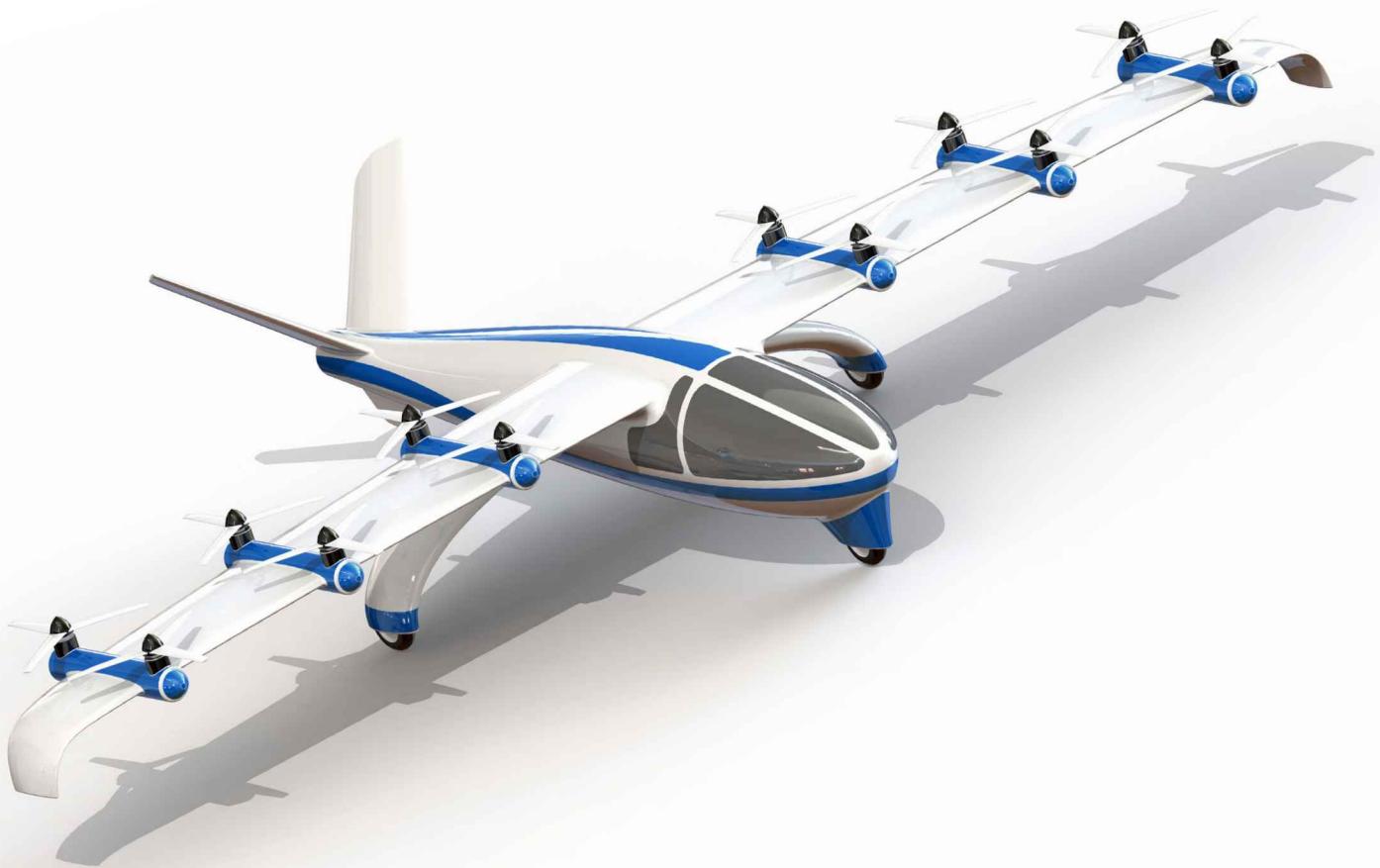


SOLIDWORKS® 2024

Advanced Techniques

Mastering Parts, Surfaces, Sheet Metal,
SimulationXpress, Top Down Assemblies,
Core & Cavity Molds



Paul Tran CSWE, CSWI



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SOLIDWORKS 2024

Advanced Techniques

Advanced Level Tutorials

Mastering Parts, Surfaces, Sheet Metal, SimulationXpress,
Top-Down Assemblies, Core - Cavity Molds & Repair Errors

Paul Tran, CSWE, CSWI



SDC Publications
P.O. Box 1334
Mission, KS 66222
913-262-2664
www.SDCpublications.com
Publisher: Stephen Schröff

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ISBN-13: 978-1-63057-635-6
ISBN-10: 1-63057-635-2

Printed and bound in the United States of America.

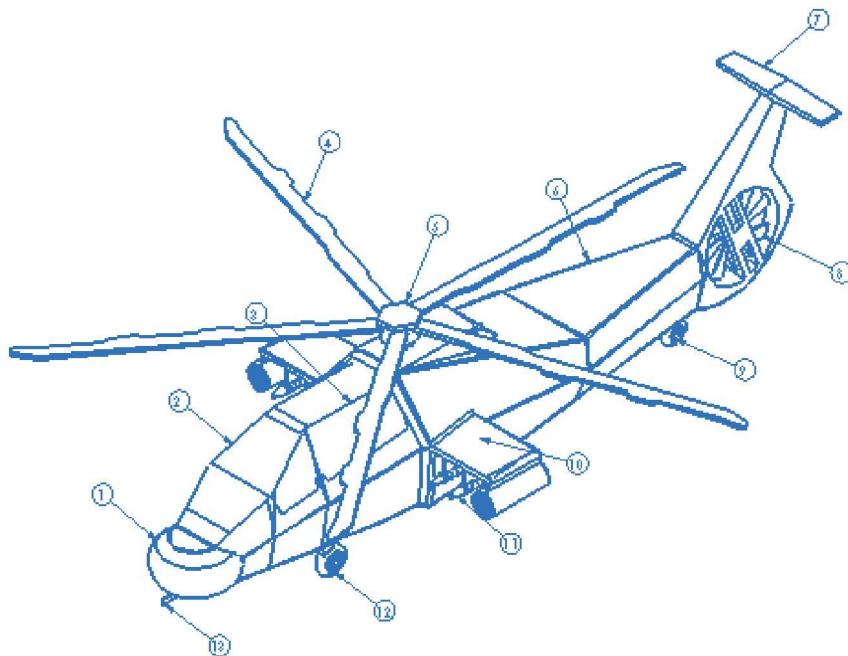
Acknowledgments

Thanks as always to my wife Vivian and my daughter Lani for always being there and providing support and honest feedback on all the chapters in the textbook.

Additionally, thanks to Peter Douglas for writing the foreword.

I also have to thank SDC Publications and the staff for its continuing encouragement and support for this edition of **SOLIDWORKS 2024 Advanced Techniques**. Thanks also to Tyler Bryant for putting together such a beautiful cover design.

Finally, I would like to thank you, our readers, for your continued support. It is with your consistent feedback that we were able to create the lessons and exercises in this book with more detailed and useful information.



Foreword

I first met Paul Tran when I was busy creating another challenge in my life. I needed to take a vision from one man's mind, understand what the vision looked like, how it was going to work and comprehend the scale of his idea. My challenge was I was missing one very important ingredient, a tool that would create a design with all the moving parts

Research led me to discover a great tool, SOLIDWORKS. It claimed to allow one to make 3D components, in picture quality, on a computer, add in all moving parts, assemble it, and make it run, all before money was spent on bending steel and buying parts that may not fit together. I needed to design and build a product with thousands of parts, make them all fit and work in harmony with millimeters tolerance. The possible cost implications of failed experimentation were daunting.

To my good fortune, one company's marketing strategy of selling a product without an instruction manual and requiring one to attend an instructional class to get it, led me to meet a communicator who made it all seem so simple.

Paul Tran has worked with and taught SOLIDWORKS as his profession for 35 years. Paul knows the SOLIDWORKS product and manipulates it like a fine musical instrument. I watched Paul explain the unexplainable to baffled students with great skill and clarity. He taught me how to navigate the intricacies of the product so that I could use it as a communication tool with skilled engineers. ***He teaches the teachers.***

I hired Paul as a design engineering consultant to create the machinery equipment with thousands of parts for my company's product. Paul Tran's knowledge and teaching skill has added immeasurable value to my company. When I read through the pages of these manuals, I now have an "instant replay" of his communication skill with the clarity of having him looking over my shoulder - ***continuously.*** We can now design, prove and build our product and know it will always work and not fail. Most important of all, Paul Tran helped me turn a blind man's vision into reality and a monument to his dream.

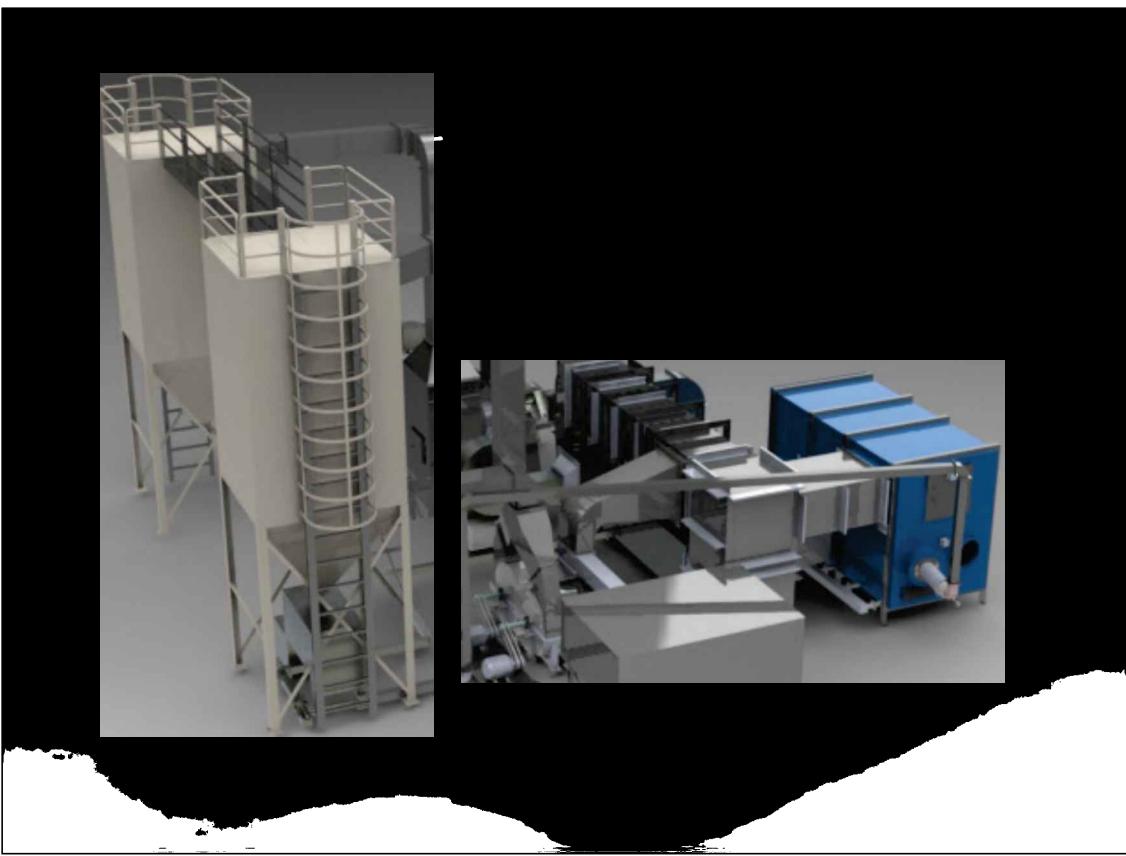
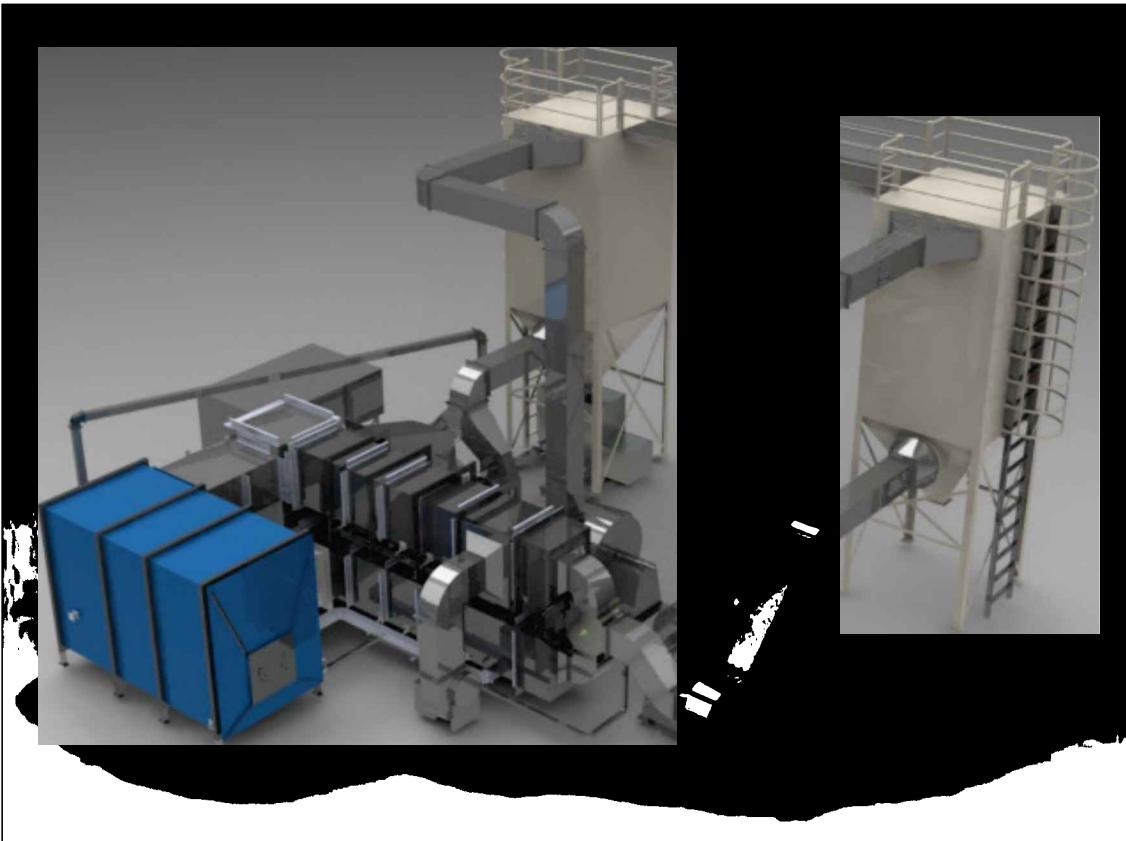
Thanks Paul.

These books will make dreams come true and help visionaries change the world.

Peter J. Douglas

CEO, Cake Energy, LLC





Images courtesy of C.A.K.E. Energy Corp., designed by Paul Tran

Author's Note

SOLIDWORKS 2024 Basic Tools, Intermediate Skills, and Advanced Techniques is comprised of lessons and exercises based on the author's extensive knowledge of this software. Paul has 35 years of experience in the fields of mechanical and manufacturing engineering. He spent 2/3 of those years teaching and supporting the SOLIDWORKS software and its add-ins.

As an active Sr. SOLIDWORKS instructor and design engineer, Paul has worked and consulted with hundreds of reputable companies including IBM, Intel, NASA, US- Navy, Boeing, Disneyland, Medtronic, Edwards Lifesciences, Terumo, Kingston and many more. Today, he has trained more than 14,000 engineering professionals, and given guidance to nearly $\frac{1}{2}$ of the number of Certified SOLIDWORKS Professionals and Certified SOLIDWORKS Expert (CSWP & CSWE) in the state of California.

Every lesson and exercise in this book was created based on real world projects. Each of these projects have been broken down and developed into easy and comprehendible steps for the reader. Learn the fundamentals of SOLIDWORKS at your own pace, as you progress from simple to more complex design challenges. Furthermore, at the end of every chapter, there are self-test questionnaires to ensure that the reader has gained sufficient knowledge from each section before moving on to more advanced lessons.

Paul believes that the most effective way to learn the “world’s most sophisticated software” is to learn it inside and out, create everything from the beginning, and take it step by step. This is what the **SOLIDWORKS 2024 Basic Tools, Intermediate Skills & Advanced Techniques** manuals are all about.

About the Training Files

The files for this textbook are available for download on the publisher's website at www.SDCpublications.com/downloads/978-1-63057-635-6. They are organized by the chapter numbers and the file names that are normally mentioned at the beginning of each chapter or exercise. In the **Built Parts** folder, you will also find copies of the parts, assemblies and drawings that were created for cross referencing or reviewing purposes.

It would be best to make a copy of the content to your local hard drive and work from these documents; you can always go back to the original training files location at any time in the future, if needed.

Who this book is for

This book is for the mid-level user, who is already familiar with the SOLIDWORKS program. It is also a great resource for the more CAD literate individuals who want to expand their knowledge of the different features that SOLIDWORKS 2024 has to offer.

The organization of the book

The chapters in this book are organized in the logical order in which you would learn the SOLIDWORKS 2024 program. Each chapter will guide you through some different tasks, from navigating through the user interface, to exploring the toolbars, from some simple 3D modeling and move on to more complex tasks that are common to all SOLIDWORKS releases. There is also a self-test questionnaire at the end of each chapter to ensure that you have gained sufficient knowledge before moving on to the next chapter.

The conventions in this book

This book uses the following conventions to describe the actions you perform when using the keyboard and mouse to work in SOLIDWORKS 2024:

Click: means to press and release the mouse button. A click of a mouse button is used to select a command or an item on the screen.

Double-click: means to quickly press and release the left mouse button twice. A double mouse click is used to open a program or show the dimensions of a feature.

Right-click: means to press and release the right mouse button. A right mouse click is used to display a list of commands, a list of shortcuts that is related to the selected item.

Click and Drag: means to position the mouse cursor over an item on the screen and then press and hold down the left mouse button; still holding down the left button, move the mouse to the new destination and release the mouse button. Drag and drop makes it easy to move things around within a SOLIDWORKS document.

Bolded words: indicated the action items that you need to perform.

Italic words: Side notes and tips that give you additional information, or to explain special conditions that may occur during the course of the task.

Numbered Steps: indicates that you should follow these steps in order to successfully perform the task.

Icons: indicates the buttons or commands that you need to press.

SOLIDWORKS 2024

SOLIDWORKS 2024 is a program suite, or a collection of engineering programs, that can help you design better products faster. SOLIDWORKS 2024 contains different combinations of programs; some of the programs used in this book may not be available in your suites.

Start and exit SOLIDWORKS

SOLIDWORKS allows you to start its program in several ways. You can either double click on its shortcut icon on the desktop or go to the Start menu and select the following: All Programs / SOLIDWORKS 2024 / SOLIDWORKS or drag a SOLIDWORKS document and drop it on the SOLIDWORKS shortcut icon.

Before exiting SOLIDWORKS, be sure to save any open documents, and then click File/Exit; you can also click the X button on the top right of your screen to exit the program.

Using the Toolbars

You can use toolbars to select commands in SOLIDWORKS rather than using the drop-down menus. Using the toolbars is normally faster. The toolbars come with commonly used commands in SOLIDWORKS, but they can be customized to help you work more efficiently.

To access the toolbars, either right click in an empty spot on the top right of your screen or select View / Toolbars.

To customize the toolbars, select Tools / Customize. When the dialog pops up, click on the Commands tab, select Category, and then drag an icon out of the dialog box and drop it on a toolbar that you want to customize. To remove an icon from a toolbar, drag an icon out of the toolbar and drop it into the dialog box.

Using the task pane

The task pane is normally kept on the right side of your screen. It displays various options like SOLIDWORKS resources, Design library, File explorer, Search, View palette, Appearances and Scenes, Custom properties, Built-in libraries, Technical alerts, and news, etc.

The task pane provides quick access to any of the mentioned items by offering the drag and drop function to all of its contents. You can see a large preview of a SOLIDWORKS document before opening it. New documents can be saved in the task pane at any time, and existing documents can also be edited and re-saved. The task pane can be resized, closed, or moved to a different location on your screen if needed.



Become a CSWP – Certified SOLIDWORKS Professional

Demonstrate and validate your competency and knowledge in SOLIDWORKS software by achieving SOLIDWORKS certifications.

A Certified SOLIDWORKS Professional is an individual who has successfully passed our advanced skills examination. Each CSWP has proven their ability to design and analyze parametric parts and moveable assemblies using a variety of complex features in SOLIDWORKS software.

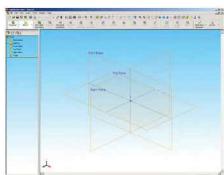
Standing out amongst SOLIDWORKS users from around the world can be challenging. Earning a SOLIDWORKS Certification can help you get a job, keep a job, or possibly move up in your current job. Be a part of our growing community of certified users. The certification center is your place to validate your certificate. This manual includes the CSWP-Exam preparation materials to help get you started.



For more information, log on to: https://solidworks.virtualtester.com/#home_button

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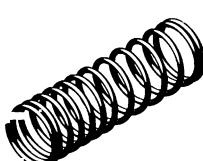


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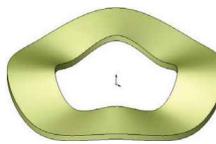
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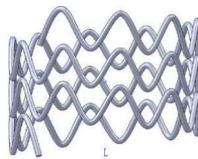


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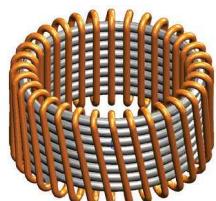
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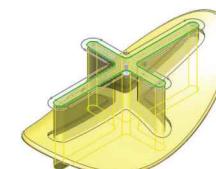
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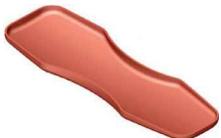


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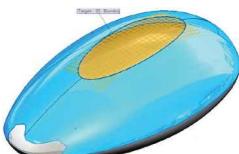
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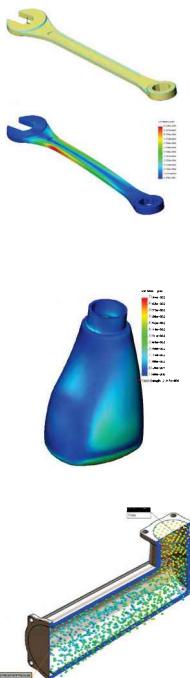
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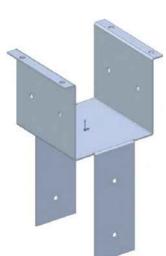


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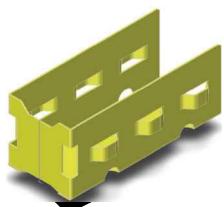
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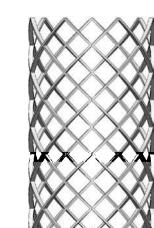
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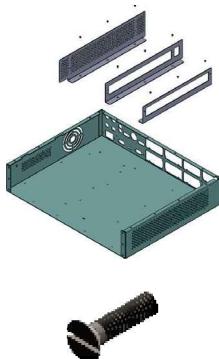
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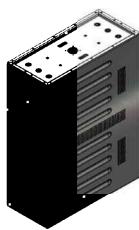
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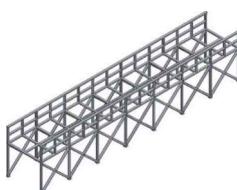
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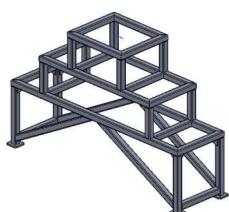
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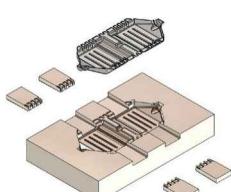
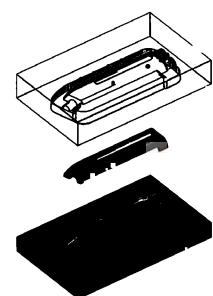
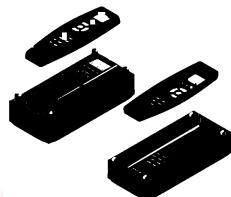


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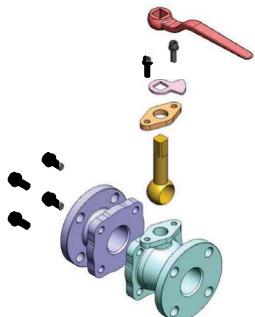
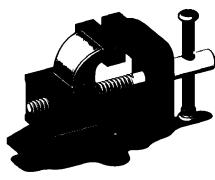


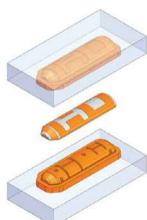
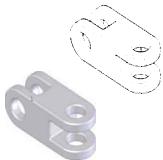
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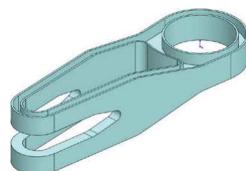
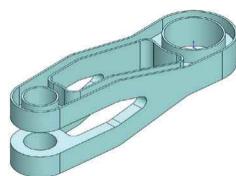
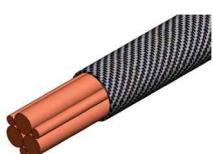
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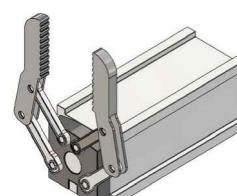
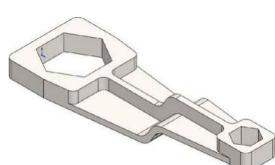
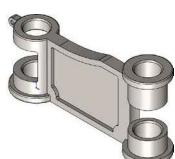
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Glossary, Index, and SOLIDWORKS 2024 Quick-Guide

Quick Reference Guide to SOLIDWORKS 2024 Command Icons and Toolbars.

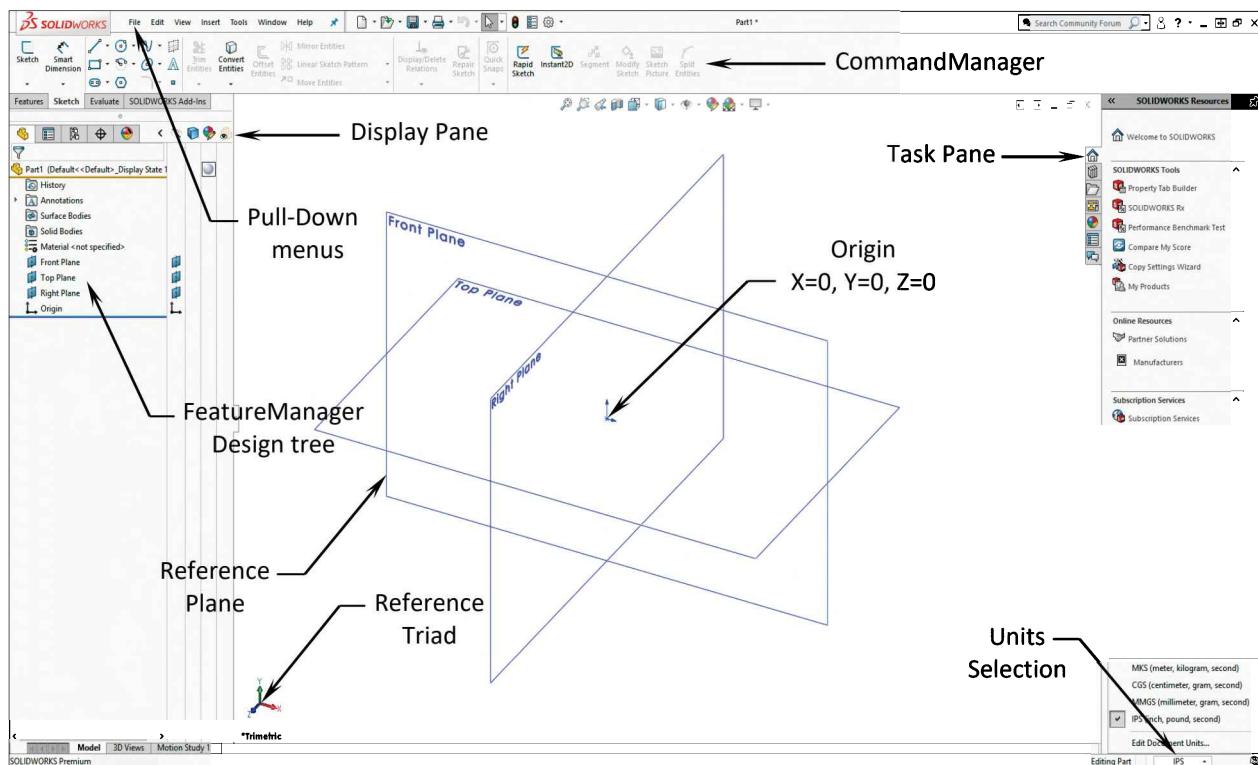


INTRODUCTION

SOLIDWORKS User Interface

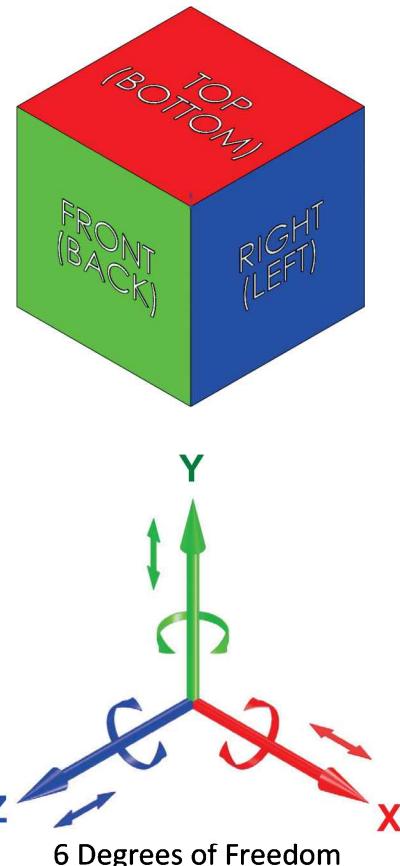
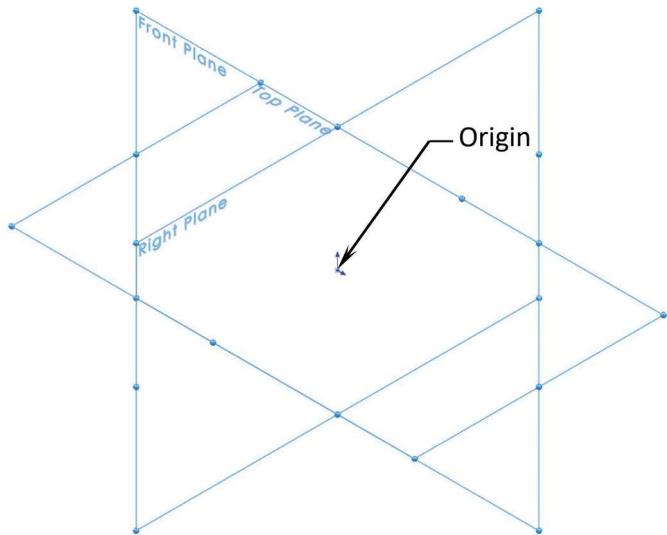


The SOLIDWORKS 2024 User Interface



The 3 reference planes:

The Front, Top and the Right plane are 90° apart.
They share the same center point called Origin.

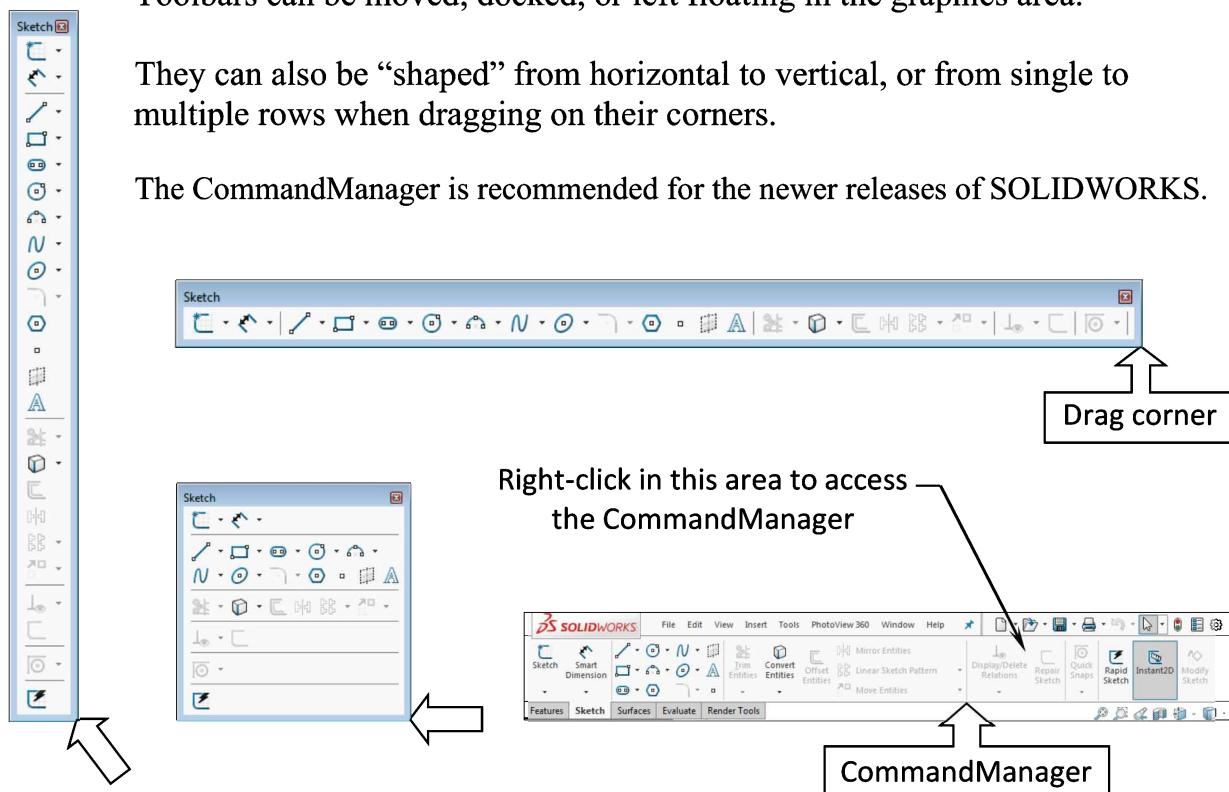


The Toolbars:

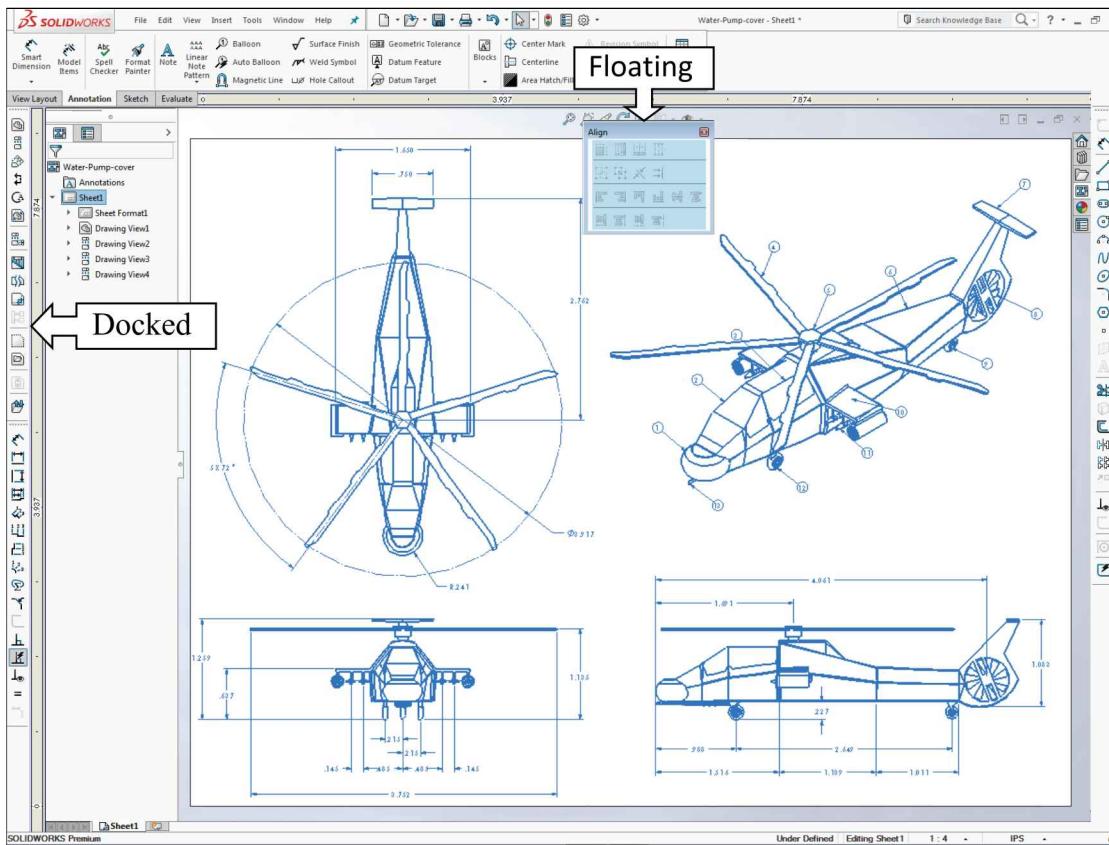
Toolbars can be moved, docked, or left floating in the graphics area.

They can also be “shaped” from horizontal to vertical, or from single to multiple rows when dragging on their corners.

The CommandManager is recommended for the newer releases of SOLIDWORKS.

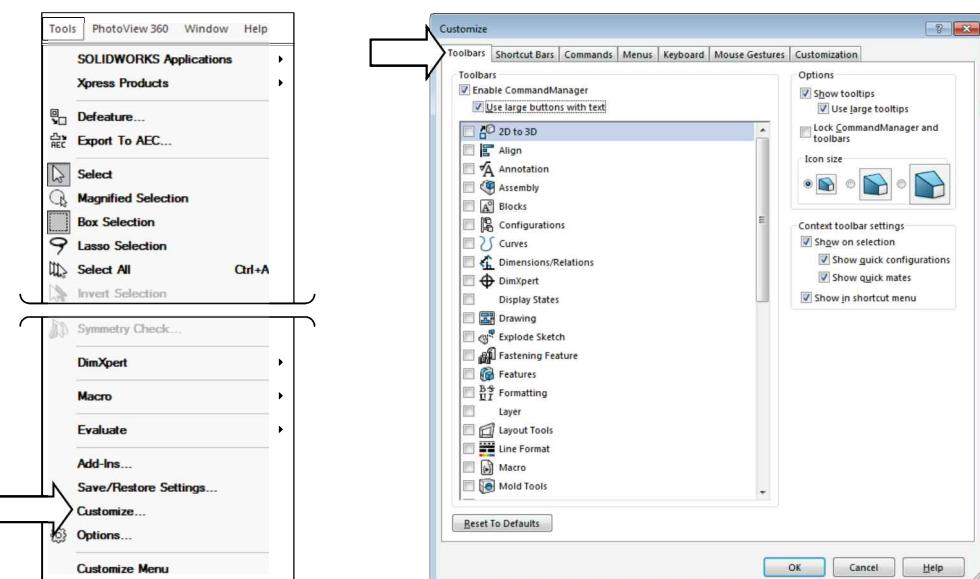


If the CommandManager is not used, toolbars can be docked or left floating.



Toolbars can be toggled on or off by activating or de-activating their check boxes:

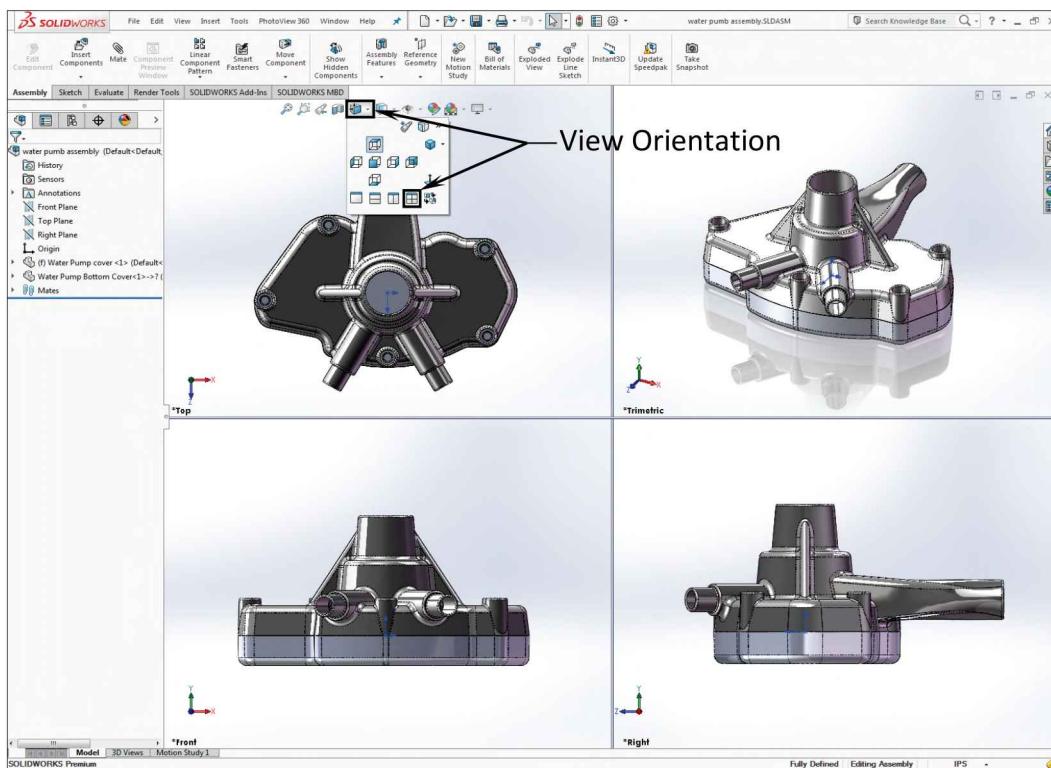
Select Tools / Customize / Toolbars tab.



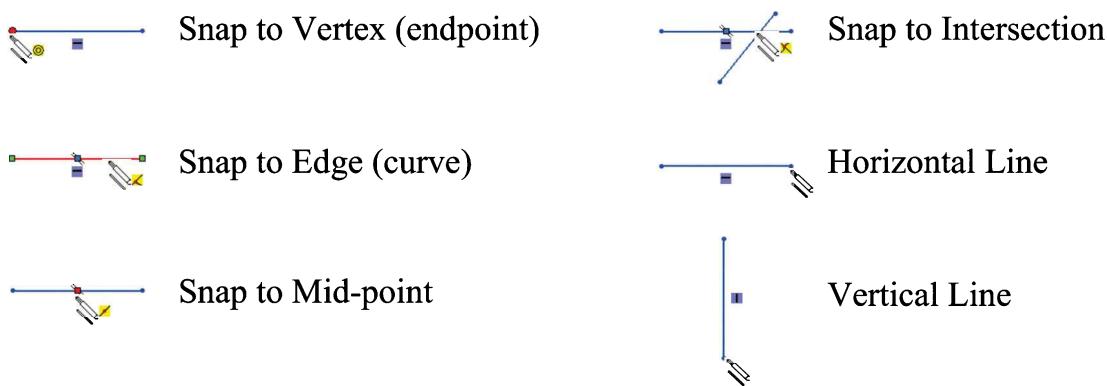
The icons in the toolbars can be enlarged when its check box is selected

Large icons

The View ports: You can view or work with SOLIDWORKS model or an assembly using one, two or four view ports.



Some of the System Feedback symbols (Inference pointers):



The Status Bar: (View / Status Bar)

Displays the status of the sketch entity using different colors to indicate:

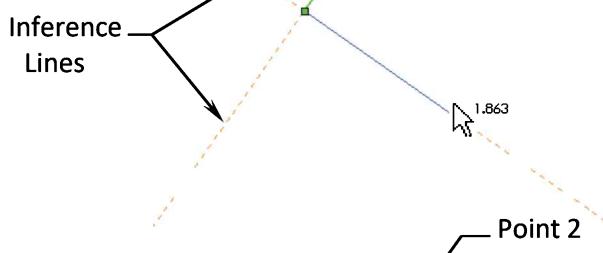
Green = Selected	Blue = Under defined
Black = Fully defined	Red = Over defined

2D Sketch examples:



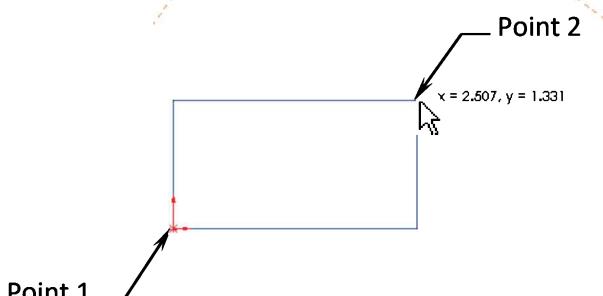
Click-Drag-Release: Single entity.

(Click Point 1, hold the mouse button, drag to Point 2 and release.)



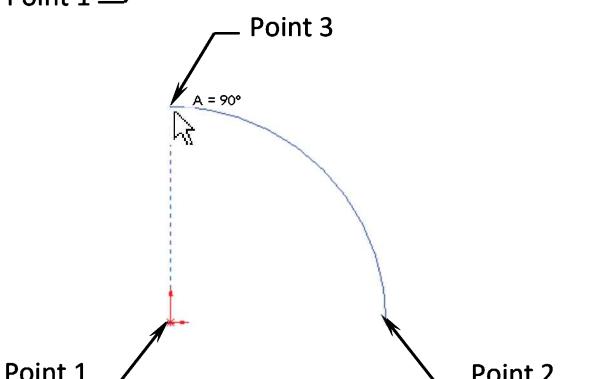
Click-Drag-Release: Continuous multiple entities.

(See Inference Lines appear when the sketch ties are Parallel, Perpendicular, or Tangent to each other.)



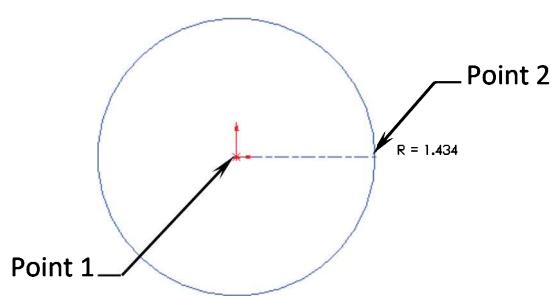
Click-Drag-Release: Single Rectangle

(Click point 1, hold the mouse button, drag to Point 2 and release.)



Click-Drag-Release: Single CenterPoint Arc

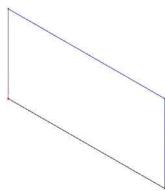
(Click point 1, hold the mouse button and drag to Point 2, release; then drag to Point 3 and release.)



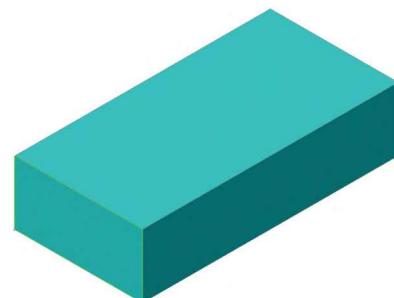
Click-Drag-Release: Single Circle

(Click point 1 [center of circle], hold the mouse button, drag to Point 2 [Radius] and release.)

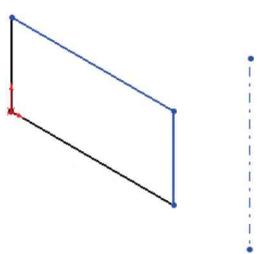
3D Feature examples:



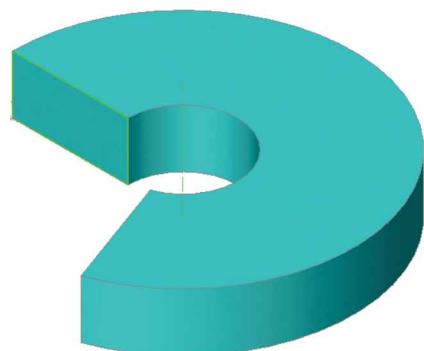
2D sketch



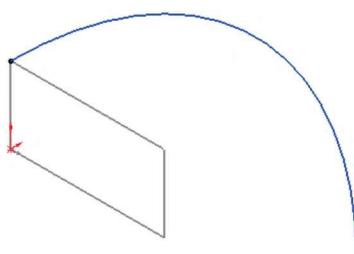
3D feature



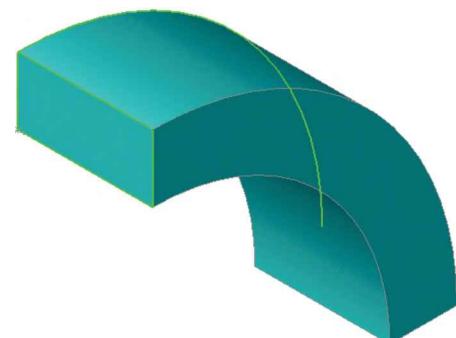
2D sketch



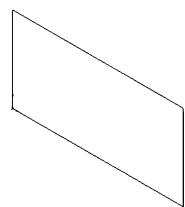
3D feature



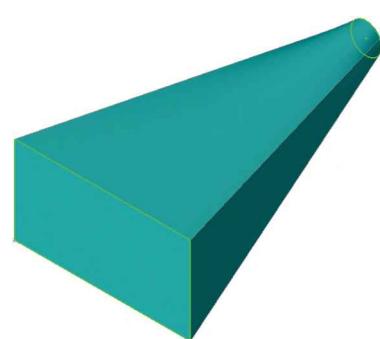
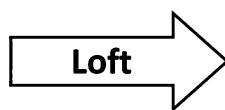
2D sketch



3D feature

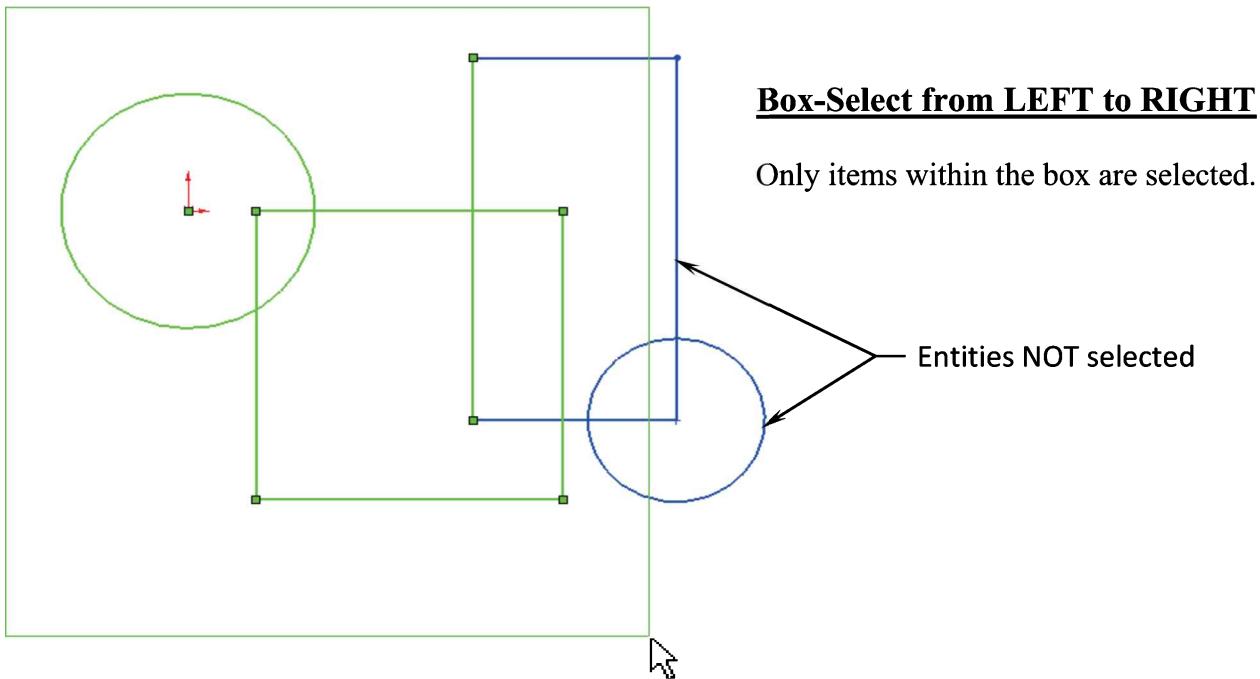


2D sketch



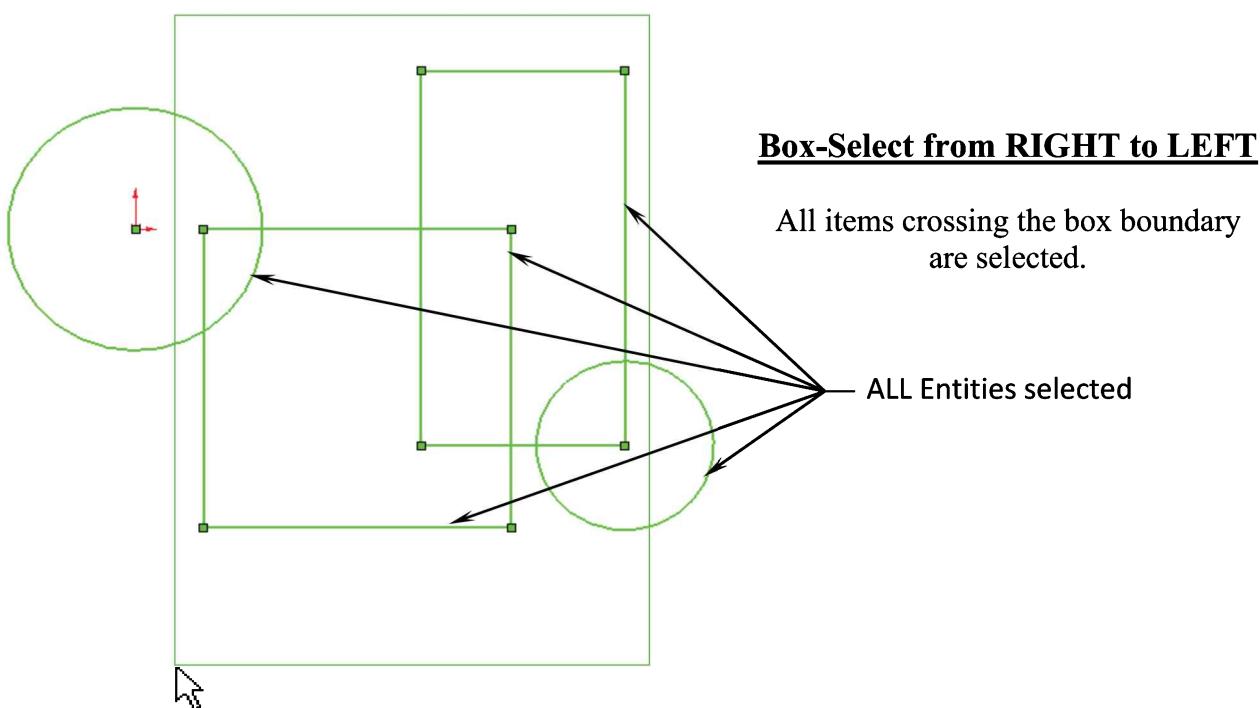
3D feature

Box-Select: Use the Select Pointer  to drag a selection box around items.



Box-Select from LEFT to RIGHT

Only items within the box are selected.



Box-Select from RIGHT to LEFT

All items crossing the box boundary are selected.

ALL Entities selected

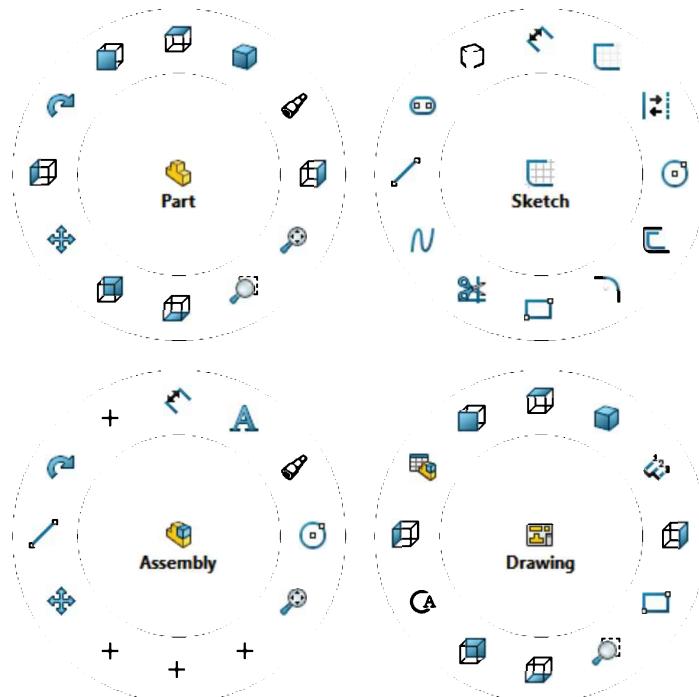
The default geometry type selected is as follows:

* Part documents – edges * Assembly documents – components * Drawing documents - sketch entities, dims & annotations. * To select multiple entities, hold down **Ctrl** while selecting after the first selection.

The Mouse Gestures for Parts, Sketches, Assemblies and Drawings

Similar to a keyboard shortcut, you can use a Mouse Gesture to execute a command. A total of 12 keyboard shortcuts can be independently mapped and stored in the Mouse Gesture Guides.

To activate the Mouse Gesture Guide, **right-click-and-drag** to see the current 12 gestures, then simply select the command that you want to use.

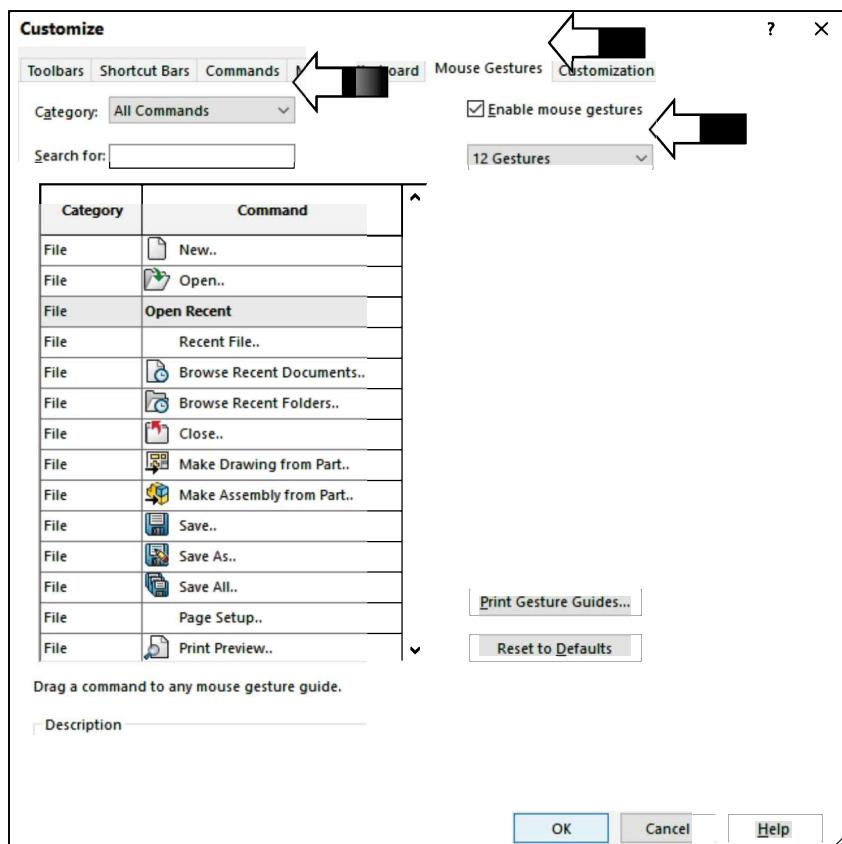


To customize the Mouse Gestures and include your favorite shortcuts, go to **Tools / Customize**.

From the **Mouse Gestures** tab select **All Commands**.

Click the **Enable Mouse Gestures** checkbox.

Select the **12 Gestures** option (arrow).



Customizing Mouse Gestures

To reassign a mouse gesture:

- With a document open, click **Tools > Customize** and select the **Mouse Gestures** tab. The tab displays a list of tools and macros. If a mouse gesture is currently assigned to a tool, the icon for the gesture appears in the appropriate column for the command.

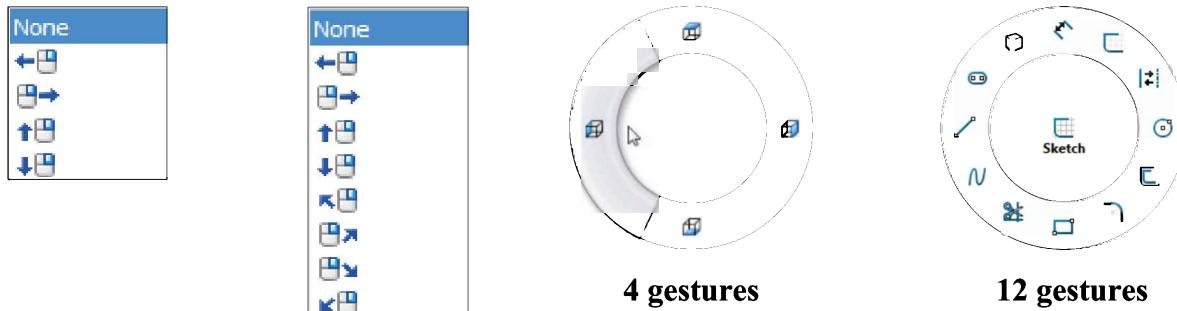
For example, by default, the right mouse gesture is assigned to the Right tool for parts and assemblies, so the icon for that gesture () appears in the Part and Assembly columns for that tool.

To filter the list of tools and macros, use the options at the top of the tab. By default, four mouse gesture directions are visible in the Mouse Gestures tab and available in the mouse gesture guide. Select 8 gestures to view and reassign commands for eight gesture directions.

- Find the row for the tool or macro you want to assign to a mouse gesture and click in the cell where that row intersects the appropriate column.

For example, to assign Make Drawing from Part to the Part column, click in the cell where the Make-Drawing-from-Part row and the Part column intersect.

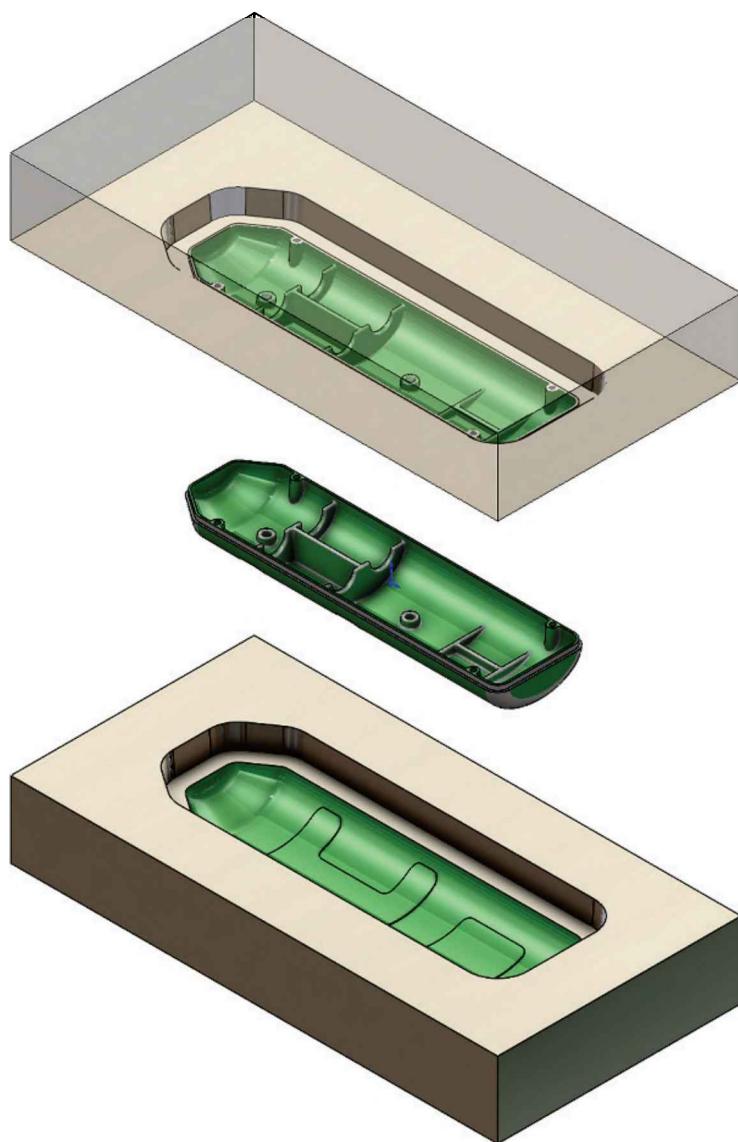
A list of either 4 or 12 gesture directions appears as shown, depending on whether you have the 4 gestures, or 12 gestures option selected.



Some tools are not applicable to all columns, so the cell is unavailable, and you cannot assign a mouse gesture. For example, you cannot assign a mouse gesture for Make Drawing from Part in the Assembly or Drawing columns.

- Select the mouse gesture direction you want to assign from the list. The mouse gesture direction is reassigned to that tool and its icon appears in the cell.

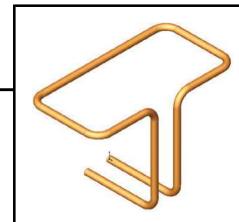
- Click OK.



Designed with SOLIDWORKS 2024, SP0

CHAPTER 1

Introduction to 3D Sketch



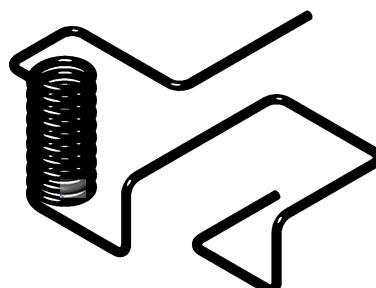
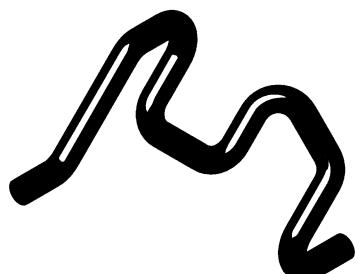
Introduction to 3D Sketch

SOLIDWORKS has 3D sketch capabilities. A 3D sketch consists of lines and arcs in series and splines. You can use a 3D sketch as a sweep path, as a guide curve for a loft or sweep, a centerline for a loft, or as one of the key entities in a piping system. Geometric relations can also be added to 3D Sketches.

Parameters

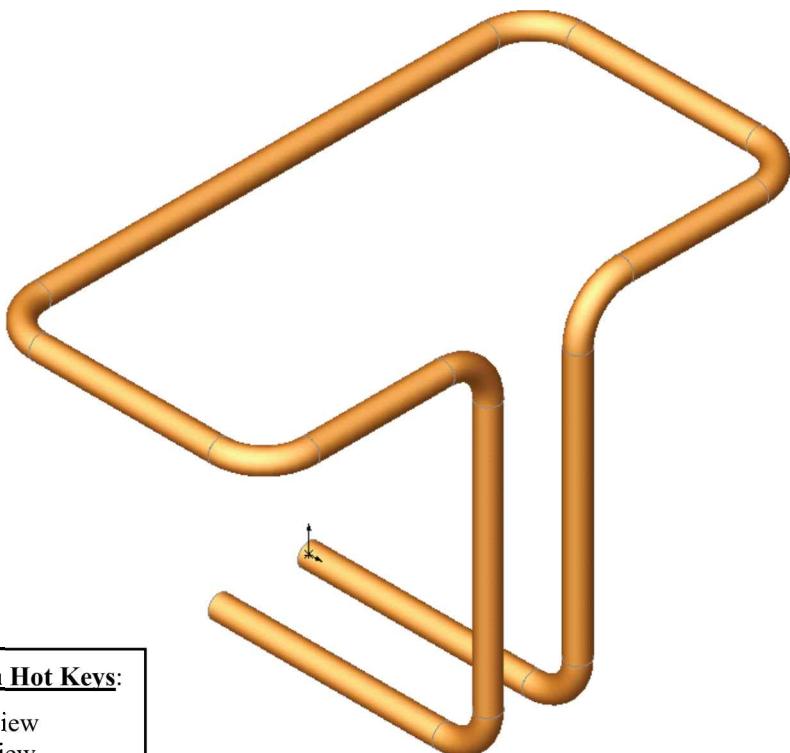
-  **X Coordinate**
-  **Y Coordinate**
-  **Z Coordinate**
-  **Curvature** (Spline curvature at the frame point)
-  **Tangency** (In the XY plane)
-  **Tangency** (In the XZ plane)
-  **Tangency** (In the YZ plane)

Space Handle



When working in a 3D sketch, a graphical assistant is provided to help you maintain your orientation while you sketch on several planes. This assistant is called a **space handle**. The space handle appears when the first point of a line or spline is defined on a selected plane. Using the space handle you can select the axis along which you want to sketch.

Introduction to 3D Sketch



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: **ANSI**
Units: **INCHES** – 3 Decimals

Tools Needed:



3D Sketch



2D Sketch



Sketch Line



Circle



Dimension



Add Geometric Relations



Sketch Fillet



Tab Key



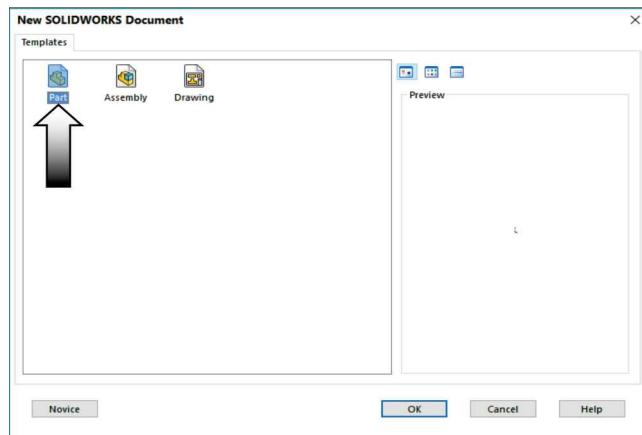
Base/ Boss Sweep

1. Starting a new part file:

Click **File / New**.

Select the **Part** template and click **OK**.

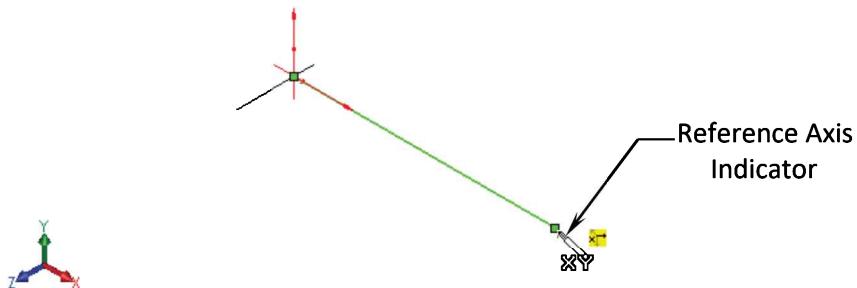
Set the Units to **IPS, 3 decimals**.



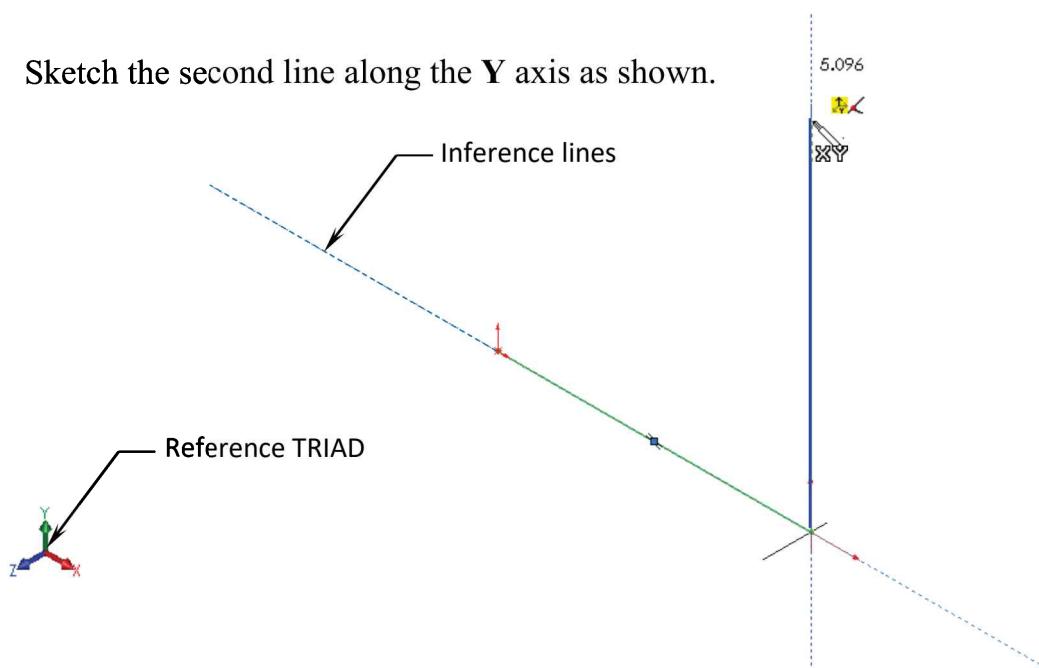
2. Creating a 3D Sketch:

Click or select **Insert / 3D Sketch** and change to **Isometric view** (Control+7).

Select the **Line** tool and sketch the first line along the **X** direction. A yellow symbol appears next to the mouse cursor when the line is drawn along the X axis; this indicates an **Along X** relation (horizontal) is being added to the line.



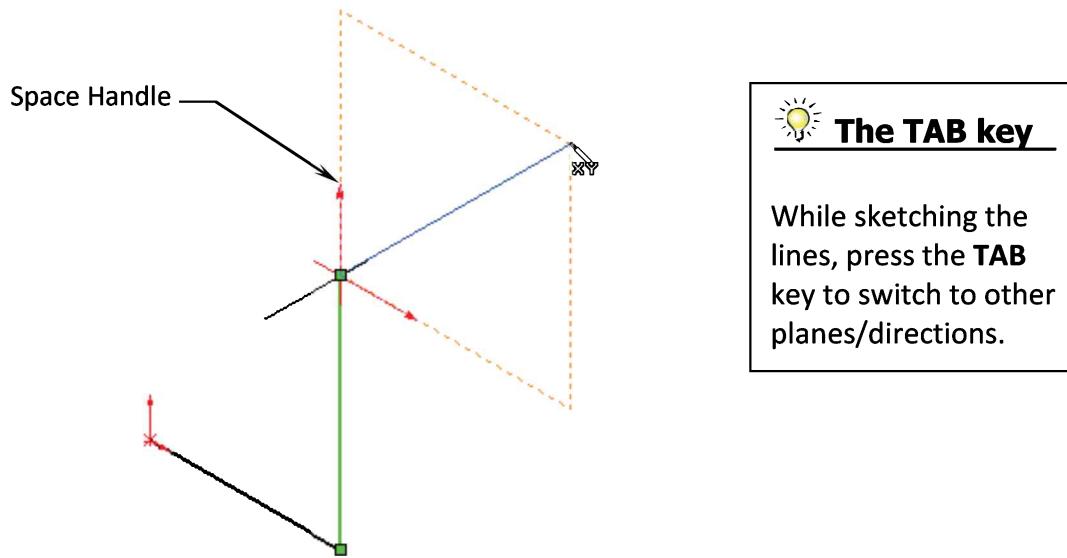
Sketch the second line along the **Y** axis as shown.



3. Changing direction:

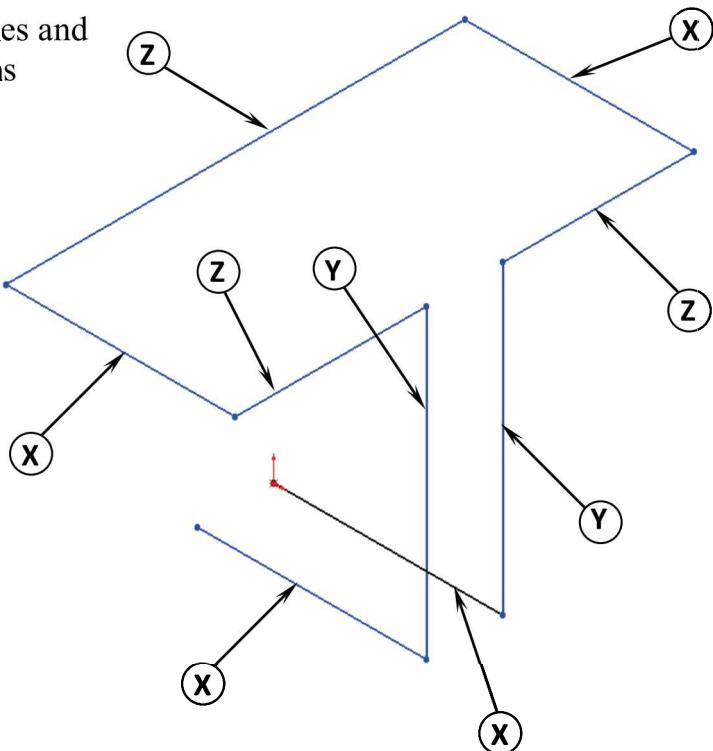
By default your sketch is relative to the default coordinate system in the model.

To switch to one of the other two default planes, press the **TAB** key and the reference origin of the current sketch plane is displayed on that plane.



4. Completing the profile:

Sketch the other lines and follow the directions as labeled, press the **TAB** key if needed to change the direction.

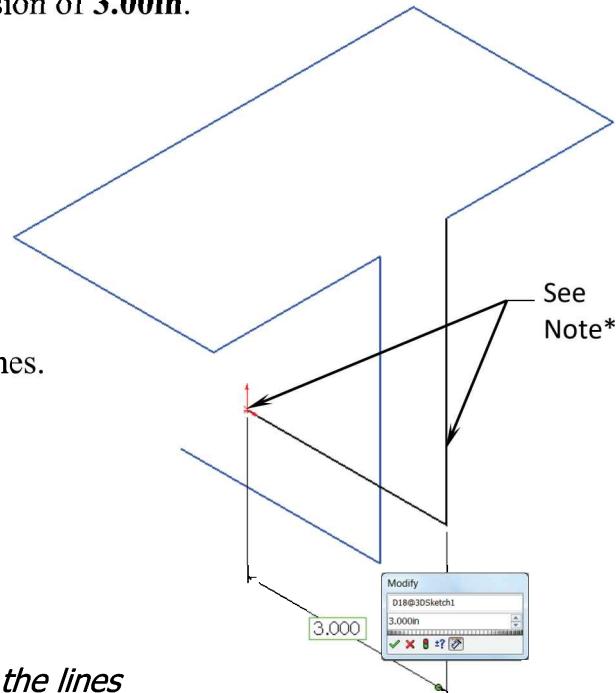


5. Adding dimensions:

Click **Smart Dimension** or select **Tools / Dimensions / Smart Dimension**.

Click the first line and enter a dimension of **3.00in**.

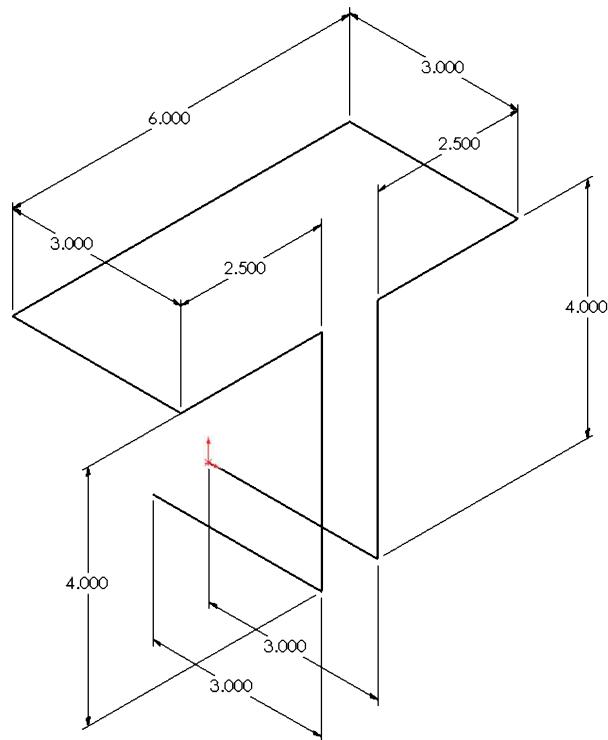
There is not a general sequence to follow when adding dimensions, so for this lesson, add the dimensions in the same order you sketched the lines.



Note: To make the dimensions parallel to the lines as shown, select the line and an endpoint instead of selecting just the endpoints.

Continue adding the dimensions to fully define the 3D sketch as shown.

Rearrange the dimensions so they are easy to read, which will make editing a little easier later on.



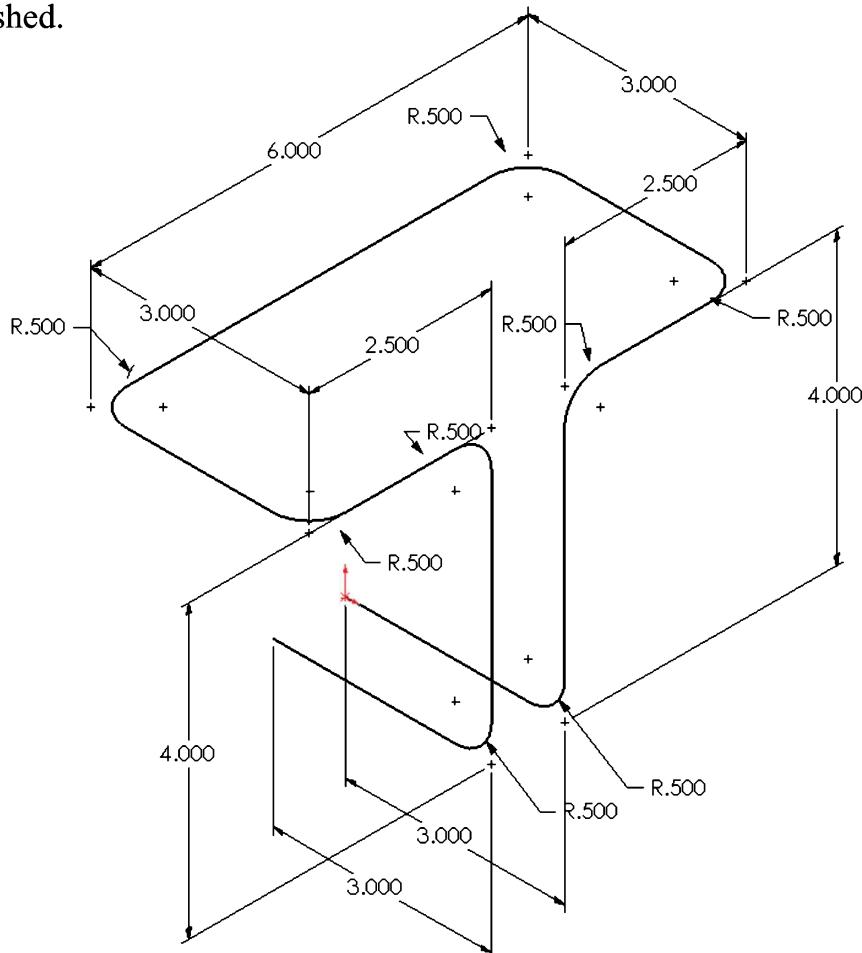
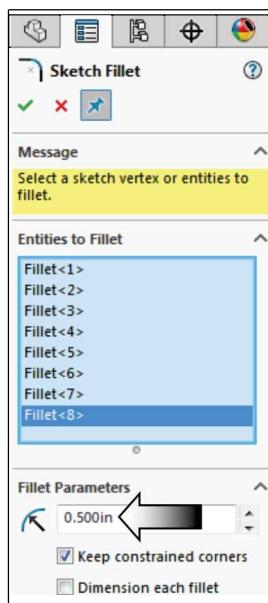
6. Adding the Sketch Fillets:

Click **Fillet** on the Features toolbar or select **Tools / Sketch Tools / Fillet**.

Add **.500"** fillets to all the intersections as indicated.

Enable the **Keep Constrained Corner** check box (to maintain the virtual intersection point if the vertex has dimensions or relations).

Click **OK** when finished.

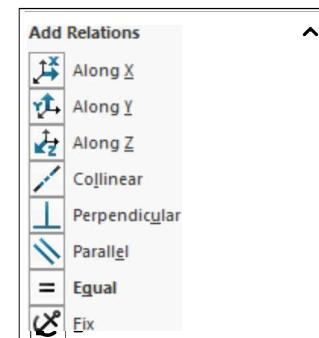


Exit the 3D Sketch
or press **Control + Q**.



Geometric Relations

Geometric Relations such as **Along X**, **Y**, **Z** and **Equal** can also be used to replace some of the duplicate dimensions.



7. Creating the Swept feature:

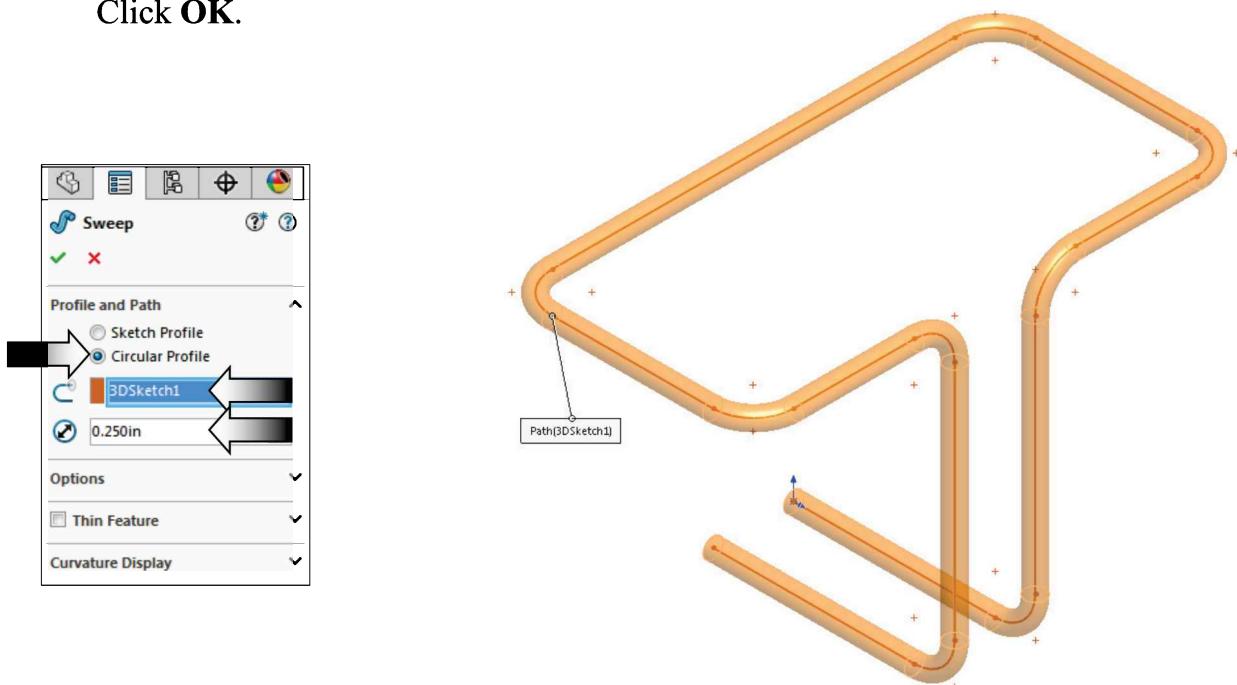
The Circular Profile option allows you to create a solid rod or hollow tube along a path, edge, or curve directly on a model without having to sketch the circular profile. This enhancement is available for Swept Boss/Base, Swept Cut, and Swept Surface features.

Click  or select **Insert / Boss-Base / Sweep**.

Select the **Circle Profile** option and enter **.250in** for the diameter of the profile .

Select the **3D Sketch** for Sweep Path (**3DSketch1**).

Click **OK**.

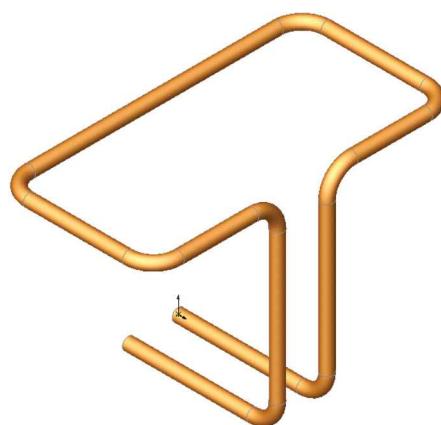


8. Saving your work:

Select **File / Save As**.

Enter: **3D Sketch** for the file name.

Click **Save**.



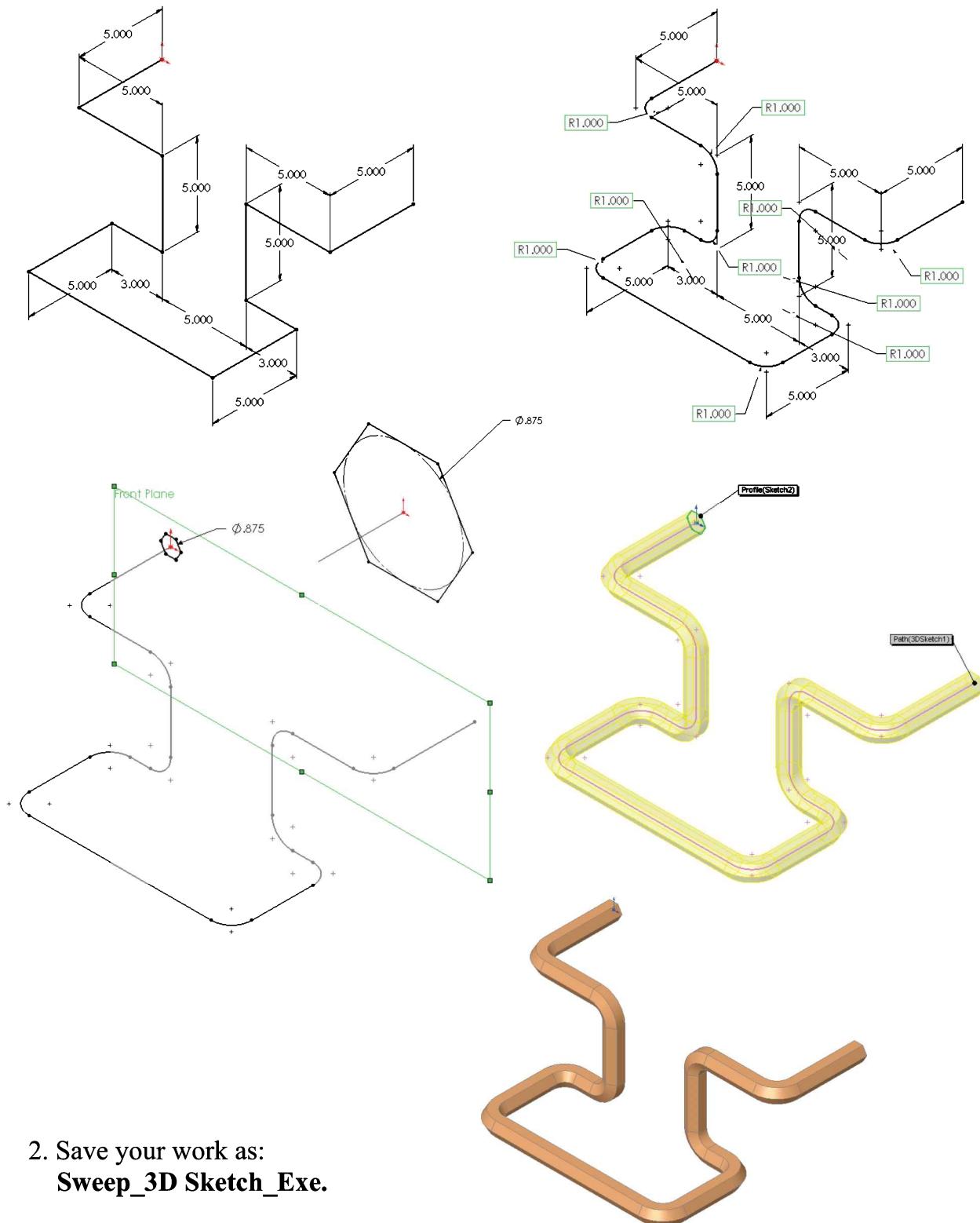
Questions for Review

1. When using 3D Sketch you do not have to pre-select a plane as you would in 2D Sketch.
 - a. True
 - b. False
2. The space handle appears only after the first point of a line is started.
 - a. True
 - b. False
3. To switch to other planes (or direction) in 3D Sketch mode, press:
 - a. Up Arrow
 - b. Down Arrow
 - c. TAB key
 - d. CONTROL key
4. Dimensions cannot be used in 3D Sketch mode.
 - a. True
 - b. False
5. Geometric Relations cannot be used in 3D Sketch mode.
 - a. True
 - b. False
6. All sketch tools in 2D Sketch are also available in 3D Sketch.
 - a. True
 - b. False
7. When adding sketch fillets, the option Keep Constrained Corner will create a virtual intersection point but will not create a radius dimension.
 - a. True
 - b. False
8. 3D Sketch entities can be used as a path in a swept feature.
 - a. True
 - b. False

1. TRUE 2. TRUE 3. C 4. FALSE 5. FALSE 6. FALSE 7. FALSE 8. TRUE

Exercise: Sweep with 3D Sketch

1. Create the part shown below using 3D Sketch.

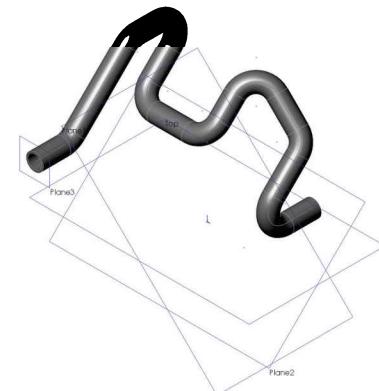


2. Save your work as:
Sweep_3D_Sketch_Exe.

Exercise: 3D Sketch & Planes

A 3D sketch normally consists of lines and arcs in series, and splines. You can use a 3D sketch as a sweep path, as a guide curve for a loft or sweep, a centerline for a loft, or as one of the key entities in a routing system.

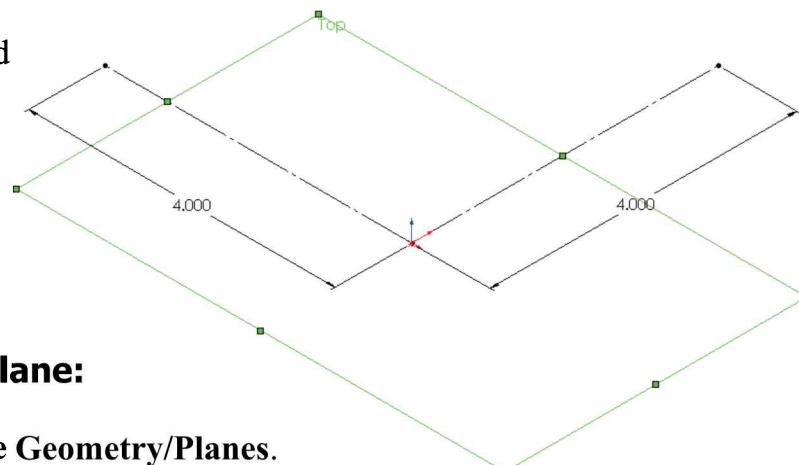
The following exercise shows how several planes can be used to help define the directions of 3D Sketch Entities.



1. Sketching the reference Pivot lines:

Select the **Top** plane and open a new sketch.

Sketch **2 Centerlines** and add dimensions as shown.

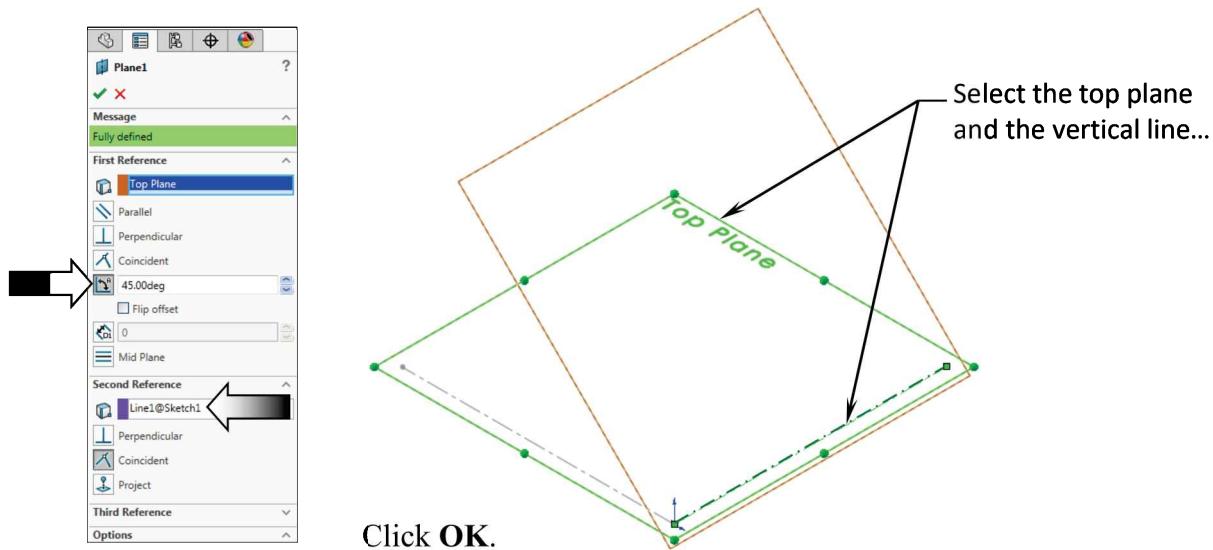


2. Creating the 1st 45° Plane:

Select **Insert/Reference Geometry/Planes**.

Click the **At Angle** button and enter **45** for Angle (arrow).

Select the **Top** plane and the **Vertical** line as noted.

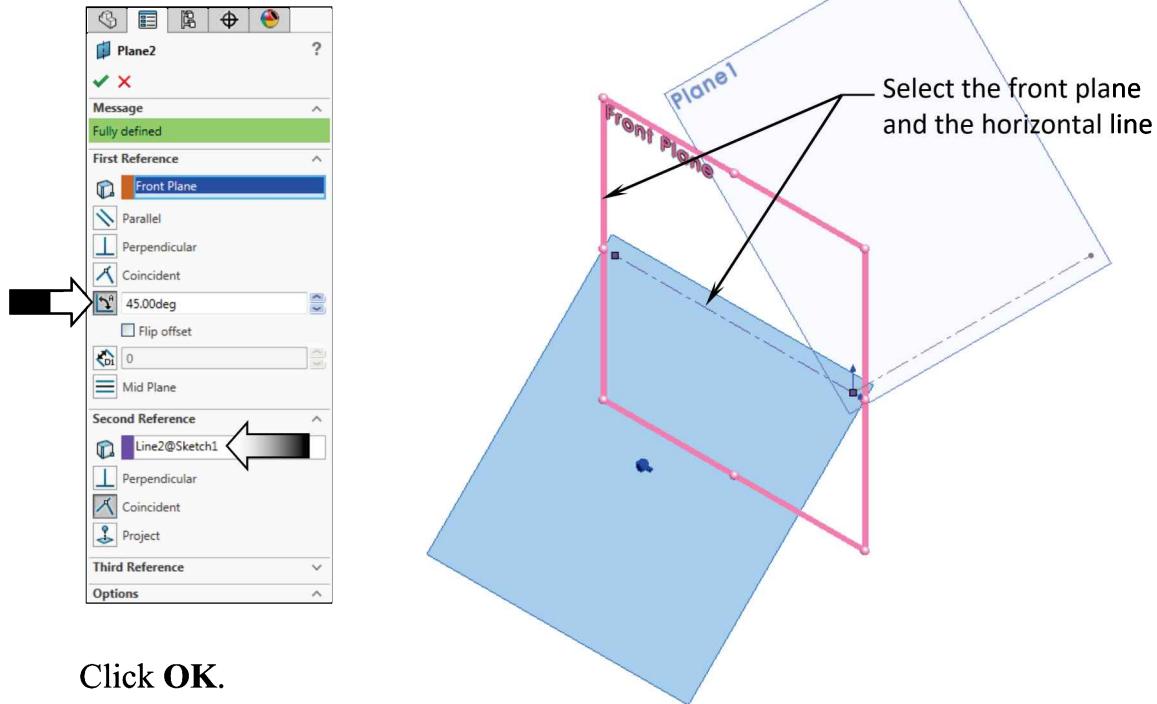


3. Creating the 2nd 45° Plane:

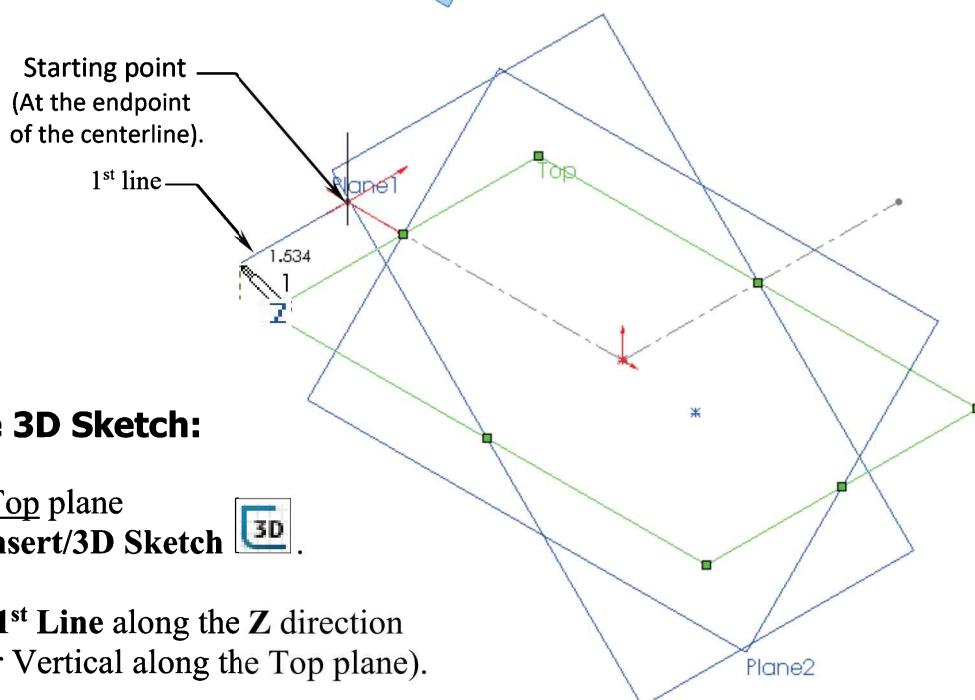
Click the **Plane** command or select **Insert/Reference Geometry/Planes** .

Click the **At Angle** option and enter **45** for Angle (arrow).

Select the **Front** plane and the **Horizontal Line** as noted.



Click **OK**.

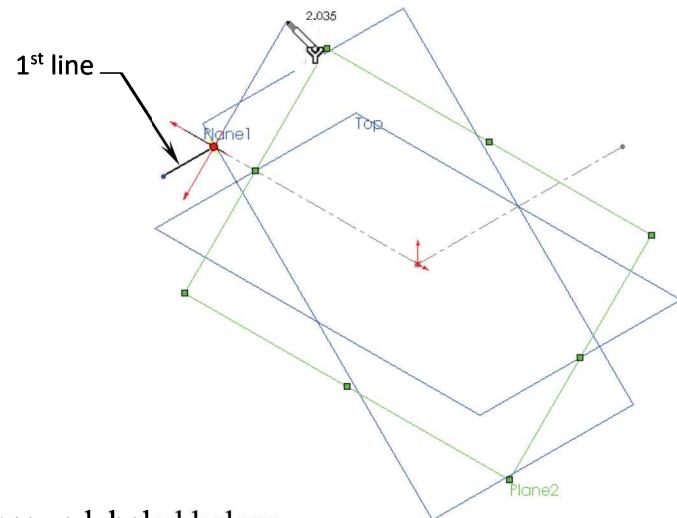
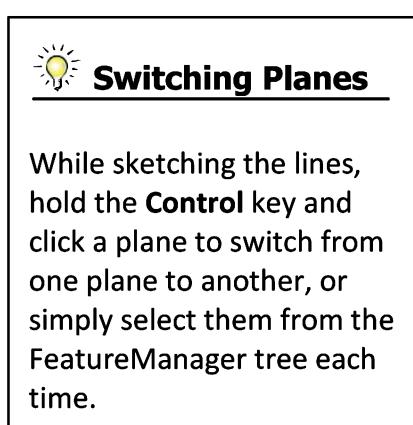


4. Creating the 3D Sketch:

Select the **Top** plane and click **Insert/3D Sketch** .

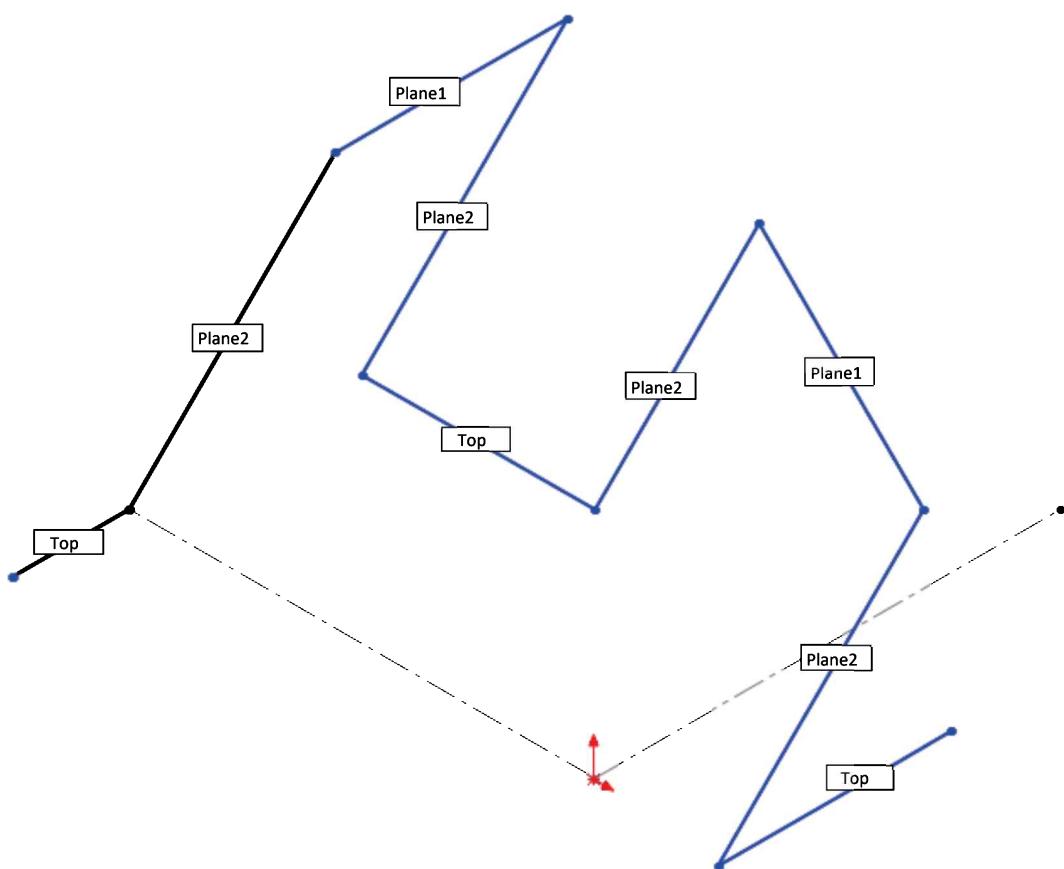
Sketch the **1st Line** along the **Z** direction as noted (or Vertical along the Top plane).

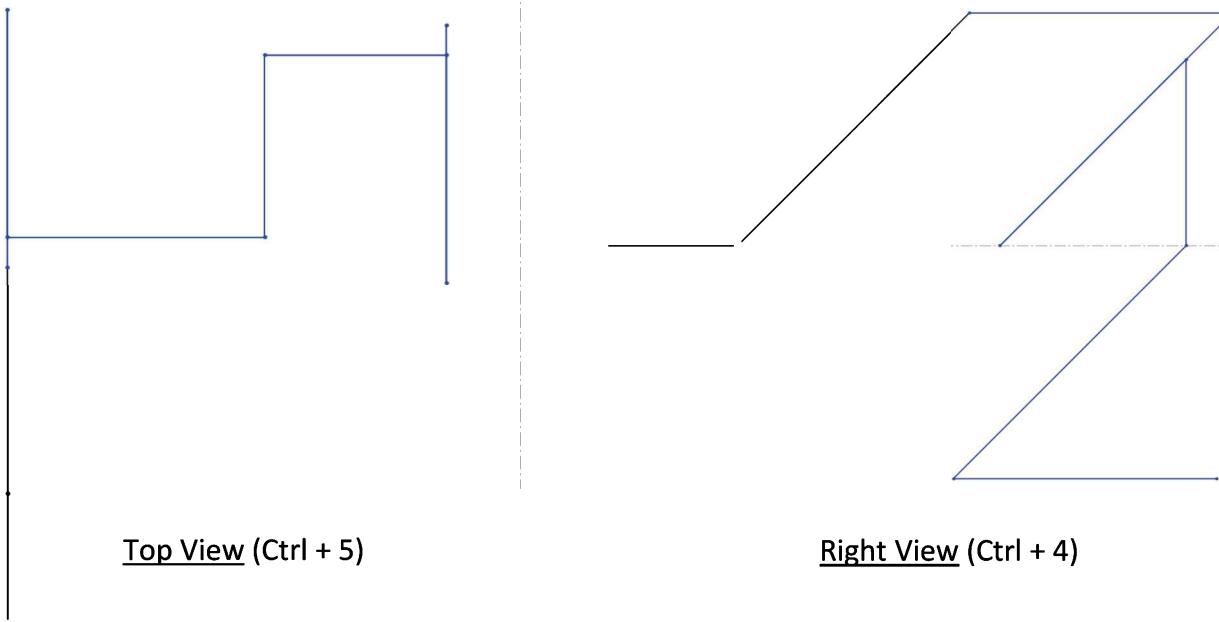
Select **Plane2** (45 deg.) from the FeatureManager tree and Sketch the **2nd Line** along the Y direction (watch the cursor feedback symbol).



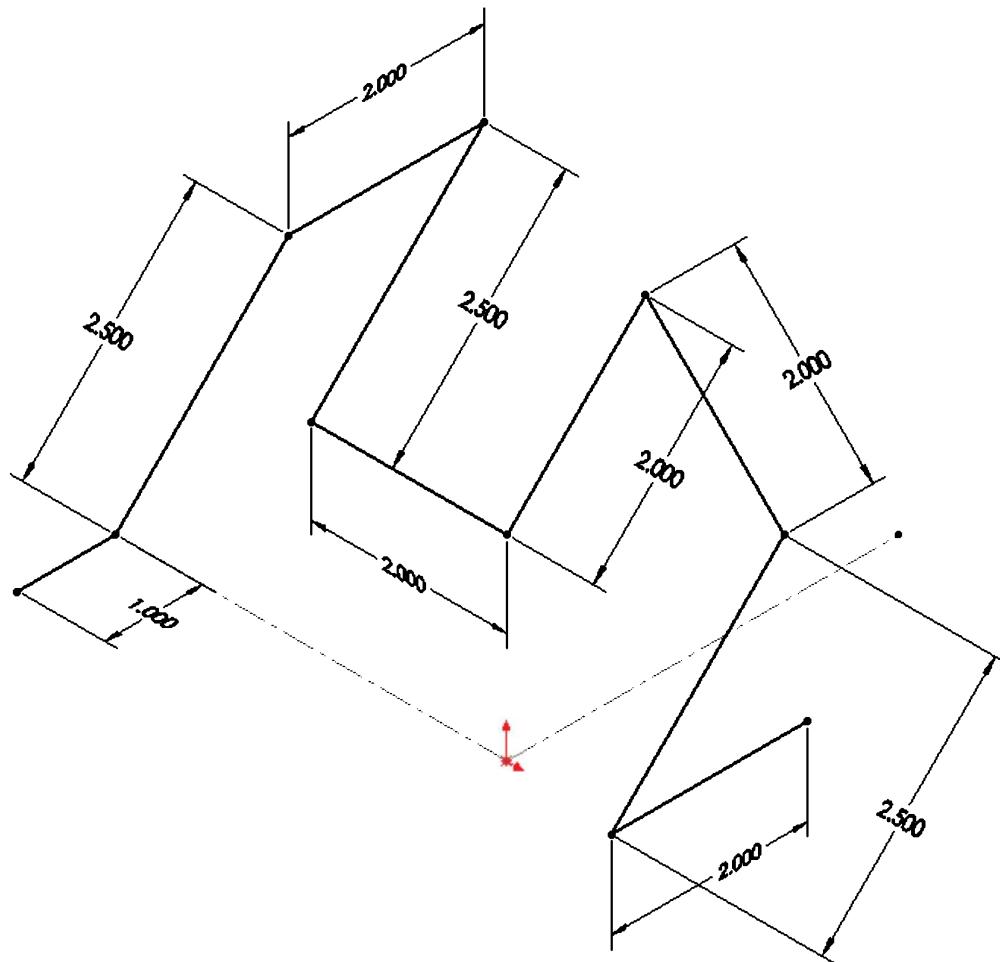
Sketch the rest of lines on the planes as labeled below.

For clarity, hide all the planes (select **View / Hide-Show** and click off **Planes**). We will select the planes from the FeatureManager tree when needed.

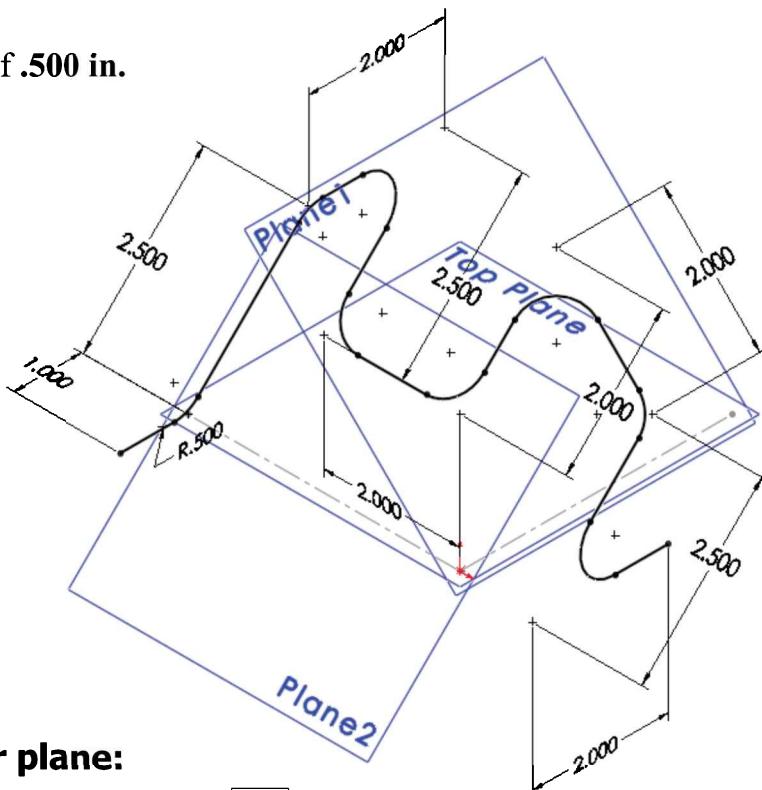
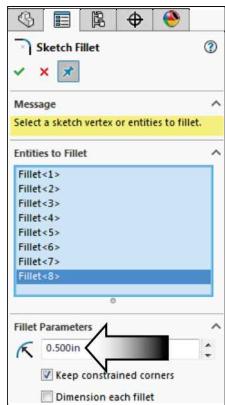




Add the dimensions  below to fully define the sketch.



Add Sketch Fillets  of .500 in.
to all corners.



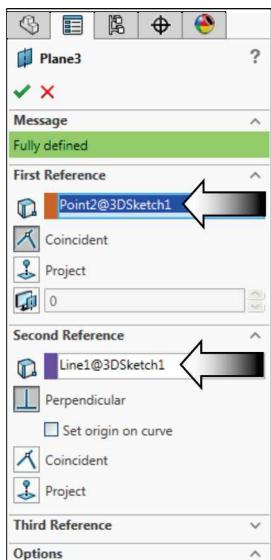
Exit the 3D Sketch or
press **Ctrl+Q**.

5. Creating a Perpendicular plane:

Select **Insert/Reference Geometry/Plane** .

Select the **line** and its **endpoint** approximately as shown.

The **Perpendicular** option should be selected by default.



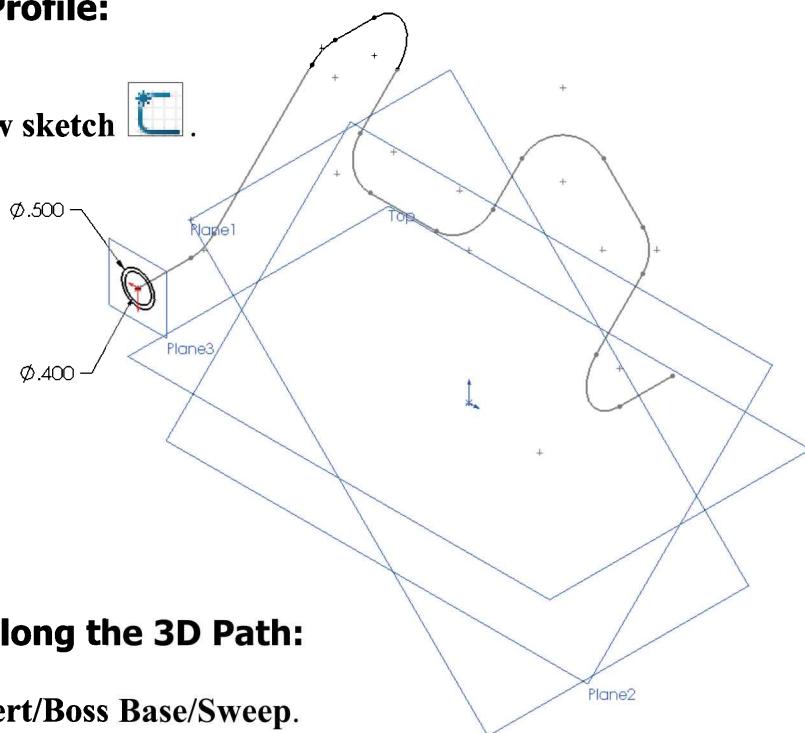
A new plane normal to the selected line is created.

Click **OK**.

6. Sketching the Sweep Profile:

Select the new plane (Plane3) and open a new sketch .

Sketch 2 Circles  on the same center and add the dimensions as shown to fully define the sketch.

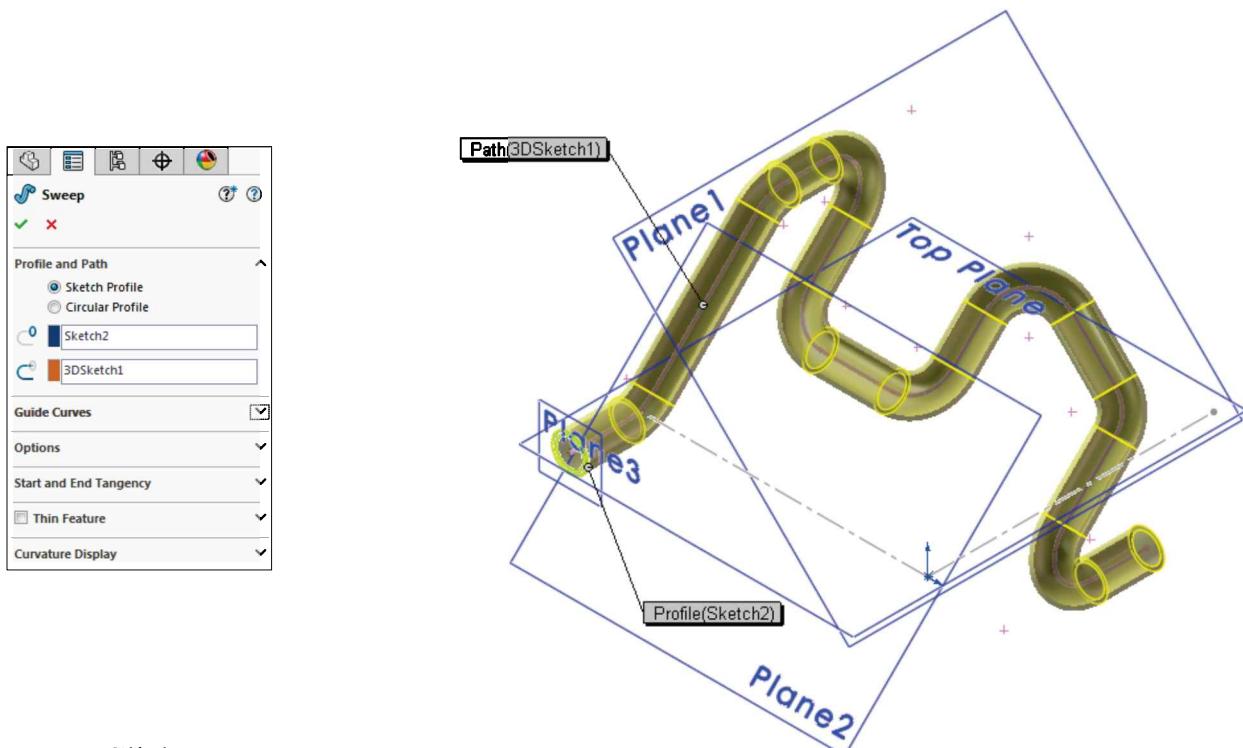


7. Sweeping the Profile along the 3D Path:

Click  or Select Insert/Boss Base/Sweep.

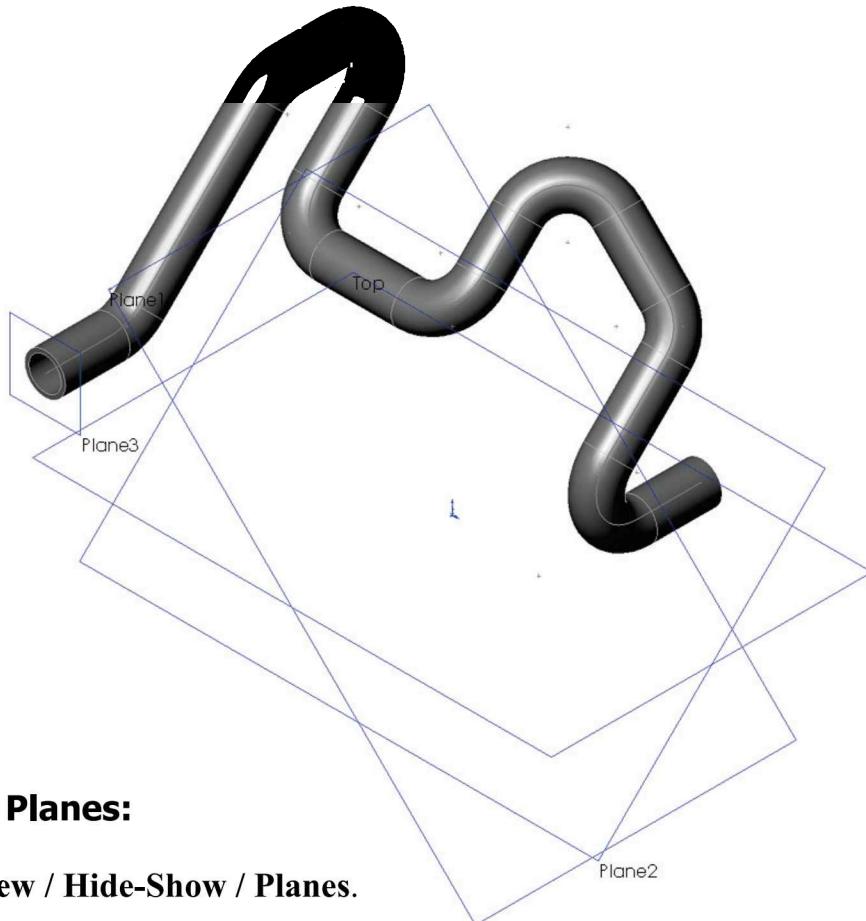
Select the Circles as the Sweep Profile .

Select the 3D Sketch as the Sweep Path .



Click **OK**.

The resulting Swept feature.



8. Hiding the Planes:

Select View / Hide-Show / Planes.

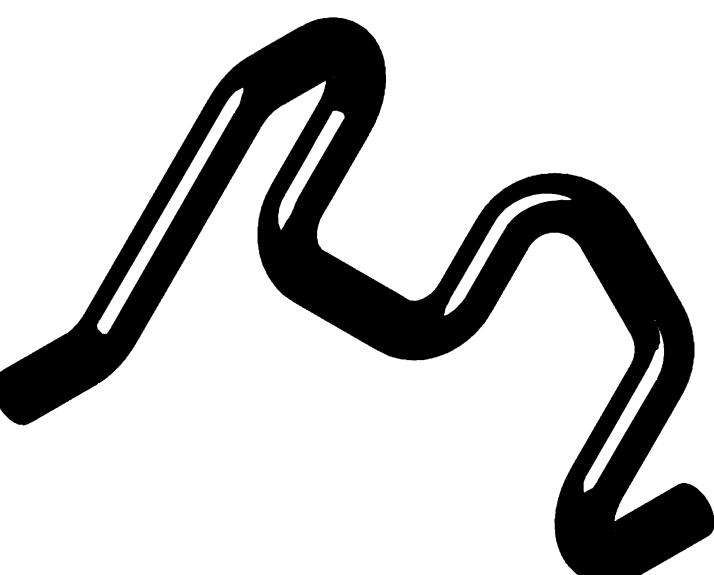
The planes are temporarily put away from the scene.

9. Saving your work:

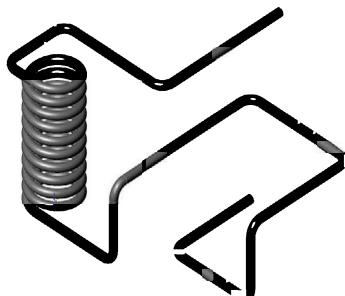
Select File / Save As.

Enter **3D Sketch_Planes** for the name of the file.

Click **Save**.

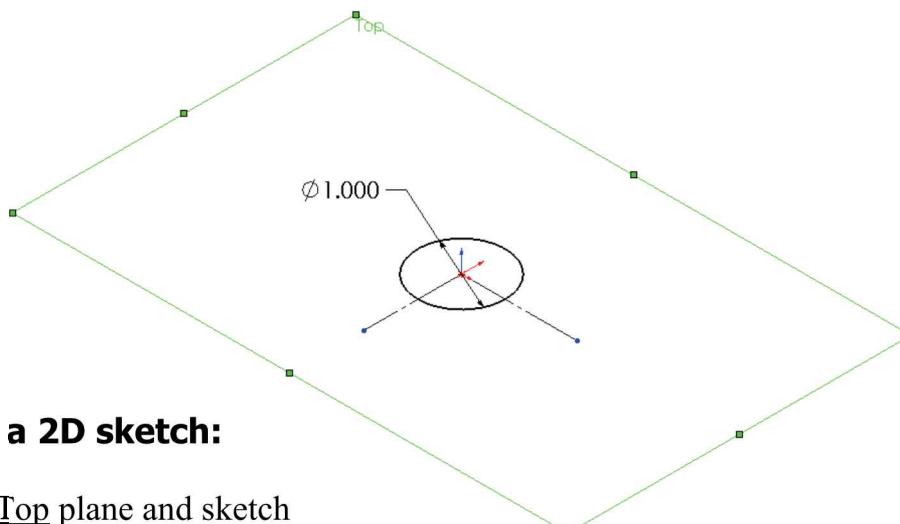


Exercise: 3D Sketch & Composite Curve



A 3D sketch normally consists of lines and arcs in series and Splines. You can use a 3D sketch as a sweep path, as a guide curve for a loft or sweep, a centerline for a loft, or as one of the key entities in a routing system.

The following exercise demonstrates how several 3D Sketches can be created, combined into 1 continuous Composite Curve, and used as a Sweep Path.



1. Creating a 2D sketch:

Select Top plane and sketch

a **1.00in** diameter **Circle**

and **2 Centerlines** .

2. Creating a Helix:

Select **Insert/Curve**/

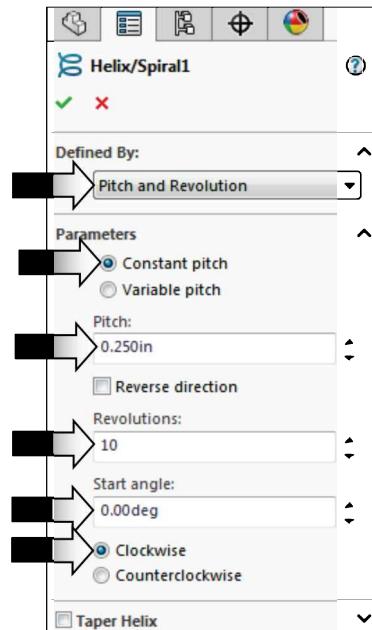
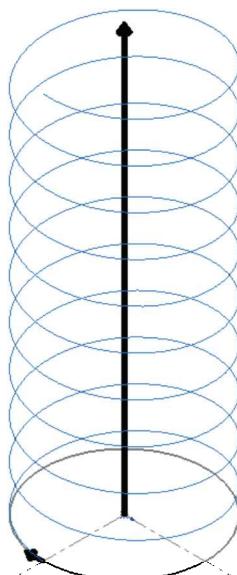
Helix-Spiral .

Pitch: **.250 in.**

Revolution: **10.**

Start Angle: **0 deg.**

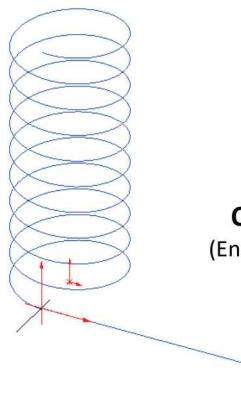
Click **OK**.



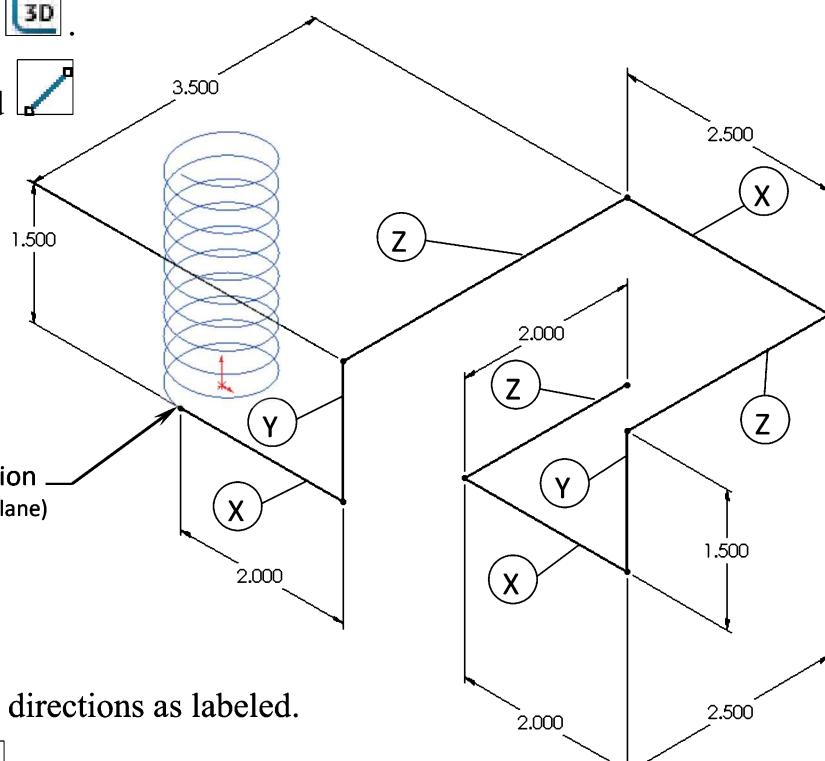
3. Creating the 1st 3D sketch:

Select Insert/3D Sketch .

Select the Line command  and sketch the 1st line along the X direction.

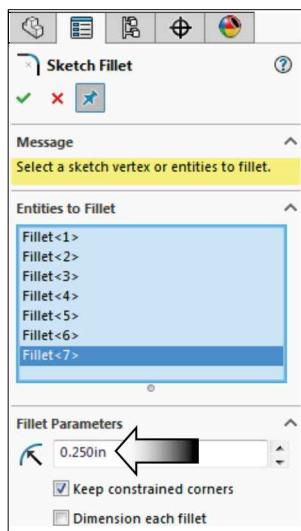


On-Plane relation
(End point & Right plane)



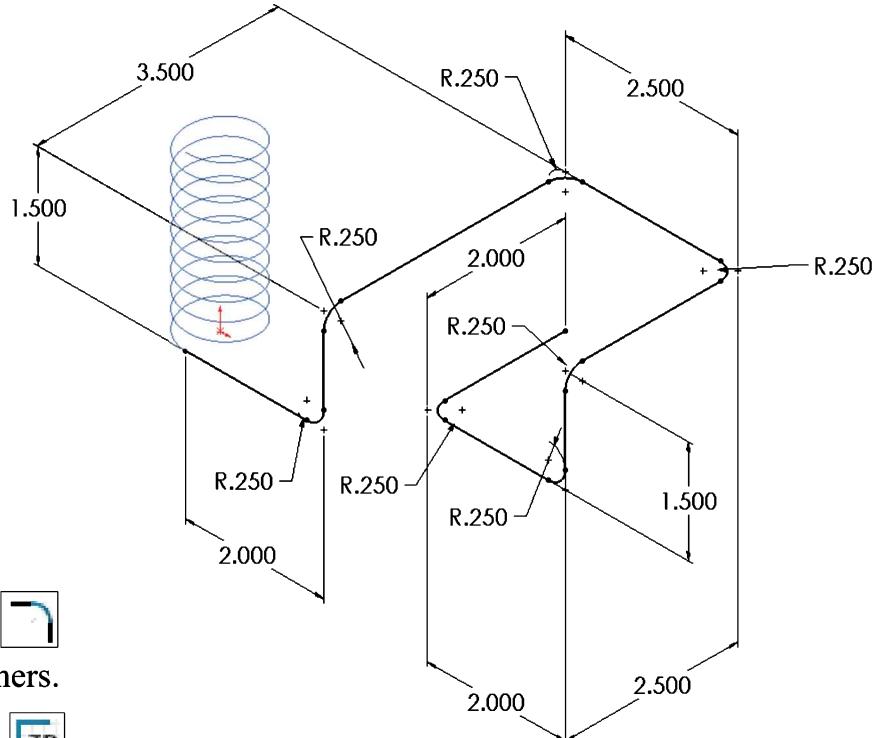
Add other lines in the directions as labeled.

Add Dimensions  to fully define the sketch.



Add Sketch Fillets  of .250in. to all corners.

Exit the 3D Sketch  or press **Ctrl + Q**.

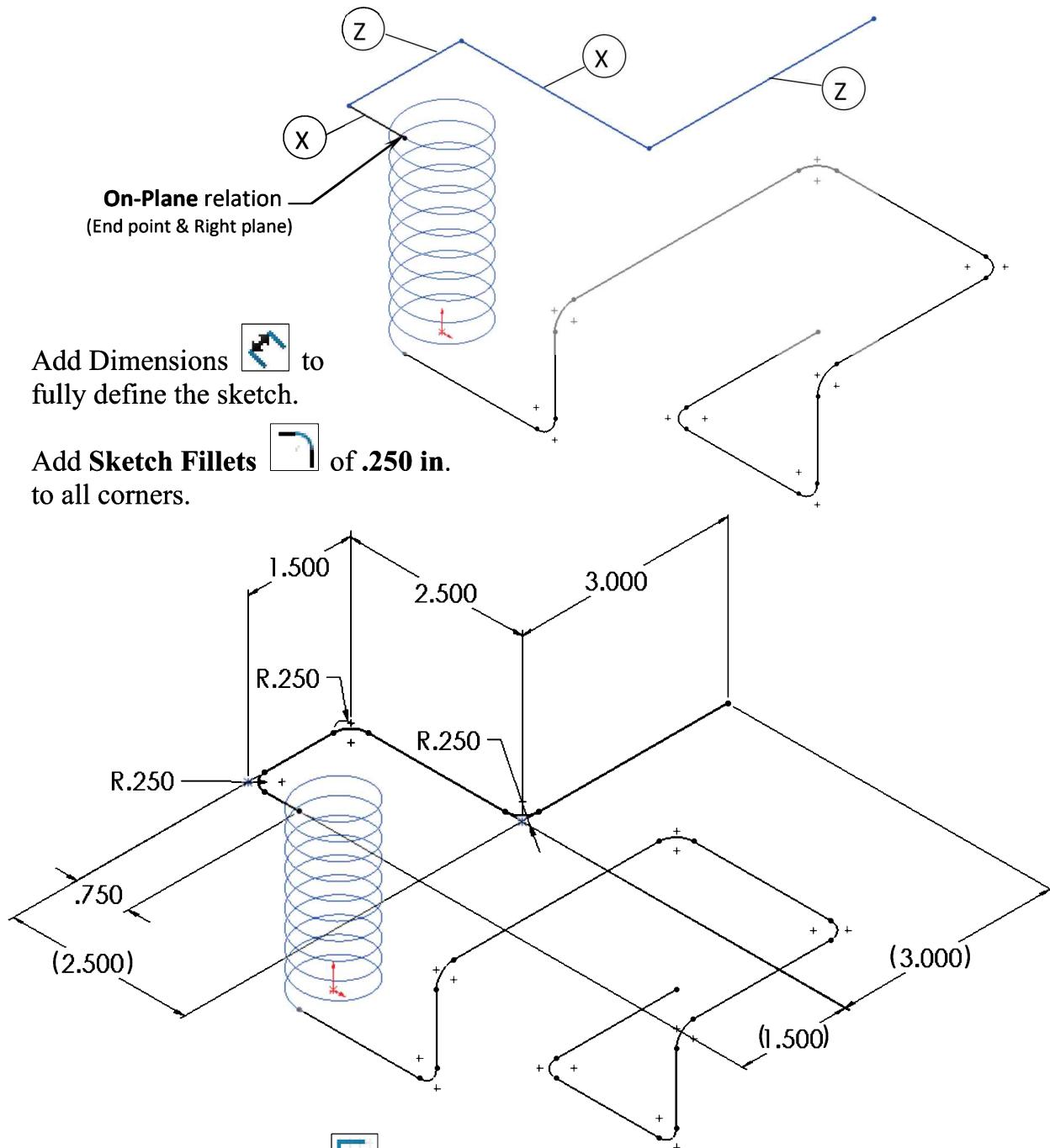


4. Creating the 2nd 3D sketch:

Select **Insert/3D Sketch** .

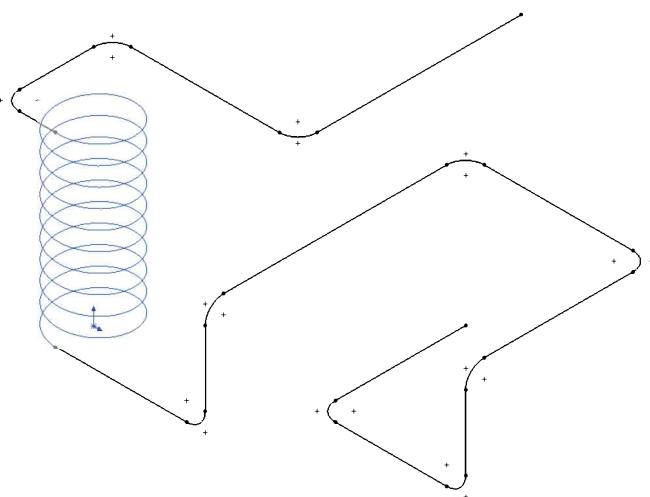
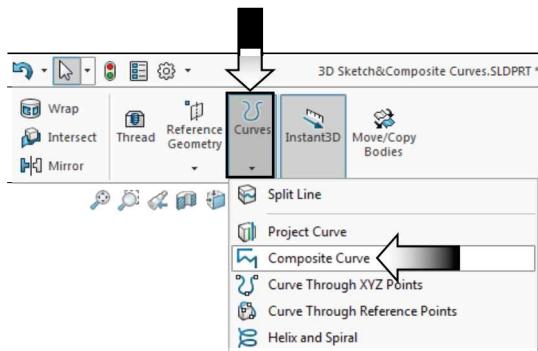
Select the **Line** command and sketch the 1st line along the X direction.

Sketch the rest of the lines following their direction shown below.

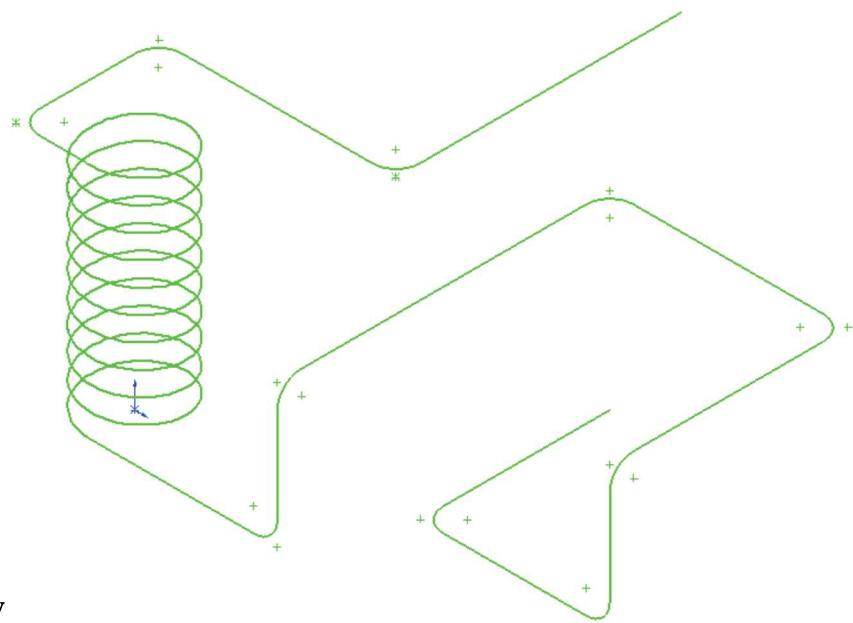
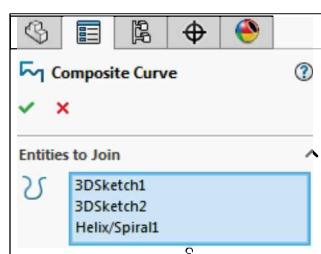


5. Combining the curves:

Select the **Composite Curve** command  below the Curves button, or select: **Insert / Curve / Composite**.



Select the 3 Sketches either from the FeatureManager tree or directly from the graphics area.



Click OK.

The sketches are now combined to 1 continuous curve.
We will use it as the sweep path in the next few steps.

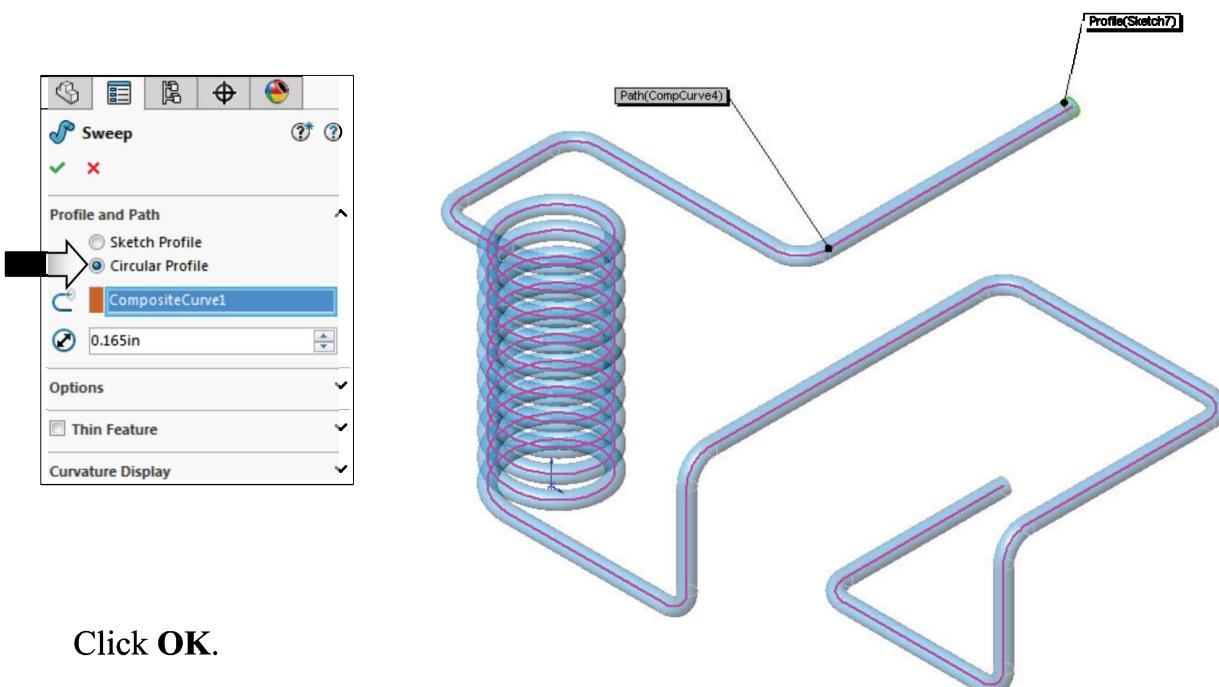
6. Creating a Sweep using Circular Profile:

Select **Insert/Boss Base/ Sweep** .

Select the **Circle Profile** option (arrow).

For sweep profile, enter **.165** in .

For Sweep Path, select the **Composite Curve** .



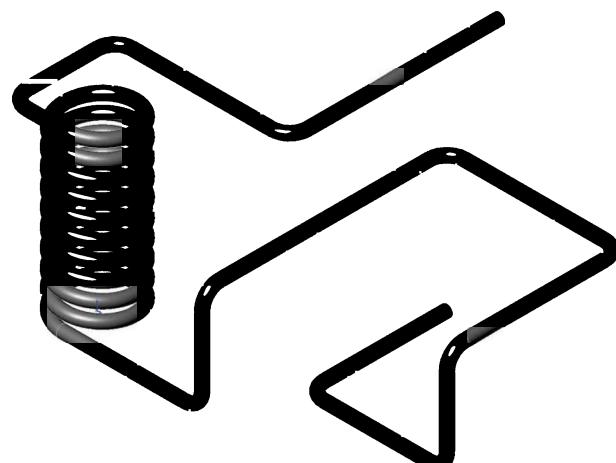
Click **OK**.

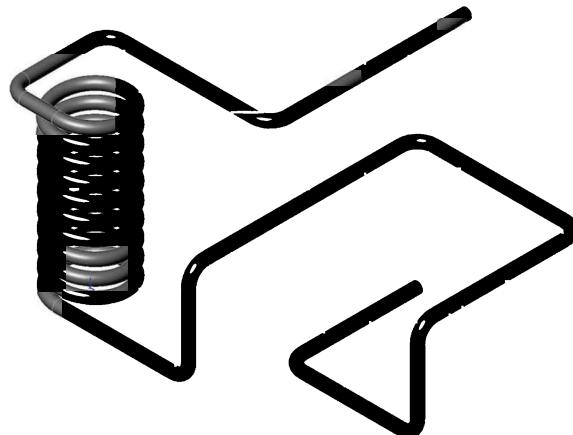
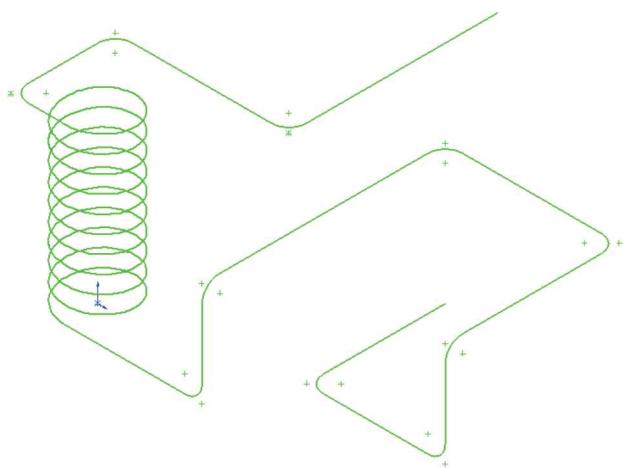
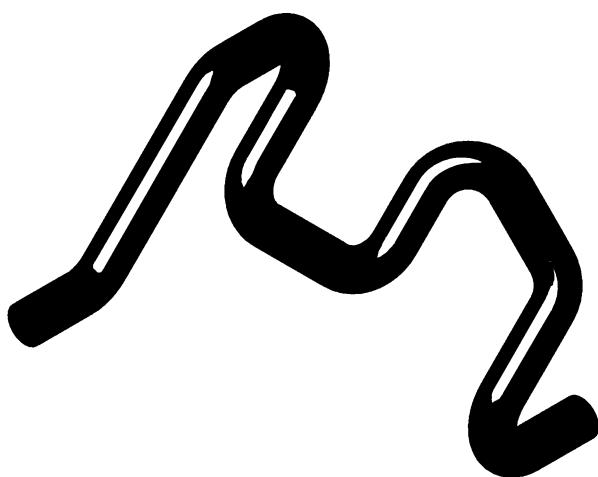
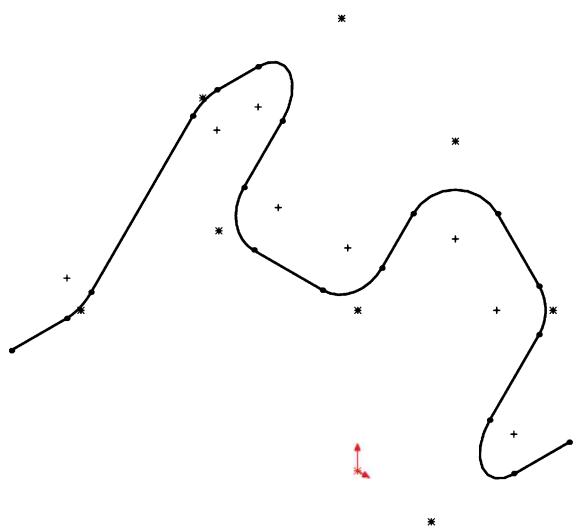
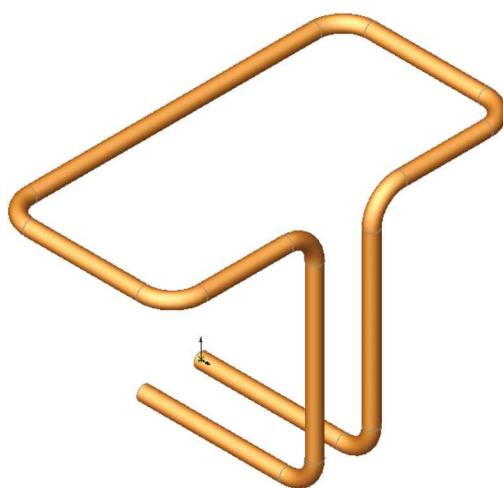
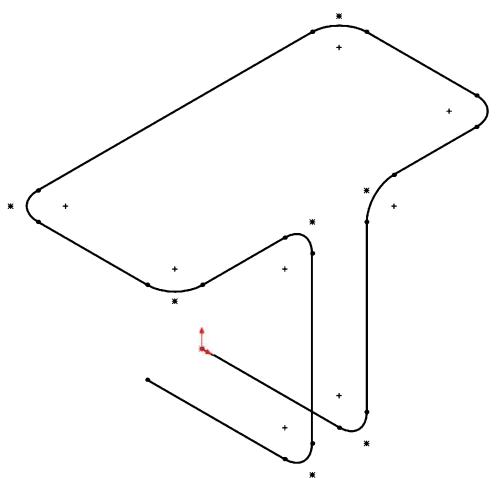
7. Saving your work:

Click **File/Save As**.

Enter **3D Sketch_ Composite Curve** for the name of the file.

Click **Save**.



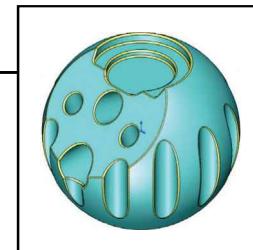


CHAPTER 2

Plane Creation

Planes

Advanced Topics



In SOLIDWORKS, planes are not only used to sketch geometry, but also used to create section views of a model or an assembly.

Planes are also used as end conditions for feature extrusion and as neutral planes to define the draft angles, etc.

There are several options to create planes:



Parallel Plane.



At Angle Plane.



Perpendicular Plane.



Offset Distance Plane.



Coincident Plane.



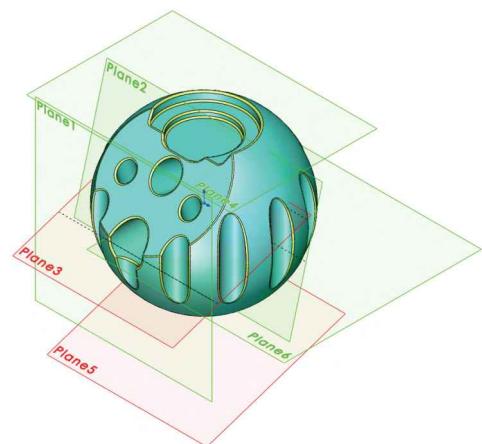
Mid Plane.



Project Plane.

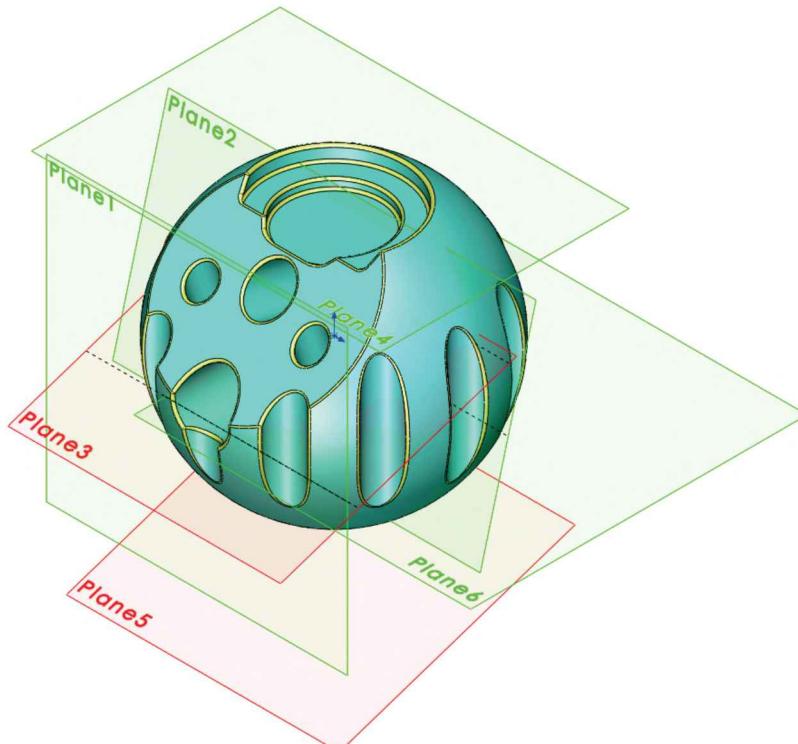
Each plane requires slightly different types of references, some of the planes may require only one reference, but some others may require two or three.

This chapter discusses how planes are created using the sketch geometry and other features that are available in the model as references.



Advanced Topics

Plane Creation



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI

Units: INCHES – 3 Decimals

Tools Needed:



Insert Sketch



Rectangle



Circle



Planes



Add Geometric Relations



Dimension



Sketch Mirror



Offset Entities



Boss/Base Revolve



Circular Pattern



Extruded Cut



Fillet/Round

1. Starting with a new Part document:

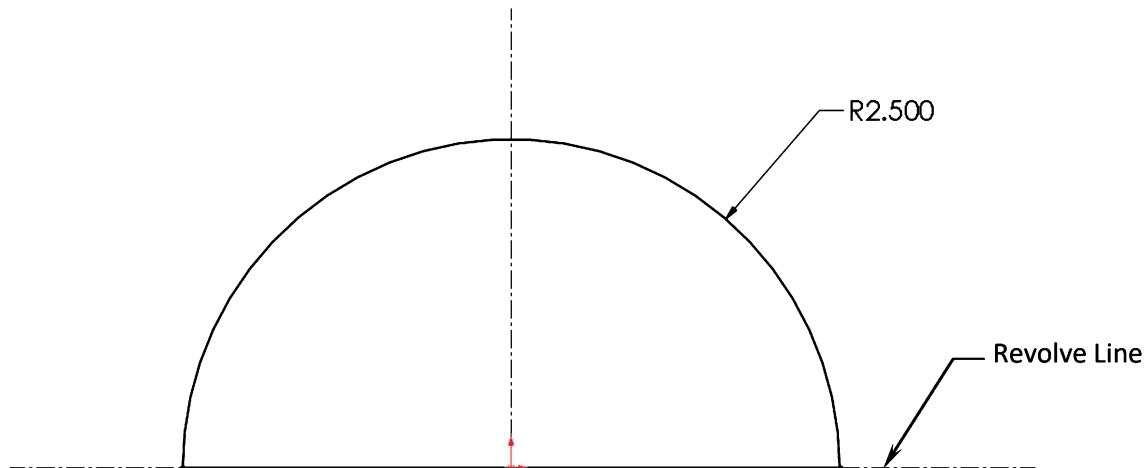
Select **File / New / Part** and click **OK**.

Select the Front plane from the FeatureManager tree.

Click  or select **Insert / Sketch**.

Sketch the profile and add dimensions as shown below.

(It may be easier to sketch a circle, instead of a center-point arc, add the 2 centerlines, and then trim away the bottom half of the circle.)



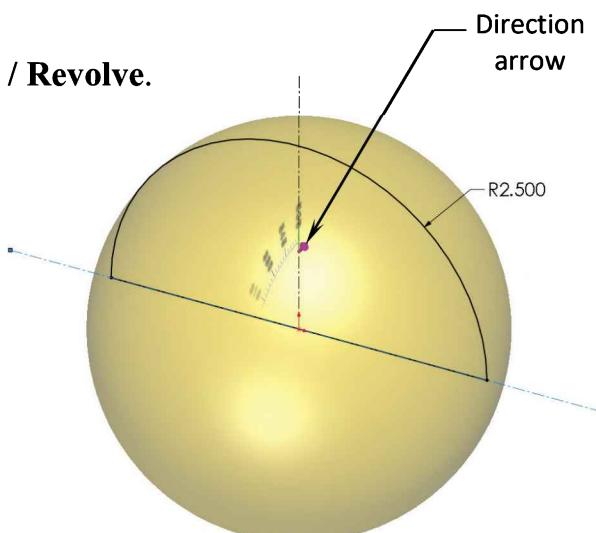
2. Revolving the Base:

Click  or select **Insert / Boss Base / Revolve**.

Set Revolve Type to: **Blind**.

Set Revolve Angle to **360 deg**.

Click **OK**.



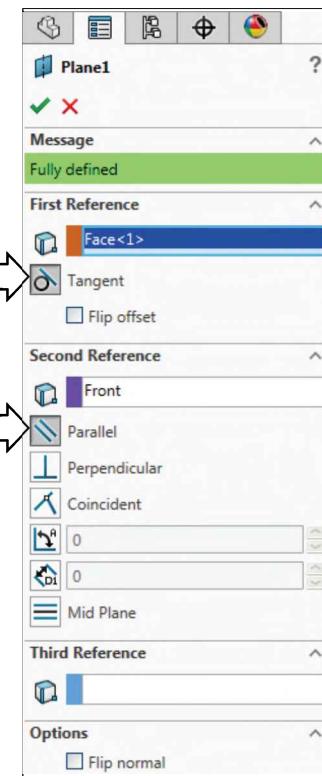
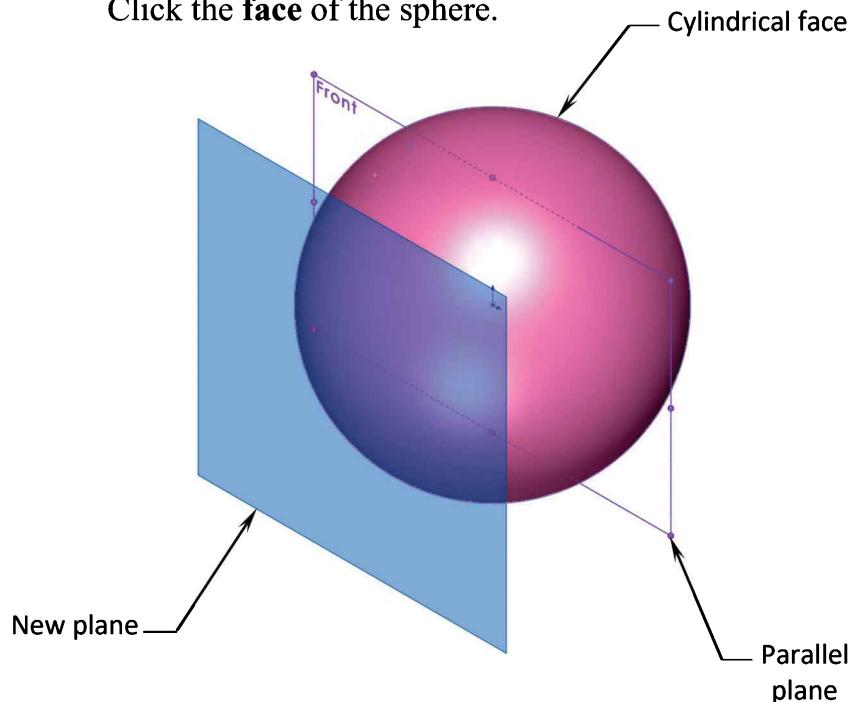
Note: Drag the Direction arrow to see the preview of the rotate angle.

3. Creating a Tangent plane:

(Requires a cylindrical face and a parallel plane).

Click  or select: Insert / Reference Geometry / Plane.

Click the **face** of the sphere.



Expand the FeatureManager tree and select the **Front** plane.

The **Tangent** option is selected automatically.

Click the **Parallel** option in the Second Reference section.

Click **OK**.

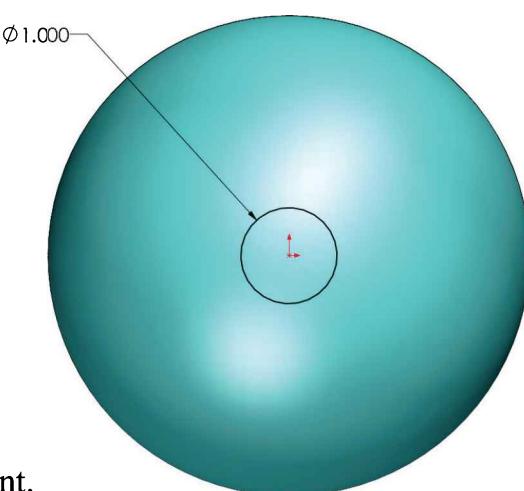
4. Adding a Center hole:

Select the new plane (Plane1) and open a **new sketch**.

Sketch a **Circle** centered on the origin.

Add a **1.000"** diameter dimension.

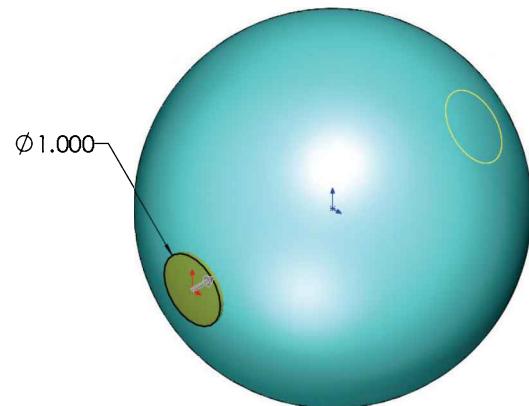
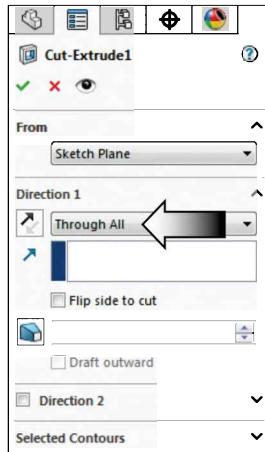
The circle should be fully defined at this point.



Click  or select:
Insert / Cut / Extrude.

Select **Through All** for
Direction 1.

Click **OK**.



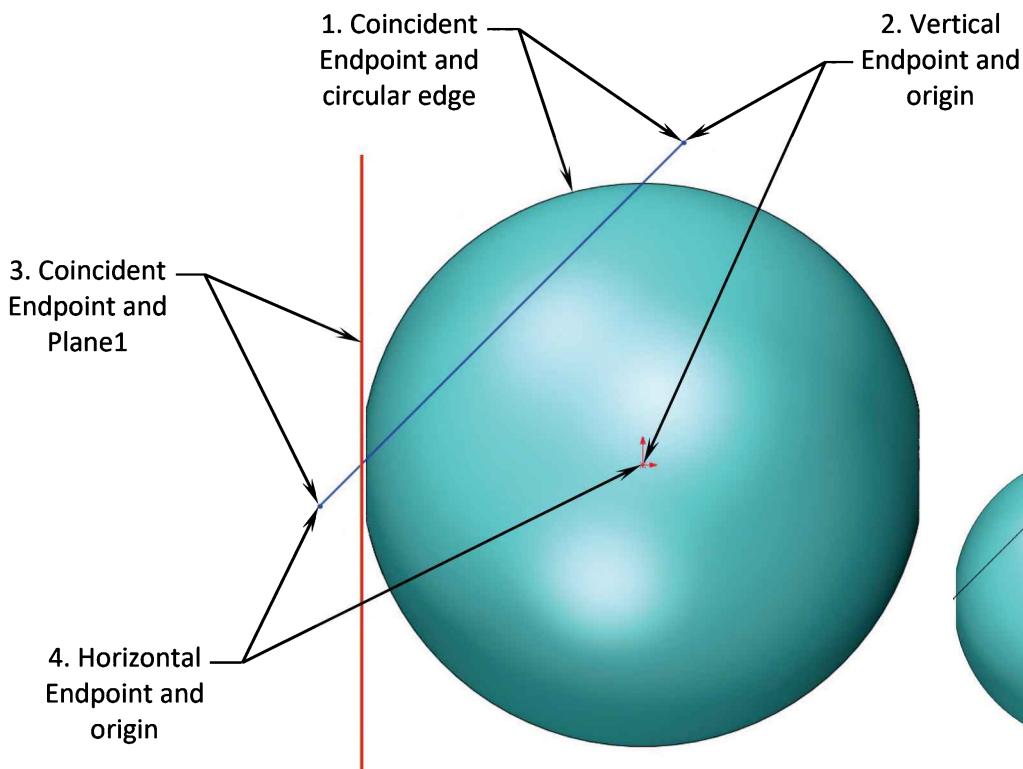
5. Creating a flat surface:

This step will demonstrate the use of geometric relations to fully define the sketch without using dimensions.

Select the Right plane from the FeatureManager tree.

Click  or select **Insert / Sketch** and change to the **Right** view (Ctrl+4).

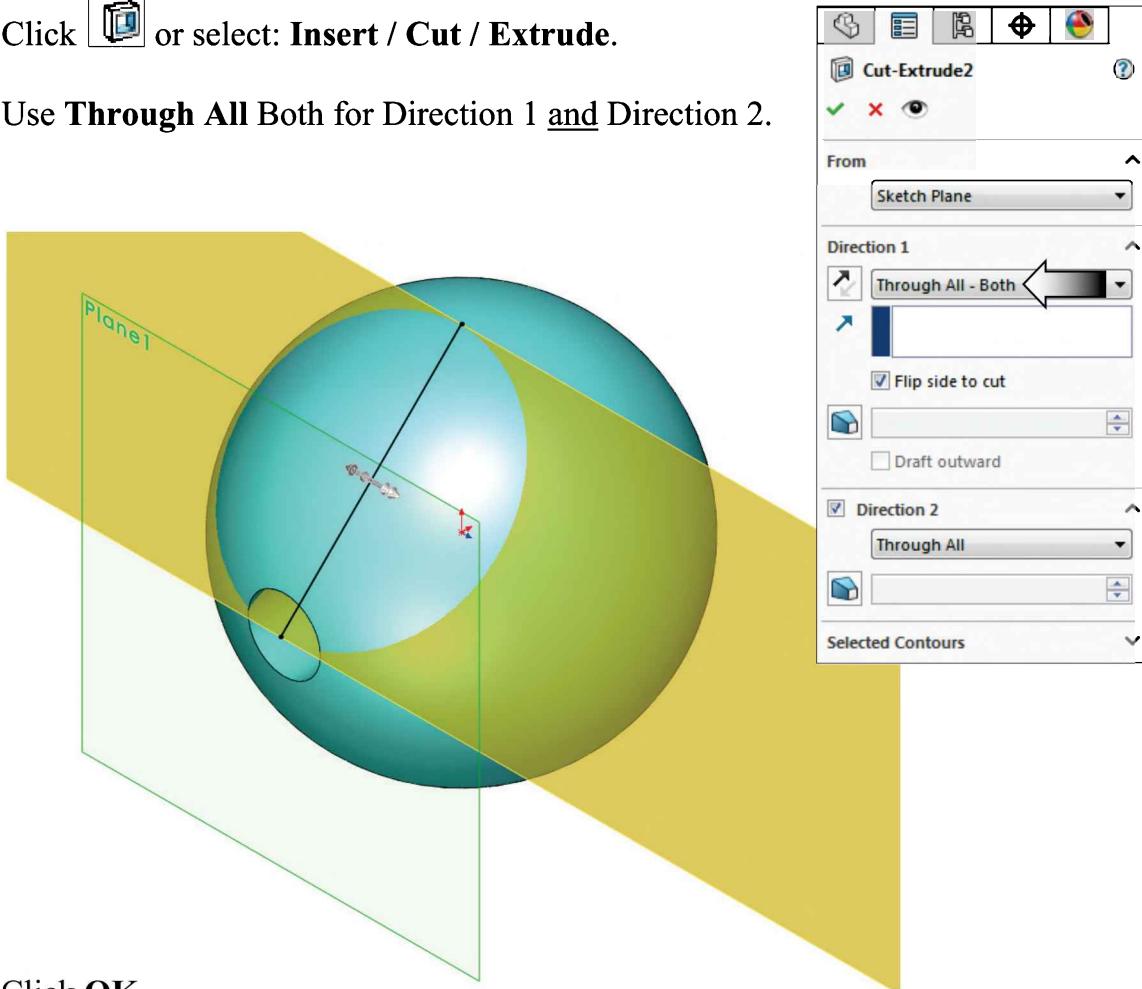
Sketch a **Line** and add the relations  as shown.



6. Extruding a Cut:

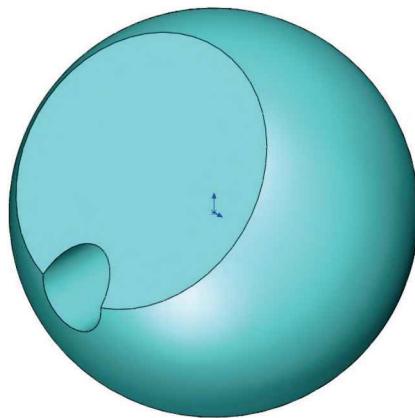
Click  or select: Insert / Cut / Extrude.

Use **Through All Both** for Direction 1 and Direction 2.

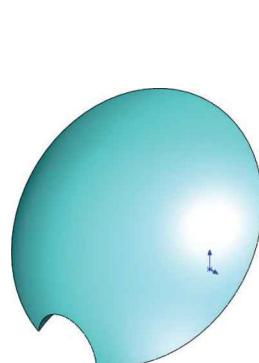


Click **OK**.

The examples below show how the option **Flip Side to Cut** works.



Flip Side to Cut Selected



Flip Side to Cut Cleared

7. Creating an At-Angle plane:

(Requires a Reference Plane, a Reference Axis, and an Angular Dimension.)

Select the **Front** plane from FeatureManager tree.

Click  or select **Insert / Reference Geometry / Plane**.

Select the **horizontal centerline** as reference axis.

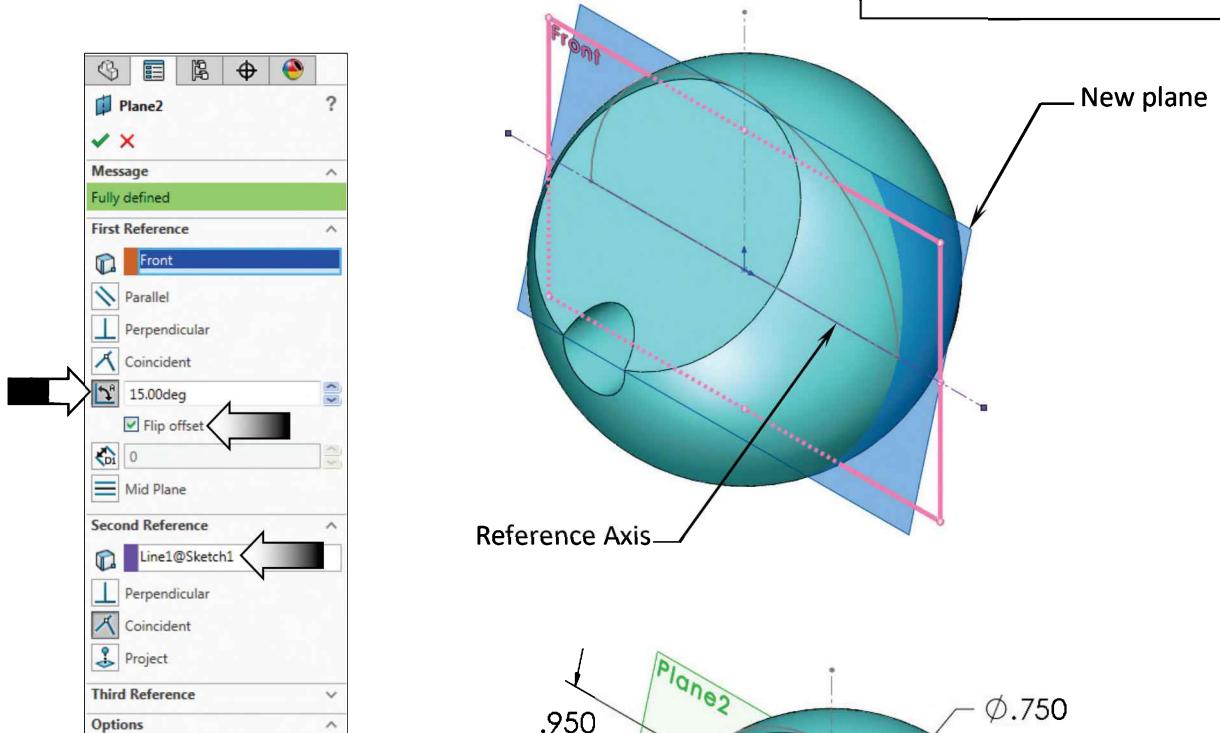
Select the **At Angle** option.

Enter **15 deg.** for angle and click **Flip Offset**.

Click **OK**.



Right-click **Sketch1**
(on the FeatureManager
tree, below the **Revolve1**)
and select **Show**.

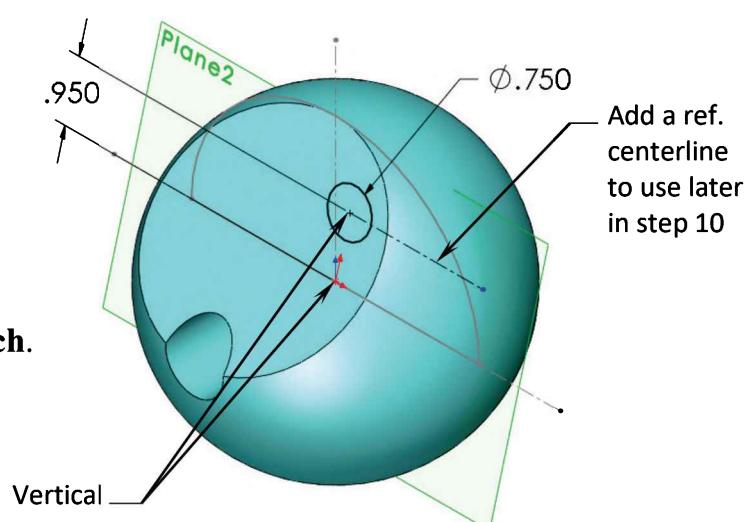


8. Creating a Ø.750 hole:

Select the new plane (Plane2).

Click  or select **Insert / Sketch**.

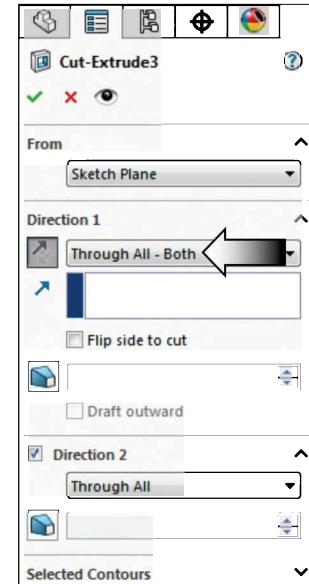
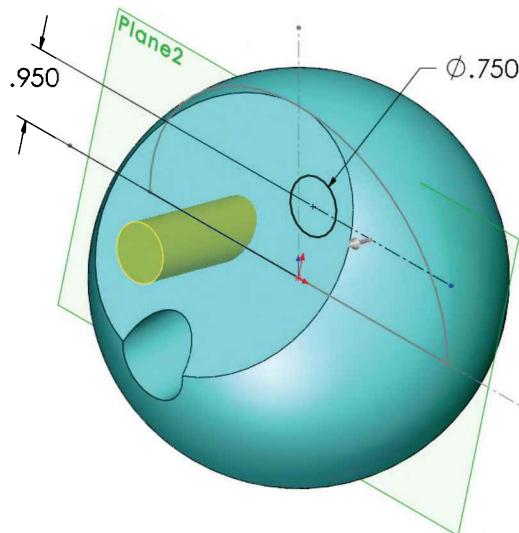
Sketch a **Circle**  and add the dimensions and a vertical relation as shown.



Click  or select Insert / Cut / Extrude.

Direction 1: Through All Both.

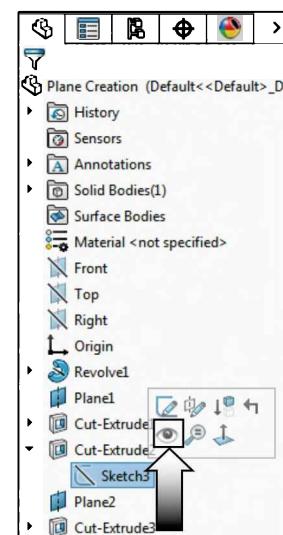
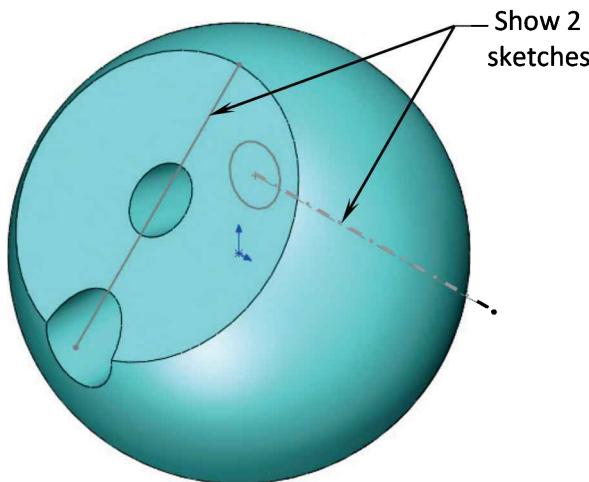
Click OK.



9. Showing the Sketches:

Expand the **Cut-Extrude1** on the FeatureManager tree (click the + symbol), right-click **Sketch2** and select **Show**.

Expand the **Cut-Extrude2** (click the + symbol), right-click **Sketch3**, and select **Show** ; also **Hide** the **Sketch1**.



10. Creating a Coincident plane:

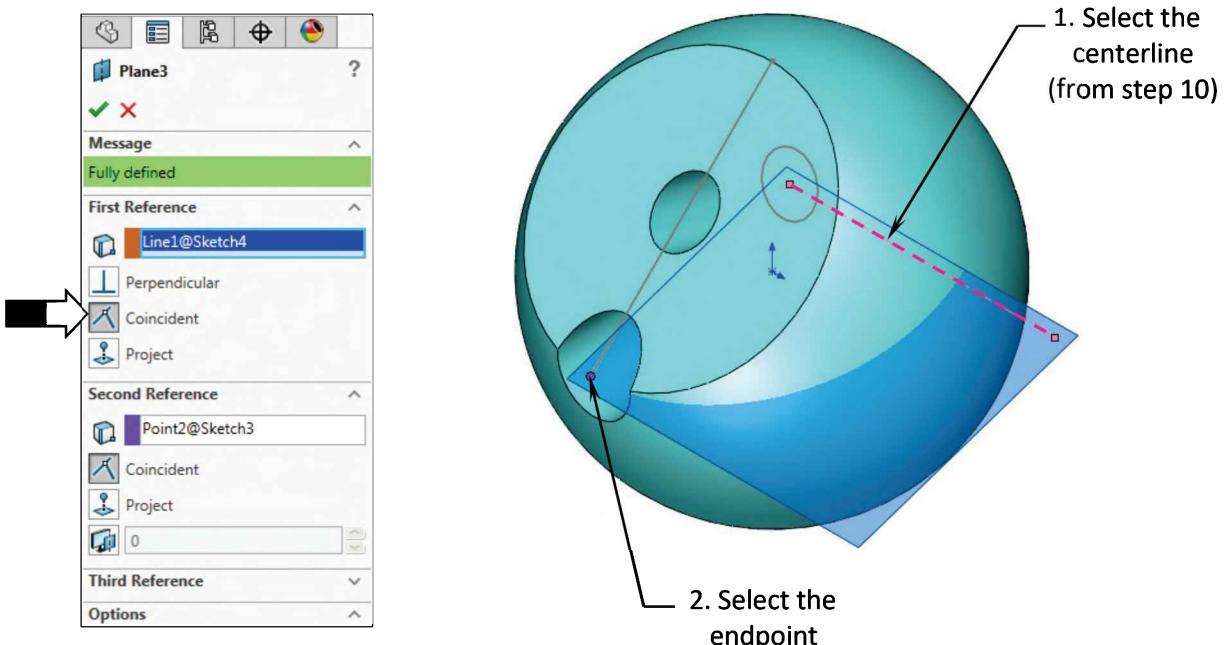
(Requires a Reference Line and a Sketch Point or a Vertex.)

Click  or select Insert / Reference Geometry / Plane.

Select the **Centerline** and the **Endpoint** as indicated.

The **Coincident** option should be selected automatically.

Click **OK**.



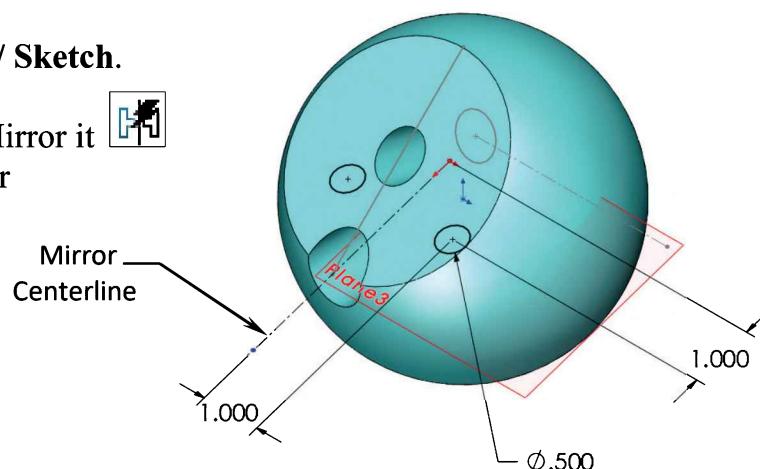
11. Creating the Ø.500 holes:

Select the new plane (Plane3).

Click  or select Insert / Sketch.

Sketch a **Circle**  and Mirror it 
(Use either Dynamic Mirror or Mirror Entity to mirror the circle.)

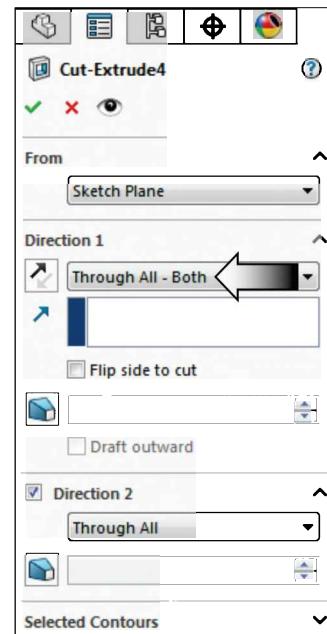
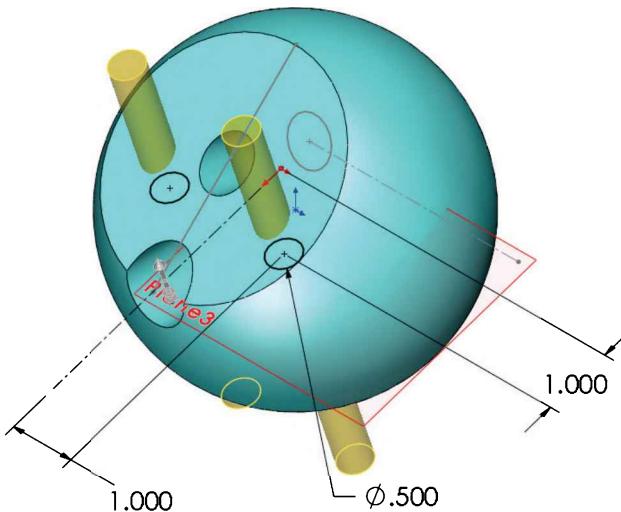
Add Dimensions  as shown to fully define the sketch.



Click  or select Insert / Cut / Extrude.

Direction 1: Through All Both.

Click OK.



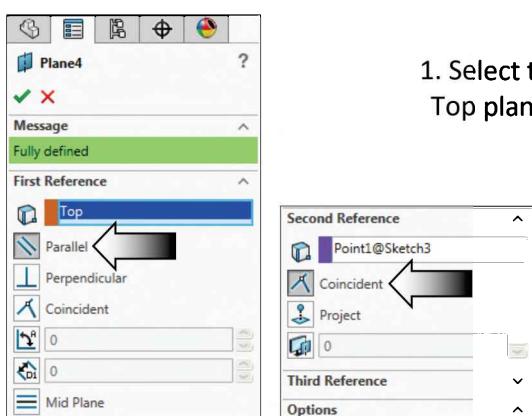
12. Creating a Parallel plane: (Requires a Reference Plane and Reference Point).

Click  or select Insert / Reference Geometry / Plane.

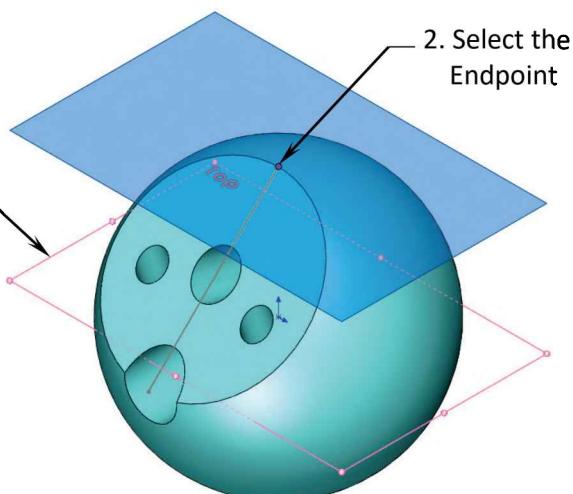
Select the Top plane and the Endpoint as indicated.

Based on the selection, the system selects the Parallel and Coincident options.

Click OK.



1. Select the Top plane

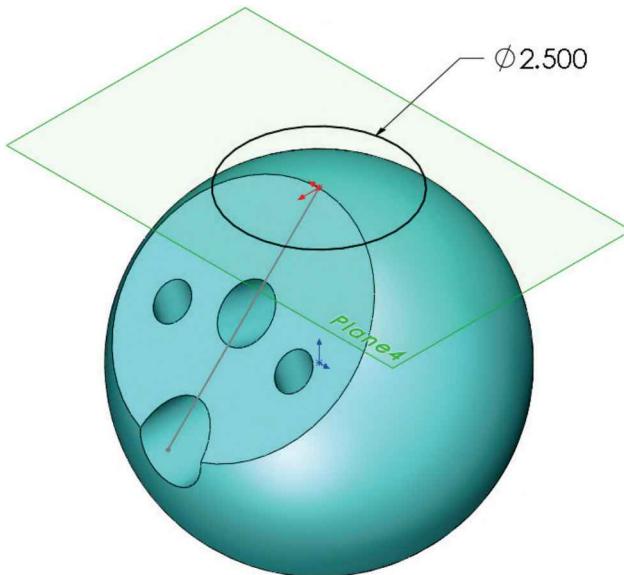


2. Select the Endpoint

13. Creating a Recess feature:

Select the new Plane (Plane4) and insert a **new sketch** .

Sketch a **Circle**  and add the diameter dimension to fully define the sketch.

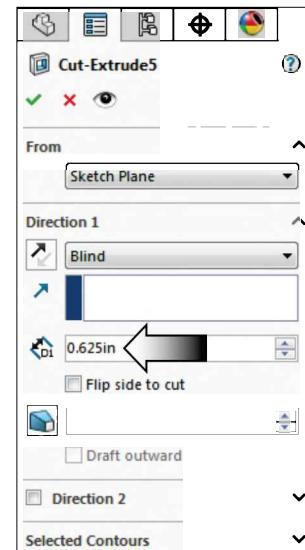
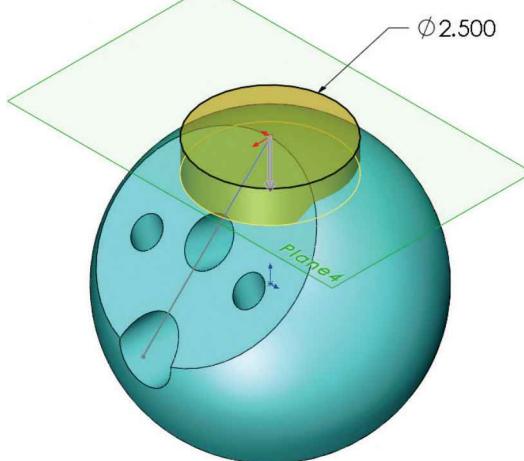


Click  or select **Insert / Cut / Extrude**.

End Condition: **Blind**.

Extrude Depth: **.625 in.**

Click **OK**.



Hide  the Sketch2, Sketch3 and all planes.

14. Creating an Offset-Distance plane: (Requires a Reference Plane and a Distance dimension.)

Click  or select **Insert / Reference Geometry / Plane**.

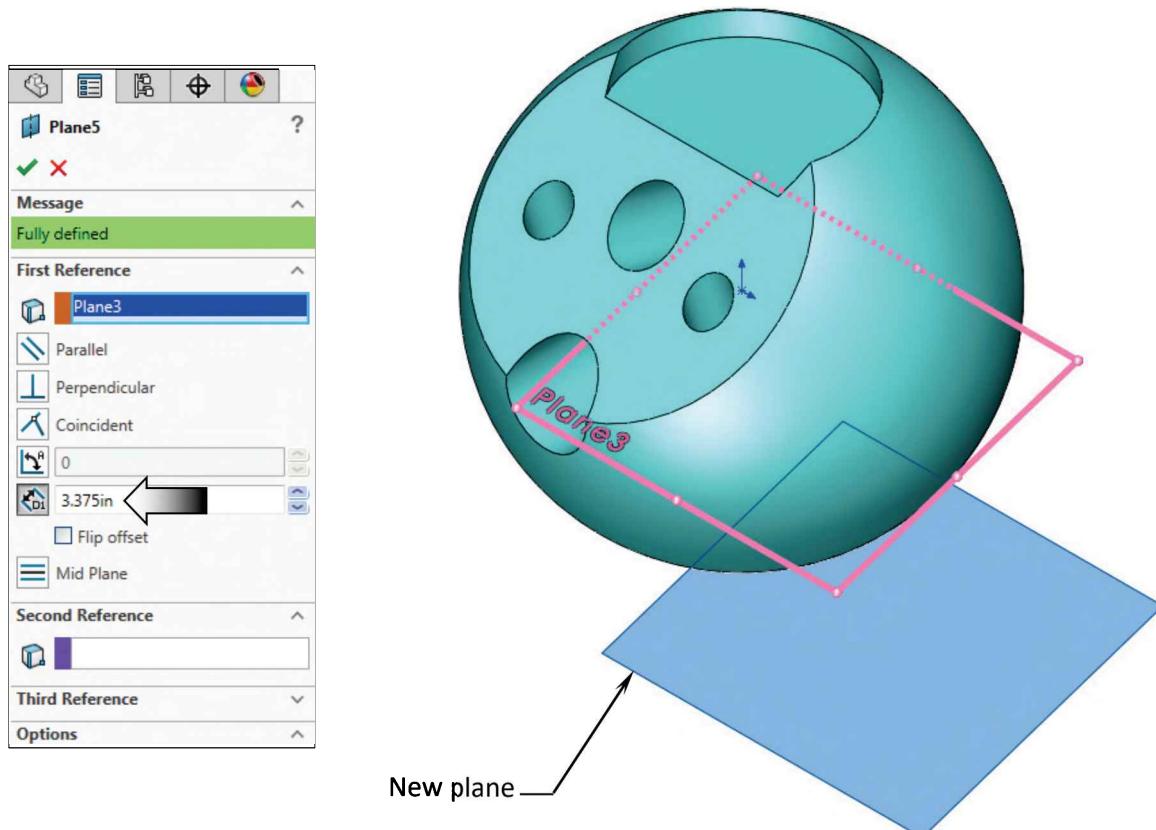
Select **Plane3** (from the FeatureManager tree) to offset from.

The **Offset Distance** option is automatically selected.

Enter **3.375** for offset value.

Make sure the new plane is placed below Plane3 (click **Flip Offset** if needed).

Click **OK**.

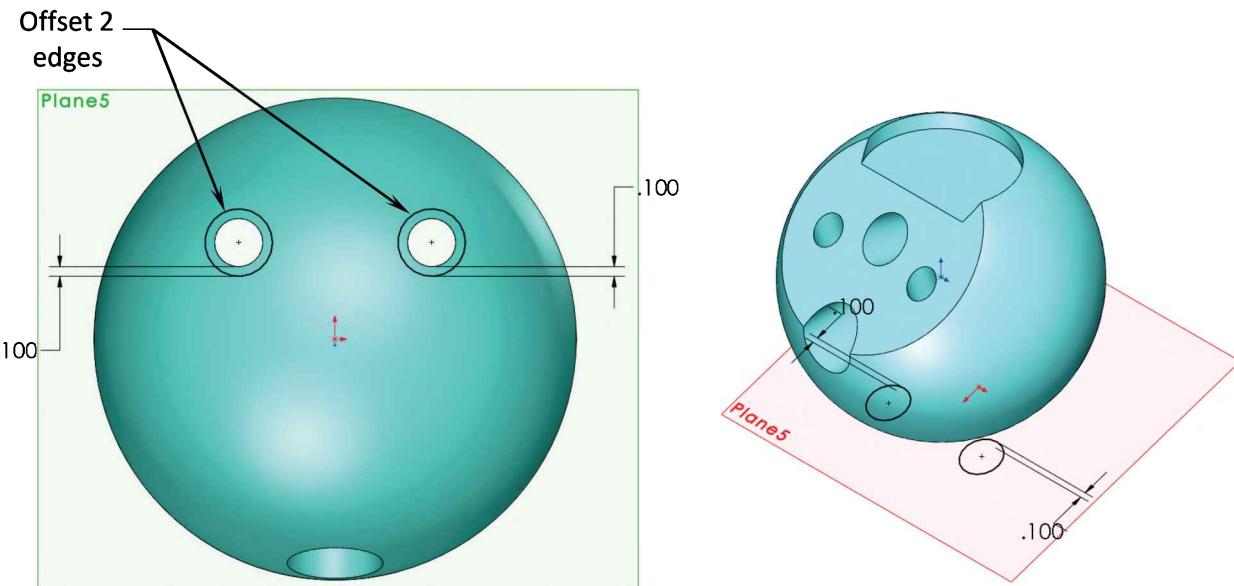


15. Creating the Bore holes:

Select the new plane (Plane5) and insert a **new sketch** .

Select the **circular edge** of the hole and press **Offset-Entities** .

Enter **.100 in.** for Offset Distance (only one offset can be done at a time, since the 2 circles are not connected to each other).



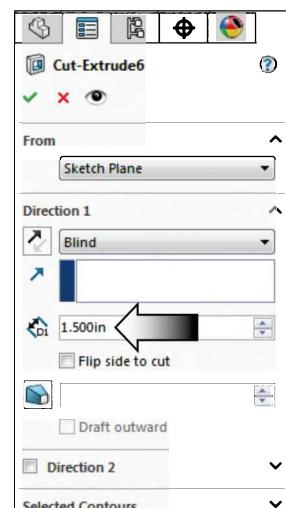
Click or select **Insert / Cut Extrude**.

End Condition: **Blind.**

Extrude Depth: **1.500 in.**

Click OK.

Hide Plane5.

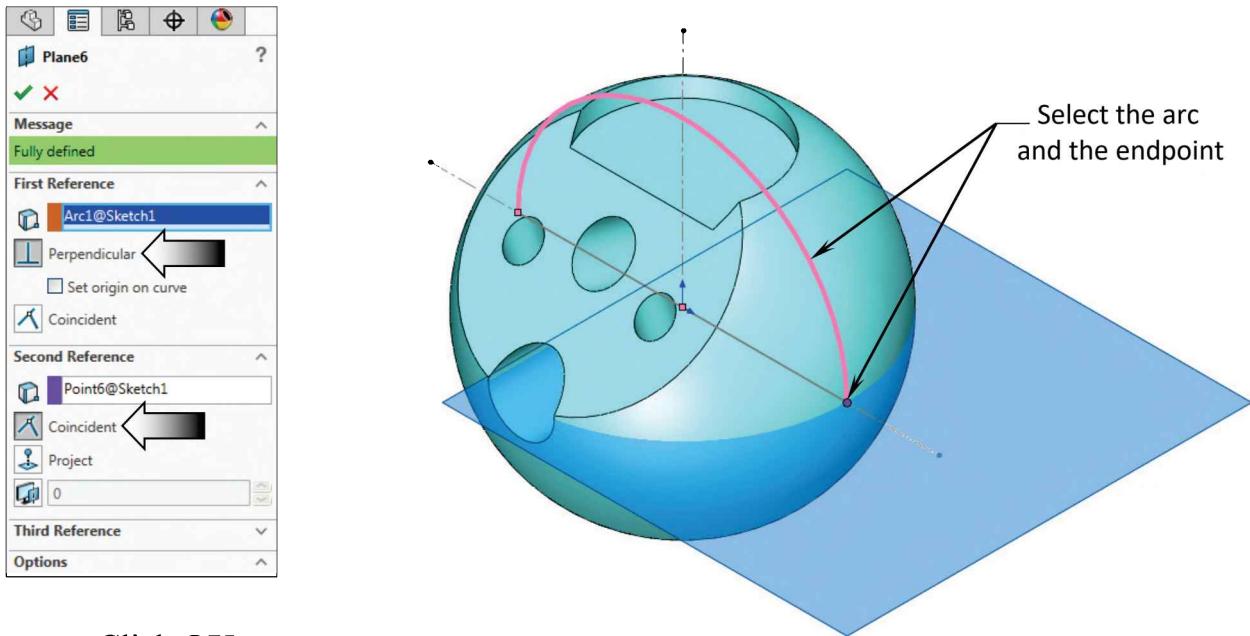


16. Creating a Perpendicular plane: (Requires a Reference Line or Curve & a Point).

Click or select **Insert / Reference Geometry / Plane**.

Show Sketch1 and select the **Arc** and its **Endpoint** on the right side as noted.

The **Perpendicular** and **Coincident** options should be selected automatically.



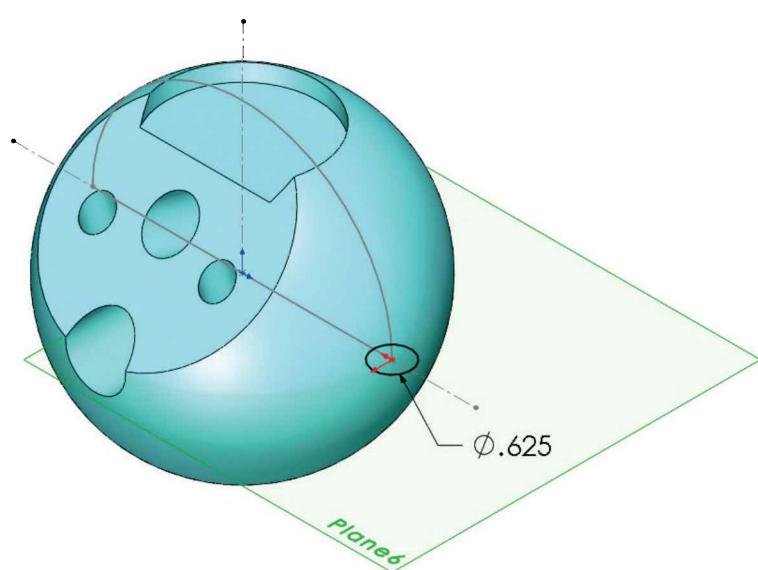
Click **OK**.

17. Creating the side-grips:

Select the new plane (Plane6) and insert a **new sketch** .

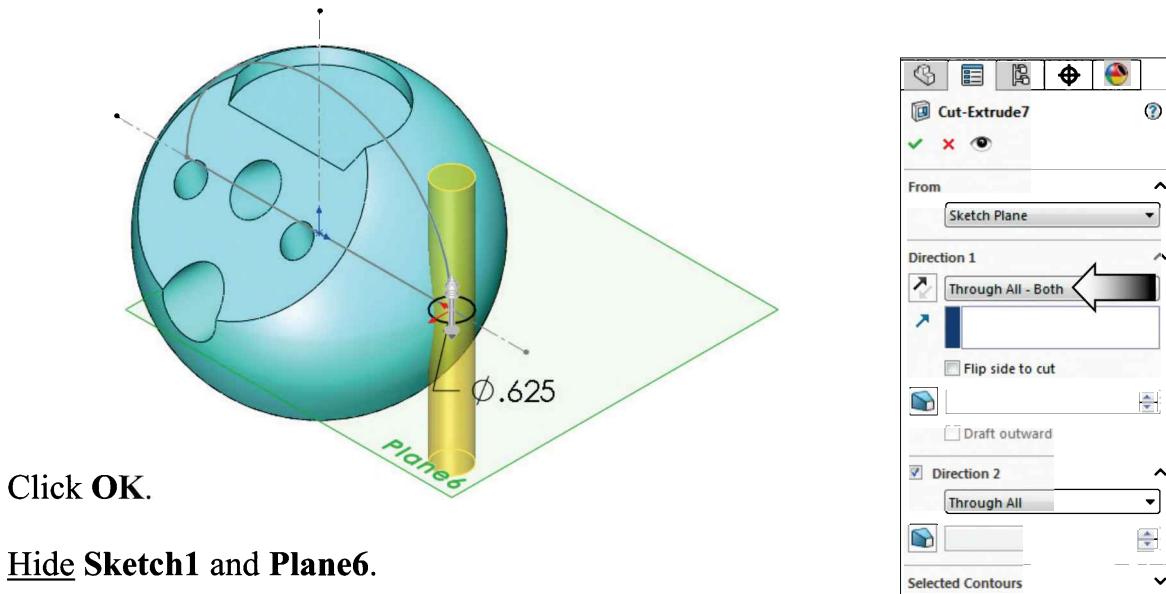
Sketch a **Circle** at the endpoint of the arc and add a **$\emptyset .625$** dimension.

Note: Use the **Coincident** relation when selecting 2 points, but use the **Pierce** relation when selecting a point and an arc.



Click  or select **Insert / Cut Extrude**.

Direction 1: **Through All Both**.



18. Creating a Circular Pattern of the Grips:

Click or select **Insert / Pattern Mirror / Circular Pattern**.

Click **View / Temporary Axis** and select the center **axis** as indicated.

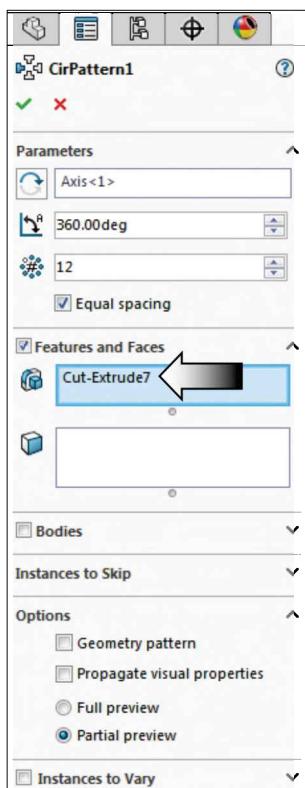
Equal Spacing: **Enabled**.

Total Angle: **360 deg.**

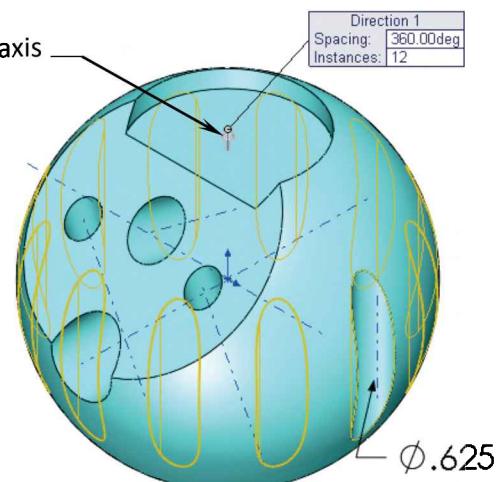
Number of instances: **12**.

For Feature To Pattern, select **Cut-Extrude6**.

Click **OK**.



Select this axis

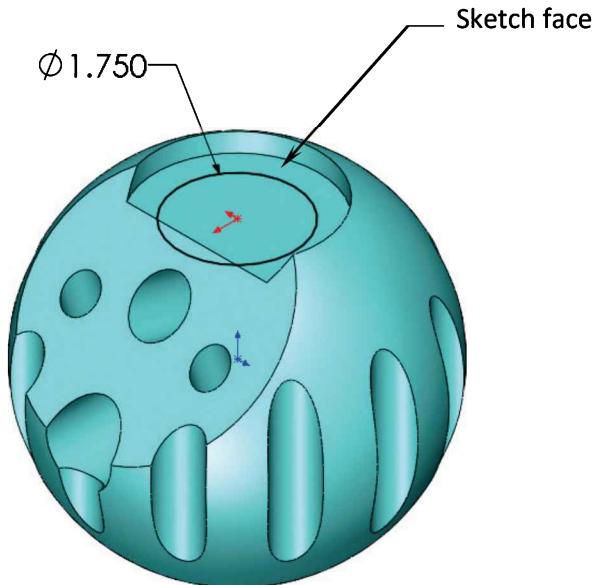


19. Adding another recess:

Select the upper face of the recess and insert a **new sketch** .

Sketch a **Circle** centered on Origin.

Add a **1.750 diameter** dimension.

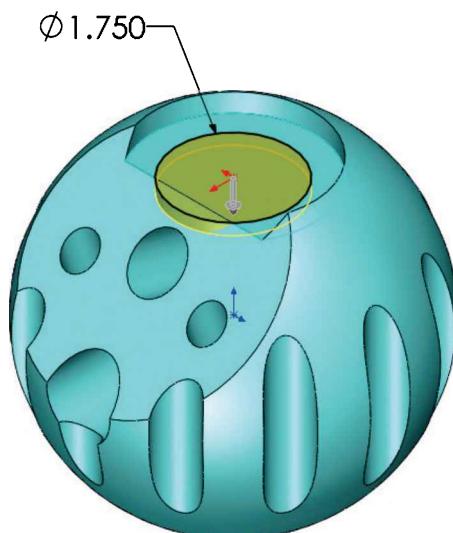
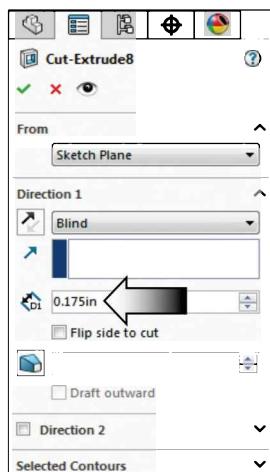


Click  or select **Insert / Cut Extrude**.

End Condition: **Blind.**

Extrude Depth: **.175 in.**

Click **OK**.



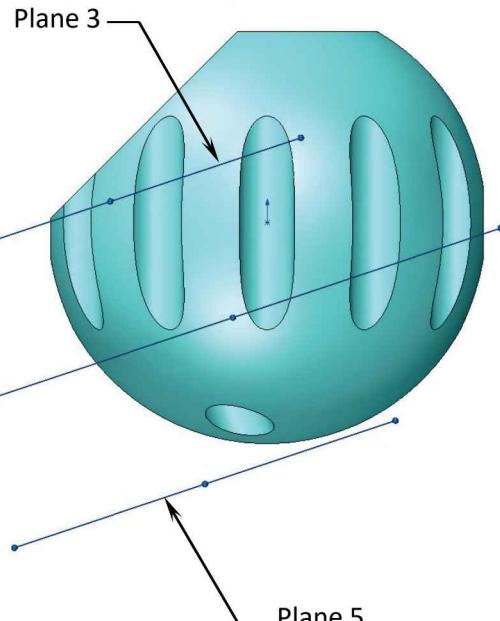
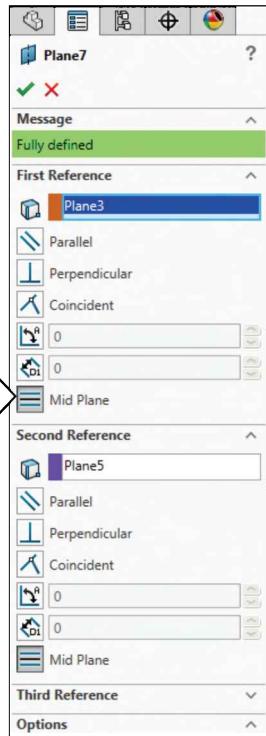
20. Creating a Mid-Plane: (Requires 2 parallel planes or 2 planar faces.)

Click  or select:
Insert / Reference Geometry / Plane.

Expand the Feature-Manager and select the **Plane3** and the **Plane5** to use as the first and second references.

The **Mid-Plane** option is selected automatically, if not, click the mid plane button (arrow).

A preview of the new plane appears in the middle of the two selected planes.

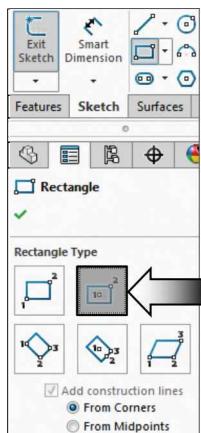


Click **OK**.

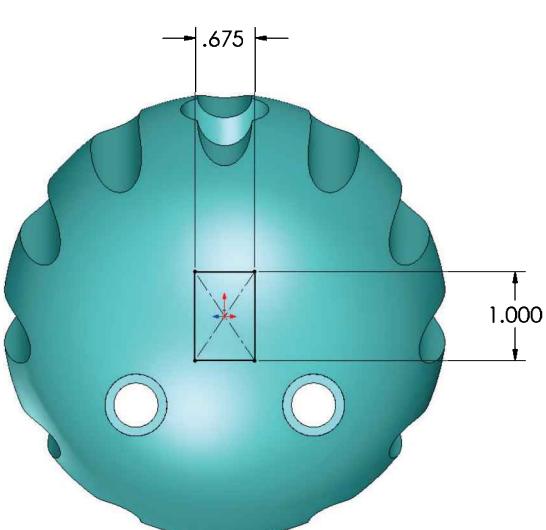
21. Creating a rectangular pocket:

Select the new plane (Plane7) and insert a **new sketch** .

Sketch a **Center Rectangle** that is centered on the origin.



Add the width and height dimensions to fully define the sketch.



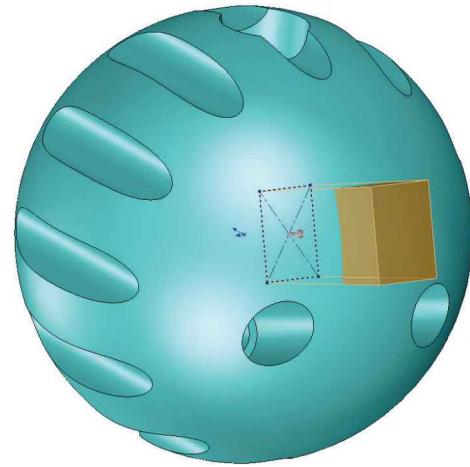
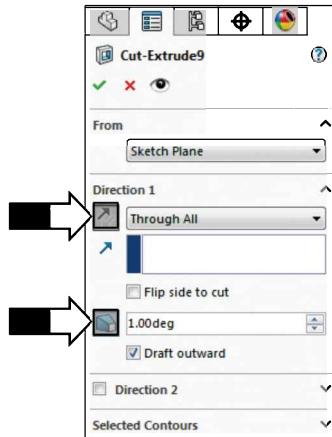
Click  or select Insert / Cut Extrude.

End Condition: Through All.

Reverse Direction: Enabled (arrow).

Draft: 1deg. Outward (arrow).

Click OK.



22. Adding fillets to the pocket:

Click  or select Insert Features / Fillet-Round.

Use the default Constant Radius option.

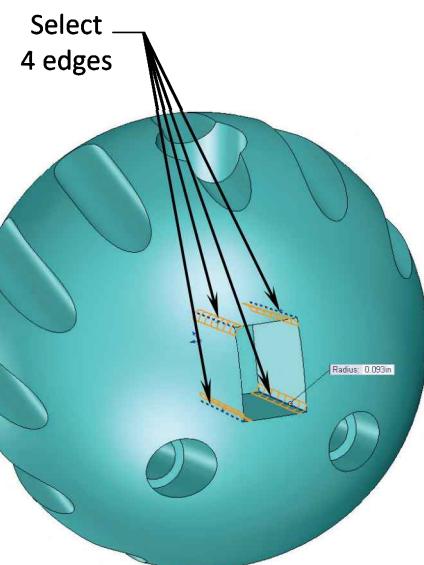
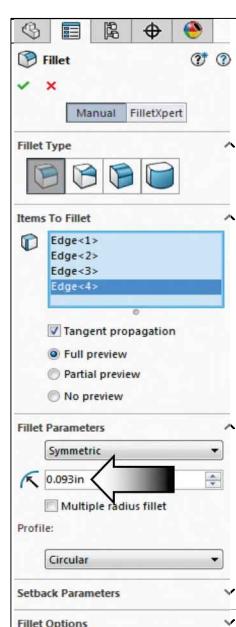
Enter .093in. for radius.

Select the 4 edges of the pocket as noted.

Note: Do not select the edges at the bottom or at the top of the pocket.

Enable the Full Preview checkbox.

Click OK.



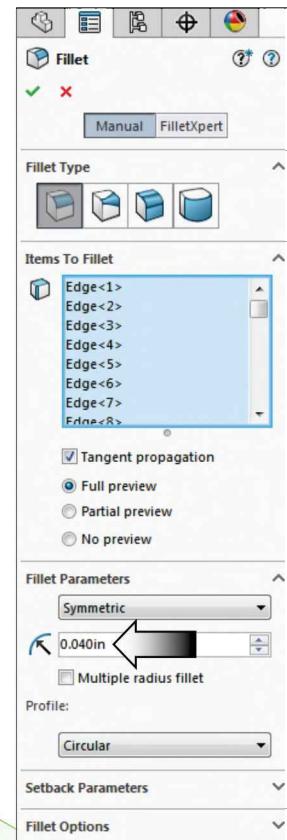
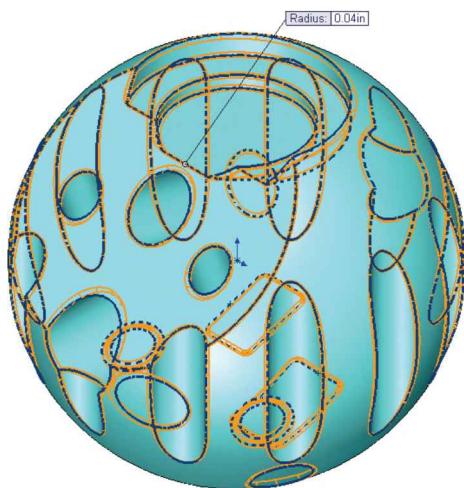
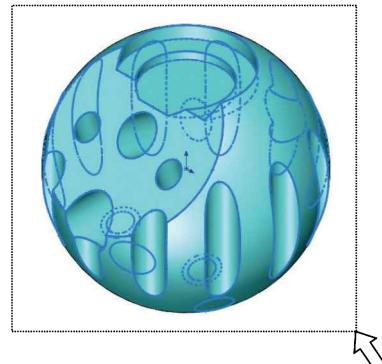
23. Adding Fillets to all edges:

Lasso or **Box-Select** around the entire part, (or press Control+A), to select all of its edges.

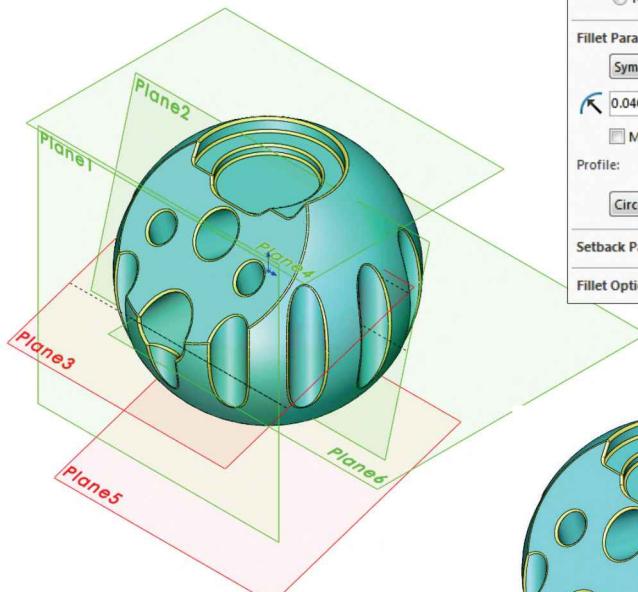
Click  or select **Insert Features / Fillet-Round**.

Enter **.040** for Radius.

Tangent Propagation: **Enabled**.



Click **OK**.



Hide the planes before saving the part. (**View / Hide-Show / Planes**).



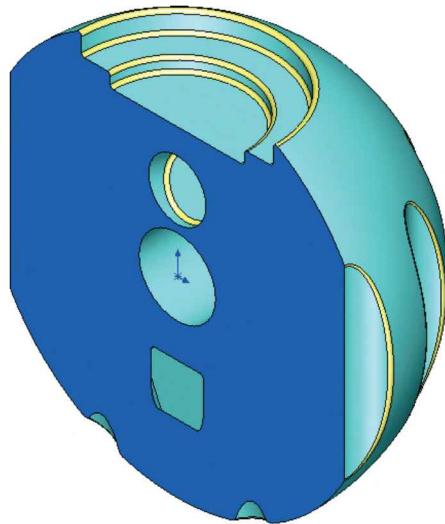
24. Saving your work:

Click **File / Save As / Planes Creation / Save**.

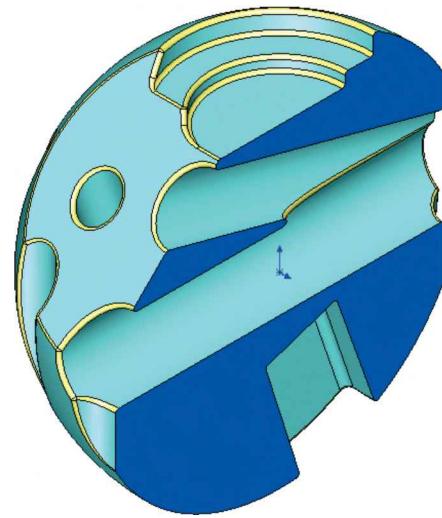
Questions for Review

1. Planes can be used to create sections in a part or an assembly.
 - a. True
 - b. False
2. A sketch can be extruded to a plane as the end condition by using the Up-To-Surface option.
 - a. True
 - b. False
3. Which one of the options below is not a valid command?
 - a. Parallel plane at Point.
 - b. Offset plane at Distance.
 - c. Perpendicular to another plane at Angle.
 - d. Normal to Curve.
4. To create a plane at Angle, you will need:
 - a. The Angle and a Reference plane.
 - b. The Angle and a pivot Line.
 - c. The Angle, a pivot Line, and a Reference plane.
5. To create a plane through Lines/Points, you will need at least:
 - a. One line and a point
 - b. Two lines and a point
 - c. Two lines and Two points
6. To create a Parallel Plane At Point, you will need a reference plane and a point.
 - a. True
 - b. False
7. When creating a Plane Normal To Curve, you can select:
 - a. A linear model edge
 - b. A straight line.
 - c. A 2D or 3D curve
 - d. All of the above

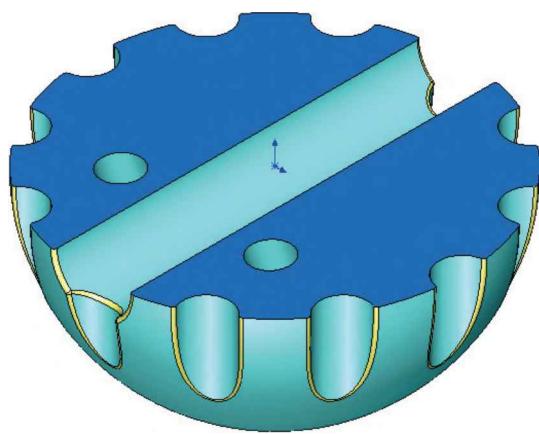
1.TRUE
2.TRUE
3.C
4.C
5.A
6.TRUE
7.D



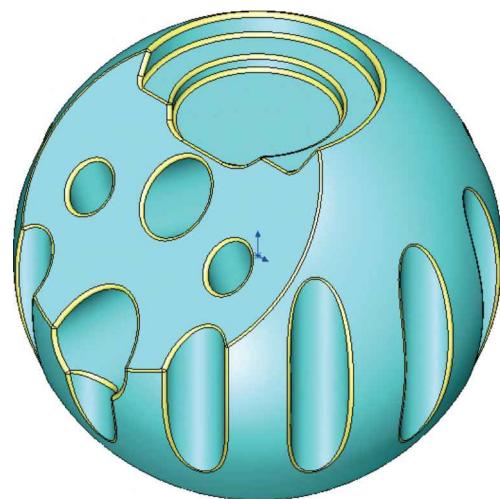
Section with **Front** plane



Section with **Right** plane



Section with **Top** plane

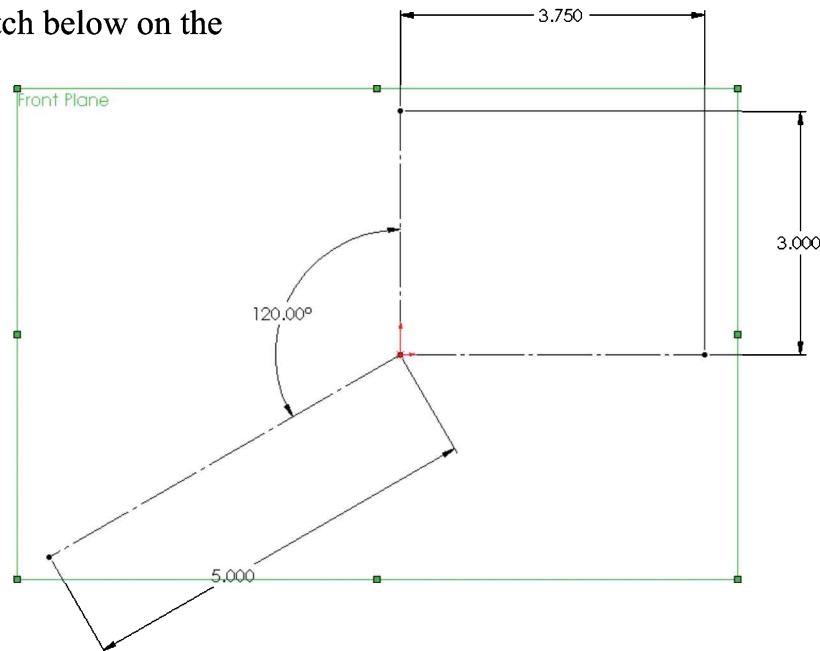


Isometric View

Exercise: Creating New Planes

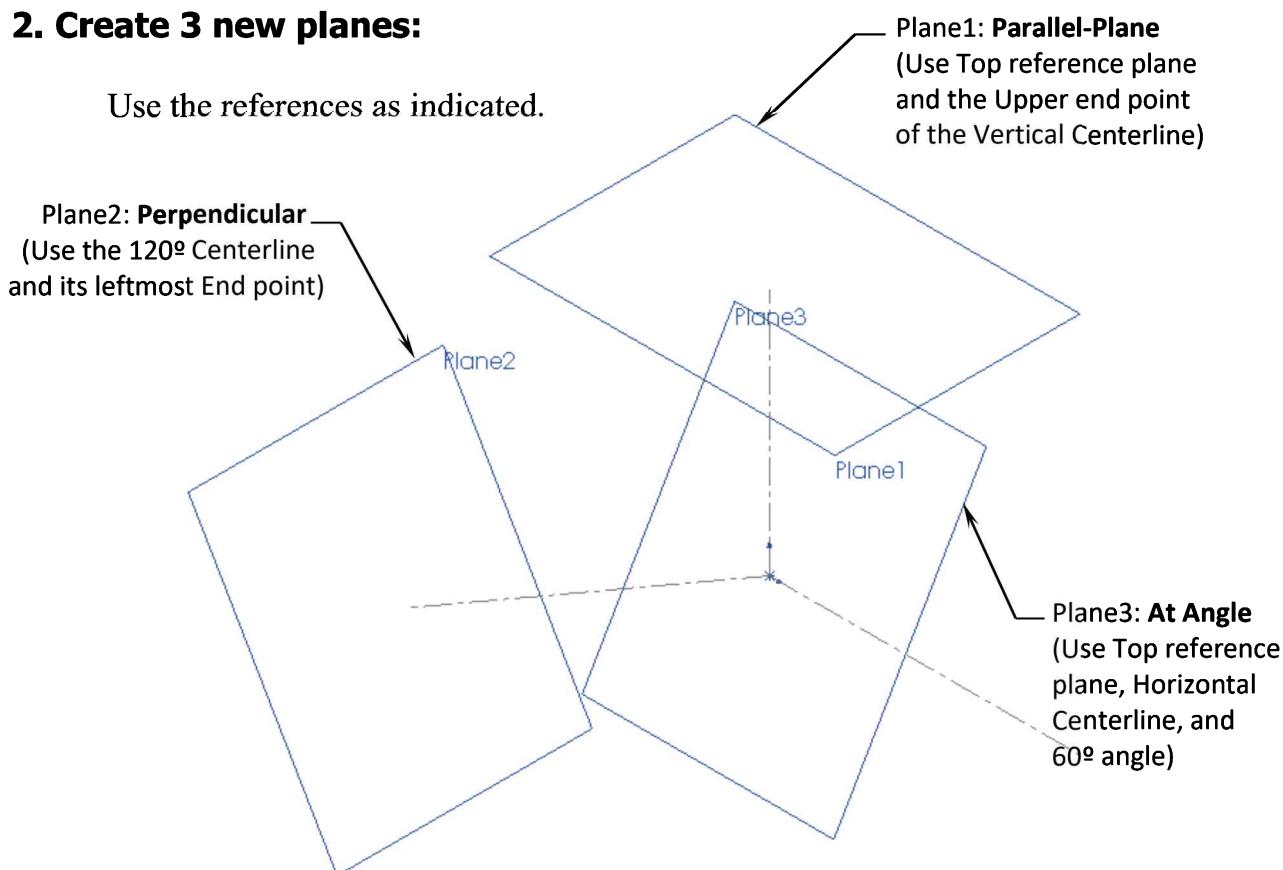
1. Create a reference sketch:

Create the sketch below on the **Front plane**.



2. Create 3 new planes:

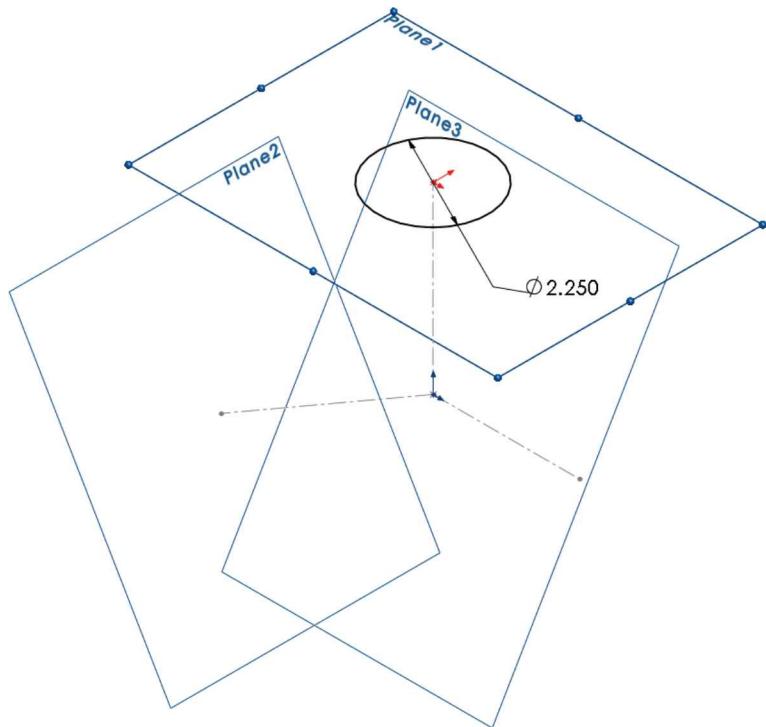
Use the references as indicated.



3. Open a new sketch on Plane1:

Sketch a **Circle** centered on the end point of the vertical centerline.

Add a the **2.250in** diameter dimension.

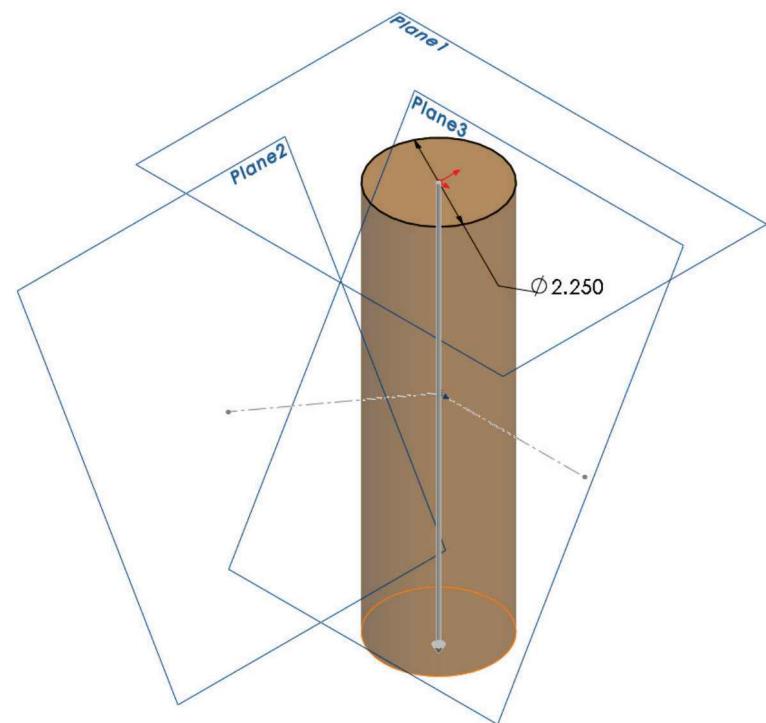
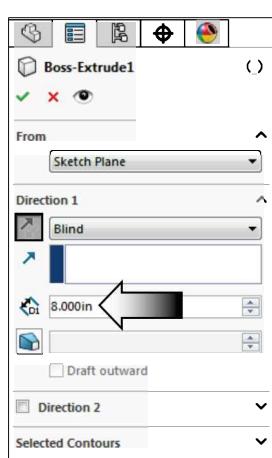


4. Extrude a boss:

Extrude Type: **Blind**.

Extrude Depth: **8.000in**

Reverse direction enabled.

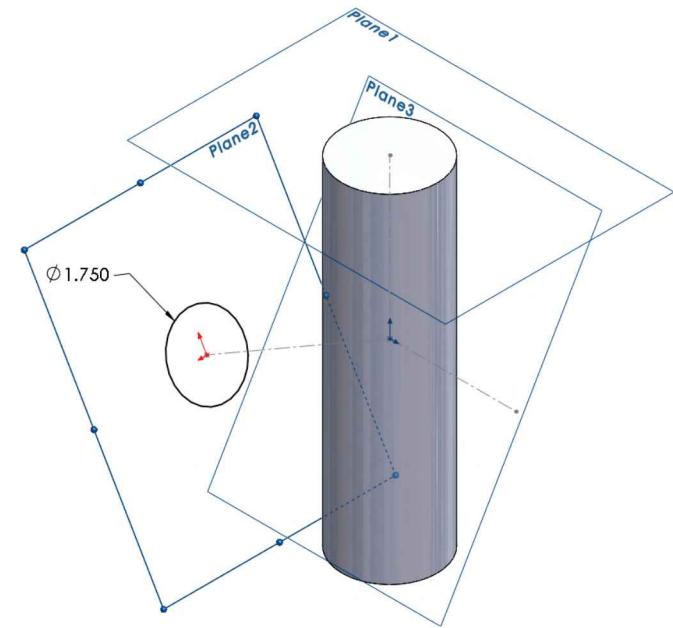


Click **OK**.

5. Open a new sketch on the Plane2:

Sketch a **Circle** centered on the left end of the 120deg. centerline.

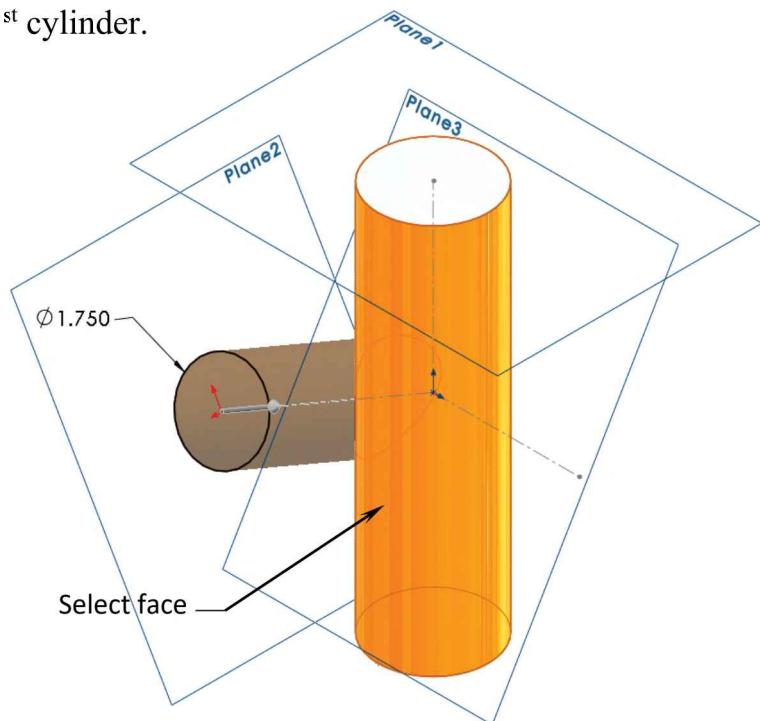
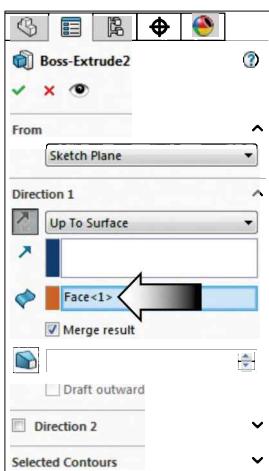
Add a the **1.750in** diameter dimension.



6. Extrude a boss:

Extrude Type: **Up To Surface**.

Select the **outer face** of the 1st cylinder.

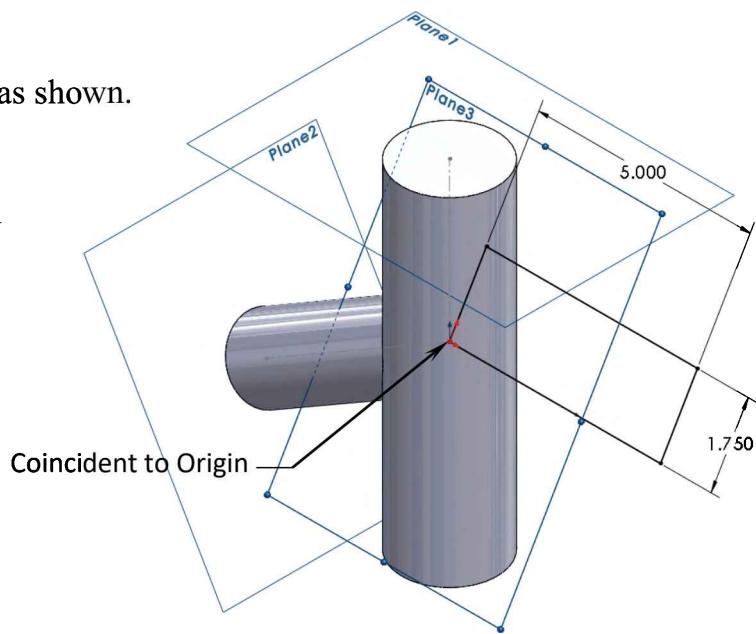


Click **OK**.

7. Open a new sketch on the plane3:

Sketch a **Corner Rectangle** as shown.

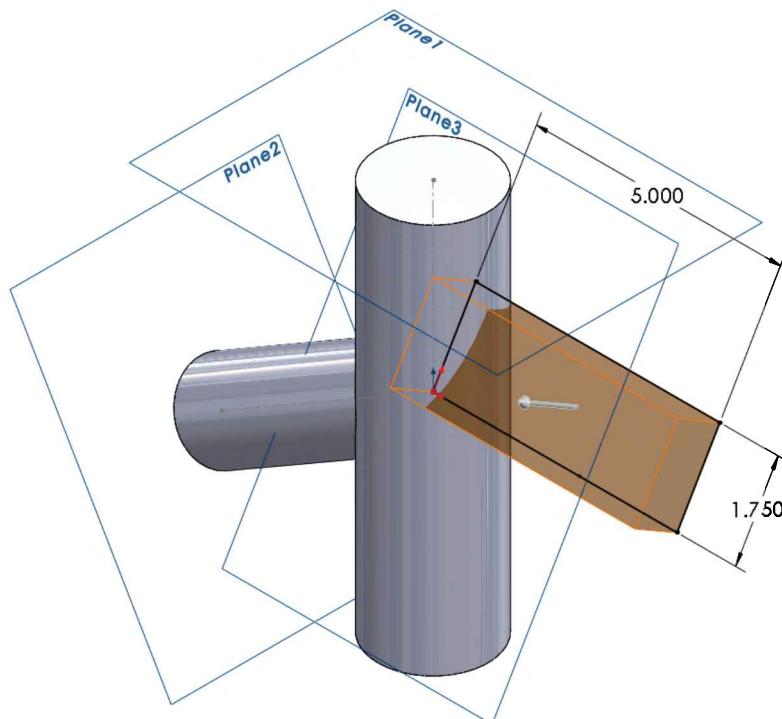
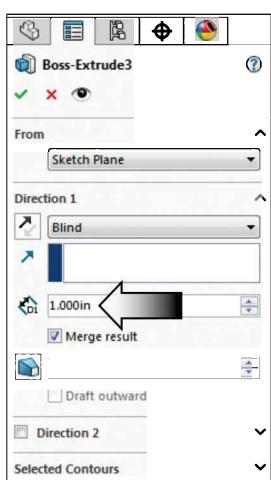
Add the height and the width dimensions.



8. Extrude a boss:

Extrude Type: **Blind**.

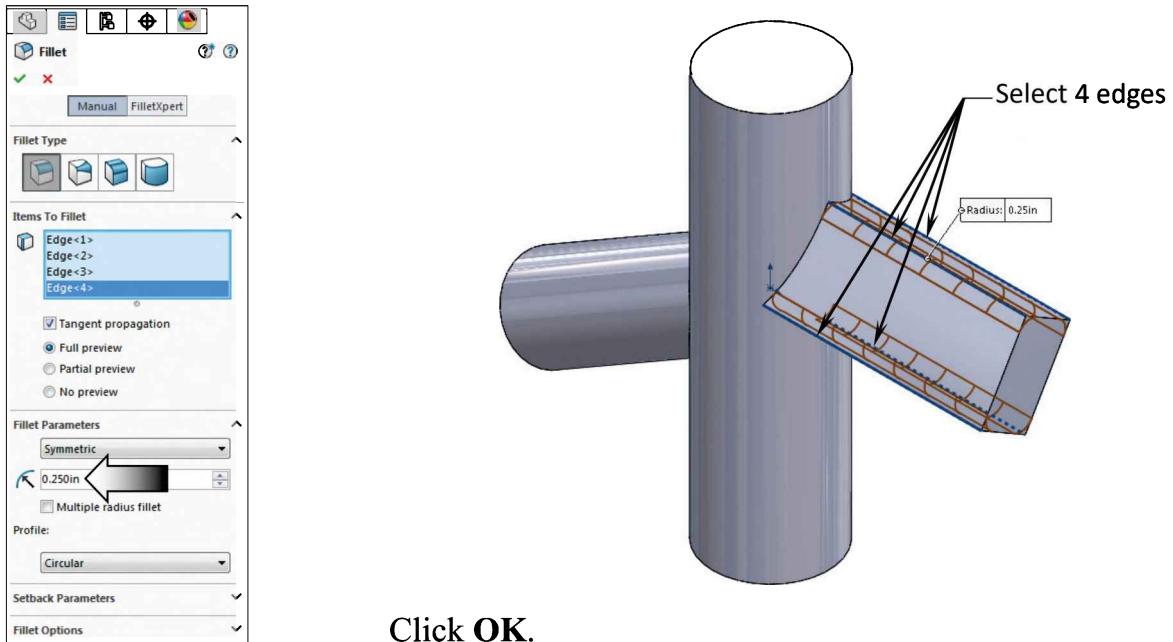
Extrude Depth: **1.000in**.



Click **OK**.

9. Add the .250in fillets:

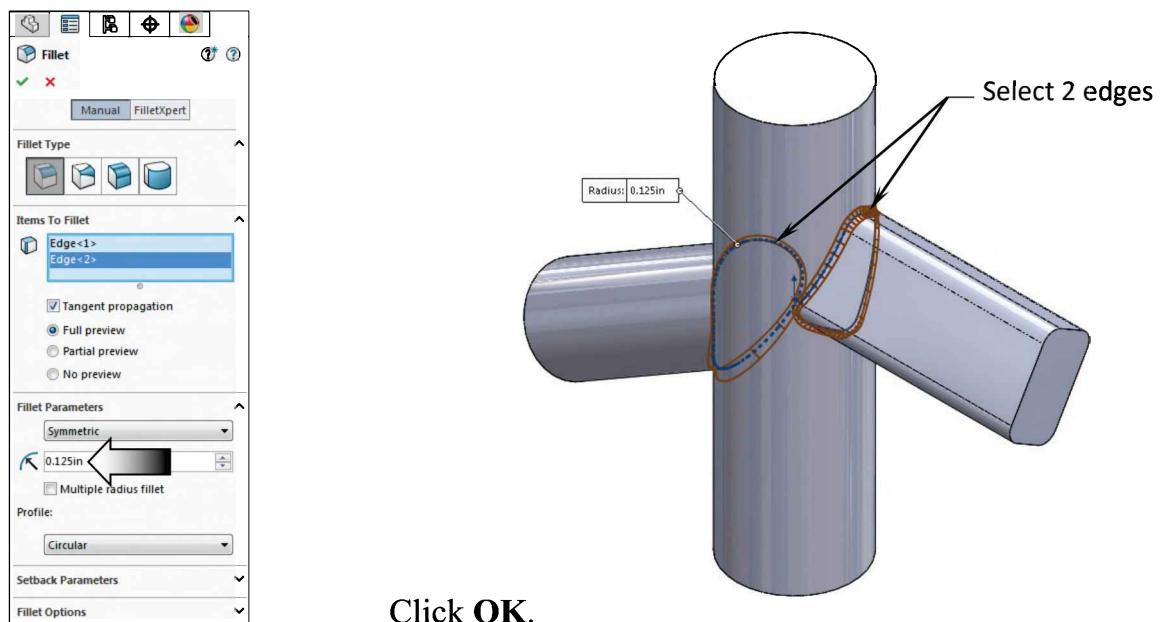
Add a fillet of **.250in** to the **4 edges** as noted.



Click OK.

10. Add the .125in fillets:

Add a fillet of **.125in** to the **2 edges** as indicated below.

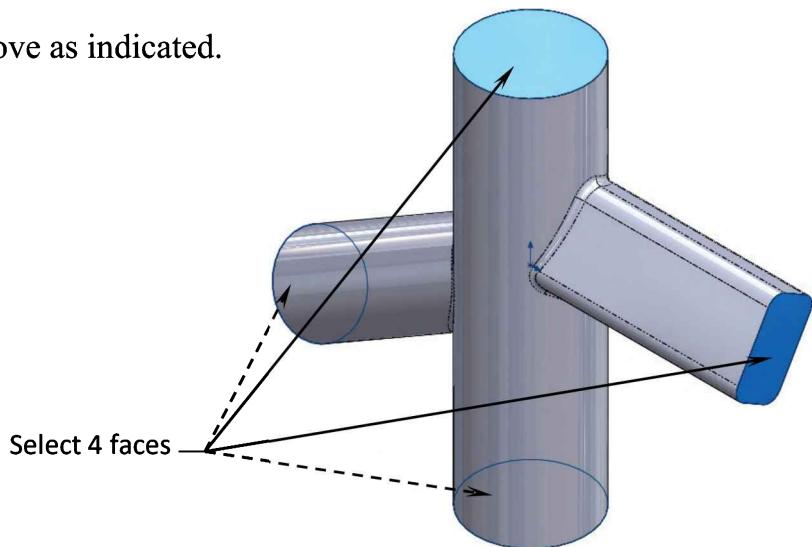
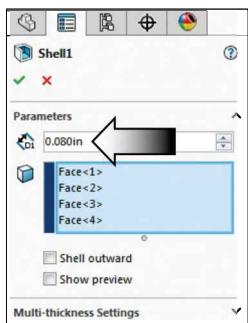


Click OK.

11. Shell the model:

Shell the model using a thickness of **.080in**.

Select the **4 faces** to remove as indicated.



Click **OK**.

12. Save your work:

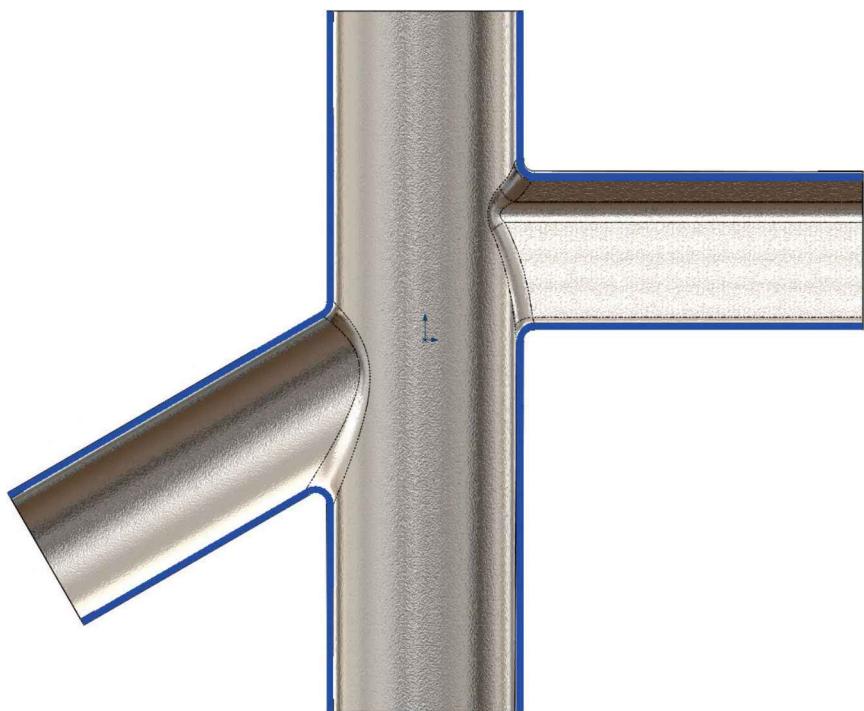
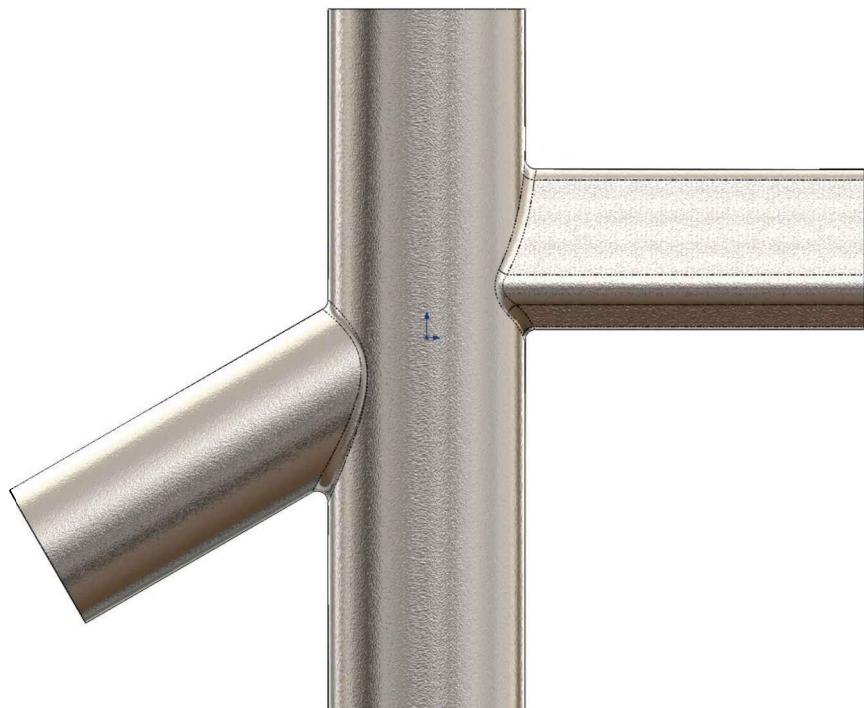
Click **File / Save As**.

Enter **Plane_Exe.sldprt** for the file name.

Click **Save**.



Close all documents.



CHAPTER 3

Advanced Modeling

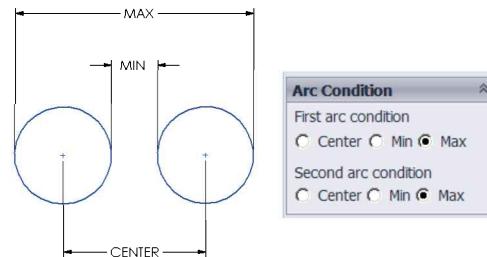
Advanced Modeling 5/8" Spanner



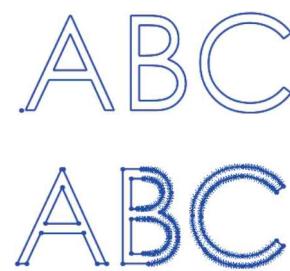
The draft option is omitted in this lesson to help focus in other areas.

The arc conditions Min / Max are options that can help with placing the dimensions on the tangents of arcs or circles. After a dimension is created, the arc conditions can be changed by right-clicking on the dimension and selecting the Leaders tab. Only two conditions can be specified at a time: either Center/Center, Min/Min, Max/Max, or Min/Max, etc.

Adding text on the model is another unique feature in SOLIDWORKS. This option allows the letters in the sketch to be extruded as an emboss or a cut, similar to other extruded features.



All letters in the same sketch are treated as one entity; they will be extruded at the same time and will have the same extrude depth. However, the option Dissolve-Sketch-Text is used to convert the sketch-text into individual sketch entities so that the shape and size of each letter can be modified.



To use this option simply right-click the text and select Dissolve-Sketch-Text.

This chapter and its exercise will guide you through some of the advanced modeling techniques as well as learn to use the Text tool to create the straight or curved extruded letters.

Advanced Modeling

5/8" Spanner



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI
Units: INCHES – 3 Decimals

Tools Needed:



Insert Sketch



Line



3 Point Arc



Text



Add Geometric Relations



Dimension



Sketch Fillet



Polygon



Plane



Base/Boss Extrude



Extruded Cut

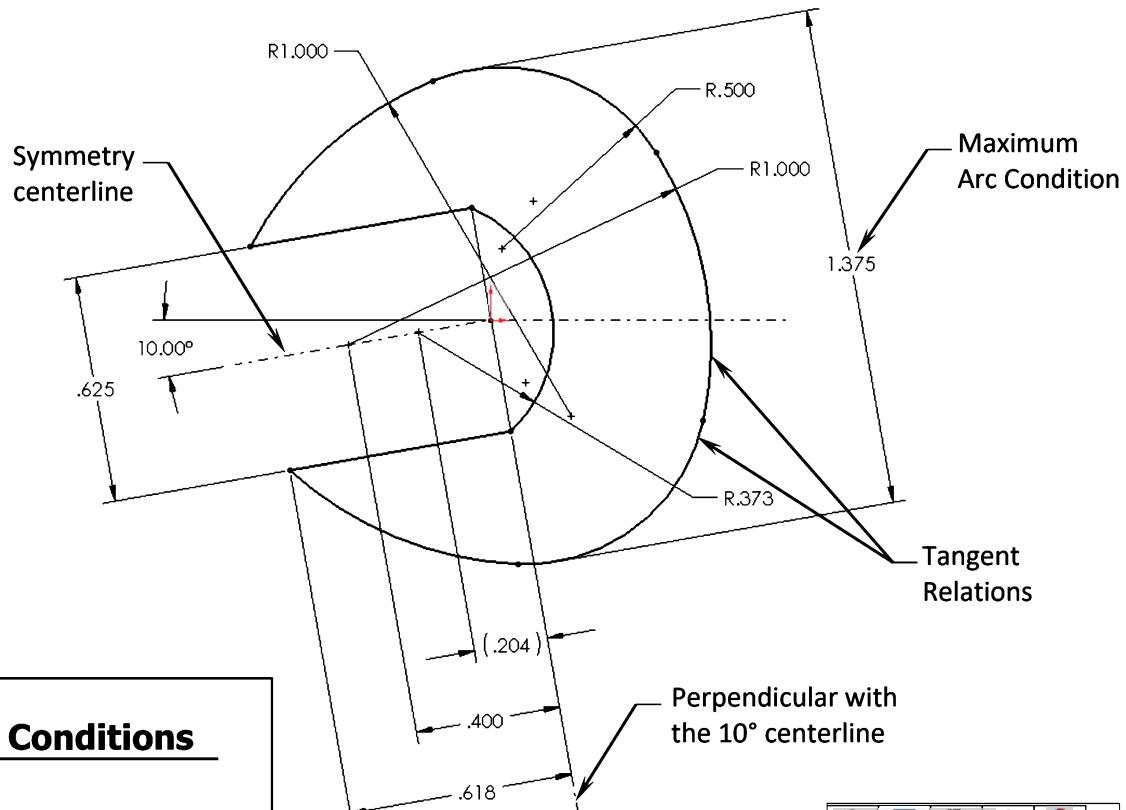


Fillet/Round

1. Opening the Spanner sketch document:

From the Training Files folder, open a document named: **Spanner Sketch**.

Edit the **Sketch1**. This is the open-end of the spanner and it is fully defined.



Arc Conditions

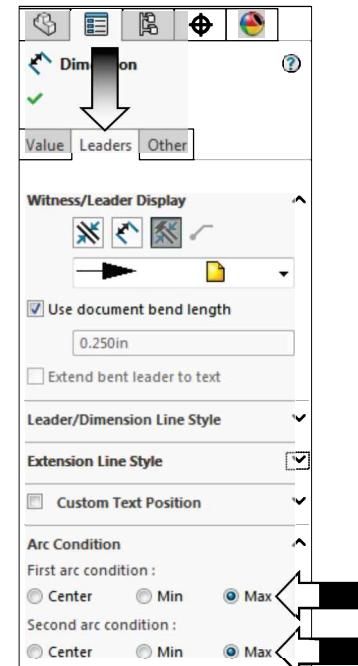
The 1.375 dimension is measured from the outer tangents of the arcs. This is called Maximum Arc Conditions*

* There are two options to create the Max/Min Arc condition dimensions:

Option 1: First, select the Smart Dimension tool, click on the 2 arcs and place the dimension.

Next, select the Leaders tab from the tree (circled). Click the Max options for both arcs (circled).

Option 2: Hold down the SHIFT key and click on the 2 arcs, the dimension's leader lines will snap to the arc tangents automatically.



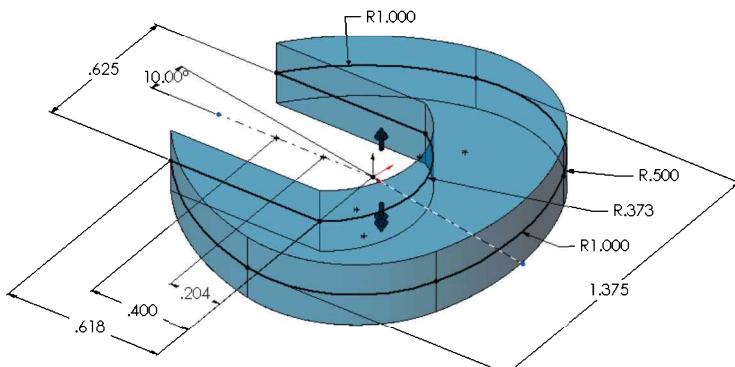
2. Extruding the base feature:

Click  or select **Insert / Boss-Base / Extrude**.

End Condition: **Mid Plane**.

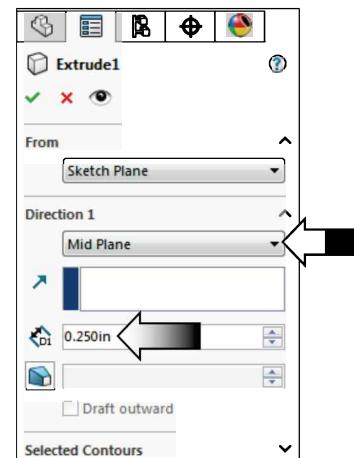
Extrude Depth: **.250 in.**

Click **OK**.



Renaming Features

Slow double-click on each feature's name (or press F2) and rename them to something more descriptive like Open-End, Transition-Body, Closed-End, etc...



3. Creating the transition sketch:

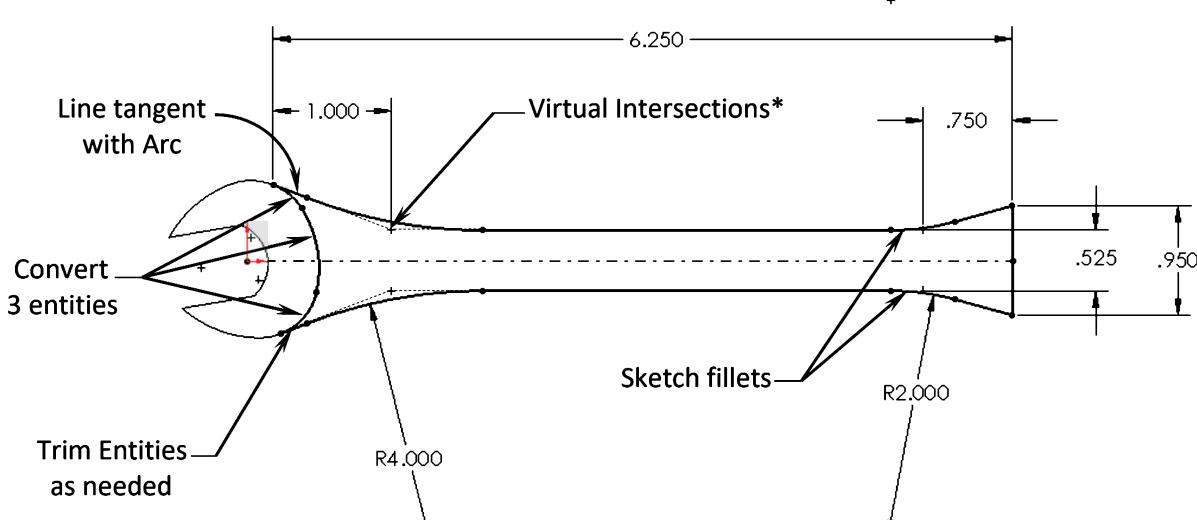
Select **Top** plane from the FeatureManager tree.

Click  or select **Insert / Sketch**.

Sketch the profile below using the **Line** command.

Add dimensions or Relations needed to fully define the sketch.

Note: Only add the *Sketch Fillets* after the sketch is fully defined.



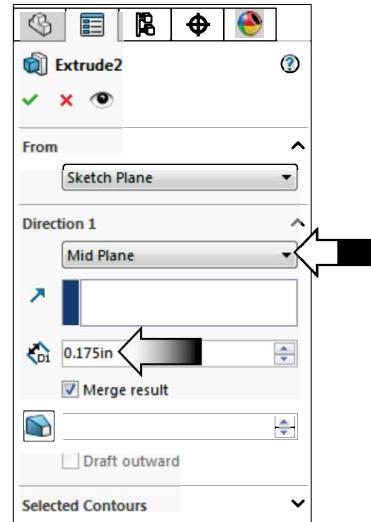
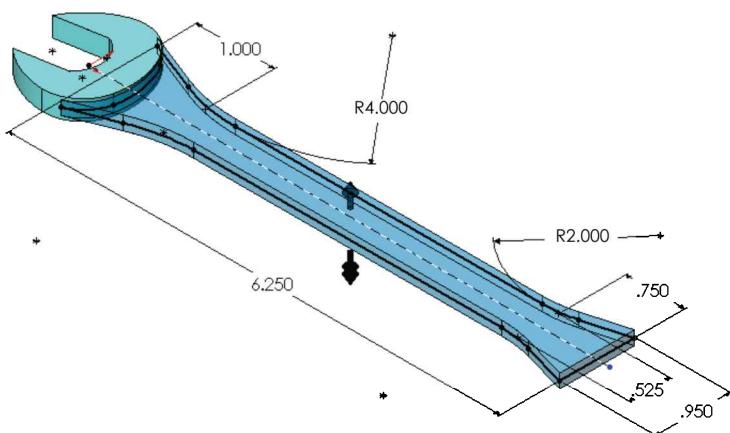
4. Extruding the Transition feature:

Click  or select **Insert / Boss-Base / Extrude**.

End Condition: Mid Plane

Extrude Depth: .175 in.

Click OK.

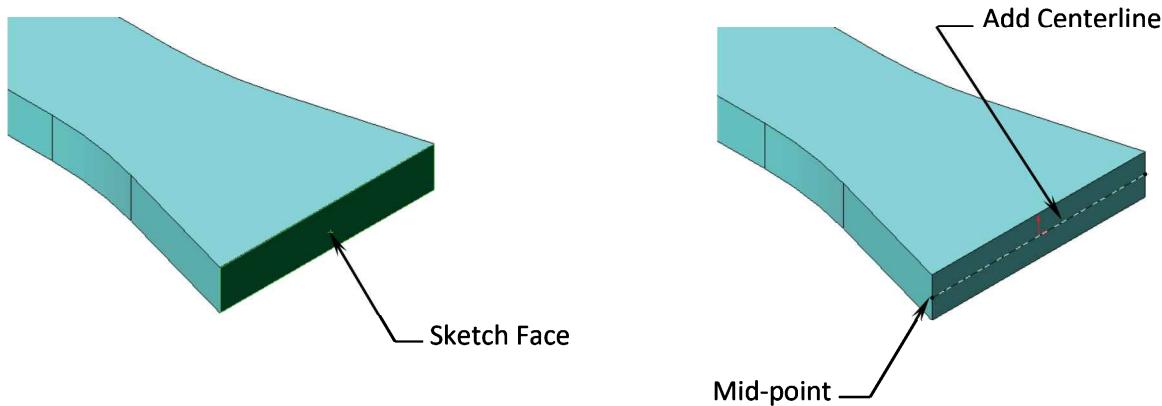


5. Adding the reference geometry:

Select the face as indicated.

Click  or select **Insert / Sketch**.

Sketch a **Centerline** at the mid-point of the two vertical edges.
This line will be used to create a new plane in the next step.



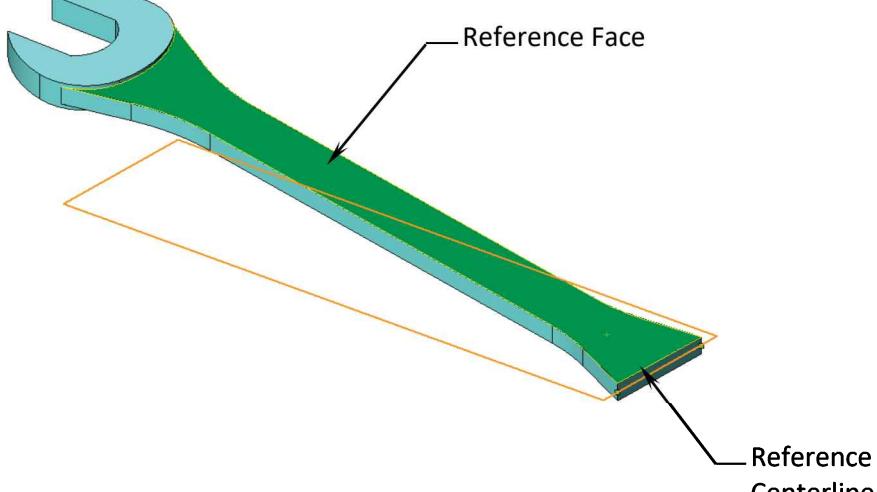
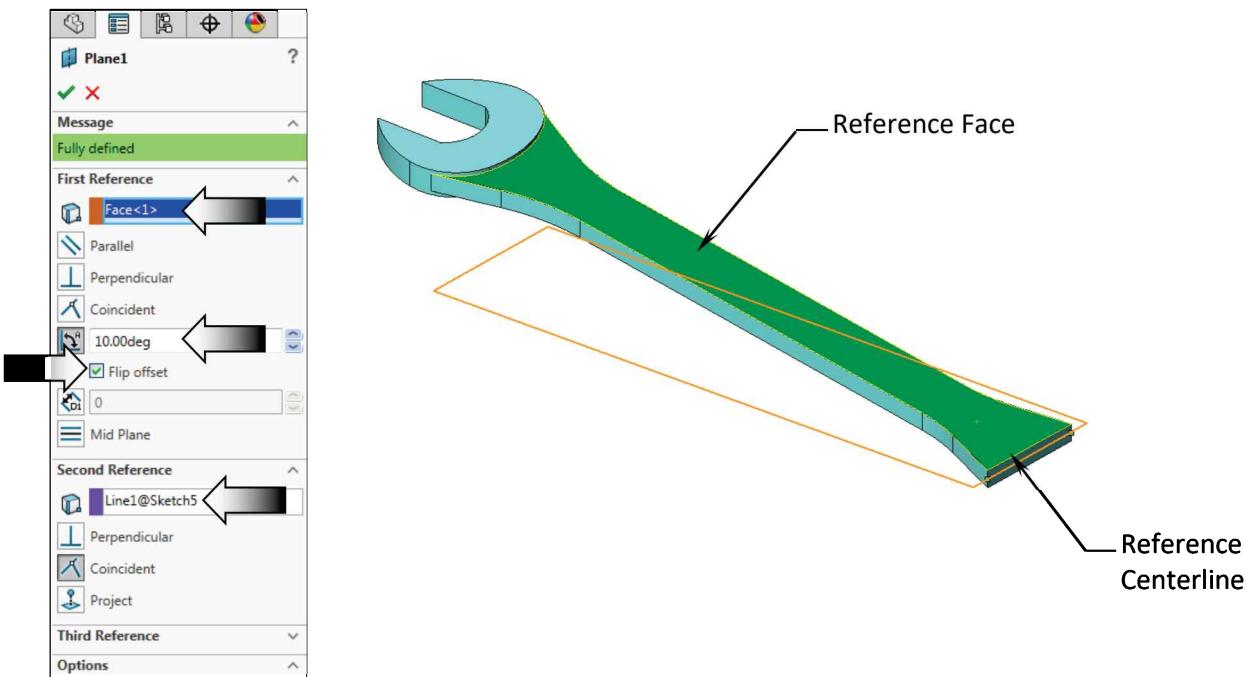
Exit the Sketch  or select **Insert / Sketch**.

6. Creating a new work plane: Plane at Angle

Click  or select **Insert / Reference Geometry / Plane**.

For Reference Entities, select the **Sketch4** (centerline) and the **upper face** of the Transition feature.

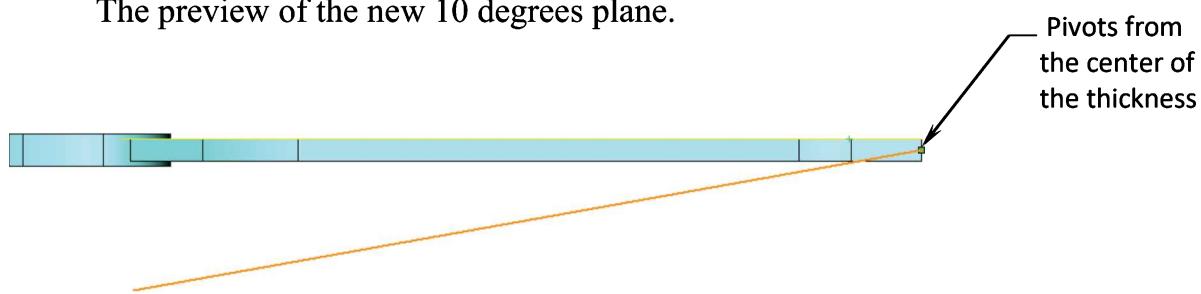
Enter **10 deg.** and click **Flip Offset** to place the new plane below the reference face.



Click **OK**.

A new plane is created using the upper reference face and pivoting about the centerline.

The preview of the new 10 degrees plane.

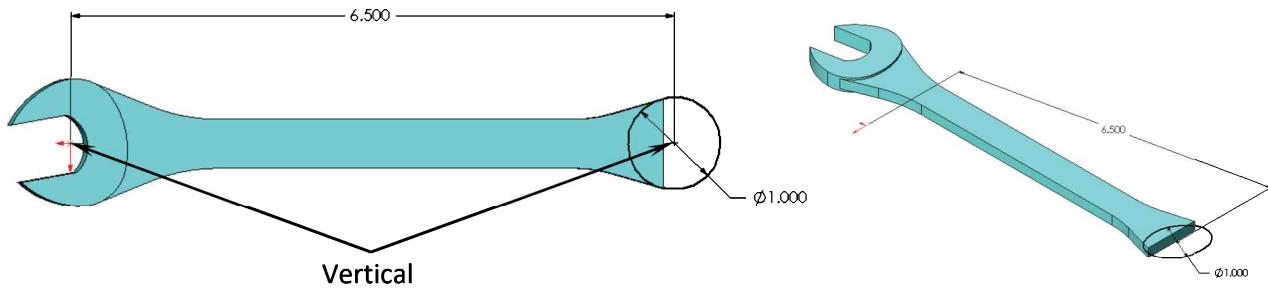


7. Creating the Closed-End sketch:

Select the new **10° plane** from the FeatureManager tree.

Click  or select **Insert / Sketch**.

Sketch a **circle** and add the dimensions and relations shown below.



8. Extruding the Closed-end feature:

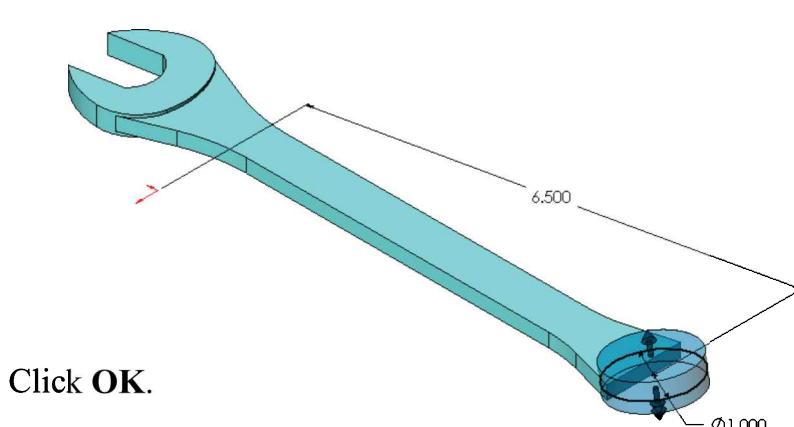
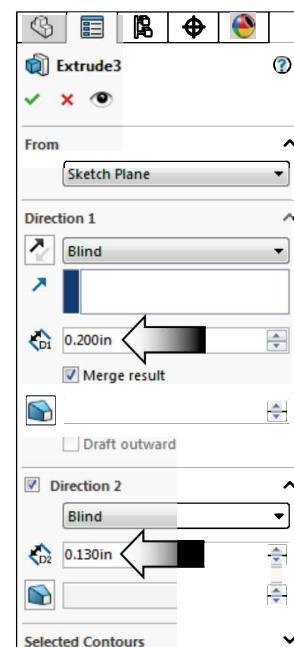
Click  or select **Insert / Boss-Base / Extrude**.

Direction 1: **Blind.**

Extrude Depth: **.200 in.**

Direction 2: **Blind.**

Extrude Depth: **.130 in.**



9. Adding a 12-sided polygonal hole:

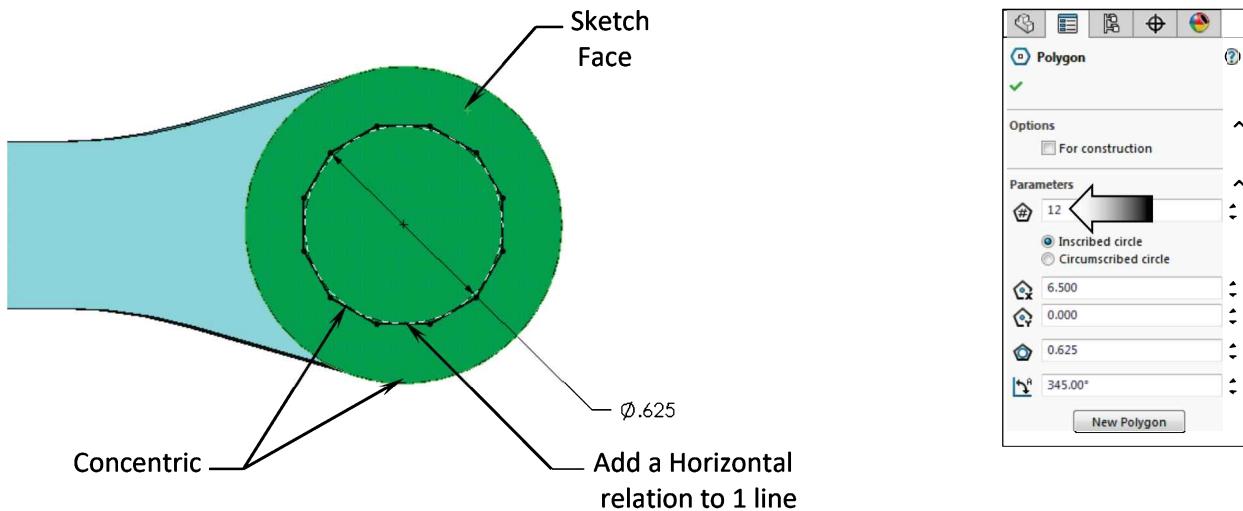
Select the face indicated as sketch plane.

Click  or select **Insert / Sketch**.

Sketch a **Polygon**  with **12 sides**  (arrow).

Add a **$\emptyset 0.625$** dimension to the construction circle on the inside.

Add a **Concentric** relation between the construction circle and the circular edge.

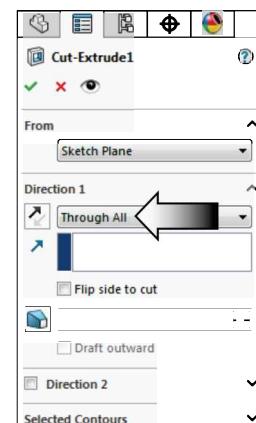
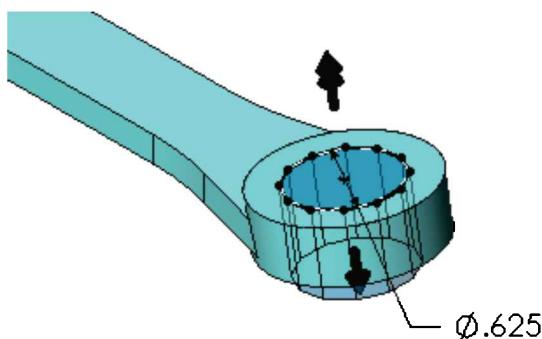


10. Extruding a cut:

Click  or select **Insert / Cut / Extrude**.

End Condition: **Through All**.

Click **OK**.

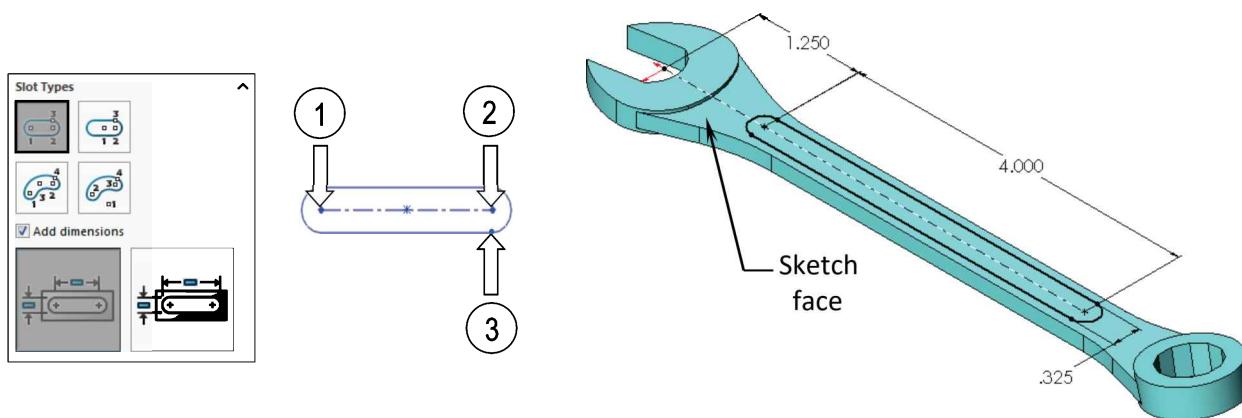


11. Creating the Recess profile:

Select the face indicated as sketch plane.

Click  or select **Insert / Sketch**.

Sketch the profile shown below using the **Straight-Slot** command .



Add dimensions or relations needed to fully define the sketch.

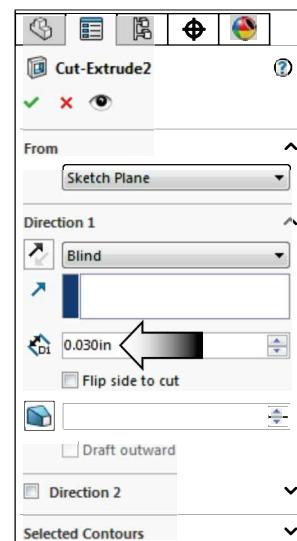
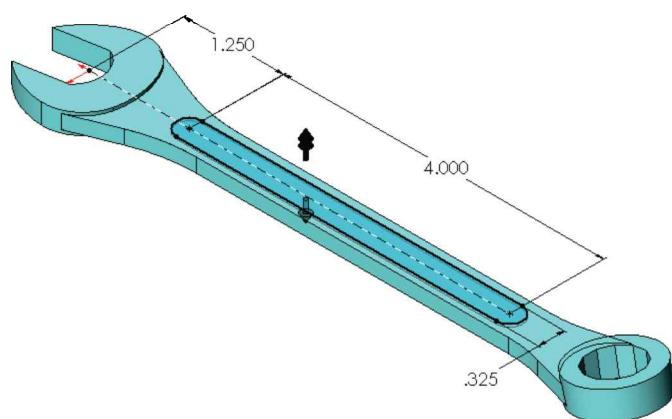
12. Extruding the Recessed feature:

Click  or select **Insert / Cut / Extrude**.

End Condition: **Blind**.

Extrude Depth: **.030 in.**

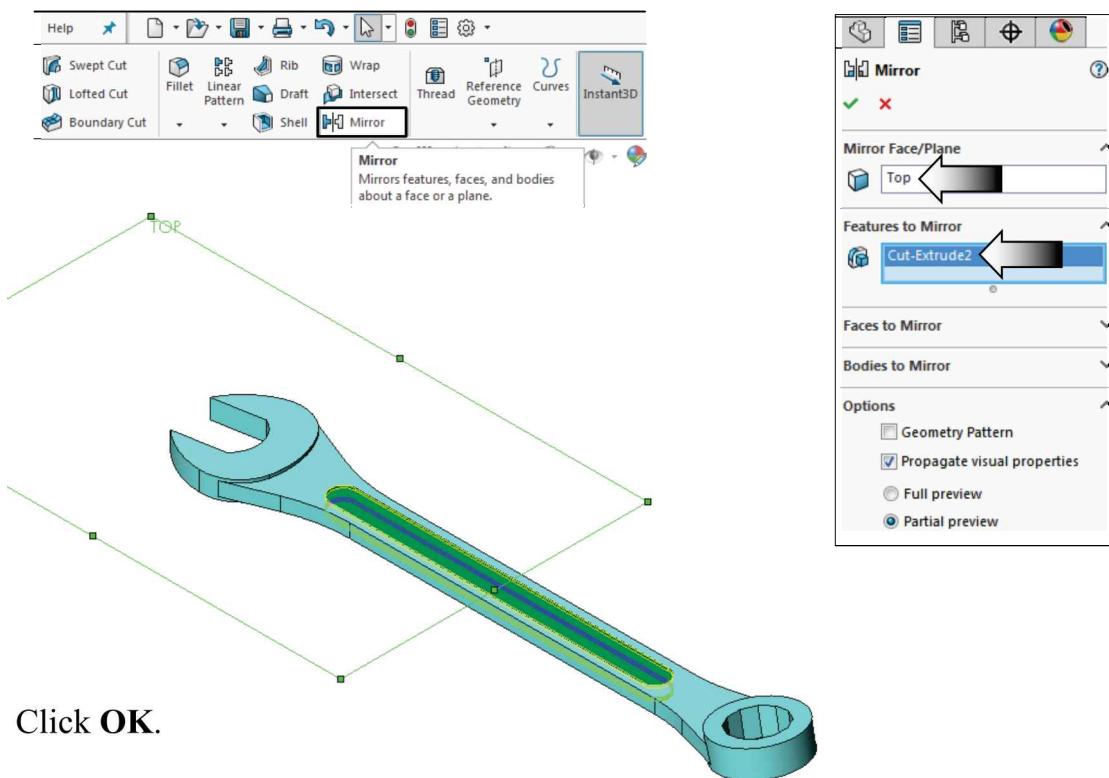
Click **OK**.



13. Mirroring the Recessed feature:

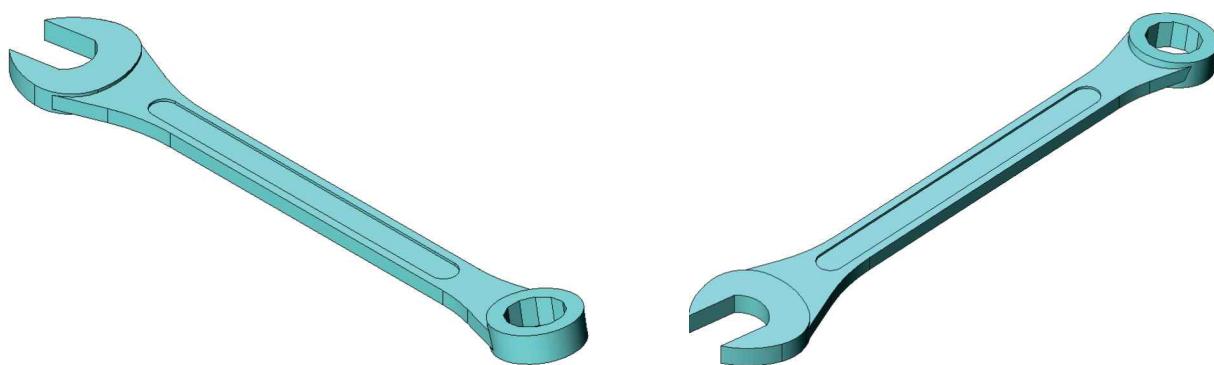
Hold the **Control** key, select the Top reference plane and the Recessed feature from the FeatureManager tree.

Click  or select **Insert / Pattern Mirror** menu and select **Mirror**.



Click **OK**.

Rotate  the model to verify the mirrored recessed feature.



14. Adding the .030" fillets:

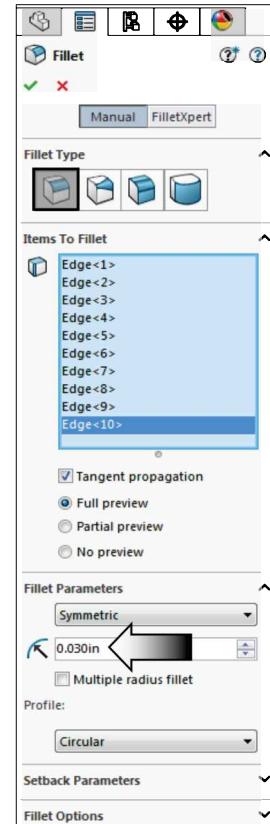
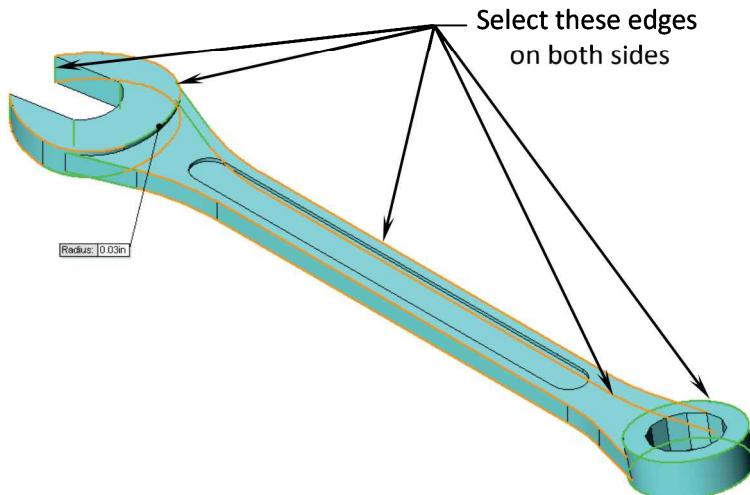
Click  or select Insert / Features / Fillet/Round.

For Radius, enter **.030 in.**

For Edges to Fillet, select the edges as indicated.

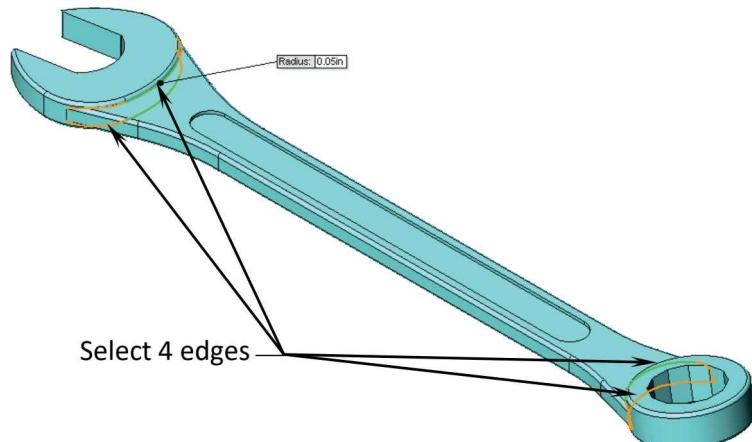
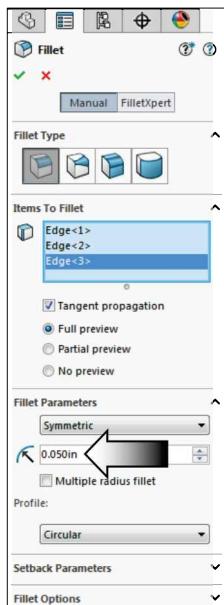
Tangent Propagation: **Enabled**.

Click **OK**.



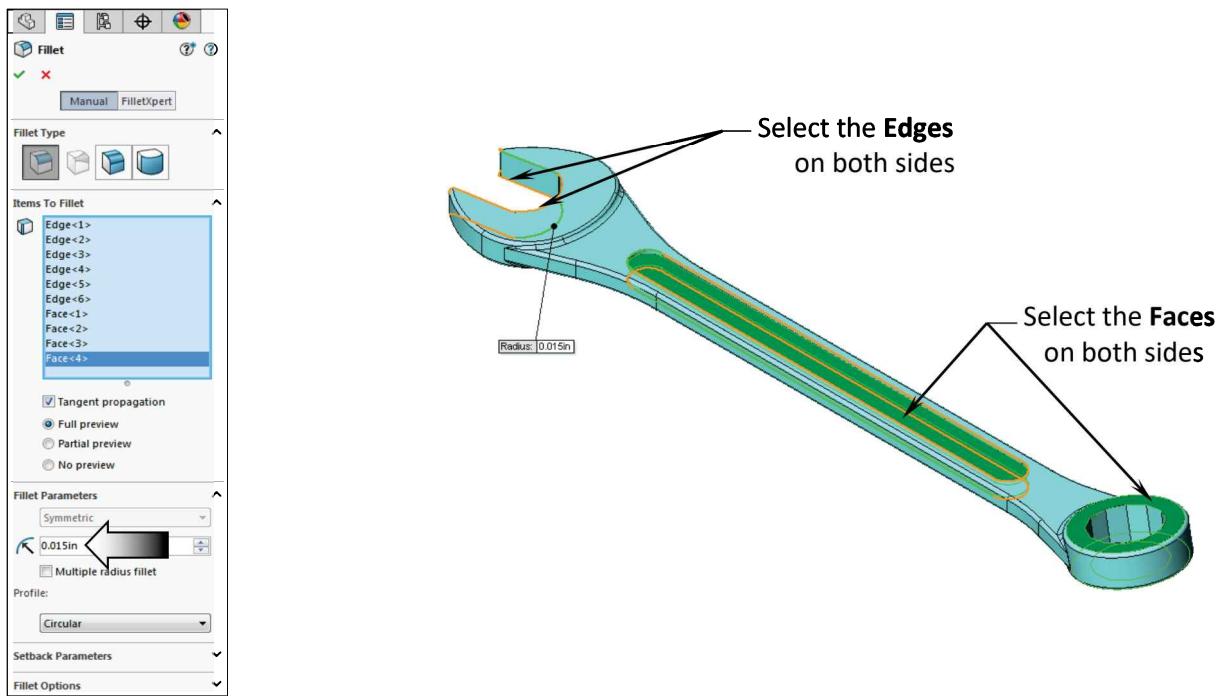
15. Adding the .050" fillets:

Repeat step 14 and add a **.050"** fillet to the **4 edges** shown below.



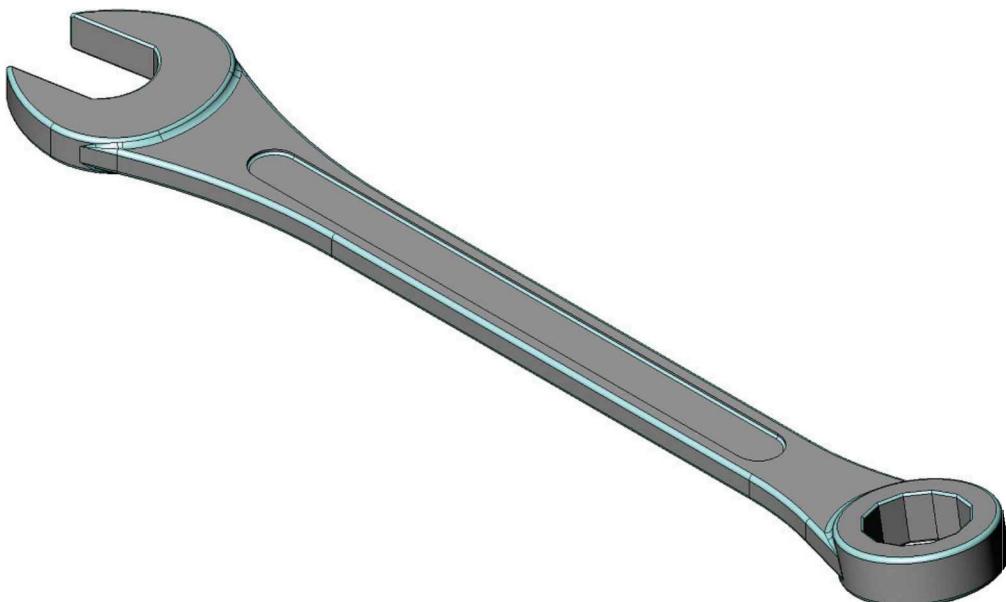
16. Adding the .015" fillets:

Click  and add a **.015"** fillet to the **edges** and **faces** shown below.



Click **OK**.

Verify your fillets with the model shown below.



17. Adding text:

Select the face indicated as sketch plane.

Click  or select **Insert / Sketch**.

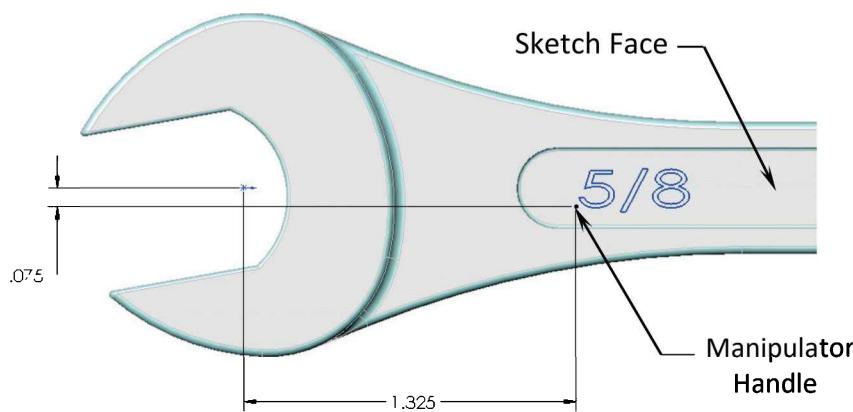
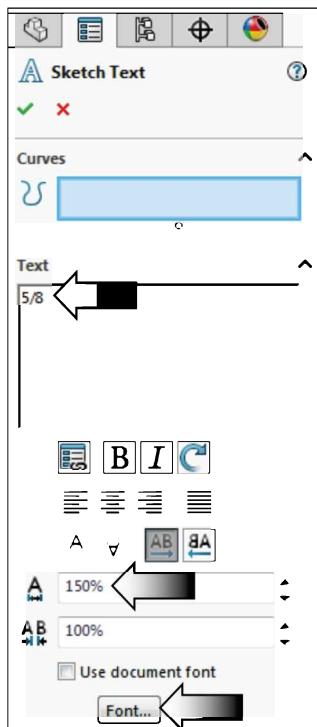
Click  and type **5/8** in the text dialog box.

Click **OK**.

Positioning Text

Each set of sketch text comes with a Manipulator Handle, dimensions can be added to this point to position the text.

Add dimensions to position the text.



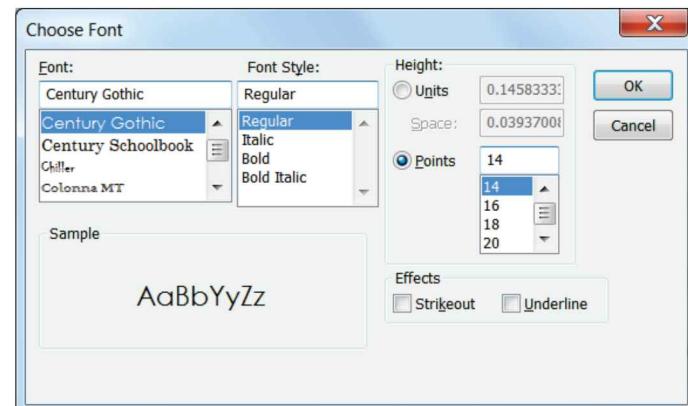
Clear Use document's font check box Use document's font .

Change Width factor to **150%** .

Leave Spacing at **100%** .

Font: **Century Gothic** .

Style: **Regular** Points size: **14 pt.**



NOTE:

Use the Curves option when you want your sketch letters to wrap along a curve.

It will work better if the curve is created in the same sketch, as construction geometry.

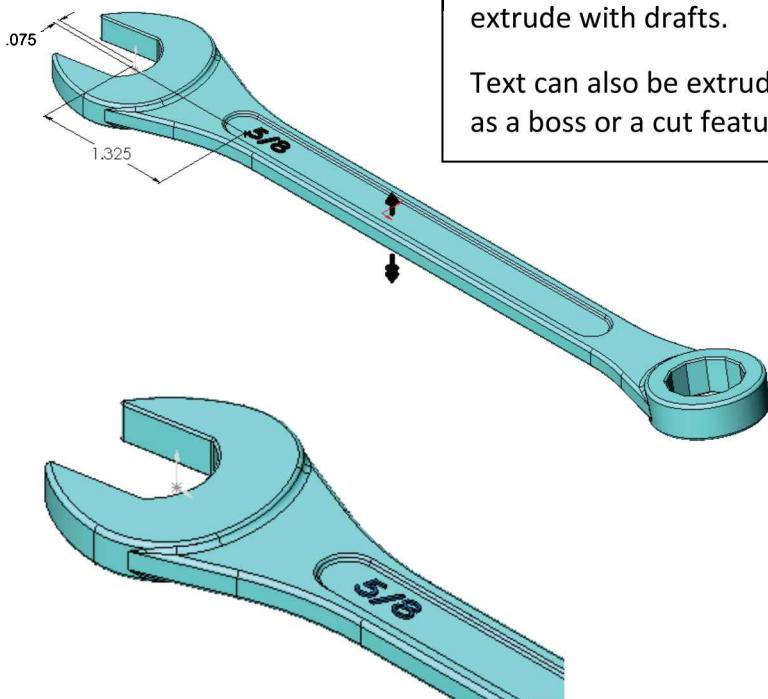
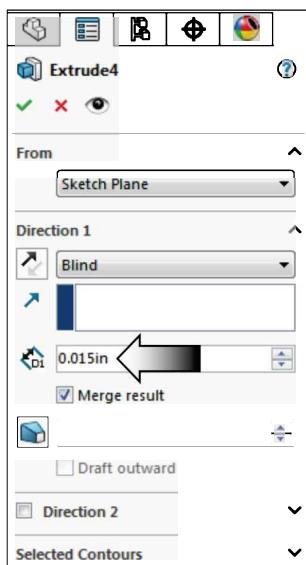
18. Extruding the text:

Click  or select **Insert / Boss-Base / Extrude**.

End Condition: **Blind**.

Extrude Depth: **.015 in.**

Click **OK**.



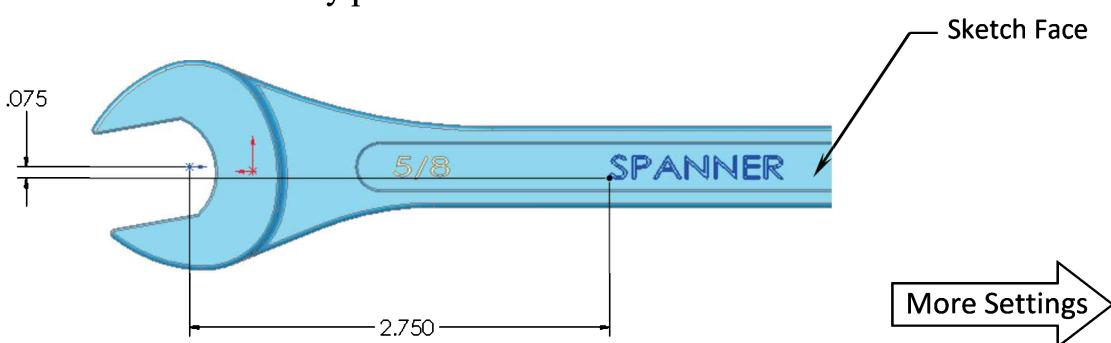
19. Adding more text:

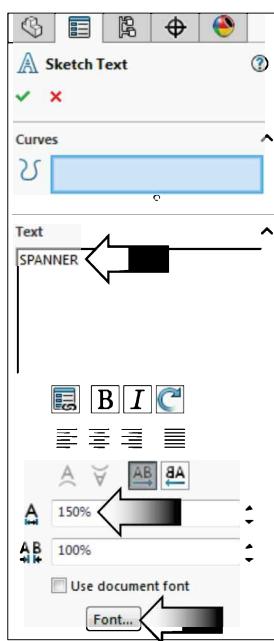
Select the face indicated as sketch plane.

Click  or select **Insert / Sketch**.

Click  and type **SPANNER** in the Text dialog box.

Add dimensions to fully position the text.





Clear Use document's font check box Use document's font.

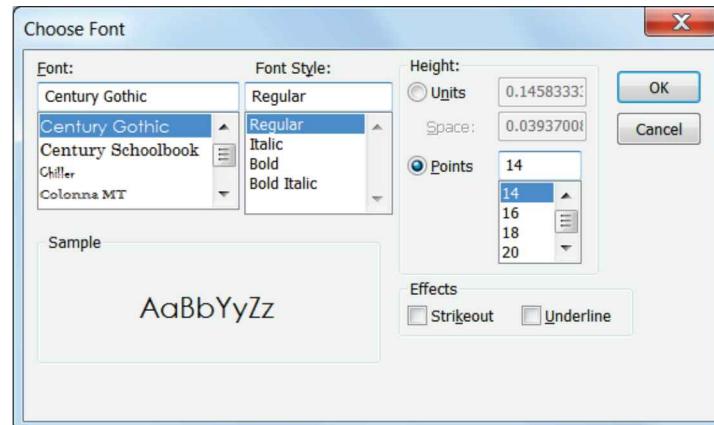
Change Width factor to **150%**

Keep Spacing at **100%**

Font: **Century Gothic** .

Style: **Regular**.

Points size: **14 pt.**



Click **OK**.

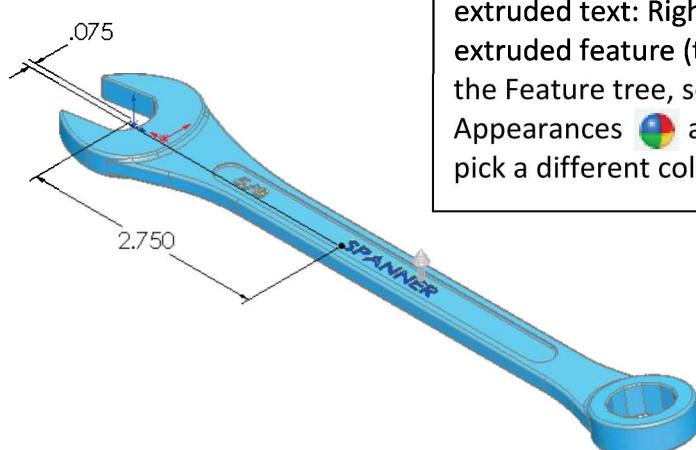
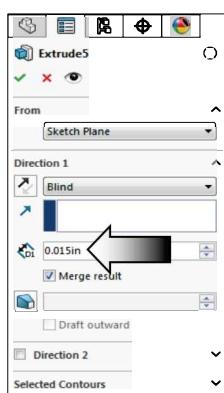
20. Extruding the text:

Click or select **Insert / Boss-base / Extrude**.

End Condition: **Blind**.

Extrude Depth: **.015 in.**

Click **OK**.



Text Color

To change the color of extruded text: Right click the extruded feature (text) from the Feature tree, select **Appearances** and pick a different color.

21. Saving your work:

Select File / Save as / Spanner / Save.



22. Optional:

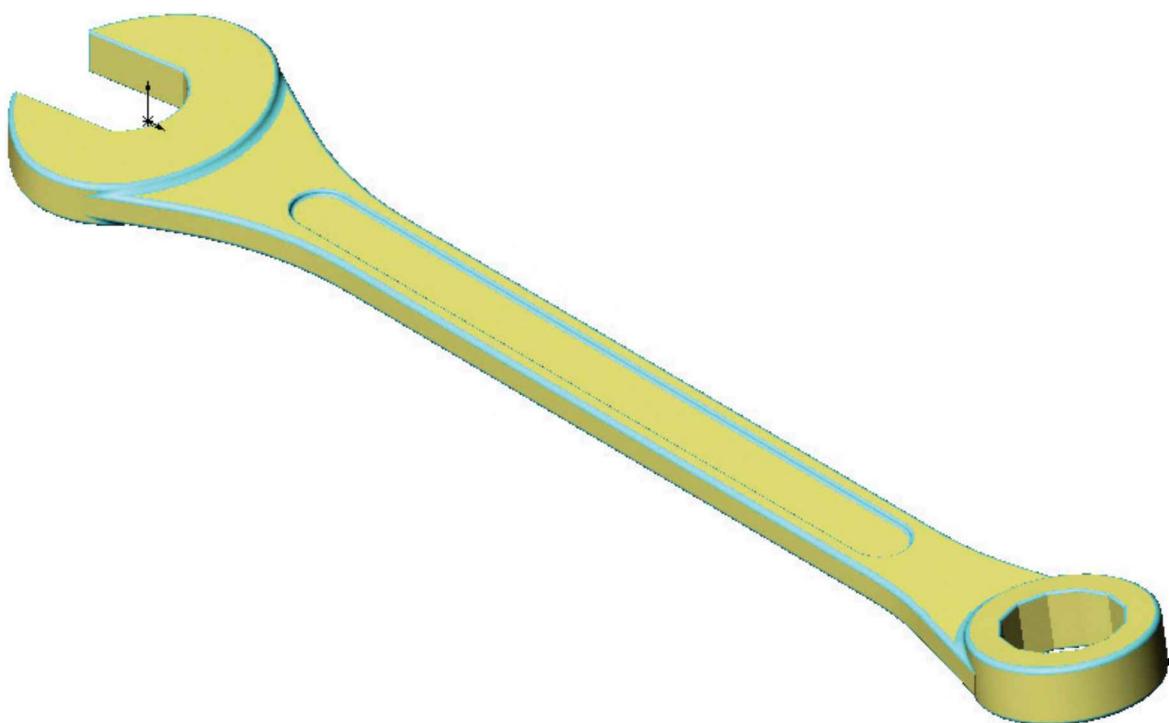
To add the same text on the opposite side of the part, repeat from step 17 through step 20.

Since the mirror option will not work correctly for text, you can either copy the sketch of the text, edit it, reposition, and extrude it again - OR - copy and paste the extruded text and then edit its sketch to reposition.

Questions for Review

1. The Min / Max conditions can be selected from the dimensions properties, under the Leaders tab.
 - a. True
 - b. False
2. The Mid-Plane extrude type protrudes the sketch profile to both directions equally.
 - a. True
 - b. False
3. It is sufficient to create a plane at an angle with a surface and an angular dimension.
 - a. True
 - b. False
4. When sketching a polygon, the number of sides can be changed on the Properties tree.
 - a. True
 - b. False
5. A 3D solid feature can be mirrored using a centerline as the center of mirror.
 - a. True
 - b. False
6. Text cannot be used to extrude as a boss or a cut feature.
 - a. True
 - b. False
7. Extruded text can be mirrored just like any other 3D features.
 - a. True
 - b. False
8. Text in a sketch can be extruded with drafts, inward or outward.
 - a. True
 - b. False

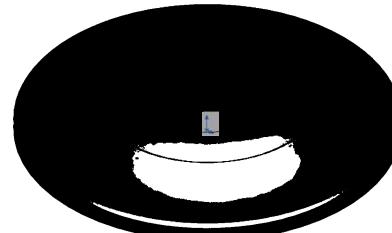
1. TRUE
2. TRUE
3. FALSE
4. TRUE
5. FALSE
6. FALSE
7. TRUE
8. TRUE



Exercise: Circular Text Wraps

1. Opening a part file:

From the Training Files folder, open an existing part document named: **Text Wrap**.



2. Adding Text:

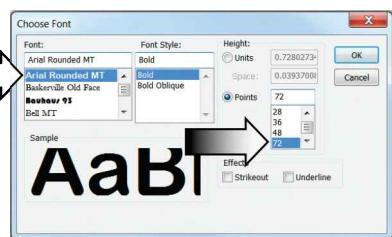
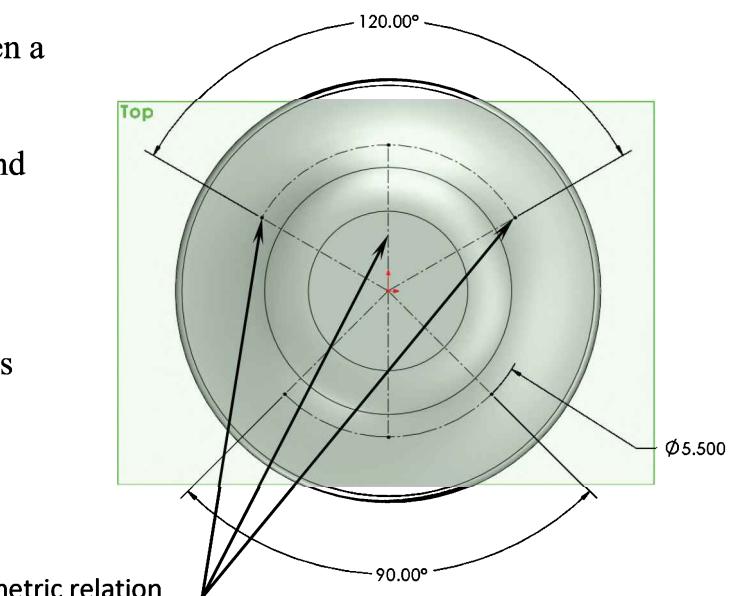
Select the **Top** plane and open a new sketch.

Sketch a circle at **$\varnothing 5.500$** and convert it into construction geometry.

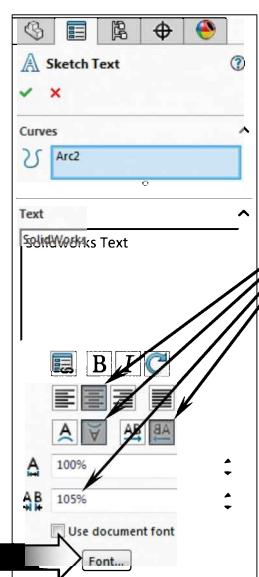
Sketch the other centerlines, trim, then add the dimensions and relations as indicated.

Click the **Text** command.

Enter the text:
SolidWorks.



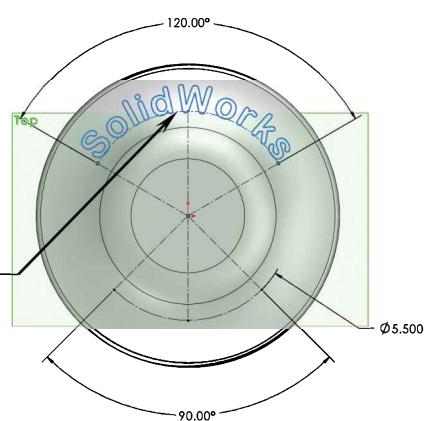
Select the upper construction curve to wrap the text around it.



Select these options

Click the **Font** button and set the size to **72 points**.

Select all other options as indicated to align the text.



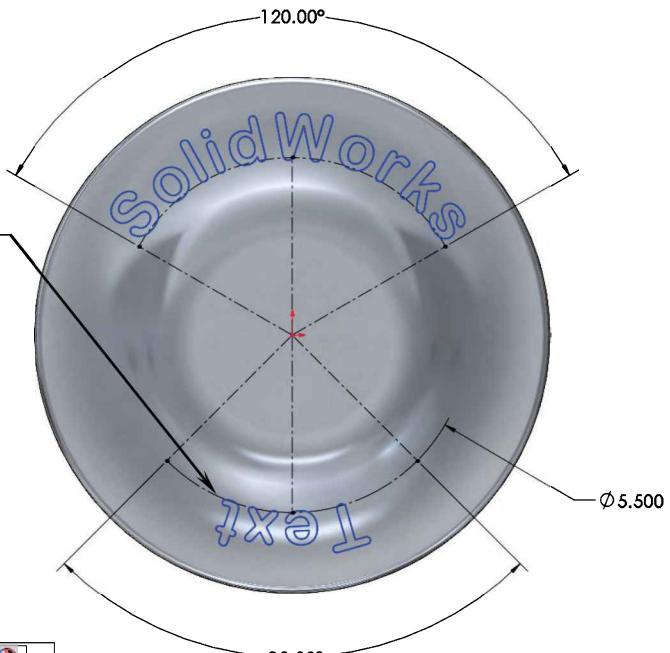
3. Repeating:

Still working in the same sketch, repeat step 2 and add the word: **Text** at the bottom.

Select this curve

Add a Symmetric relation between the 2 endpoints of the construction curve and the vertical centerline.

Use the same text settings as the last text.



4. Extruding the text:

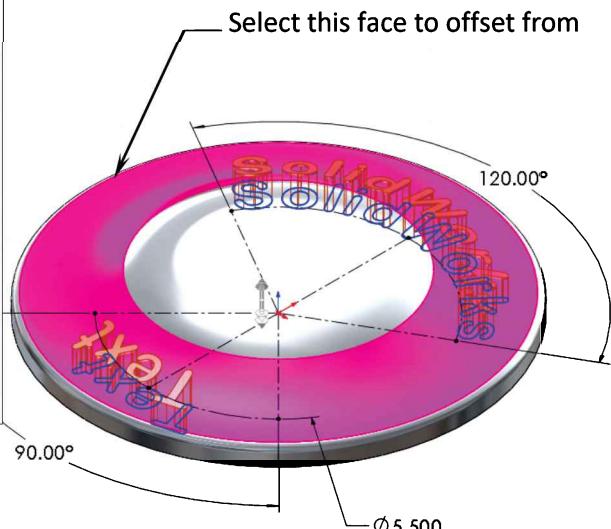
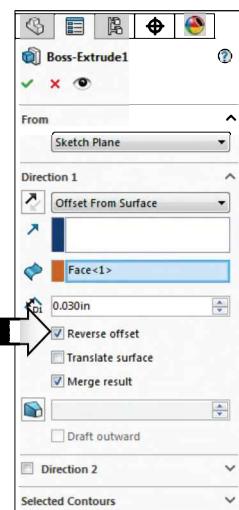
Click **Extruded Boss-Base**.

Change Direction 1 to **Offset From Surface**.

Enter **.030"** and click **Reverse Offset**.

Select the **face** as indicated to offset from.

Click **OK**.



5. Saving your work:

Click **File / Save As**.

Enter **Circular Text Wrap**.

Press **Save**.

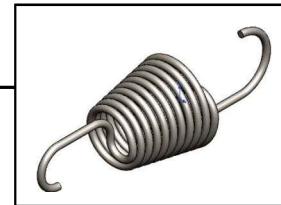


CHAPTER 4

Sweep With Composite Curves

Sweep **with Composite Curves**

Helical Extension Spring



Unlike extruded or revolved shapes, the sweep option offers a more advanced way to creating complex geometry, where a single profile can be swept along a 2D path or a 3D curve to define the feature's characteristics.

To create a sweep feature the Sweep Path should be created first, then a single closed sketch Profile is created after.

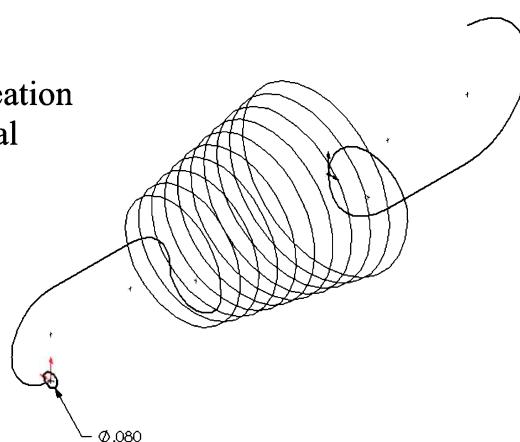
The Profile will be related to the Sweep Path with a Pierce or a coincident relation.

When the Profile is swept along the Sweep Path, the Guide Curves can help control the feature's accuracy and its behaviors like twisting, tangencies, etc.

The Composite Curve  option allows multiple sketches or model edges to be jointed into one continuous path to use in sweep features.
(The sketches must be connecting with one another in order for the composite curve to work.)

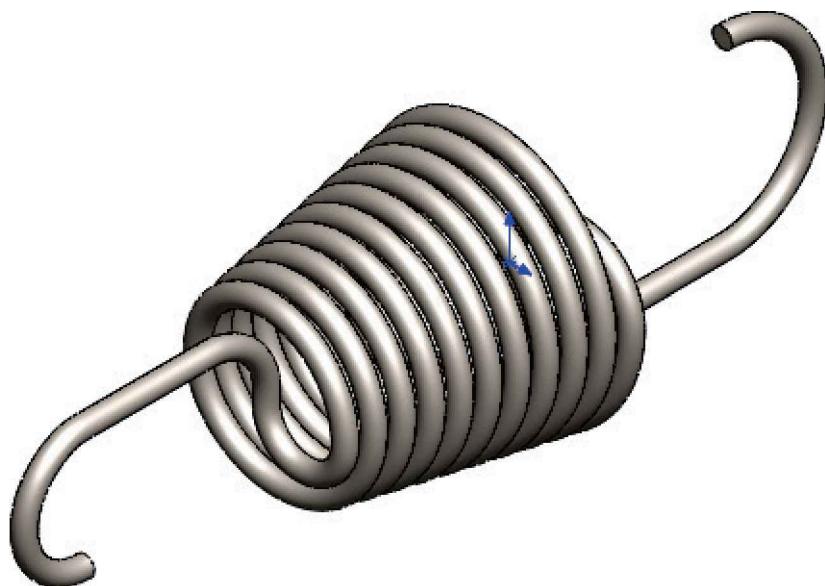
This lesson will guide you through the creation of a helical extension spring, where several 2D sketches and a 3D helix are combined into a single curve and then used as the sweep path.

A circular profile that represents the diameter of the wire is attached to one end and swept along the path to define the final shape of the spring.



Sweep with Composite Curves

Helical Extension Spring



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI

Units: INCHES – 3 Decimals

Tools Needed:



Insert Sketch



Line



Circle



Tangent Arc



3 Point Arc



Add Geometric Relations



Dimension



Composite Curve



Sweep

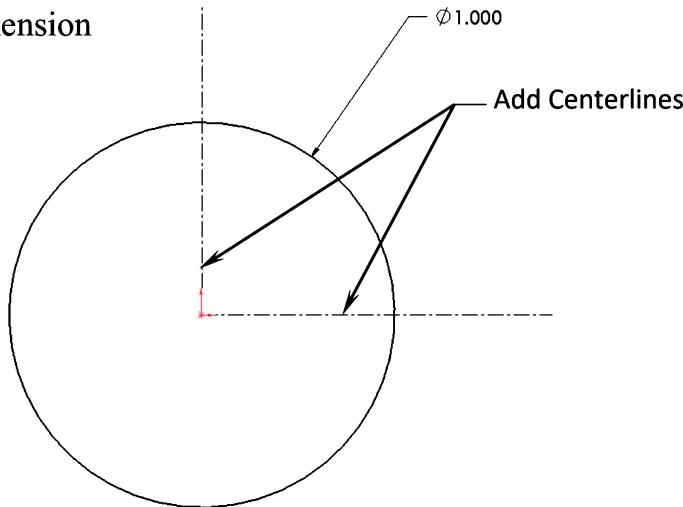
1. Sketching the first profile:

Select the Front plane from the FeatureManager Tree.

Click  or **Insert / Sketch**.

Sketch a **Circle** and **two centerlines**.

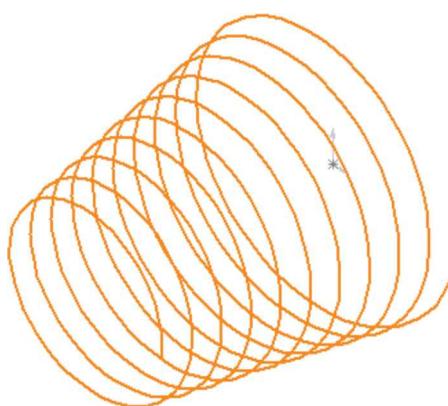
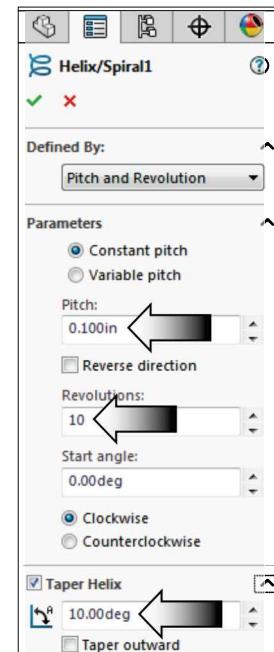
Add the diameter dimension to fully define the sketch.



2. Converting to a Helix:

Select **Insert / Curve / Helix / Spiral**.

Defined by: **Pitch and Revolution**
 Pitch: **.100**
 Revolution: **10**
 Starting angle: **0°**
 Taper helix: **Enabled**
 Taper angle: **10°**

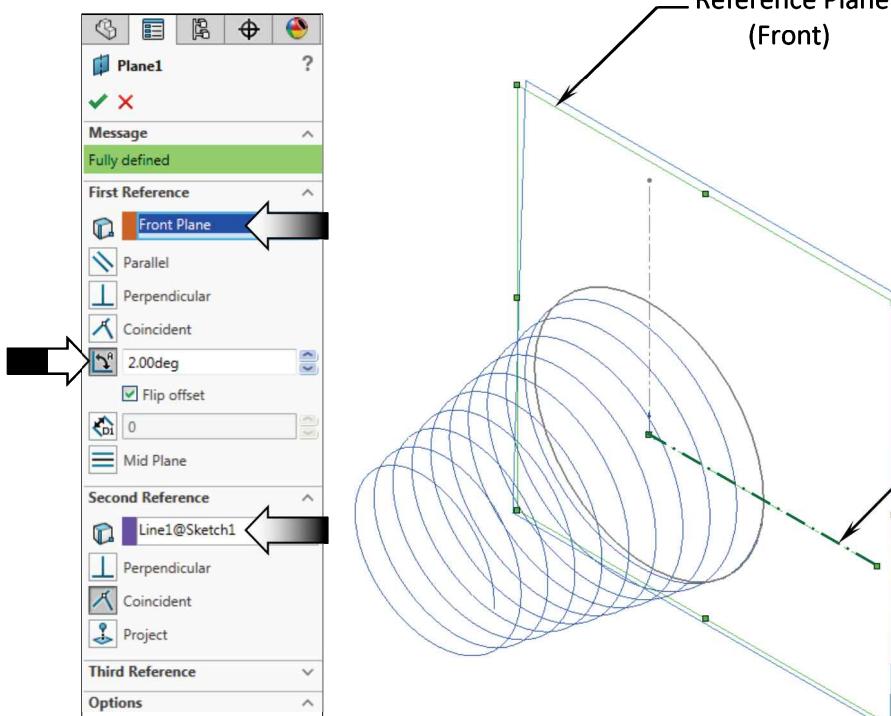


Click **OK**.

3. Creating a 2-degree plane:

Show the previous sketch (Sketch1).

Click or select **Insert/Reference Geometry/Plane**.



Show Sketch

Sketches can be made visible for use with other operations such as: Convert Entities, Sweep Path, Loft Profiles, new Plane creation, etc. From the FeatureManager Tree, click Sketch1 and select SHOW.

Select the **Horizontal Centerline** as the Pivot Line.

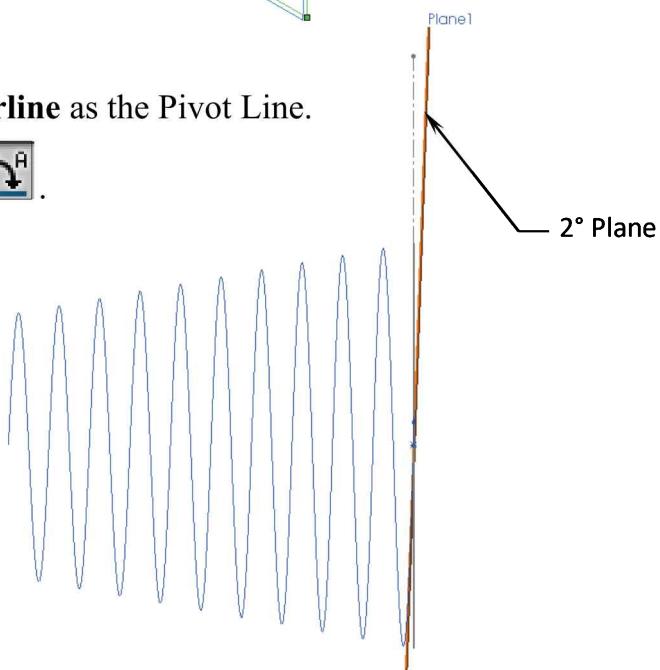
Select the **at Angle** option .

Enter **2.00deg**. for Angle.

Enable **Flip Offset**.
(The new plane should lean to the right. Change to the **Right** view Ctrl+4.)

Click **OK**.

Hide the Sketch1.

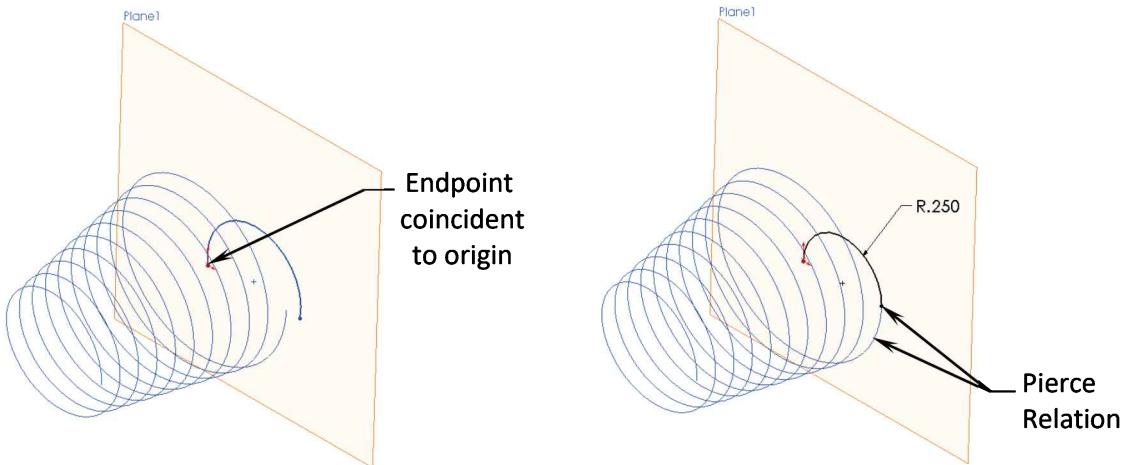


4. Sketching the large loop:

Select Plane1 from the FeatureManager Tree.

Click  or select **Insert / Sketch**.

Sketch a **3-point Arc** and add dimension shown:



Add a **Pierce relation** between the end point of the Arc and the Helix.

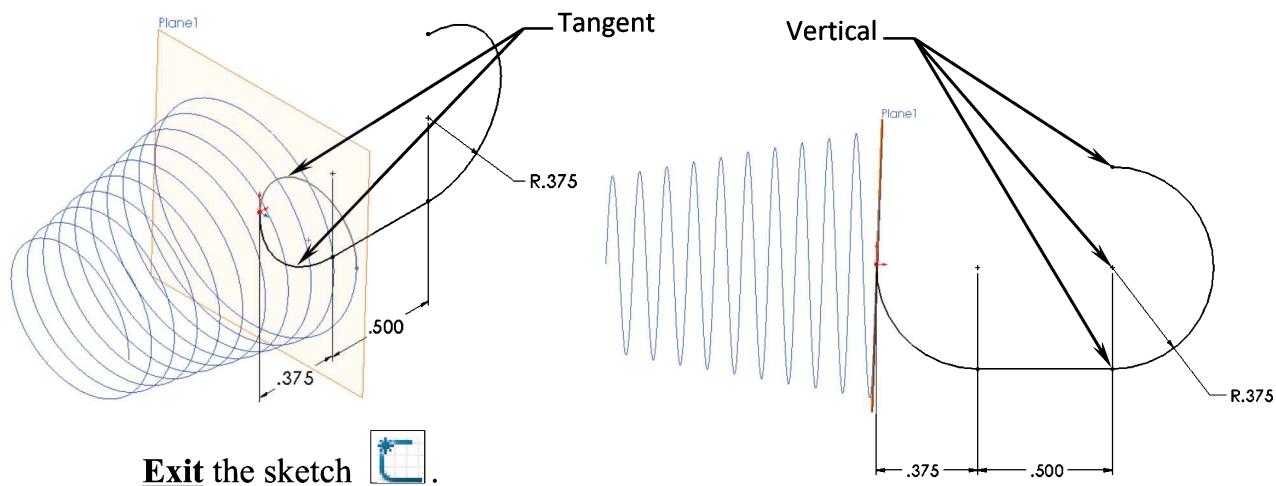
Exit the sketch  or select **Insert / Sketch**.

5. Sketching the large hook:

Select the Right plane from the FeatureManager Tree.

Click  or select **Insert / Sketch**.

Sketch the profile and add dimension and relations as shown below:

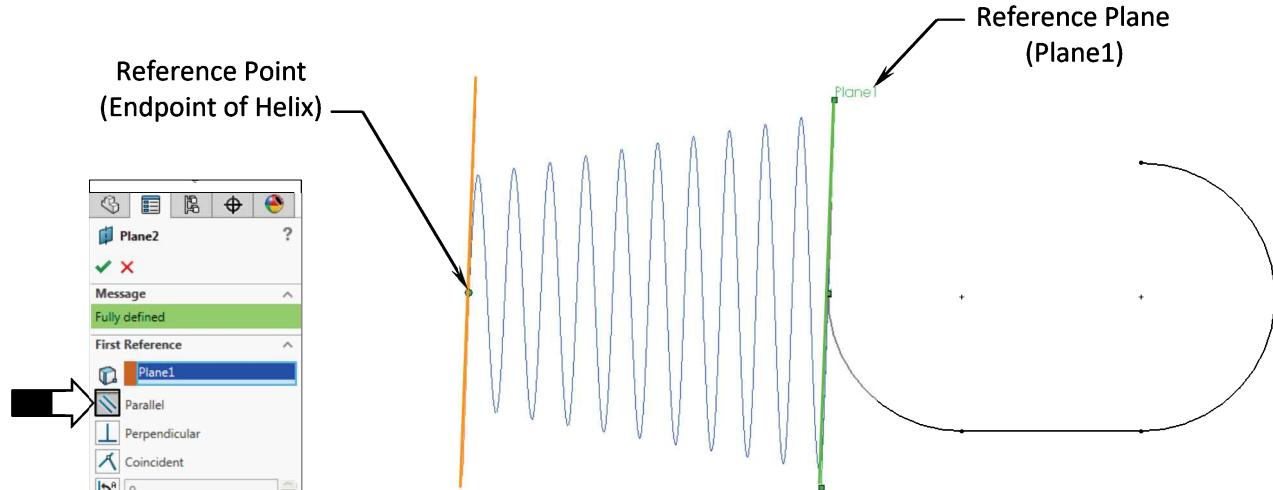


Exit the sketch .

6. Creating a Parallel plane:

Select Plane1 from the FeatureManager Tree.

Click  or Insert / Reference Geometry / Plane.



Select Plane1 as Reference Plane.

Click the left **Endpoint** of the helix as Reference Point.

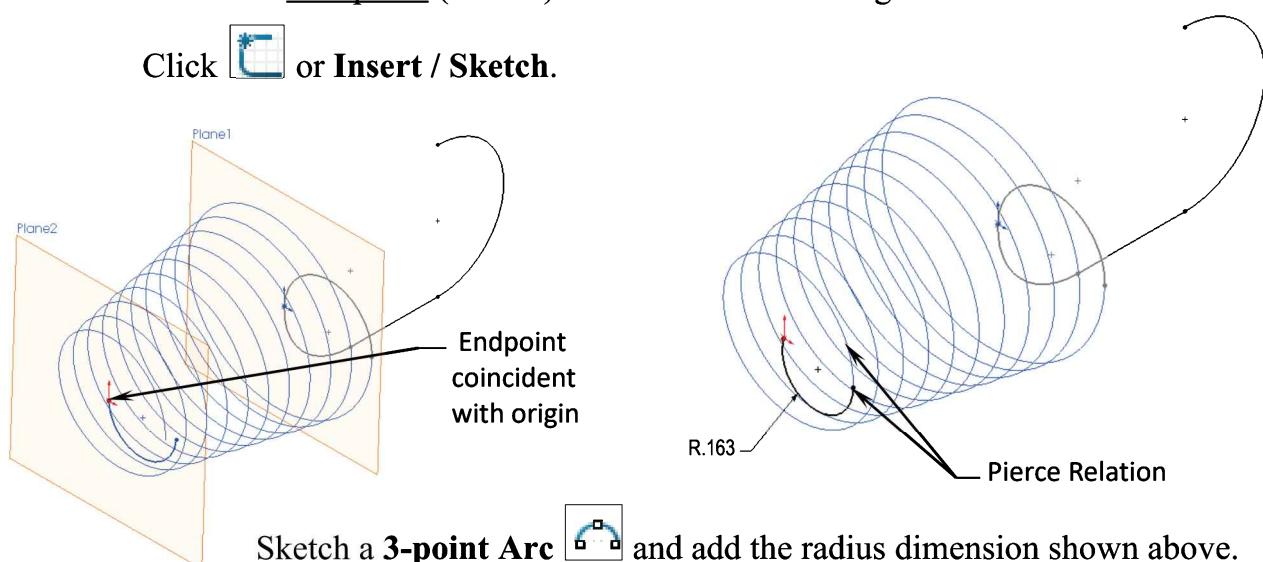
Select the **Parallel** relation .

Click **OK**.

7. Adding the small loop:

Select the new plane (Plane2) from the FeatureManager Tree.

Click  or Insert / Sketch.



Sketch a **3-point Arc**  and add the radius dimension shown above.

Add a **Pierce** relation between the endpoint of the Arc and the Helix.

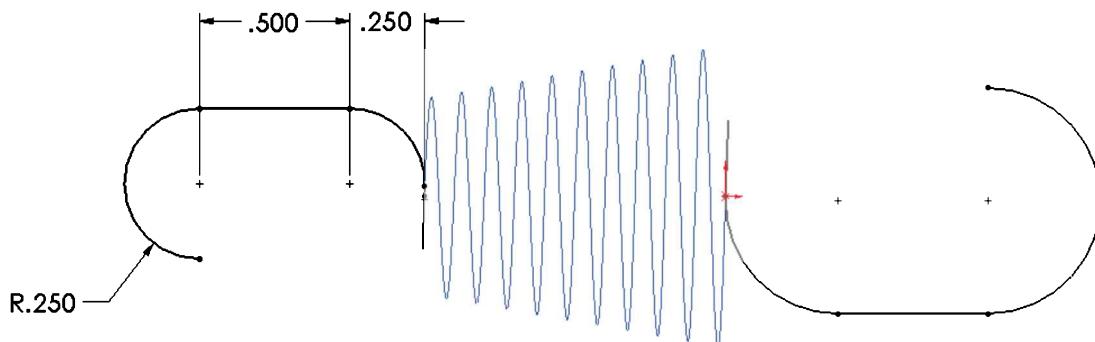
Exit the sketch  or **Insert / Sketch**.

8. Creating a small hook:

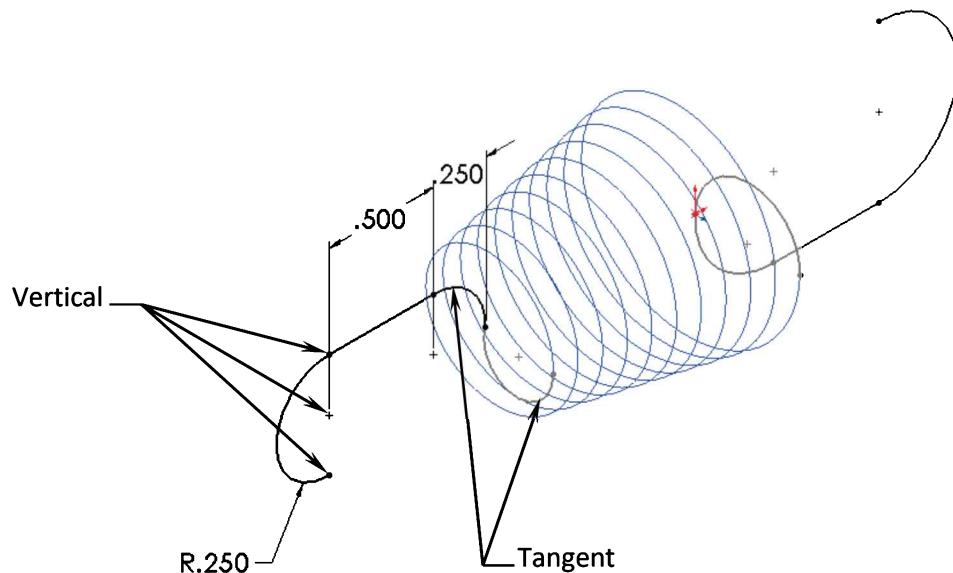
Select the Right plane from the FeatureManager Tree.

Click  or **Insert / Sketch**.

Sketch the profile and add the dimensions shown below.



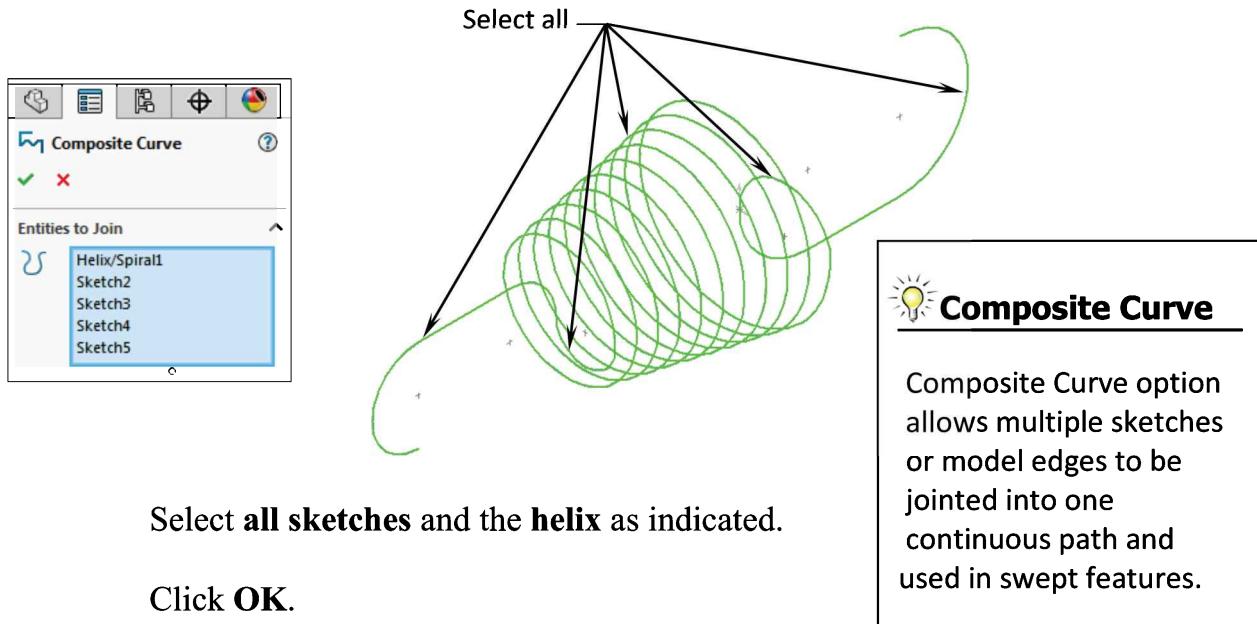
Add the **Vertical** and **Tangent** relations to the indicated entities.



Exit the sketch  or click **Insert / Sketch**.

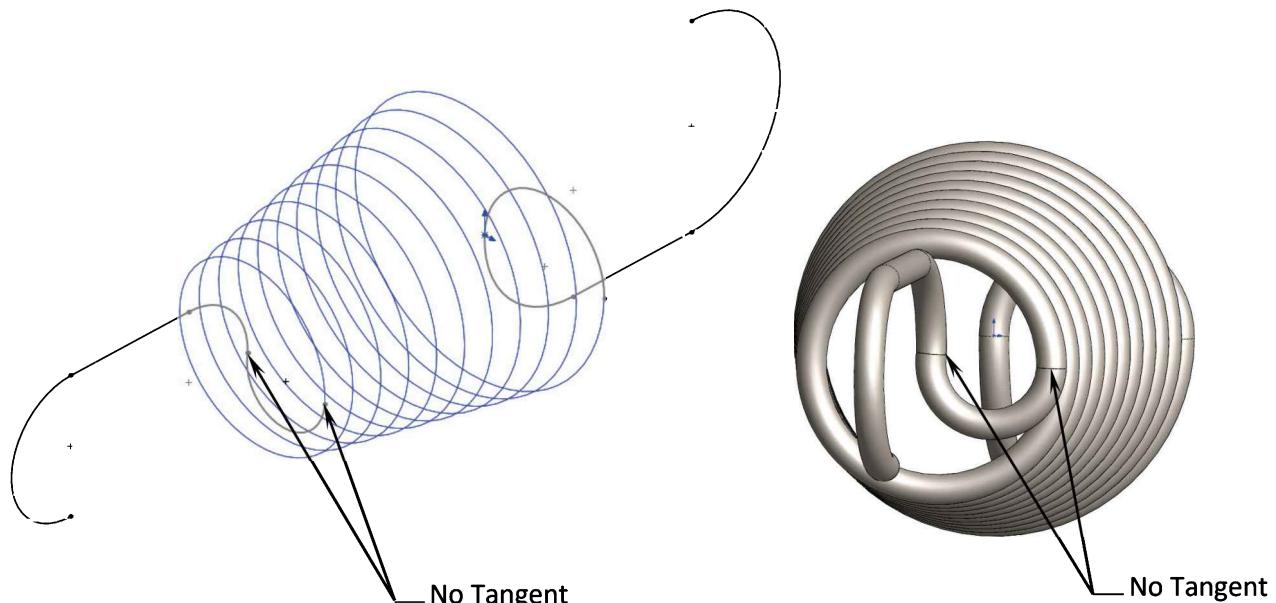
9. Creating a Composite Curve:

Click  under the Curves drop-down or select: Insert / Curve / Composite.



NOTE:

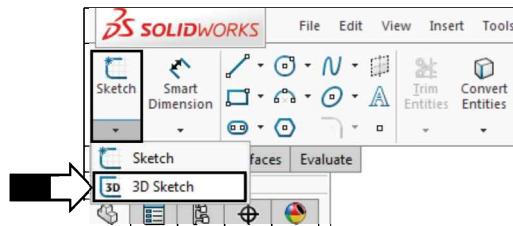
The transitions sketch between the helix and the sketch of the hook is not perfectly tangent. Since we cannot add a fillet between the helix (3D) and the hook (2D) we will have to try another approach called *Fit Spline*, to smooth out any tangency issues in the model.



10. Converting to 3D sketch:

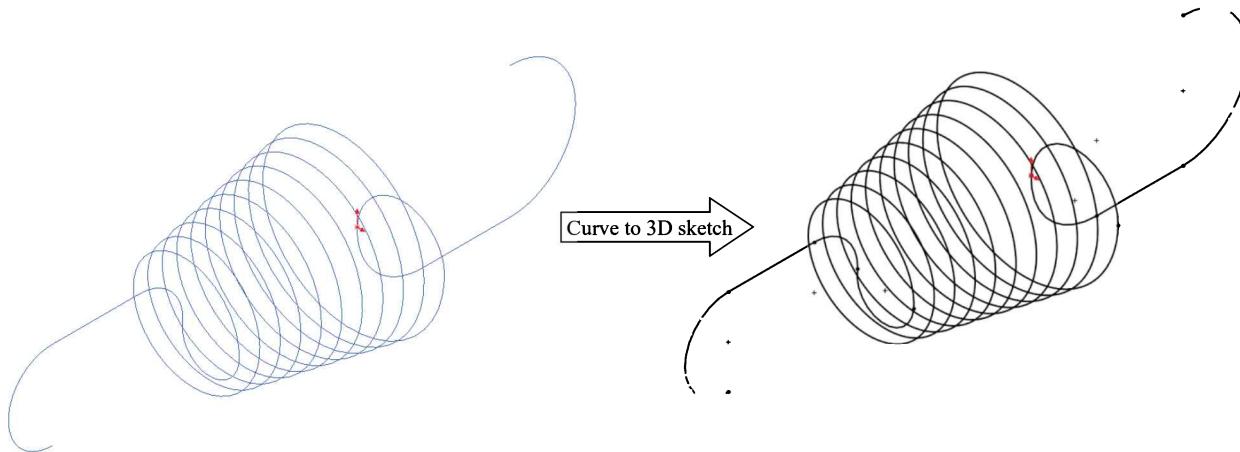
The **Fit Spline**  command can only be used with sketch entities, not 3D curves.

So the next step is to convert the composite curve to a 3D sketch.

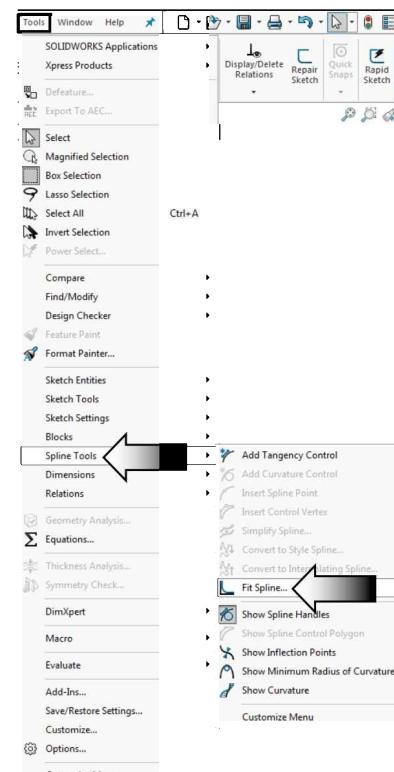
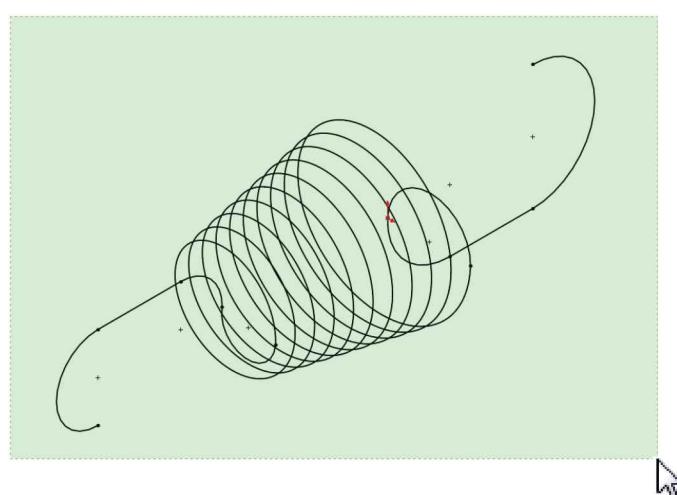


Click **3D Sketch** under the Sketch drop-down menu (arrow).

Select the Composite Curve and click **Convert Entities** .



The composite curve is converted to a 3D sketch.

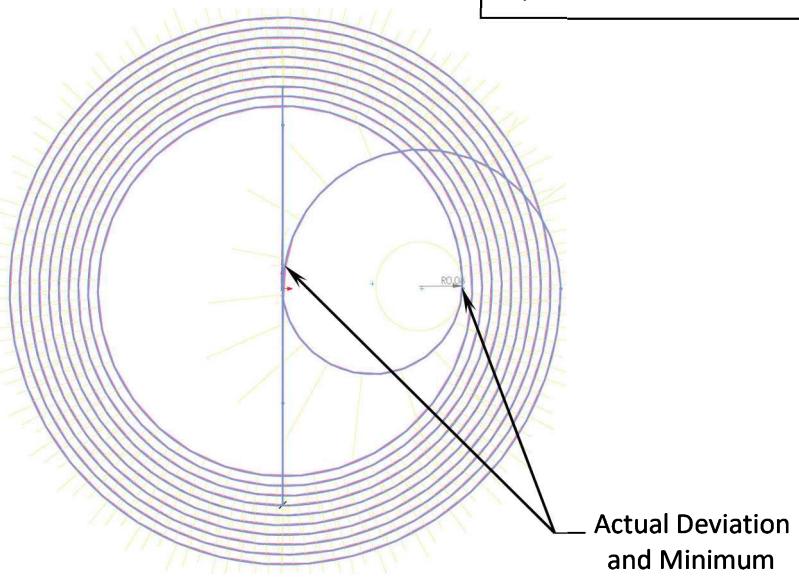
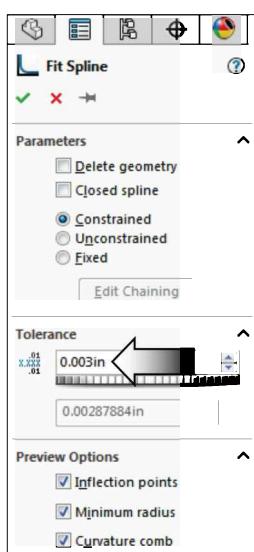


Box-select the entire 3D sketch (or press Control + A) and click:

Tools / Spline Tools / Fit Spline .

Select / enter the following:

- * **Constraint.**
- * **Tolerance: .003in.**
- * **Inflection Points.**
- * **Minimum Radius.**
- * **Curvature Comb.**



Fit Spline tool

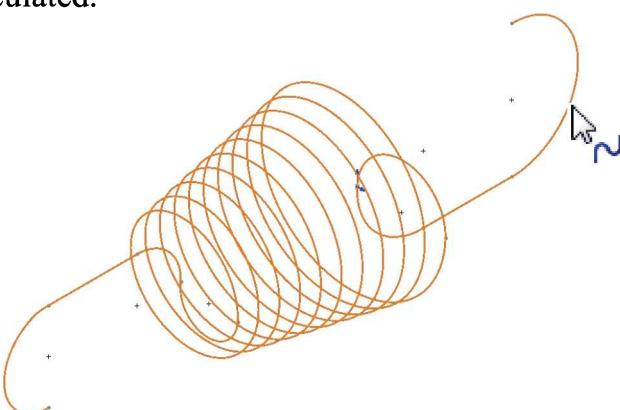
The Fit Spline tool to fit sketch segments to a spline. Fit splines are parametrically linked to underlying geometry so that changes to the geometry update the spline.

Tolerance: Specifies the maximum deviation allowed from the original sketch segments. Use the thumbwheel to adjust the tolerance so you can see changes to the geometry in the graphics area.

Actual Deviation: Updates based on the Tolerance value and the geometry selected. This is automatically calculated.

Click **OK**.

Hover the mouse cursor over one of the sketch segments. All entities have been fitted into a single spline.

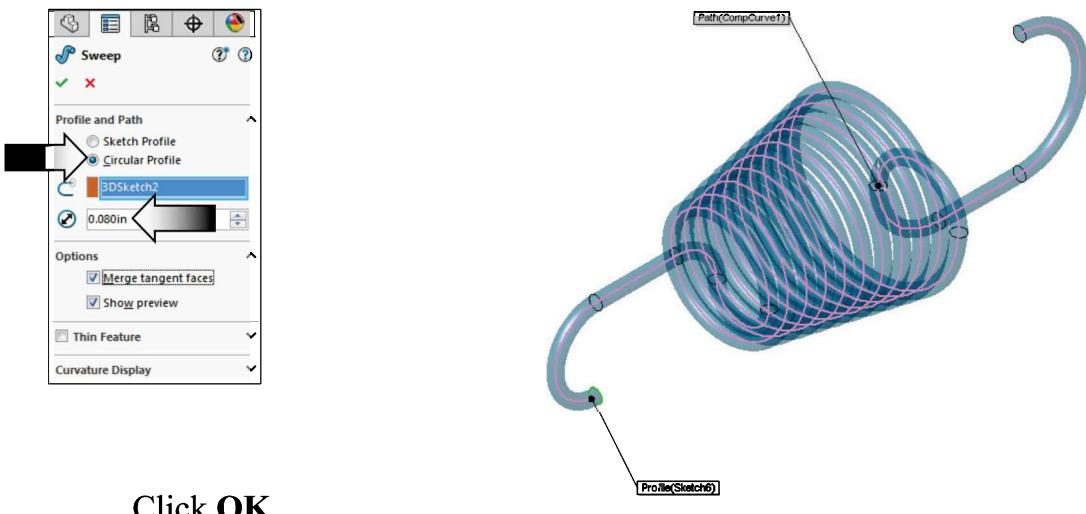


11. Sweeping the profile along the path:

Click  on the Features toolbar or select: **Insert / Boss-Base / Sweep**.

For Profile Diameter, click the **Circular Profile** button  and enter **.080in**.

For Sweep Path, select the **Composite Curve** in the graphics area.



Click **OK**.



Before Fit Spline

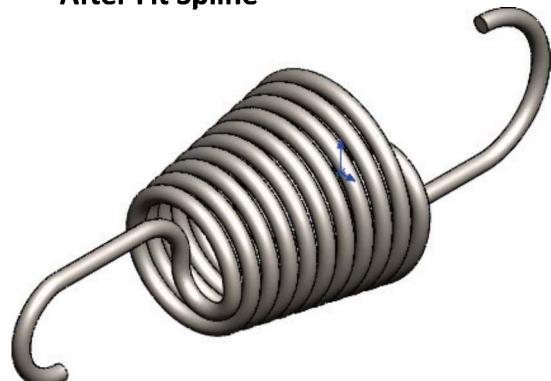


After Fit Spline

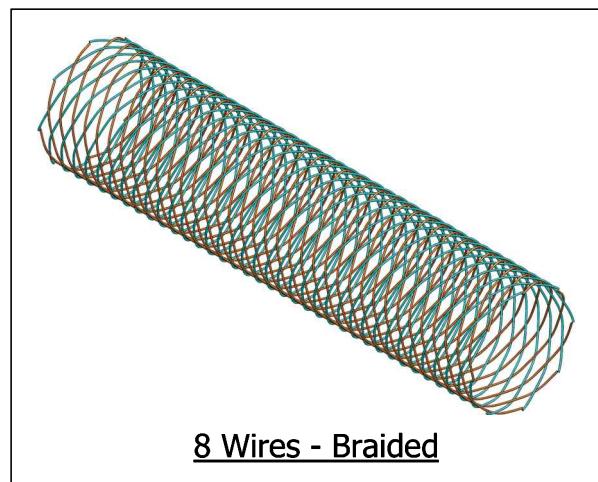
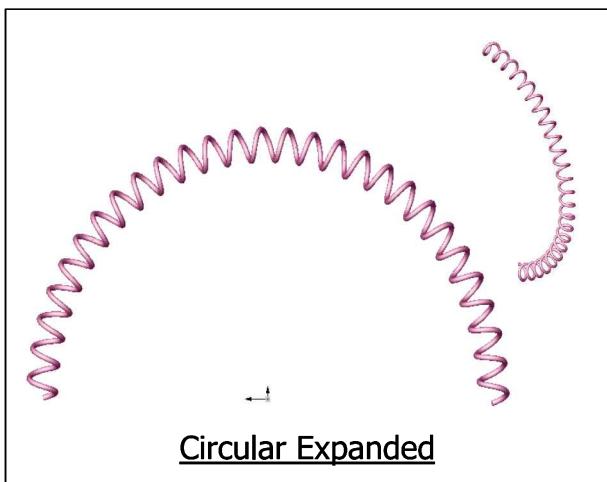
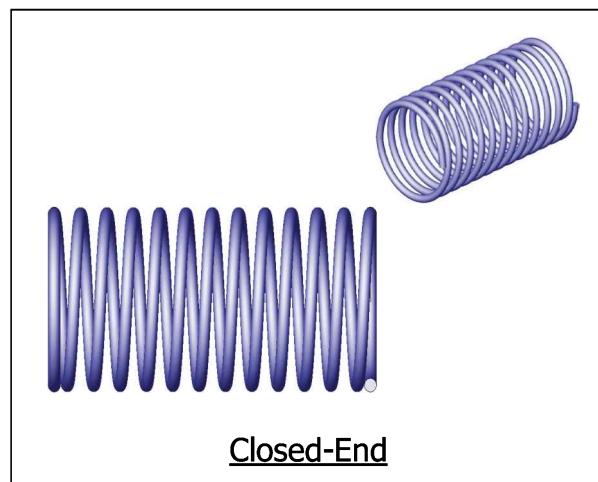
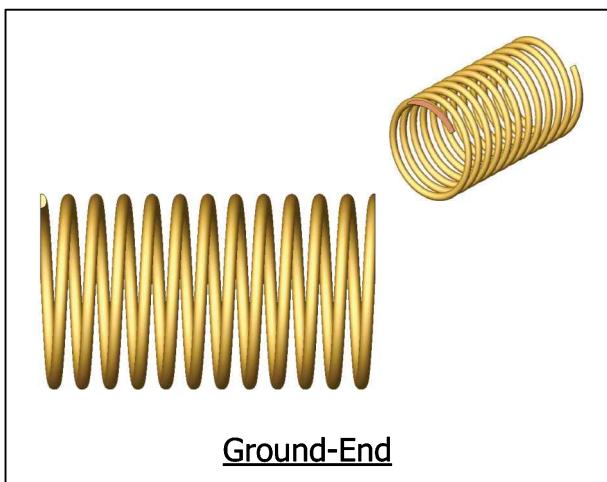
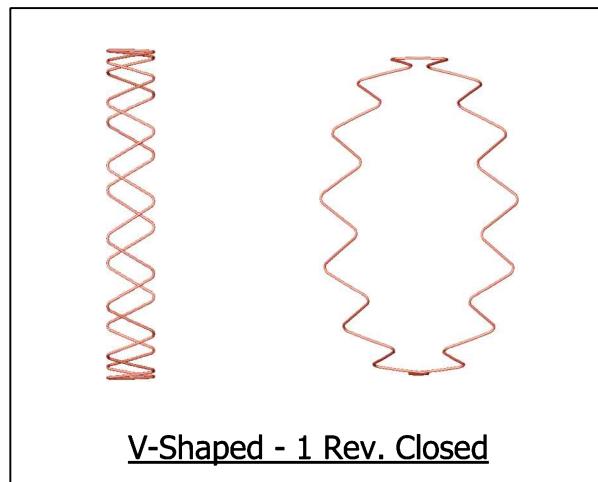
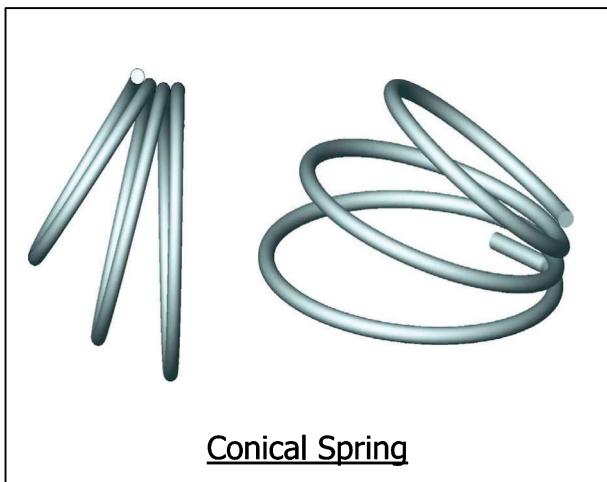
12. Saving your work:

Click **File / Save As /**

Enter **Helical Extension Spring** for the file name and click **Save**.



Other Examples:

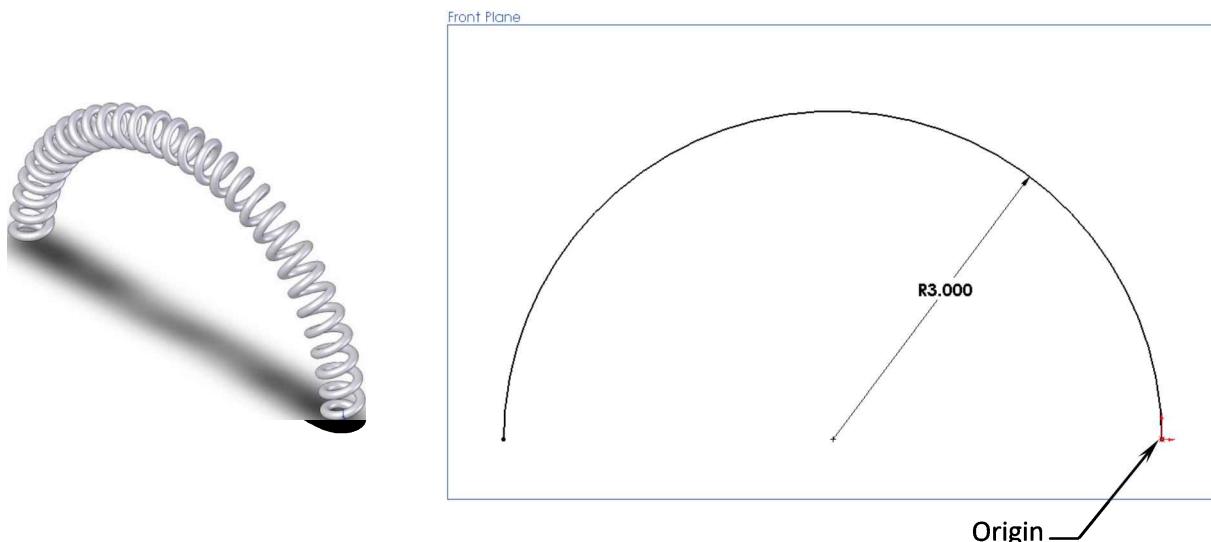


Questions for Review

1. Besides the Pitch and Revolution option, a helix can be defined with Pitch and Height.
 - a. True
 - b. False
2. It is sufficient to create an Offset Distance plane using a reference plane and a distance.
 - a. True
 - b. False
3. The sweep profile should have a Pierce relation with the sweep path.
 - a. True
 - b. False
4. Several sketches or model edges can be combined to make a Composite curve.
 - a. True
 - b. False
5. A Composite curve cannot be used as a sweep path.
 - a. True
 - b. False
6. The composite curve combines all sketches and model edges into one continuous curve, even if they are not connected.
 - a. True
 - b. False
7. In a sweep feature, SOLIDWORKS allows only one sweep path, but multiple guide curves can be used.
 - a. True
 - b. False
8. Several sketch profiles can be used to sweep along a path.
 - a. True
 - b. False

1. TRUE	2. TRUE	3. TRUE	4. TRUE	5. FALSE	6. FALSE	7. TRUE	8. FALSE
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Exercise: Circular Spring – 180deg.



1. Sketching the Sweep Path:

Select the Front plane and open a **new sketch**.

Sketch a **3-Point Arc** as shown and add a Horizontal relation between the left and the right endpoints, and then add a radius dimension.

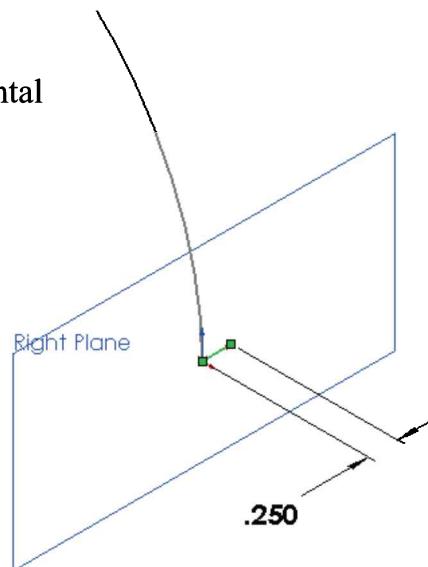
Exit the sketch.

2. Sketching the Sweep Profile:

Select the Right plane and open a **new sketch**.

Sketch a horizontal **Line** towards the right.

Add a **.250 in.** dimension.



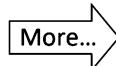
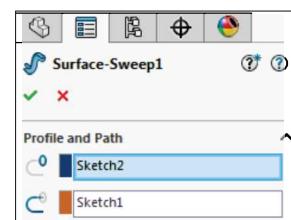
Exit the sketch.

3. Creating a Swept Surface:

Click  or select **Insert / Surface / Sweep**.

For Sweep Profile, select the horizontal **Line**.

For Sweep Path, select the **Arc**.



Expand the **Options** dialog box.

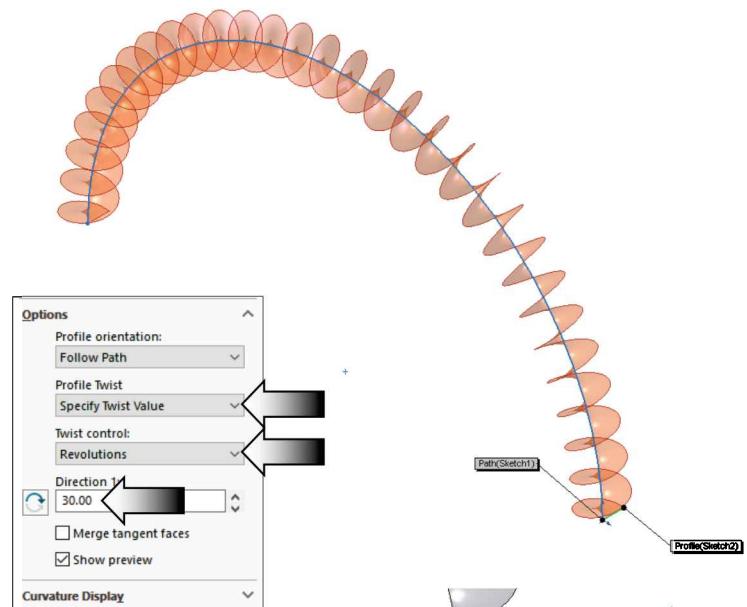
Under Profile Twist, select **Specify Twist Value**.

For Twist Control: Select **Revolutions**.

For Direction 1: Enter **30**.

Click **OK**.

The line is swept and twisted 30 revolutions along the path.



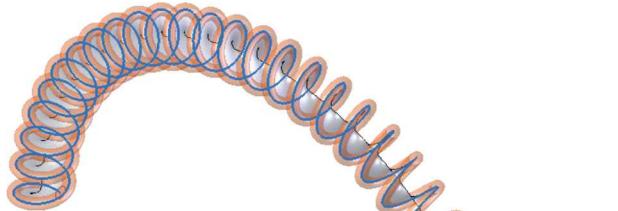
4. Sketching the Wire-Diameter:

Select the Right plane and open a new sketch.

Sketch a **Circle** at the right end of the swept surface.

Add a **Ø.125 in.** diameter dimension.

Exit the sketch.



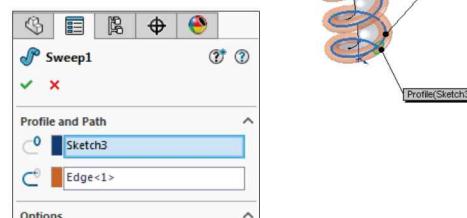
5. Creating a Swept Boss-Base: (Solid)

Click or select **Insert / Boss-Base / Sweep**.

For Sweep Profile, select the **Circle**.

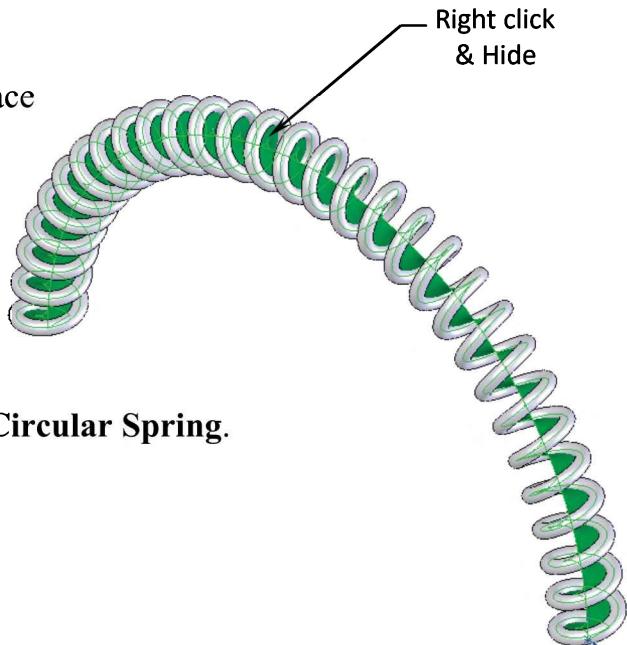
For Sweep Path, select the **Edge** of the Swept-Surface.

Click **OK**.



6. Hide the Swept-Surface:

Right-click over the Swept-Surface and select **Hide**.

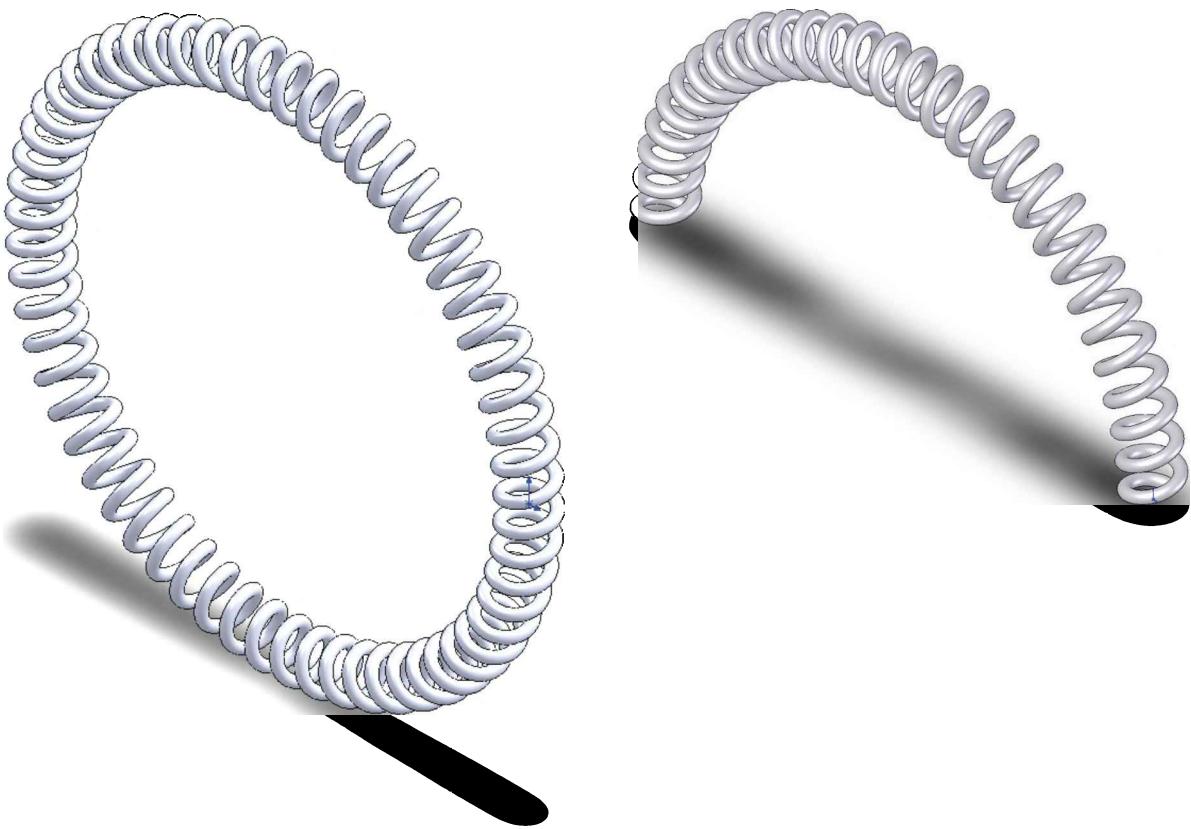


7. Save your work:

Select **File / Save As**.

For file name, enter **Expanded Circular Spring**.

Click **Save**.

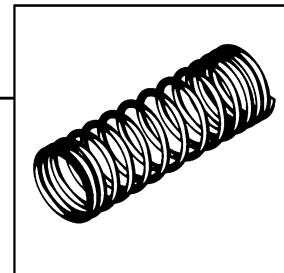


Using Variable Pitch

Multi-Pitch Spring with Closed Ends

Sweep

with Variable Pitch Helix



In a Sweep feature, there is only one Sweep Profile, one Sweep Path, and one or more Guide Curves.

The Sweep Profile describes the feature's cross-section; the Sweep Path helps control how the Sweep Profile moves along the path.

The Guide Curve(s) keeps the profile from twisting while it moves along the sweep path.

The Sweep path can either be a 2D or a 3D sketch, the edges of the part, or a Composite Curve.

Beside the Pitch and Revolution option, the Helix / Spiral command offers other options to create more advanced curves such as:

- * Height and Revolutions.
- * Height and Pitch.
- * Spiral.
- * Constant Pitch.
- * Variable Pitch.

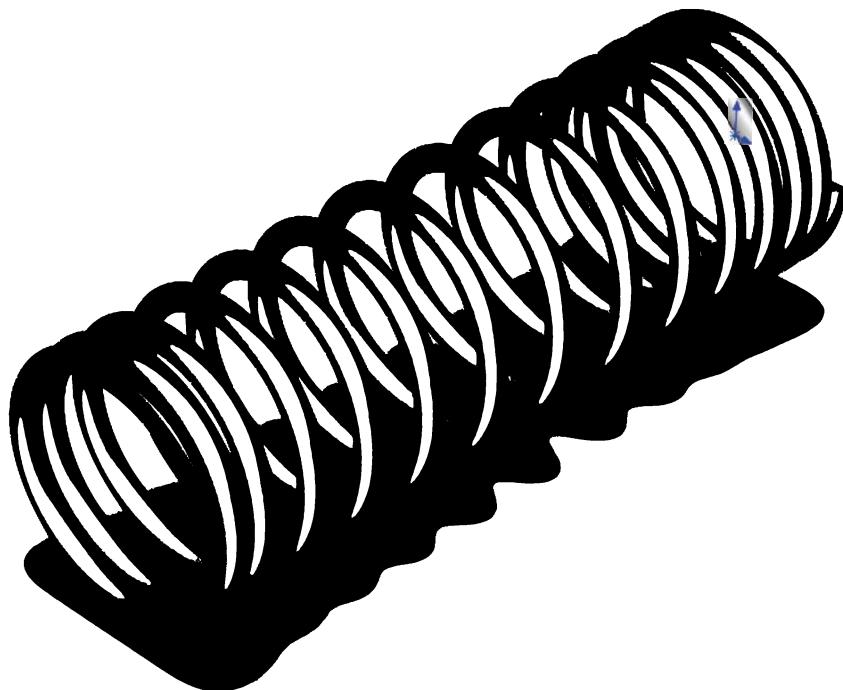
Region parameters:				
	P	Rev	H	Dia
1	0.115in	0	0in	1in
2	0.115in	1.5	0.1725	1in
3	0.375in	6.5	1.3975	1in
4	0.25in	11.5	2.96in	1in
5	0.115in	12.5	3.1425	1in
6	0.115in	14	3.315i	1in
7				

We will take a look at the Variable Pitch option in this lesson and learn how a helix with variable pitch is created using a table to help control the changes of the dimensions.

To create the flat ground ends, an extruded cut feature is added to the two ends of the spring.

Using Variable Pitch

Multi-Pitch Spring with Closed Ends



Dimensioning Standards: ANSI

Units: INCHES – 3 Decimals

Tools Needed:



Insert Sketch



Helix / Spiral



Add Geometric
Relations



Dimension



Base/Boss
Sweep



Extruded Cut

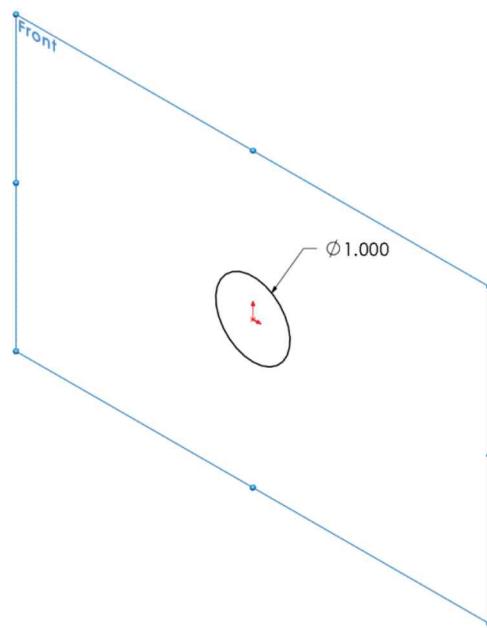
1. Creating the base sketch:

Select the **Front** plane from the FeatureManager tree.

Click  or select: **Insert / Sketch**.

Sketch a **Circle** centered on the origin.

Add a diameter dimension of **1.000"**.



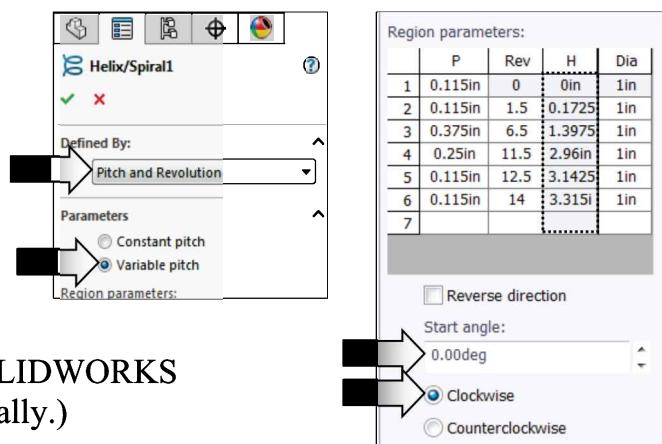
2. Creating a helix with Variable Pitch:

Click  or select **Insert / Curve / Helix-Spiral**.

Under Define By, select:
Pitch and Revolution.

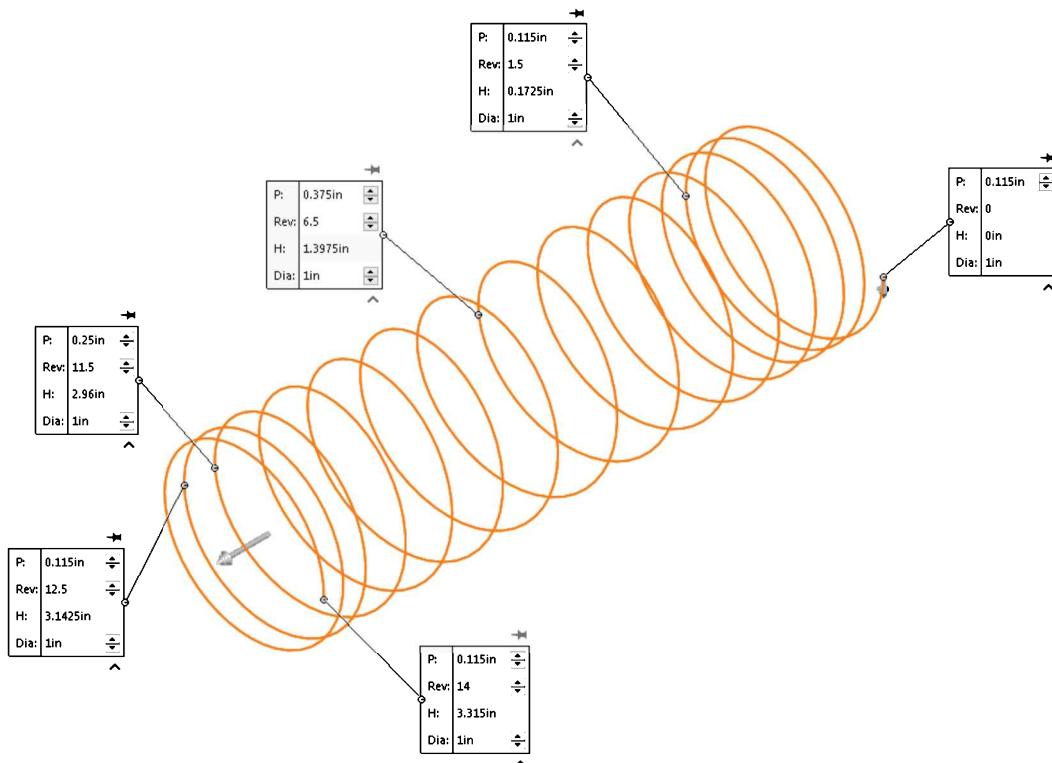
Under Parameter, select:
Variable Pitch.

For Region Parameters,
enter the values of the **Pitches**,
Revolutions, and **Diameters**.
(Ignore the Height column; SOLIDWORKS
will fill in the values automatically.)



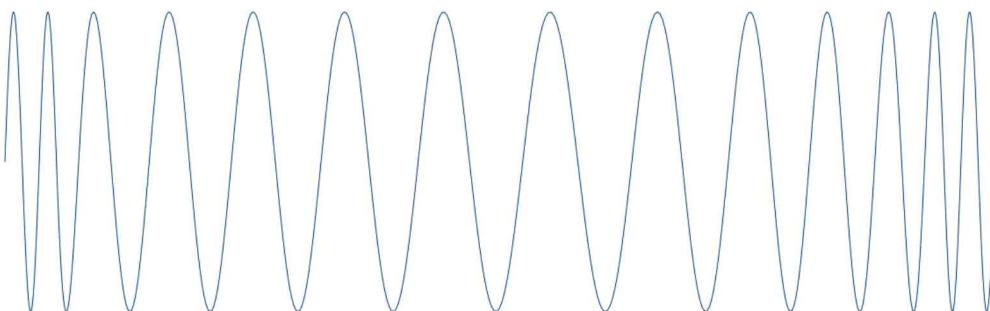
Set Angle to **0deg** and **Clockwise** direction.

Your Variable Pitch helix should look like the image below.



Click **OK**.

Press **Control + 4** to view the Variable-Pitch Helix from the right side.



3. Sweeping the profile along the path:

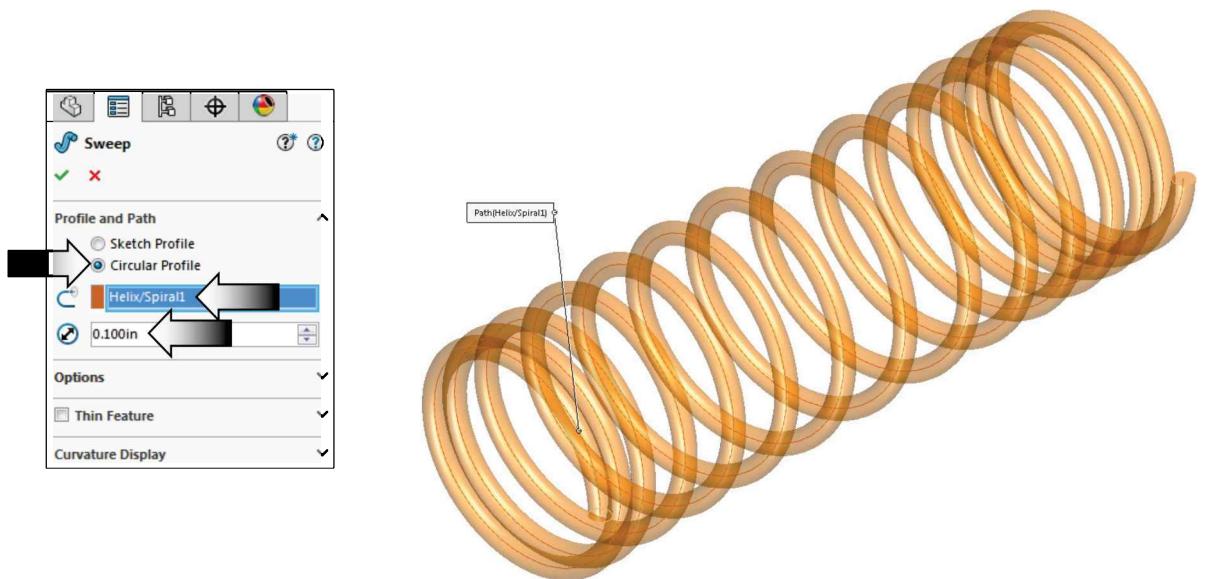
Click  or select: **Insert / Boss-Base / Sweep**.

Select the **Circular Profile** option.

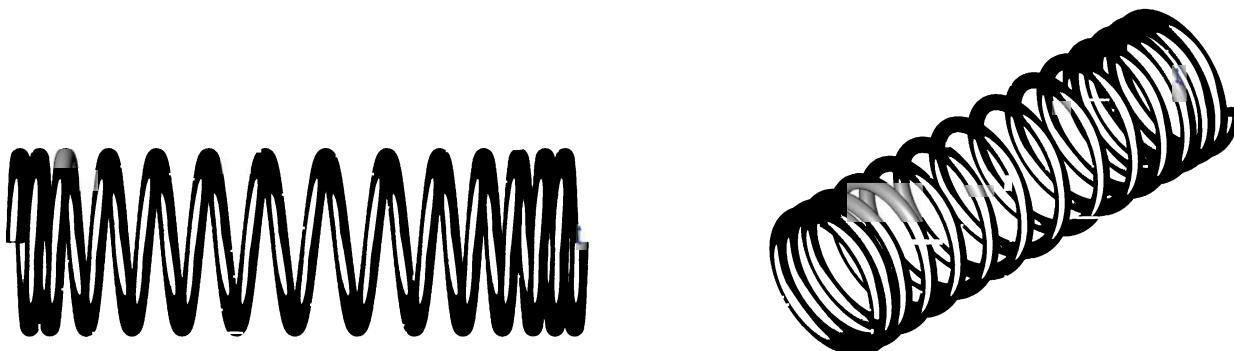
For profile diameter, enter **.100in**.

For sweep path, select the **Helix** either from the feature tree or from the graphics area.

Click **OK**.



Change to different orientations to inspect the Swept feature.

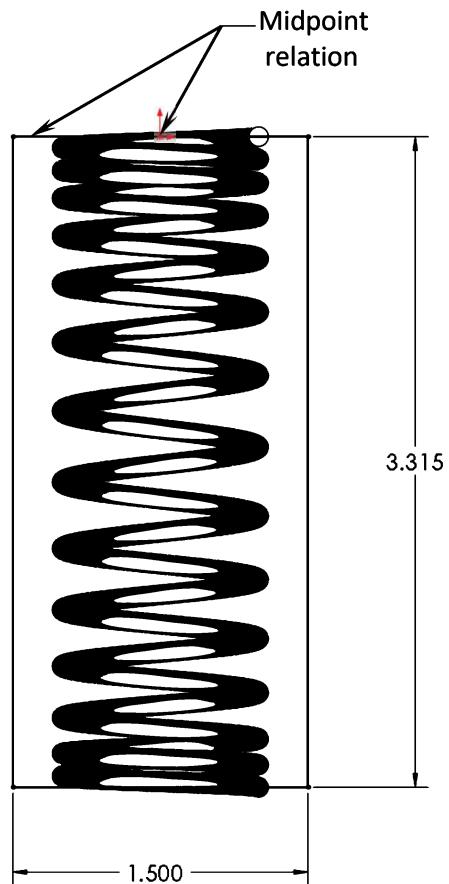


4. Creating a trimmed sketch:

Select the Top plane from the Feature-Manager tree and open a new sketch.

Sketch a **Rectangle** and add a **Midpoint** relation between the line on top and the origin.

Add the width and height dimensions. The sketch should now be fully defined.



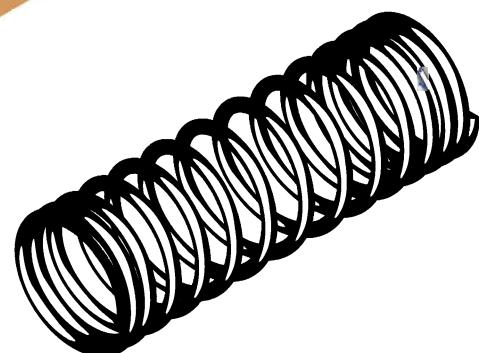
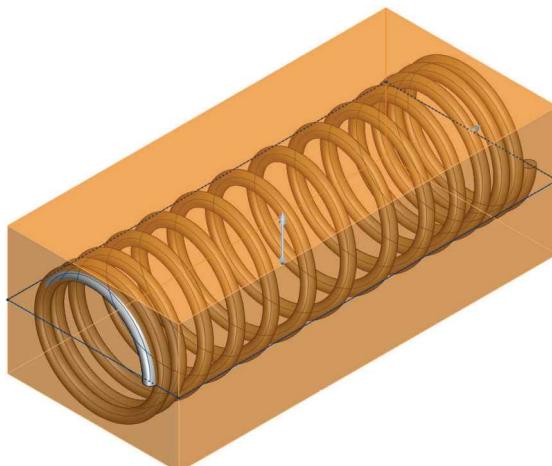
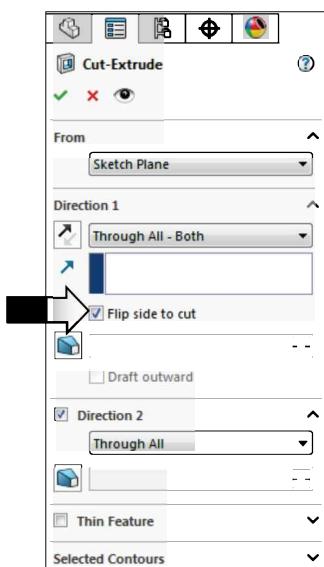
5. Extruding a cut:

Click or select: **Insert / Cut / Extrude**.

Set the Direction 1 to **Through All Both**.

Enable the **Flip Side to Cut** checkbox.

Click **OK**.



6. Saving your work:

Click **File / Save As**.

Enter **Variable Pitch Spring** and press **Save**.

Questions for Review

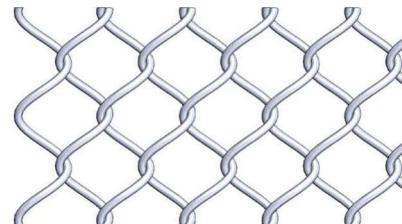
1. Multiple Sweep Profiles can be used in a sweep feature.
 - a. True
 - b. False
2. Multiple Sweep Paths can be used in a sweep feature.
 - a. True
 - b. False
3. Only one Sweep Profile and one Sweep Path can be used in a sweep.
 - a. True
 - b. False
4. A Helix can be defined by:
 - a. Pitch and Revolution
 - b. Height and Revolution
 - c. Height and Pitch
 - d. Spiral
 - e. All of the above
5. Several connected Helices can be combined into one single Composite Curve.
 - a. True
 - b. False
6. The Sweep Profile sketch should be related to the Sweep Path using the relation:
 - a. Perpendicular
 - b. Parallel
 - c. Coincident
 - d. Pierce
7. The Sweep Path controls the twisting and how the Sweep Profile moves along.
 - a. True
 - b. False
8. The Edges of the part can also be used as the Sweep Path.
 - a. True
 - b. False

1. FALSE	2. FALSE	3. TRUE	4. E	5. TRUE	6. D	7. TRUE	8. TRUE
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Exercise: Using Equation Driven Curve

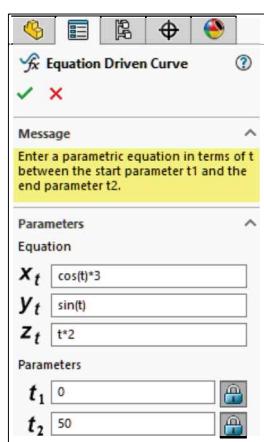
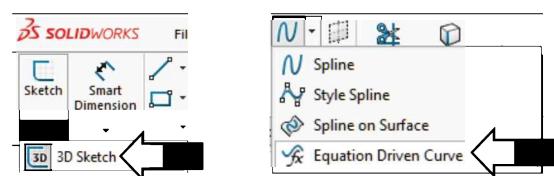
The Equation Driven Curve tool in SOLIDWORKS allows us to create a curve by defining the equation for the curve. The values in the equation driven curves must be in radians.

This exercise will teach us how to create an equation that can be used to model the chain link fence.



1. Creating an Equation Driven Curve:

Start with a new **Part Template**, set the units to **Millimeter** and open a new 3D Sketch.



Select **Equation Driven Curve** under the Spline drop-down list.

Enter the equations and the parameters shown in the dialog box.

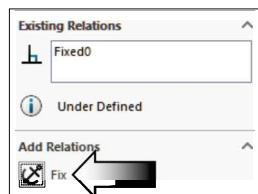
Click **OK**.

Equation
 $X_t: \cos(t)*3$
 $Y_t: \sin(t)$
 $Z_t: t^2$
Parameters
 $t1: 0$
 $t2: 50$

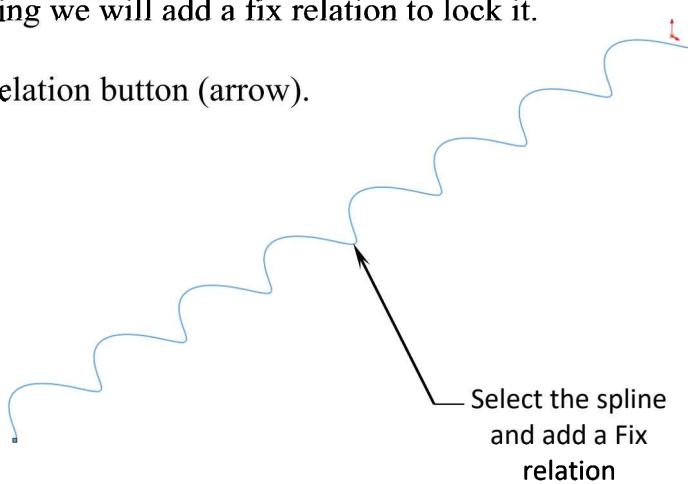
2. Adding a Fix relation:

To prevent the 3D Spline from shifting we will add a fix relation to lock it.

Click the Spline and select the **Fix** relation button (arrow).



Exit the 3D Sketch.

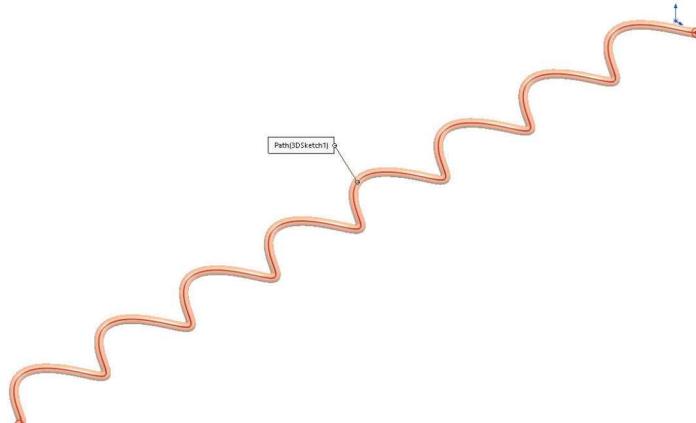
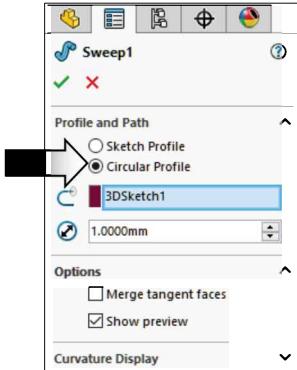


3. Making a swept feature:

Switch to the **Features** tab.



Click **Swept Boss/Base**.

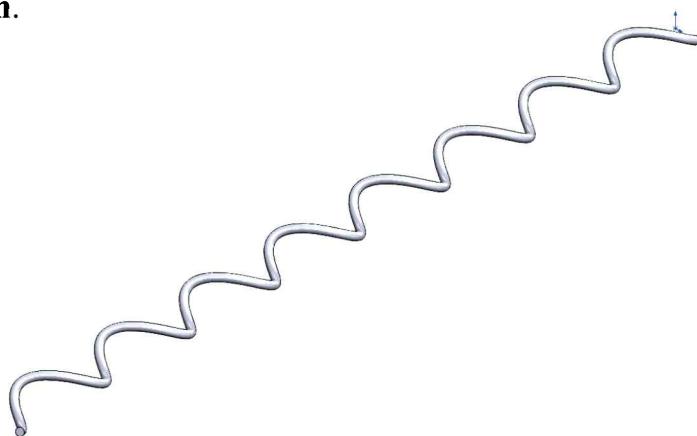


For Profile and Path, select the **Circular Profile** option.

For Path, select the **3D Spline** from the graphics area.

For Profile Diameter, enter **1.00mm**.

Click **OK**.

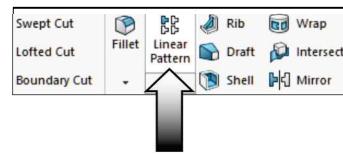


A 1mm circular profile is created and swept along the 3D Spline.

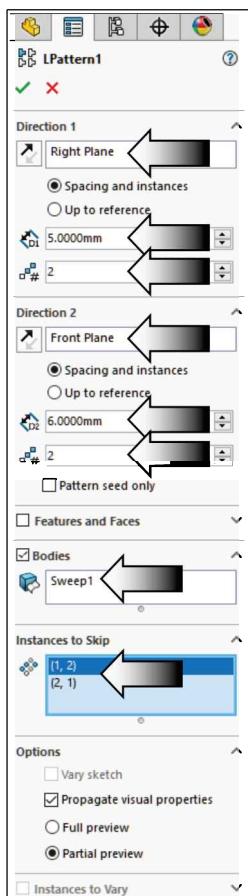
The Swept feature will be patterned a couple times in the next step to create the chain link fence model.

4. Creating the 1st Linear Pattern:

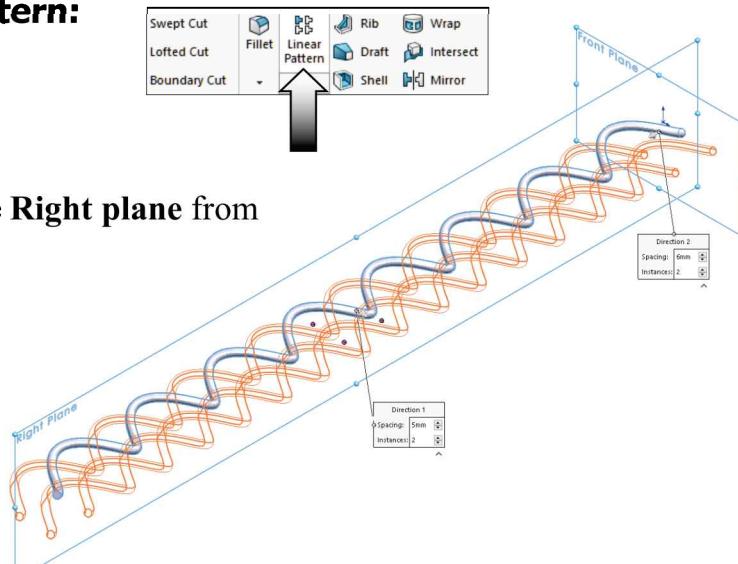
Click **Linear Pattern**.



For Direction 1, select the **Right plane** from the FeatureManager tree.

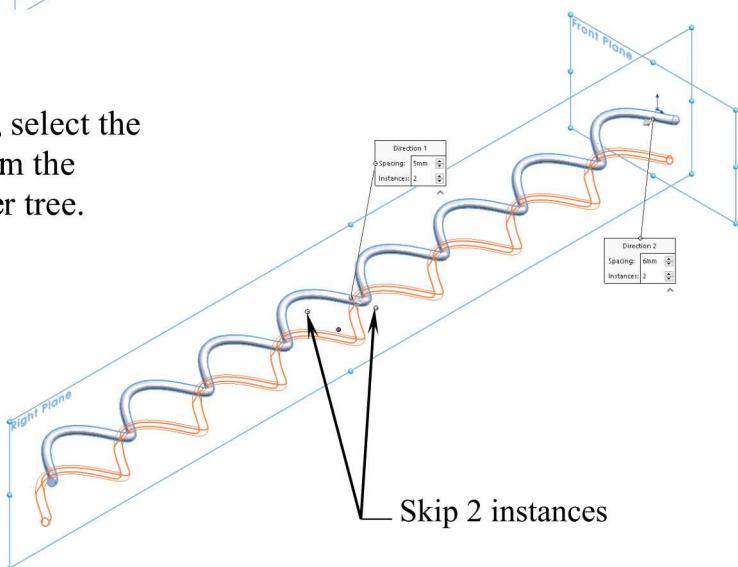


Enter **5.00mm** for Spacing.



Enter **2** for Instances.

For Direction 2, select the **Front plane** from the Feature Manager tree.



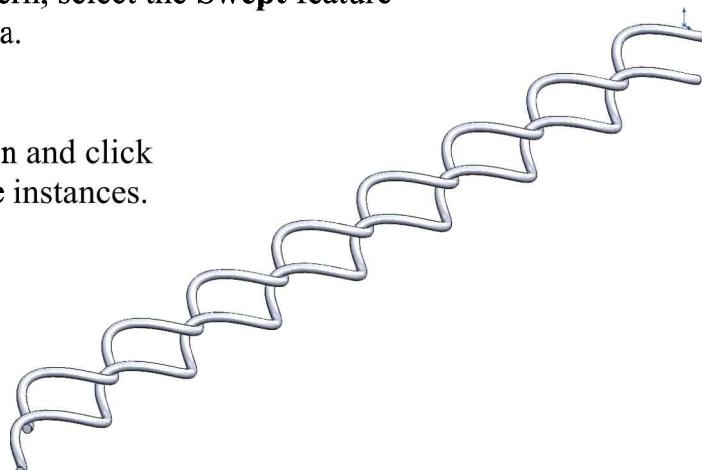
Enter **6.00mm** for Spacing

Enter **2** for Instances.

For Bodies to Pattern, select the **Swept** feature in the graphics area.

Expand the Instances to Skip section and click the **2 dots** as indicated to skip those instances.

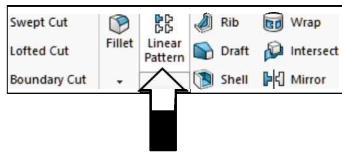
Click **OK**.



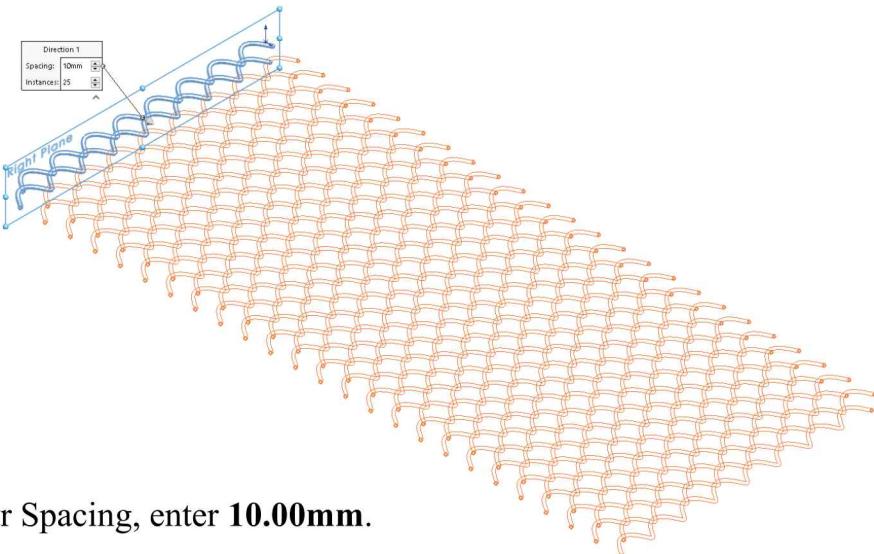
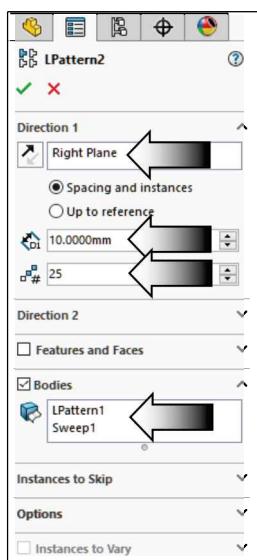
Compare your model against the one shown on the right.

5. Creating the 2nd Linear Pattern:

Click **Linear Pattern** once again.



For Direction 1, select the **Right plane** from the FeatureManager tree.

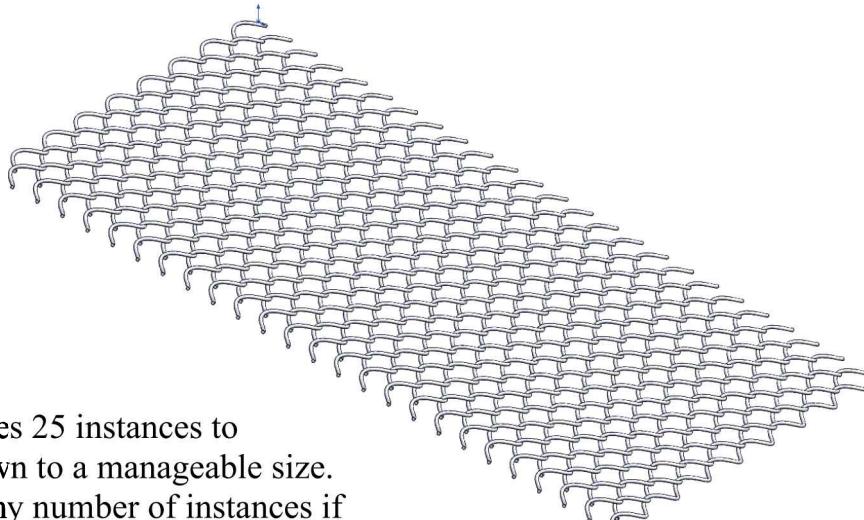


For Spacing, enter **10.00mm**.

For Number of Instances, enter **25**.

For Bodies to Pattern, select the **Sweep1** and the **Pattern1** bodies either from the graphics area or from the FeatureManager tree.

Click **OK**.



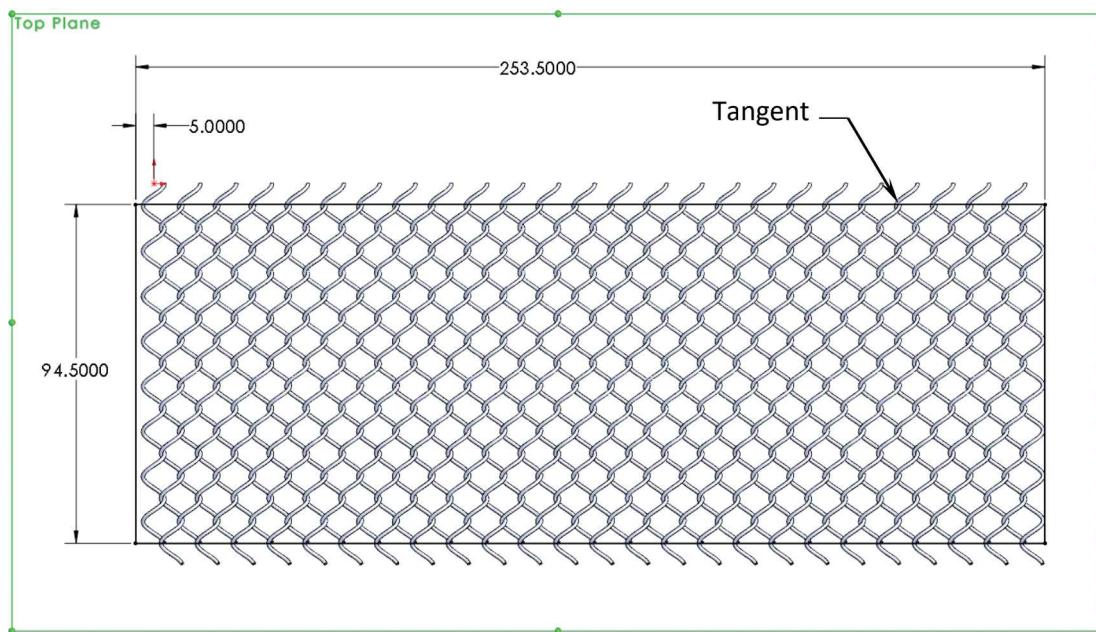
This exercise uses 25 instances to keep the file down to a manageable size. You can enter any number of instances if desired.

6. Trimming the ends:

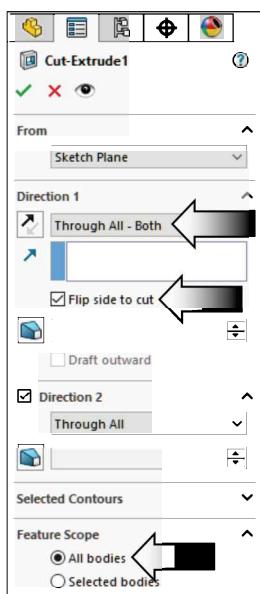
Open a new sketch on the Top plane.

Sketch a **Corner Rectangle** and add the dimensions shown below.

Add the dimensions and relation to fully define the sketch.



Switch to the **Features** tab and click **Extruded Cut**.

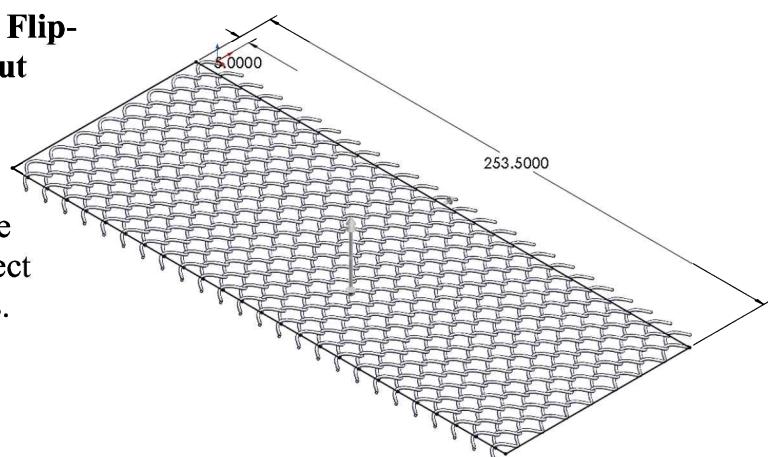


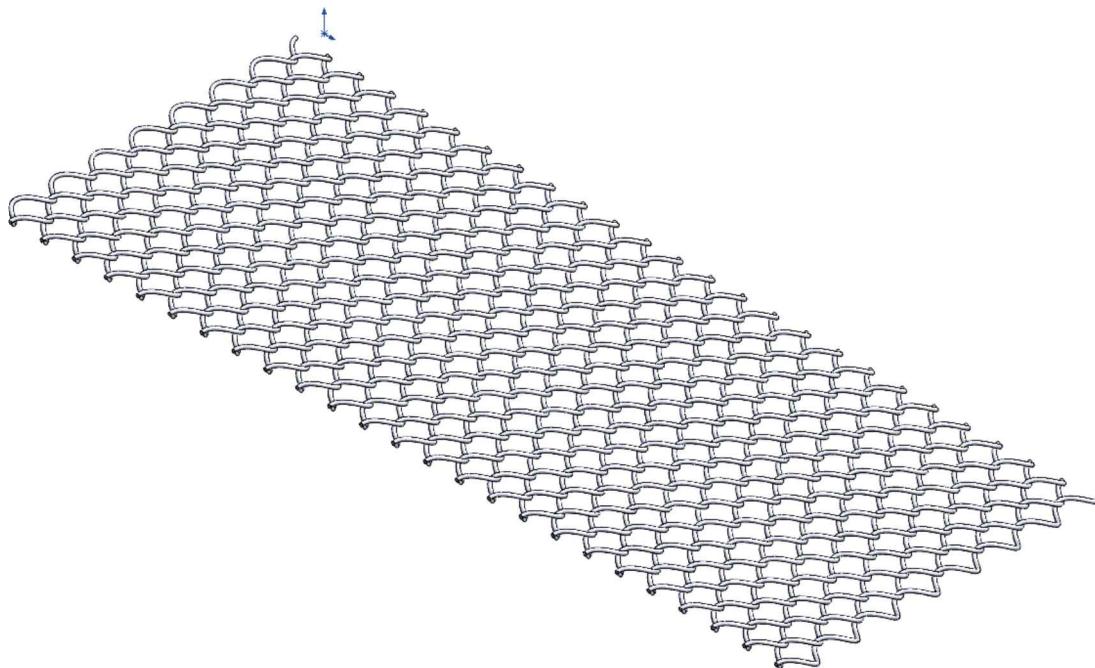
For Direction 1, select **Through All – Both**.

Enable the **Flip Side To Cut** checkbox.

For Feature Scope, select **All Bodies**.

Click **OK**.



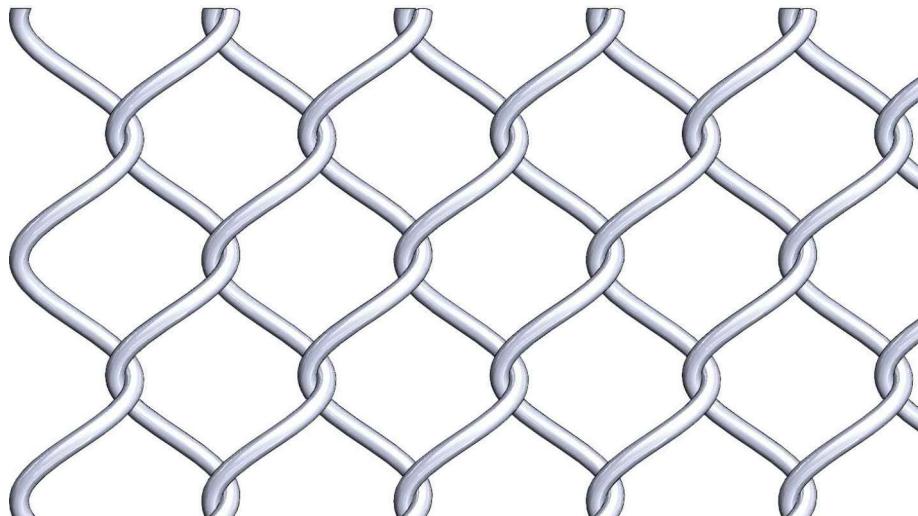


7. Saving your work:

Select File, Save As.

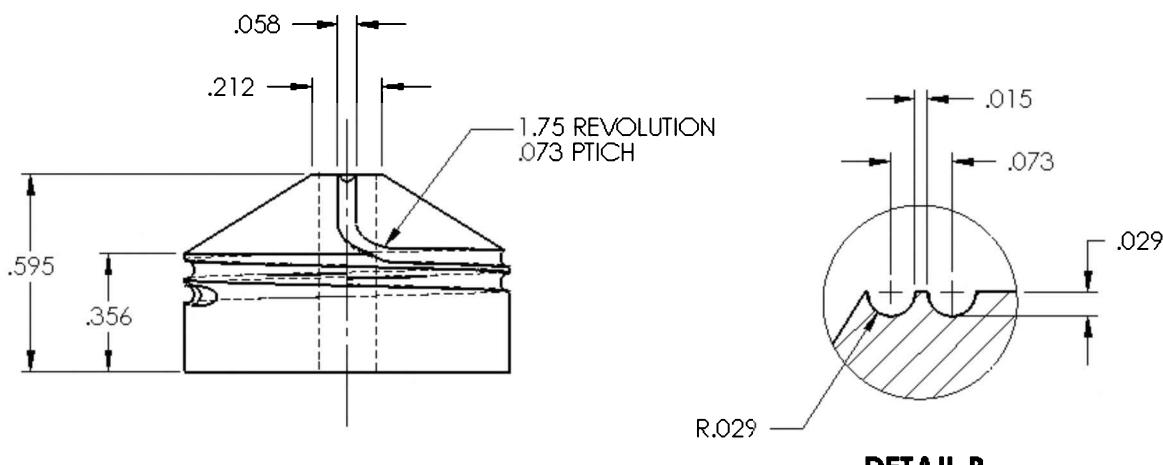
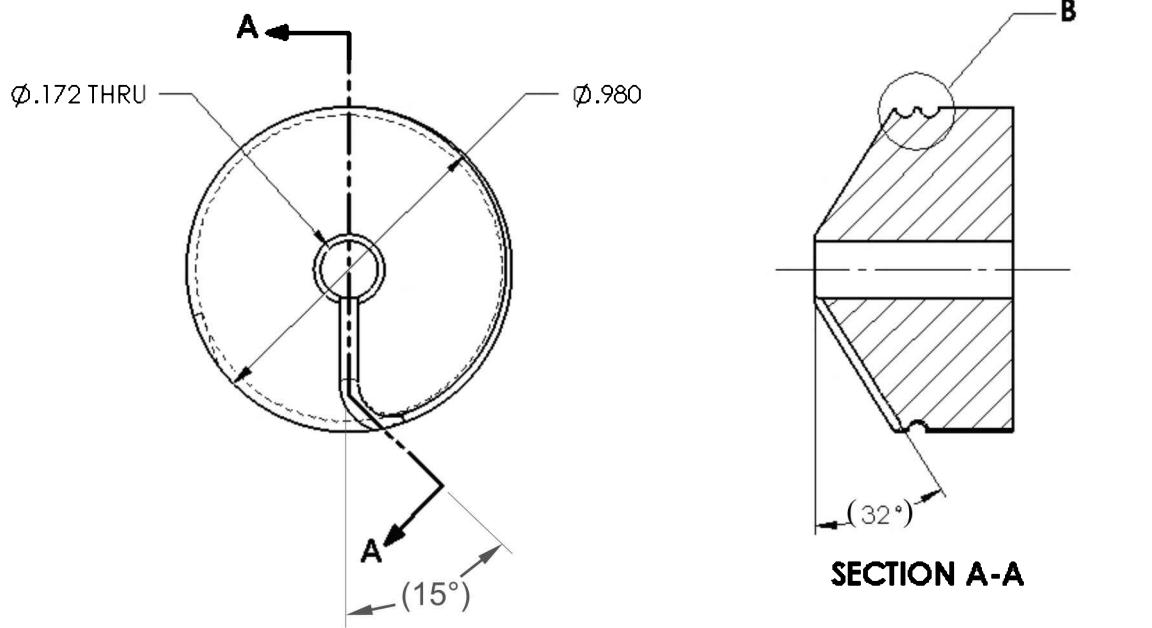
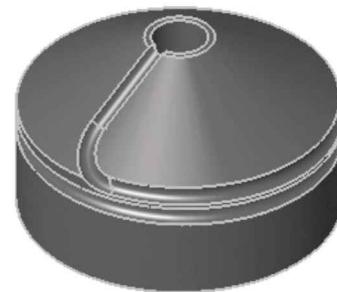
Enter **Equation Driven Curve.sldprt** for the file name.

Click Save.



Exercise: Projected Curve & Composite Curve

1. Create the part using the drawing provided below.
2. Dimensions are in inches, 3 decimal places.
3. Focus on Projected Curve & Composite Curve options.

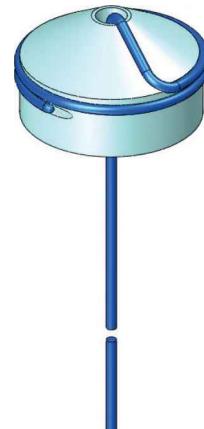


4. Follow the instructions on the following pages, if needed.

1. Opening a part document:

From the Training Files folder, locate and open the part document named:
Project and Composite Curves.

This is an actual die to form the shape of a catheter tip. We will create the groove that wraps around this die, using the Project and Composite Curve options.



2. Creating a helix:

Select the **Top** plane and open a **new sketch**.

Select the **bottom edge** (or top edge) and press **Convert- Entity**.

Click or select:

Insert / Curve / Helix-Spiral.

Enter the following:

Convert the
edge into
a circle

* **Constant Pitch.**

* **Pitch: .073"**

* **Reverse Direction**

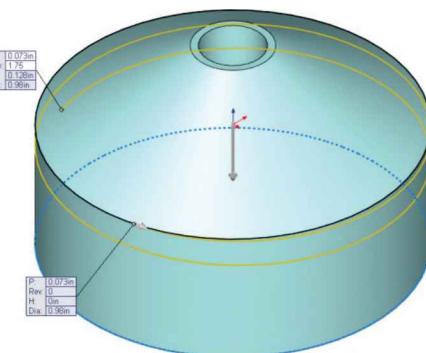
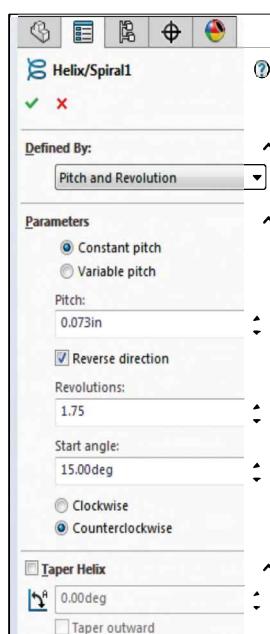
* **Revolutions: 1.75**

* **Start Angle: 15deg.**

* **Clockwise**

Click **OK**.

Notice the helix starts at a 15° angle! This way the end of the helix will align with the bend radius in the next step.



3. Sketching the upper transition:

Select the small upper face as indicated and open a **new sketch**.

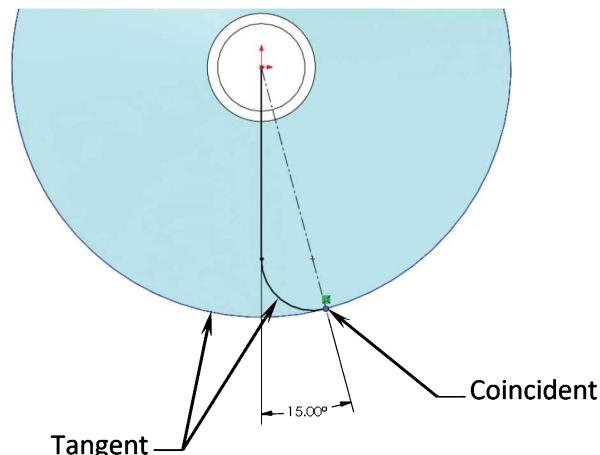
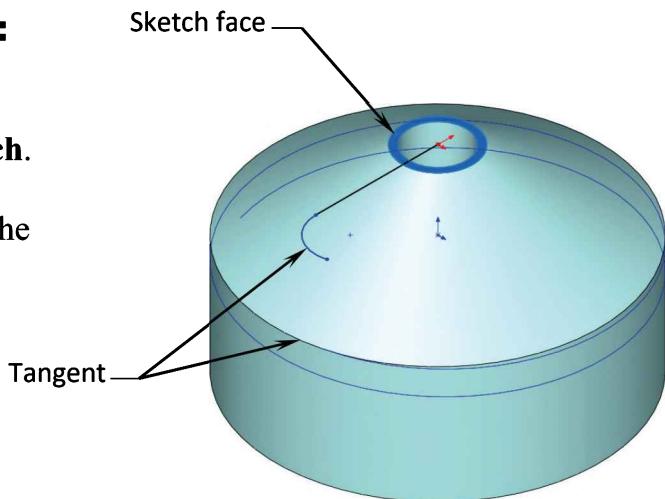
Sketch a **Line** that starts from the origin and connects with a **Tangent Arc**.

Add a **Tangent** relation between the arc and the circular edge of the part.

Sketch a **centerline** that starts from the origin and connects to the right endpoint of the arc.

Add a **Coincident** relation between the right endpoint of the arc and the circular edge of the part, then add a **15°** angular dimension.

The sketch should be fully defined at this point. Exit the sketch.



4. Creating a projected Curve:

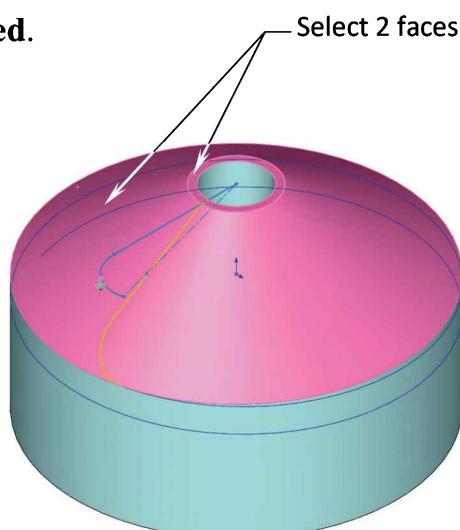
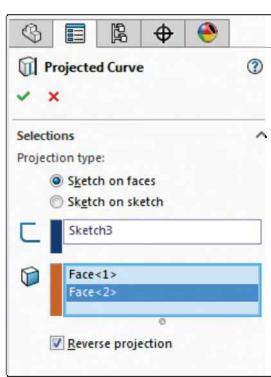
Click  or select: **Insert / Curve / Projected**.

Select the **Sketch on Faces** option.

For Sketch to Project, select Sketch3.

For Projection Faces, select the **2 faces** as noted, click **Reverse**.

Click **OK**.

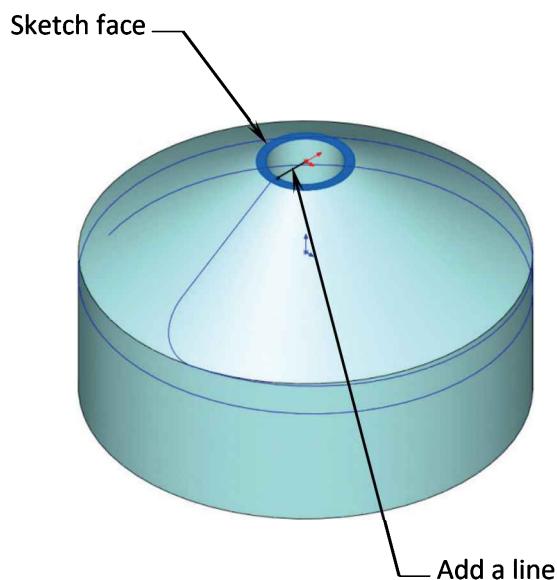


5. Adding a sketch line:

We want the cut to start from the origin, a sketch line is needed to connect the projected curve to the origin.

Sketch a **Line** that starts from the origin to the endpoint of the projected curve.

Exit the sketch.

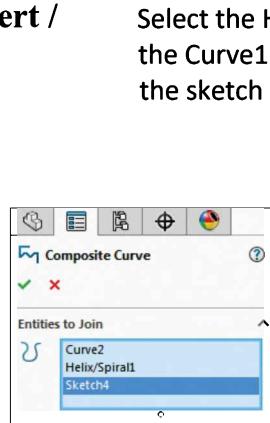


6. Creating a Composite Curve:

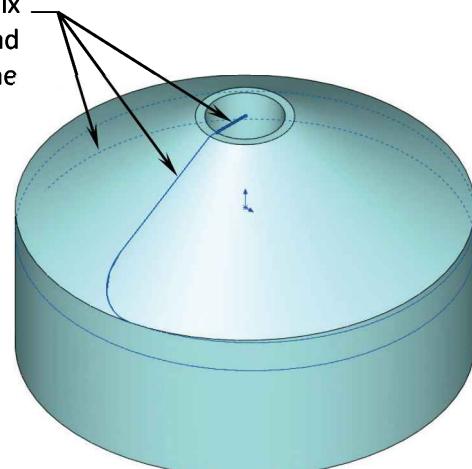
Click  or select: **Insert / Curve / Composite**.

Select either from the Feature tree or from the graphics: the **Helix**, the **Curve1** and the **Sketch4** (the line).

Click **OK**.



Select the Helix, the Curve1 and the sketch Line



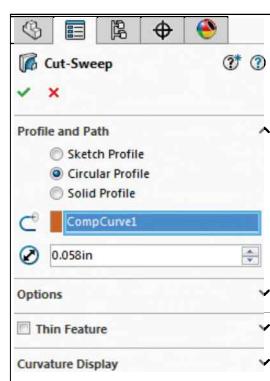
7. Creating a swept cut:

Click  or select: **Insert / Cut / Sweep**.

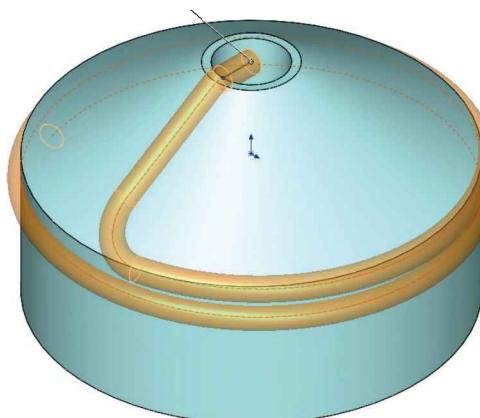
Select the **Circular Profile** option.

For Profile Diameter enter **.058in**.

For Sweep Path, select the **Composite Curve**.



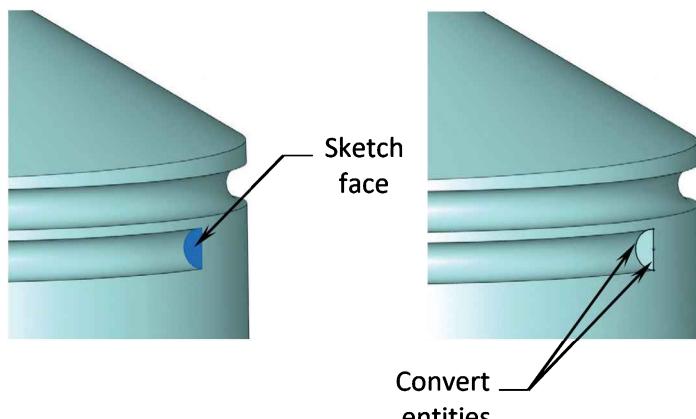
Click **OK**.



8. Removing the undercut:

If the swept feature stops short, we will need to clean it up.

Select the face as noted and open a **new sketch**.

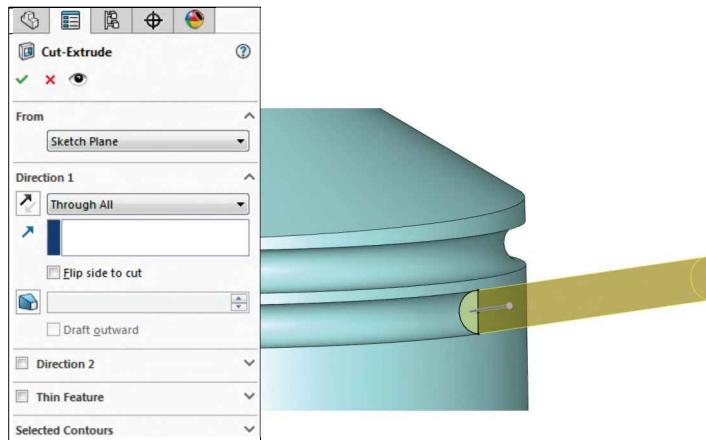


Convert the face into a sketch.

Click or select:
Insert / Cut / Extrude.

Set Direction 1 to:
Through All.

Click **OK**.

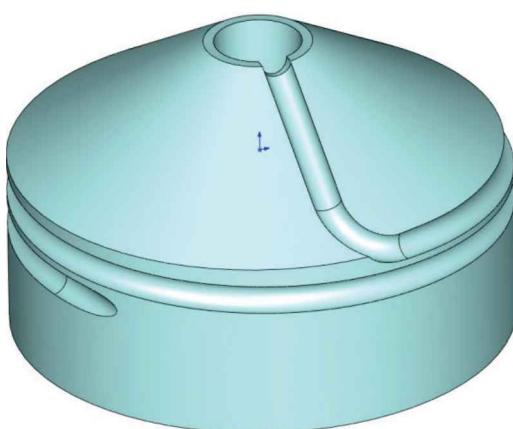


9. Saving your work:

Click **File / Save As**.

Enter **Project and Composite Curves** for the name of the file.

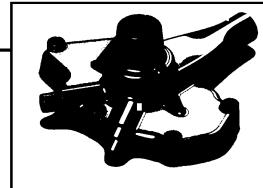
Press **Save**.



CHAPTER 5

Advanced Modeling with Sweep & Loft

Advanced Modeling with Sweep & Loft Water Pump Housing



The Sweep option creates a solid, thin, or surface feature by moving a single sketch profile along a path and guiding with one or more guide curves.

In order to create a sweep feature properly, a set of rules should be taken into consideration:



- The Sweep option uses only one sketch profile, and it must be a closed non-intersecting contour for a **solid** feature.
- The sketch profile can be either closed or open for a **surface** feature.
- Only one path is used in a sweep, and it can be open or closed.
- One or more guide-curves can be used to guide the sketch profile.
- The sketch profile must be drawn on a new plane starting at the end point of the path.

The Loft option creates a solid, thin, or surface feature by making a transition between the sketch profiles.

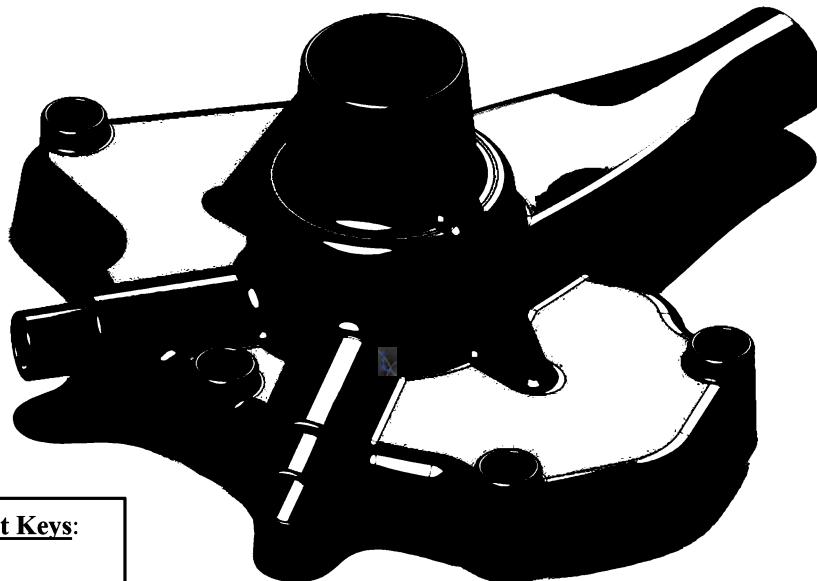
Keep in mind the following requirements when creating a loft feature:



- The Loft option uses multiple sketch profiles that must be closed, and non-intersecting for a **solid** feature.
- The sketch profiles can be either closed or open, for a **surface** feature.
- Use the Centerline Parameter option to guide the profiles from the inside.
- Use Guide Curves option to guide the sketch profiles from the outside.
- The Guide Curves can be either 2D or a 3D sketch.

Advanced Modeling with Sweep & Loft

Water Pump Housing



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI

Units: INCHES – 3 Decimals

Tools Needed:



Insert Sketch



Split Entities



Add Geometric Relations



Rib



Plane



Revolved Boss/Base



Extruded Boss/Base



Swept Boss/Base



Lofted Boss/Base

Understanding the Draft Options

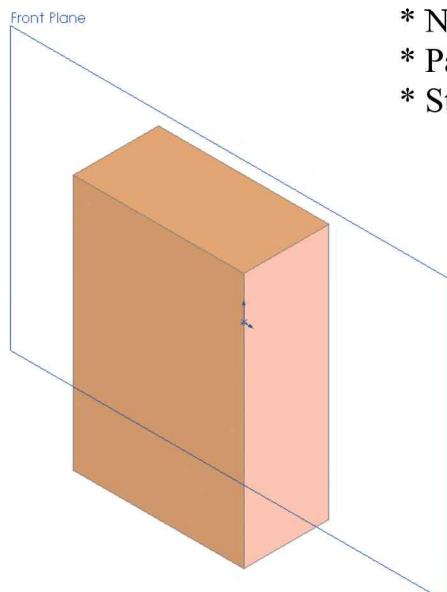
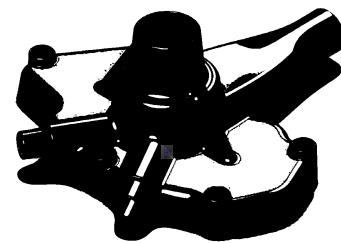
Drafts are usually required in most plastic injection molded parts to ensure proper part removal from the mold halves.

The Draft option in SOLIDWORKS adds tapers to the faces using the angles specified by the user.

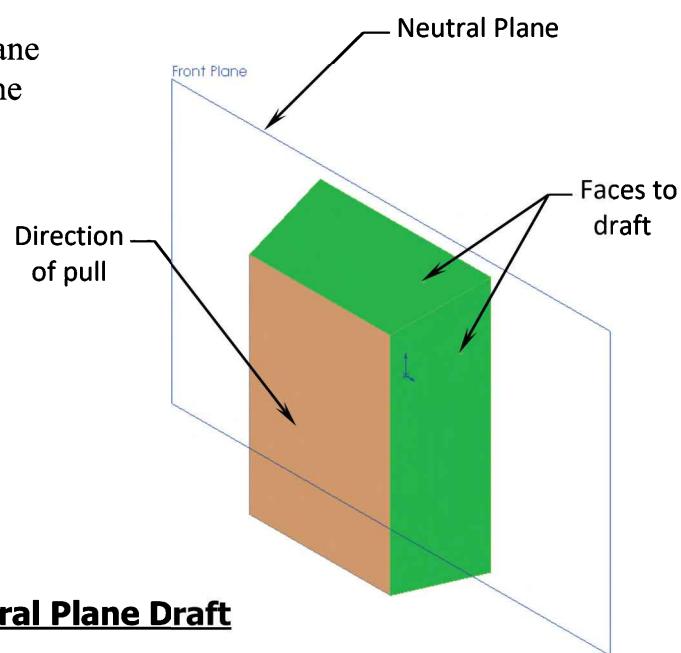
Drafts can be inserted in an existing part or added to a feature while being extruded.

Drafts can be applied to solid parts as well as the surface models.

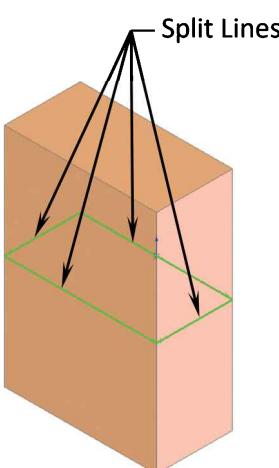
There are several types of draft available:



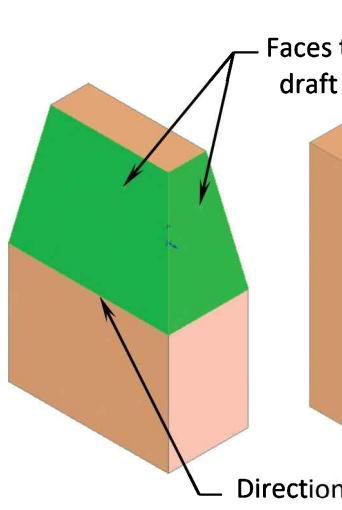
- * Neutral Plane
- * Parting Line
- * Step Draft



Neutral Plane Draft

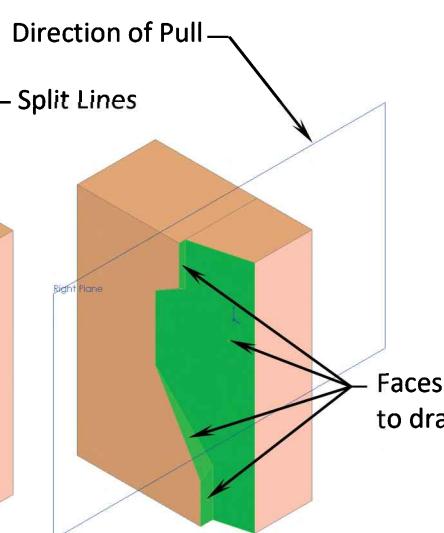
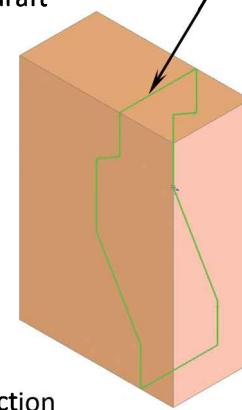


Split Line Draft



Direction
of pull

Faces to
draft



Faces
to draft

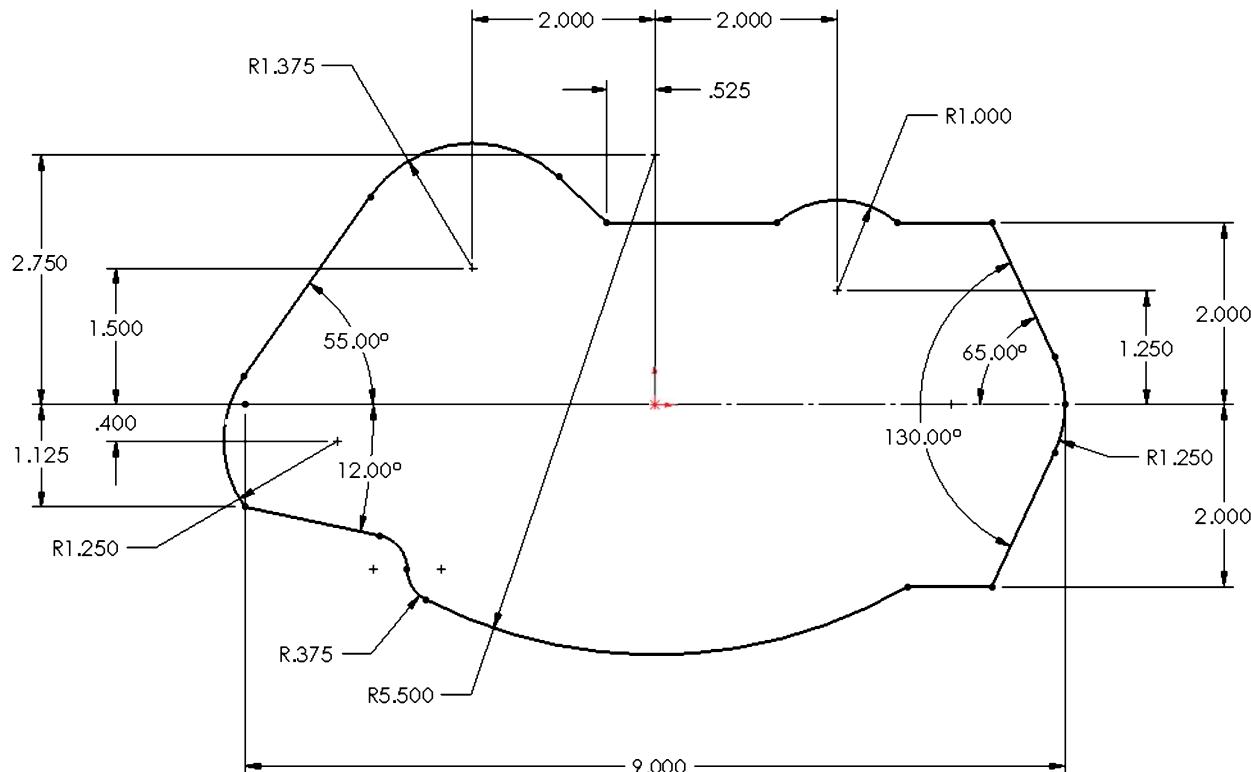
Step Draft

1. Opening a part document:

From the Training Files folder, locate and open a part document named: **Water Pump Sketch**.

The sketch was created ahead of time to help focus on the key features of this lesson: Sweep and Loft.

Edit the Sketch1.



2. Extruding the Base with Draft:

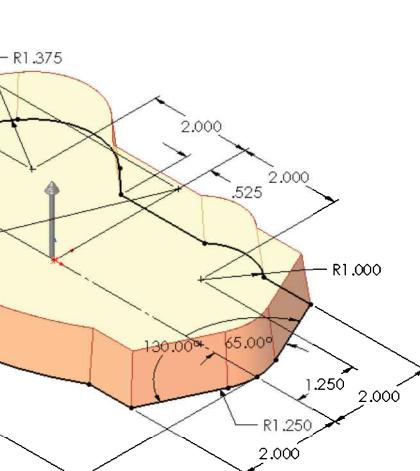
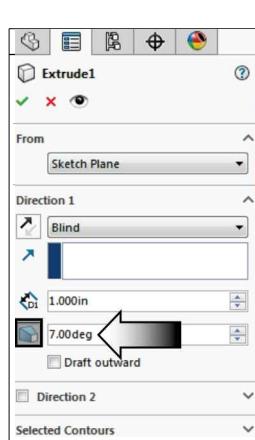
Click **Extruded Boss/Base**.

End Condition: **Blind**.

Depth: **1.00in**.

Draft: **7 deg. inward**.

Click **OK**.



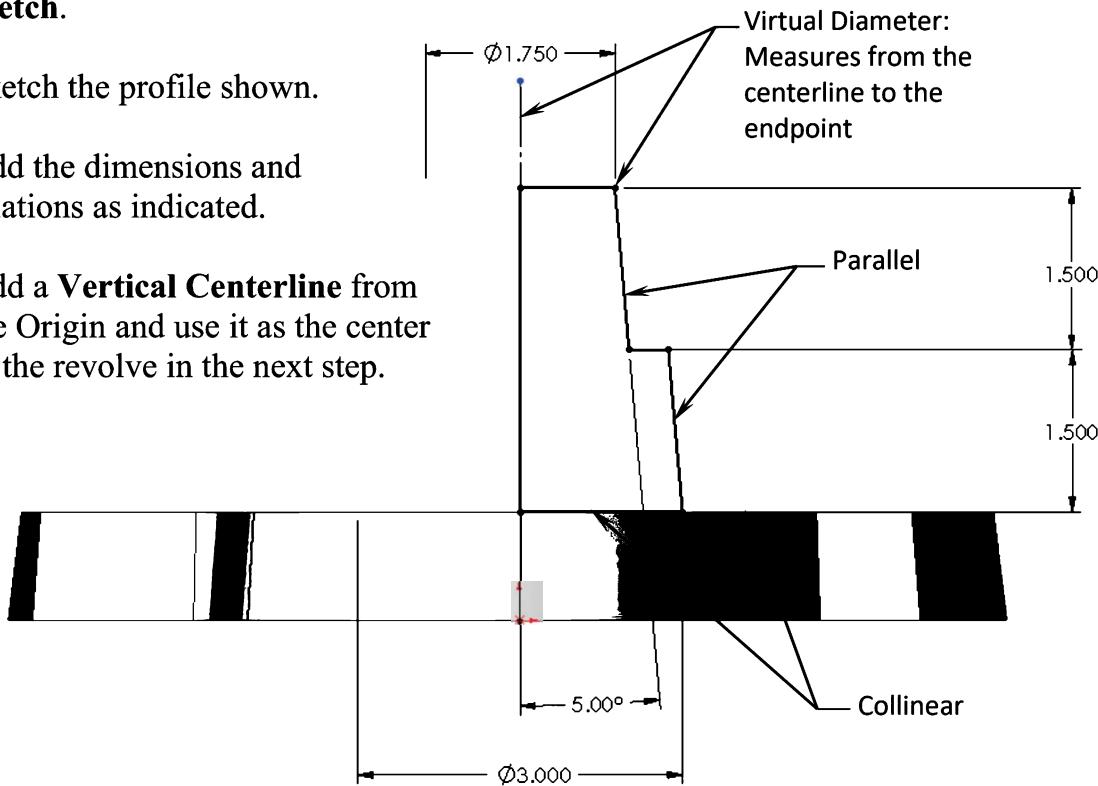
3. Sketching the upper Inlet Port:

Select the **Front** plane and open a **new-sketch**.

Sketch the profile shown.

Add the dimensions and relations as indicated.

Add a **Vertical Centerline** from the Origin and use it as the center of the revolve in the next step.



4. Revolving the upper Inlet Port:

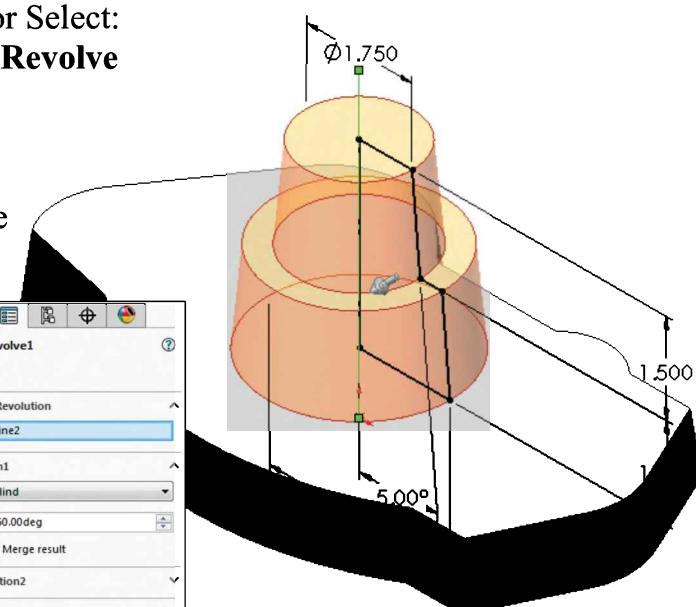
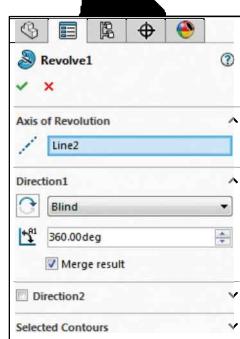
Click **Revolve Boss/Base**  or Select: **Insert / Features / Boss-Base / Revolve** from the drop-down menus.

The vertical centerline should be selected automatically to use as the Axis of Revolution.

Revolve Direction: **Blind**.

Revolve Angle: **360 deg.**

Click **OK**.



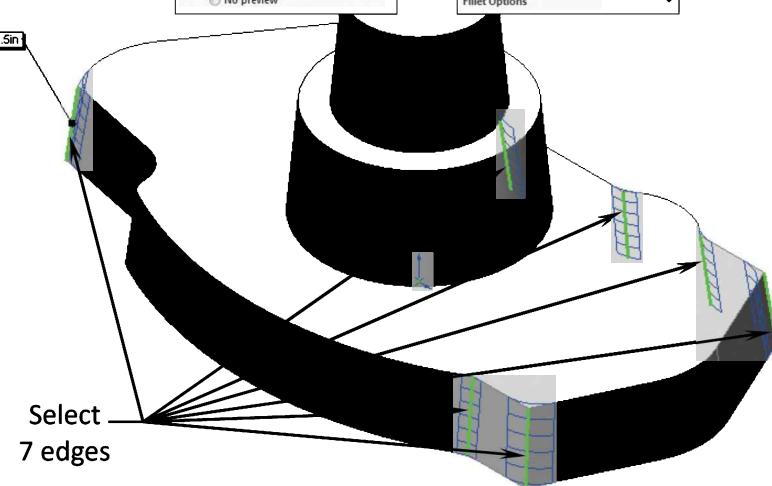
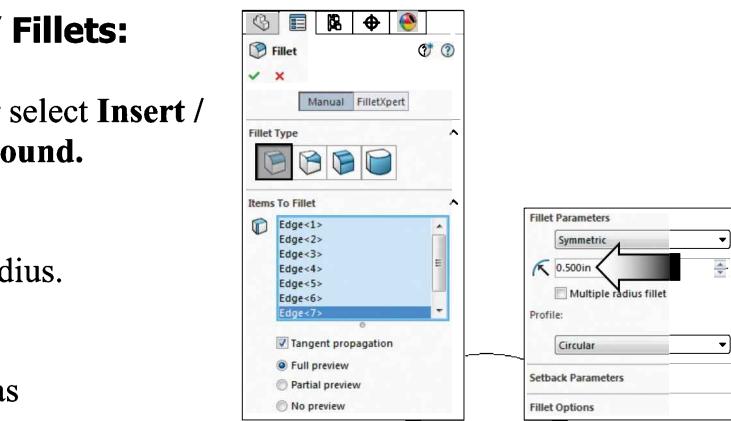
5. Adding the .500" Fillets:

Click **Fillet**  or select **Insert / Features / Fillet-Round**.

Enter **.500in.** for radius.

Select the **7 edges** as indicated.

Click **OK**.



6. Adding the .275" Fillets:

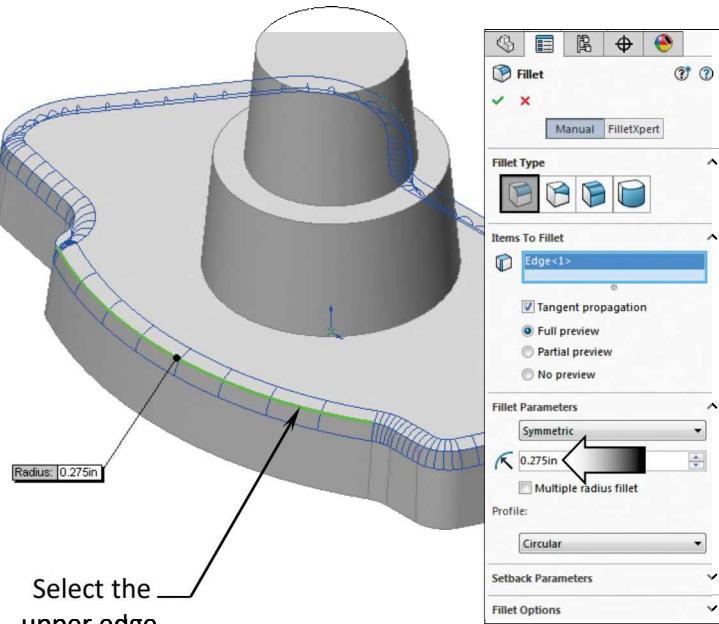
Click **Fillet**  or select **Insert / Features / Fillet-Round**.

Enter **.275in.** for radius.

Select one of the **upper edges** of the base.

Enabled the option **Tangent Propagation** to allow the fillet to propagate itself to all connecting edges.

Click **OK**.



Select the upper edge

7. Creating the 1st Offset-Distance Plane:

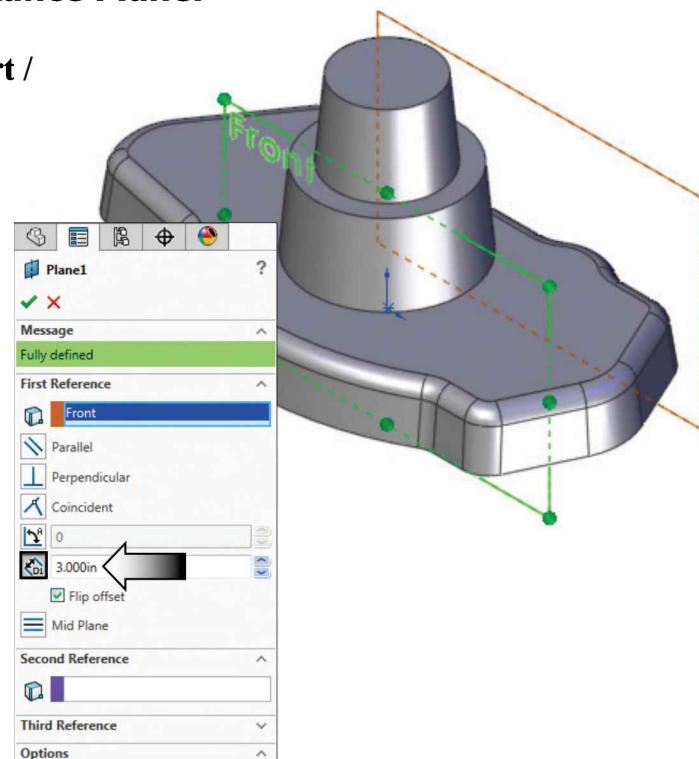
Click **Plane**  or select **Insert / Reference Geometry / Plane**.

From the FeatureManager tree, select the **Front** plane to offset from.

Enter **3.000in.** for distance.

Place the new plane on the **right side** of the front plane.

Click **OK**.



8. Creating the 2nd Offset-Distance Plane:

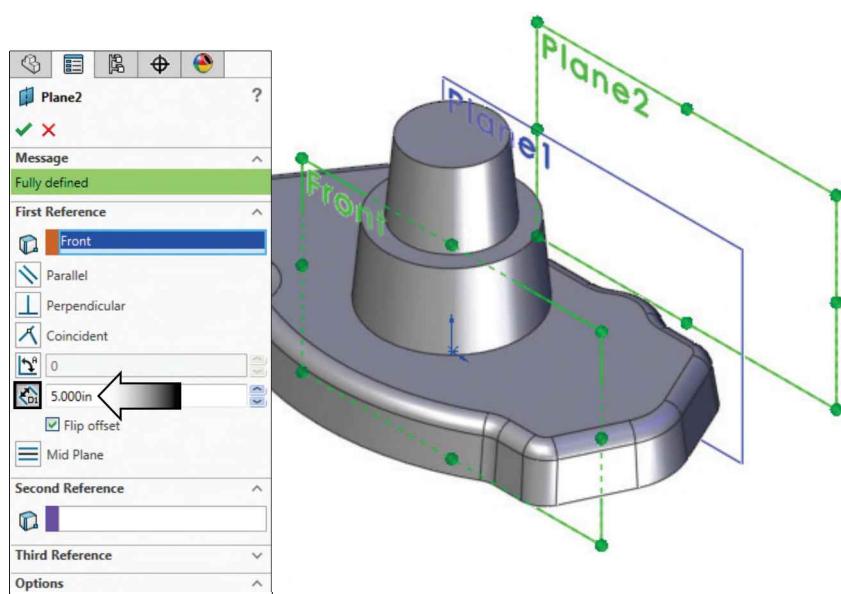
Click **Plane**  again or select **Insert / Reference Geometry / Plane**.

Select the **Front** plane from the FeatureManager tree to offset from.

Enter **5.000in.** for offset distance.

Place the new plane also on the **right side** of the front plane.

Click **OK**.



9. Creating the 3rd Offset-Distance Plane:

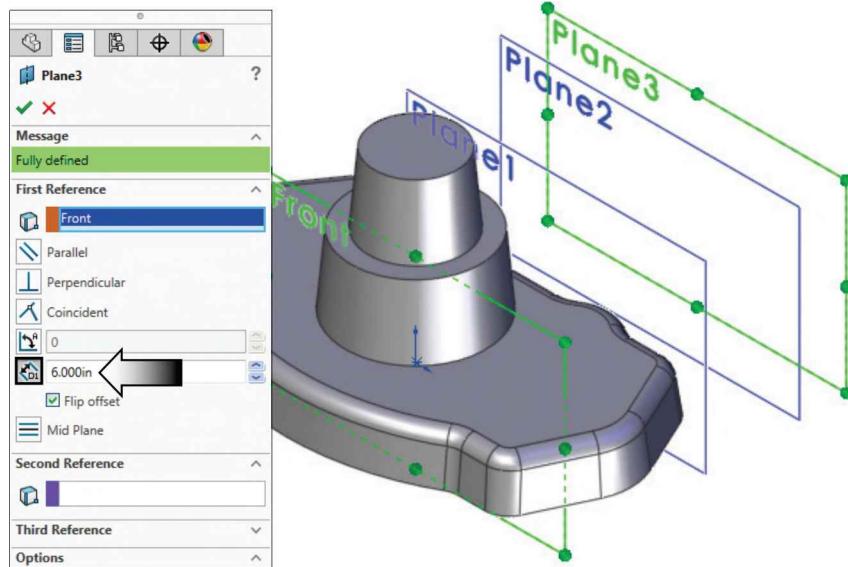
Click **Plane**  or select **Insert / Reference Geometry / Plane**.

Select the Front plane once again from the FeatureManager tree to offset from.

Since the distance of each plane is not the same, we will have to create them individually.

Enter **6.000in.** for offset distance.

Place the new plane also on the **right side**.



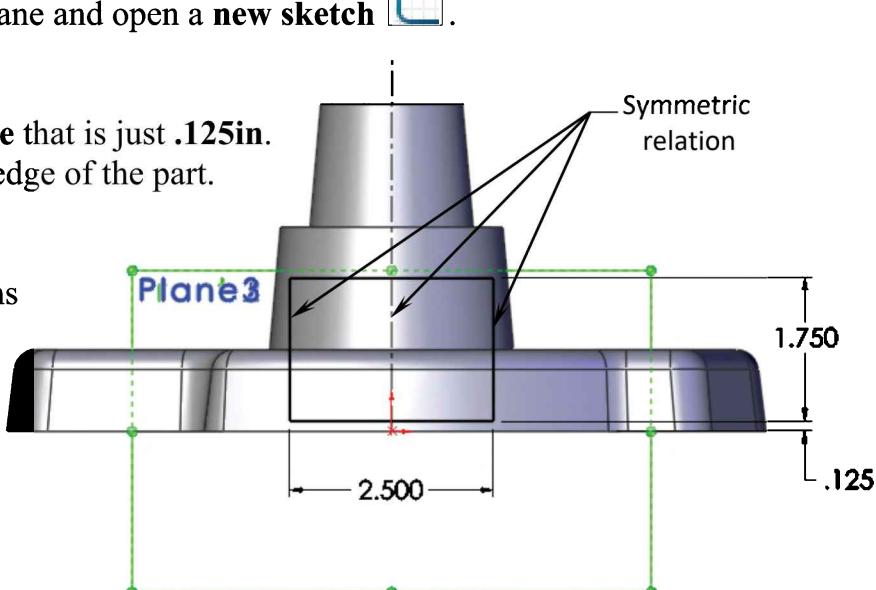
Click **OK**.

10. Sketching the 1st loft profile:

Select the Front plane and open a **new sketch** .

Sketch a **Rectangle** that is just **.125in.** above the bottom edge of the part.

Add the dimensions and relations as shown to fully define the sketch.



Exit the sketch.

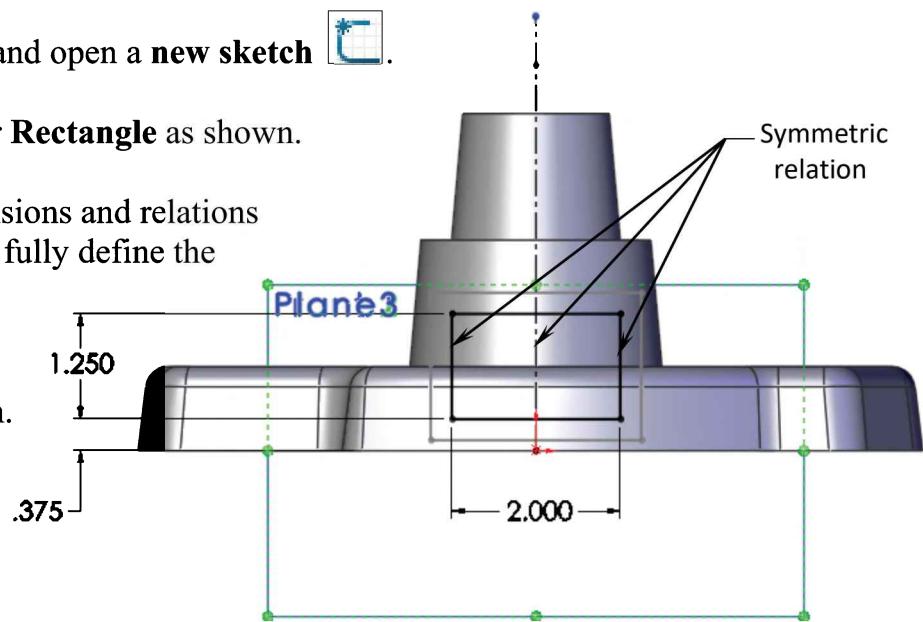
11. Sketching the 2nd loft profile:

Select Plane1 and open a new sketch .

Sketch another **Rectangle** as shown.

Add the dimensions and relations as indicated to fully define the sketch.

Exit the sketch.



12. Sketching the 3rd loft profile:

Select Plane2 and open a new sketch .

Sketch a **Circle** just above the Origin as shown below.

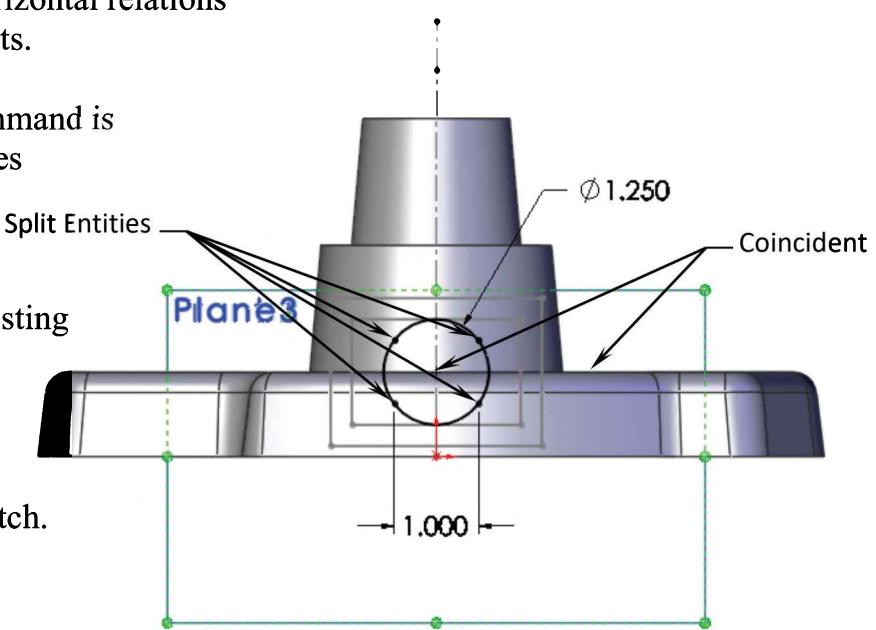
Use the **Split-Entities**  command and split the circle into **4 segments**.

Add **Vertical** and **Horizontal** relations between the split points.

The Split Entities command is used to split the entities in each sketch to an even number of connecting points to help control the twisting issue in a loft feature.

Add the dimensions and relations needed to fully define the sketch.

Exit the sketch.

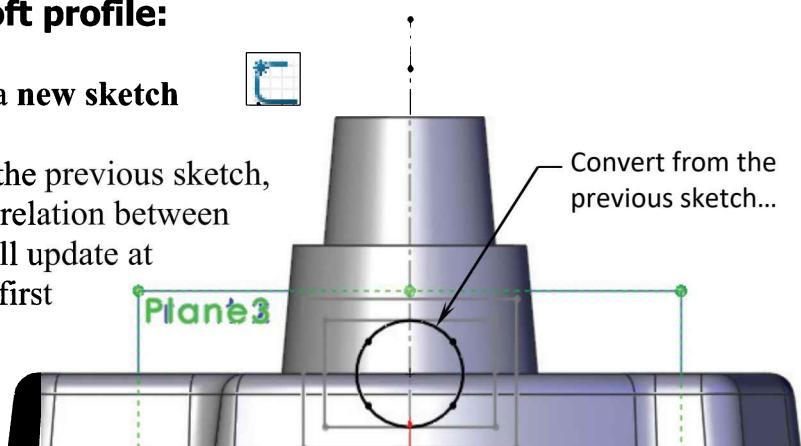


13. Sketching the 4th loft profile:

Select Plane3 and open a new sketch



Convert the circle from the previous sketch, this creates an On-Edge relation between the 2 circles and they will update at the same time when the first circle is changed.



Exit the sketch.

14. Creating a loft feature:

Click Loft or select: Insert / Boss-Base / Loft from the drop-down menus.

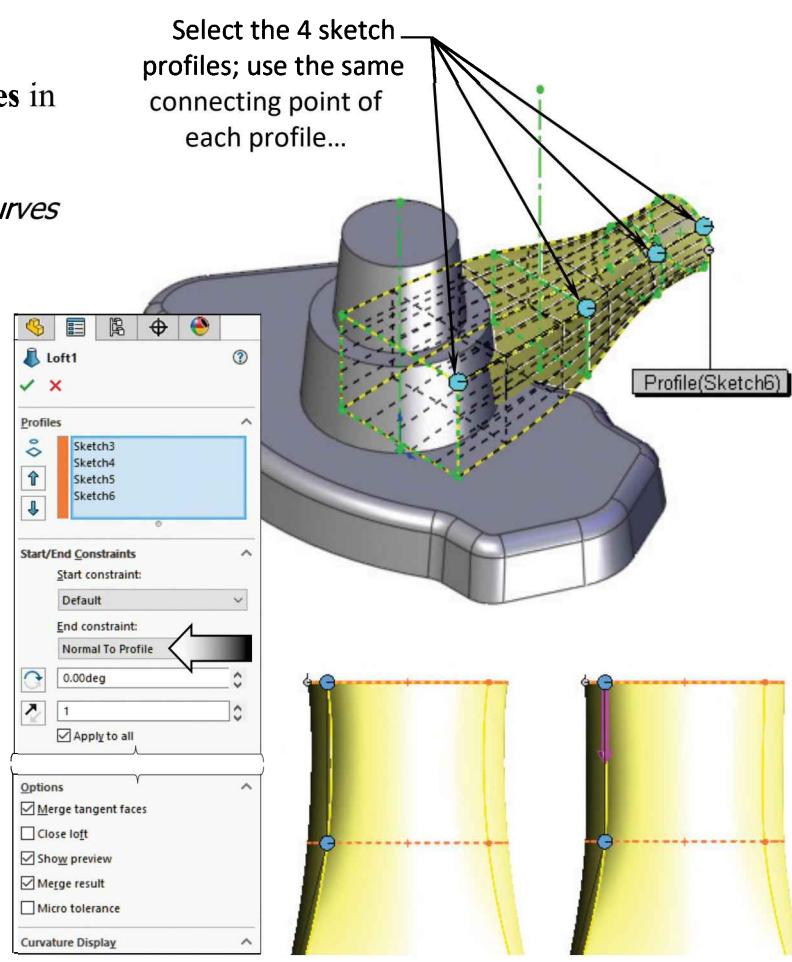
Select the 4 sketch profiles in the graphics area.

(Since there are no guide curves to help control the loft, the profiles should be selected from the same connector each time to prevent them from twisting.)

For clarity, right-click in the yellow shaded area and select the following options:

- * Opaque Preview
- * Clear Meshed Faces

Expand the Start/End Constraints section and Set the End Constraint to **Normal to Profile**.



Click OK.

15. Creating the mounting bosses:

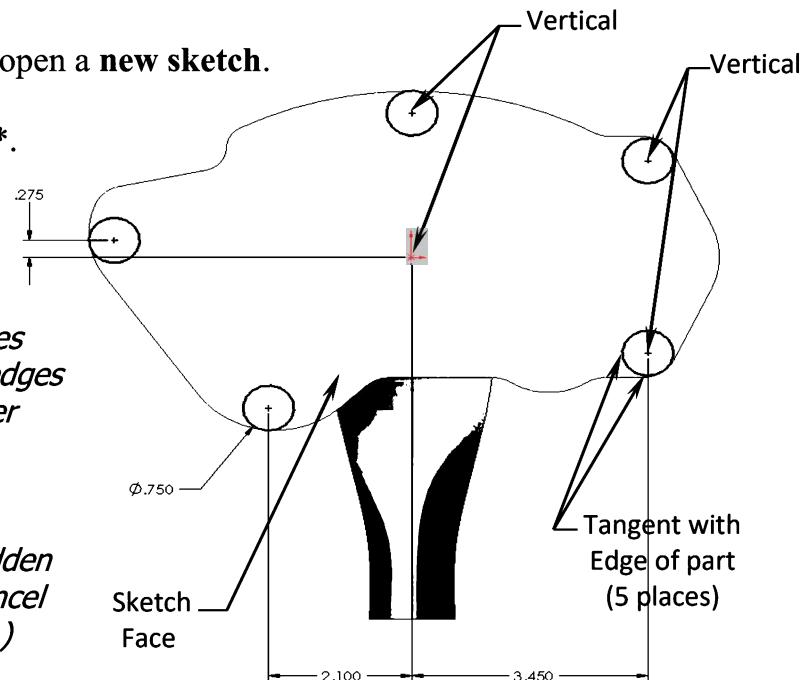
Select the bottom face and open a **new sketch**.

Sketch **5 Circles** as shown*.

* Avoid the hidden entities.

(When sketching, your circles may snap to some hidden edges in the model causing an over defined error when adding the dimensions.

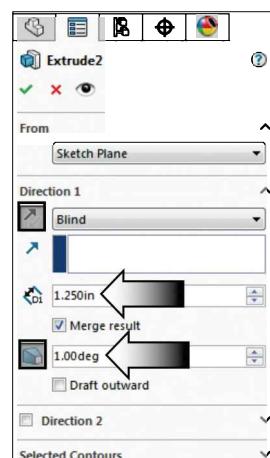
To overcome this, hold the control key every time a hidden edge highlights, this will cancel the Auto-Relation snapping.)



Add a **Tangent** relation for each circle to the outer edge of the part.

Add an **Equal** relation to all 5 circles.

Add dimensions to fully position the 5 circles.



16. Extruding the 5 mounting bosses:

Click **Extrude Boss-Base** or select: **Insert / Extrude / Boss-Base**, from the drop-down menus.

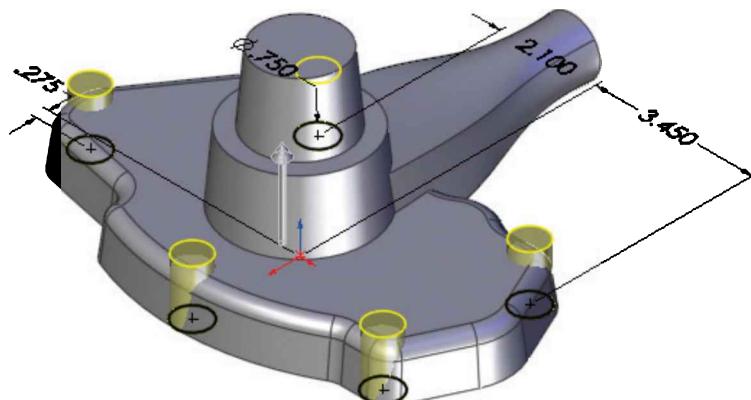
Set the following:

End Condition: **Blind**

Depth: **1.250in**.

Draft: **1 deg. Inward**.

Click **OK**.



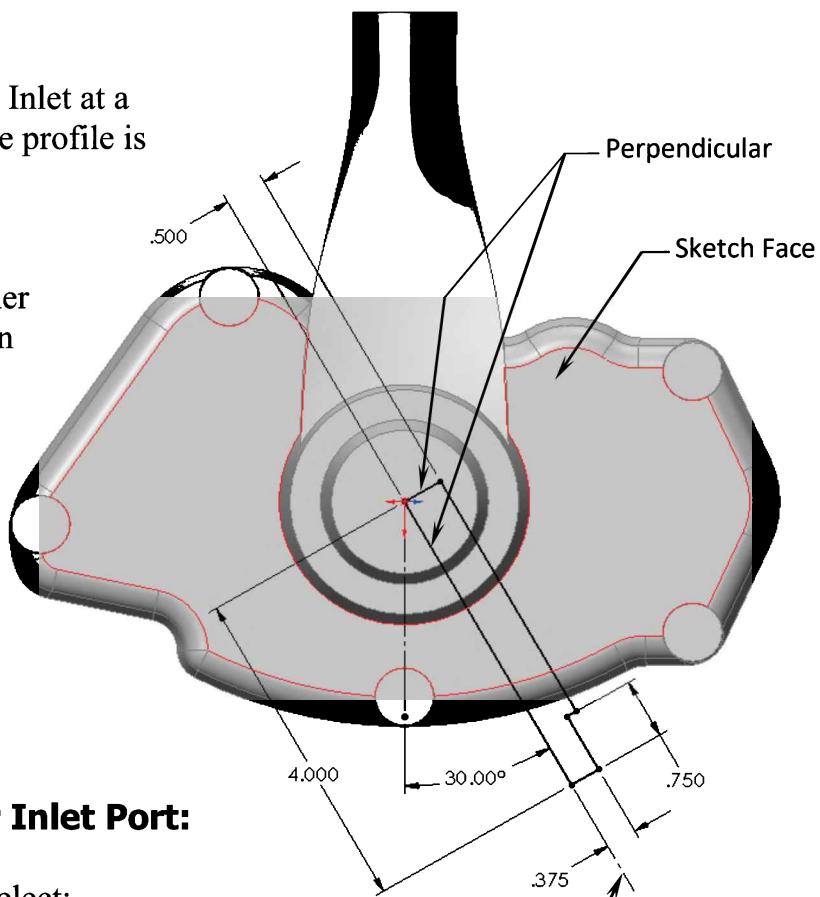
17. Sketching the rear Inlet Port:

Select the upper face and open a **new sketch**.

Sketch the profile of the Inlet at a **30° angle**; one end of the profile is locked on the Origin.

Add dimensions and other relations to fully position the sketch.

There is more than one centerline in this sketch; select the 30° centerline before clicking the revolve command.



18. Revolving the Rear Inlet Port:

Click **Revolve** or select:
Insert / Boss-Base / Revolve.

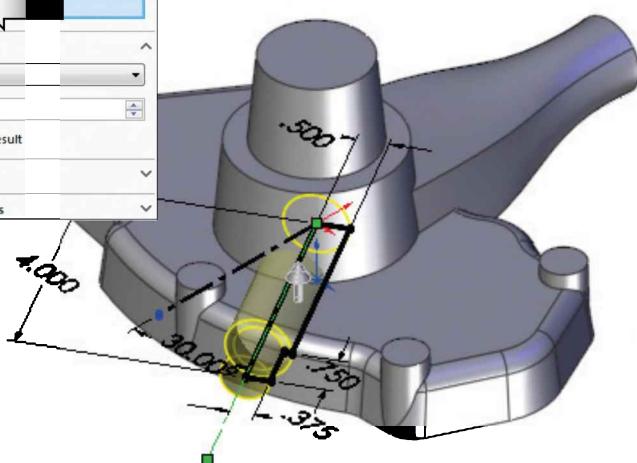
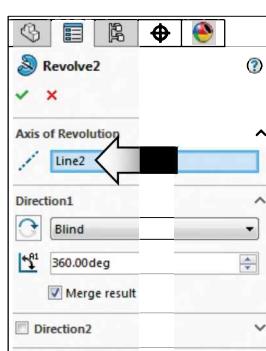
Select the **30° centerline** and set the following:

Revolve Type: **Blind**.

Revolve Angle: **360 deg**.

Merge Result: **Enabled**

Click **OK**.



19. Adding the 1st Face Fillet:

Click **Fillet**  or select:
Insert / Features / Fillet-Round.

Select the **Face Fillet** option.

Enter **.250in.** for radius.

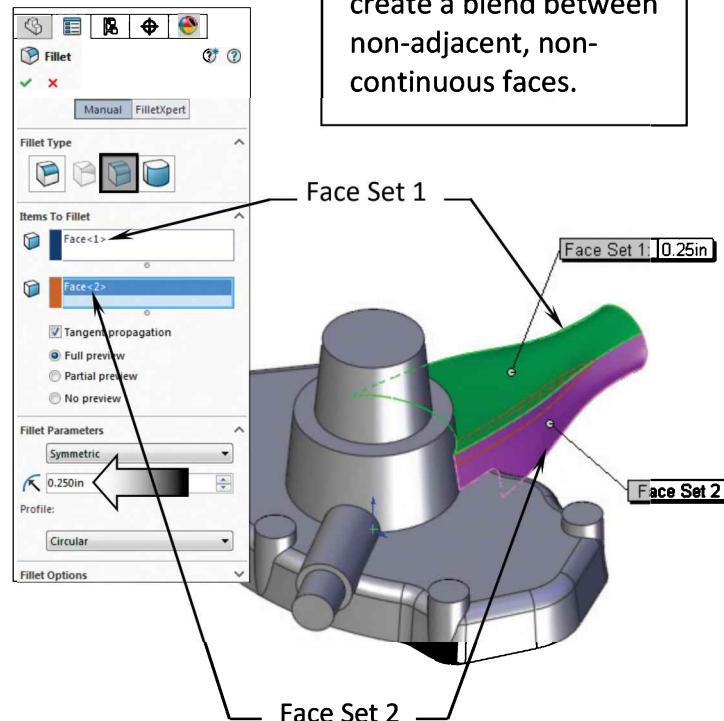
For **Face Set 1**, select the **upper face** of the lofted feature.

For **Face Set 2**, select the **side face** of the lofted feature.

Click **OK**.

Face Fillet

A Face Fillet is used to create a blend between non-adjacent, non-continuous faces.



20. Adding the 2nd Face Fillet:

Click **Fillet**  or select: **Insert / Features / Fillet-Round.**

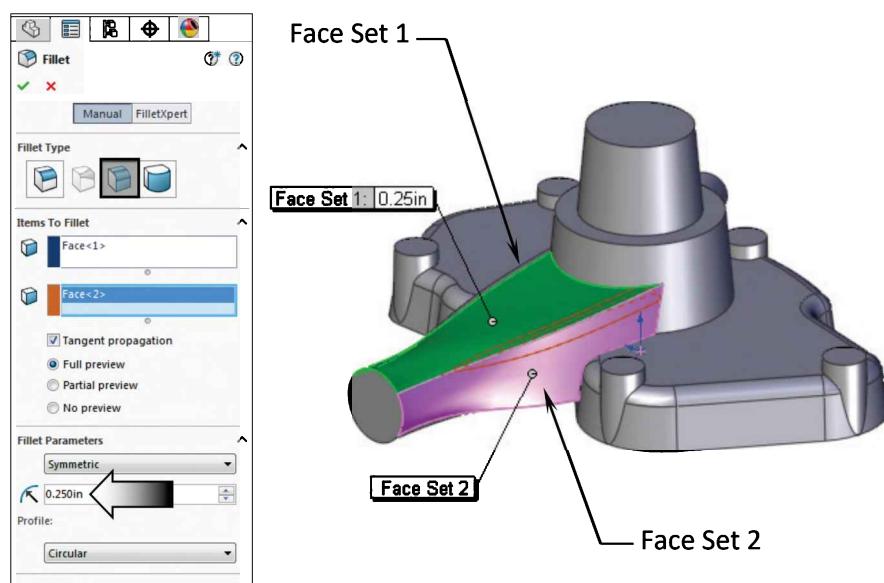
Select the **Face Fillet** button again.

Enter **.250in.** for radius.

For **Face Set 1**, select the **upper face** of the lofted feature.

For **Face Set 2**, select the **side face** of the lofted feature.

Click **OK**.



21. Adding the 3rd Face Fillet:

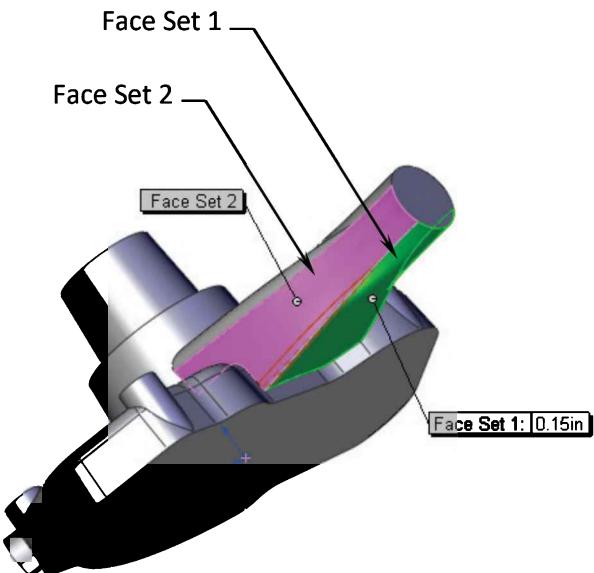
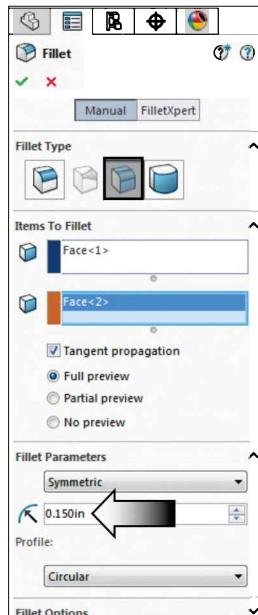
Click **Fillet**  or select:
Insert / Features / Fillet-Round.

Click Face Fillet.

Enter **.150in.** for radius.

For **Face Set 1**, select the **bottom face** of the lofted feature.

For **Face Set 2**, select the **side face** of the lofted feature.



Click OK.

22. Adding the 4th Face Fillet:

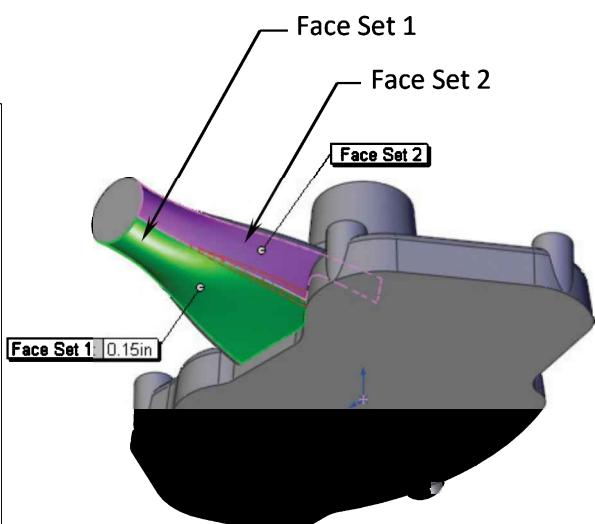
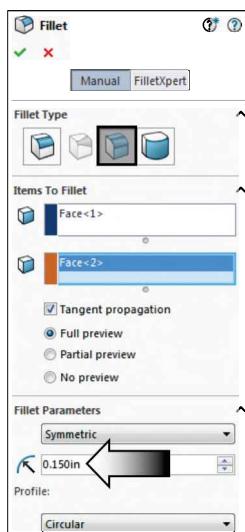
Click **Fillet**  or select:
Insert / Features / Fillet-Round.

Click Face Fillet.

Enter **.150in.** for radius.

For **Face Set 1**, select the **bottom face** of the lofted feature.

For **Face Set 2**, select the **side face** of the lofted feature.



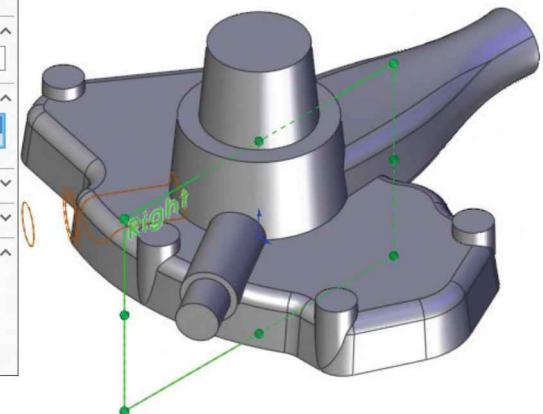
Click OK.

23. Mirroring the rear Inlet Port:

Click **Mirror**  or select:
Insert / Pattern Mirror / Mirror.

For Mirror Face/Plane,
select the **Right** plane from
the FeatureManager tree.

For Features to Mirror,
select the **Rear Inlet Port**
either from the graphics
area or from the feature tree.



Click **OK**.

24. Adding the .175" Fillets:

Click **Fillet**  or select: **Insert / Features / Fillet-Round.**

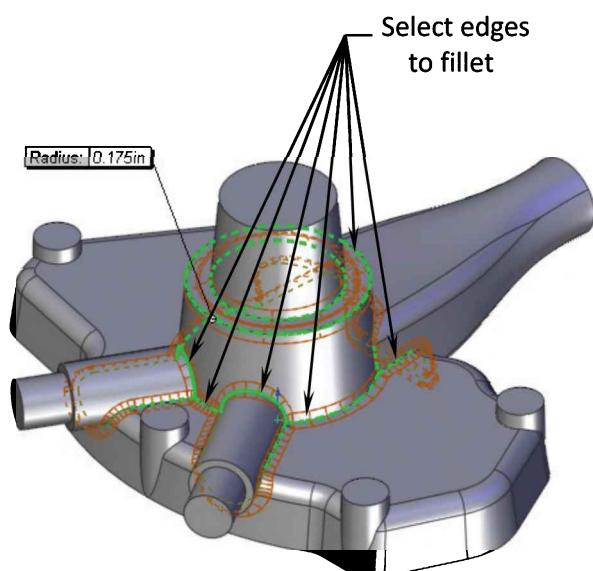
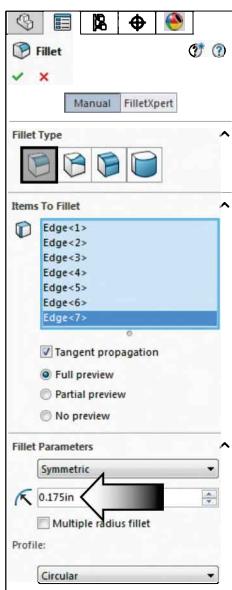
Select the **Constant Size** fillet option.

Enter **.175in.** for
radius.

Select the **edges** as
noted to add the fillets.

The option **Tangent Propagation** should be
enabled by default.

Click **OK**.



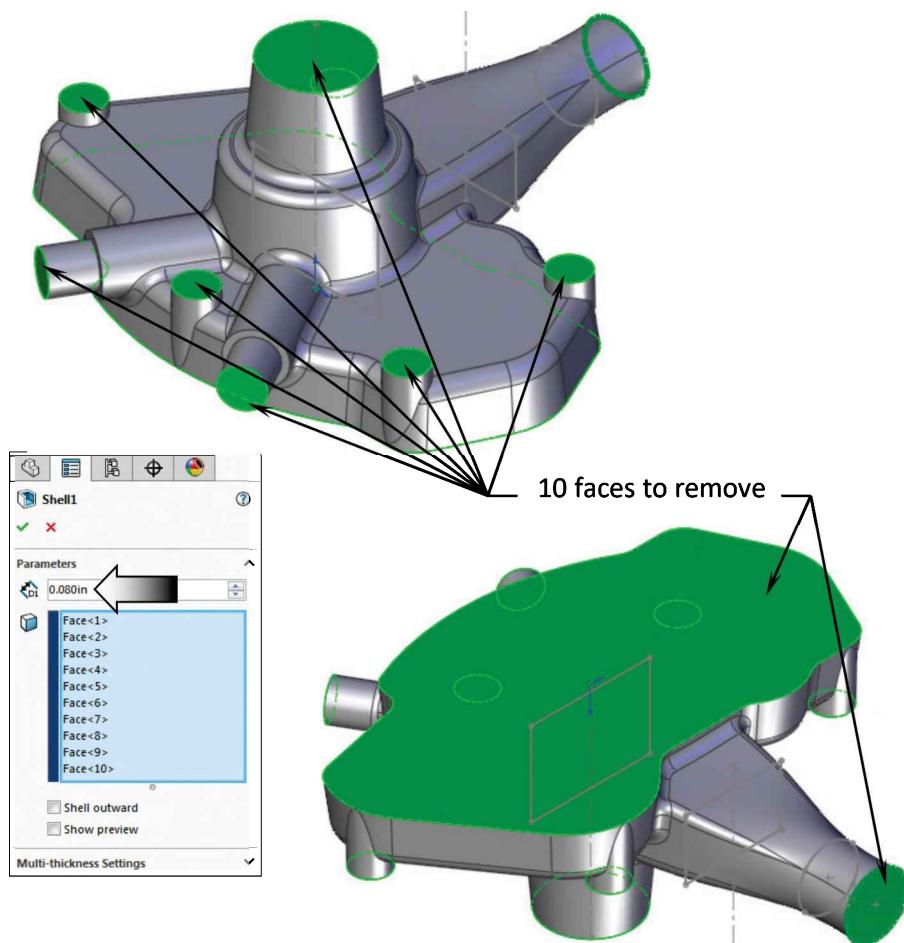
25. Shelling the part:

Click **Shell**  or select: **Insert / Features / Shell**.

Under the Parameters section, enter **.080in.** for wall thickness.

Select a total of **10** faces to remove.

Click **OK**.



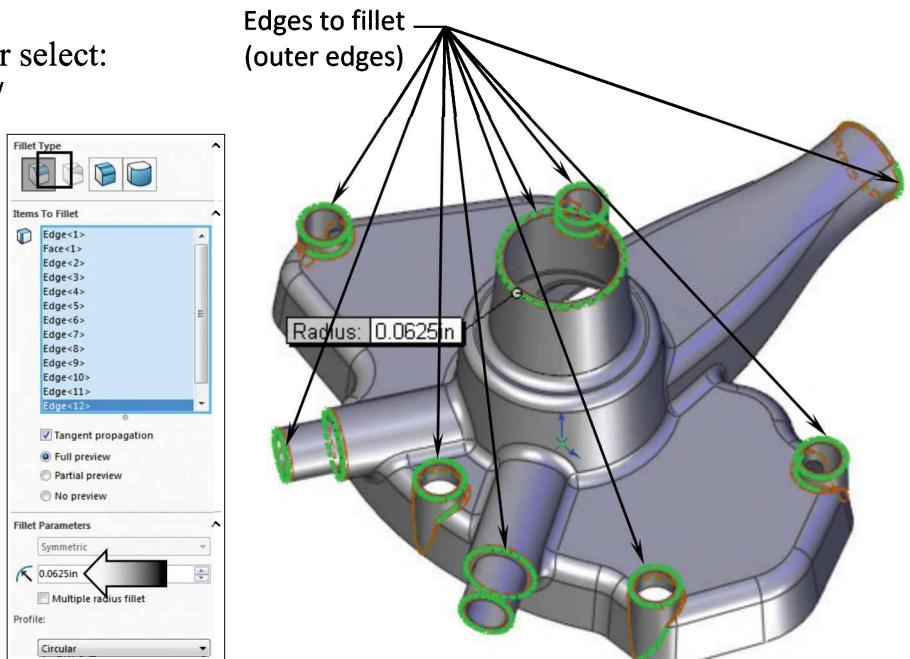
26. Adding the .0625" Fillets:

Click **Fillet**  or select: **Insert / Features / Fillet-Round**.

Enter **.0625in.** for radius size.

Select the **edges** as noted to add the fillets.

Click **OK**.



27. Adding a Rib:

Select the **Front** plane from the FeatureManager tree and open a **new sketch** .

Sketch a **Line** as shown below.

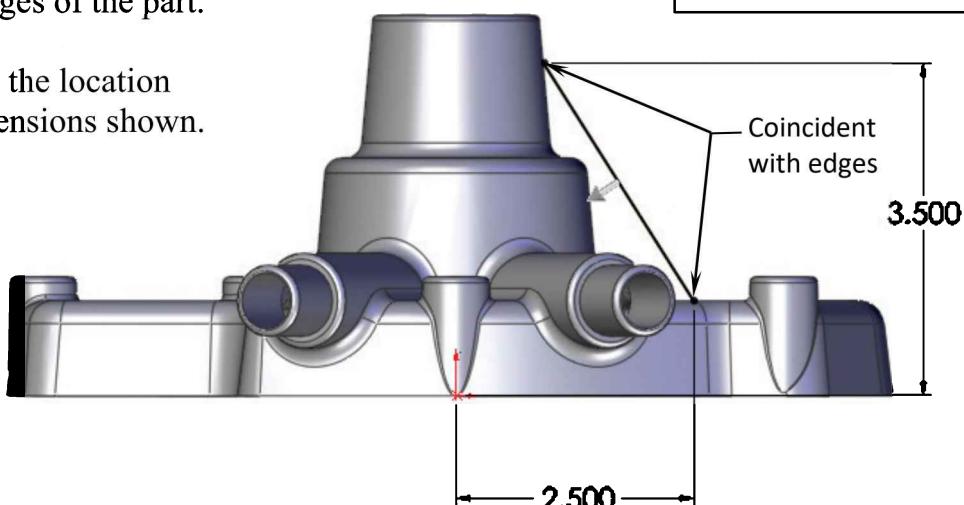
Add the coincident relations between the 2 endpoints of the line and the 2 edges of the part.

Add the location dimensions shown.



Rib Features

A rib is an extruded feature which adds material of a specified thickness in a specified direction. Drafts can also be added to the faces of the rib.



28. Extruding the Rib:

Click **Rib**  or select: **Insert / Features / Rib**.

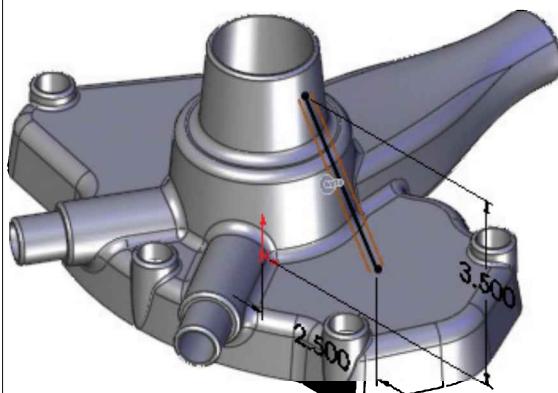
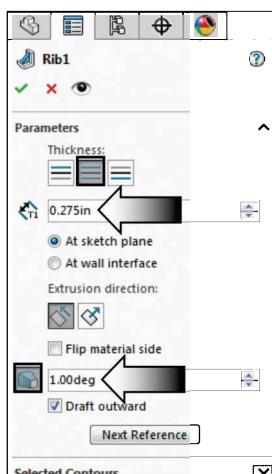
Select **Both Directions** under the Thickness section.

Enter **.275in.** for the thickness of the rib.

Enable the **Draft** option and enter **1.00deg**.

Enable the **Draft Outward** check box.

Click **OK**.



29. Creating a Full-Round fillet:

A Full-Round creates fillets that are tangent to three adjacent face sets.

Click **Fillet**  or select: **Insert / Features / Fillet-Round**.

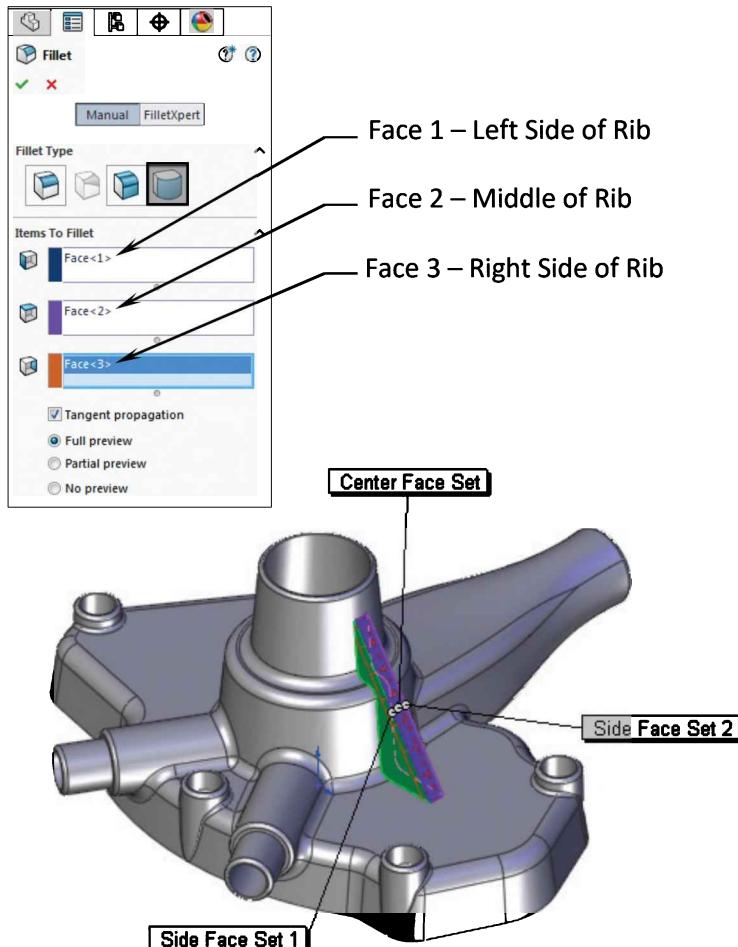
Select the **Full Round** fillet option.

For Face 1, select the **left face** of the Rib.

For Face 2, select the **middle face** of the Rib.

For Face 3, select the **right face** of the Rib.

Click **OK**.



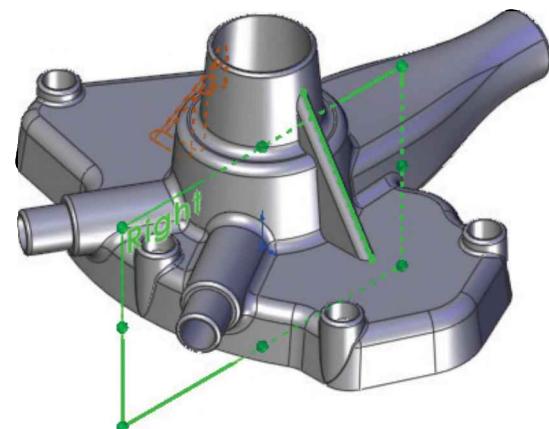
30. Mirroring the Rib:

Click **Mirror**  or select: **Insert / Pattern Mirror / Mirror**.

For Mirror Face/Plane, select the **Right** plane.

For Features to Mirror, select the **Rib** and its **fillet**.

Click **OK**.



31. Adding the .025" fillets:

Click **Fillet**  or select: **Insert / Features / Fillet-Round.**

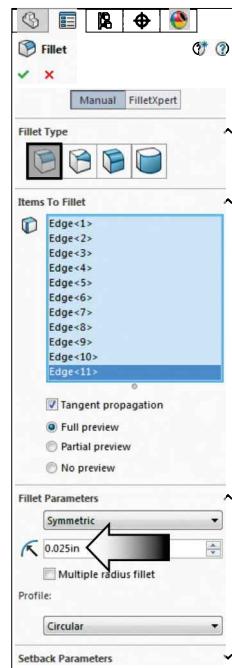
Select the **Constant Radius** fillet option.

Enter **.025in.** for radius value.

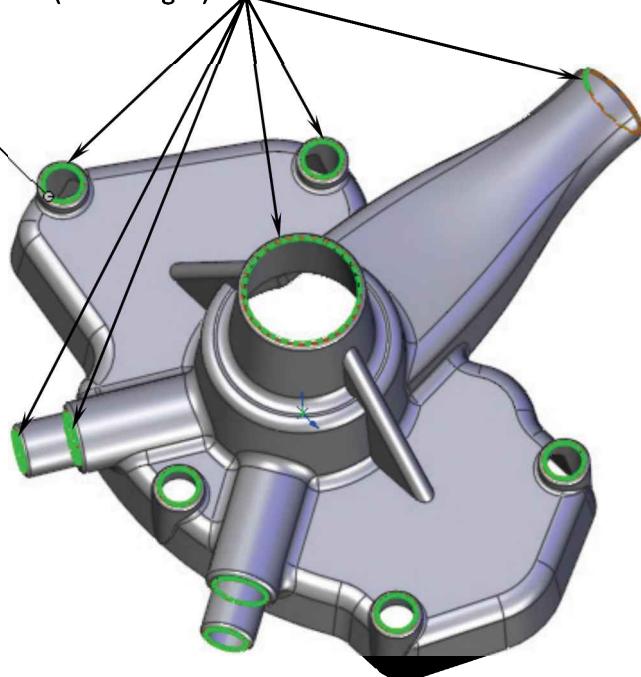
Select the **edges** as indicated to add the fillets.

Enable the **Tangent-Propagation** checkbox.

Click **OK.**



Select edges
(Inner Edges)



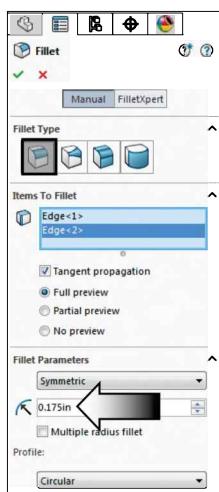
32. Adding the .175" fillets:

Click **Fillet**  or select: **Insert / Features / Fillet-Round.**

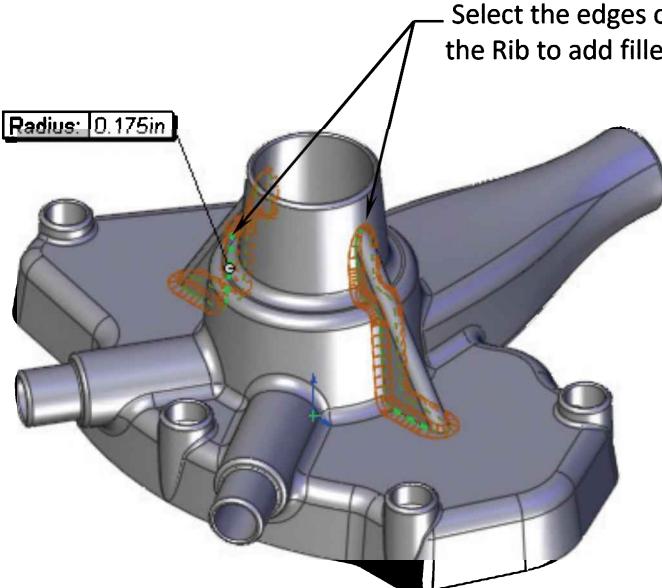
Click **Constant Radius.**

Enter **.175in.** for radius value.

Select the **edges** of the 2 ribs as indicated.



Select the edges of
the Rib to add fillets



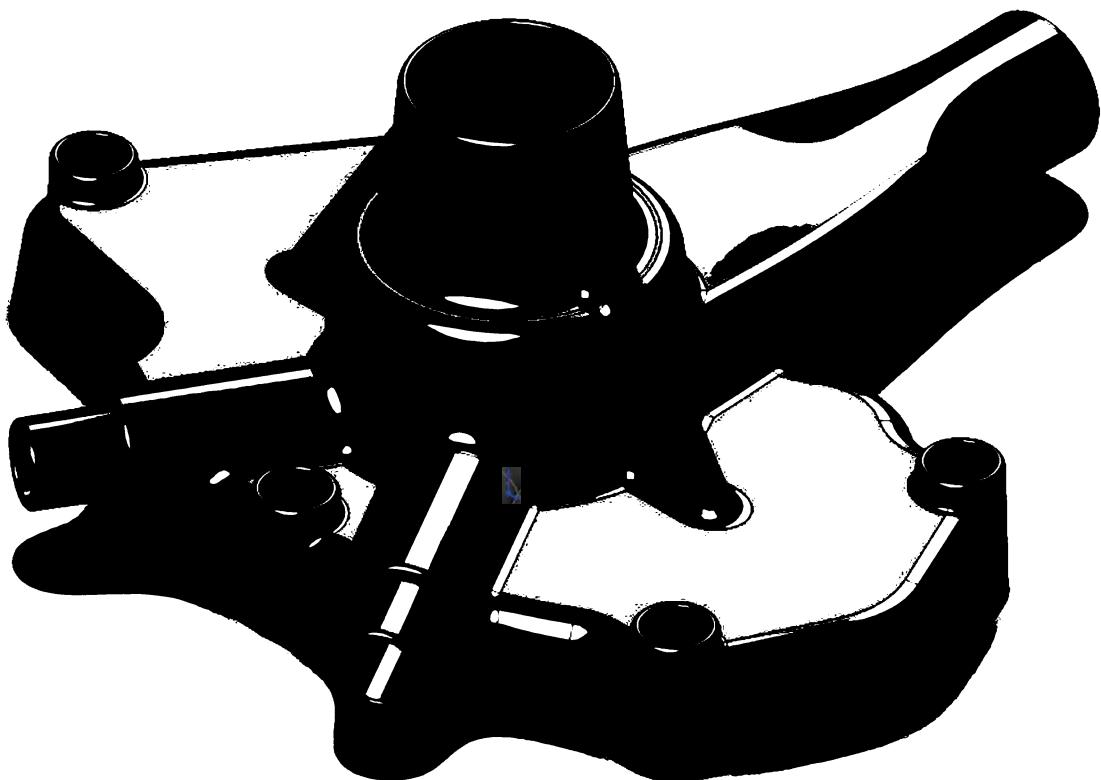
Click **OK.**

33. Saving your work:

Click **File / Save As.**

Enter **Water Pump Cover** as the name of the file.

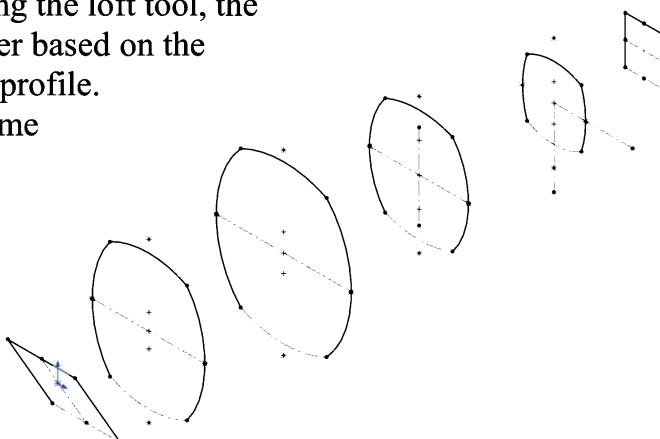
Click **Save.**



Exercise: Loft Without Guide Curves

When creating an advanced shape using the loft tool, the loft profiles are connected to each other based on the number of connectors created in each profile.

If the number of connectors are the same for each profile, a loft feature can be created without any twist issues.



1. Opening a part document:

Browse to the training folder and open a part document named:

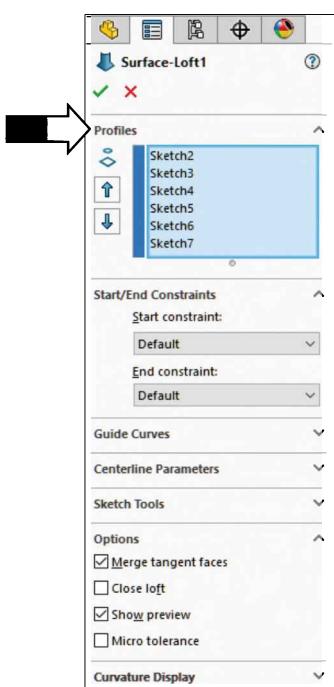
Loft Without Guide Curves.sldprt.

There are 6 loft profiles in this document and no guided curves.

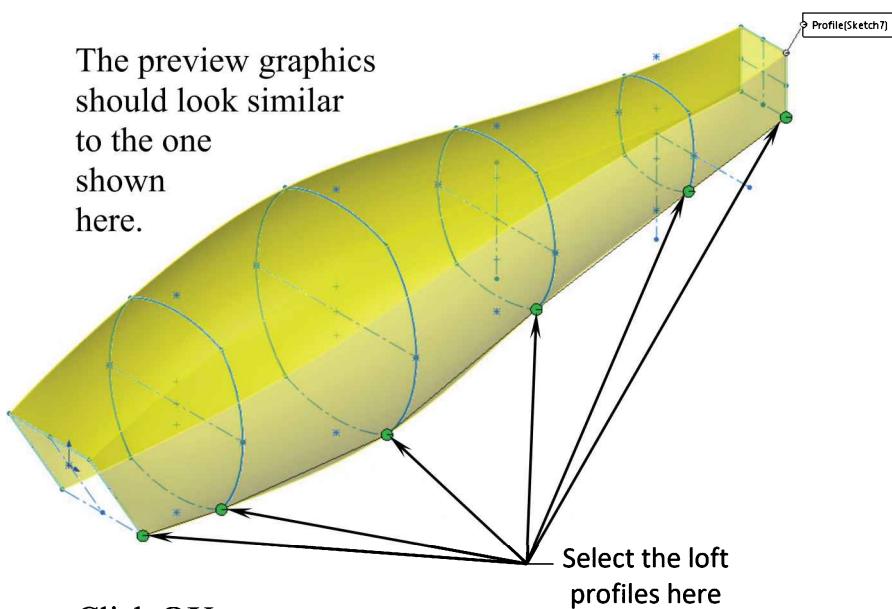
2. Creating a lofted surface:

Switch to the **Surfaces** tab and click **Lofted Surface**.

For Loft Profiles, select the **connectors** at the bottom right of each profile as indicated.



The preview graphics should look similar to the one shown here.



Click OK.

3. Running the Deviation Analysis:

Use Deviation Analysis to calculate the angle between faces.

Switch to the **Evaluate** tab and click: **Deviation Analysis**.

Drag the slider for the Number of Sample Points to about halfway.

Select the edge as indicated to examine.

Click **Calculate**.

The colored arrows display the amount of deviation. The results appear for the following criteria between the adjacent faces:

The maximum deviation error along the selected edge is **90.4°**.

The minimum deviation error along the selected edge is **15.18°**.

The average between the maximum and the minimum values along the selected edge is **35.88°**.



4. Saving your work:

Select File, Save As.

Enter: **Loft Without Guide Curves Completed** for the file name.

Click **Save**.

Close all documents.

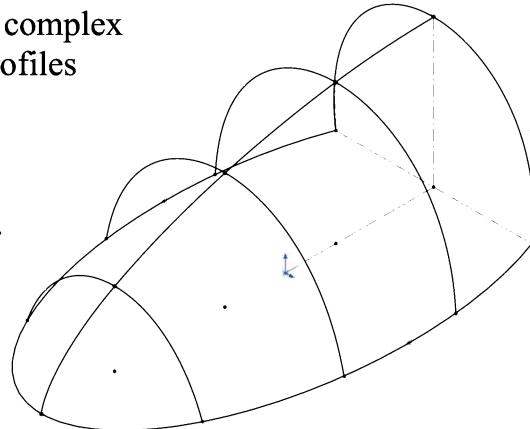
Exercise: Loft with Guide Curves

One of the recommended methods to create a complex shape more precisely is to use multiple loft profiles along with one or more guide curves.

This exercise will teach us how guide curves can help to create this model more accurately.

1. Opening a part document:

Browse to the training folder and open a part document named:
Loft With Guide Curves.sldprt.



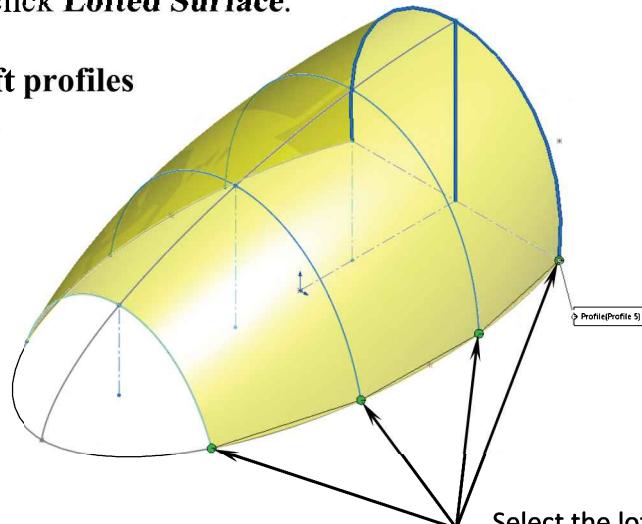
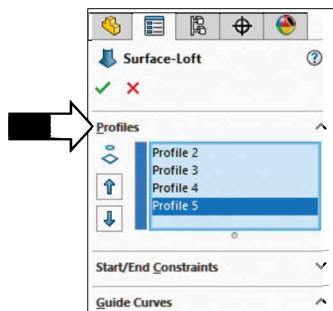
There are 5 loft profiles and 3 guides in this document. We will use them to create 2 different loft surfaces.

Focus on the use of the guide curves to see how they accurately control the transitions between the loft profiles and how the 2 surfaces are perfectly blended when Tangency is applied to them.

2. Creating the 1st lofted surface:

Switch to the **Surfaces** tab and click **Lofted Surface**.

For Loft Profiles, select the **4 loft profiles** in the graphics area as indicated.

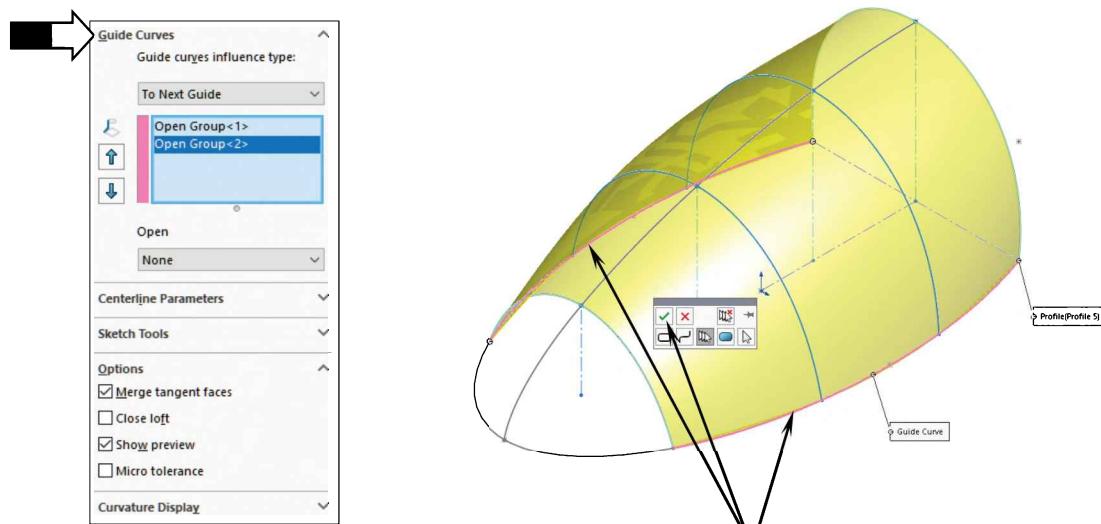


When selecting the profiles, SOLIDWORKS automatically selects the nearest endpoint (connector) in that profile.

Select the loft profiles here

Expand the Guide Curves section.

Select the 1st Guide Curve on the right. The SelectionManager dialog box pops up, click **OK** in the SelectionManager to accept the 1st guide.



Select a Guide Curve
and click OK in the
SelectionManager

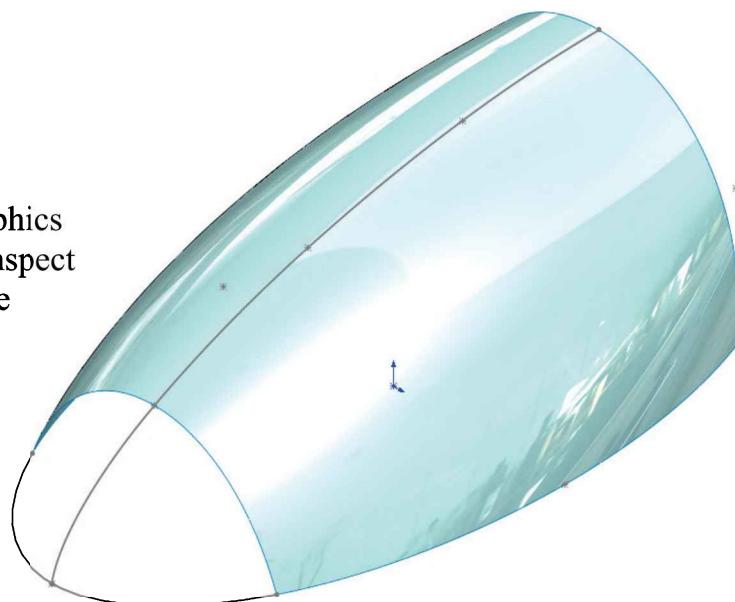
Select the 2nd Guide Curve on the left and click
OK in the SelectionManager dialog box to
accept the 2nd guide.

The preview graphics shows a high-quality lofted surface is
being created from 4 loft profiles and 2 guide curves.

Click **OK**.

Enable the RealView Graphics option (if available) and inspect your model against the one shown here.

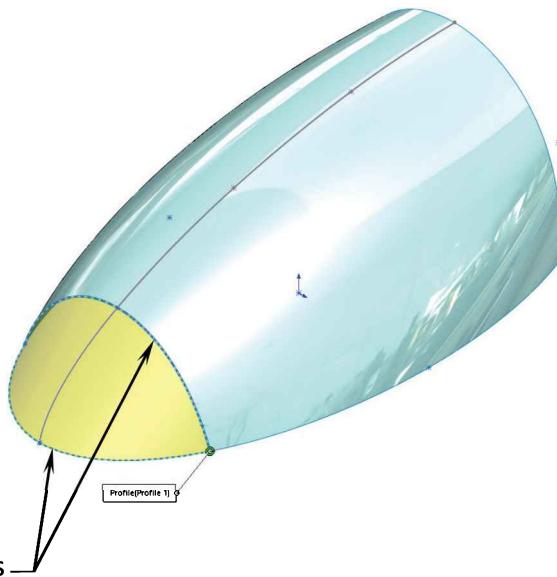
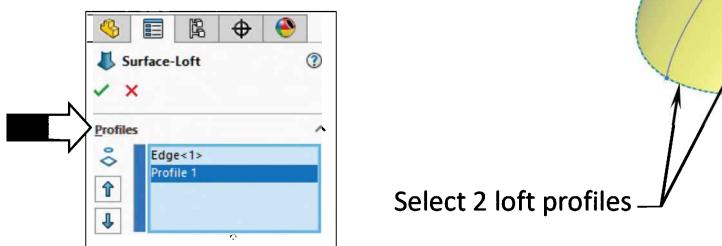
The other sketches
will be used to create
the 2nd lofted surface
in the next step.



3. Creating the 2nd lofted surface:

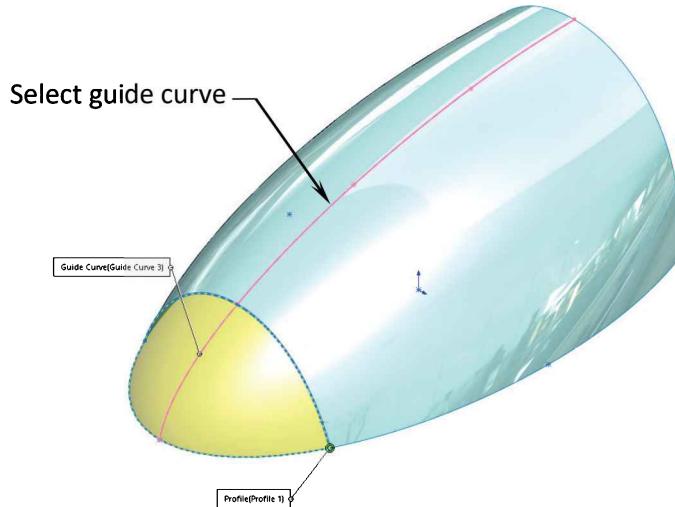
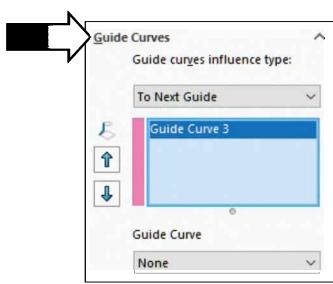
Click **Lofted Surface** again.

For Loft Profiles, select the **sketch** and the model **edge** as indicated.



Expand the Guide Curves section.

For Guide Curves, select the **sketch** in the center of the model as noted.

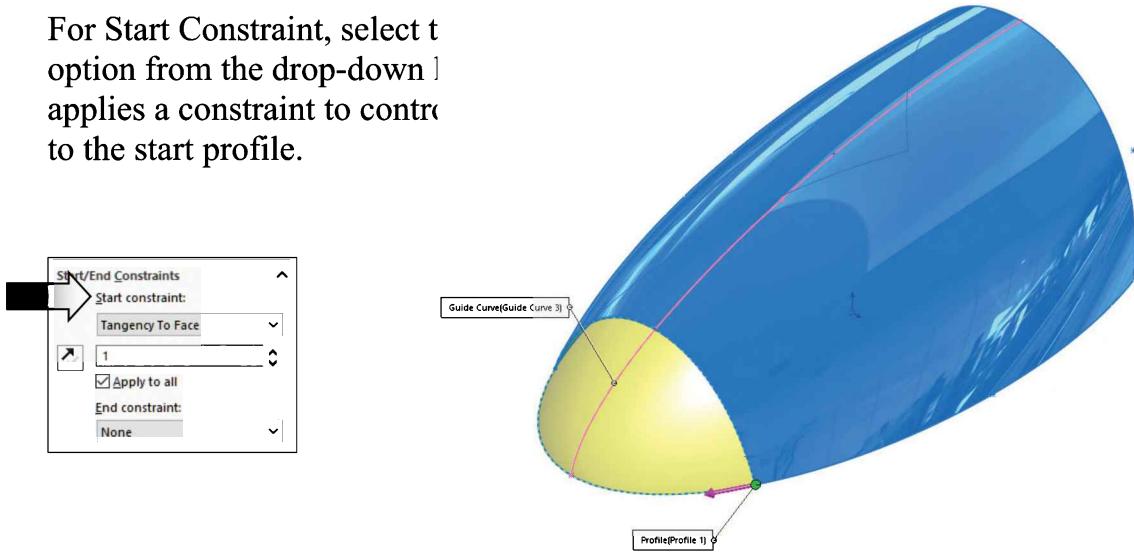


The preview graphics shows a high-quality surface being generated from 2 loft profiles and 1 guide curve.

This surface is acceptable in most cases, but to improve the blend between the 2 surfaces even further, let us take a look at a Tangent blend option in the Start and End Constraint section.

Expand the **Start / End Constraints** section.

For Start Constraint, select the **Tangency To Face** option from the drop-down menu. This applies a constraint to control the start profile.



Leave the End Constraint at **None**.

Click **OK**.

Inspect your model against the one shown below.

4. Saving your work:

Select **File, Save As**.

Enter: **Loft With Guide Curves_Completed** for the file name.

Click **Save**.

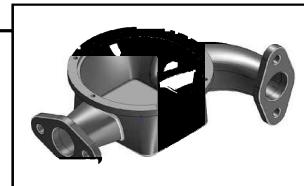
Close all documents.

CHAPTER 6

Loft Vs. Sweep

Loft vs. Sweep

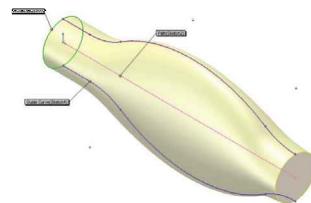
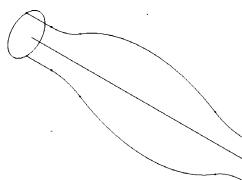
Water Meter Housing



Both Loft and the Sweep commands are usually used to create advanced, complex shapes. The differences between the two are:

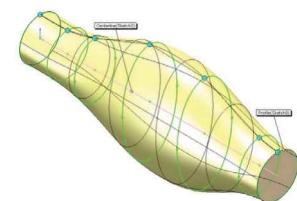
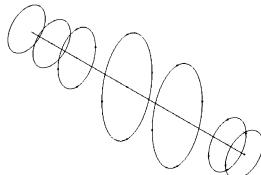
- Sweep uses a single sketched profile to sweep along a path and is controlled by one or more guide curves.

Sweep uses one profile, one path and multiple guide curves



- Loft uses multiple sketched profiles to loft between the sections and is controlled by 1 or more guide curves or 1 Centerline Parameter.

Loft uses multiple profiles, one centerline parameter, and multiple guide curves



In order to create a solid feature, each sketch profile must be a single, closed, and non-intersecting shape.

The guide curves can be either a 2D sketch or a 3D curve, open or closed.

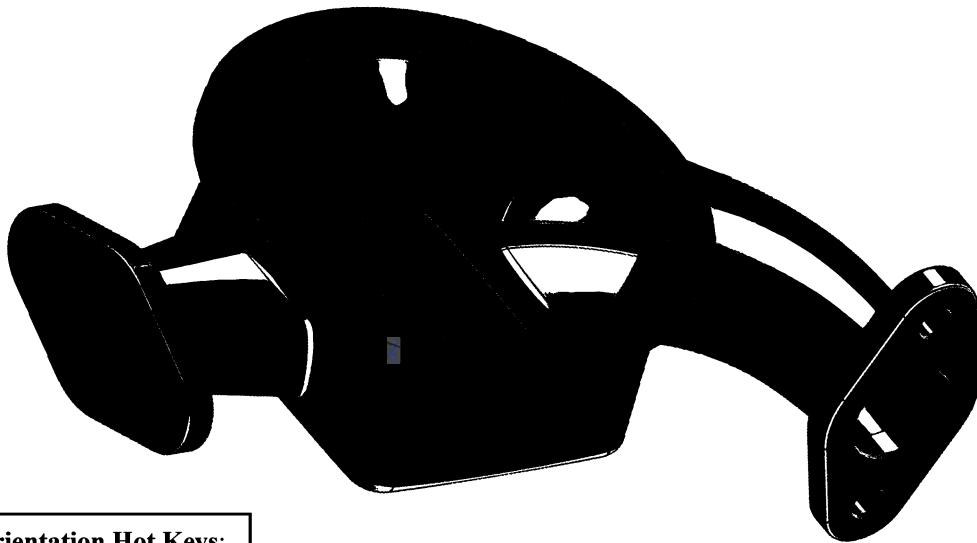
The sweep path and guide curves must be related to the sketch profiles with either a Coincident or Pierce relation.

The loft profiles should have the same number of entities or segments.

The Split-Entities  commands can be used to split the sketch entities and add the necessary connectors to help control the loft more accurately.

Loft vs. Sweep

Water Meter Housing



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI

Units: INCHES – 3 Decimals

Tools Needed:



Insert Sketch



Line



Split Entities



Sketch Fillet



Mirror



Add Geometric Relations



Dimension



Base/Boss Extrude



Extruded Cut



Plane



Base/Boss Sweep



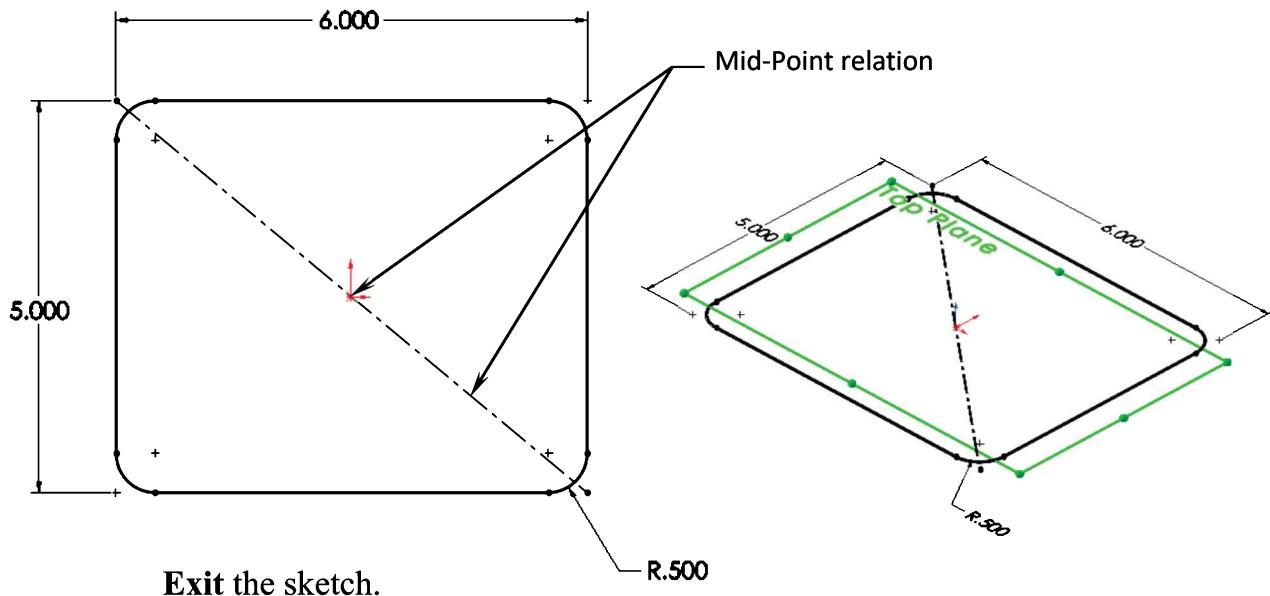
Base/Boss Loft

1. Sketching the 1st Loft Profile:

Select the Top plane and open a **new sketch** .

Sketch a **Rectangle** centered on the Origin and add the dimensions shown.

(Note: Only add the corner fillets after the sketch is fully defined.)



Exit the sketch.

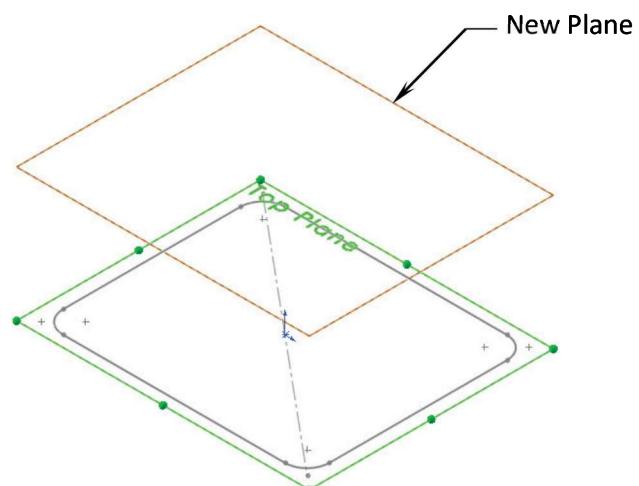
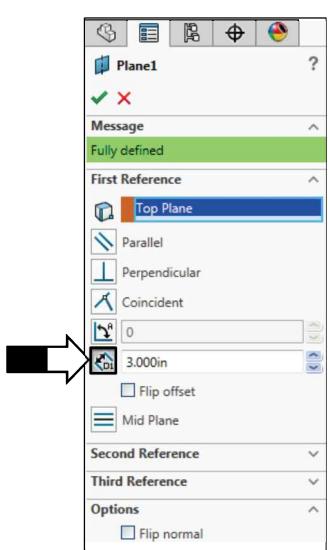
2. Creating a new plane:

Click  or select **Insert / Reference Geometry / Plane**.

Select the Top reference plane from the FeatureManager tree.

Select the **Offset Distance** option and enter **3.000in**.

Click **OK**.

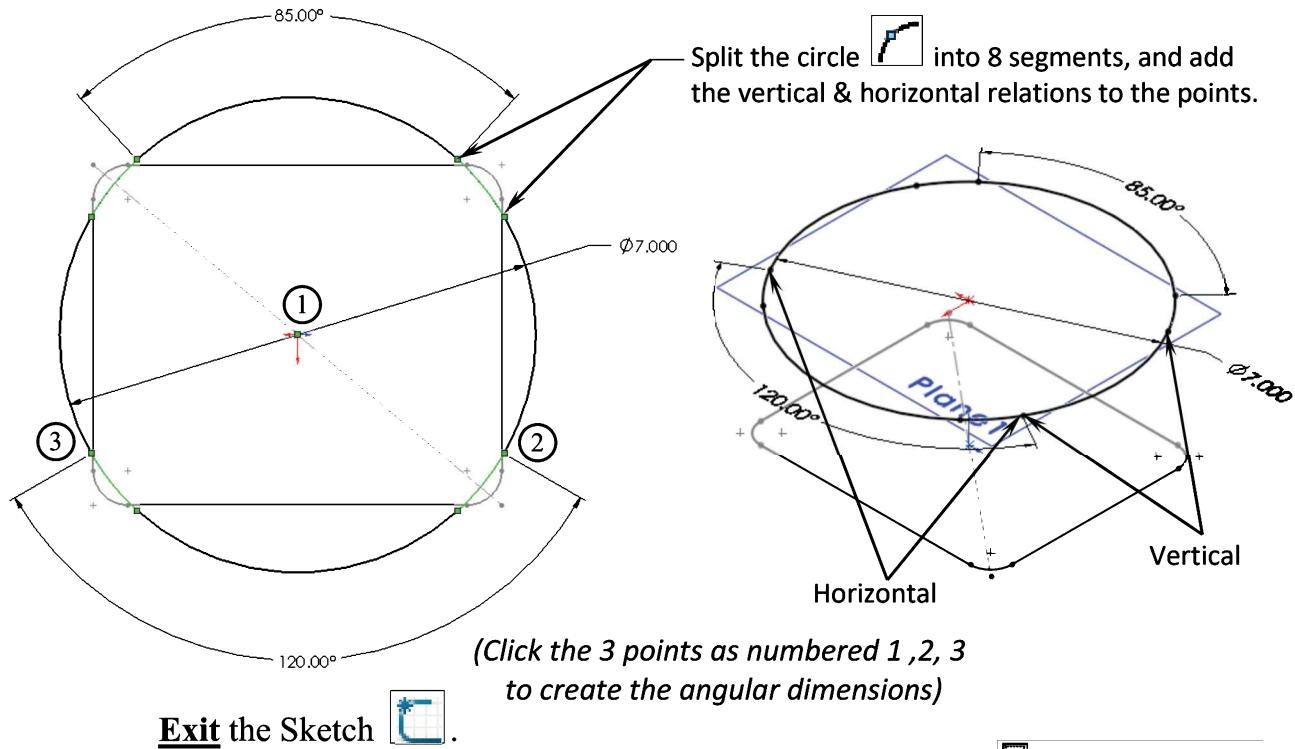


3. Sketching the 2nd Loft Profile:

Select the new plane (**Plane1**) and open a **new sketch** .

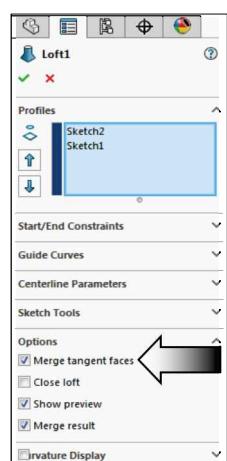
Sketch a **Circle** and split it into 8 segments (Tools / Sketch Tools / Split Entities).

Add dimensions and the vertical / horizontal relations between the split points.



4. Creating the Lofted Base feature:

Click  or select **Insert / Boss-Base / Loft**.

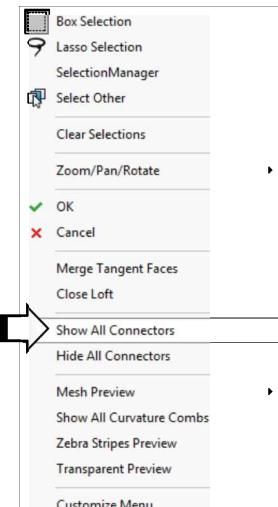
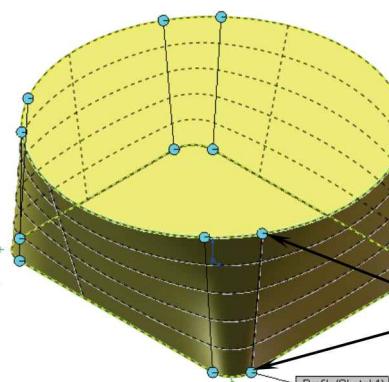


For Loft Profiles, select the **2 sketches** in the graphics area.

Show all connectors as noted.

Enable **Merge Tangent Faces**.

Click **OK**.

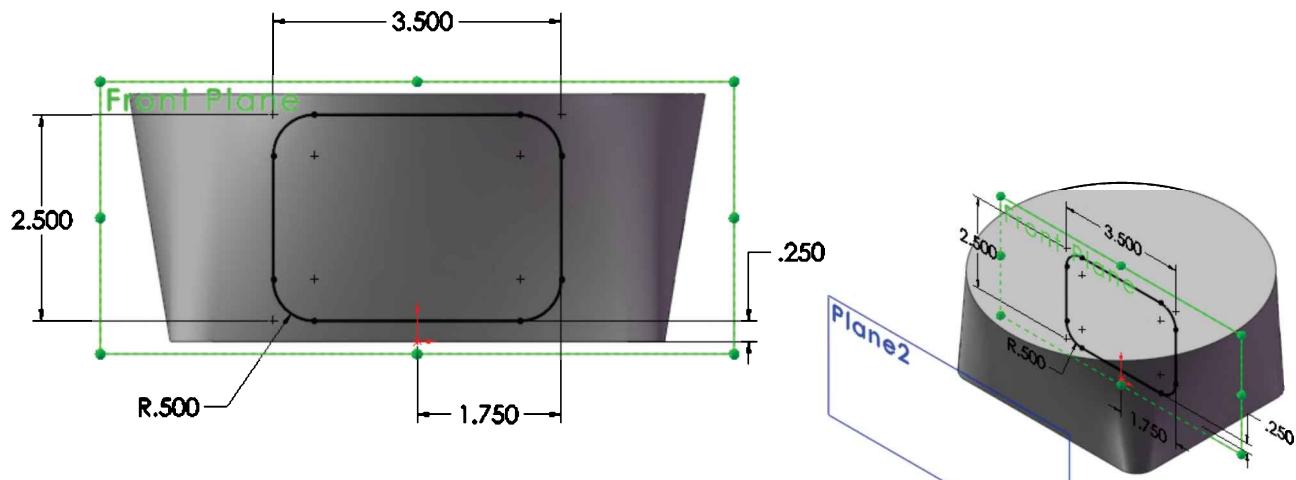


Right-click and pick:
Show All Connectors

5. Constructing the Inlet's 1st Loft Profile:

Select the Front plane and open a new sketch .

Sketch a **Rectangle**, add the dimensions shown below. Only add the sketch fillets after the sketch is fully defined.



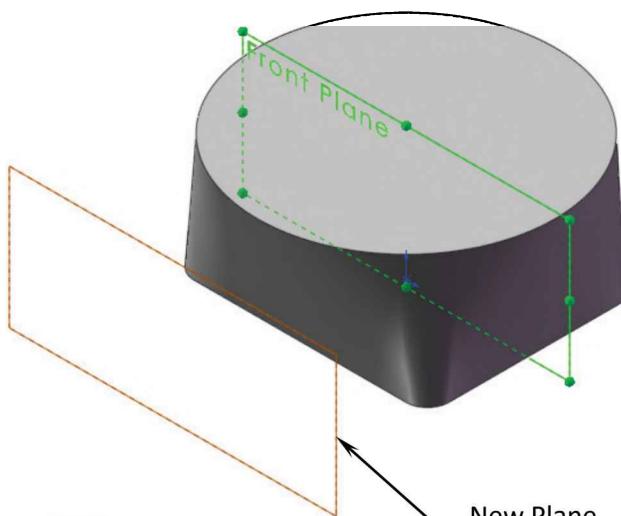
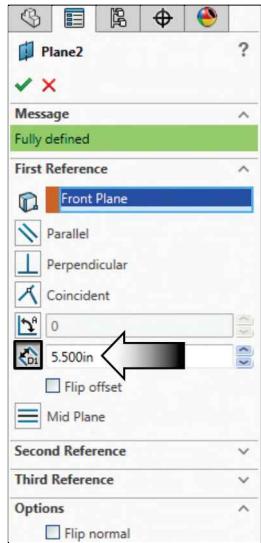
6. Creating an Offset Distance plane at 5.500":

 Click  or select **Insert / Reference Geometry / Plane**.

Select the **Front** reference plane from the FeatureManager tree.

Select **Offset Distance** option and enter **5.500in.** for distance.

Click **OK**.



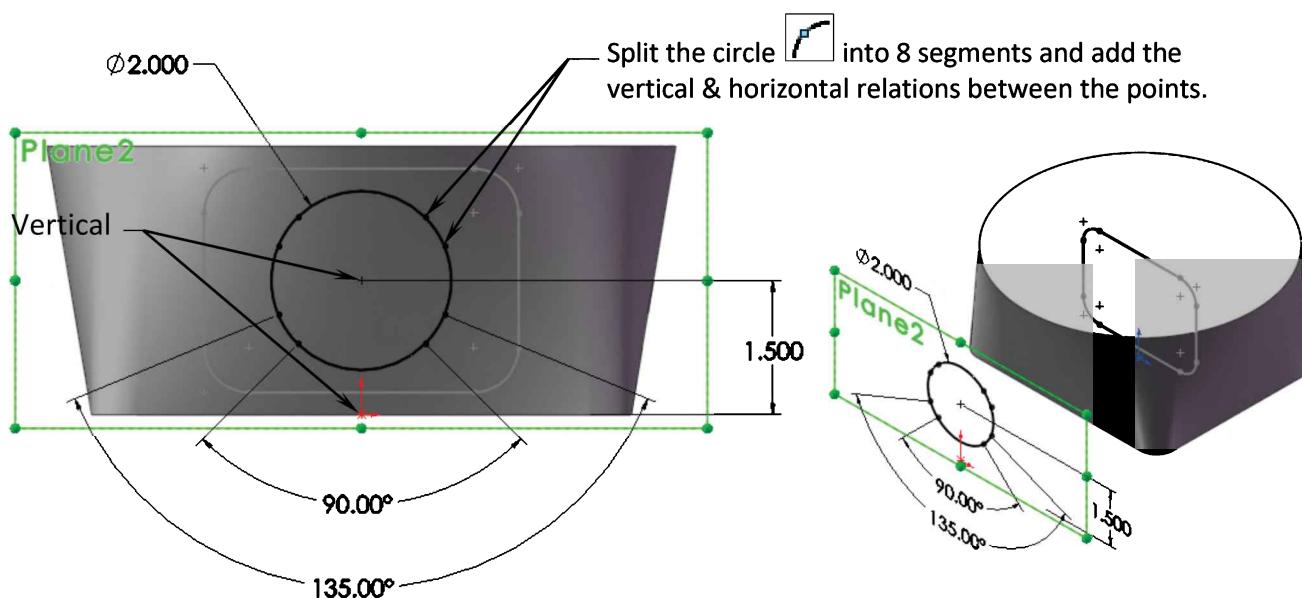
Exit the Sketch .

7. Constructing the Inlet's 2nd Loft Profile:

Select Plane2 and open a new sketch .

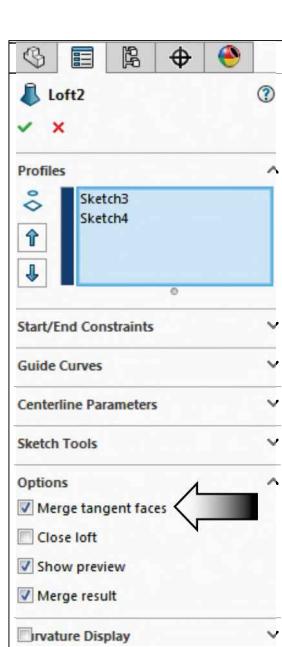
Sketch a **Circle** and add dimensions to fully define it.

Click **Split Entities**  and split the circle into 8 segments (Tools, Sketch Tools, Split Entities).



Exit the Sketch .

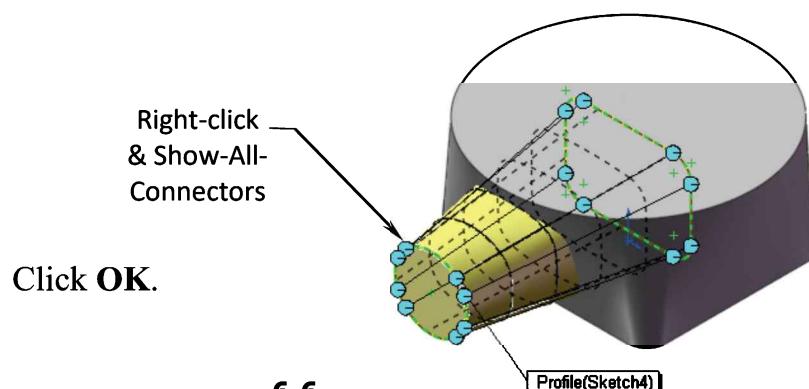
8. Creating the Inlet feature:



Click  or select **Insert / Boss-Base / Loft**.

For profiles, select the **2 sketches** from the graphics area.

Show all connector points to ensure a straight transition between the 2 profiles. Enable the **Merge Tangent Faces** option.



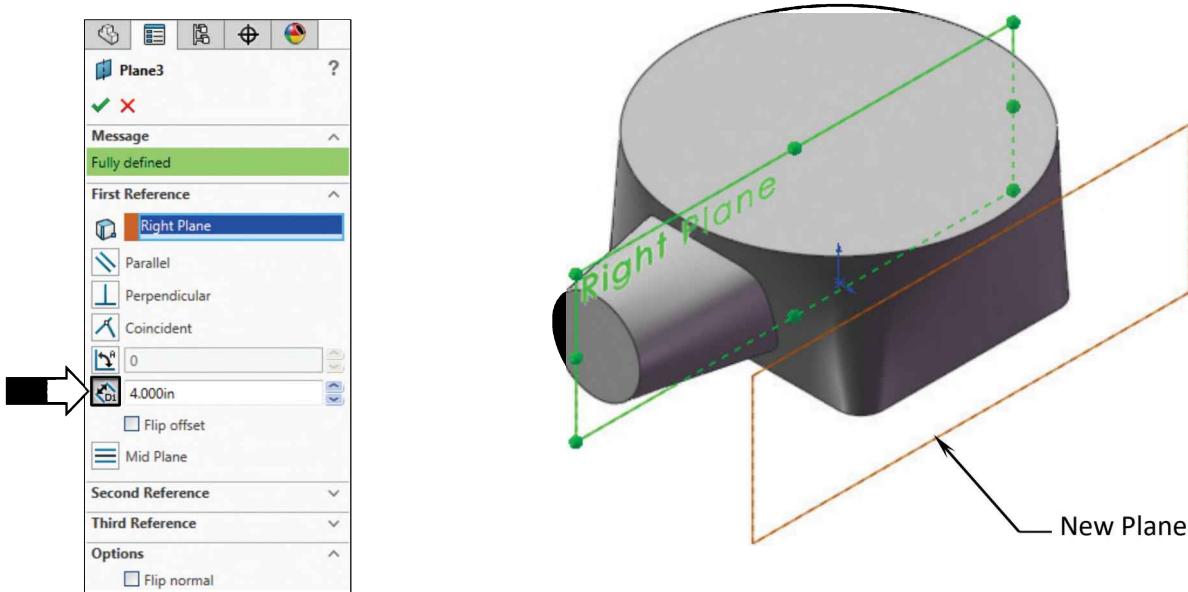
9. Creating an Offset Distance plane at 4.00":

Click  or select Insert / Reference Geometry / Plane.

Select the **Right** reference plane from the FeatureManager tree.

Select **Offset Distance** and enter **4.000in.** for distance and place it on the right.

Click **OK**.

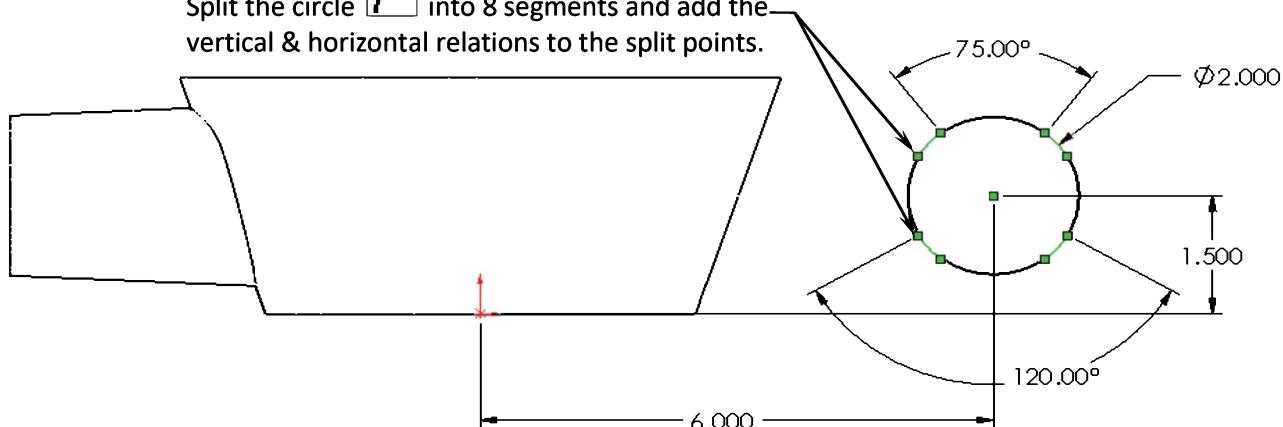


10. Constructing the Outlet's 1st Loft profile:

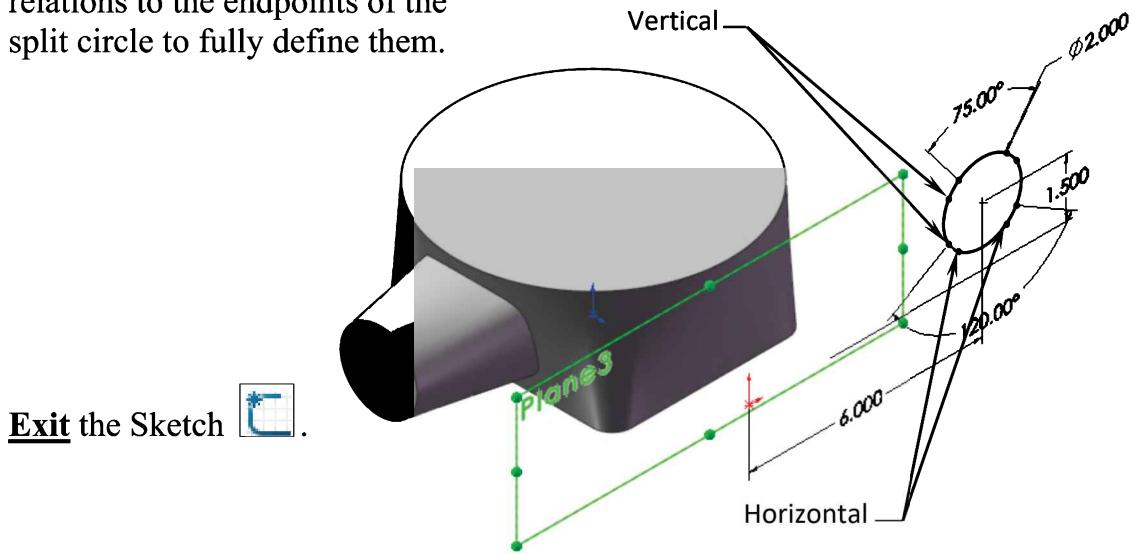
Select the new plane (Plane3) and open a new sketch .

Sketch a **Circle** and split it  into 8 segments. Add the dimensions as noted.

Split the circle  into 8 segments and add the vertical & horizontal relations to the split points.



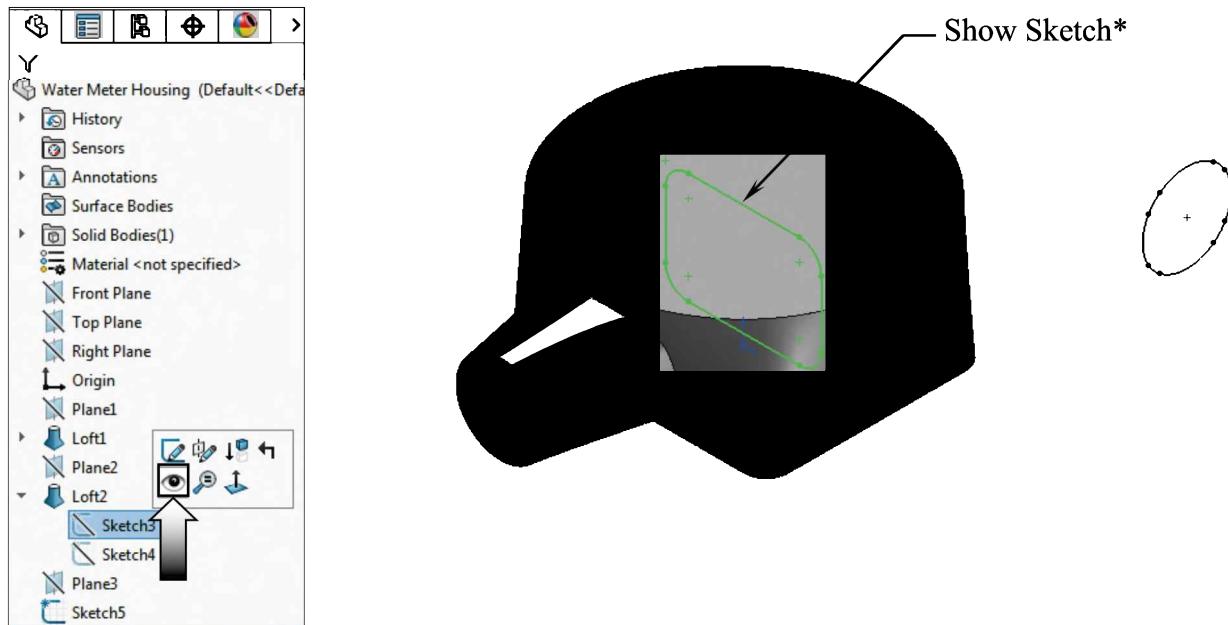
Add the **Vertical** and **Horizontal** relations to the endpoints of the split circle to fully define them.



11. Showing the previous sketch*:

Expand the feature **Loft2** from the FeatureManager tree (click the + sign).

Locate **sketch3** (the sketch of the Rectangle), click it and select **Show**



* The previous sketch is now visible in the graphics area.
It will be used again in the next loft operation.

12. Creating a Parallel plane:

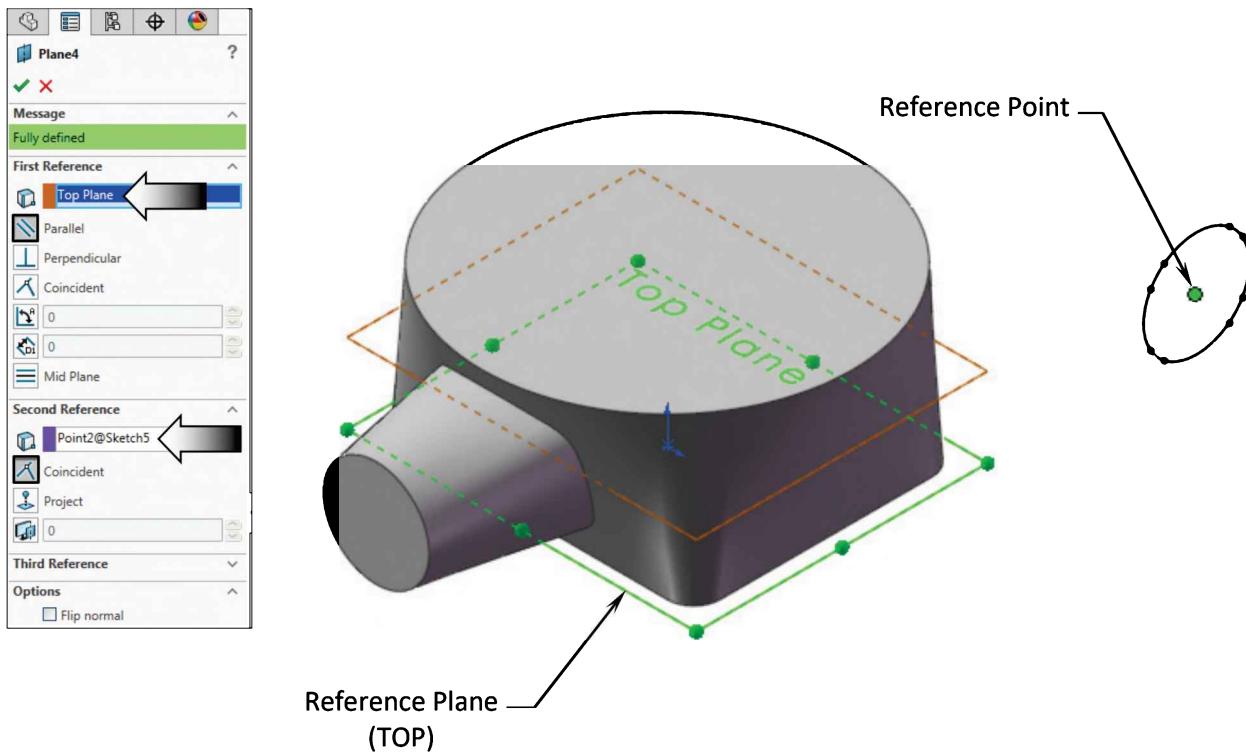
Click  or select Insert / Reference Geometry / Plane.

For 1st Reference, select the **Top** reference plane from the FeatureManager tree.

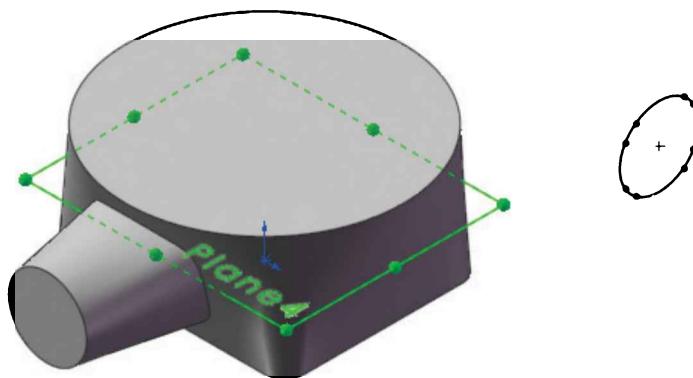
Click the **Parallel** option (arrow).

For 2nd Reference, click the **Center Point** of the circle as noted.

Click **OK**.



The new plane is created (Plane4).



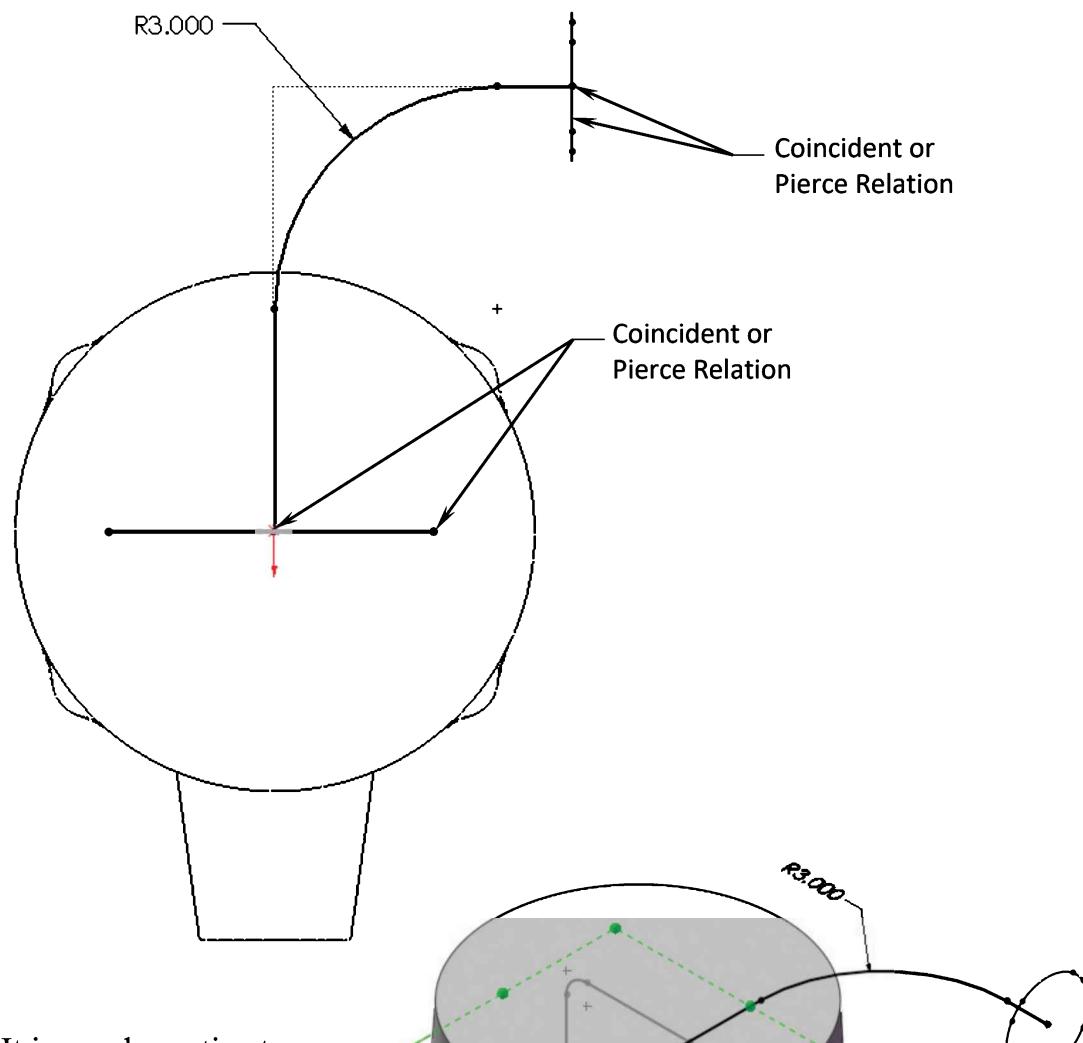
13. Constructing the Centerline Parameter:

Select the new plane (Plane4) and open a **new sketch** .

Switch to the **Top** view orientation  (Ctrl+5).

Sketch 2 Lines and add a **Sketch Fillet** as shown below.

The end points of the lines must be coincident or Pierced to the centers of the other sketches.



Note: It is good practice to make the sketch fully defined before adding the fillets.

Exit the Sketch .

14. Creating the Outlet loft feature:

Click  or select Insert / Boss / Loft.

Select the 2 Sketch Profiles (Rectangle and Circle) from the graphics area.

Expand the **Centerline Parameters** option and select **Sketch6** from either the graphics area or from the FeatureManager tree.

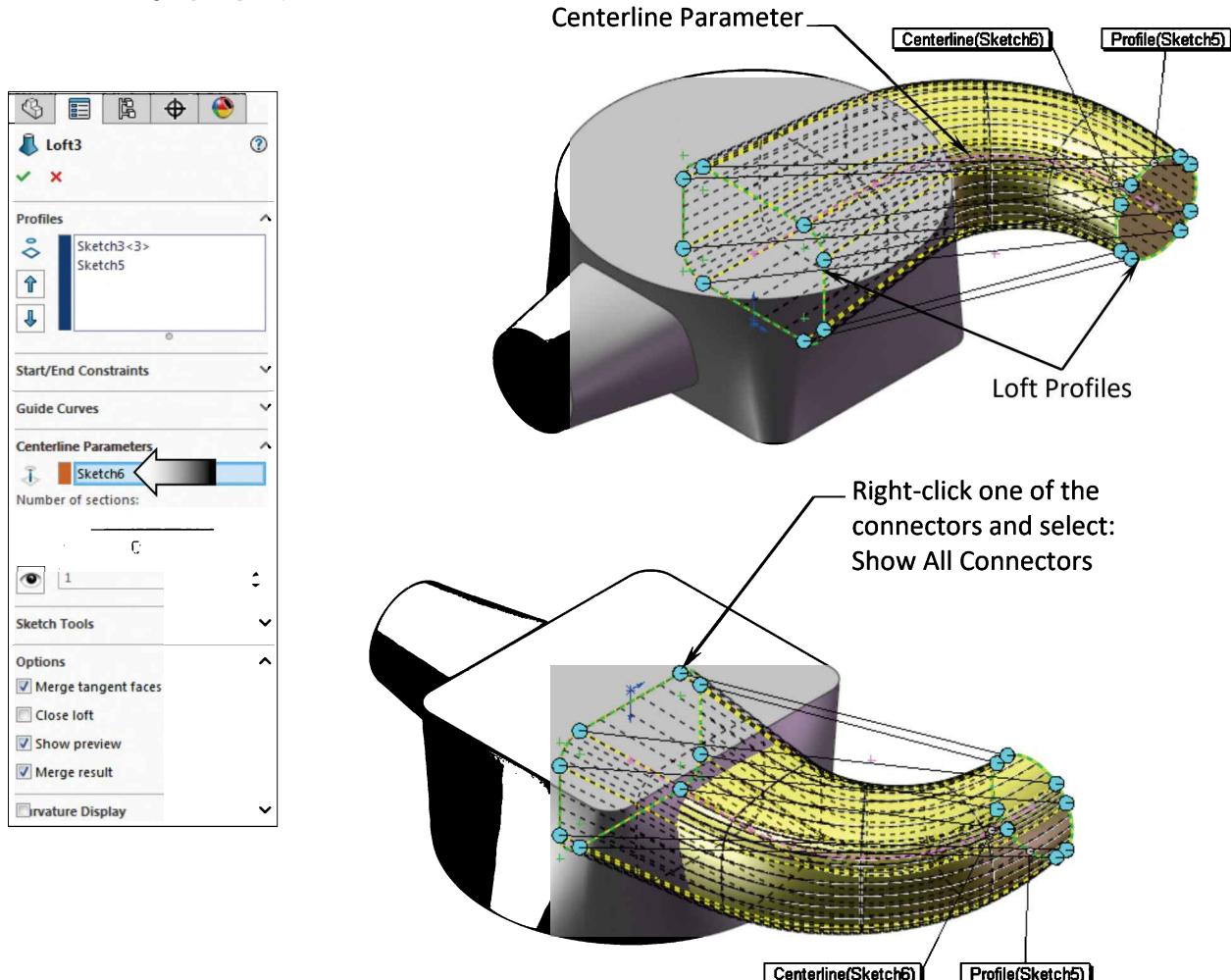
When the Centerline Parameter is used to guide the loft, the sketch planes of all the intermediate sections will be rotated normal to the centerline.

After the preview appears, check the connectors to ensure a proper loft transition.

Centerline Parameters

If the number of entities in each sketch is the same, a “Centerline Parameter Sketch” can be used instead of the guide curves.

Click **OK**.

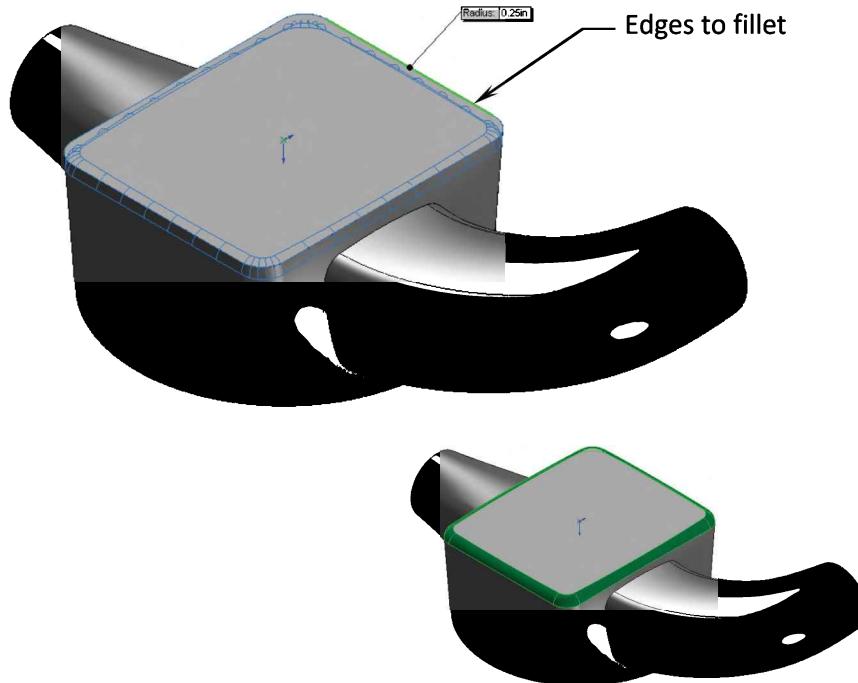
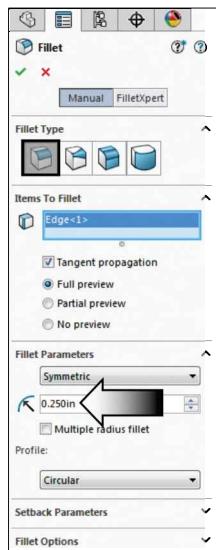


15. Adding .250" fillet to the Bottom:

Click  or select Insert / Features / Fillet-Round.

Enter **.250in.** for radius and select the **bottom edges** as indicated.

Click **OK**.

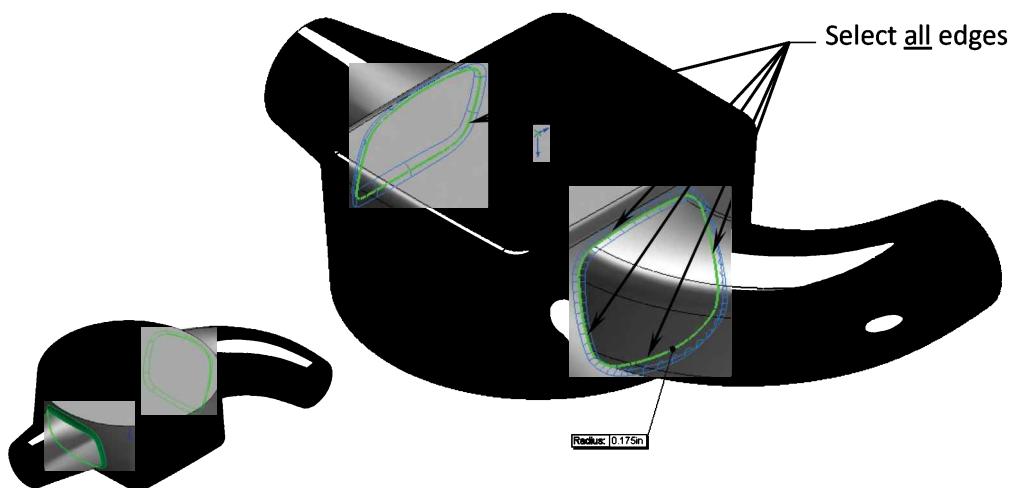
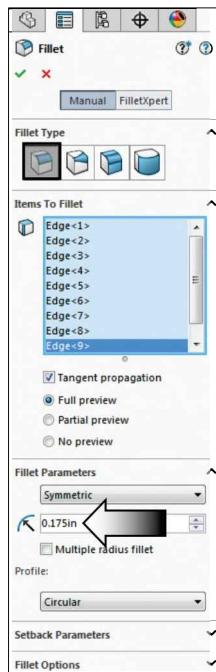


16. Add .175" fillets to the sides:

Click  or select Insert / Features / Fillet-Round.

Enter **.175in.** for Radius and select all side-edges as noted.

Click **OK**.



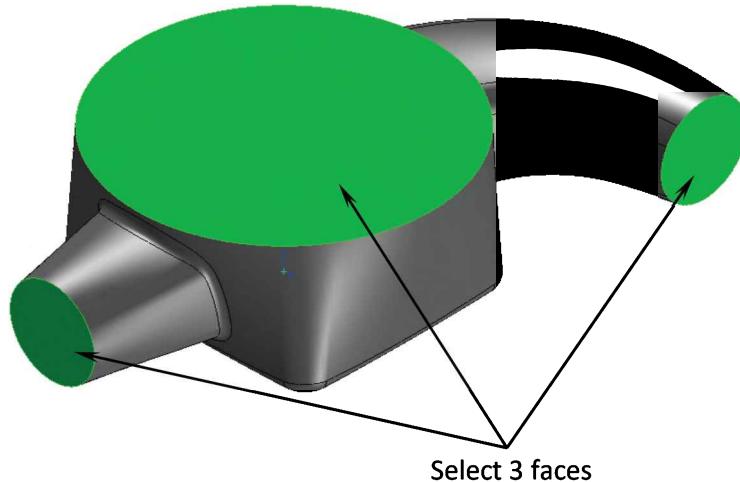
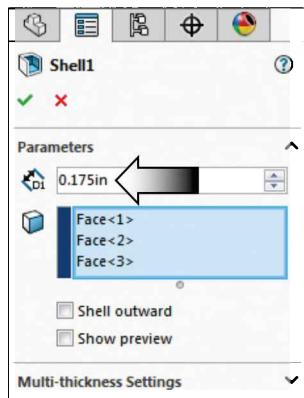
17. Shelling the part:

Click  or select **Insert / Features / Shell**.



Enter **.175in.** for wall thickness and select the **3 faces** as noted.

Click **OK**.

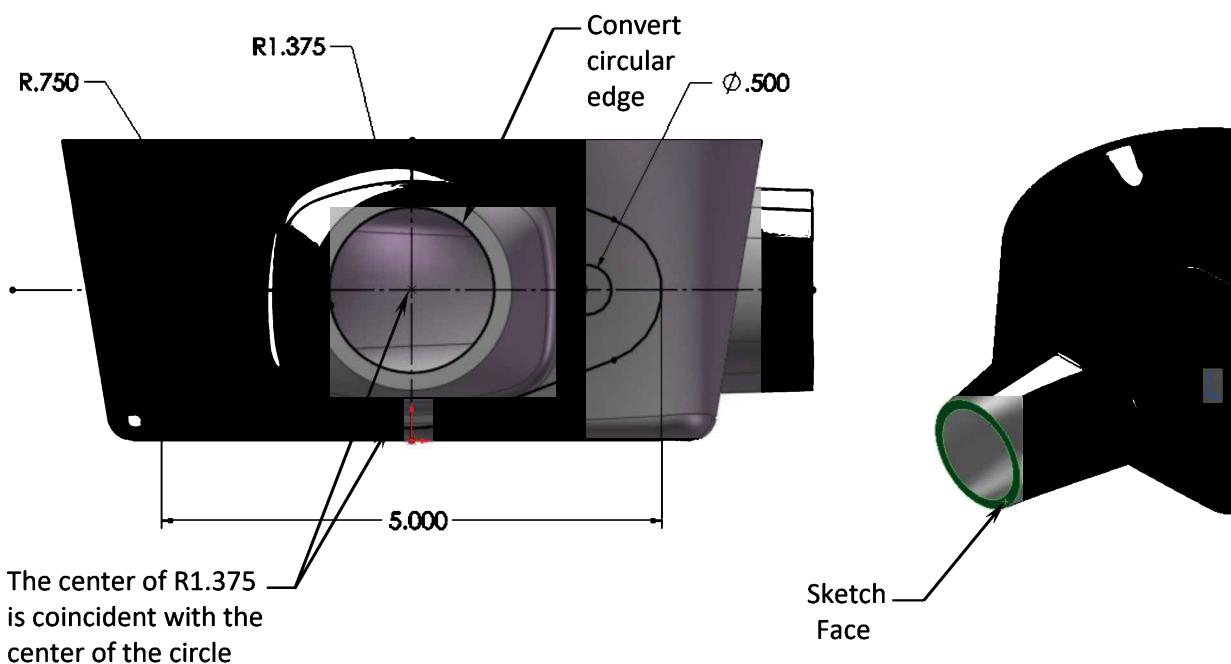


18. Creating the 1st Mounting bracket:

Select the Face as noted and open a **new sketch** .

Sketch the profile shown below; use **Convert Entities**  where applicable.

Add dimensions and relations needed to fully define the sketch.

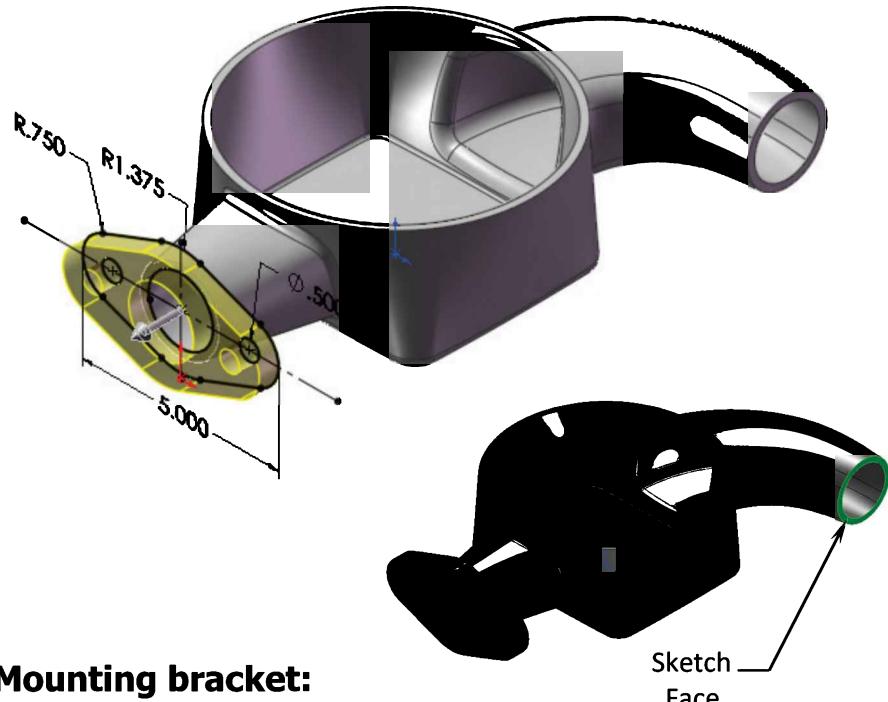
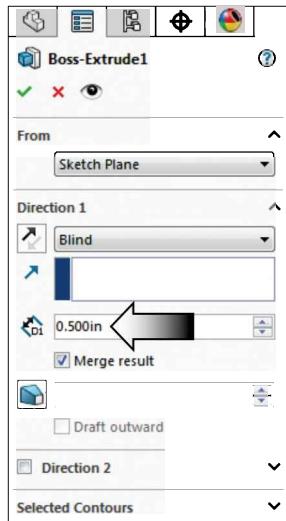


19. Extruding the Left bracket:

Click  or select Insert / Boss-Base / Extrude.

End Condition: **Blind** and Depth: **.500in**.

Click **OK**.

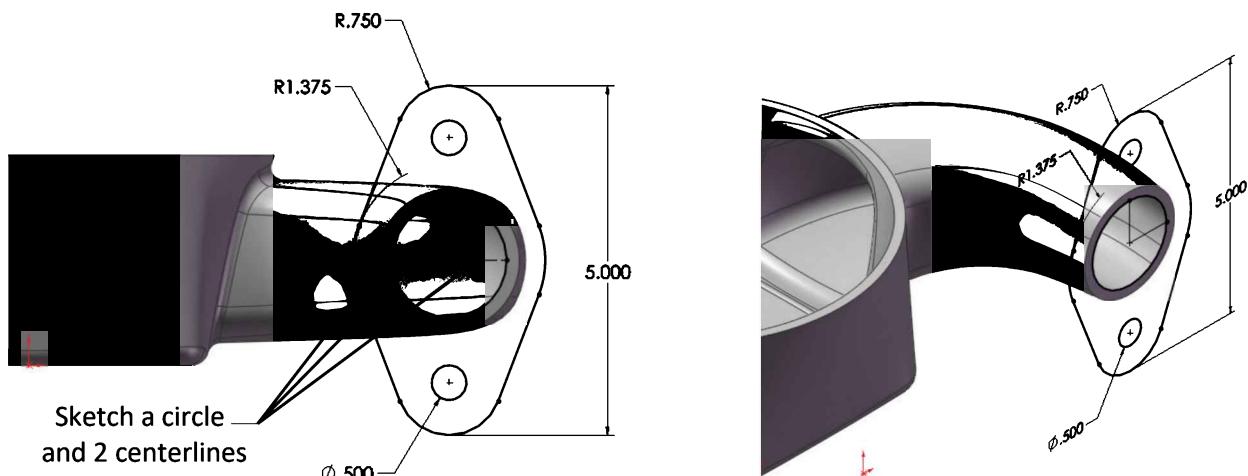


20. Creating the 2nd Mounting bracket:

Select the Face as noted and open a new sketch .

Either copy the previous sketch – or – recreate the same sketch again on the right side. (Derived-Sketch is also a good option to make the 2nd bracket.)

Add the dimensions and relations needed to fully define the sketch.

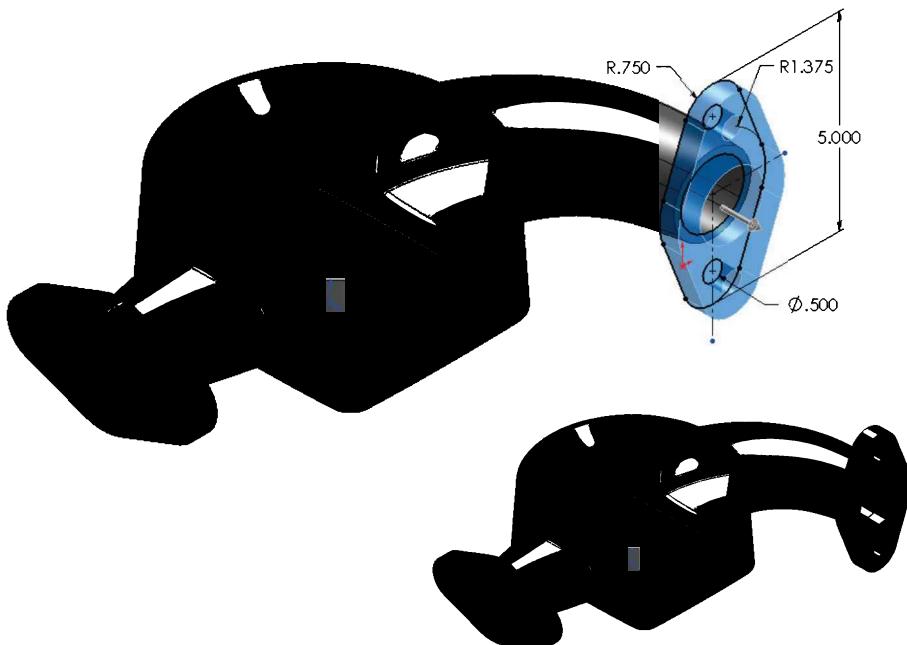
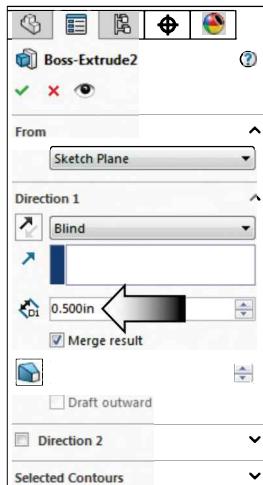


21. Extruding the Right bracket:

Click  or select Insert / Boss-Base / Extrude.

End Condition: **Blind**. Depth: **.500in**.

Click **OK**.

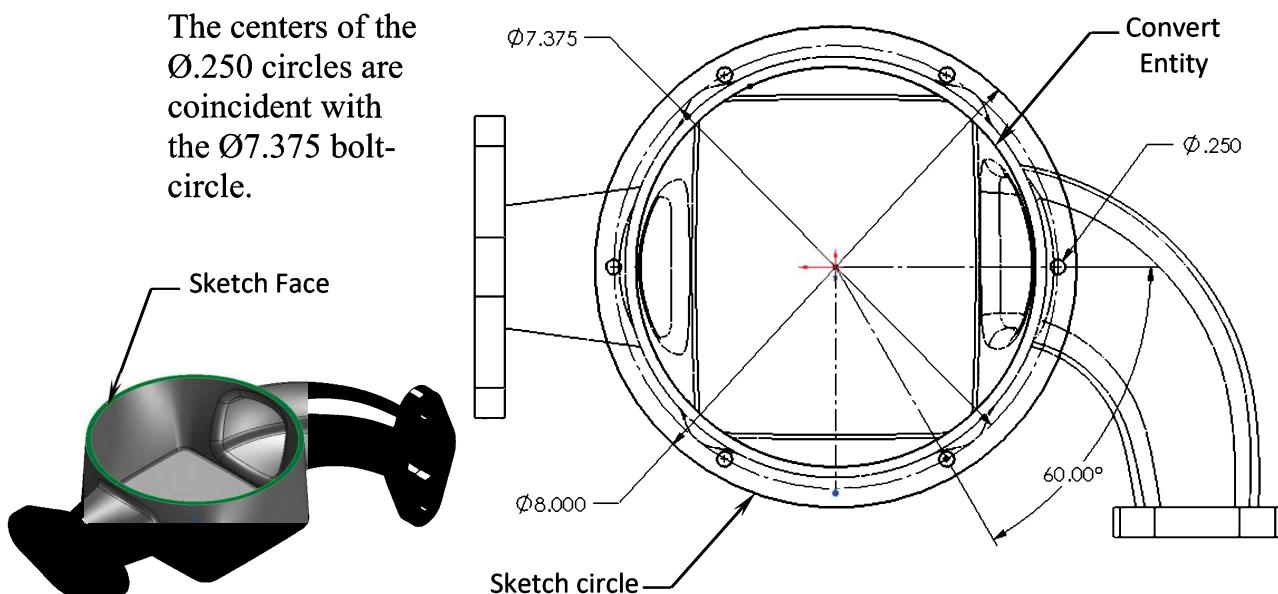


22. Constructing the Upper Ring:

Select the Face as indicated and open a new sketch .

Sketch the profile below; use **Convert Entities**  where needed.

The centers of the $\varnothing 0.250$ circles are coincident with the $\varnothing 7.375$ bolt-circle.

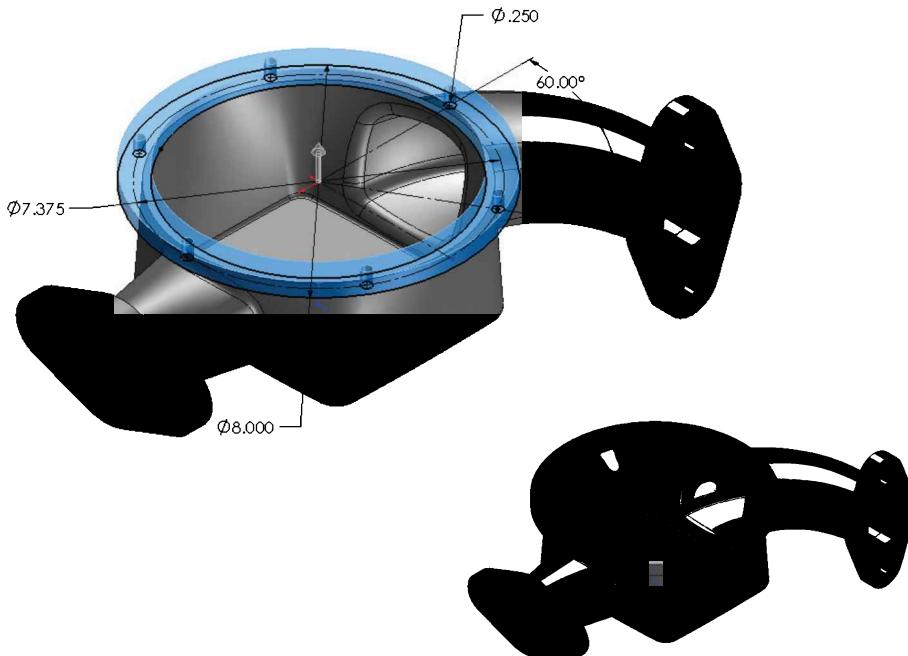
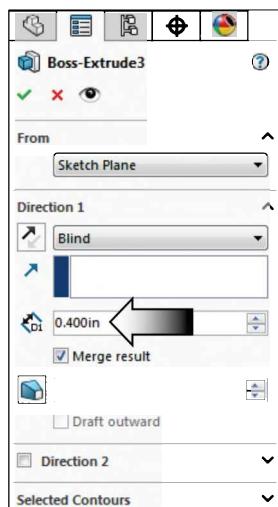


23. Extruding the Upper Ring:

Click  or select Insert / Boss-Base / Extrude.

End Condition: **Blind**. Depth: **.400in.** (upward)

Click **OK**.



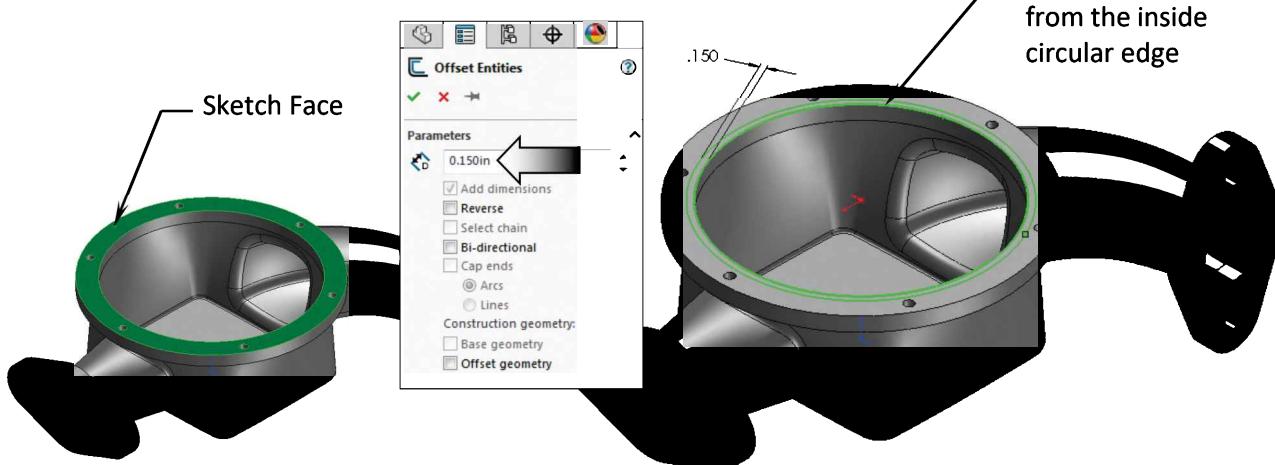
24. Adding a Seal-Ring bore:

Select the Face as noted and open a new sketch .

Select the inside circular edge and click **Offset Entities** .

Enter **.150in.** for Offset Value (larger diameter).

Click **OK**.



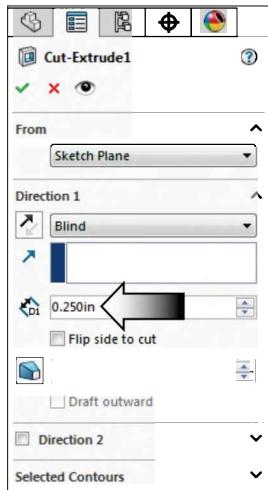
25. Cutting the Seal Ring bore:

Click  or select Insert / Cut / Extrude.

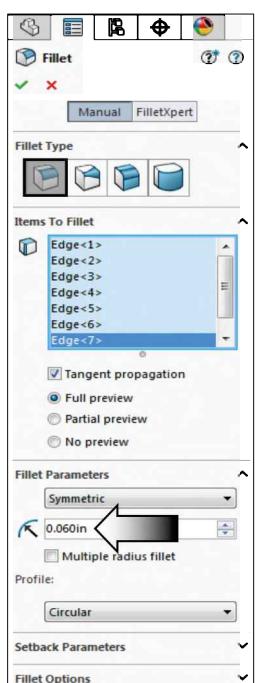
End Condition: **Blind**.

Depth: **.250in**.

Click **OK**.



26. Adding fillets:

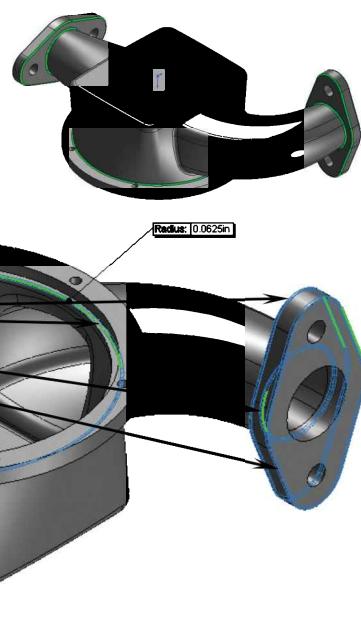


Click  or select Insert / Features / Fillet-Round.

Enter **.060in** for radius value.

Select the **7 edges** as noted.

Click **OK**.



27. Adding Chamfers:

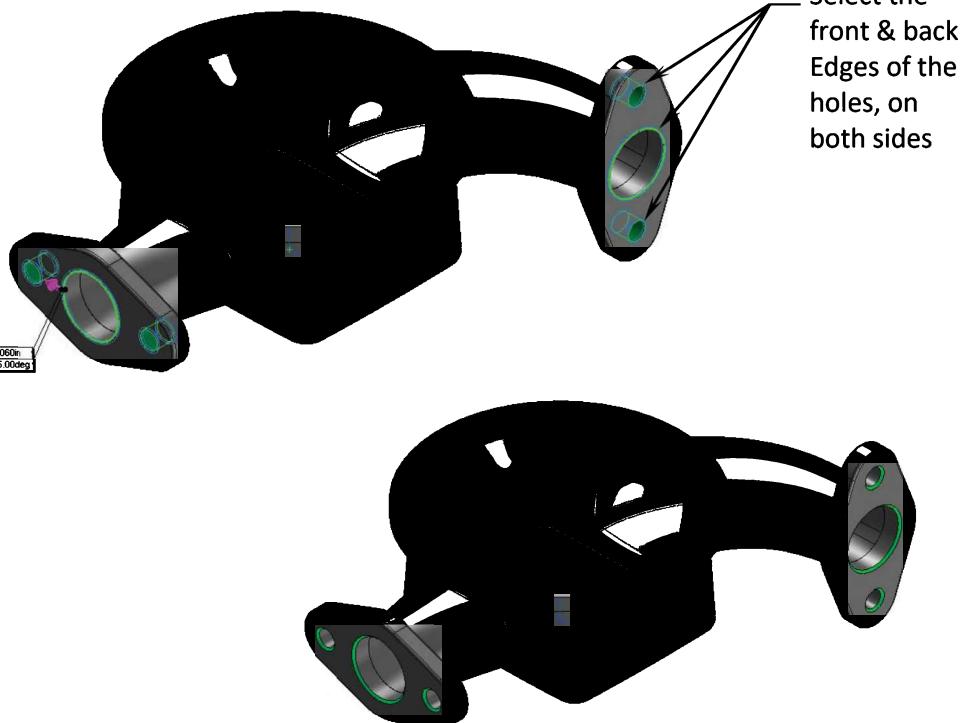
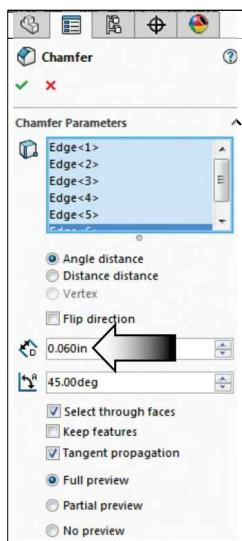
Click  or select Insert / Features / Chamfer.

Enter **.060in.** for Depth.

Enter **45 deg.** for Angle.

Select the **edges** of the 6 holes.

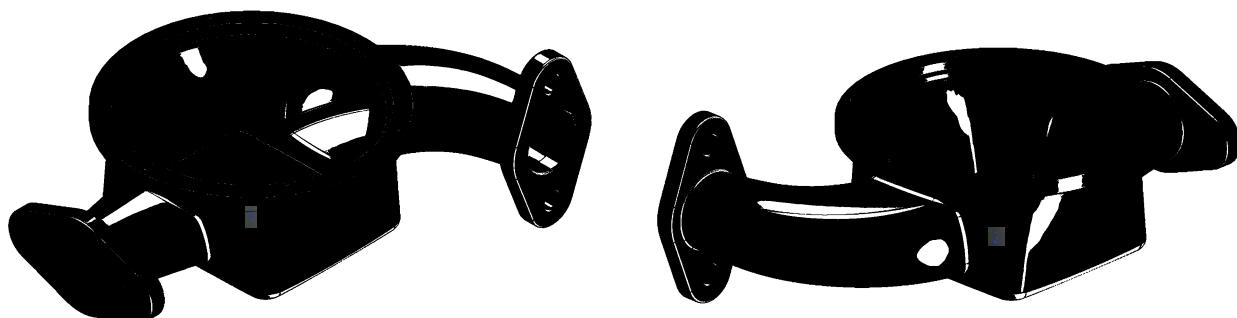
Click **OK**.



Select the front & back Edges of the holes, on both sides

28. Saving your work:

Select File / Save As / Water Meter Housing / Save.



Questions for Review

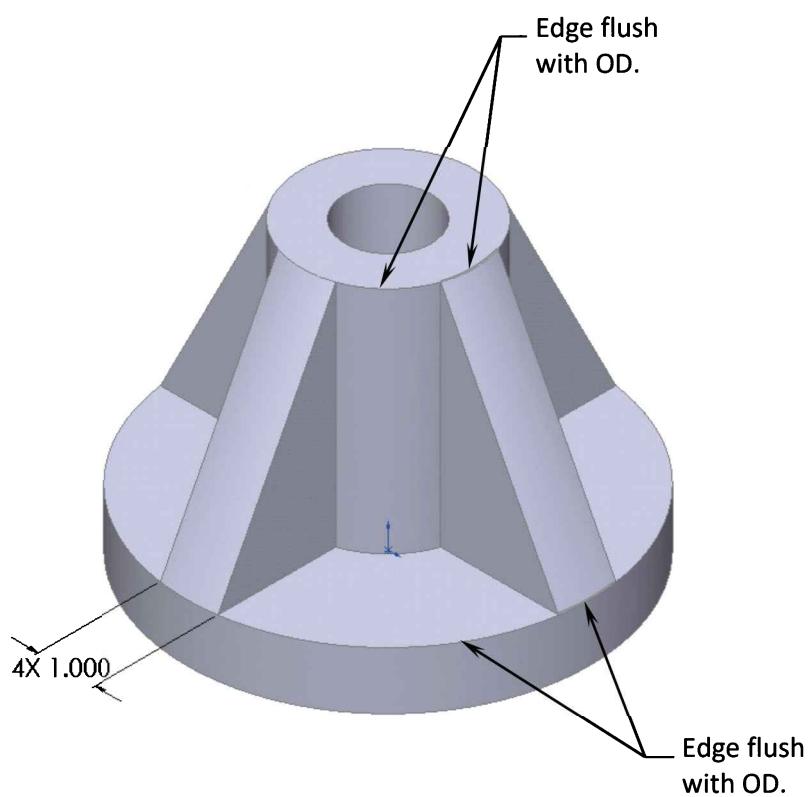
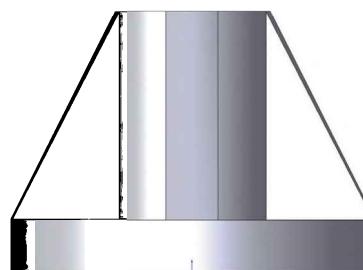
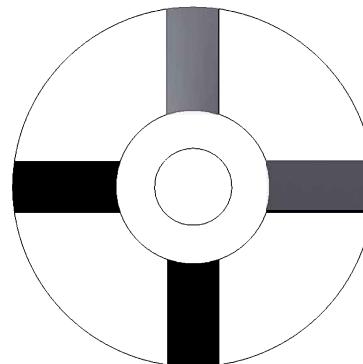
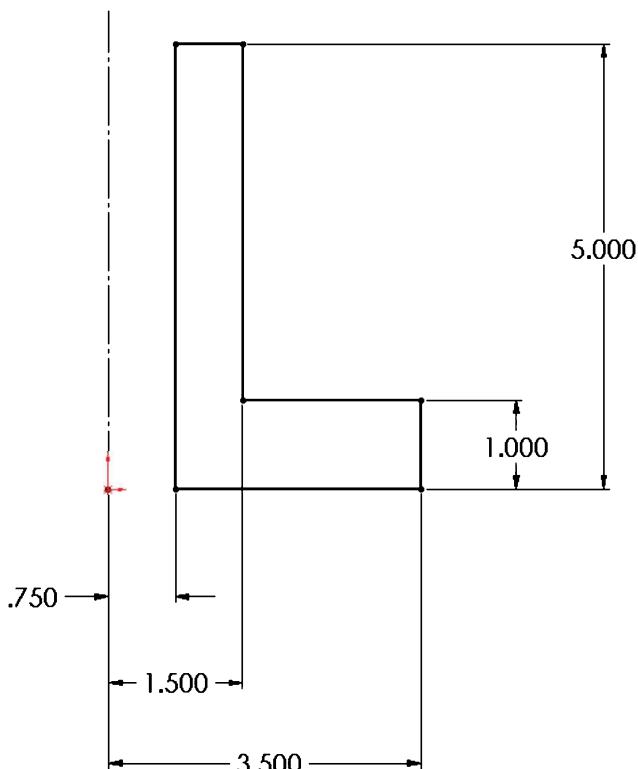
1. In a new part mode, when the Sketch button is selected first, SOLIDWORKS will prompt you to select a sketch plane.
 - a. True
 - b. False
2. It is sufficient to create a Parallel-Plane-At-Point with a Reference Plane and a Reference Point.
 - a. True
 - b. False
3. A loft feature uses two or more sketch profiles to define its shape.
 - a. True
 - b. False
4. Only one guide curve can be used in each loft feature.
 - a. True
 - b. False
5. Multiple guide curves can be used to connect and control the loft transition.
 - a. True
 - b. False
6. The guide curves can be either a 2D sketch or a 3D curve.
 - a. True
 - b. False
7. The loft profiles and the guide curves should be related with Coincident or Pierce relations.
 - a. True
 - b. False
8. The loft profiles should be created before the guide curves.
 - a. True
 - b. False

1. TRUE
2. TRUE
3. TRUE
4. FALSE
5. TRUE
6. TRUE
7. TRUE
8. TRUE

Exercise: Loft

There are several different ways to model this part, but this exercise will focus on the Loft technique instead.

1. Create the part below, use the Loft and the Circular Pattern features.



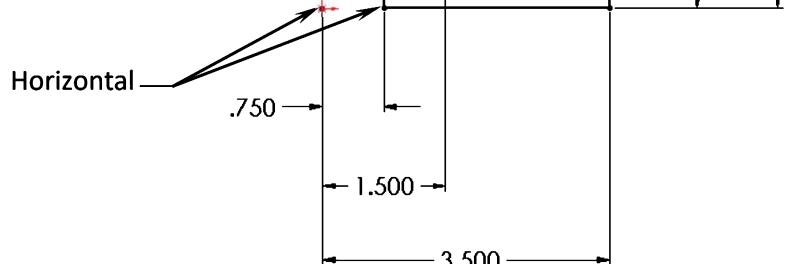
2. Use the instructions on the following pages, if needed.

1. Starting with the base sketch:

Select the Front plane and open a new sketch.

Sketch the profile shown on the right.

Add the dimensions to fully define the sketch.



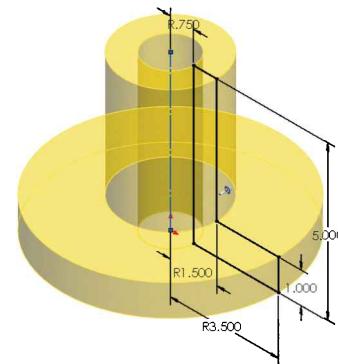
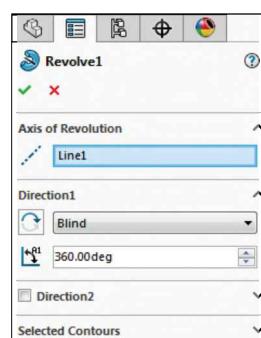
2. Revolving the base:

Click or select **Insert / Boss-Base / Revolve**.

Direction 1: **Blind**.

Revolve Angle: **360deg**.

Click **OK**.

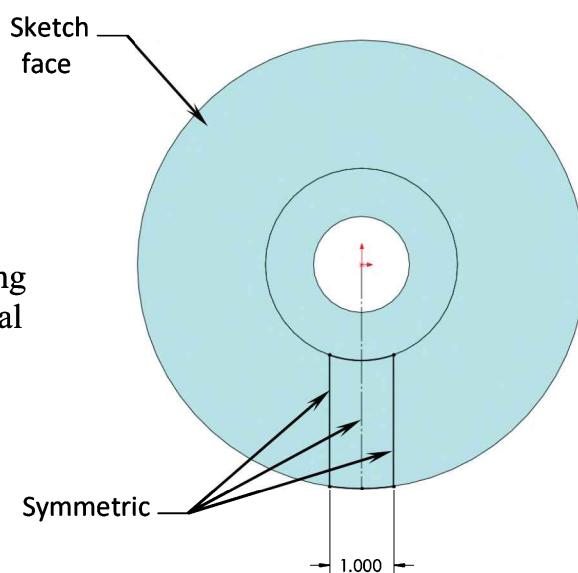


3. Creating the 1st loft profile:

Select the face indicated and open a new sketch.

Create the sketch profile by converting the 2 circular edges then add 2 vertical lines.

Trim the circles to form one continuous closed profile.



Exit the sketch.

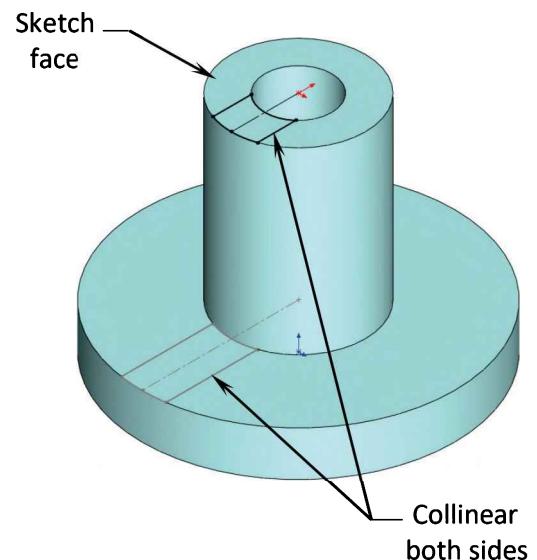
4. Creating the 2nd loft profile:

Select the upper face as indicated and open a **new sketch**.

Using the same techniques as in the previous sketch, construct this new sketch the same way.

Add the Collinear relations to both sides, between the 2 sketches.

Exit the sketch.

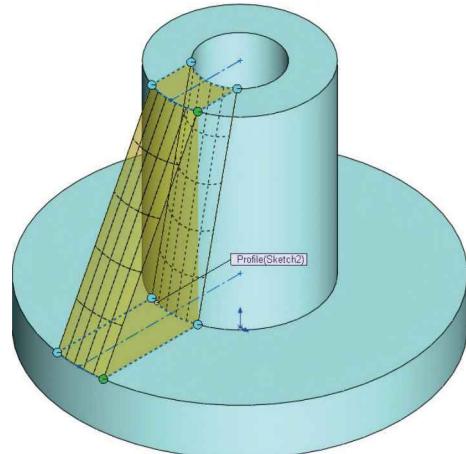
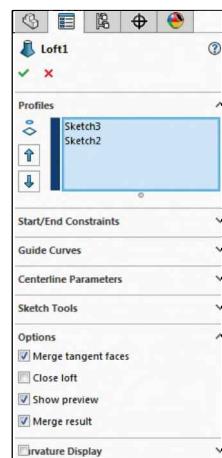


5. Creating the 1st loft feature:

Click  or select: **Insert** **Boss-Base Loft**.

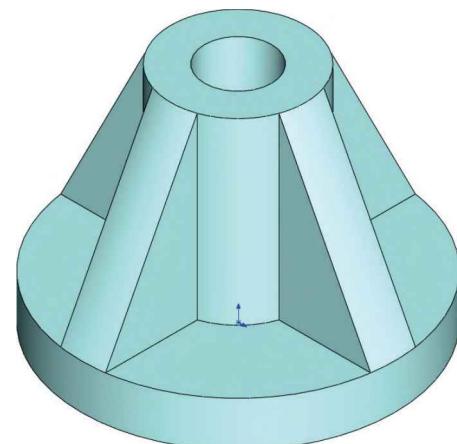
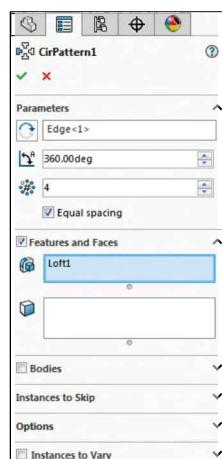
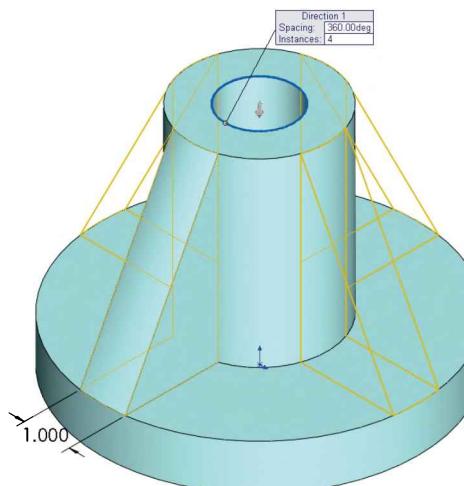
Select the outer corner of each sketch profile to prevent the loft from twisting.

Click **OK**.



6. Pattern and Save:

Create a **circular pattern** of the lofted feature with a total of **4 instances**, then save the part as: **Loft_Exe**.



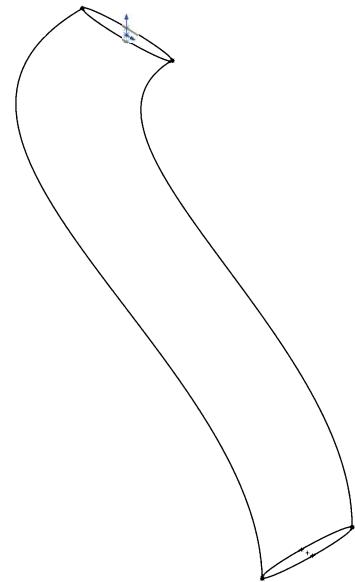
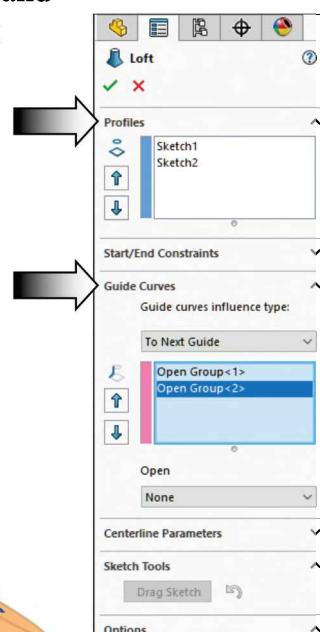
Exercise: Loft vs. Boundary

The Loft and the Boundary tools can produce very high quality and accurate features. They both use multiple profiles and guide curves to define the shape of the feature.

1. Opening a part document:

Browse to the training folder and open a part document named:
Loft vs. Boundary.sldprt.

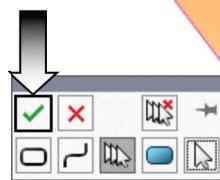
There are 2 sketch profiles and 2 guide curves in this document. They will be used to create a Loft and a Boundary feature so that we can compare the results side by side.



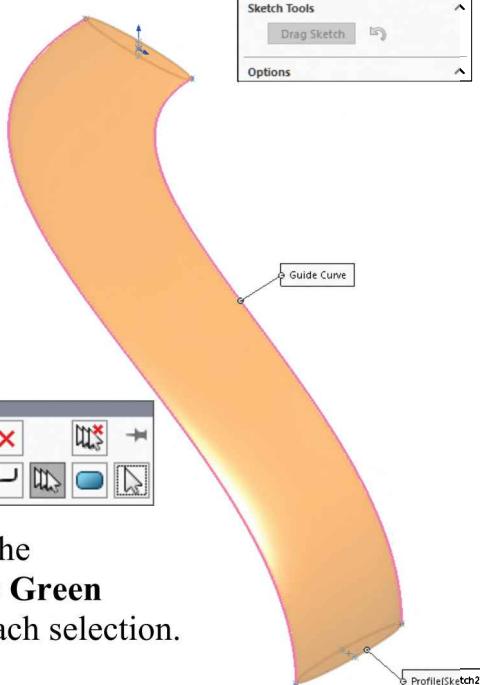
2. Creating a Loft feature:

Switch to the **Features** tab and click **Lofted-Boss-Base**.

For Loft Profiles, select the **2 elliptical** sketches.



Expand the Guide Curves section and select the **2 curves**. Click the **Green** check mark after each selection.



Click **OK**.

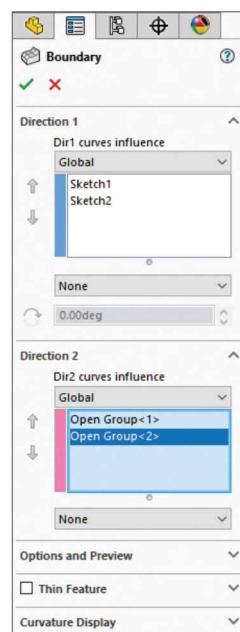
3. Creating a Boundary feature:

Save the model as **Loft1** and open the **Loft vs. Boundary** part document again to use in this 2nd half.

Switch to the Features tab and click **Boundary Boss-Base**.



For Direction 1, select the **2 elliptical** sketches.



For Direction 2, select the **2 curves**. Click the **Green** check mark inside the SelectionManager after selecting curve.

Click **OK**.

Save the model as **Boundary1**.

As mentioned earlier, the Loft and the Boundary tools can produce high quality and accurate features, but in some cases, depending on the complexity of the shape, the Boundary tool may offer a slightly better, smoother looking surface than the loft tool.

Let us take a look at them side by side.



4. Verifying the curvatures:

Open the **Loft1** document that was saved earlier and select:
Window, Tile Vertically.



Switch to the **Evaluate** tab and click **Curvature**.

Press **Control+7** to switch to the Isometric view.

Hover the mouse cursor over the lower left area as indicated. The Curvature and Radius are about the same for both features.

Loft Feature

Hover over this area

Curvature: 0.173626 Rad. of Curvature: 2.67647

Boundary Feature

Hover over this area

Curvature: 0.18984 Rad. of Curvature: 2.63863

Press **Control+4** to switch to the Right view.

The lower left area looks good in the Loft but looks a little better in the Boundary.

Good blended area

Loft Feature

(Note: This area can be improved even further if the Start/End Constraints are set to Normal-To-Profile.)

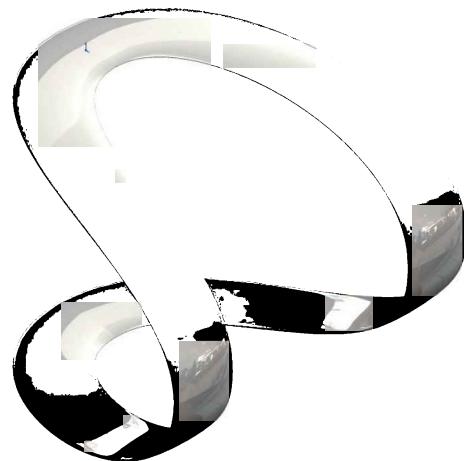
Better blended area

Boundary Feature

Exercise: Mirror using 2 Planes

In SOLIDWORKS 2022 and newer, you can mirror about two planes at once. Previously, you needed to create multiple features to achieve this.

Available in parts only. You can mirror a feature about two planes or 2 faces at a time in one feature.



1. Opening a part document:

Browse to the training folder and open a part document name: **Mirror with 2 Planes.sldprt**.

This feature represents one-quarter of the whole part. We will create the entire part by using the new option called: Mirror about 2 planes.

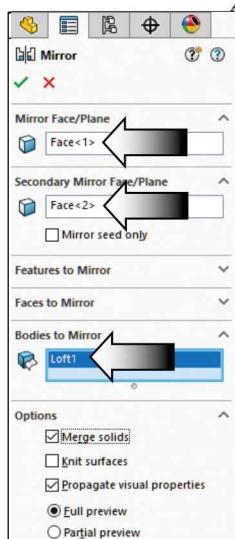
Switch to the **Features** tab and click **Mirror**.

For Mirror Face/Plane, select the **upper face** of the model as indicated.

Expand the Second Mirror Face/Plane and select the **lower face** as noted.

For Bodies to Mirror, select the **model** in the graphics area.

Enable **Merge-Solids** option.



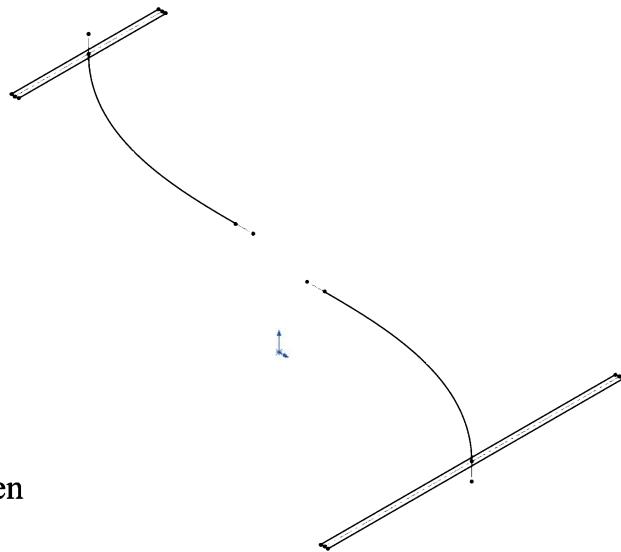
Click **OK**.



Exercise: Tangent vs. Curvature

When creating a Sweep, Loft, or Boundary feature, the option Tangency-To-Face or Curvature-To-Face applies a smooth, visually appealing curvature continuous feature at the selected start or end profile.

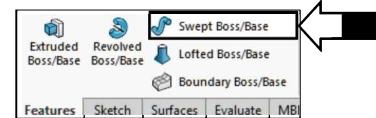
This exercise provides an opportunity to explore those options.



1. Opening apart document:

Browse to the training folder and open a part document named:
Tangent vs. Curvature Blends.sldprt.

There are 4 sketches in this model. 2 of them will be used as Sweep Profiles, and the other 2 will be used as Sweep Paths.



2. Creating the 1st swept feature:

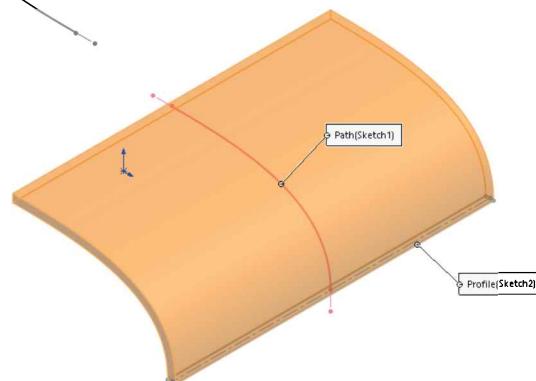
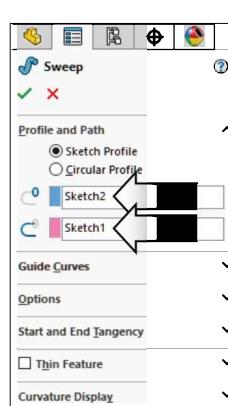
Switch to the **Features** tab and click **Swept Boss/Base**.

For Sweep Profile, select **Sketch2** either from the FeatureManager tree or directly from the graphics area.

For Sweep Path, select **Sketch1**.

Keep all other parameters at their defaults.

Click **OK**.



3. Creating the 2nd swept feature:

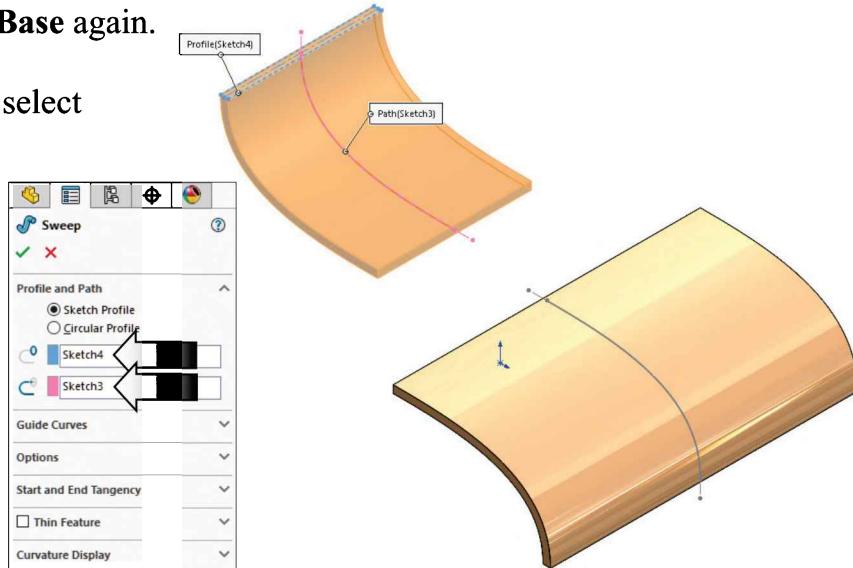
Click Swept Boss-Base again.

For Sweep Profile, select Sketch4.

For Sweep Path, select Sketch3.

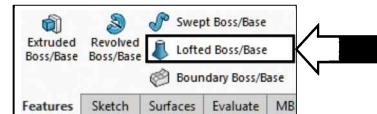
Leave all other parameters at their default values.

Click OK.



4. Creating a loft feature:

Two closed sketch profiles or two faces in a model can be used to create a loft feature.



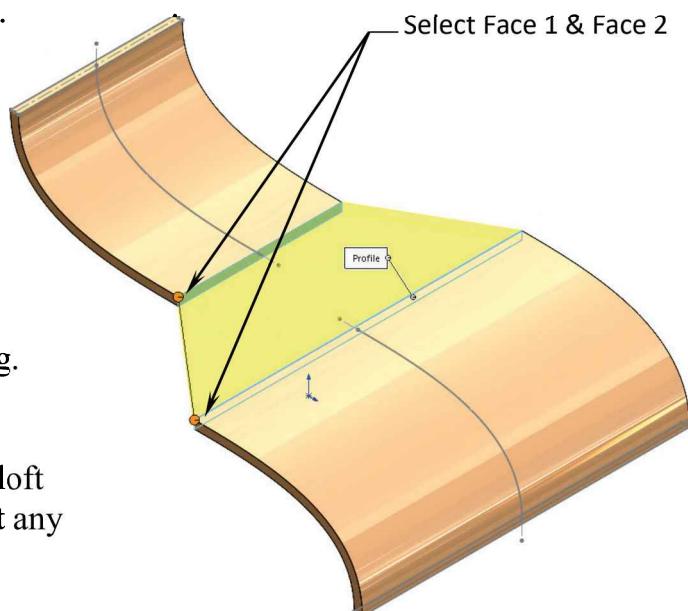
Click Lofted Boss-Base.

For Loft Profiles, select the **2 rectangular faces** of the model as indicated.

Note: Zoom closer when selecting the 2 faces. Selecting the edges will not work for solid features.

Move the connectors if needed to prevent the loft from twisting.

The preview graphics shows a loft feature is being created without any tangent or curvature controls.



Expand the Start/End Constraints section.

Set both Start and End to **Tangency to Face**.

Expand the Feature Scope section and enable the **All Bodies** option.

Click OK.

Press **Control+Z** to undo the Loft.

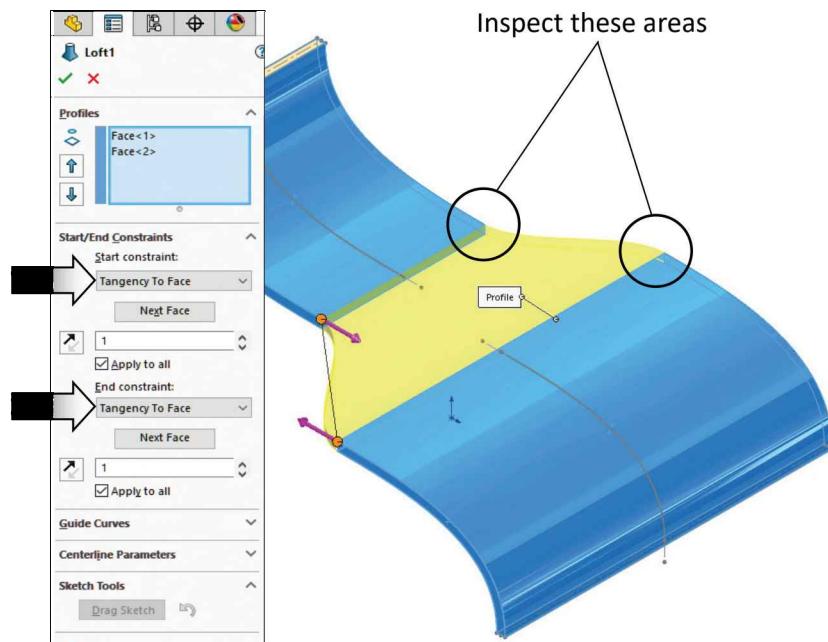
We will re-create this feature using the Boundary tool this time.

Click Boundary Boss-Base.

Select the **2 rectangular faces** as shown in the previous step.

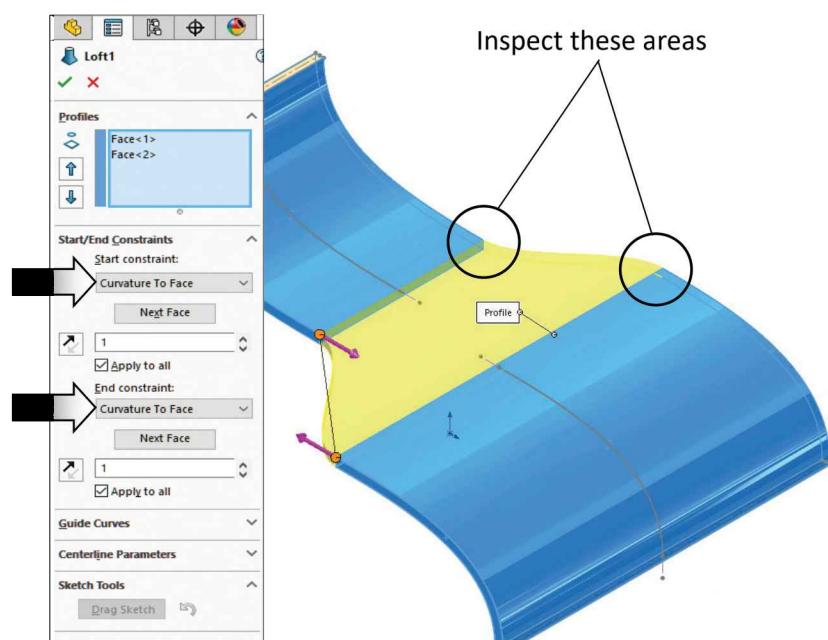
For Start/End Constraints, select **Curvature To Face** for both.

Click OK.



Inspect these areas

Examine the circled areas.
The Tangency to Face option creates a very good blend between the 2 solid bodies



Inspect these areas

Examine the circled areas.
The Curvature to Face option creates a smooth blend between the 2 solid bodies

5. Adding fillets:

We will go with the Boundary feature for this exercise.

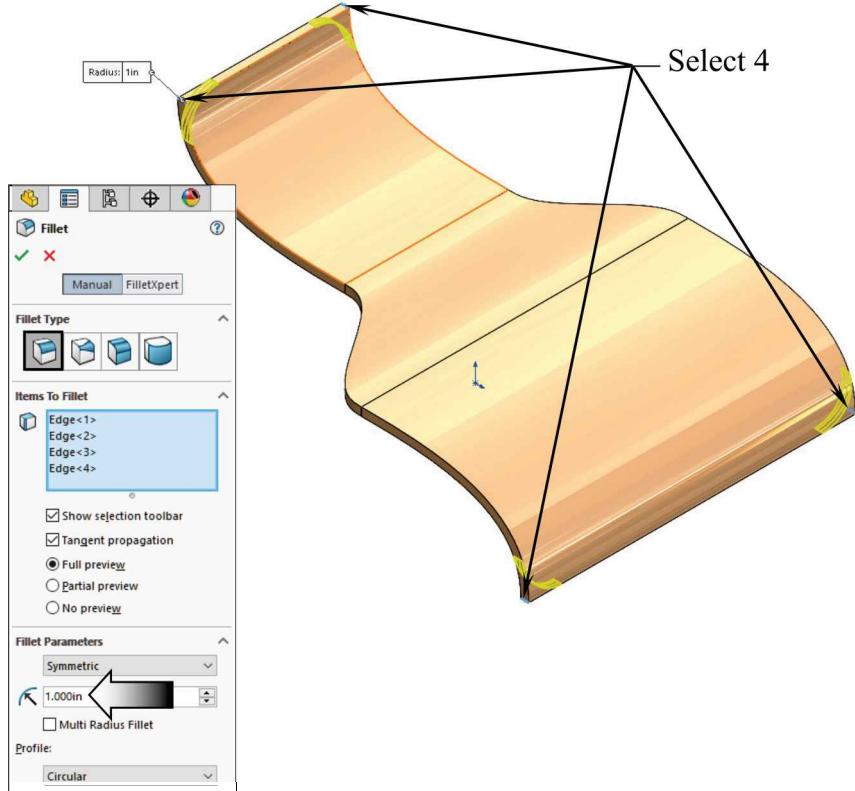
Click **Fillet**.

Use the default
Constant Size
Fillet option.

For Radius Size,
enter **1.000in**.

For Items to Fillet,
select the 4 edges
as indicated.

Click **OK**.



6. Saving your work:

Select **File, Save As**.

Enter: **Tangent vs. Curvature**
Completed.sldprt for the file name

Click **Save**.

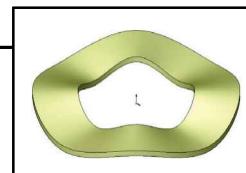
Close all documents.

CHAPTER 7

Loft With Guide Curves

Loft with Guide Curves

Waved Washer



This lesson demonstrates the creation of a waved washer using the loft technique, where 4 sketch profiles and a single 3D guide curve are used.

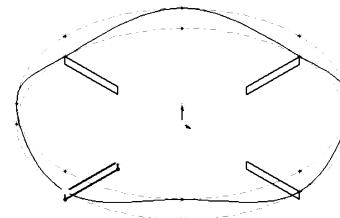
A loft feature usually contains several profiles, one centerline parameter and one or more guide curves.

In this case study, four identical sketches will be used as the loft profiles and a single guide curve will be used to connect the profiles.

Since the loft profiles are identical, the derived-sketch option will be used to show how the sketches can be derived or copied. A derived sketch is driven by the original sketch. It can only be positioned with relations or dimensions, but its sketched entities cannot be changed.

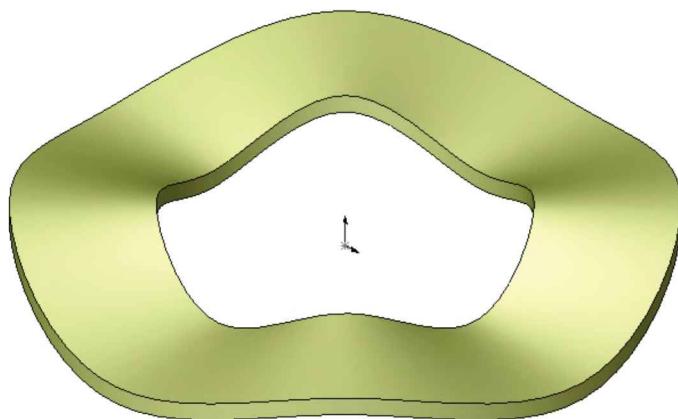
When the Original sketch is changed, the derived sketch will be updated automatically; however, the derived sketches can be Un-derived to break their associations with the Original sketch.

The loft profiles are connected with a 3D curve. The 3D curve will be generated using the command called Curve-Through-Reference-Points. The loft profiles are also Pierced to these reference points.



The Curve-Through-Reference-Points will be used to guide and control the transition between the loft profiles.

Waved Washer Loft with Guide Curves



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI

Units: INCHES – 3 Decimals

Tools Needed:



Insert Sketch



Line



Circle



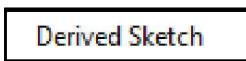
Sketch Point



Add Geometric Relations



Dimension



Curve Through Reference Points



Base/Boss Loft

1. Creating the 1st Construction profile:

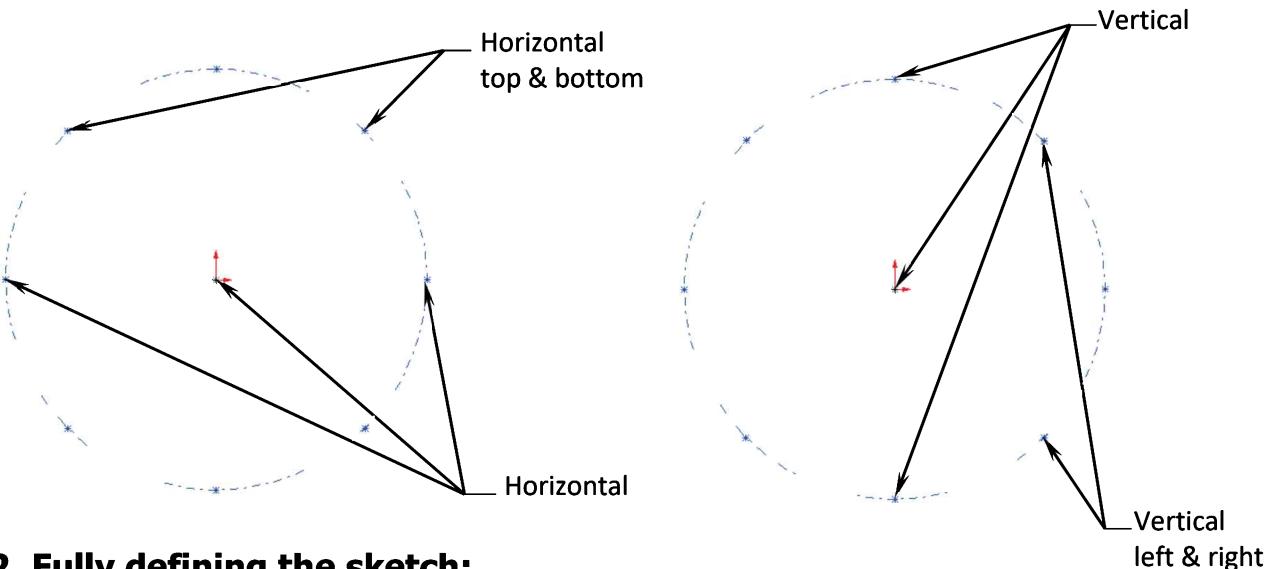
Select the Top plane from the FeatureManager tree.

Click  or select **Insert / Sketch**.

Sketch a **Circle** and convert it to a construction circle



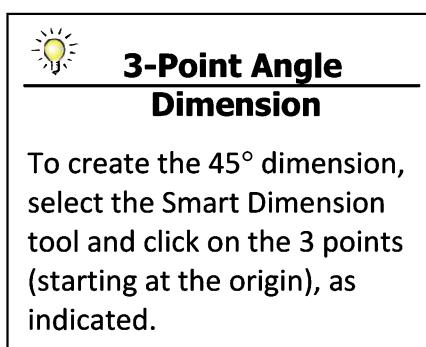
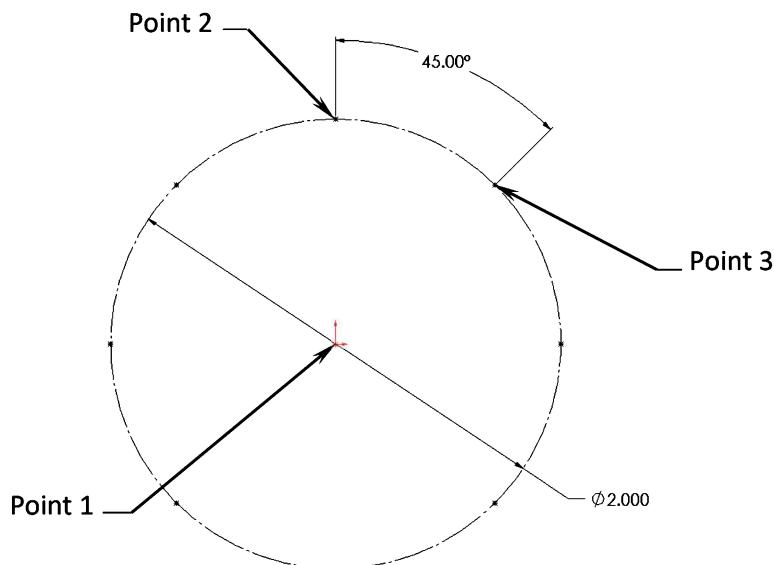
Add **8 sketch points**  approximately as shown.



2. Fully defining the sketch:

Add an angular dimension between the sketch points.

Add a vertical and a horizontal relation as indicated.



Exit the Sketch .

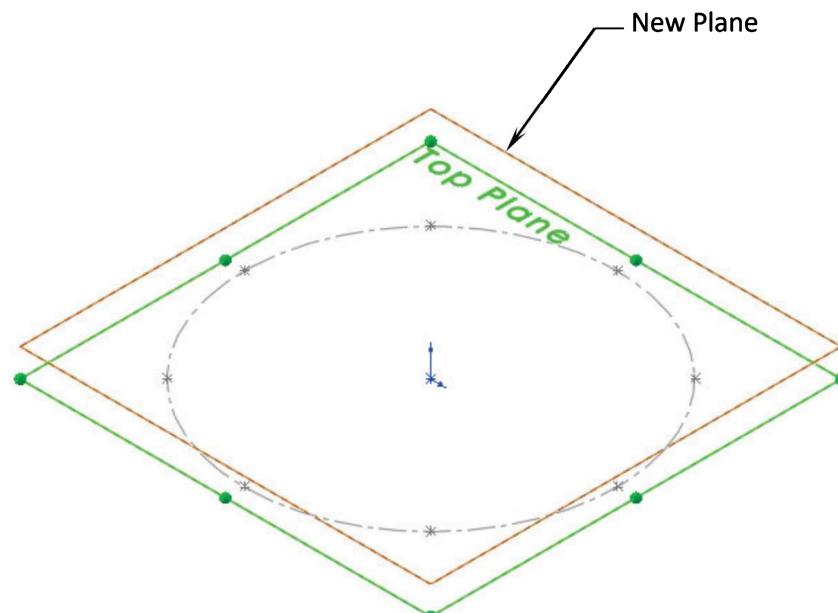
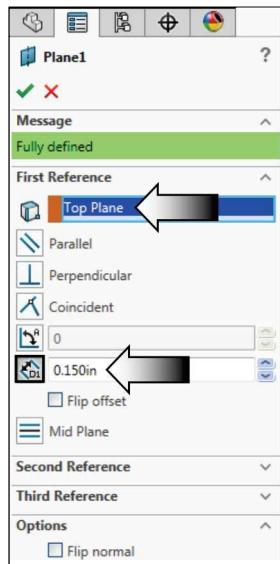
3. Creating an Offset Distance plane:

Click  or select Insert / Reference Geometry / Plane.

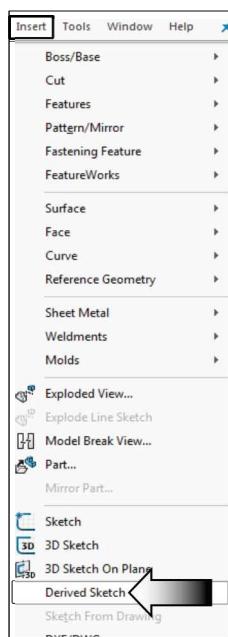
Select the Top plane as the First Reference.

Select Offset Distance option and enter .150in.

Click OK.

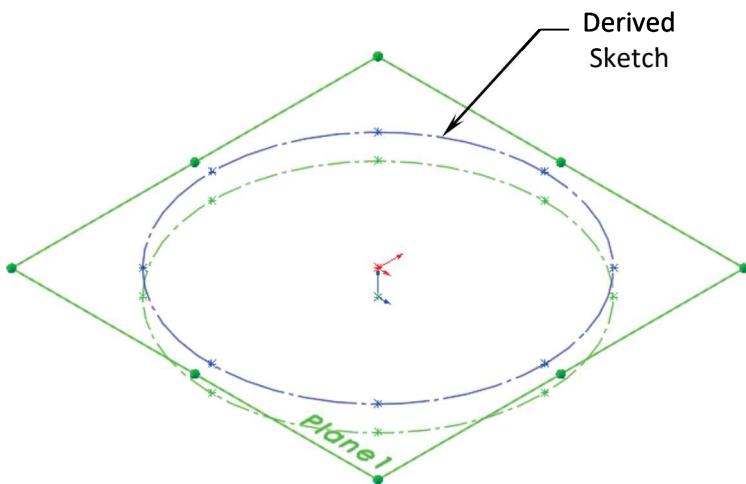


4. Creating the 2nd construction profile using Derived Sketch:



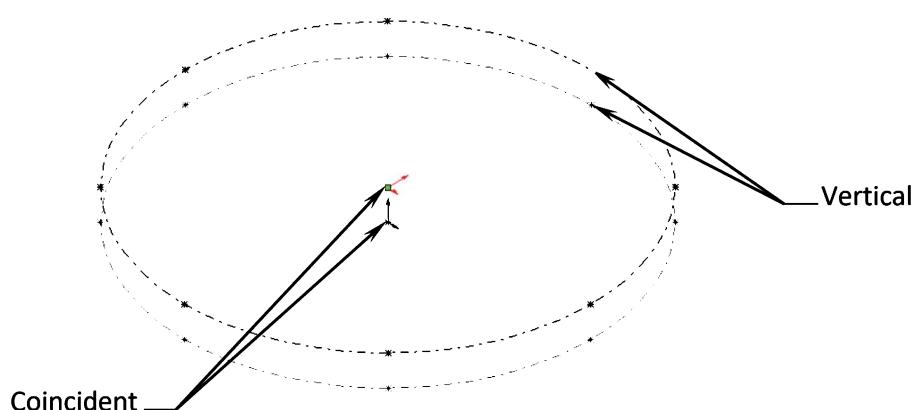
Hold the **Control** key, select the **new Plane (plane1)** and **Sketch1** from the FeatureManager tree.

Click **Insert / Derived Sketch**.



5. Positioning the Derived Sketch:

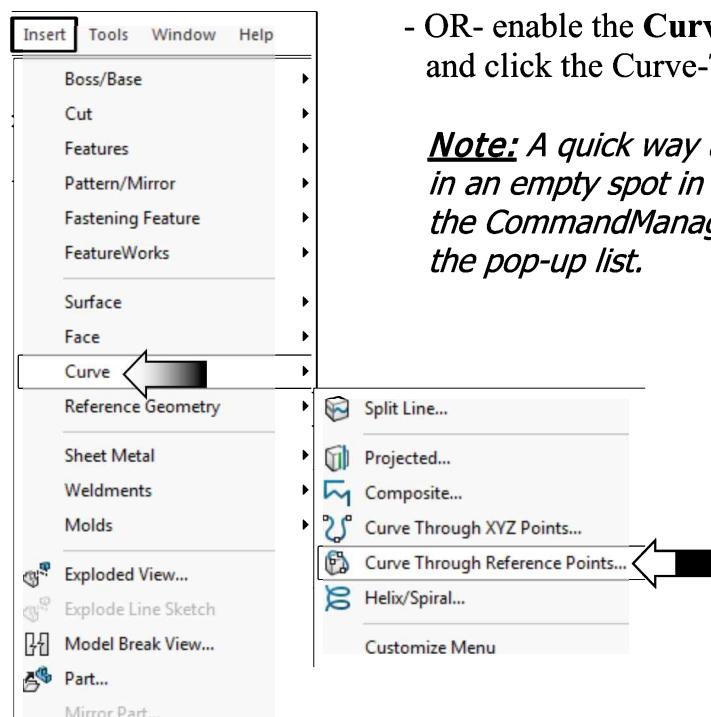
Add the Vertical & Coincident relations between the indicated points.



Exit the Sketch  or select **Insert / Sketch**.

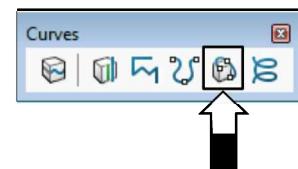
6. Creating a Curve through Reference Points:

 Click  or select **Insert / Curve / Curve-Through-Reference-Points**.



- OR- enable the **Curves** toolbar (View / Tool Bars / Curves), and click the Curve-Through-Reference-Points icon.

Note: A quick way to access the toolbars is to right click in an empty spot in the top right area of the screen (inside the CommandManager) and select the Curves toolbar from the pop-up list.

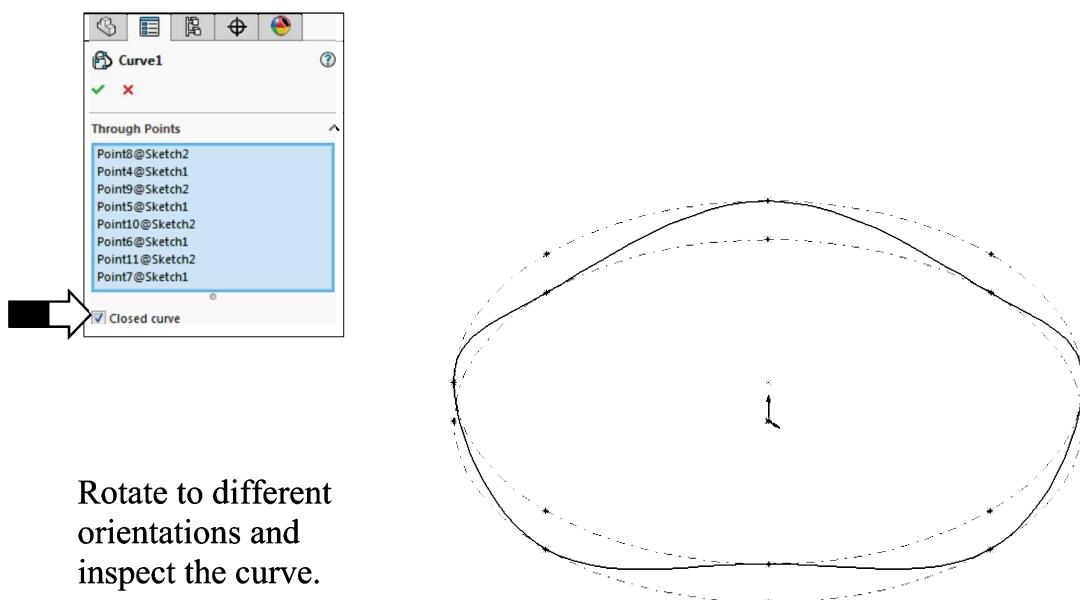
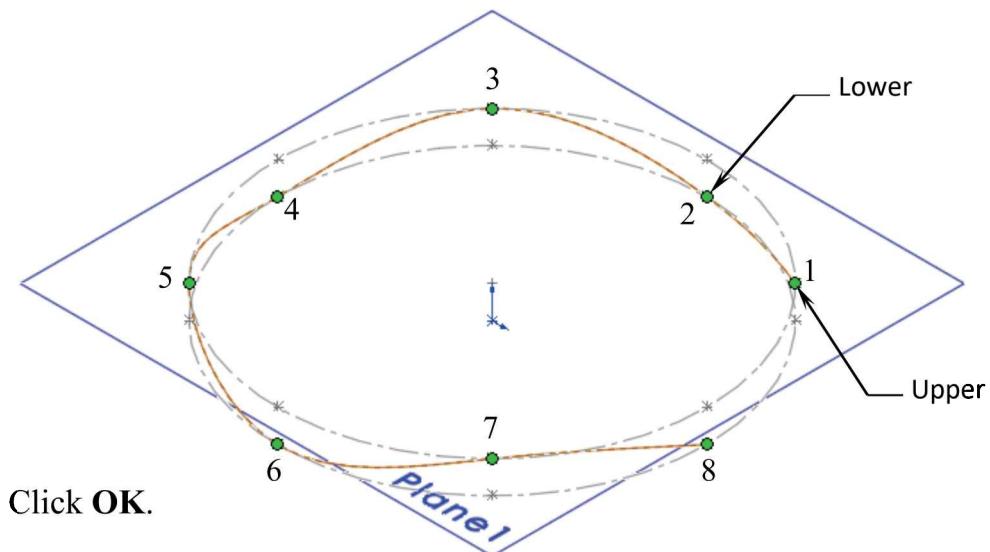


Curve Through Reference Points

Select the sketch points in the order as noted (required for this lesson only). (Point 1 is on top, point 2 is on the bottom, 3 on top, 4 on bottom, and so on).

Click **Closed curve** to close the curve.

The **Closed Loft** option connects the 1st and the last points and forms a closed loop.

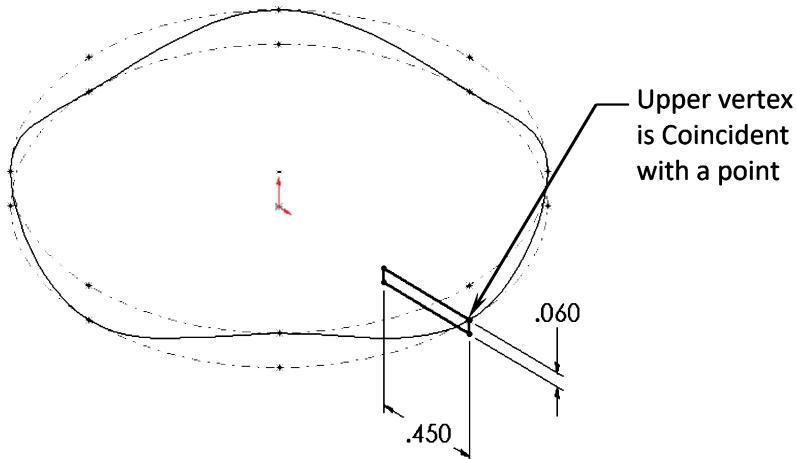


Keep both sketches visible to use in the next couple of steps.

7. Sketching the 1st loft section:

Select the **Front** plane from the FeatureManager tree and open a **new sketch**.

Sketch a **Rectangle** and add the dimensions / relations as shown.



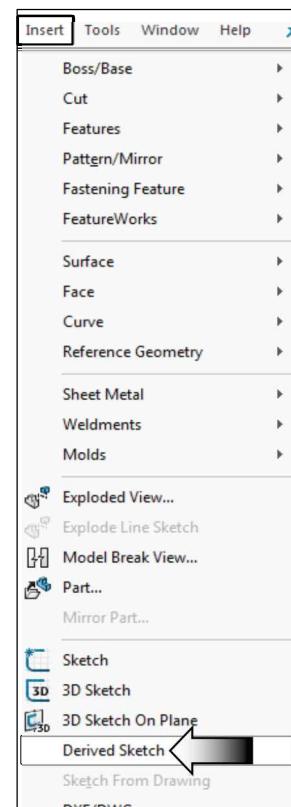
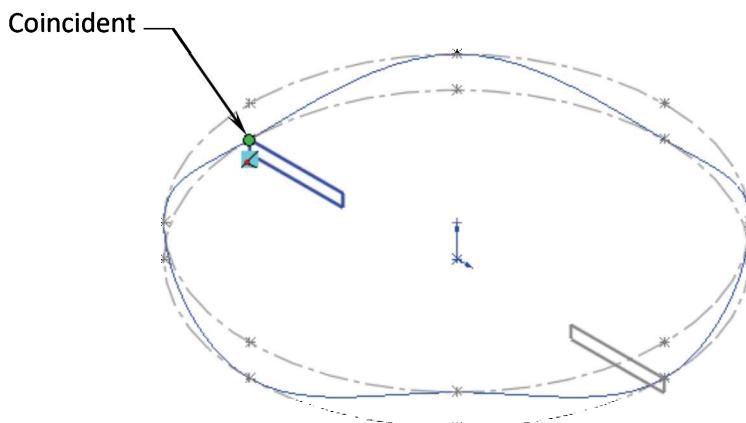
Exit the sketch or select **Insert / Sketch**.

8. Creating the 2nd loft section using Derived-Sketch:

Hold the **Control** key, select the **Front** plane and the sketch of the **rectangle** from the Feature tree.

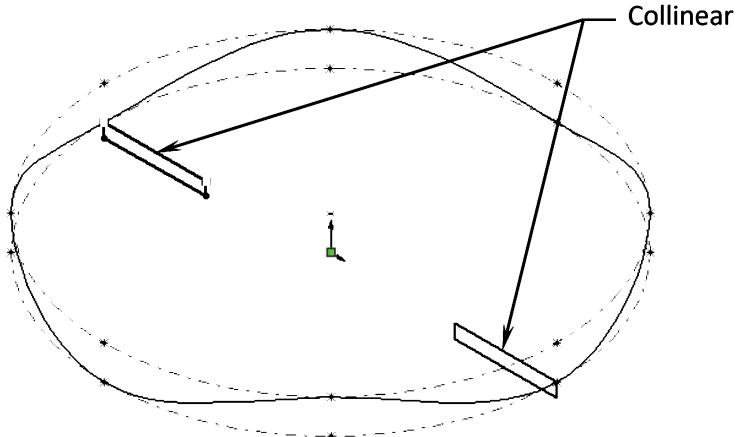
Select **Insert / Derived Sketch**.

A derived sketch is created and placed on top of the original sketch; drag it to the left side and then add a **coincident** relation to the point on the bottom as noted.



9. Fully defining the Derived Sketch:

Add a **Collinear** relation between the 2 lines as indicated.



Derived Sketch

A derived sketch is a dependent copy of the original sketch. It can only be moved or positioned on the same or different plane with respect to the same model.

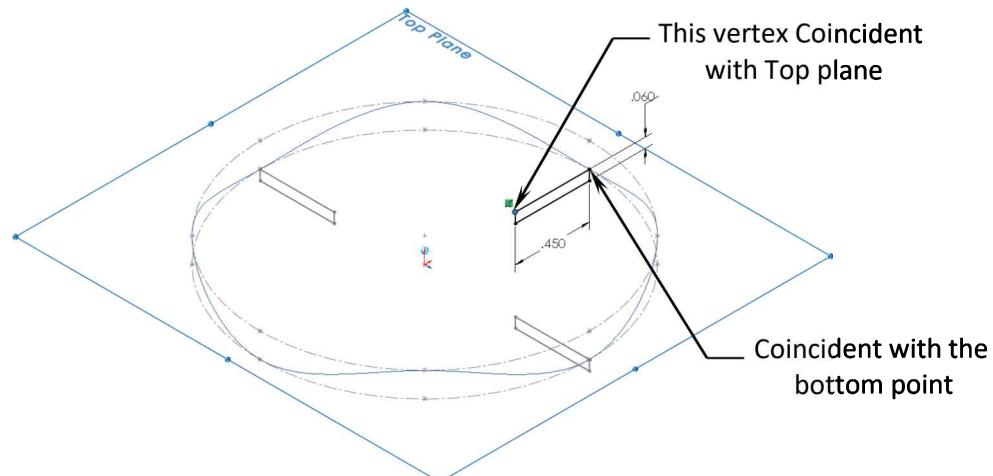
Exit the Sketch or select **Insert / Sketch**.

10. Sketching the 3rd loft section: (or use the Derived Sketch option)

Select the Right plane from the FeatureManager tree and open a **new sketch**.

Sketch a **Rectangle** (or copy and paste the previous sketch).

Add the dimensions and relations needed to fully define the sketch.

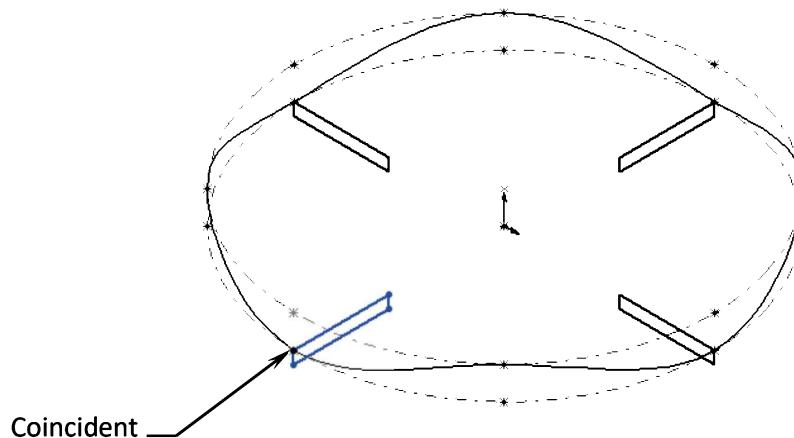


Exit the Sketch or select **Insert / Sketch**.

11. Creating the 4th loft section using Derived-Sketch:

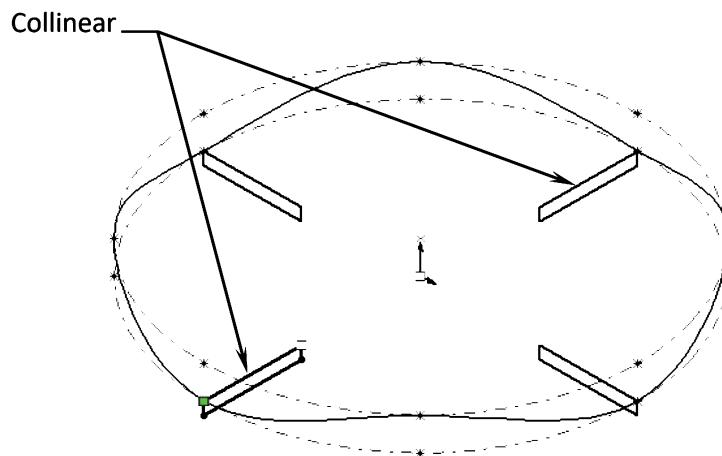
Hold the **Control** key; select the **Right** plane and the sketch of the 3rd rectangle from the FeatureManager tree.

Select **Insert / Derived Sketch**.



12. Constraining the Derived sketch:

Add a **Collinear** relation between the 2 lines as shown.

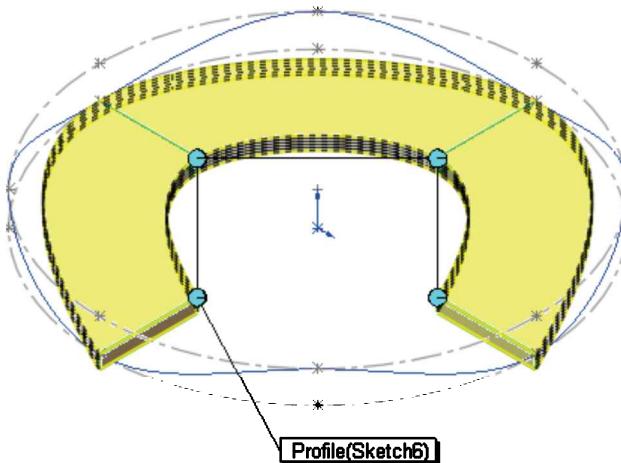


 Exit the Sketch or select **Insert / Sketch**.

13. Creating a Loft with Guide curve:

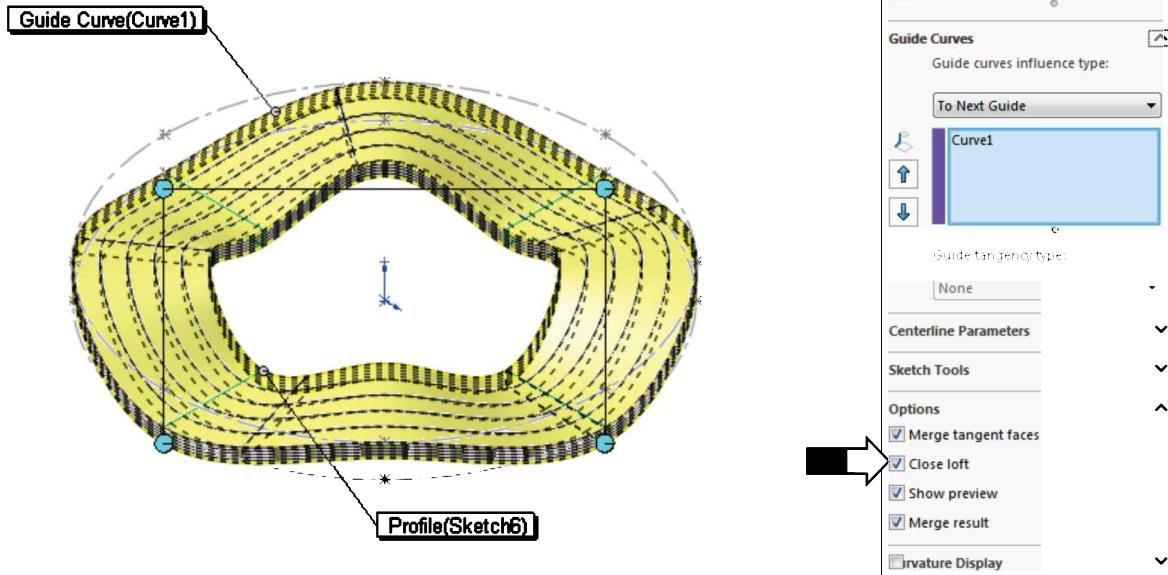
Click  or select Insert / Boss-Base / Loft.

For Loft Profiles: Select the **four** rectangular sketches.



For Guide Curve, select the **3D Curve**.

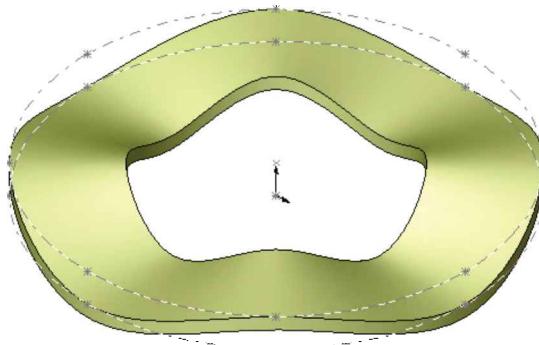
Click **Close Loft** under Options (arrow).



Click **OK**.

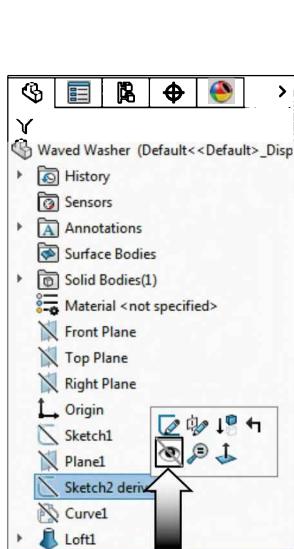
The completed Waved Washer (with the construction sketches still visible).

Note: Use 8 sketch points to create 4 waves, 16 sketch points will make 8 waves, and so on.

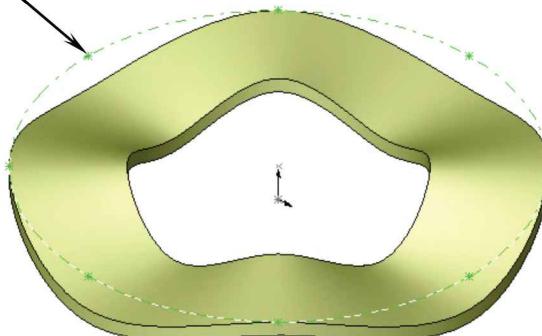


14. Hiding the construction sketches:

Click the construction sketches and select **Hide** .

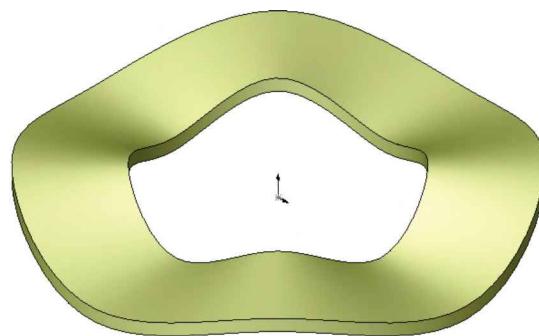


Hide sketch



15. Saving your work:

Select **File / Save As**.



Enter **Waved Washer** for the file name.

Click **Save**.

Questions for Review

1. A sketch profile can be copied onto another plane or a planar surface.
 - a. True
 - b. False
2. Sketch points can be added in any sketch to help define the sketch geometry or locations.
 - a. True
 - b. False
3. If a derived sketch is driven by the original sketch, its entities cannot be changed.
 - a. True
 - b. False
4. A 3D curve can be created using the reference points in the sketches or model's vertices.
 - a. True
 - b. False
5. The loft sections should either be Pierced or Coincident to the guide curves.
 - a. True
 - b. False
6. The guide curves can also be used to control the loft sections from twisting.
 - a. True
 - b. False
7. Only two guide curves can be used in each loft feature.
 - a. True
 - b. False
8. Four sketch profiles are required to create a loft feature, no more, no less.
 - a. True
 - b. False
9. The construction sketches can be toggled (Show/Hide) at any time.
 - a. True
 - b. False

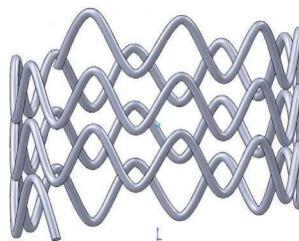
1. TRUE	2. TRUE	3. TRUE	4. TRUE	5. TRUE	6. TRUE	7. FALSE	8. FALSE	9. TRUE
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Exercise: Using Curve Driven Pattern

1. Creating the Base sketch:

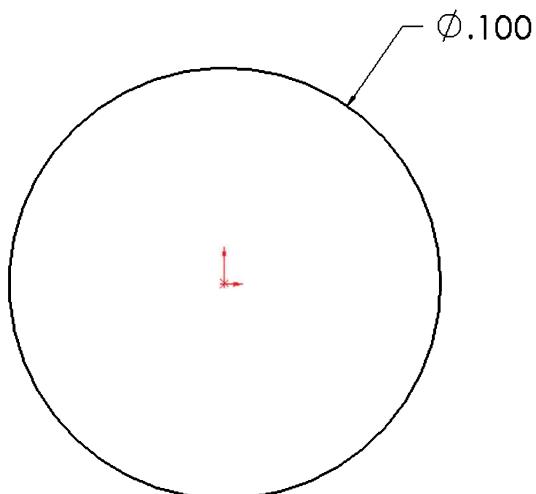
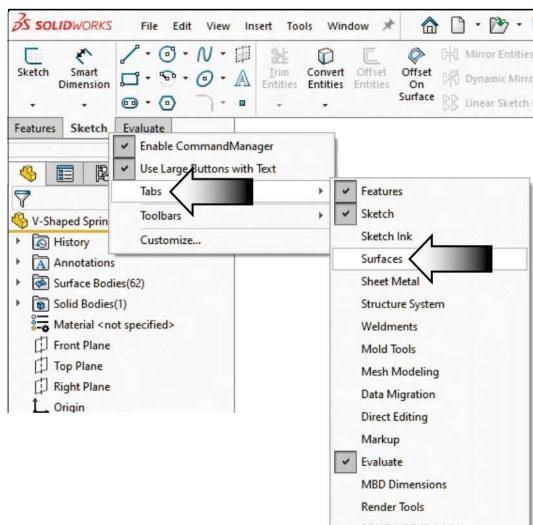
From the **Top** plane sketch a **circle**, centered on the origin.

Add a diameter dimension of **.100"**.



2. Activating the Surfaces toolbar:

Right-click the **Sketch Tab** and select **Tabs, Surfaces** to enable the toolbar.

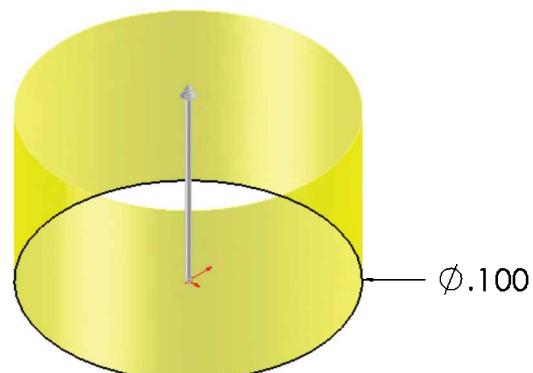
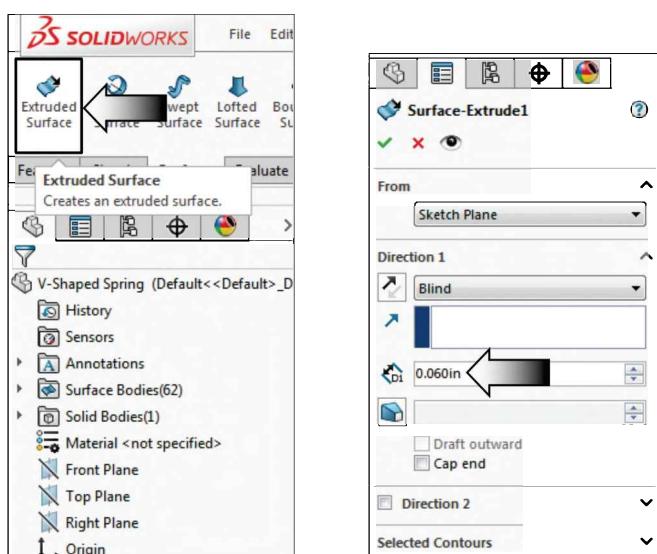


3. Extruding a surface:

From the Surfaces tab click: **Extruded Surface**.

Use the **Blind** type and enter **.060"** for extrude depth.

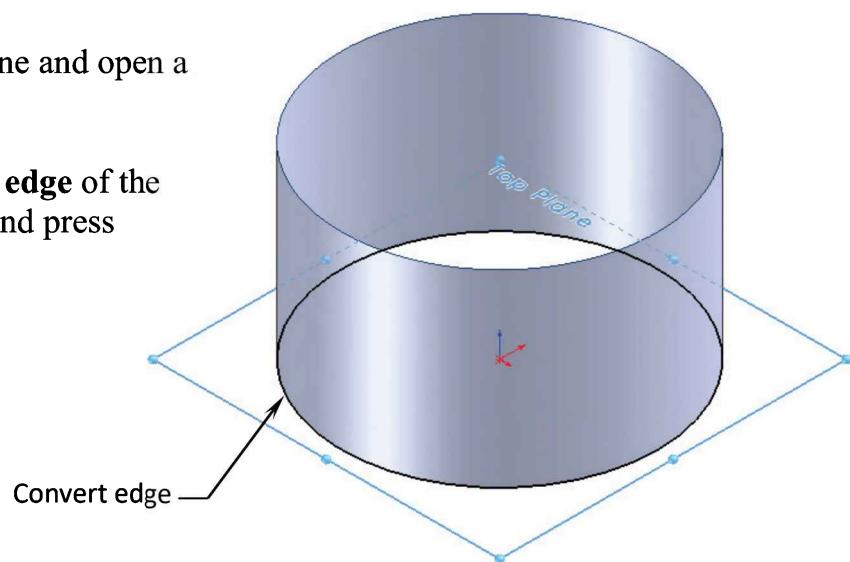
Click **OK**.



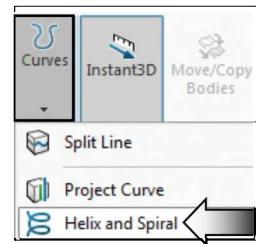
4. Creating a helix:

Select the Top plane and open a new sketch.

Select the **bottom edge** of the extruded surface and press **Convert Entities**.



From the Features toolbar click: **Curves / Helix and Spiral**.



Enter the following:

* Defined by: **Pitch and Revolution**

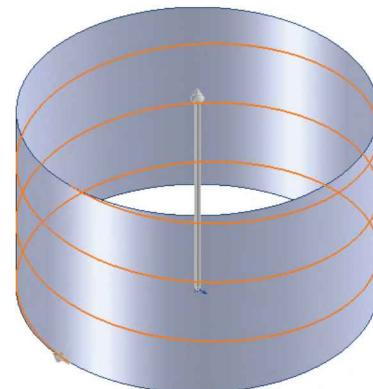
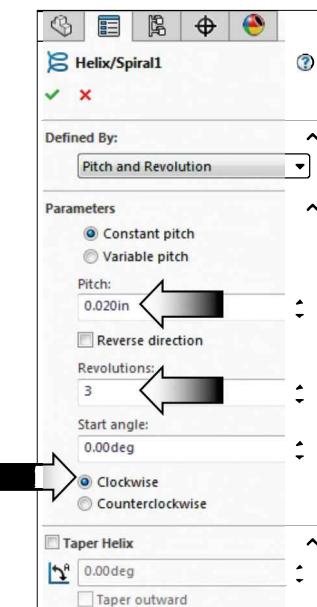
* Constant Pitch

* Pitch: **0.020"**

* Revolutions: **3**

* Start Angle: **0**

* **Clockwise**



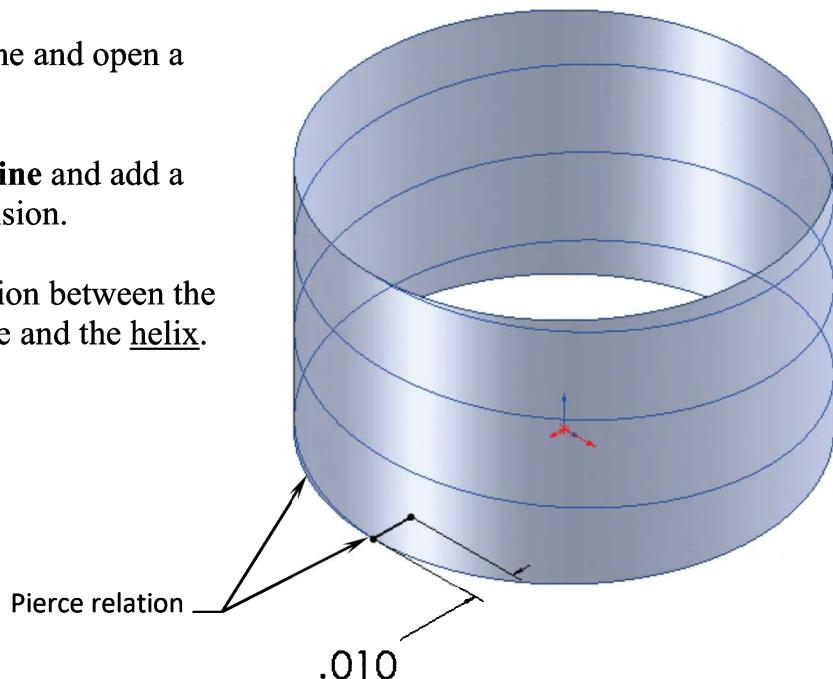
Click **OK**.

5. Creating a surface profile:

Select the Top plane and open a new sketch.

Sketch a **vertical line** and add a **.010"** linear dimension.

Add a **Pierce** relation between the endpoint of the line and the helix.

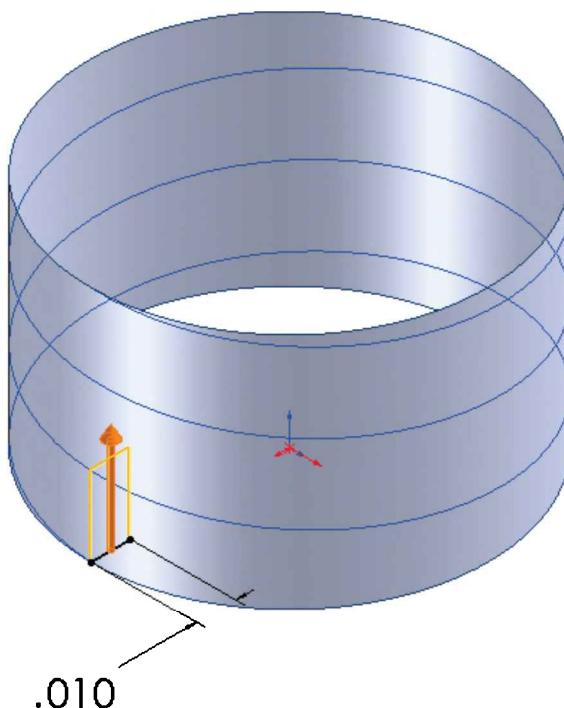
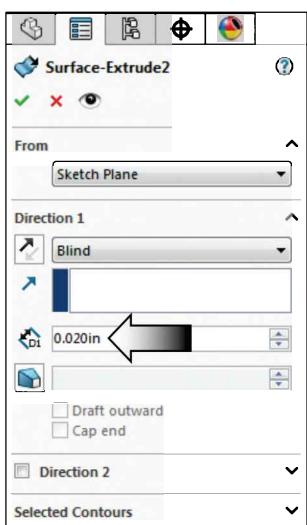


Switch to the **Surfaces** toolbar and click **Extruded Surface**.

For extrude type, use the default **Blind** type.

Enter **.020"** for extrude depth.

Click **OK**.



6. Creating a Curve Driven Pattern:

From the Features tab click:
Linear Pattern / Curve Driven Pattern.

For Direction 1, select the Helix (Edge 1).

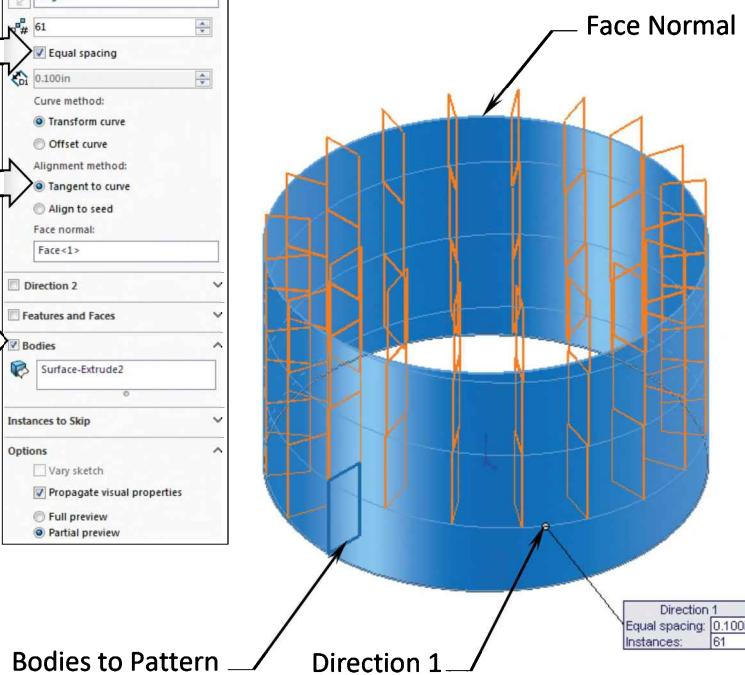
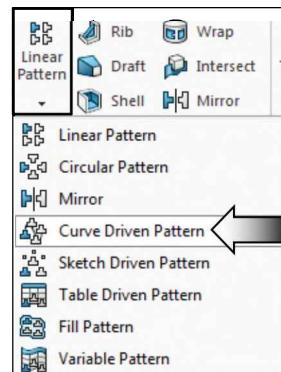
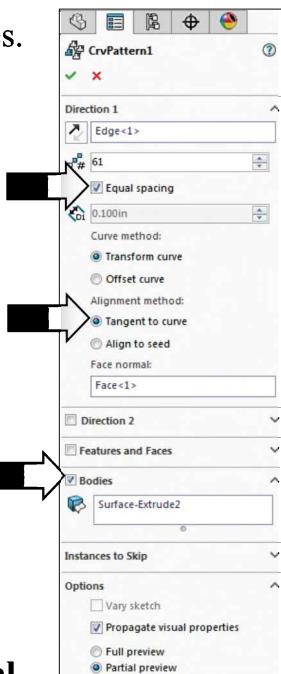
Enter **61** for Instances.

Enable the **Equal Spacing** checkbox.

Under Curve Method, select:
Transform Curve.

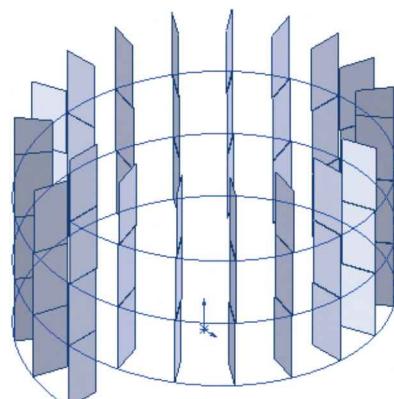
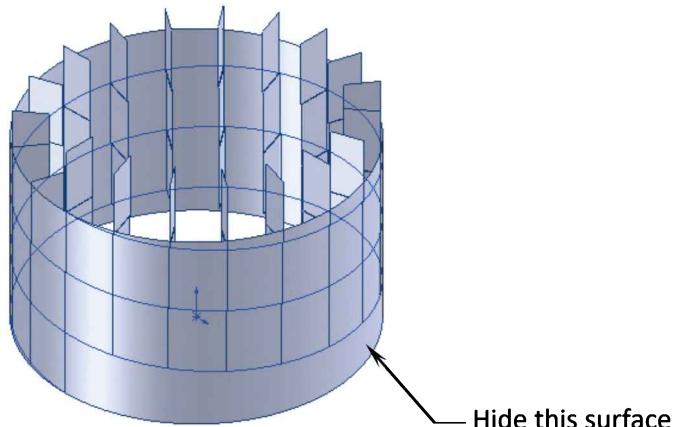
For Alignment Method, select:
Tangent to Curve.

Under Face Normal select the **Cylindrical** surface of the cylinder.



Expand the **Bodies to Pattern** section and select the **Extruded Surface** in step 5.

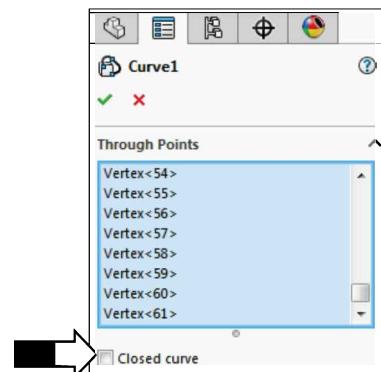
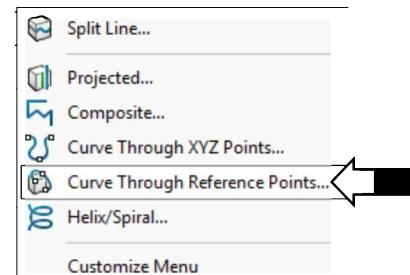
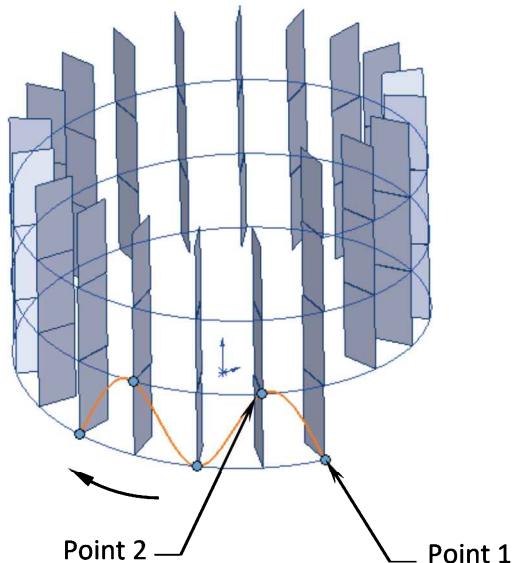
Click **OK**.



7. Creating a Curve Through Reference Points:

From the **Features** tab select:
Curves / Curve Through Reference points.

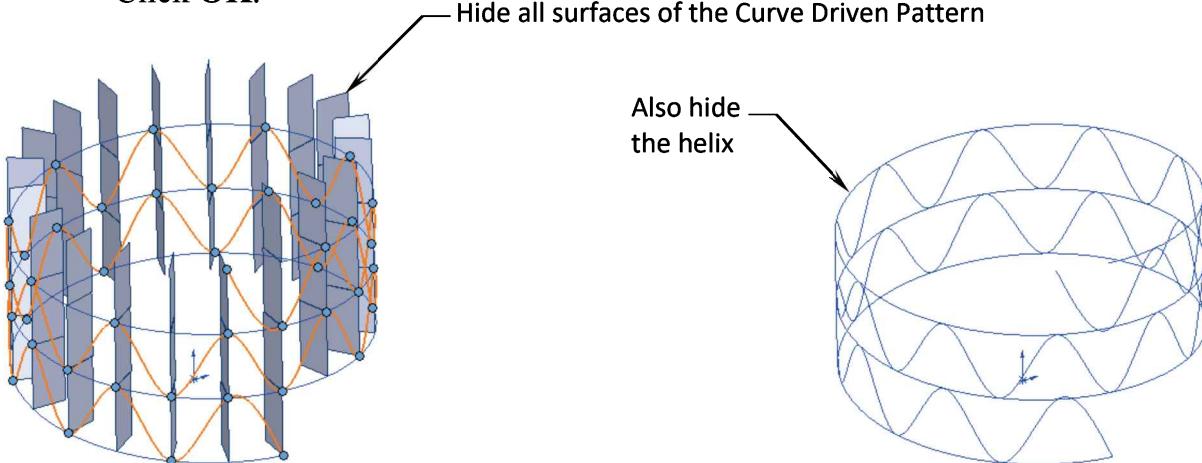
Click the starting point (point 1) at the end of the helix and go clockwise to point 2, then point 3 as indicated.



Continue going around the helix and select all connecting points. (If you make a mistake simply delete the point in the Through Points dialog box and reselect it again.)

Be sure to uncheck the **Close Curve** checkbox since the two ends of this curve are not supposed to connect.

Click **OK**.



8. Creating the final sweep:

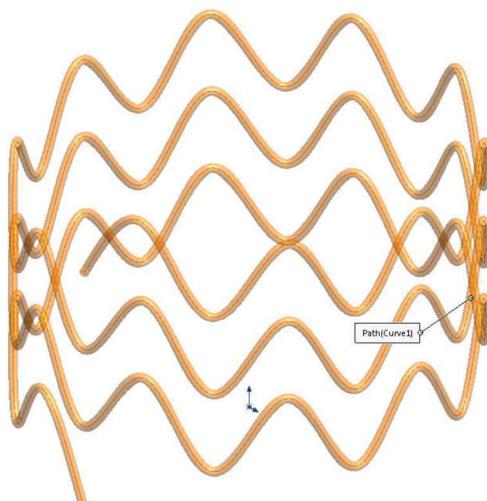
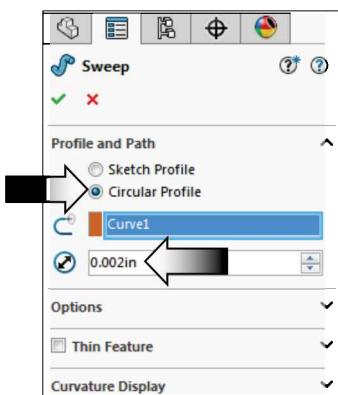
Switch to the **Features** tab and click: **Swept Boss-Base**.

Select the **Circular Profile** option (arrow).

Enter **.002in** for Profile Diameter (arrow).

Select **Curve1** as the sweep path.

Click **OK**.



9. Saving your work:

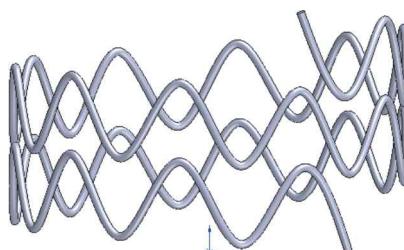
Click **File / Save As**.

Enter **V-Shape Spring** for the name of the file.

Click **Save**.



Close all documents.

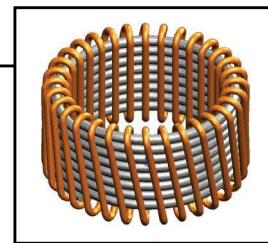


Advanced Sweep

Using 3D-Sketch Sweep Path

Advanced Sweep

Using 3D-Sketch Sweep Path

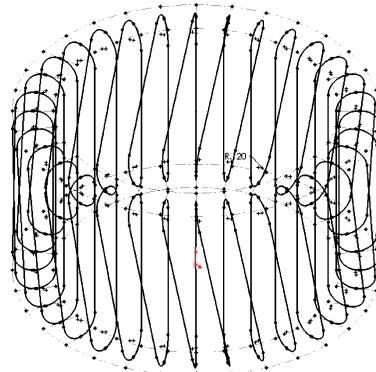


This second half of this chapter discusses one of the advanced techniques on creating a continuous wrapped wire around a coil.

There will be two separate sweep features created in this design. The first sweep feature is the coil, shaped like a spring, and the sweep profile is made tangent to the sweep path. To achieve this, a couple of construction lines are used to help locate one of its quadrant points on the path.

The second sweep feature, due to its complex shape, is done by using a 3D sketch as a sweep path. It is going to wrap around the first sweep feature in a unique way, and 3D sketch is one of the few options used to create it with.

The method in this lesson demonstrates the use of location points, created in two different sketches to guide a 3D sketch. The entities in the 3D sketch are connected to the pre-defined endpoints, and the geometry of the sweep path is formed.

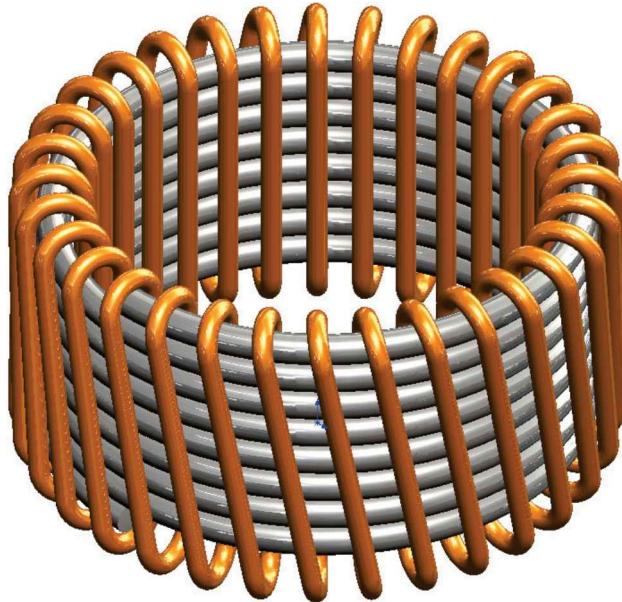


To ensure the location points are identical between the two sketches, the option Derived-Sketch is used. That way if one of the endpoints in the original sketch is moved, the same point in the derived sketch will also move.

The entities in the derived sketch are driven by the original sketch. Changes done to the original sketch are reflected in the derived sketch. To break the link between the derived sketch and its parent sketch, right-click the derived sketch from the FeatureManager tree and select **Underived**. After the link is broken, the derived sketch will no longer update when the original sketch is changed.

Advanced Sweep

3D-Sketch Sweep Path



Dimensioning Standards: ANSI
Units: INCHES – 3 Decimals

Tools Needed:



Insert Sketch



3D Sketch



Line



Add Geometric Relations



Sketch Fillet



Circle



Dimension



Centerline



Fillet/Round



Sketch Circular Pattern



Helix/Spiral



Sweep Boss/Base

1. Starting a new part document:

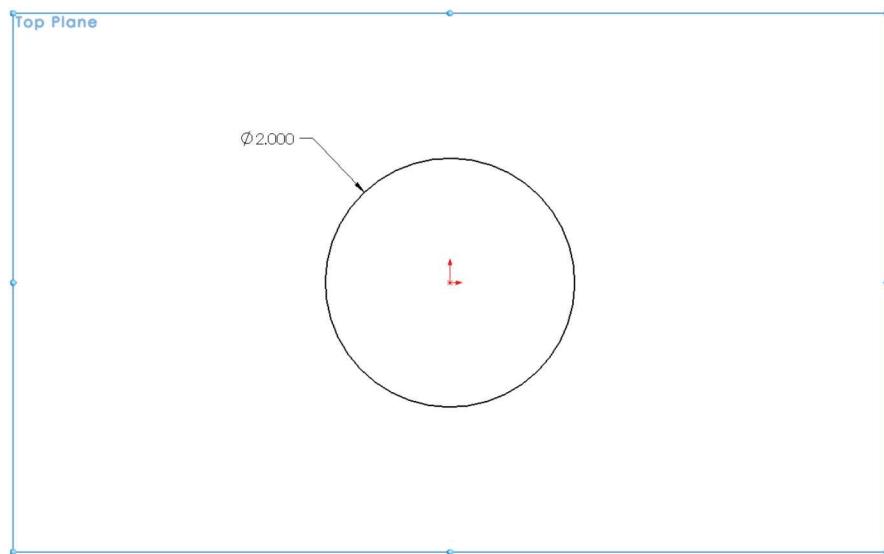
Go to File / New / Part.

Set the Drafting Standard to ANSI and the Units to Inches, 3 decimals.

Select the Top plane and open a new sketch .

Sketch a **Circle** centered on the origin as shown.

Add a Ø2.000" diameter dimension to fully define the sketch.

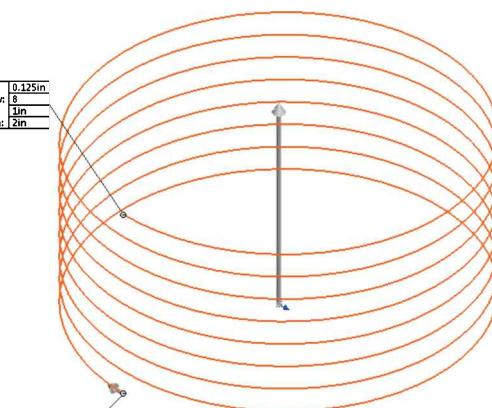
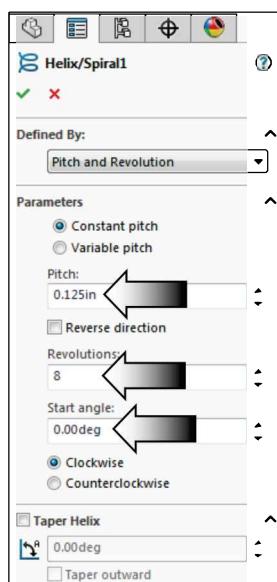


2. Creating the sweep path:

 Click the **Helix / Spiral** command on the Curves drop-down menu, or select: **Insert / Curve Helix-Spiral**.

Set the following:

- * **Constant Pitch**
- * **Pitch = .125"**
- * **Revolutions: 8**
- * **Start Angle = 0deg**
- * **Clockwise**



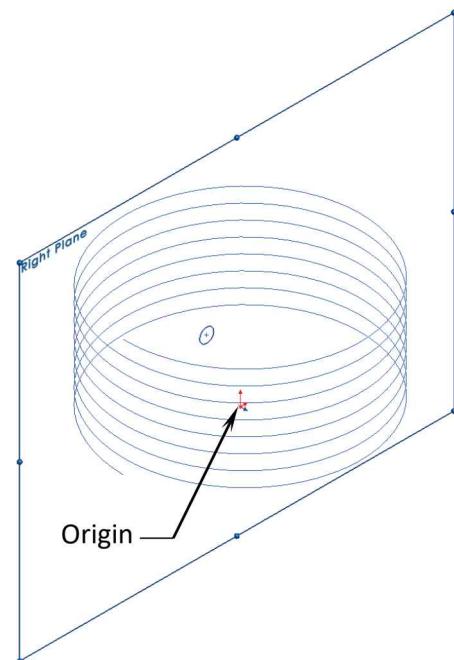
Click **OK**.

3. Creating the sweep profile:

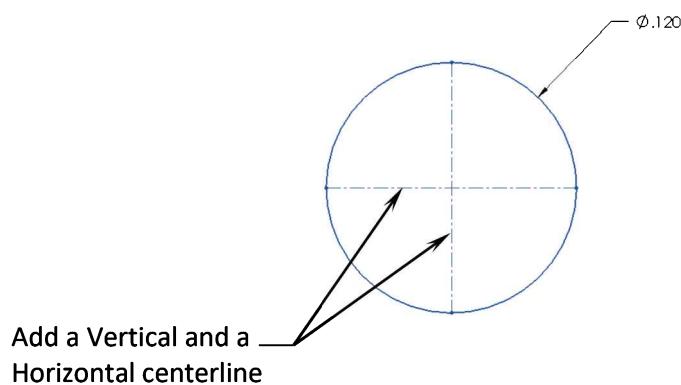
Select the **Right** plane and open a **new sketch**

Sketch a **Circle** away from the Origin.

The lower quadrant point of the circle must be pierced to the helix, and one way to achieve that is to create a couple of centerlines and then use one of the endpoints of the line and pierce it to the helix.

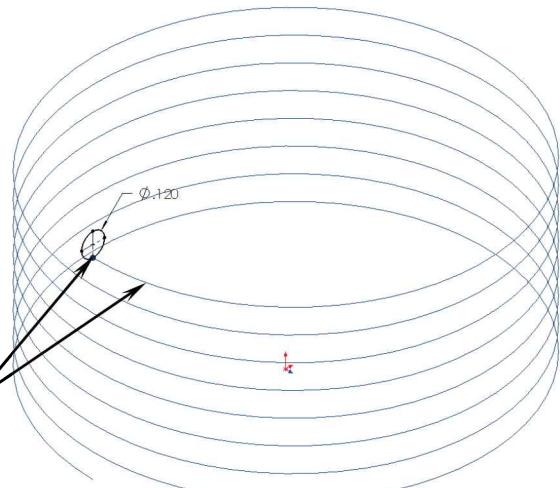
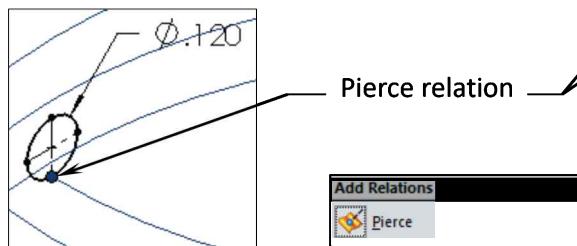


Add a **vertical** and a **horizontal centerline** as illustrated.



Add the dimension **$\text{Ø}.120"$** .

Add a **Pierce relation** between the endpoint at the bottom of the vertical centerline and the helix.



4. Creating the sweep feature:

Switch to the **Features** tab.

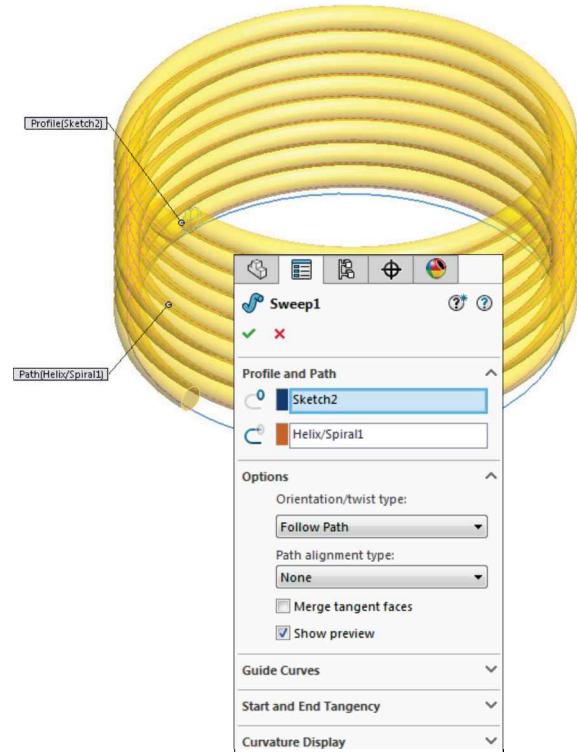
Click the  **Swept Boss /Base** command.

For Sweep Path, select the **Helix**.

For Sweep Profile, select the small **Circle**.

Keep all other parameters at their default settings.

Click **OK**.



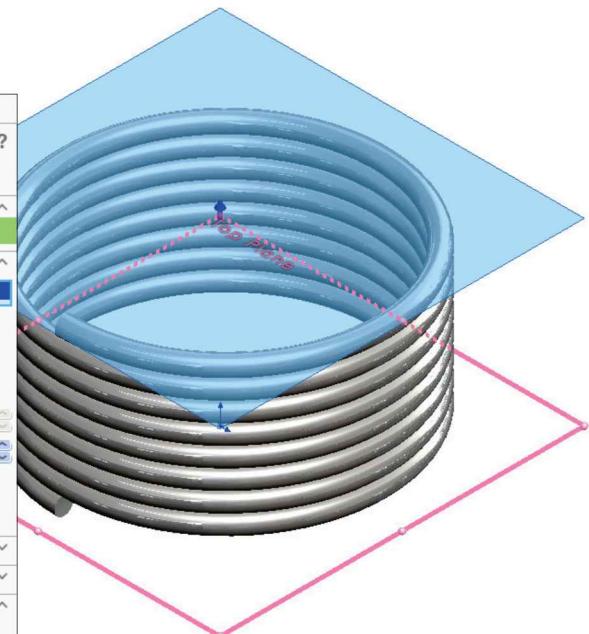
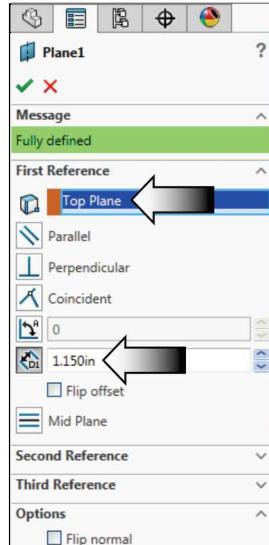
5. Creating the 1st offset plane:

From the **Features** tab, click **Reference Geometry, Plane** or select: **Insert, Reference Geometry, Plane**.

For First Reference select the **Top** plane from the FeatureManager tree.

For Offset-Distance enter: **1.150"**.

Place the new plane above the Top plane.



Click **OK**.

6. Creating the 2nd offset plane:

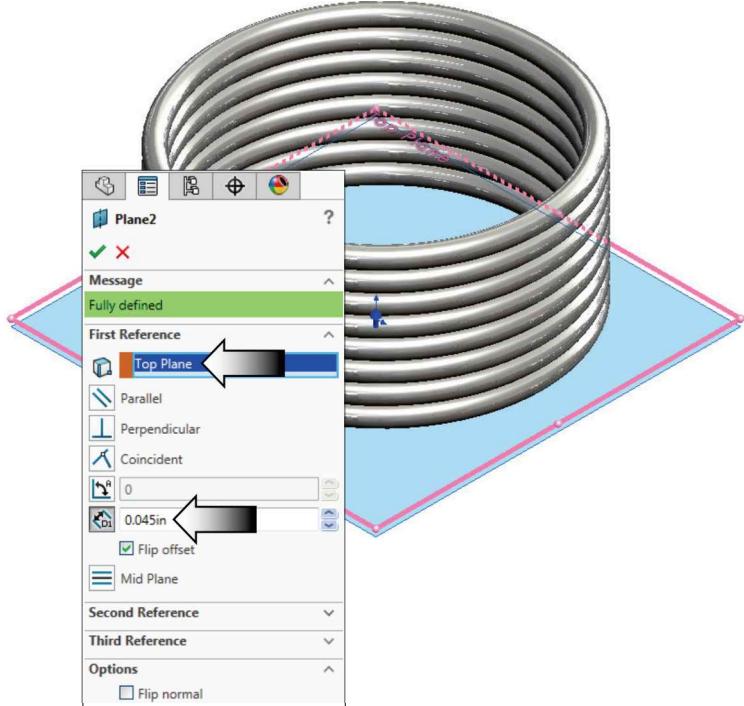
Click the **Plane** command  once again.

For First Reference
select **Top** plane.

Enter **.045"** for
Offset Distance.

Place the new plane
below the Top plane.

Click **OK**.



7. Sketching on the 1st offset plane:

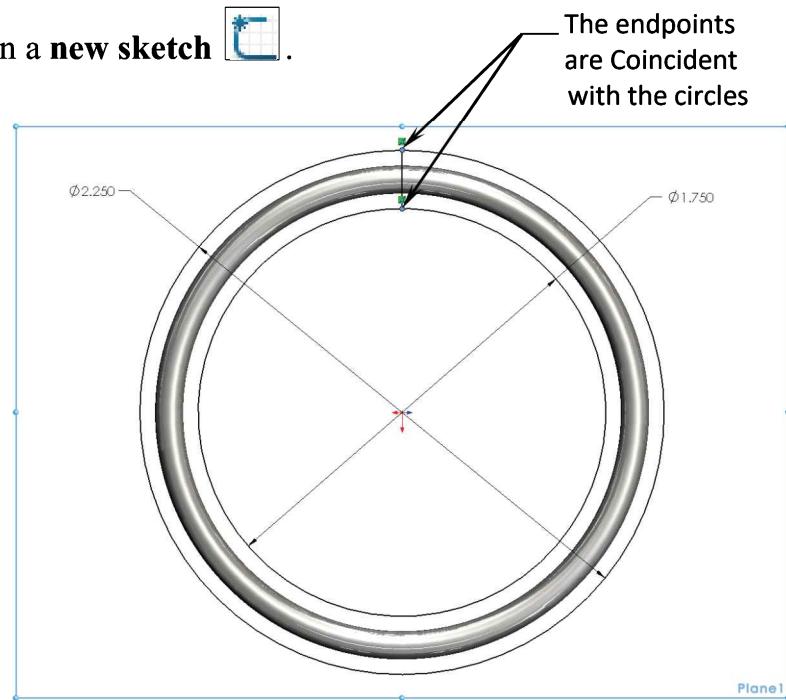
Select Plane1 and open a new sketch .

Sketch 2 Circles
that are centered
on the origin.

Sketch a vertical
Line that is
Coincident to
the two circles
as indicated.

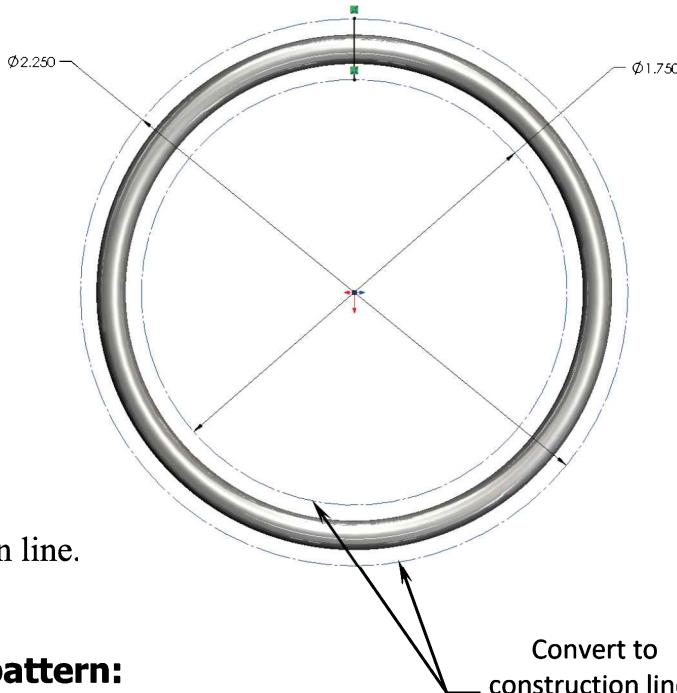
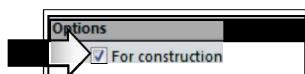
Add the two diameter
dimensions shown.

The endpoints
are **Coincident**
with the circles



Hold the **Control** key and select both circles.

Release the control key and click the **For Construction** checkbox on the Property tree.



This option toggles the selected entities back and forth between a line and a construction line.

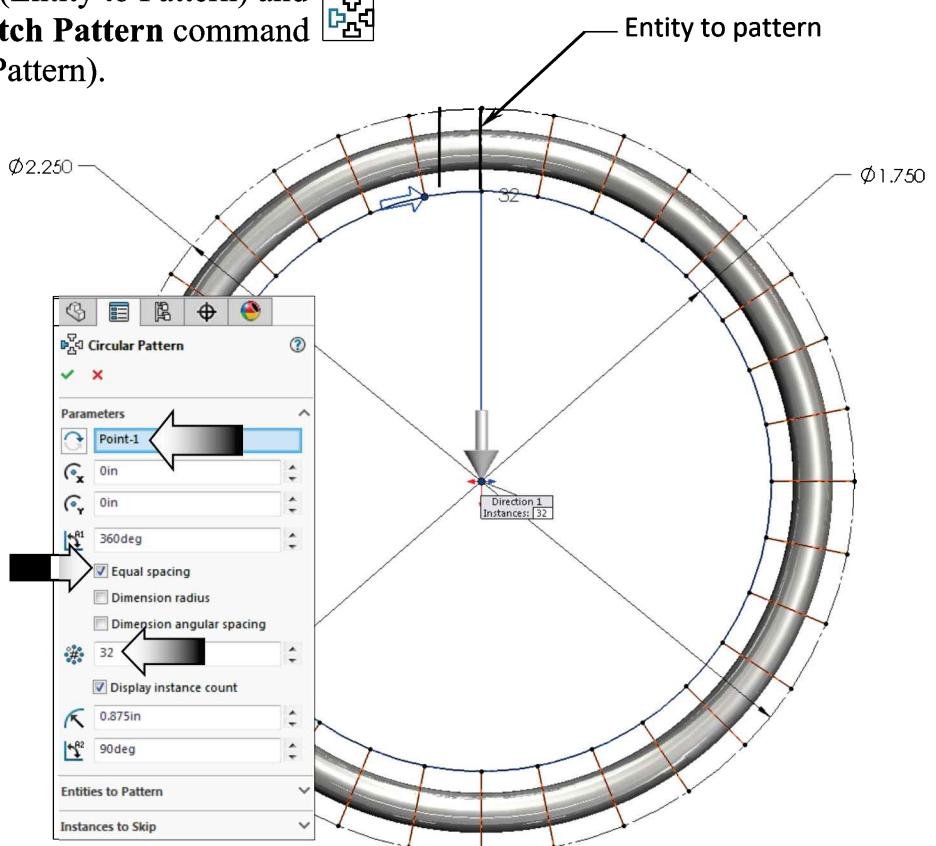
8. Creating a Circular Sketch pattern:

Select the vertical line (Entity to Pattern) and click the **Circular Sketch Pattern** command 

The **origin** is automatically selected as the center of the pattern.

Enable the **Equal-Spacing** checkbox.

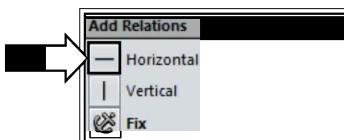
Enter **32** for the number of instances.



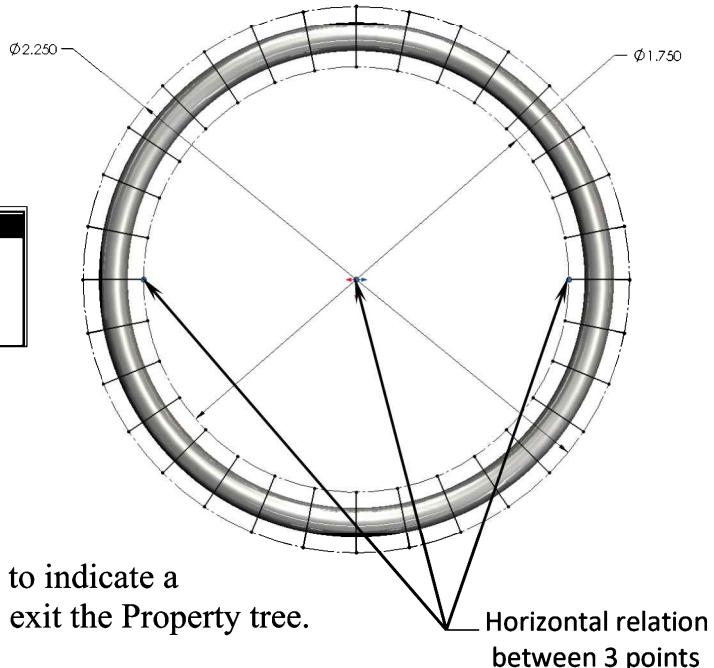
Click **OK**.

The pattern is still under defined at this point.

Hold the **Control** key and select the 3 points as noted.



Select the **Horizontal** relation from the Property tree.



The sketch color changes to black to indicate a Fully Defined status. Click **OK** to exit the Property tree.

Horizontal relation between 3 points

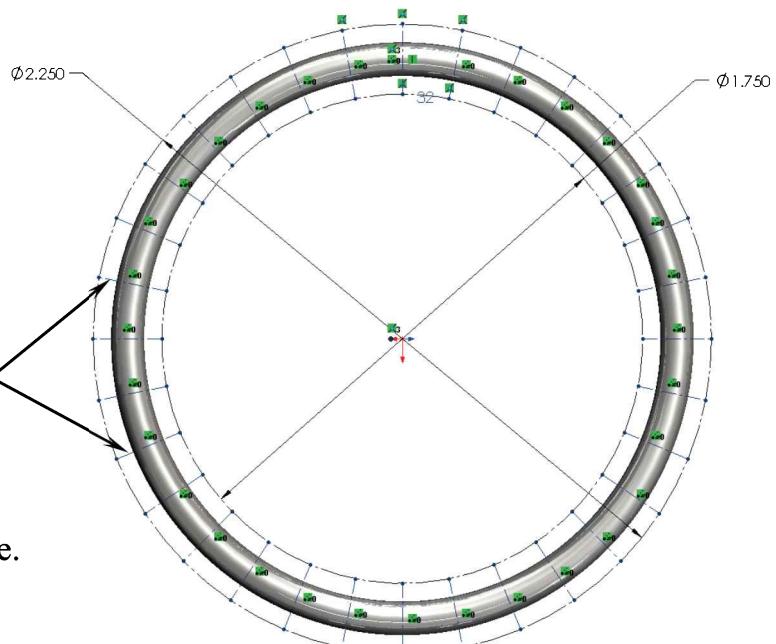
9. Converting to construction geometry:

The endpoints of each line will be used as location points, we should convert all the lines to construction lines at this point.

Either hold the Control key and select **all the lines**, or press Control+A to select all entities, then hold the control key and deselect the 2 circles.

Convert all lines to construction

Enable the **For Construction** checkbox on the Property tree.

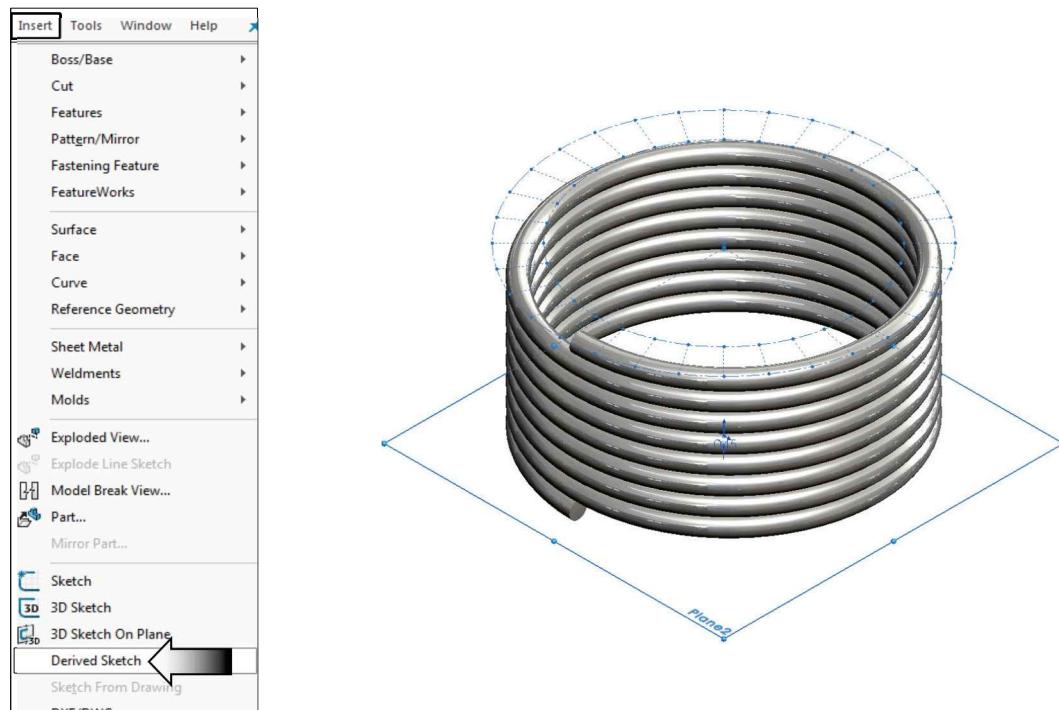


Exit the sketch.

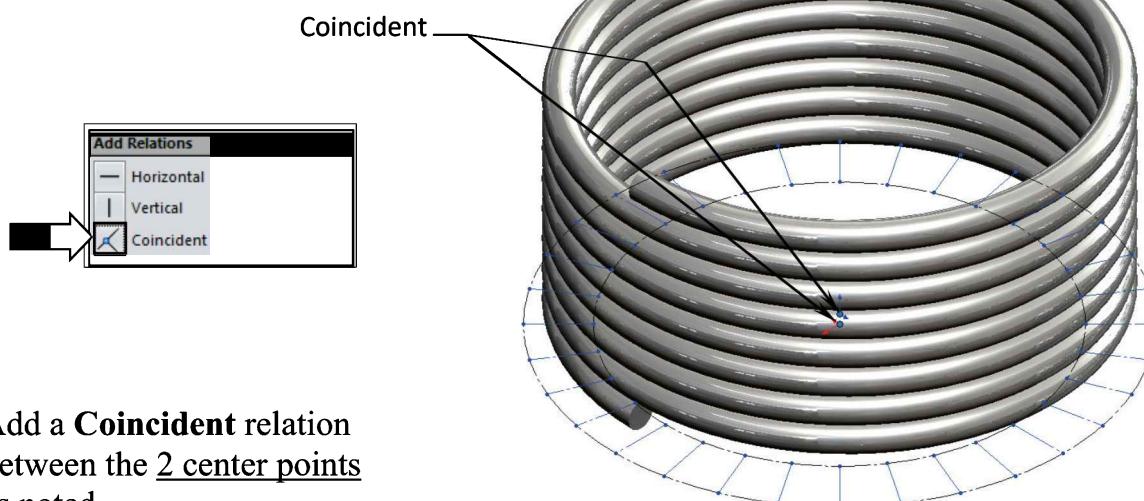
10. Creating a derived sketch:

Hold the **Control** key and select **Plane2** and **Sketch3** from the feature tree.

Click **Insert / Derived Sketch** (arrow).

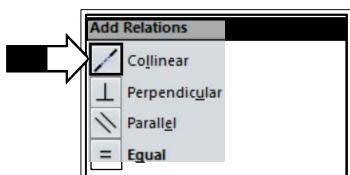


A copy of the sketch is created and placed on the selected plane.



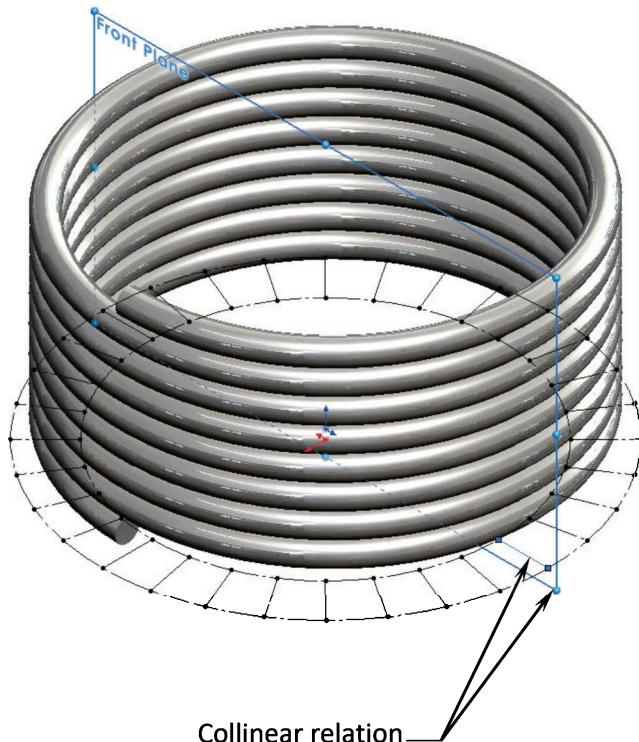
Add a **Coincident** relation between the 2 center points as noted.

Add a **Collinear** relation between the horizontal line on the right and the Front plane as indicated.



All entities in the sketch should change to the black color at this time (Fully Defined).

Exit the sketch.



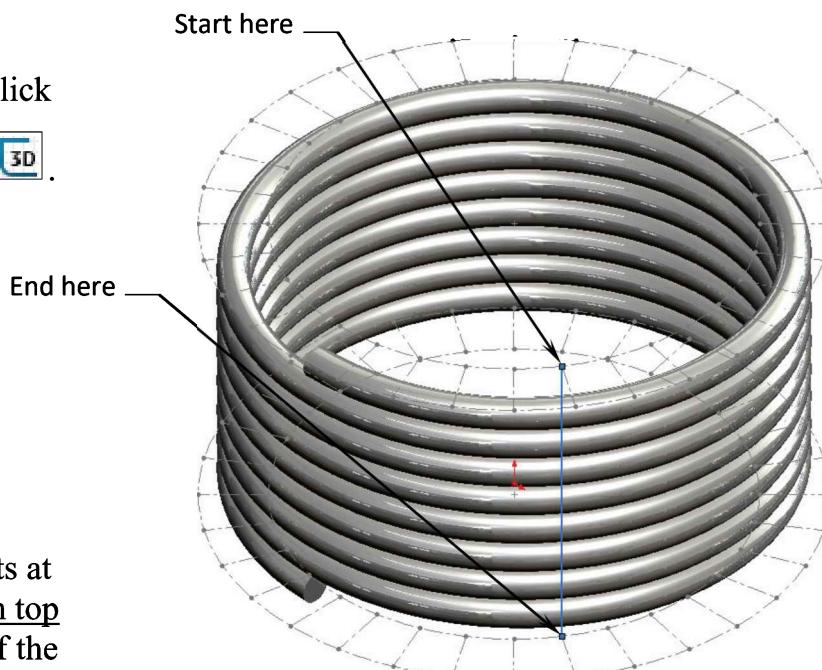
11. Creating a 3D sketch:

The endpoints of the lines will help us create the 3D sketch a little easier and also more accurately. Be sure both sketches are visible for this step.

From the **Sketch** tab, click the drop-down arrow and select **3D Sketch** .

Select the **Line** tool from the 3D-Sketch toolbar.

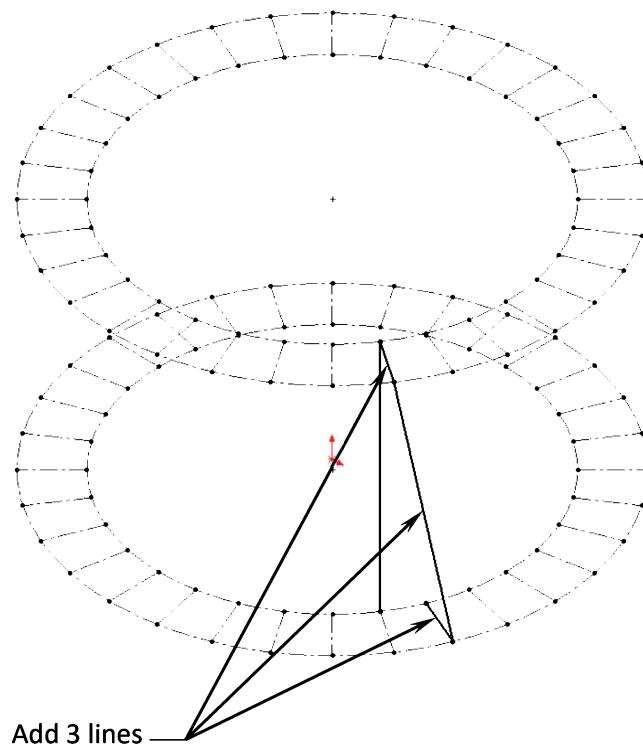
Sketch a **Line** that starts at one of the endpoints on top and connect it to one of the endpoints on the bottom.



The feature Swept1 is hidden for clarity.

(Use the Click + Release technique to create multiple lines, while the Click + Hold + Drag creates only 1 line each time.)

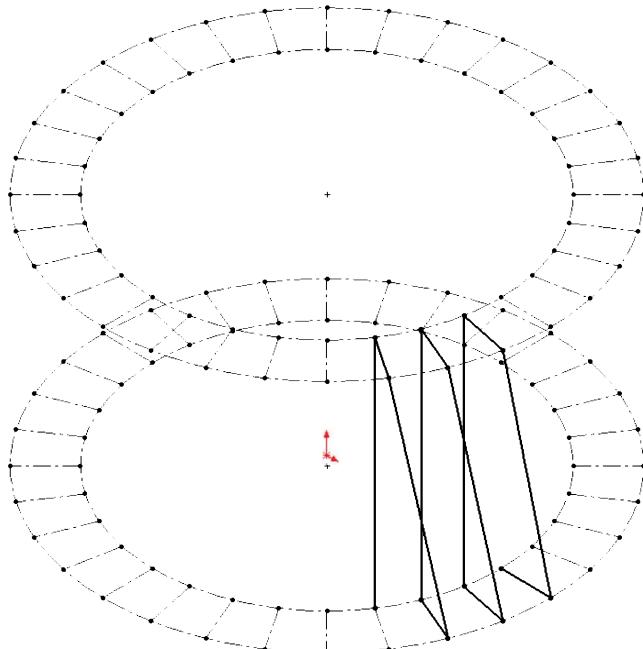
Add 3 more lines as shown.
Be sure to snap to each existing endpoint when sketching the lines.



Continue to add the other lines as shown below. If you run into an error, simply press Escape and start sketching the lines again.

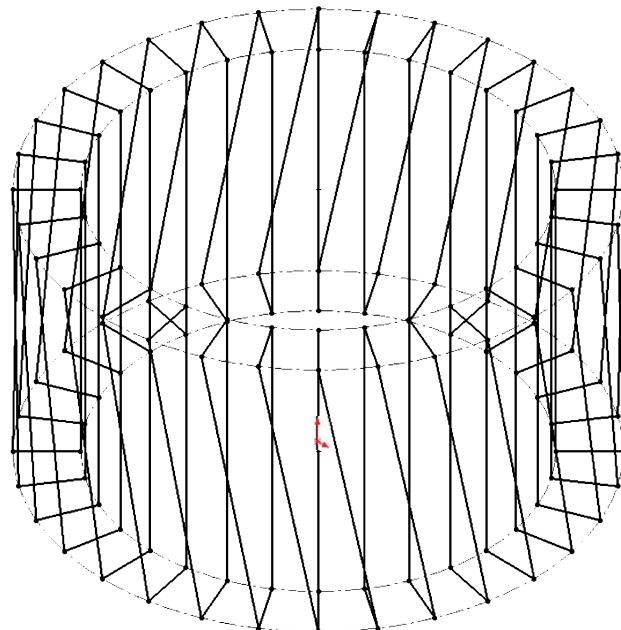
It is maybe a little easier if you could change the orientation of the view after adding a few lines.

While in the middle of the line mode you can press and hold the scroll-wheel to rotate slightly to a different angle, then continue with your sketch.



Once completed, your sketch should look like the image shown on the right.

Do not exit the sketch just yet; the sketch fillets will be added next.

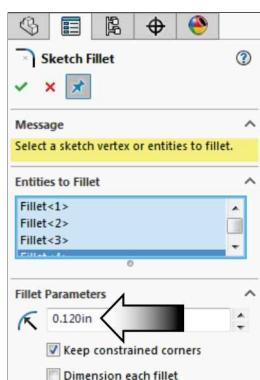


12. Adding the sketch fillets:

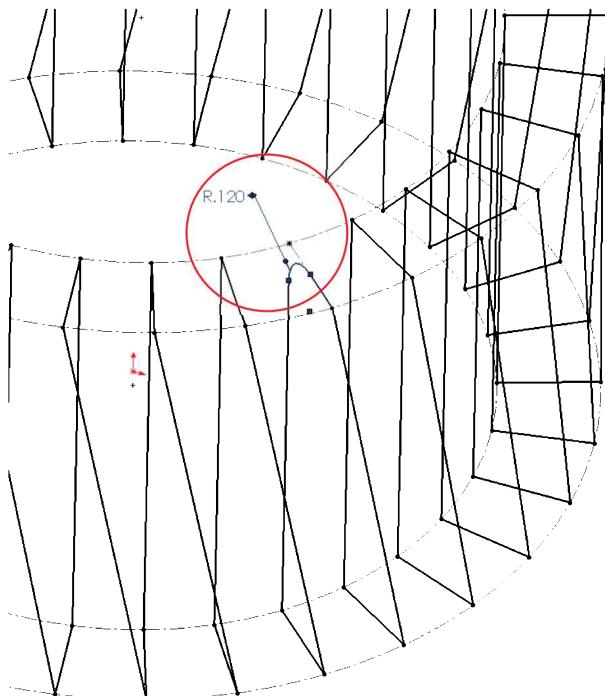
When creating the Sketch Fillets you can either select the 2 lines, or click directly at the intersection point between the 2 lines to add the fillets. If a fillet gets deleted the 2 lines will be extended and form a sharp corner once again.

Click the **Sketch Fillet** command from the **Sketch Tools** tab.

Enter **.120"** for radius value.

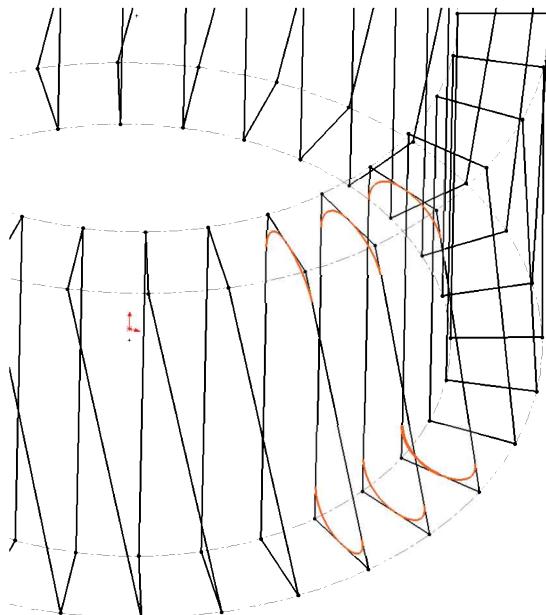


Begin adding the fillets by clicking at the intersections of the lines.

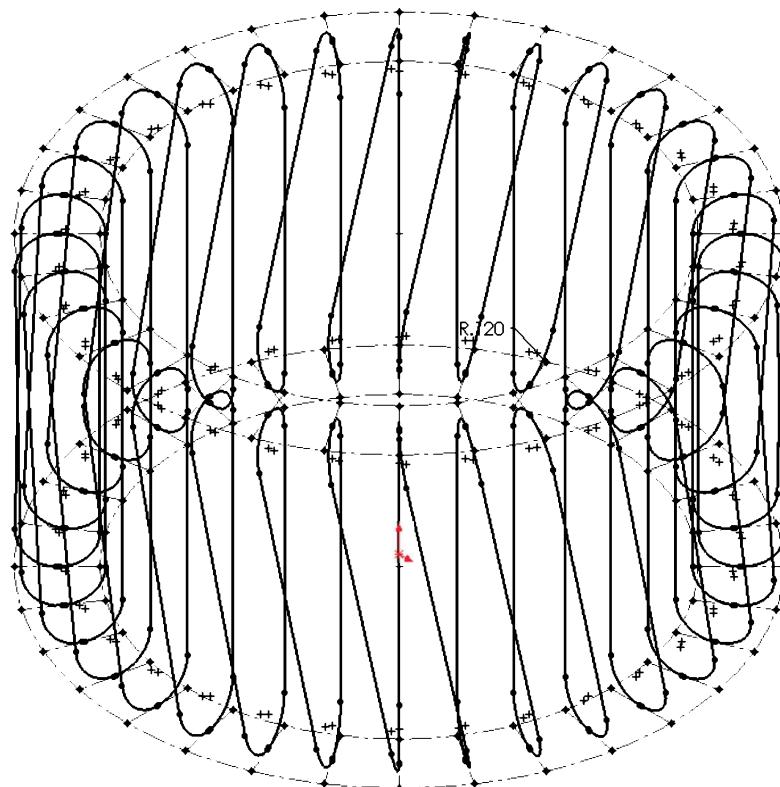


Continue with adding the same fillet to the intersection of the other lines.

It does not really matter which direction you want to go when adding the fillets (clockwise or counterclockwise), but it will be a little easier if you go around 1 direction and not to jump back and forth as you may accidentally skip one or more vertices.



After completed, your sketch should look like the image shown below. This 3D-sketch will be used as the sweep path in the next few steps.



Exit the 3D sketch.

13. Creating the sweep feature:

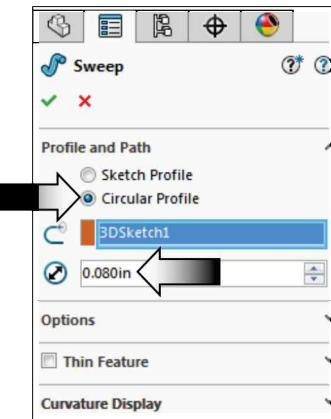
Switch to the **Features** tab.

Select the **Swept Boss/Base** command .

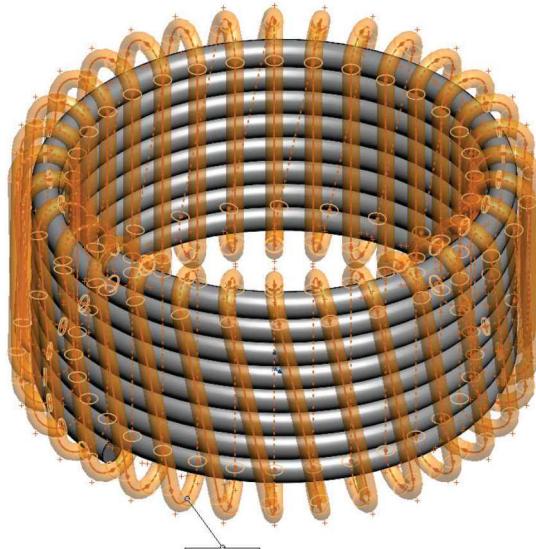
Select the **Circular Profile** option (arrow).

Enter **.080in** for Profile Diameter (arrow).

For Sweep Path, select the **3D Sketch**.



Click **OK**.

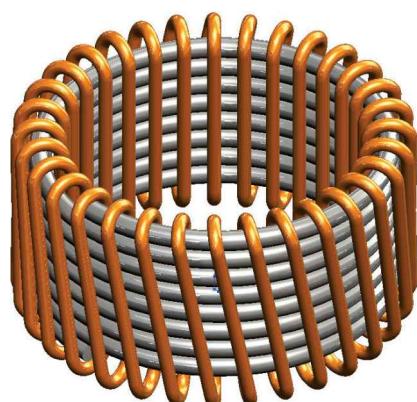
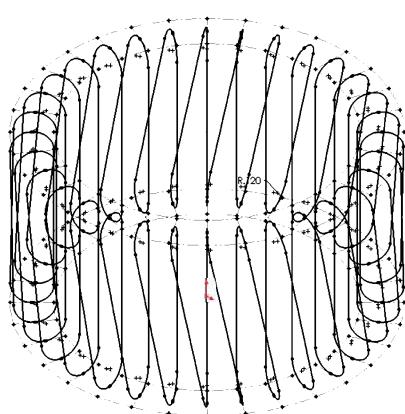


14. Saving your work:

Click **File / Save As**.

For the file name, enter: **Advanced Sweep_Wire Form**.

Select a location to save the file and click **Save**.



Exercise: Using Curve Through Reference Points

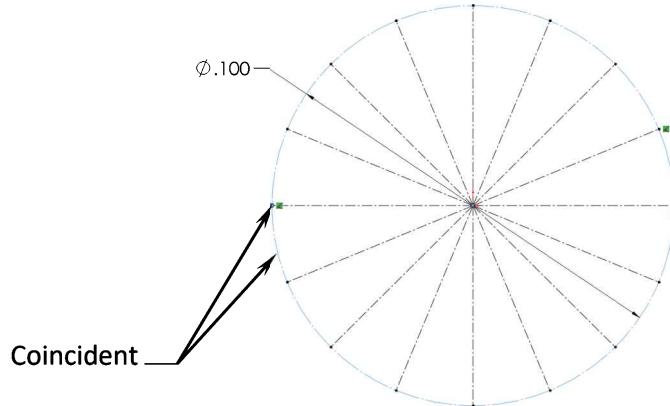
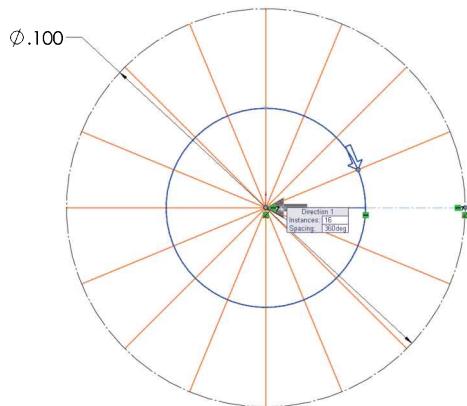
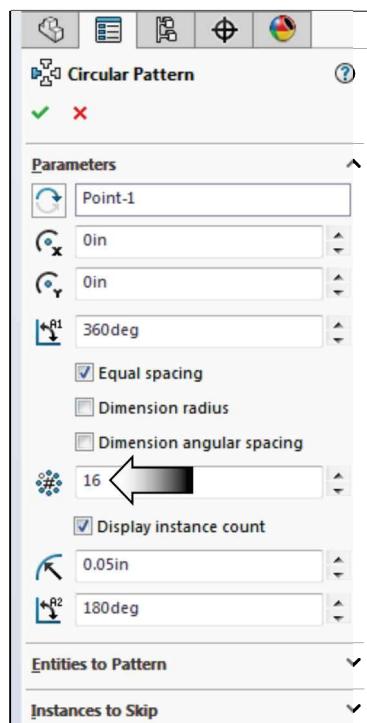
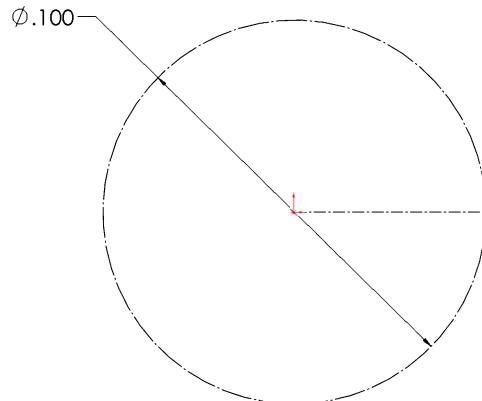
1. Creating the Base sketch:

From the **Top** plane sketch a **Circle** and a horizontal **Centerline**, then convert both entities to **Construction** geometry.

Add a diameter dimension of **.100"**.

Select the horizontal **Centerline** and click **Circular Sketch Pattern**.

Enter **16** instances, **Equal Spacing**.

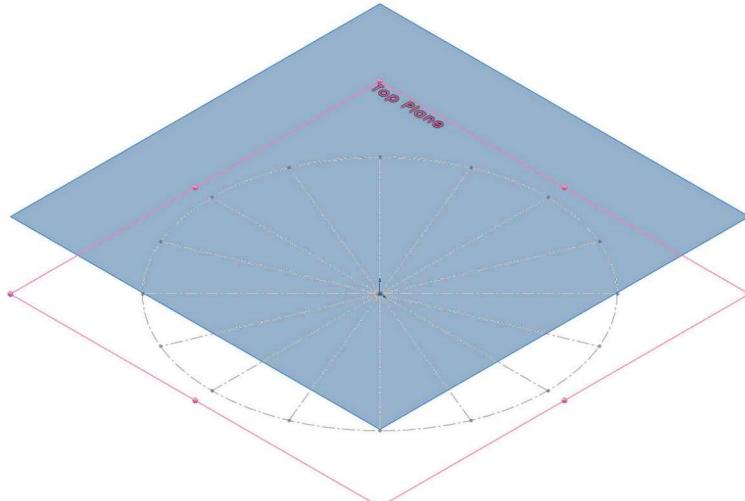
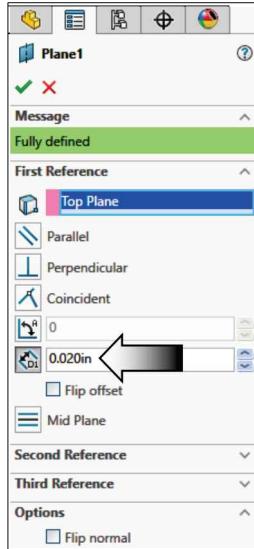


Add a couple of **Coincident** relations between the endpoints of any of the instances and the circle, to fully define the sketch.

Exit the sketch (or press **Control +Q**).

2. Creating an offset plane:

Create a new plane that is **.020"** above the **Top** plane.



Click **OK**.

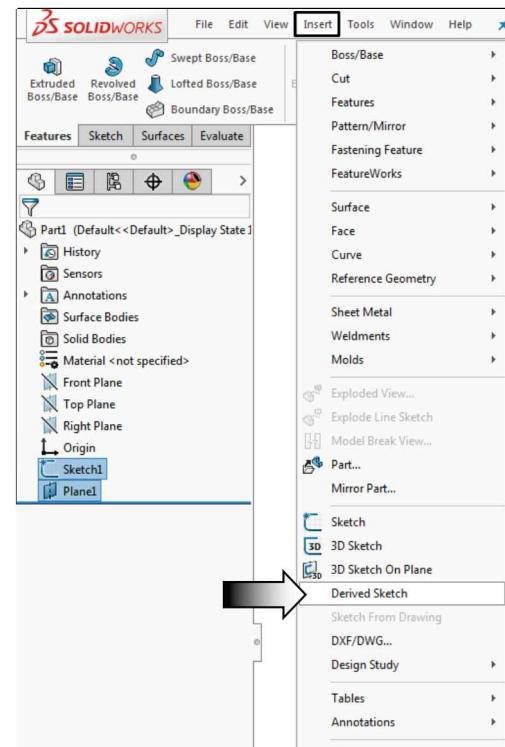
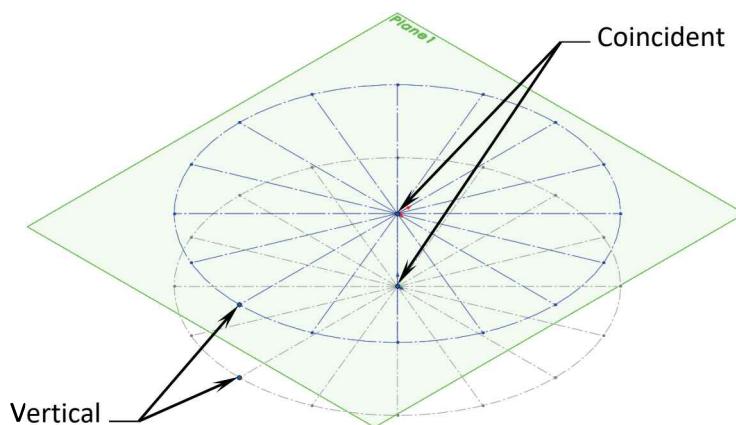
3. Creating a Derived sketch:

Hold the **Control** key and select **Sketch1** and **Plane1**.

Click **Insert / Derived Sketch**.

Add a **Coincident** relation between the centers of the 2 circles and a **Vertical** relation between any 2 endpoints.

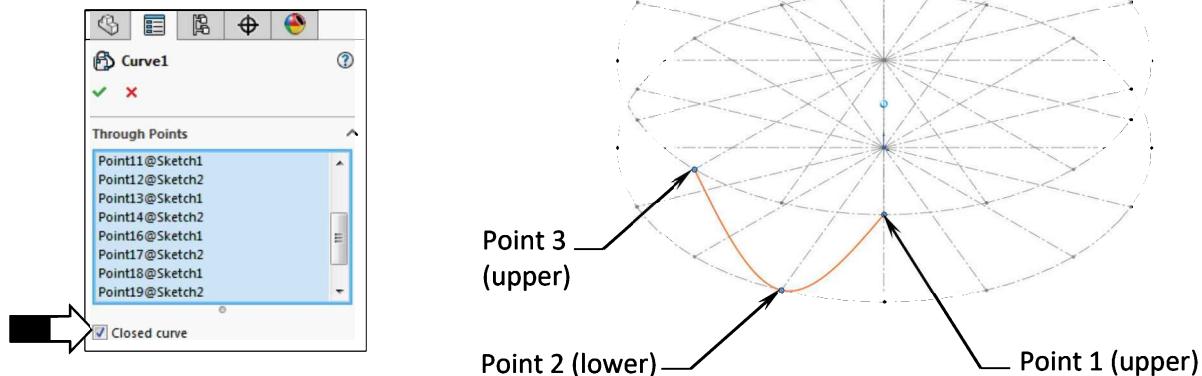
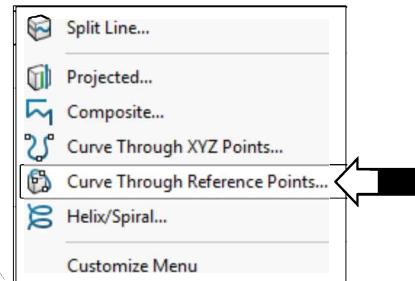
Exit the sketch.



4. Creating a Curve Through Reference Points:

From the **Features** tab, click:
Curves / Curve Through Reference Points.

Click the **starting point** (point 1), move clockwise and click **point 2**, then **point 3** so on.

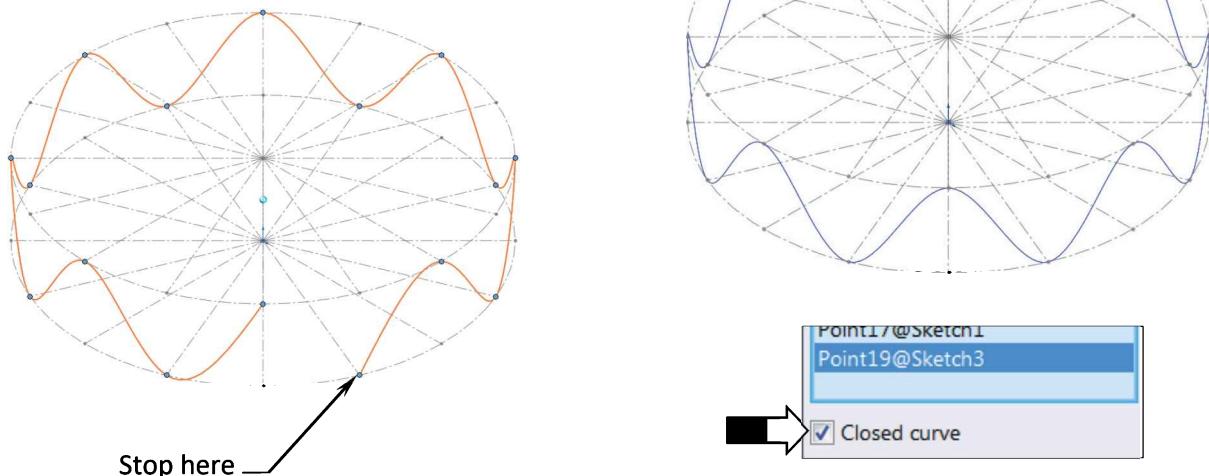


Continue to go around a full revolution and select all the connecting points in the 2 sketches.

At the end of the path, do not click point 1 again as the **Closed Curve** option will join the 2 ends together automatically.

Enable the **Closed Curve** checkbox.

Click **OK**.

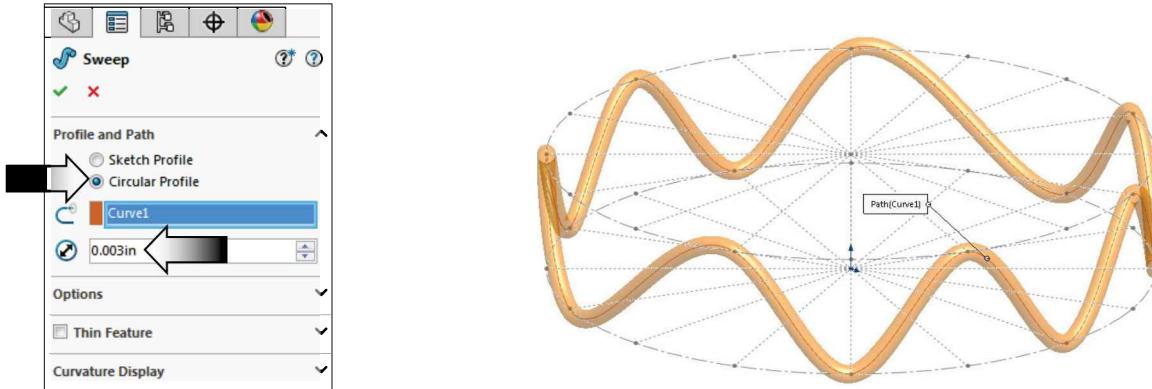


5. Creating a swept feature:

From the Features tab, click **Swept Boss Base**.

Select the **Circular Profile** option and enter **.003in** for diameter (arrows).

Select the **3D curve** as the Sweep path.



Click **OK**.

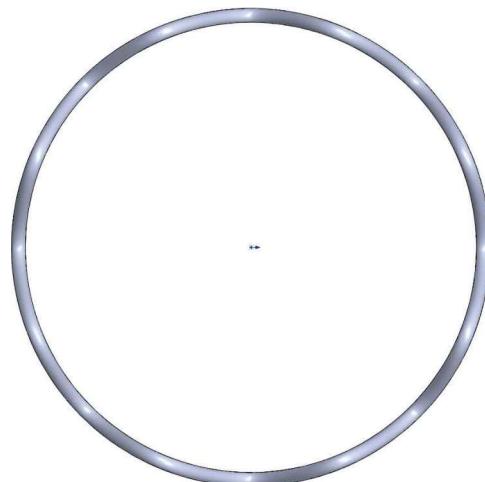
Hide the construction sketches and the 3D curve.

6. Saving your work:

Click **File / Save as:**

Enter: **Curve Through Reference Points**
for the name of the file.

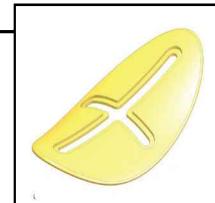
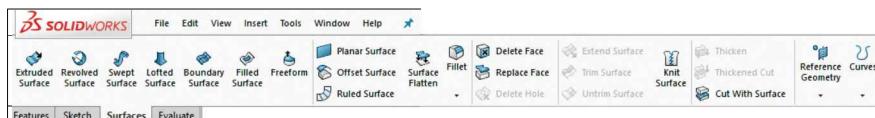
Click **Save**.



CHAPTER 8

Using Surfaces

Advanced Modeling - Using Surfaces



Surfaces are a type of geometry that can be used to create solid features.

The surface options are used to form complex free-form shapes and to manipulate files imported from other CAD formats.

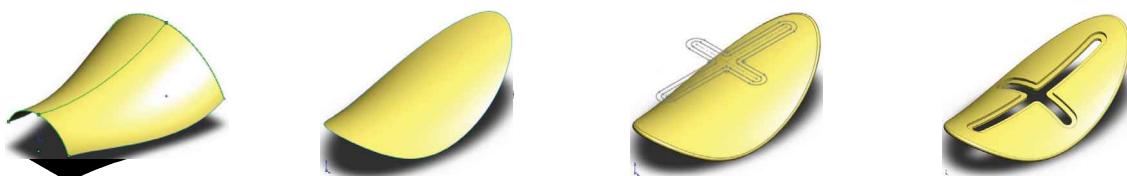
Unlike solid models, surfaces can be opened, overlapped, and have no thickness. Each surface can be constructed individually and then knitted together. A solid feature is created by thickening the surfaces that have been knitted into a closed volume.

Surfaces can be modeled in any shape and their sketches can either be extruded, revolved, swept, or lofted into a surface. These surfaces can also be replaced and filled with other surfaces.

The edges of the surfaces can be extended and trimmed. Surfaces can be moved, rotated, and copied.

The angle between the faces of the surfaces can be calculated using the Draft-Analysis tool: Positive Drafts, Negative Drafts, and Required Drafts are reported on screen.

This 1st half of the chapter discusses the use of some surfacing tools in the newer releases of SOLIDWORKS.



Advanced Modeling Using Surfaces



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI

Units: INCHES – 3 Decimals

Tools Needed:



Insert Sketch



3 Point Arc



Ellipse



Dimension



Add Geometric Relations



Split Line



Plane



Lofted Surface



Surface Thicken

1. Constructing a new work plane:

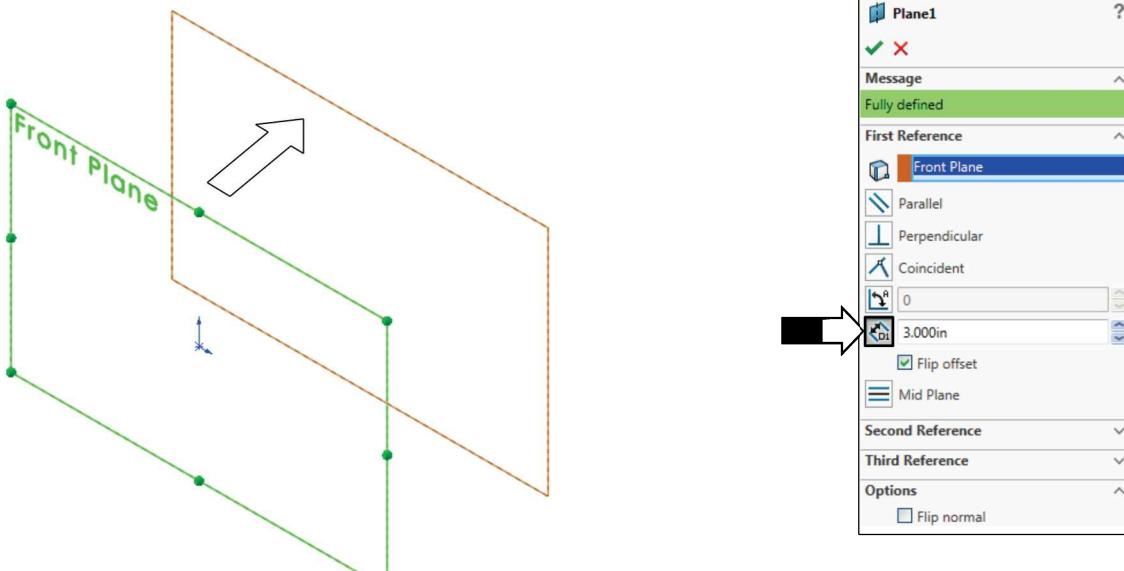
Select the Front plane from the FeatureManager tree.

Click  or select **Insert / Reference Geometry / Plane**.

Click the **Offset Distance** button and enter **3.000in**.

Enable the **Flip Offset** check box to reverse the direction.

Click **OK**.



2. Sketching the 1st profile:

Select the new plane (Plane1).

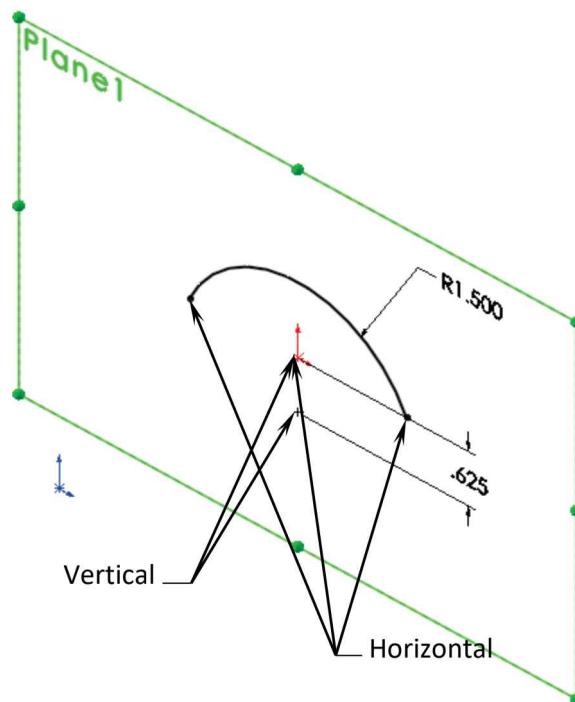
Click  or select **Insert / Sketch**.

Sketch a **3-Point-Arc** and add the dimensions shown on the right.

Add a **Vertical** and **Horizontal** relation as noted.

NOTE: The center of the Arc is .625" below the origin.

Exit the Sketch .



3. Sketching the 2nd profile:

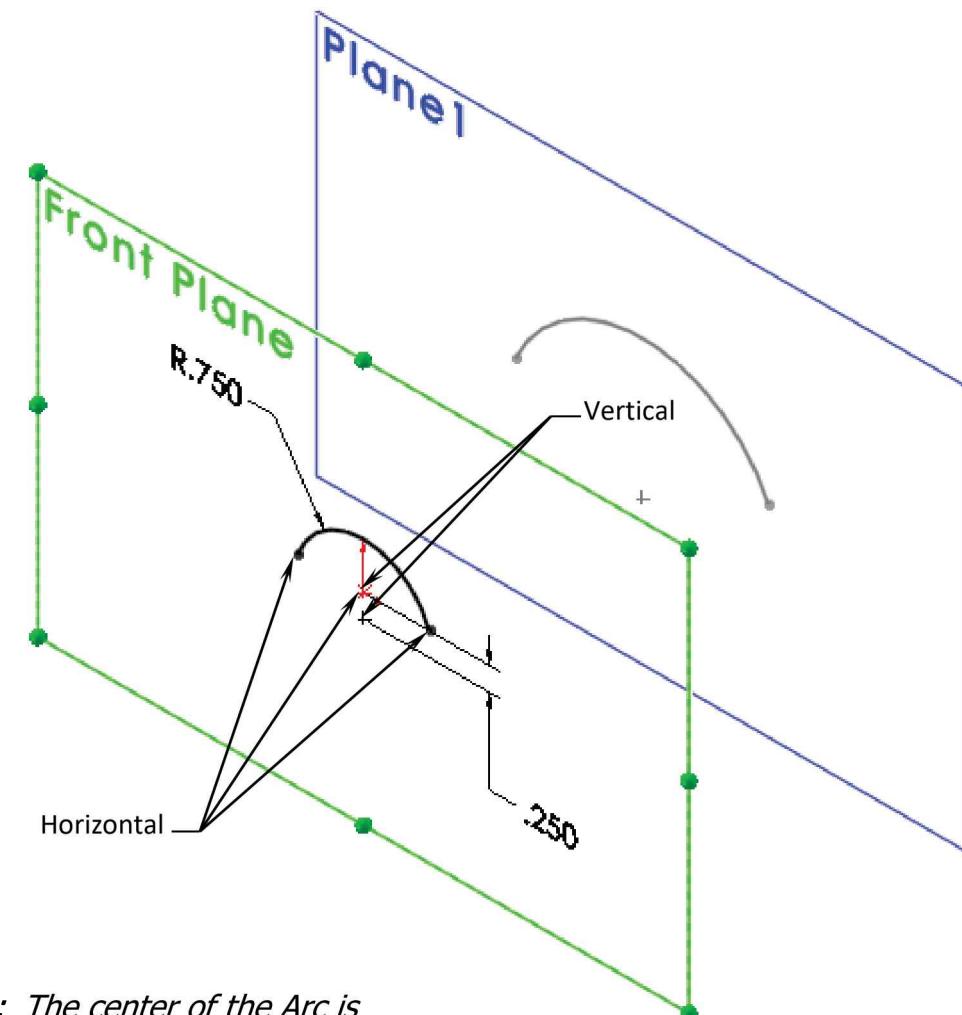
Select the Front plane from the FeatureManager tree.

Click  or select **Insert / Sketch**.

Sketch a **3-Point-Arc** as shown.

Add a **.750in** radius dimension to the arc.

Add a **Horizontal** and a **Vertical** relation as indicated.



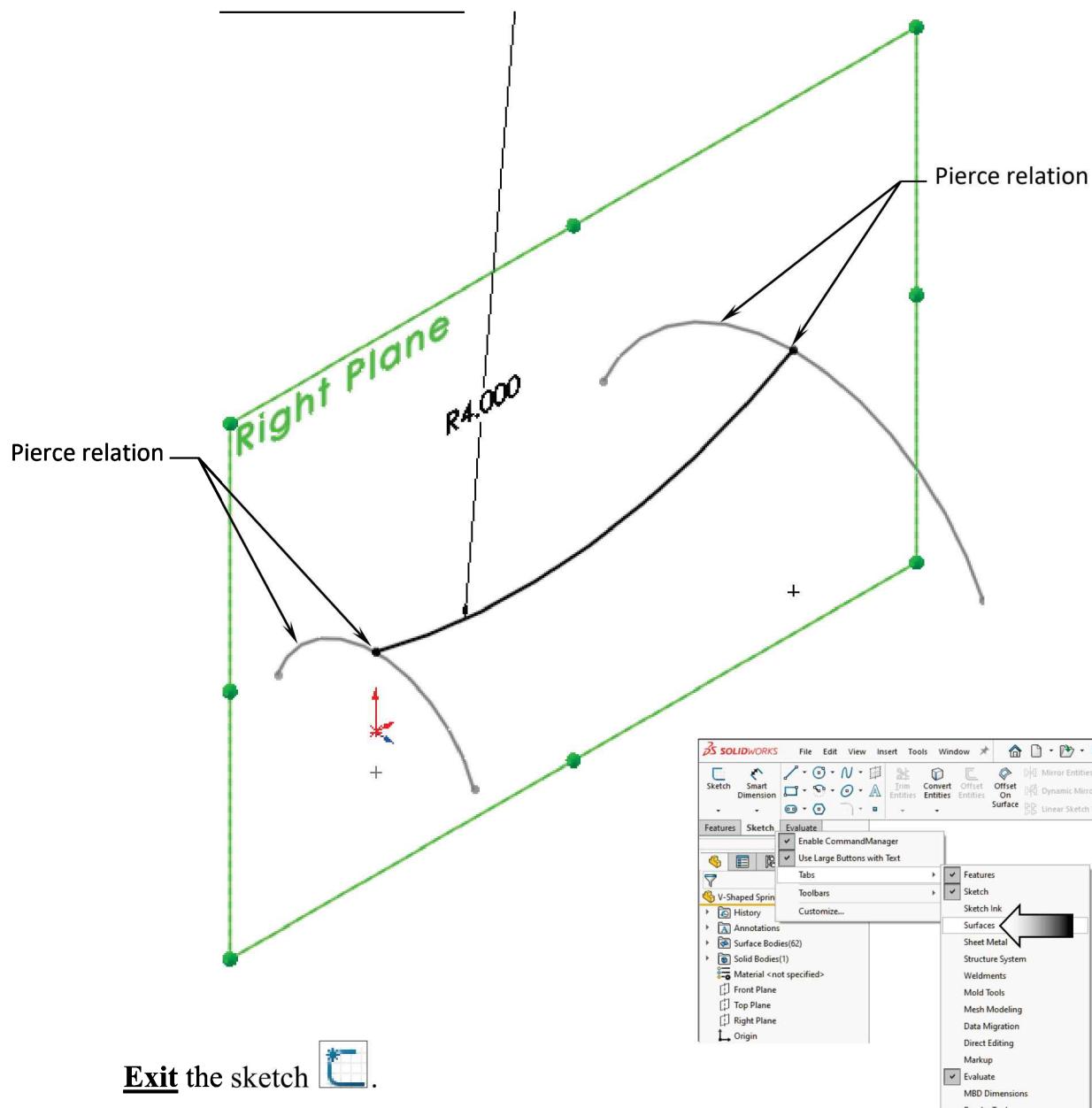
Exit the sketch .

4. Sketching the Guide Curve:

Select the Right plane from the FeatureManager tree and open a **new sketch**.

Sketch a **3-Point-Arc** and add a **4.00in.** dimension.

Add the **Pierce** relations as noted.



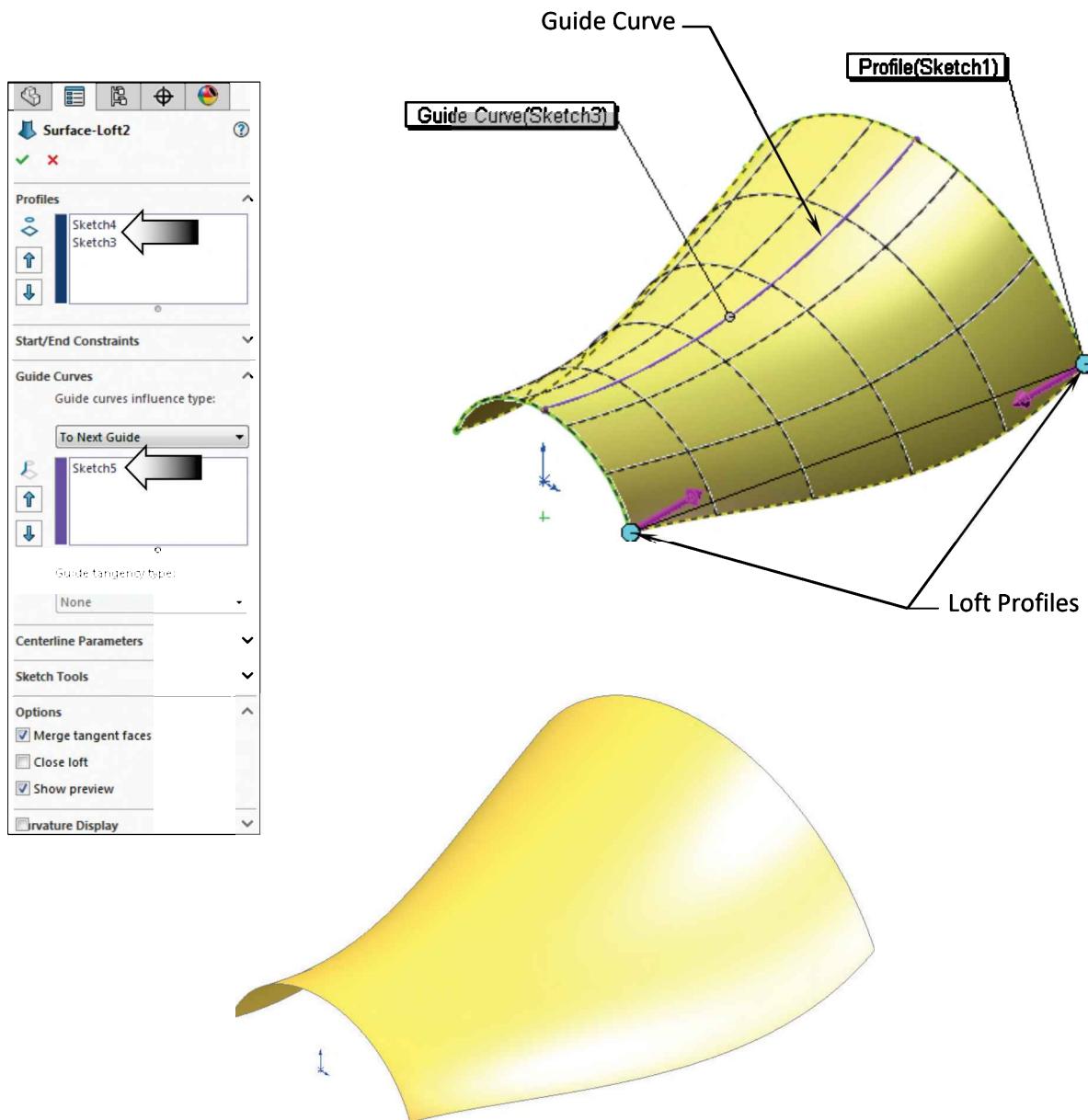
Enabling the Surfaces toolbar: Right-click one of the tabs (Features, Sketch, etc.) and enable the Surfaces option, then switch to the Surfaces tab.

5. Creating a Surface-Loft:

Click  on the **Surfaces** tab or select: **Insert / Surface / Loft**.

Select the **2 Sketched Profiles** by clicking on their *right-most endpoints*.

Expand the **Guide-Curve** section and select the **Arc** in the middle (Sketch3).



Click **OK**.

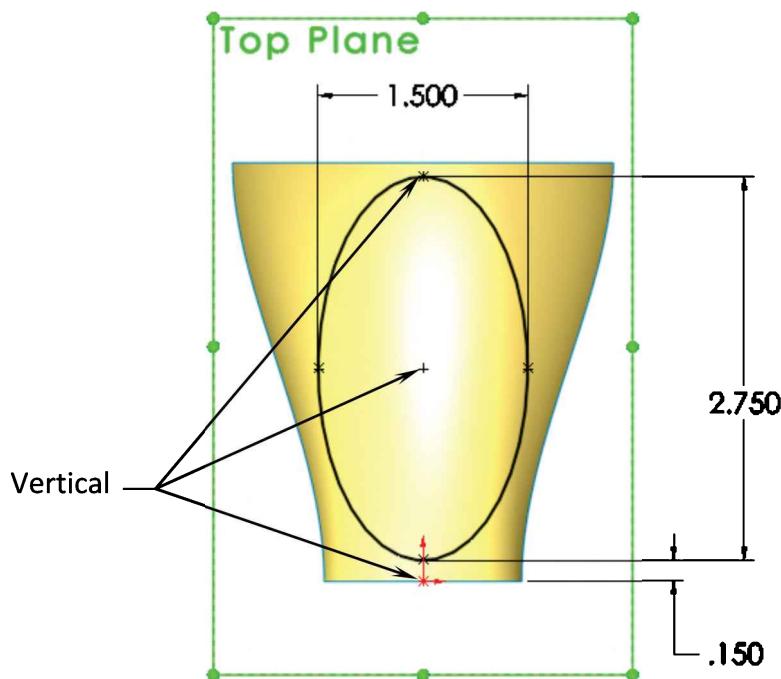
6. Sketching the Split profile:

Select the **Top** plane from the FeatureManager tree.

Click  or **Insert / Sketch**.

Change to the **Top** view orientation (Control+5).

Sketch an **Ellipse**  and add dimensions/relations as shown below.



Split Lines

The **Split Lines** command projects a sketch entity onto a face (or a group of faces) and divides the selected face(s) into multiple separate faces, enabling you to select and work with each face individually.

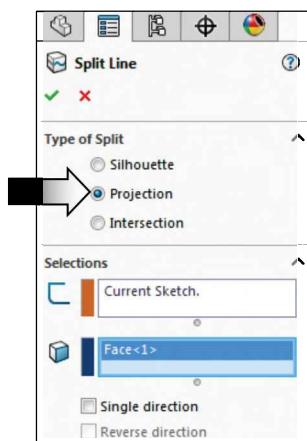
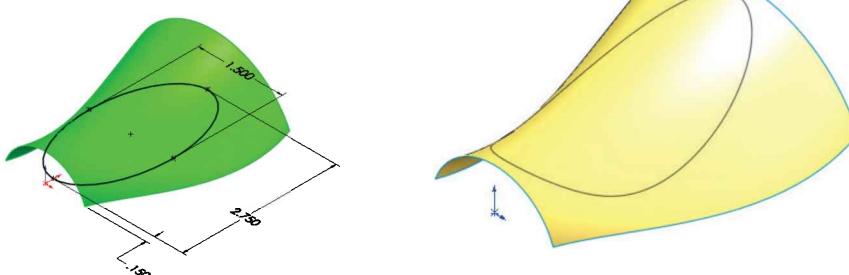
7. Splitting the surface:

Click  on the **Curves** tab OR select **Insert / Curve / Split Line**.

For Sketch-to-Project, select the **Ellipse**.

For Faces-to-Split, select the **Surface-Loft1**.

Click **OK**.



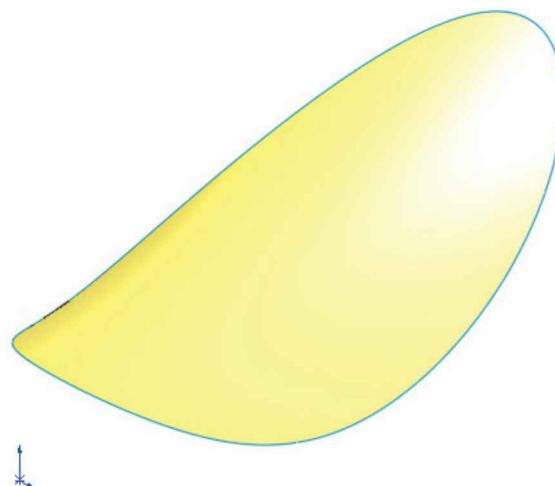
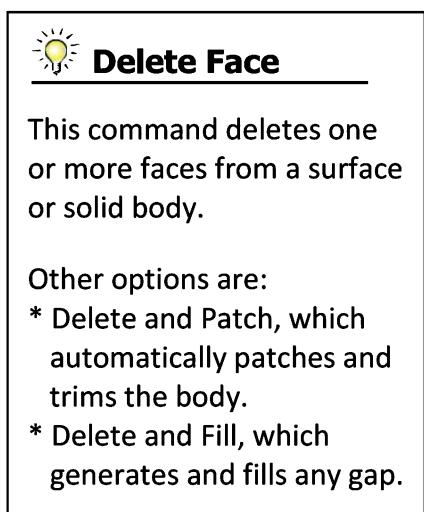
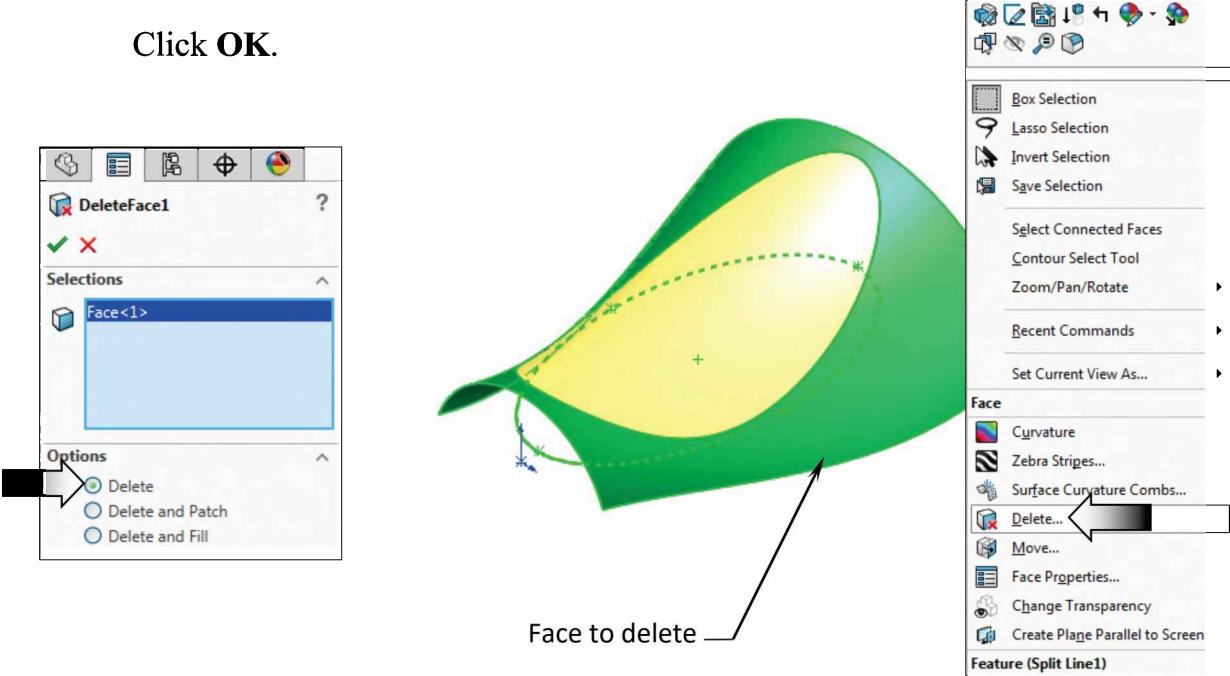
8. Deleting Surfaces:

Right-click on the **outer portion** of the surface.

Select **Face / Delete** from the menu.

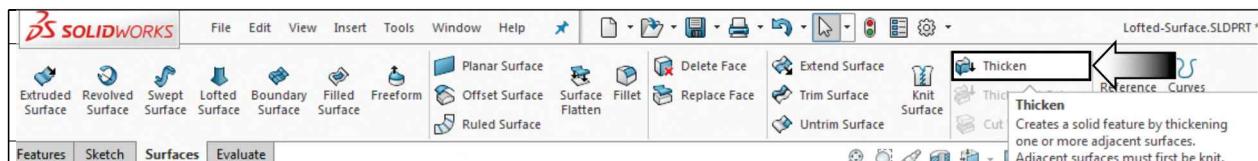
Click **Delete** under Options (circled).

Click **OK**.



Your trimmed surface should look like the one shown here.

9. Thickening the surface:

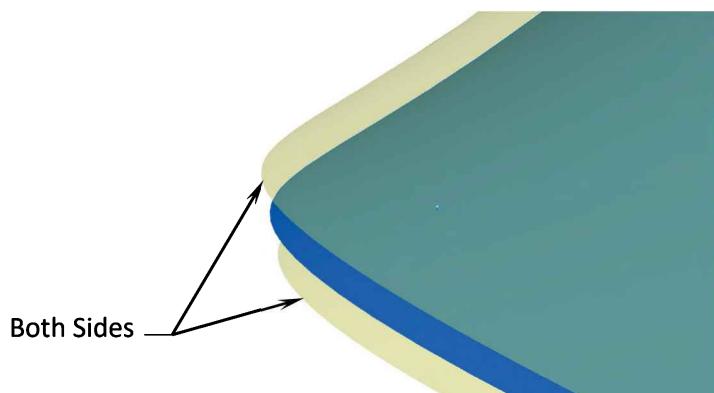


Click or select Insert / Boss-Base / Thicken.

For Surface-To-Thicken, select the **Trimmed Surface**.

Select the **Thicken Both Sides** option .

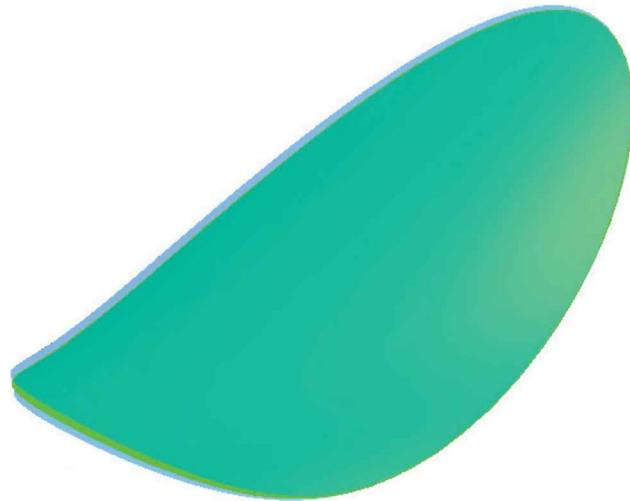
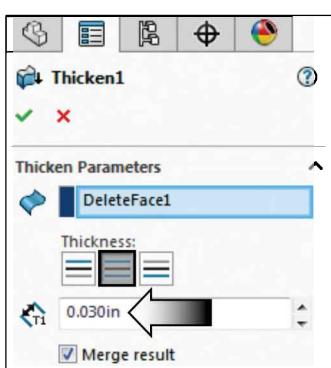
For Thickness, enter **.030** in. (.060 total thickness).



Thicken Surfaces

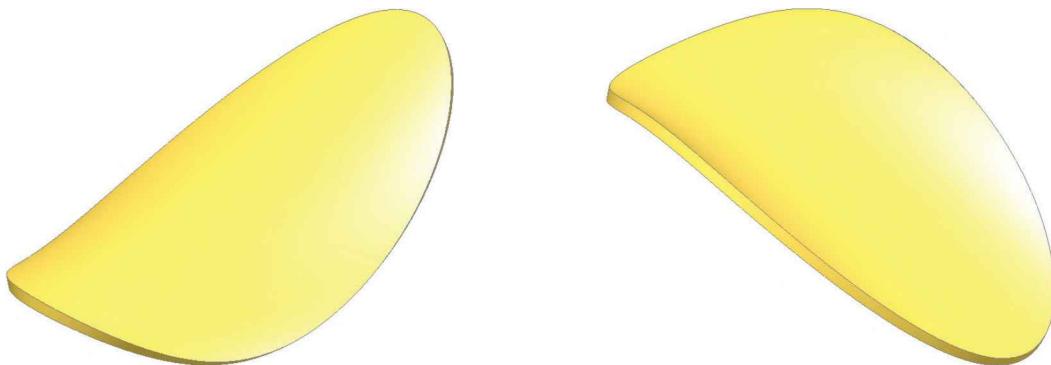
In order to create a solid volume, all surfaces have to form a closed shape.

If the shape is open, a wall thickness can be added to the surface to close.



Click **OK**.

The surface model is thickened into a solid model.



10. Calculating the angles between the faces:

Switch to the **Evaluate** tab and click the **Draft Analysis**  command.

For Direction of Pull, select the **Top** plane.

Enter **1.00deg.** for Draft Angle.

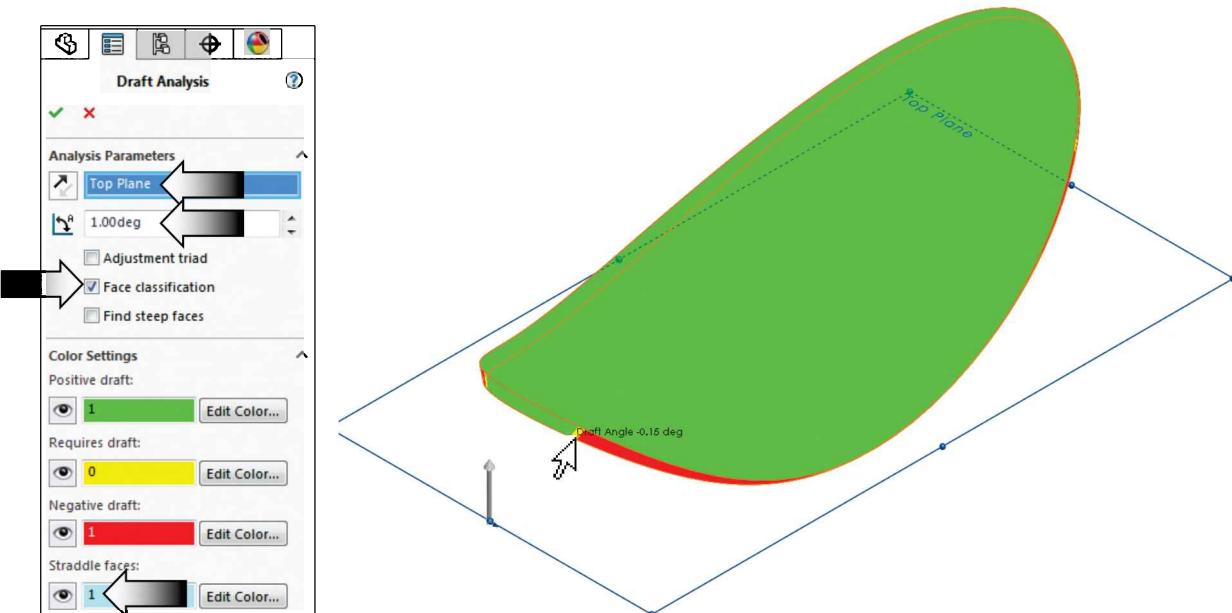
Enable the **Face Classification** checkbox and click **Calculate**.

The **light blue color** indicates the surfaces that have both positive and negative drafts on them. This can be eliminated by creating a split line in the middle, or by adding a full round fillet around the parameter.



Draft Analysis

Using the settings in draft analysis, you can verify the draft angles on model faces, or you can examine angle changes within a face.



Draft Analysis Explained



The **Draft Analysis** is a tool to check the correct application of draft to the faces of each part. With draft analysis, you can verify draft angles, examine angle changes within a face, as well as locate parting lines, injection, and ejection surfaces in parts.

Draft analysis results listed under Color Settings are grouped into four categories, when you specify Face classification:

Positive draft: Displays any faces with a positive draft based on the reference draft angle you specified. A positive draft means the angle of the face, with respect to the direction of pull, is more than the reference angle.

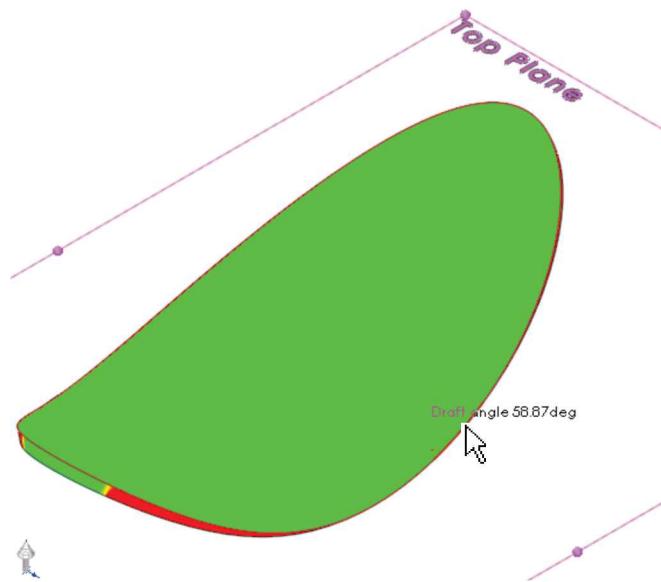
Negative draft: Displays any faces with a negative draft based on the reference draft angle you specified. A negative draft means the angle of the face, with respect to the direction of the pull, is less than the negative reference angle.

Draft required: Displays any faces that require correction. These are faces with an angle greater than the negative reference angle and less than the positive reference angle.

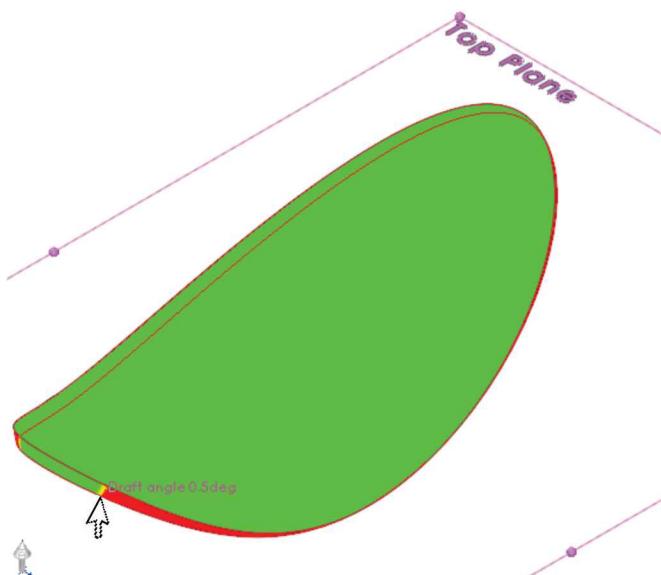
Straddle faces: Displays any faces that contain both positive and negative types of draft. Typically, these are faces that require you to create a split line.

Note: When analyzing the draft for surfaces, an additional Face classification criterion is added: Surface faces with draft. Since a surface includes an inside and an outside face, surface faces are not added to the numerical part of the classification (Positive draft and Negative draft). Surface faces with draft lists all positive and negative surfaces that include draft.

- * Hover the cursor over the upper surface to see the draft angle for that particular area.



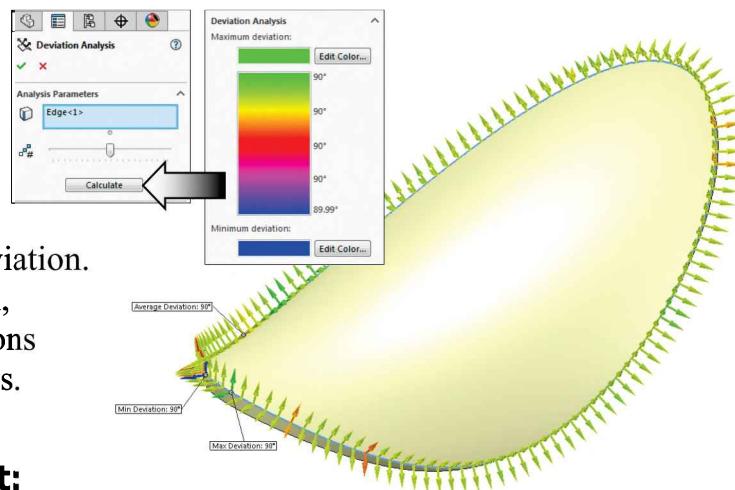
- * Position the mouse cursor over the yellow areas (required drafts) and see if the draft angles actually meet your draft requirements.



- * Click OK.

The option Deviation-Analysis can be used to diagnose and calculate the angle between faces.

The colored arrows display the amount of deviation. The results show the Max, Min and Average deviations between the adjacent faces.

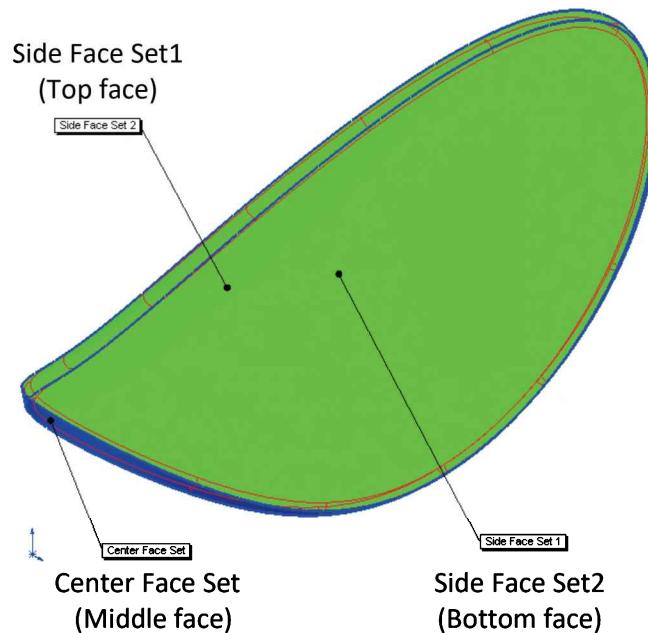
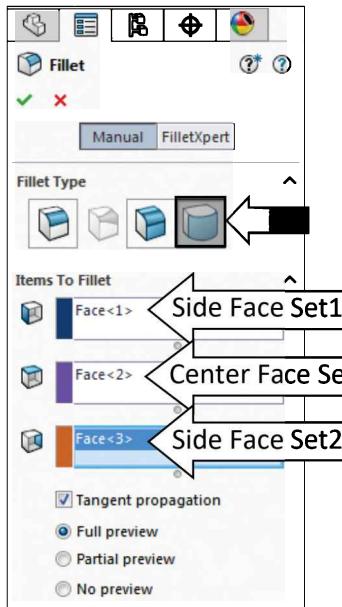


11. Adding a Full Round Fillet:

Click or select Insert / Features / Fillet-Round.

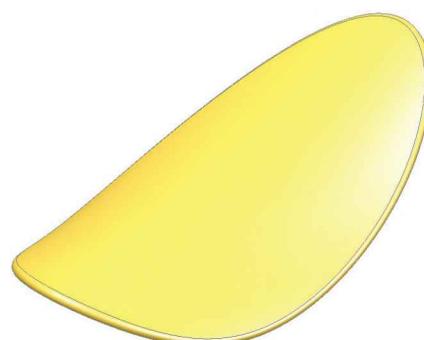
Select the **Full Round Fillet** button.

Select the Side-Face-Set1, Center-Face-Set, and Side-Face-Set2 as noted.



Click **OK**.

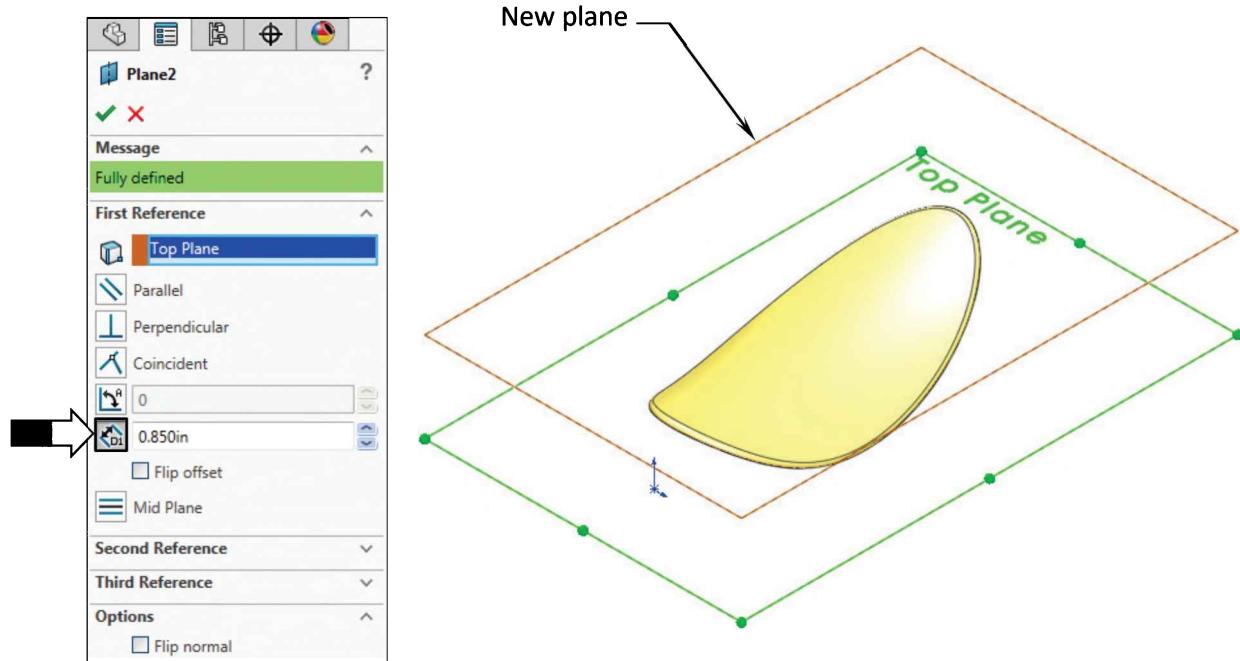
Rotate the model to verify the full round fillet from different orientations.



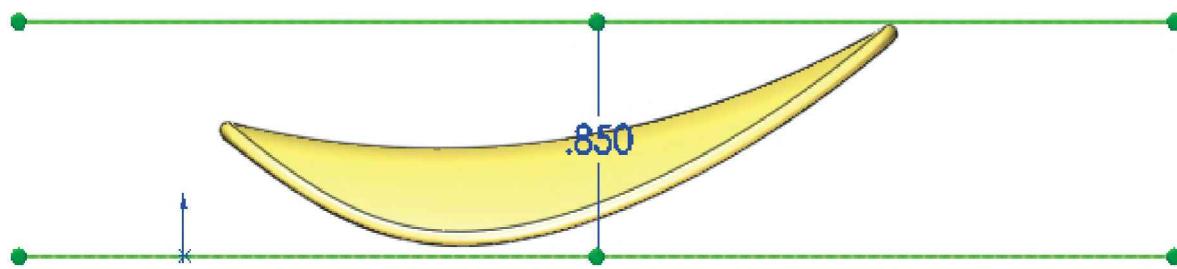
12. Creating an Offset Distance plane:

Select the **Top** plane from the FeatureManager tree and click  , or select **Insert / Reference Geometry / Plane**.

Enter **.850 in.** for Distance and place the new plane above the Top plane.



Click **OK**.



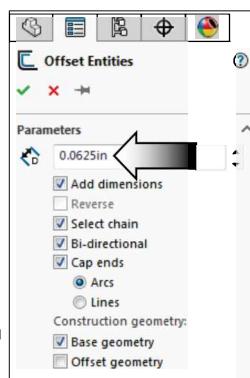
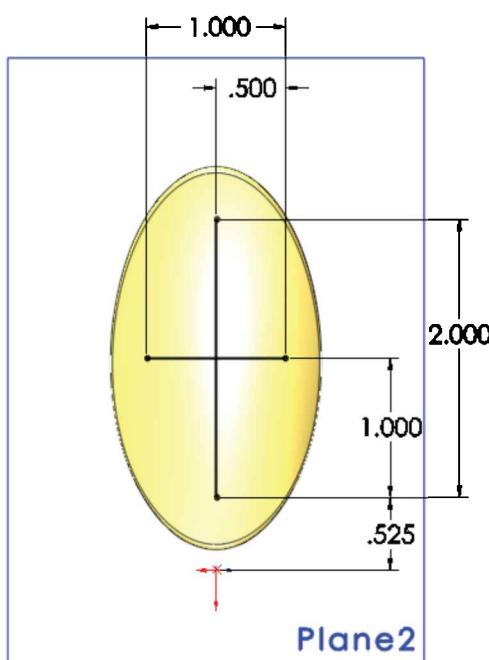
13. Sketching the Slot Contours:

Select the new plane (**Plane2**) and open a new sketch .

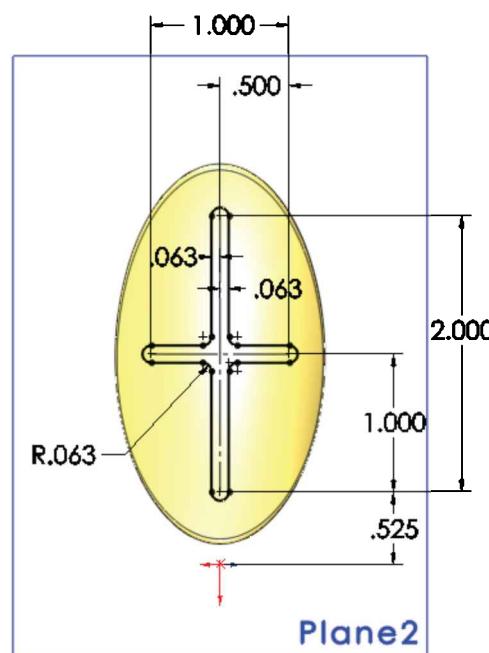
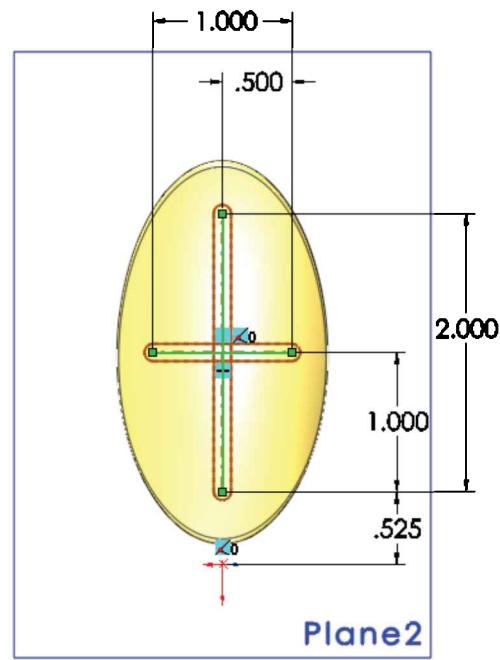
Create the sketch using either Mirror or Offset options.

Sketch the profile shown below and add dimensions to fully position the sketch.

Create an offset  of **.0625in**. from the 2 sketch lines, using the settings below.

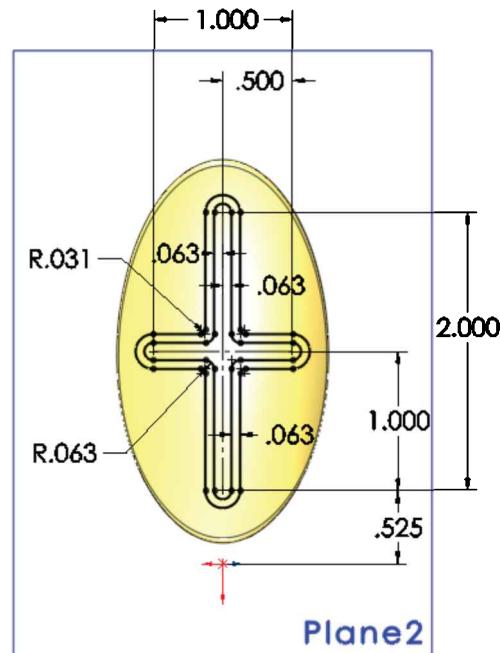


(Add the corner radius after the sketch is fully defined.)



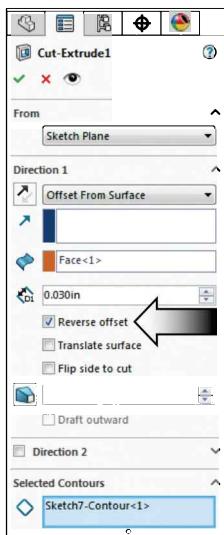
Trim the inner intersections and create a second offset also at **.0625in** as shown.

Clean up the corners and add the **.031** radius as indicated.



14. Extruding Cut the 1st Contour:

Click  or select Insert / Cut / Extrude.



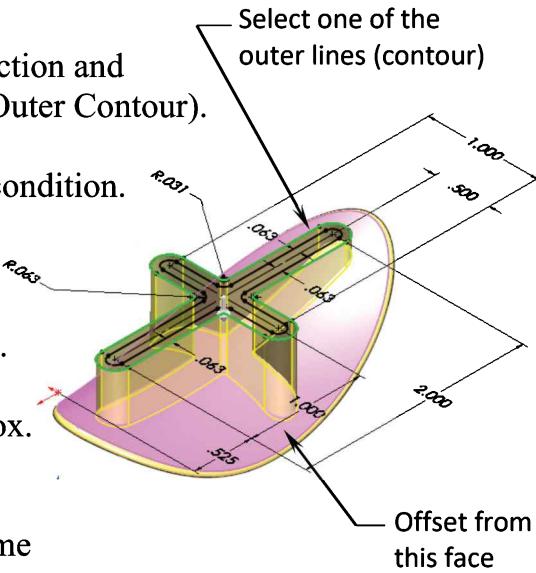
Expand the Selected Contour section and select one of the Outer Lines (Outer Contour).

Use Offset From Surface end condition.

Enter .030 in. for Depth.

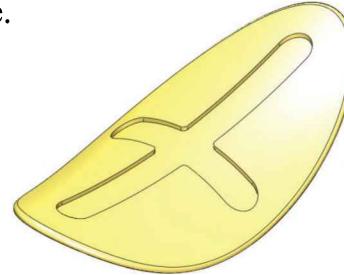
Click the Top face of the model.

Enable Reverse Offset check box.



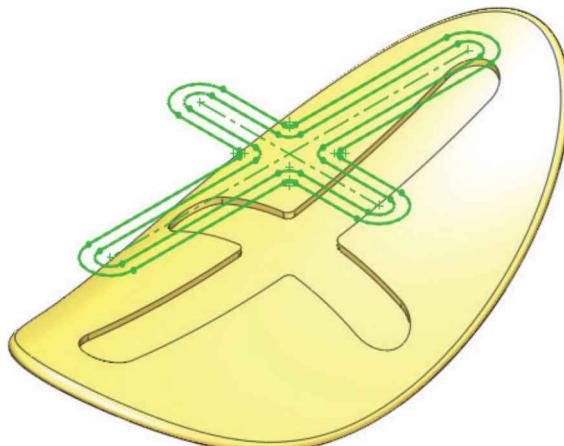
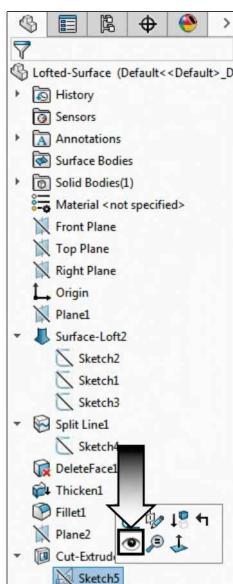
The slot is cut, following the same contours of the upper surface.

Click OK.



15. Extruding Cut the 2nd Contour:

Expand the Cut-Extrude1 from the FeatureManager tree, right-click Sketch5  and select Show .

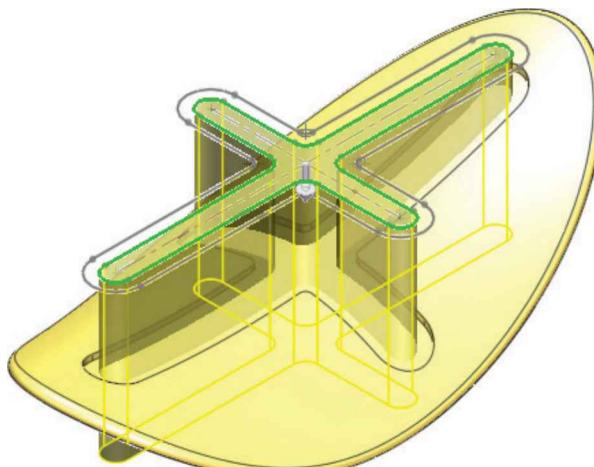
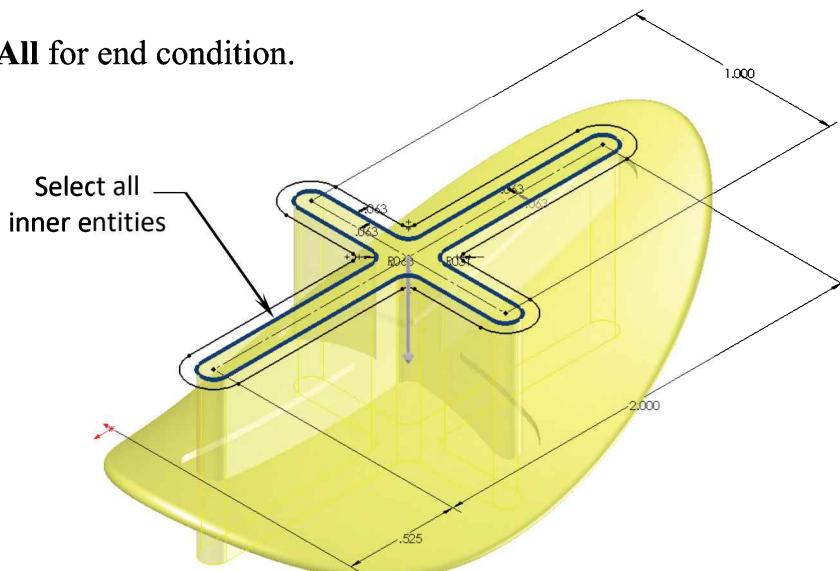
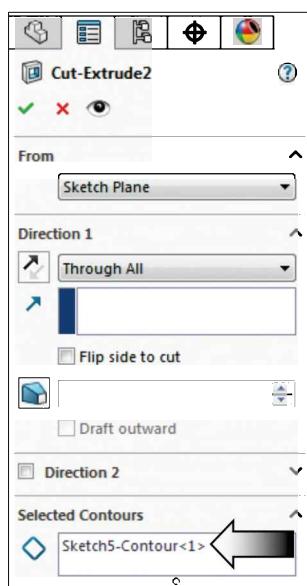


Hover the mouse cursor over one of the inner lines and select the entire contour when it highlights.

Click  or select **Insert / Cut / Extrude**.

Select **Through All** for end condition.

Click **OK**.



Hide  **Sketch5** when finished.

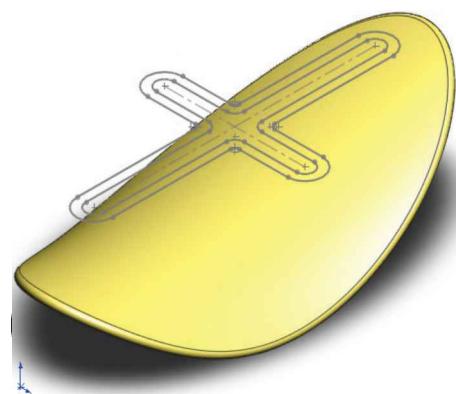
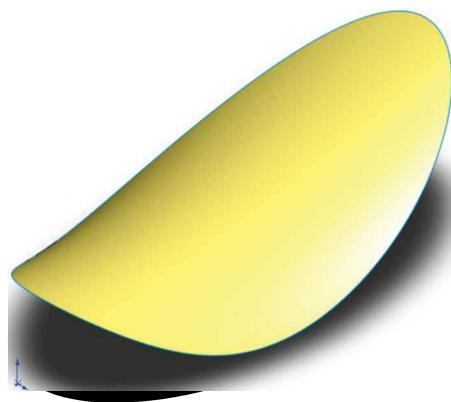
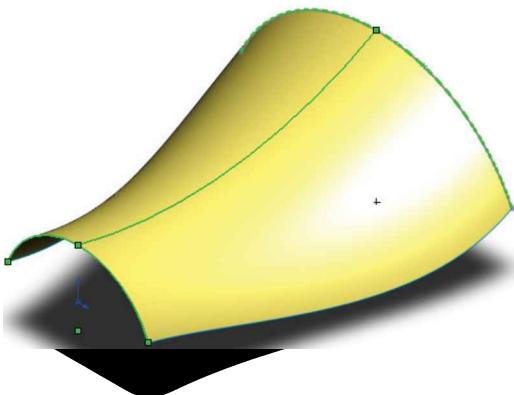
16. Saving your work:

Select **File / Save As / Lofted-Surface / Save**.

Questions for Review

1. Surfaces are a type of geometry that can be used to create complex shapes.
 - a. True
 - b. False
2. Surfaces can be opened, overlapped, and can have no thickness.
 - a. True
 - b. False
3. Surfaces can be modeled into any shape and can be extruded, revolved, swept, or lofted.
 - a. True
 - b. False
4. The split line option can be used to “divide” a surface into two or more surfaces.
 - a. True
 - b. False
5. Several surfaces can be used to create a solid loft feature.
 - a. True
 - b. False
6. Surfaces cannot be moved or copied in a part document.
 - a. True
 - b. False
7. Each surface can be created individually and then knitted together as one surface.
 - a. True
 - b. False
8. The same Sketched profile can be re-used to create different extruded contours.
 - a. True
 - b. False
9. Offset From Surface (extrude option) only works with surfaces, not solid features.
 - a. True
 - b. False

1. TRUE	2. TRUE	3. TRUE	4. TRUE	5. TRUE	6. FALSE	7. TRUE	8. TRUE	9. FALSE
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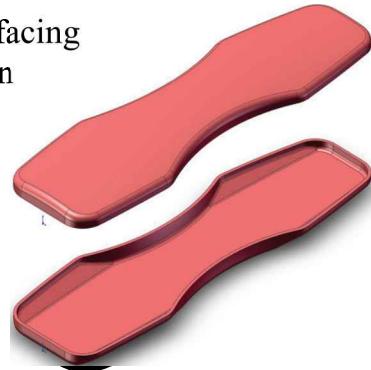


Lofted Surface

Let us take a look at a couple of techniques when modeling advanced shapes.

We will start out by creating some surfaces using various surfacing tools. These surfaces will get knitted into one surface and then thickened into a solid part.

Finally, the part is split into two halves and assembled in an assembly document.



1. Creating new offset planes:

Select the Front plane from the FeatureManager tree.

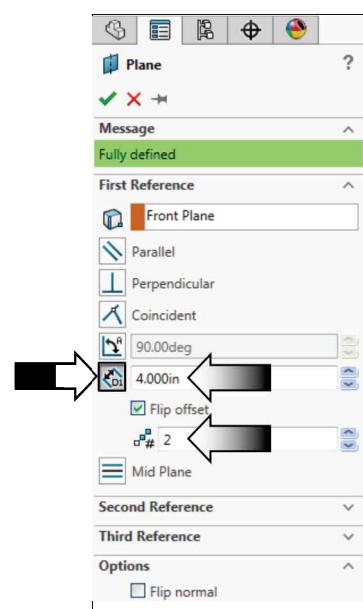
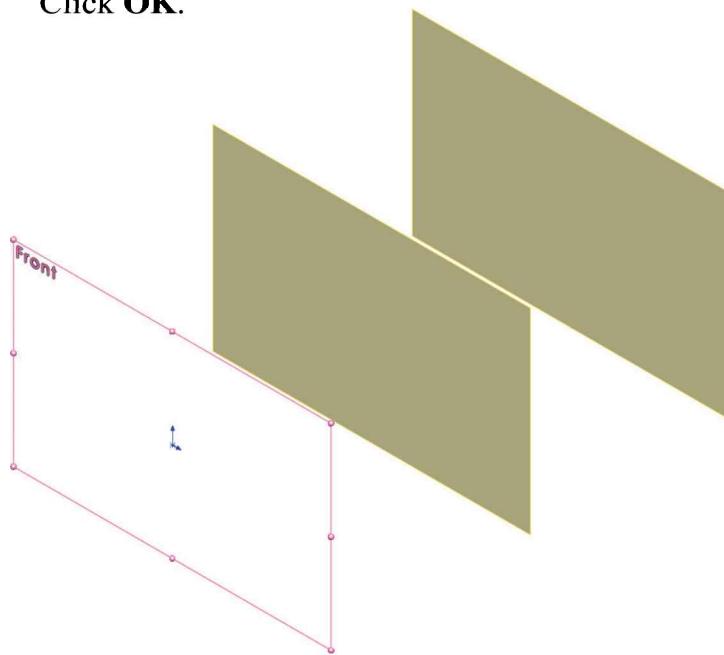
Click or select **Insert / Reference Geometry / Plane**.

Select the **Offset Distance** option and enter **4.00in**.

Click **Flip Offset** if needed to place the new plane on the *back side*.

Enter **2** for number of instances.

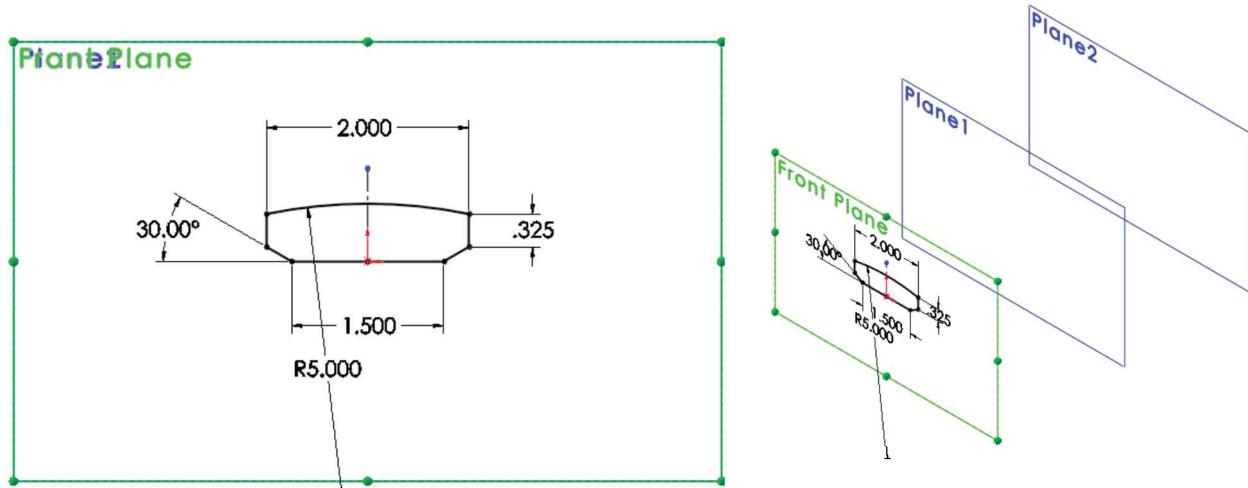
Click **OK**.



2. Sketch the first profile: (the front section)

Select the Front plane and open a **new sketch** .

Sketch the profile and add the dimensions shown below.



Exit the Sketch  or select **Insert / Sketch**.

Copy & Paste

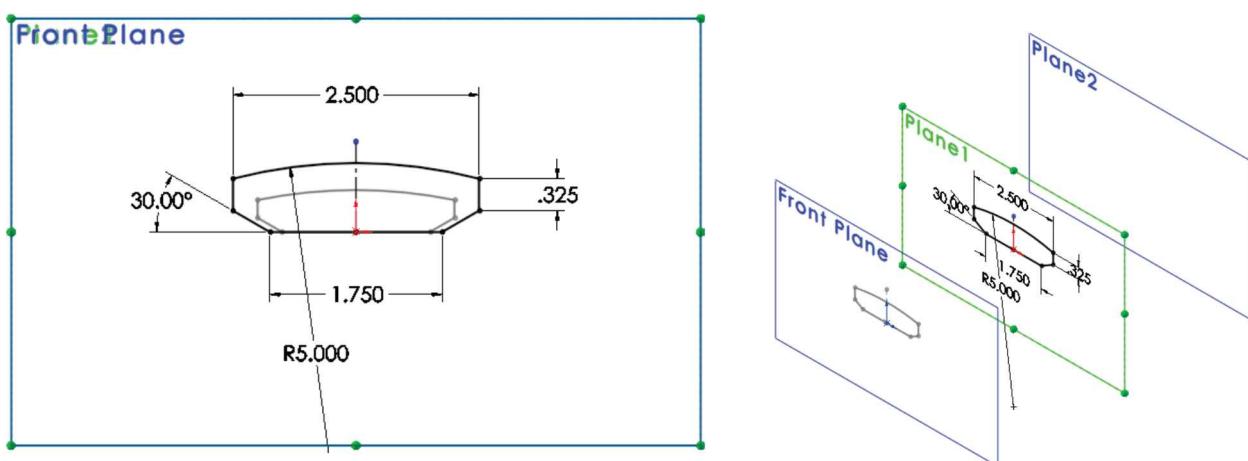
The 1st profile can be copied and pasted to make the next 2 sketches. The dimensions will then be adjusted to the final size.

3. Sketching the second profile: (the middle section)

Select Plane1 from FeatureManager tree.

Click  or select **Insert / Sketch**.

Sketch the profile and add the dimensions shown.



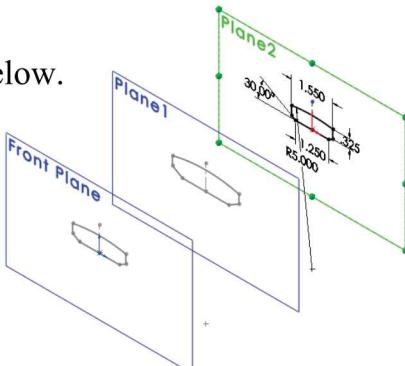
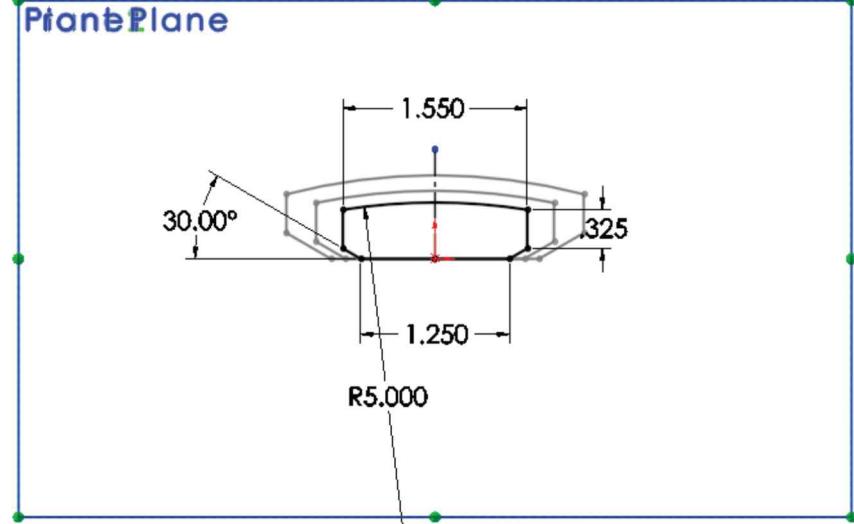
Exit the Sketch  or select **Insert / Sketch**.

4. Sketching the third profile: (the end section)

Select Plane2 from the FeatureManager tree.

Click  or select **Insert / Sketch**.

Sketch the profile and add dimensions as shown below.



Lofted Surface

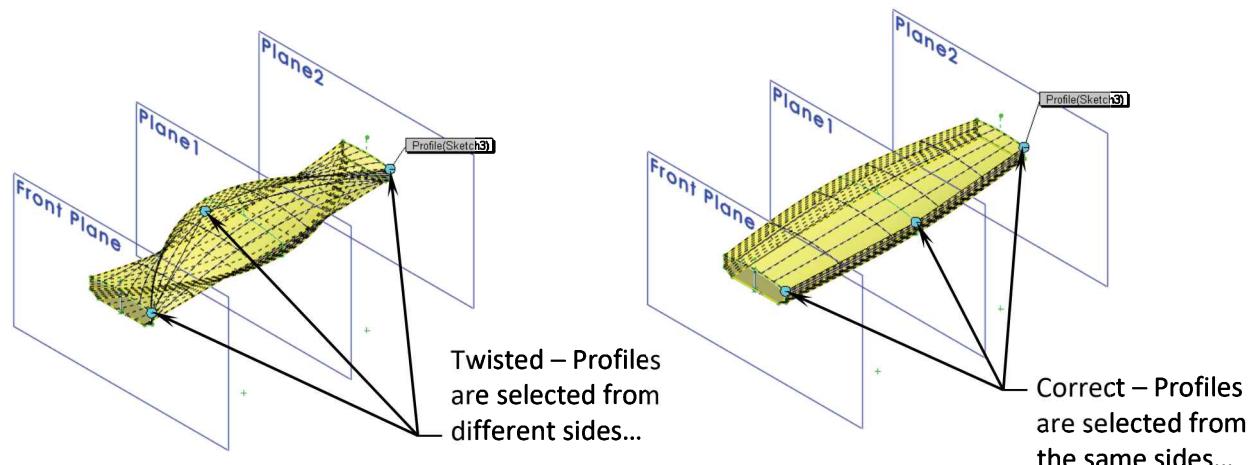
Lofted Surface creates a surface by making transitions between the sketch profiles.

Two or more profiles are needed to create a loft.

Exit the Sketch  or select **Insert / Sketch**.

5. Selecting the loft profiles:

To prevent the loft feature from being twisted, try and select the same vertex in each profile.



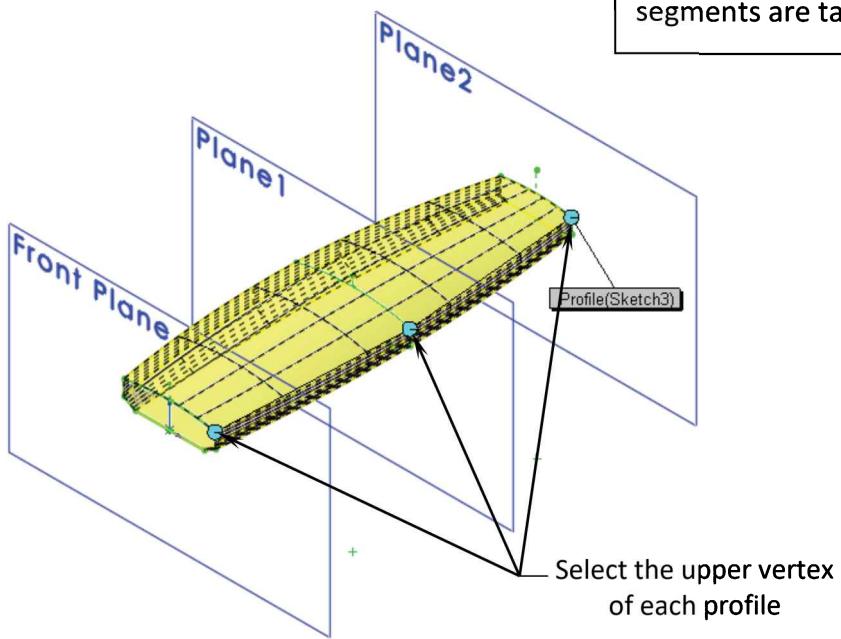
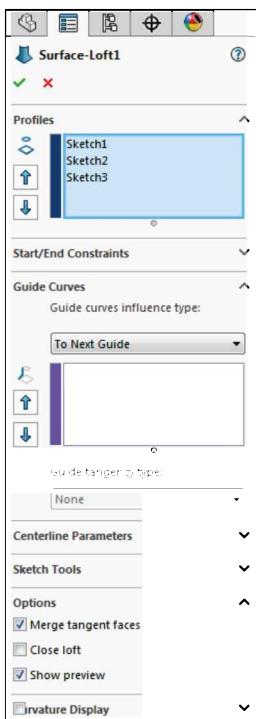
6. Lofting between the profiles:

Click  or select Insert / Surface / Loft.

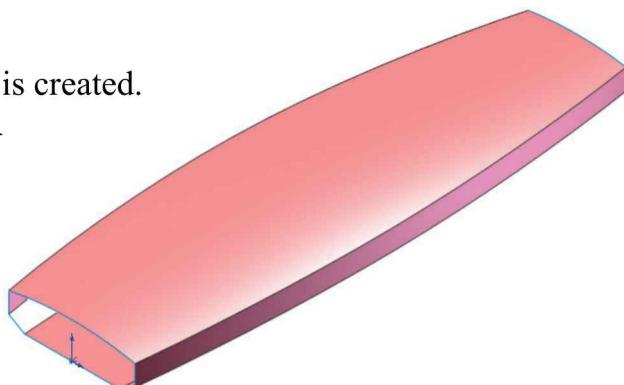
Select the **Upper-right vertex** of each profile.

Enable **Merge Tangent Faces** checkbox.

Click **OK**.



The surface model is created.
Inspect your model
against the one
shown here.

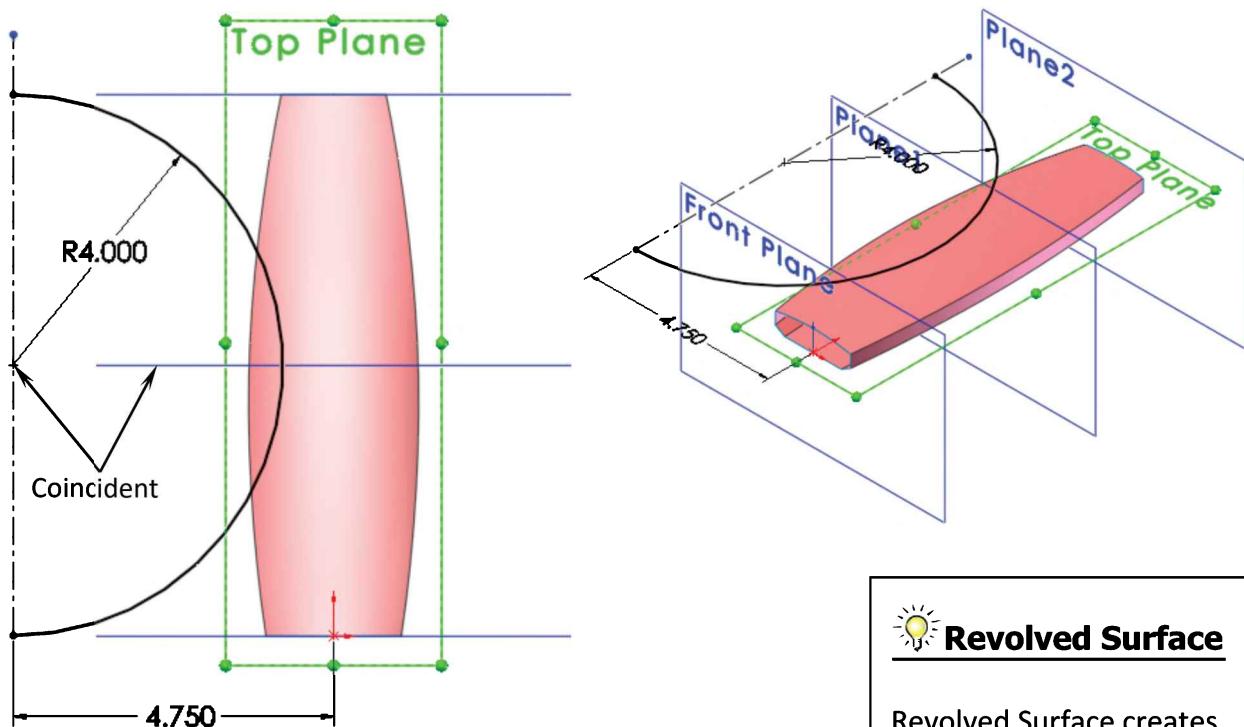


7. Creating a Revolved sketch:

Select the **Top** plane from the FeatureManager tree.

Click  or select **Insert / Sketch**.

Sketch the revolve profile and add the dimensions shown.



Revolved Surface

Revolved Surface creates a surface by rotating a sketch profile around a centerline (or the Axis of Revolution).

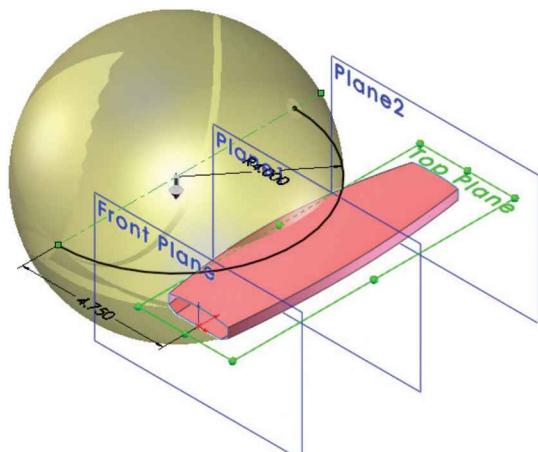
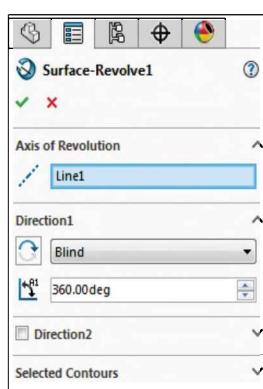
8. Revolving the Spherical surface:

Click  or select **Insert / Surface / Revolve**.

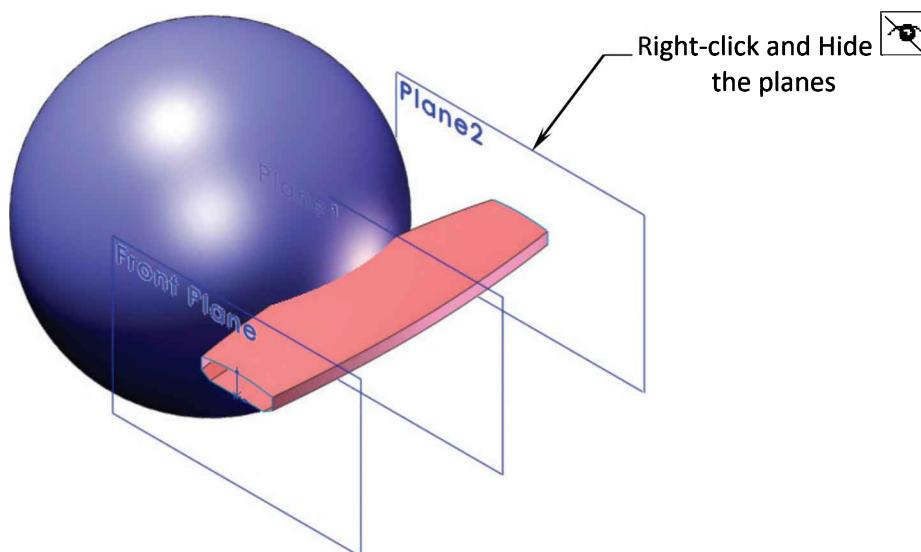
Revolve Type:
Blind.

Revolve Angle:
360 deg.

Click **OK**.



The Revolved Surface.



9. Copying the Revolved Surface:

Click or select **Insert / Surface / Move/Copy**.

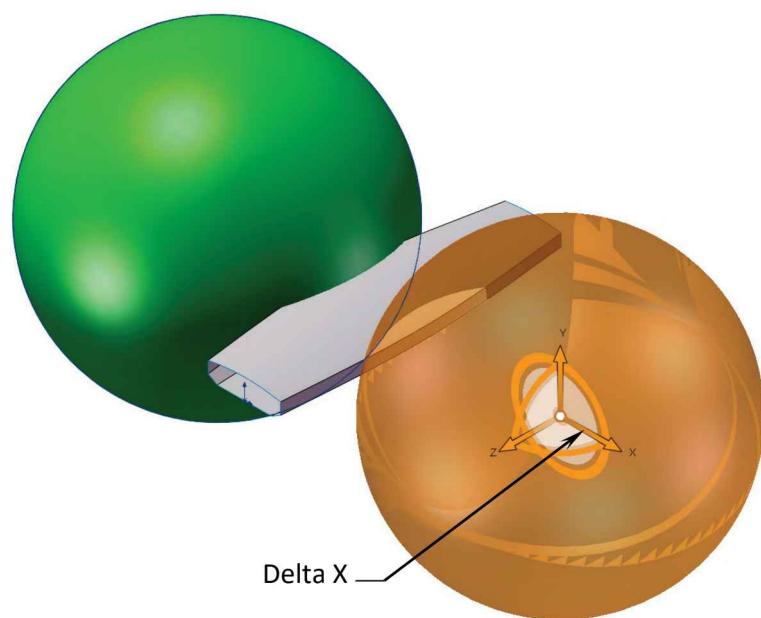
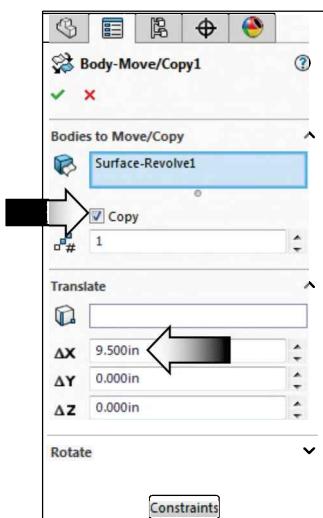
Select **Surface-Revolve1** under Surfaces to Move/Copy.

Enable the **Copy** check box.

Enter **1** for Number of Copies.

Enter **9.500** in the **Delta X** distance box.

Click **OK**.



10. Trimming the Base part:

Click  or select **Insert / Surface / Trim**.

For Trim-Type, click **Mutual Trim** .

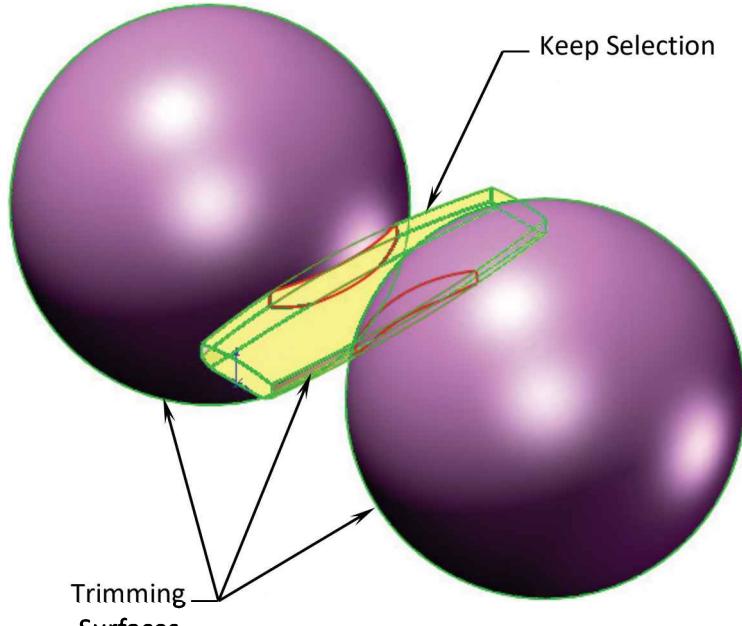
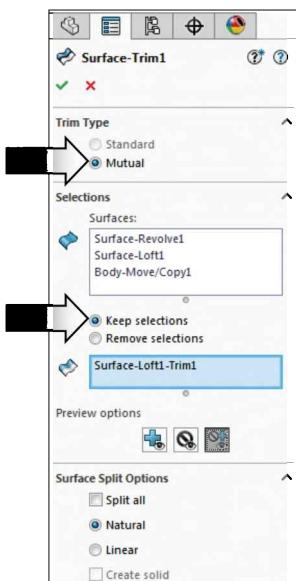
For **Trimming-Surfaces**, select all 3 surfaces.

For **Keep Selection**, select the Surface-Loft1.

Trim Surface

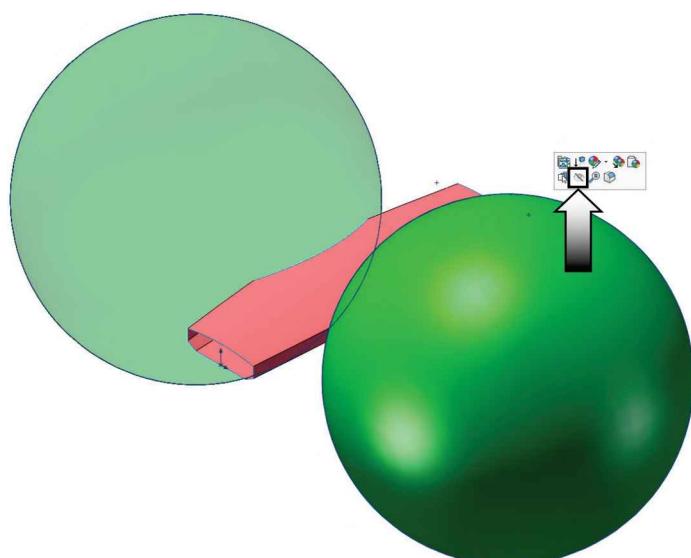
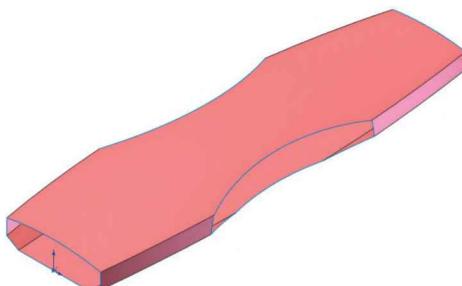
A surface or a sketch can be used as a trim tool to trim the intersecting surfaces.

Click **OK**.

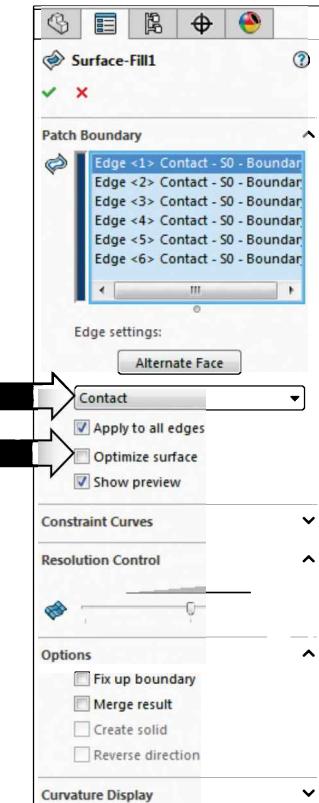


11. Hiding the surfaces:

Right-click on the two revolved surfaces and select **Hide** .



12. Patching the right side opening:



Click Filled Surface .

For Patch Boundary: select the **6 edges** as indicated.

For Curvature Control use: **Contact**.

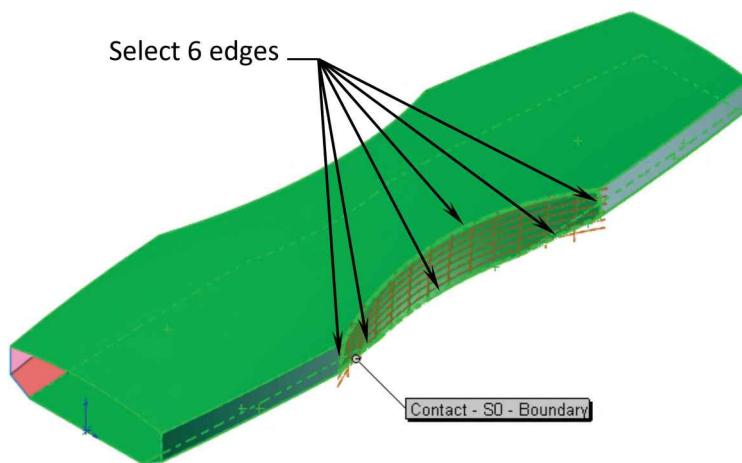
Enable Apply to all edges.

Clear the Optimize Surface
option.

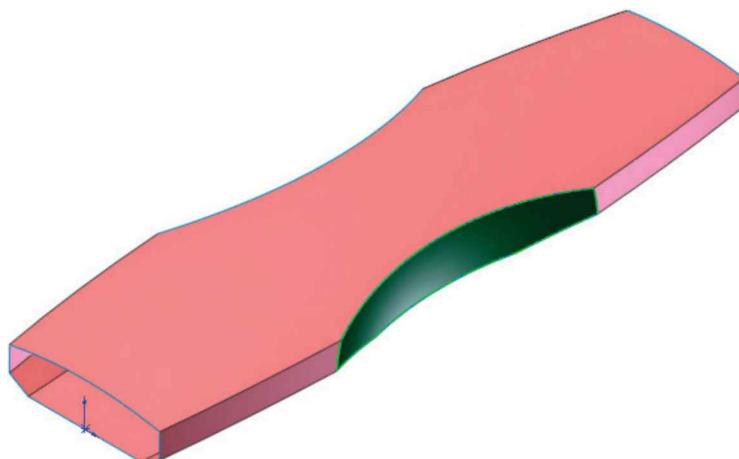
Click **OK**.

Filled Surface

Filled surface constructs a surface patch to fill a non-planar opening in a model.



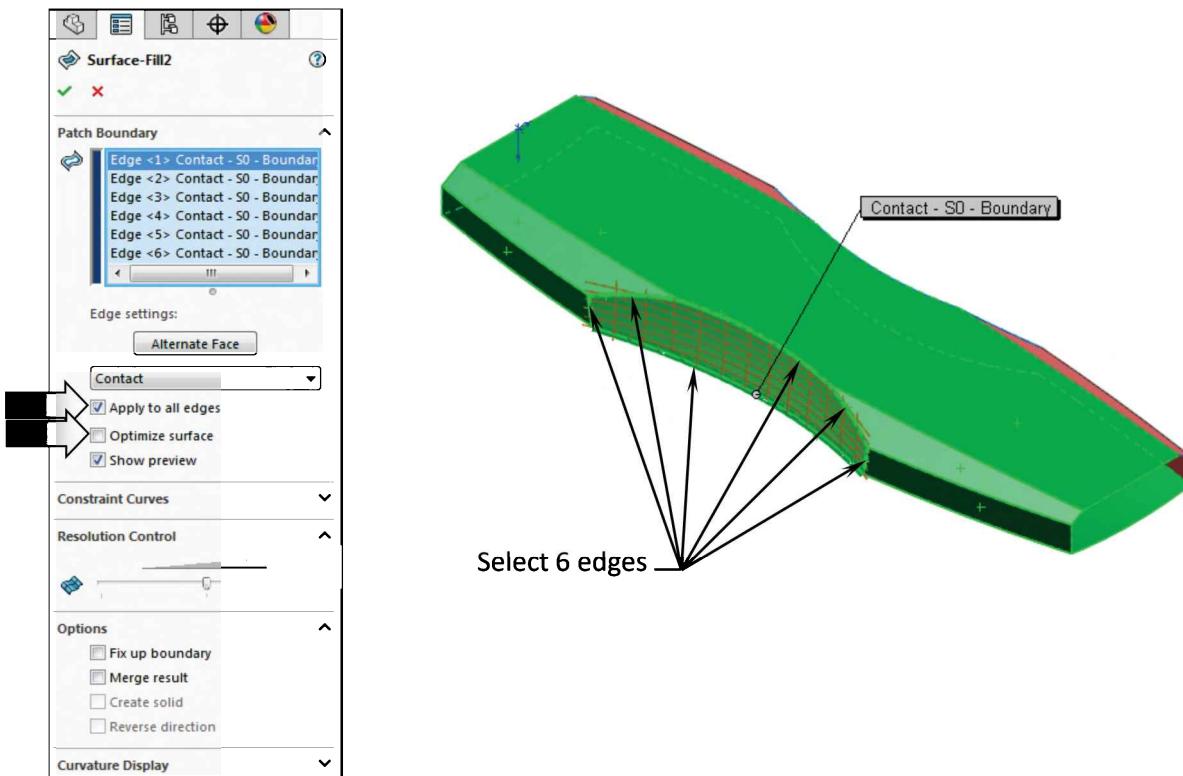
The right side cutout is filled with a non-planar surface (Surface-Fill1).



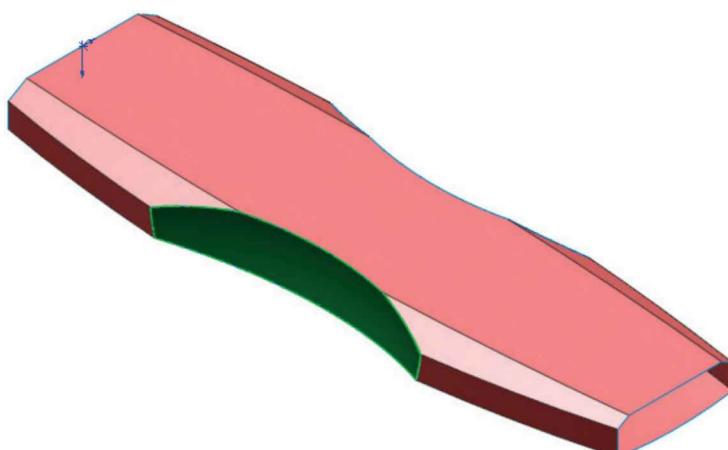
13. Filling the left side opening:

Rotate the view  to the opposite side - or - Hold the Shift key and press the Up-Arrow key twice (this hotkey rotates 90° each time).

Repeat step 13 and fill the left side opening with a new surface.



The opening on the left side is filled with a new surface (Surface-Fill2).

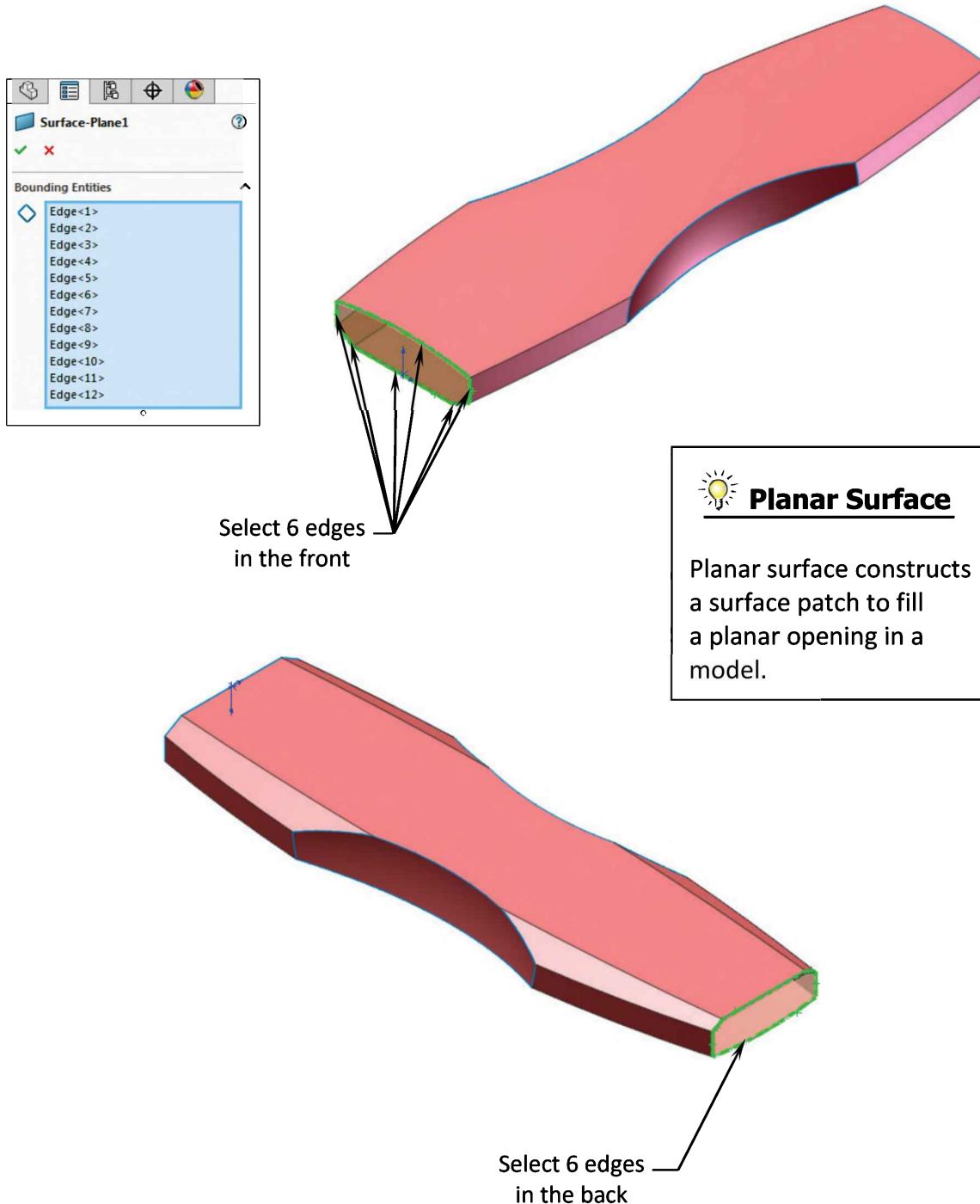


14. Filling the front and back openings:

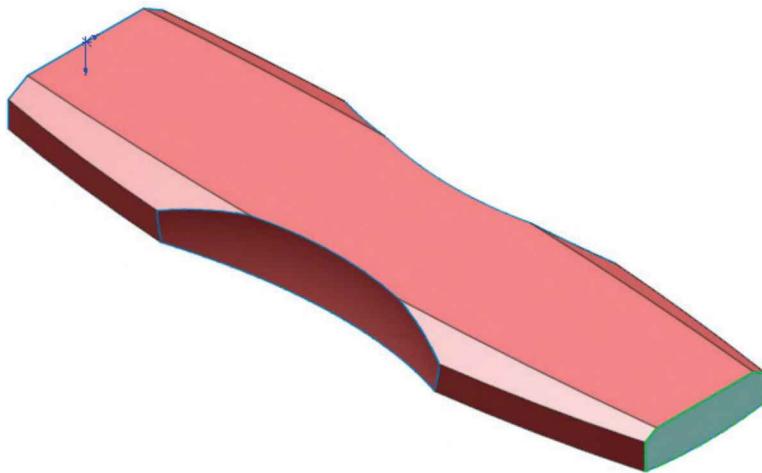
Click  or select **Insert / Surface / Planar**.

For Boundary Entities: select all **12 edges** in the front and back openings.

Click **OK**.



The front and back openings are filled with planar surfaces (Surface-Plane2).



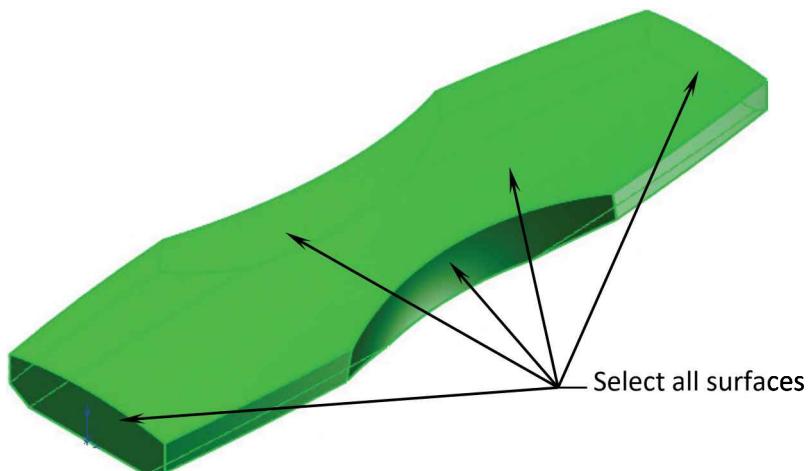
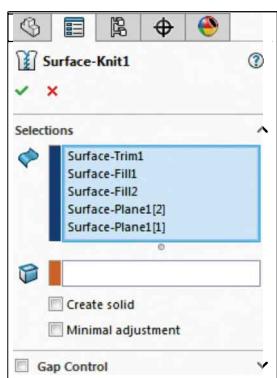
15. Creating a Surface-Knit:

Click or select **Insert / Surface / Knit**.

For Surfaces/Faces-To-Knit, select all **5 surfaces**.

Click **OK**.

Knit Surface
Combines two or
more faces or
surfaces into one.



All 5 surfaces are knitted and combined into a single surface (Surface-Knit1).

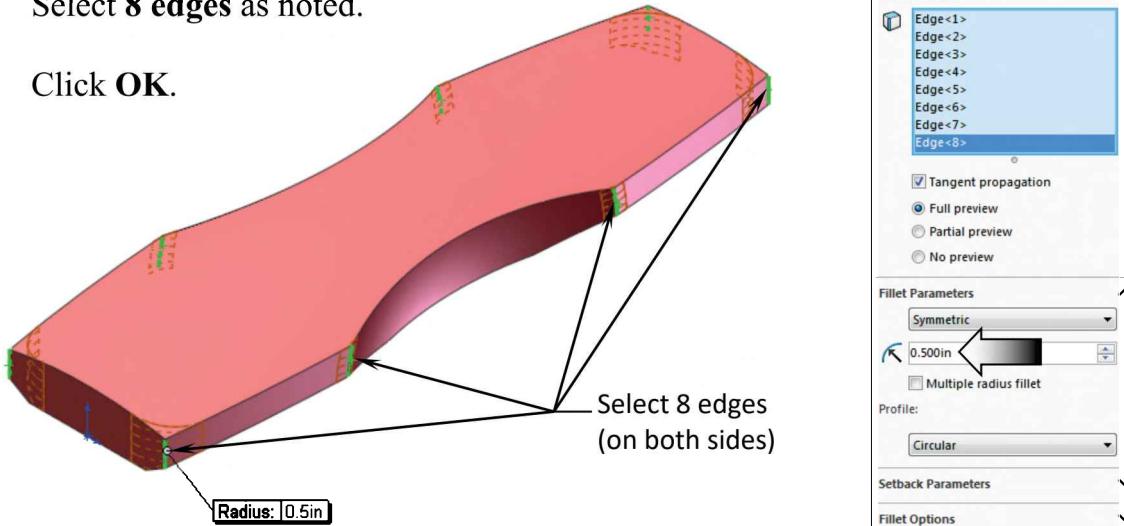
16. Adding .500" fillets:

Click  or select Insert / Features / Fillet/Round.

Type **.500in.** for Radius.

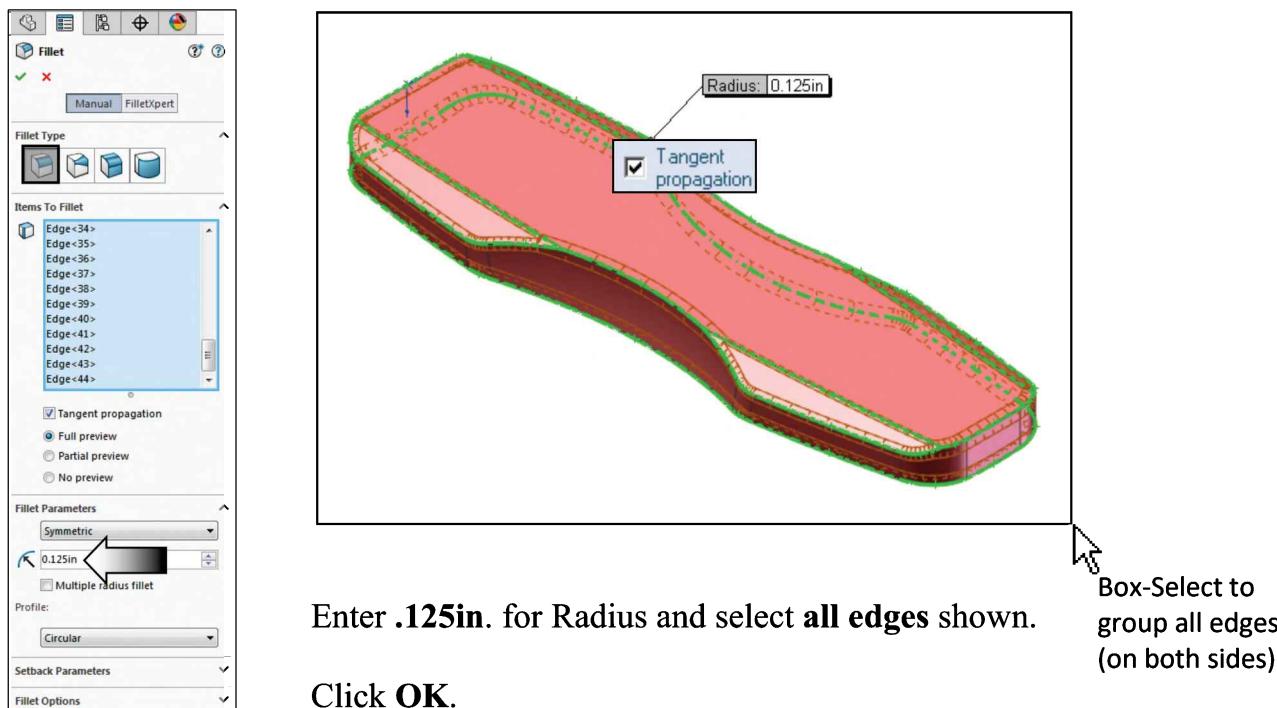
Select **8 edges** as noted.

Click **OK**.

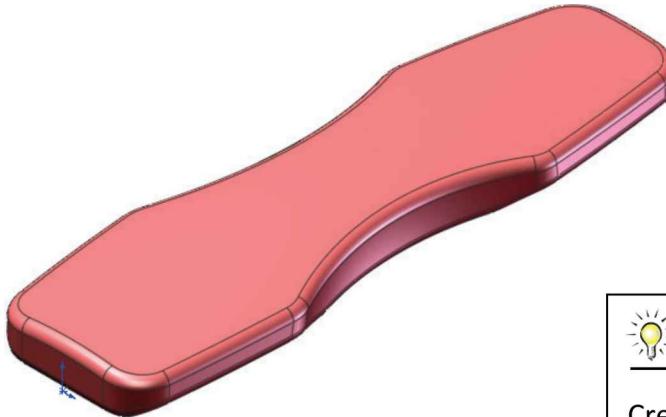


17. Adding .125" fillets:

Click  or select Insert / Features / Fillet/ Round.



Inspect your model against the image below. Rotate the model and make sure all edges are filleted.

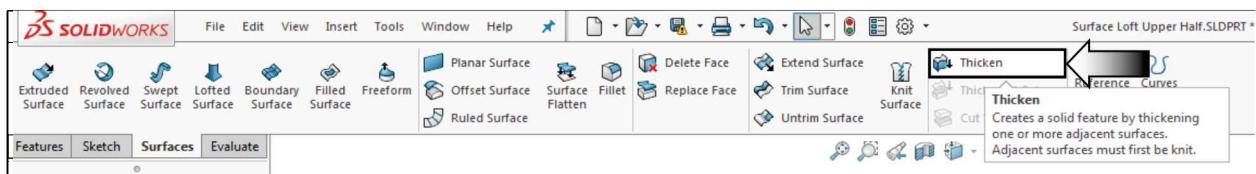


Thicken Surface

Creates a solid feature by thickening one or more adjacent surfaces.

18. Creating a solid from the surface model:

Click on the Surfaces tool tab or select Insert / Boss-Base Thicken.



For Surface-To-Thicken: Select the model from the graphics area.

For thickness Direction: Select **Thicken Side 2** (Inside).

For Thickness: Enter **.060 in.**

Click **OK**.

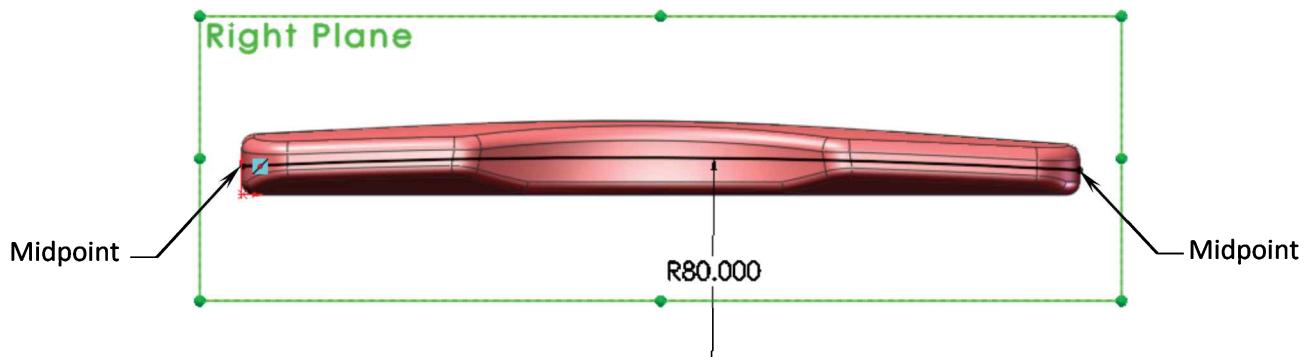
19. Sketching the split profile: (to split the part into 2 halves)

Select the Right plane from the FeatureManager tree.

Click  or select **Insert / Sketch**.

Sketch a **3-Point-Arc** and add the radius dimension shown below.

Add a **Mid-point** relation between the 2 end points of the arc and the outer edges of the model.



20. Removing the Upper Half:

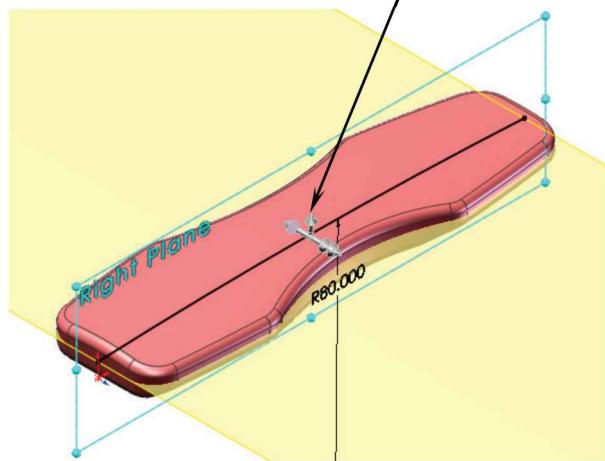
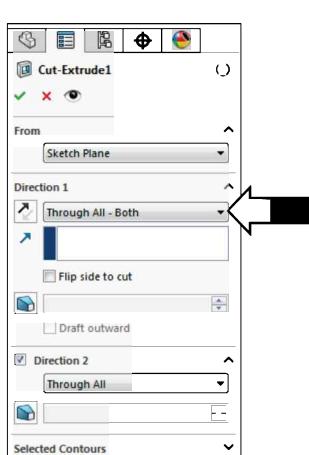
Click  or select **Insert / Cut / Extrude**.

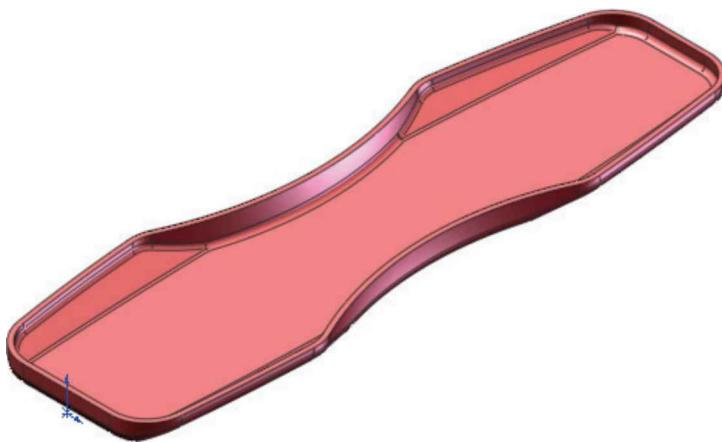
Extrude Type: **Through All Both** (use **Flip side-to-Cut** if needed).

Direction 2: **Through All** (default).

Click **OK**.

The arrow in the middle indicates which half is going to be removed by the cut





The resulted cut, the lower half of the model is kept.

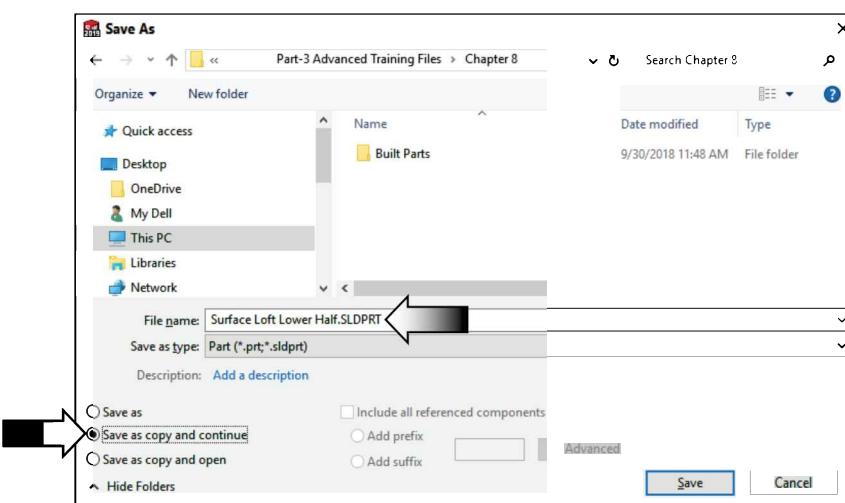
21. Saving the lower half:

Select File / Save As / Surface Loft Lower-Half / Save.

22. Saving the lower half of the part:

Select File / Save As and enter: Surface Loft Lower Half for the file name.

Enable the Save As Copy and Continue* check box and click Save.

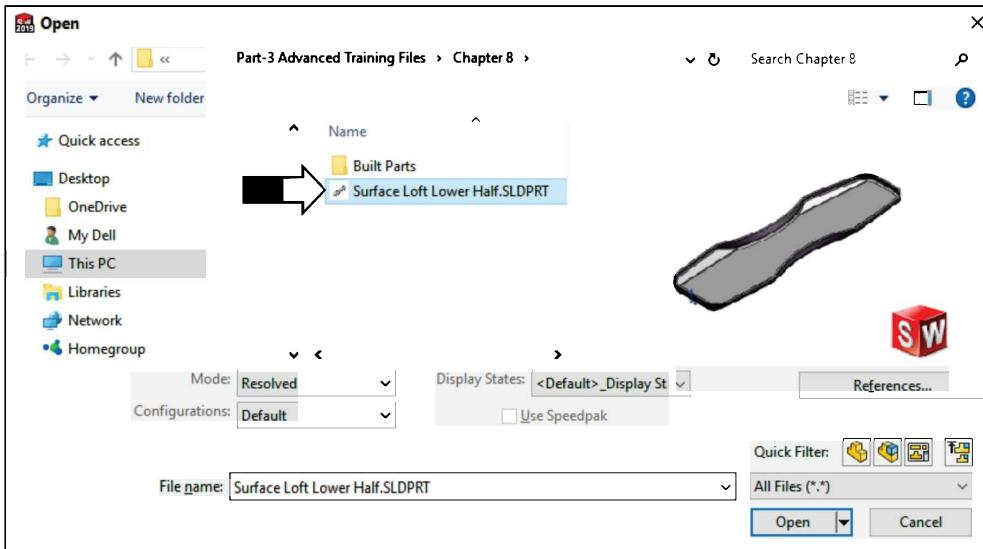


* This saves an exact copy of the same part but with a different name, so that we still have a full feature tree to create the second half of the part.

23. Modifying the copied file:

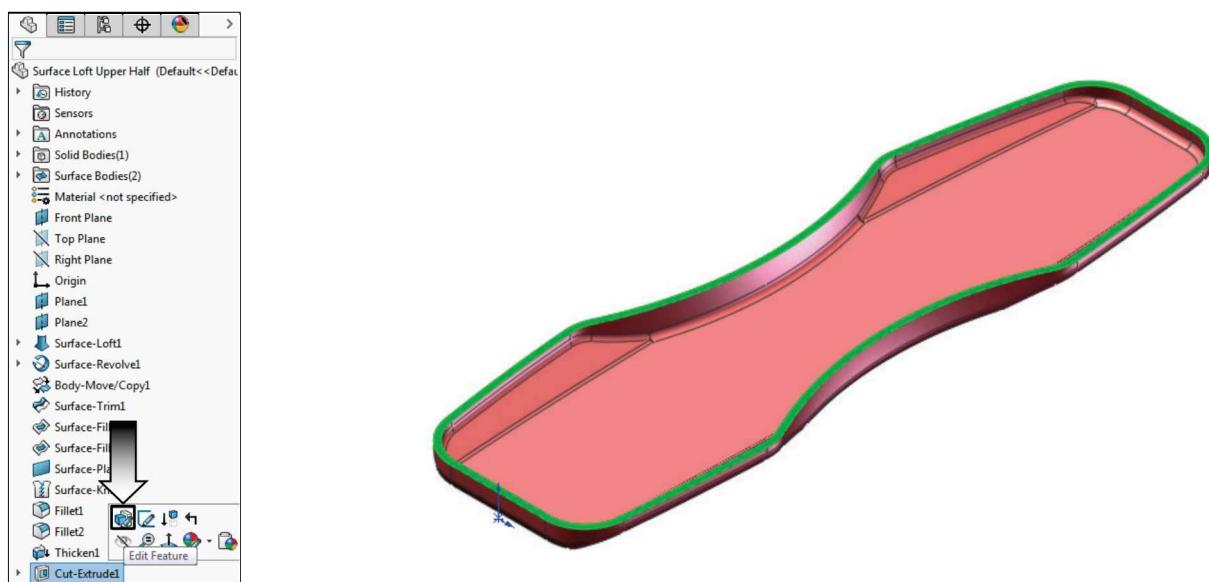
Select File / Open.

Select the document **Surface Loft Lower Half** and click Open.



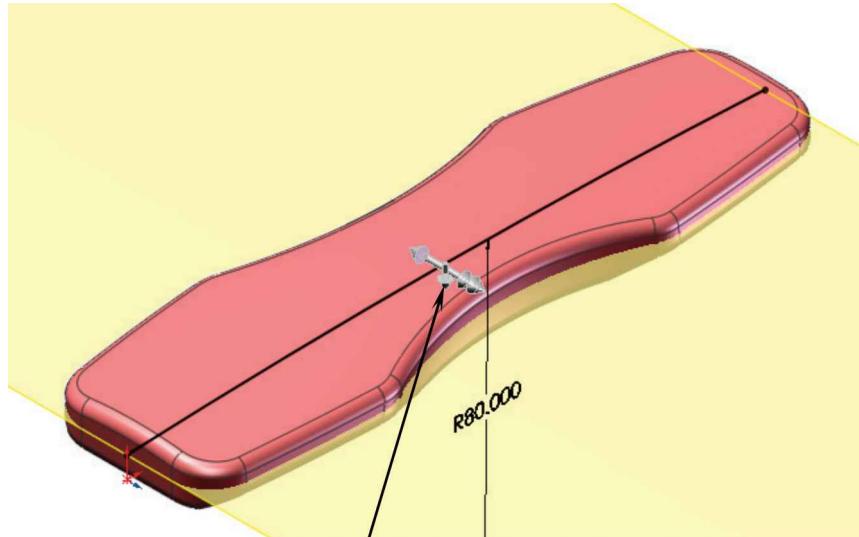
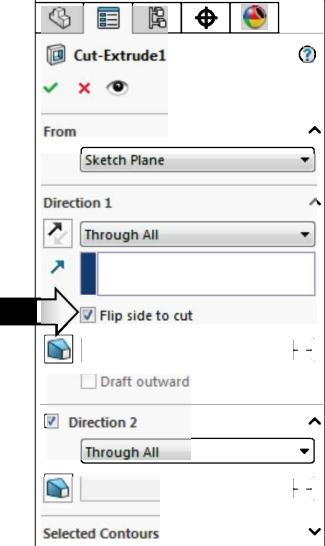
24. Changing the direction of the cut:

Right-click the **Cut-Extrude1** (the last feature on the tree) and select:



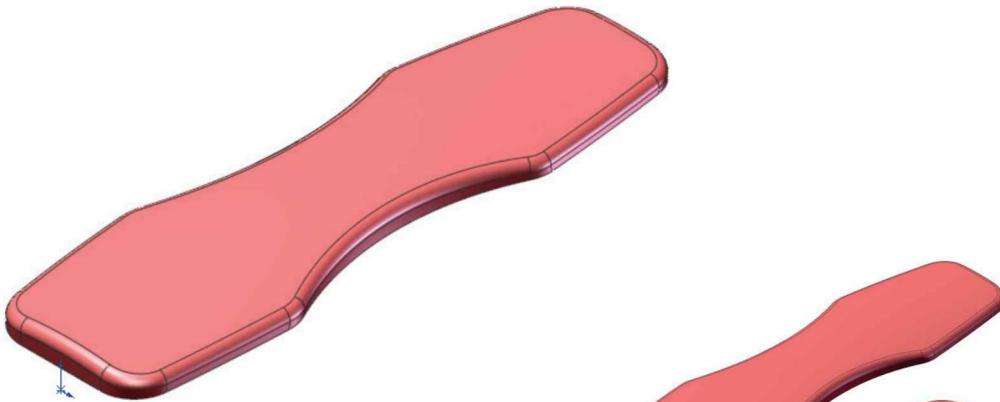
Select **Flip Side To Cut** option **Flip side to cut**.

Click **OK**.



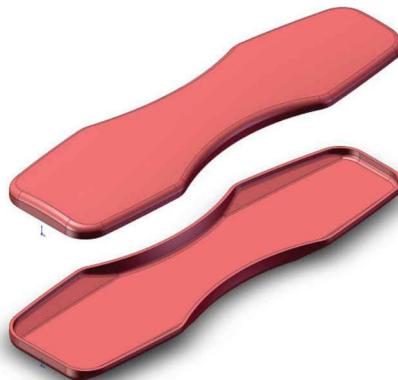
The middle arrow pointing downwards indicates the lower half is being removed

The Lower Half of the part is removed, leaving the Upper-Half as the result of the Flip Side cut.



25. Saving the upper half:

Select **File / Save As**. Enter: **Surface Loft Upper Half** for the file name and click **Save**.



OPTIONAL:

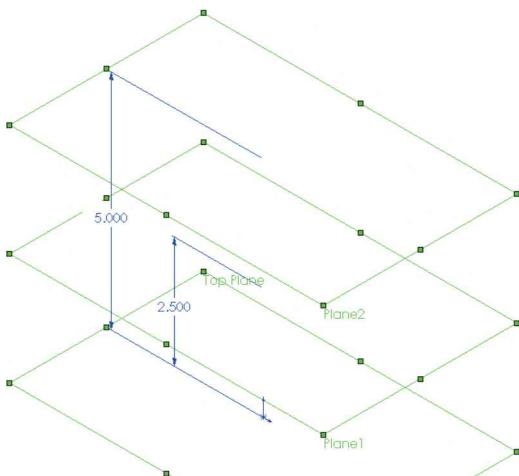
Insert the 2 halves into an assembly document and assemble them as shown above.

Questions for Review

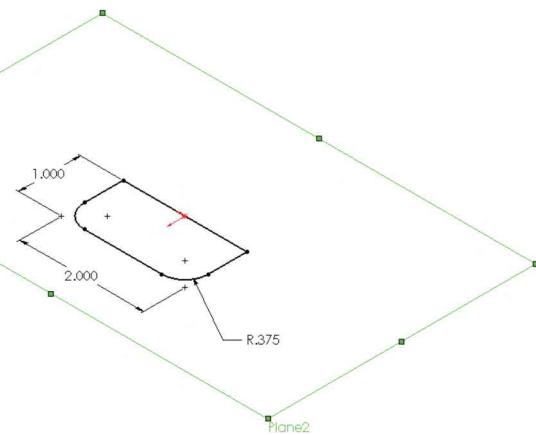
1. There are no limits on how many profiles you can have in a loft feature.
 - a. True
 - b. False
2. Each loft profile should be created on a different plane.
 - a. True
 - b. False
3. The guide curves used in a loft feature must be Coincident or Pierced to the profiles.
 - a. True
 - b. False
4. Only solid features can be mirrored, but surfaces cannot.
 - a. True
 - b. False
5. Only two surfaces can be used for knitting at a time.
 - a. True
 - b. False
6. Fillets cannot be used with surfaces; they are only available in solid models.
 - a. True
 - b. False
7. Surfaces can be thickened after they are knitted together.
 - a. True
 - b. False
8. Mass properties options such as volume are available for all surfaces.
 - a. True
 - b. False
9. Surfaces can be knitted into a closed volume and then thickened into a solid.
 - a. True
 - b. False

1. TRUE	2. TRUE	3. TRUE	4. FALSE	5. FALSE	6. FALSE	7. TRUE	8. FALSE	9. TRUE
---------	---------	---------	----------	----------	----------	---------	----------	---------

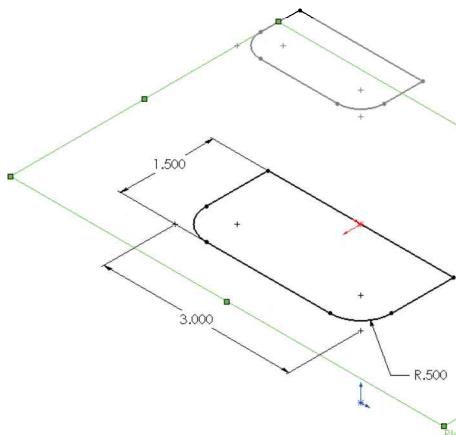
Exercise: Loft & Delete Face



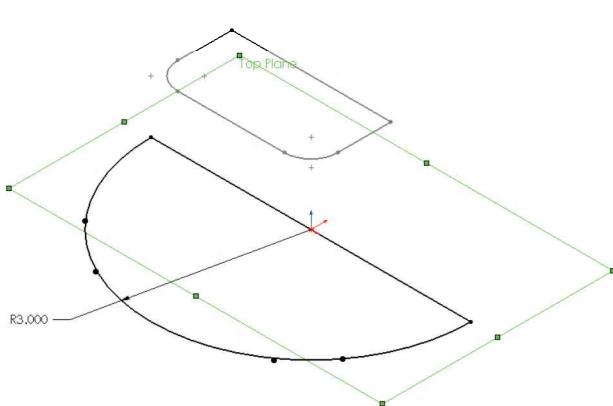
1. Create 2 new Planes, offset from Top plane.



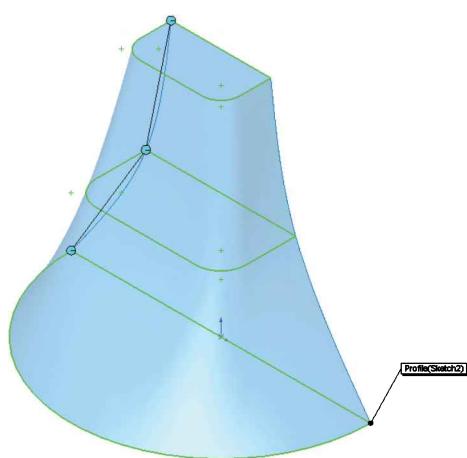
2. Sketch the 1st Profile on Plane2.



3. Sketch the 2nd Profile on Plane1.



4. Sketch the 3rd Profile on Top plane & add the connector points.



5. Solid-Loft the Profiles.



6. Add the Raised features (any size), use Delete Face command and remove 3 Faces (top, bottom & back).

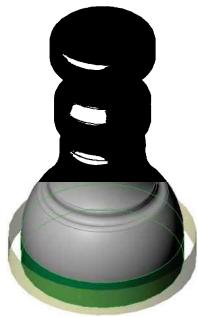


CHAPTER 9

Offset Surface & Ruled Surface

Offset Surface & Ruled Surface

The Offset Surface command creates a new surface from a **single face** or a **group of faces**, with a distance of zero or greater. The Offset Surface can be created inward or outward.



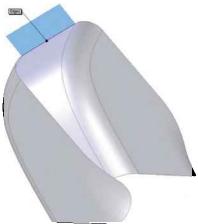
Offset from a single face

Offset Surface



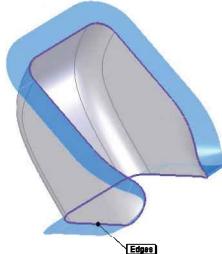
Offset from a group of faces

The Ruled Surface command creates a new surface from a single edge or a group of edges. The Ruled Surface can either be perpendicular or tapered from the selected edges.



Ruled surface from a single edge

Ruled Surface



Ruled surface from a group of edges

The Offset Surface and the Ruled Surface are used to create reference surfaces that help define the solid features in a part. In most cases, these surfaces should be knitted together before the next operation such as extruded cuts, fillets, etc., can be performed.

Advanced Surfaces Using Offset Surface & Ruled Surface



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI
Units: INCHES – 3 Decimals

Tools Needed:



Lofted
Boss/Base



Split Line



Offset Surface



Ruled Surface



Knit Surface



Shell

Advanced Surface Modeling Using Offset & Ruled Surface options

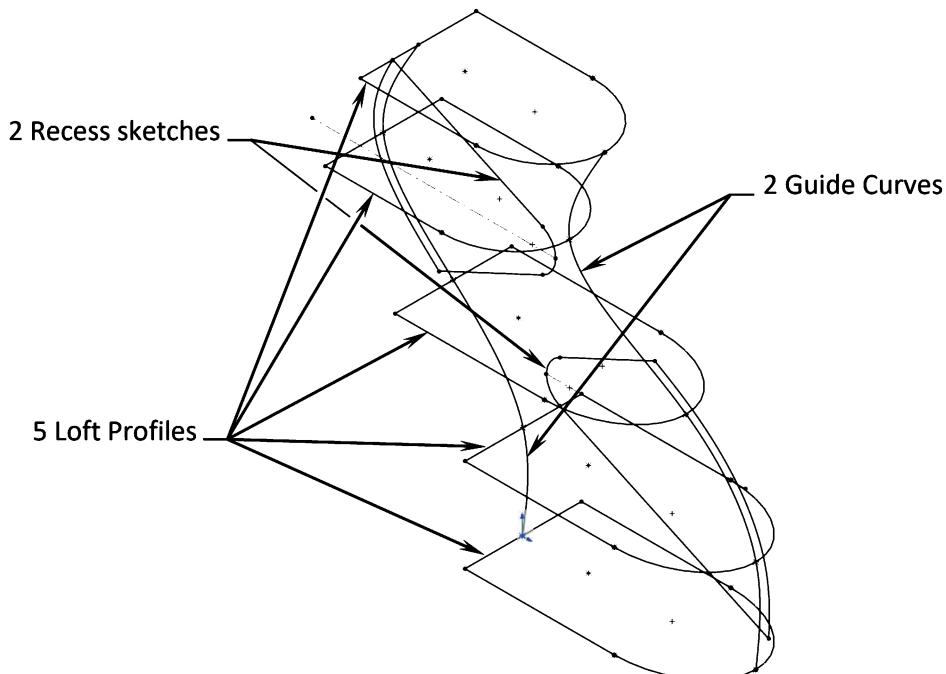
1. Opening the existing file:

Browse to the Training Files folder
and open a part document named:
Surface_Offset_Ruled.sldprt

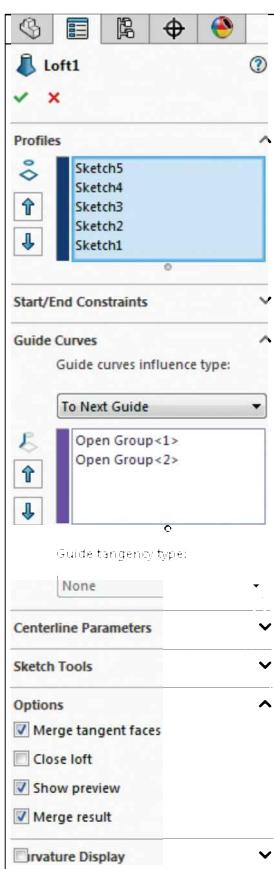
This part document contains several sketches to be used as the Loft Profiles, and 2 other sketches used as the Guide Curves that help control the transition between each profile.



This case study focuses on the use of the Offset and the Ruled Surface commands.

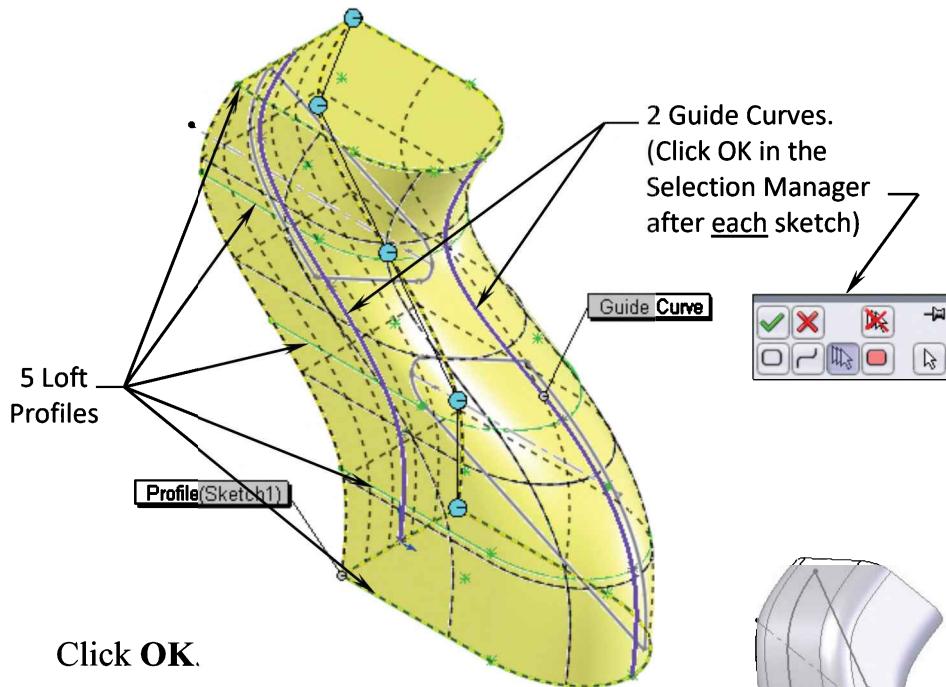


2. Creating the Base Loft:



Click or select Insert / Boss-Base / Loft.

Select the **5 Loft Profiles** and the **2 Guide Curves** as noted.
(The SelectionManager appears when disjointed or overlapped entities are found in the sketch.)



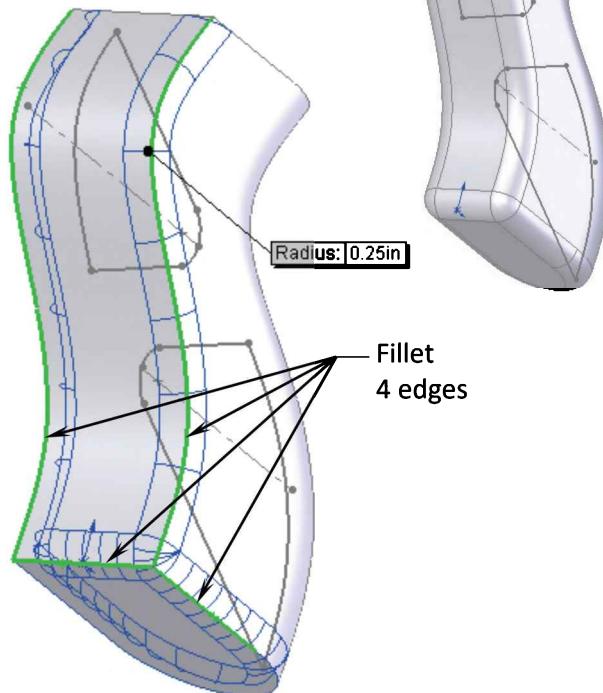
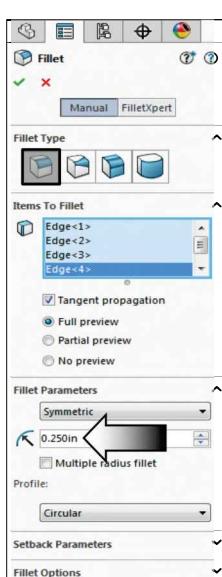
3. Adding .250" fillets:

Click or select
Insert / Features /
Fillet-Round.

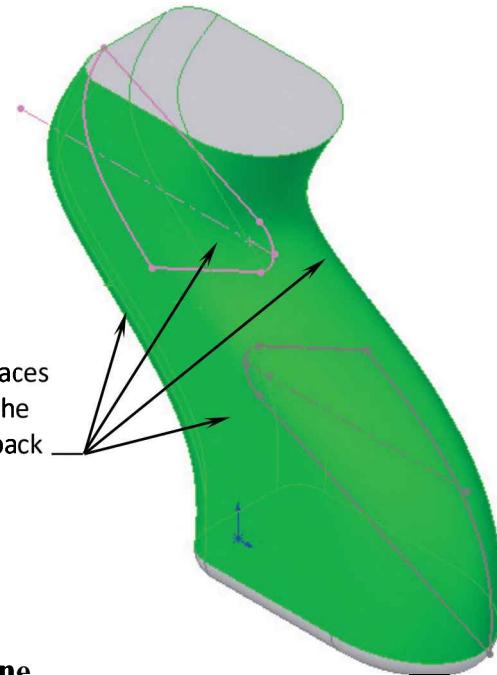
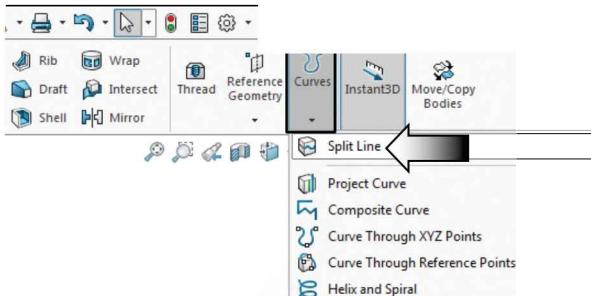
Enter **.250in.** for
radius value.

Select the **4 edges**
indicated to add the
fillets.

Click **OK**.



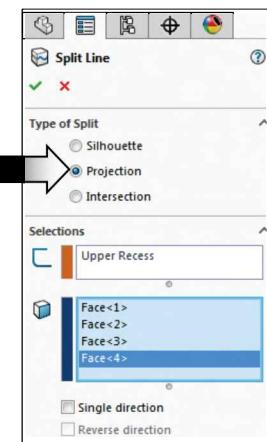
The Split Line command projects an entity to the surface(s) and divides the surface into multiple, separate faces.



4. Creating the 1st Split Line:

Click or select Insert / Curves / Split Line.

For Type-of-Split, select the **Projection** option (Default).

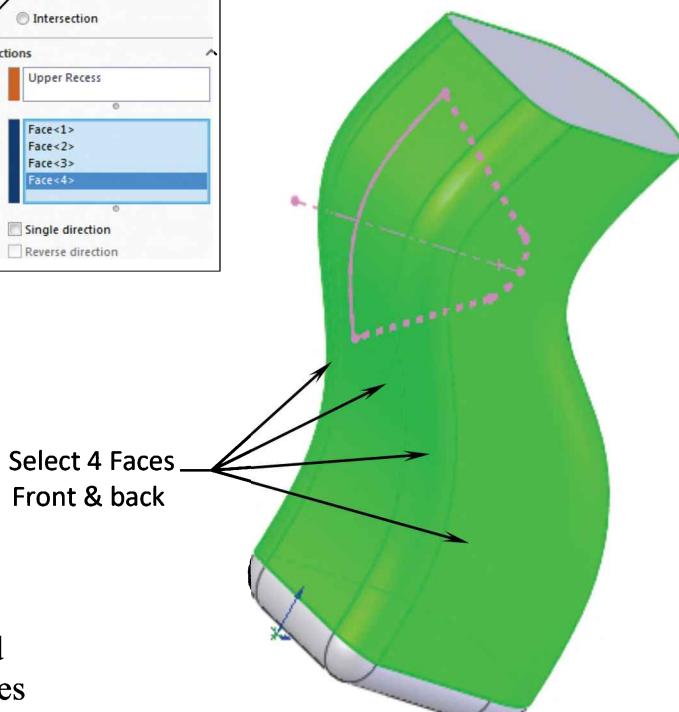


For Sketch-To-Project, select the sketch **Upper Recess** from the Feature-Manager tree.

For Faces-To-Split, select the **4 faces** as indicated.

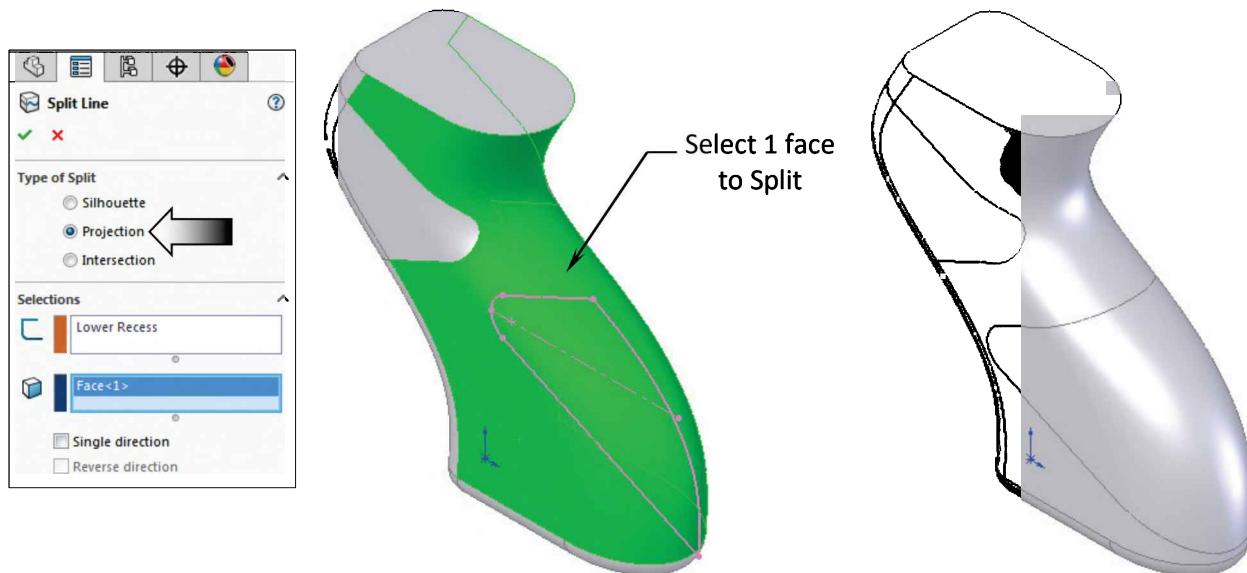
Click **OK**.

The handle body now has a new group of faces; they will be used to create the two recessed features in the next few steps.



5. Creating the 2nd Split Line:

Using the sketch **Lower Recess** from the FeatureManager tree, repeat step number 3 to create the 2nd Split Line.



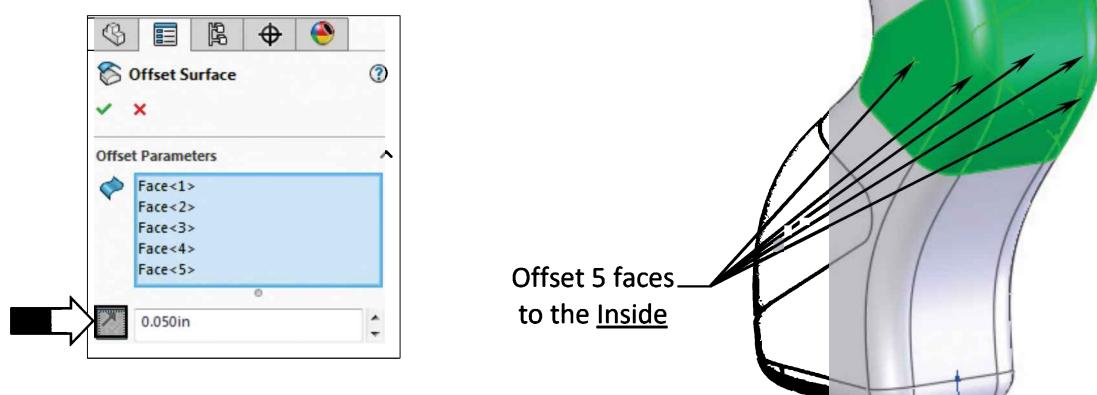
6. Creating the 1st Surface-Offset:

Click or select **Insert / Surface / Offset**.



Select the **5 Split-Faces** to offset.

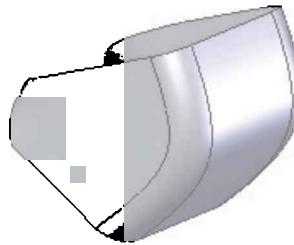
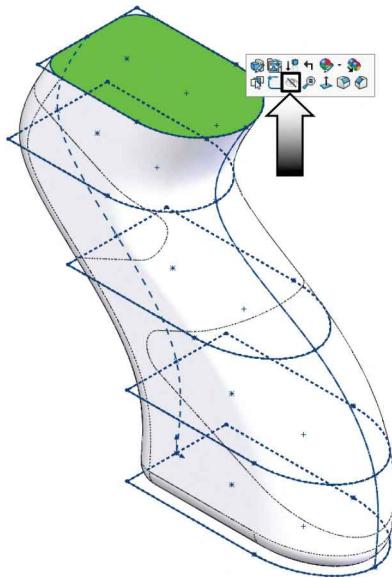
For Offset Distance, enter **.050in**. and click the **Reverse** button (Inside).



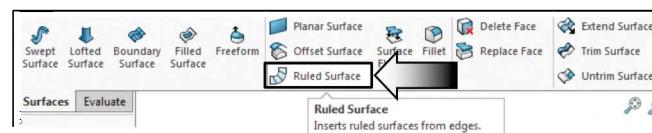
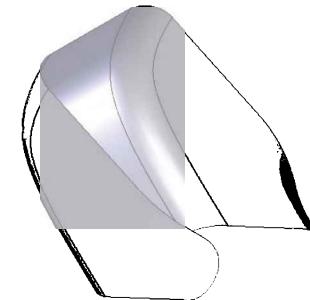
Click **OK**.

7. Hiding the Solid Body:

Right-click on the **upper surface** of the solid body and select **Hide** .

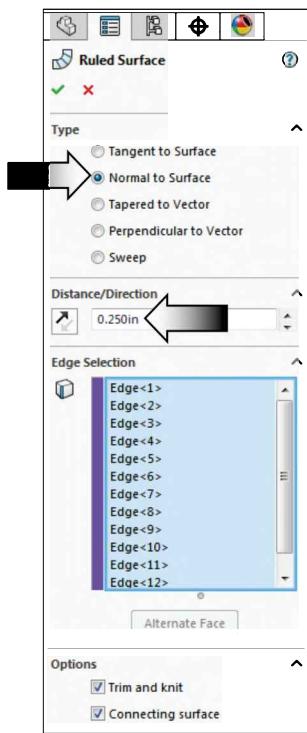


Inspect your surface offset against the one shown here.



8. Creating the 1st Ruled Surface:

Click  on the **Surfaces** tab or select: **Insert / Surface / Ruled Surface**.

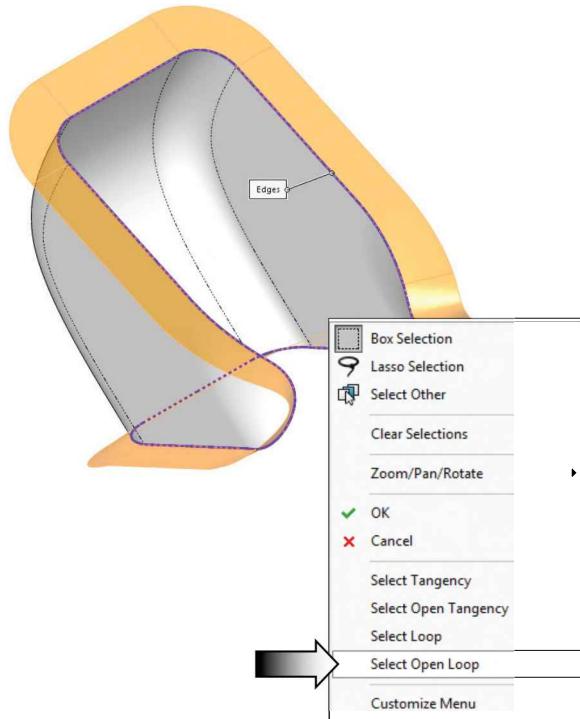


Select the **Normal-To-Surface** option.

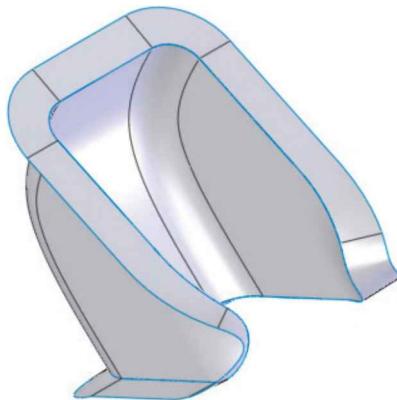
Enter **.250in.** for Offset Distance.

Right-click one of the **outer edges** and select: **Select-Open-Loop**.

Click **OK**.



Rotate to different orientations and inspect the ruled surface.

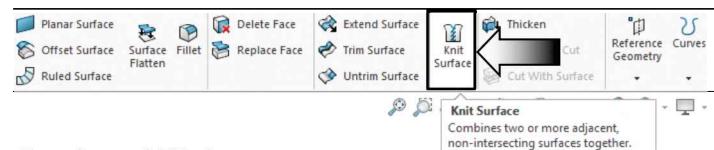


Ruled Surfaces

The Ruled Surfaces creates a set of surfaces that either perpendicular or taper from the selected edges. These surfaces can also be used as the Interlock Surfaces in molded parts.

9. Knitting the 2 surfaces:

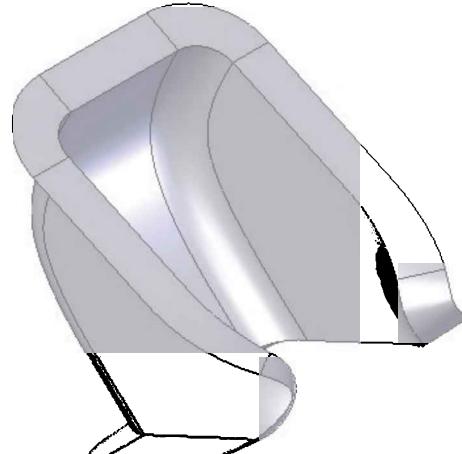
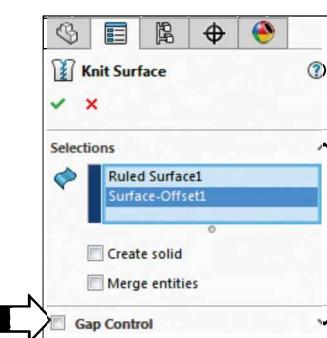
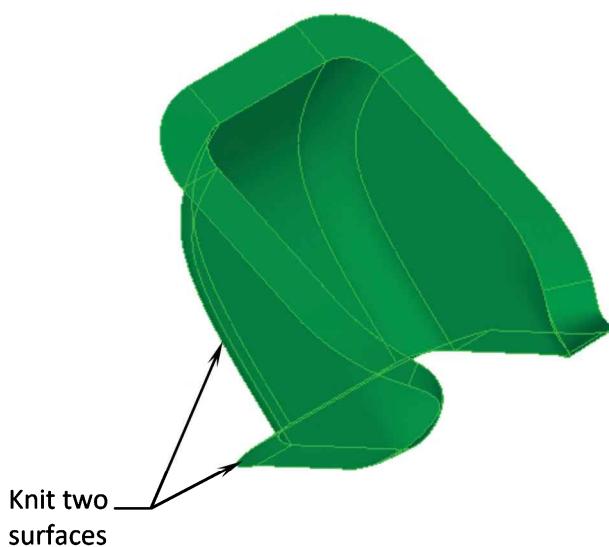
Click or select Insert / Surface / Knit.



Select the Surface-Offset and the Ruled-Surface to knit.

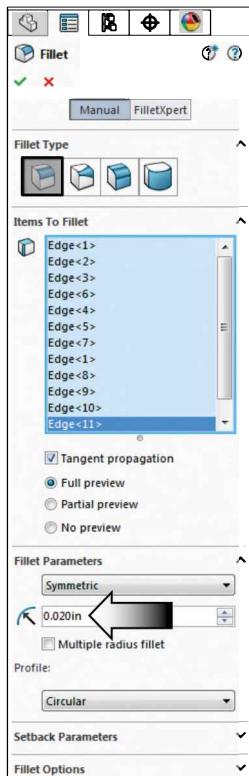
Clear the Gap Control checkbox (arrow).

Click OK.



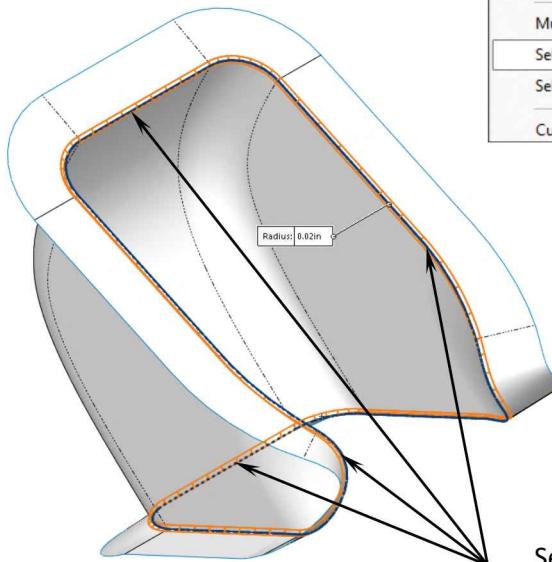
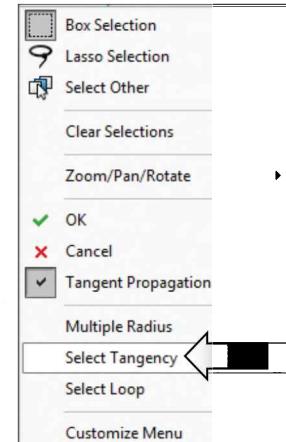
10. Adding .020" Fillets:

Click  or select: Insert / Features / Fillet-Round.



Enter **.020in.** for Radius value.

Select all **Inner Edges** to fillet.
(Right click / Select Tangency.)



Click **OK**.

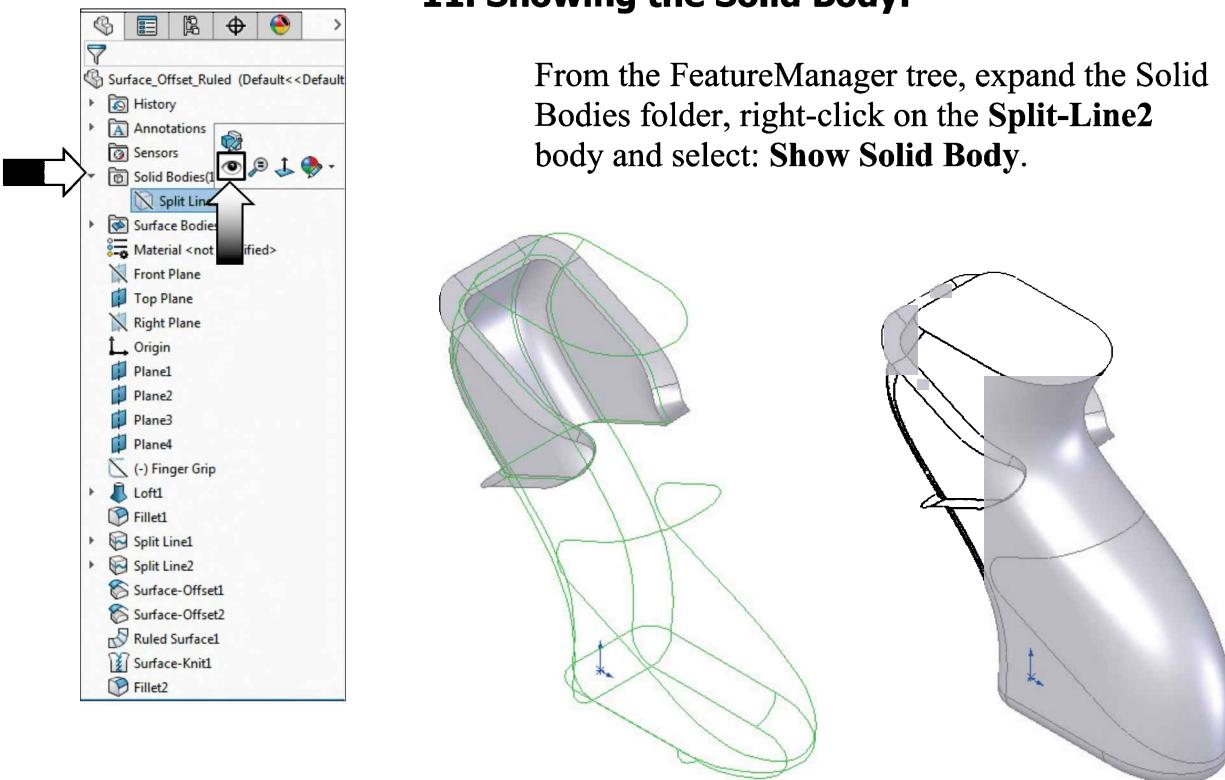
Change to different orientations to inspect the new fillets.



Verify that all inner edges are filleted.

11. Showing the Solid Body:

From the FeatureManager tree, expand the Solid Bodies folder, right-click on the **Split-Line2** body and select: **Show Solid Body**.

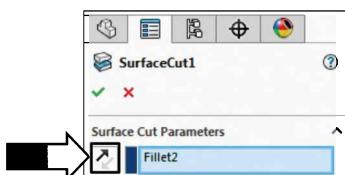


12. Creating the Surface Cut:

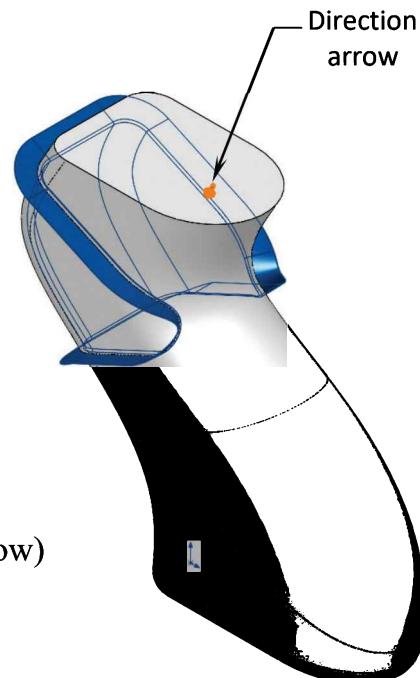
Click or select **Insert / Cut / With Surface**.



Select the **Surface-Knit1** either from the graphics area or from the FeatureManager tree.



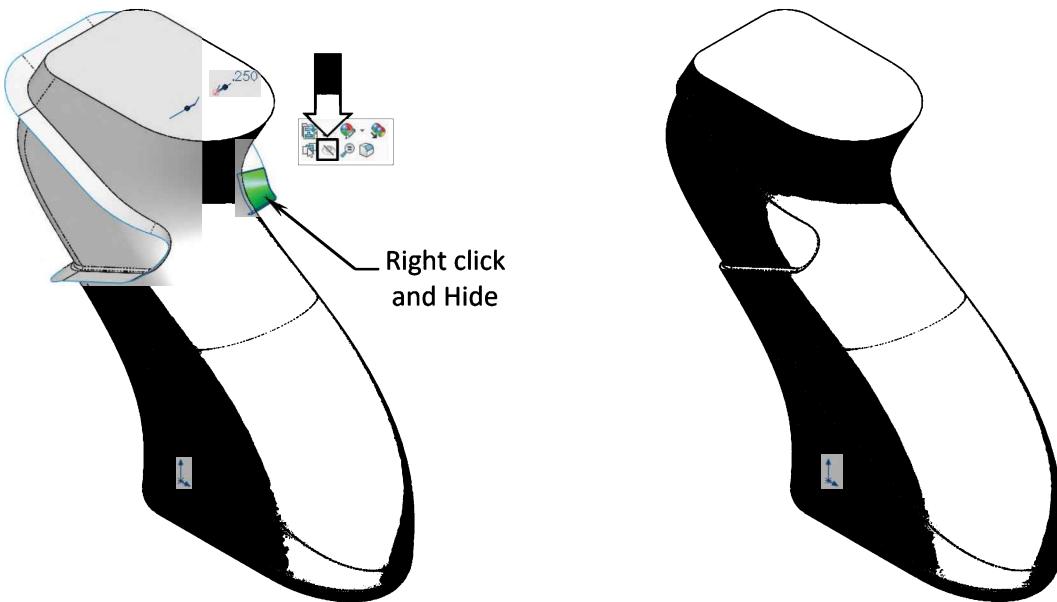
The Direction Arrow indicates the side that will be removed by the cut. Click Reverse (arrow) if needed.



Click **OK**.

13. Hiding the Knit Surface:

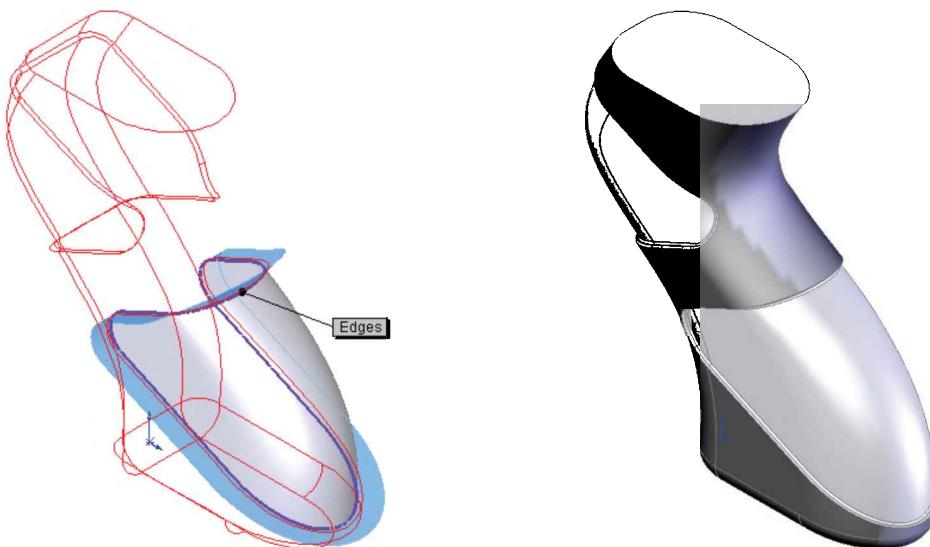
Right-click on the **Knit Surface** and select **Hide** .



14. Creating the 2nd Ruled Surface:

Repeat from step number 8 to create the 2nd Ruled Surface.

Create another **Surface Cut** as indicated in step 12.



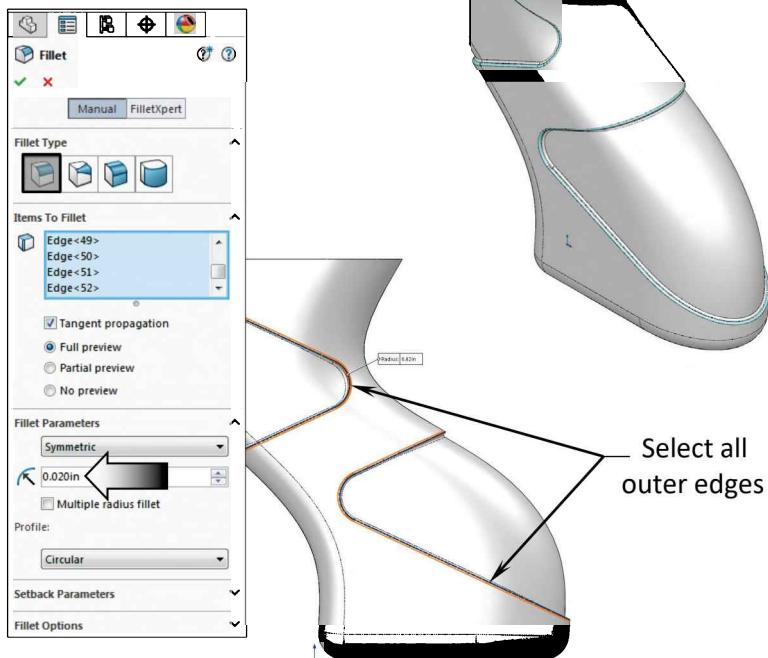
15. Adding more Fillets:

Click  or select:
Insert / Features /
Fillet-Round.

Enter **.020in.** for
radius.

Select the outer edges
of both upper and
lower cuts to add
the fillet.

Click **OK**.



16. Shelling the part:

Select the upper face as noted.

Click  or select Insert /
Features / Shell.

Enter **.020in.** for thickness.

Click **OK**.



17. Saving your work:

Click File / Save As.

Enter: **Surface_Offset_Ruled** for file name.

Click **Save**.

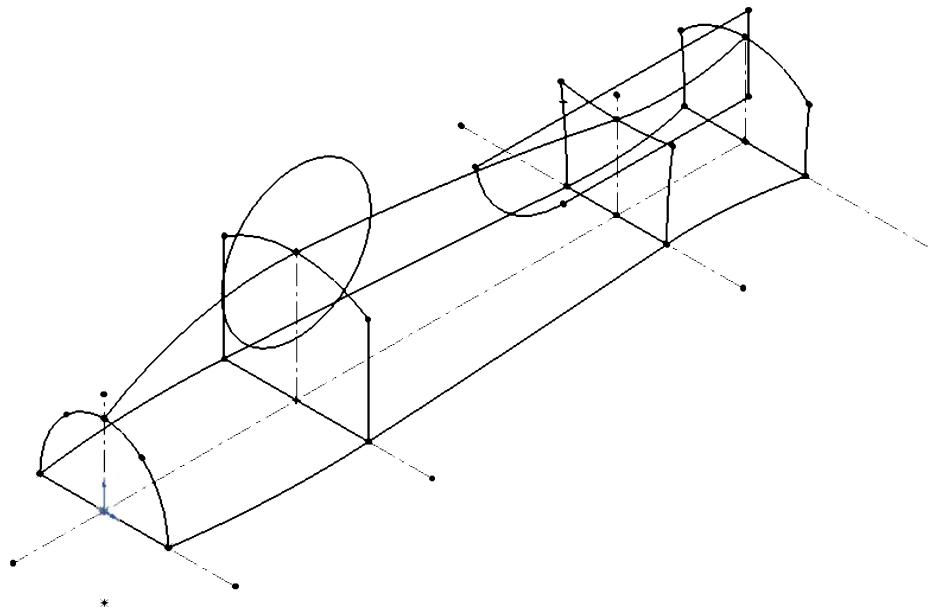
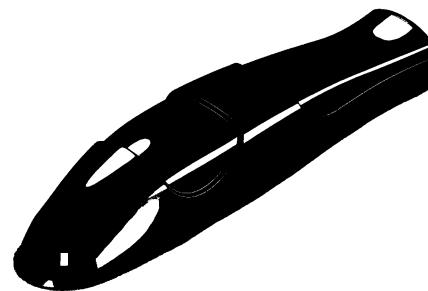


Exercise: Using Loft

1. Opening an Existing file:

Browse to the Training Files folder.

Open the document named:
Advanced Surfaces Exercise.

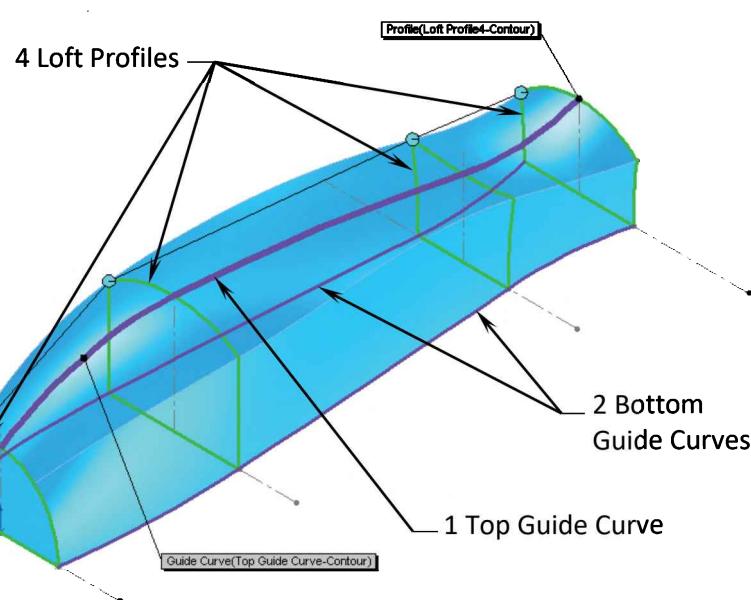


2. Creating the Loft body:

Create a **Solid Loft** from the **4 profiles** as indicated.

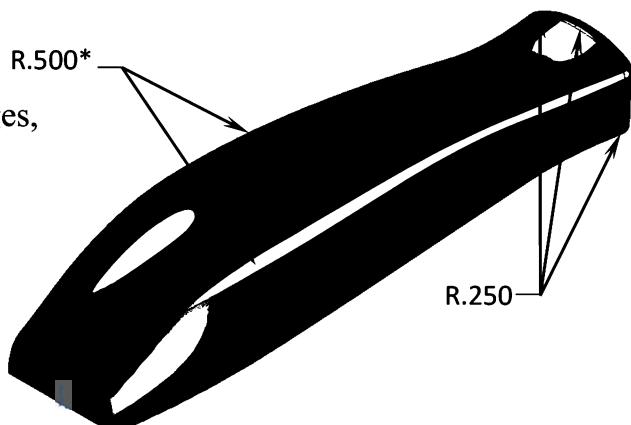
Use the **2 bottom Guide Curves** to control the 2 sides.

Use the top **Guide Curve** to control the upper curvatures.



3. Adding Fillets:

Add a **.500in.** fillet to the upper edges, and a **.250in.** fillet to the edges on the right end, as indicated.

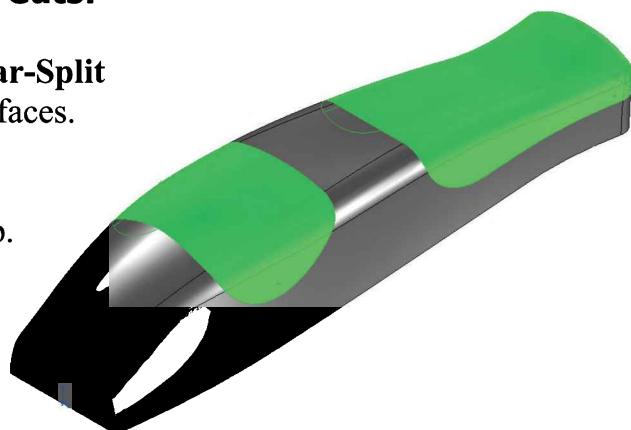


* The tangent edges can be eliminated by enabling the Merge-Tangent-Faces in the loft options.

4. Creating the Split Lines & Lofted-Cuts:

Use the 2 sketches named: **Circular-Split** and **Side-Split** to create 2 split surfaces.

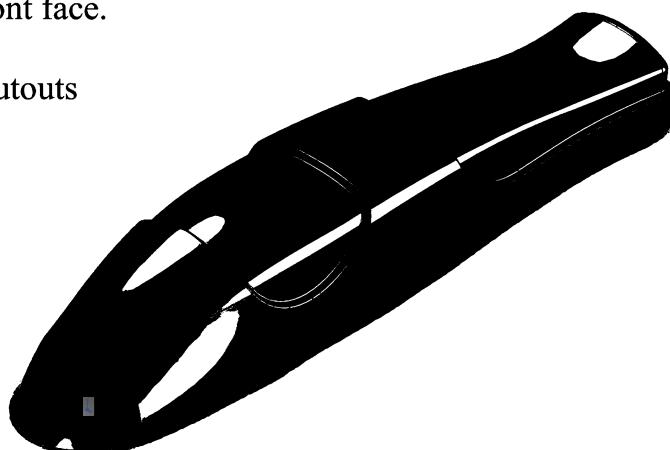
Create 2 lofted cuts at **.090in.** deep. Use either the **Offset** or **Ruled** surface options to make the cuts.



5. Adding the Nose and Fillets:

Add the Tip feature that measured between **1.250in.** to **1.500in.** from the front face.

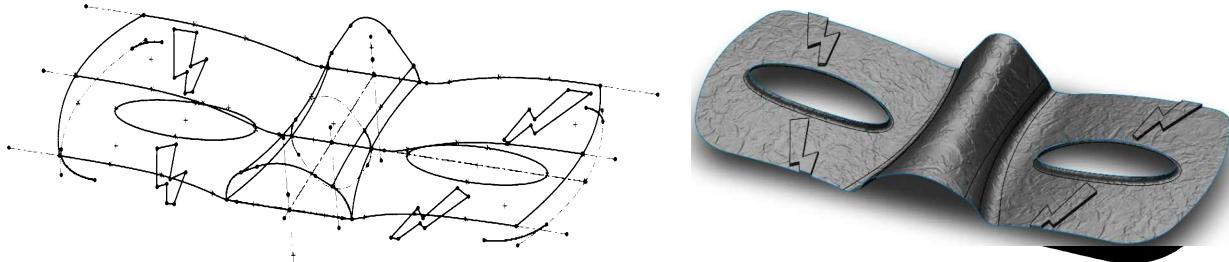
Replace all sharp edges of the cutouts with **.040in.** fillets.



6. Saving your work:

Save the exercise as:
Advanced_Surfaces_Exe.

Exercise: Advanced Surfacing Techniques



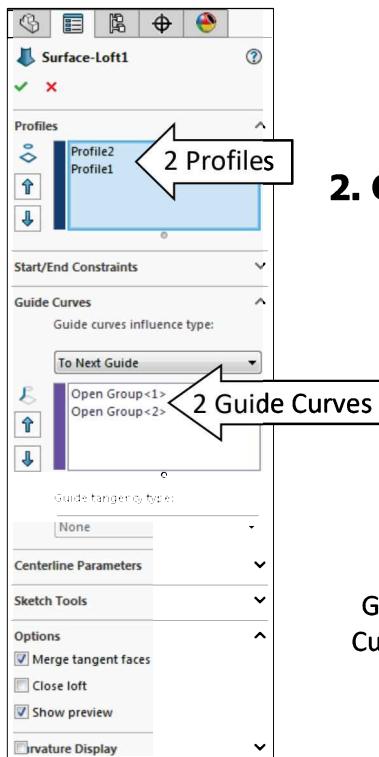
Surfaces

Surfaces are a type of geometry that can be used to create solid features. Surfaces can be created in a variety of different ways, from a sketch or multiple sketches, a surface can be made by extruding, revolving, sweeping, and lofting.

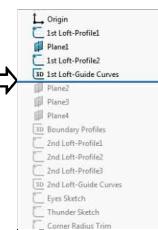
Surfaces are normally created individually and knitted together so that an enclosed volume or a solid feature can be generated afterwards.

This exercise discusses some advanced techniques on surfacing such as Lofted Surface, Boundary Surface, Trimmed Surface, Offset Surface, Extrude From, and variable fillets.

1. Opening an existing document named:



Advanced Surfacing Techniques from the Training Files folder. Rollback below the sketch: 1st Loft Guide-Curves.

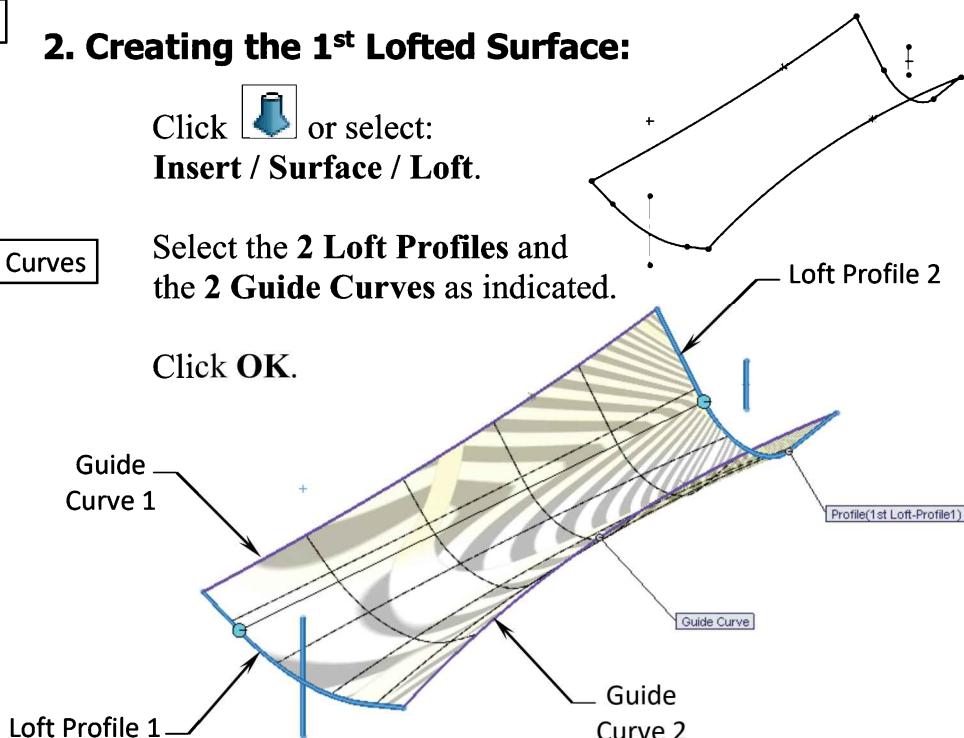


2. Creating the 1st Lofted Surface:

Click or select: **Insert / Surface / Loft**.

Select the **2 Loft Profiles** and the **2 Guide Curves** as indicated.

Click **OK**.



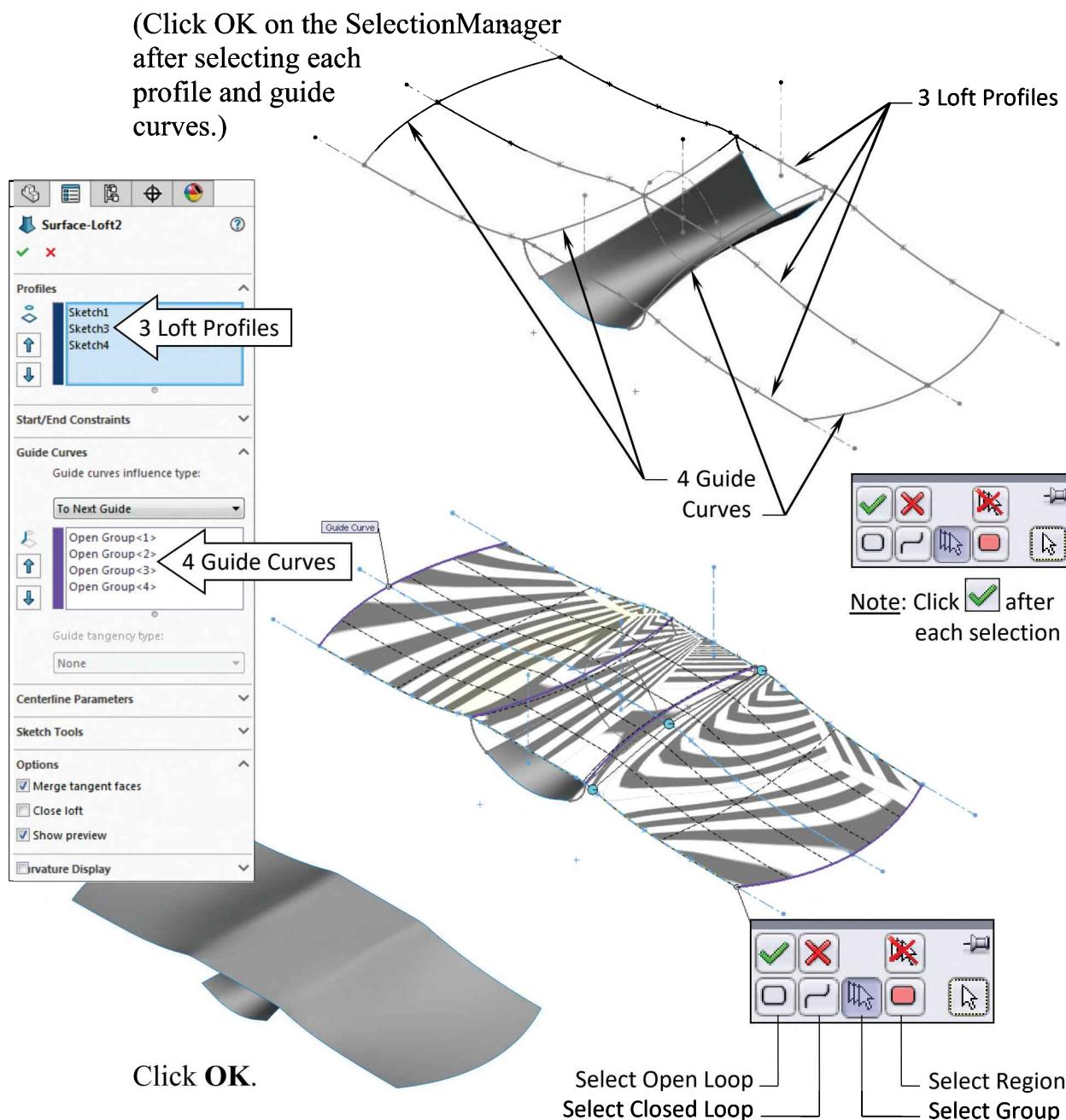


Lofted-Surface creates a feature by making transitions between two or more profiles. A loft can either be a surface or solid and one or more Guide Curves can be used to guide the transitions between the profiles.

3. Creating the 2nd Lofted Surface:

Click or select: **Insert / Surface / Loft**.

Select the **3 Loft Profiles** and the **4 Guide Curves** as indicated.

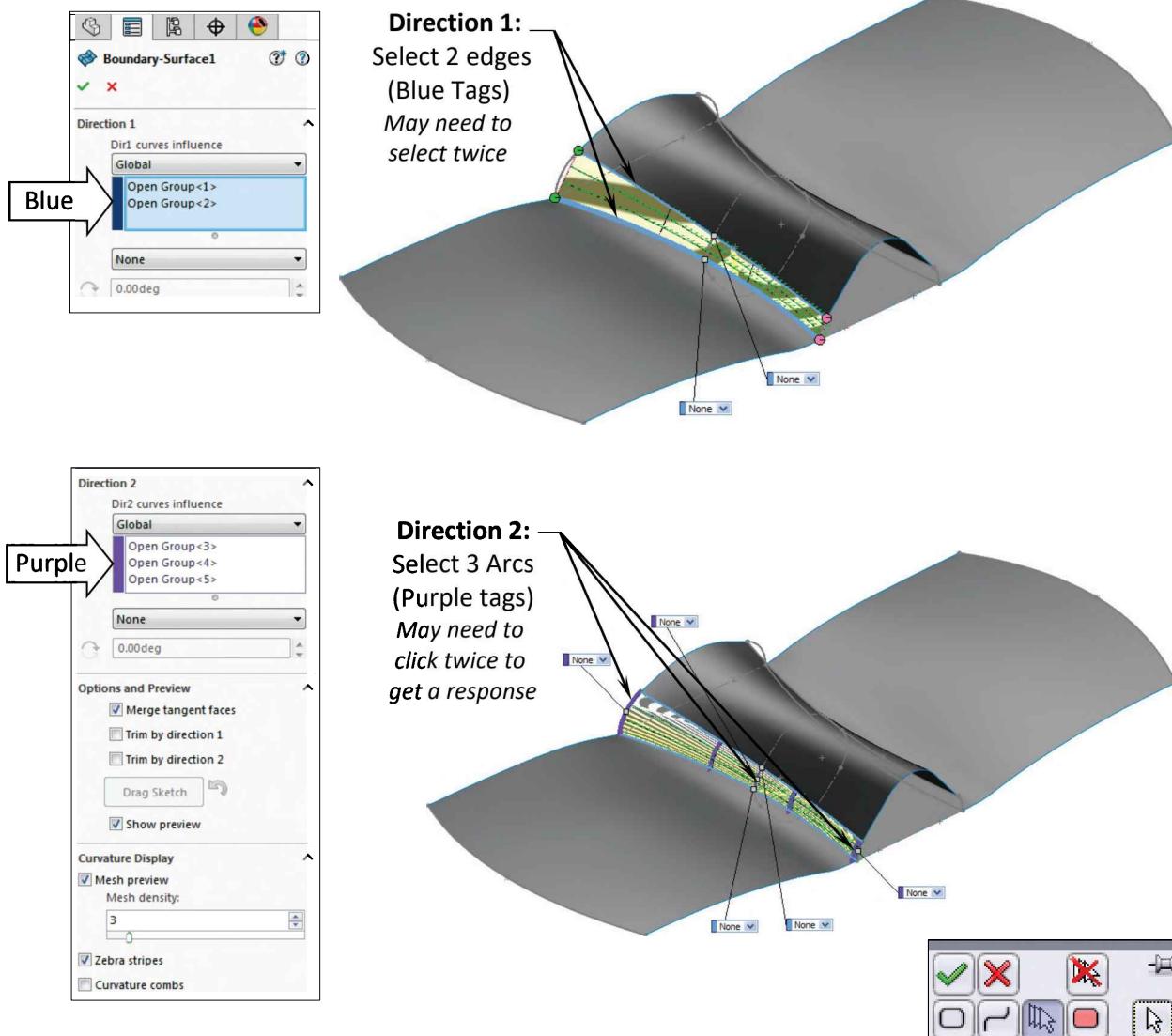
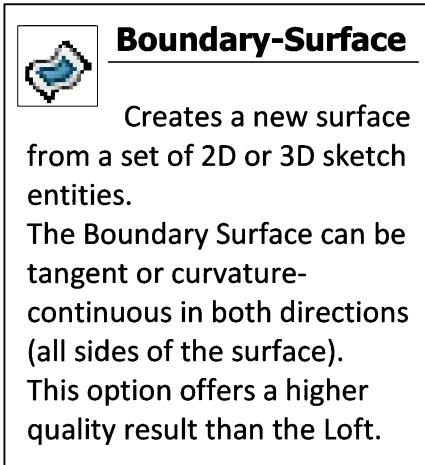


4. Creating the Boundary-Surfaces:

Click  or select Insert / Surface / Boundary Surface.

For **Direction 1**, select the **2 edges** as shown Below (Blue tags).

For **Direction 2**, select the **3 Arcs** in the Boundary Sketch as shown (Purple tags).

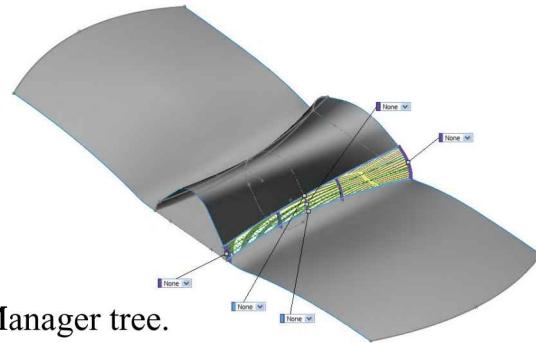


Click **OK**.

Note: Click  after each selection

5. Repeating:

Repeat step number 4 and create another Boundary Surface on the opposite side.



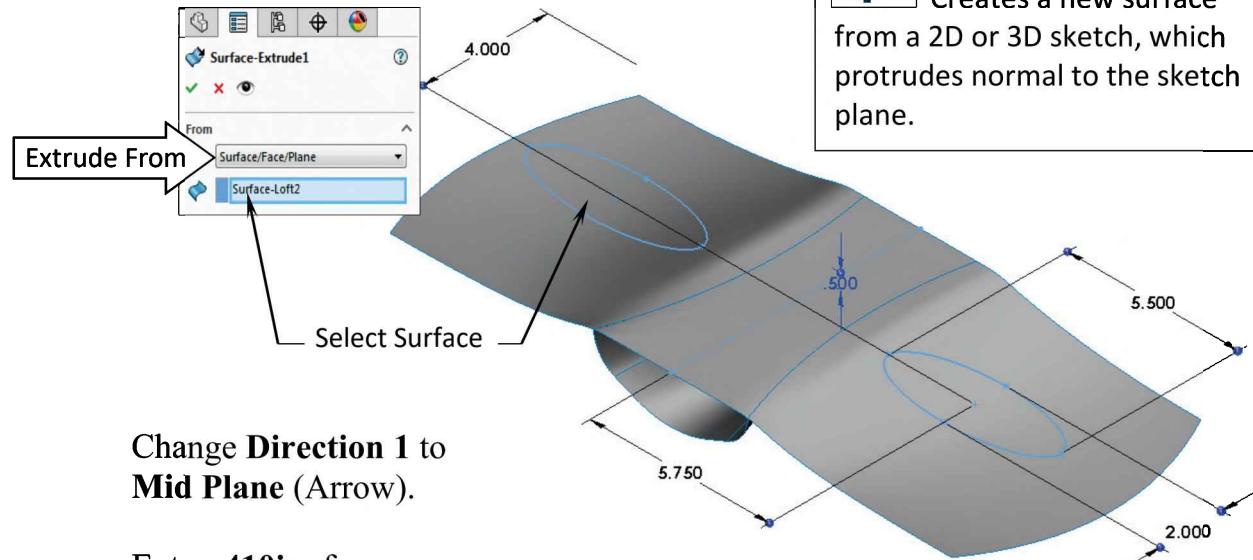
6. Creating an Extruded-Surface:

Select the **Eyes Sketch** from the FeatureManager tree.

Click or select: **Insert / Surface / Extrude**.

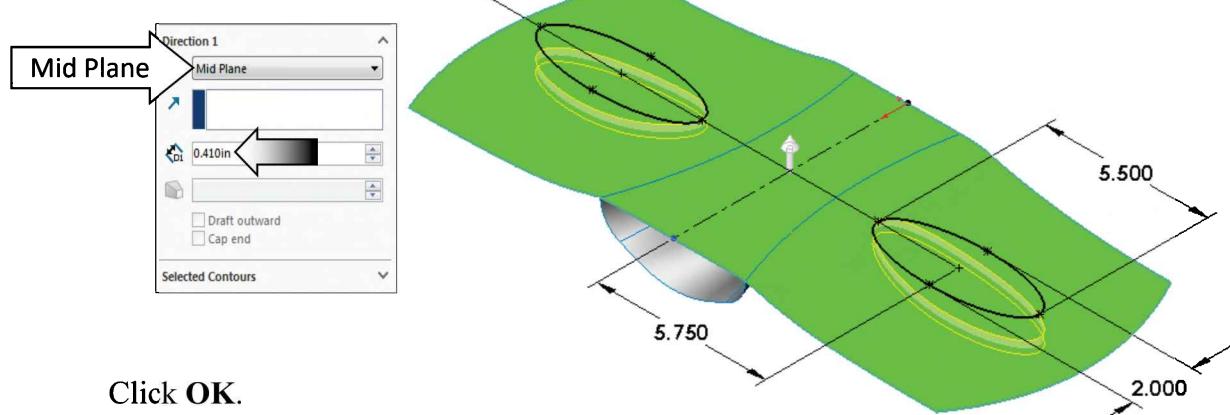
Change the option **Extrude From** to **Surface/Face/Plane** (Arrow).

Select **Surface-Loft2** as noted to extrude from.



Change **Direction 1** to **Mid Plane** (Arrow).

Enter **.410in.** for extrude depth.



Click **OK**.

7. Creating a Trimmed Surface:

Click  or select Insert / Surface / Trim.

Select **Mutual** under Trim Type (Arrow).

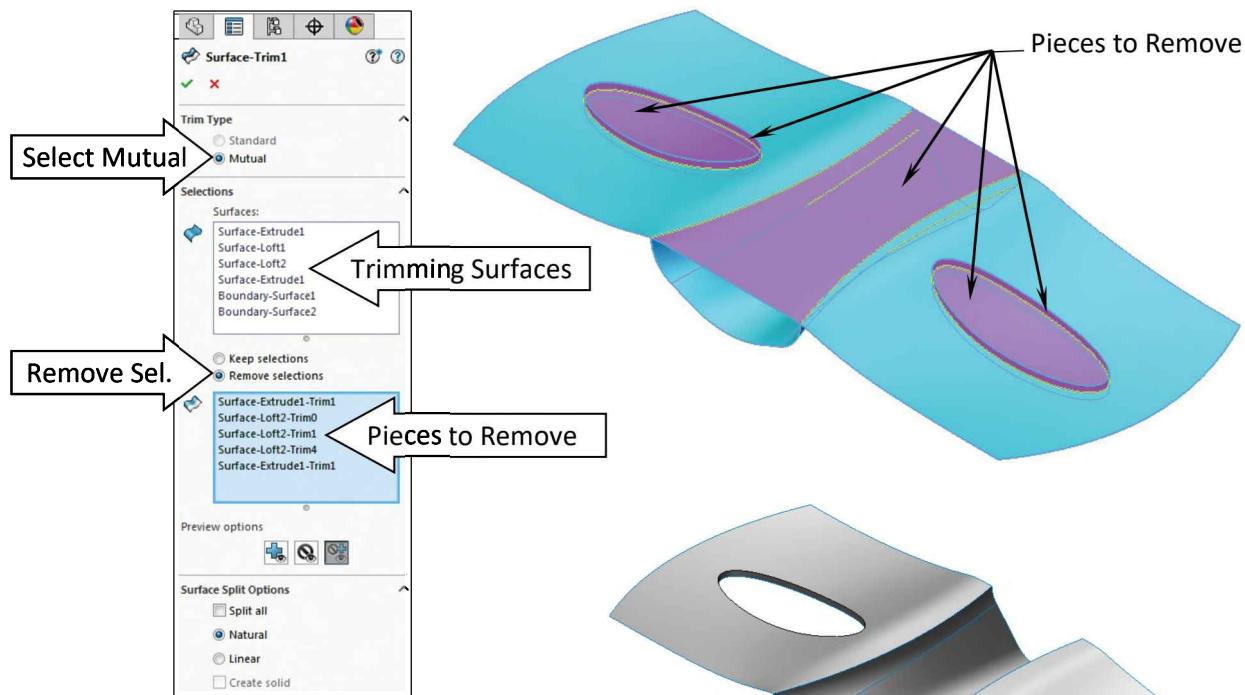


Trimmed-Surface

Uses a plane, a surface, or a sketch as a trim tool to trim the intersecting surfaces.

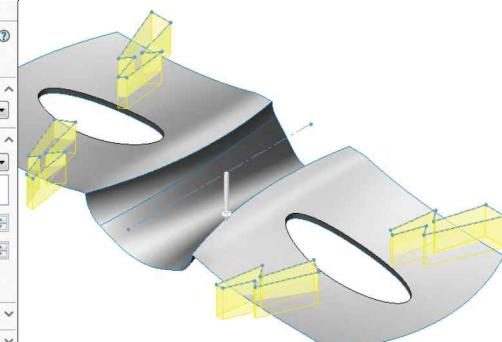
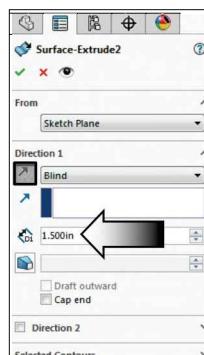
For Trimming Surfaces, select all surfaces of the model.

For Remove Selections, select the **5 Faces** as shown (Arrow).



8. Creating another Extruded Surface:

Select the **Thunder Sketch** and click  or select Insert / Surface /Extrude.



Set the **Direction 1** to **Blind** and click Reverse Direction.

Set Extrude Depth to:
1.250in.

Click **OK**.

9. Creating an Offset-Surface:

Click  or select Insert / Surface / Offset.

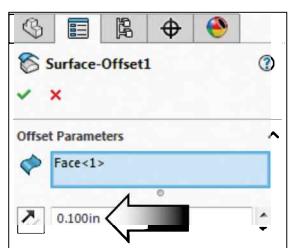
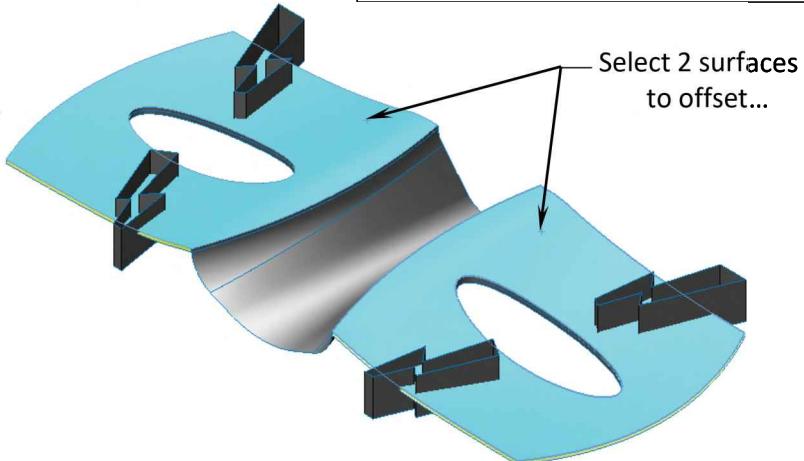
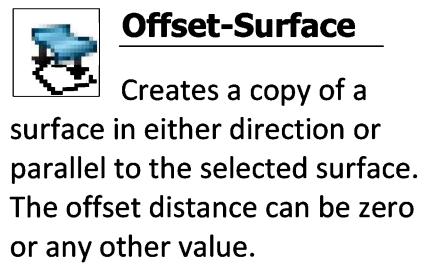
Select the **2 surfaces** as shown (arrow).

Enter **.100in.** for Offset Distance.

Place the copy on the **bottom** of the original.

Note: Create 2 offset surfaces separately if the next trim failed.

Click **OK**.

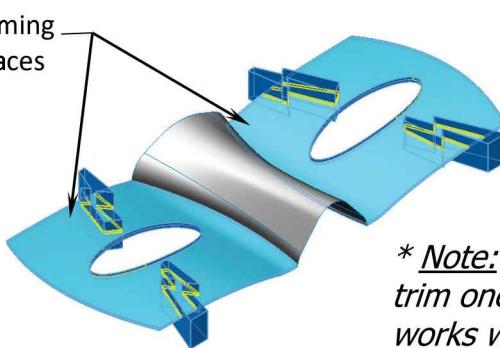
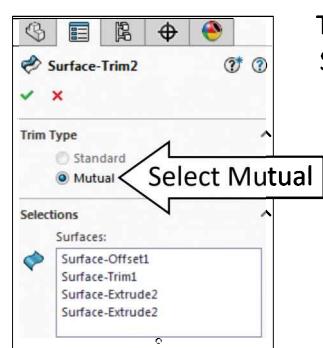
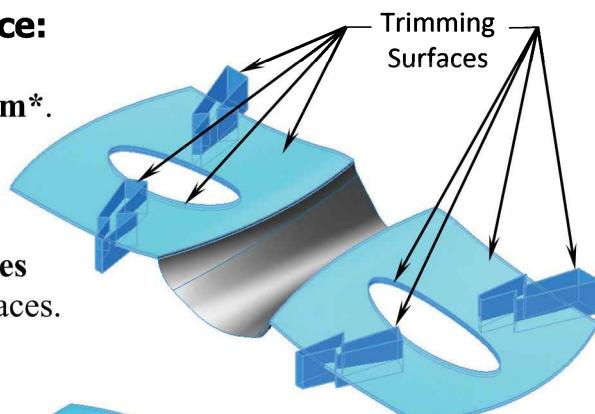


10. Creating a Mutual Trimmed Surface:

Click  or select Insert / Surface / Trim*.

Select **Mutual** under Trim Type (Arrow).

For **Trimming Surfaces**, select all surfaces of the Thunder Sketch and the Offset Surfaces.

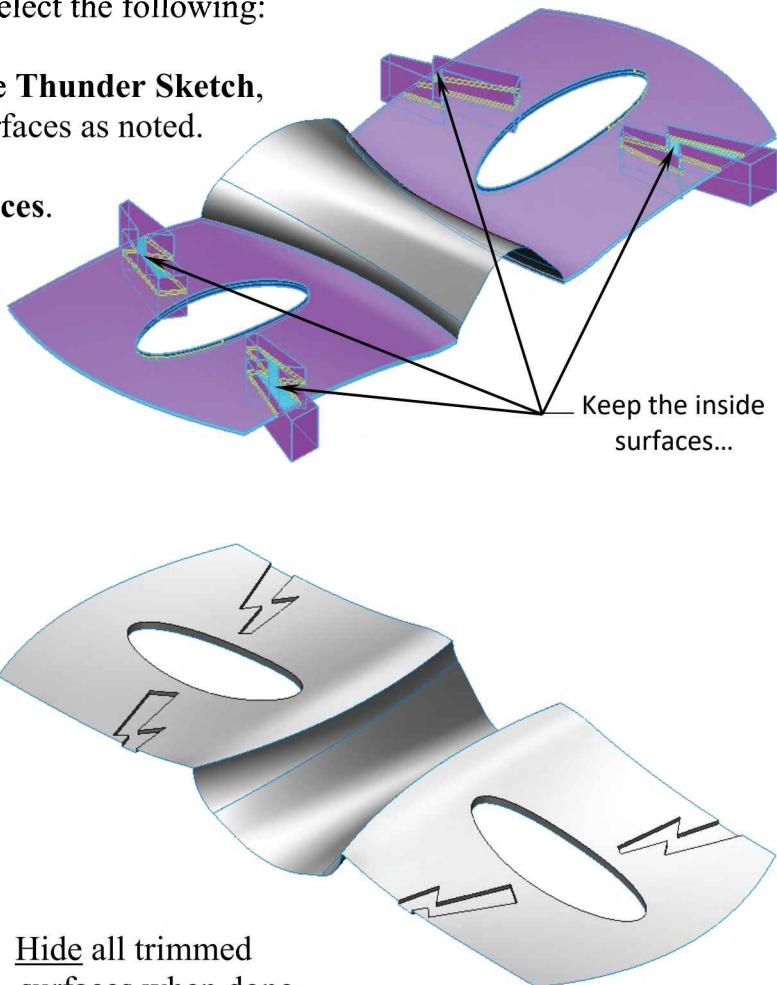
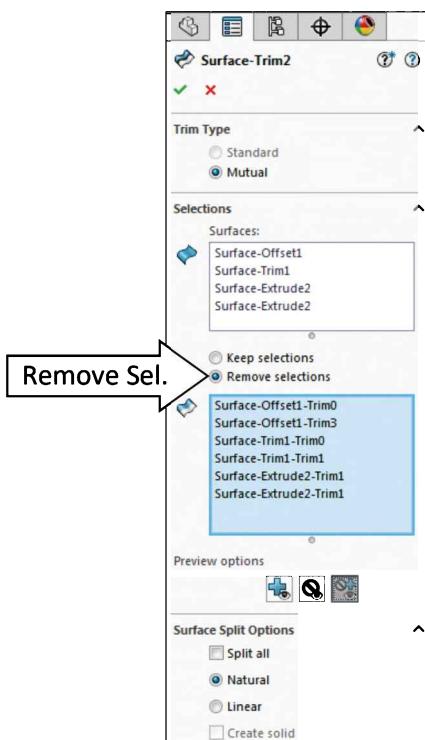


* Note: Using Standard-Trim to trim one feature each time also works well.

For Remove Selection, select the following:

- * The surfaces of the Thunder Sketch,
Keep the inside surfaces as noted.

- * The 2 Offset Surfaces.



Hide all trimmed
surfaces when done.

Click OK.

11. Creating an Extruded Surface :

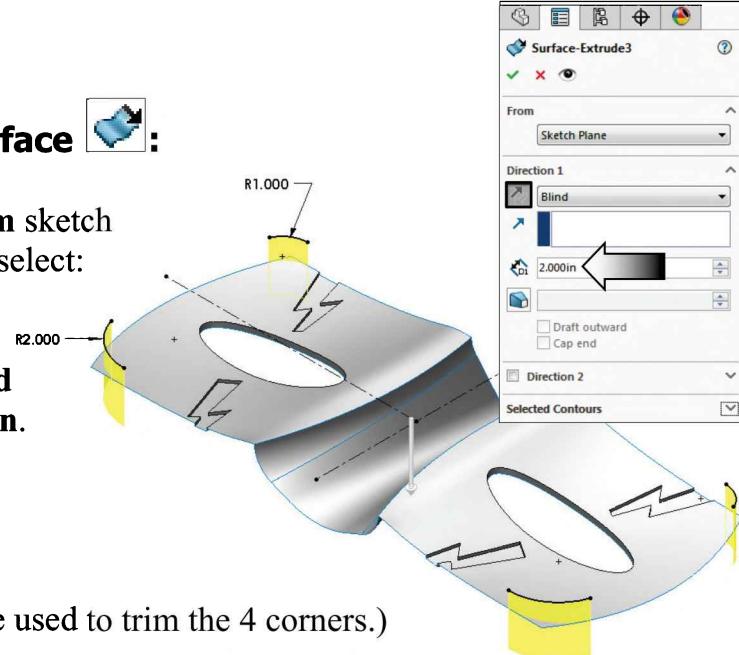
Select Corner Radius Trim sketch
and click Trim Surface or select:
Insert / Surface / Extrude.

Set the **Direction 1** to **Blind**
and click **Reverse Direction**.

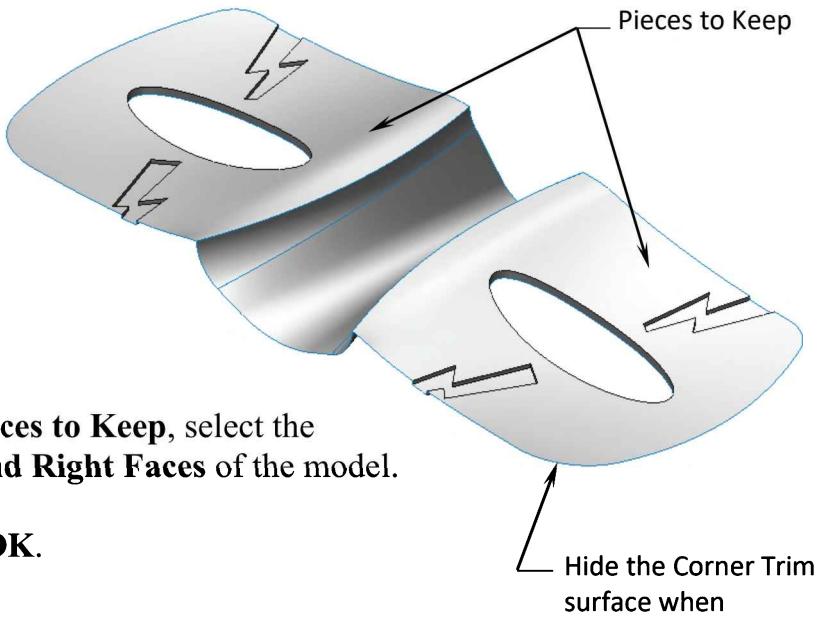
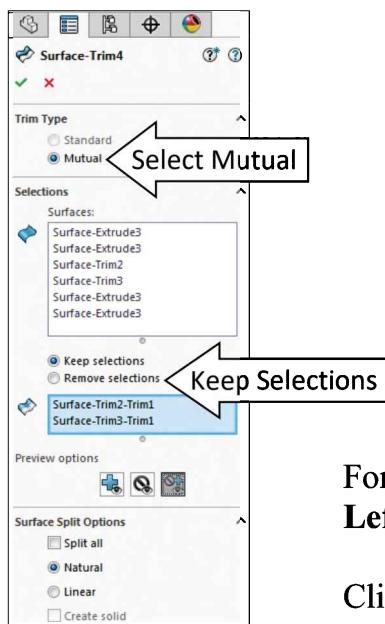
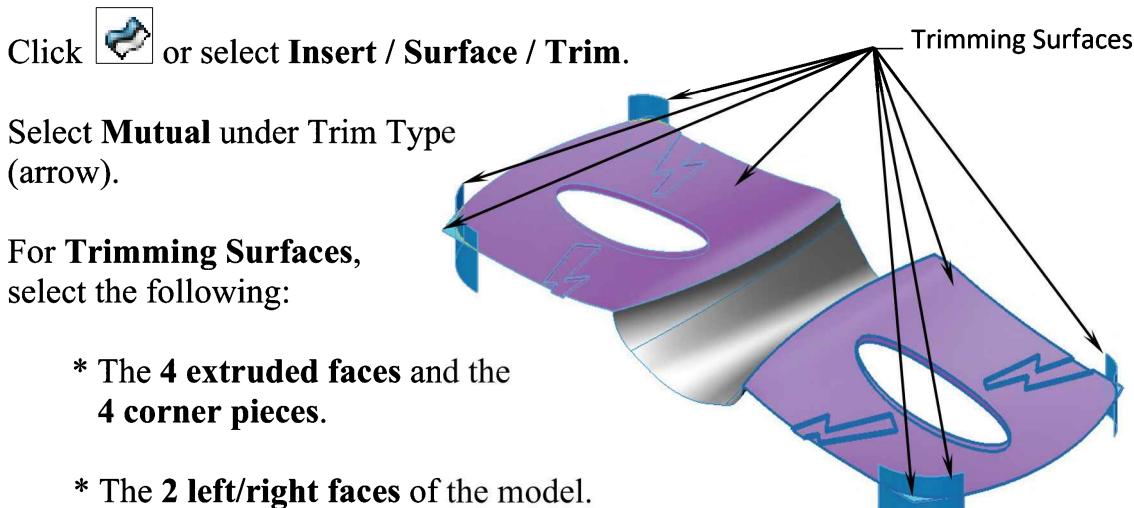
Set the Depth to **2.000in**.

Click OK.

(These new surfaces will be used to trim the 4 corners.)



12. Creating a corner Trimmed Surface:



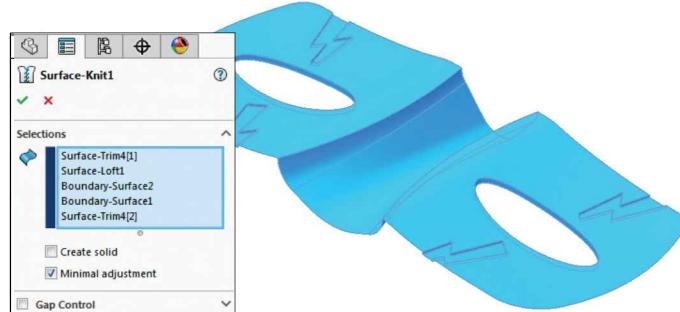
13. Creating a Knit-Surface:

Click or select Insert / Surface / Knit.

Select all surfaces either from the FeatureManager tree or from the graphics area.

Clear the Gap Control box.

Click OK.



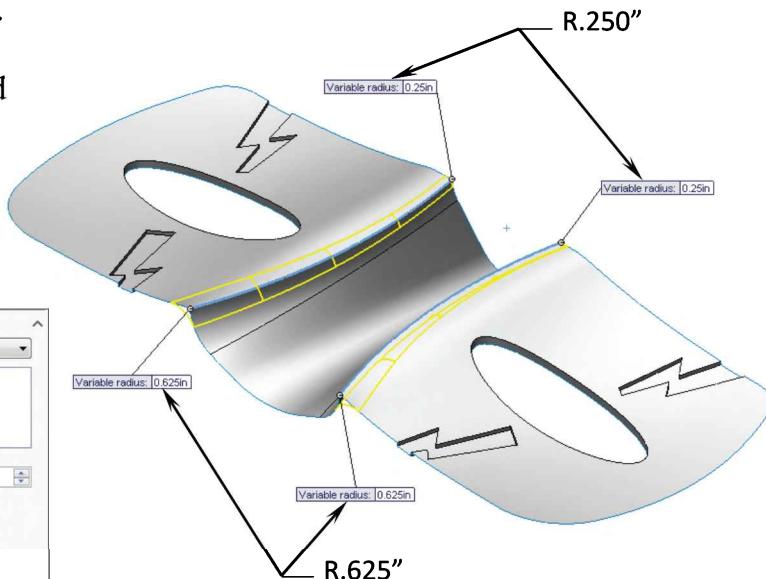
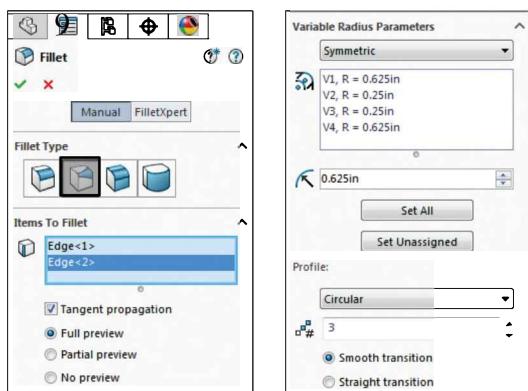
14. Adding a Variable Fillet:

Click  or select Insert / Features / Fillet-Round.

Select the 2 edges shown.

Use the Call-out tags and enter the radius values as noted.

Click OK.



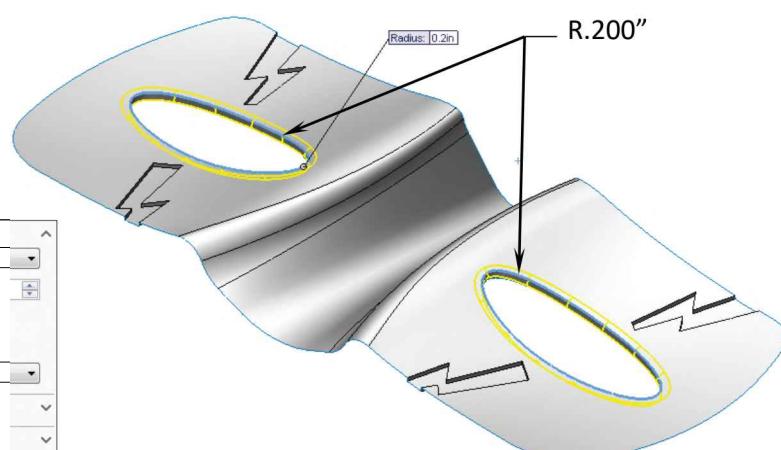
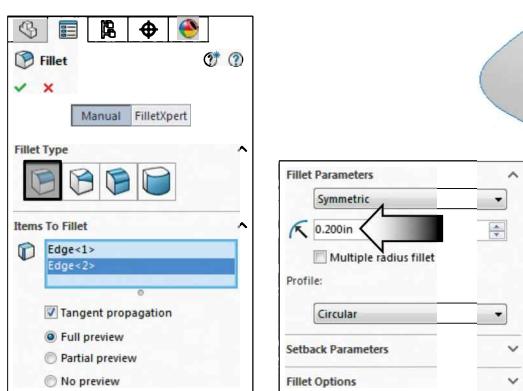
15. Adding a Constant Fillet:

Click  or select Insert / Features / Fillet-Round.

Select the 2 edges as shown below.

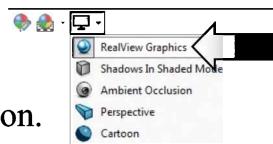
Enter **.200in.** for Radius values.

Click OK.



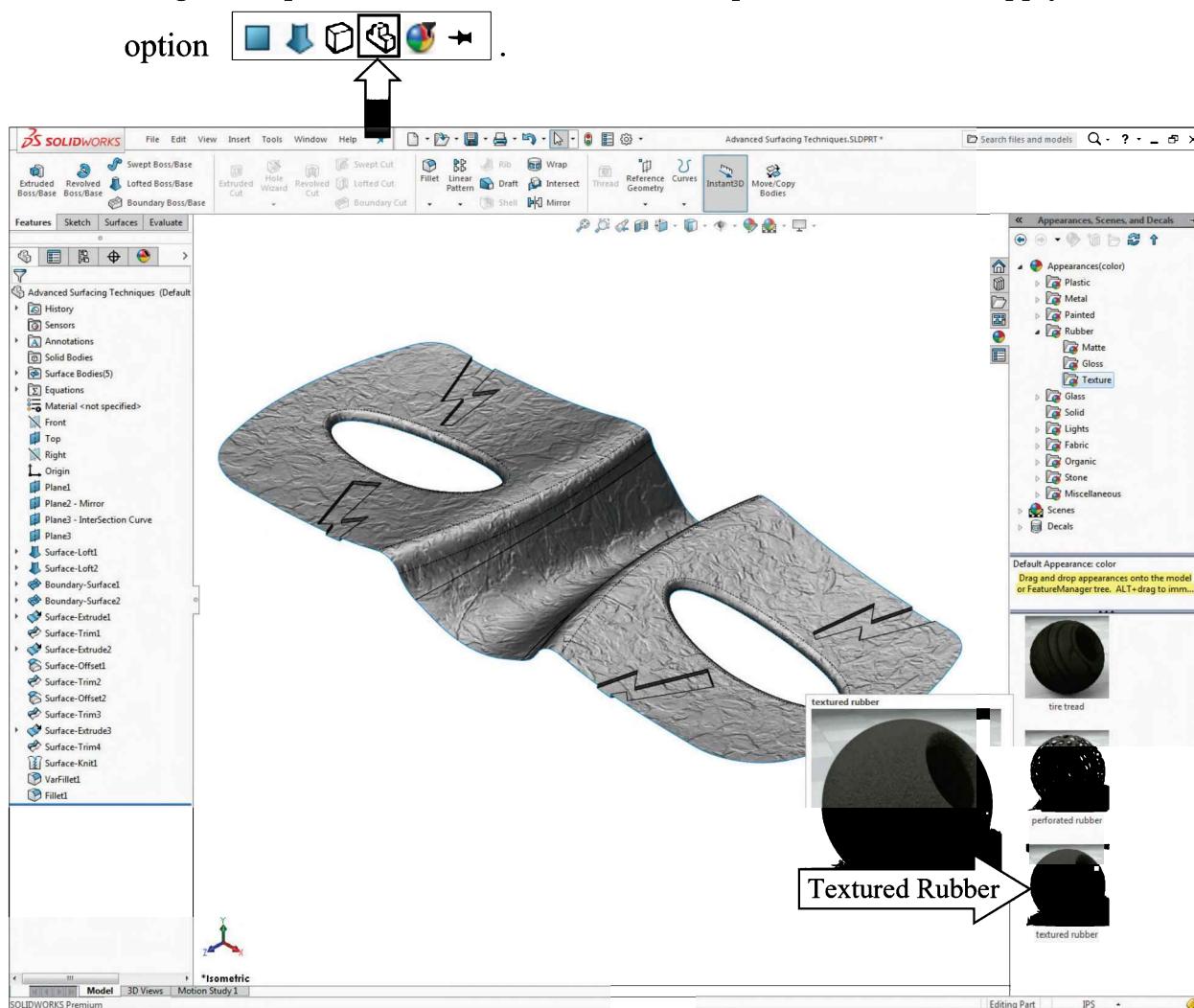
16. Optional: Adding texture

Enable the **RealView Graphics** option.



From the Task Pane expand the **Appearances** folder and locate the **Rubber / Texture** folder.

Drag & Drop the **Textured Rubber** onto the part and select the Apply to Part option



17. Saving your work:

Click **File / Save As**.

Enter **Advanced Surfacing Techniques** for the name of the file.

Click **Save**.

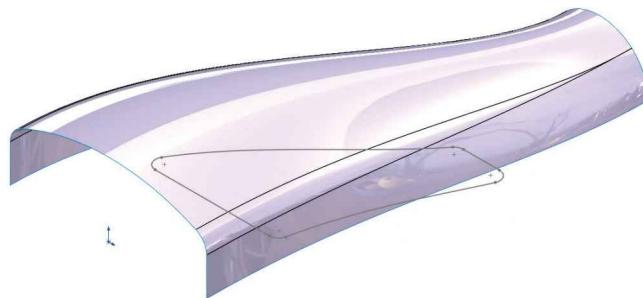
Exercise: Using Split

1. Opening a part document:

Browse to the Training Files folder and open a part document named:
Using Split. Sldprt.

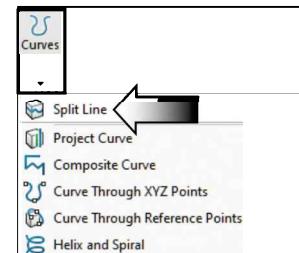
This model contains a single surface body and a 2D sketch.

The 2D sketch will be used to create the split line in the next step.



2. Creating a split line:

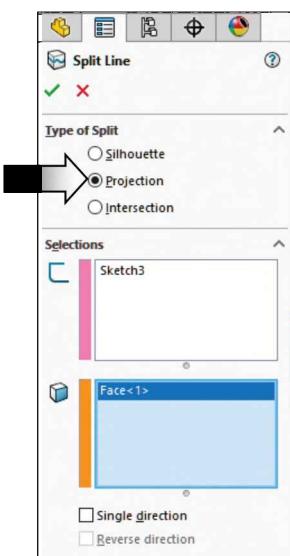
Switch to the **Surfaces** tab.



Expand the **Curves** drop-down and select **Split Line**.

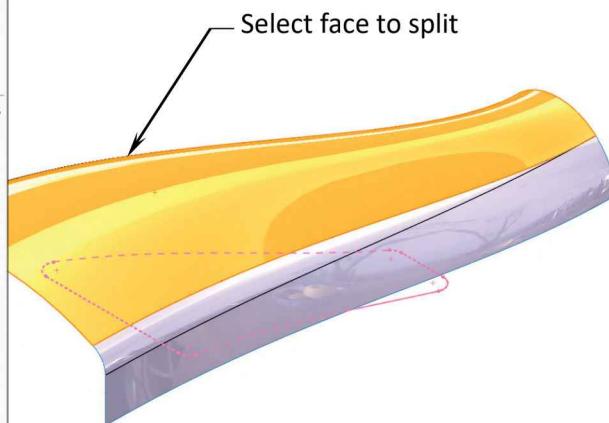
For Type of Split, use the default **Projection** option.

For Selections select **Sketch3**.



For Faces to Split, select the **upper face** of the model as indicated.

Click **OK**.

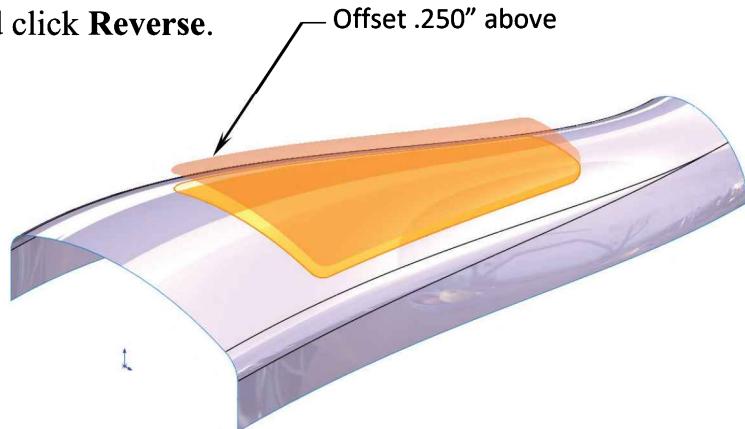
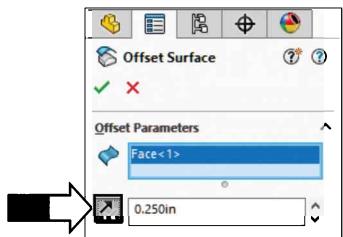
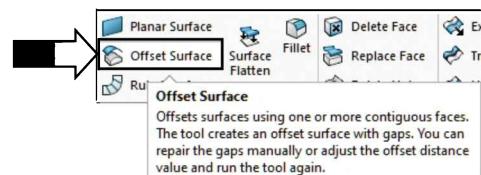


3. Making an offset surface:

Click **Offset Surface** on the Surfaces tab.

For Offset Parameters, select the **inside** of the split surface.

For Distance, enter **.250"** and click Reverse.



Click **OK**.

4. Adding a ruled surface:

Click **Ruled Surface**.

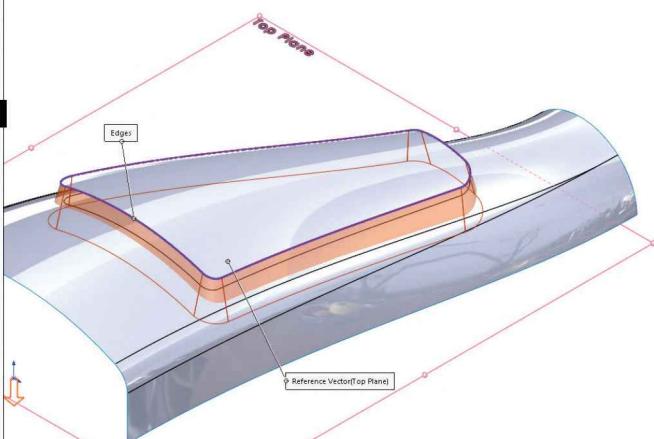
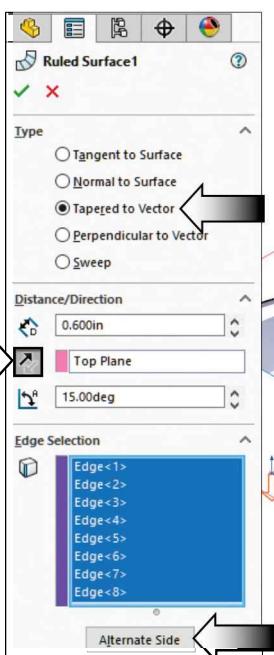
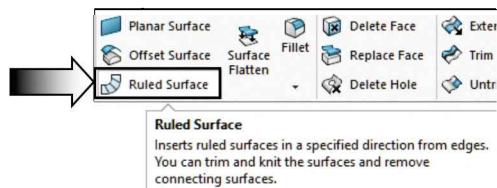
For Type, click **Tapered to Vector**.

For Distance enter **.600in**.

For Direction select **Top plane** and click **Reverse**.

For Angle, enter **15deg**.

For Edge Selection, select all edges of the **Offset Surface**, click Alternate Side to protrude the ruled surface outward, intersecting with the model. Click **OK**.



5. Deleting a surface:

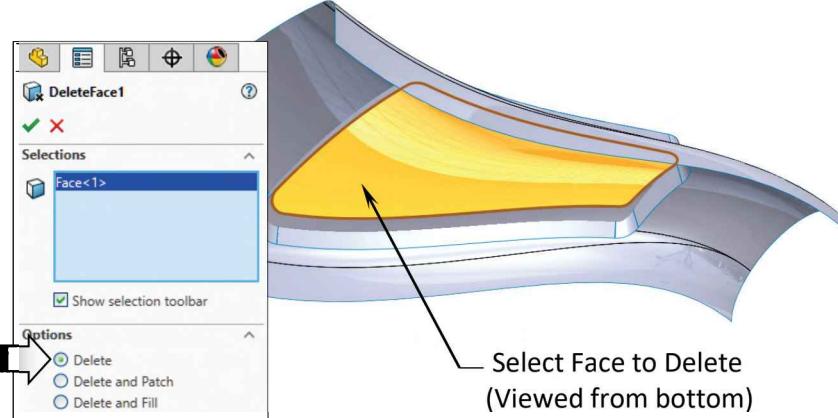
The surface that was used to create the offset surface is no longer needed.

Click **Delete Face**.

For Options, select **Delete**.

For Faces to Delete, select the face in the middle of the model as noted.

Click **OK**.



Select Face to Delete
(Viewed from bottom)

6. Creating the 1st surface trim:

Click **Trim Surface**.

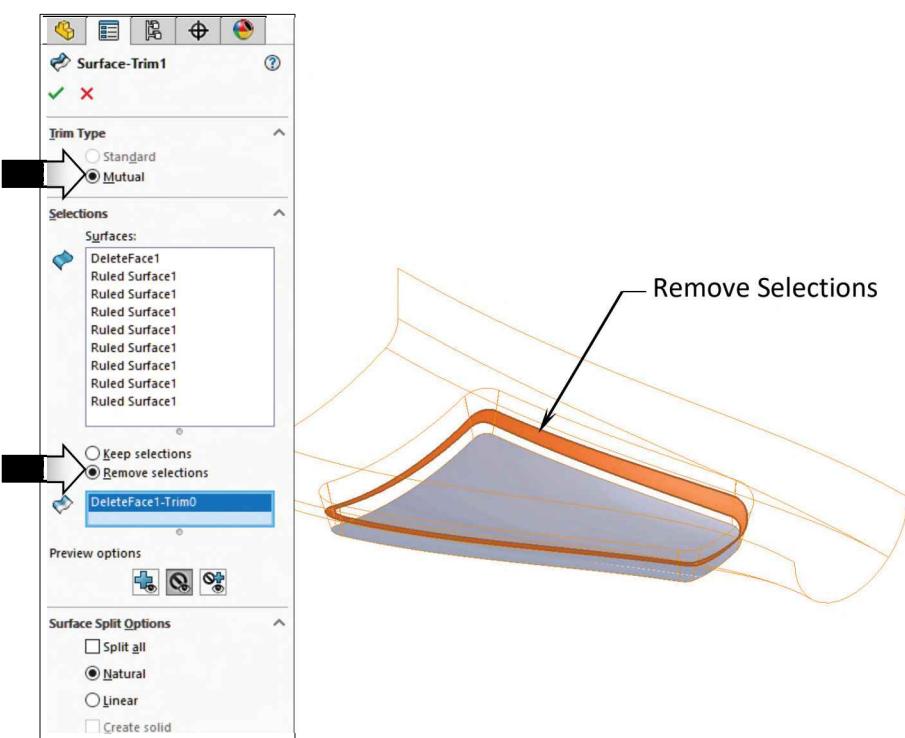
Select **Mutual** for Trim Type.



For Selections, select the **main surface body** and the **ruled surface**.

Click Remove Selections button and select the surfaces as indicated.

Click **OK**.



Remove Selections

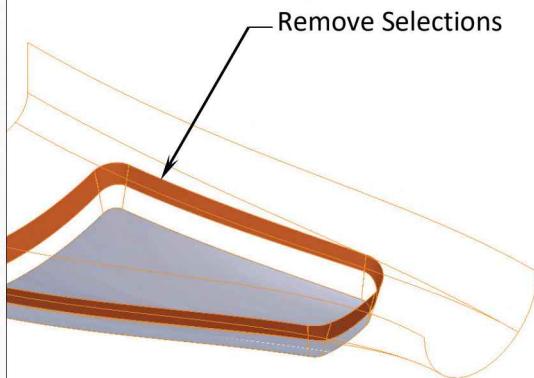
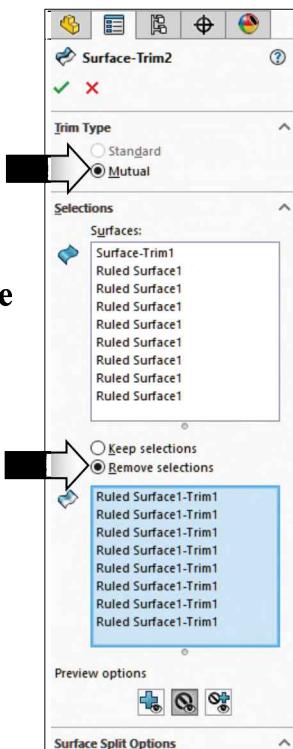
7. Creating the 2nd surface trim:

Click Trim Surface.

Click Mutual for Trim Type

Select the **main surface body** and the **Ruled Surface** for Selections.

For Remove-Selections, select the inner portion of the **ruled surface** that is protruding outward.



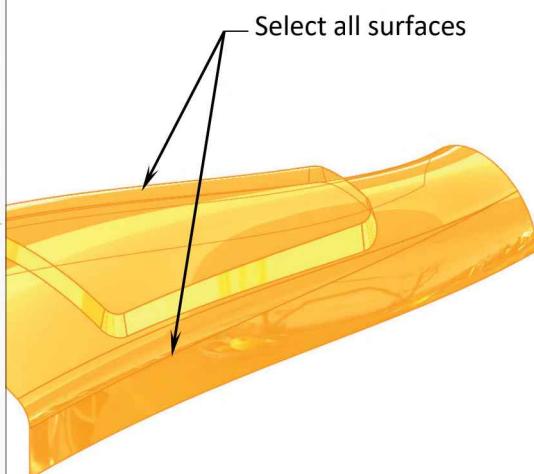
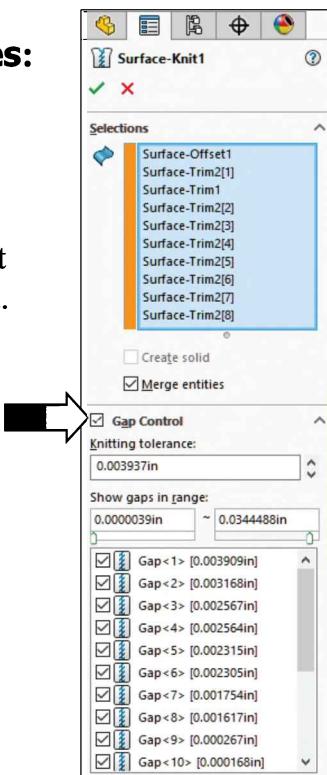
Click OK.

8. Knitting the surfaces:

Click Knit Surface.

For Selections, select **all surfaces** as noted.

Click the **Gap- Control** checkbox and enable all gap checkboxes to allow the software to heal them.



Click OK.

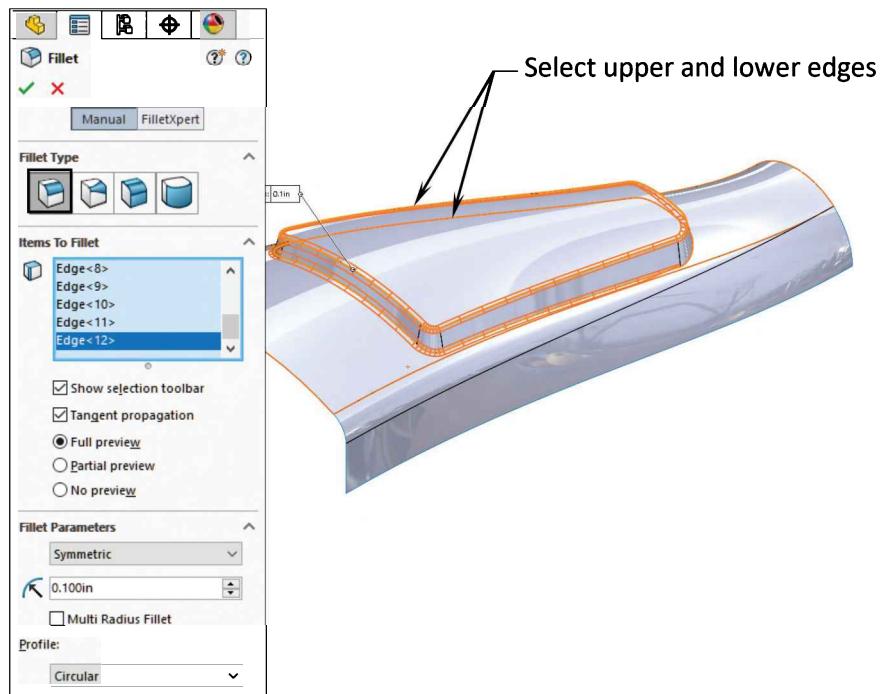
9. Adding fillets:

Click Fillet.

Use the default Constant Size Radius option.

For Items to Fillet, select all edges of the ruled surface, both upper and lower edges.

For radius size, enter **.100in**.



Click OK.

10. Saving your work:

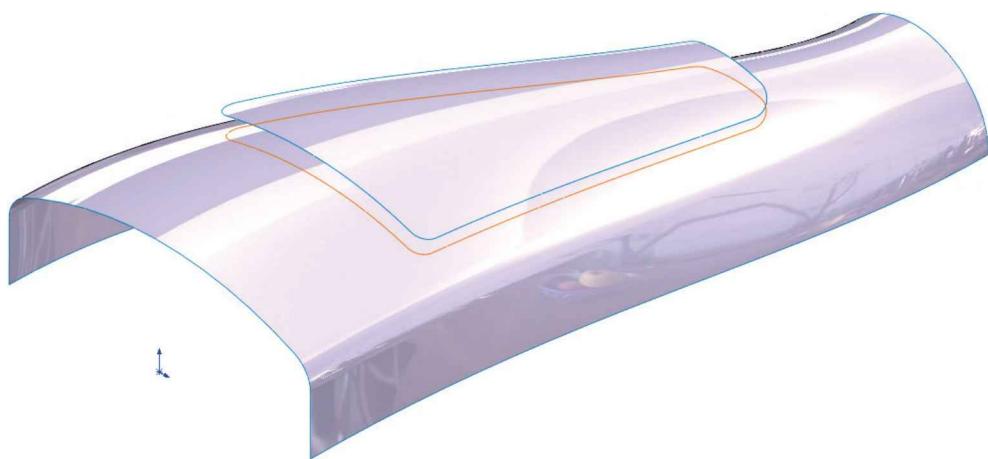
Select File, Save As.

Enter **Using_Split_Completed** for the file name.

Click Save.



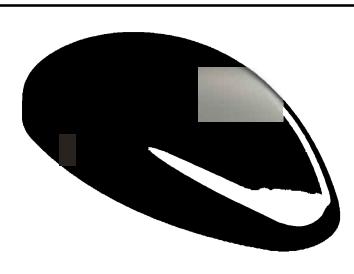
Close all documents.



CHAPTER 10

Advanced Surfaces

Advanced Surfaces Computer Mouse



Surfaces are a type of geometry that can be used to create solid features. Surface tools are available on the Surfaces toolbar.

You can use surfaces in the following ways:

- * Select surface edges and vertices to use as a sweep guide curve and path.
- * Create a solid or cut feature by thickening a surface.
- * Extrude a solid or cut feature with the end condition Up to Surface or Offset from Surface.
- * Create a solid feature by thickening surfaces that have been knit into a closed volume.
- * Replace a face with a surface.

Surface body is a general term that describes connected zero-thickness geometries such as single surfaces, knit surfaces, trimmed, and filleted surfaces, and so on. You can have multiple surface bodies in a single part.

Keep in mind that when modeling with surfaces, every feature should be easily edited. One way to achieve that is to try and create one surface at a time, and towards the end, knit all surfaces to a single surface, then thicken to a solid model.

This lesson will teach us one of the more convenient methods where multibody surfaces are used to create the computer mouse body. Each surface will be created individually and then trimmed and knitted into a single surface body.

After the surface knit, a wall thickness can be added to the surface body to convert it into a solid body so that other solid features can be added to it from that point on.

Advanced Surfaces

Computer Mouse



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI
Units: INCHES – 3 Decimals

Tools Needed:



Extruded Surface



Trim Surface



Mirror Surface



Filled Surface



Planar Surface

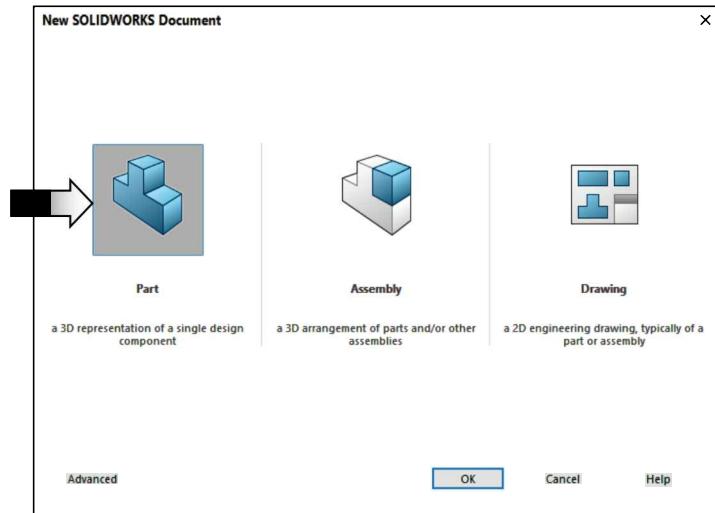


Knit Surface

1. Starting a new part document:

Click **File / New**.

Select the **Part** template and click **OK**.

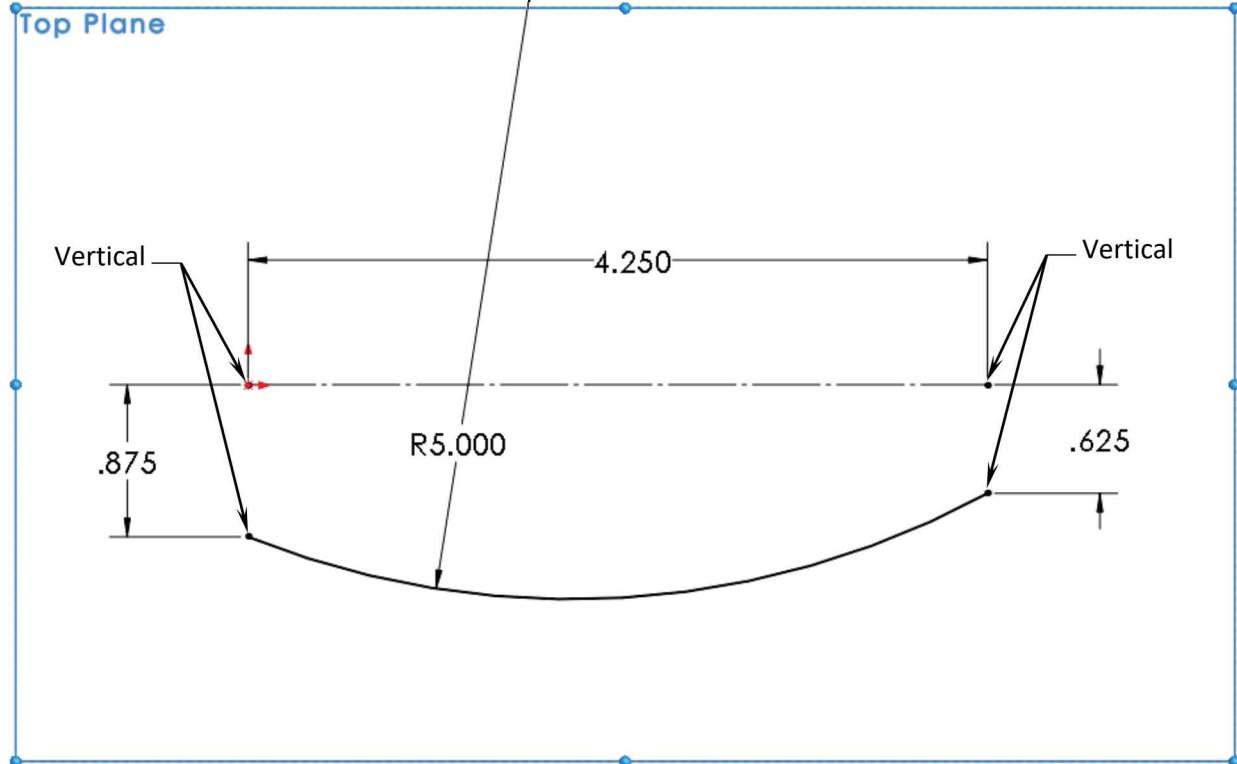


2. Sketching the 1st profile:

Select the Top plane and open a **new sketch**.

Sketch a **Centerline** and a **3-Point Arc** shown below.

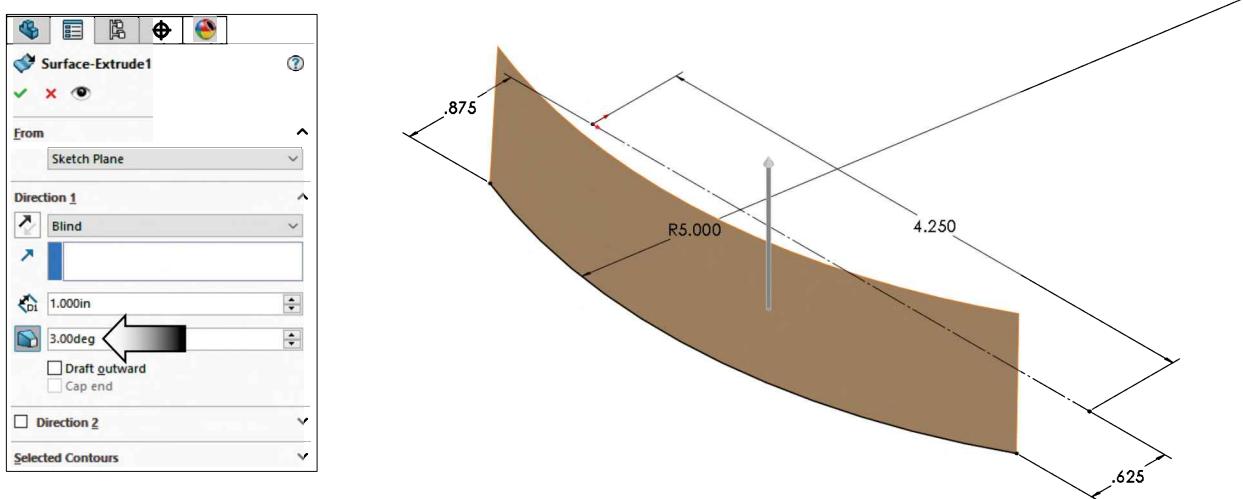
Add the dimensions and relations as shown below to fully define the sketch.



3. Extruding the 1st surface:

Switch to the **Surfaces** tab and click **Extruded Surface** .

Use the default **Blind** type and enter a Depth of **1.00in**.



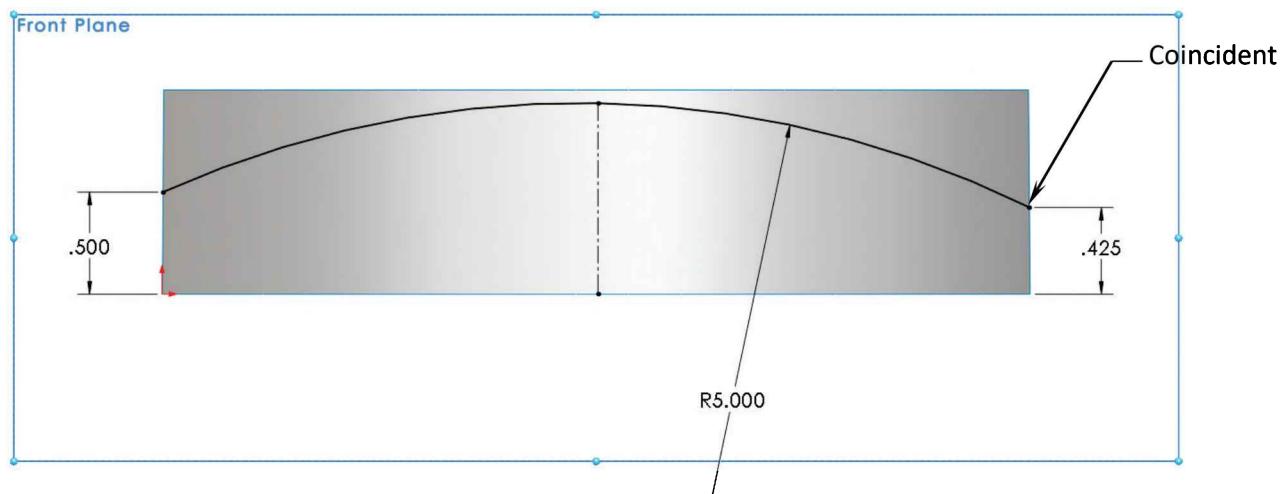
Enable the **Draft On/Off** button and enter **3.00deg** for Draft Angle.

Click **OK**.

4. Sketching the 2nd profile:

Select the Front plane and open a **new sketch**.

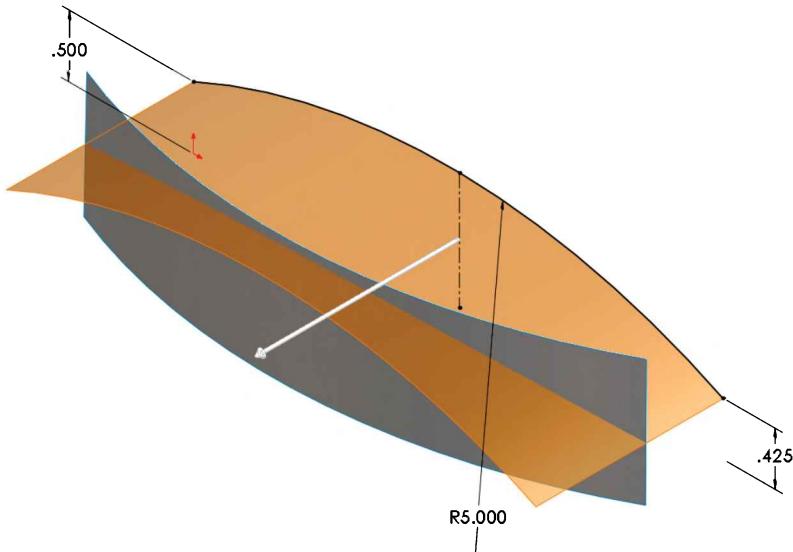
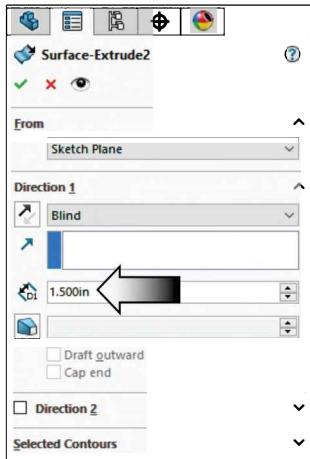
Sketch a **3-Point-Arc** and add the dimensions / relations as indicated.
(Note: The dimension .500 can be replaced with a Midpoint relation).



5. Extruding the 2nd surface:

Switch to the **Surfaces** toolbar and click **Extruded Surface** .

Use the default **Blind** type and enter a Depth of **1.50in**.

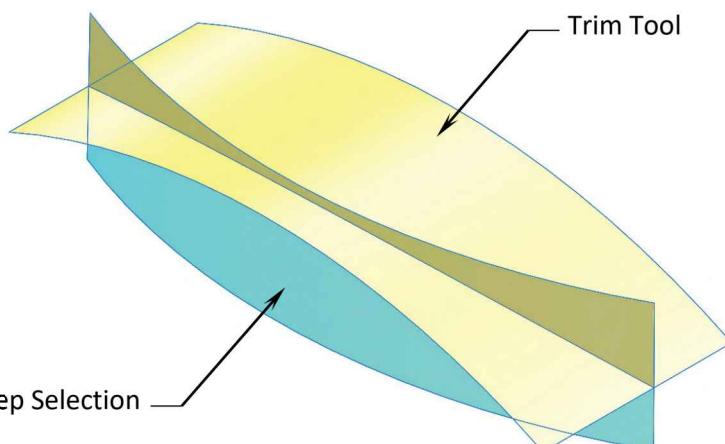
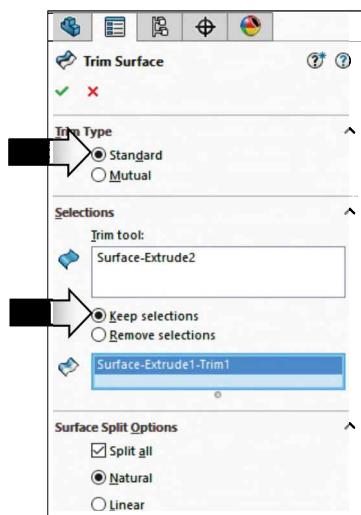


Click **OK**.

6. Trimming the surfaces:

Select the **Trim Surface** command .

Use the default **Standard Trim** option. For Trim Tool, click **Surface-Extrude2**.



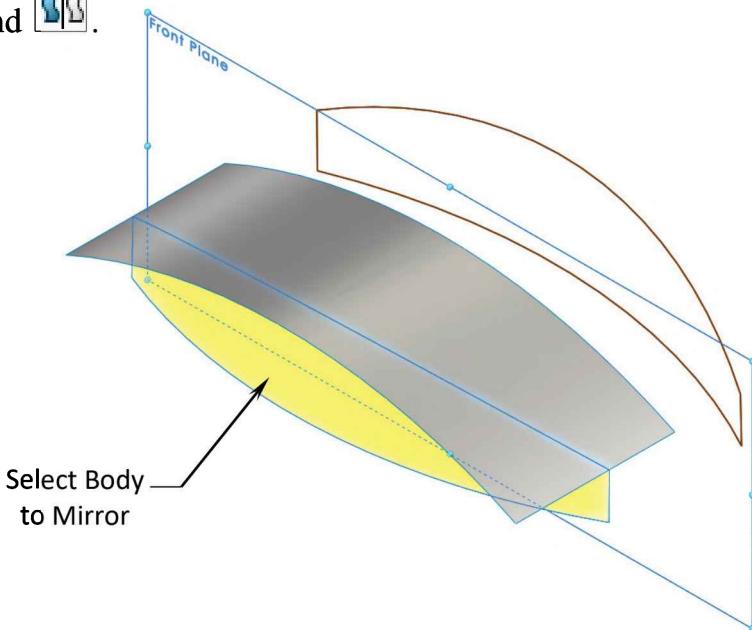
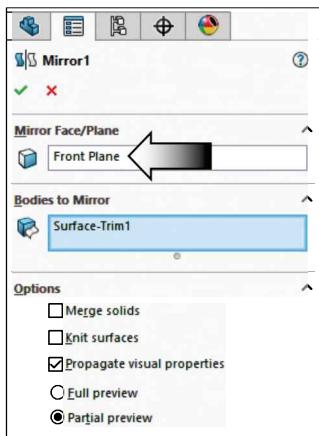
For **Keep Selections**, select the lower portion of the **Surface Extrude1** as indicated.

Click **OK**.

7. Mirroring the surfaces:

Switch to the **Features** tab.

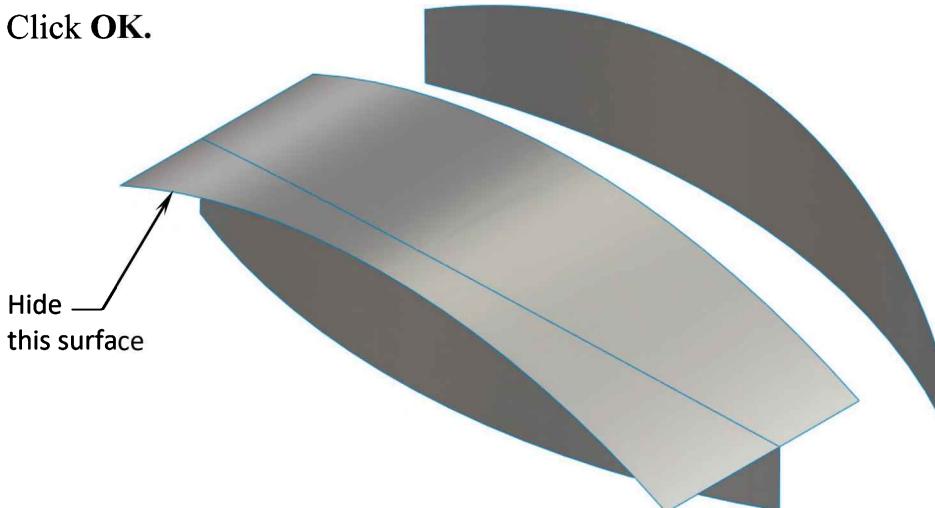
Click the **Mirror** command .



For **Mirror Face/Plane**, select the **Front** plane from the FeatureManager tree.

Expand the **Bodies to Mirror** section and select the lower portion of the **Surface-Trim1** as noted.

Click **OK**.



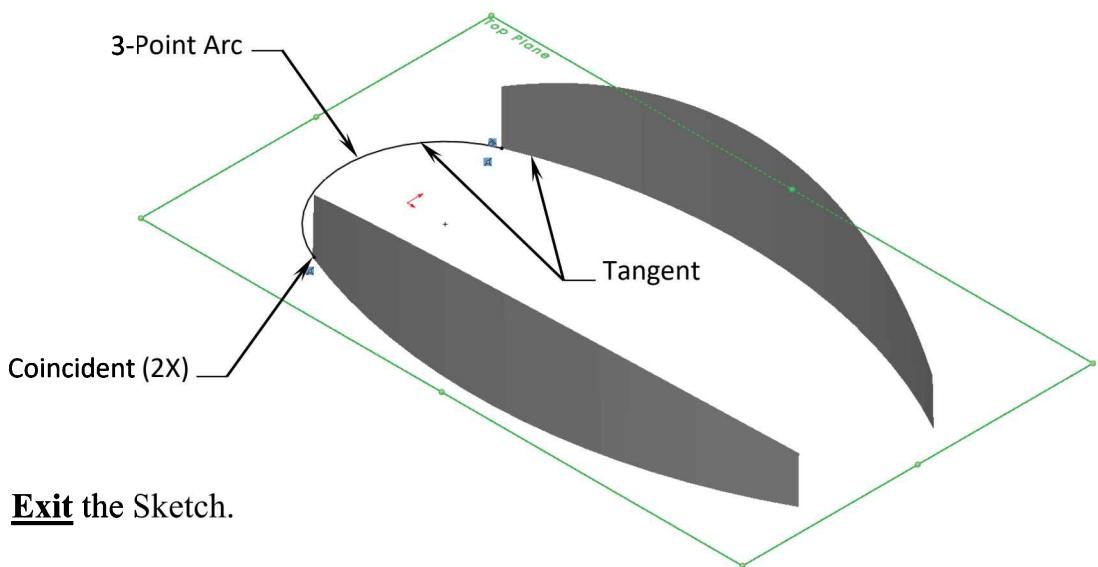
Inspect your model against the image above.

8. Creating the 1st lower sketches:

Hide the Surface-Extruded2.

Open a **new Sketch** on the Top plane.

Sketch a **3-Point Arc** as shown below and add the **Tangent** relations as noted.



9. Creating the 2nd lower sketches:

Open a **new sketch** on
the Top plane again.

Sketch another
3-Point-Arc and
add the relations
as indicated.

Exit the Sketch.

Coincident
(2X)

Tangent

10. Creating the 1st upper sketches:

Open a new 3D Sketch.

Select the
3-Point Arc command.

Click Point 1 and **Point 2**
as indicated.

(Do not click Point 3 just yet.)

Press **Control + 5** to change
to the **Top View** orientation.

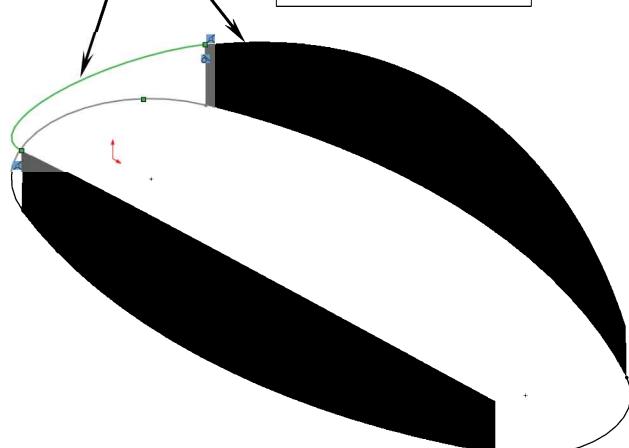
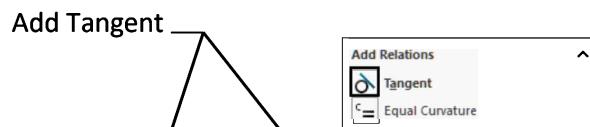
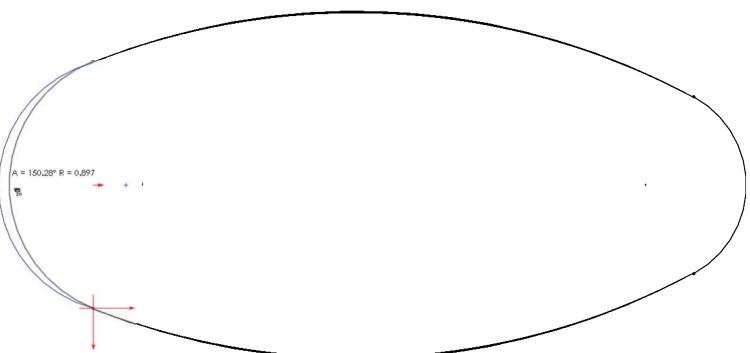
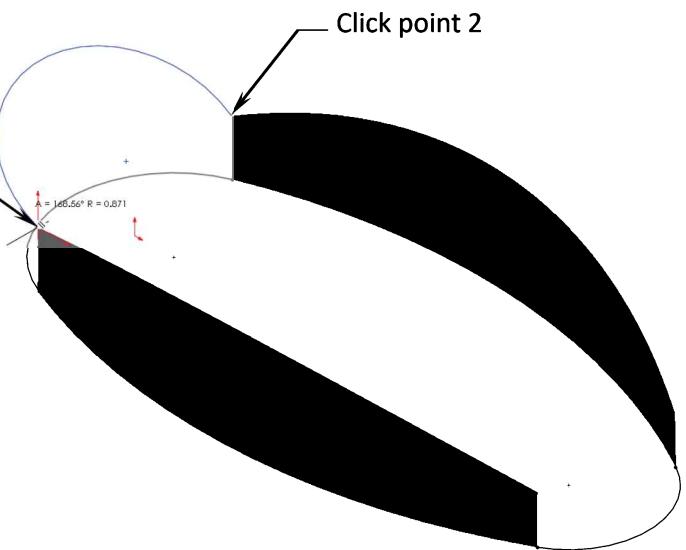
Move the mouse cursor
outward approximately
as shown and click
Point 3.

Push **Escape** to deselect the
3 Point Arc command.

Add a **Tangent** relation
between the 2 entities as
indicated.

Exit the 3D Sketch.

Repeat the step 10 and add the
2nd upper arc on the right side.

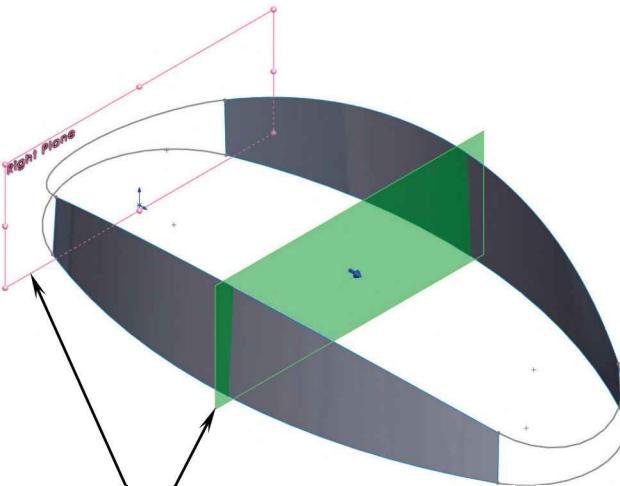
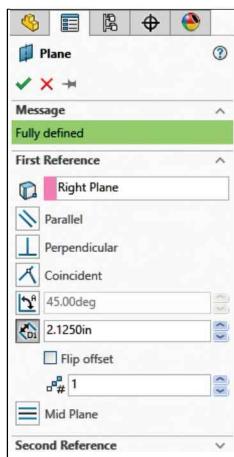


11. Creating a new plane:

Select the **Right** plane from the Feature-Manager tree.

Hold the **Control** key and drag the **Right** plane to the right side to make a copy.

Enter **2.125in.** for Distance.



Hold Control key & drag the Right plane to make a copy

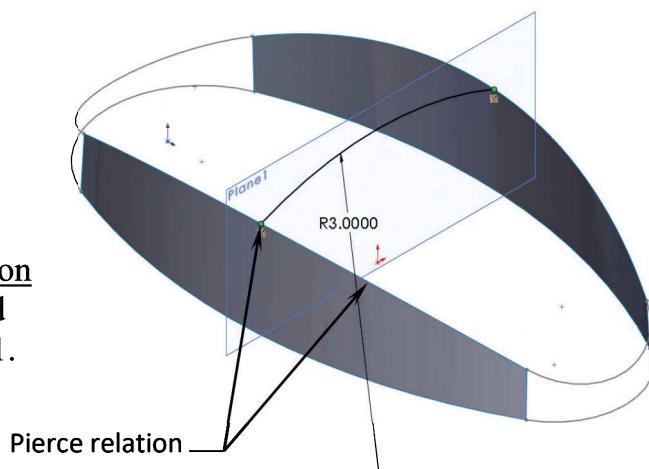
Click **OK**.

12. Making a 3-Point Arc:

Open a new sketch on the **new plane** (Plane1).

Sketch a **3-Point-Arc**.

Add a dimension and a Pierce relation between the end point of the arc and the upper edge of the Surface-Trim1.

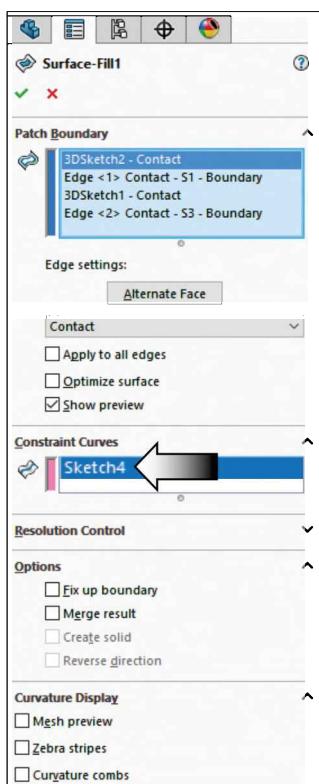


Exit the sketch.

13. Creating the 1st Filled Surface:

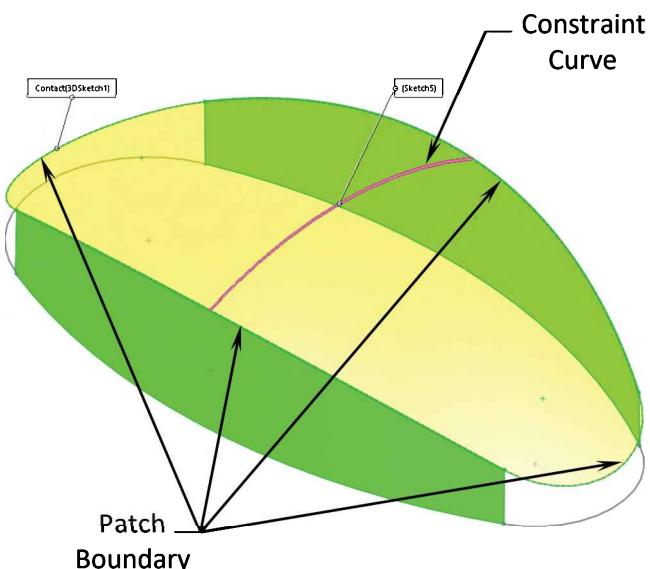
Switch back to the **Surfaces** tab.

Select the **Filled Surface** command .



Filled Surface

The Filled Surface feature constructs a surface patch with any number of sides, within a boundary defined by existing model edges, sketches, or curves, including composite curves.



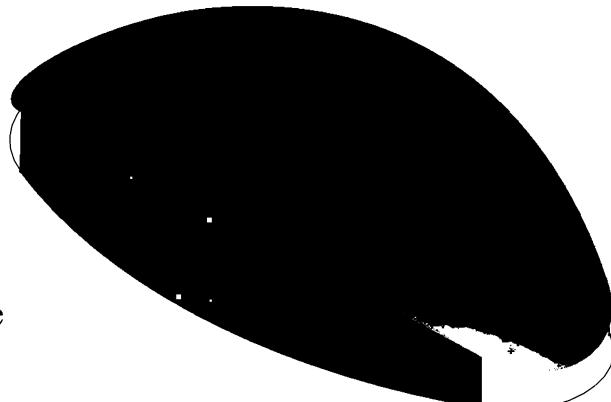
For Patch Boundary, select the **4 edges** on top as noted.

Expand the **Constraint Curves** section and select **Sketch4** in the middle of the model as indicated, to help control the curvature of the patch.

Leave all other parameters at their default settings.

Click **OK**.

Inspect the result of your Filled Surface against the image shown here.



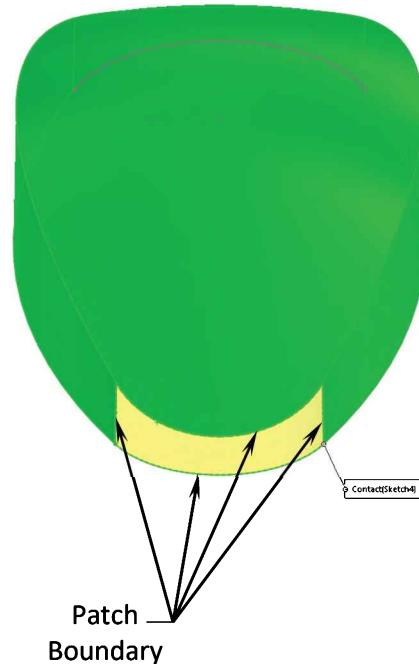
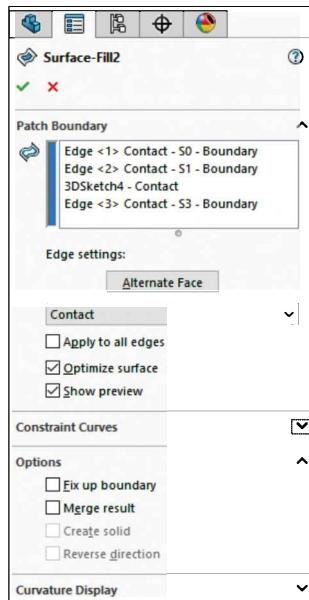
14. Creating the 2nd Filled Surface:

Rotate and bring the right end of the model to a similar position shown below.

Click the **Filled-Surface** command again.

For Patch Boundary, select the **4 edges** on the right end as indicated.

Click **OK**.



15. Creating the 3rd Filled Surface:

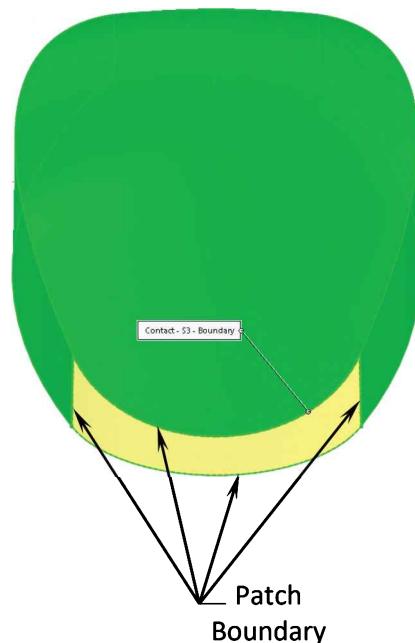
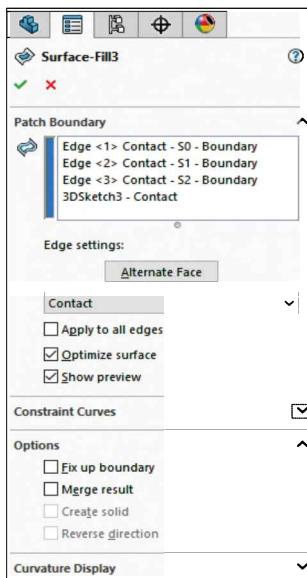
Select the **Filled Surface** command once again.

Rotate and bring the left end of the model to look similar to the image shown on the right.

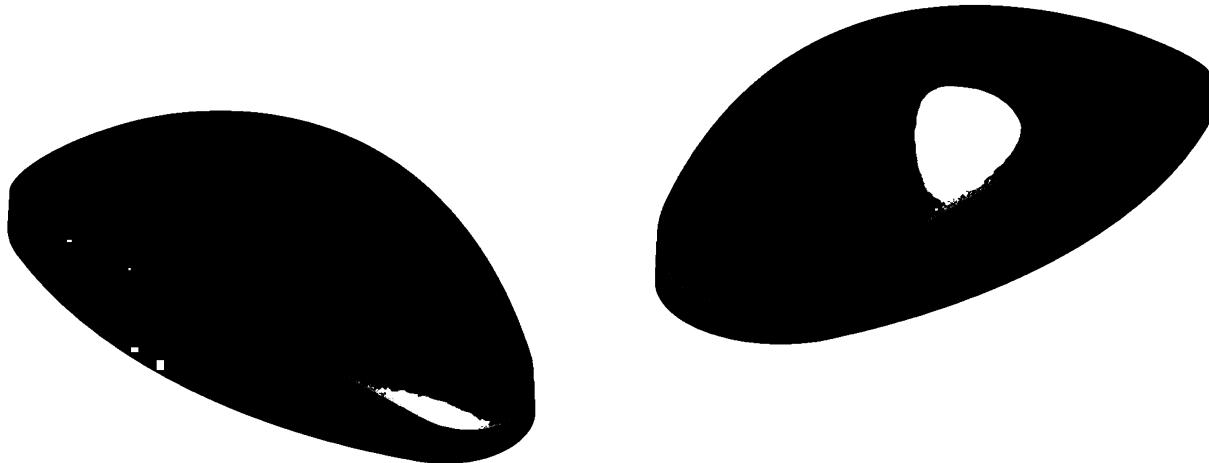
For Patch Boundary, select the **4 edges** as noted in the image.

Keep all other parameters at their default settings.

Click **OK**.



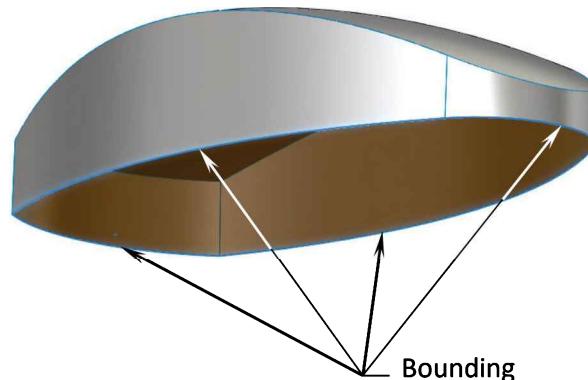
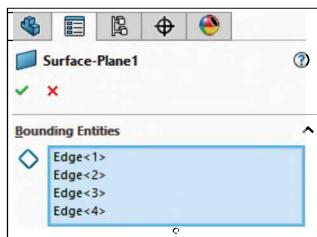
Inspect your model against the ones shown below.



16. Creating a Planar Surface:

Rotate the surface model so that the bottom opening is visible.

Select the **Planar Surface** command .



For Bounding Entities, select the **4 edges** on the bottom of the surface model.

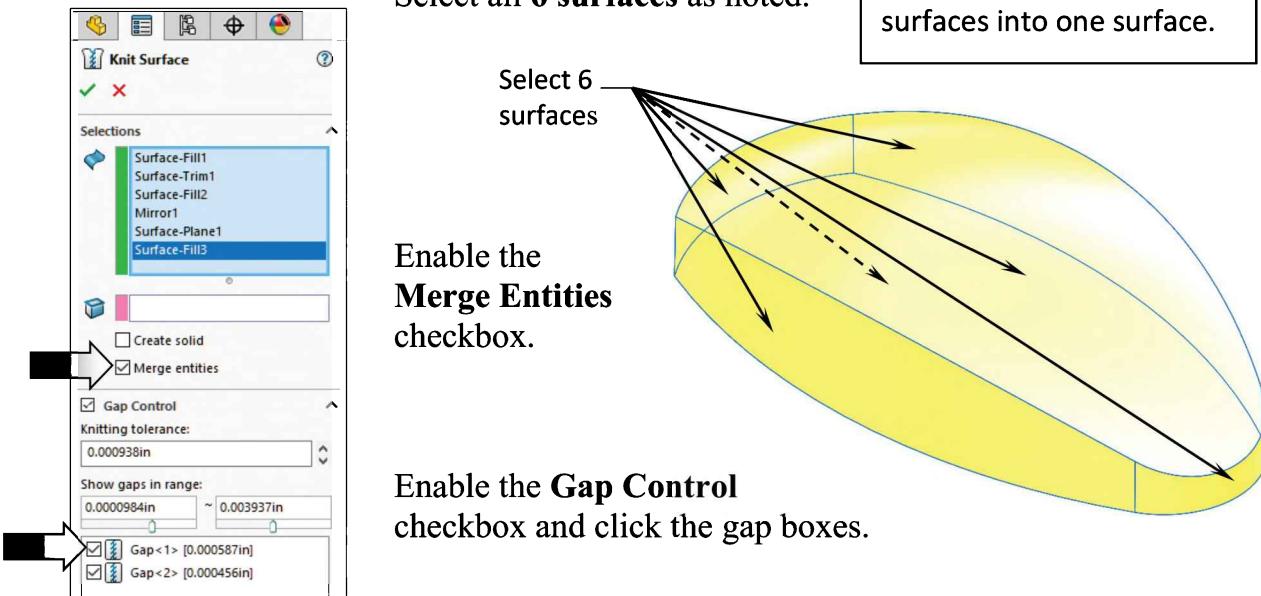
The preview of a planar surface showing the bottom opening is being closed-off.

Click **OK**.

17. Creating a Knit Surface:

Click the Knit Surface command .

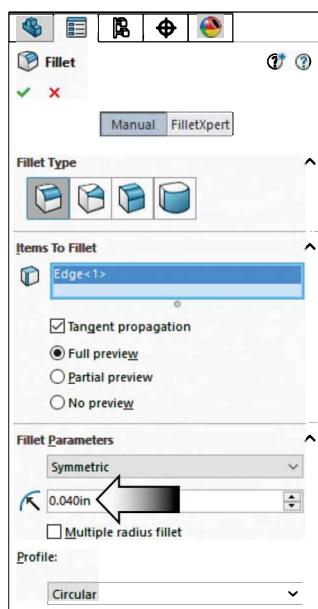
Select all **6 surfaces** as noted.



Click **OK**.

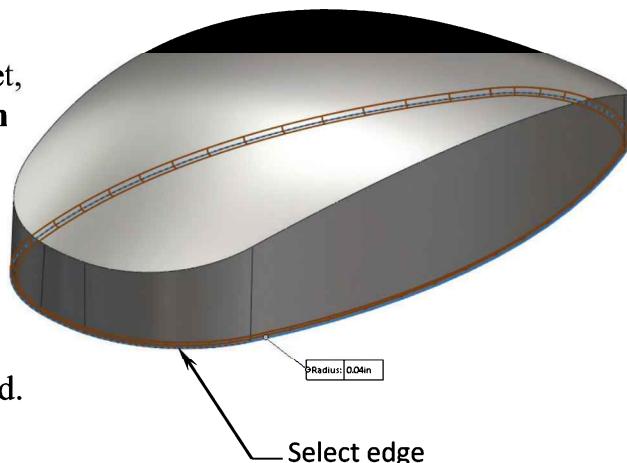
18. Adding the .040" fillet:

Select the Fillet command  again.



For radius size, enter **.040in**.

For Items to Fillet, select the **bottom edge** of the model.

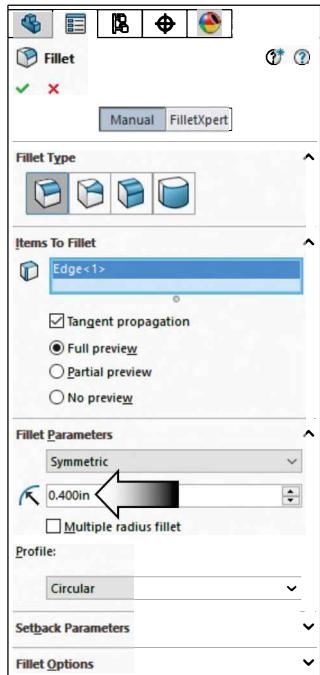


The Tangent-Propagation should be enabled.

Click **OK**.

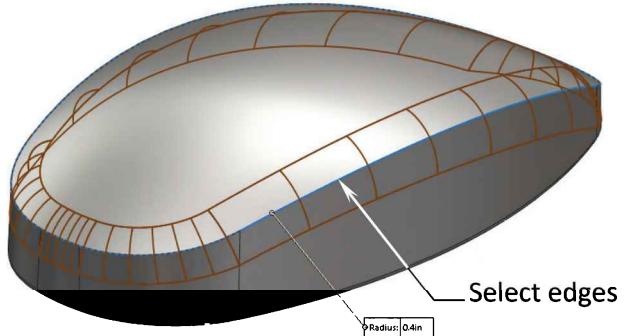
19. Creating the .400" fillet:

Select the **Fillet** command  once again.



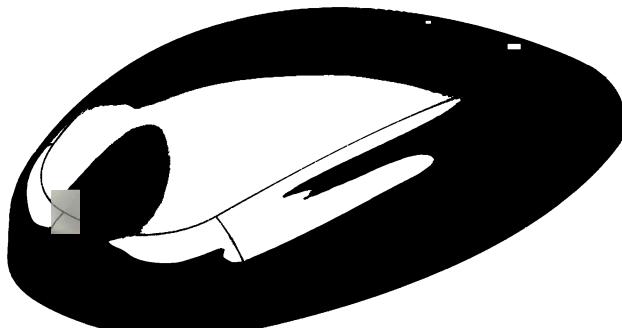
For Items to Fillet, select the **top edge** of the surface model.

For radius size,
enter **.400in**



Enable the
Tangent-
Propagation
checkbox to allow the
fillet to propagate all around the bottom of the surface
model.

Click **OK**.



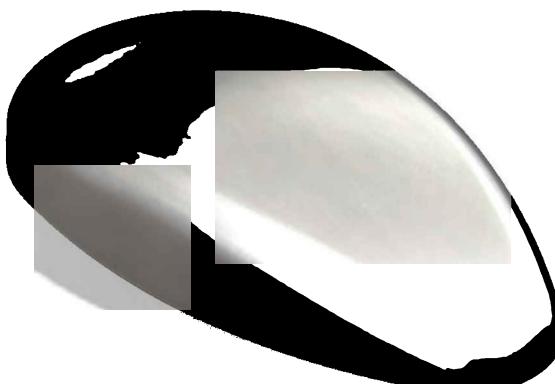
Shaded with Edges (Front Isometric)

20. Saving your work:

Click **File / Save As**.

For the file name, enter:
Computer Mouse.sldprt

Click **Save**.



Shaded with Ambient Occlusion (Back Isometric)

Using Filled Surfaces

Using Filled Surfaces

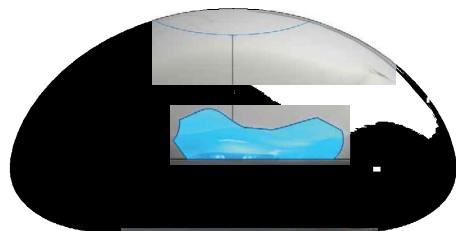
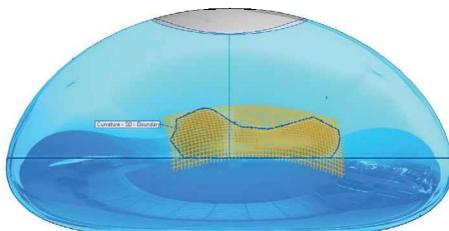
Curvature Controls



Use Filled Surface and Planar Surface commands to fill or patch a surface boundary with any number of sides.

The boundary can be a set of existing model edges, sketches, or curves, including composite curves. The boundary should be closed for the patch to work properly.

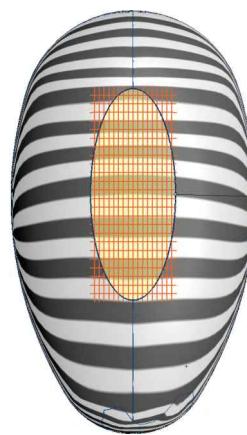
There are several options to help you control the curvatures when patching a surface boundary such as Contact, Tangent, and Curvature. These options are explained later in the lesson.



Other than the Curvature Control options the Apply-to-All-Edges check box enables you to apply the same curvature control to all edges. If you select the function after applying both **Contact** and **Tangent** to different edges, it applies the current selection to all edges.

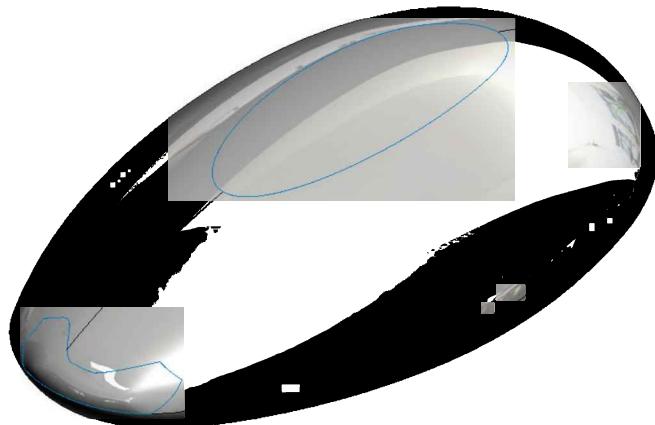
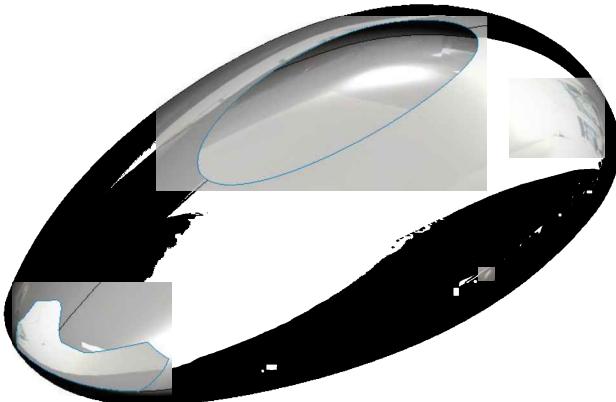
If your surface model has two or four-sided surfaces, try using the **Optimize surface** option. The Optimize surface option applies a simplified surface patch that is similar to a lofted surface. Potential advantages of the optimized surface patch include faster build times and increased stability when used in conjunction with other features in the model.

This lesson will teach us the use of the Filled Surface and Planar Surface commands.



Using Filled Surfaces

Patch with Curvature Controls



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI

Units: INCHES – 3 Decimals

Tools Needed:



Planar Surface



Filled Surface

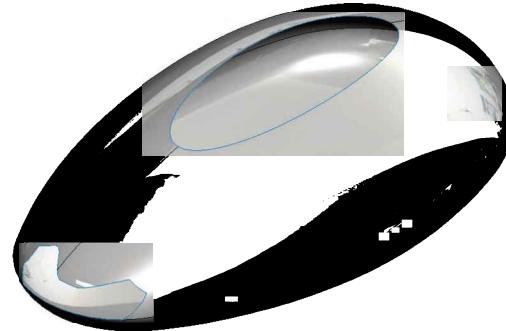


Knit Surface

1. Opening a part document:

Click **File / Open**.

Browse to the Training Files folder,
open a part document named:
Filled Surfaces.sldprt



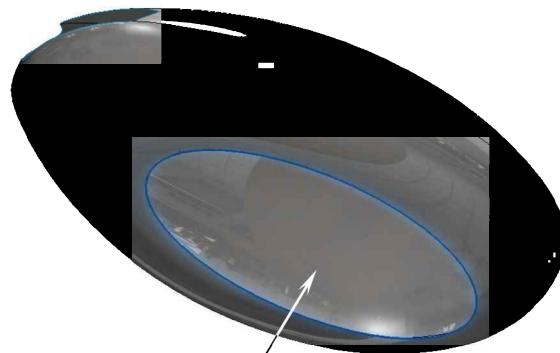
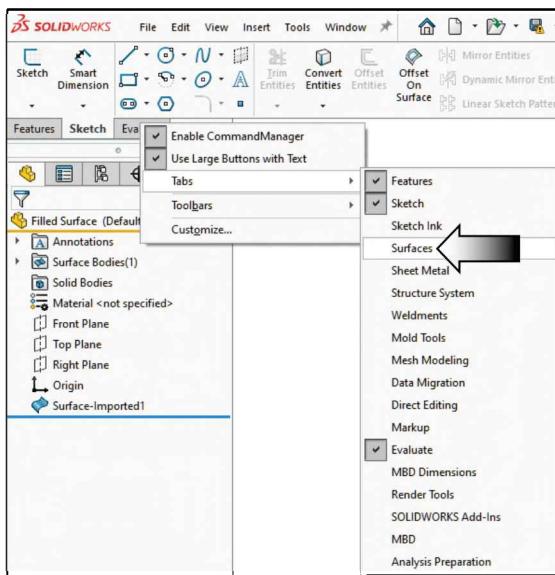
This part document was previously saved as a different file format. There is no feature history available on the FeatureManager tree.

There are three openings in the part that we need to fill using different options available in the Filled-Surface command.

If the Feature Recognition dialog pops up, click **NO** to close it.

2. Enabling the Surfaces toolbar:

Right-click one of the tool tabs and select **Tabs, Surfaces**.



Rotate the model and locate the elliptical hole at the bottom of the model.

The opening should form a flat surface. We can use the Planar-Surface command to patch it up.

3. Creating a Planar surface:

Switch to the **Surfaces** tool tab.

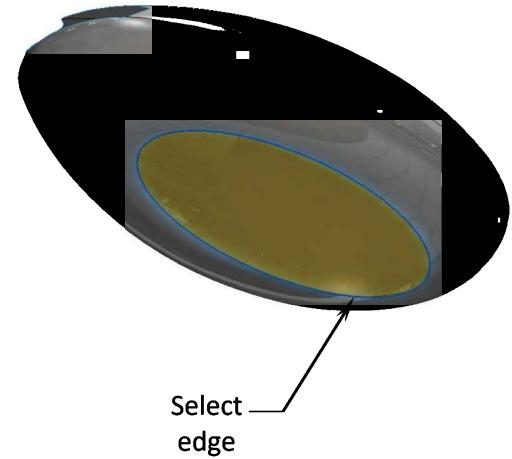
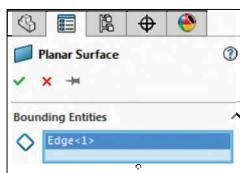
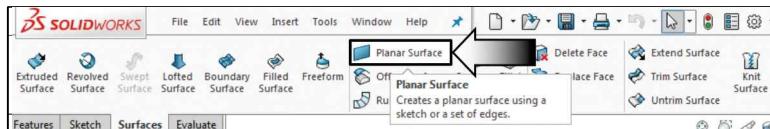
Select the **Planar-Surface** command.

Select the edge of of the elliptical hole.

The preview of a planar surface appears.

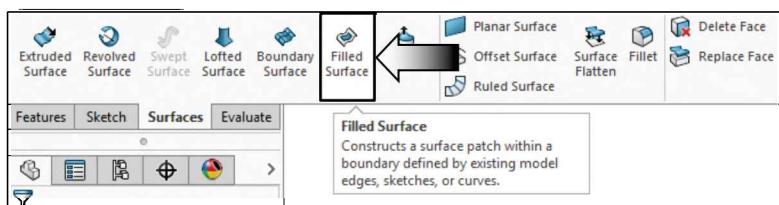
Click **OK**.

A planar surface can be created from a sketch, a set of closed edges, or a pair of planar entities such as curves or edges.



4. Creating a Surface Fill with Tangent Control:

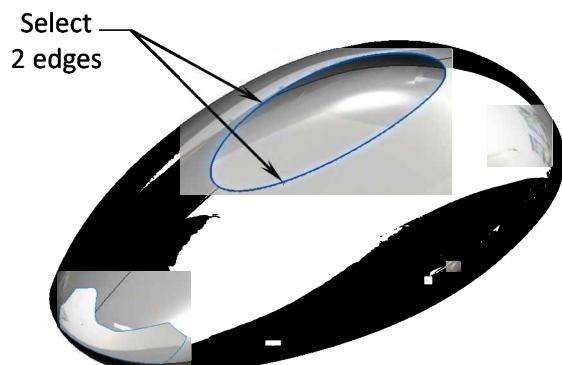
Switch back to the **Isometric view** (or press the hotkey Control + 7).



Click the **Filled Surface** command.

The Filled Surface command is used to patch a closed boundary. You can define the boundary by selecting a set of 2D or 3D sketch entities, model edges or composite curves.

Select the two edges of the elliptical hole on the top of the model, as noted.

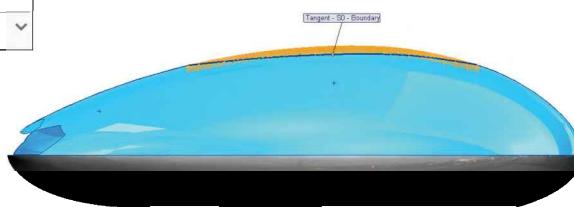
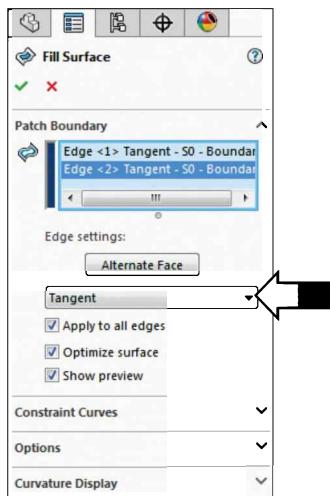


A preview mesh is displayed to help you visualize the curvature of the new surface.

Change the Curvature Control* to **Tangent** (arrow).

Enable the check-boxes as shown in the dialog box.

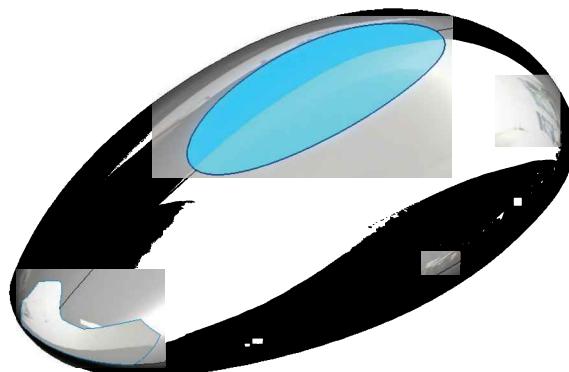
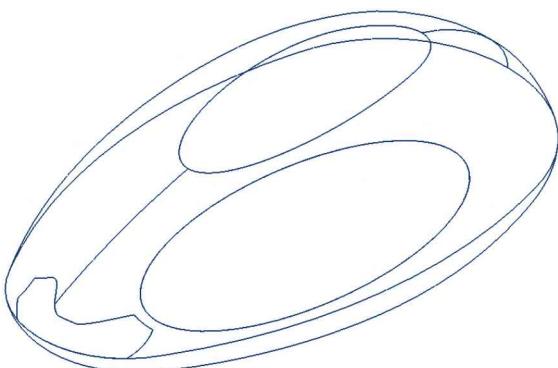
Click **OK**.



Curvature Controls Explained:

The **Curvature Control** defines the type of control you want to exert on the patch you create. The types of **Curvature Control** include:

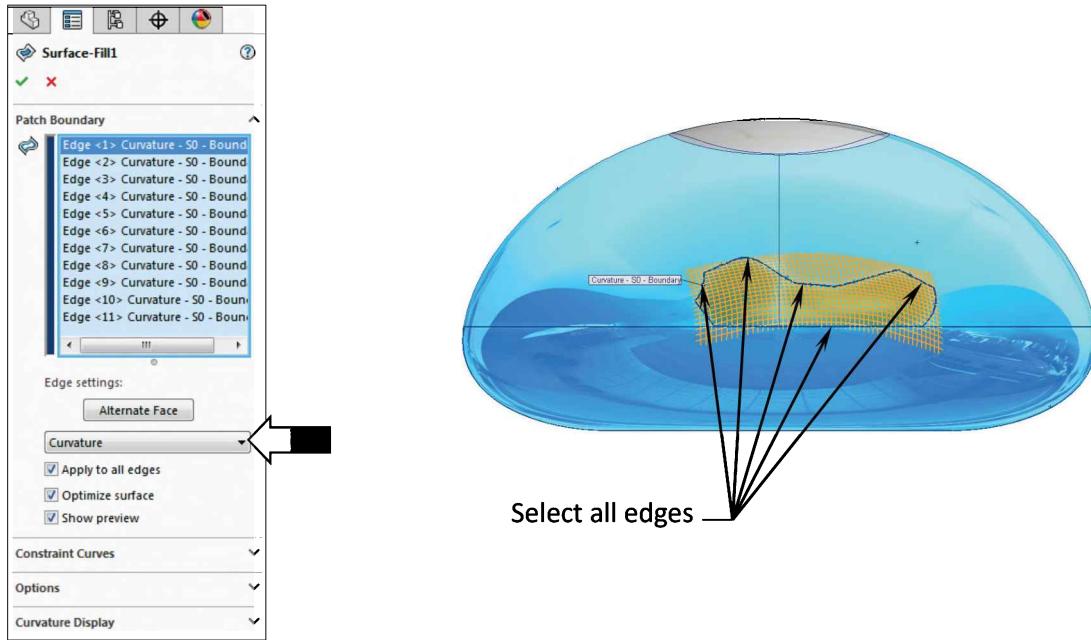
- * **Contact:** Creates a surface within the selected boundary.
- * **Tangent:** Creates a surface within the selected boundary, but maintains the tangency of the patch edges.
- * **Curvature:** Creates a surface that matches the curvature of the selected surface across the boundary edge with the adjacent surface.



5. Creating a Surface Fill with Curvature Control:

Click the **Filled Surface** command once again.

Change to the **Front** view orientation (or press Control + 1).



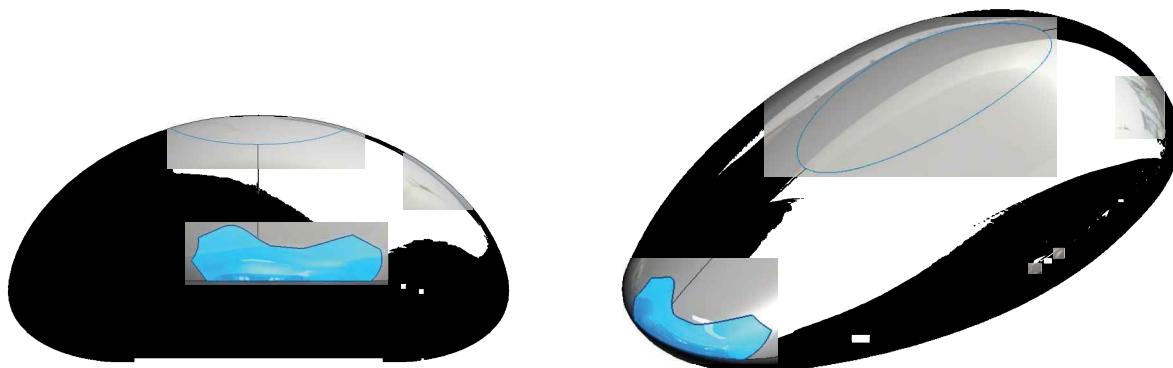
Select all edges of the opening in the front as noted.

The preview mesh appears indicating a closed boundary is found. (Enable the Preview Mesh checkbox, under Curvature Display, if the preview is not visible.)

Under the Curvature Control change the Contact option to **Curvature** (arrow).

Enable the other checkboxes as shown in the dialog box.

Click **OK**.

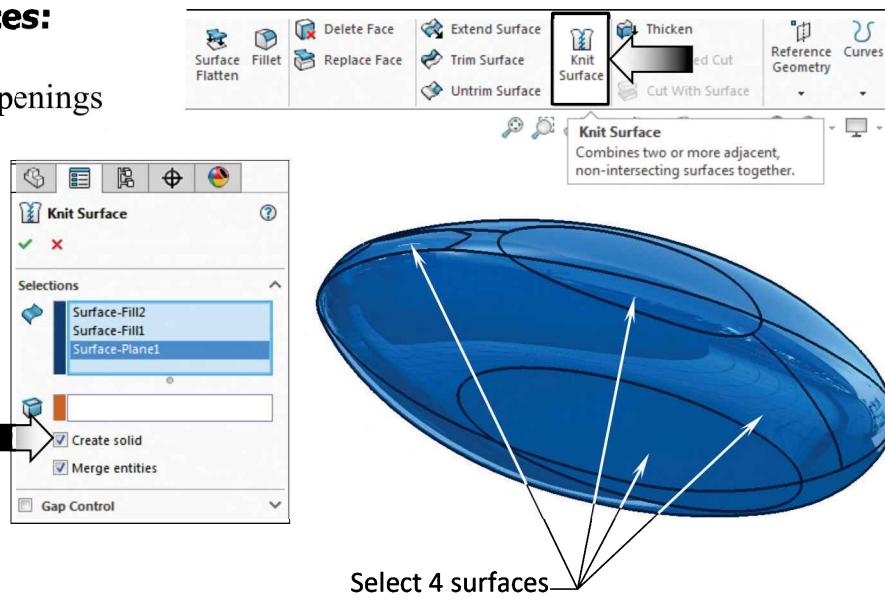


6. Knitting all surfaces:

At this point, all openings have been filled.

We can now combine all surfaces into one by using the Surface-Knit option.

Click the **Knit Surface** on the Surfaces tab.



Select all 4 surfaces in the graphics area as indicated.

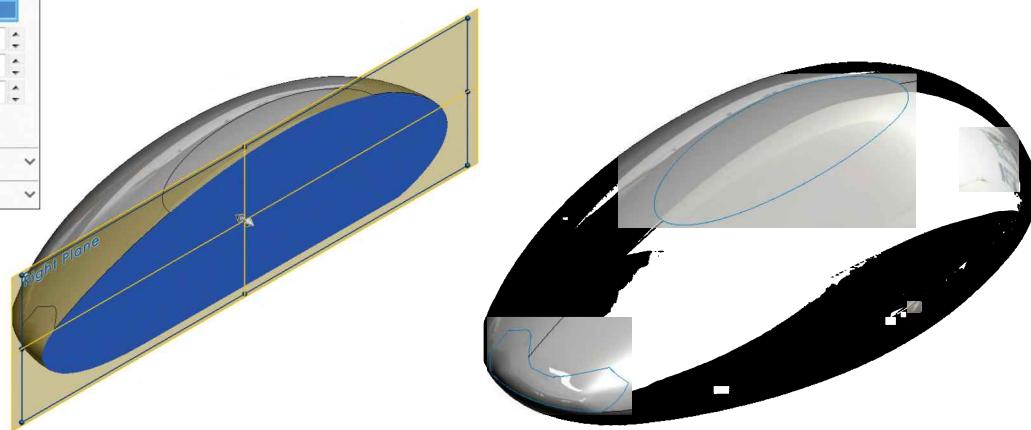
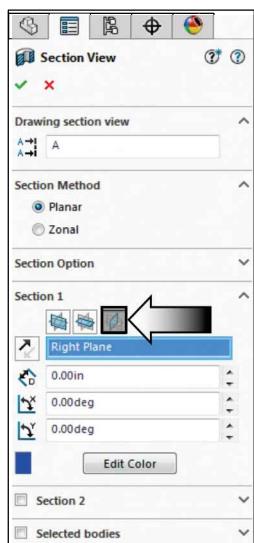
Click the **Try To Form Solid** checkbox (arrow) to convert the part to a solid model.

Click **OK**.

To verify the interior of the part create a **Section View** using the **Right** plane as the cutting plane. Click **Cancel** when done.

7. Saving your work:

Save your work as **Using Filled Surface**.

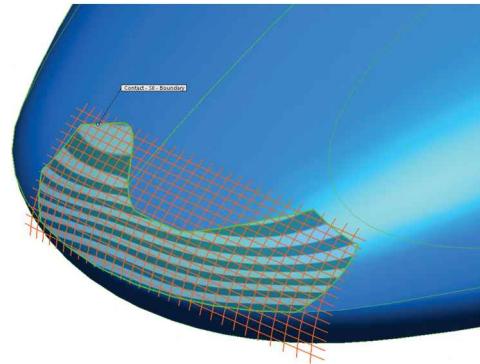
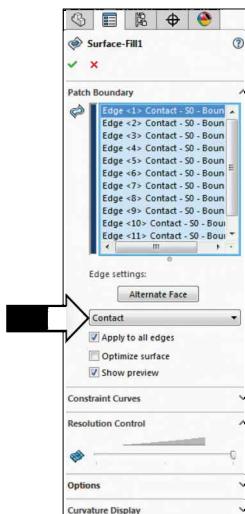


Patch Types:

A closer look at the Edge Settings: Use the Edge Settings options to define the type of control you want to apply on the patch that you create.

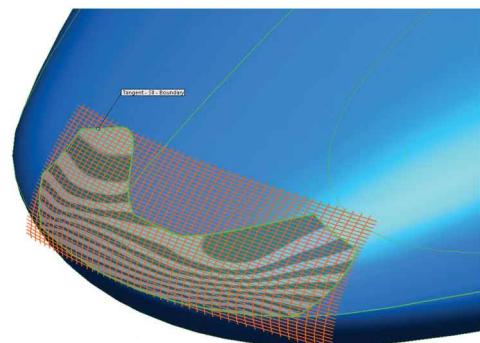
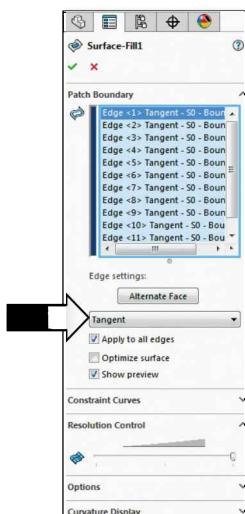
Contact Patch:

Creates a surface within the selected boundary.



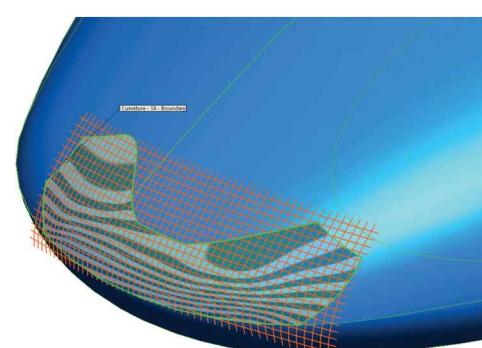
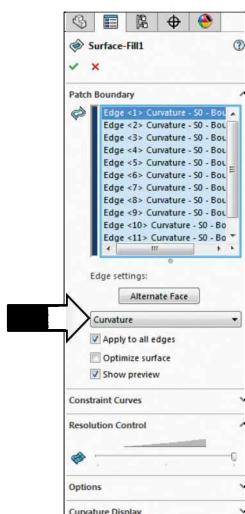
Tangent Patch:

Creates a surface within the selected boundary but maintains the tangency of the patch edges.



Curvature Patch:

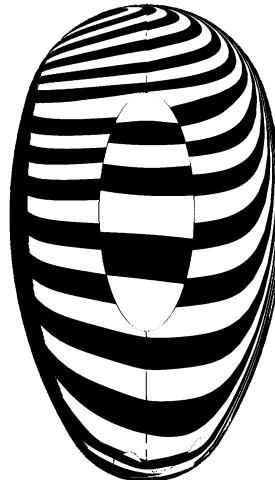
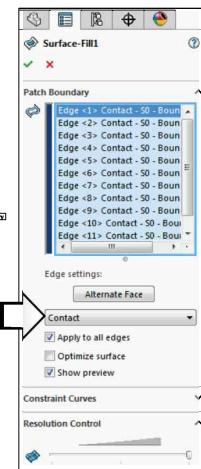
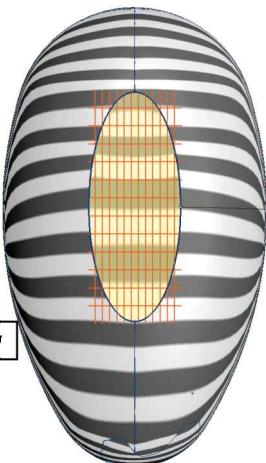
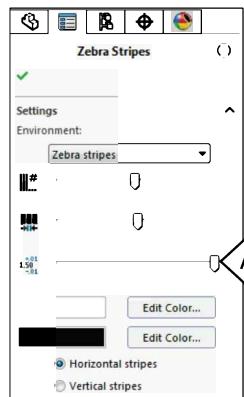
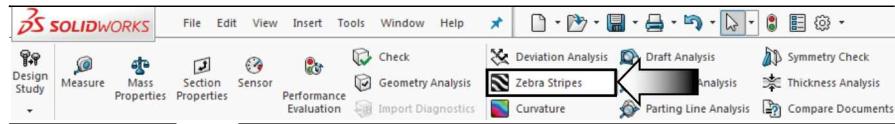
Creates a surface that matches the curvature of the selected surface across the boundary edge with the adjacent surface.



Optional:

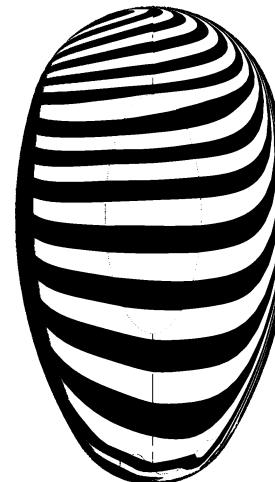
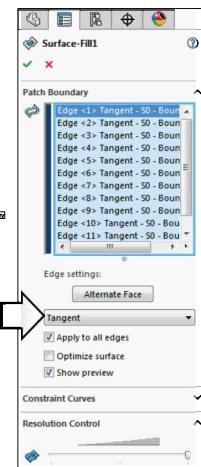
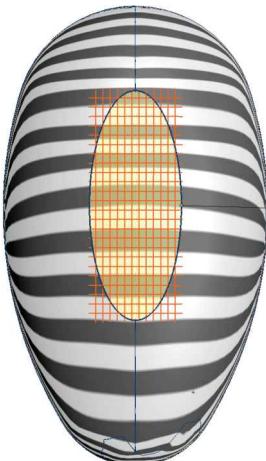
A closer look at the Zebra Stripes:

Use zebra stripes to visually determine which type of boundary to use between surfaces.

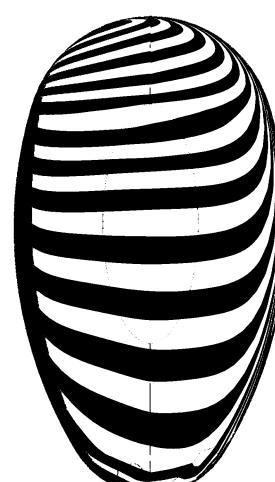
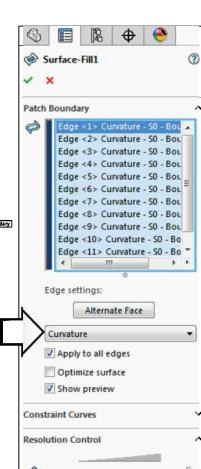


Zebra Stripes examples:

You can see small changes in a surface that may be hard to see with a standard display. Zebra stripes simulate the reflection of long strips of light on a very shiny surface.



With zebra stripes, you can easily see wrinkles or defects in a surface, and you can verify that two adjacent faces are in contact, are tangent, or have continuous curvature.



Questions for Review

1. When opening a part document created from another CAD software, all of its features and sketches will appear on the FeatureManager tree.
 - a. True
 - b. False
2. Tool tabs can be added when needed by right clicking on one of the existing tool tabs and selecting them from the list.
 - a. True
 - b. False
3. A planar surface can be created from a closed sketch or a group of closed edges.
 - a. True
 - b. False
4. An open sketch or a group of open edges can also be patched using the Planar surface command.
 - a. True
 - b. False
5. The Filled Surface command is used to patch a non-planar closed boundary.
 - a. True
 - b. False
6. The Filled Surface command will fail if the boundary is not closed.
 - a. True
 - b. False
7. The Preview Mesh can be toggled on/off during the creation of the filled surface.
 - a. True
 - b. False
8. The Knit Surface command can only Knit the surfaces; it cannot form a solid from the surfaces.
 - a. True
 - b. False

1. FALSE
2. TRUE
3. TRUE
4. FALSE
5. TRUE
6. TRUE
7. TRUE
8. FALSE

Using Ruled Surface

1. Opening a part document:

Select File, Open.

From the Training Files folder, open the part document named: **Using Ruled Surface.sldprt**.

The main body of the Detergent Container has already been created to help focus on the creation of a ruled surface.



2. Creating a Split Line feature:

Edit the **Sketch8** from the FeatureManager tree.

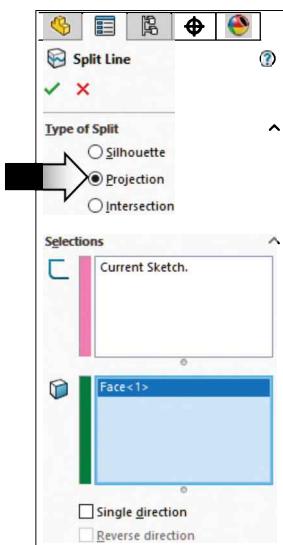
This sketch was created on the Front plane and has already been fully defined.

Switch to the **Surfaces** tab and select:
Curves, Split Line.

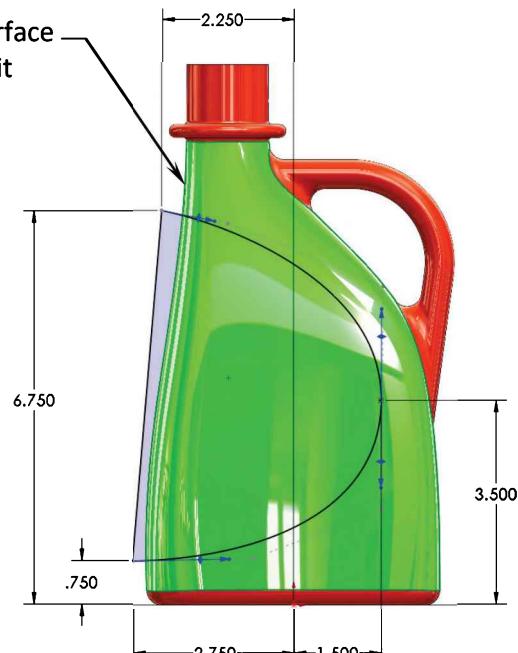
For Type of Split, select **Projection**.

For Split Sketch, select **Sketch8**.

For Faces to Split, select the **face** of the container as noted.



Click **OK**.



3. Creating an offset surface:

Click **Offset Surface**.

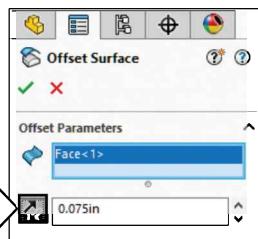
For Faces to Offset,
select the **face** as
indicated.

Select face
to offset

For Offset Distance,
enter **.075in**.

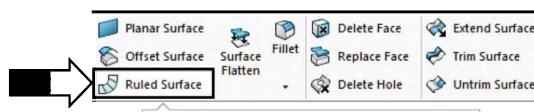
Click **Reverse** to
place the copy on
the inside.

Click **OK**.



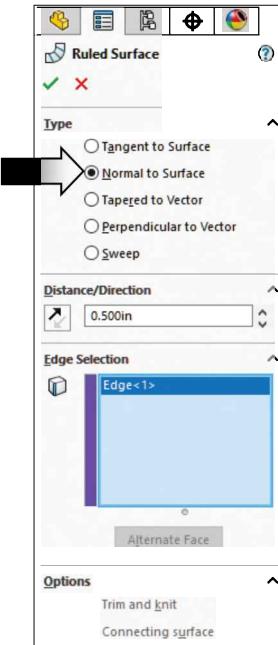
4. Adding a Ruled Surface:

Ruled surface creates surfaces that extend out in a specified direction from selected edges.



Ruled Surface
Inserts ruled surfaces in a specified direction from edges.
You can trim and knit the surfaces and remove connecting surfaces.

Hide the main body and
click **Ruled Surface**.



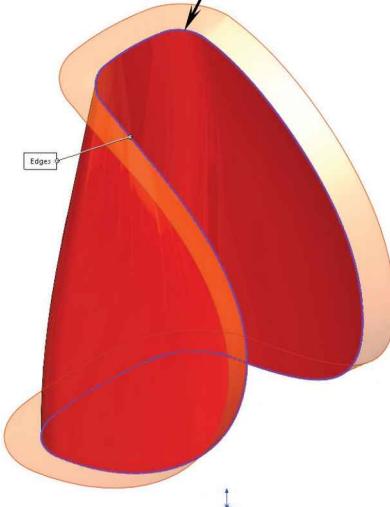
Click **Normal to Surface**.

For Distance, enter **.500in**.

For Edge Selection, select
all **edges** as noted.

Click **OK**.

Select all
edges



5. Knitting the surfaces:

The Offset Surface and the Ruled Surface must be knitted into a single surface in order to use as the trim tool.

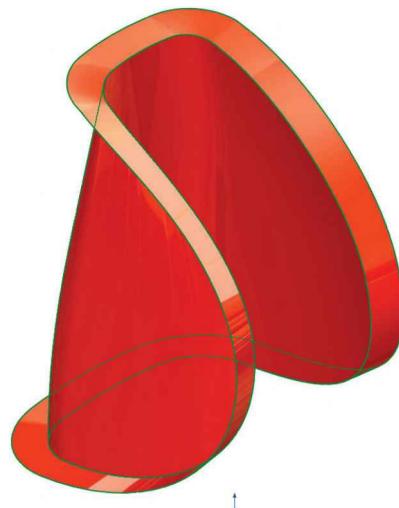
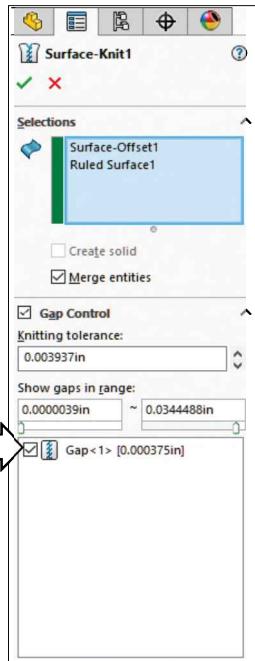
Click **Knit Surface**.

Select both surfaces, the **Offset** and the **Ruled** surfaces.

Click **Merge Entities**.

Enable the **Gap Control** checkbox and check the gap-box to allow the software to close any gaps automatically.

Click **OK**.



6. Creating a Surface Cut:

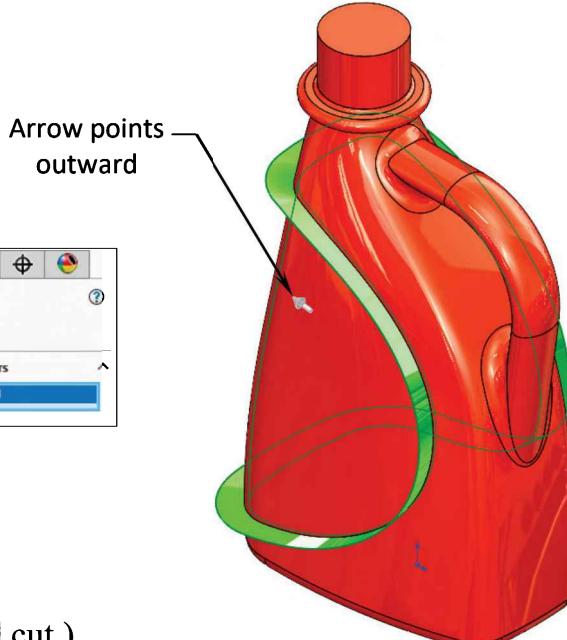
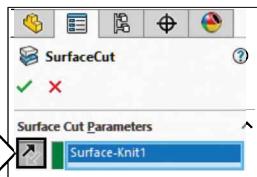
The Knit Surface can now be used as the trim tool to cut into the container.

Show the main body and click **Cut with Surface**.

For Surface Cut Parameters, select the **Knit Surface**.

Click **Reverse**.

The direction arrow should be pointing outward.



Click **OK**.

(Hide the Knit Surface to see the recessed cut.)

7. Adding the .060" fillet:

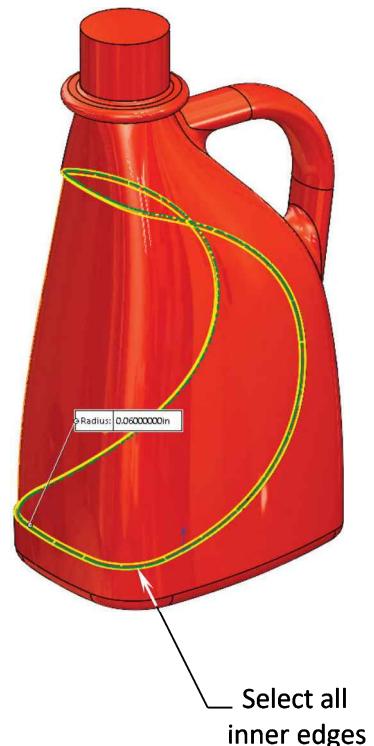
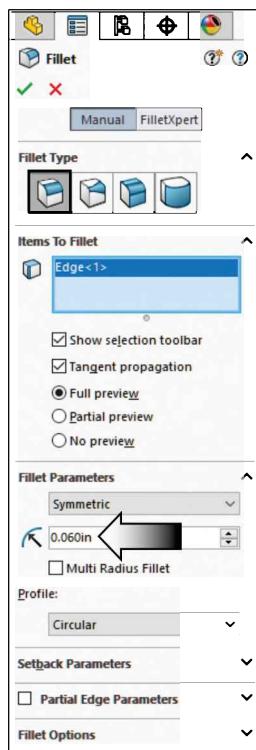
Click **Fillet**.

For Fillet Type, use the default **Constant Size**.

For Items to Fillet, select one of the **inner edges** of the recessed feature.

For Radius, enter **.060in**.

Click **OK**.



8. Adding the .125" fillet:

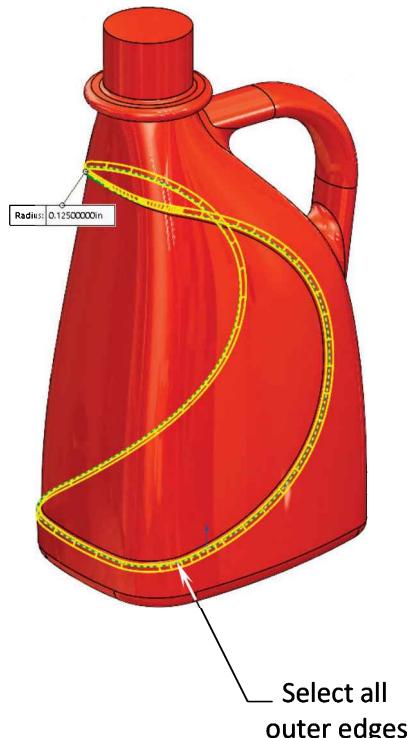
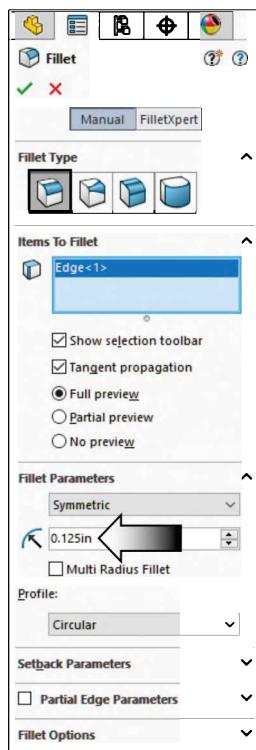
Click **Fillet** again.

For Fillet Type, use the default **Constant Size**.

For Items to Fillet, select one of the **outer edges** of the recessed feature.

For Radius, enter **.125in**.

Click **OK**.



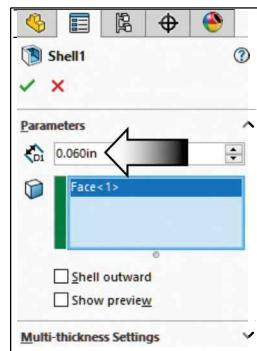
9. Shelling the model:

Switch to the **Features** tab.

For Wall Thickness, enter **.060in**.

For Faces to Remove, select the **top surface** of the model.

Click **OK**.

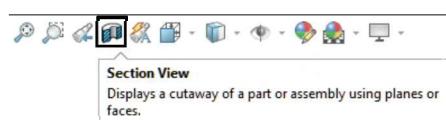


Select face to remove



10. Verifying the wall thickness:

Section View is one of the quick ways to inspect the wall thickness and to view the interior features.



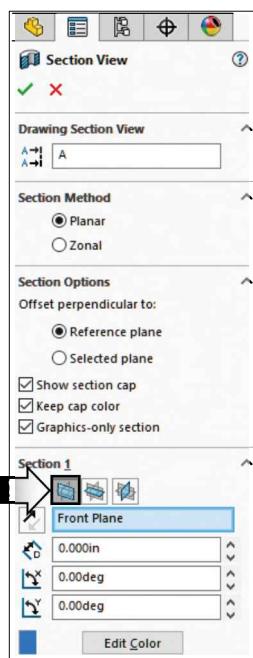
Click **Section View** on the View-Heads Up tool bar.

For Section 1, use the default **Front** plane.

For Distance, use the default **0.00** dimension.

Click **OK**.

Zoom in closer to inspect the wall thickness, especially around the recessed area.



Exit the Section View command.

11. Saving your work:

Click **File / Save As**.

Enter **Using Ruled Surface_Completed** for the name of the file.

Click **Save**.



CHAPTER 11

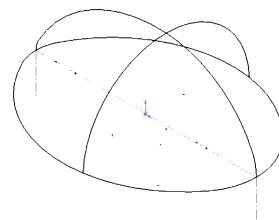
Surface Vs. Solid Modeling

Surface vs. Solid Modeling

Surfaces are a type of geometry that can be used to create solid features. Surface tools are available on the Surfaces tab.

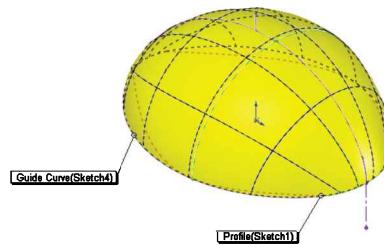
You can create surfaces using these methods:

- Insert a planar surface from a sketch or from a set of closed edges that lie on a plane
- Extrude, revolve, sweep, or loft from sketches
- Offset from existing faces or surfaces
- Import a file
- Create mid-surfaces
- Radiate surfaces



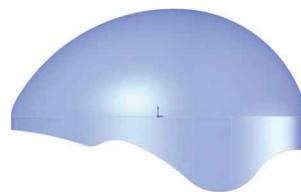
You can modify surfaces in the following ways:

- Extend
- Trim existing surfaces
- Un-trim surfaces
- Fillet surfaces
- Repair surfaces using Filled Surface
- Move/Copy surfaces
- Delete and patch a face
- Knit surfaces



You can use surfaces in the following ways:

- Select surface edges and vertices to use as a sweep guide curve and path
- Create a solid or cut feature by thickening a surface
- Extrude a solid or cut feature with the end condition Up to Surface or Offset from Surface
- Create a solid feature by thickening surfaces that have been knit into a closed volume
- Replace a face with a surface



Surface vs. Solid Modeling

Safety Helmet



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: **ANSI**
Units: **INCHES – 3 Decimals**

Tools Needed:



Insert Sketch



Plane



Lofted Surface



Swept Surface



Planar Surface



Knit Surface



Revolve Cut



Sweep Cut

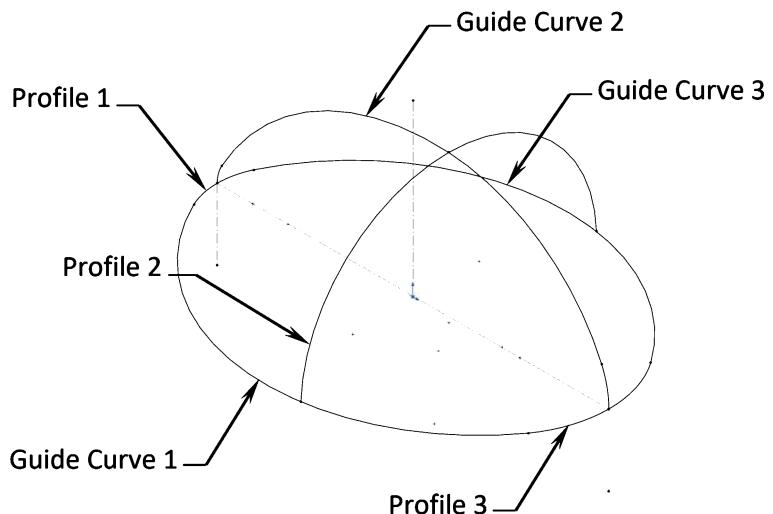


Surface Thicken

1. Opening the Existing file:

Browse to the Training Files folder and open a part document named: **Helmet**.

This part file has 3 sketch profiles and 3 guide curves previously created.

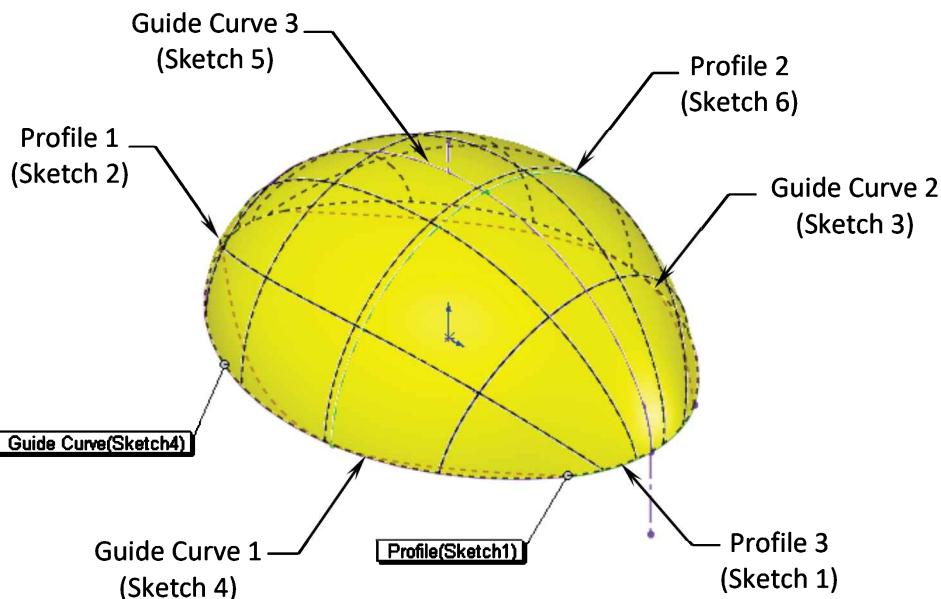
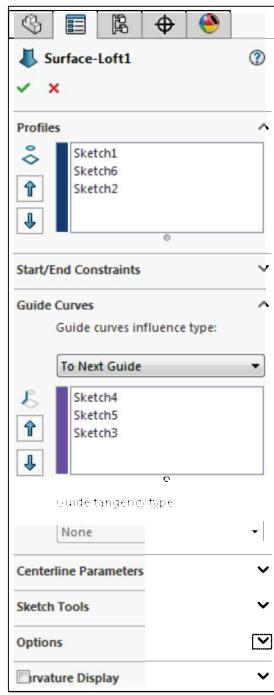
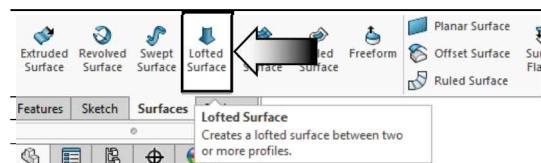


2. Constructing the Body of the Helmet:

Click **Lofted Surface** or select **Insert, Surface, Loft**.

Select the **3 Sketch Profiles**.

Select the **3 Guide Curves** as noted.



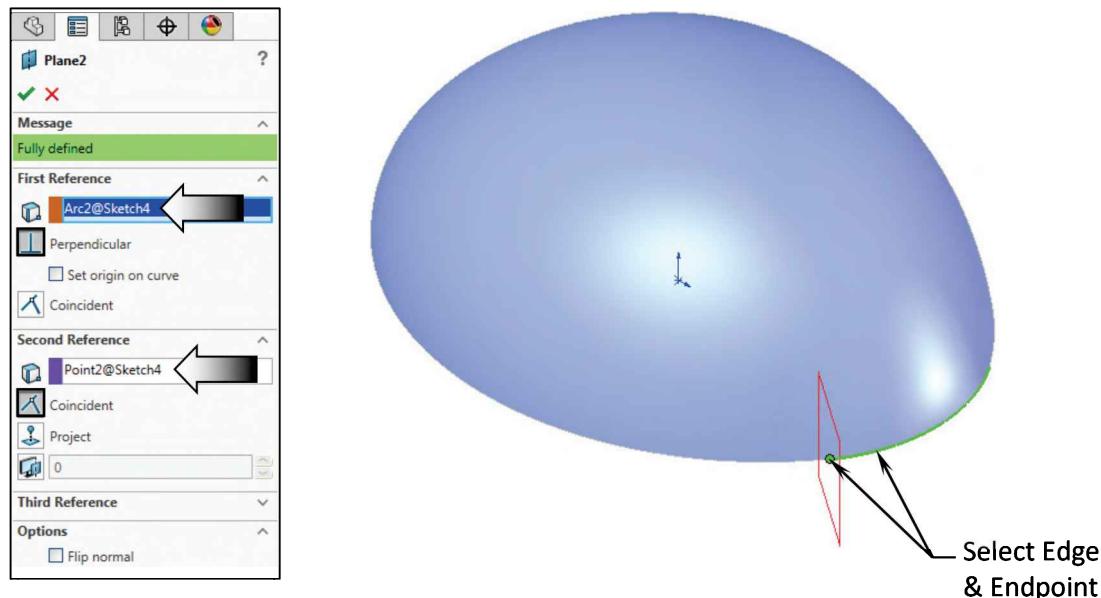
Click **OK**.

3. Creating a new work plane:

Create a plane **Perpendicular** as illustrated.

Click **Insert, Reference Geometry, Plane** .

Select the **circular edge** and its **endpoint** as noted.



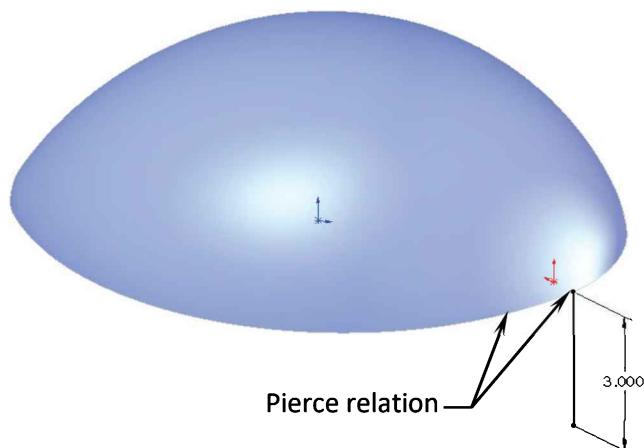
Click **OK**.

4. Sketching the Sweep-Profile:

Open a **new sketch**  on the **new plane** and sketch a **Vertical Line** as shown.

Add a **3.000in.** dimension and a **Pierce** relation between the end point of the line and the bottom edge.

Exit the Sketch .



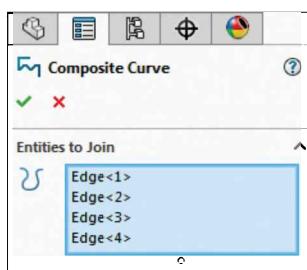
5. Creating the Sweep Path (Composite Curve):

Click **Composite** on the **Curves** toolbar, or select:

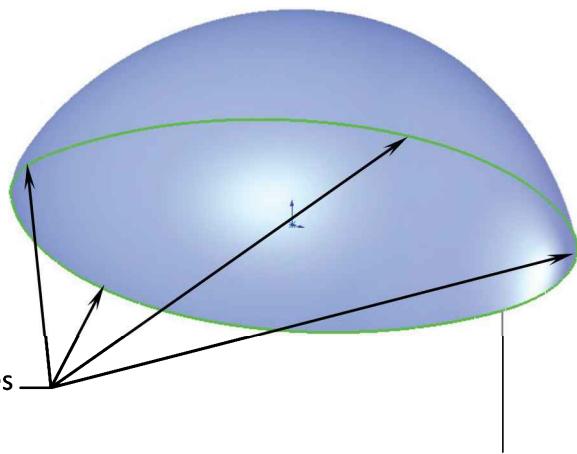
Insert / Curve /Composite .

Select all edges as noted.

Click **OK**.



Select all edges



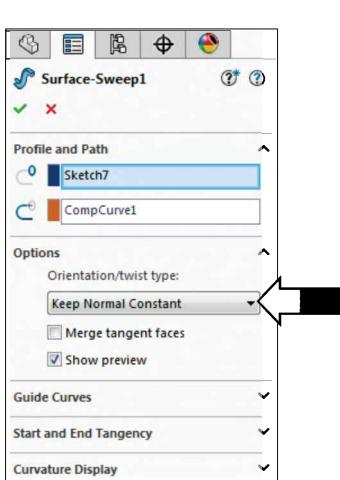
(The alternative option is to use the SelectionManager to select the 4 bottom edges and use as the sweep path instead of the Composite Curve.)

6. Creating a Swept-Surface:



Click **Swept Surface** or select: **Insert, Surface, Sweep**.

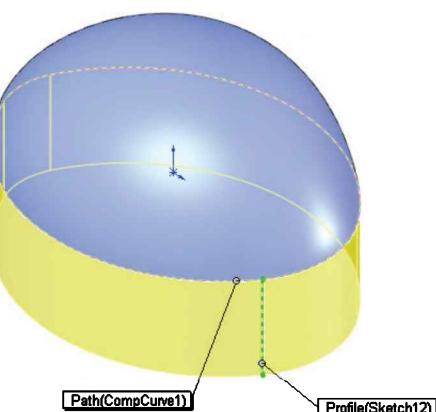
For Sketch Profile, select the **Vertical Line**.



For Sweep Path, select the **Composite-Curve** .

Click **Keep Normal Constant** under Options.
(This option keeps the line perpendicular to the sweep path, all around.)

Click **OK**.



7. Adding a Planar Surface:

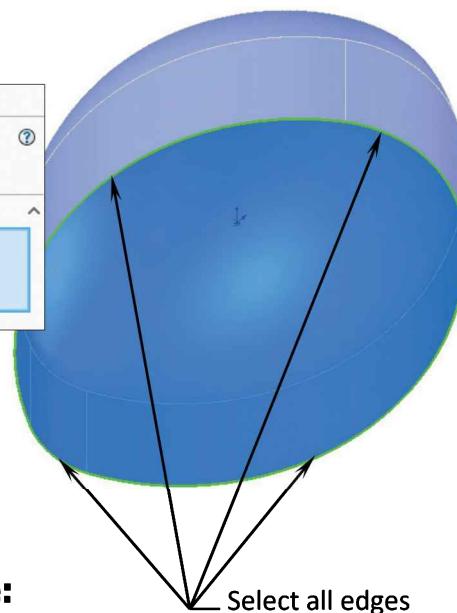
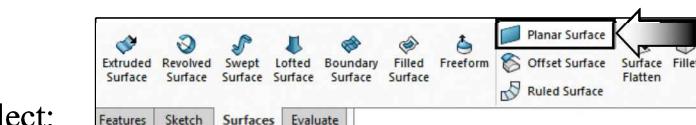
Click **Planar Surface** or select:
Insert / Surface / Planar.

For Bounding Entities, select
all edges at the bottom.

Click **OK**.

The new planar surface
 is created and it closes
 off the bottom of the part.

This model has 3 surfaces
 at this time: the Lofted surface,
 the Swept surface, and the Planar surface.



Select all edges

8. Knitting the three Surface-Bodies into one:

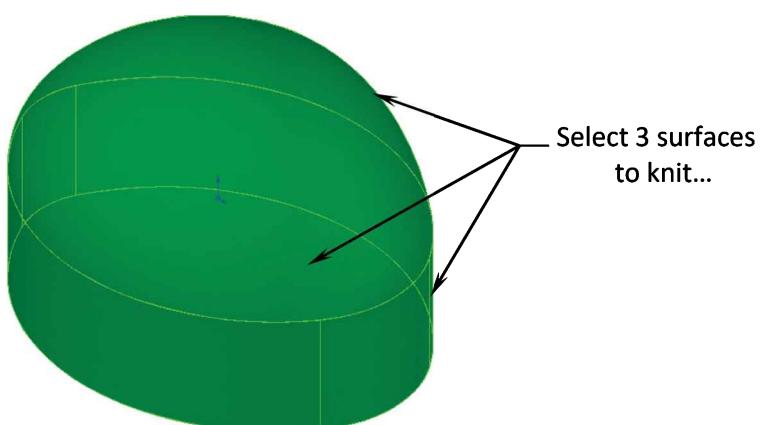
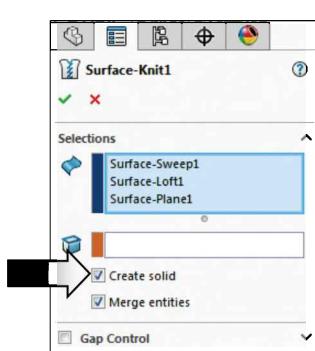
Click **Knit Surface** or select **Insert / Surface / Knit**.

Select the **Lofted-Surface**, the **Swept-Surface**, and the **Planar-Surface**
 either from the FeatureManager tree or from the graphics area.

Clear the Gap Control checkbox.

Enable the **Create Solid** option to convert the surface into a solid body.

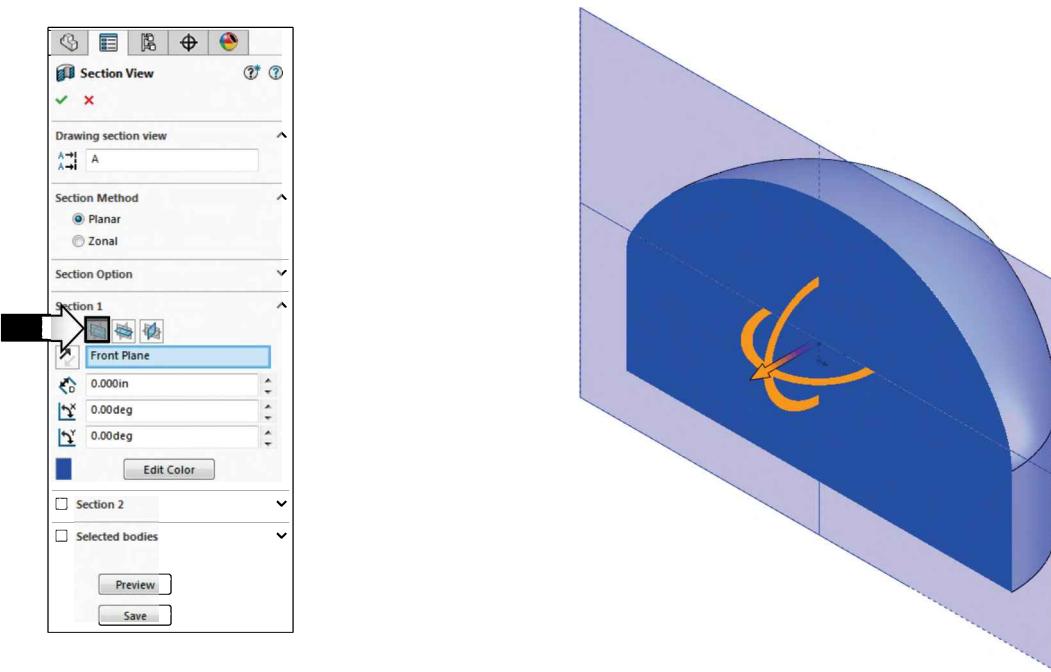
Click **OK**.



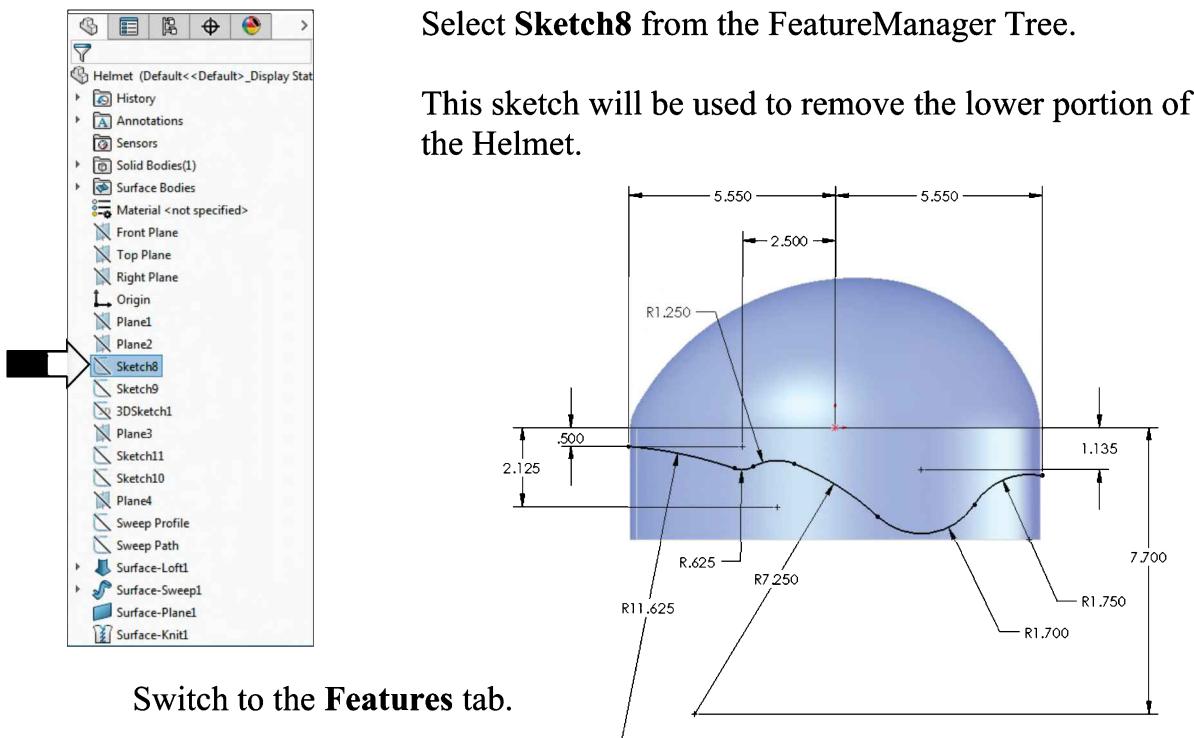
Select 3 surfaces
 to knit...

9. Creating a section view:

Select the Front plane from the FeatureManager and click **Section View** (arrow) to verify the solid material. Click-off the section view command when finished viewing.



10. Adding an Extruded Cut feature:

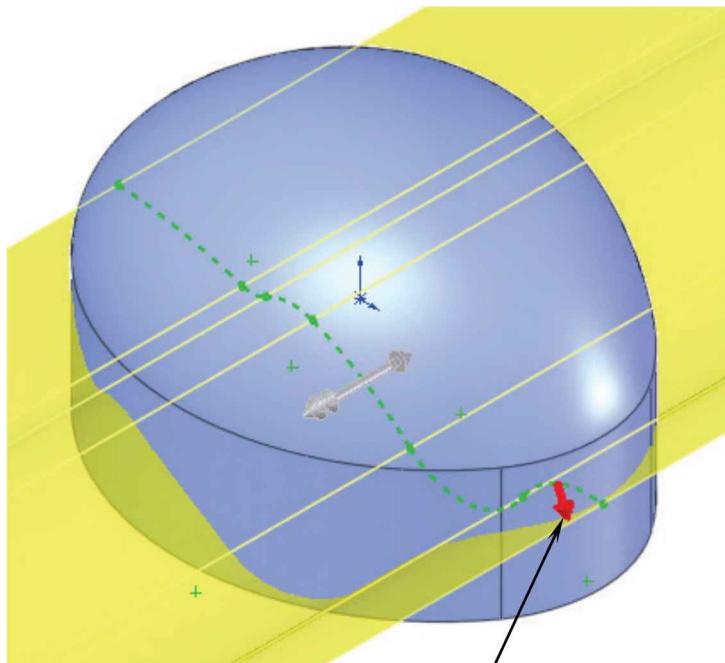
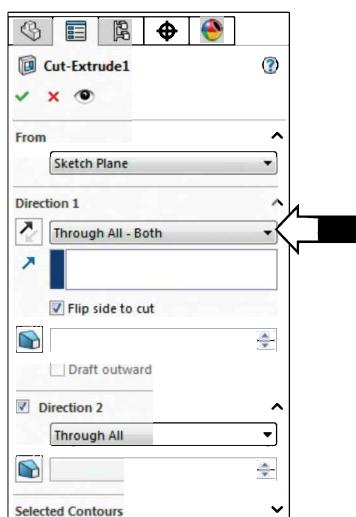


Click **Extruded-Cut**  or select: **Insert / Cut / Extrude**.

For End Condition, select **Through-All Both** from the drop-down list.

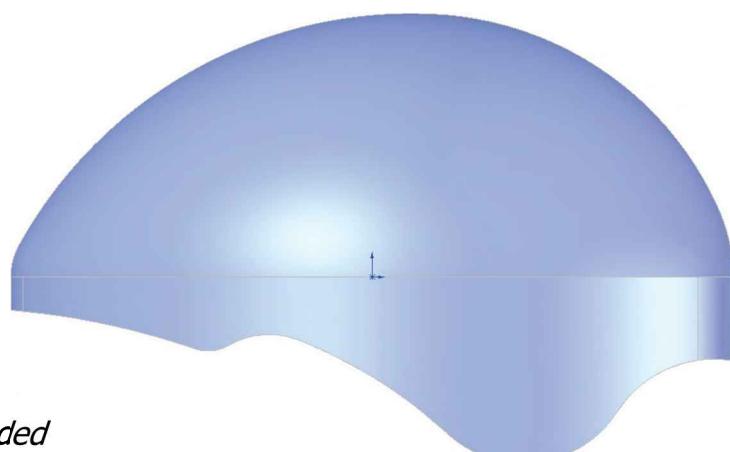
Enable **Flip-Side-To-Cut** if needed, to remove the lower portion of the part.

Click **OK**.



Hide Sketch8.

Inspect your model against the image shown here before moving to the next step.



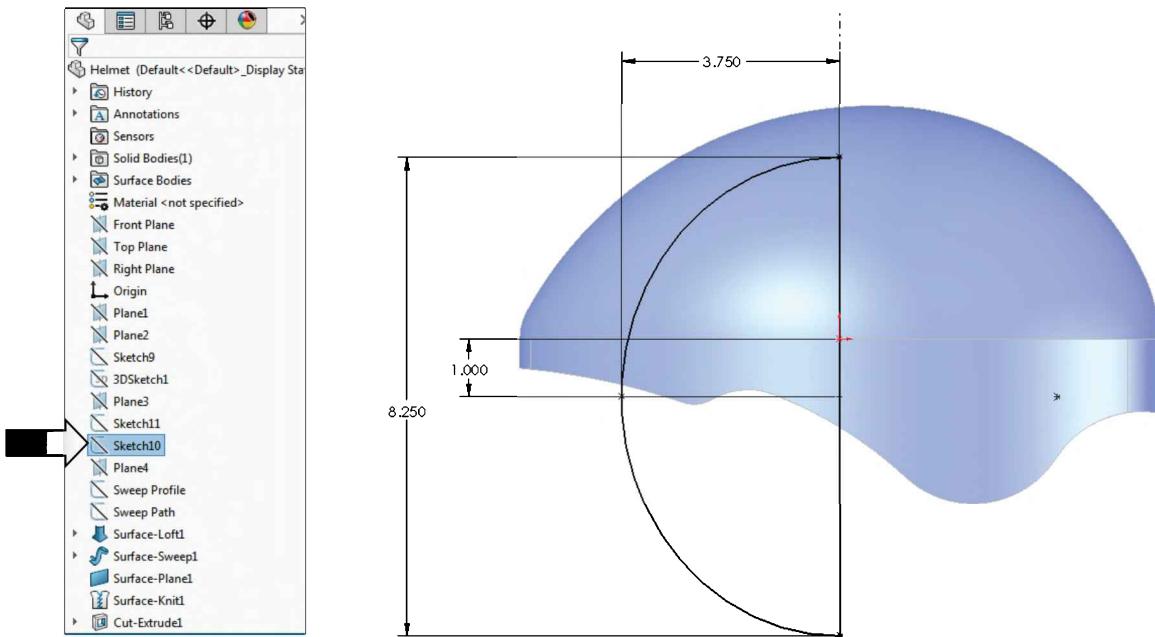
NOTE: Change from Shaded to Shaded With-Edges to be able to see the model edges more clearly.

The resulted cut.

11. Adding a Revolve-Cut feature:

Select Sketch10 from the FeatureManager Tree.

This sketch will be used to remove the material on the inside of the Helmet.



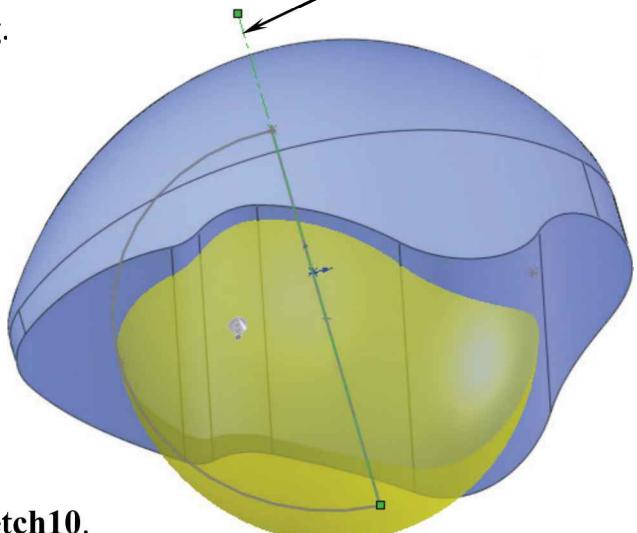
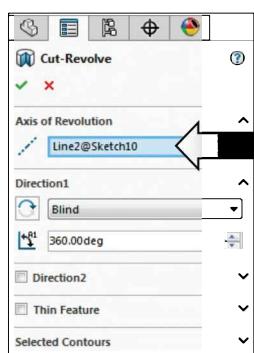
Switch to the **Features** tab.

Click **Revolve-Cut** or select **Insert / Cut / Revolve**.

Select the **vertical centerline** as Revolve-Direction.

Revolve Angle: **360.00deg.**

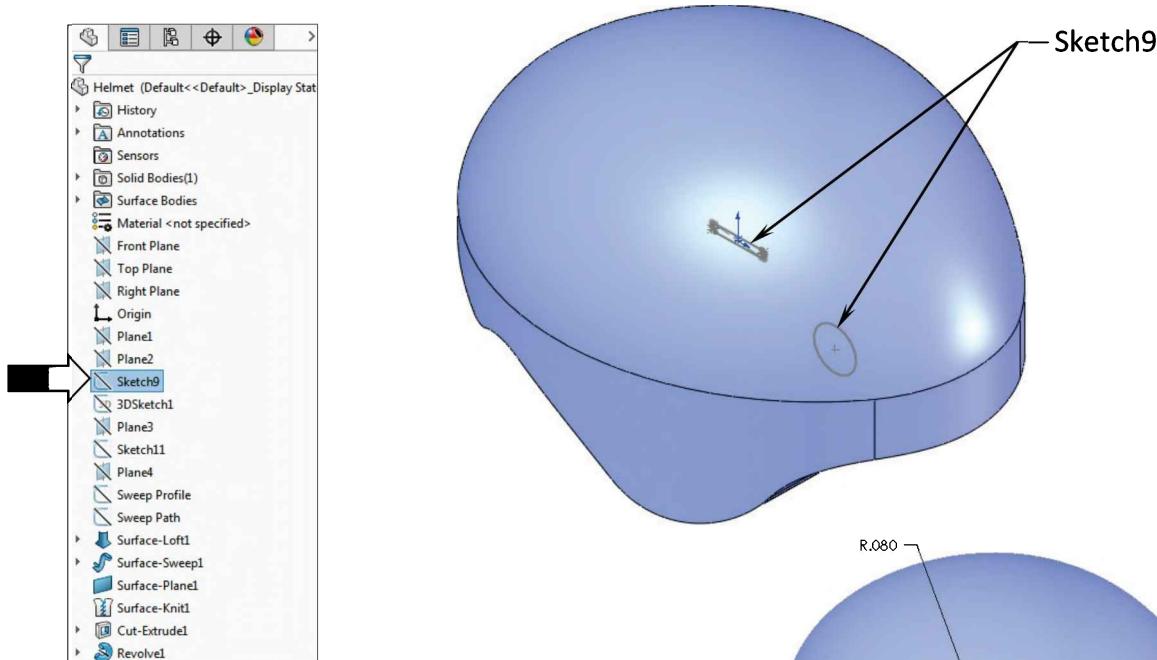
Click **OK**.



12. Adding the side cut features:

Select Sketch9 from the FeatureManager Tree.

This sketch is used to cut from the inside with a **1.00deg.** draft angle.



Switch to the **Features** tab.

Click **Extruded-Cut** or select: **Insert / Cut / Extrude**.

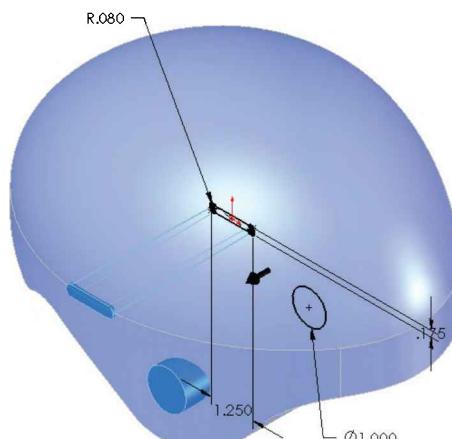
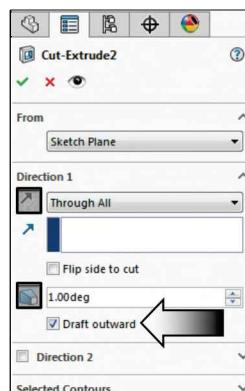
Select **Through-All** for end condition.

Enable Draft Angle and enter: **1.00deg.**

Enable **Draft Outward** option.

Click **OK**.

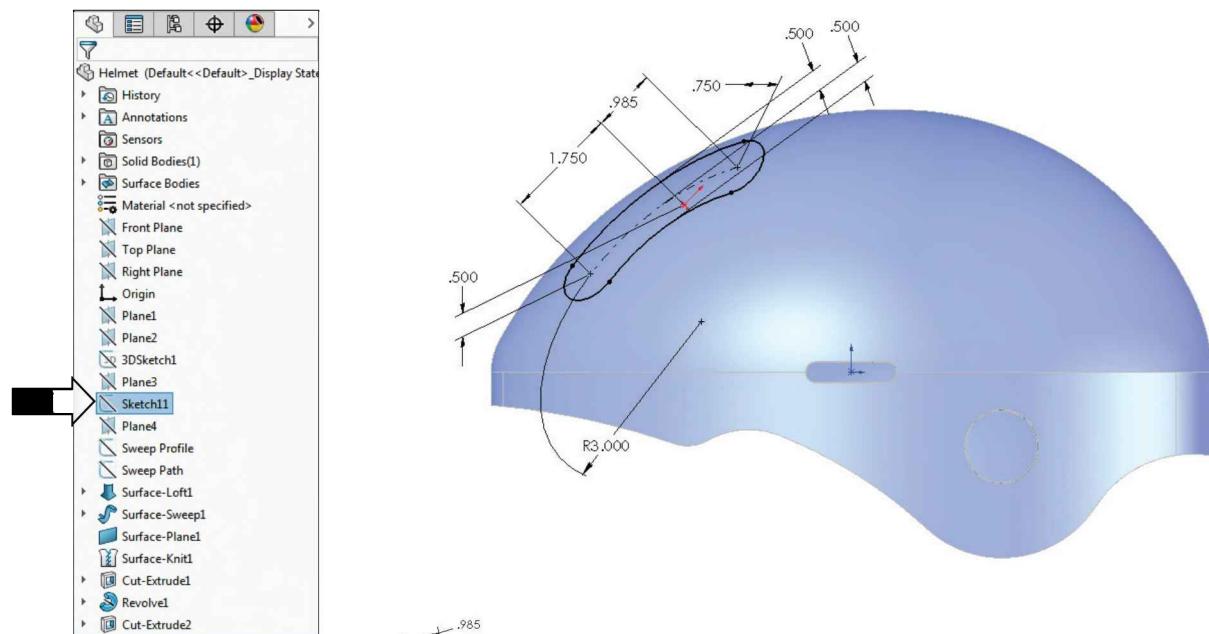
Hide the Sketch9.



13. Creating the cutout slot:

Select Sketch11 from the FeatureManager Tree.

This sketch will be used to cut a slot from the outside of the model and with a **10.00deg.** draft angle.



Switch to the Features tab.

Click **Extruded-Cut** or select: **Insert, Cut, Extrude**.

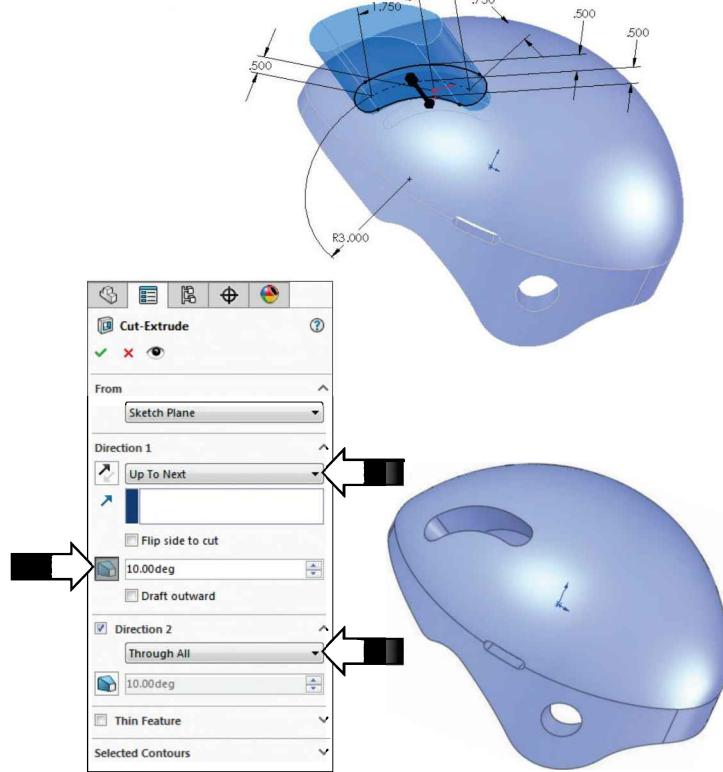
For Direction 1, select **Up-To-Next** end condition.

Enable Draft Angle and enter: **10.00deg.** (inward).

For Direction 2, select **Through-All** end condition, no draft. (The 2nd direction is needed to remove the material on the mirrored side.)

Click **OK**.

Hide Sketch11.

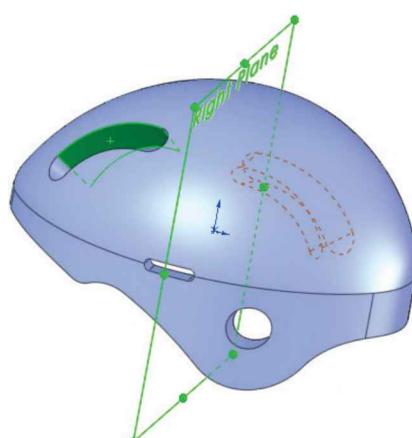
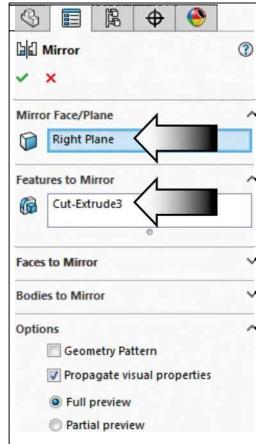


14. Mirroring the slot:

Click **Mirror** – OR – select **Insert, Pattern-Mirror, Mirror** .
For Mirror Plane, select the **Right** plane.

For Features to Mirror,
select the **Cut Extrude3**
(the slot).

Click **OK**.



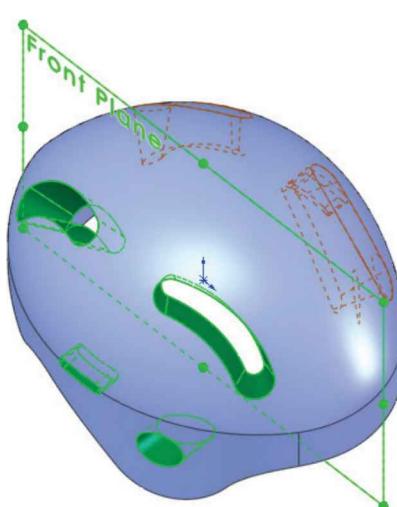
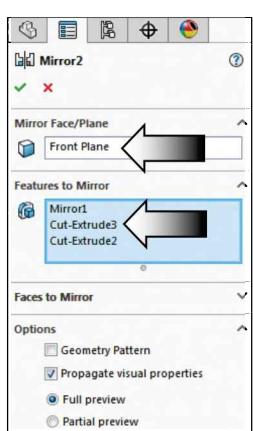
15. Mirroring the cuts:

Click **Mirror**  or
select: **Insert / Pattern-Mirror / Mirror**.

For Mirror Plane, select the **Front** plane.



For Features to Mirror, select **both Slots**
and the **Side-Holes** as Features to Mirror.



Click **OK**.

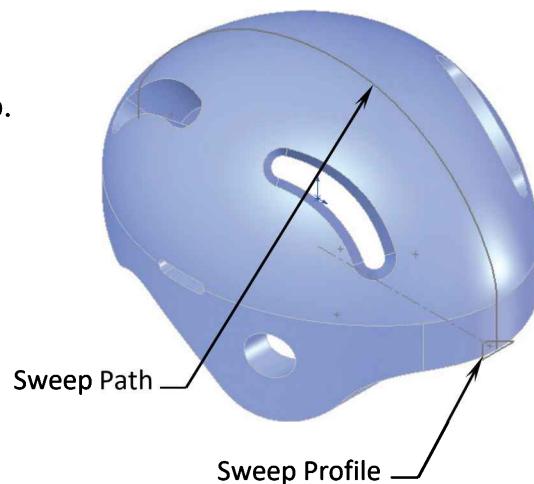
Inspect your model against the image shown above.

16. Creating the Sweep-Cut:

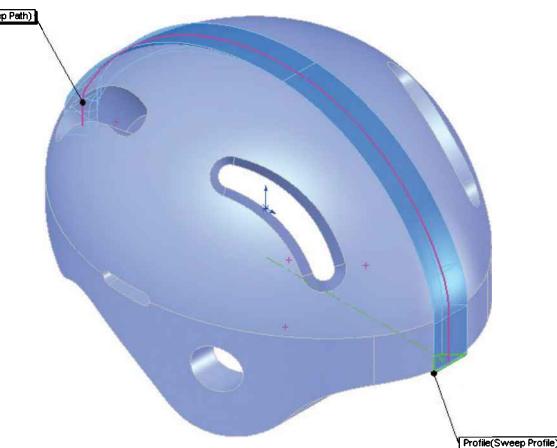
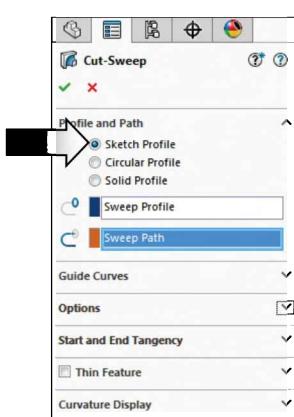
Click  or select: **Insert, Cut, Sweep**.

Select the sketch **Sweep-Profile** and the sketch **Sweep Path** from the FeatureManager tree.

The preview graphics shows the proper transition of the sweep feature.

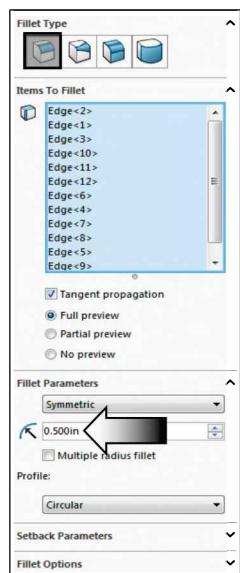


Click **OK**.



17. Adding the .500" fillets:

Click **Fillet**  or select **Insert, Features, Fillet-Round**.



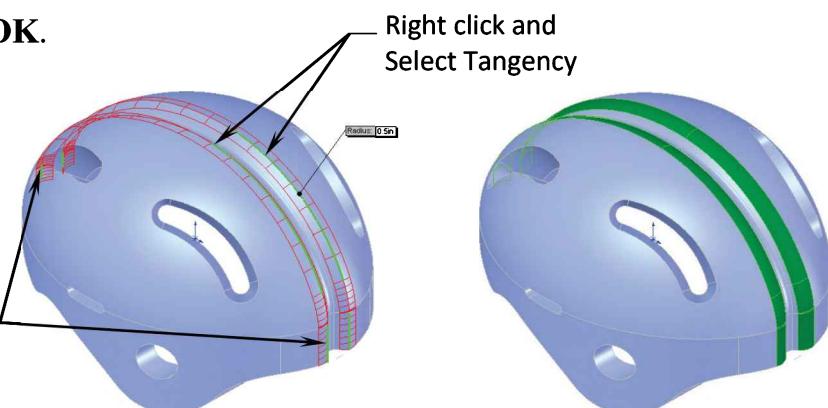
Enter **.500in** for radius.

Select the **12 edges** of the Swept feature.

Click **OK**.

Right click and Select Tangency

Select 4 edges on both sides

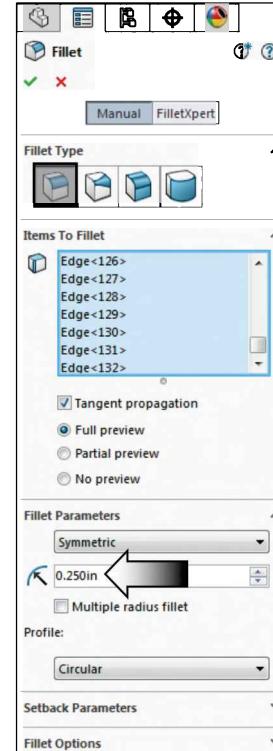
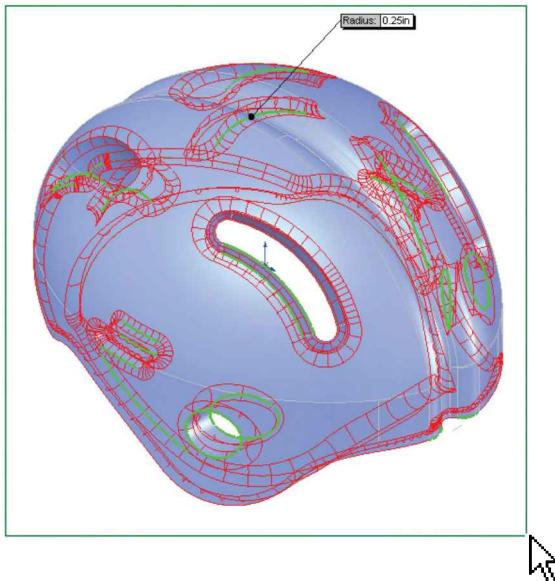


18. Adding the .250" fillets:

Click **Fillet**  or select: **Insert, Features, Fillet-Round.**

Enter **.250"** for radius.

Select **all edges** of the part (Control+A).



Click **OK**.

19. Saving your work:

Click **File / Save As / Helmet / Save**. (Save on the Desktop.)



Front Isometric



Back Isometric

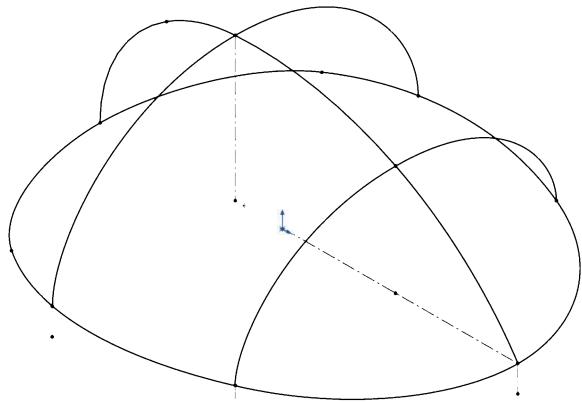
Exercise: Helmet_Surfaces

This exercise provides you with an opportunity to apply what you have learned from the previous lesson, where most of the surfacing tools will be used again to create another helmet design.

1. Opening a part document:

Browse to the training folder and open a part document named:
Helmet_Surface_Exe.sldprt.

All required sketches have already been created to help focus on the use of the surfacing tools.



2. Creating a boundary surface:

Switch to the **Surfaces** tab and click: **Boundary Surface**.

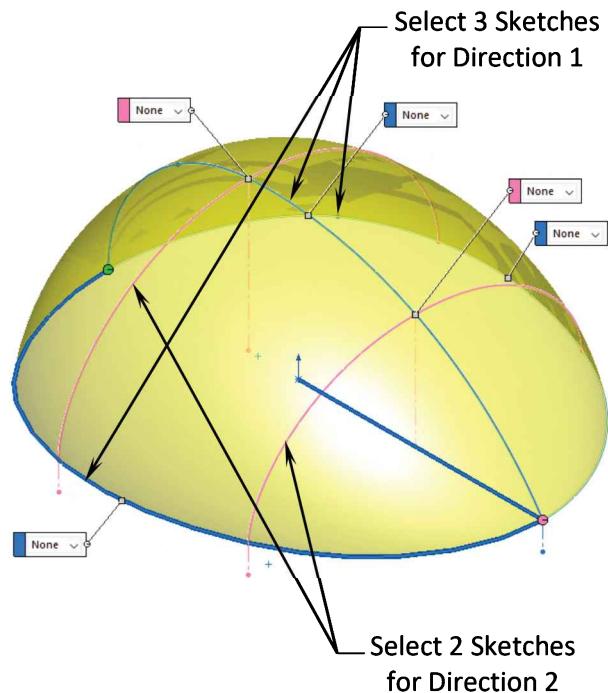
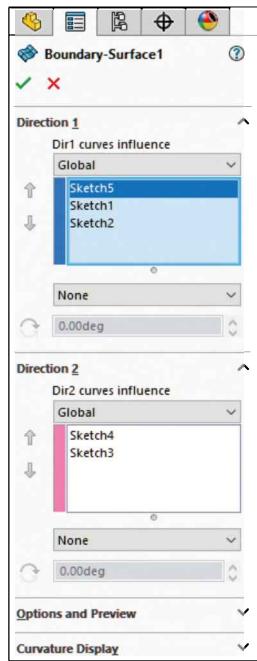
For Direction 1, select **Sketch5**, **Sketch1**, and **Sketch2** as indicated.

For Direction 2, select **Sketch3** and **Sketch4** in the graphics area.

Drag the connectors to reposition them, if the surface is twisting.

Keep all other parameters at their defaults.

Click **OK**.



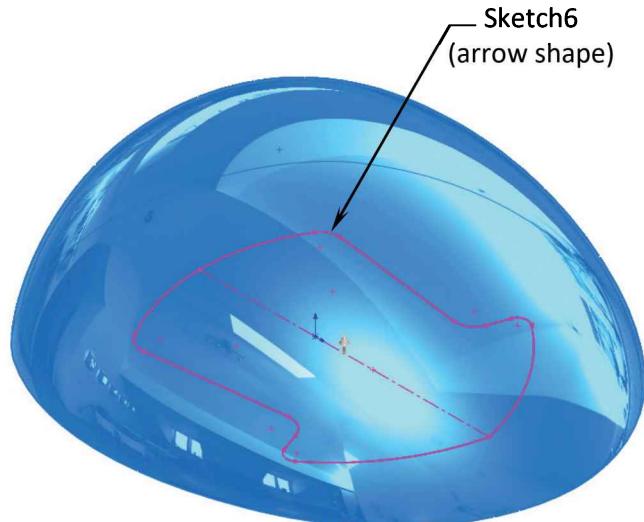
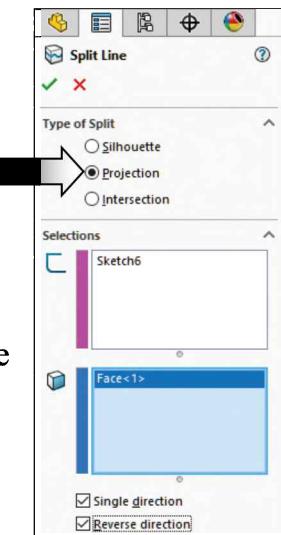
3. Creating a split line feature:

Select Curves,
Split Line.

Click the
Projection
option.

For Selection,
select Sketch6
from the Feature
Manager tree.

For Faces to
Split, select the
surface model
in the graphics area.



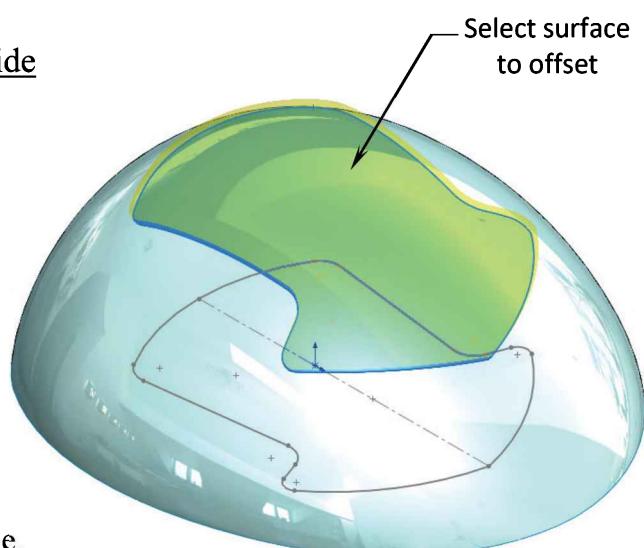
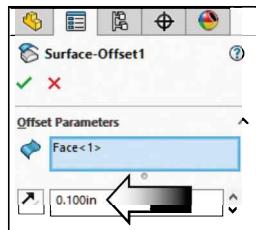
Enable the Single Direction and Reverse Direction checkboxes.

Click OK.

4. Creating the 1st offset surface:

Click Offset Surface on the Surfaces tab.

For Offset Parameters, select the inside
portion of the split surface as noted.



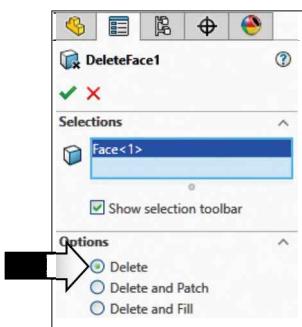
For Offset Distance, enter **.100in**.
Place the offset surface on the outside.

Click OK.

5. Deleting a surface:

Click **Delete Face** on the **Surfaces** tab.

For Selections, select the surface below the offset surface.



For Options, click **Delete** (arrow).

Click **OK**.

Select the surface below

6. Adding a swept surface:

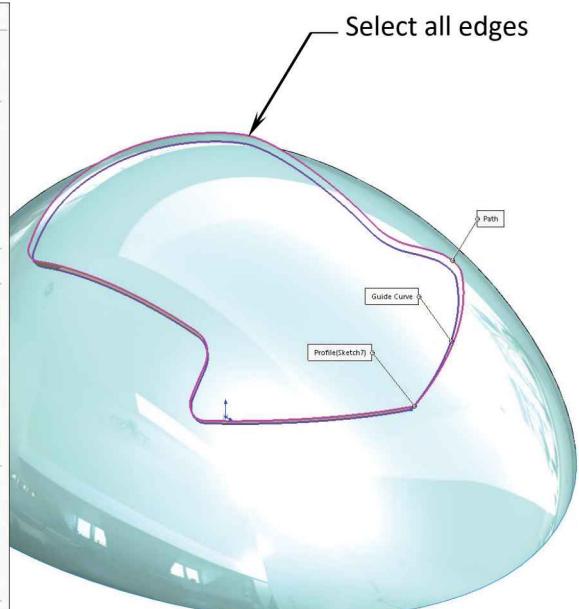
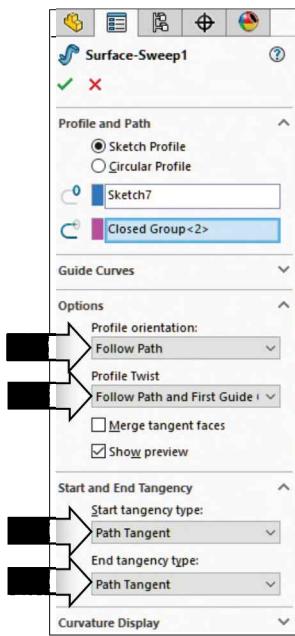
To close-off the gap between the helmet body and the offset surface, a surface-swept is needed.

Click **Swept-Surface**.

For Sweep Profile, select **Sketch7** from the Feature-Manager tree.

For Sweep Path, use Selection-Manager to select all edges of the Offset Surface.

Set the options as shown in the dialog box and click **OK**.



7. Creating another split line feature:

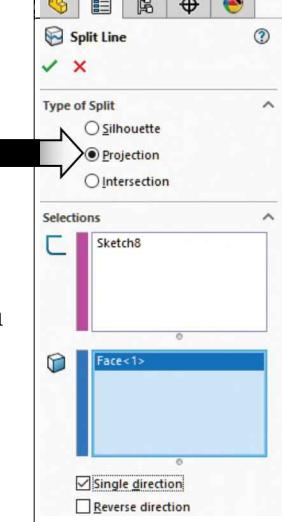
Click Curves, Split Line.

For Type of Split, use the default **Projection** type.

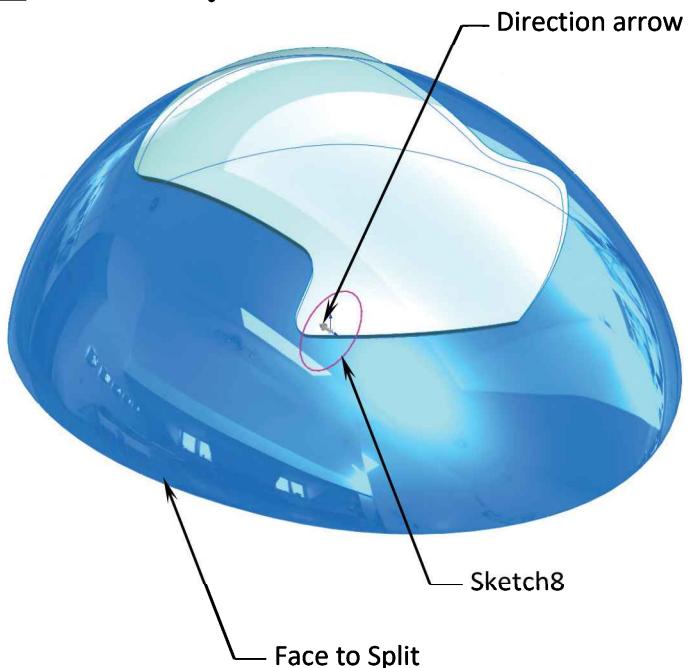
For Sketch to Split, select **Sketch8** from the FeatureManager tree (the Circle).

For Faces to Split, select the helmet surface body.

Enable the **Single-Direction** checkbox, ensure that the direction arrow is pointing towards the left side.



Click **OK**.



8. Creating the 2nd offset surface:

Click **Offset Surface**.

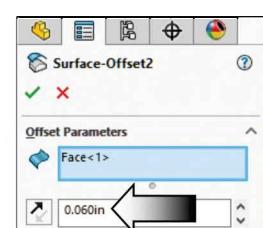
For Surface to Offset, select the new-circular **split surface**.

For Offset Distance, enter: **.060in**.

Place the offset surface on the outside.

Offset to outside

Click **OK**.



9. Deleting another surface:

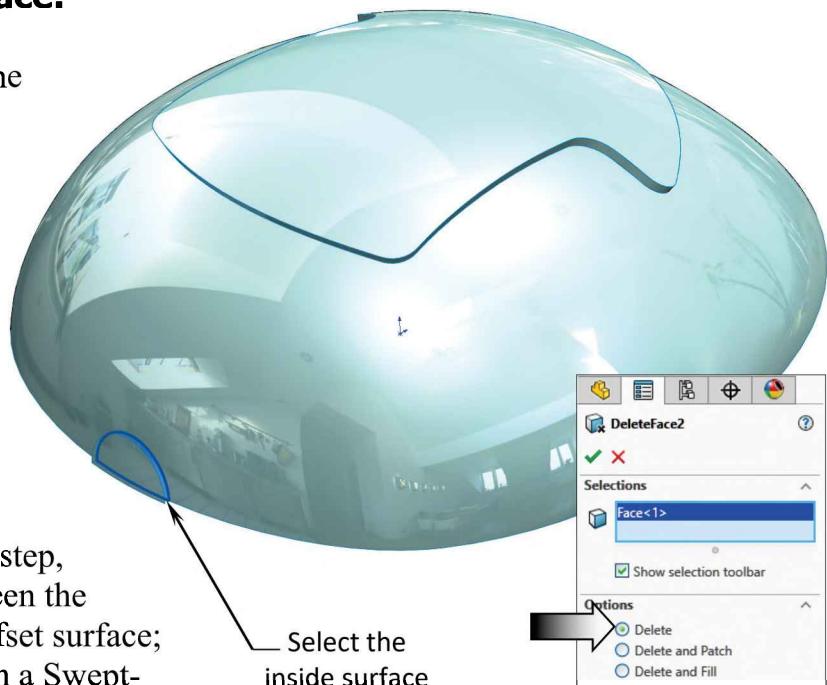
Click **Delete Face** on the Surfaces tab.

For Selections, select the surface behind the Offset Surface2.

For Options, select **Delete**.

Click **OK**.

Similar to the previous step, this creates a gap between the helmet body and the offset surface; it needs to be filled with a Swept-surface.



10. Adding another swept surface:

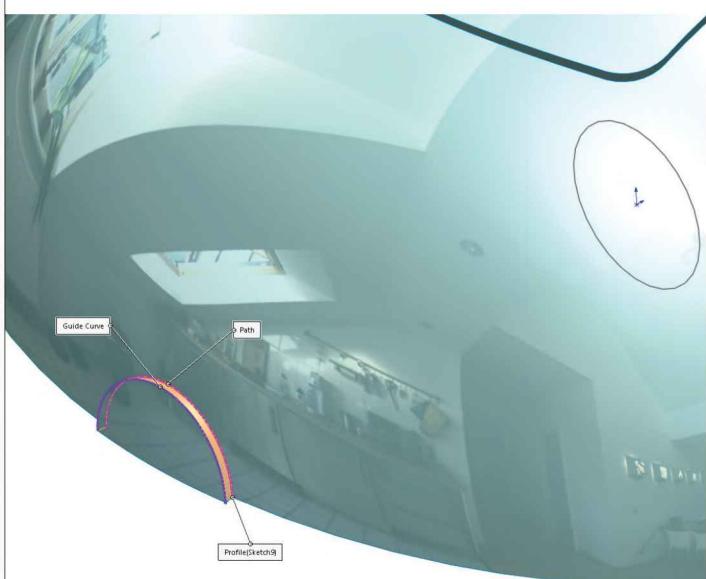
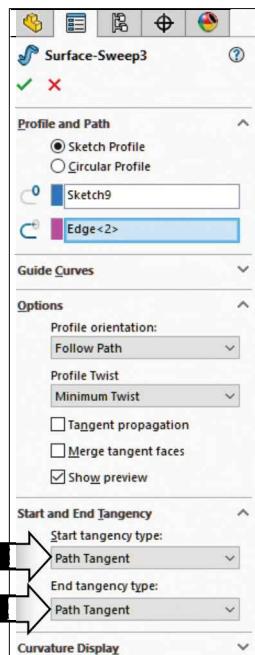
Click **Swept Surface** on the Surfaces tab.

For Sweep Profile, select **Sketch9** from the FeatureManager tree.

For Sweep Path, select the curved edge of the offset-surface.

Enable the options shown in the dialog box.

Click **OK**.



11. Trimming a surface:

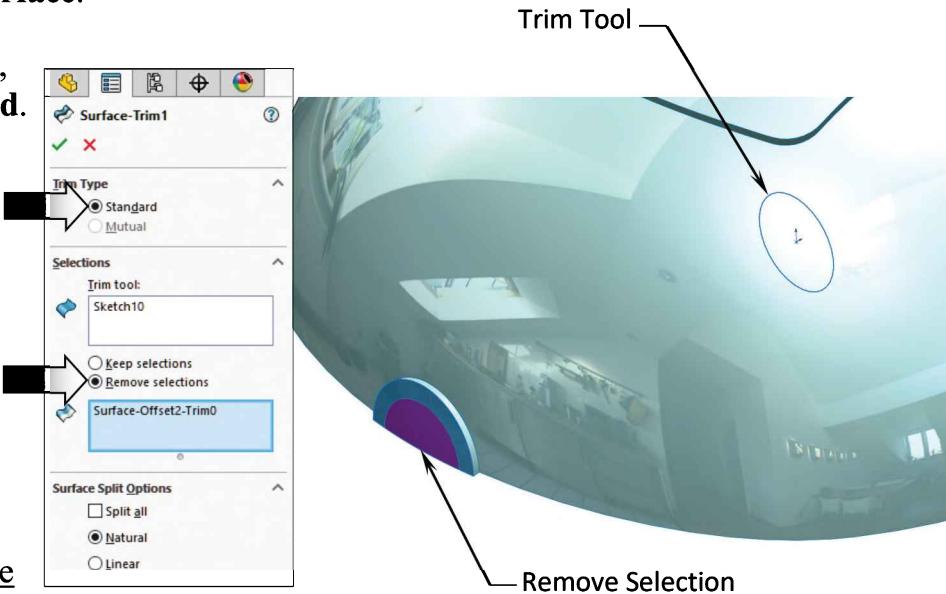
We deleted a couple of surfaces so far, let us try and trim a surface this time.

Click **Trim Surface**.

For Trim Type, select **Standard**.

For Trim-Tool, select **Sketch10** from the Feature-Manager tree.

Click **Remove Selection** and select the inside portion of the Split Surface and click **OK**.



12. Creating a surface filled:

Click **Filled Surface** on the **Surfaces** tab.

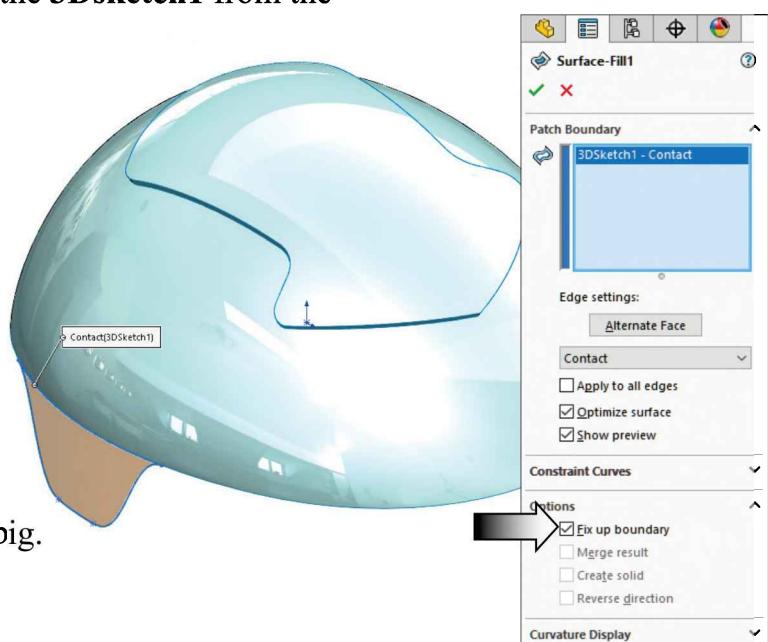
For Patch Boundary, select the **3DSketch1** from the FeatureManager tree.

For Edge Settings, use the default **Contact** option.

Enable the options:
Optimize Surface and
Fix Up Boundary.

Click Fix Up Boundary to construct a valid boundary by automatically building or trimming missing pieces that are too big.

Click **OK**.



13. Mirroring a surface:

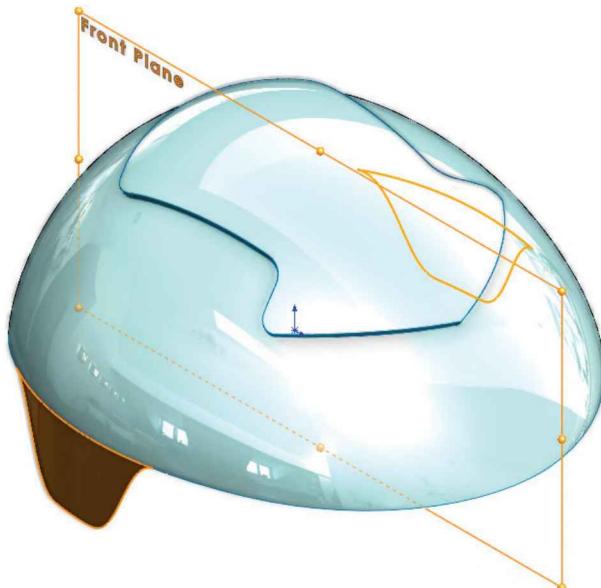
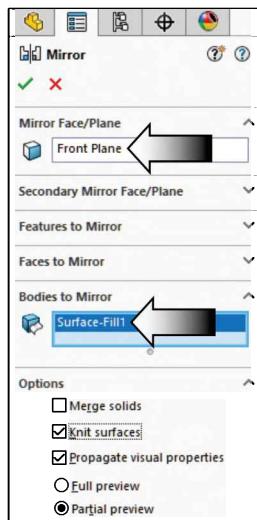
Switch to the **Features** tab and click **Mirror**.

For Mirror Face/Plane, select the **Front** plane from the FeatureManager tree.

Expand the Bodies to Mirror section and select the **Filled Surface** created from the last step.

Leave other parameters at their defaults.

Click **OK**.



14. Trimming the surfaces:

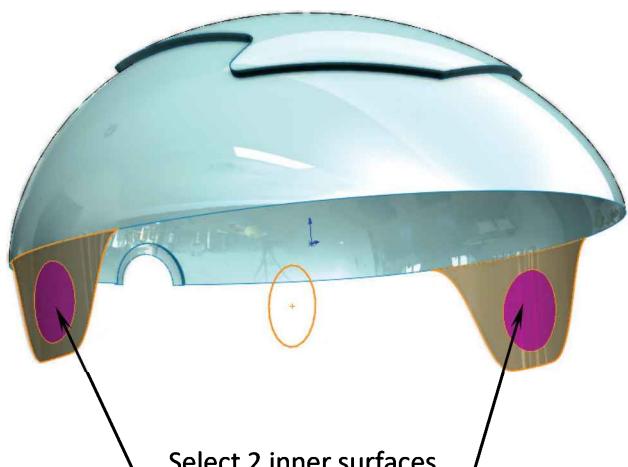
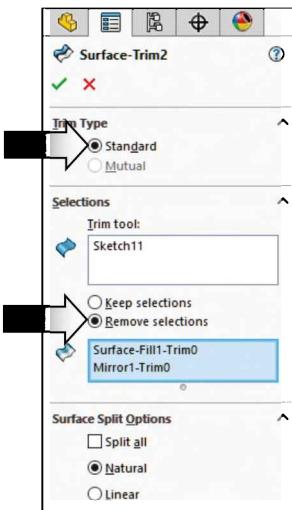
Switch back to the **Surfaces** tab and click **Trim Surface**.

For Trim Type, click **Standard**.

For Trim Tool, select **Sketch11** from the FeatureManager tree.

For Remove-Selection, select the inside portions of the **Filled Surfaces** on both sides as indicated.

Click **OK**.



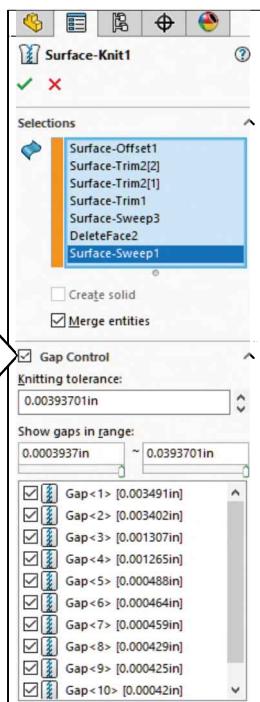
15. Knitting all surfaces:

Click Knit Surface.

Expand the Surface Bodies folder and select all **7 surfaces** inside.

Enable the Gap Control checkbox and check all boxes.

Click OK.



16. Adding the .050in fillets:

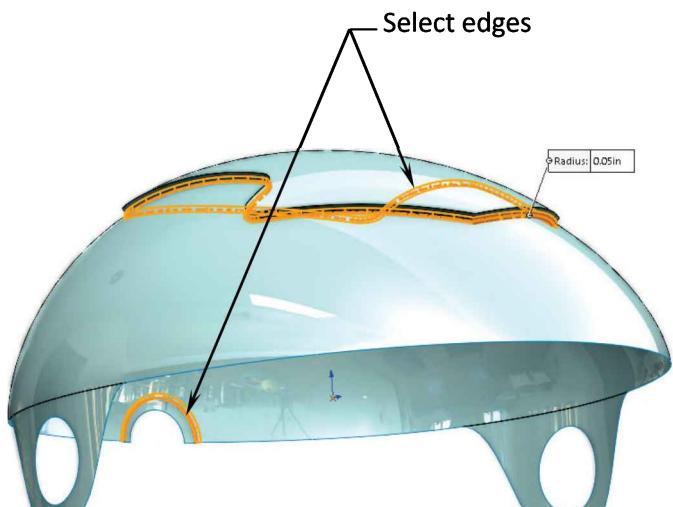
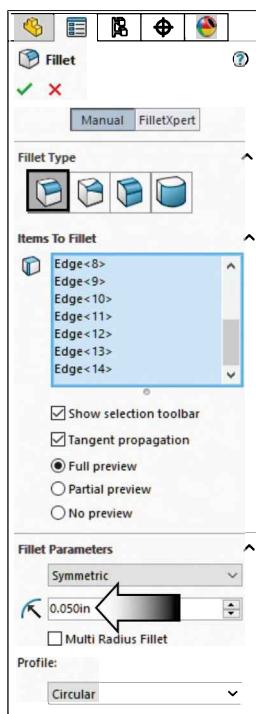
Click Fillet.

Use the default **Constant Size fillet**.

Enter: **.050”** for radius.

Select the bottom edges of the swept surface and the upper edge of the rear opening.

Click OK.



17. Adding the .040in fillets:

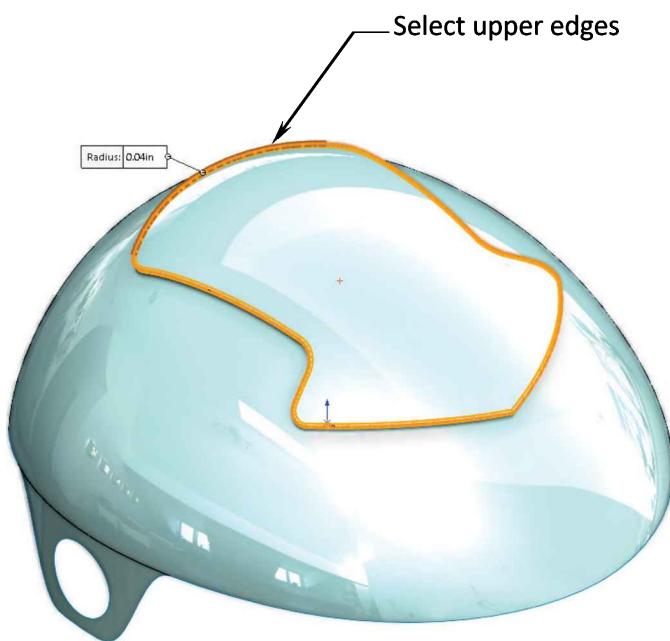
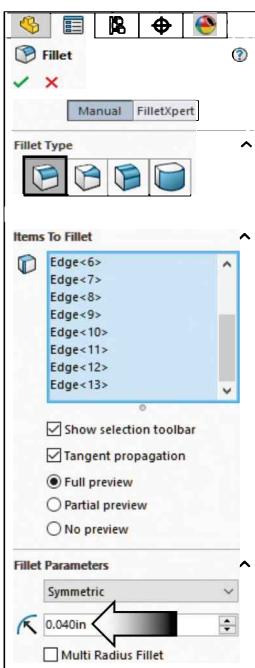
Click **Fillet**.

Use the default Constant Size fillet.

Enter **.040"** for radius.

Select the upper edges of the offset surface as indicated.

Click **OK**.



18. Adding the .030in fillets:

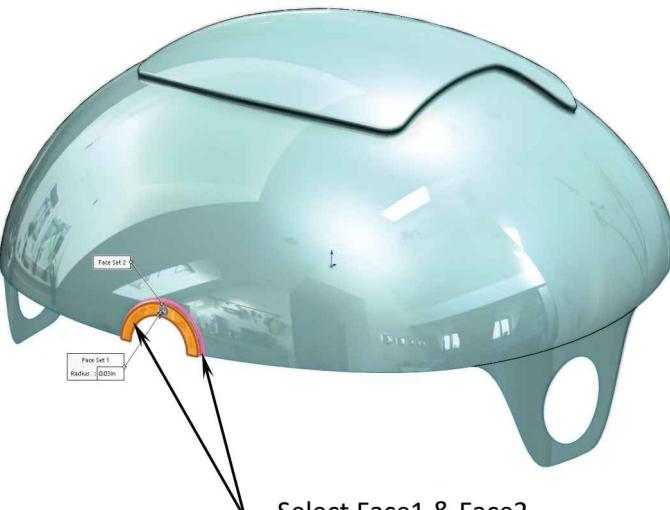
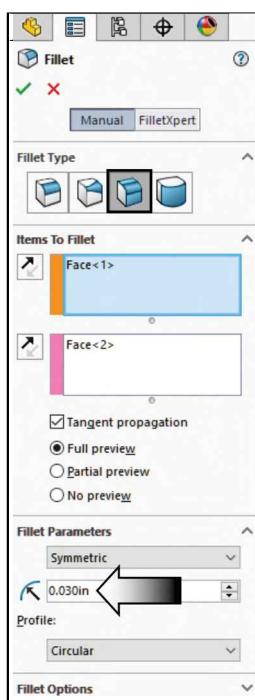
Click **Fillet** again.

For Fillet Type, select the **Face-Fillet** option.

Enter: **.030"** for radius.

For Face-Group 1, select the **outer face** of the offset surface.

For Face-Group 2, select **Fillet2**.

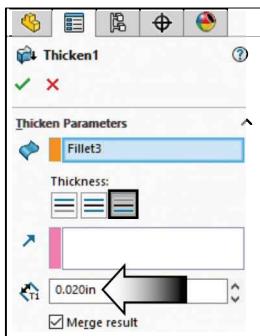


Click **OK**.

19. Thickening a surface body:

Click **Surface Thicken**.

For Thick Parameters, select the Surface body in the graphics area.



For Direction, select **Thicken Side 2 (inside)**

For Thickness, enter **.020in**.

Click **OK**.

20. Saving your work:

Select **File, Save As**.

Enter: **Helmet_Surfaces Completed.sldprt** for the file name.

Click **Save**.



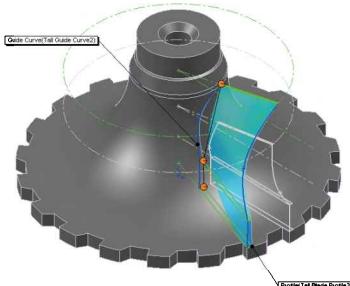
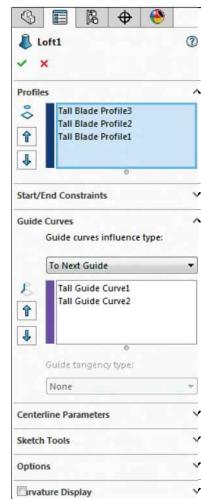
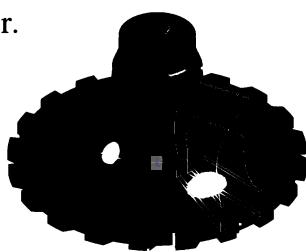
Close all documents.

Exercise: Advanced Loft – Turbine Blades

1. Open a part document:

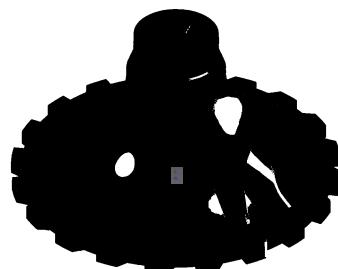
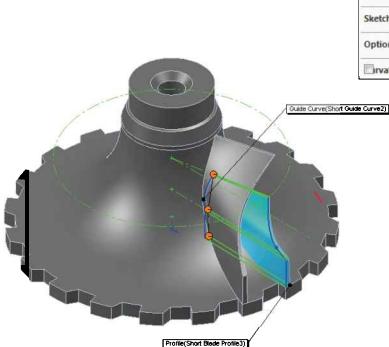
Turbine.sldprt from the Training Files folder.

This part file has 6 sketch profiles and 4 guide curves previously created.



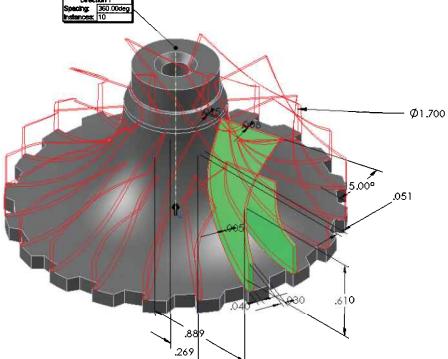
2. Create the 1st loft:

Select the 3 Tall Blade sketches for Profiles.
Select the 2 Tall Guide Curve sketches for Guide Curves.



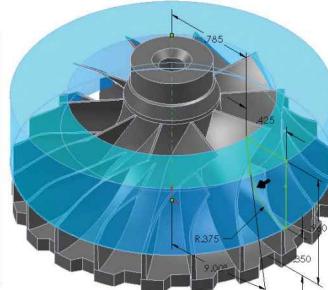
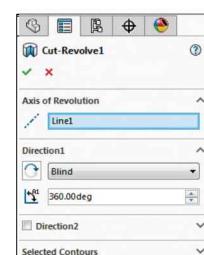
3. Create the 2nd loft:

Select the 3 Short Blade sketches for Profiles.
Select the 2 Short Guide Curve sketches for Guide Curves.



4. Circular pattern the Blades:

Create a circular pattern of the Lofted Blades.
Enter 10 for number of instances.



5. Create a Revolve Cut:

Use the Blade Trim Sketch and create a Revolve-Cut.

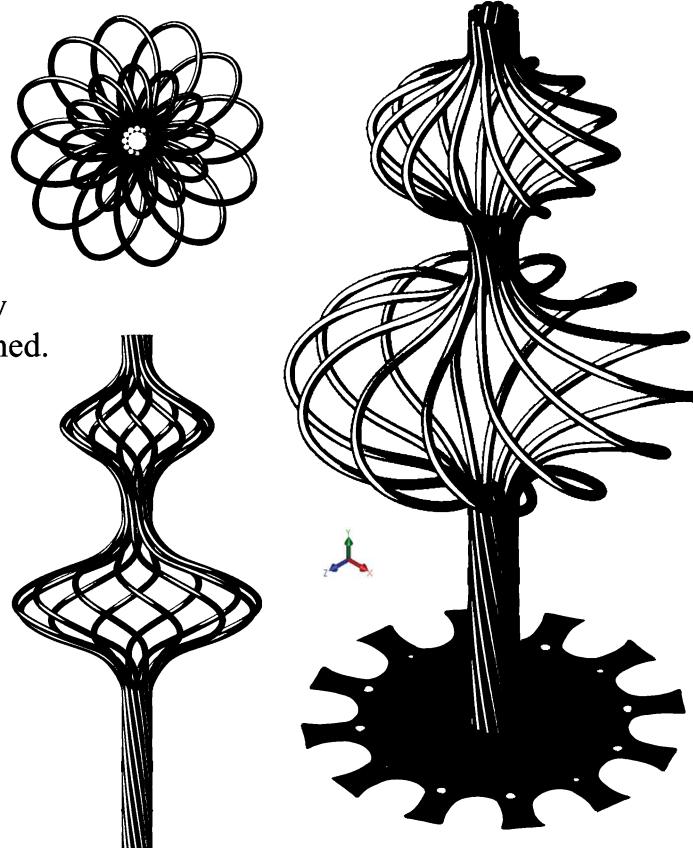
6. Save your work as:

Turbine Blades-Exe.

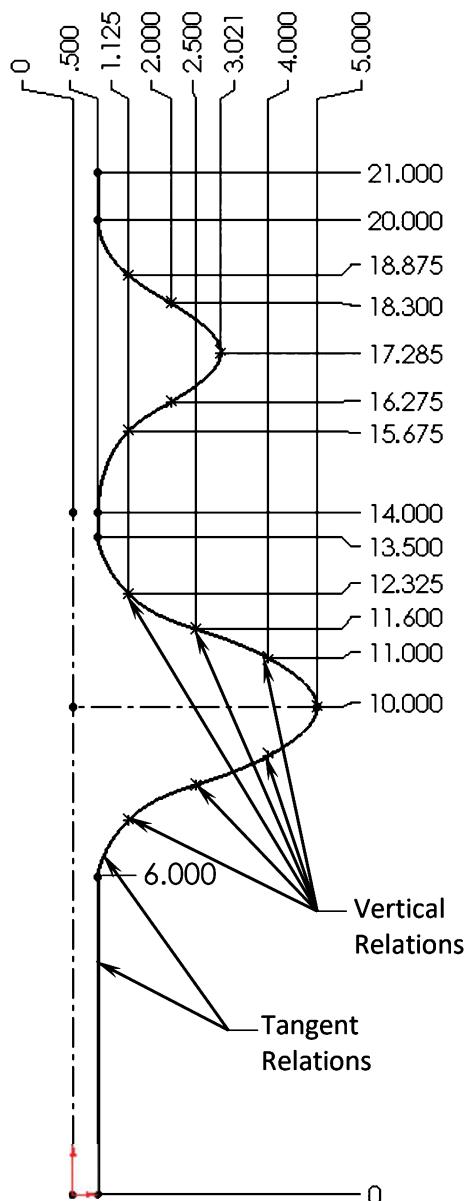
Exercise: Advanced Sweep – Candle Holder

1. Opening the main sketch:

From the Training Files folder open the part document named:
Candle Holder Sketch.



Edit the **Sketch1** and verify that the sketch is fully defined.

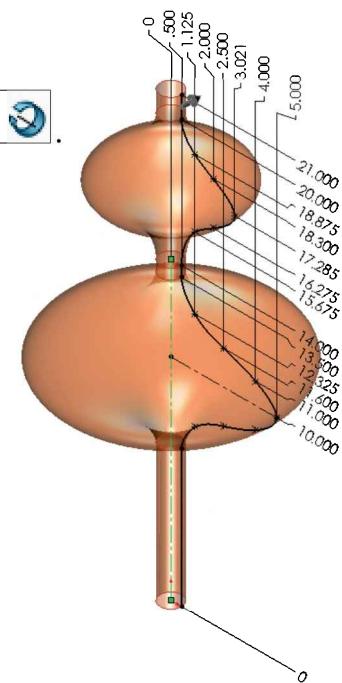
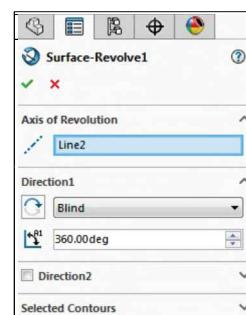


2. Revolving a surface:

Click **Revolve Surface**

Use **Blind** and **360 deg**.

Click **OK**.



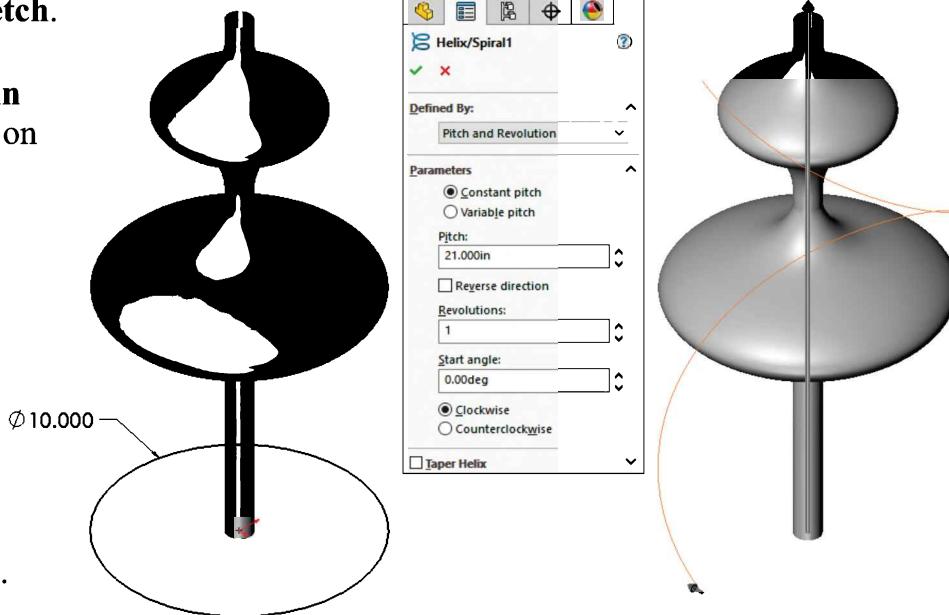
3. Creating a Helix:

Select the **Top** plane and open a **new sketch**.

Sketch a **10.00in** circle centered on the origin.

Convert the circle into a Helix using the settings shown in the dialog box.

Exit the Sketch.



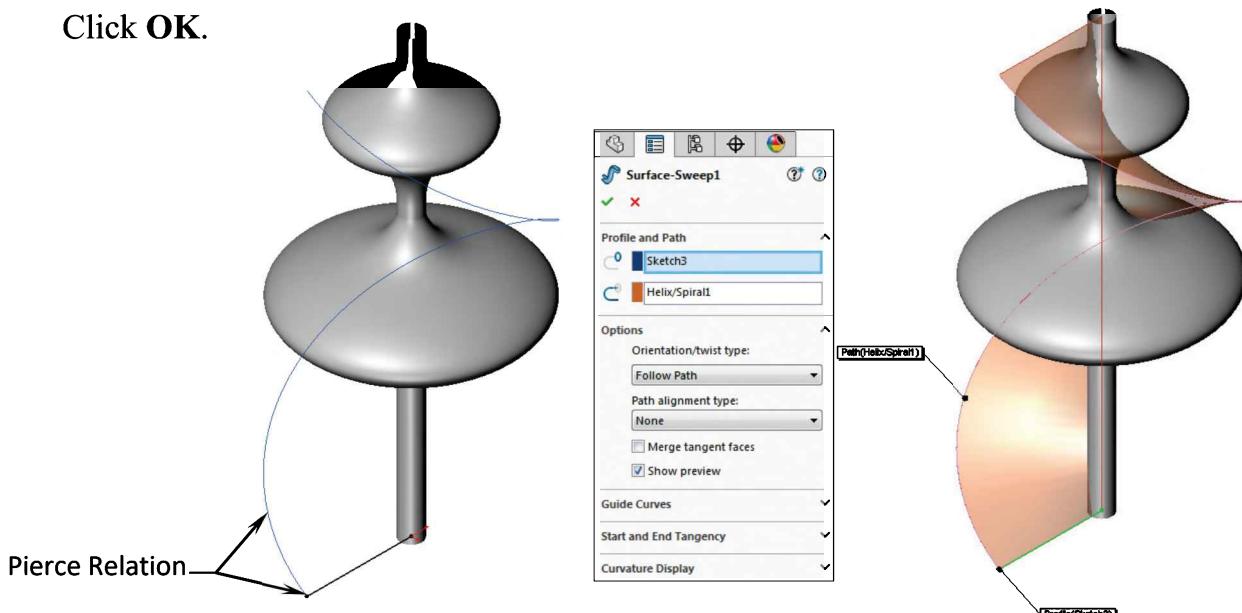
4. Creating a swept surface:

Select the **Top** plane once again and open a **new sketch**.

Sketch a **Line** from the Origin and **Pierce** the other end of the line to the Helix.

Click and sweep the Line along the Helix using the **Swept-Surface** option.

Click **OK**.



5. Create a new Axis: (to be used in step 9)

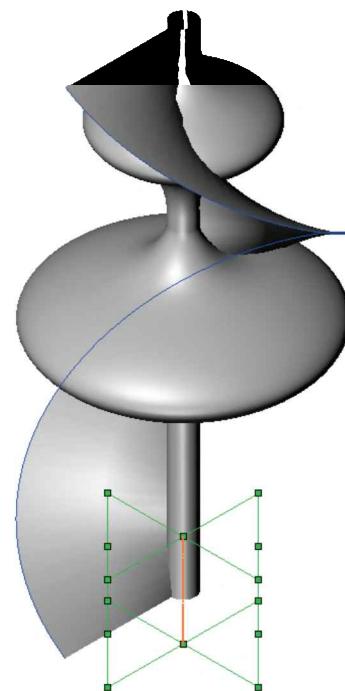
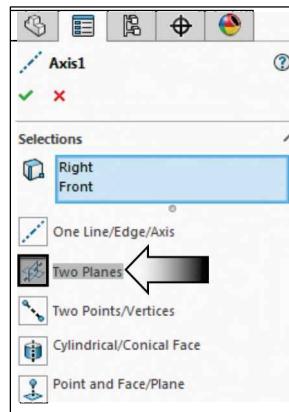
Select **Insert, Reference Geometry, Axis** .

Click the **Two-Planes** option.

Select the **Front** and the **Right** planes from the FeatureManager tree.

A preview of the new axis appears in the center of the part.

Click **OK**.



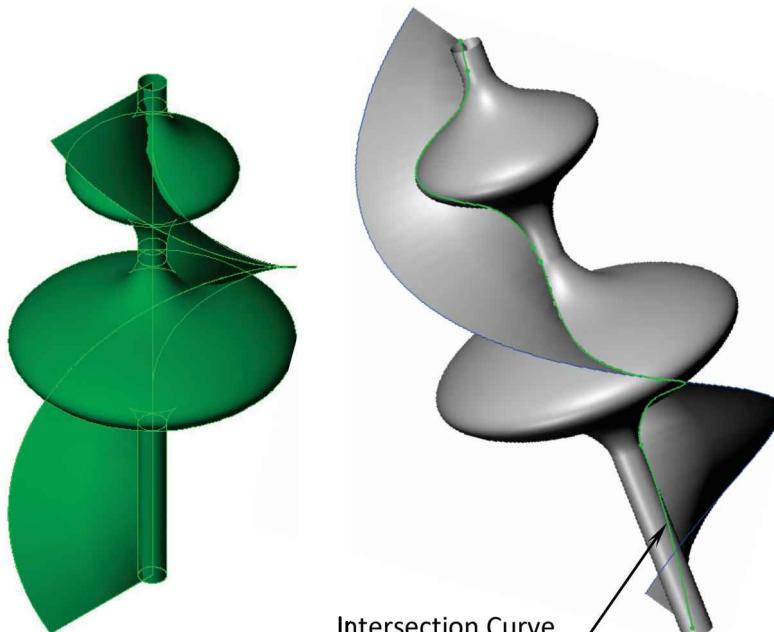
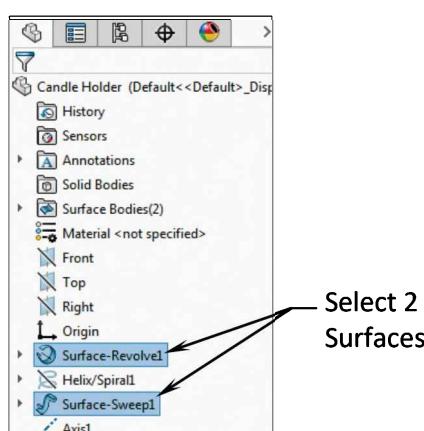
6. Create the Intersection Curve:

Hold the **Control** key and select the **Surface-Revolve1** and the **Surface Sweep1** from the FeatureManager tree.

Click  or select **Tools / Sketch Tools / Intersection Curve**.

A 3D-Sketch is created from the intersection of the two surfaces.

Exit the 3D Sketch or press Control+Q.

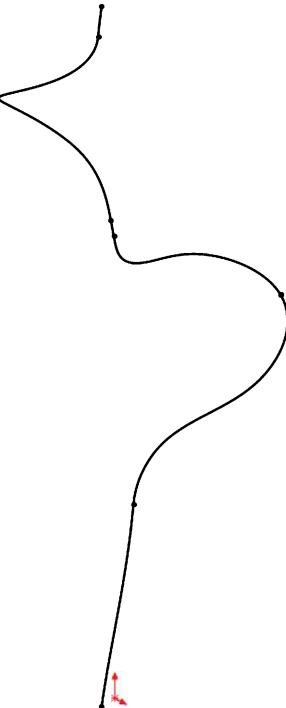
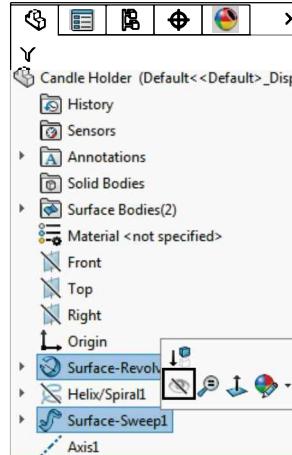


7. Hide the 2 Surface Bodies:

From the FeatureManager tree, right-click each Surface and select **Hide**.

This 3D sketch will be used as the sweep path in the next step.

Click **OK**.

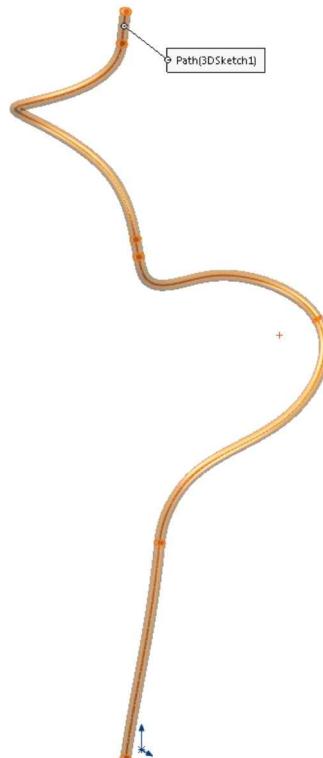
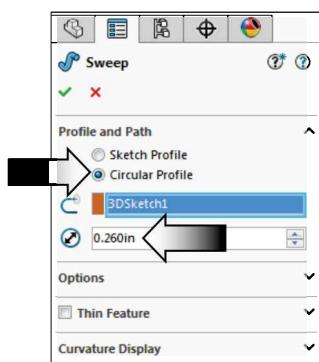


8. Create a solid swept feature:

Switch to the **Features** tool tab and click or select **Insert / Boss-Base/ Sweep**.

Select the **Circular Profile** option and enter **.260in** for Profile Diameter (arrow).

Select the **3D-Sketch** for Sweep Path.

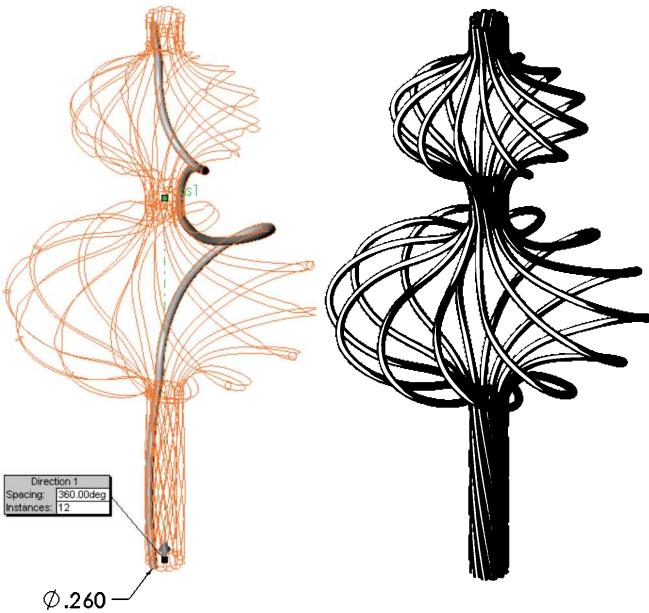
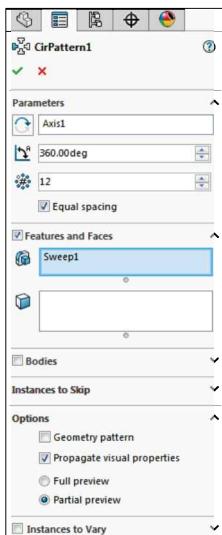


Click **OK**.

9. Create the Circular Pattern :

Using the **Axis** created in step 5 as the Center of the Pattern, replicate the Swept Feature 12 times.

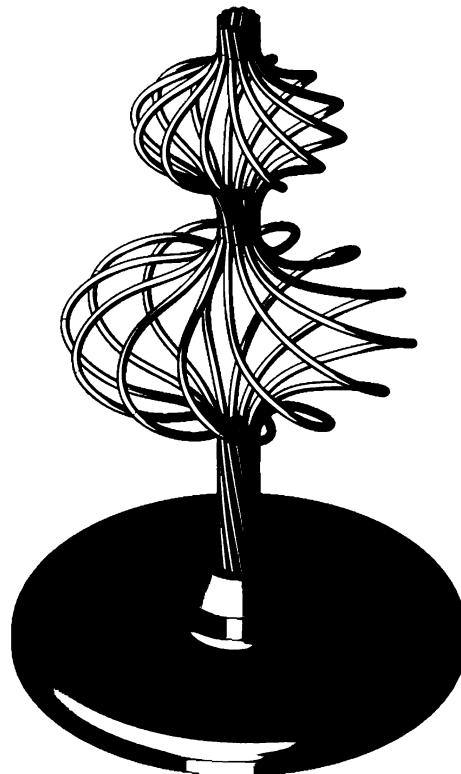
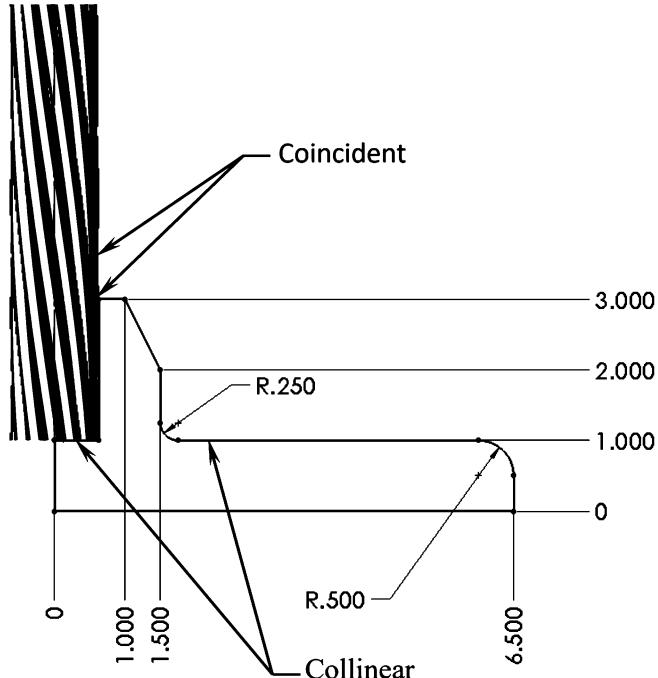
Click **OK**.

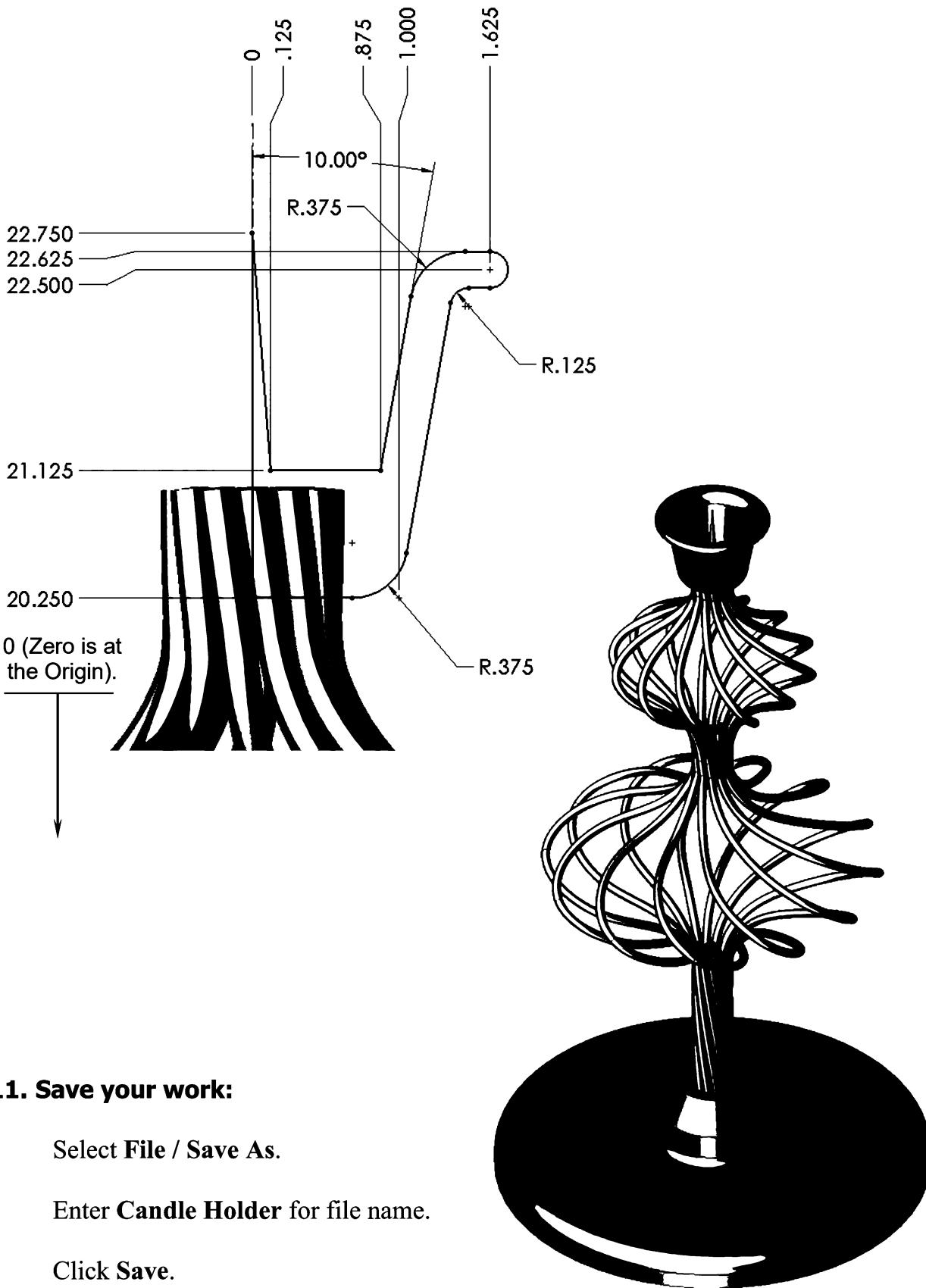


10. OPTIONAL:

Create the **Base** and the **Candle Holder** solid features as shown below.

Modify or design your own shapes if needed.



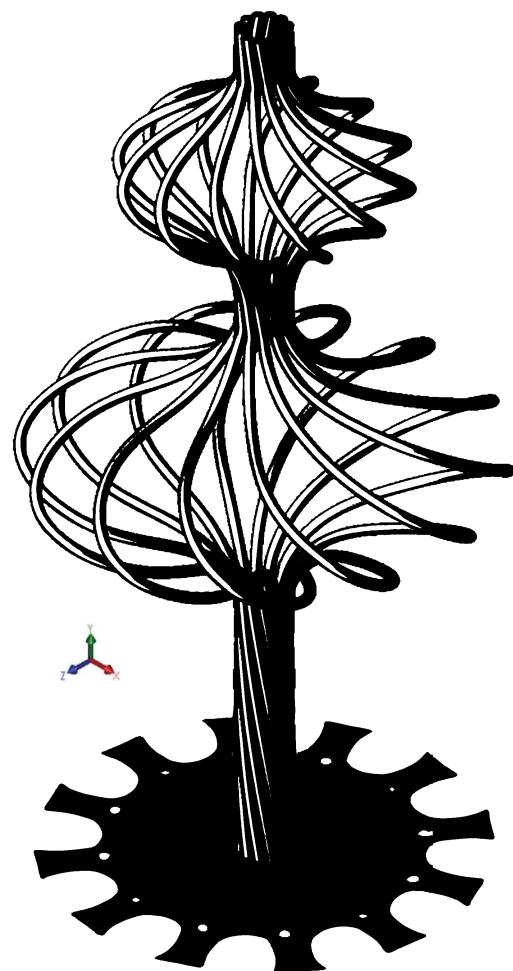
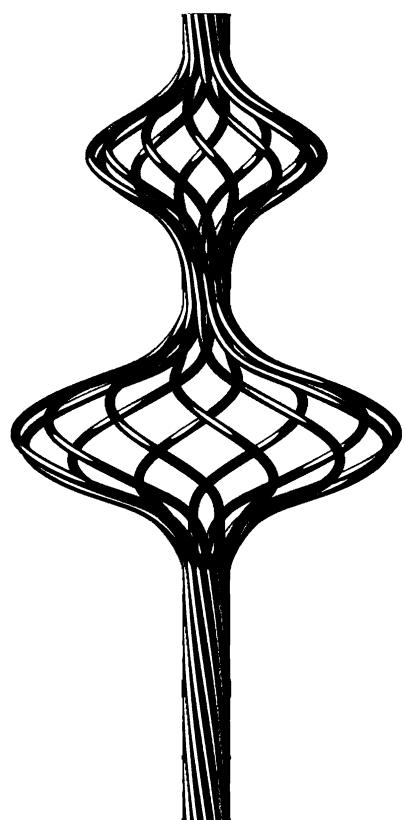
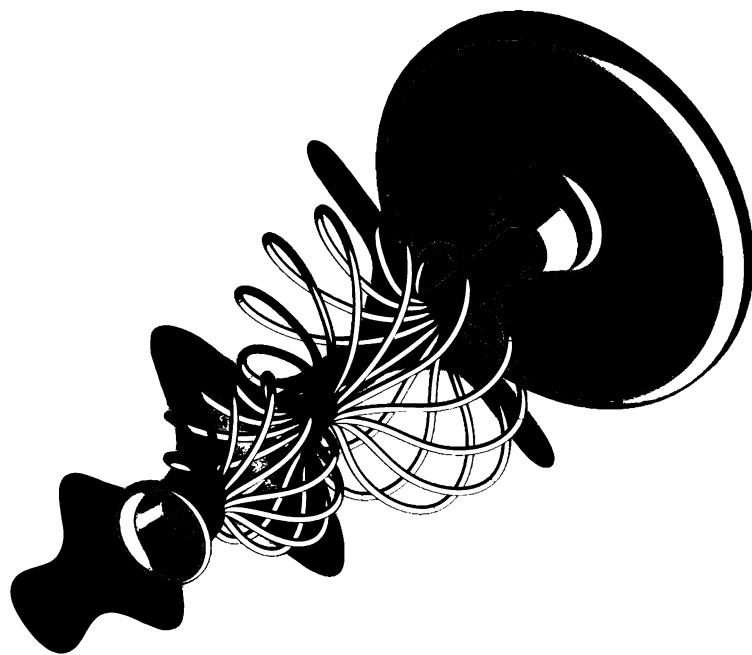
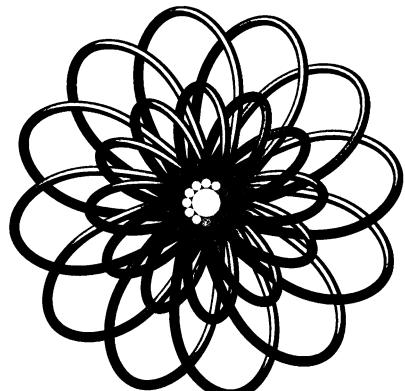


11. Save your work:

Select **File / Save As**.

Enter **Candle Holder** for file name.

Click **Save**.



Final Exam

Create the part **Bottle** using the LOFT and SWEEP options where noted.

All sketch profiles must be fully defined.

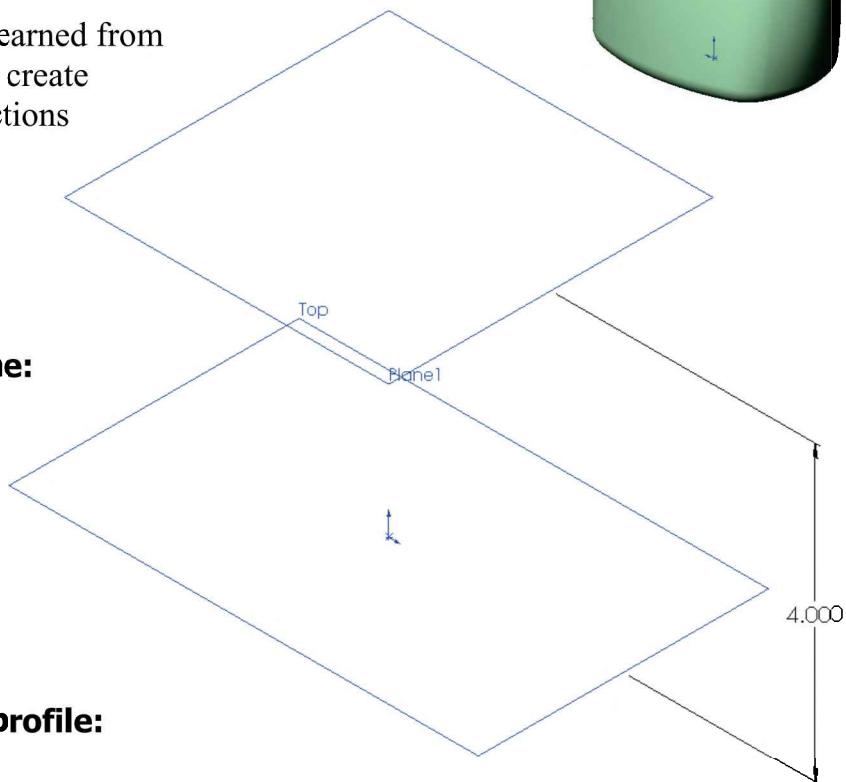
The part must have no errors when finished.

Apply what you have learned from the previous lessons to create this model. The instructions on the following pages are for reference only.



1. Creating an Offset plane:

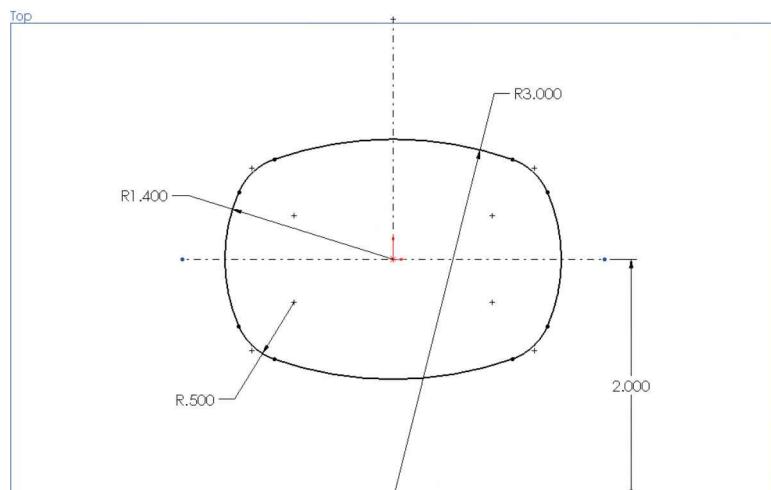
Use the Top reference Plane to create a new plane at **4.000in.** offset distance.



2. Sketching the bottom profile:

Select the Top plane and sketch the profile as shown.

Use the Mirror option to create the **Symmetric** relations between entities.



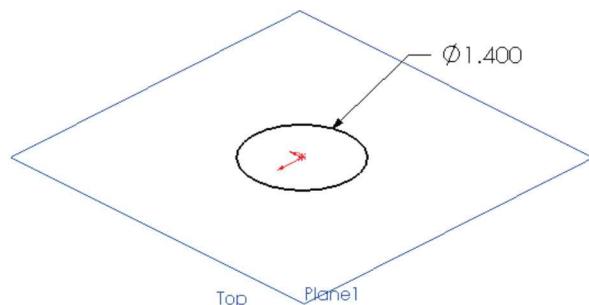
Exit the Sketch.

3. Creating the top profile:

Use the Plane1 and sketch a **Circle** centered on Origin.

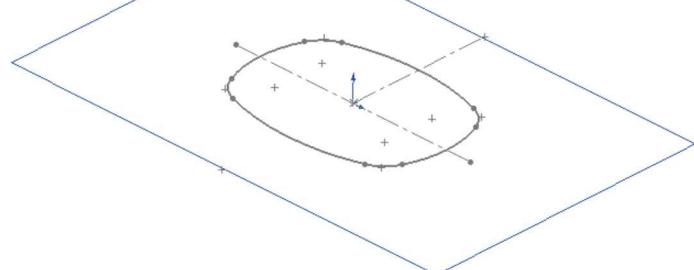
Add a diameter dimension to fully define the sketch.

Exit the sketch.

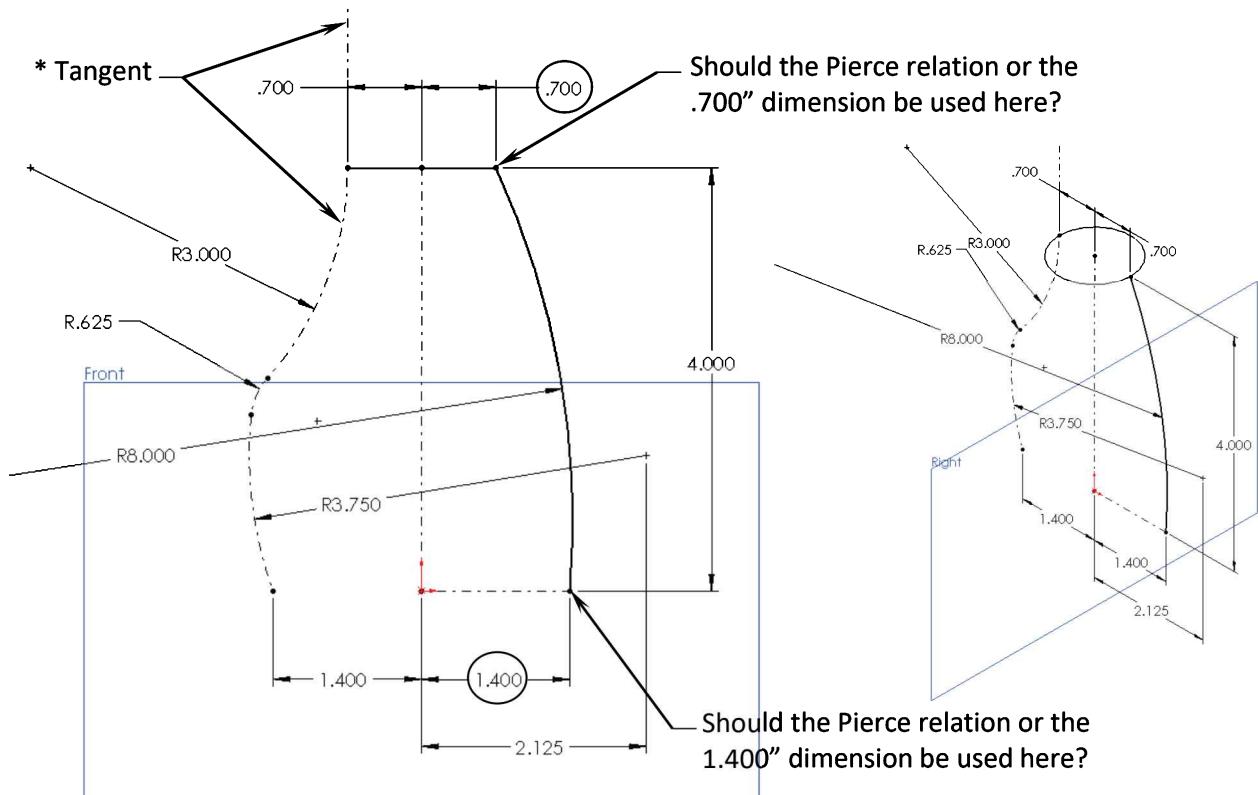


4. Creating the 1st Guide Curve:

Select the Front plane and sketch the profile shown below.

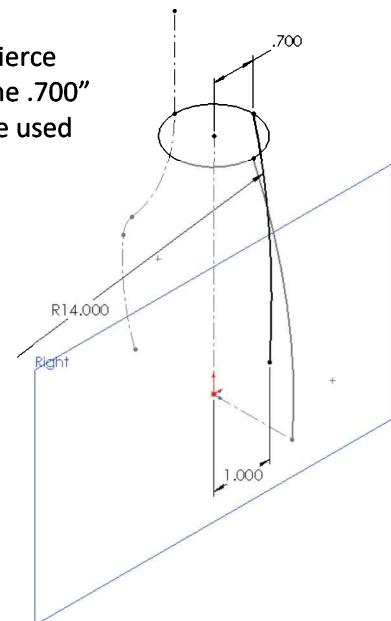
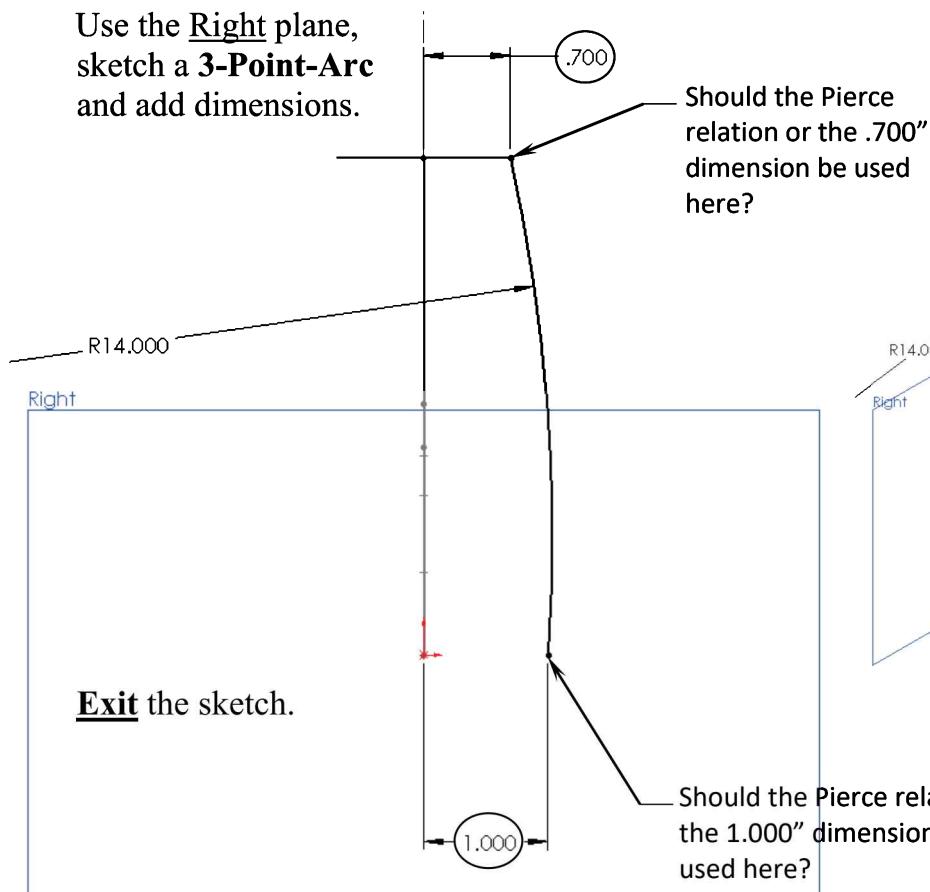


* *Note:* The construction lines will be used later to create the Derived-Sketch and the Guide Curves.



5. Creating the 2nd Guide Curve:

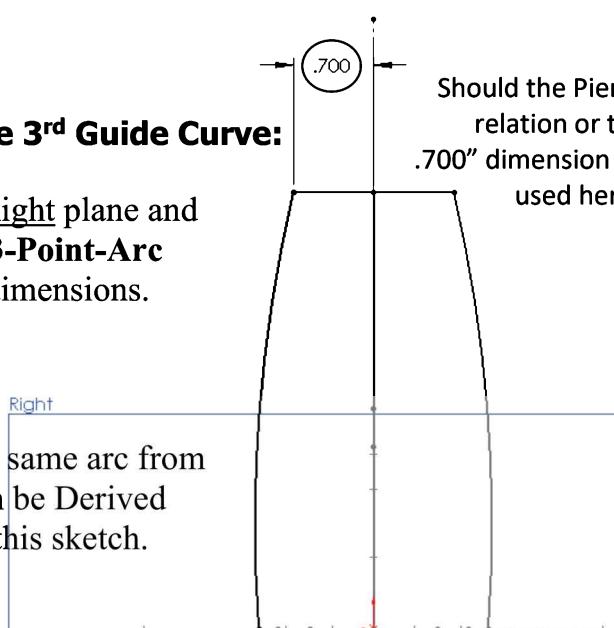
Use the Right plane, sketch a **3-Point-Arc** and add dimensions.



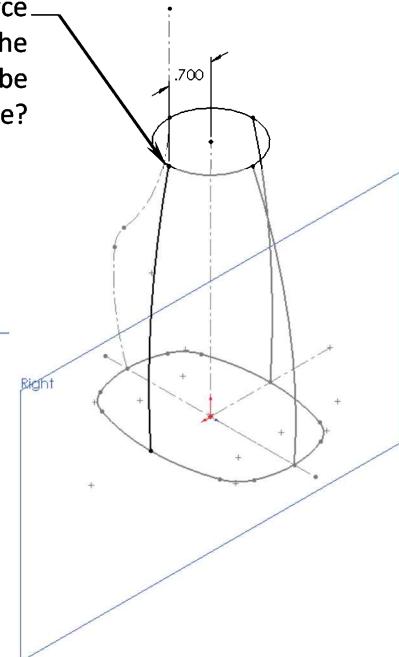
6. Creating the 3rd Guide Curve:

Use the Right plane and sketch a **3-Point-Arc** and add dimensions.

Hint: The same arc from step 5 can be Derived to create this sketch.



Exit the sketch.

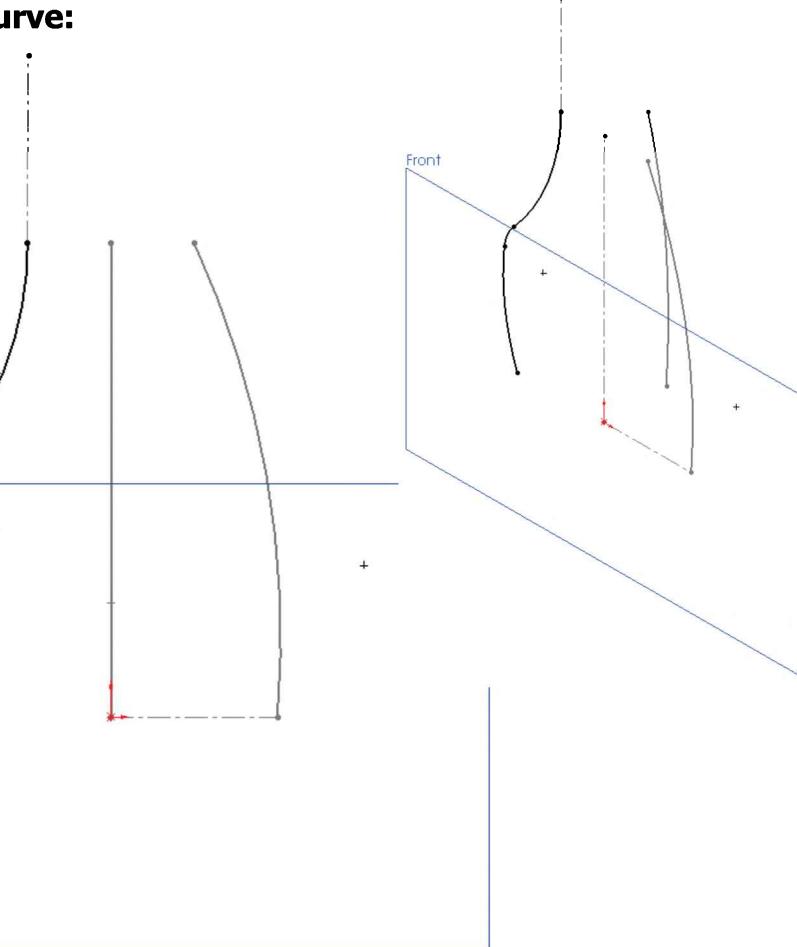


7. Creating the 4th Guide Curve:

Select the **Front** plane, open a **new sketch** and **convert** the **3 entities** as noted.

Convert entities

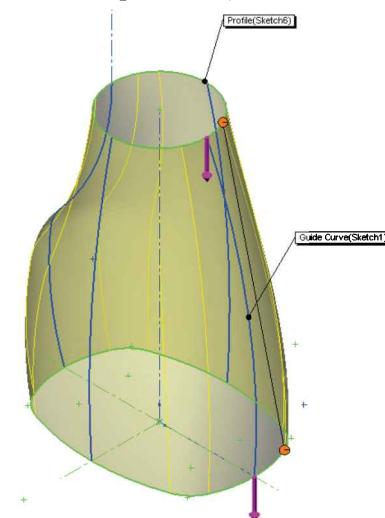
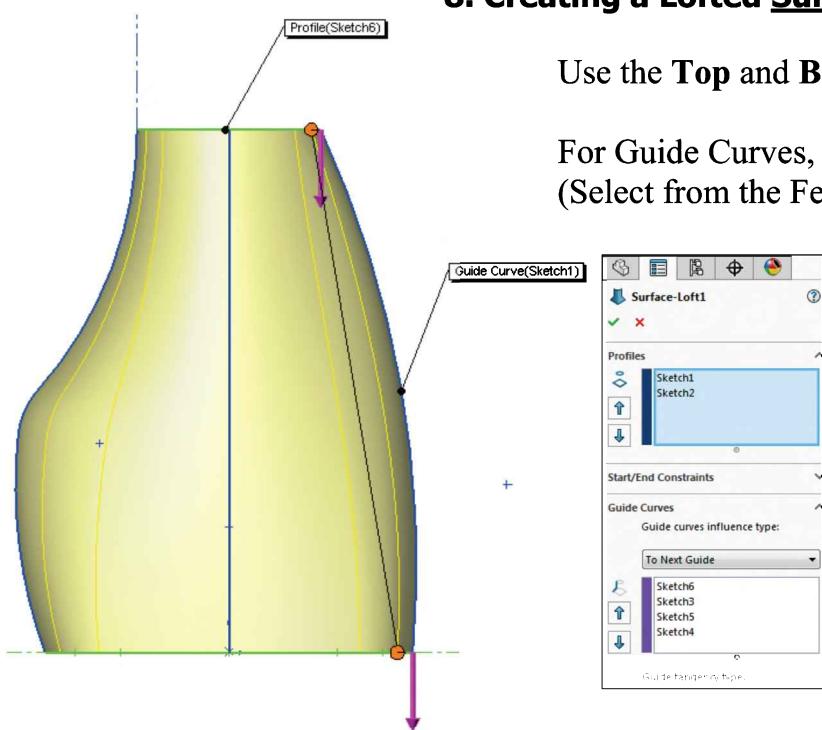
Exit the sketch.



8. Creating a Lofted Surface:

Use the **Top** and **Bottom** sketches as Loft Profiles.

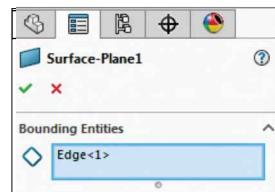
For Guide Curves, select the next **4 sketches**.
(Select from the FeatureManager tree.)



9. Filling the bottom surface:

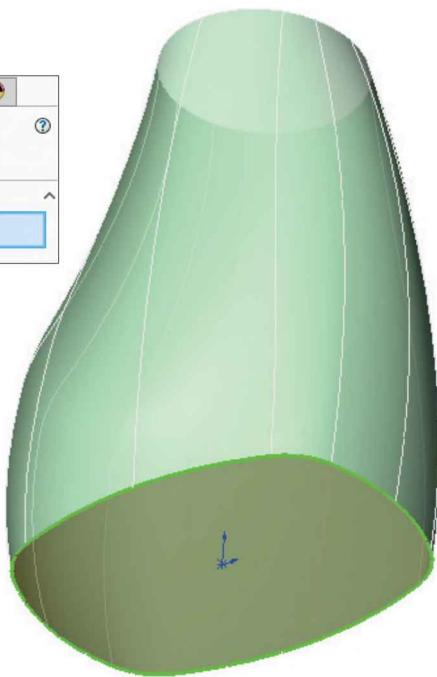
Select Insert / Surface / Planar.

Select **all edges** at the bottom for this operation.



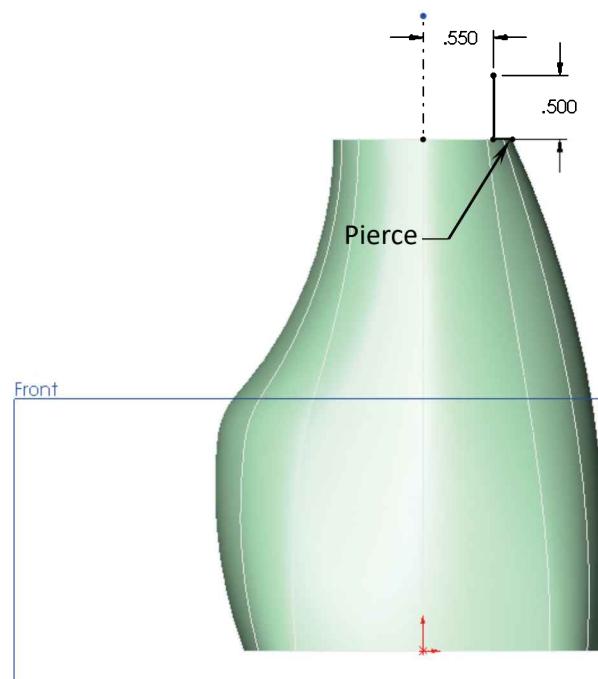
When finished, the bottom surface should be completely covered.

Click **OK**.



10. Sketching the Neck profile:

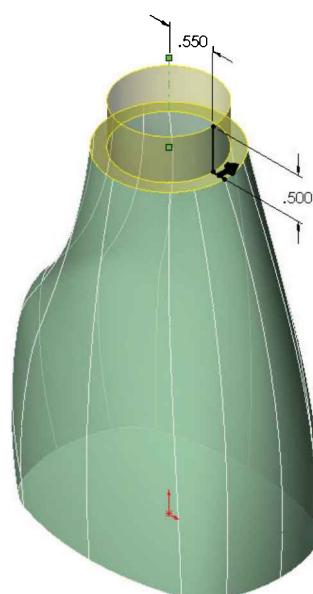
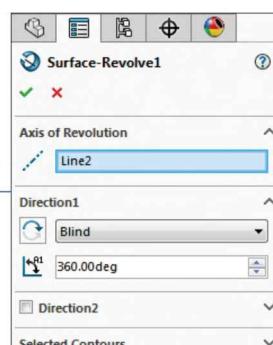
Select the Front plane and sketch the profile below (2 lines).



Revolve the sketch profile as a Surface.

Revolve **One Direction**.

Revolve a complete **360°**.



11. Knitting all surfaces into one:

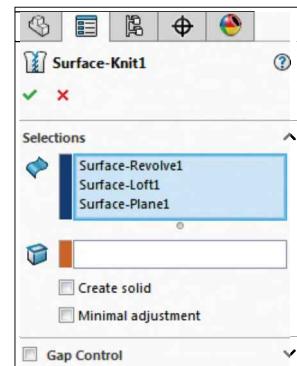
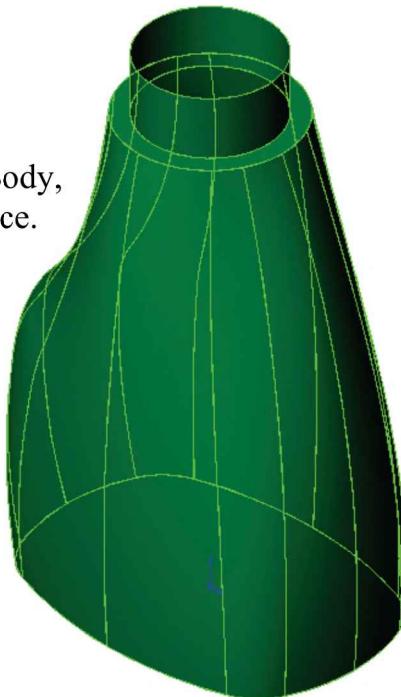
Select Insert / Surface / Knit.

Select all **three surfaces**: the Body, the Neck, and the Bottom surface.

Clear the **Gap Control** option.

Click **OK**.

When finished, all 3 surfaces should become a single surface body.

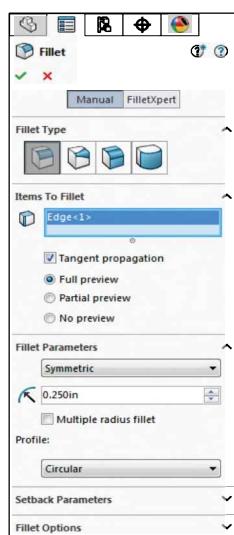
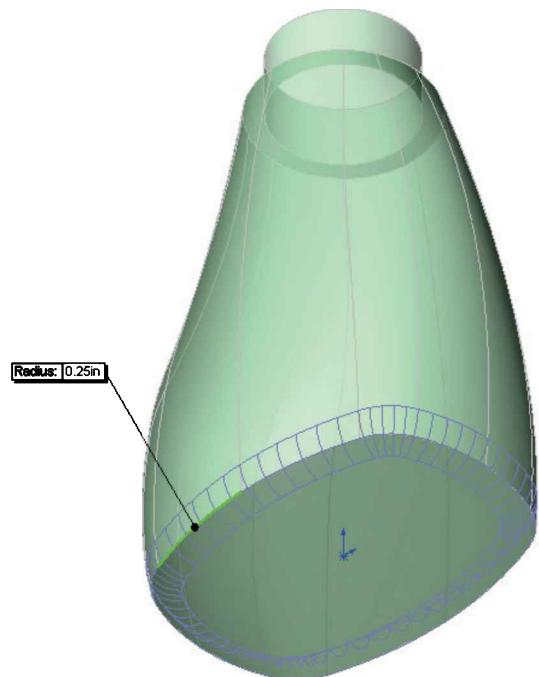


12. Adding fillet to the bottom edges:

Add a **.250in.** fillet to the bottom edges as shown.

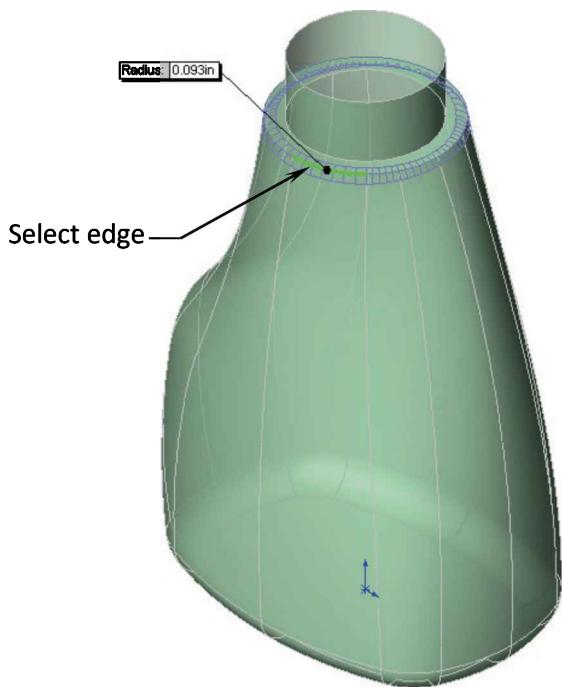
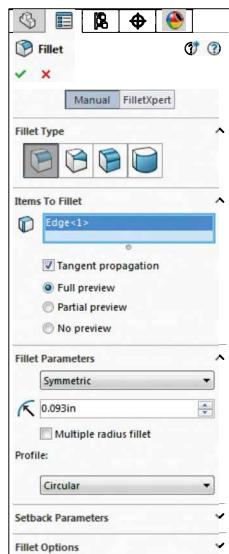
Tips:

Right click one of the edges and pick: Select Tangency; it is one of the faster ways to select all edges at the same time.



13. Adding fillet to the upper area:

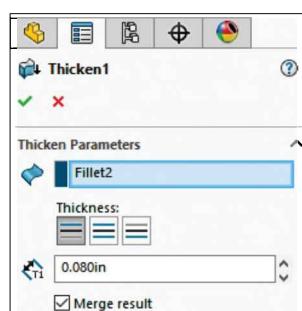
Add a **.093in.** fillet to the upper edge as indicated.



14. Thickening the surface body:

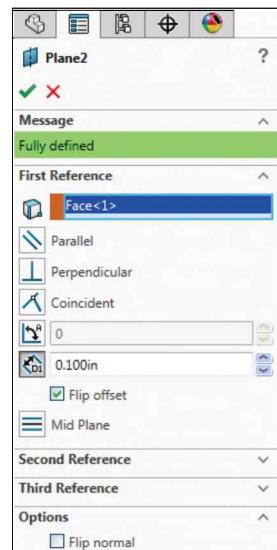
Select **Insert / Boss-Base / Thicken**.

Enter a wall thickness of **.080 in.** to the **inside** of the bottle.



15. Creating a new Offset plane:

Create a new plane that is **.100in.** offset from the Top surface of the bottle. Place the new plane below the face.



16. Creating the Sweep Path of the Thread (the Helix):

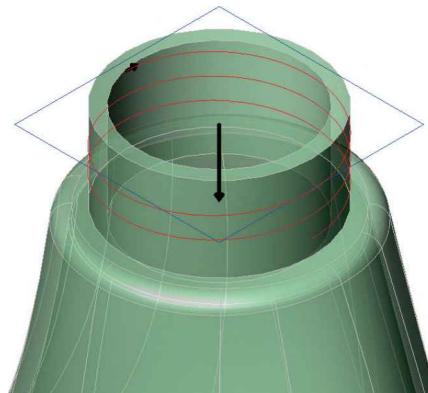
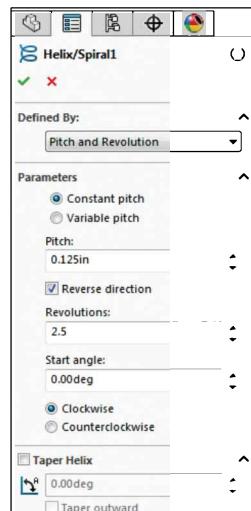
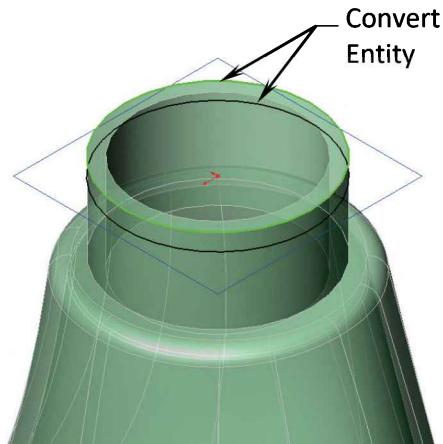
Convert the upper **circular edge** and then click **Insert / Curves / Helix-Spiral**.

Pitch = .125 in.

Revolution = 2.5.

Starting Angle = 0 deg.

Click OK.

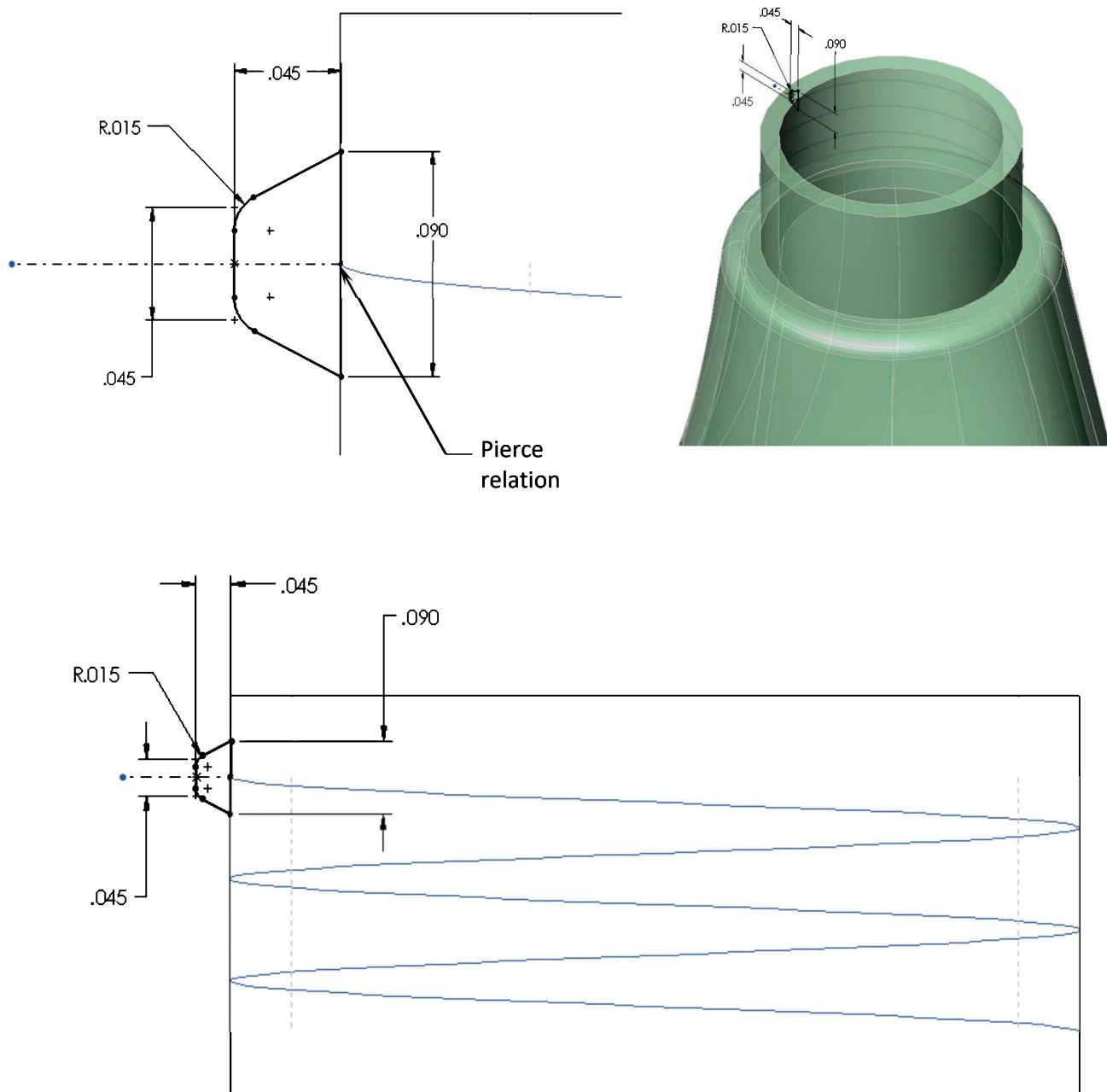


17. Creating the Sweep Profile of the Thread:

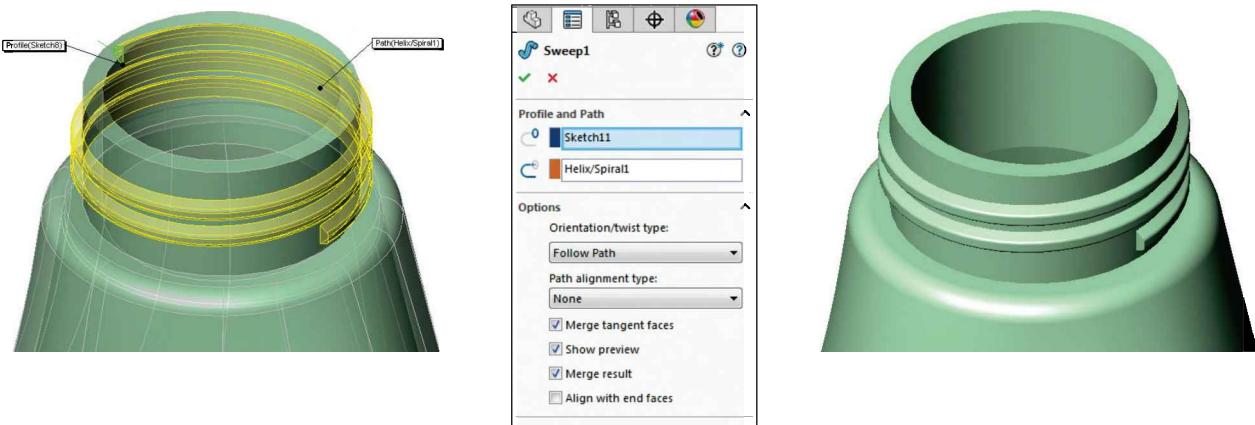
Select the Right plane and sketch the thread profile as shown below.

Add dimensions and relations needed to fully define the sketch.

Add the **R.015"** sketch fillets after the sketch is fully defined.



Exit the sketch.

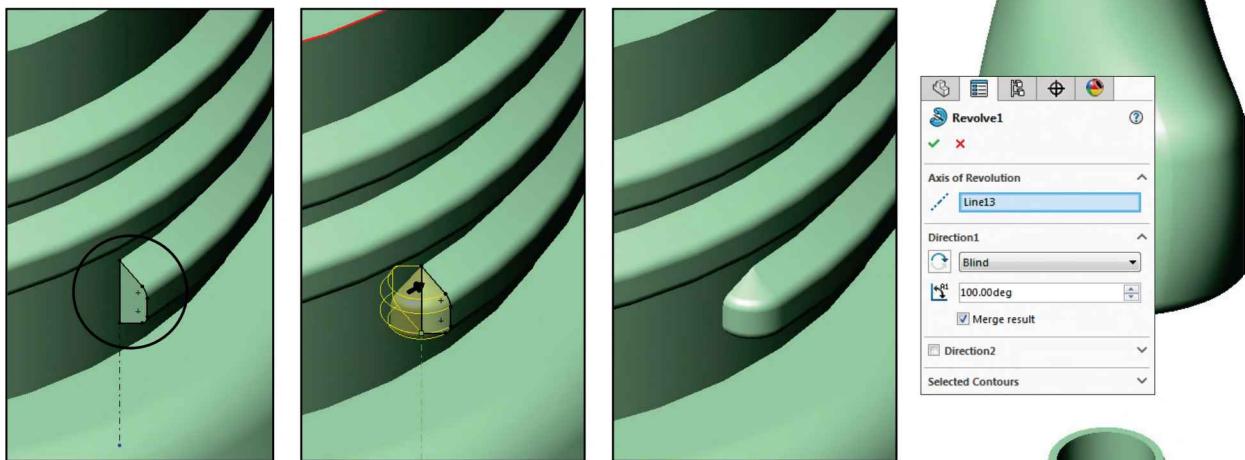


18. Sweeping:

Sweep the thread profile along the helix to create the external threads.

19. Revolving:

Convert the faces at the end of the thread and revolve them about the vertical centerlines to round off the two ends.



20. Applying dimension changes:

Change the dimension **R1.400** in Sketch1 to **R1.500**.

Change the **$\varnothing 1.400$** in Sketch2 to **$\varnothing 1.500$** .

Repair any errors caused by the changes.

21. Saving your work:

Save your work as: **Level 3 – Final Exam**.



Designed by a CSWE student



Designed by a CSWE student



Designed by a CSWE student



Designed by a CSWE student

CHAPTER 12

SimulationXpress

SimulationXpress Using the Analysis Wizard



SimulationXpress is a design analysis technology that allows the SOLIDWORKS users to perform first-pass stress analysis. SimulationXpress can help you reduce cost and time-to-market by testing your 3D designs within the SOLIDWORKS program, instead of expensive and time-consuming field tests.

There are five basic steps to complete the analysis using SimulationXpress:

1. Apply restraints (Fixture)

Users can define restraints. Each restraint can contain multiple faces. The restrained faces are constrained in all directions due to rigid body motion; you must at least restrain one face of the part to avoid analysis failure.

2. Apply loads

User inputs force and pressure loads to the faces of the model.

3. Define material of the part

- * EX (Modulus of elasticity).
- * NUXY (Poisson's ratio). If users do not define NUXY, SimulationXpress assumes a value of 0.
- * SIGYLD (Yield Strength). Used only to calculate the factors of safety (FOS).
- * DENS (Mass density). Used only to include mass properties of the part in the report file.

4. Analyze the part

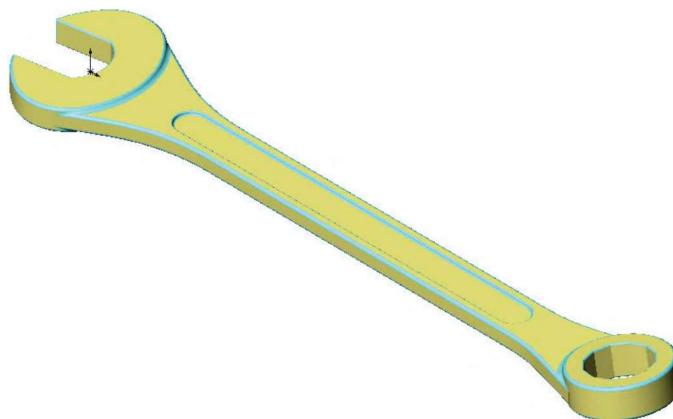
SimulationXpress prepares the model for analysis, then calculates displacements, strains, and stresses.

5. View the results

After completing the analysis, users can view results. A check mark on the Results tab indicates that results exist and are available to view for the current geometry, material, restraints, and loads. A report can also be created in MS-Word format.

SimulationXpress

Using the Analysis Wizard



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI

Units: INCHES – 3 Decimals

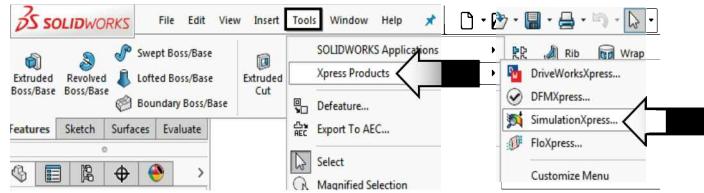
Tools Needed:

SimulationXpress is part of SOLIDWORKS Basic, SOLIDWORKS Office Professional, and SOLIDWORKS Premium.



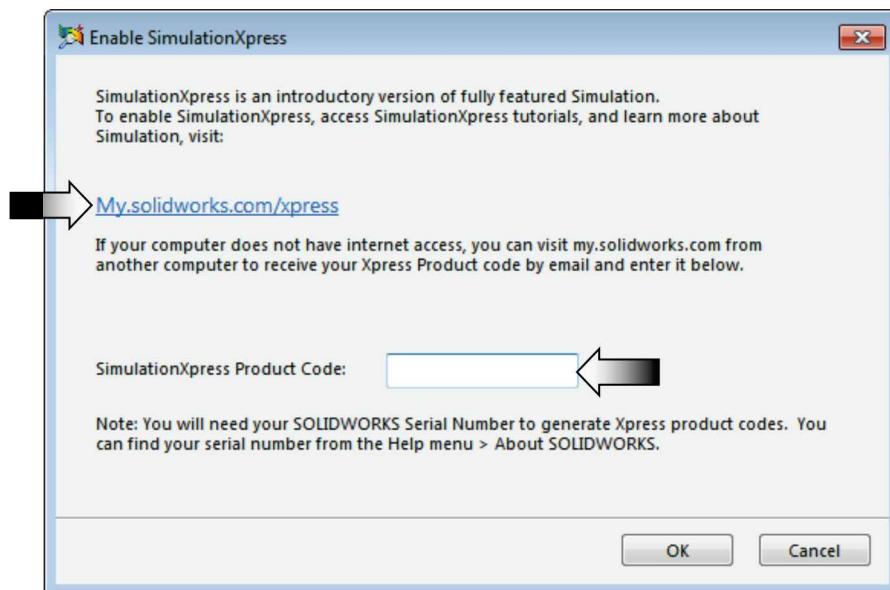
1. Starting SimulationXpress:

SimulationXpress is an introductory version of fully featured simulation.

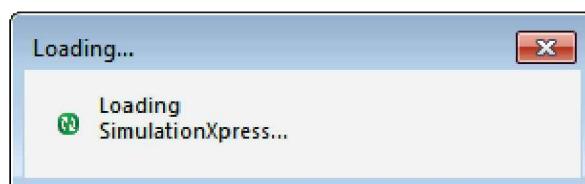


SimulationXpress is a design analysis application that is fully integrated with SOLIDWORKS. It is used by designers, analysts, engineers, students, and others worldwide to design safe, efficient, and economical products.

To enable SimulationXpress, visit: www.my.solidworks.com/xpress, log in or create a user account and enter your SOLIDWORKS Serial Number to generate the **Xpress Product Codes**. You can find your serial number from the **Help (?)** menu **About SOLIDWORKS**.



Select: **Tools / Xpress Products / SimulationXpress**. After entering the Xpress Product Code, click **OK** to launch the SimulationXpress application.



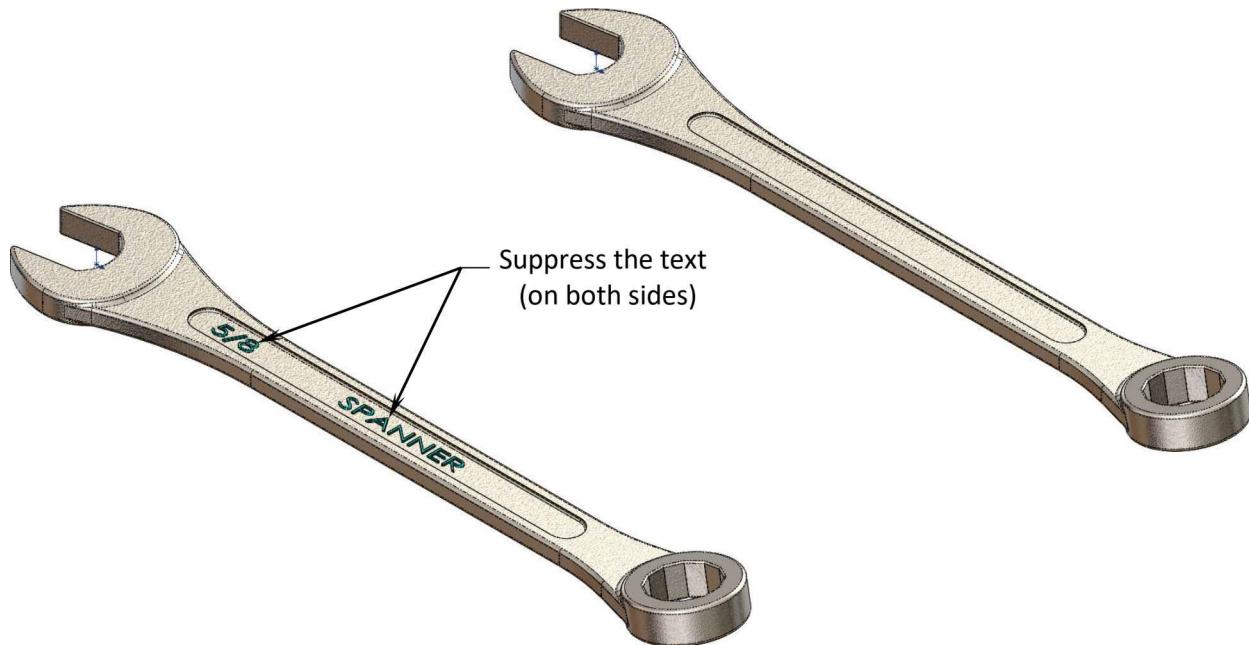
The SimulationXpress application is launched and appears on the right side of the screen.

2. Opening a part document:

Open a part document that was created earlier: **Spanner** (or open a copy from the Training Files folder).

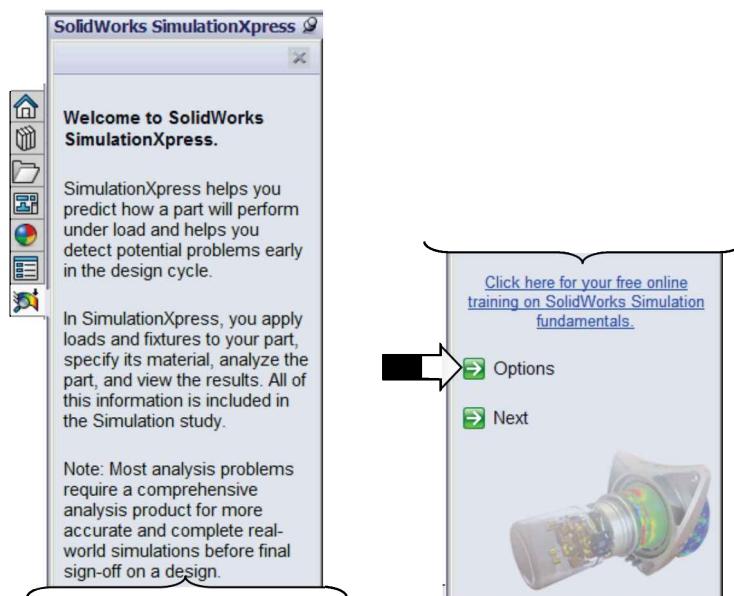
SUPPRESS the extruded text on both sides (the **5/8"** and the **Spanner** text).

From the **Tools** drop-down menu, select **SimulationXpress** (Arrow).



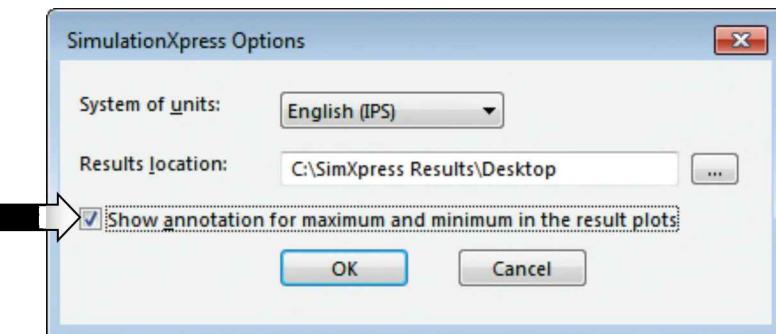
3. Setting up the Units:

Click **Options** (arrow) to set the system of units for the analysis.



Select **English (IPS)** for System of Units (Inch, Pound, Second).

Select the folder and the location to save the analysis results.



Enable the option:

Show Annotation for Maximum and Minimum in the Result Plot.

Creating a new folder for each study is recommended.

Click **OK** .

Click **Next** .

4. Adding a Fixture (restraint):

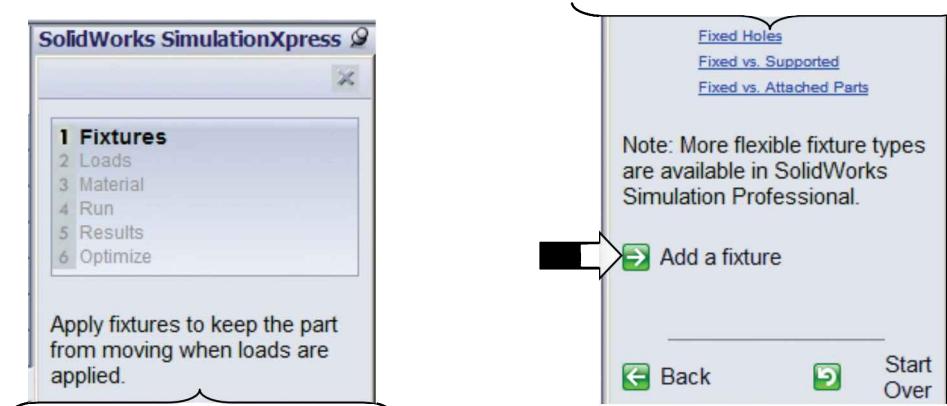
The next step is to create the restraint area(s).

Each restraint can contain one or multiple faces. The restrained faces are constrained in all directions. There must be at least one fixed face of the part to avoid analysis failure due to rigid body motion.

Restraints

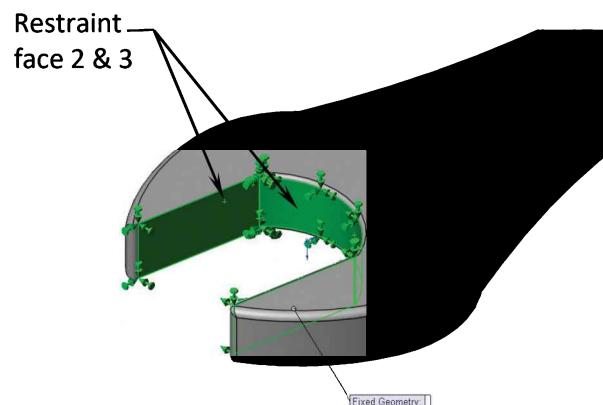
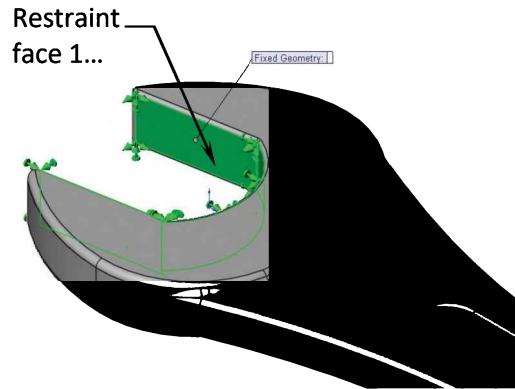
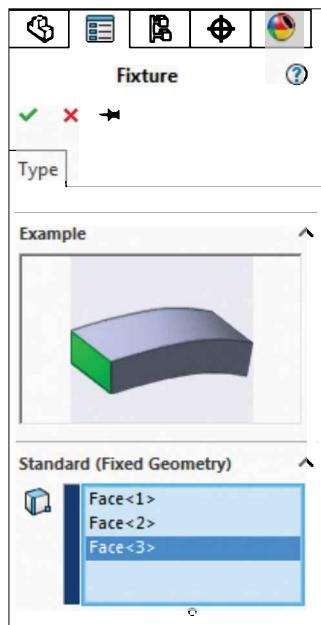
Restraint is used to anchor certain areas of the model so that they will not move or shift during the analysis. At least one face should be restrained prior to running the analysis.

Click **Add a Fixture** .



Select the 3 faces as indicated to use as restraint faces.

The Restraint faces are locked in all directions to avoid failure due to rigid body motion.

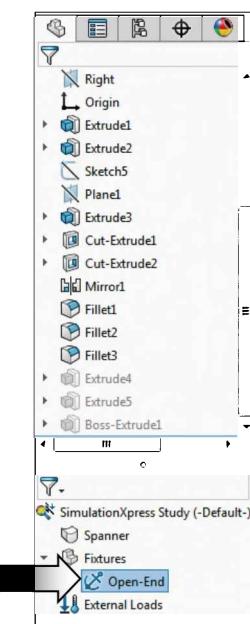


Click **OK**.

When the restraint faces are selected, more faces can be added to create different restraint sets. They can also be edited or deleted at any time.

The information regarding the settings and parameters for this study is recorded on the lower half of the FeatureManager tree (arrows).

Rename the fixture Fixed 1 to Open-End.

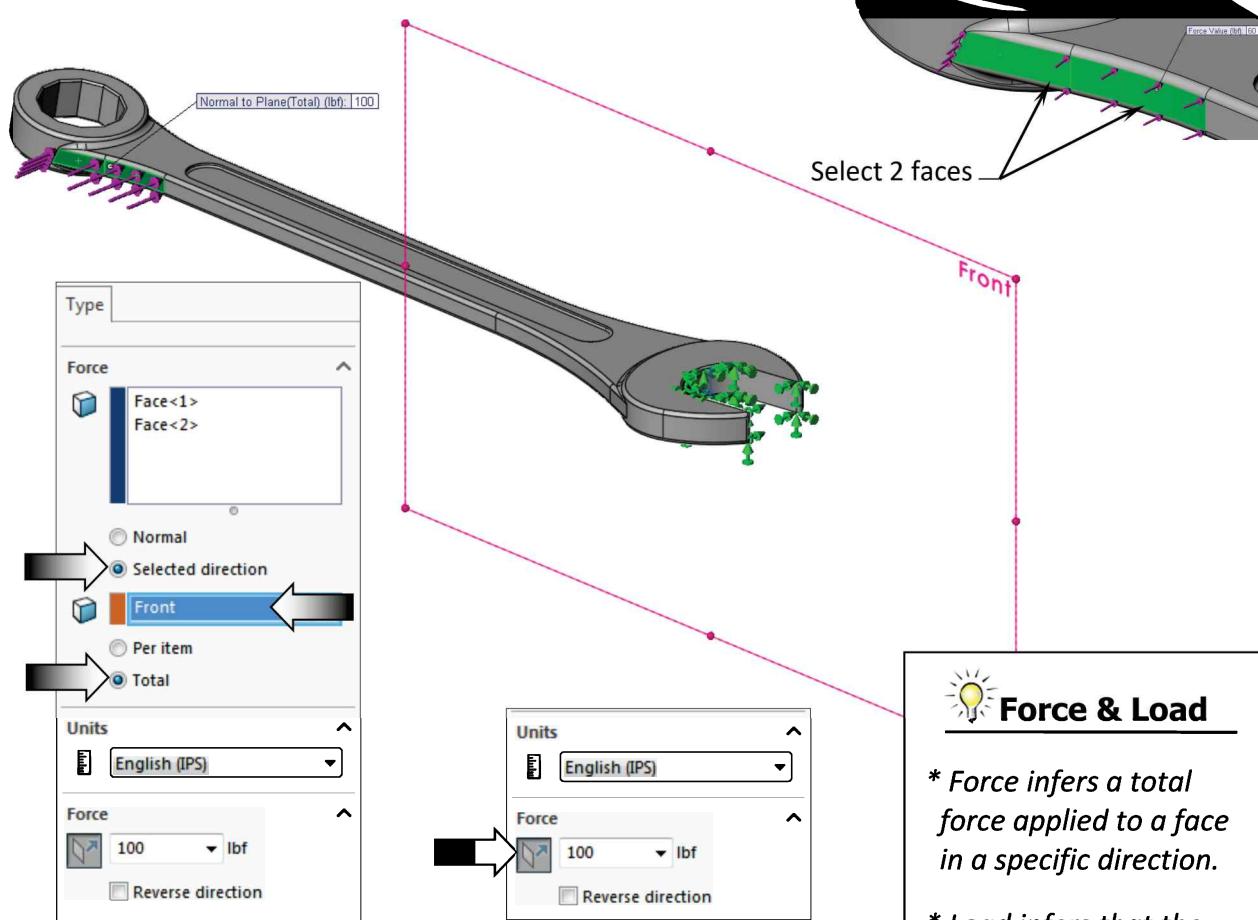


5. Applying a Force:

Click Add a Force →.

The **Forces** and **Pressures** options allow SOLIDWORKS users to apply force or pressure loads to faces of the model. Multiple forces can be applied to a single face or to multiple faces.

Select the 2 faces from the back side of the closed end as shown below.

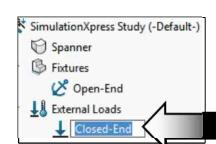


Click the **Selected Direction** checkbox and select the **Front** plane from FeatureManager tree (arrow).

Click the **Total** option (arrow). Enter **100** lbs. for Total Force value (arrow).

The direction arrows must be pointing towards the 2 faces.

Click **OK**. Rename Force-1 to **Closed-End**



We will need to specify a material at this point so that SimulationXpress can predict how the model will respond to the loads.

6. Selecting the material:

Click Next →.

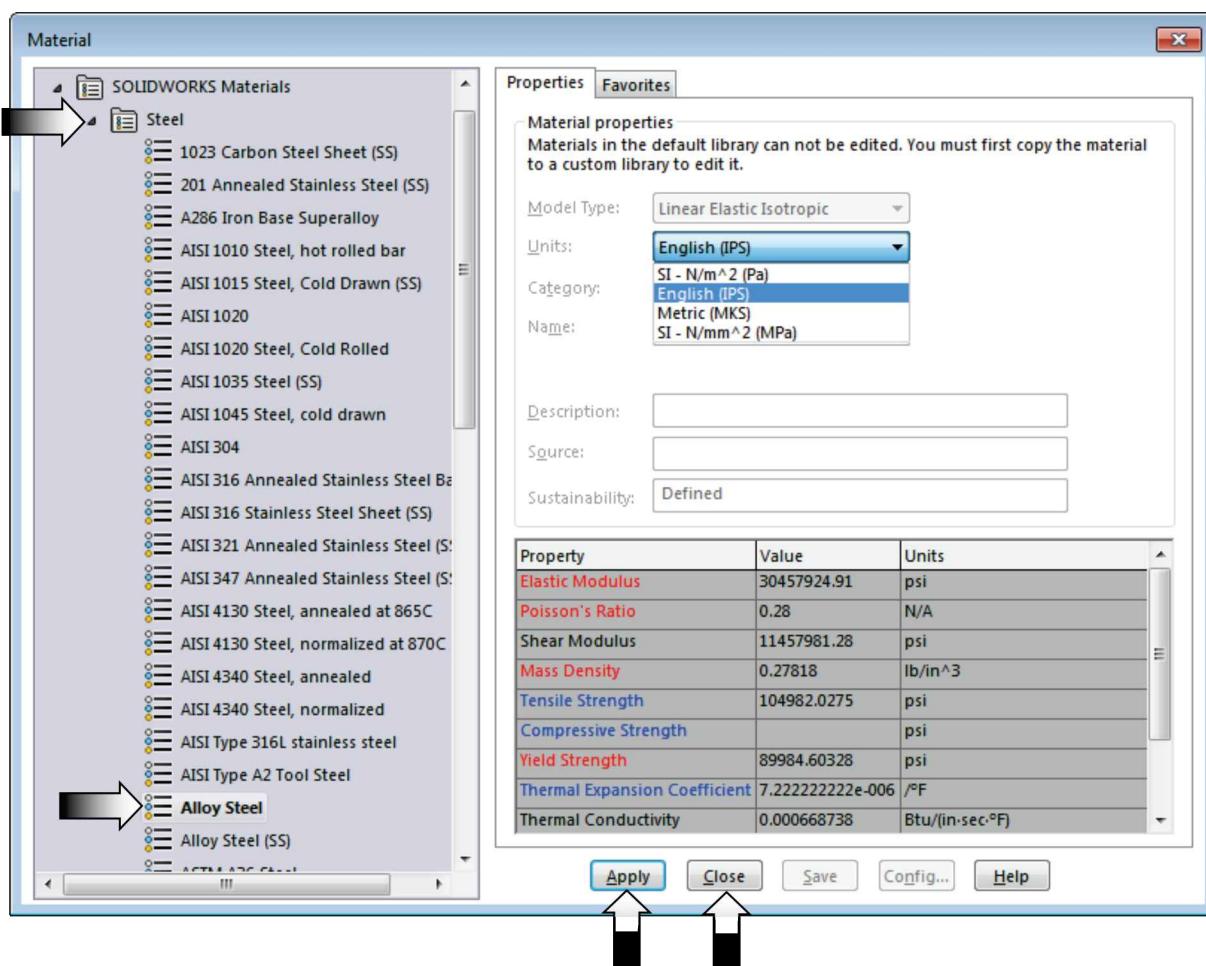
Select Choose Material →.

Expand the Steel folder.

Choose Alloy Steel from the list (arrow).

Material Editor

Material can be assigned to the part using the **Material Editor** PropertyManager. The material will then appear in SimulationXpress.



The analysis results are dependent upon the material selection. SimulationXpress needs to know the Visual and the Physical properties to run the analysis.

Click **Apply** [Apply] and **Close** [Close].

The material **Alloy Steel** is now assigned to the part. SimulationXpress assumes that the material deforms in a linear fashion with increased load. Non-Linear materials (such as many plastic materials) require the use of Simulation Premium.



The material assigned to this part is:

Alloy Steel

Young's Modulus:
3.04579e+007psi

Yield Strength:
89984.6psi

Change material

Next

Back

Start Over

The Modulus of Elasticity and the Yield Strength for the selected material are reported on the right side pane.

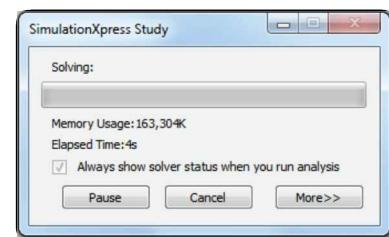
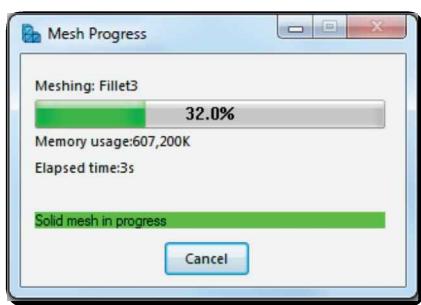
Click **Next** .

SimulationXpress is ready to analyze the model based on the information provided. Displacements, Strains, and Stresses will then be calculated.

7. Analyzing the model:

Click **Run Simulation** .

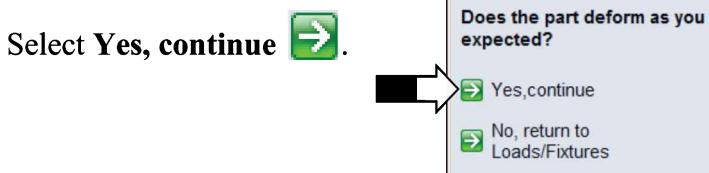
SimulationXpress automatically tries to mesh the model using the default element size. The smaller the element size the more accurate the results, but more time is needed to analyze the model.



To change mesh settings: click **Change settings** then click **Change mesh density**.

* Drag the slider to the right for a finer mesh (more accurate but takes longer).

* Drag the slider to the left for a coarser mesh (quicker).



8. Viewing the Results:

Click Show Von Mises Stress .

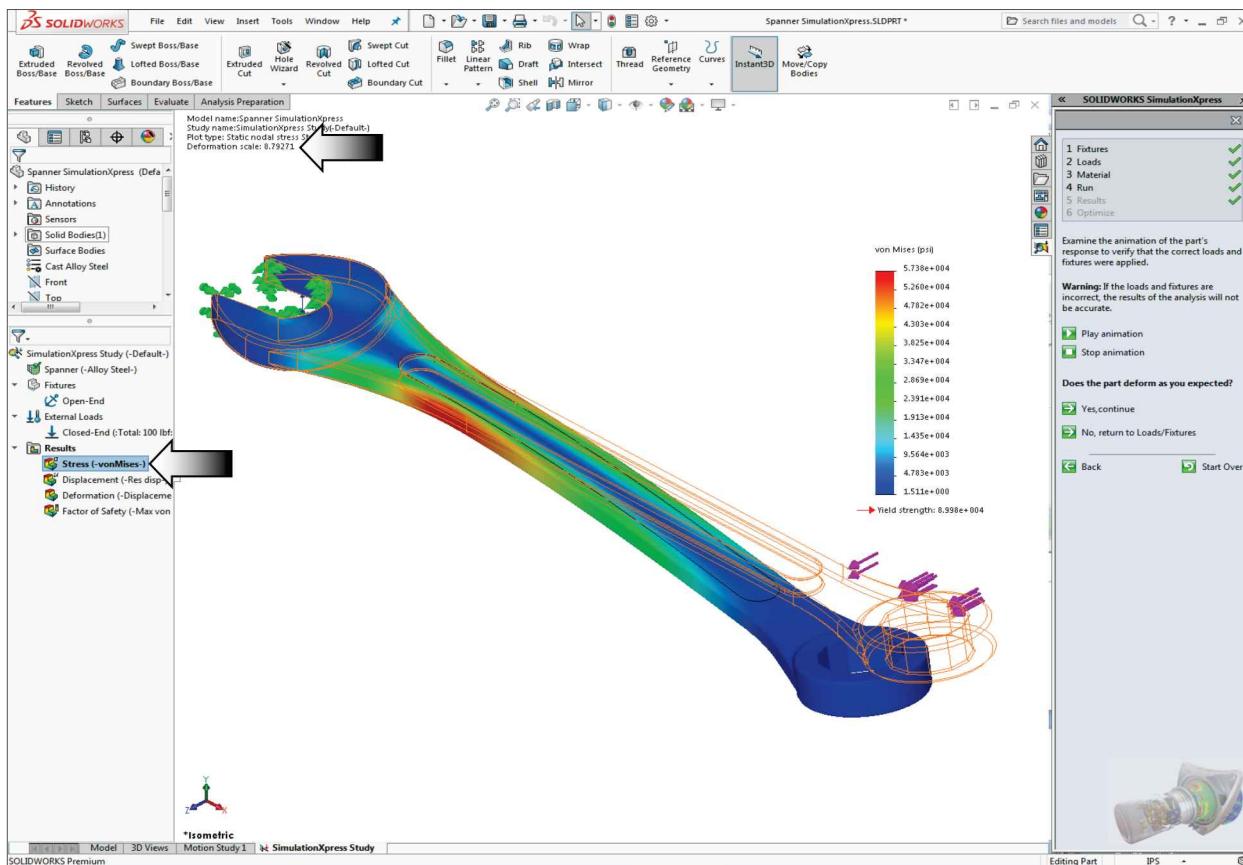
SimulationXpress plots stresses on the deformed shape of the part.

In most cases, the actual deformation is so small that the deformed shape almost coincides with the un-deformed shape, if plotted to scale.

SimulationXpress exaggerates the deformation to demonstrate it more clearly.

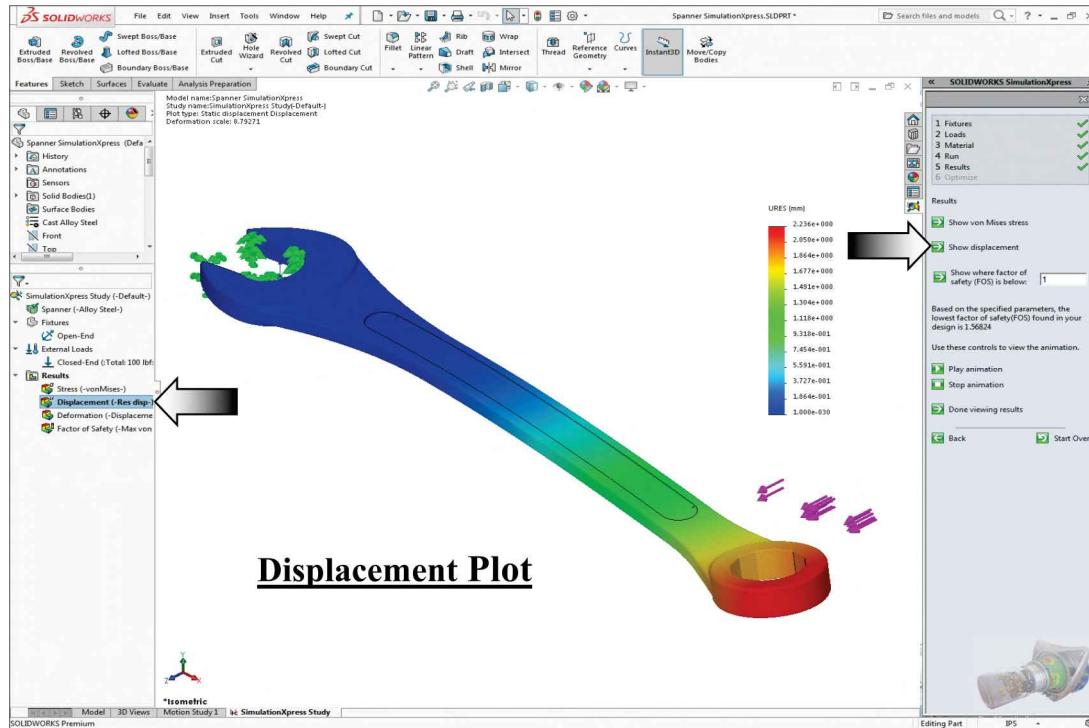
The Deformation Scale shown on the stress and deformed shape plots is the scale used to rescale the maximum deformation to **4.34794%** of the bounding box of the part.

The Stress Distribution Plot is displayed below.



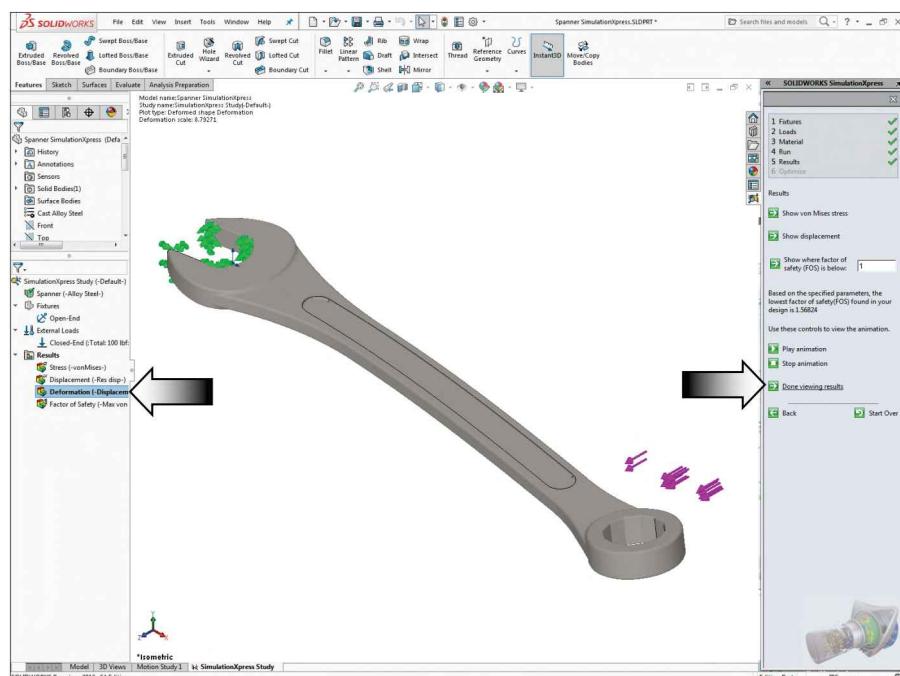
To see the resultant displacement plot click **Show Displacement** .

Click **Play Animation**  or **Stop Animation**  when finished viewing.



To view regions of the model with a factor of safety less than a given value (1), click: **Show Where Factor Of Safety (FOS) Is Below: 1** (or enter any value).

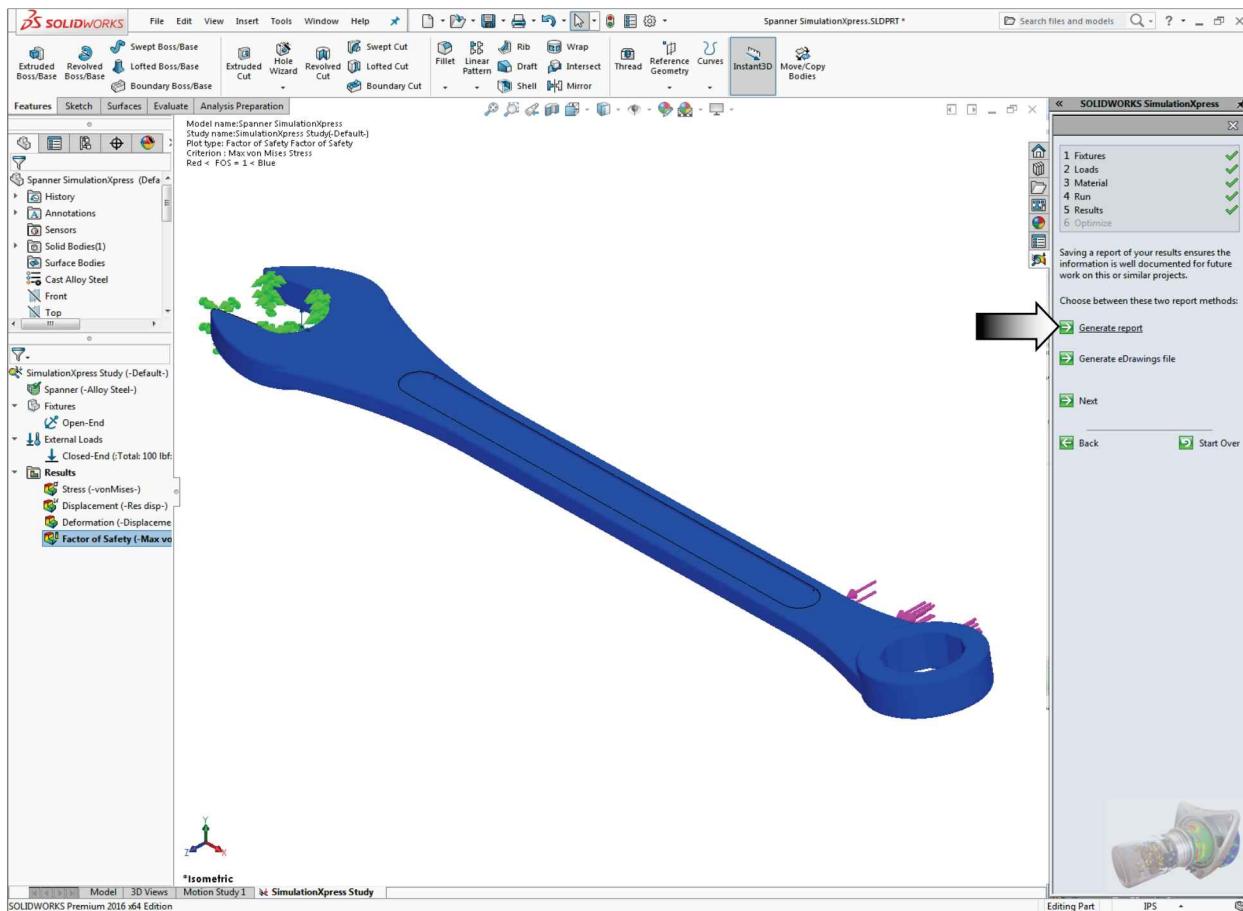
SimulationXpress displays regions of the model with factors of safety less than the specified value in red (unsafe regions) and regions with higher factors of safety in blue (safe regions).



9. Creating the report:

Click Generate Report →.

SimulationXpress cycles through the results, generates a report in Word format, and the MS-Word application is launched to display the full report.



The report includes:

- | |
|---|
| <ol style="list-style-type: none"> 1. Cover Page 2. Model Information 3. Load/Fixture Details 4. Slid Mesh Information 5. Stress Results 6. Displacement Results 7. Deformation Results 8. Factor of Safety Results |
|---|

In the **Report Settings** dialog box, enable the **Description** checkbox and enter the following:

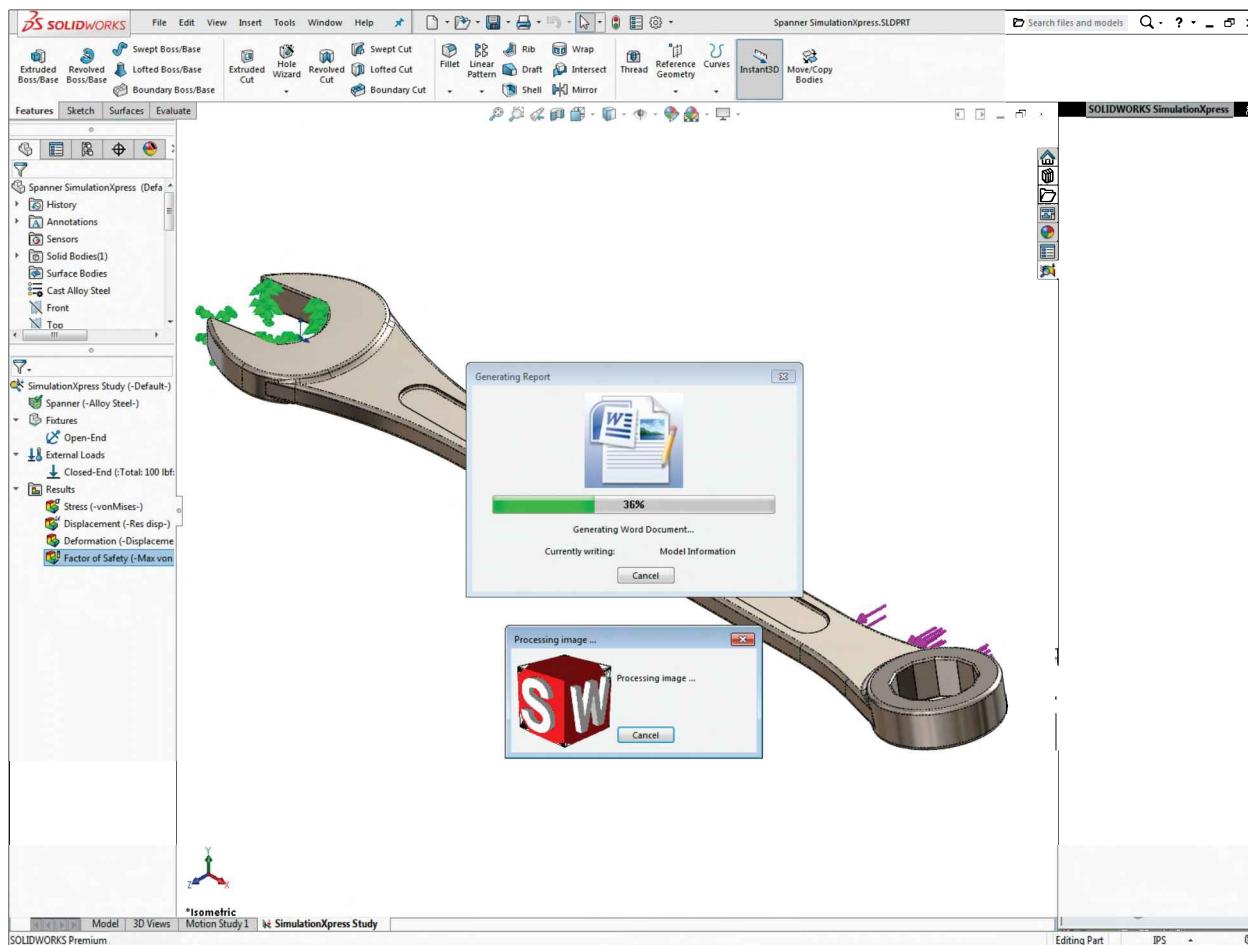
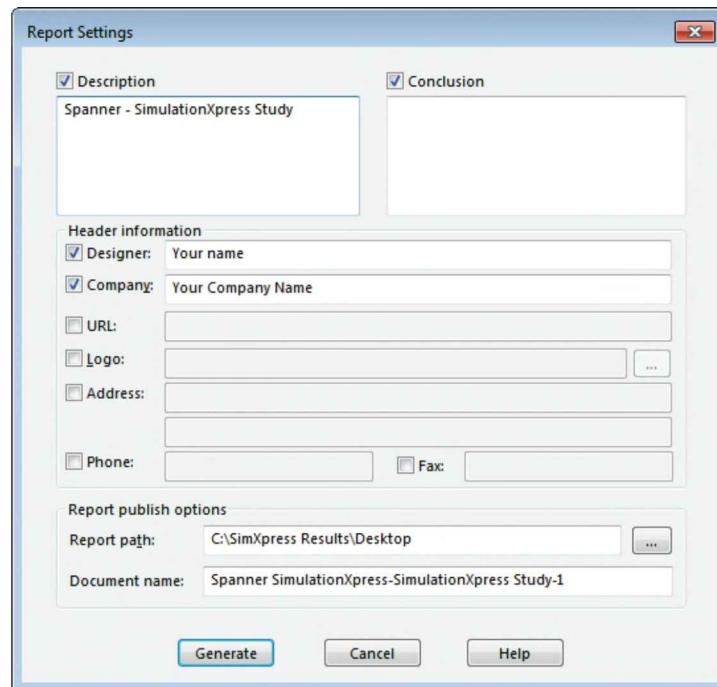
* **Spanner - SimXpress Study.**

* **Your Name.**

* **Your Company Name** and any information that you wish to include in the report.

Select the location that you want to save the report.

Click **Generate**.



SOLIDWORKS 2024 I Advanced Techniques I SimulationXpress

The Report Cover Page

Your Company
www.yourwebaddress.com



Description
Spanner - SimulationXpress Study

Simulation of Spanner

Date: Tuesday, March 02, 2010
Designer: Your Name
Study name: SimulationXpress Study
Analysis type: Static

Table of Contents

Description.....	1
Assumptions	2
Model Information.....	2
Material Properties.....	3
Loads and Fixtures.....	4
Mesh Information.....	5
Study Results.....	6
Conclusion	10

SolidWorks Analyzed with SolidWorks Simulation Simulation of Spanner 1

The Model Information

Your Company
Your Name
3/2/2010

Assumptions

Model Information



Model name: Spanner
Current Configuration: Default

Solid Bodies

Document Name and Reference	Treated As	Volumetric Properties	Document Path/Date Modified
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SolidWorks Analyzed with SolidWorks Simulation Simulation of Spanner 2

The Model Information cont.

Your Company
Your Name
3/2/2010

Fillet3	Solid Body	Mass:0.240191 lb Volume:0.86143 / in ³ Density:0.27818 lb/in ³ Weight:0.240028 lbf	C:\Users\Gateway\Desktop\pSW-2011 Advanced Techniques Training Files\Built Parts\Spanner.SLDPRPart Apr 02 00:24:01 2010
---------	------------	---	---

Material Properties

Model Reference	Properties	Components
	Name: Alloy Steel Model type: Linear elastic isotropic Default: Max von Mises Stress criterion Yield strength: 89984.0 psi Tensile strength: 104982 psi	SolidBody 1(Fillet3)(Spanner)

SolidWorks Analyzed with SolidWorks Simulation Simulation of Spanner 3

The Loads and Fixtures Details

Your Company
Your Name
3/2/2010

Loads and Fixtures

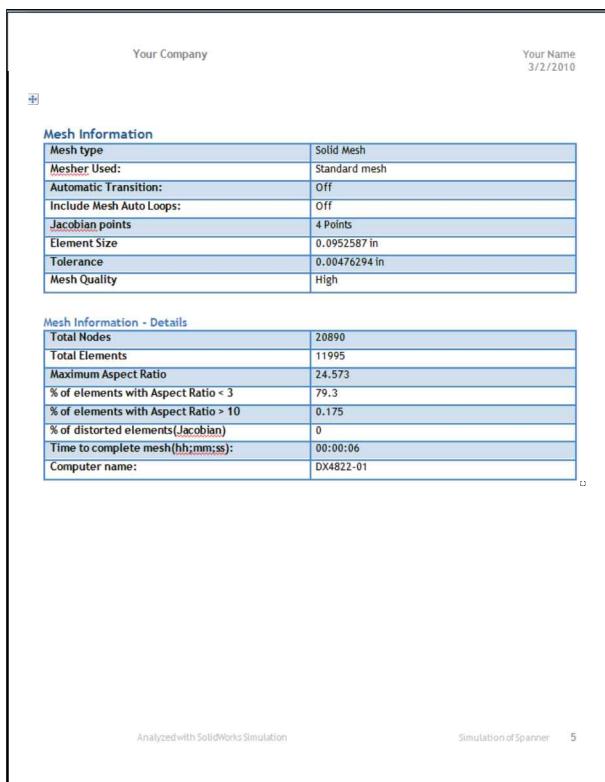
Fixture name	Fixture Image	Fixture Details
Fixed-1		Entities: 3 face(s) Type: Fixed Geometry

Load name

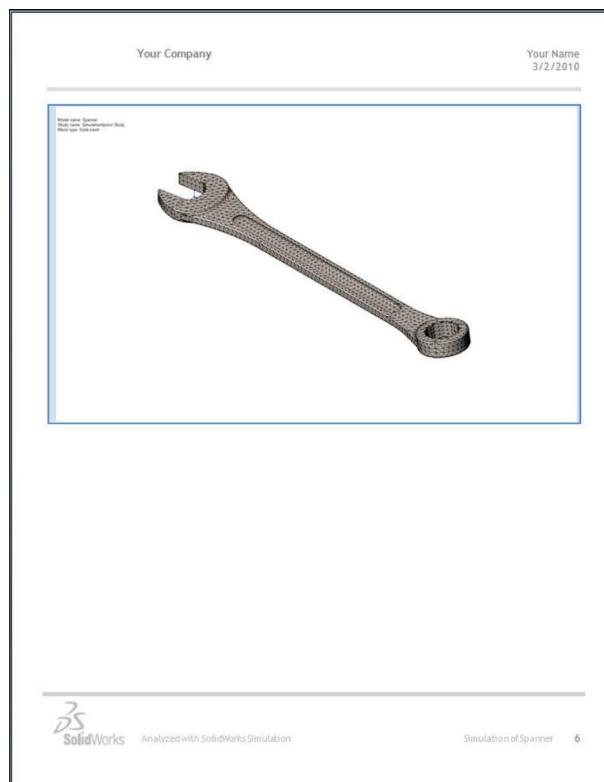
Load name	Load Image	Load Details
Force-1		Entities: 2 face(s), 1 plane(s) Reference: Front Type: Apply force Values: ..., ..., 100 lbf

SolidWorks Analyzed with SolidWorks Simulation Simulation of Spanner 4

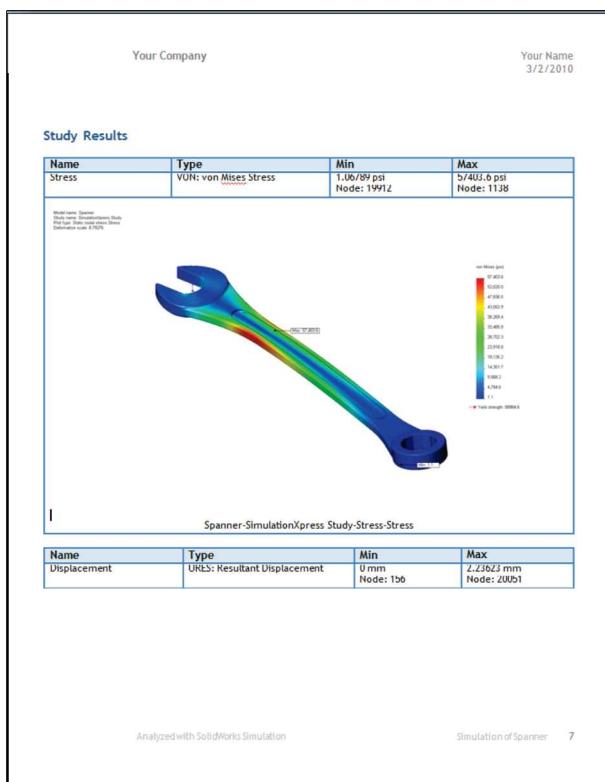
The Mesh Information



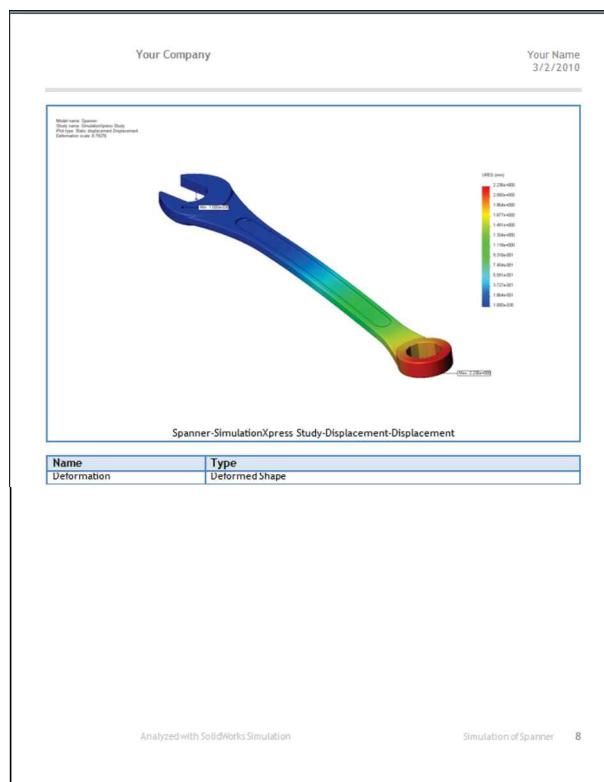
The Solid Mesh Plot



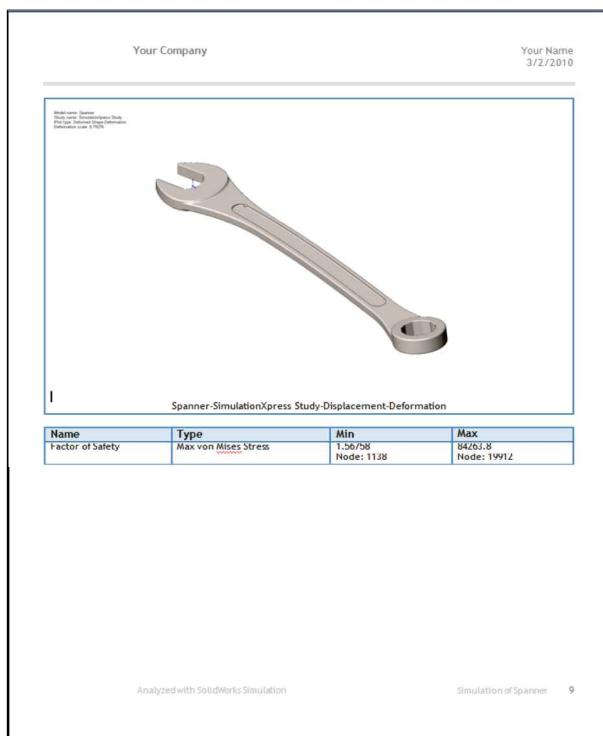
The Von Mises Stress Plot



The Displacement Plot



The Deformation Plot



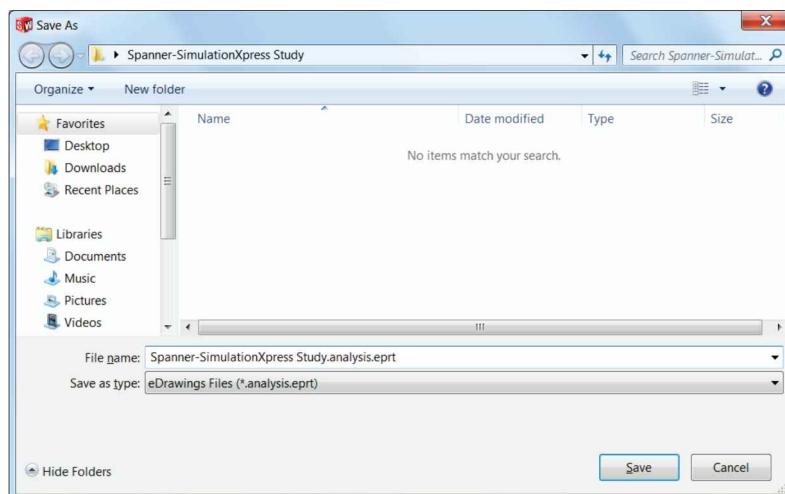
The Factor of Safety Plot



When finished with viewing the Report, click: **Generate eDrawings File** →.

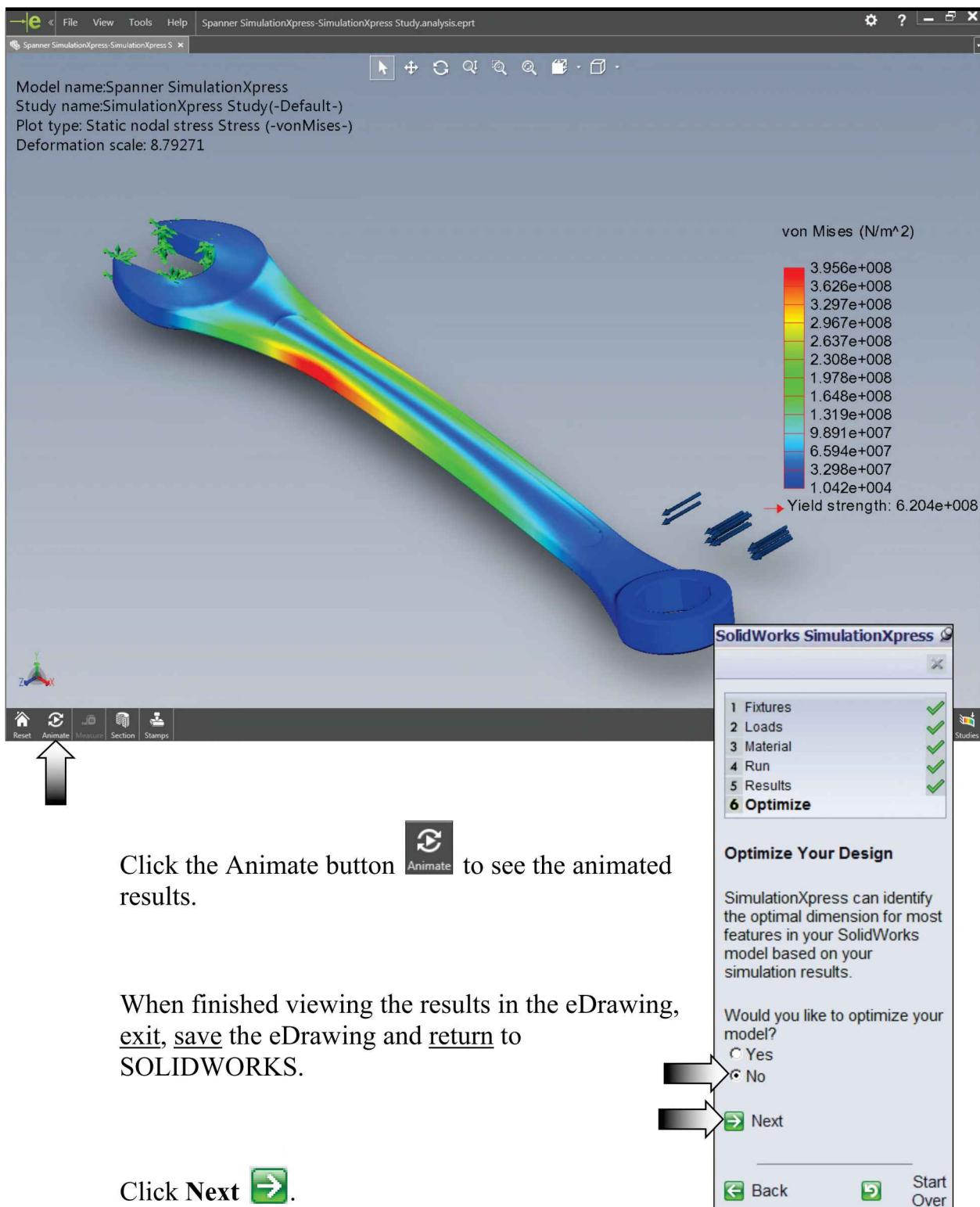
10. Generating the eDrawings file:

An eDrawings file can be created for the SimulationXpress result plots. The eDrawings file allows you to view, animate and print your analysis results.



When prompted, **Save** the analysis study in the default folder.

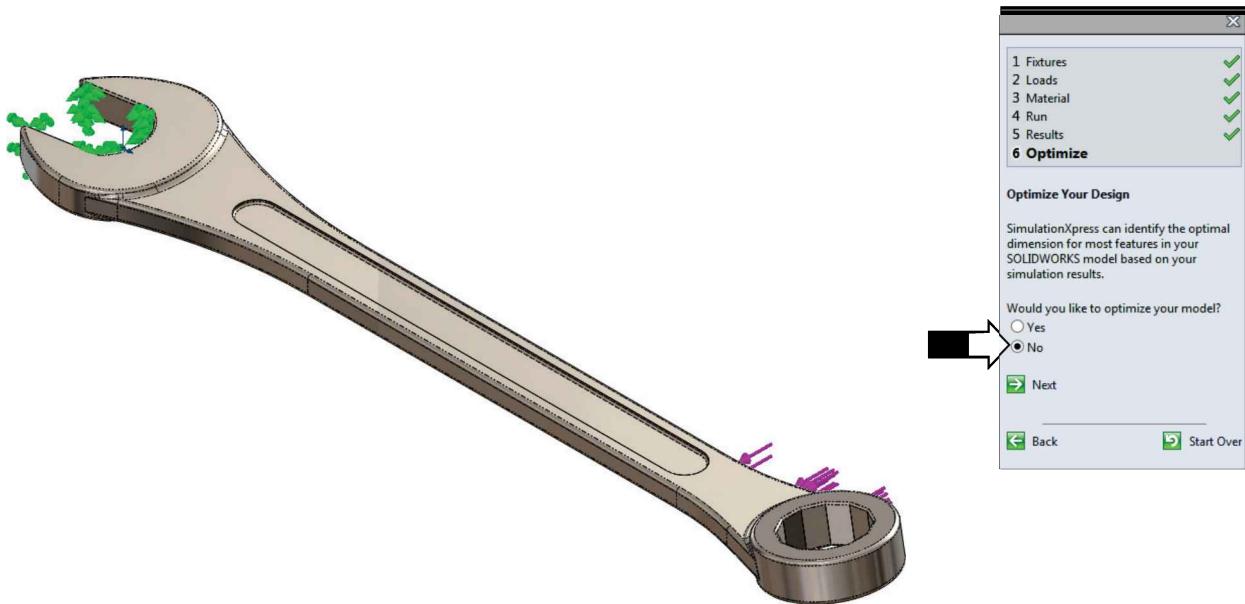
SimulationXpress creates an eDrawing file with the .eprt extension. The file contains von Mises stress, displacement, deformation, and Factor of Safety plots. By default, the von Mises stress plot is displayed.



Click **No** under **Optimize Your Design**. (Only if the analysis fails then the optimization step is needed.)

Click **Next ➔** again.

At this point, you are prompted that the analysis has been completed. All 5 steps on the SimulationXpress property tree (right side) have the check marks in front of them.



11. Saving your work:

Click **File / Save As**.

Enter **Spanner Study** for file name and click **Save**.

Isotropic, Orthotropic & Anisotropic Materials:

Isotropic Material: If its mechanical properties are the same in all directions. The elastic properties of an Isotropic material are defined by the Modulus of Elasticity (E_X) and Poisson's Ratio (ν_{XY}).

Orthotropic Material: If its mechanical properties are unique and independent in the directions of three mutually perpendicular axes.

Anisotropic Material: If its mechanical properties are different in different directions. In general, the Mechanical properties of the anisotropic materials are not symmetrical with respect to any plane or axis.

SimulationXpress supports Isotropic materials only.

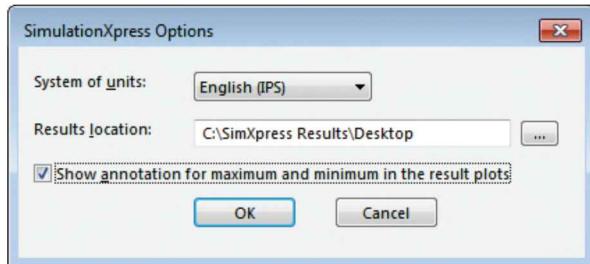
Questions for Review

1. SimulationXpress can be accessed from the Tools pull down menu.
 - a. True
 - b. False
2. System Of Units SI (Joules) is the only type that is supported in SimulationXpress.
 - a. True
 - b. False
3. The material of the part can be selected from the built-in library or input directly by the user.
 - a. True
 - b. False
4. SimulationXpress supports Isotropic, Orthotropic, and Anisotropic materials.
 - a. True
 - b. False
5. Restraints/Fixture are used to anchor certain areas of the part so that it will not move during the analysis.
 - a. True
 - b. False
6. Only one surface/face should be used for restraint in each study.
 - a. True
 - b. False
7. The elements size (mesh) can be adjusted to a smaller value for more accurate results.
 - a. True
 - b. False
8. The types of results reported are:
 - a. Stress Distribution
 - b. Deformed Shape
 - c. Deformation
 - d. Factor of Safety
 - e. All of the above

1. TRUE	2. FALSE	3. TRUE	4. FALSE	5. TRUE	6. FALSE	7. TRUE	8. E
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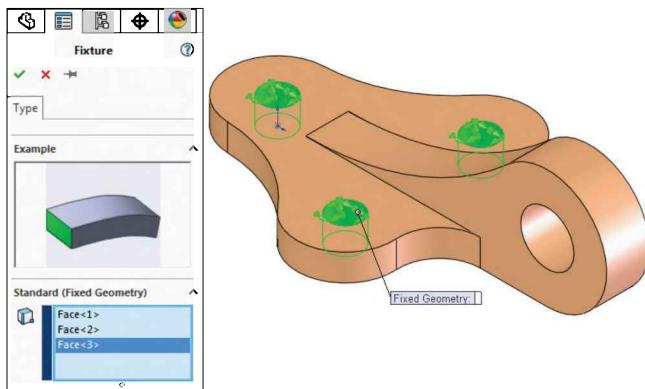
Exercise 1: SimulationXpress: Force

1. Open the existing part: **Extrude Boss & Extrude Cut.**

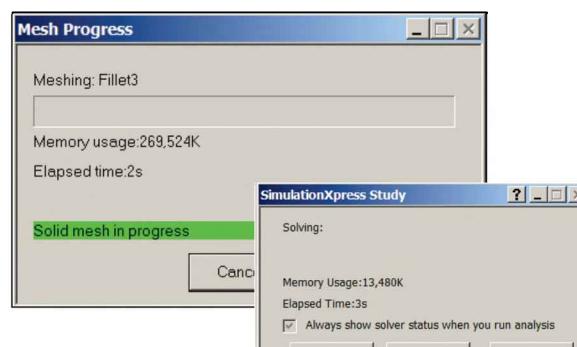


Property	Value
Elastic Modulus	304575
Poissons Ratio	0.28
Shear Modulus	114579.00
Density	0.27818
Tensile Strength	104982.01
Compressive Strength in X	psi
Yield Strength	89984.59
Thermal Expansion Coefficient	1.3e-005 /°F
Thermal Conductivity	0.000668738 Btu/(in sec °F)
Specific Heat	0.109869 Btu/(lb °F)
Material Damping Ratio	N/A

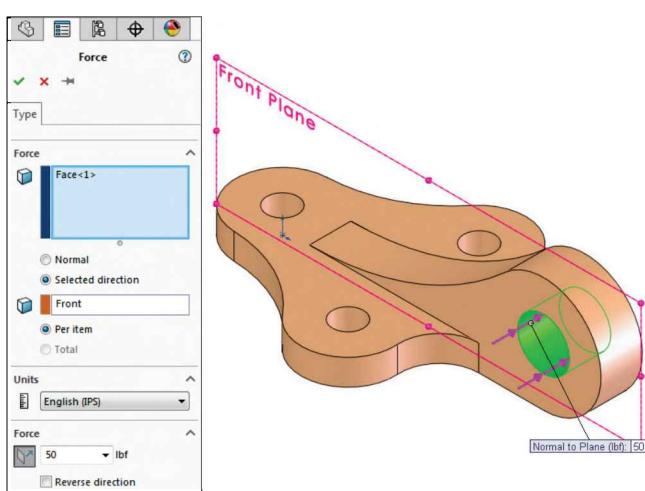
2. Set Unit to: **English (IPS).**



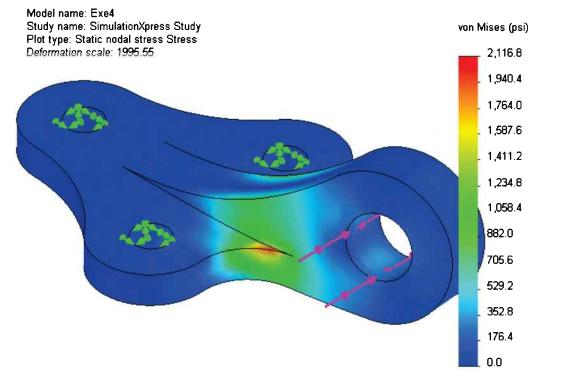
5. Select Material: **Alloy Steel.**



3. Apply Restraint: to 3 Holes.



6. Run the Analysis.



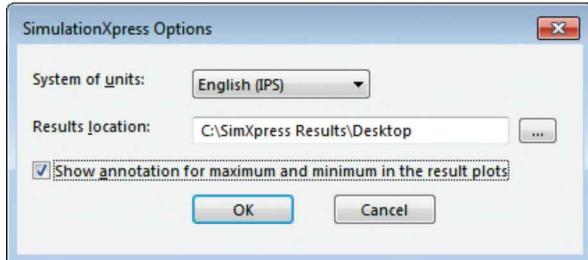
4. Apply Force of **50 Lbs.** to the side hole, normal to the **Front Plane**.

7. Check the von Mises Stress results.

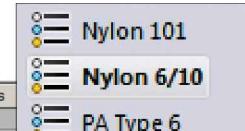
8. Save a copy as: **Simulation_Force.**

Exercise 2: SimulationXpress: Pressure

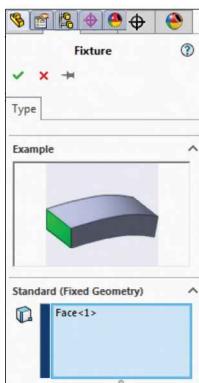
1. Open the existing part: **Bottle_SimulationXpress**.



Property	Value	Units
Elastic Modulus in X	1203812.99	psi
Poisson's Ratio in XY	0.28	N/A
Shear Modulus in XY	464120.67	psi
Mass Density	0.0505782	lb/in^3
Tensile Strength in X	20676.43	psi
Compressive Strength in X		psi
Yield Strength	20166.48	psi
Thermal Expansion Coefficient in X	3e-005	/°F
Thermal Conductivity in X	7.08862e-006	Btu/(in·sec·°F)
Specific Heat	0.358269	Btu/(lb·°F)
Material Damping Ratio		N/A



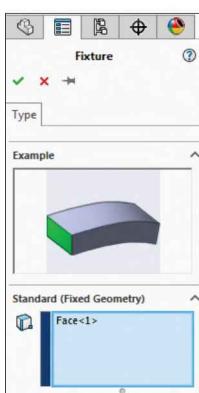
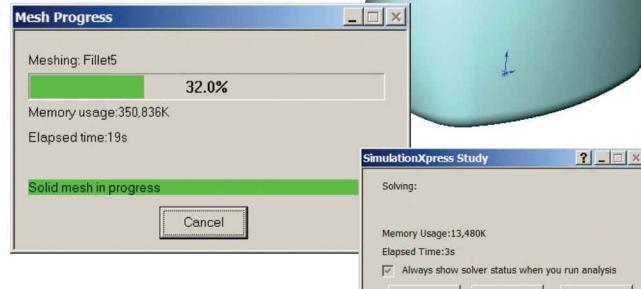
2. Set Unit to: **English (IPS)**.



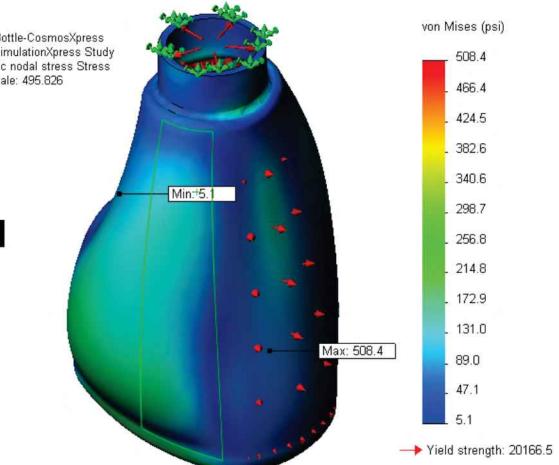
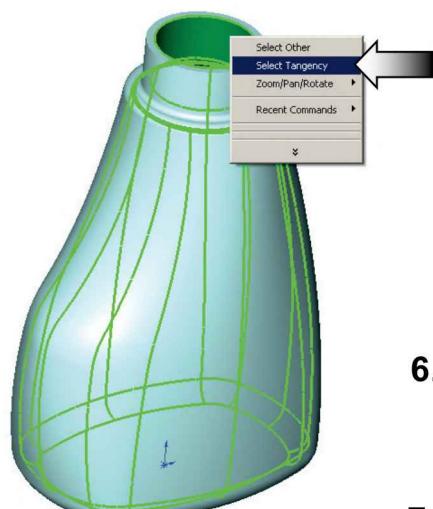
3. Apply Restraint:
to the **Top** face.



5. Select Material:
Nylon 6/10.



4. Apply Pressure
of **5 psi**. and
select all Inside faces.



6. Run the Analysis and show the
Von-Mises Stress.

7. Save a copy as:
Simulation_Pressure.



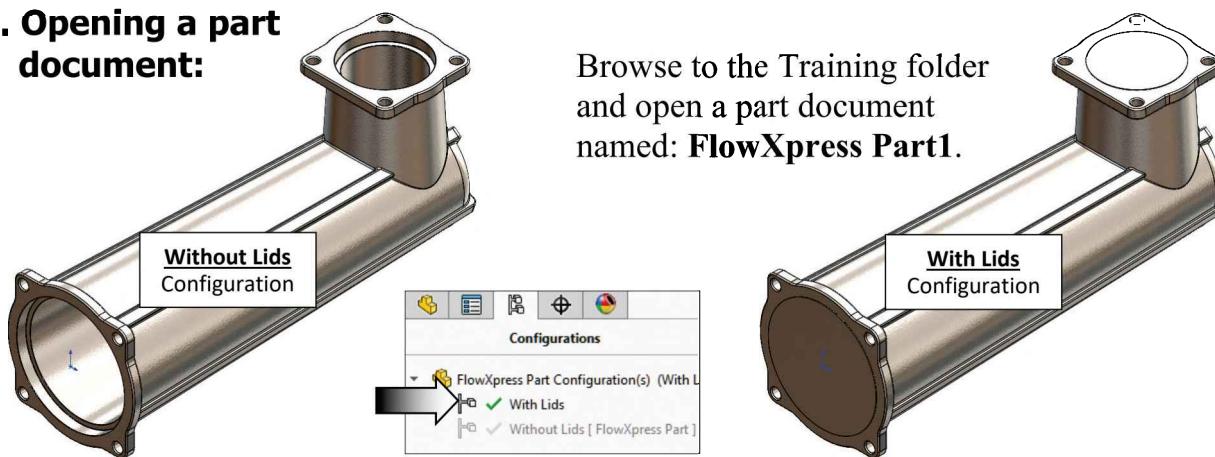
SOLIDWORKS FlowXpress

Using the Flow Wizard

FloXpress is a fluid dynamics application that calculates how fluid flows through your parts or assemblies. Based on the calculated velocity field, you can find problem areas in your models and improve them during the early stages of the design process.

The FlowXpress Wizard guides you through the steps required to create the analysis such as: Create Lids to close-off all through openings, Check Geometry, Select a Fluid, Set the Flow Inlet and Outlet, Solve the Model, and View the Results.

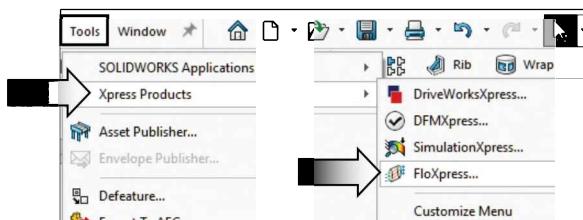
1. Opening a part document:



Browse to the Training folder and open a part document named: **FlowXpress Part1**.

2. Switching Configuration:

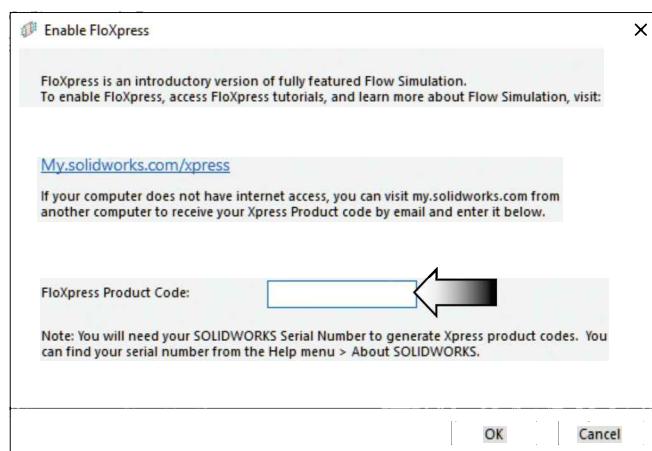
Switch to the ConfigurationManager tree and double-click the configuration named: **With Lids**.



3. Enabling FlowXpress:

Select Tools, Xpress Products.

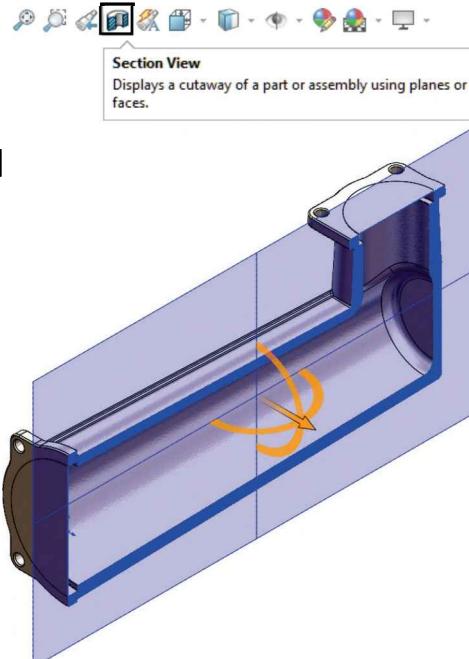
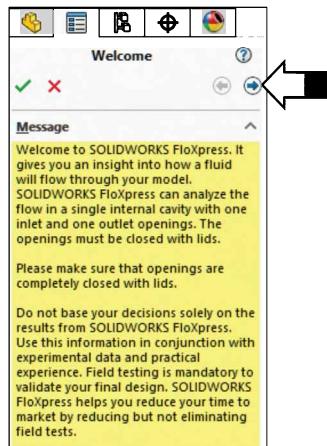
(If needed, click the blue link: My.solidworks.com/xpress to register and obtain the FloXpress Product Code, then enter the code in the dialog box.)



4. Using the Flow Wizard:

Create a **Section View** using the **Right** plane so selecting the inside faces can be done more easily.

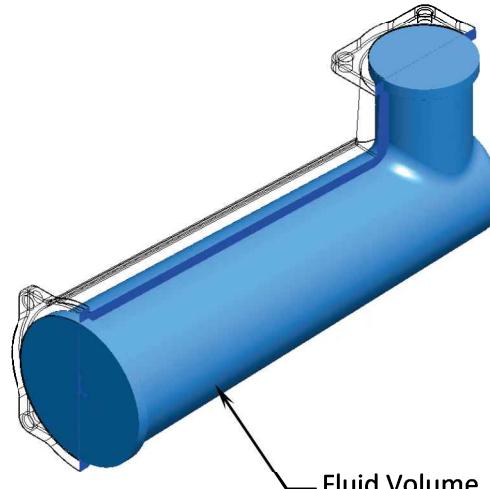
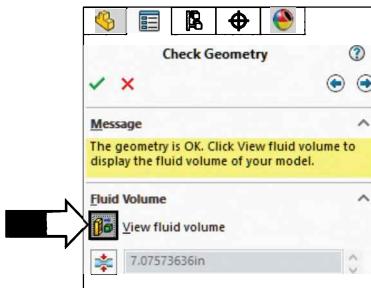
Click the **Next** arrow  on the Welcome tree.



5. Checking Geometry:

In the Check Geometry section, click the **View Fluid Volume** button. A preview of the fluid volume is displayed in the blue color.

Note: All through openings must be closed in order to view the fluid volume.

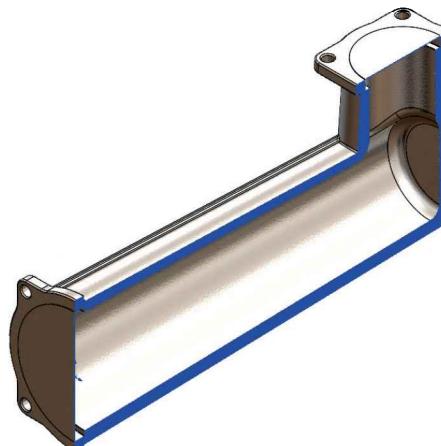
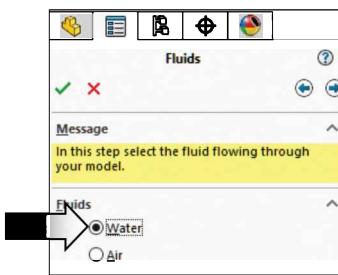


Click **Next** .

6. Selecting Fluid Type:

For Fluid Type, select **Water**.

Click **Next** .

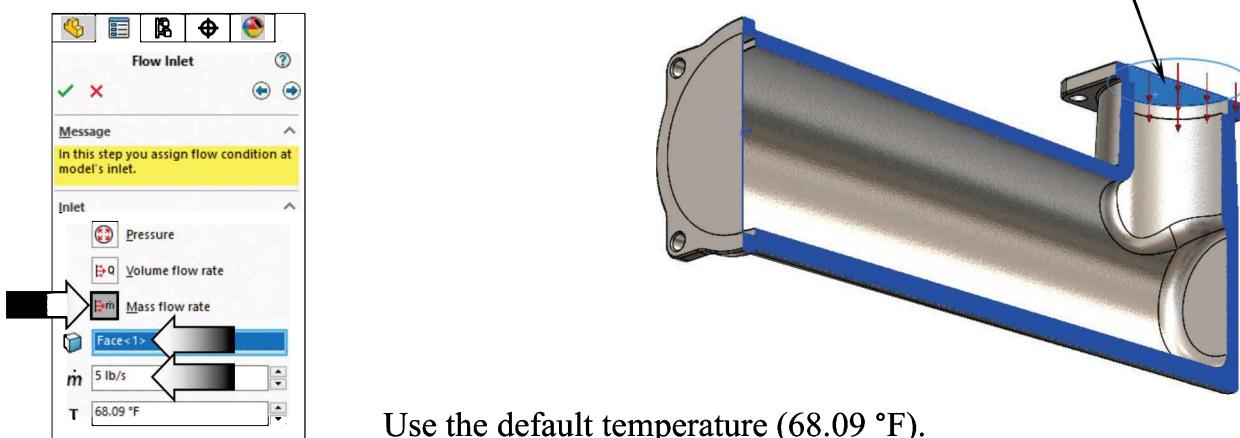


7. Setting the Flow Inlet:

For Flow Inlet, select the **Mass Flow Rate** button.

For Face to Apply Inlet Boundary Condition, rotate the model and select the inside-face of the Inlet Lid, as indicated.

For Mass Flow Rate, enter **5 lb/s**.

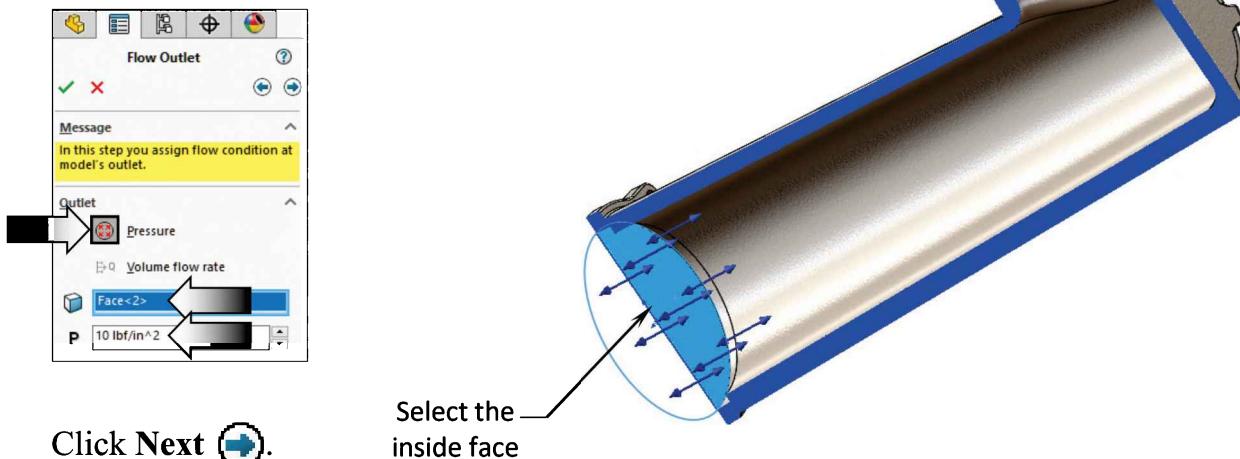


Click Next .

8. Setting the Flow Outlet:

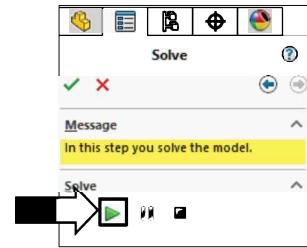
For Flow Outlet, rotate the model and select the inside face of the Outlet Lid as noted.

For Pressure, enter **10 lbf/in²**.



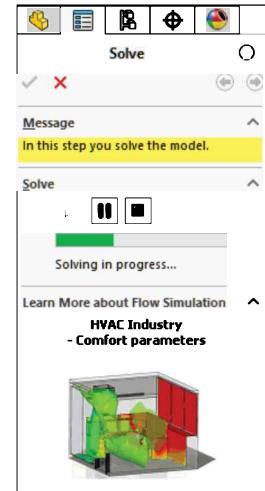
9. Solving the model:

FlowXpress has collected all information needed to analyze the model at this point.



Click **Solve** .

FlowXpress starts the analysis to calculate the flow velocity in the model.

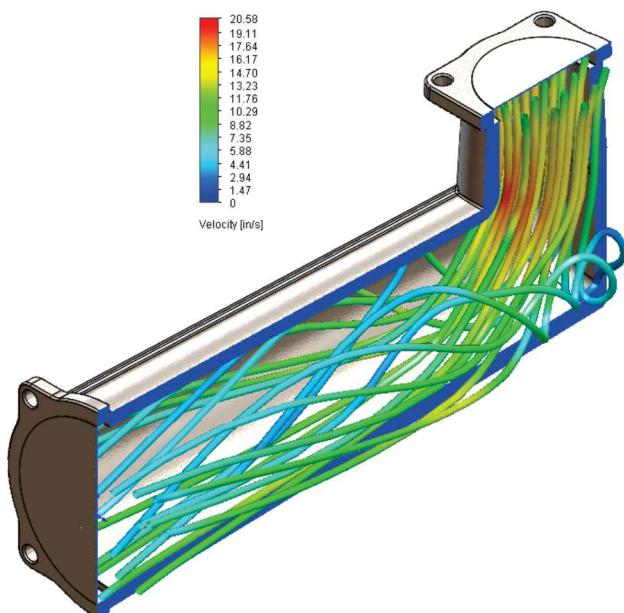


When the analysis is complete, the **View Results** Property Manager opens automatically.

FloXpress uses Trajectories to display results as flow streamlines.

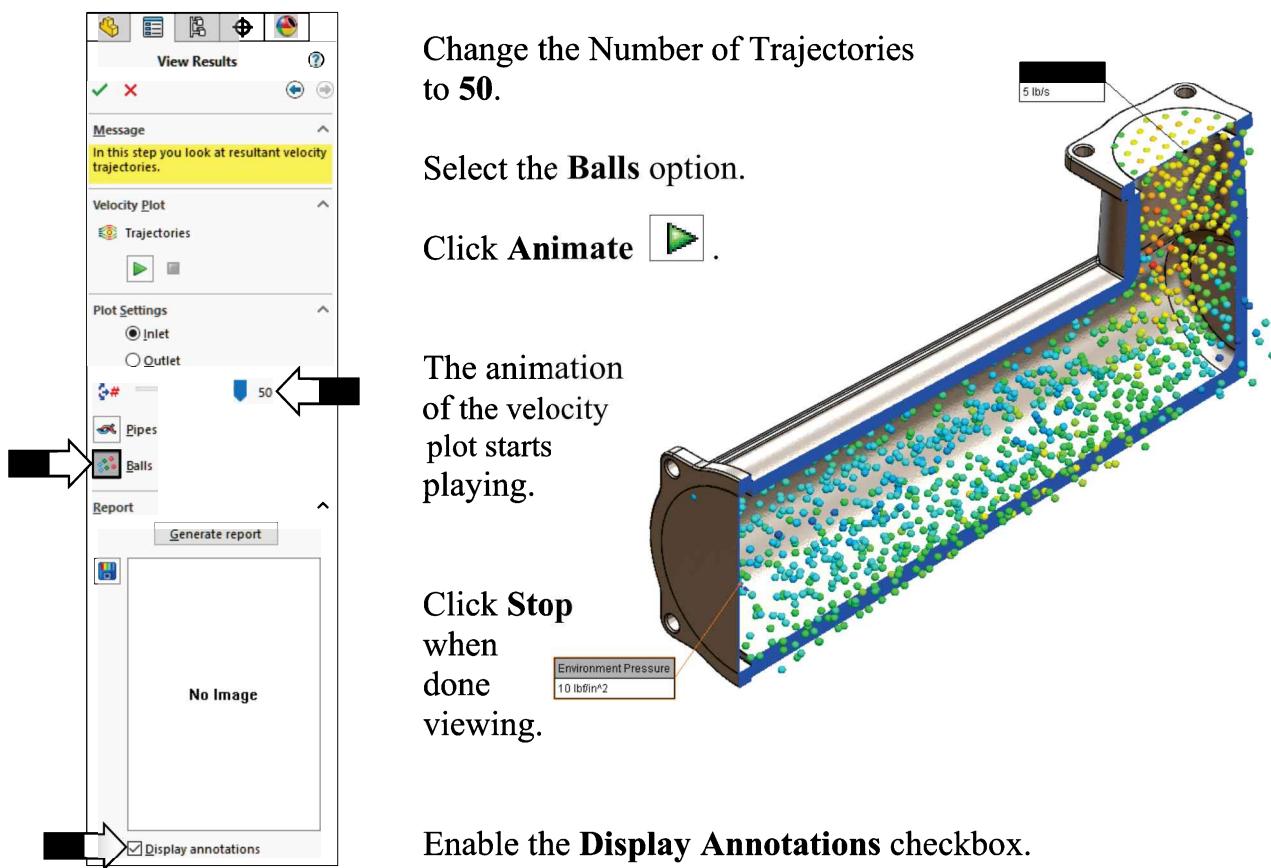
The colors represent changes in velocity along the trajectories.

Based on your input, the color of the trajectories shows how any parameter changes along the trajectories.

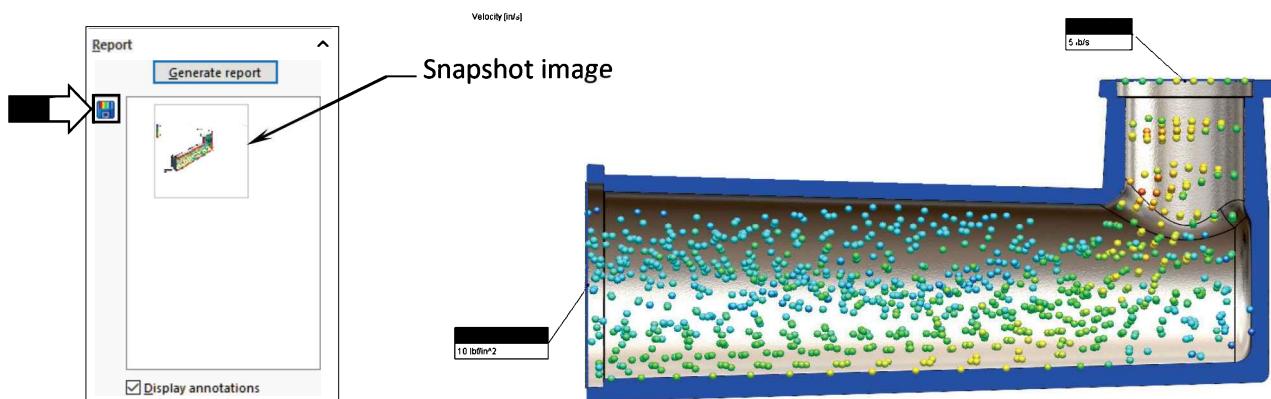


10. Viewing the results:

When FloXpress completes the analysis, you can examine flow trajectories, which are the flow lines (or balls) between the inlet and outlet openings. Trajectories are tangent to the flow velocity at every point.



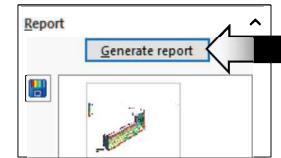
Click **Save a Snapshot** to save an image of the flow trajectories as a JPEG image. The images are automatically saved in a folder named fxp1 located in the same folder as the model.



11. Creating a report:

Click **Generate Report**.

A report is automatically generated and opened in MS-Word.



SOLIDWORKS FloXpress Report

SOLIDWORKS FloXpress is a first pass qualitative flow analysis tool which gives insight into water or air flow inside your SOLIDWORKS model. To get more quantitative results like pressure drop, flow rate etc you will have to use SOLIDWORKS Flow Simulation. Please visit www.solidworks.com to learn more about the capabilities of SOLIDWORKS Flow Simulation.

Model
Model Name: FlowXpress Part2 (Completed).SLDPRT

Fluid
Water

Inlet

Type	Mass Flow Rate
Faces	Lid_Inlet//Face<2>
Value	Mass flow rate: 5.0000 lb/s Temperature: 68.09 °F

Outlet

Type	Environment Pressure
Faces	Lid_Outlet//Face<2>
Value	Environment pressure: 10.00000 lbf/in^2 Temperature: 68.09 °F

Results

Name	Unit	Value
Maximum Velocity	in/s	20.41

12. Saving your work:

Select **File, Save As**.

Enter: **FlowXpress Part2.sldprt** for the file name. Save and close all documents.

Exercise: Flow Simulation

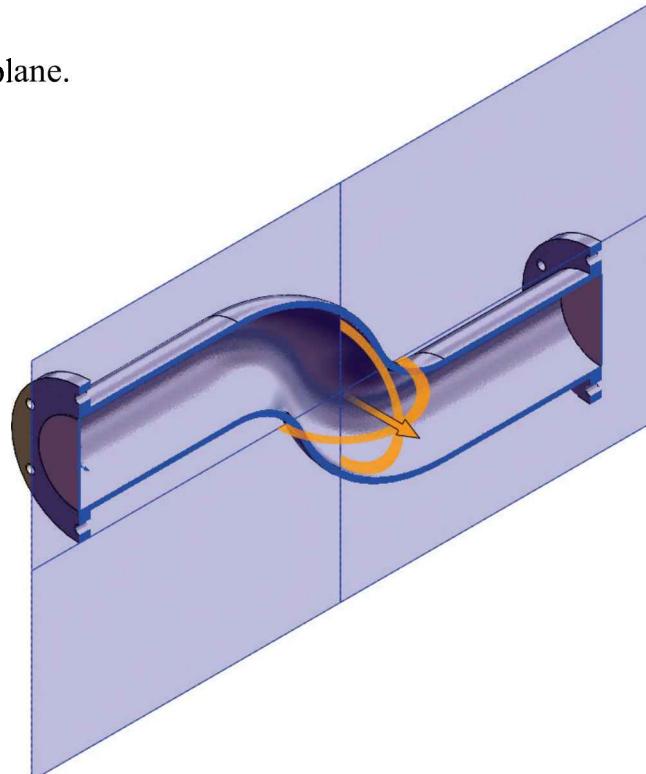
1. Opening a part document:

FlowXpress Part 2.sldprt



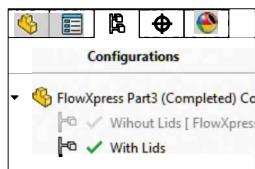
The model contains 2 configurations:
With Lids and Without Lid.

Create a Section View using the **Right** plane.



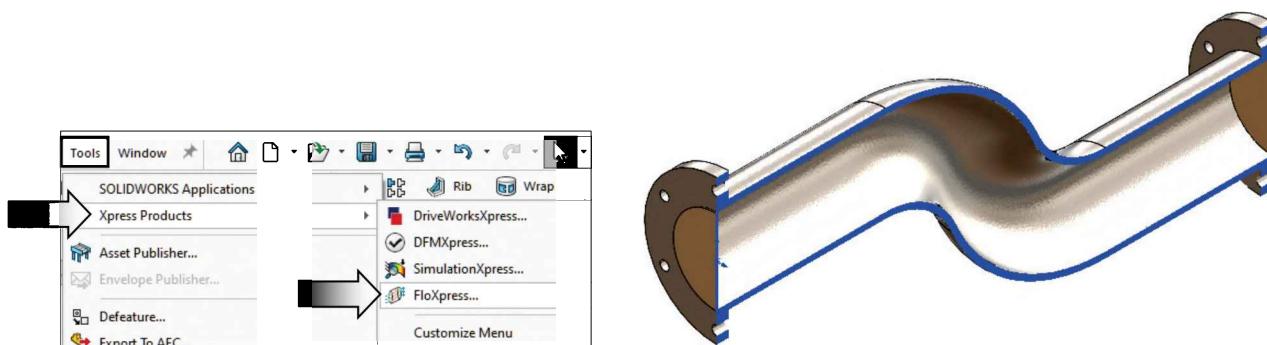
2. Switching Configuration:

Switch to the ConfigurationManager and double-click the **With Lids** configuration to activate it.



3. Enabling FlowXpress:

Select Tools, Xpress Products, FlowXpress.



4. Setting the conditions:

For Flow Inlet, select **Mass Flow Rate** and select the inside face of the lid on the right side.

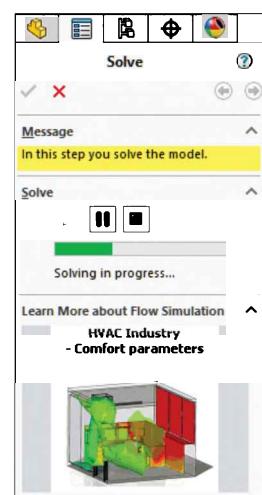
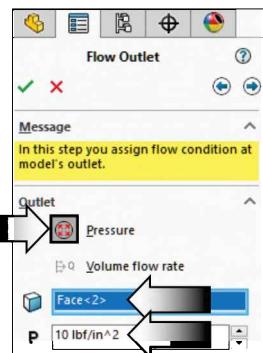
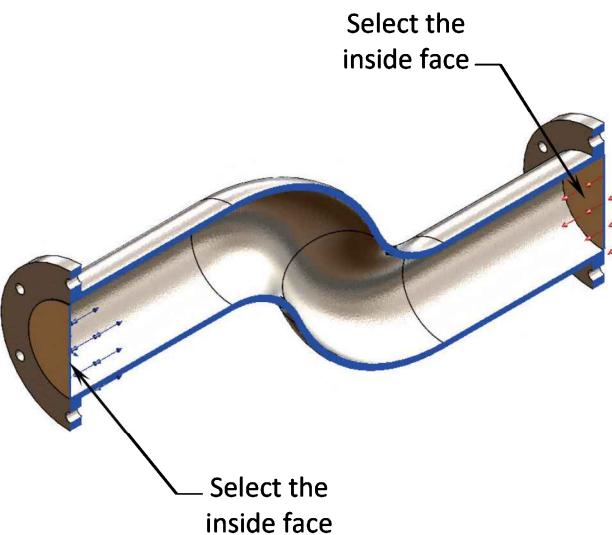
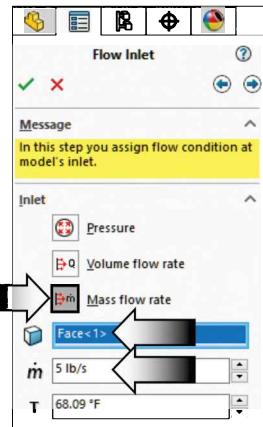
For Mass Flow-Rate, enter **5 lb/s**.

Use the default Temperature of **68.09°**.

For Flow Outlet, select **Pressure** and select the inside face of the lid on the left side.

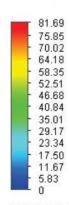
For Pressure, enter **10 lbf/in²**.

Click **Solve**.



Optional:

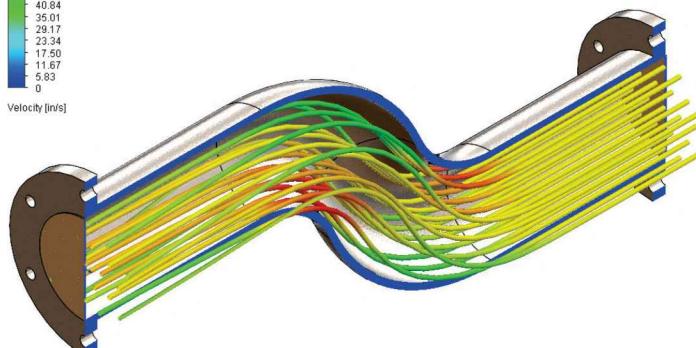
Generate a report for the flow study.



5. Saving your work:

Save your work as:
FlowXpress Part4.sldprt.

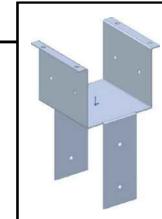
Close all documents.



CHAPTER 13

Sheet Metal Parts

Sheet Metal Parts Post Cap



This chapter discusses the introduction to designing sheet metal parts.

Create a sheet metal part in the folded stage and add the sheet metal specific flange features such as:

- * Base Flange.
- * Edge Flanges.
- * Sketch Bends.
- * Cut with Link to Thickness.
- * Normal Cuts.



There are several options for specifying the setback allowance, or the length difference between the fold and the flat patterns of a sheet metal part.

* **Bend Table:** You can specify the bend allowance or bend deduction values for a sheet metal part in a bend table. The bend table also contains values for bend radius, bend angle, and part thickness.

* **K-Factor:** Is a ratio that represents the location of the neutral sheet with respect to the thickness of the sheet metal part.

Bend allowance using a K-Factor is calculated as follows:

$$BA = \Pi (R + KT) A / 180$$

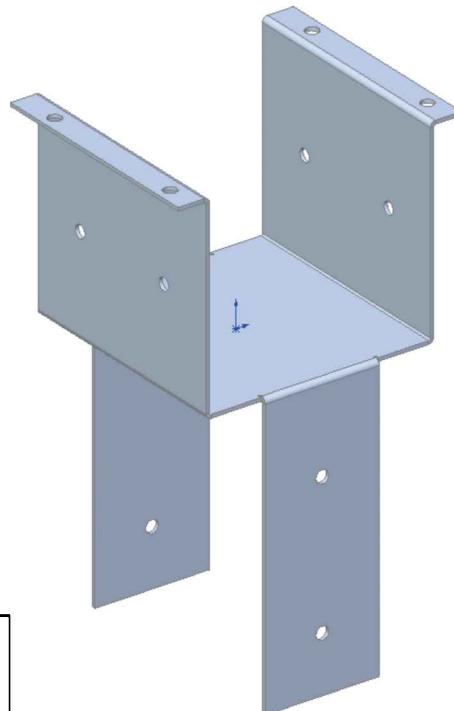
* **Use Bend Allowance:** Enter your own bend value based on your shop experience.

* **Bend allowance Calculations:** The following equation is used to determine the total flat length when bend allowance values are used: $L_t = A + B + BA$

* **Bend Deduction Calculations:** The following equation is used to determine the total flat length when bend deduction values are used: $L_t = A + B - BD$

Sheet Metal Parts

Post Cap



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI

Units: INCHES – 3 Decimals

Tools Needed:



Insert Sketch



Line



Circle



Dimension



Add Geometric Relations



Base Flange



Edge Flange



Sketch Bend



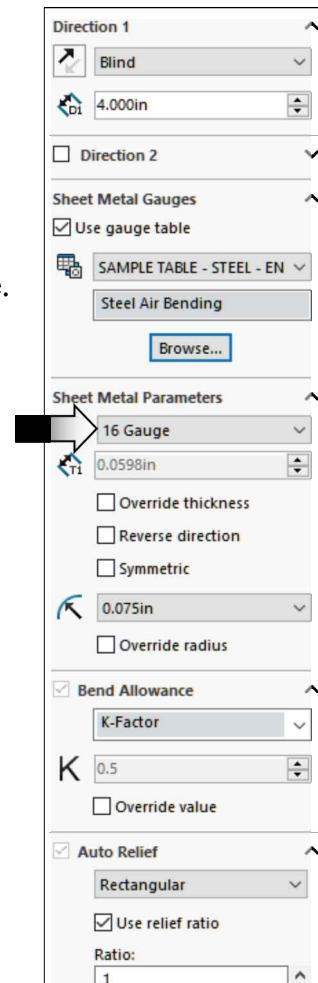
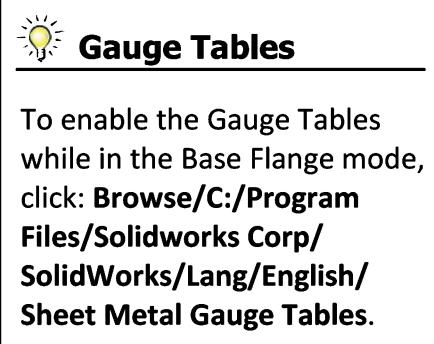
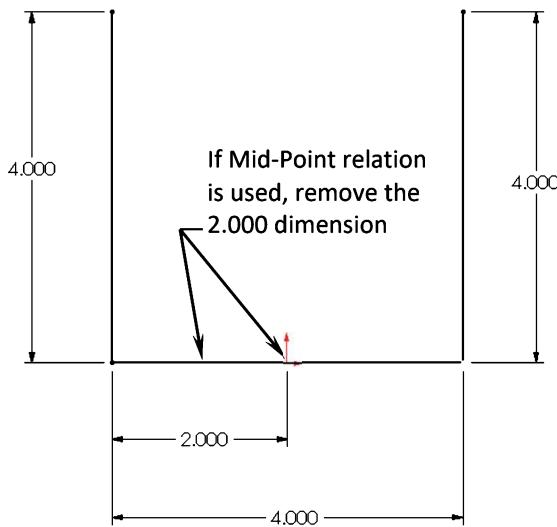
Flat Pattern

1. Starting with the base profile:

Select the Right plane from FeatureManager tree and insert a new sketch .

Sketch the profile below using the **Line** tool.

Add dimensions and relations as indicated to fully define the sketch.



2. Extruding the Base Flange:

 or select **Insert / Sheet Metal / Base Flange**.

Direction 1: **Mid-Plane**.

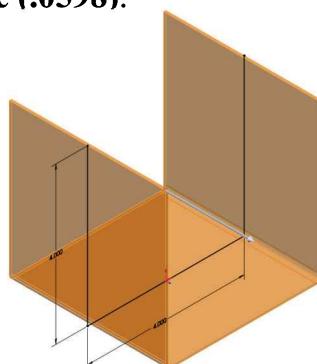
Extrude Depth: **4.00 in.**

Use Gauge Table: **Sample Table - Steel**.

Mat'l Thickness: **16 Gauge (.0598)**.

Bend Radius: **.075in**.

(Set the material thickness to inside.)

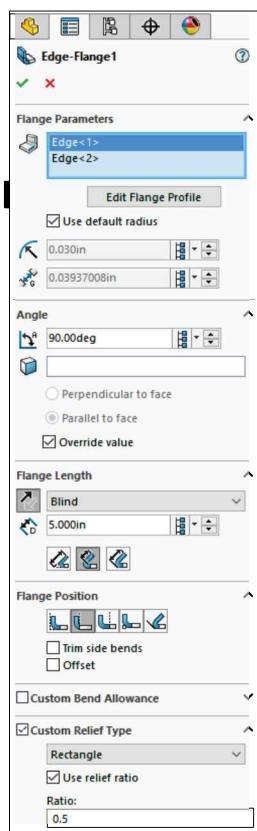
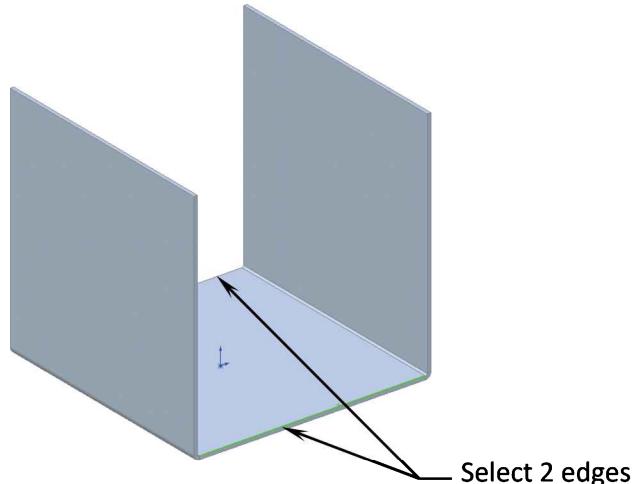


Click **OK**.

3. Creating an Edge Flange:

Hold the **Control** key and select the **2 edges** as indicated.

Click  or select **Insert / Sheet Metal / Edge Flange**.



Drag the cursor downwards and click anywhere to lock the direction of the 2 Edge Flanges.

Select **Material Outside** under the Flange Position.

Enter **90°** for Angle (default).

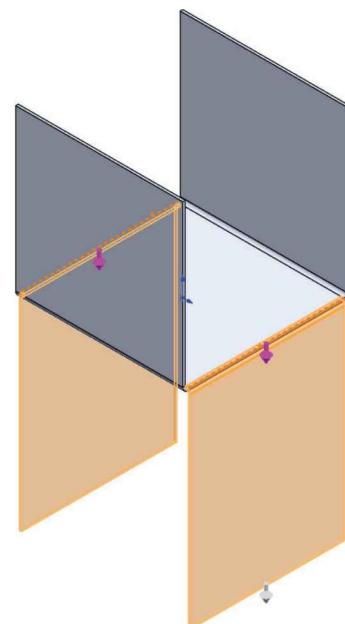
Flange Length: **Blind**.

Enter **5.000in.** for depth.

Relief Type: **Rectangle**.

Relief Ratio: **.5** (1/2 thickness).

Click **Edit Flange Profile** (arrow).



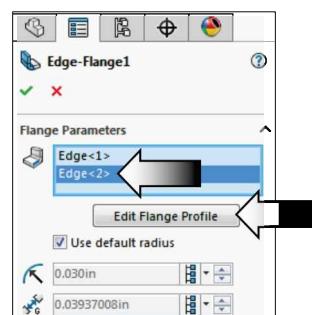
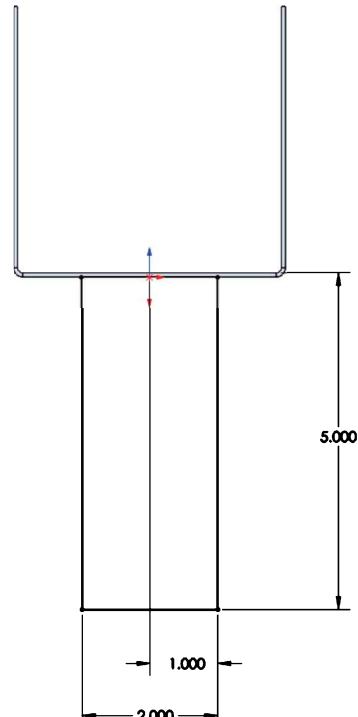
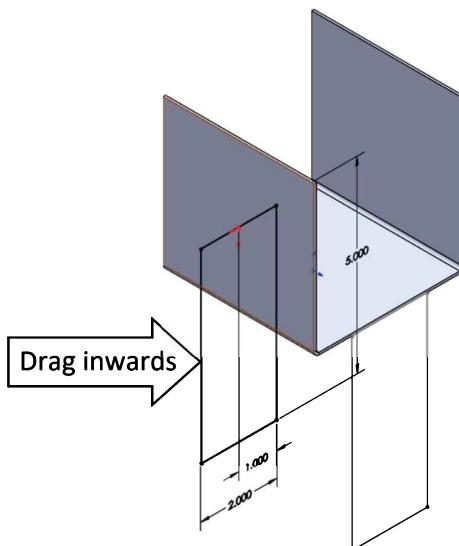
** The flange length will be modified in the next steps.*

4. Editing the Edge Flange Profile:

Move the Profile Sketch dialog out of the way.

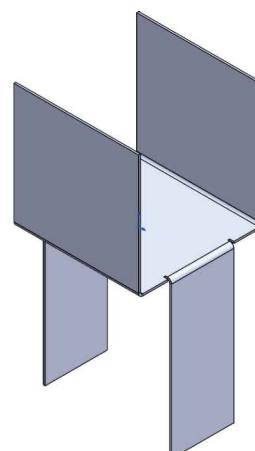
Drag the 2 outer lines inward as noted.

Add the dimensions shown below to fully define the sketch.



After the 1st edge flange is fully defined, click the **Back** button (arrow) to switch back to the previous screen.

Select **Edge 2** under the Flange Parameters section and click the **Edit Flange Profile** once again (arrow).

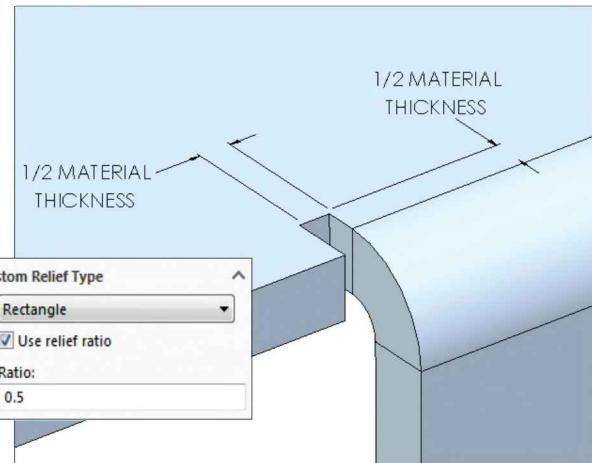


Repeat the step above to make the 2nd flange exactly the same as the 1st flange.

Click **Finished** [Finish] after the 2nd sketch is fully defined.

Zoom in on the upper end of the edge flange to see the relief details.

Beside the Rectangular relief, there are two other types available: Obound and Tear. Select the **Rectangular** type.



5. Viewing the Flat Pattern:

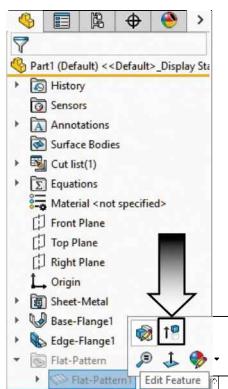
Click **Flatten**  on the **Sheet Metal** toolbar:

The flat pattern of the part is displayed.

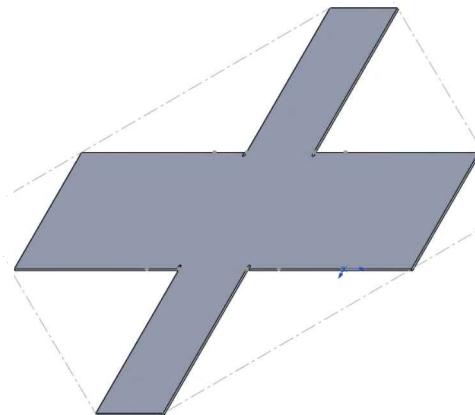
An alternative to viewing the flat pattern is to right click on the **Flat-Pattern1** feature and select **Unsuppress** (arrow).

Auto-Relief is added automatically after exiting the Sketch.

Relief Width and Depth are defaulted to one-half (0.5) the Material Thickness.

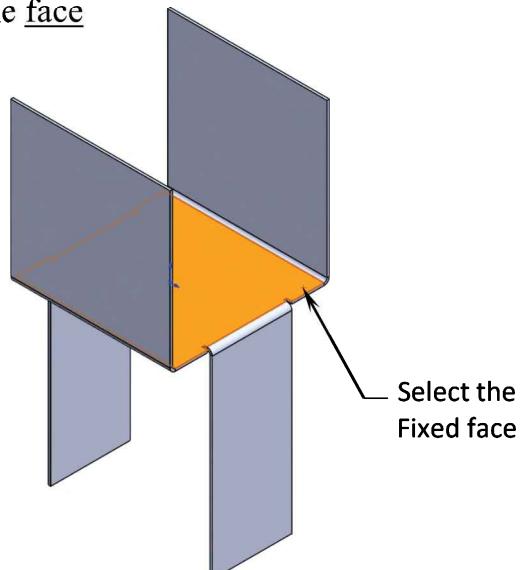
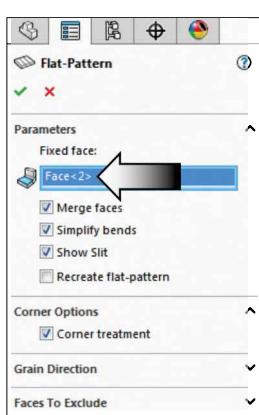
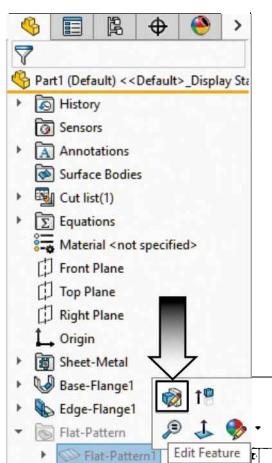


Exit the Flat Pattern.



6. Changing the Fixed face:

Edit the **Flat-Pattern** feature and select the face indicated to use as the **Fixed** face.

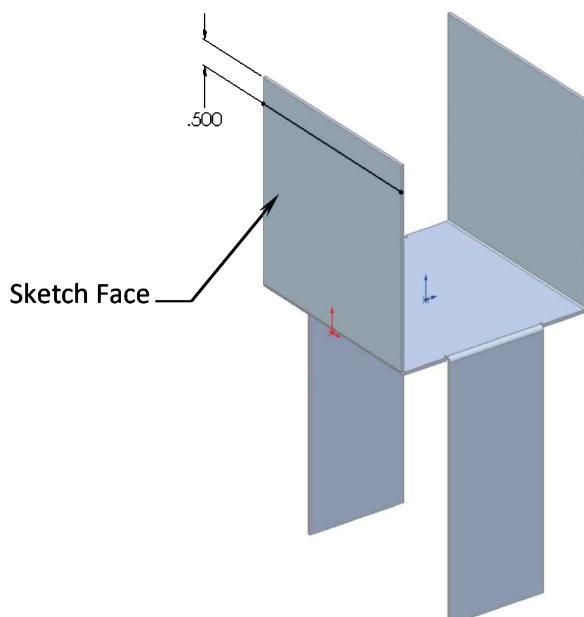


7. Creating a Sketch Bend:

Select the side face and open a **new Sketch** .

Sketch a **Line** starting at one edge and **Coincident** with the other.

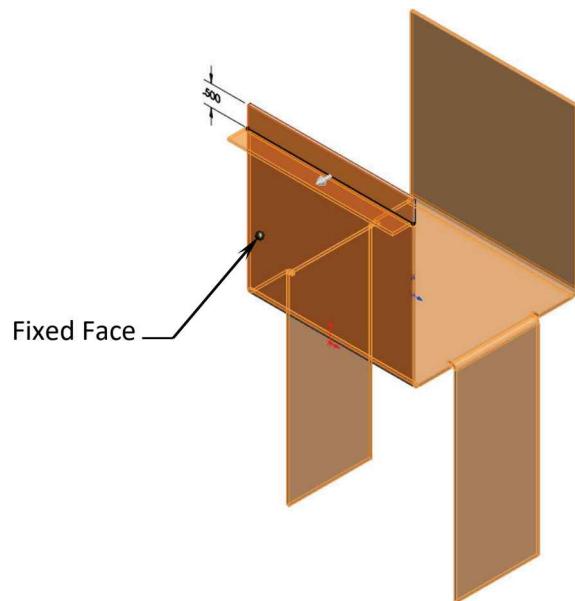
Add dimensions to fully define the sketch.



Sketch Bends

This command adds Bends or Tabs to the Sheet metal part with the sketch lines.

Only sketch lines are allowed, but more than one line can be used in the same sketch to create multiple bends.



Click  or select **Insert / Sheet Metal / Sketch Bends**.

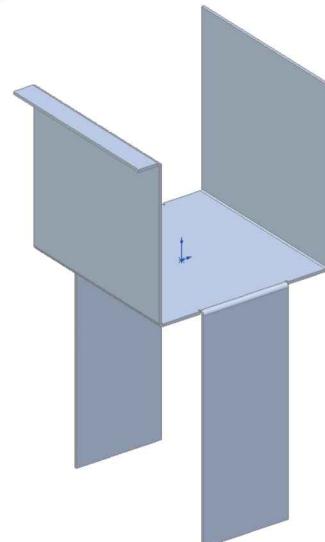
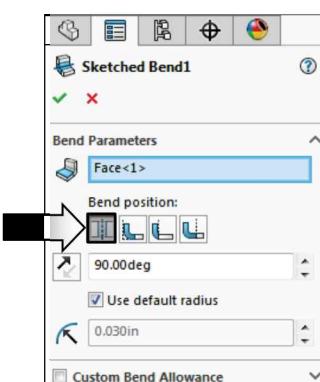
For Fixed Face  select the **lower portion** below the line.

For Bend Position select **Bend Centerline** (default).

For Bend Angle enter **90.00 deg**.

Enable **Use Default Radius**.

Click **OK**.

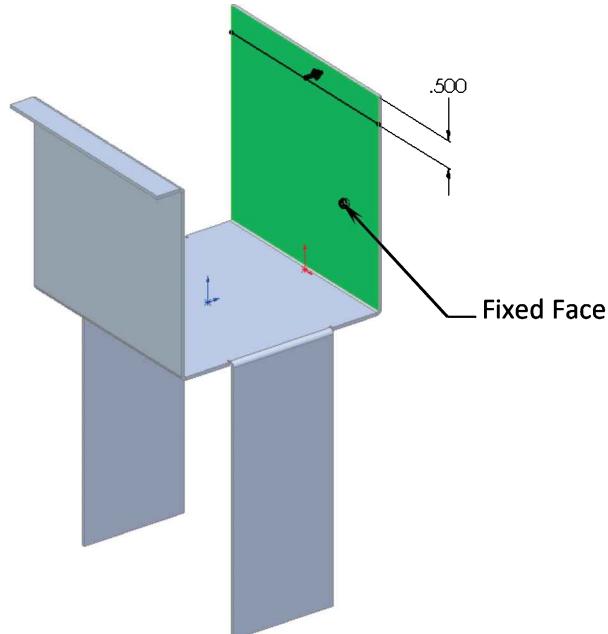
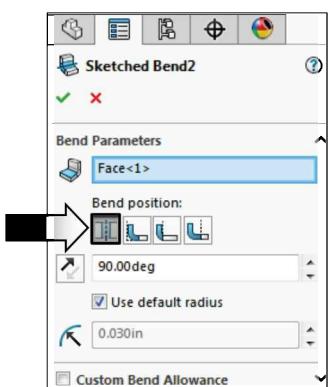


8. Creating the 2nd Sketch Bend:

Select the face on the right and open a **new sketch**  or select **Insert / Sketch**.

Sketch a **Line** as shown and add the spacing dimension to fully define the sketch.

Click  or select **Insert / Sheet Metal / Sketch Bend**.



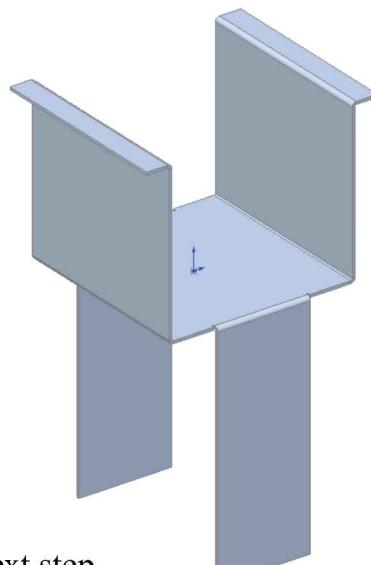
Fixed Face  select the **lower portion** of the surface, below the line.

For Bend Position, select **Bend Centerline** (default).

For Bend Angle, enter **90.00deg** (default).

Enable **Use Default Radius**.

Click **OK**.

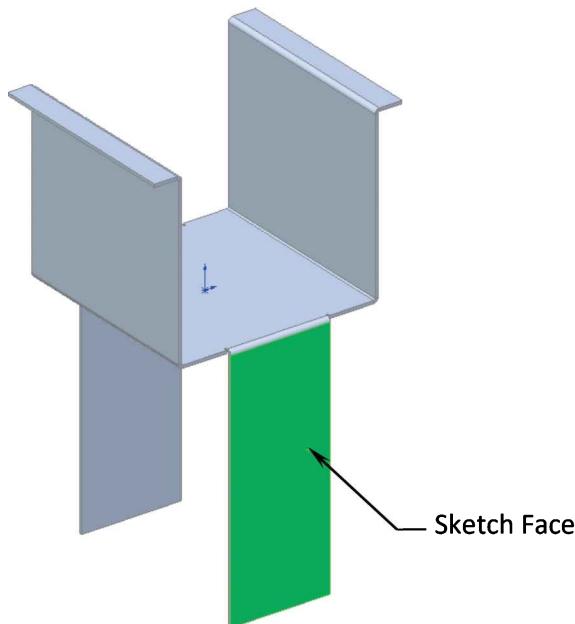


The upper portion of the flange is bent outward 90°, leaving the lower portion fixed.

Inspect your model against the image shown here.
Make any adjustments needed before going to the next step.

9. Adding holes:

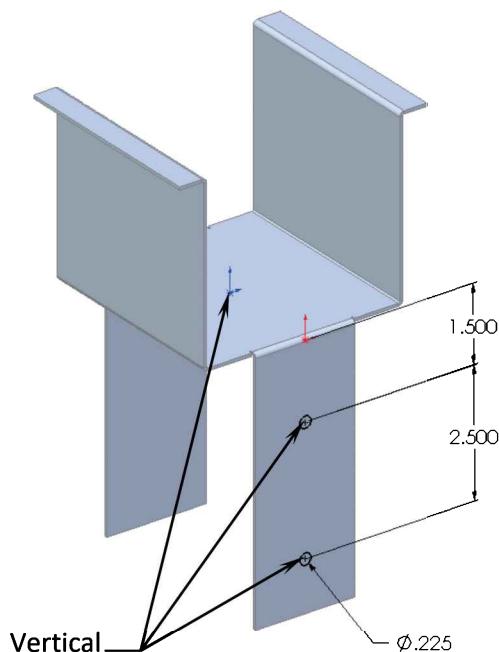
Open a **new sketch** on the side face as indicated  or select: **Insert / Sketch**.



Sketch two circles  on the face.

Add dimensions as shown to size and to locate the circles.

Add **Vertical** relations between the centers of the circles and the Origin, to fully define the sketch.



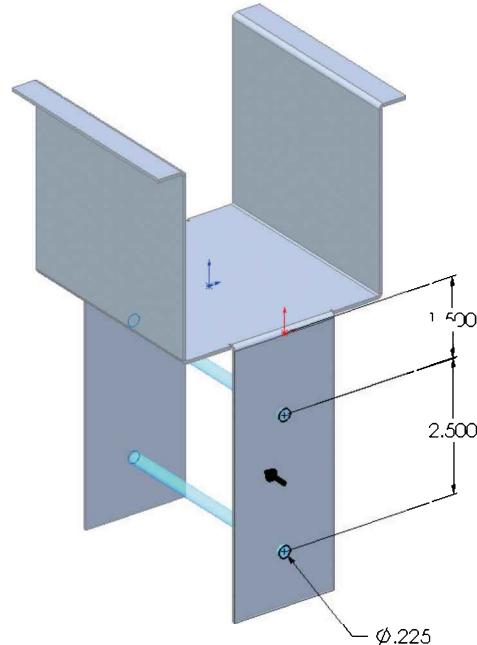
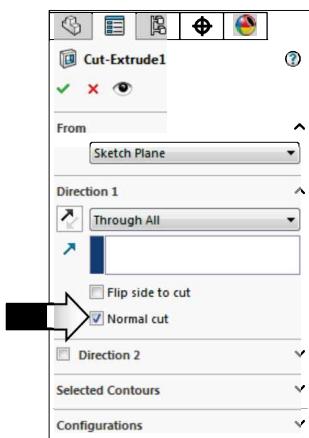
Switch to the **Sheet Metal** tab.

Click  or select Insert / Cut / Extrude.

End Condition: **Through All**.

Enable **Normal Cut** (default).

Click **OK**.

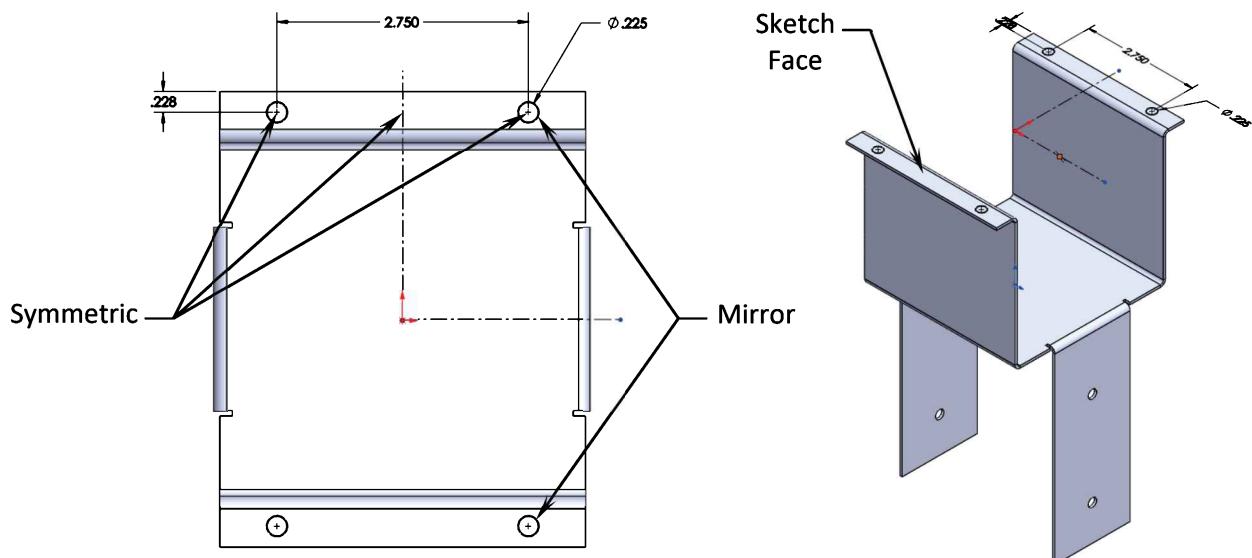


10. Adding holes on the Sketch Bend Flanges:

Select the face as noted and open a **new sketch**  or select Insert / Sketch.

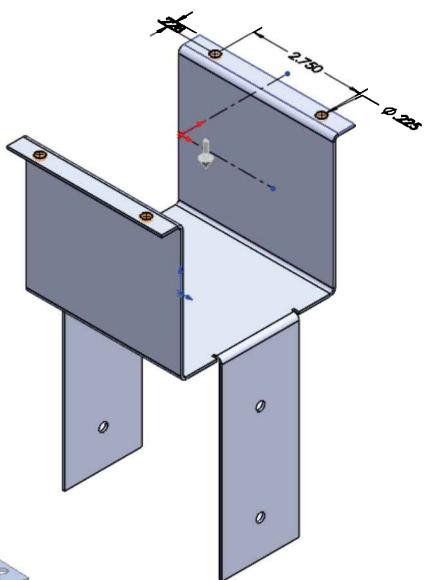
Sketch 2 Circles and mirror them as noted below.

Add the dimensions and relations to fully define the sketch.



Switch to the Sheet Metal tab and click  or select **Insert / Cut / Extrude**.

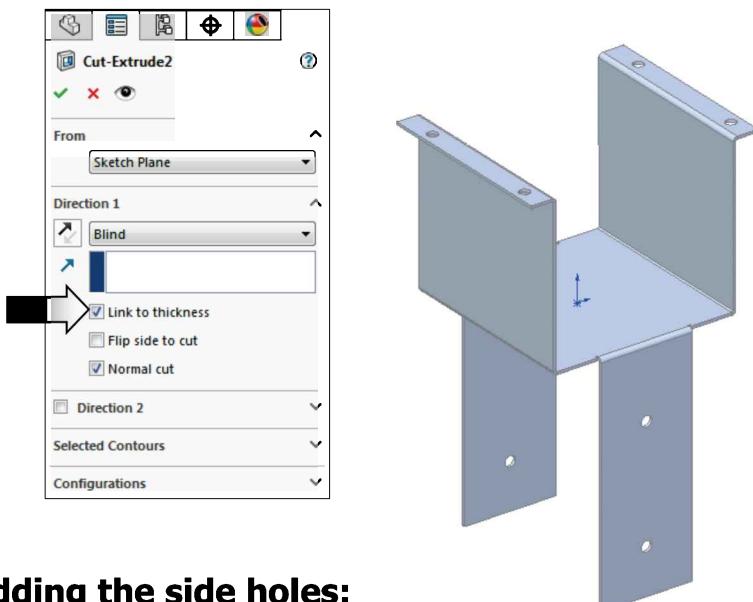
End Condition: **Blind**.



Enable the **Link To Thickness** option.

Enable **Normal Cut** (default).

Click **OK**.



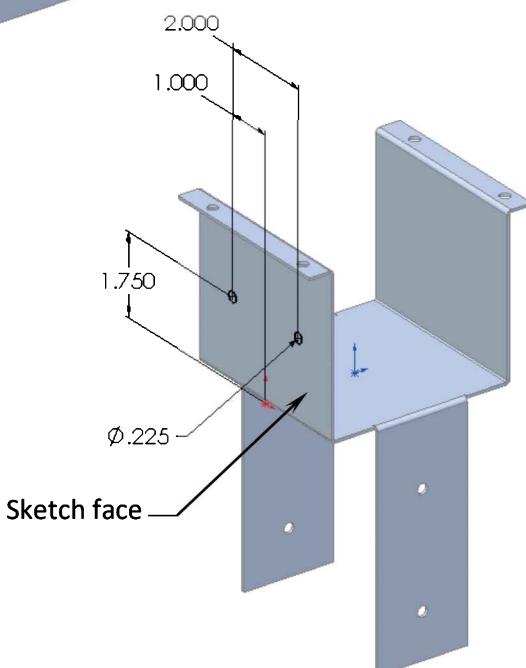
11. Adding the side holes:

Select the side face as indicated and

 or select **Insert / Sketch**.

Sketch **2 Circles** and add the dimensions as shown in the image on the right.

Add a **Horizontal** relation between the centers of the two circles to fully define the sketch.



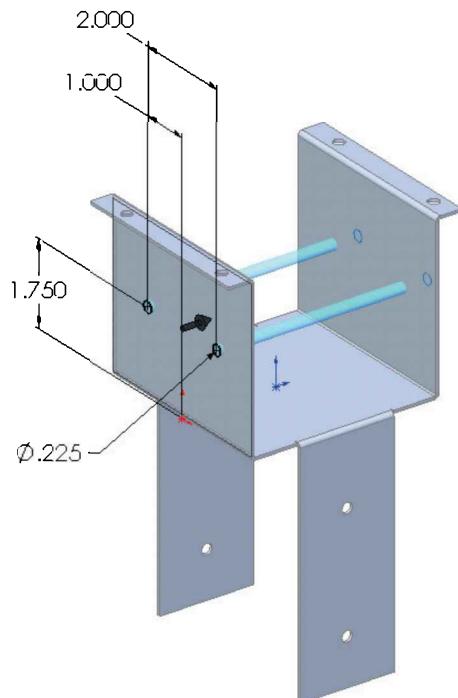
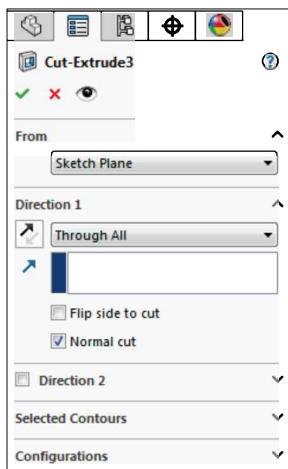
Switch to the **Sheet Metal** tab.

Click  or select **Insert / Cut / Extrude**.

End Condition: **Through All**.

Enable **Normal Cut** (default).

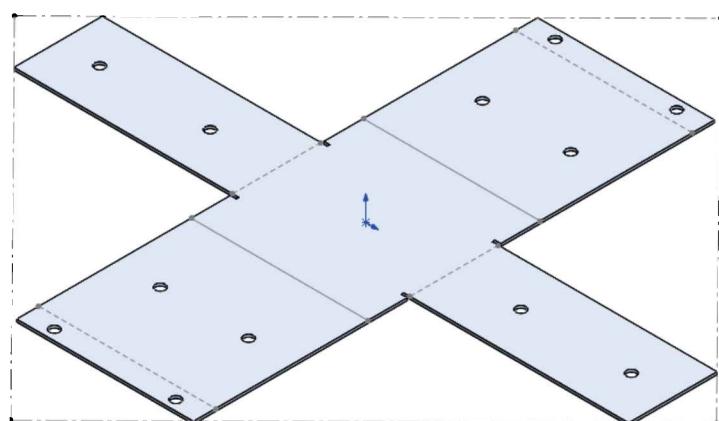
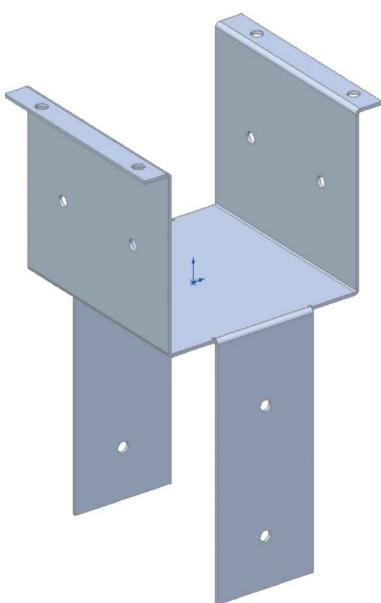
Click **OK**.



12. Switching to the Flat Pattern:

Click **Flat Pattern**  on the Sheet Metal tab.

The sheet metal part is flattened with the bend lines displayed.



13. Saving your work:

Select **File / Save As / Post Cap / Save**.

Using Sheet Metal Costing

Use SOLIDWORKS sheet metal Costing tools to determine the cost of a sheet metal part. Costing provides a comprehensive breakdown and comparison of manufacturing and material costs for sheet metal parts.

Costing Template Editor for Sheet Metal Parts

You can create and edit costing templates for sheet metal parts or bodies from the Costing Template Editor.

You can specify rates and costs for the procedures required to manufacture a sheet metal part or body in the sheet metal template. You can include customized information in the template, such as material cost and thicknesses, cost of manufacturing, and manufacturing setup costs.

You can determine how manufacturing operations affect the cost of your design. For example, you can set up templates for vendors that use different manufacturing operations.

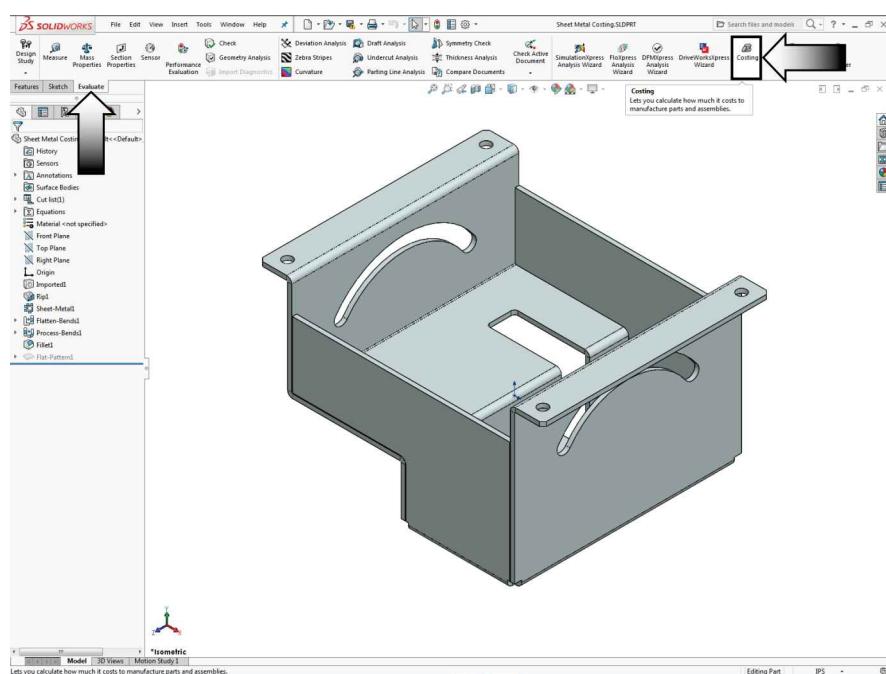
You can specify the file location for Costing templates in Tools > Options > System Options > File Locations. In Show folders for, select Costing templates to add or delete a location. The default Costing template folder is *installation directory/lang/language/Costing templates*.

1. Opening an existing sheet metal part:

Click File / Open.

Browse to the Training Files folder, locate and open the **Sheet Metal Costing** part document.

Click **NO** on the Feature-Works dialog box to close it.



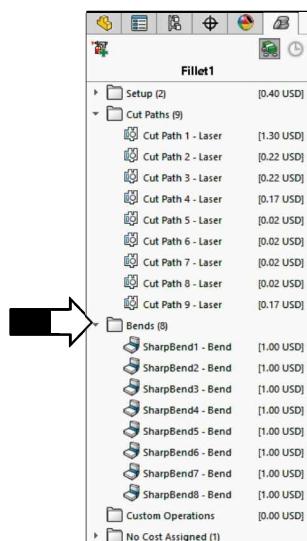
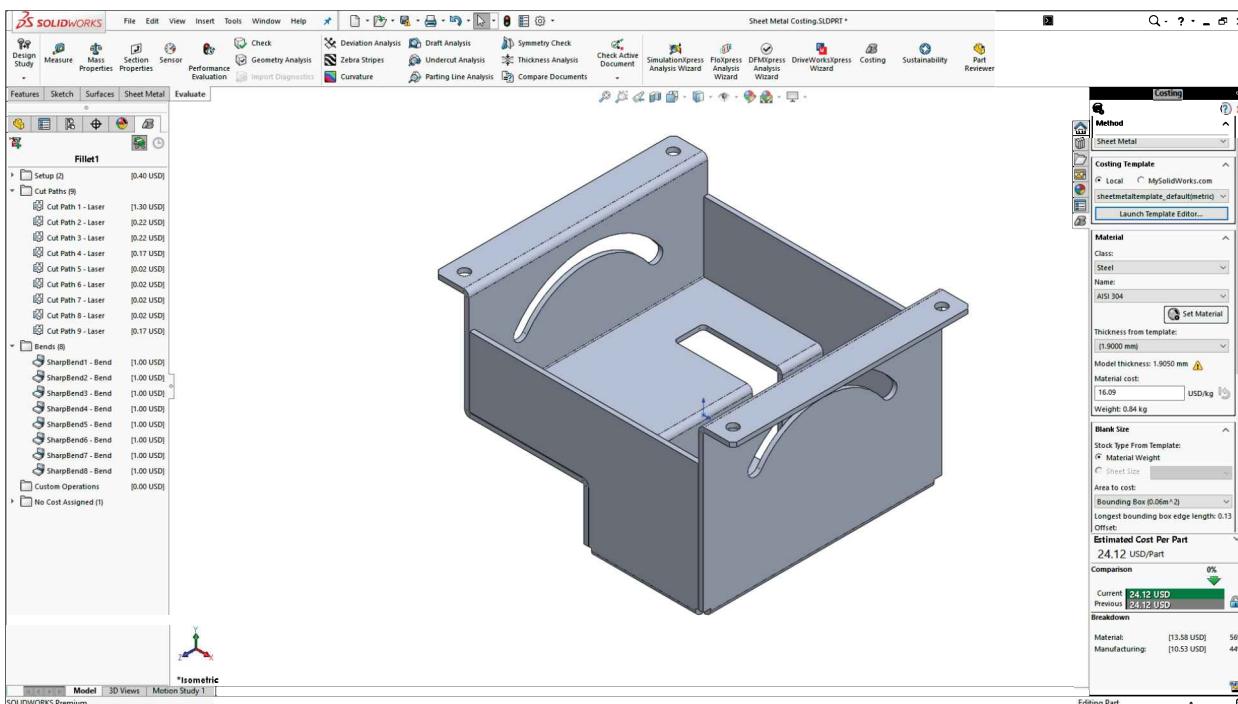
2. Inputting the information:

Switch to the **Evaluate** tab and click **Costing**.



Click the **CostingManager** tab (arrow) on the left side tree to see how the Costing tool categorizes each operation required for manufacturing the part.

Set the **Material** to **Steel AISI 304** and the **Thickness** to **1.90mm (.075in)**.
The sheet metal part is evaluated based on the selected material and processing costs as set in the default template.



The Costing tab (on the right) indicates the cost for the selected material and thickness is roughly **\$16.09 USD per kg**.

The total cost per part is **24.12 USD**.

The major processing cost (on the left) is **\$2.16 USD** for 9 laser cut paths and **\$8.00 USD** for eight bends (\$1.00 USD per bend).

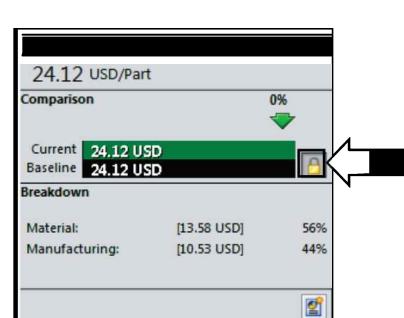
3. Setting the Baseline:

The **Set Baseline** is used to set a baseline cost for comparison. If you change the design later on, the cost is compared to the baseline cost. When you set a baseline cost, any changes to the part are considered Current and the difference is displayed. While the baseline price is set, the part is rotated, flattened, and refolded because the software is capturing images for the Costing report.

Click the **Set Baseline** button  at the lower right, under Comparison.

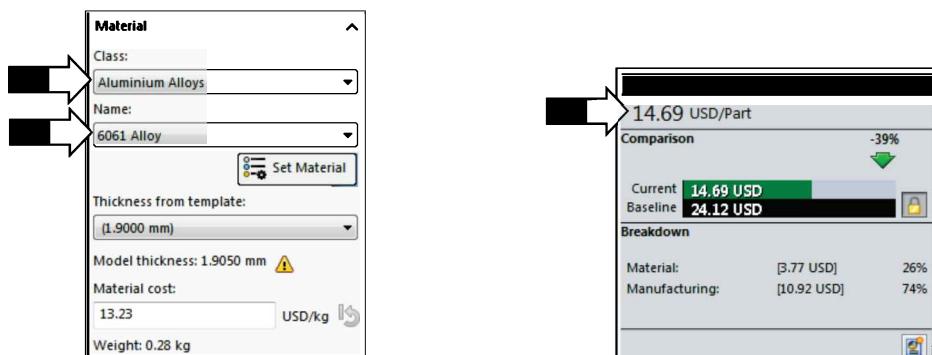
The final cost is based on the values for manufacturing steps.

The values that we are using here are for samples only.



Change the material to:
Aluminum Alloy 6061

Use the same thickness (1.90mm or .075in).



SOLIDWORKS Costing recalculates the cost based on the change in material.

The recalculated cost is now almost half of what it was for Steel (\$14.69 instead of \$24.12).

4. Saving your work:

Click **File / Save As**.

Enter **Sheet Metal Costing** for the name of the document and click **Save**.

- **General**

Use the General screen in the Costing Template Editor to set the units and currency options.

- **Material**

Use the Material screen in the Costing Template Editor to set the materials you need to manufacture the sheet metal part.

- **Thickness**

Use the Thickness screen in the Costing Template Editor to set the thickness and cost values for each class and material combination.

- **Cut**

Use the Cut screen in the Costing Template Editor to define the cost of cutting methods based on length or stroke.

- **Bend**

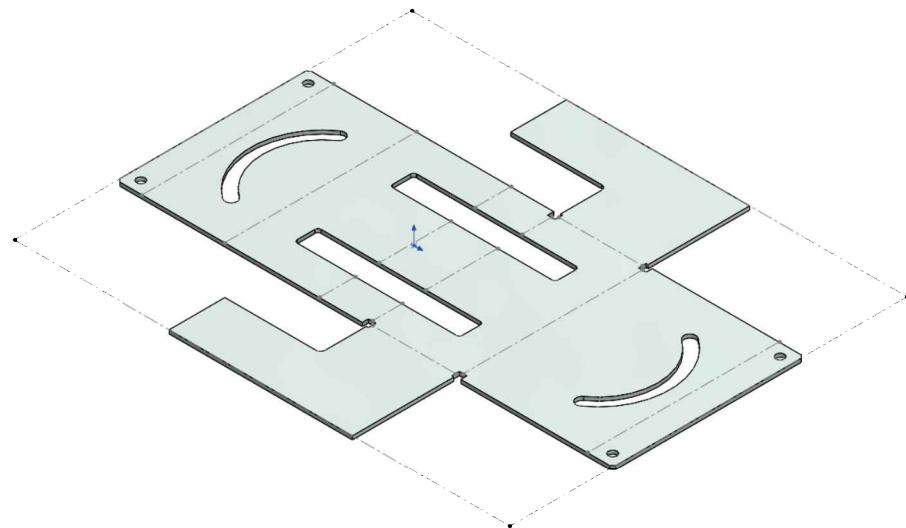
Use the Bend screen in the Costing Template Editor to define the cost of bending methods based on regular bends or hem bends.

- **Library Features**

Use the Library Features screen in the Costing Template Editor to define the cost of library features, punch features, and forming tools in the part.

- **Custom**

Use the Custom screen in the Costing Template Editor to define additional operations that contribute to a part's manufacturing cost, such as powder coating.



Questions for Review

1. A sheet metal part can have multiple thicknesses.
 - a. True
 - b. False
2. An Edge Flange can be mirrored just like any other feature.
 - a. True
 - b. False
3. A sheet metal part can be created right from the beginning using the Base flange option.
 - a. True
 - b. False
4. Auto relief option is not available when extruding a Base Flange.
 - a. True
 - b. False
5. When the Sketched Bend option is used to create a bend, you will have to specify at least two parameters:
 - a. A fixed side and a sketched line.
 - b. Fixed side and a bend angle value.
 - c. A bend radius and a bend angle value.
6. The only time when the K-Factor option can be changed to Bend Table is when extruding the Base Flange.
 - a. True
 - b. False
7. A sheet metal part can be designed from a flat sheet and other bends can be added later using Sketched Bend, Edge Flange, etc.
 - a. True
 - b. False
8. Link-to-Thickness option allows all sheet metal features in a part to have the same wall thickness and they can all be changed at the same time.
 - a. True
 - b. False

1. FALSE
2. TRUE
3. TRUE
4. FALSE
5. TRUE
6. FALSE
7. TRUE
8. TRUE

Sheet Metal Parts

Sheet Metal Parts

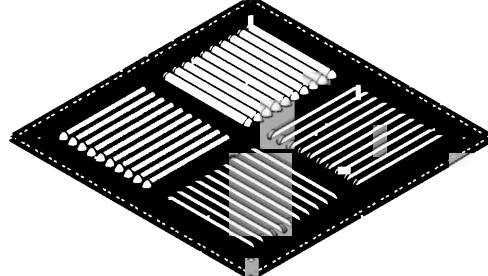
A/C Vents



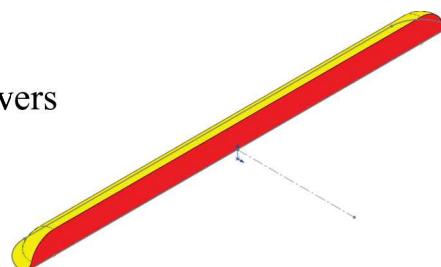
Sheet metal parts can be created using one of the following methods:

- * Create the part as a solid and then insert the sheet metal parameters such as rips, bend radius, material thickness, bend allowance, and cut relief so that the part can be flattened.
- * Create the part as a sheet metal part from the beginning by using the Base Flange command to extrude the first feature.
- * Sheet metal parameters can be applied onto the sheet metal part during the extrusion or after the fact.

This chapter will guide you through the design development of a sheet metal part as well as the use of the sheet metal and forming tool commands to create a louver form tool:

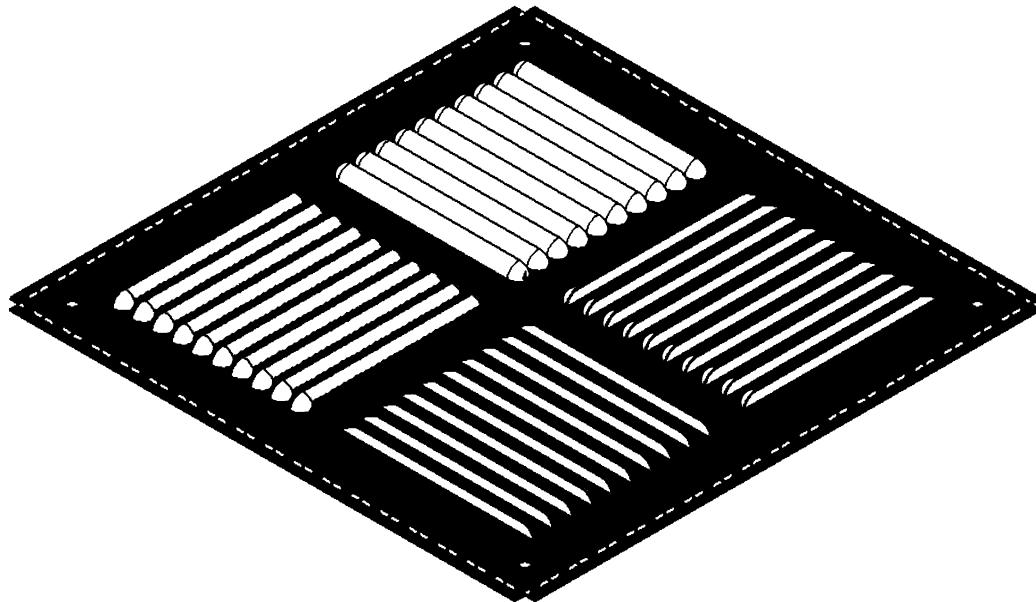


- * Creating the parent feature with the Base Flange command
- * Using the Miter Flange command
- * Create a sheet metal part in the flat or folded stage
- * Create revolved features
- * Accessing the Design Library
- * Using the forming tool to create the louvers
- * Create a linear pattern of features
- * Create a circular pattern of features
- * Create a pattern of patterned features
- * Flatten the sheet metal part



Sheet Metal Parts

A/C Vents



Dimensioning Standards: ANSI

Units: INCHES – 3 Decimals

Tools Needed:



Insert Sketch



Line



Dimension



Rectangle



Add Geometric
Relations



Linear Pattern



Base Flange



Miter Flange



Circular Pattern



Flat Pattern



Extruded Cut



Design Library

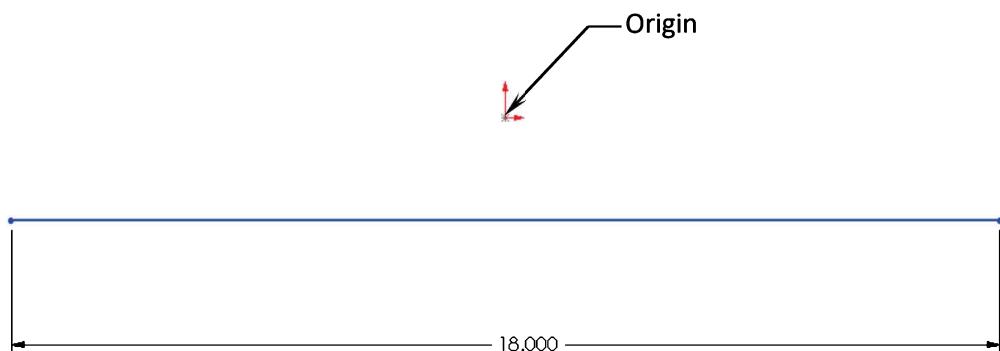
1. Sketching the first profile:

Select Front plane from the FeatureManager tree.

Click  on the Sketch toolbar or select **Insert / Sketch**.

Sketch a horizontal Line below the Origin.

Add the dimension **18.00in** for the length of the line.



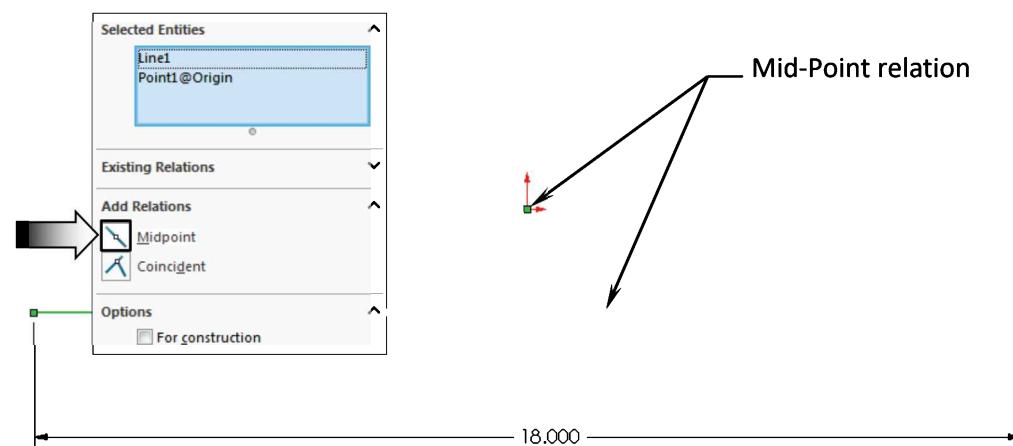
2. Adding a Midpoint relation:

Click  from the Sketch toolbar OR select **Tools / Relations / Add**.

Select the **origin point** and the **line** as indicated below.

Select **Midpoint** from the dialog box.

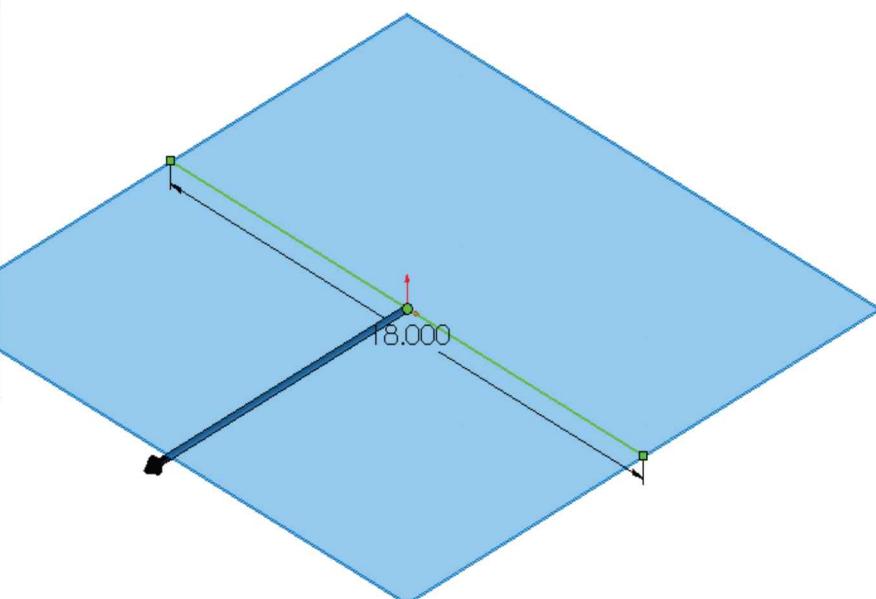
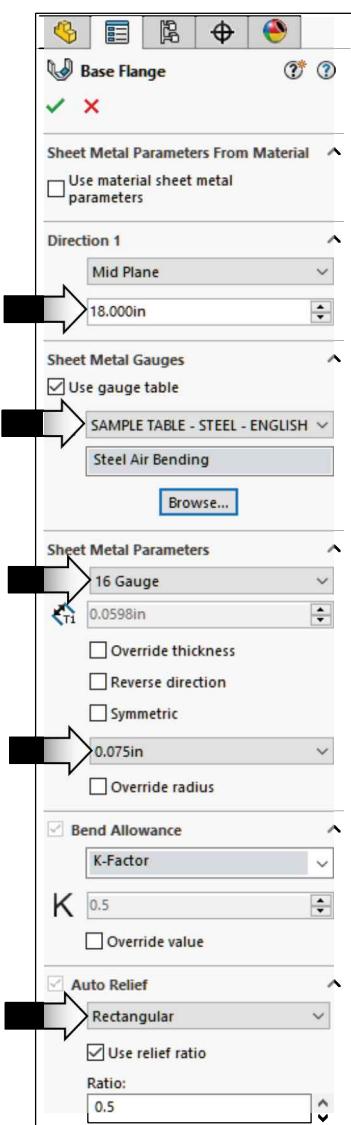
Click **OK**.



3. Extruding the Base-Flange:

Click  (Base Flange) from the Sheet Metal tab or select: **Insert /Sheet Metal / Base Flange**.

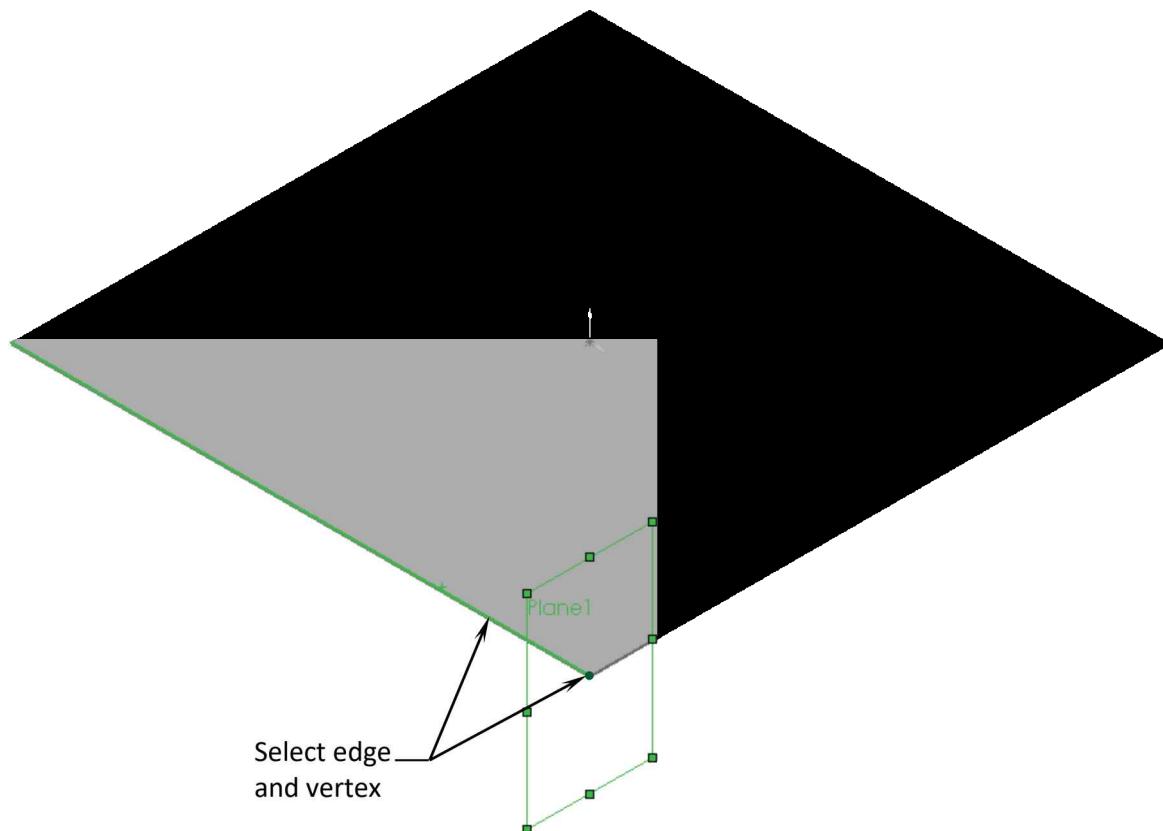
Enter / select the following:



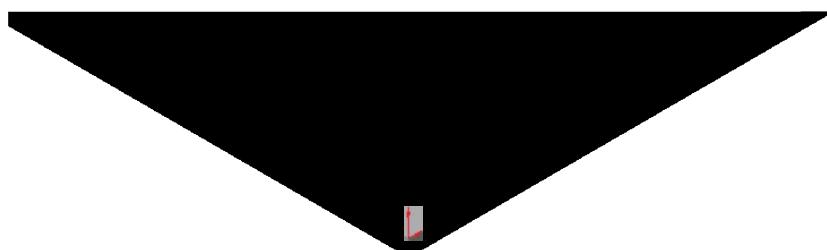
4. Creating the Miter-Flanges:

Hold the Control key, select the edge and the vertex as indicated below, and click  or select: **Insert / Sketch**.

SOLIDWORKS automatically creates a new plane normal to the selected edge and coincident to the vertex.

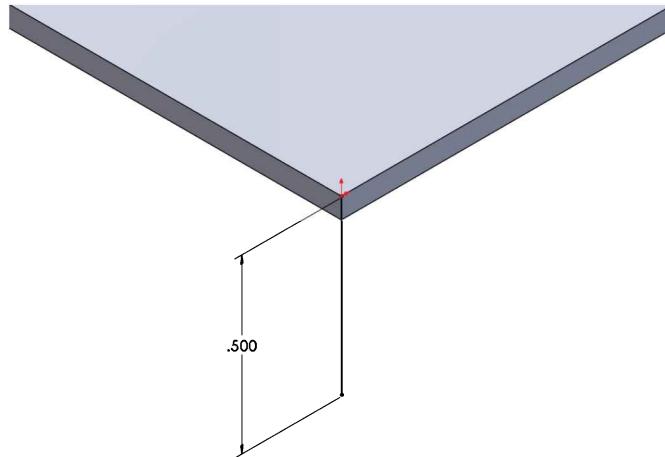


Click **Zoom-to-Area**  from the View toolbar and zoom in on the corner as shown below.



Sketch a vertical **Line** starting at the upper corner.

Add a **.500in** dimension for the length of the line.



Click  (**Miter-Flange**) from the Sheet Metal toolbar or select:
Insert / Sheet Metal / Miter Flange.

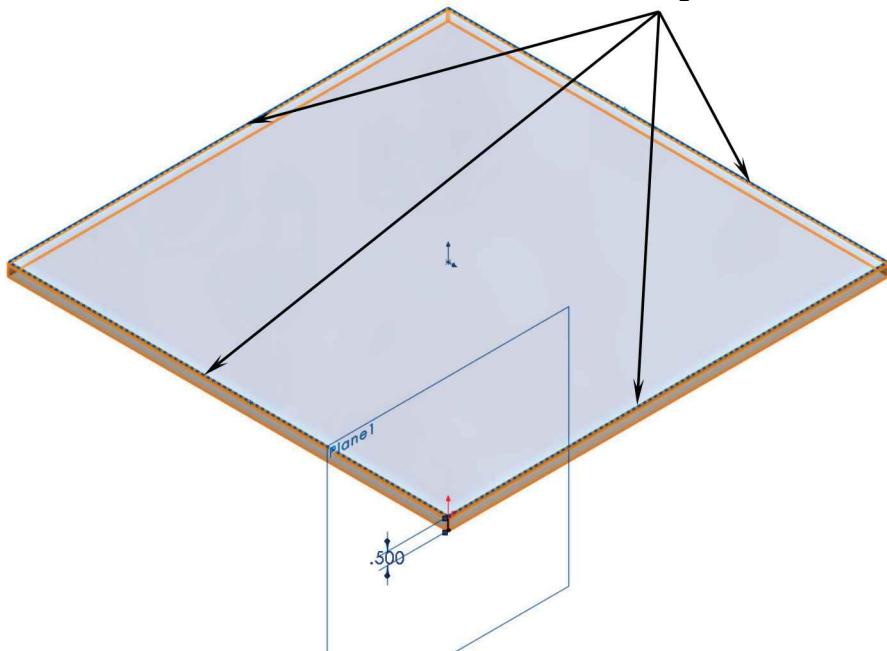
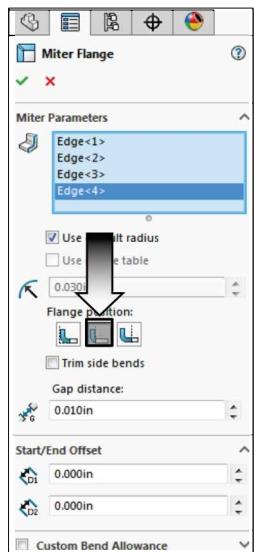
Select the **4 upper edges** as indicated.

Click **Material Outside** under Flange Position.

Set Gap Distance to **.010 in**.

Click **OK**.

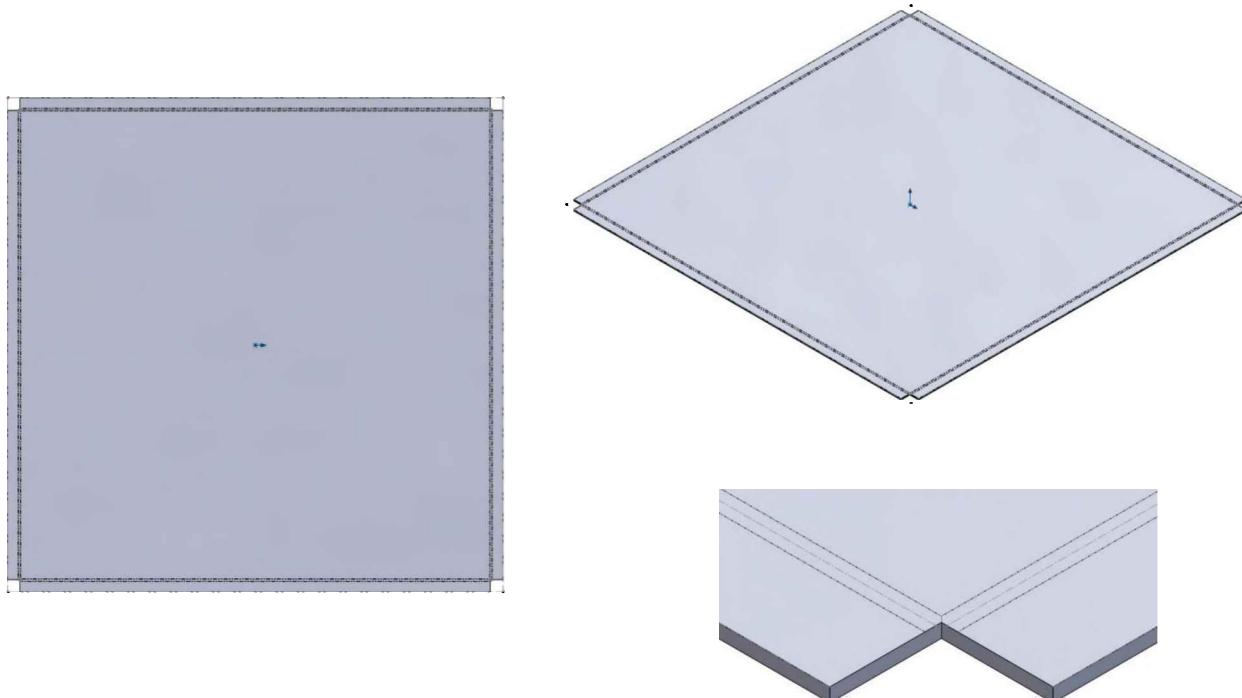
Select the top
4 edges



5. Flattening the part:

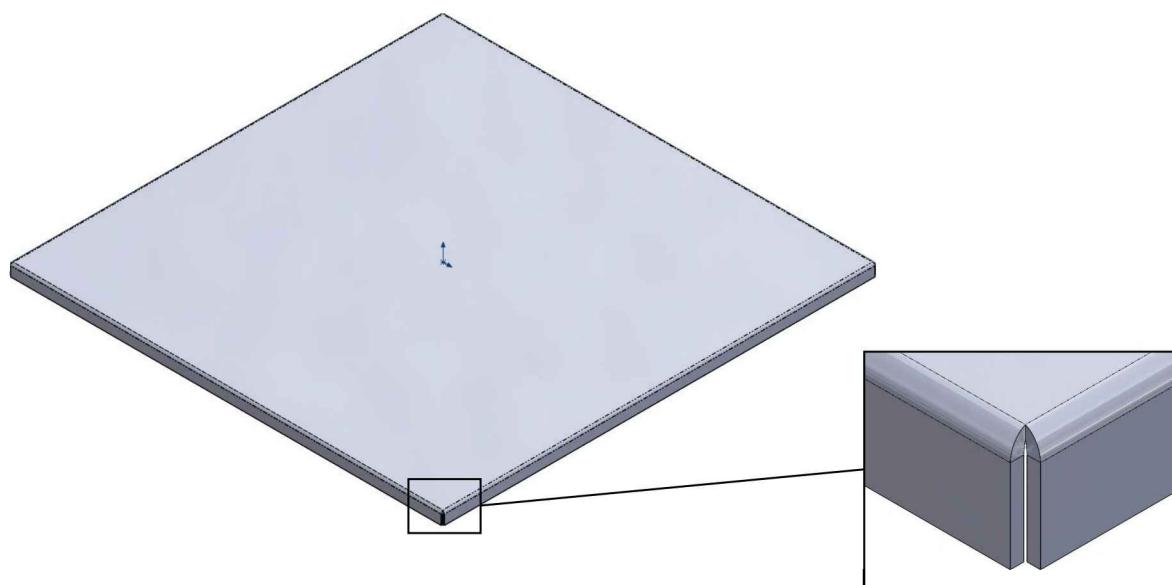
The Flat Pattern can be toggled at any time during or after the part is created.

Click  (Flatten) from the Sheet Metal tab to flatten the part.

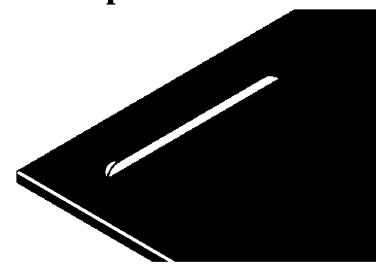


6. Switching back to the folded model:

Click  (Flatten) again to return the model back to the folded stage.



7. Saving your work: Save your model as **Sheet Metal Vents.sldprt**



8. Creating a new Forming Tool – The Louver:

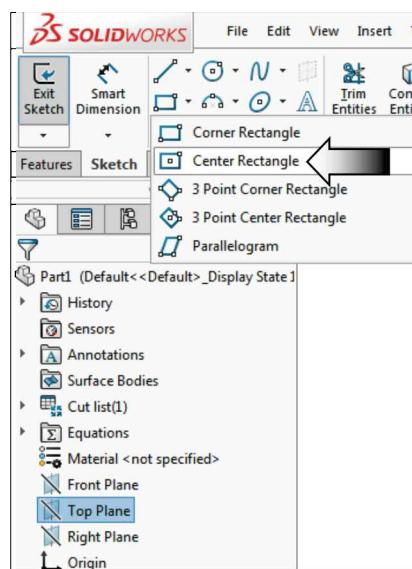
Start a new Part file: click **File / New / Part / OK**.

Set Units to **Inches – 3 Decimals** (Tools/Options/Document Properties/ Units).

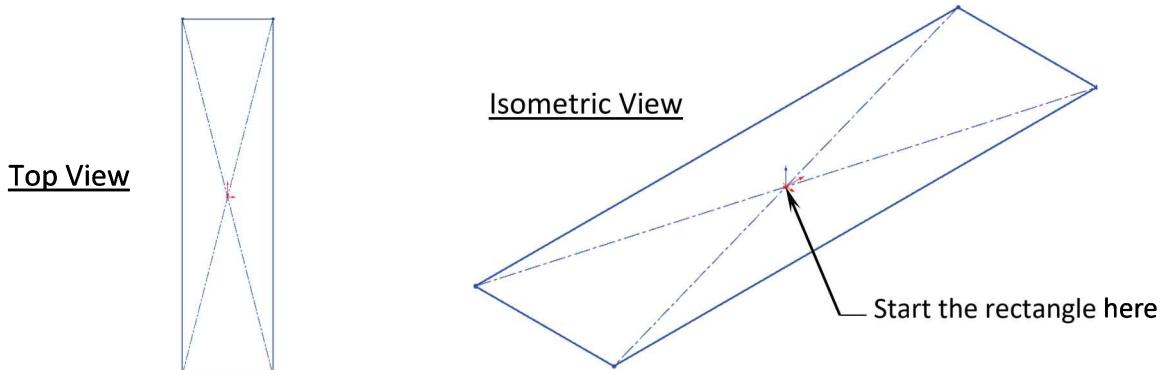
9. Sketching on the TOP reference plane:

Select the Top plane and click **Insert / Sketch** from the Sketch toolbar.

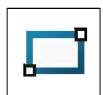
Select the **Center Rectangle** from the Sketch-Tools tab.



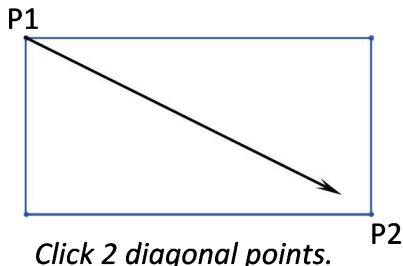
Sketch a **Center Rectangle** that is centered on the Origin, as shown below.



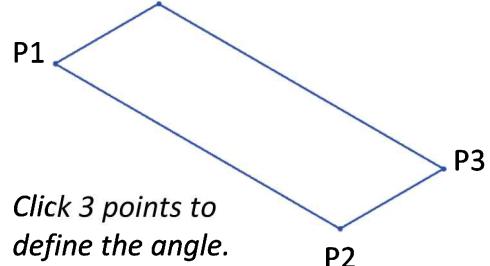
10. Other Rectangle options:



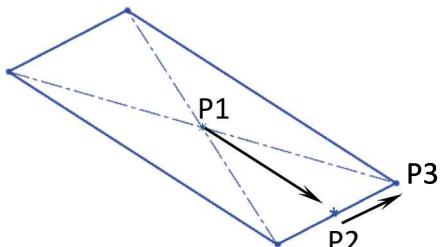
Corner Rectangle



3-Point Corner Rectangle



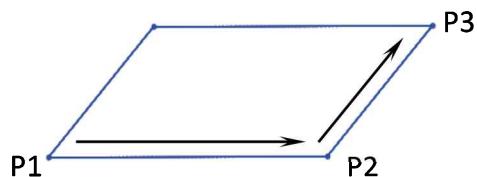
3-Point Center Rectangle



Start at Center point and click 2 other points to define the angle.



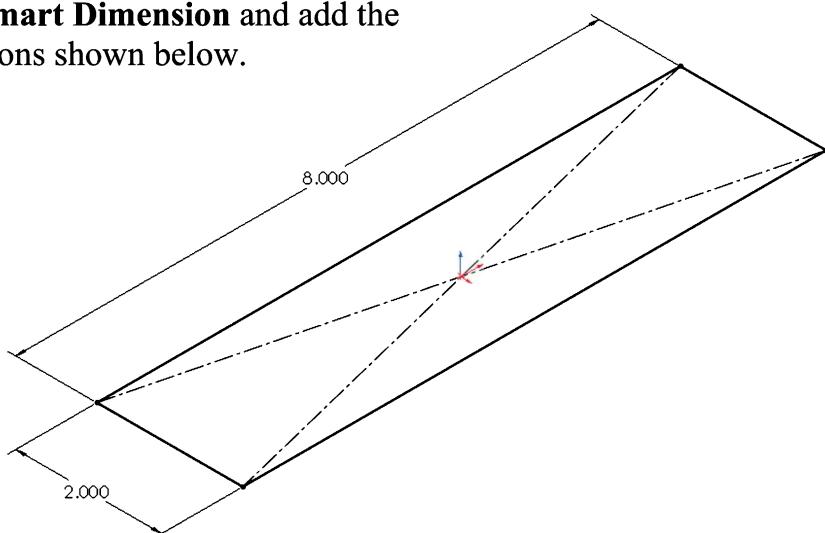
Parallelogram



Click 3 points to define the parallelogram.

11. Adding dimensions:

Click **Smart Dimension** and add the dimensions shown below.



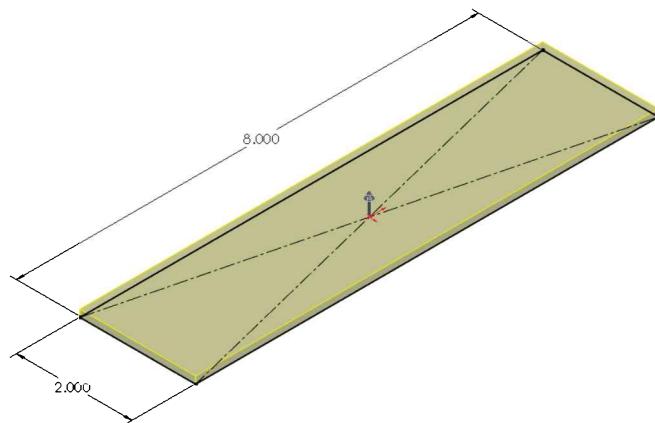
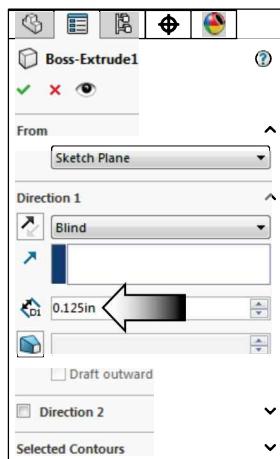
12. Extruding the Base:

Click  (Extruded Boss/Base) and fill in the following parameters:

End Condition: **Blind**.

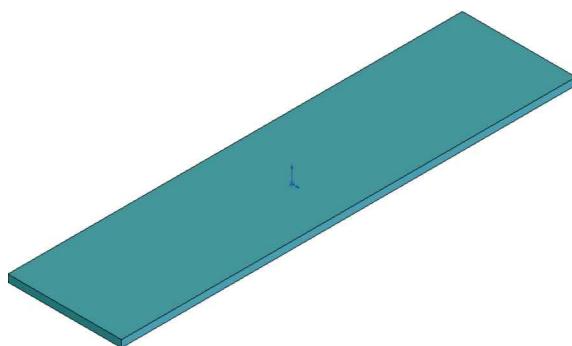
Depth: **.125in**.

Click **OK**.



Forming Tools

Forming tools are solid parts used to bend, stretch, and form sheet metal.

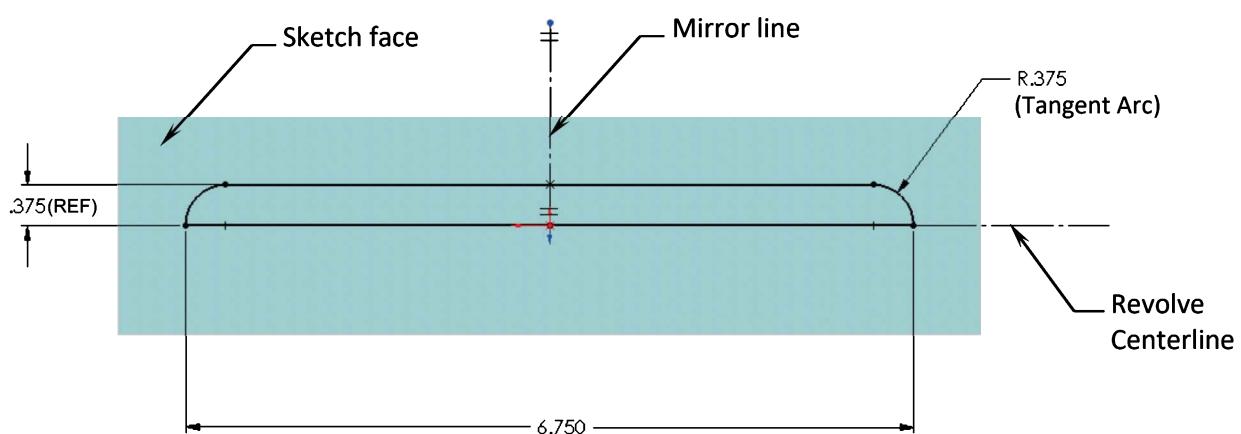


13. Building the Louver's body:

Select the upper face of the part and open a new sketch .

Change to the **Top View Orientation**  (Ctrl+5).

Sketch the profile of the Louver-Forming tool and add dimensions shown below:



14. Revolving the louver body:

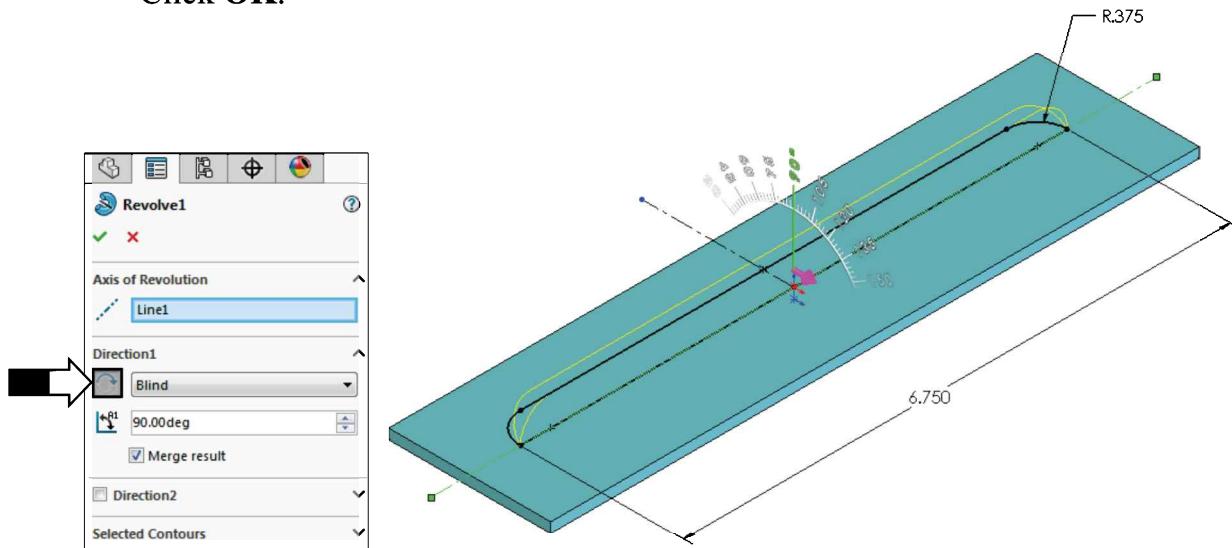
Select the **horizontal centerline** and click  (Revolve Boss/Base).

For Revolve Type, select **Blind**.

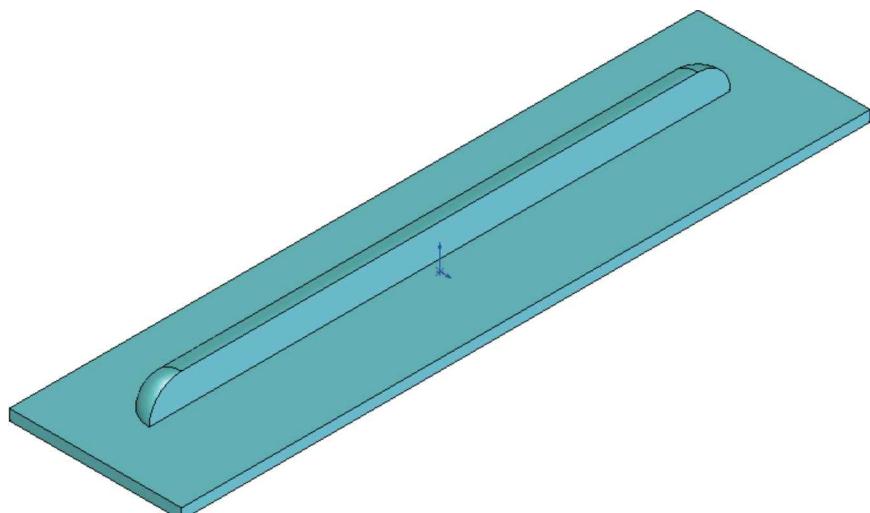
For revolve Angle, enter **90°**.

Toggle  (Reverse) and make sure the preview looks like the one below.

Click **OK**.



Rotate the model and inspect the model from different angles to ensure it is protruded to the correct side.



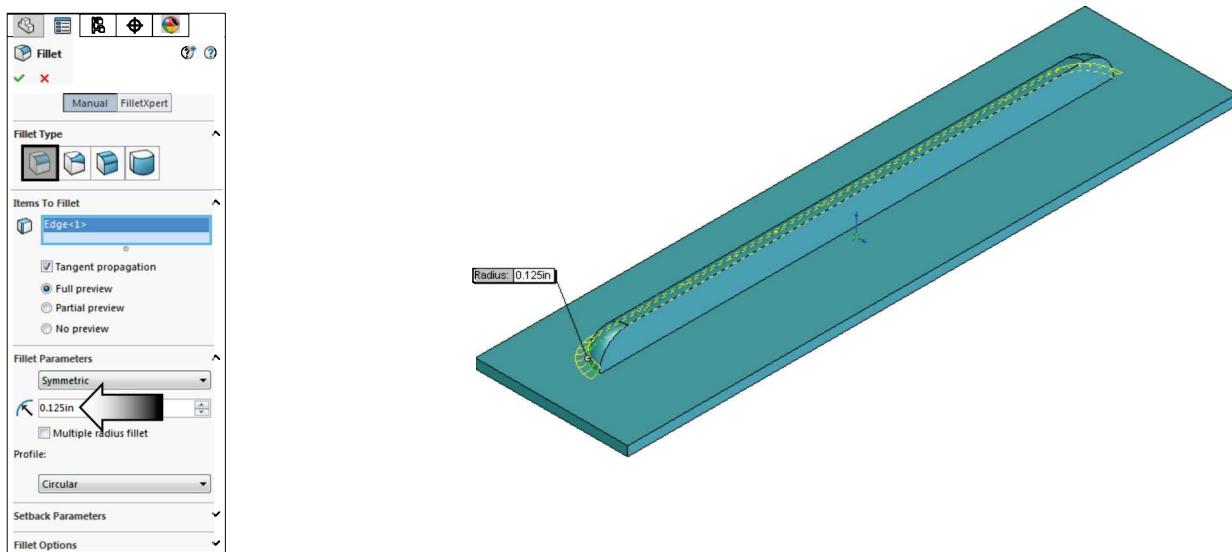
15. Adding a fillet at the base:

Click **Fillet**  and enter **.125 in.** for Radius.

Select the edge as indicated.

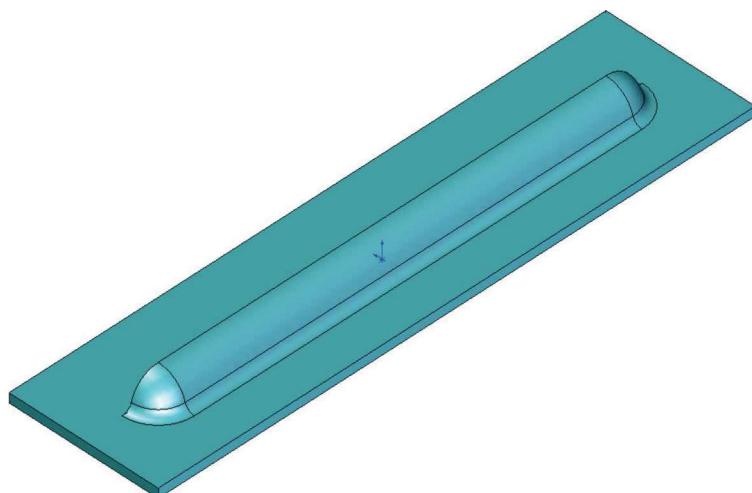
Select **Tangent Propagation** checkbox (default); the system applies the same fillet to all connected tangent edges.

Click **OK**.



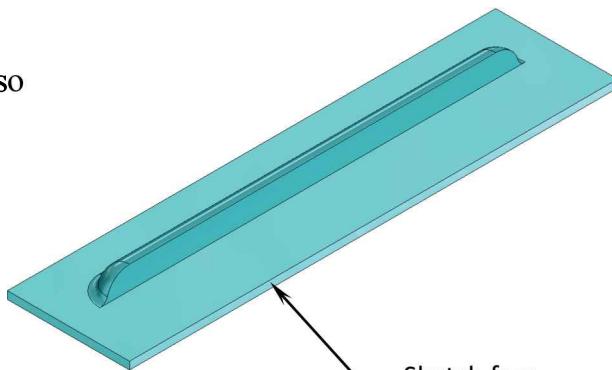
Rotate the model to see the resulting fillet from the backside.

The new fillet should run around the back side but not the front.



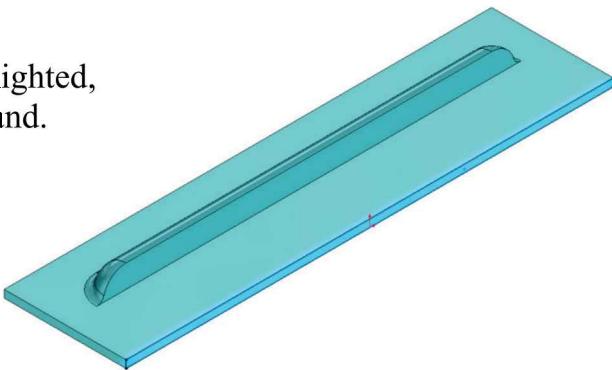
16. Removing the base:

The rectangular plate was created so that a fillet can be added between the two features. We no longer need it at this point.



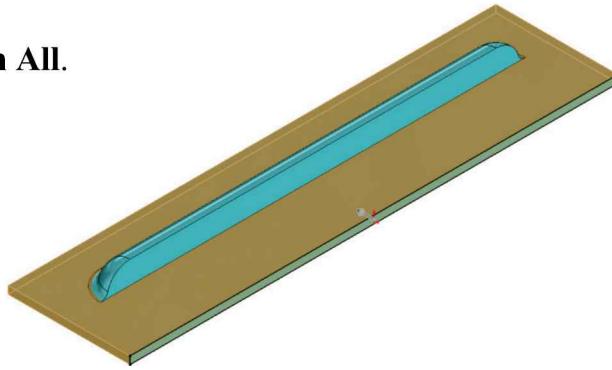
Select the side face of the part and open a **new sketch**.

While the side surface is still highlighted, press the **Convert Entities** command. The selected face is converted to a rectangle.

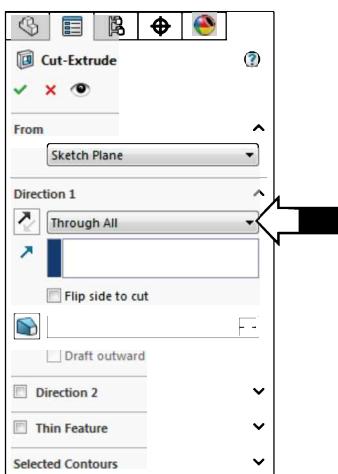


Switch to the **Features** tab and click the **Extruded Cut** command.

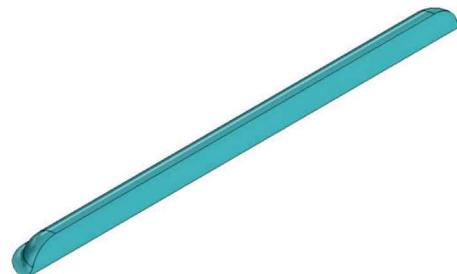
For End Condition select **Through All**.



Click **OK**.



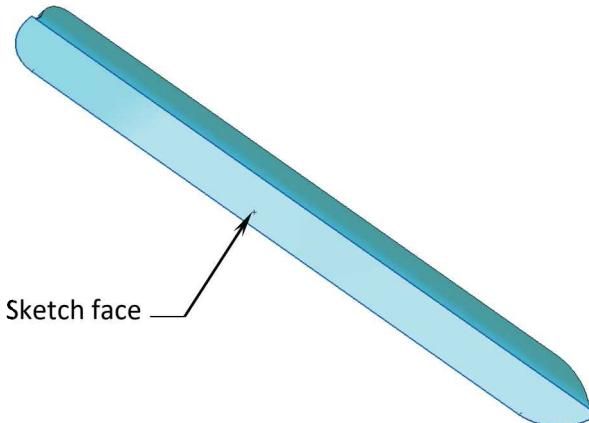
The base is removed and only the form tool portion is kept.



17. Creating the Positioning Sketch:

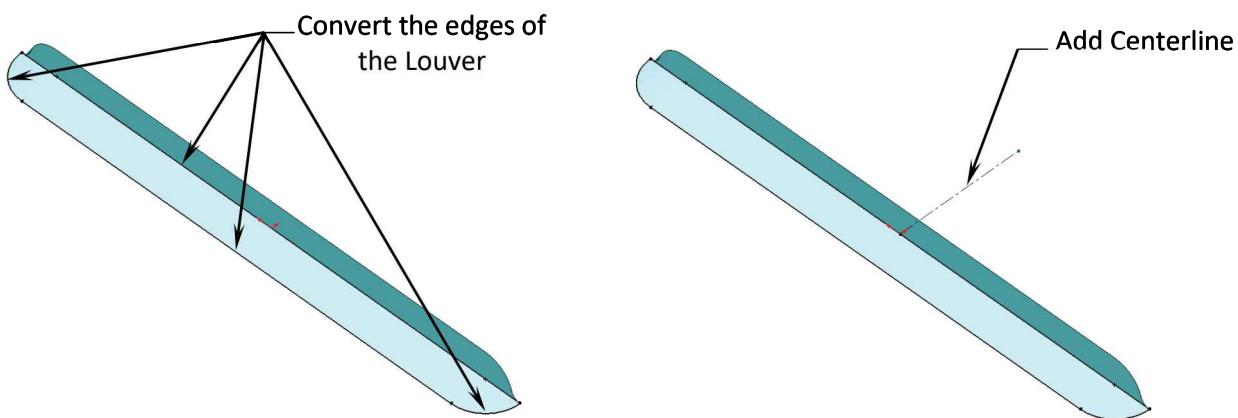
The Positioning-Sketch displays the preview of the Form tool while it is being dragged from the Design Library. Its sketched entities can be dimensioned to position the Formed feature.

Select the bottom face of the part and open a new sketch .



Select the bottom face of the part and click **Convert Entities** .

Add a **Centerline**  as shown to assist with positioning the tool when it is placed on a sheet metal part.



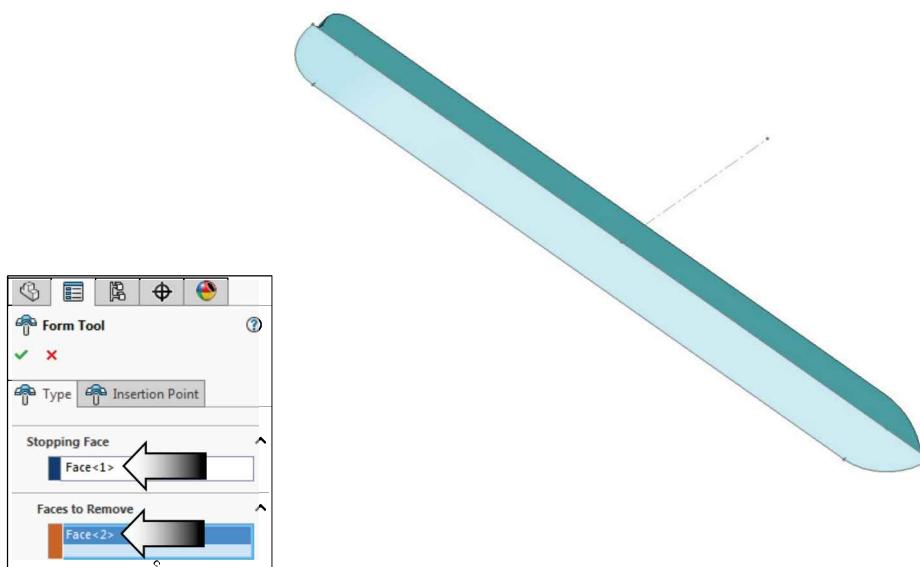
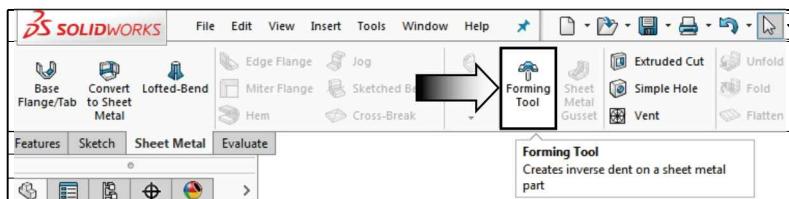
Exit the sketch or click .

Rename the sketch to **Position Sketch** from the FeatureManager tree.

18. Establishing the Stop and Remove faces:

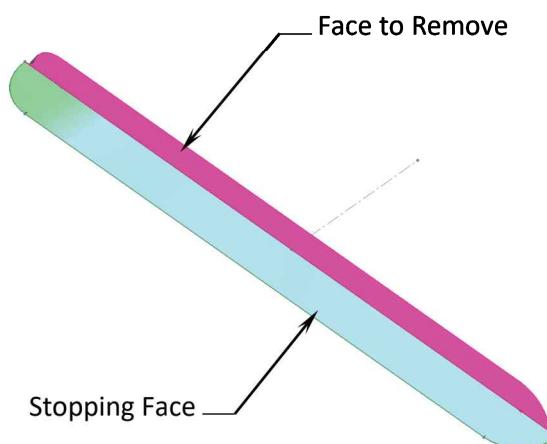
In order for the forming tools to work properly, a set of Stopping Face and Removing Faces will have to be established prior to saving as a forming tool.

Change to the Sheet Metal tab and click the **Forming Tool** command.



For **Stopping Face**, select the **bottom face** of the part.

For **Faces to Remove**, select the **face** on the right side of the part.



Click **OK**.

19. Saving the Forming Tool:

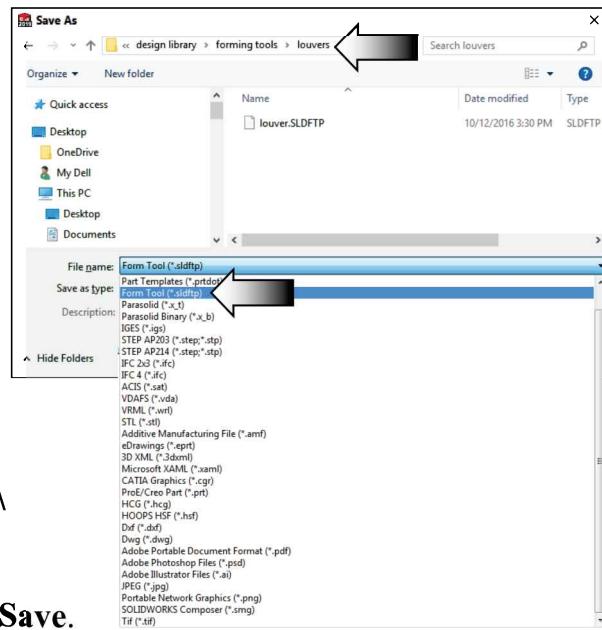
Click: File / Save As.

Enter **Long Louver** for the name of the file.

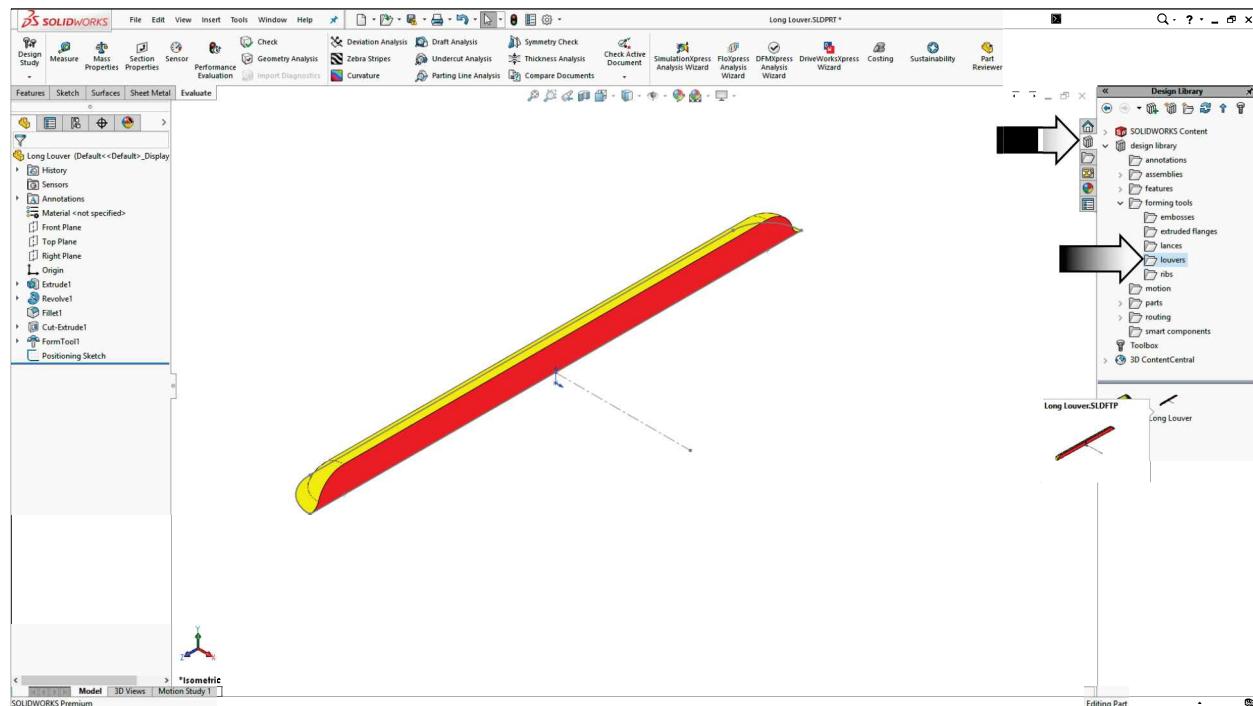
Select **Form Tool (*.sldftp)** in the Save As Type.

Browse to the following directories:
Program Data\ SolidWorks
SolidWorks 2024\ Design Library
Forming Tools\Louvers.

Expand the Louver folder and click **Save**.



NOTE: The Design Library is a Hidden Folder (Explorer / Organize / Folder and Search Options / View / Show Hidden Files, Folders and Drives).

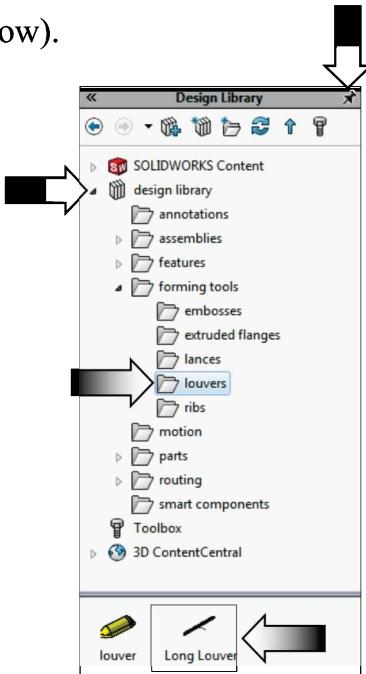
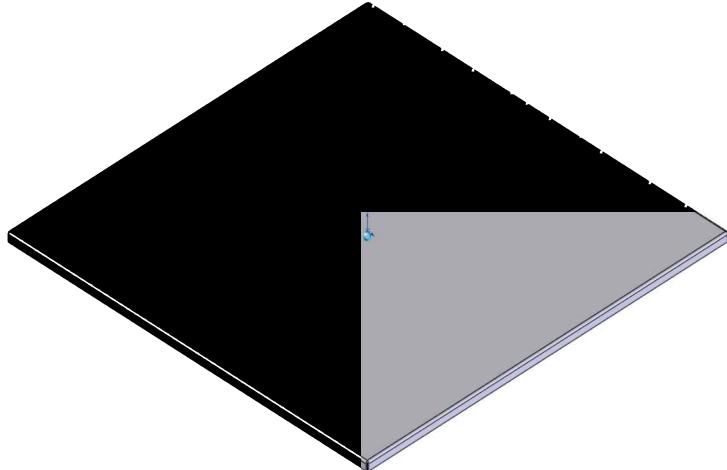


After the forming tool is saved, it can be accessed through the Task Pane by dragging and dropping it from the Design Library folder.

20. Opening the previous part:

Open the **Sheet Metal Vents** that was saved earlier (**Alt+Tab**).

Click the pushpin to lock the Task Pane in place (arrow).



Expand the **Design Library** folder

Expand the **Forming Tool** folder and double click on the **Louvers** folder to see its content.

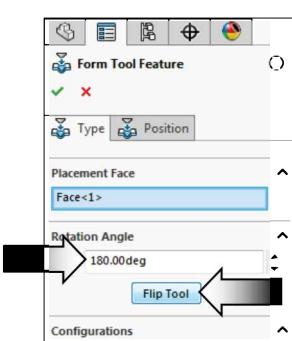
Hover the mouse cursor over the name **Long Louver** to see the preview of the Forming tool.

21. Applying the form tool:

Drag the **Long Louver** form tool from the Design Library and drop it approximately as shown.

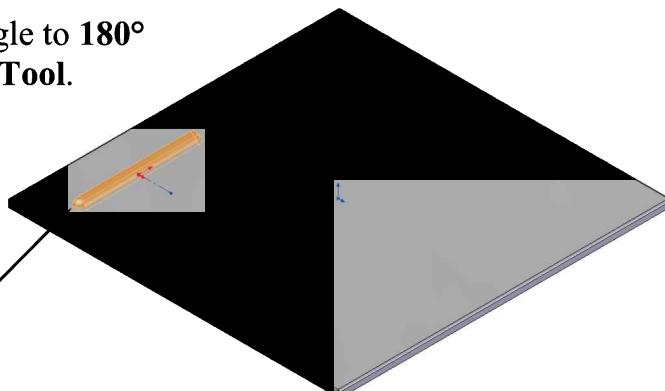
Forming Tools

Forming tools should be inserted from the Design Library window and applied onto parts with sheet metal parameters such as material thickness, bend allowance, fixed face, cut relief, etc...



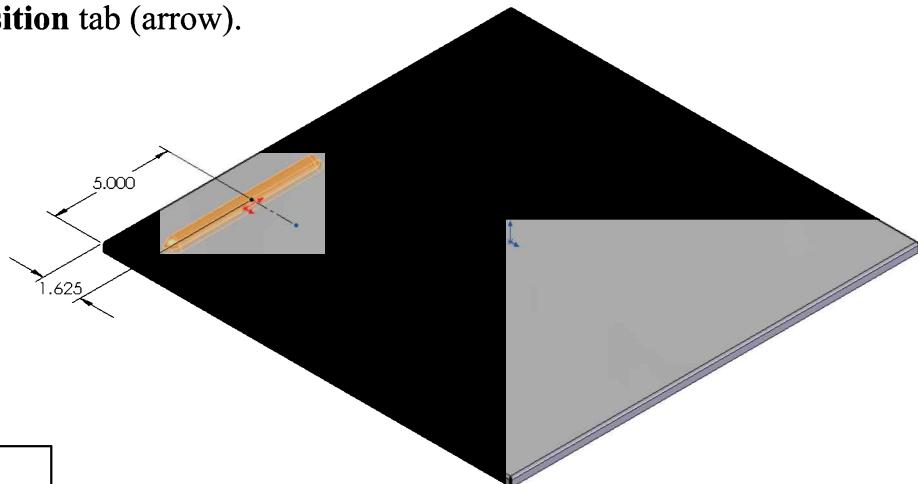
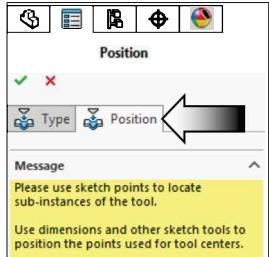
Change the angle to **180°** and click **Flip Tool**.

Place the Louver here



22. Positioning the form tool:

Click the **Position** tab (arrow).



Push / Pull

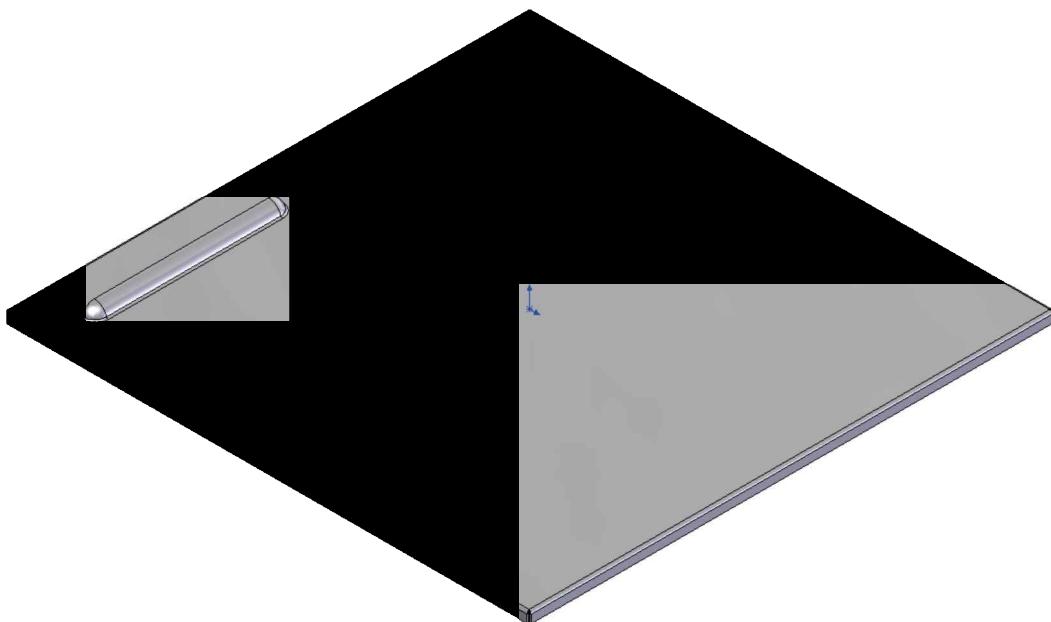
While dragging the Forming Tool from the Design Library, (still holding the mouse button) press the TAB key to reverse the direction from push to pull.

Add the locating dimensions to the outer edges of the model to fully define this sketch.

Use the outer edges of the sheet metal part when adding the dimensions.

Click **OK**.

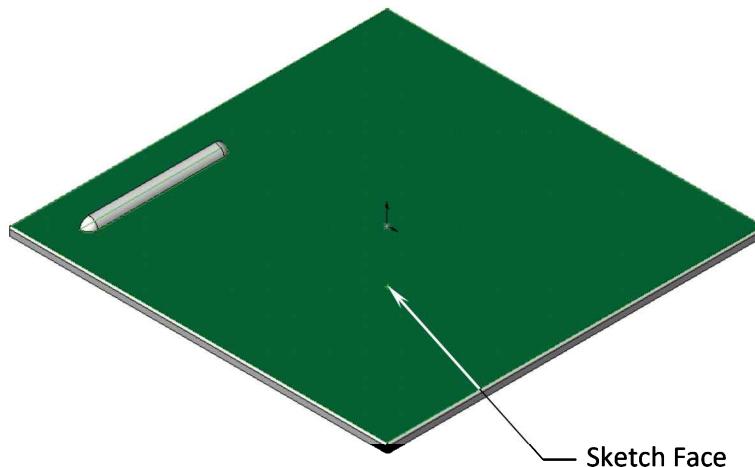
The Louver feature is formed.



23. Adding a mounting hole:

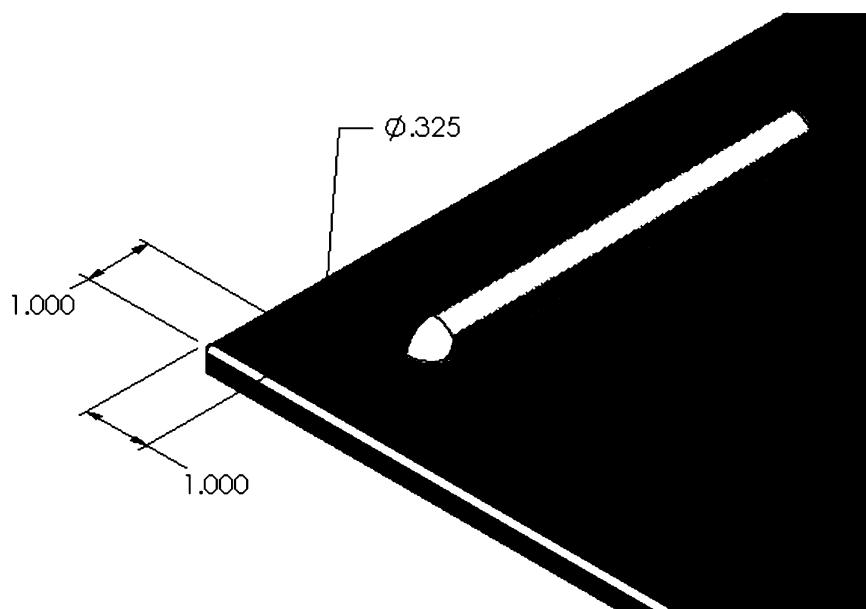
Select the upper face of the part as indicated.

Click  to open a new sketch or select **Insert / Sketch**.



Sketch a **Circle** on the upper left side of the louver.

Add dimensions to fully define the sketch.



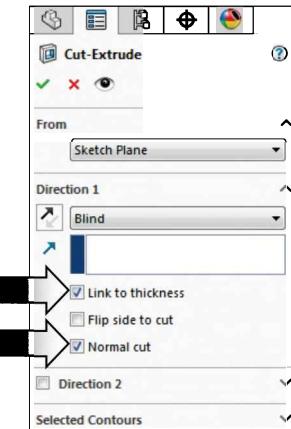
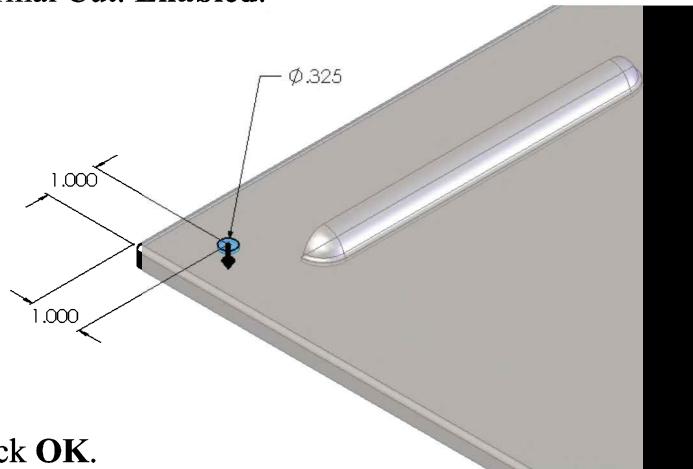
24. Extruding a cut:

Click  or select Insert / Cut / Extrude.

End Condition: **Blind**.

Link to Thickness: **Enabled**.

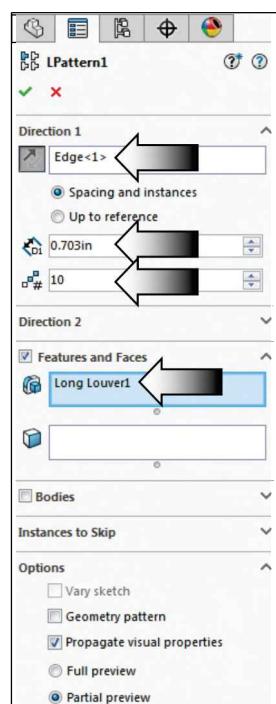
Normal Cut: **Enabled**.



Click **OK**.

25. Creating a Linear Pattern:

Click  or select Insert / Pattern Mirror / Linear Pattern.



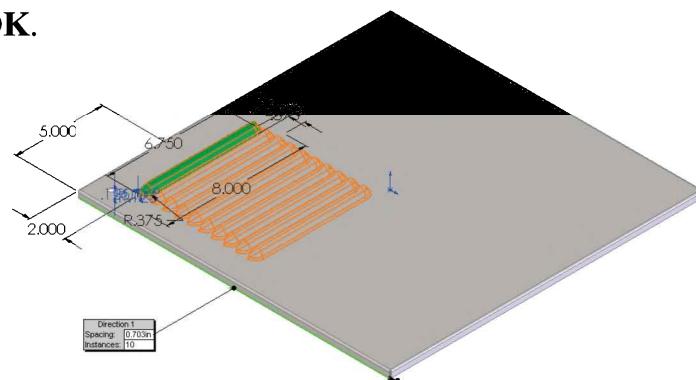
Select the **bottom edge** as Pattern Direction.

Enter **.703in** as Spacing.

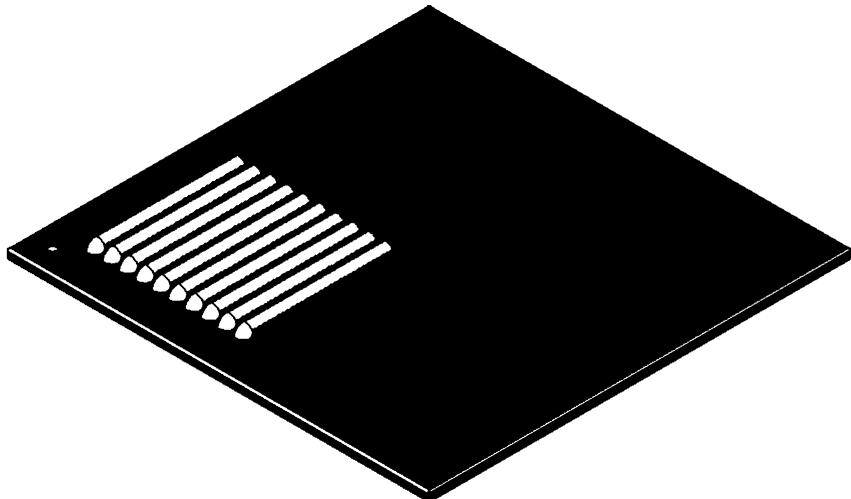
Enter **10** as Number of Instances.

Select the **Long Louver** as Features to Pattern.

Click **OK**.



The completed Linear Pattern.



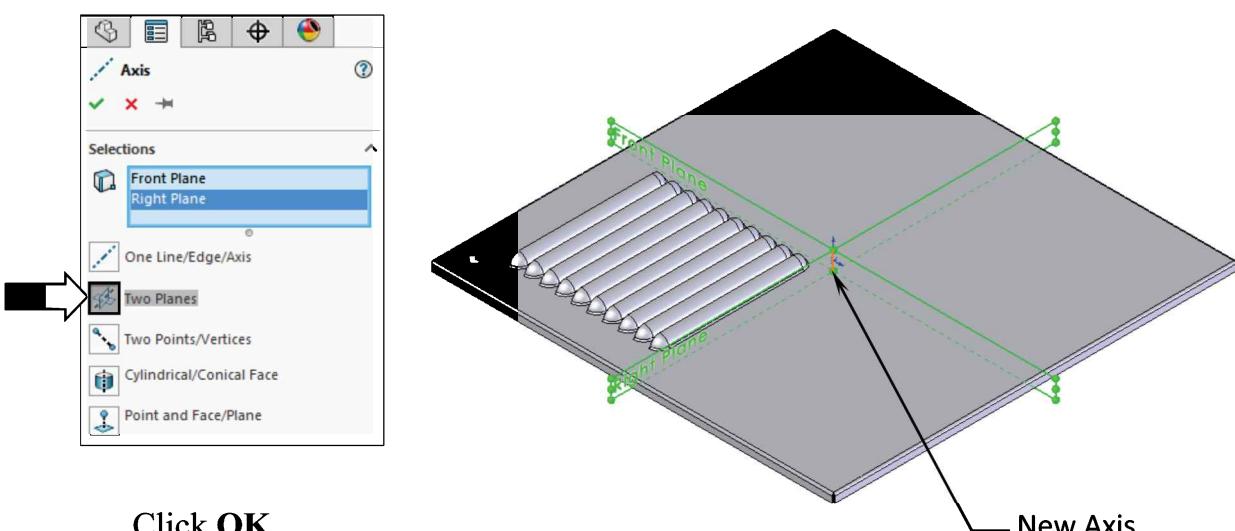
26. Creating an Axis:

An axis can be created at any time so features can be arrayed around it. In this case, an axis in the center of the part will be made and used as the center of the next Circular Pattern.

Click or select **Insert / Reference Geometry / Axis**.

Click the **Two Planes** option .

Select **Front** and **Right** planes from FeatureManager tree.



Click **OK**.

27. Creating a Circular Pattern:

Click  or select Insert / Pattern Mirror / Circular Pattern.

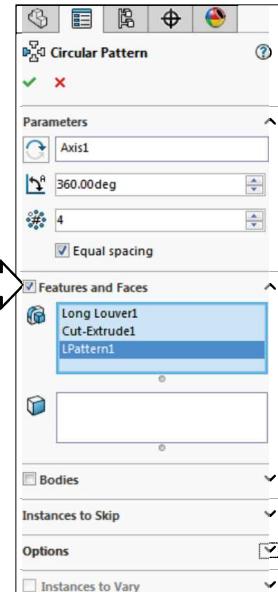
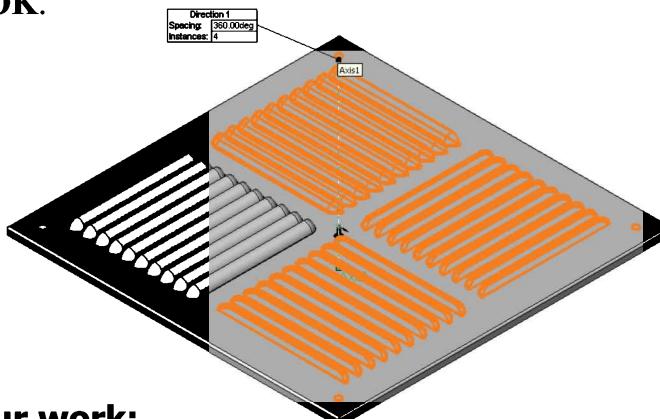
Select the new Axis for Pattern Axis.

Enter 360deg. for Pattern Angle.

Enter 4 for Number of Instances.

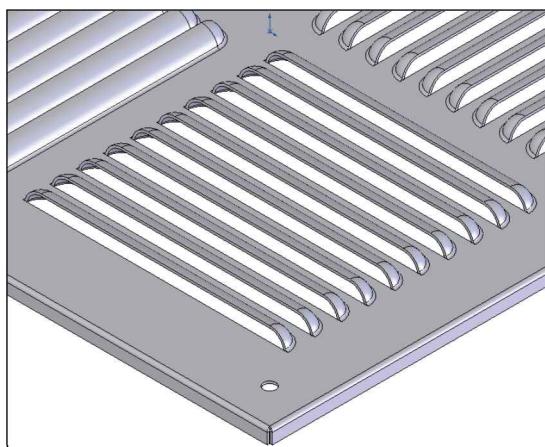
For Feature to Pattern select the **Cut-Extrude1**, **LPattern1** and the **Long Louver** from the Feature tree.

Click **OK**.

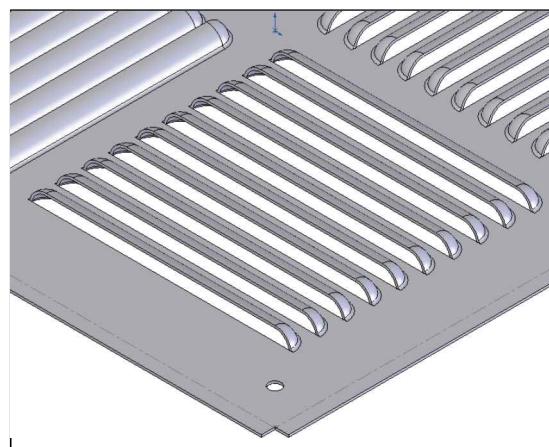


28. Saving your work:

Save the model using the same file name and override the previous document.



Finished Part (Folded)



Flat Pattern*

* After the formed features are created, they will remain in their formed shapes even when toggled back and forth between Folded or Flattened.

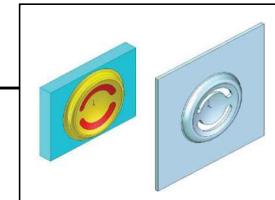
Questions for Review

1. The mid-point relation can be used to center a line on the Origin.
 - a. True
 - b. False
2. The base flange command can also be selected from Insert / Sheet Metal / Base Flange.
 - a. True
 - b. False
3. When a linear model edge is selected and the sketch is activated, SOLIDWORKS creates a plane Normal To Curve automatically.
 - a. True
 - b. False
4. The Miter Flange feature can create more than one flange in the same operation.
 - a. True
 - b. False
5. The Flat and the Folded patterns cannot be toggled until the part is completed and saved.
 - a. True
 - b. False
6. An existing forming tool cannot be edited or changed; forming tools are fixed by default.
 - a. True
 - b. False
7. When applying a form tool onto a sheet metal part, the push or pull direction can be toggled when pressing:
 - a. Up arrow
 - b. Tab
 - c. Control
8. The Modify Sketch command can be used to rotate or translate the entire sketch.
 - a. True
 - b. False
9. A formed feature(s) cannot be copied or patterned.
 - a. True
 - b. False

1. TRUE	2. TRUE	3. TRUE	4. TRUE	5. FALSE	6. FALSE	7. B	8. TRUE	9. FALSE
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CHAPTER 14

Sheet Metal Forming Tools



Sheet Metal Forming Tools

Forming tools act as dies that bend, stretch, or otherwise form sheet metal features.

SOLIDWORKS includes some sample forming tools to get you started. They are stored in: ***Installation_Directory/Data/Design Library/FormingTools/folder_name***.

Some types of form features, such as louvers and lances, create openings on sheet metal parts. To indicate which faces of the form tool will create the openings, the system changes the color of these faces to **red**, the stopping faces to **blue** and the rest to **yellow**.

The user can only insert (drag & drop) forming tools from the **Design Library** window and apply them only to sheet metal parts. The Design Library window gives you quick access to the parts, assemblies, library features, and form tools that are used most often.

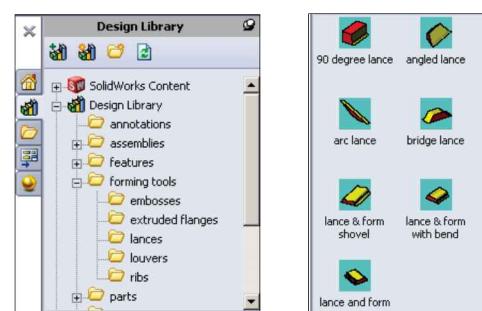
Users can create their own forming tools and apply them to sheet metal parts to create form features such as louvers, lances, flanges, and ribs.

The Design Library window has several default folders. Each folder contains a group of palette items displayed as Thumbnail Graphics.

Design Library can include:

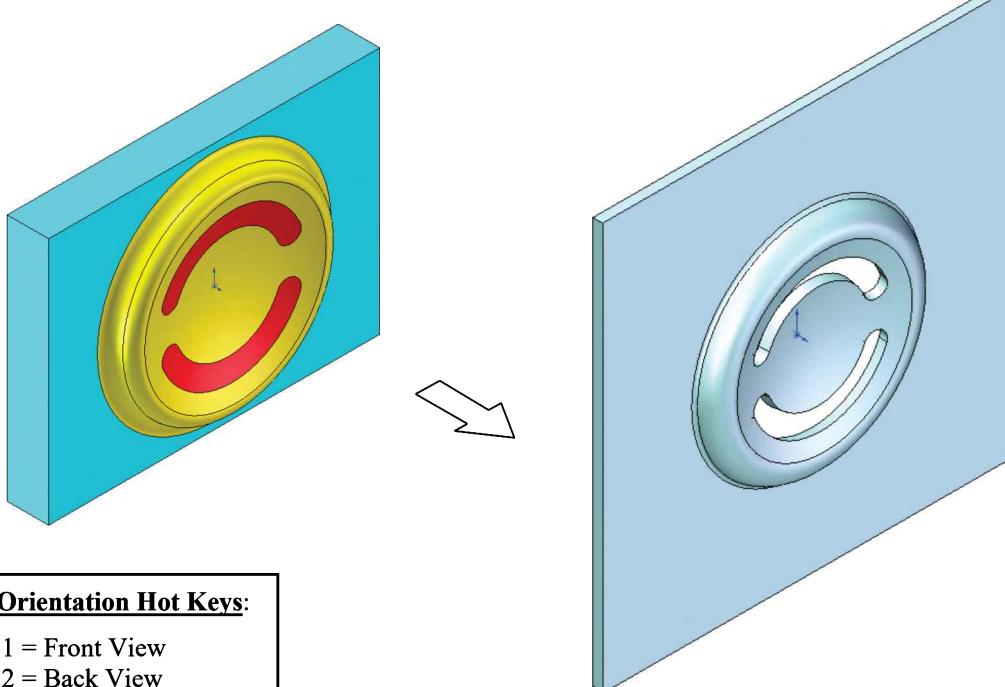
- * Parts (.sldprt)
- * Assemblies (.sldasm)
- * Sheet Metal Forming Tools (.sldftp)
- * Library Features (.sldlfp)

In this 1st half of the chapter we will learn how to create and save a forming tool.



Button with Slots

Sheet Metal Forming Tools



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI

Units: INCHES – 3 Decimals

Tools Needed:



Insert Sketch



Boss/Base Revolve



Convert Entities



Dimension



Add Geometric Relations



Extrude Cut



Split Line



Fillet/Round



Forming Tool

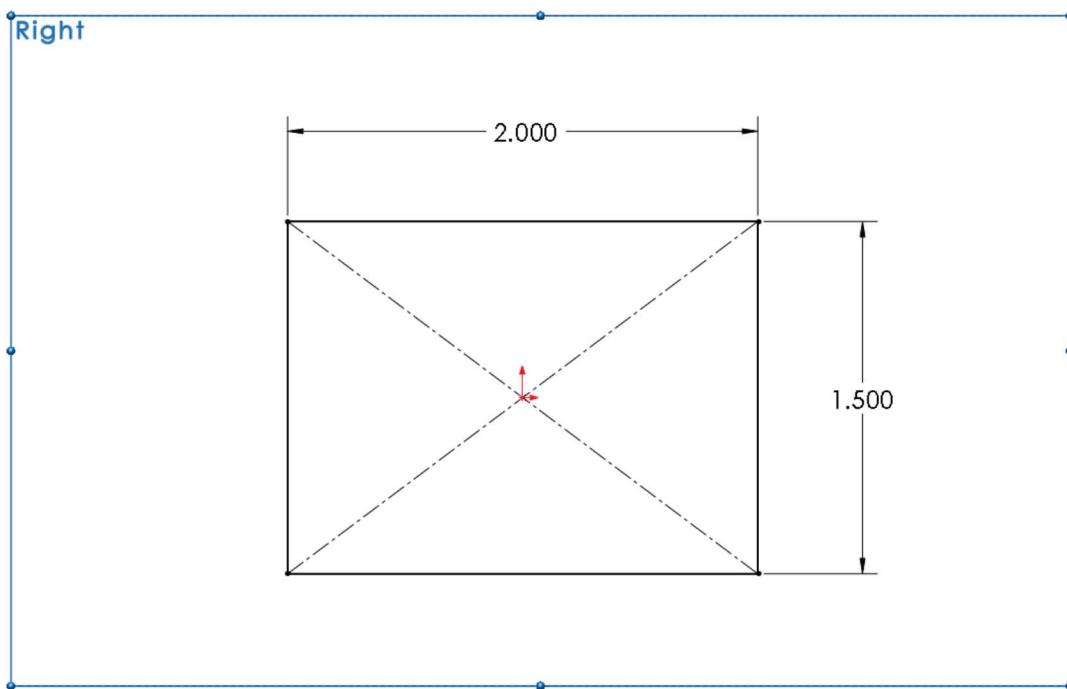
1. Creating the base block:

Select the Right plane and insert a **new Sketch** .

Sketch a **Center-Rectangle**  that is centered on the origin.

Add the width and height dimensions shown below.

(The sketch should be fully defined at this point.)



2. Extruding the base:

Click **Extruded Boss/Base** .

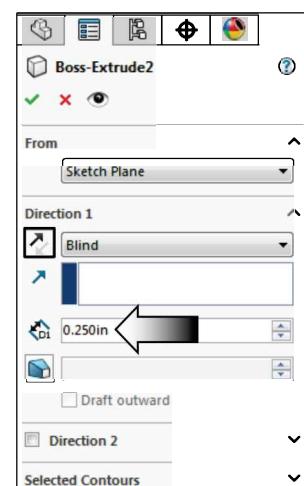
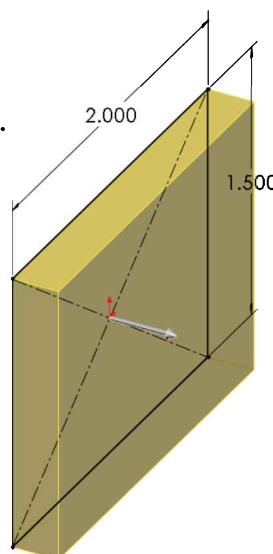
Enter the following:

Direction 1: Blind.

Reverse Direction.

Depth: .250in.

Click OK.

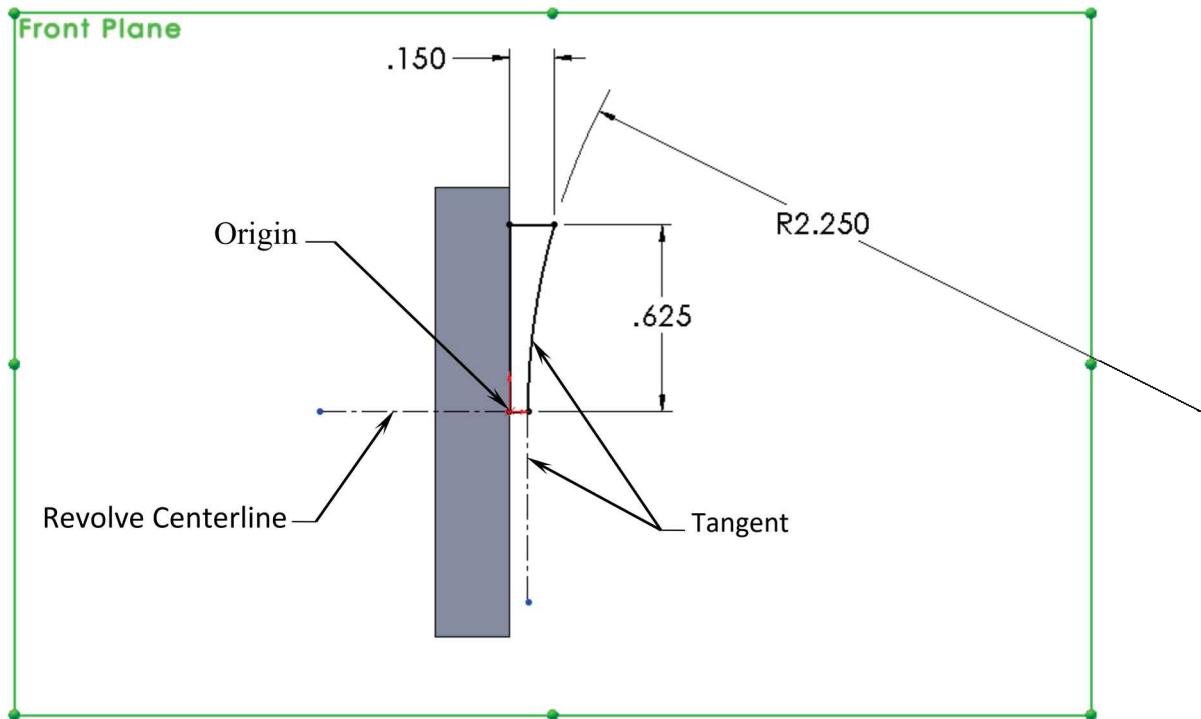


3. Creating the forming tool body:

Select the Front plane and open a **new sketch** .

Sketch the profile shown below.

Add dimensions and relations to fully define the sketch.



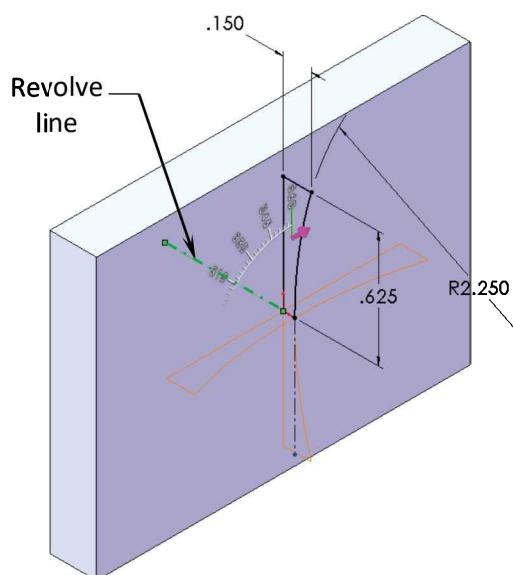
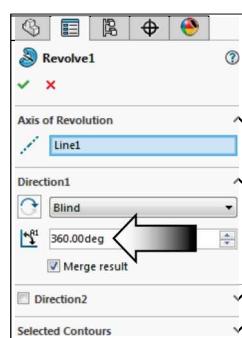
4. Revolving the body:

Click **Revolved Boss/Base** .

Revolve Blind.

Angle: 360°

Click OK.

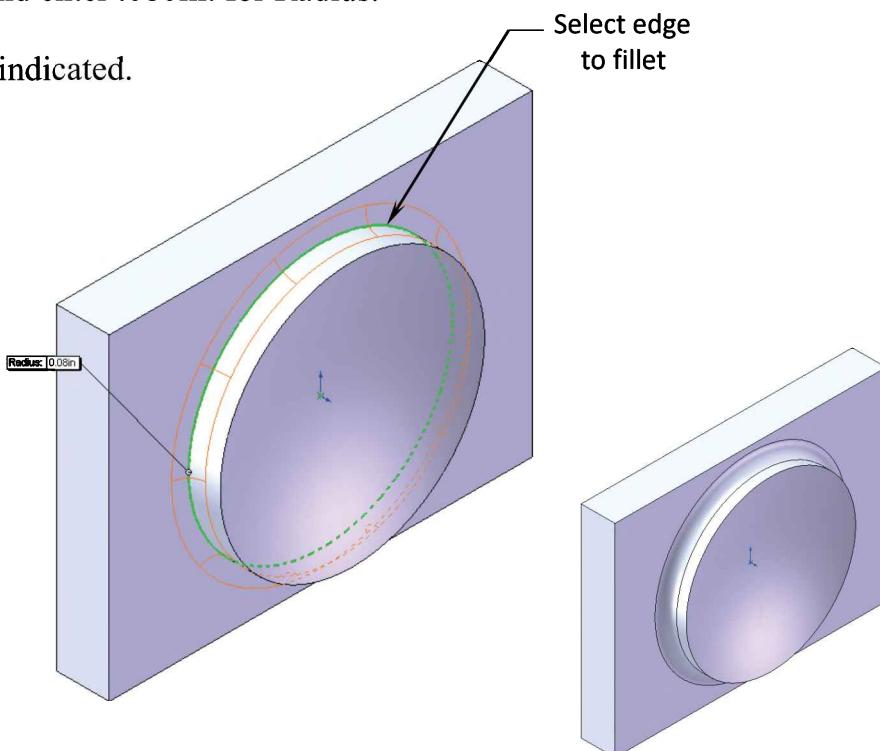
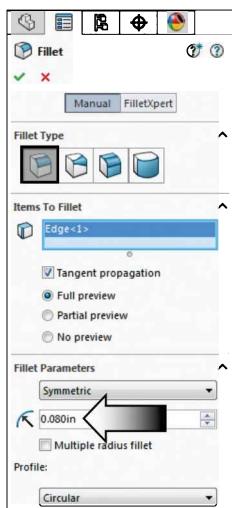


5. Adding a fillet:

Click **Fillet**  and enter **.080in.** for Radius.

Select the **edge** as indicated.

Click **OK**.

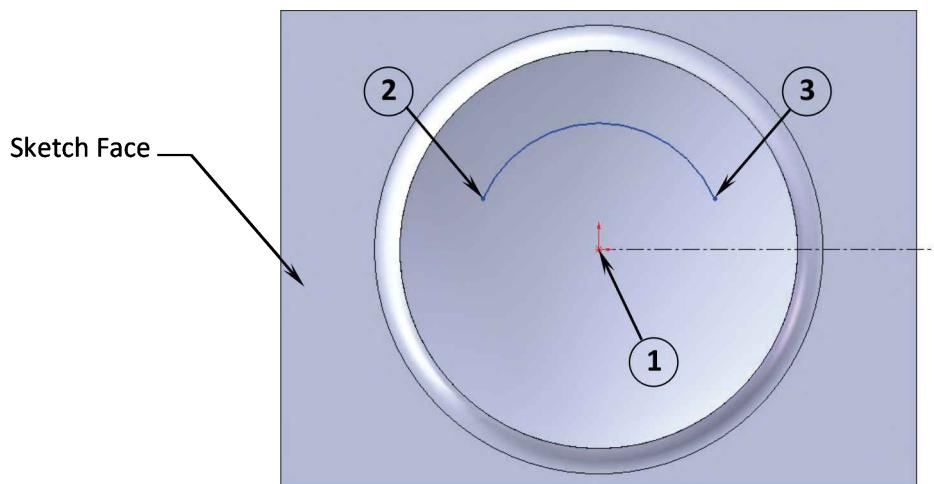


6. Sketching the 1st slot profile:

We will take a look at two different methods to create the slots.

For the 1st Arc Slot let us try the **Offset Entities** option.

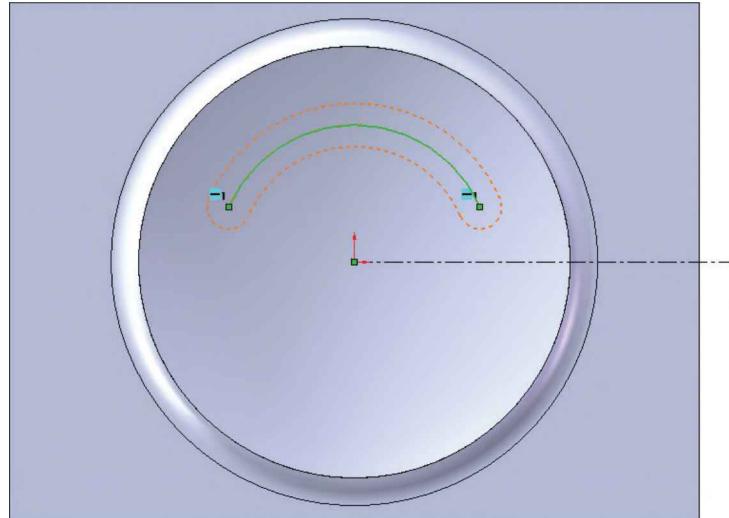
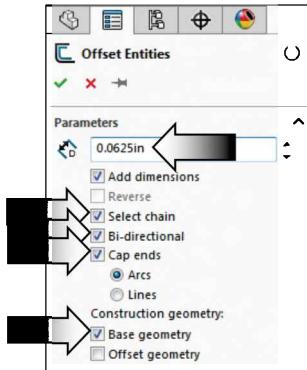
Select the face as indicated and open a **new sketch**. Draw a **Center-Point-Arc**  following the 3-click as shown below.



While the arc is still highlighted, click the **Offset Entities** command.



Enter **.0625in.** for Offset-Distance.



Enable the following:

*** Add Dimensions**

*** Select Chain**

*** Bi-Directional**

*** Cap End / Arcs.**

*** Base Geometry** (to convert the arc to construction)

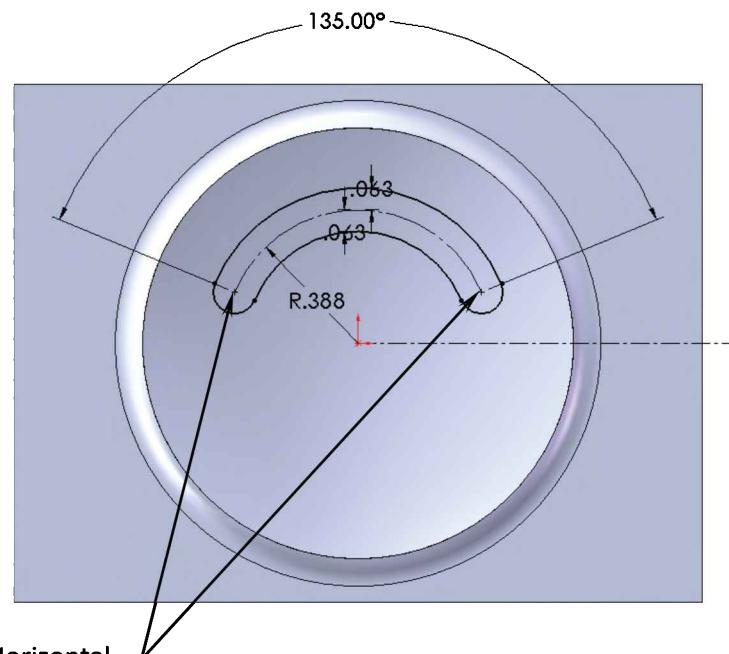
Click **OK**.

Add the dimensions and a horizontal relation as noted to fully define the sketch.

To create the angular dimension: First click the origin and then click the centers of the 2 arcs.

Note:

We will use the Arc-Slot command to create the 2nd slot in step 8.



7. Creating the 1st Split Line:

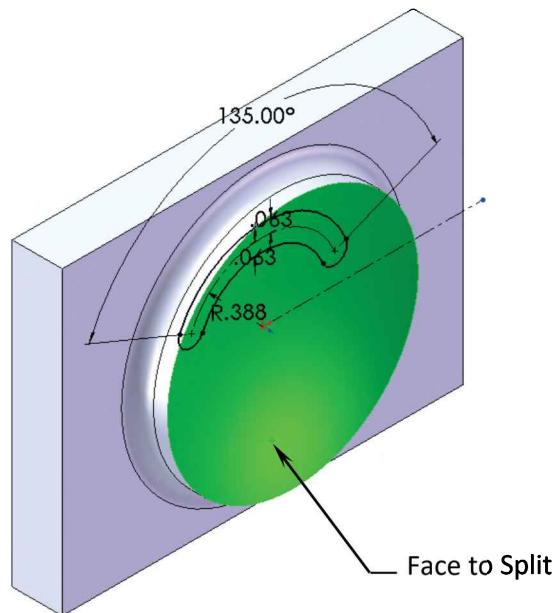
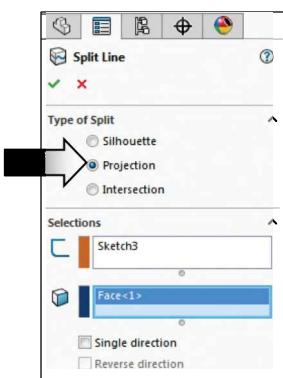
Click **Split Line**  from the **Curves** toolbar or select: **Insert / Curve / Split Line**.

Select the face as indicated to split.

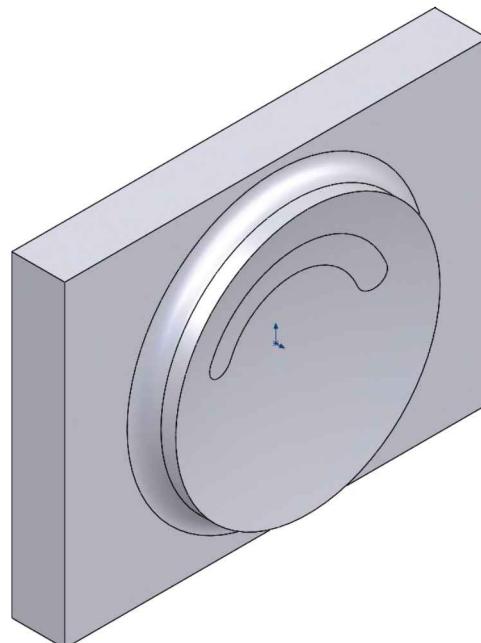
Click **OK**.

Split Line

The Split Line command projects an entity to the face and divides it into multiple faces.



The selected face is split into a new, separate surface.



This new surface can now be used as Faces-to-Remove when the form tool is inserted onto a sheet metal part.

The Faces-to-Remove option specifies what features/area will get a through cut.

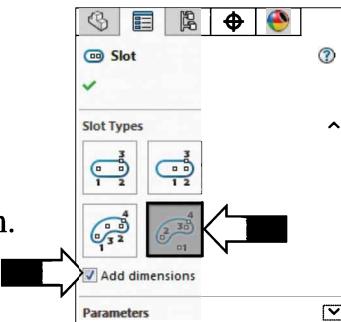
8. Creating the 2nd slot profile:

This time we will try another method to create the 2nd curved slot. (The Copy & Paste option also works well.)

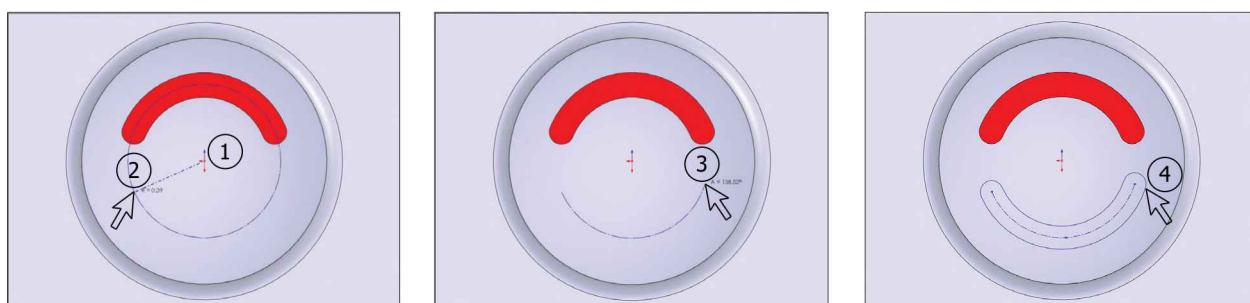
Select the Face as noted* and open a **new sketch**.

Click **Straight-Slot** and select the **Center Arc Slot** option.

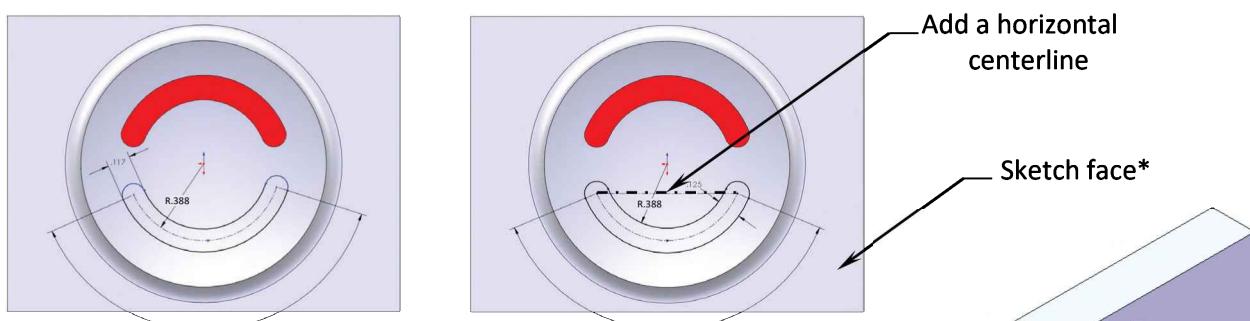
Enable the **Add Dimensions** checkbox.



Start at the Origin and click **point 1**, move outward and click **point 2**, move the cursor to the other side and click **point 3**, then drag down (or upward) to **point 4**.



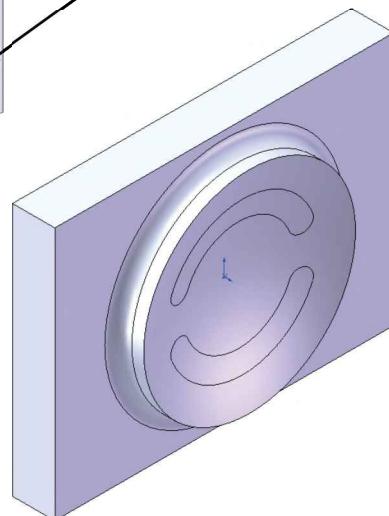
The arc-slot is completed with the Radius, Width, and Angular dimensions.



Change the values of the dimensions to:

- * Width: **.125** in.
- * Radius: **R.388** in.
- * Angle: **135** deg.

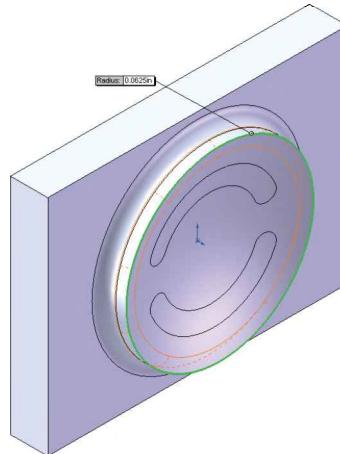
Create the **2nd split line** as shown in step 8.



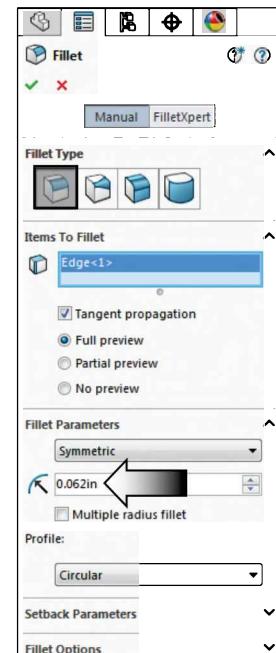
9. Adding more fillets:

Click **Fillet**  and enter **.062in.** for radius.

Select the edge as indicated.



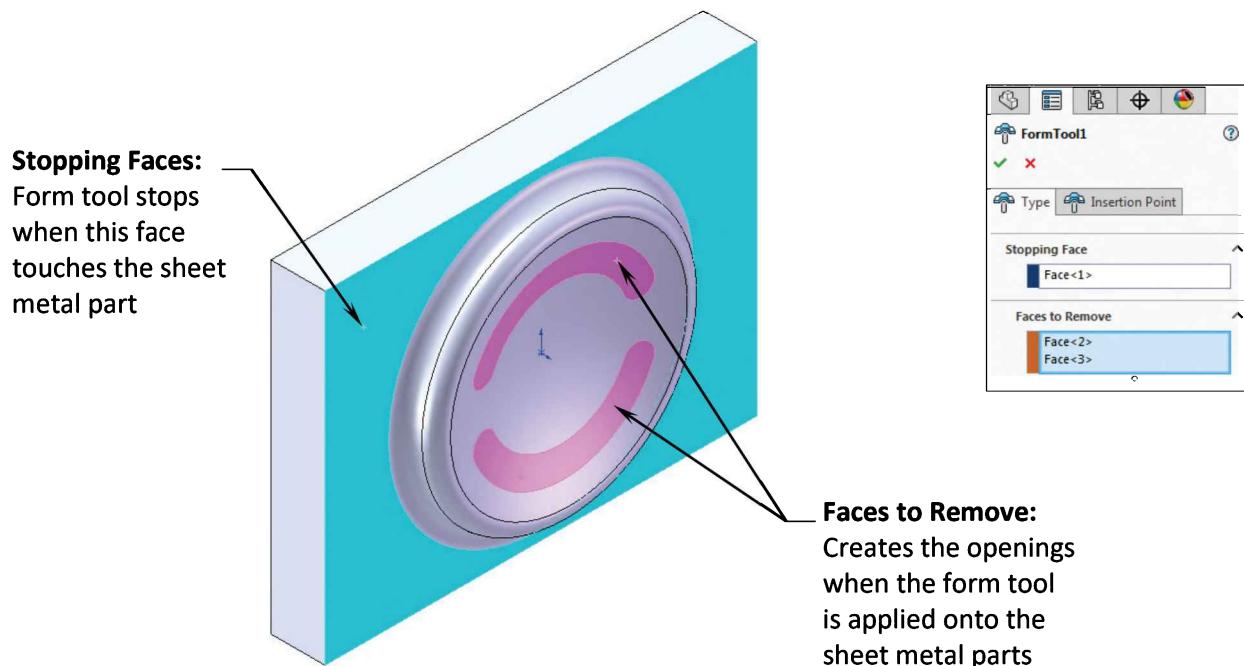
Click **OK**.



10. Inserting the Forming Tool feature:

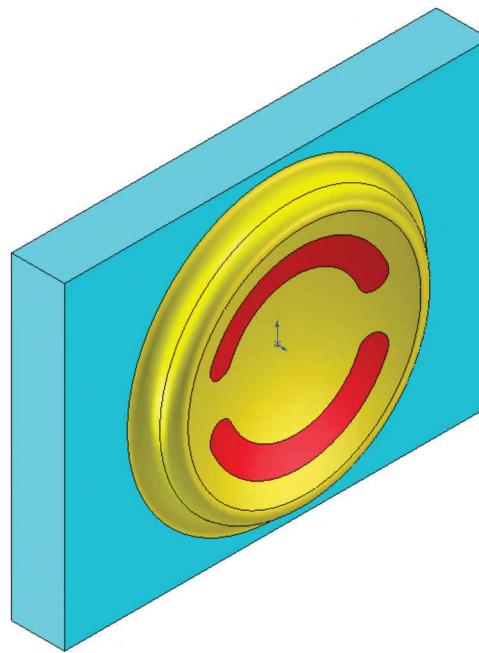
Select **Insert / Sheet Metal / Forming tool**  from the pull-down menu.

Select the **Stopping-Face** and the **Faces-to-Remove** as indicated.



Click **OK**.

The completed form tool.



11. Saving the Forming tool:

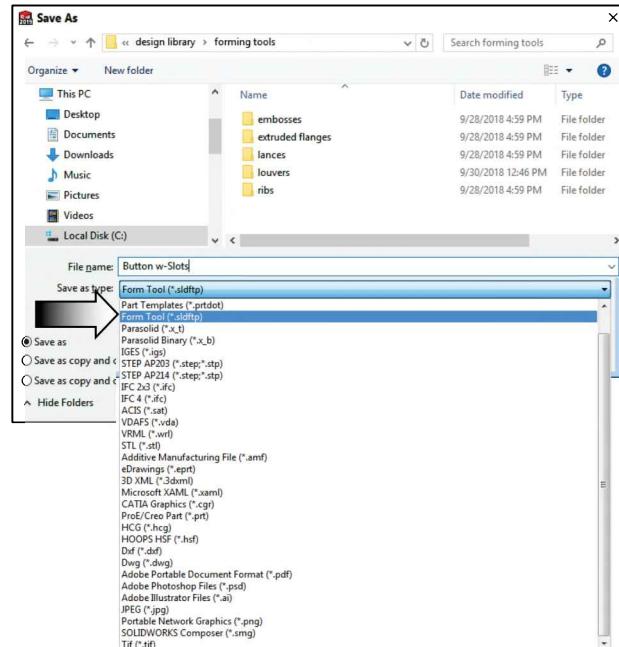
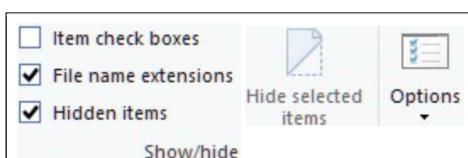
Click **File / Save As / Button w-Slots**, change the Save as Type to **Form Tool** and save the part in the following directories:

**C:/Program Data/SOLIDWORKS/
SOLIDWORKS 2024/Design-
Library/Forming Tools.**

Note:

If the Design Library Folder is Hidden, go to:

**Windows Explorer (Windows + E), click the View tab and enable the option:
Show Hidden Items**



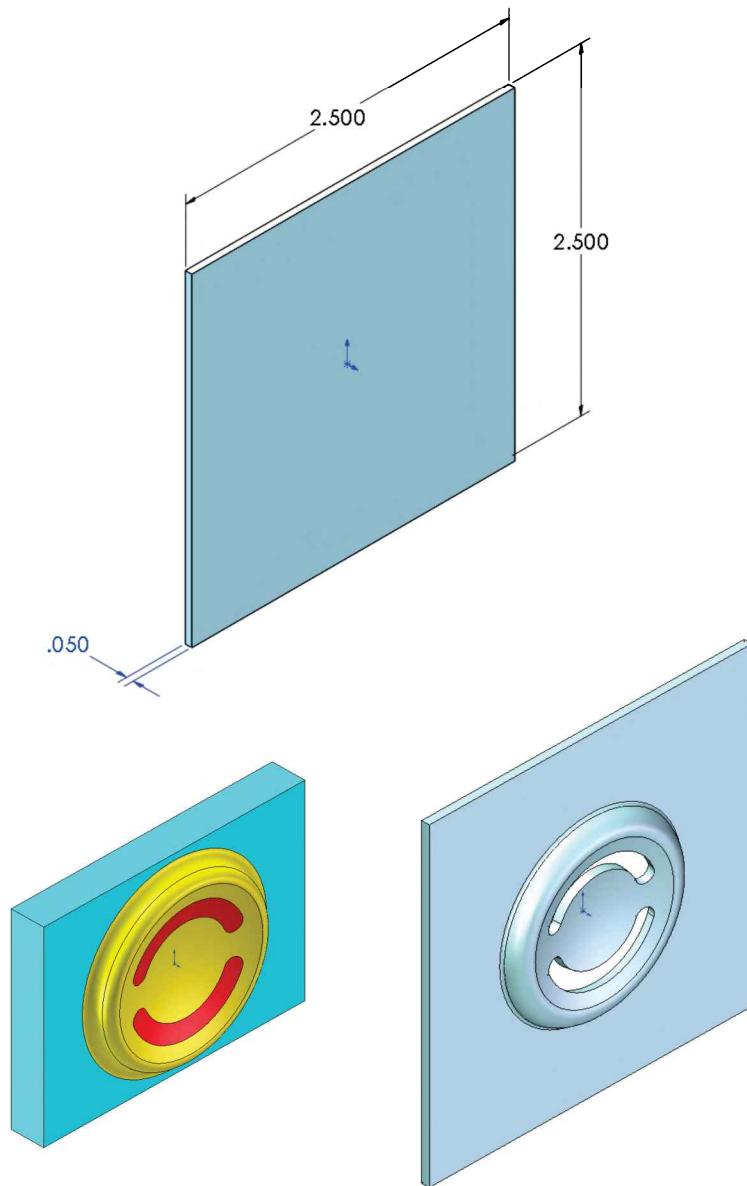
Note:

The Sheet metal forming tools can also be saved by dragging and dropping from the FeatureManager tree to a folder (i.e. Forming Tools) inside the Design Library.

The file name, file type, and path can be selected to save the forming tool at this time.

12. Applying the new forming tool: (Optional)

Create a sheet metal part using the drawing below and test out your new forming tool.



Questions for Review

1. Forming tools can bend or stretch sheet metal parts.
 - a. True
 - b. False
2. Forming tools can be stored in the Design Library window using the file extension:
 - a. slddrw
 - b. sldftp
 - c. dwg
3. Forming tools can be dragged and dropped from the Design Library window.
 - a. True
 - b. False
4. Forming tools can be used to form surfaces and solid parts as well.
 - a. True
 - b. False
5. The Red color on the face(s) of the forming tool creates openings on the sheet metal parts.
 - a. True
 - b. False
6. The Split Line command divides a selected face into multiple faces.
 - a. True
 - b. False
7. Only one single closed sketch can be used with the Split Line command.
 - a. True
 - b. False
8. The _____ key is used to reverse the direction of the forming tool (push/pull):
 - a. Shift
 - b. Tab
 - c. Alt

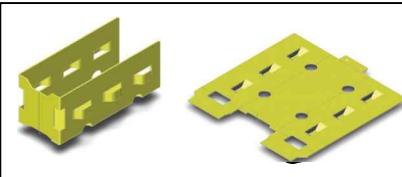
1. TRUE	2. B	3. TRUE	4. FALSE	5. TRUE	6. TRUE	7. TRUE	8. b
---------	------	---------	----------	---------	---------	---------	------

Designing Sheet Metal Parts

Tool Holder

Designing Sheet Metal Parts

Tool Holder



Sheet metal components are normally used as housings or enclosures or to help strengthen and support other parts.

A Sheet Metal part can be created as a single part, or it can also be designed in the context of an assembly that has the enclosed components.

Forming tools are dies that can bend, stretch, or form sheet metal.

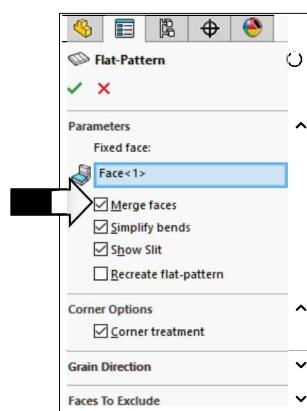
In SOLIDWORKS, forming tools are applied using the “Positive Half” (the raised side) to form features.

When inserting a forming tool, its direction can be reversed using the TAB key (Push or Pull).

The Sheet Metal part can be flattened either by using the Unfold or Flattened button, and drawings can be made to show views of the bent or flattened part. Bend lines are also visible in the drawing views.

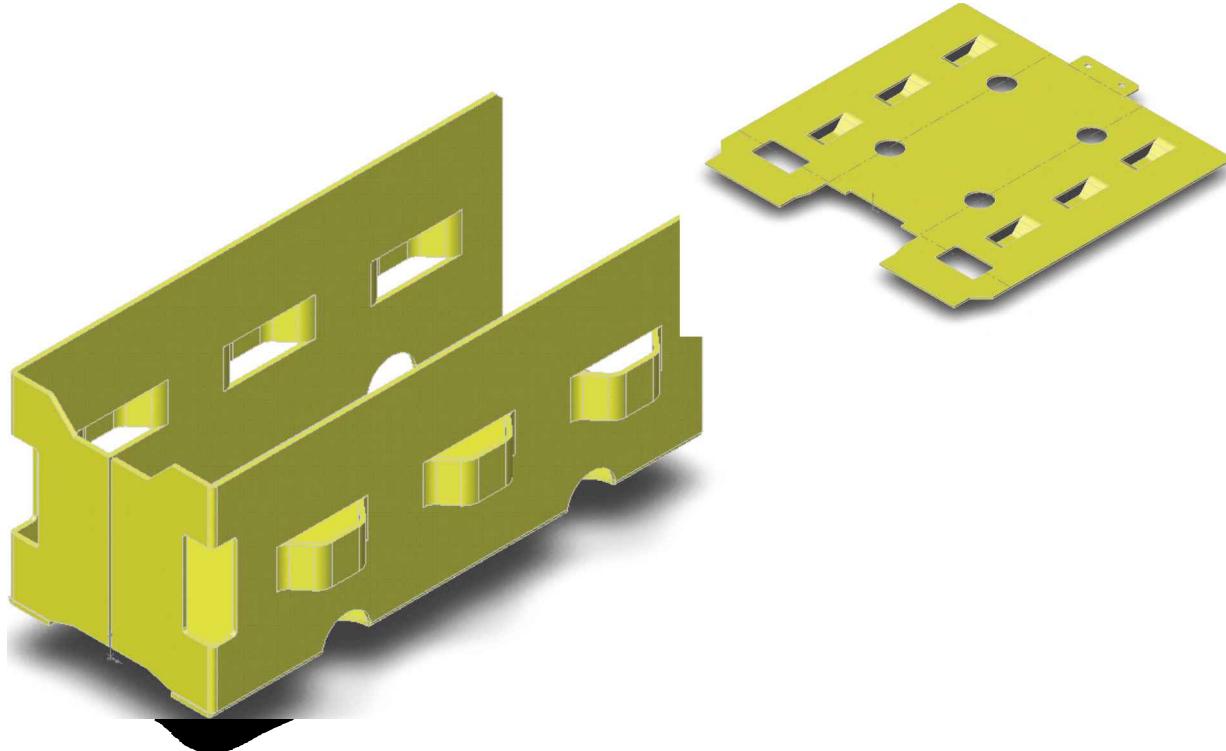
By default, only the Bend-Lines are visible at all times but not the Bend-Regions. To show the Bend Regions, right click the Flat Pattern1 icon at the bottom of the FeatureManager tree, then select Edit Feature and clear the Merge Faces check box.

In this 2nd half of the chapter, besides learning how to create a sheet metal part, we will also learn how to apply the form tool that was created earlier in the 1st half of the lesson.



Designing Sheet Metal Parts

Tool Holder



Dimensioning Standards: ANSI

Units: INCHES – 3 Decimals

Tools Needed:



Insert Sketch



Base Flange



Edge Flange



Unfold



Fold



Extruded Cut



Linear Pattern



Flattened



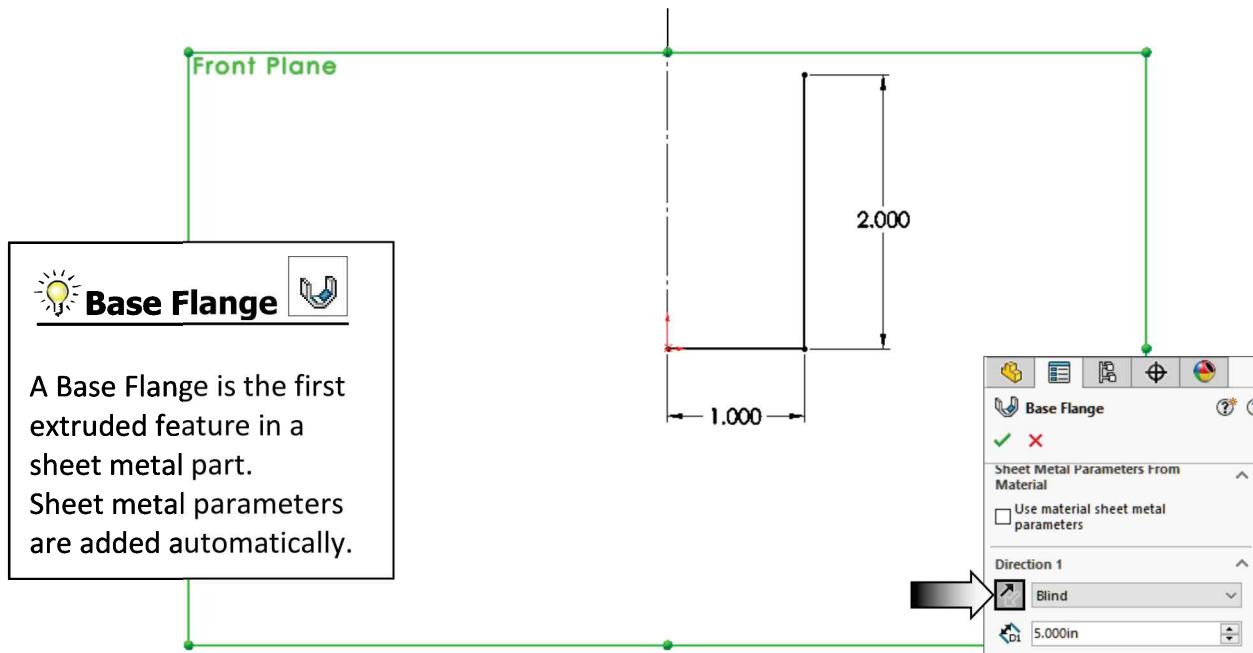
Break Corner

1. Starting with the base sketch:

Select the Front plane from FeatureManager Tree.

Click  or select **Insert / Sketch**.

Sketch the profile shown below and add dimensions to fully define the sketch.



2. Extruding the Base Flange:

Click  on the Sheet Metal toolbar, or select:

Insert / Sheet Metal / Base Flange.

End Condition: **Blind, Material Outside.**

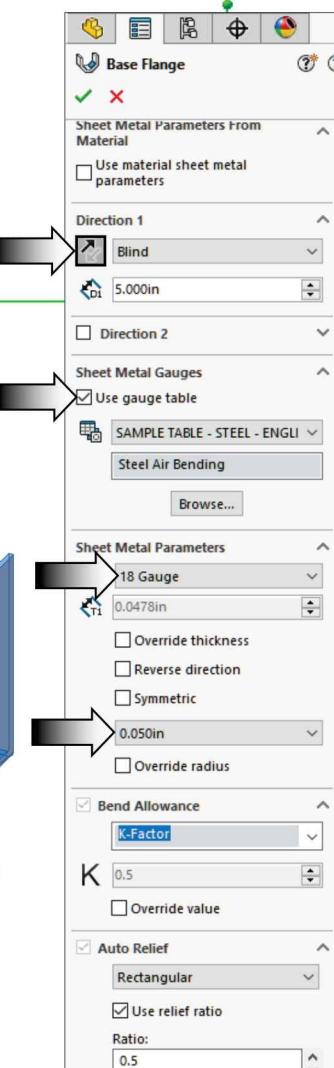
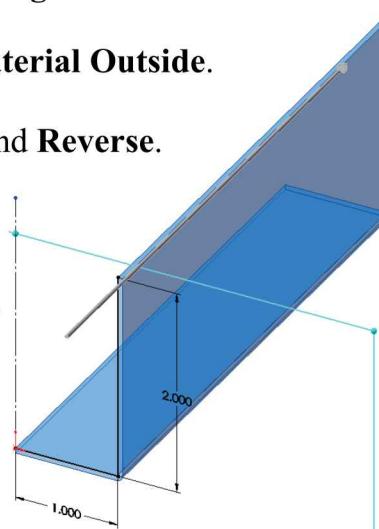
Extrude Depth: **5.000in. and Reverse.**

Use Gauge Table: **Steel.**

Thickness: **18 Gauge.**

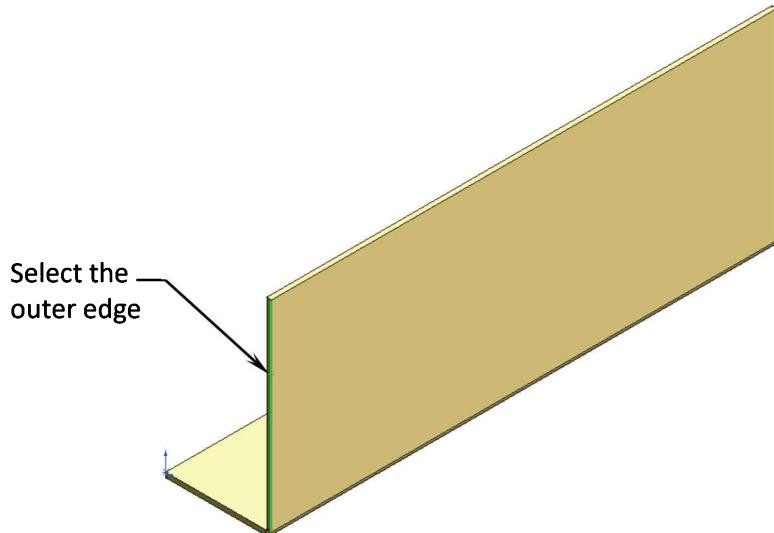
Bend Radius: **.050in.**

Click **OK**.



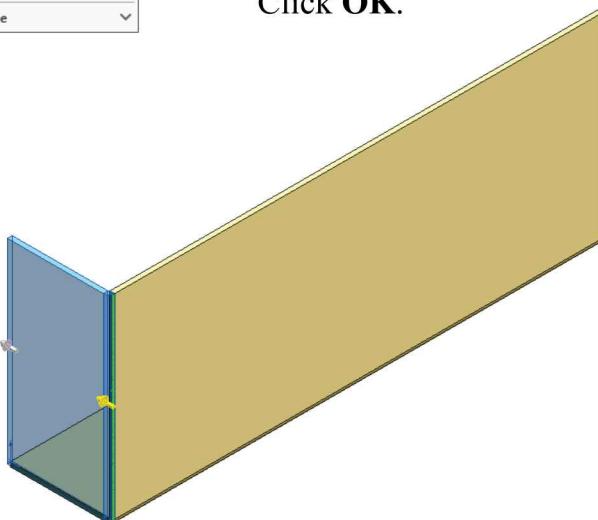
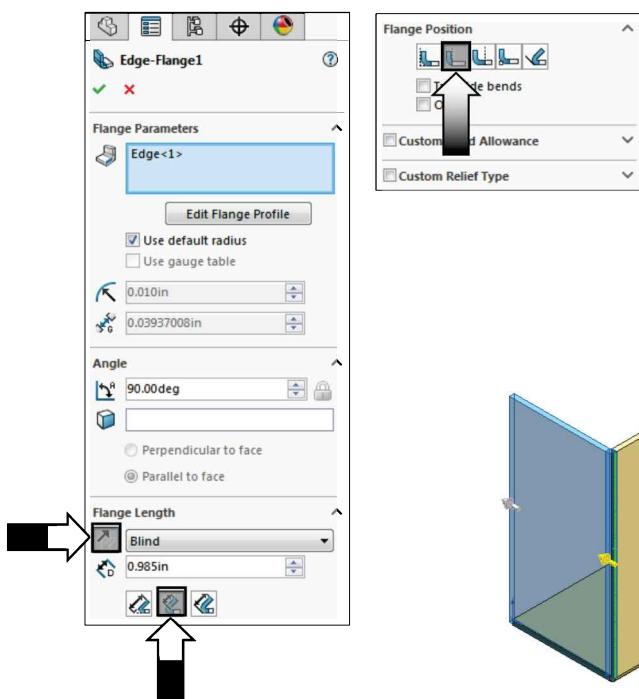
3. Creating an Edge Flange:

Select the **outer edge** as shown and click  on the Sheet Metal tab or select: **Insert / Sheet Metal / Edge Flange**.



Position the flange towards the left side and set the following:

- * Use Default Radius: **Enabled**.
- * Flange Direction: **Blind**.
- * Bend Angle: **90deg**.
- * Flange Depth: **.985**.
- * Use **Inner Virtual Sharp** (arrow).
- * Flange Position: **Material Outside**.

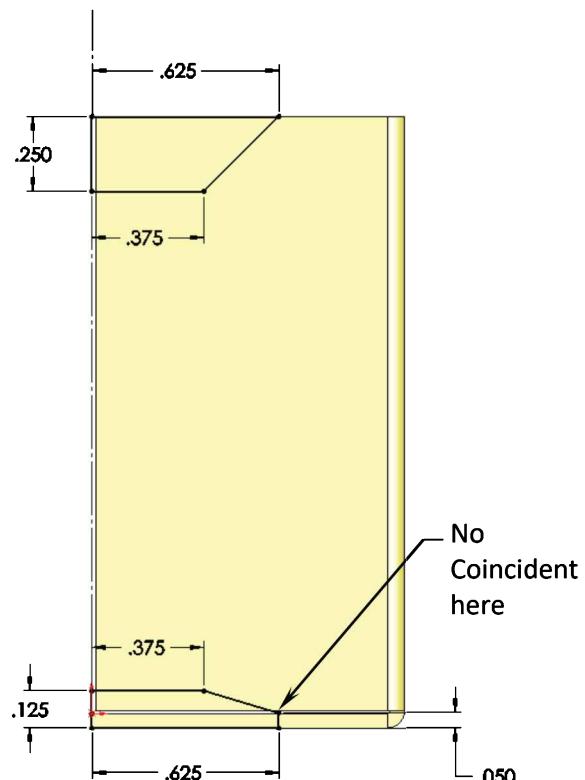
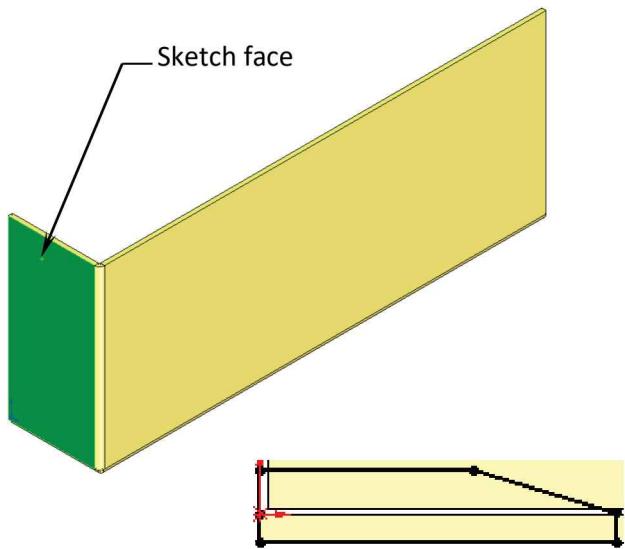


4. Adding cut features:

Select the face as noted and insert a **new sketch** .

Sketch the profile and add dimensions.

All horizontal dimensions are measured from the centerline.



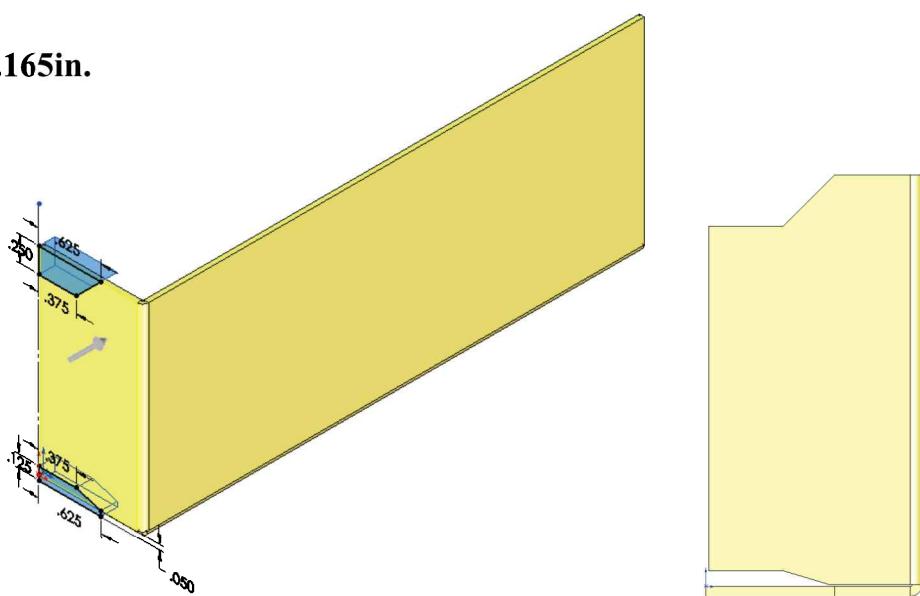
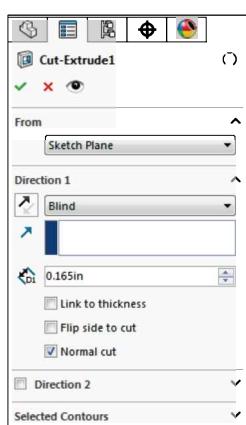
5. Extruding a cut:

 Click  or select **Insert / Cut / Extrude**.

End Condition: **Blind**.

Extrude Depth: **.165in**.

Click **OK**.

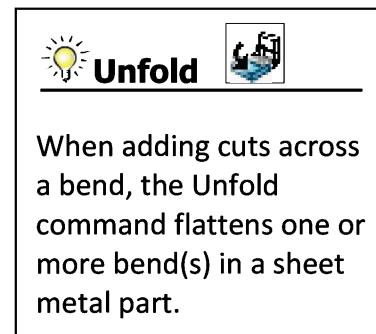


6. Using the Unfold command:

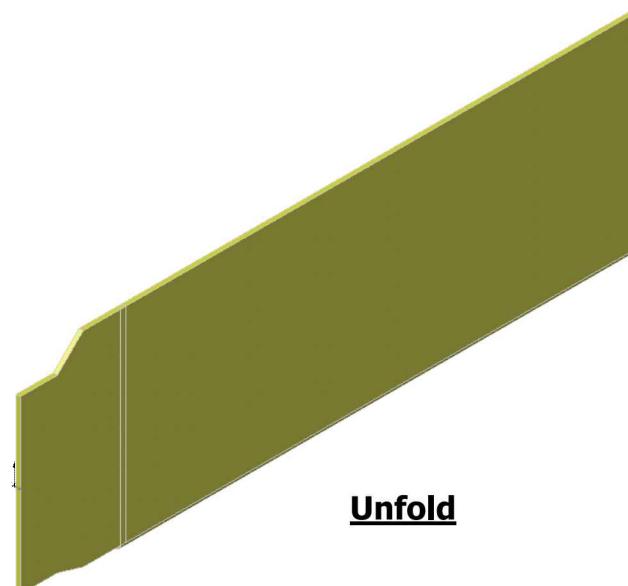
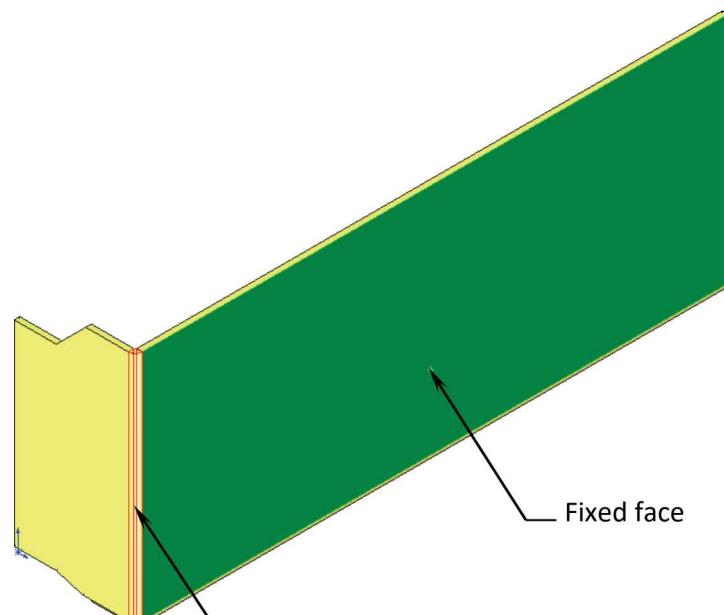
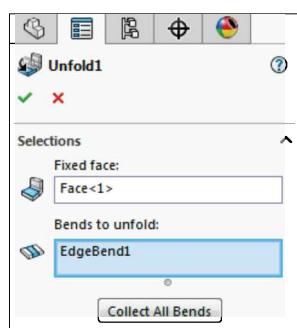
Click  on the Sheet metal tab or select:
Insert / Sheet Metal / Unfold.

For Fixed face select the **right side face** of the model.

For Bends to Unfold select the **bend radius** as indicated. (Click Collect All Bends if you want to flatten the entire part.)



Click **OK**.

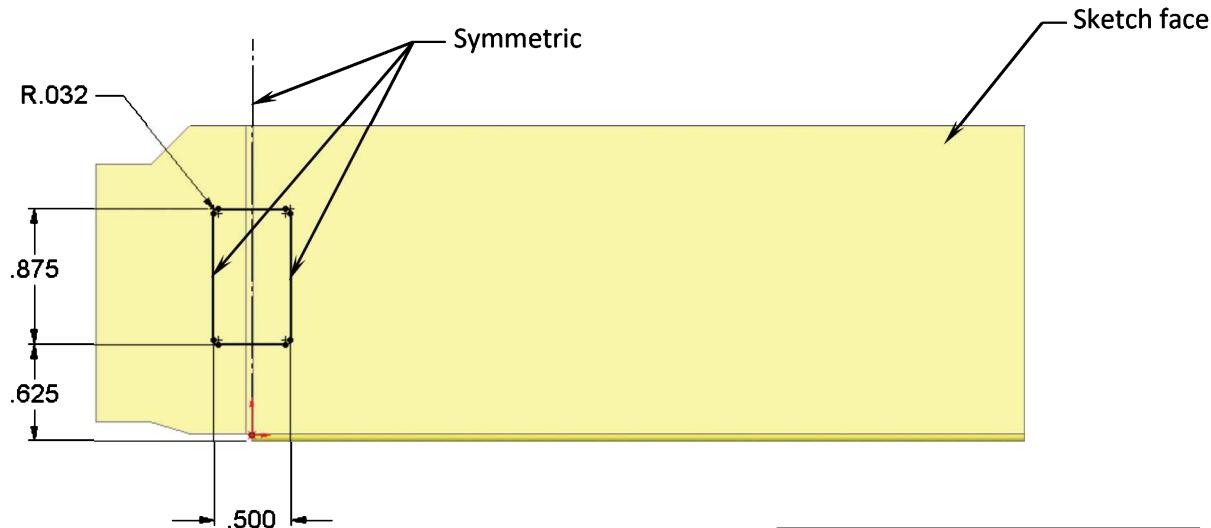


7. Creating a Rectangular Window:

Select the face as indicated and insert a **new sketch** .

Sketch a **Corner Rectangle** and a vertical **Centerline** as shown below.

Add dimensions, relation and **4 Sketch Fillets**.



8. Extruding a Cut:

Click  or select **Insert / Cut / Extrude**.

End Condition: Blind.

Link to Thickness: Enabled (default).

Normal Cut: Enabled.

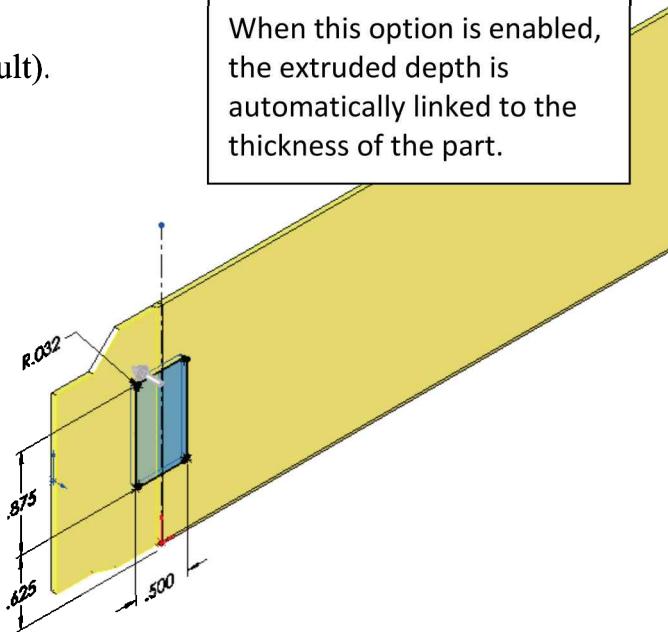
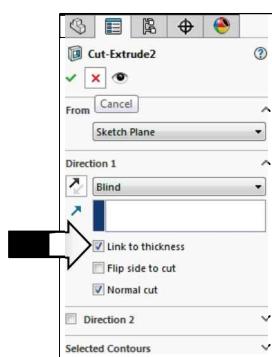


Link To Thickness

The Link-to-Thickness option is only available in sheet metal parts.

When this option is enabled, the extruded depth is automatically linked to the thickness of the part.

Click **OK**.



9. Using the Fold command:

Click  on the Sheet Metal tab or select:
Insert / Sheet Metal / Fold.

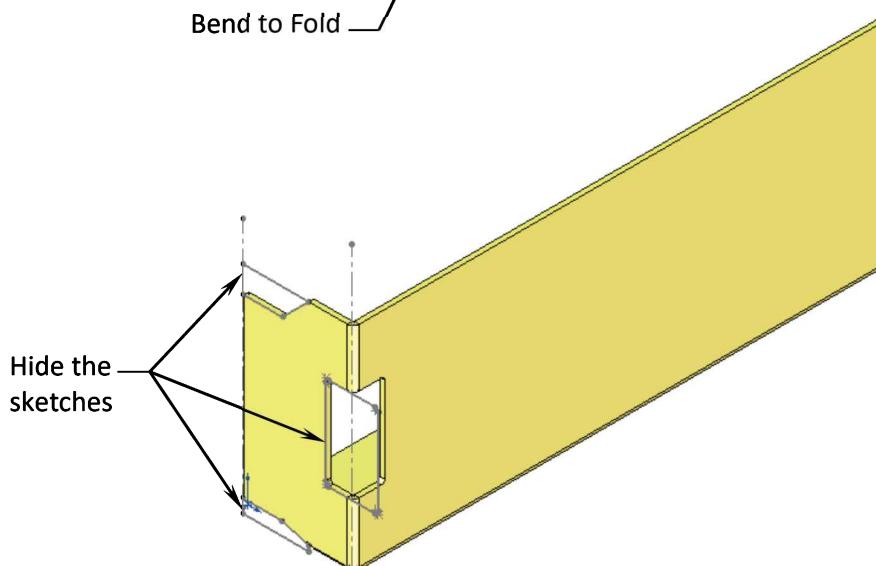
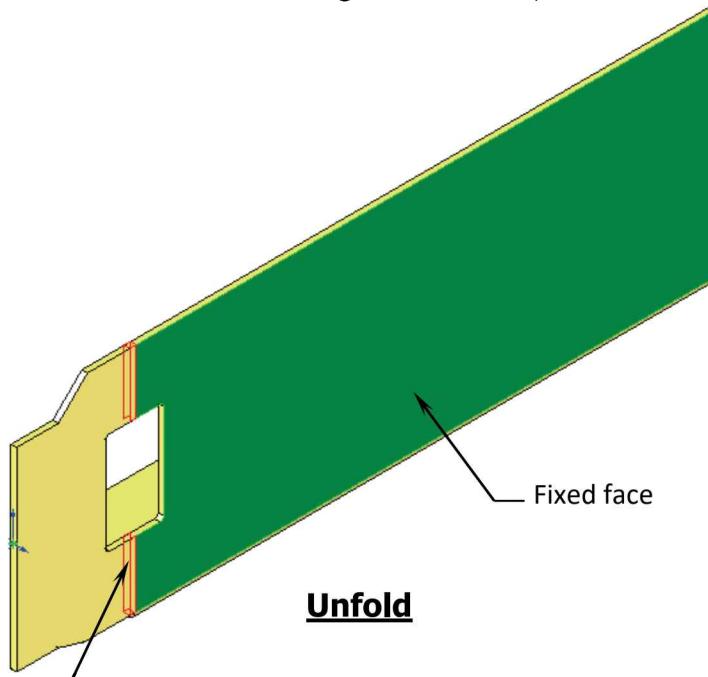
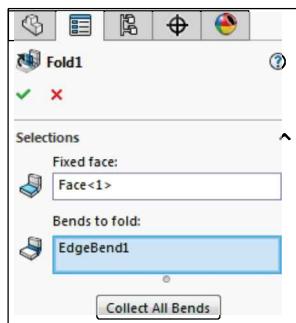
For Fixed face select the **right face** as noted.



The Fold command
returns the bends to
their folded state.

Select the **bend radius** as indicated for Bends to Fold.
(If Collect-All-Bends was selected last time, click it again this time.)

Click **OK**.



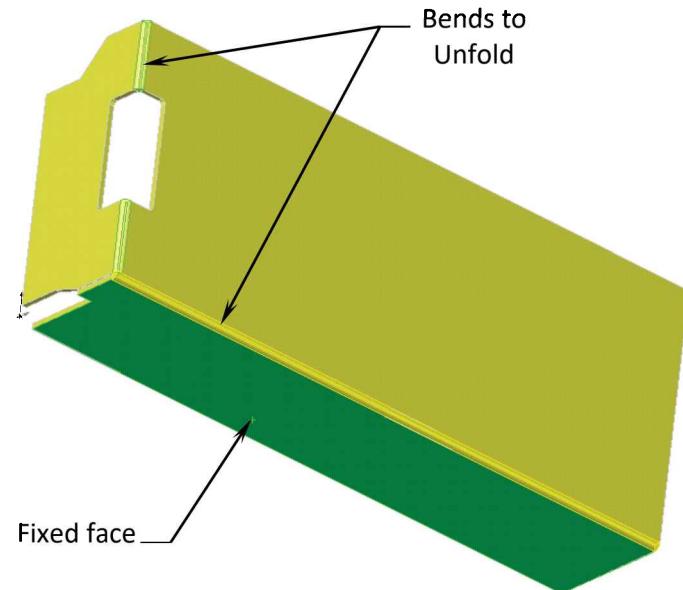
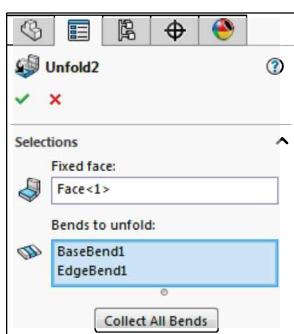
10. Unfolding multiple bends:

Click  or select **Insert / Sheet Metal / Unfold**.

For Fixed face select the bottom face of the model.

For Bends-To-Unfold select the **faces** of the 2 bends as shown
(or click Collect All Bends).

Click **OK**.

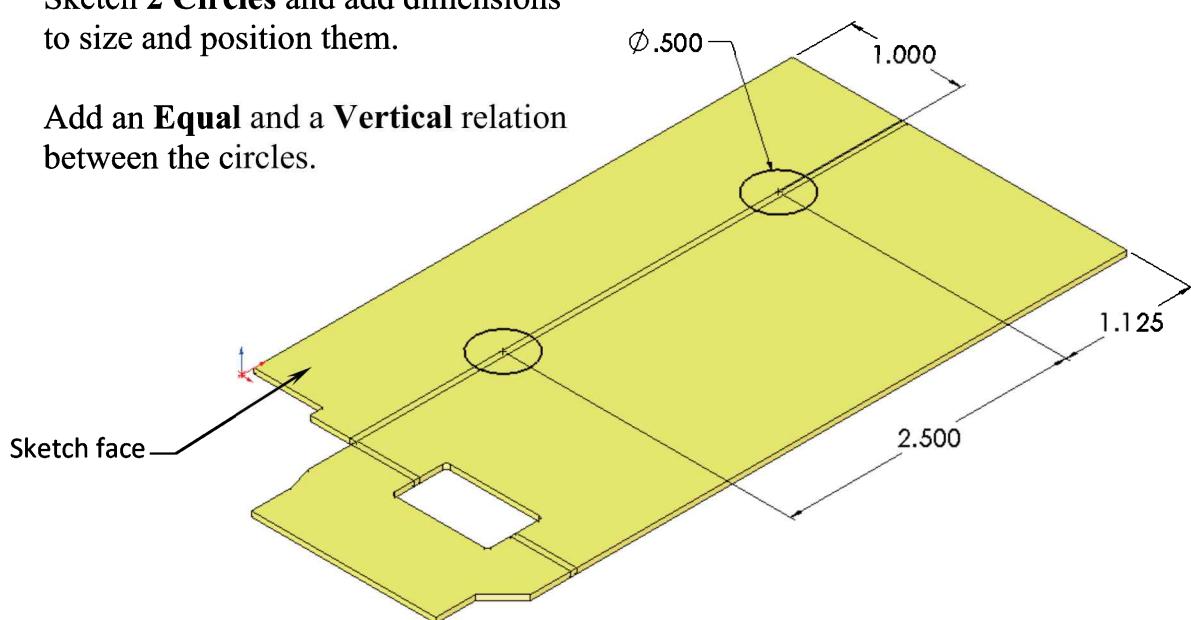


11. Adding more Cuts:

Select the **upper face** as noted and insert a **new sketch** .

Sketch 2 Circles and add dimensions to size and position them.

Add an **Equal** and a **Vertical** relation between the circles.

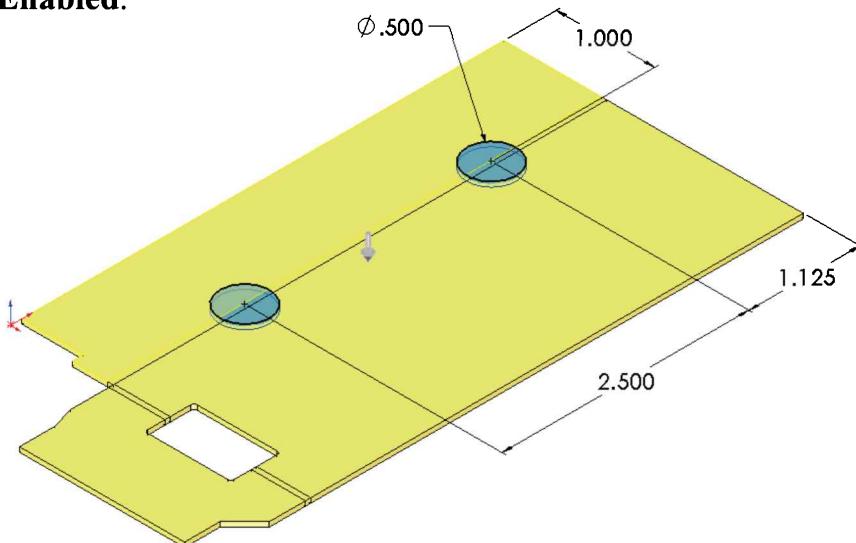
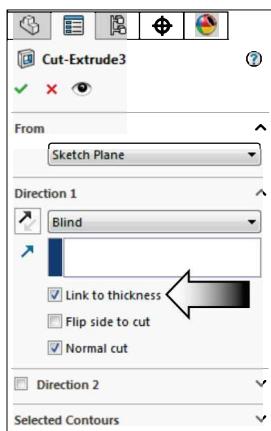


Click  or select **Insert / Cut / Extrude**.

End Condition: **Blind**.

Link To Thickness: **Enabled**.

Click **OK**.



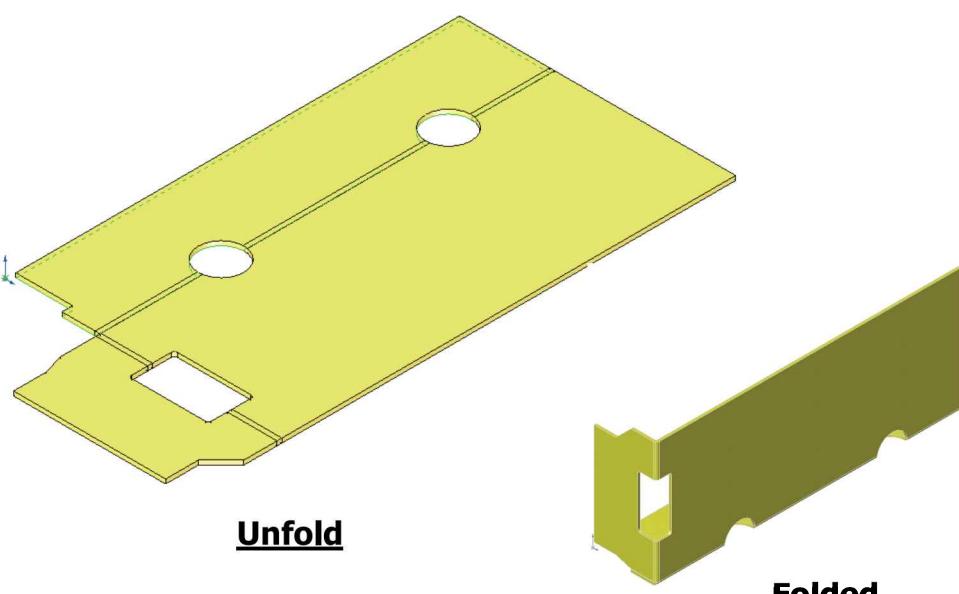
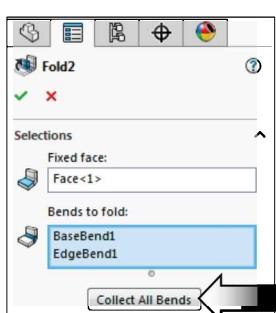
12. Folding multiple bends:

Click  on the Sheet Metal tab or select: **Insert / Sheet Metal / Fold**.

The Fixed face is still selected by default.

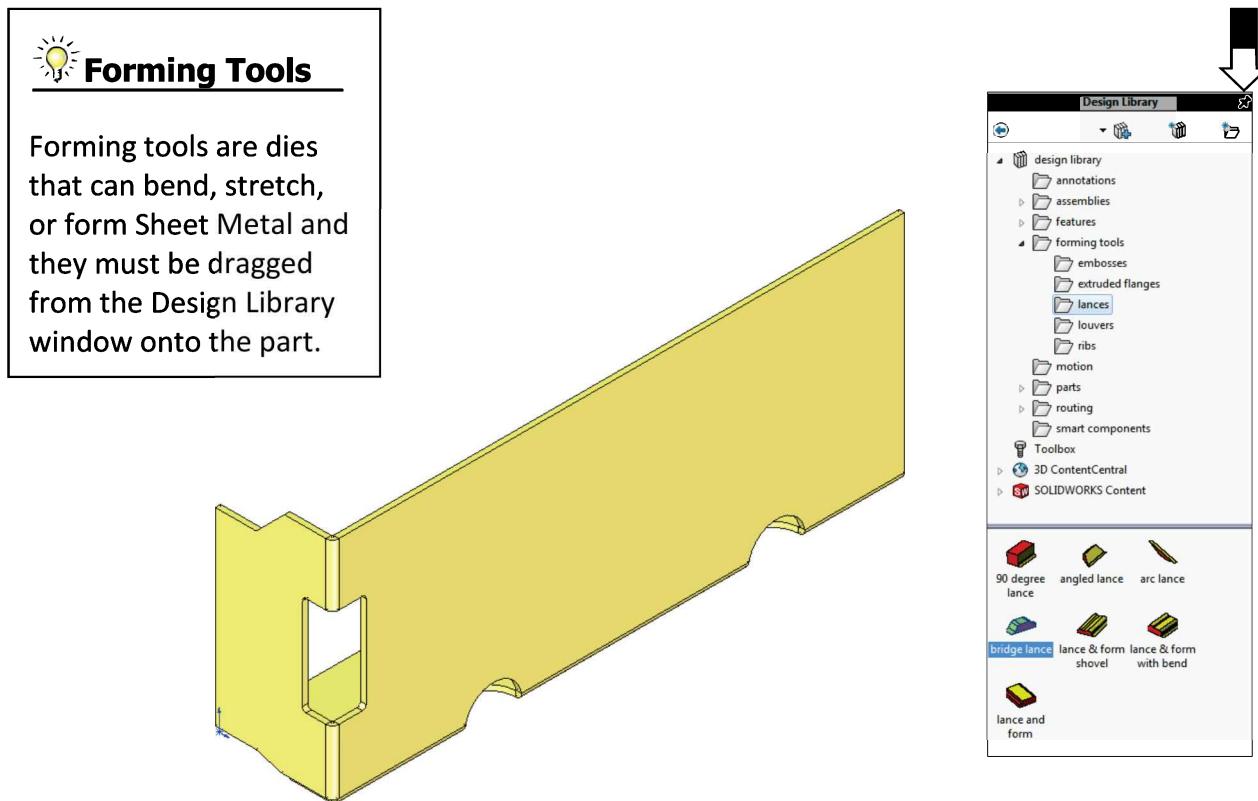
Under Bends to Fold, click **Collect-All-Bends**.

Click **OK**.



13. Inserting a Sheet Metal Forming Tool:

Click the **Design Library** icon  and click the push pin  to lock it.



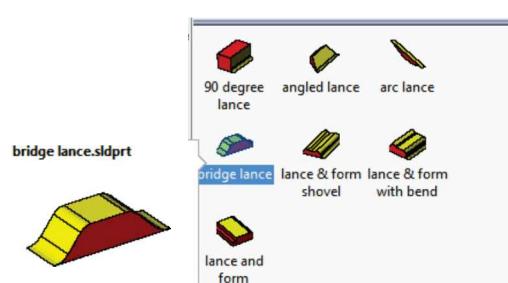
Expand the **Design Library** folder .

Click on the **Forming Tools** folder.

Click on the **Lances** folder.

Locate the **Bridge Lance** form tool.

Hover the mouse cursor over the Bridge Lance icon to see its preview graphics.

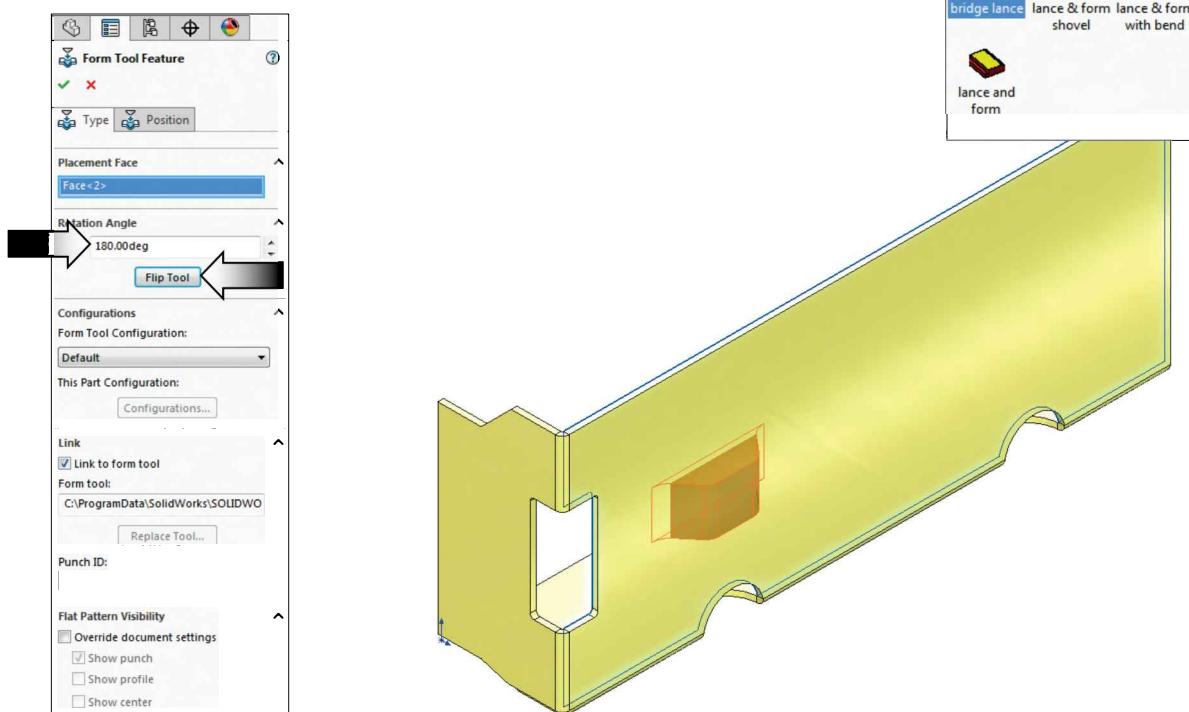
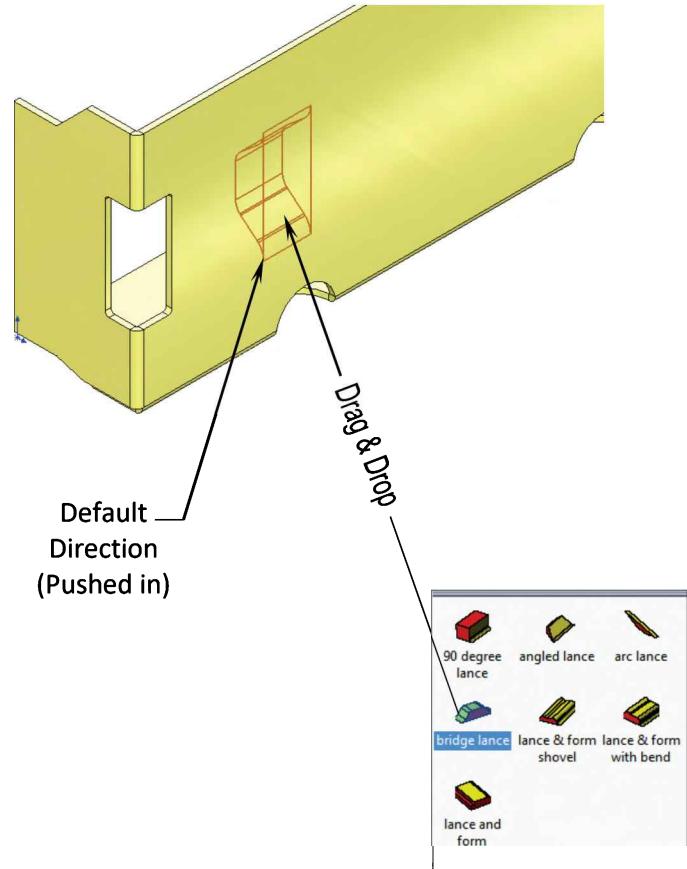


SOLIDWORKS includes some sample forming tools to get you started. These form tools can be customized and used in different sheet metal parts.

Drag the **Bridge Lance*** from the Task Pane and drop it on the sheet metal part approximately as shown.

By default, this form tool is inserted inwards (pushed in) and orientated vertically.

To correctly position the form tool, change the Rotation Angle to **180°** and click the **Flip Tool** button (arrows).



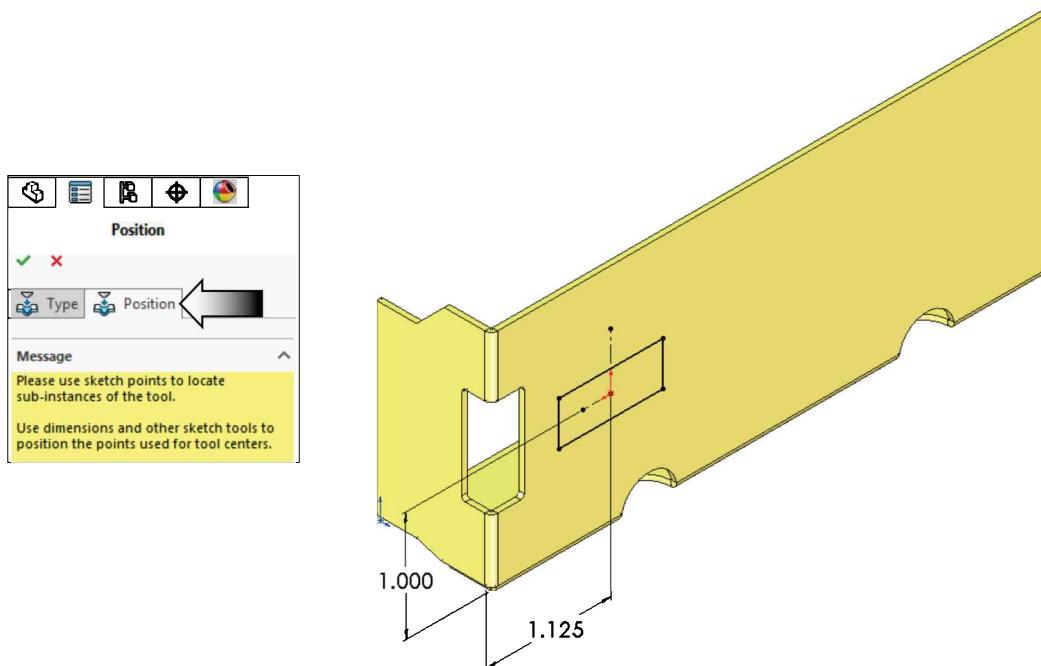
* If the Bridge Lance fails to form, double click on its icon to open the actual part, and re-save it in the same location but using the new Forming Tool extension (.sldftp)

14. Locating the Bridge Lance:

Click the **Position** tab (arrow).

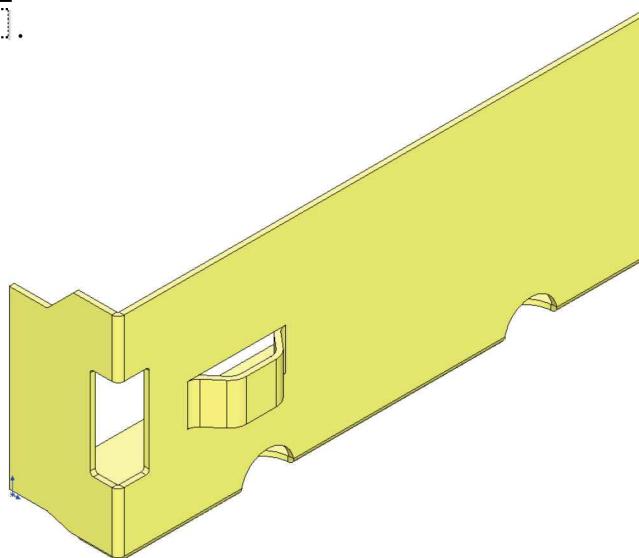
Click-off the Sketch Point command and select **Smart Dimension**.

Add the dimensions shown below to correctly position the formed feature.



Dimensions from the centerline to the outer-most edges of the part.

Click **Finish**.



Unpin the Design Library tree to put it away temporarily.

15. Creating the Linear Pattern of the Bridge Lance:

Click **Linear Pattern**  or select **Insert / Pattern Mirror / Linear Pattern**.

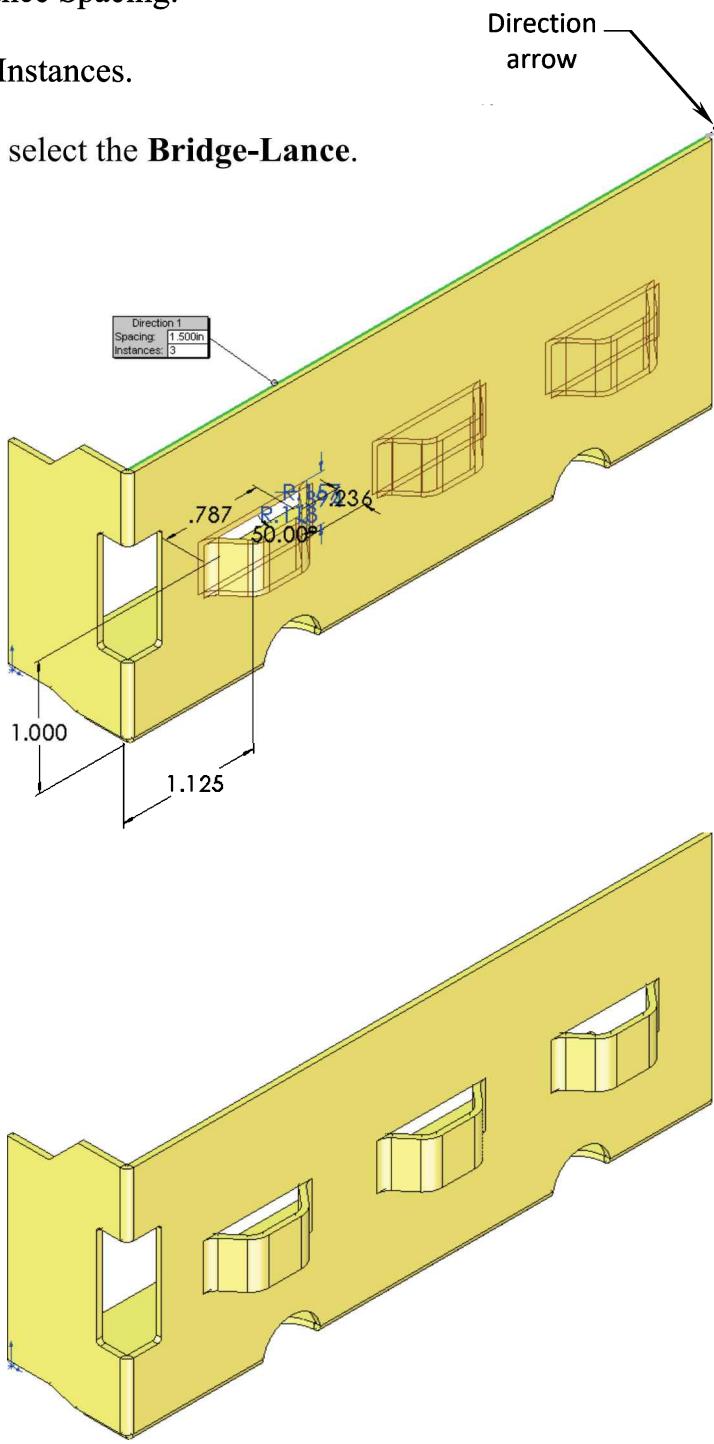
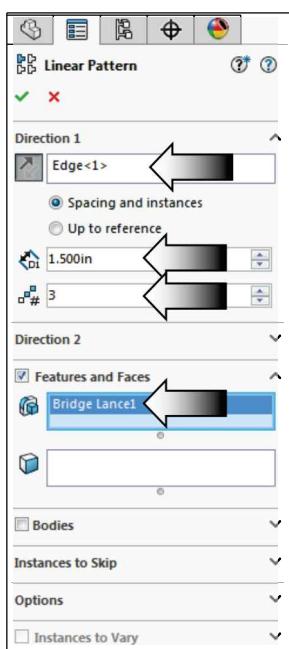
Select the top **horizontal edge** of the part as Pattern Direction.

Enter **1.500in.** for Instance Spacing.

Enter **3** for Number of Instances.

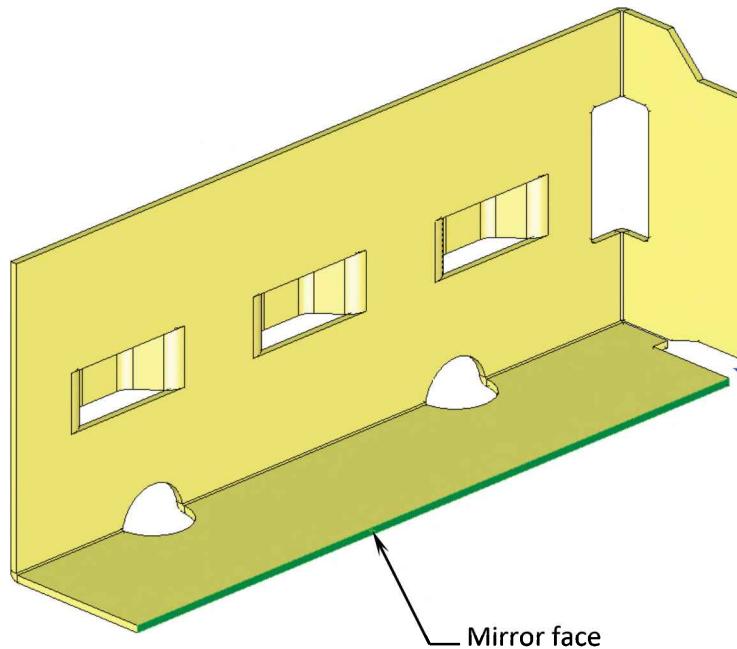
For Features to pattern, select the **Bridge-Lance**.

Click **OK**.



16. Mirroring the body:

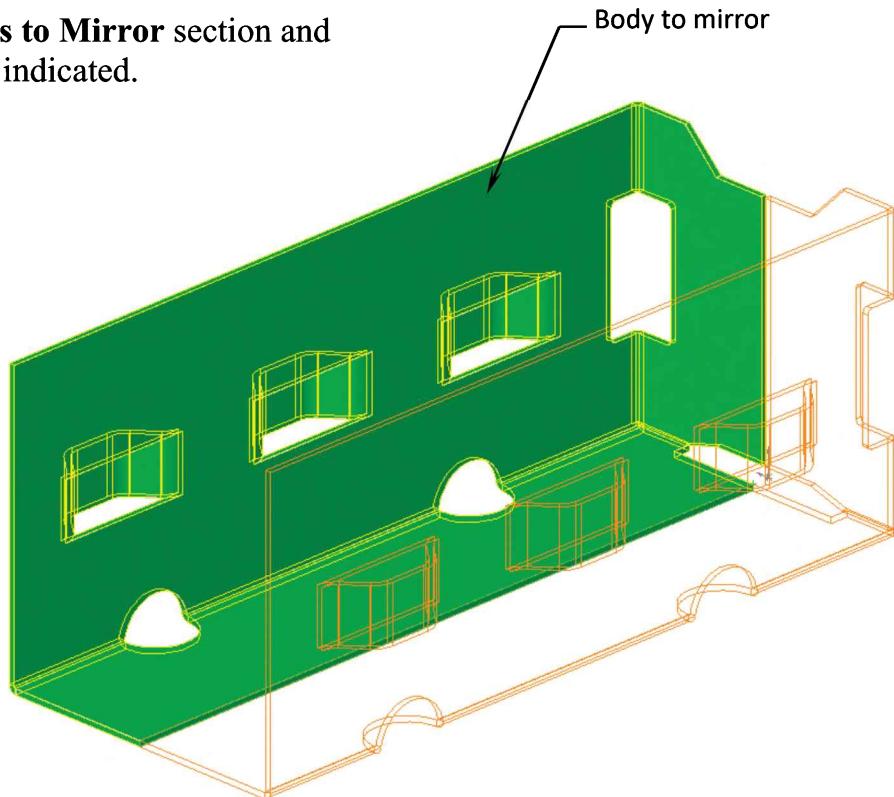
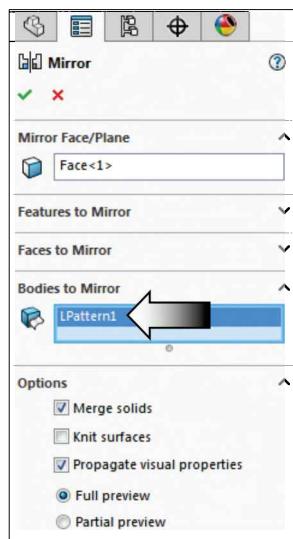
Rotate the part and select the face as indicated for mirror face.



Click **Mirror** or select **Insert / Pattern Mirror / Mirror**.

Expand the **Bodies to Mirror** section and select the body as indicated.

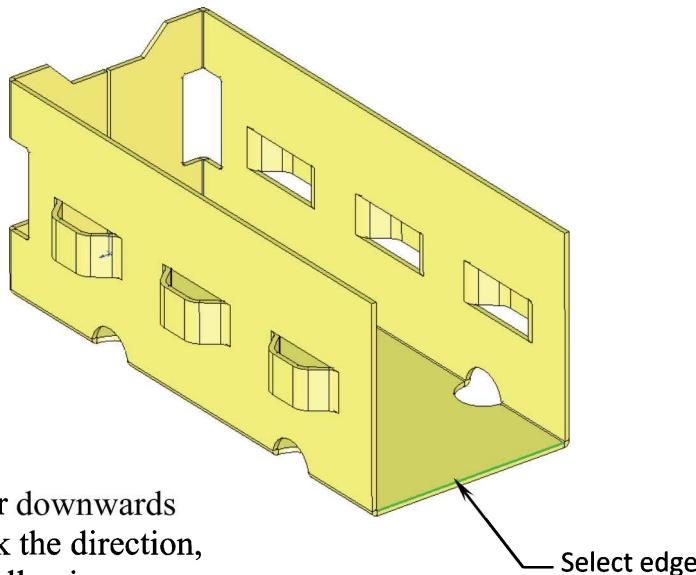
Click **OK**.



17. Adding the rear Edge Flange:

Select the edge as indicated.

Click  or select Insert / Sheet Metal / Edge Flange.



Move the cursor downwards
and click to lock the direction,
then enter the following:

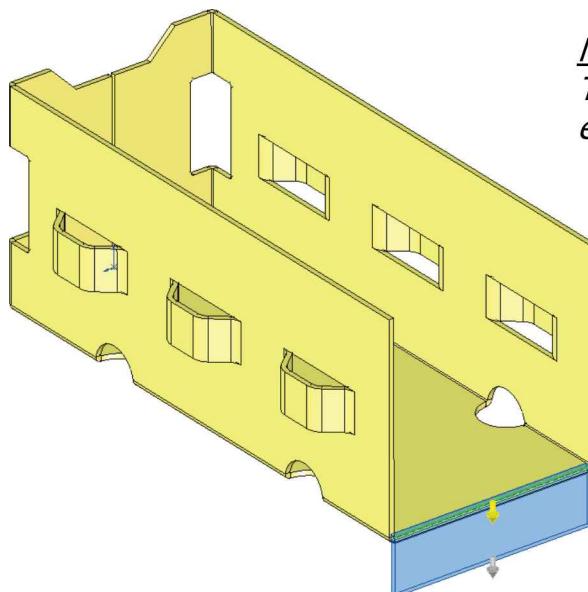
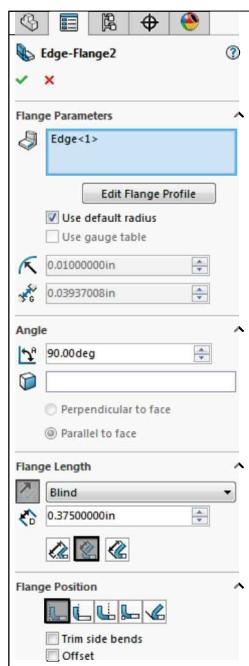
Use Default Radius: **Enabled**.

Flange Length: **Blind**.

Bend Angle: **90deg**.

Flange Position: **Material Inside**.

Use **Inner Virtual Sharp**.



*Note:
The Flange depth will be
edited in the next step.*

18. Resizing the Edge Flange:

Click the **Edit Flange Profile** button (arrow below).

The 2D sketch of the flange is activated; its shape and size can now be modified.

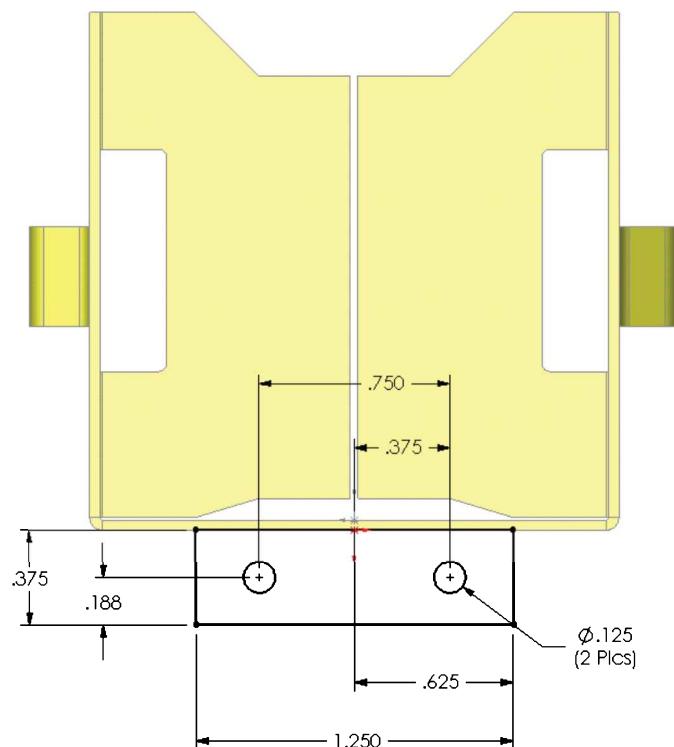
Drag the 2 vertical lines inward.



Sketch 2 Circles and add the Dimensions as shown.

Click to exit the sketch,

or click **Rebuilt** .



19. Adding Chamfers:

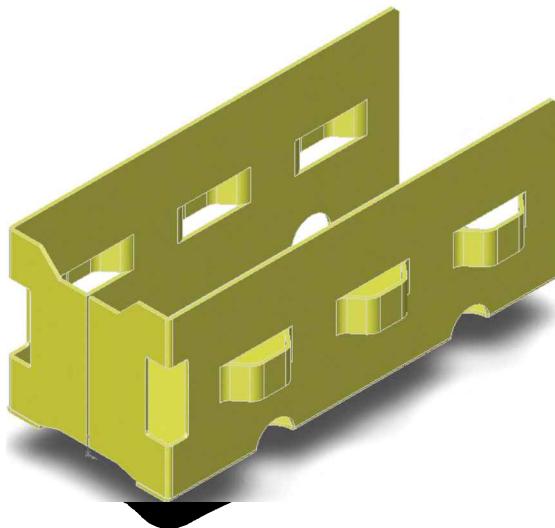
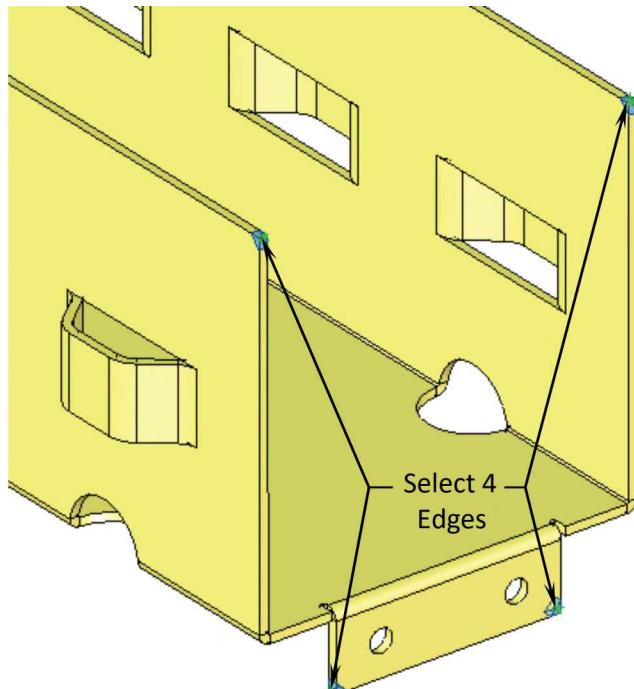
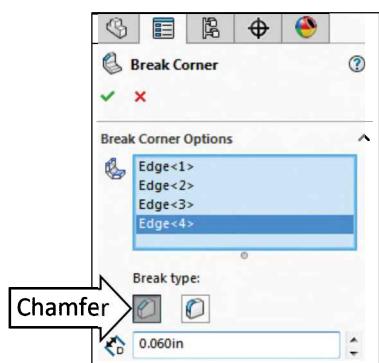
Click  or select Insert / Sheet Metal / Break-Corner.

Break Type: **Chamfer**.

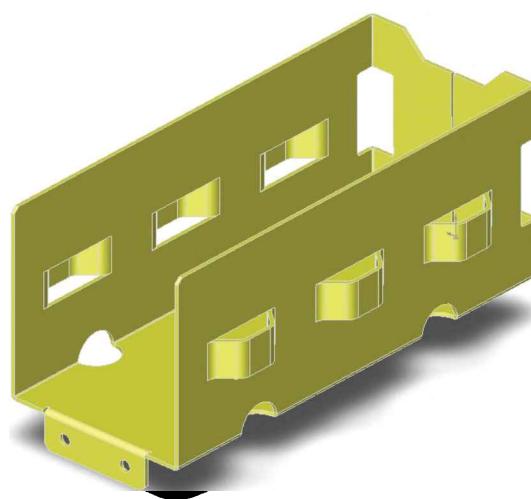
Enter **.060in.** for chamfer depth.

Select the **4 Edges** as shown.

Click **OK**.



Front Isometric



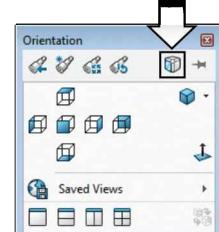
Back Isometric

See note below

Rotate Option 1: Shift + Up or Down Arrow twice to rotate 90° each time.
Rotate Option 2: Set the View rotation to 15 degrees – Press the Right arrow

12 times and the Down arrow 4 times.

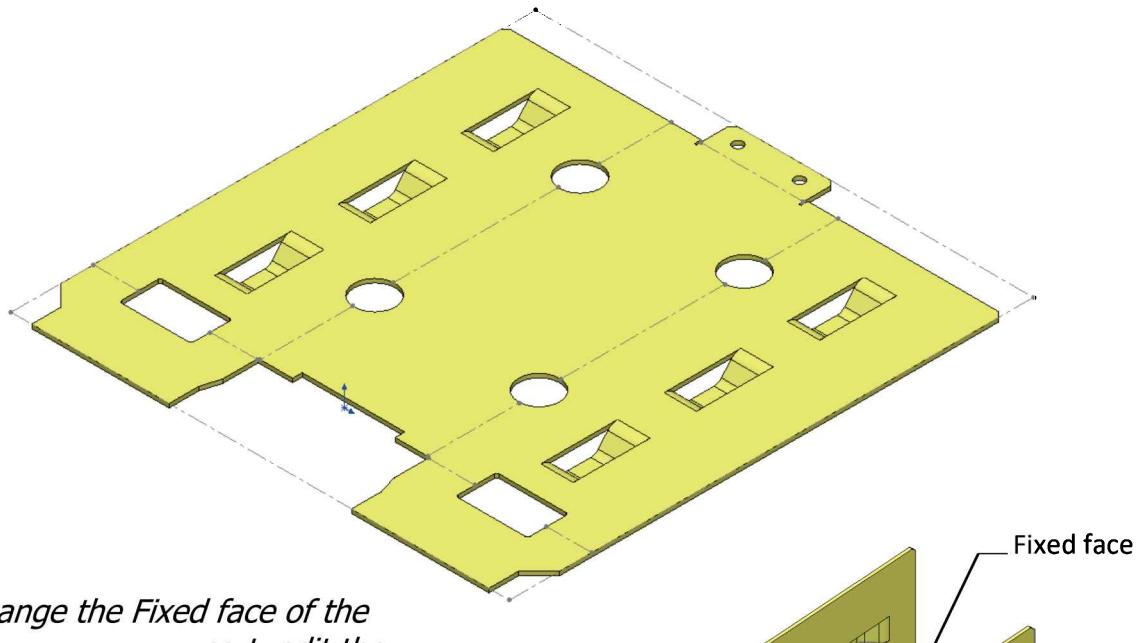
OR: use the new option **View Selector** (Space Bar) and click one of the faces
 (or projected faces) on the cube to rotate the model to that orientation.



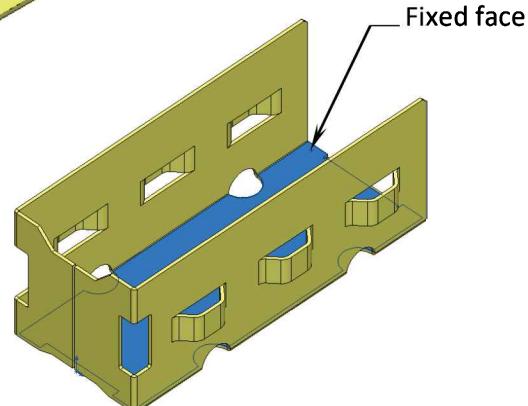
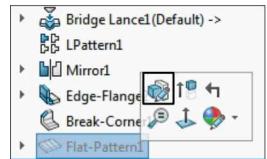
20. Switching to the Flat Pattern:

Click  or select **Insert / Sheet Metal / Flattened***.

Verify that the part is flattened properly and there are no rebuild errors.

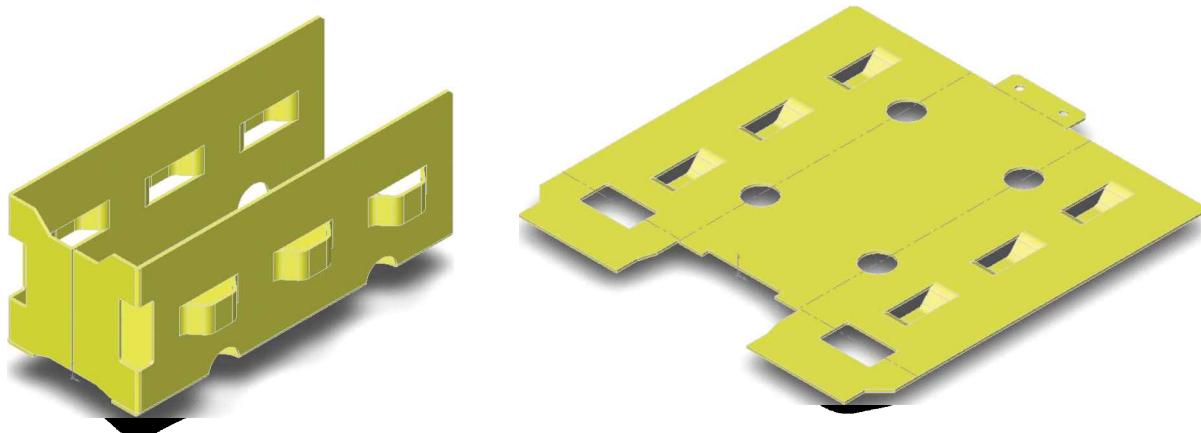


* To change the Fixed face of the part, edit the **Flat-Pattern1** feature and select the face as noted then press OK.



21. Saving your work:

Select **File / Save As / Tool Holder / Save**.



Questions for Review

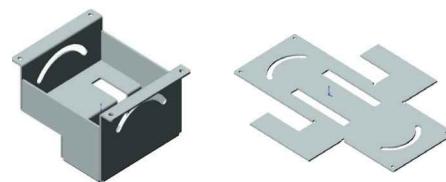
1. A Sheet Metal part can be created as a single part or in context of an assembly with enclosed components.
 - a. True
 - b. False
2. A Base Flange is the first extruded feature in a sheet metal part. Sheet metal parameters are added automatically.
 - a. True
 - b. False
3. A sheet metal part designed in SOLIDWORKS can have multiple wall thicknesses.
 - a. True
 - b. False
4. The Edge Flange command adds a flange to the selected linear edge and shares the same material thickness of the sheet metal part.
 - a. True
 - b. False
5. Only one bend can be flattened at a time using the Unfold command.
 - a. True
 - b. False
6. Forming tools need to be drag/drop from the Feature Palette window.
 - a. True
 - b. False
7. To reverse the direction of the forming tool while being dragged from the Feature Palette window, press:
 - a. Tab
 - b. Control
 - c. Shift
8. After the features are created by the forming tools, their sketches can only be moved or re-positioned, and their dimension values cannot be changed.
 - a. True
 - b. False

1. TRUE	2. TRUE	3. FALSE	4. TRUE	5. FALSE	6. TRUE	7. A	8. TRUE
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CHAPTER 15

Sheet Metal Conversions

Sheet Metal Conversions From IGES to SOLIDWORKS



Parts created from other CAD systems and saved as IGES (or Initial Graphics Exchange Specification) can be imported and converted into SOLIDWORKS Sheet Metal.

When importing other CAD formats into SOLIDWORKS, the software recognizes them as follows:

- * If there are blank surfaces, they are imported and added to the Feature-Manager design Tree as surface features.
- * If the attempt to knit the surfaces into a solid succeeds, the solid appears as the base feature (named **Imported1**) in a new part file.
- * If the surfaces represent multiple closed volumes, then one part is generated for each closed volume.
- * If the attempt to knit the surfaces fail, the surfaces are grouped into one or more surface features (named **Surface-Imported1...**) in a new part file.
- * If you import a .dxf or .dwg file, the **DXF/DWG import wizard** appears to guide you through the import process.

The imported parts must have a uniform thickness to fold and unfold properly.

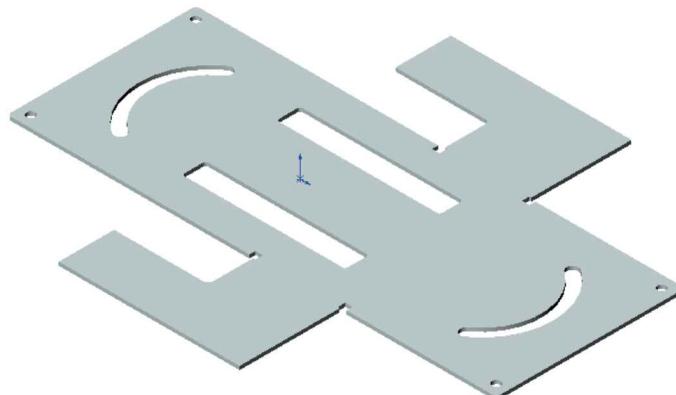
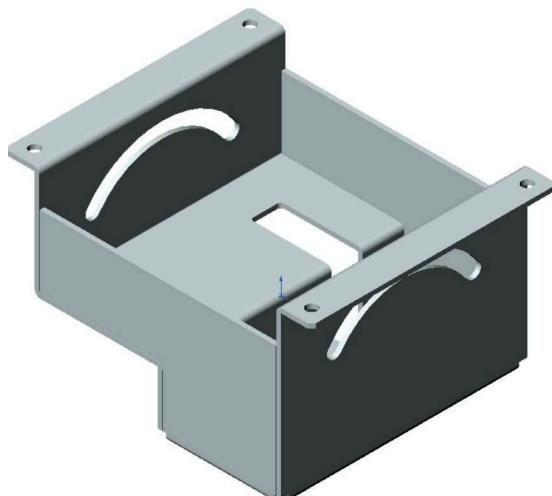
After the part is opened in SOLIDWORKS, there are several methods to convert it to a sheet metal part, but the sheet metal parameters such as Rip, Fixed face or edge, Bend radius, etc., must be added before the Flat Pattern can be created.

The converted part appears on the Feature Manager Design tree; it contains the features Sheet Metal1, Flatten Bend1, and Process Bend1.

The sheet metal part can now be Flattened and Folded by toggling the Suppression state of the Process Bends.

Sheet Metal Conversions

From IGES to SOLIDWORKS Flat Pattern



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI
Units: INCHES – 3 Decimals

Tools Needed:



Convert to Sheet Metal



Insert Bend



Flat Pattern



Sheet Metal Gusset



Flatten Bend

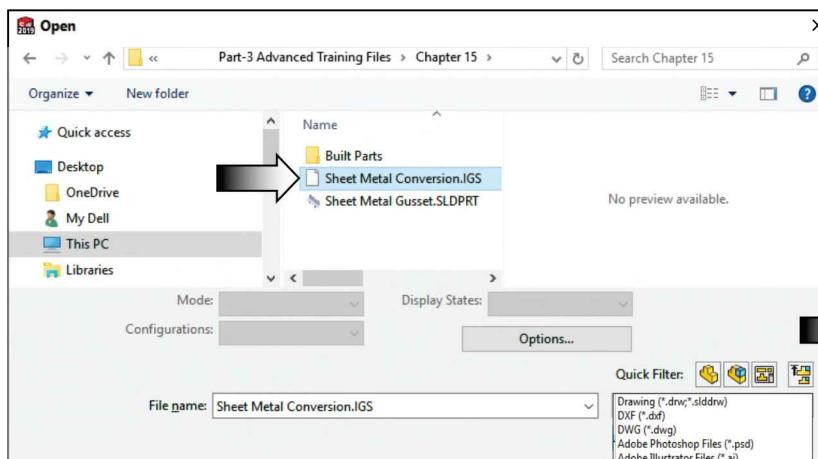


Process Bends

1. Opening an IGES document:

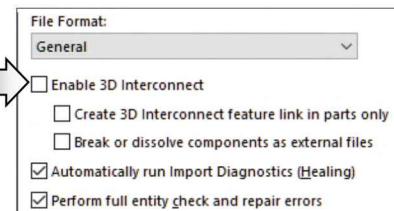
Go to **File / Open**. Change Files of Type to **IGES**.

Browse to the Training Files folder and open: **Sheet Metal Conversion**.



NOTE: The **3D-Interconnect*** option should be disabled if you have trouble opening the iges document.

Go to: Tools, Options, System Options, Import to disable it.

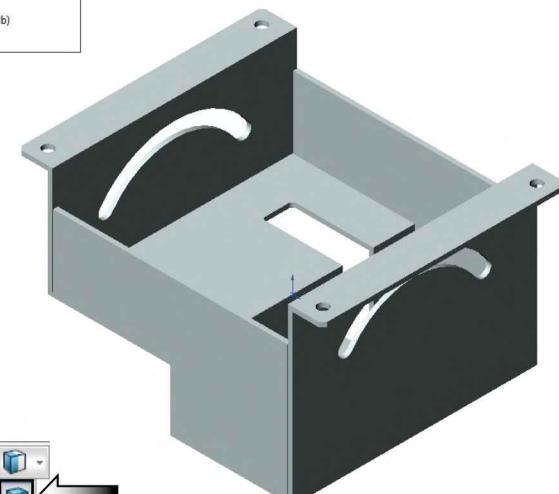
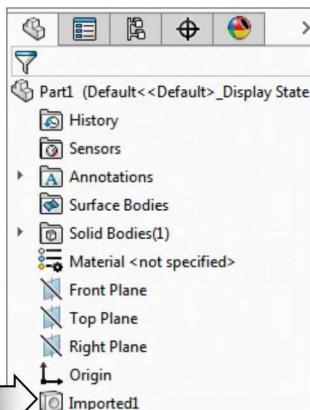


*The **3D Interconnect** option opens the proprietary 3D CAD format in the SOLIDWORKS software with its associative link to the original part.

Click **No** [No] to skip the Import-Diagnostics option.

Click **No** [No] to skip the Feature-Recognition option.

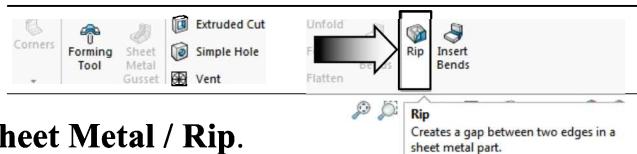
The part is imported into **SOLIDWORKS** as solid body with no feature history.



Change the Display Style to: **Shaded-With Edges**.

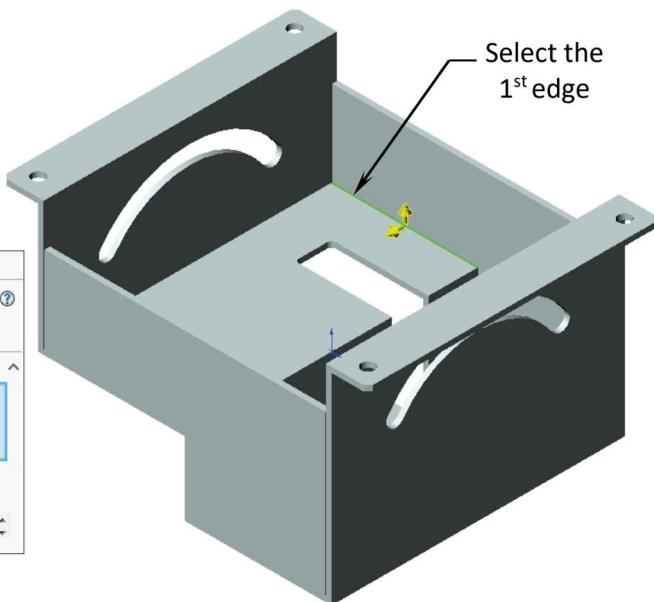
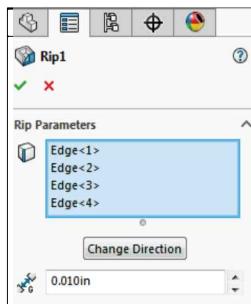
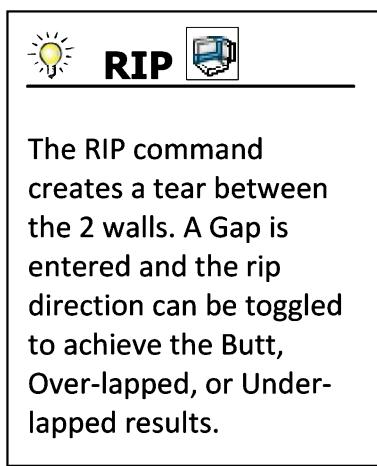
2. Creating the Rips:

Click **Rip**  on the sheet metal toolbar or select **Insert / Sheet Metal / Rip**.

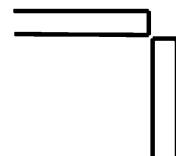


Select the **inner edge** as shown.

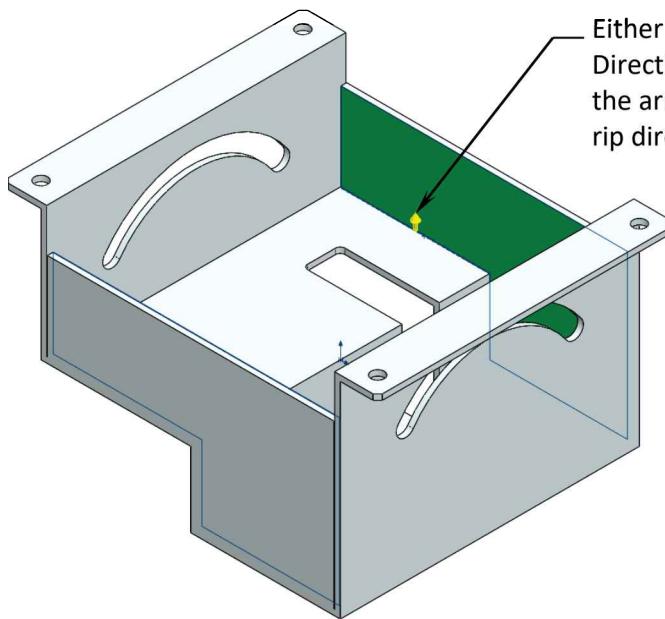
The 2 arrows indicate that the Rip command is going to cut both walls.



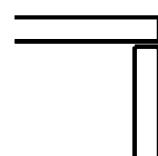
Click on the direction arrow as noted, to rip only the side where the arrow is.



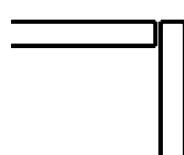
Use the **Default Gap** (.010).



Default



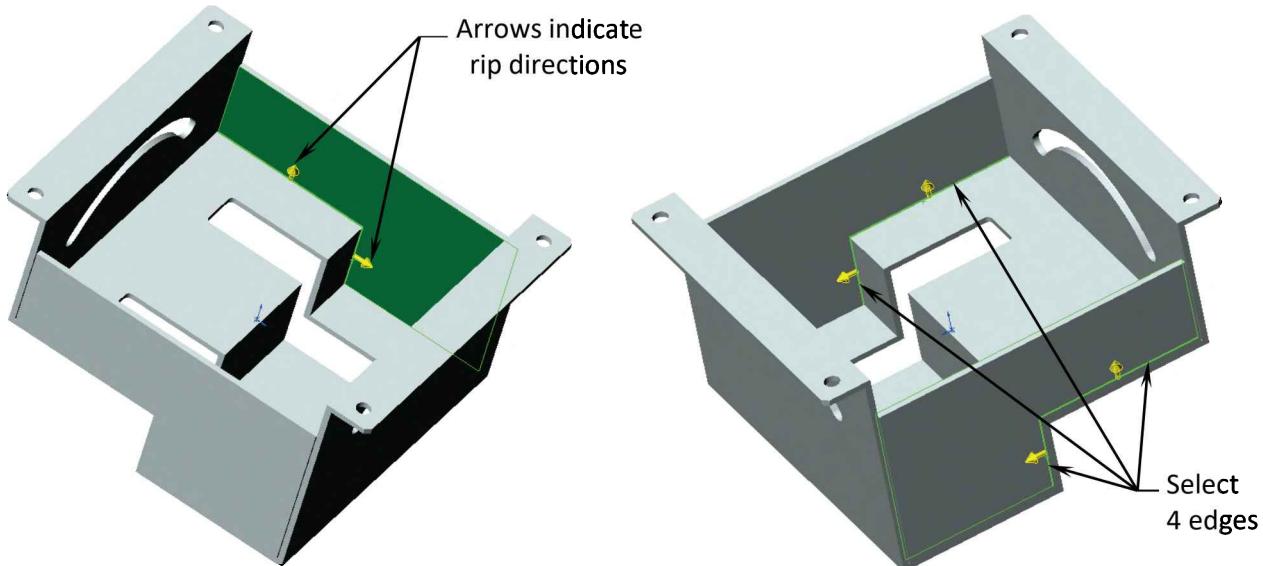
**Direction 1
(Over lapped)**



**Direction 2
(Under lapped)**

Select a total of **4 edges** (2 on each side) as indicated.

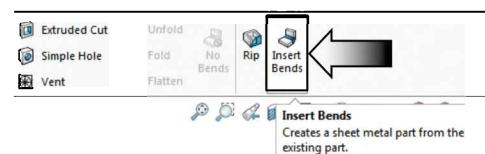
Click **OK**.



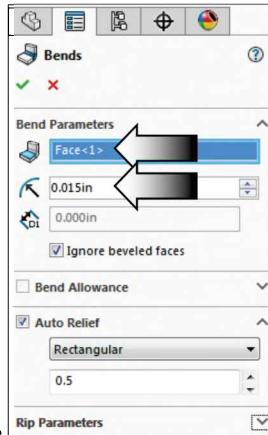
3. Inserting the Sheet Metal Parameters:

Click **Insert Bends**  command or select **Insert / Sheet Metal / Bends**.

Select the inside face to use as the Fixed Face.

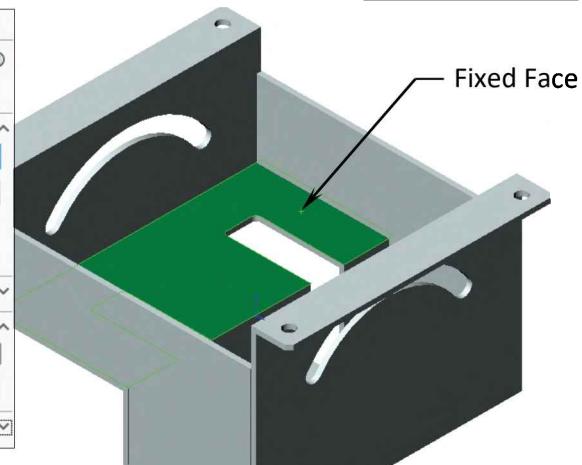


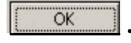
Enter **.015in.** for inside Bend Radius.



Click **OK**.

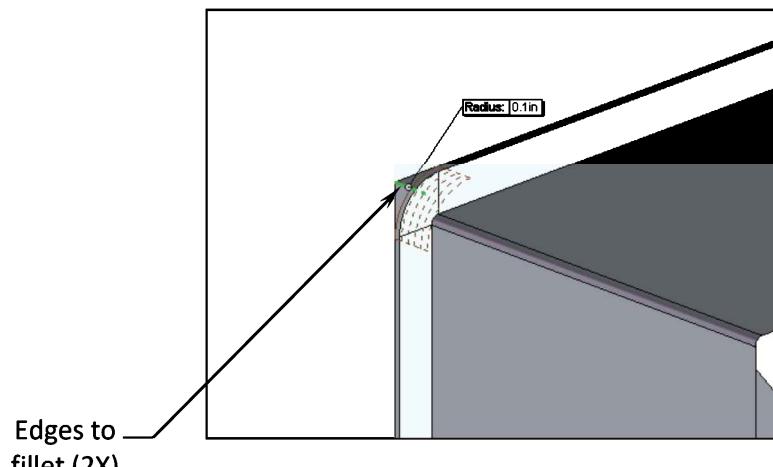
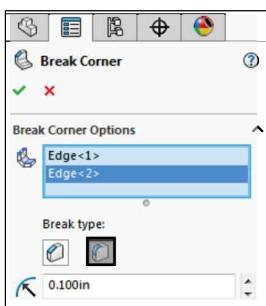
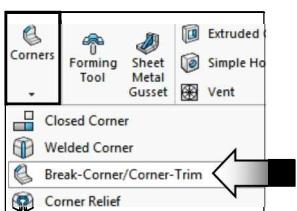
A message pops up indicating some Auto Relief Cuts were added.



Click .

4. Adding Fillets:

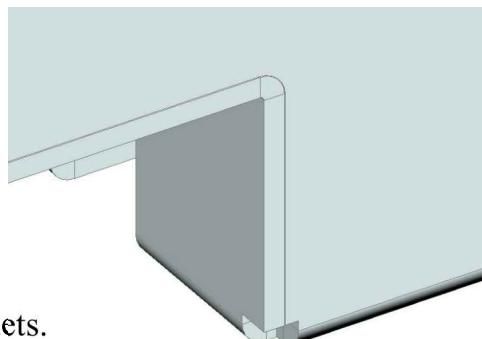
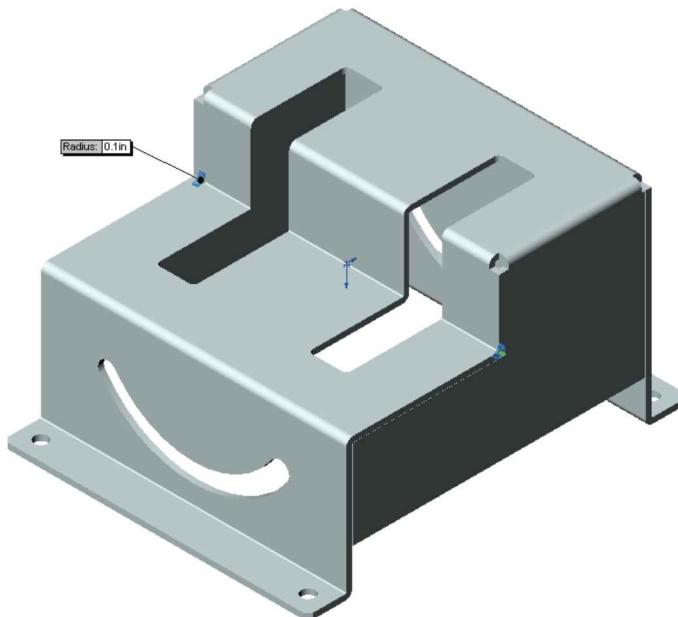
Click **Break Corner / Corner Trim** command  and select the **Fillet**  option.



Enter **.100 in.** for Radius.

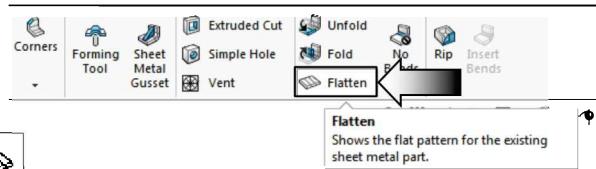
Select the **2 edges** as noted.

Click **OK**.

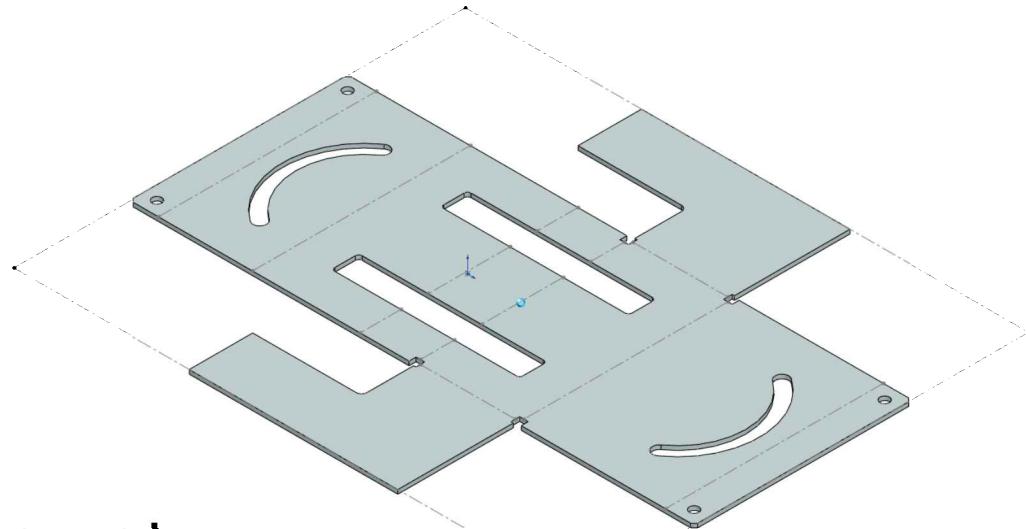


Rotate the model to verify the resulted fillets.

5. Switching to the Flat pattern:



To examine the part in the flattened view, click the **Flatten** command on the Sheet Metal tab (arrow).

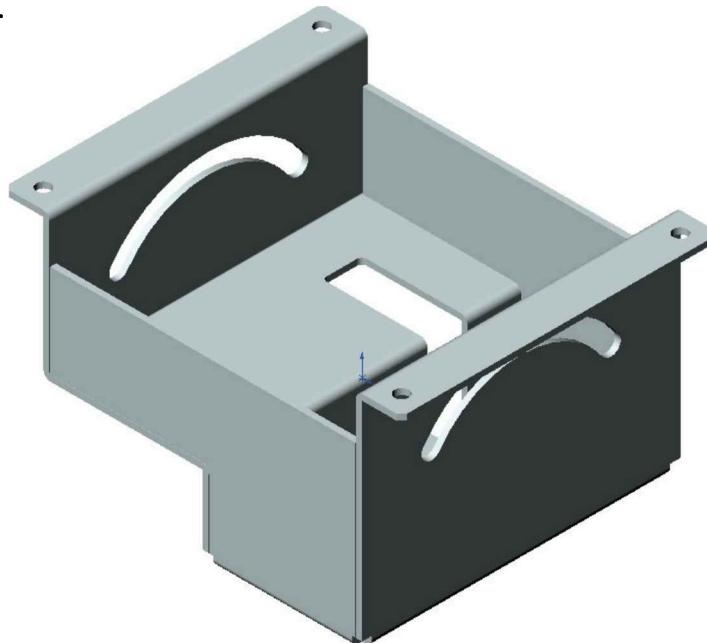


6. Saving your work:

Click **File / Save As**.

Enter **Sheet Metal Conversion** for the file name.

Click **Save**.



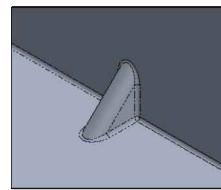
Questions for Review

1. An IGES file can be imported into SOLIDWORKS and converted into a sheet metal part.
 - a. True
 - b. False
2. DXF and DWG are imported into SOLIDWORKS as 2D Sketches, using the DXF/DWG Import-Wizard.
 - a. True
 - b. False
3. After being imported into SOLIDWORKS, the IGES file can be flattened instantly.
 - a. True
 - b. False
4. The imported parts must have a uniform thickness to fold and unfold properly.
 - a. True
 - b. False
5. The Rip feature removes 1 material thickness based on the direction arrow that you select.
 - a. True
 - b. False
6. When applying the sheet metal parameters, you do not have to specify a fixed face.
 - a. True
 - b. False
7. The width and depth of the relief cuts are fixed and cannot be changed.
 - a. True
 - b. False
8. The Folded and the Flat pattern can be toggled by moving the Rollback Line up or down.
 - a. True
 - b. False

1. TRUE
2. TRUE
3. FALSE
4. TRUE
5. TRUE
6. FALSE
7. FALSE
8. TRUE

Sheet Metal Gussets

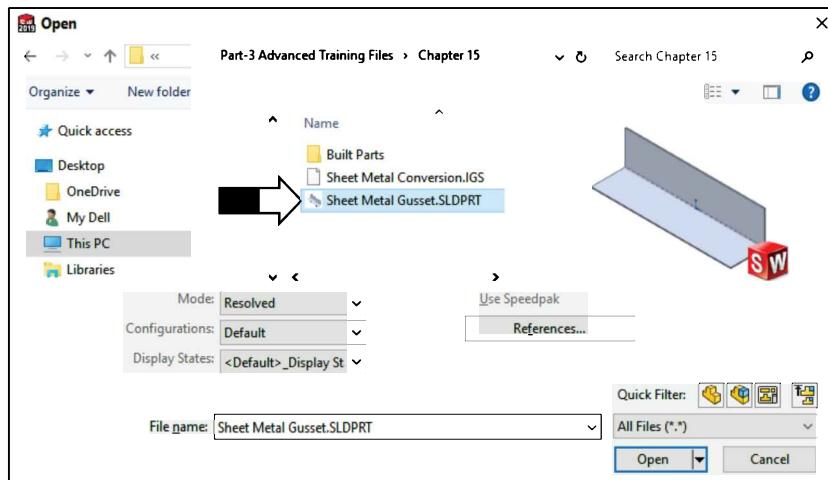
Sheet Metal gussets can be created in SOLIDWORKS with specific indents that go across bends. This exercise will guide us through the creation of a gusset in a sheet metal part.



1. Opening a sheet metal part document:

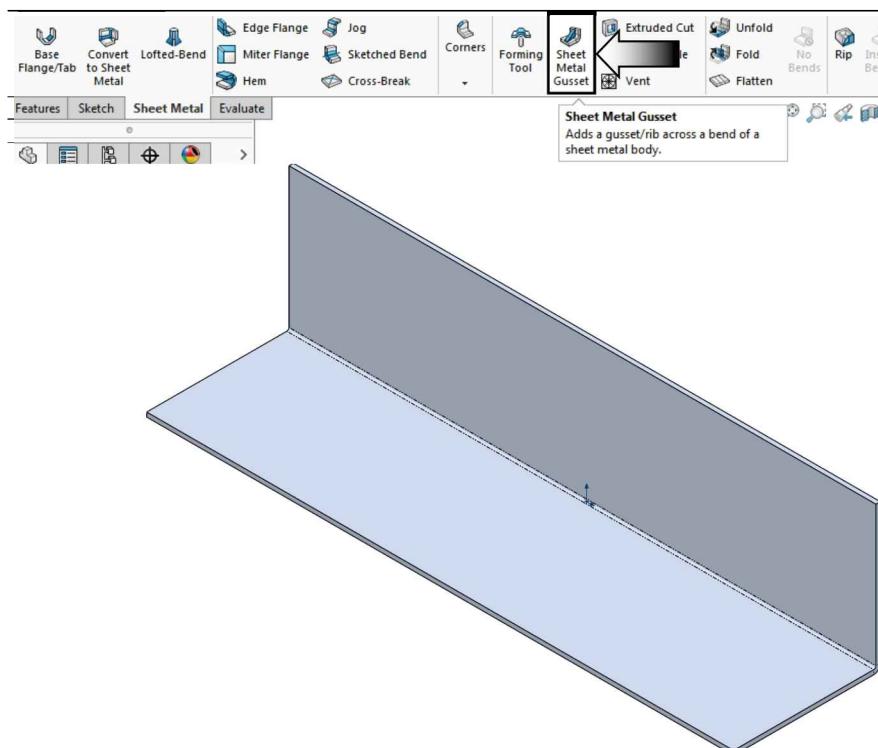
Click File / Open.

Browse to the Training files folder and open a part document named: **Sheet-Metal Gusset**.

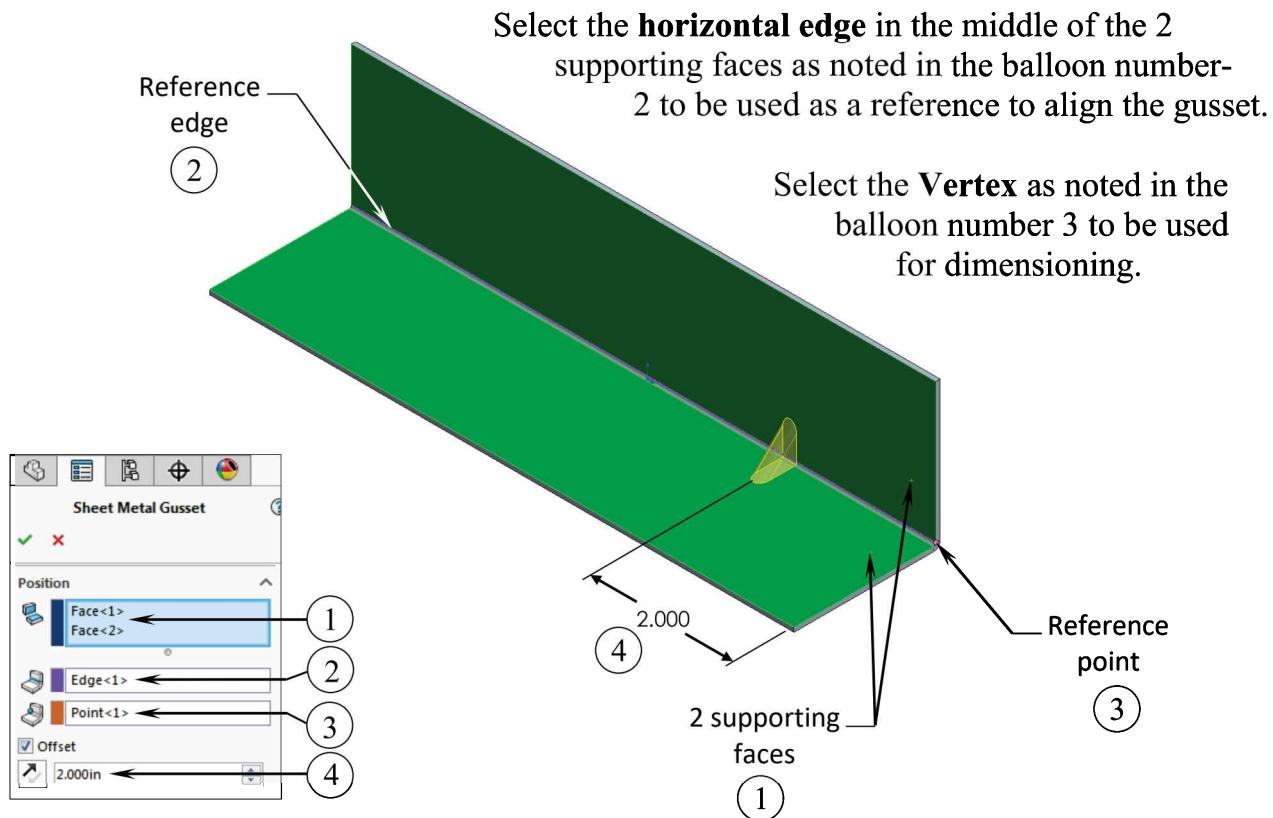


2. Creating a gusset:

Click the **Sheet Metal Gusset** command from the Sheet Metal tool tab (arrow).



Under the **Position** section, select the **2 Faces** as indicated in the balloon number 1 for **Supporting Faces**.

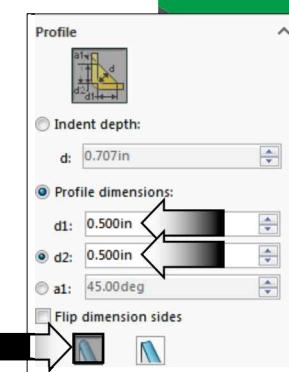
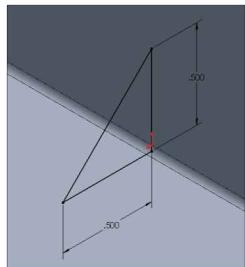


Move down to the **Profile** section and click the **Profile Dimensions** option (arrow).

Enter **.500in.** for Profile Length.

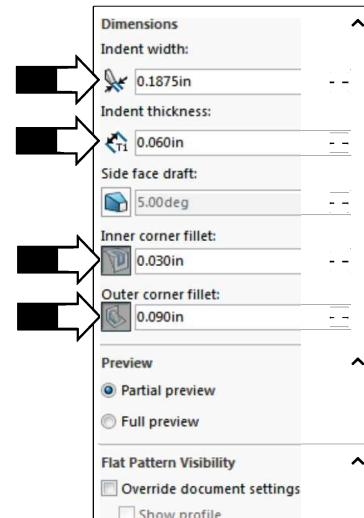
Enter **.500in.** for Profile Height.

Click the **Rounded Gusset** button to create a gusset with a rounded edge.



Scroll down to the Dimensions section and enter the following:

- * Indent Width Dimension: **.1875in.**
- * Indent Thickness Dimension: **.060in.**
- * Inner Corner Fillet: **.030in.**
- * Outer Corner Fillet: **.090in.**



Enable the **Full Preview** option in the Preview section, if needed.

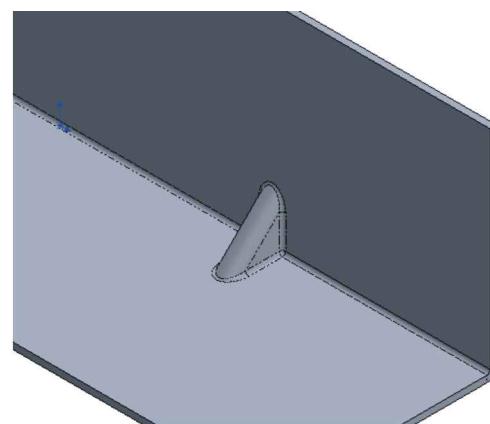
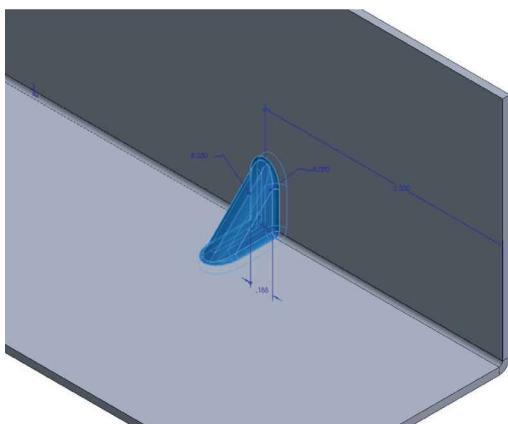
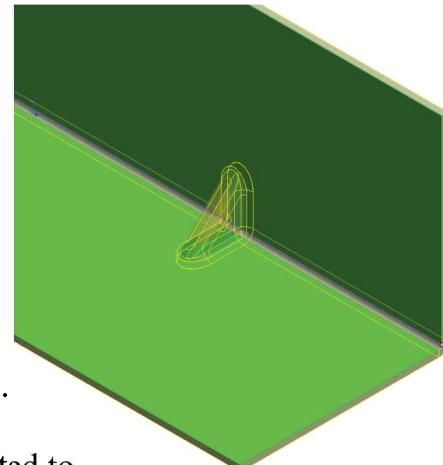
Click **OK** to create the gusset.

3. Viewing the resulted gusset:

Zoom in on the new gusset to see its details.
Also rotate the view to see the indent from the back side.

Click on the feature itself to see its dimensions.

The gusset has a built-in sketch that can be edited to change to a custom profile if needed.



4. Mirroring the gusset:

Switch to the Features tab and click the **Mirror** command.

Select the **Right** plane from the Feature tree for Mirror-Plane.

For Features-to-Mirror, select **Gusset1** also from the FeatureManager tree.

Click **OK**.

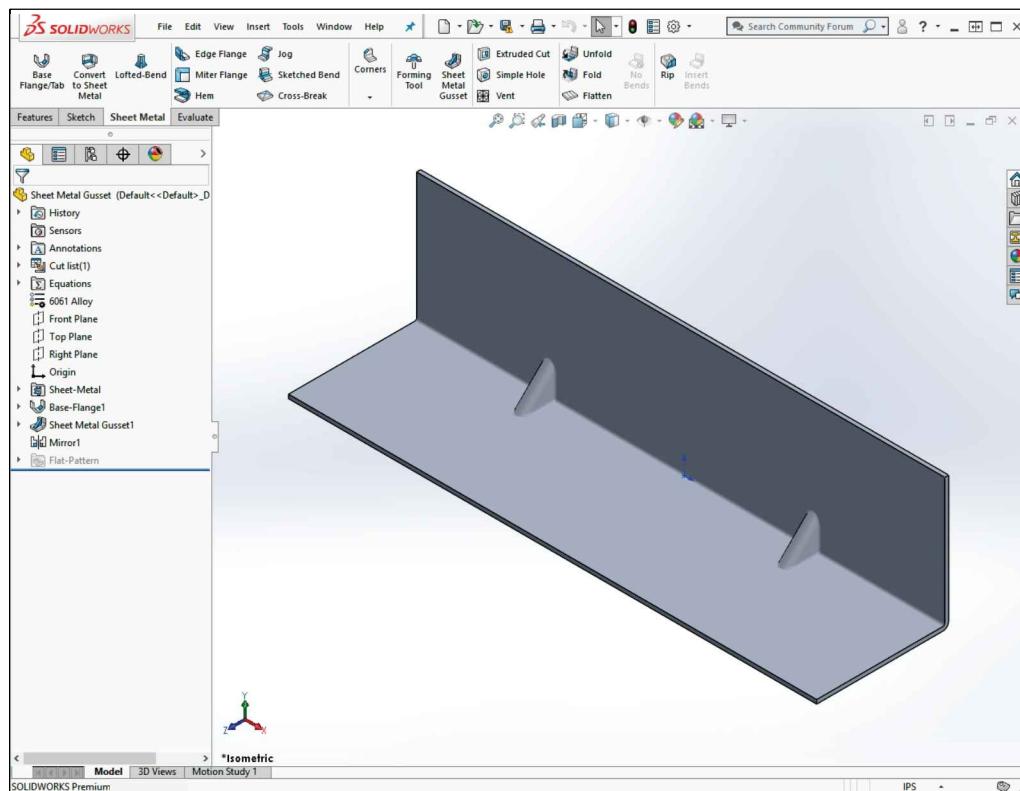
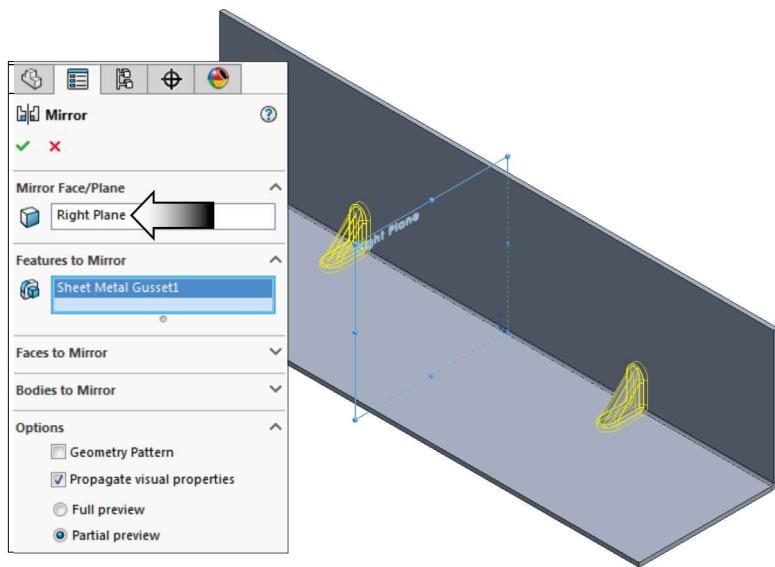
5. Saving your work:

Click **File / Save As**.

Enter: **Sheet Metal Gusset** for the name of the file.

Click **Save**.

Click **Yes** to replace the old file with the new when prompted.



Exercise: Hem & Vent Features

The Hem tool creates a Hem feature from a linear edge. There are 3 types of Hem available: Closed, Open, and Tear Drop.

The Vent feature uses a sketch to define the Boundary, Spars, Ribs, etc.

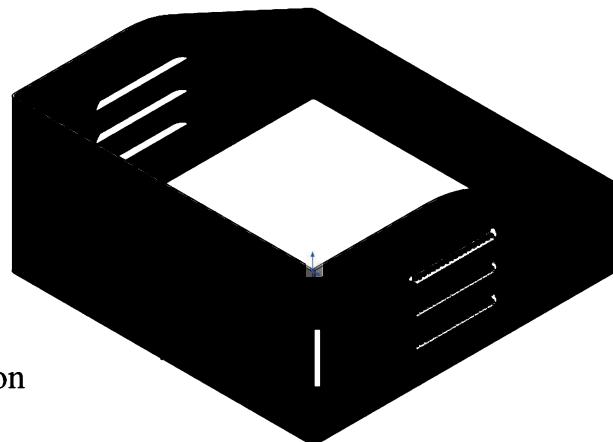
This exercise focuses on the use of the Hem and Vent Sheet Metal features.

1. Opening a part document:

Select **File, Open**.

Browse to the Training folder and open a part document named:
Hem & Vent.sldprt.

The sketch for the Vent feature has already been created to help focus on the use of the Vent tool.

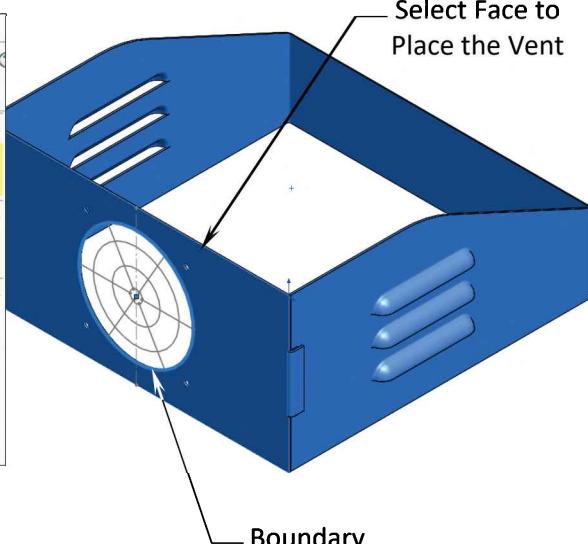
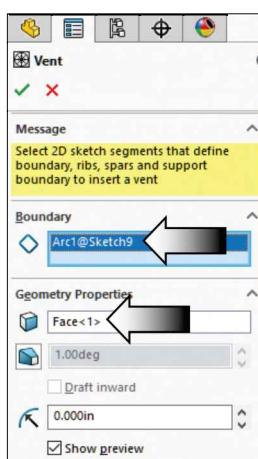


2. Creating a Vent feature:

Select **Sketch9** from the FeatureManager tree and click:
Insert, Fastening Features, Vent
(or click the Vent icon on the Sheet Metal tab).

For Boundary,
select the **large circle** as noted.

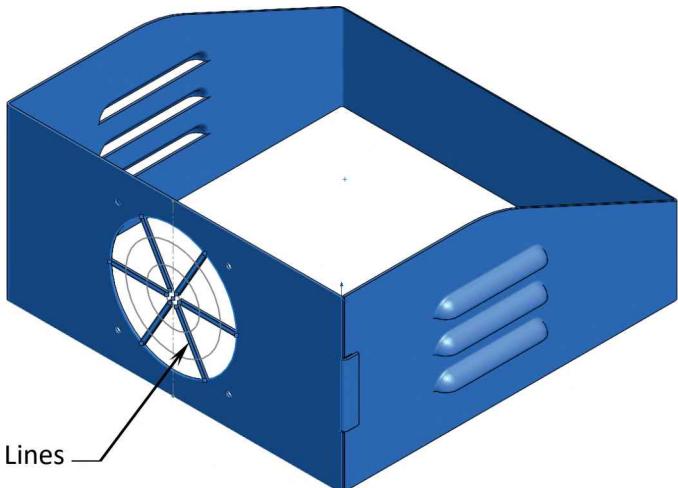
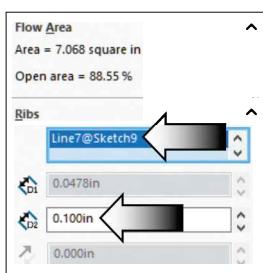
For Face to place
the Vent feature,
select the **front face** of the part.



We will change
the radius in the next few steps.

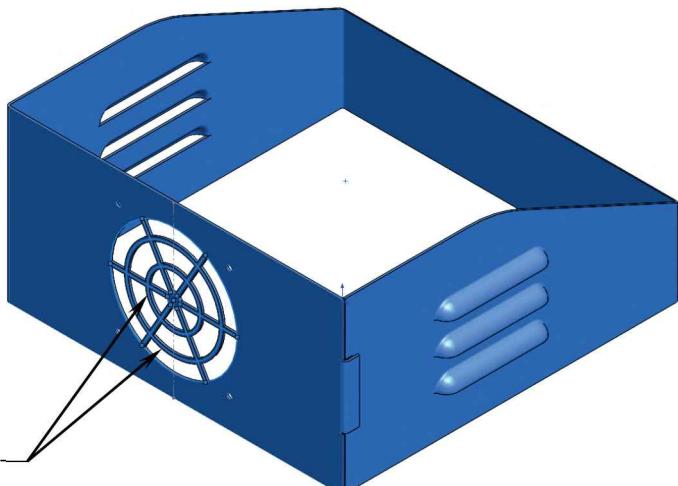
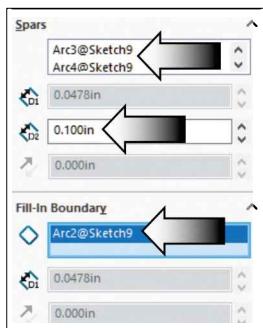
For Ribs, select the **6 lines** in the graphics area as noted.

For Rib Thickness, enter **.100in**.

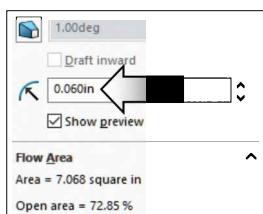


For Spars, Select the **2 circles** as indicated.

For Spar Thickness, Enter **.100in**.

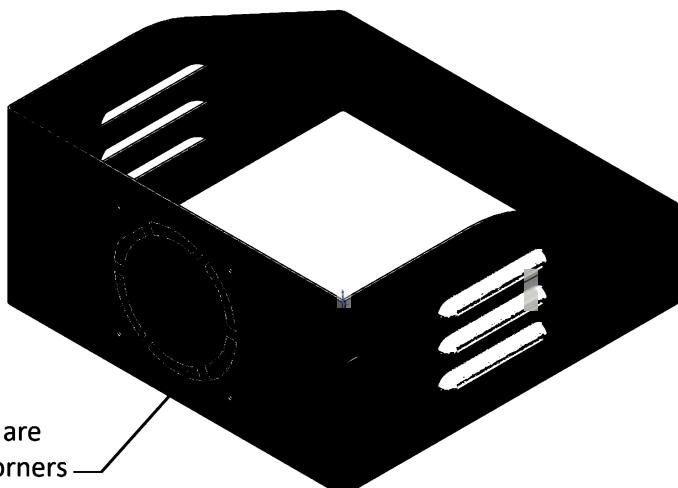


Move back up the tree, click in the **Radius** section and enter **.060in**.



Click **OK**.

.060" Fillets are added to all corners



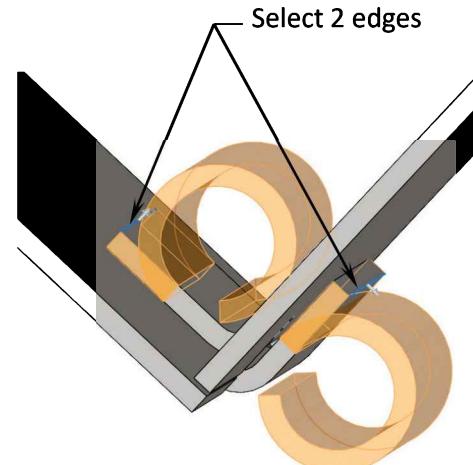
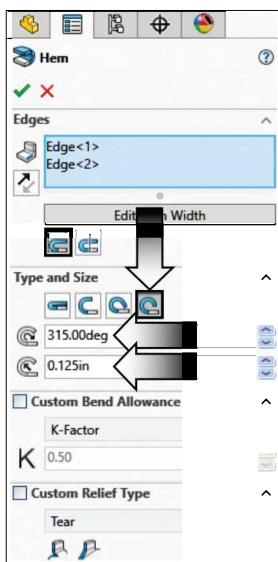
3. Creating a Hem feature:

Select **Insert, Sheet Metal, Hem** (or click the Hem icon on the Sheet Metal tab).

For Edges, select the **2 edges** of the 2 tabs as indicated.

For Angle, enter : **315.00deg**.

For Radius, enter: **.125in**.

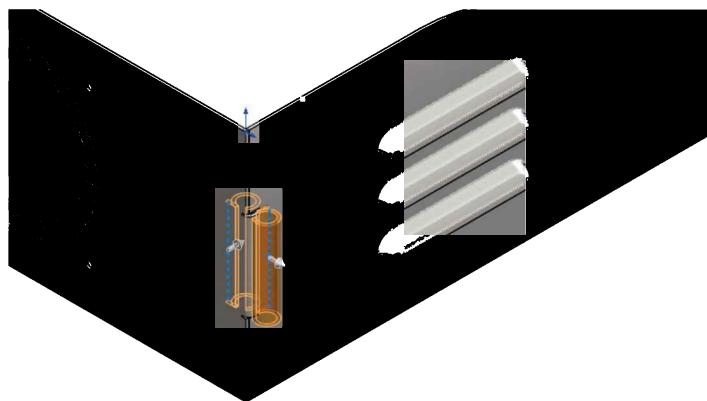


Set other parameters as follows:

K-Factor: .5

Custom Relief Type: **Tear**.

Click **OK**.

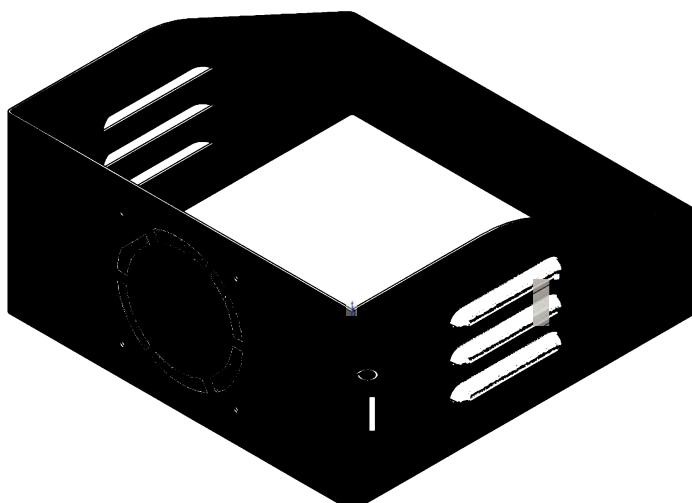


4. Saving your work:

Select **File, Save As**.

Enter: **Hem & Vent (completed).sldprt** for the file name.

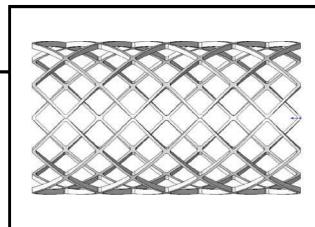
Click **Save**.



Close all documents.

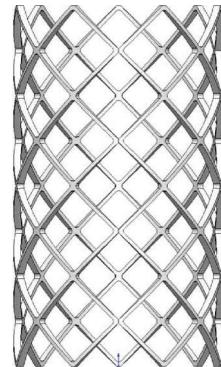
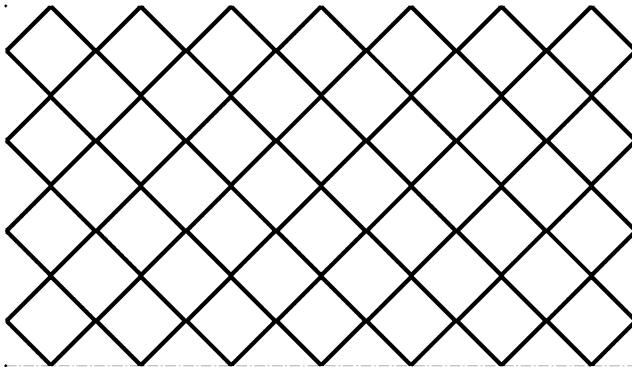
Flat Pattern Stent

Flat Pattern Stent A Different Approach



Using the built-in Sheet Metal features in SOLIDWORKS you can flatten or roll solid models such as wire mesh screens, grill meshes, or stent patterns.

When designing a sheet metal part, the material setback is something we must keep in mind: The Bend allowance and bend deduction calculations are methods you can choose to determine the flat length of sheet stock to give the desired dimension of the bent part. This lesson uses the default settings of the K-Factor to calculate the bend allowance ($BA=P(R + KT) A/180$).

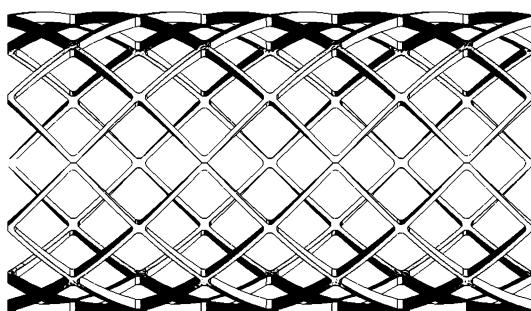
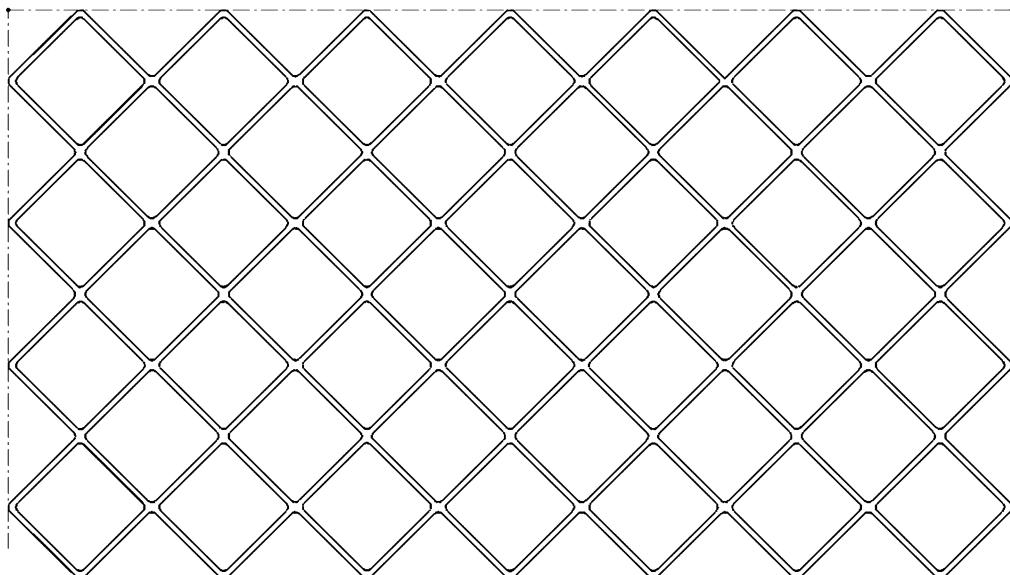


K-Factor is a ratio that represents the location of the neutral sheet with respect to the thickness of the sheet metal part. When you select K-Factor as the bend allowance, you can specify a K-Factor bend table. The SOLIDWORKS application also comes with a K-Factor bend table in Microsoft Excel format.

There are several known methods to create these types of patterns; this lesson will walk you through the use of rolling and unrolling a cylinder and its pattern, using the Sheet Metal functions in SOLIDWORKS.

Flat Pattern Stent

A Different Approach



Dimensioning Standards: **ANSI**

Units: **INCHES – 3 Decimals**

Tools Needed:



Revolve Boss-Base



Flatten



Unfold



Fold



Extruded Cut



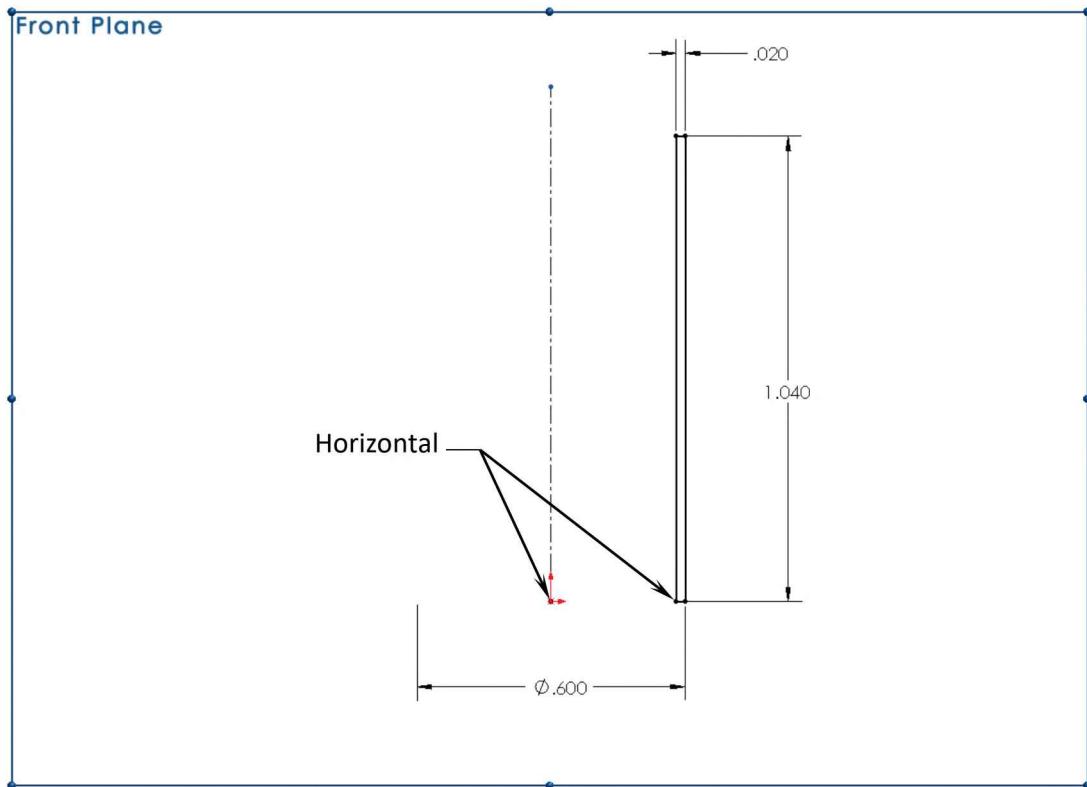
Fillet / Round

1. Starting with a new part document:

Click **File / New / Part**, set the units to **Inches, 3 decimal places**.

Select the Front plane and open a **new sketch**.

Create the sketch and add the dimensions / relation shown below.
(The dimensions are scaled up for ease of modeling purposes.)



Click **Revolve / Boss-Base** .

The revolve centerline is selected automatically.

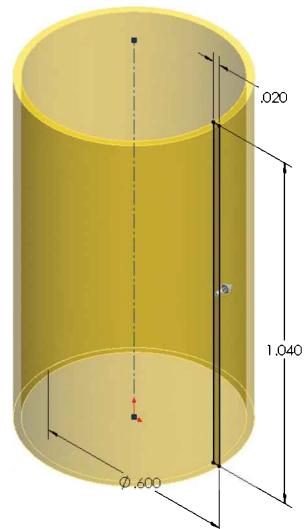


Use the default **Blind** type.

Enter **359.9deg** for angle.

Click **OK**.

(The gap is needed to flatten the sheet metal part later on.)



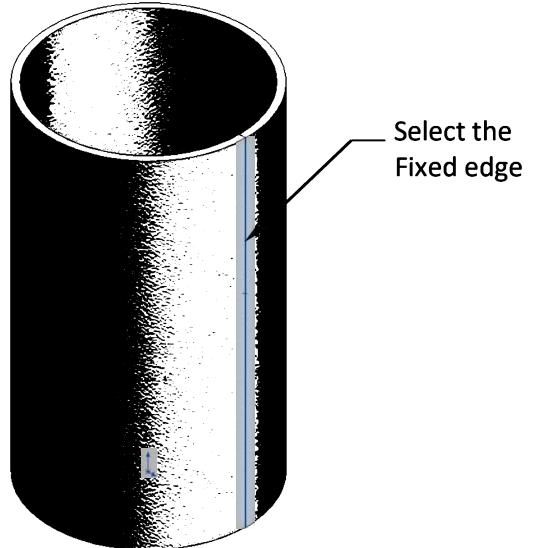
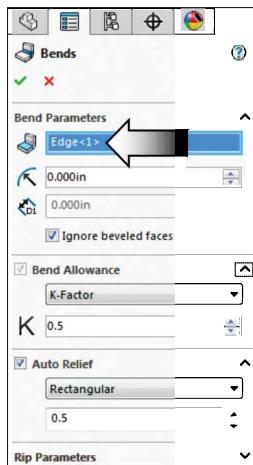
2. Converting to Sheet Metal :

Click the **Insert-Bends** command on the Sheet Metal tab.

For Fixed Edge/Face select the edge on the left side as noted.

Enter **0** for bend radius.

Use the default K-Factor and Auto Relief settings.



Click **OK**.

The solid model is converted to a Sheet Metal part. Press the **Flatten** command on the Sheet Metal toolbar to see its flat pattern.

Click the **Flat Pattern** command again to roll the part back to its default stage.

3. Unfolding the part:

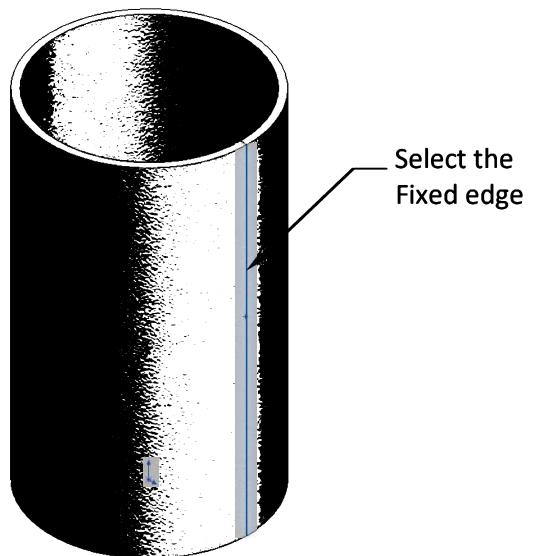
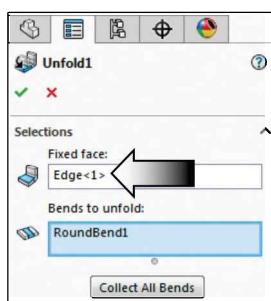
Select the **Unfold** command  from the Sheet Metal toolbar.

Select the same edge to keep as the Fixed Edge/Face.

Click the **Collect-All-Bends** button (arrow).

Click **OK**.

The part is flattened but this time new features can be added and they will roll back when the part is folded.

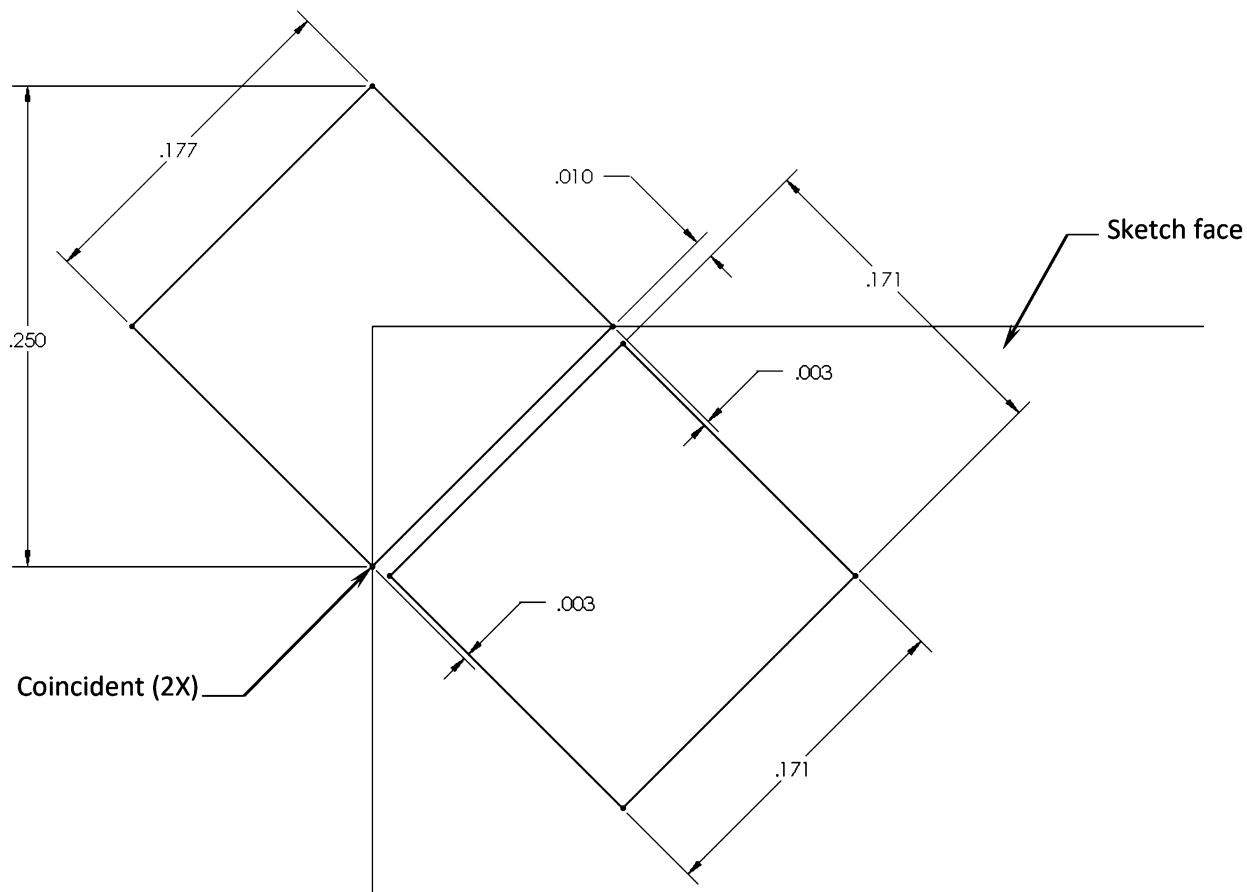


4. Adding the sketch pattern:

Select the face as noted and open a **new sketch**.

Sketch a couple of **squares** (notice the upper square is slightly larger than the lower one by .006").

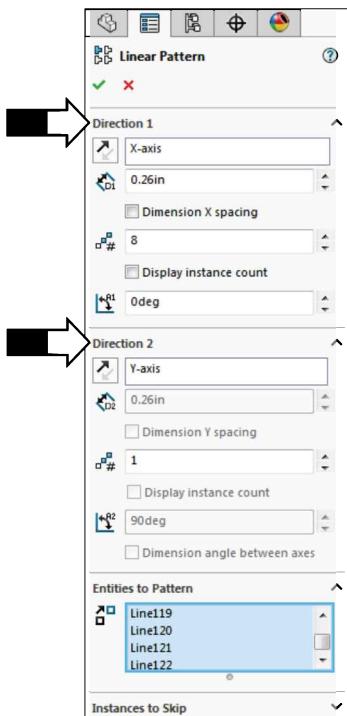
Add the dimensions and relations shown in the image to fully define the sketch.



Additional relations such as **Parallel**, **Equal**, or **Perpendicular** can also be used to help eliminate some of the redundant dimensions.

We are going to use the Linear Sketch Pattern command to repeat the two squares several times, so there are a few things to keep in mind:

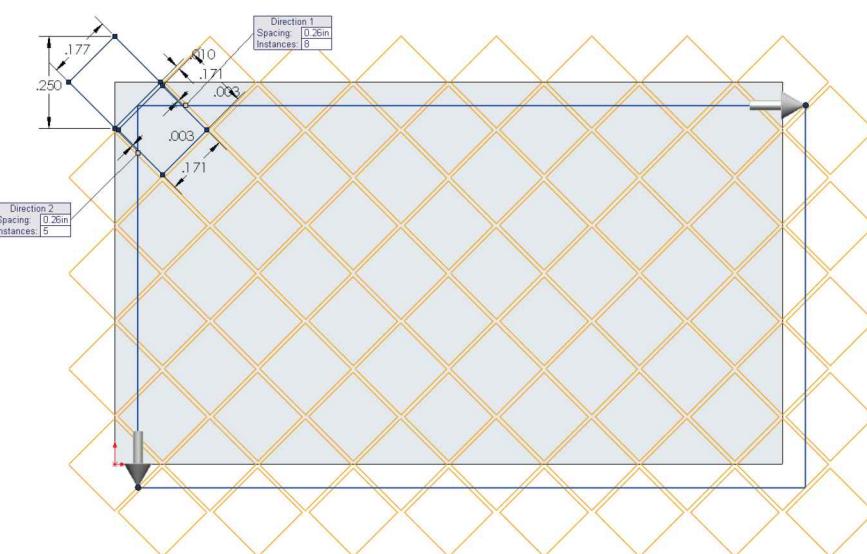
- * Pre-select the entities to pattern.
- * Auto add spacing dimensions.
- * Pattern along 2 directions, use angle (0deg and 270deg for directions).



Select both squares and click the **Linear-Sketch-Pattern** command from the Sketch toolbar, or select it from the drop-down menus: **Tools / Sketch Tools**.

Under **Direction 1**, enter / select the following:

- * .260in * Dimension X Spacing enabled.
- * 8 Instances * Display Instance Count enabled.
- * 0deg

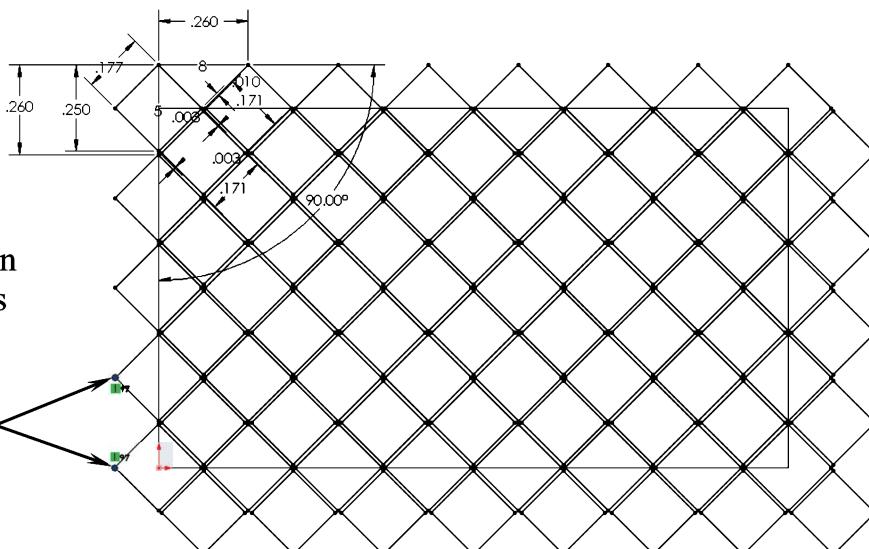


Under Direction 2, enter or select the following:

- * .260in * Dimension Y Spacing enabled.
- * 5 Instances * Display Instance Count enabled.
- * 270deg * Dimension between Axes enabled.

Add a vertical relation between the 2 end points as indicated.

Vertical

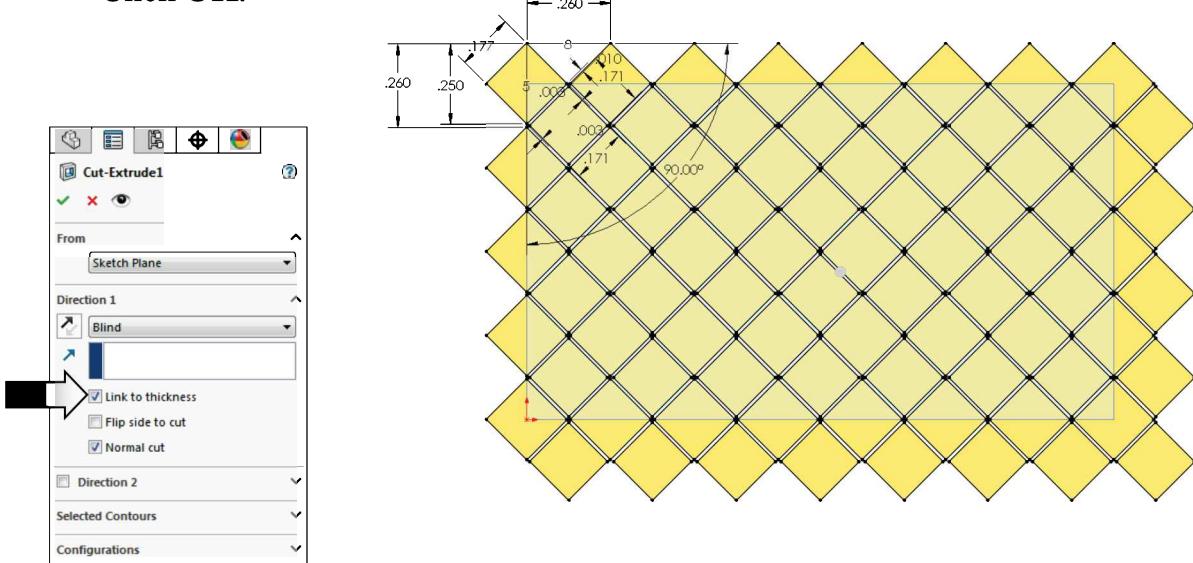


5. Creating a cut with Link to Thickness:

Select the **Extruded Cut**  command from the Sheet Metal toolbar.

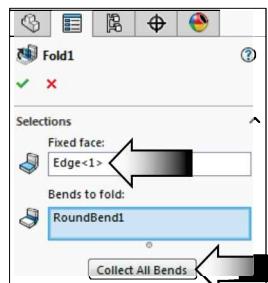
Use the default **Blind** option and enable the **Link To Thickness** checkbox.

Click OK.



6. Folding the part:

Click the **Fold**  command from the Sheet Metal toolbar.

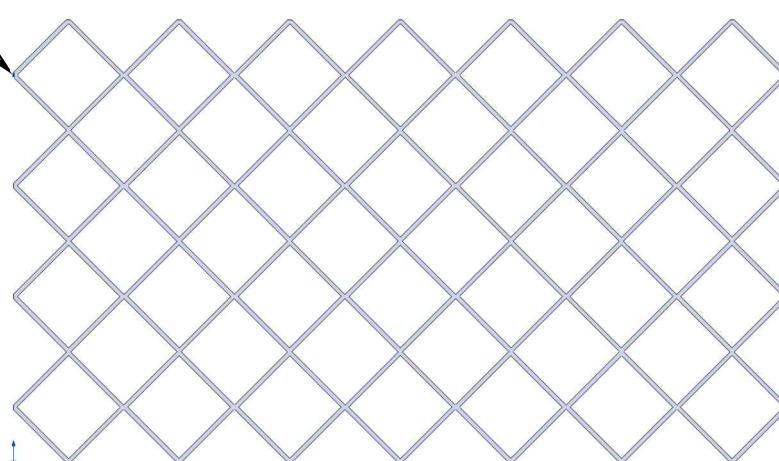


Select the fixed edge

Select the small vertical edge as noted for Fixed Face/Edge.

Click the **Collect-All-Bends** button.

Click OK.



7. Creating a new configuration:

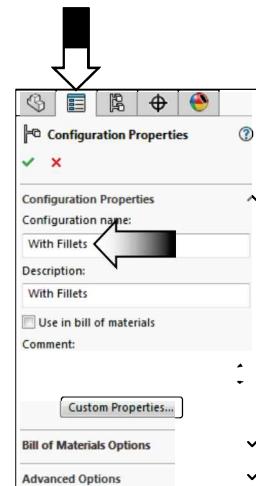
Before adding the fillets to all sharp corners, we will need to create a configuration so that the fillets can be added and captured in a separate configuration.

Switch to the ConfigurationManager (arrow).

Right-click the name of the part (on the top of the tree) and enter: **With Fillets** (arrow) for the name of the new configuration.

Click **OK**.

Rename the Default configuration to:
Without Fillets (arrow).



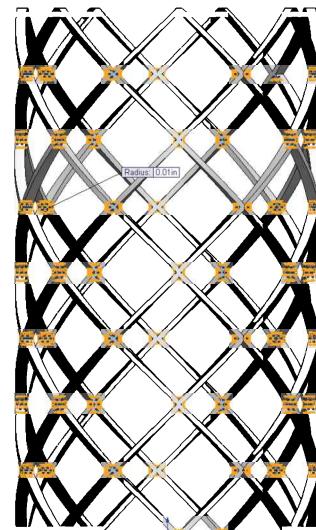
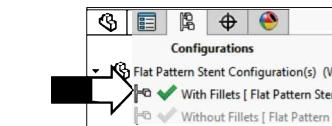
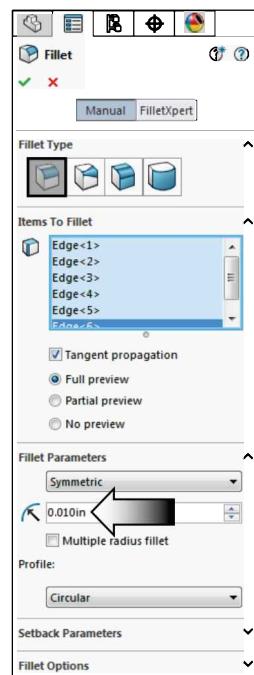
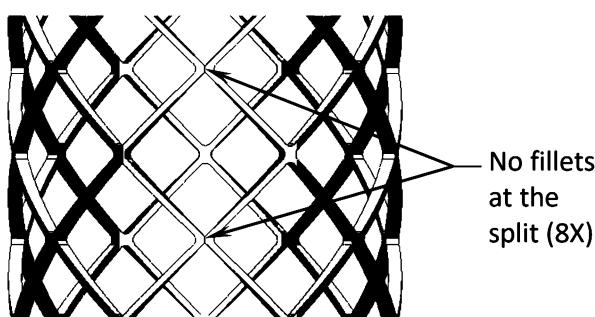
8. Adding the .010" fillets:

Click the **Fillet**  command from the Features toolbar.

Use the default **Constant Radius** option.

Enter **.010in** for radius.

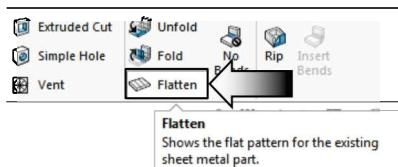
Select all edges except for the ones at the two ends as indicated.



Click **OK**.

9. Switching to Flatten mode:

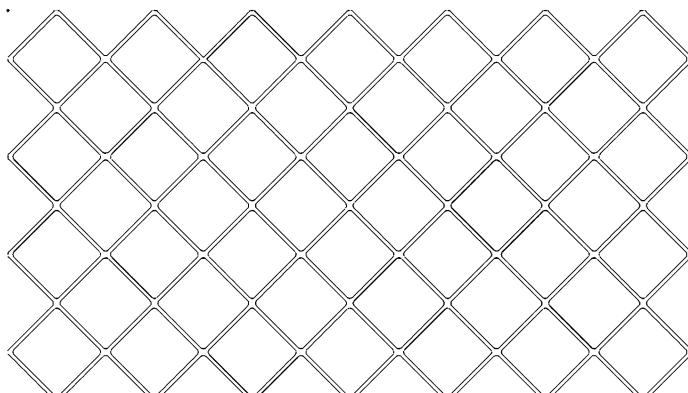
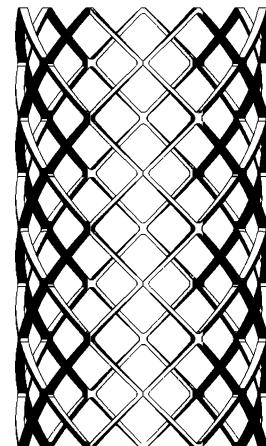
Click **Flatten**  on the Sheet Metal toolbar.



The part is flattened and the cut pattern is unrolled with it.

Use the Flatten command to flatten the sheet metal part to check its dimensions, get a printout from it, or export it as a DXF or DWG to use in manufacturing.

Use the Fold and Unfold commands to flatten the part and add new features, so that these features can roll or unroll with the part.



10. Switching configuration:

Switch back to the **Without Fillets** configuration by double-clicking on its name.

Click the **Flatten** command again to verify the pattern.

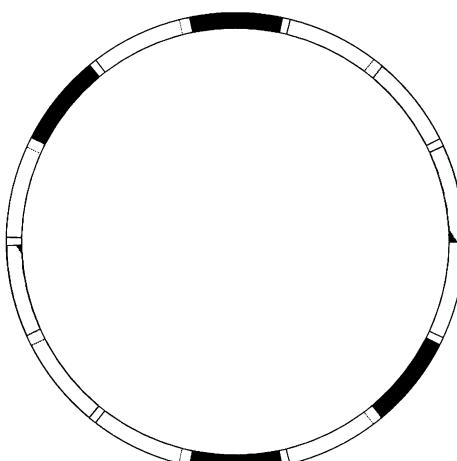
At this point, the pattern can be exported, or a drawing can be made from it for inspection or documentation purposes.

11. Saving your work:

Click **File / Save As**.

Enter **Flat Pattern Stent** for the name of the file.

Click **Save**.



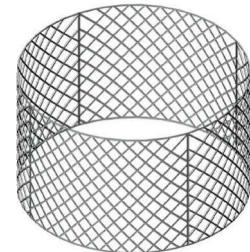
Questions for Review

1. Using SOLIDWORKS, a cylinder or a cone can be unrolled into a Sheet Metal flat pattern.
 - a. True
 - b. False
2. There must be a gap or a slit along the length of the cylinder for it to flatten.
 - a. True
 - b. False
3. A sheet metal part can have more than one thickness.
 - a. True
 - b. False
4. A sheet metal part must have one uniform thickness.
 - a. True
 - b. False
5. The Link-to-Thickness option links the depth-of-cut to the thickness of the part.
 - a. True
 - b. False
6. The K-Factor value is locked to .5; this ratio cannot be changed.
 - a. True
 - b. False
7. Use the Flatten command to flatten the part and add new features.
 - a. True
 - b. False
8. Use the Unfold command to flatten the part and add new features.
 - a. True
 - b. False

1. TRUE
2. TRUE
3. FALSE
4. TRUE
5. TRUE
6. FALSE
7. FALSE
8. TRUE

Exercise: Screen Mesh - Sheet Metal Approach

There are many known methods for creating the screen Mesh. This exercise will show the one that uses the combination of Patterns, Ribs and Combine Common options to create the model shown above. (The dimensions in the model are scaled up for visual purposes.)

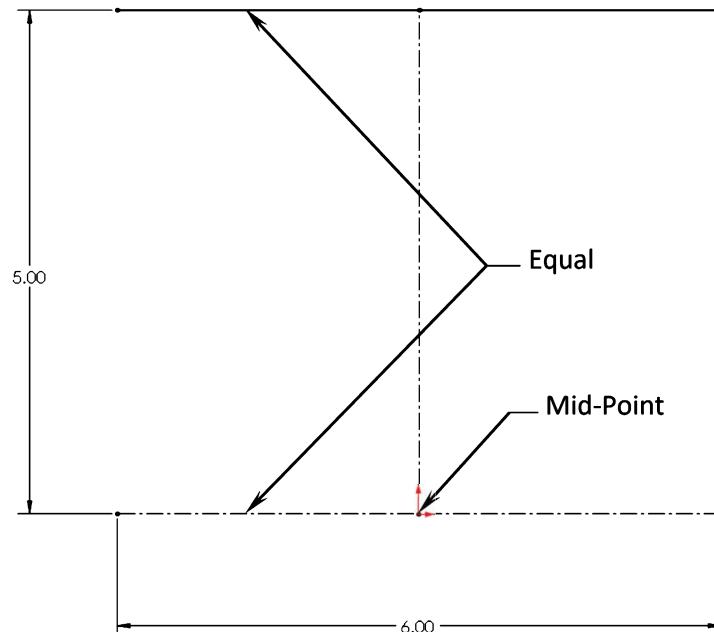


1. Creating the main sketch:

Select the **Front** plane and open a new Sketch.

Sketch a **Line** centered on the origin and **2 Centerlines** as shown.

Add the dimensions to fully define the sketch.



2. Revolving the main body:

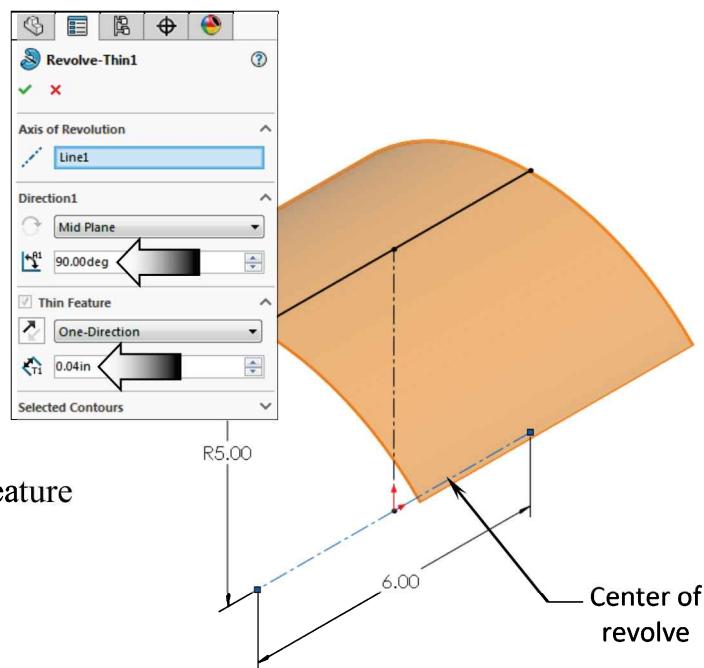
From the Features tab, click **Revolve Boss-Base**.

Set Direction 1 to: **Mid-Plane**.

Set Revolve Angle to: **90deg**.

Set the Thickness under Thin Feature to: **.040in**.

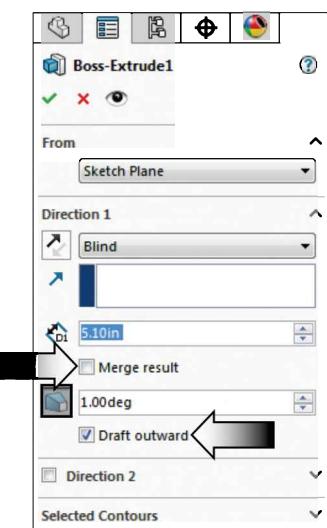
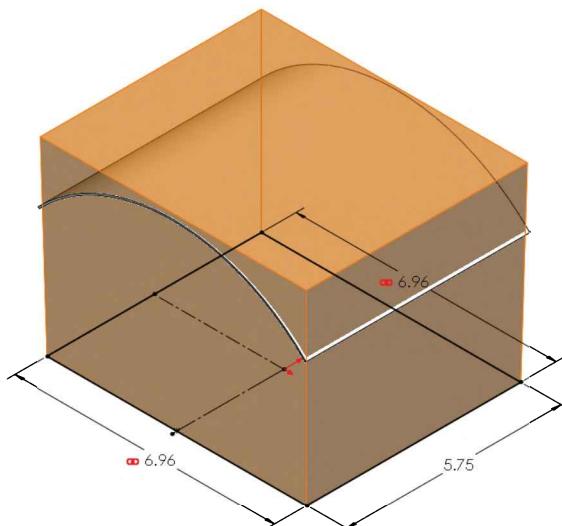
Click **OK**.



3. Creating the 2nd body:

Select the **Top** plane and open a new sketch.

Sketch a **Center Rectangle** and add the dimension shown.



Click Extruded Boss-Base:

Blind: **5.10in**

Merge Result: **Cleared** (arrow).

Draft: **1deg** (arrow).

Draft Outward **Enabled**.

Click OK.

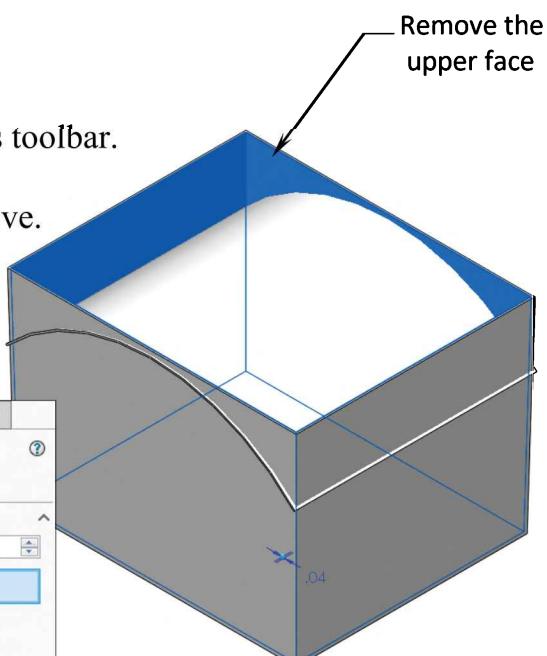
4. Shelling the body:

Click the **Shell** command from the Features toolbar.

Select the **upper face** of the **body2** to remove.

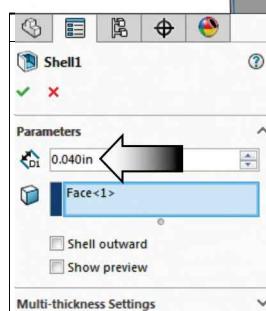
Enter **.040in.** for thickness.

Click OK.



Note:

*Only Body2 is shelled.
The Body1 is set below
the top surface by .100".*

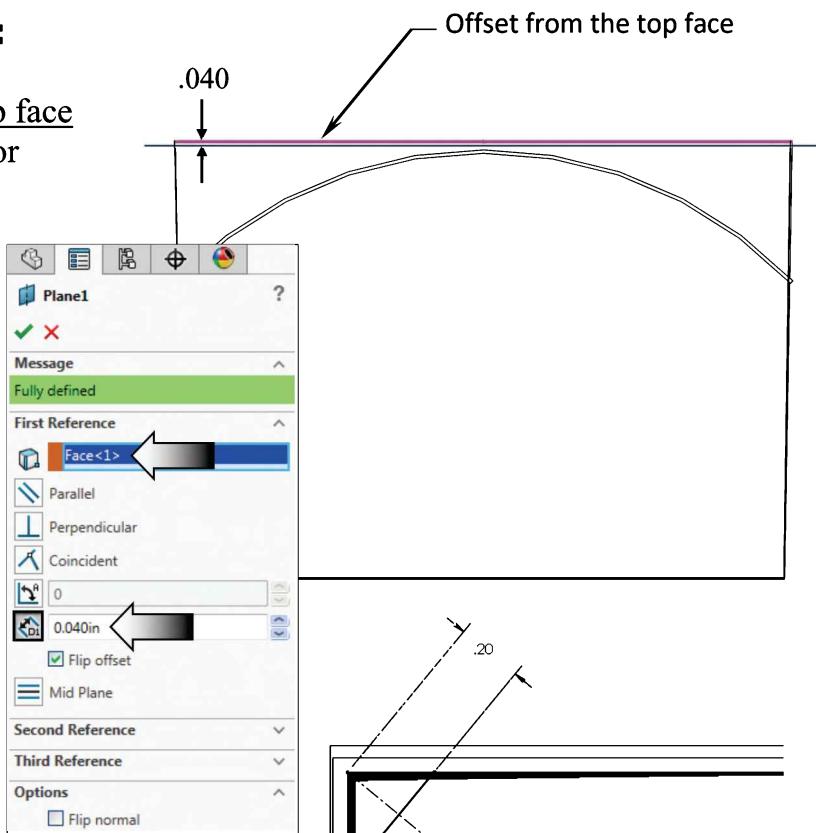


5. Creating an offset plane:

Create a plane from the top face of the part. Enter **.040in.** for the **Offset Distance**.

Click the **Flip** checkbox to place the new plane below the surface.

Click **OK**.



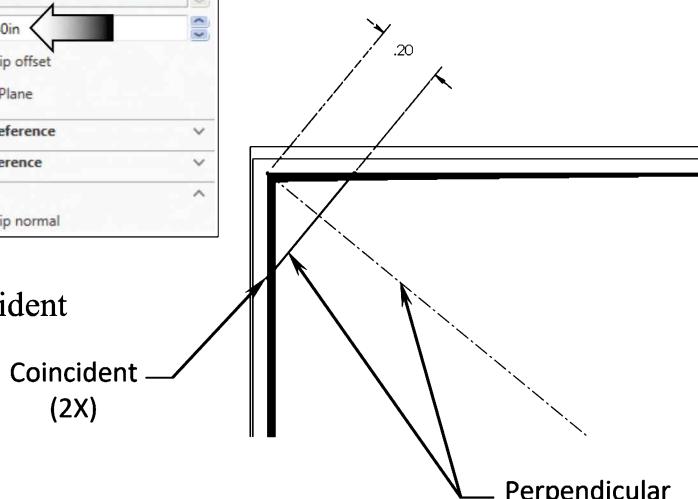
6. Creating the 1st Rib:

Open a **new sketch** on the new plane.

Add a **line** across the walls, near the upper left corner of the part.

Sketch a **centerline** that is coincident to the 2 diagonal corners.

Add the dimension and relations as noted.



Click the **Rib** command from the Features toolbar.

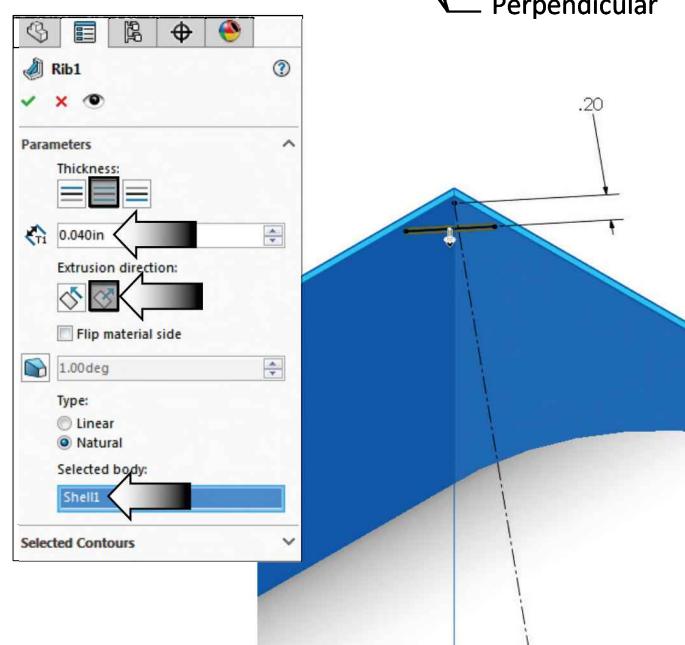
Set the thickness to **Mid Plane**.

Set the wall to: **.040in**.

Click the **Normal To Sketch** button (arrow).

Under the Selected Body section, click the **Shell1** body (arrow).

Click **OK**.



7. Patterning the Rib:

Click **Linear Pattern**.

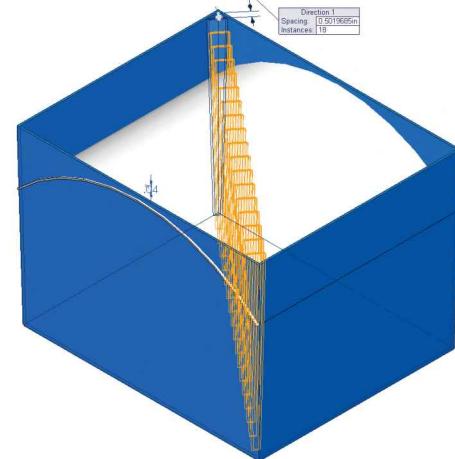
Click the **front face** of the rib to see its dimensions.

For Pattern Direction: select the **.200in** dimension.

For Spacing, enter: **.502in**.



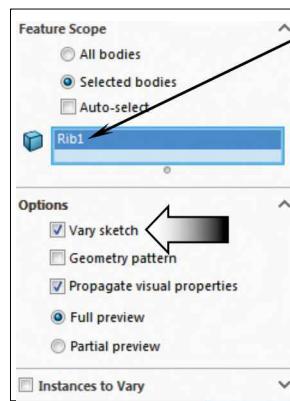
Select this dimension for direction



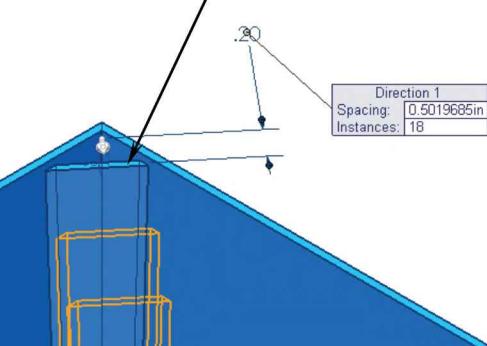
For Number of Instances, enter: **18**.

For Features to Pattern, select the **Rib1** either from the graphics area or from the tree.

Expand the Feature-Scope section, clear the Auto-Select check box, and select the face of the Rib as noted.



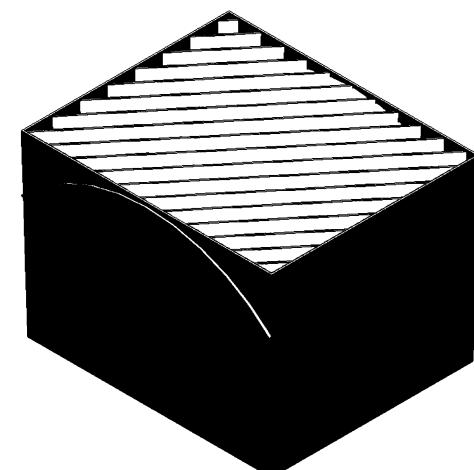
Click the face of the Rib1



Enable the **Vary Sketch** checkbox.

Click **OK**.

The rib feature is repeated 18 times along the direction that was specified by the **.200in** dimension.



This is the 1st set of the ribs. We will repeat steps 6 and 7 again to create similar ribs on the opposite side.

8. Creating the 2nd Rib:

Select the upper face of the part and open a **new sketch**.

Sketch a **Line** across the walls as shown.

Add a **Centerline** that is coincident to the 2 diagonal corners.

Add the dimension and relations as indicated.

Create another rib using the same settings as the first one.

Click **OK**.

9. Patterning the 2nd set of the ribs:

Click **Linear Pattern**.

Double-click the front face of the rib to see its dimensions.

For Pattern Direction:
click the **.20in.** dimension.

For Spacing, enter: **.502in**.

For Number of Instances, enter: **18**.

For Features to Pattern, select **Rib1**.

Clear the Auto-Select checkbox in the Feature-Scope section and select the face of the Rib. Also enable **Vary Sketch**.

Click **OK**. The rib feature is repeated 18 times.

10. Using Combine Common:

Select **Insert / Features / Combine**.

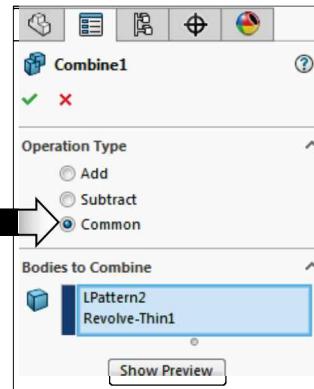
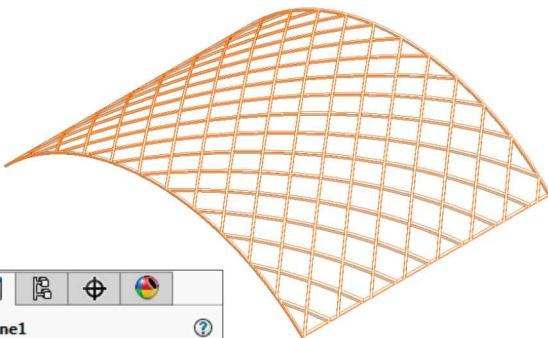
Click the **Common** option.

Select **all bodies** either from the graphics area or from the Feature-Manager tree.

Click the **Show Preview** button.

Click **OK**.

The Combine-Common removes all material except that which overlaps.



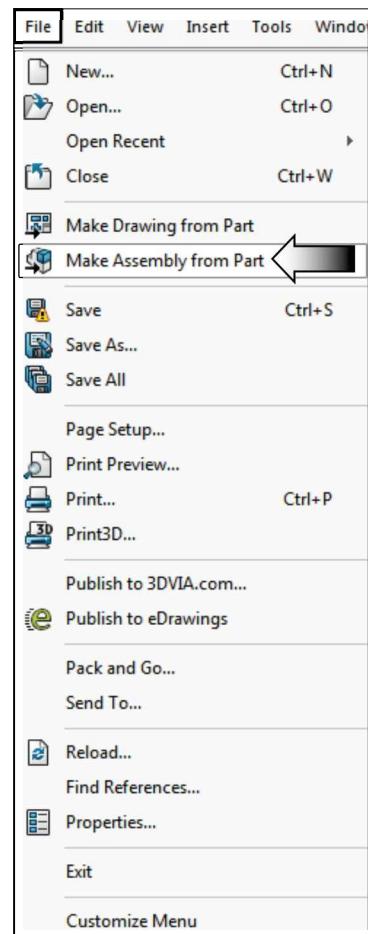
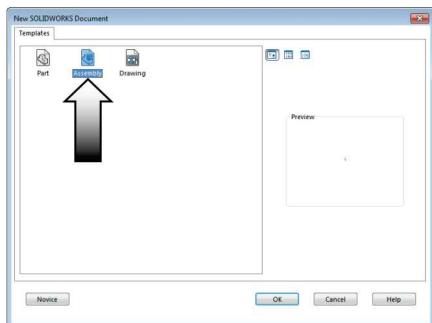
11. Saving the part:

Save the model as **Screen Mesh.sldprt**

12. Making an assembly from the part:

The first one-quarter of the part is completed. We are going to place it in an assembly document and create 3 more instances to make the complete part.

Click **File / Make Assembly From Part**.



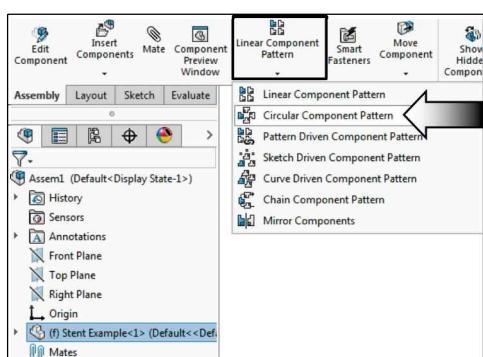
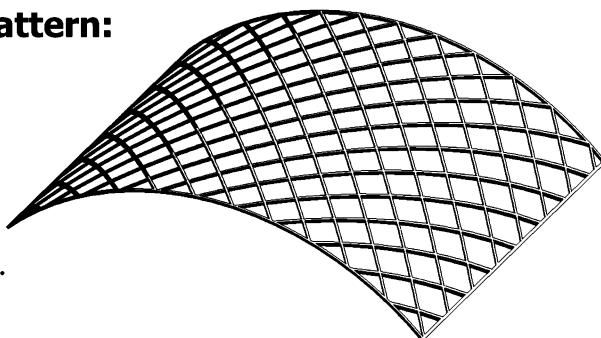
Select the **Assembly Template** and click **OK**.

Click the **Green check** to place the component on the **Origin**.

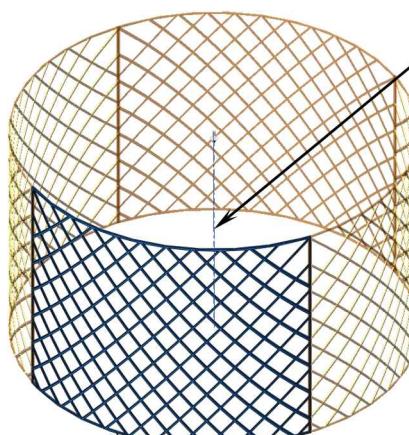
13. Creating a Circular Component Pattern:

Enable the Temporary Axis under the View / Hide/Show pull-down menus.

On the Assembly tool tab, select: **Circular Component Pattern** (arrow).



Part's Origin is coincident with Assembly's origin



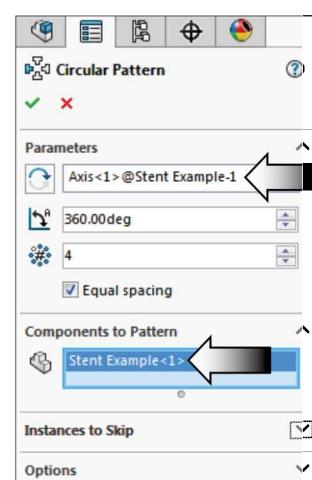
For Pattern Axis, select the **Temporary Axis**.

Enable the **Equal Spacing** checkbox.

Enter **4** for number of instances.

For Components to Pattern, select the **Screen Mesh** from the graphics area.

Click **OK**.



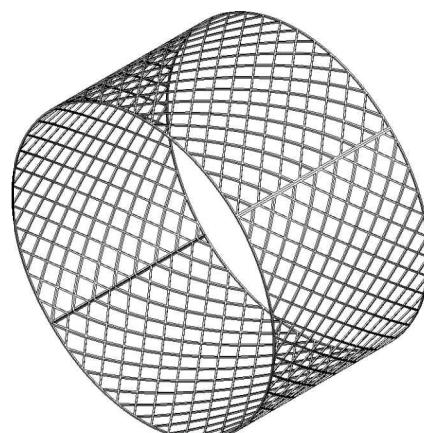
Component to pattern

14. Saving the assembly:

Click **File / Save As**.

Enter **Stent Sample Assembly.sldasm** for the name of the file.

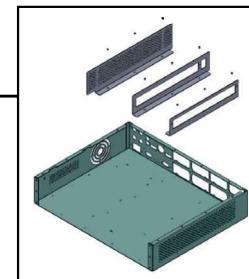
Click **Save**.



CHAPTER 16

Working with Sheet Metal STEP Files

Working with Sheet Metal STEP Files



STEP file extension is short for: STandard for the Exchange of Product data.

The STEP translator supports import and export of body, face, and curve colors from STEP AP214 files.

The STEP AP203 standard does not have any color implementation.

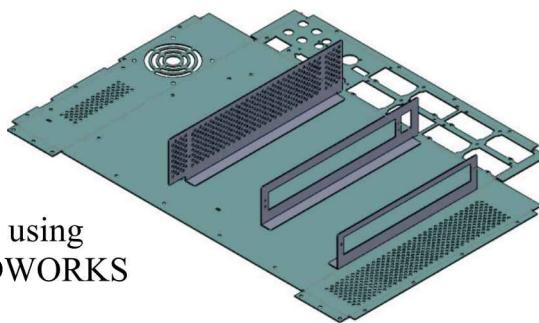
The STEP translator imports STEP files as SOLIDWORKS part or assembly documents.

The STEP translator exports SOLIDWORKS part or assembly documents to STEP files. You can select to export individual parts or subassemblies from an assembly tree, limiting export to only those parts or subassemblies.

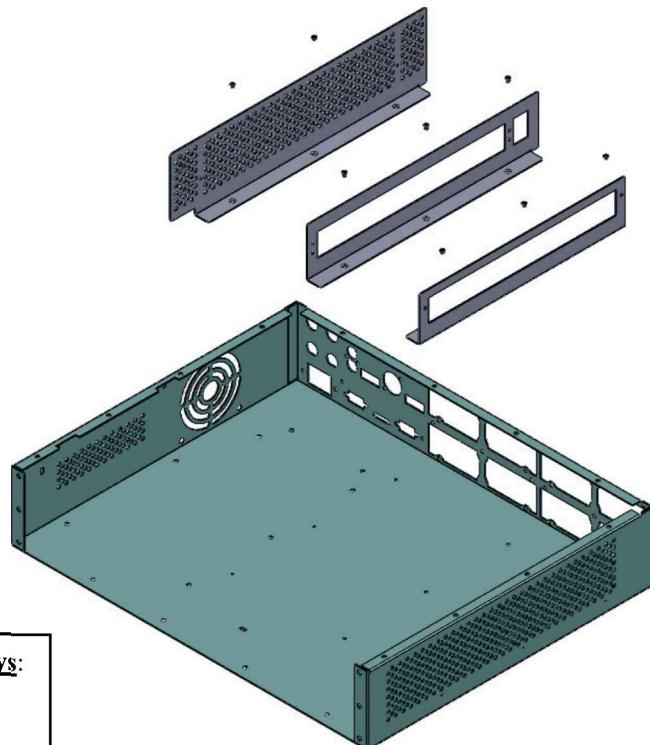
If you select a subassembly, all of its components are automatically selected. If you select a component, its descendants are partially selected, preserving the assembly structure.

This lesson discusses one of the methods to convert an Assembly STEP file to SOLIDWORKS Sheet Metal parts.

After the components are converted, some of the Assembly Features such as the Hole Series and Hole Wizards are used to add the new holes in the assembly mode, then the Fasteners are inserted automatically using the Smart Fasteners feature (requires SOLIDWORKS Toolbox).



Working with Sheet Metal STEP Files



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI

Units: INCHES – 3 Decimals

Tools Needed:



Dimension



Insert Bends



Flat Pattern



Hole Series

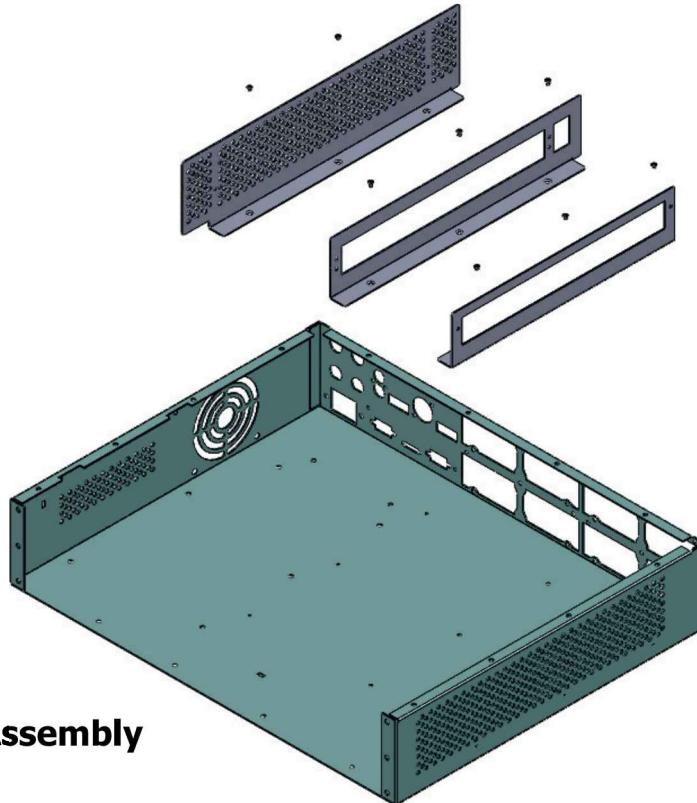


Hole Wizard



Smart Fasteners

Sheet Metal **STEP** Files and Smart Fasteners

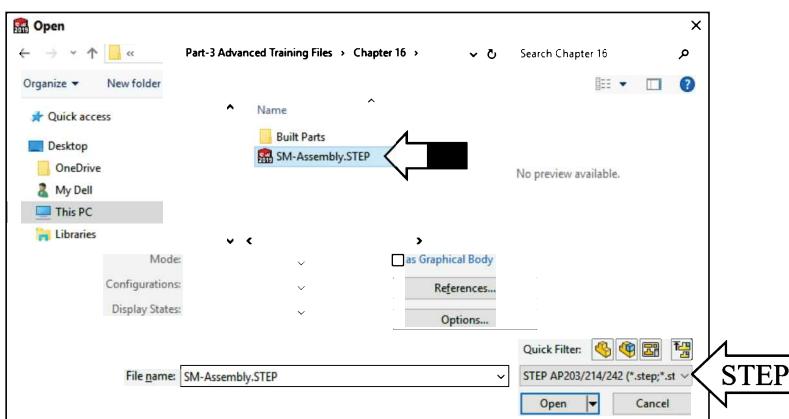


1. Opening an Assembly Step File:

Select to **File / Open**.

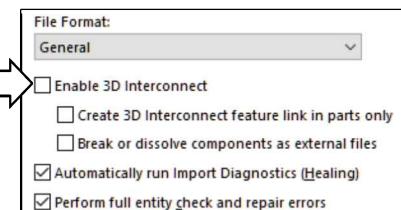
Browse to the Training Files folder, change the Files of Type to **STEP** and open a STEP document named: **SM-Assembly.step**.

The part files from the STEP document will appear as SOLIDWORKS documents on the FeatureManager tree without any model history.



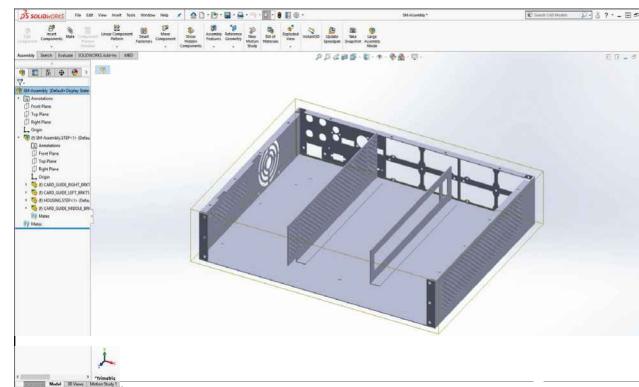
NOTE: The **3D-Interconnect** option should be disabled if you have trouble opening the STEP document.

Go to: Tools, Options, System Options, Import to disable it.

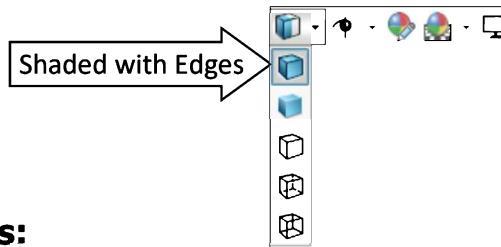


There are 4 components in this assembly and they have not yet been constrained.

The Housing will be used as the Fixed Component and the 3 Card Guides will be left un-constrained for the purpose of this exercise.

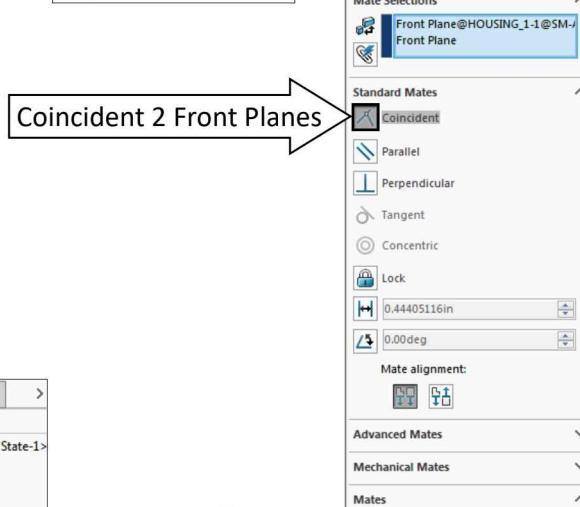


Change the Shading option to:
Shaded with Edges
(arrow).

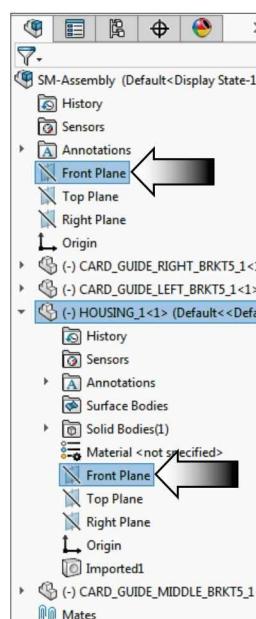


2. Mating the components:

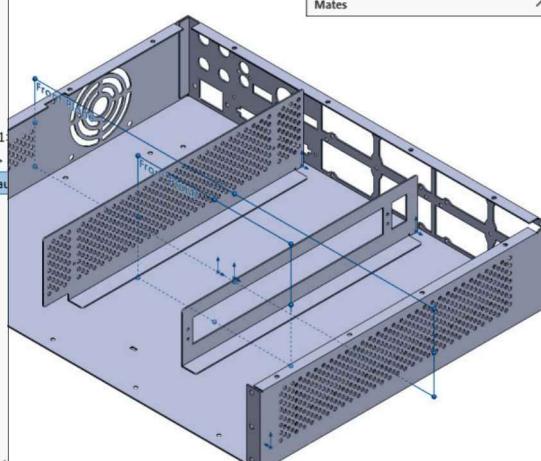
In order to mate the Housing component to the assembly's Origin, we will need to align the Front, Top, and Right planes.



Click the **Mate** command from the Assembly toolbar.



Select the **Front** plane of the Housing and the **Front** plane of the Assembly.

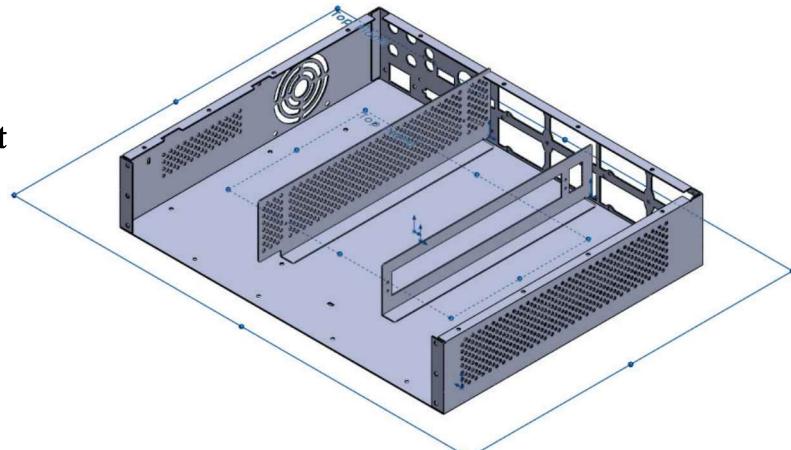


Select **Coincident** from the list.

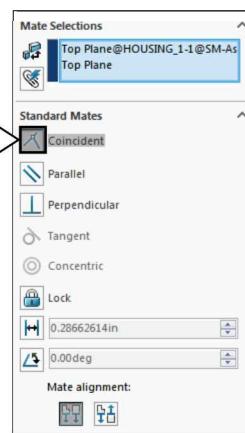
Click **OK**.

(More on next page...)

Add the same **Coincident** mate to the **Top** plane of the Housing and the **Top** plane of the Assembly.

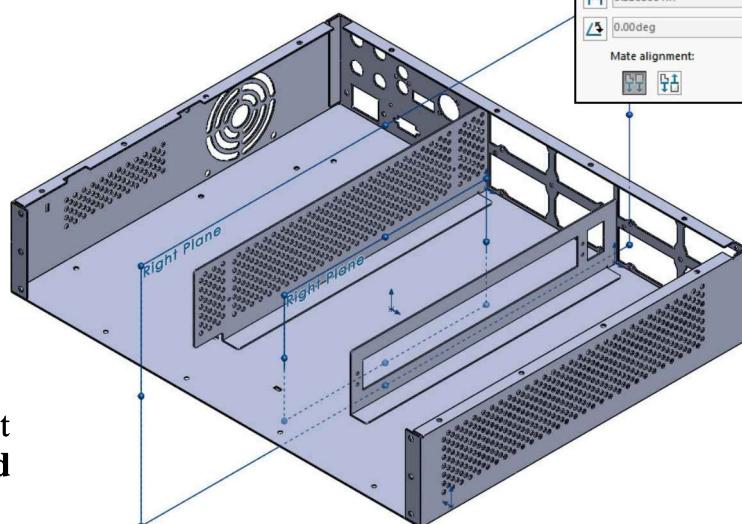
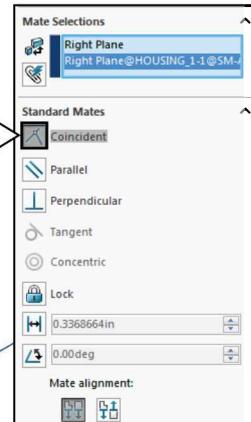


Coincident 2 Top Planes



Repeat the **Coincident** mate to the **Right** plane of the Housing and the **Right** plane of the Assembly.

Coincident 2 Right Planes

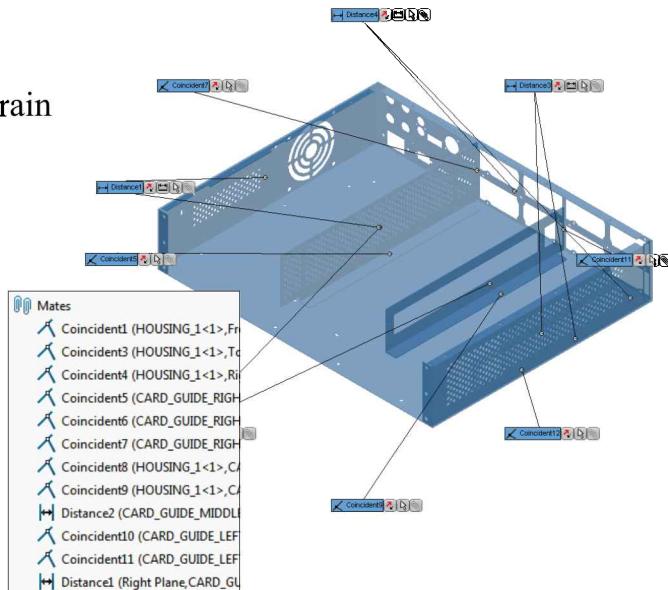


The Housing component should be **Fully Defined** at this point.

3. Adding other Mates:

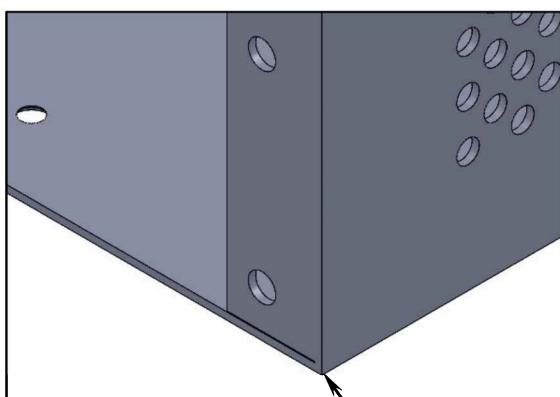
Other mates can be added to constrain the other components, but later on they will need to be suppressed so that the final sheet metal components can be flattened properly.

In this exercise we will leave the components un-constrained to help focus in other areas.

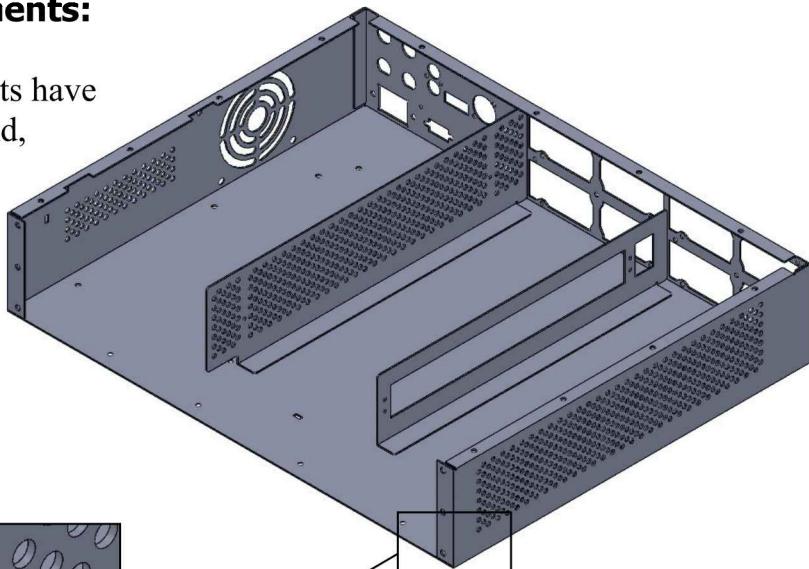


4. Examining the components:

The imported components have **Sharp corners** all around, which is not suitable for the Sheet Metal processes.



Sharp Corner(s)*



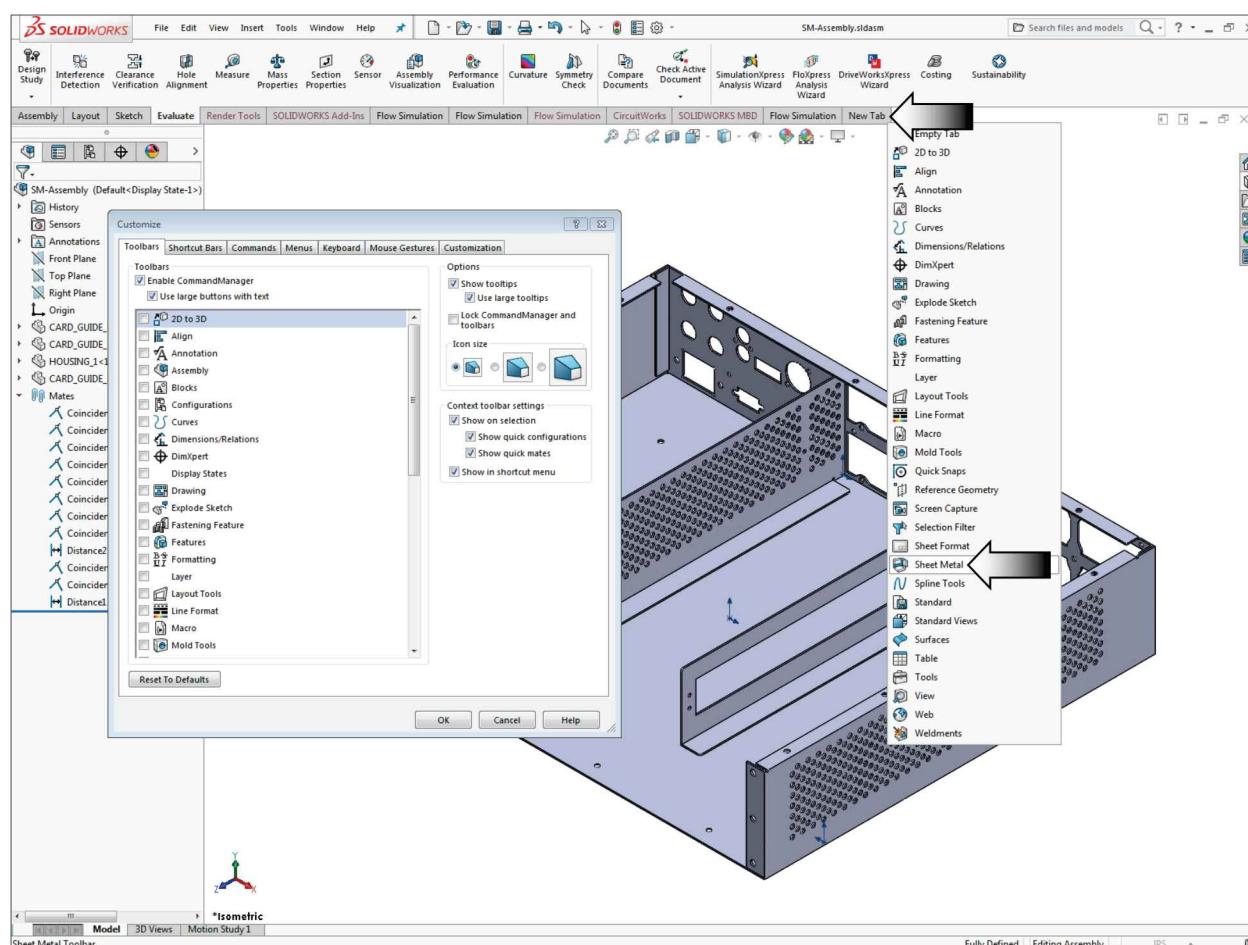
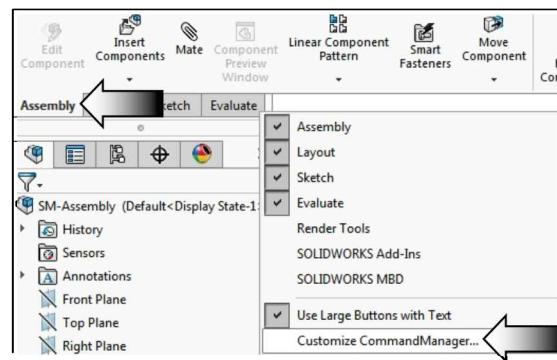
Sheet Metal parameters* must be added to fully convert the imported part into a SOLIDWORKS Sheet Metal part.

5. Adding the Sheet Metal tool tab:

If the Sheet Metal tool tab is not yet enabled on the CommandManager, do the following to add it:

Right-click the Assembly tab and select the **Customize CommandManager** option (arrow).

Click the **New Tab** and pick: **Sheet Metal** (arrow) and click **OK**.



Switch to the **Sheet Metal** tab.



6. Inserting Sheet Metal parameters*:

Select the component **Housing** and click the **Edit Component** command .

From the **Sheet Metal** tab
click the **Insert Bends**  command.

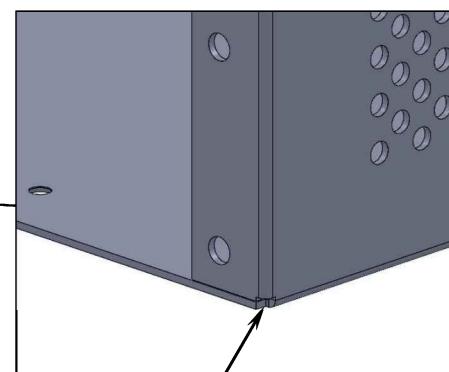
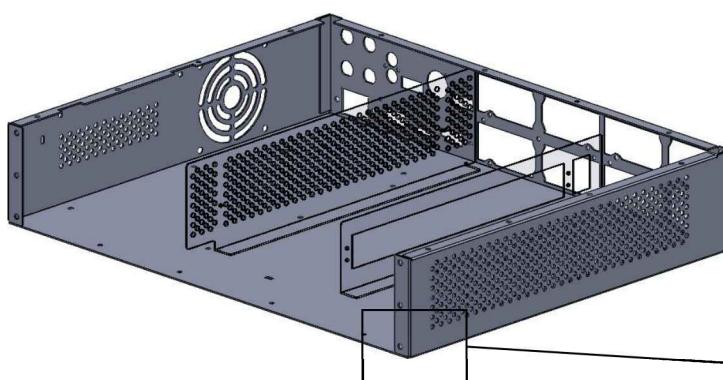
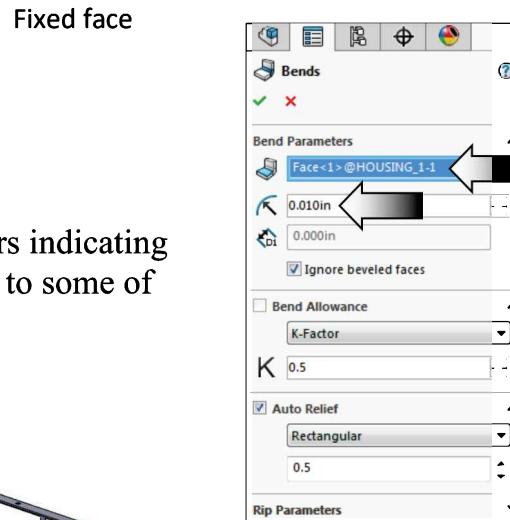
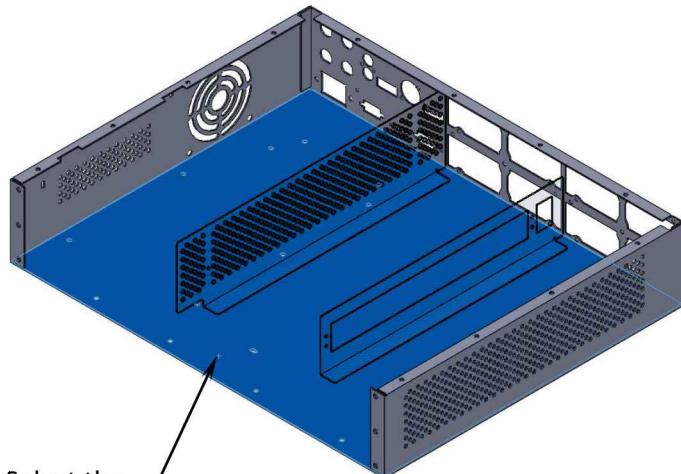
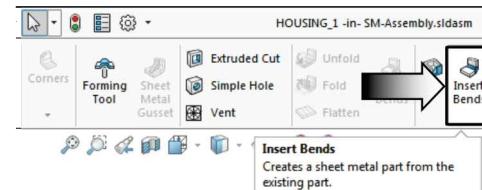
Select the **Fixed face** as noted.

Enter **.010in** for Bend Radius.

Use the **default settings** for
Bend Allowance and **Auto-
Relief**.

Click **OK**.

Click OK when a message appears indicating the Auto-Relief-Cuts were added to some of the corners of the part.



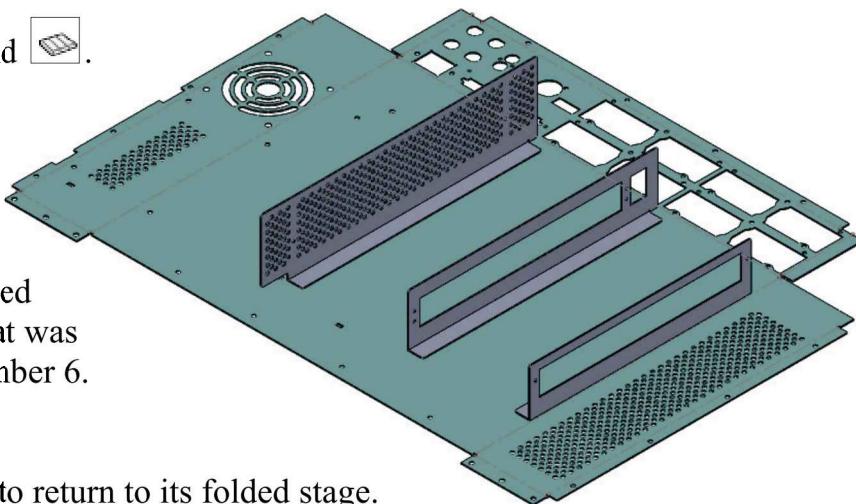
Zoom closer to inspect the relief corners.

7. Viewing the Flat Pattern:

From the **Sheet Metal** tab, click

the **Flatten** command .

The Housing is flattened. The orientation of the flattened view is based on the Fixed face that was specified in step number 6.



Click **Flatten** again to return to its folded stage.

Click off the **Edit Component**  command.

8. Converting the 2nd component:

Select the **Card Guide Left** as shown and click the

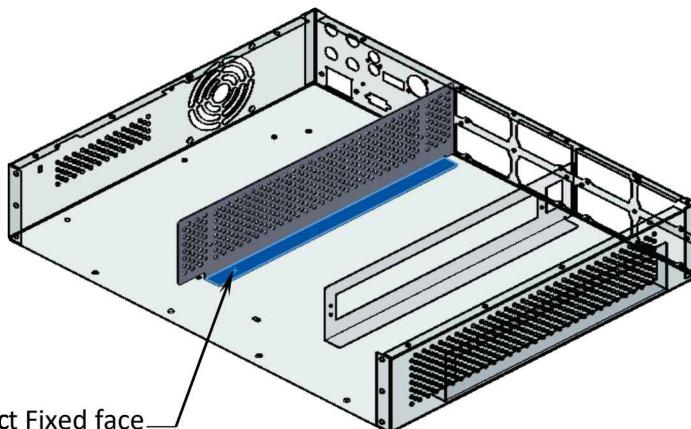
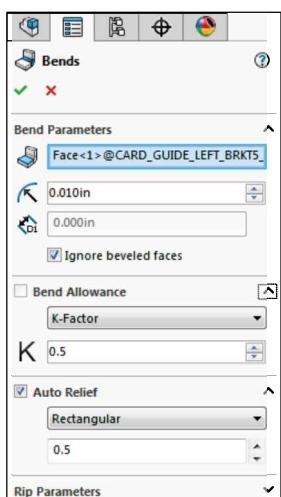
Edit Component  command.

From the Sheet Metal tab, click **Insert Bends** .

Select the **Fixed face** as noted.

Enter **.010"** for Bend Radius. Keep all other default parameters.

Click **OK**.



Click off the **Edit Component**  command.

Select Fixed face

9. Converting the 3rd component:

Select the Card Guide Middle and click the **Edit Component** .

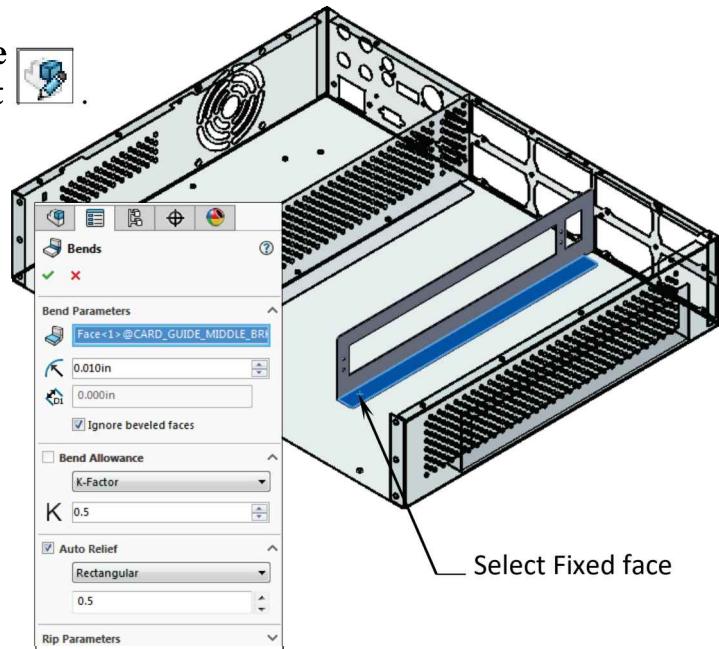
From the **Sheet Metal** tool

tab click **Insert Bends** .

Select the **Fixed face** as noted.

Enter **.010"** for Bend Radius and use the **default settings** for the Bend Allowance and K-Factor.

Click **OK**.

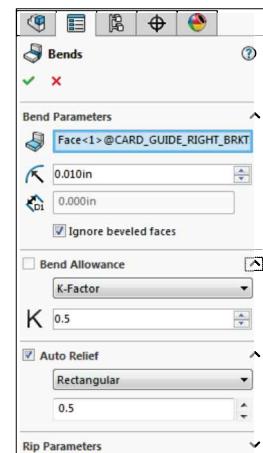


Click off the **Edit Component**  command.

10. Converting the 4th component:

Select the Card Guide Right and click **Edit Component** .

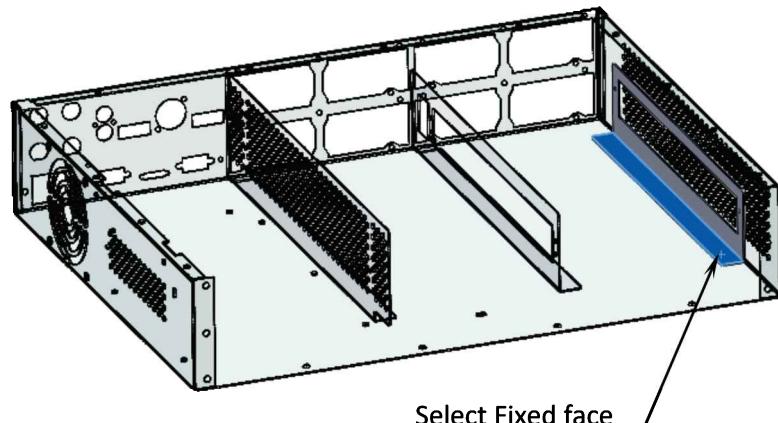
From the Sheet Metal tab, click: **Insert Bends** .



Select the **Fixed face** as noted.

Enter **.010"** for Bend Radius. Keep all other default parameters the same.

Click **OK**.



Click off the **Edit Component**  command.

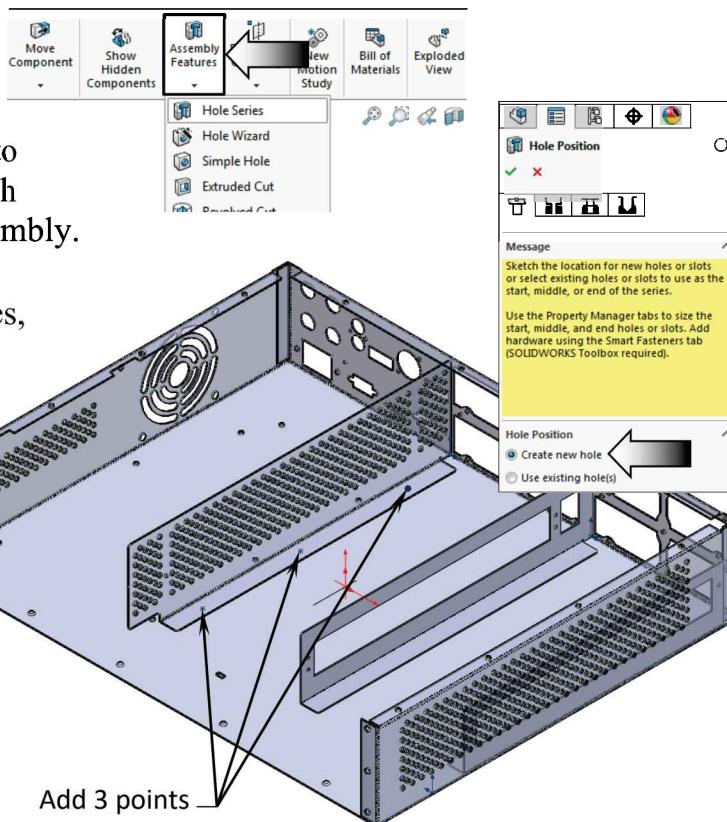
11. Using the Hole-Series:

The Hole Series is an Assembly-Feature; it is used to create a series of holes through the individual parts of an assembly.

Unlike other assembly features, the holes are contained in the individual parts as externally referenced features. If you edit a hole-series within an assembly, the individual parts are modified.

From the Assembly tool tab, select **Assembly Features /**

Hole Series .



From the FeatureManager, click: **Create New Hole** (arrow).

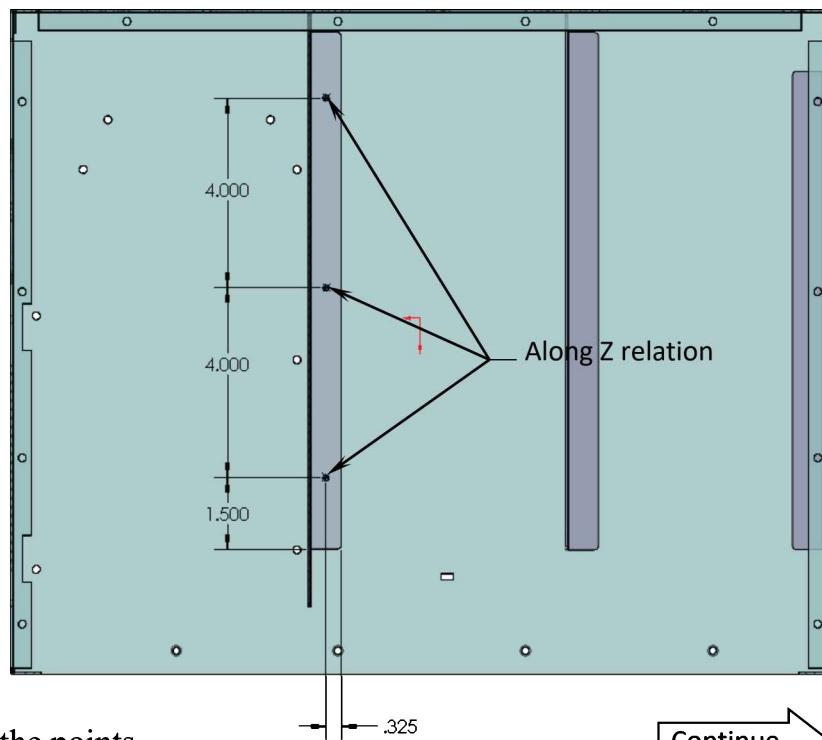
The mouse cursor changes to the  **Sketch Point** command.

Add 3 Sketch Points approximately as shown.

Each point is the center of a hole.

Add an **ALONG Z** relation (vertical) between the 3 points.

Add the dimensions as indicated to fully define the positions of the points.



Click the **First Part** tab.

Click the **Countersink** option.

Select the following:

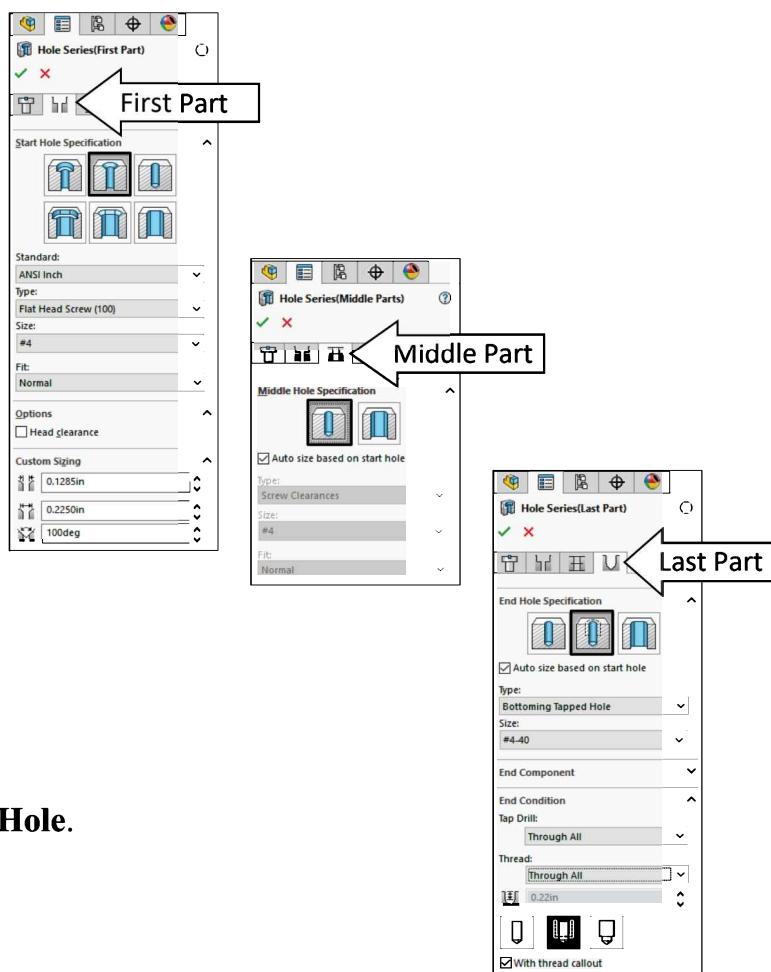
Standard: Ansi Inch

Type: Flat Head Screw

Size: #4

Fit: Normal

Use the default settings for Custom Sizing.



Click the **Middle Part** tab.

Select the **Hole** button.

Enable the check box:

Auto Size based on Start Hole.

Click the **Last Part** tab.

Select the **Straight Tap** button.

Enable **Auto Size Based On Start Hole**.

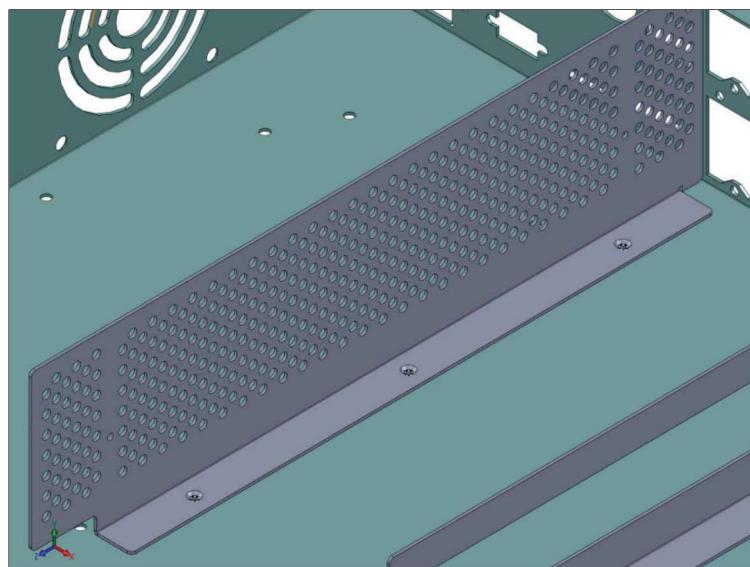
Set Type to **Tapped Hole**.

Set Size to **#4-40**.

Set both End Conditions to **Through All**.

Enable the checkbox: **With Thread Callout**.

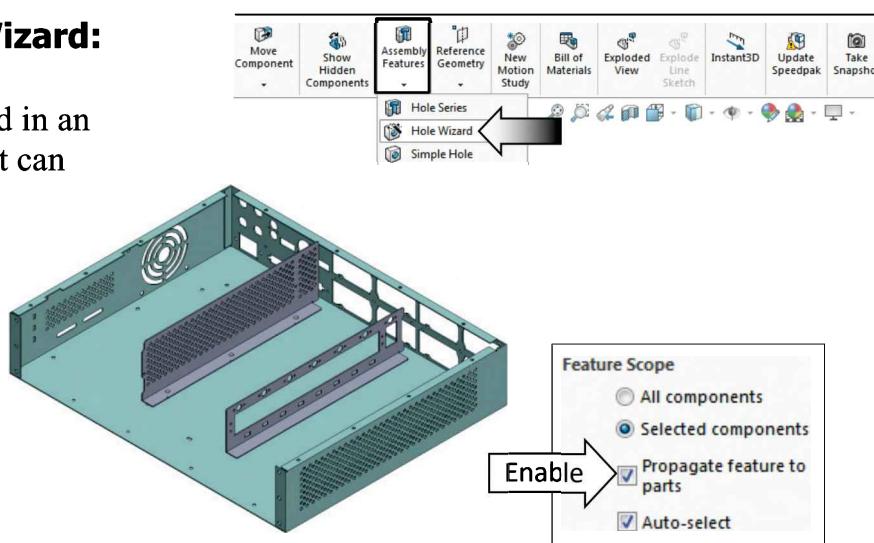
Click **OK**.



12. Using the Hole Wizard:

Hole wizard created in an assembly document can be populated to the part for future editing.

This feature can be toggled on/off under the Feature-Scope section.



From the Assembly tool tab, click Assembly Features / Hole Wizard.



Select the **Countersink** option under the Type tab.

Set the options to match the last 3 holes in step number 11.

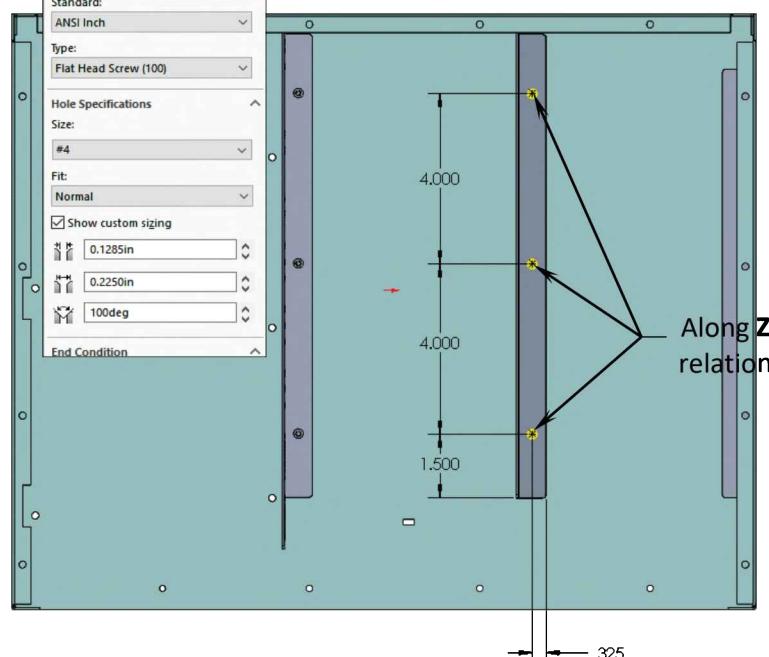
Switch to the Positions tab (arrow).

Click approximately as shown to create 3 points.

Each point represents the center of that hole.

Add the relation and dimensions as indicated.

Click OK.

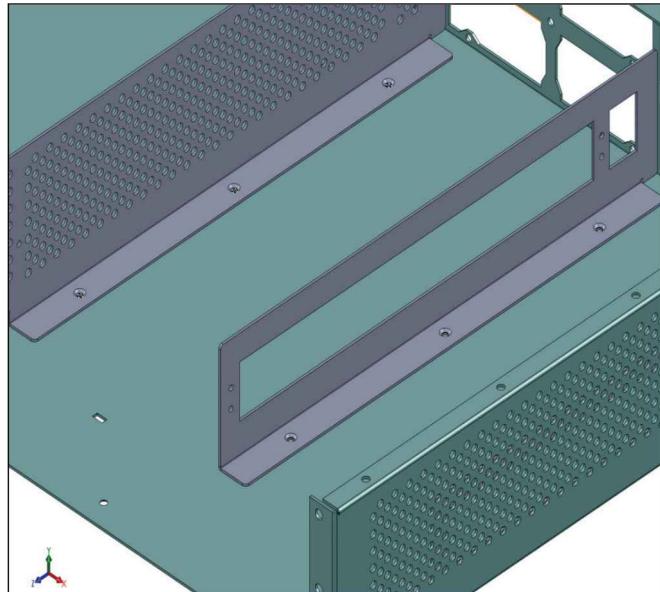


13. Verifying the two hole types:

Although the 6 holes were created with 2 different hole options, they are exactly identical.

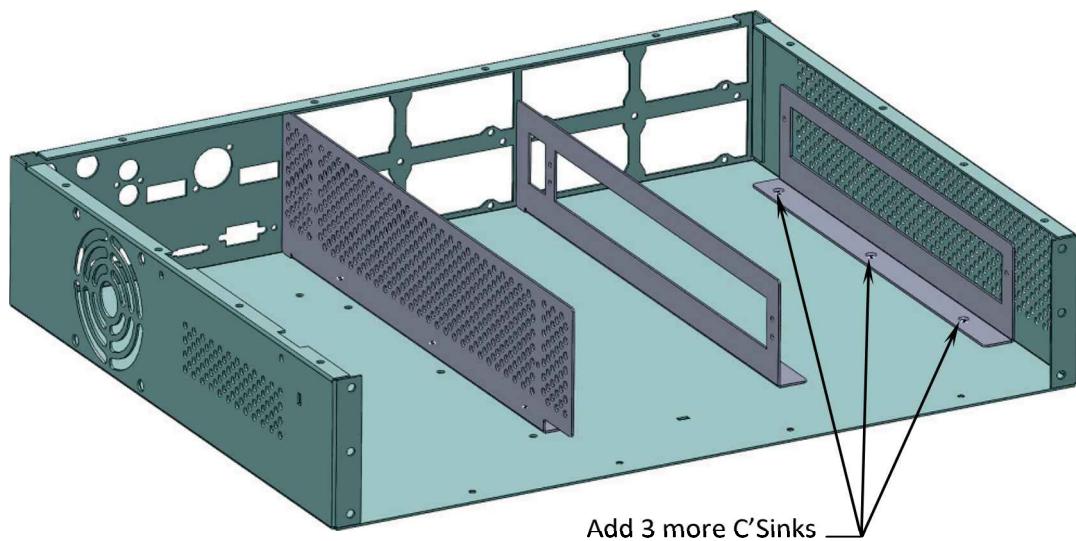
Open the Card Guide Middle to verify that the holes are actually there on the part.

The unique option in step 12 (propagate feature to parts) allows these Assembly Features to appear in the part mode as well.



14. Adding holes on the Card Guide Right:

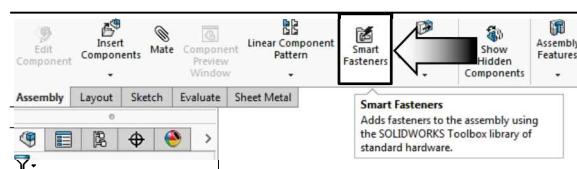
Repeat either step number 11 (Hole Series) or step number 12 (Hole Wizard) and create 3 more holes for the last Card Guide.



Use the same dimensions from the previous step to position the holes.

15. Adding the Smart Fasteners:

Click **Smart Fasteners** from the **Assembly** tab.

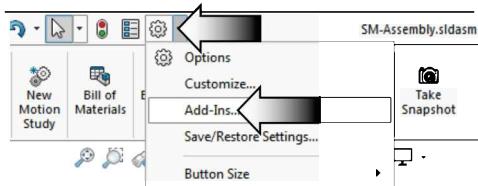


An error message appears inquiring for SOLIDWORKS Toolbox to be activated. (Requires SOLIDWORKS Professional or SOLIDWORKS Premium.)



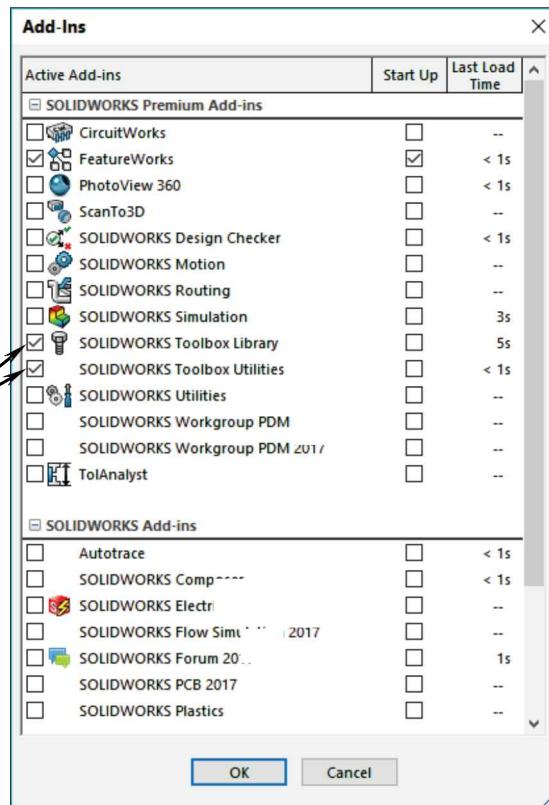
To activate Toolbox, select the following:

Tools / Options / Add Ins.



Enable Toolbox Library & Toolbox Utilities

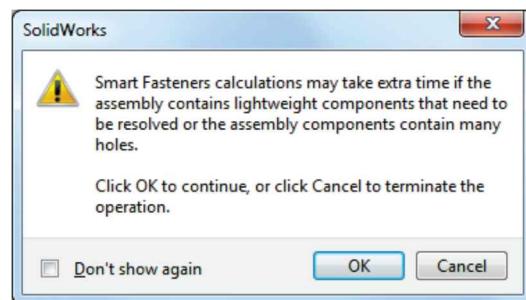
Enable SOLIDWORKS Toolbox Library and SOLIDWORKS Toolbox Utilities.



Click **OK**.

Click the **Smart Fasteners** button once again.

Another message pops up indicating that the Smart Fasteners calculation may take extra time; click **OK**.

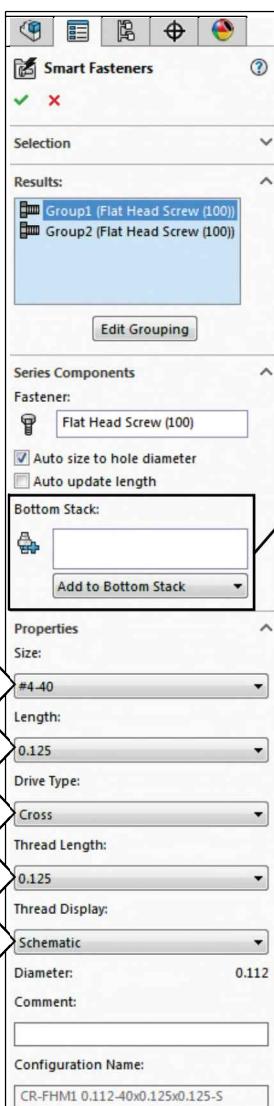
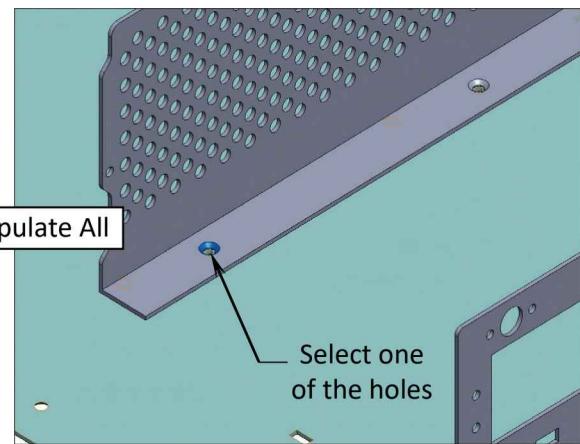
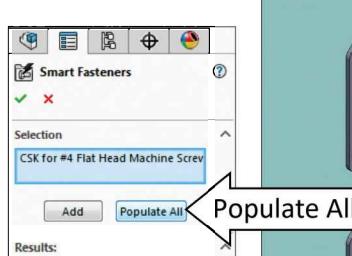


Select one of the **Countersink** holes in the graphics area.

Click Populate All (arrow).

The system searches for the best matched screws from its Toolbox library and automatically inserts them into each hole.

Set the following properties:



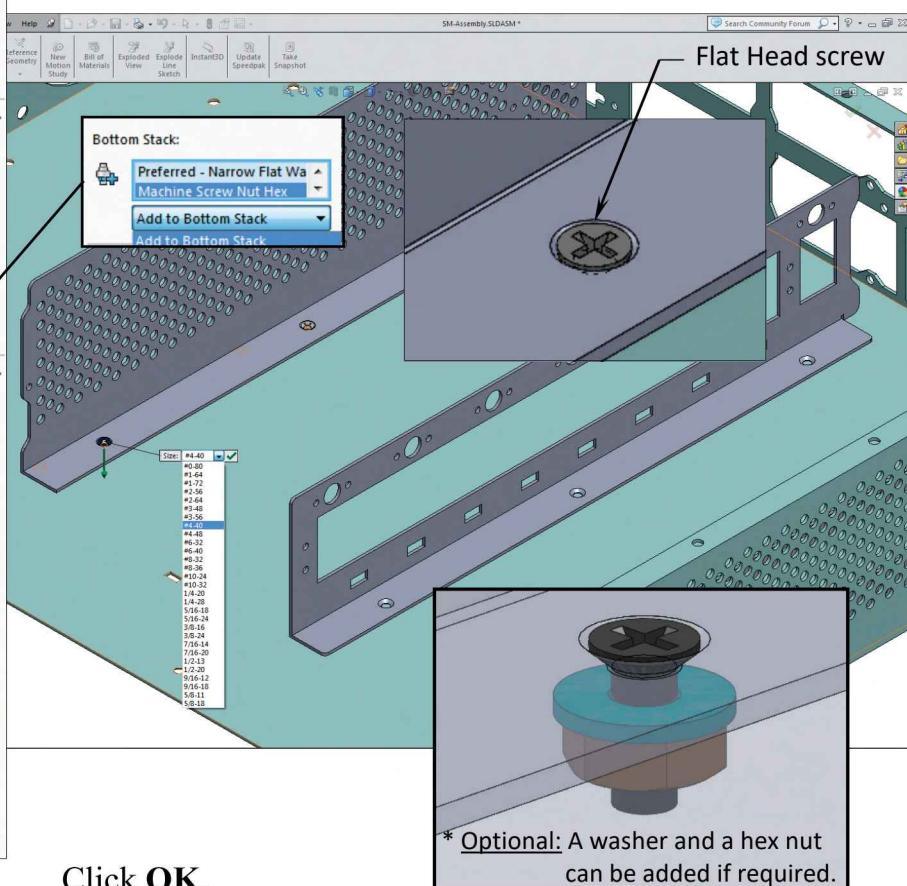
Size: #4-40

Length: .125in*

Drive Type: Cross

Thread Display: Schematic

* Change the screw length to .250in if adding a washer and a nut.



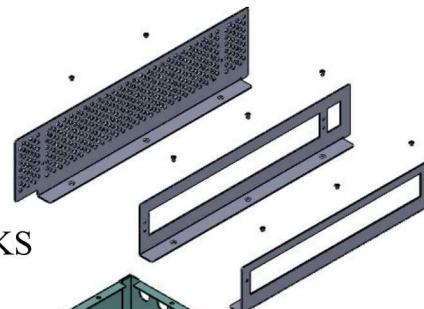
Click OK.

16. Creating an Exploded View:

Option 1

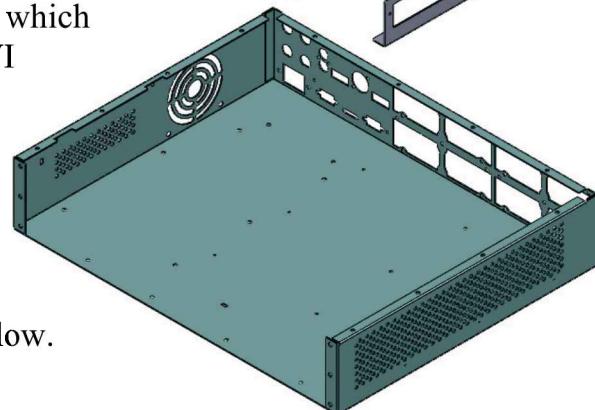
Create an exploded view with all 4 components shown in folded stage as shown.

When an exploded view is created, SOLIDWORKS also creates an animated configuration, which can be played back and saved as an AVI file format.



Option 2

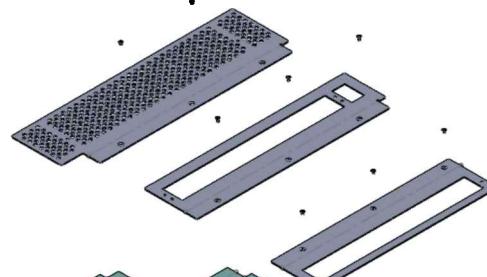
Create a 2nd exploded view with all 4 parts as shown in the Flatten view below.



NOTES:

Edit each component in order to switch from the Folded to flatten stage.

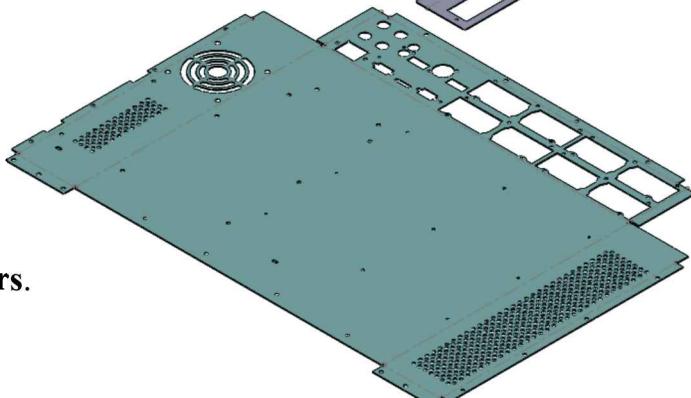
Configurations can also be used to capture the flat pattern of each component.



17. Saving your work:

Click File / Save As.

For the name of the file enter:
SM_Assembly_Smart Fasteners.



Close all documents.

Adding Parts to the Toolbox Library

Customized parts or fasteners can be added to existing Toolbox folders.

For parts that are stored in the shared library, administrators can control who can add or change parts by creating a Toolbox password and setting permissions for Toolbox functions for each user (see Toolbox Permission in the SOLIDWORKS Help section).

1. Starting the Toolbox Settings Utility:

From the Windows desktop select the following:
Start / All Programs / SOLIDWORKS 2024 / SOLIDWORKS Tools 2024 / Toolbox Settings (arrow).

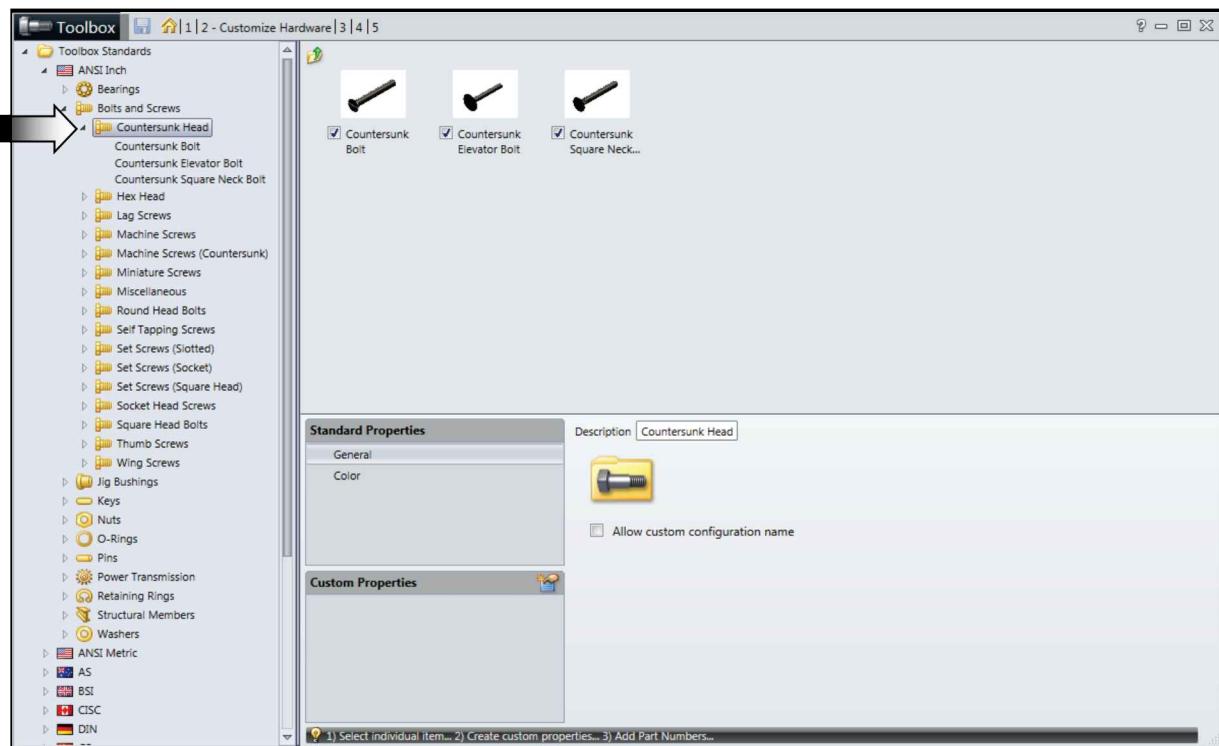
Select the option number 2 (arrow):
Customize your hardware.



2. Adding a part to a folder:

From the upper left side of the tree, select the following:

ANSI Inch.
Bolts and Screws
Countersunk Head

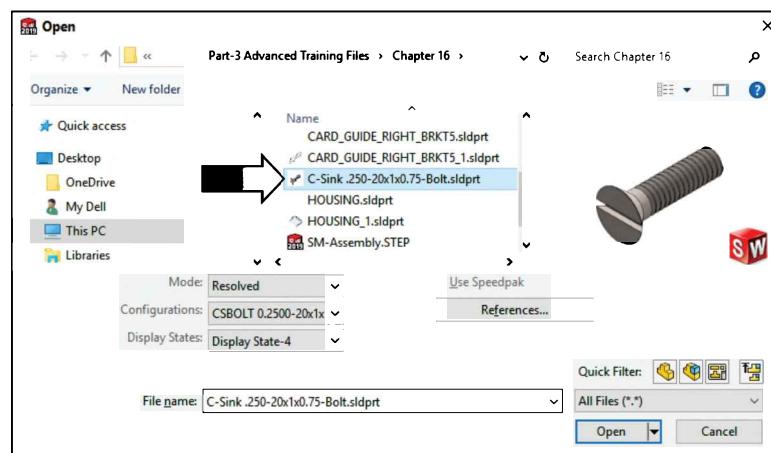


In this folder there are 3 existing screw/bolt types: **Countersunk Bolt**, **Countersunk Elevator Bolt**, and **Countersunk Square Neck Bolt**.

Right-click the **Countersunk Head** folder and select **Add File** (arrow).

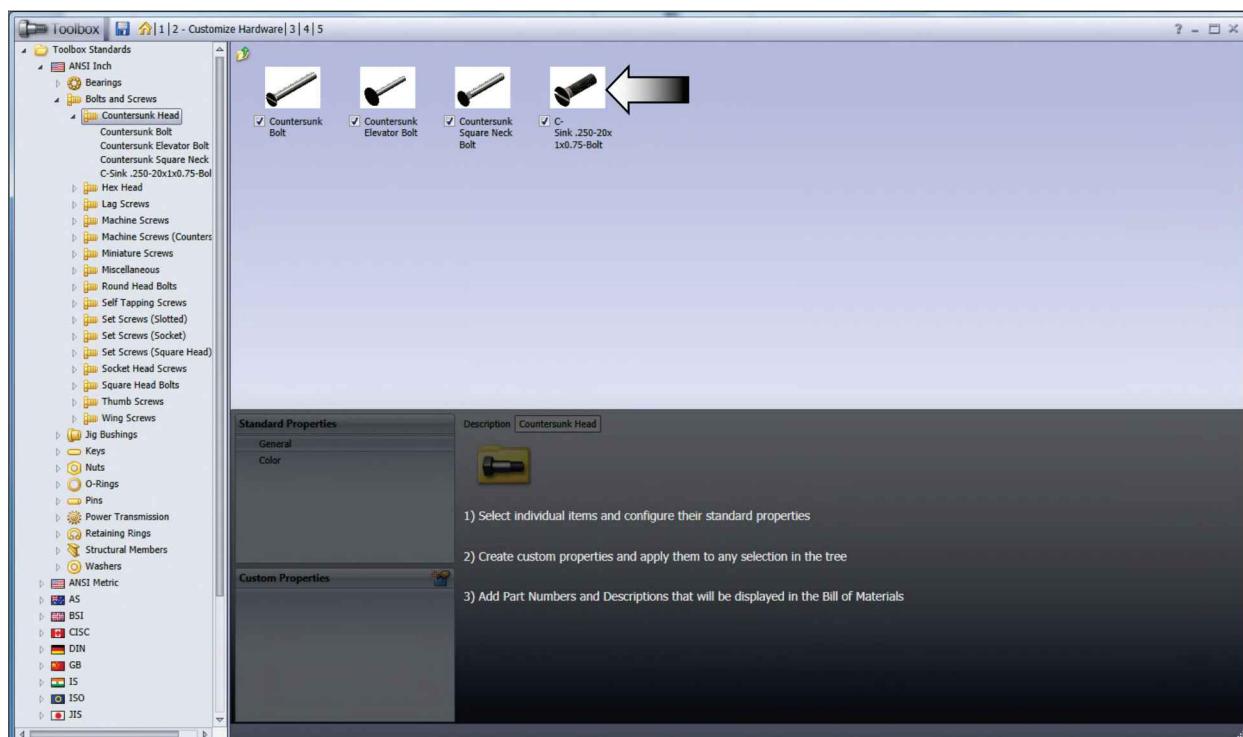
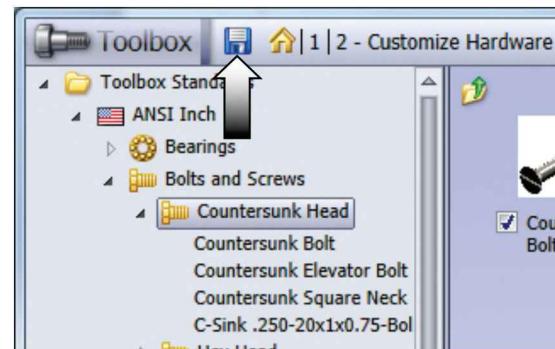


Browse to the Training Files folder and select the document named: C-Sink .250-20x1x0.75-Bolt.sldprt and open it.



Click the Save icon (arrow) on the top left of the dialog box to save the newly added part.

The new part and its name appear in the display window (arrow).



3. Activating Toolbox:

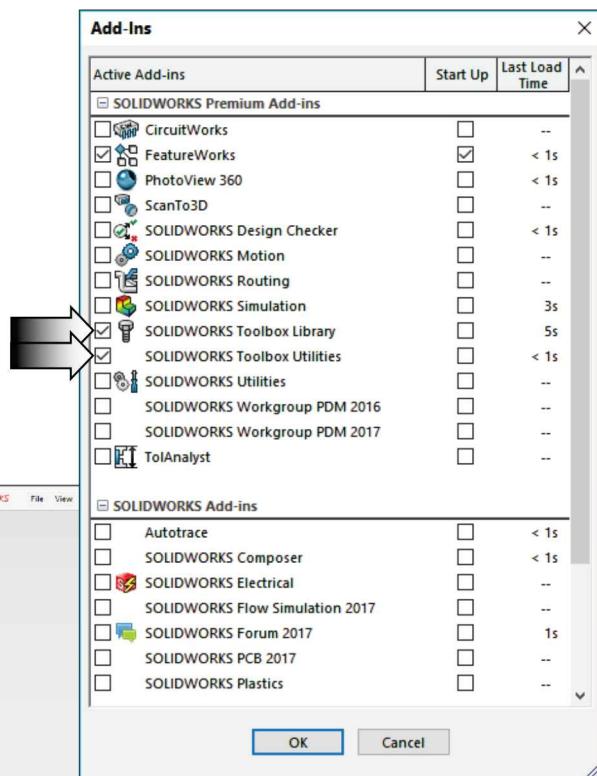
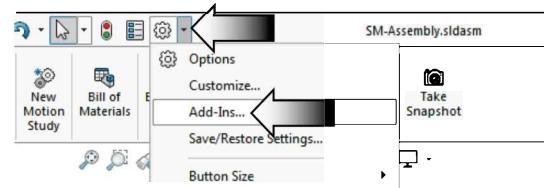
Launch the SOLIDWORKS application.

Select: Options / Add-Ins (arrow).

Enable the 2 options:

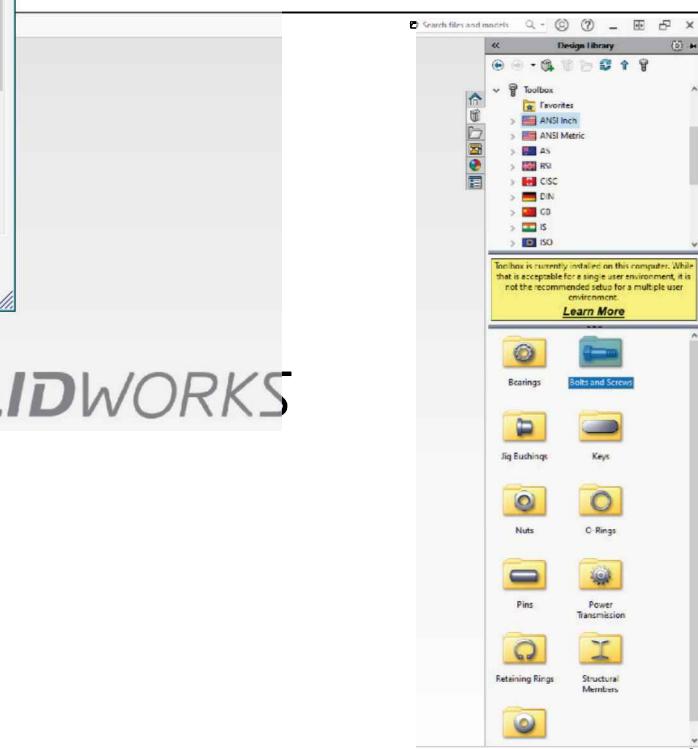
- * SOLIDWORKS Toolbox Library
- * SOLIDWORKS Toolbox Utilities

Click OK.



From the **Task Pane** (on top right hand corner) click the **Design Library** folder and then expand the **Toolbox** folder (arrow).

NOTE: Toolbox is only available in SOLIDWORKS Professional and SOLIDWORKS Premium.



4. Locating the new part:

Select the following under the Toolbox folder:

- * ANSI Inch
- * Bolts and Screws
- * Countersunk Head

Locate the new part **C-Sink .250-.20x1x0.75-Bolt** on the lower right side of the Task Pane (arrow).

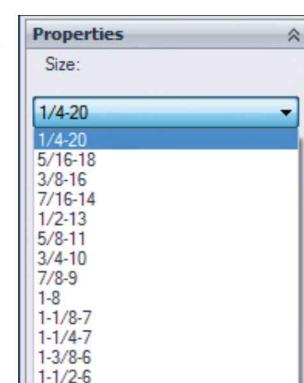
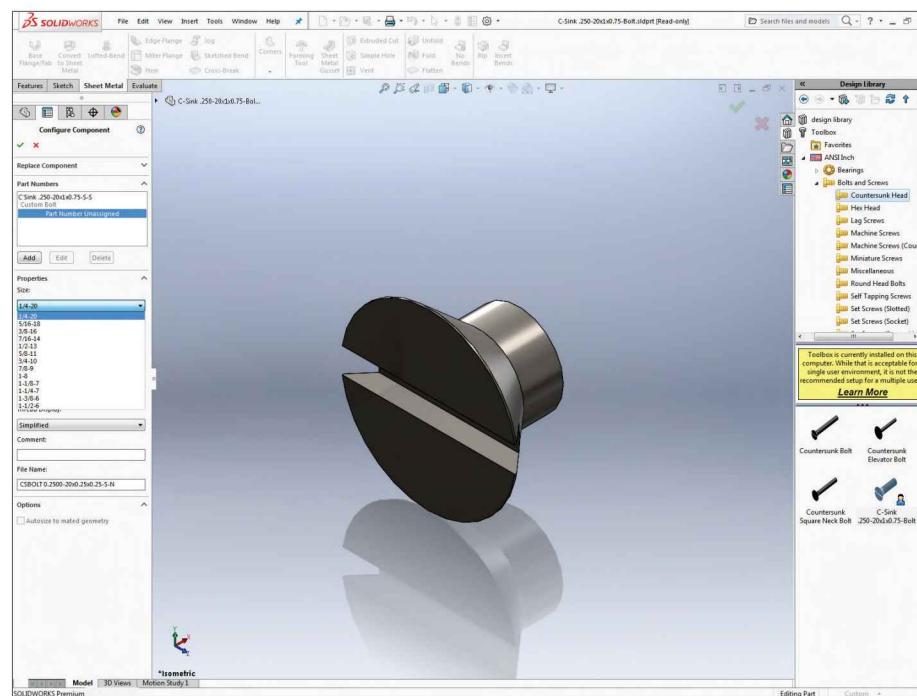
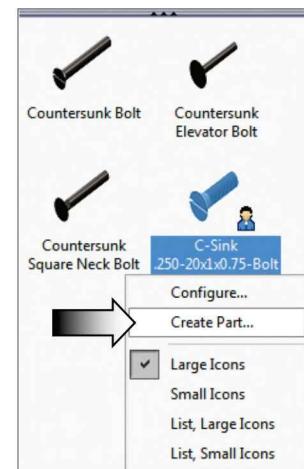
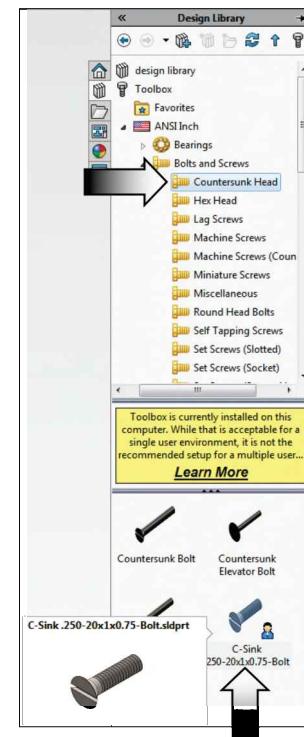
5. Viewing the new part:

Right-click on the new part and select:
Configure Part.

NOTE:

*There is no configuration created for this custom screw.
Its length will have to be modified manually.*

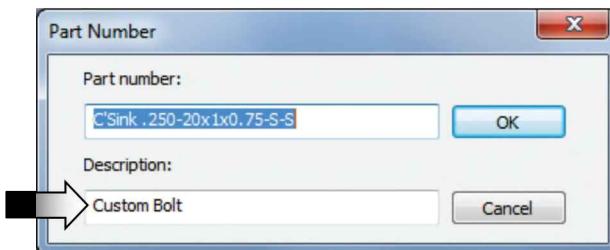
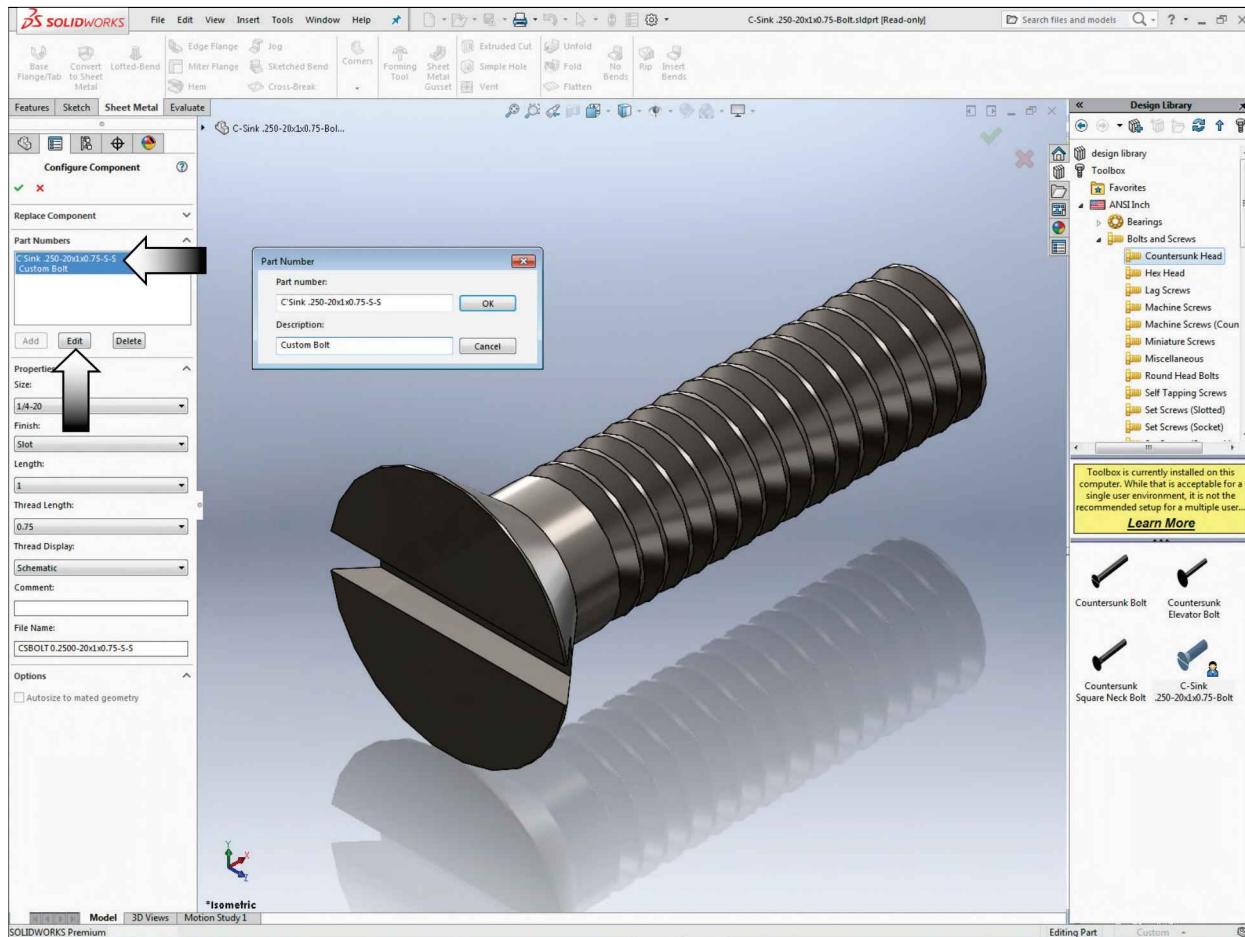
Existing Toolbox parts will have several configurations such as Size, Length, Thread Display, etc., to choose from.



The new Toolbox part appears in its own window.

6. Adding a Part Number and Description:

Click the **Edit** button (arrow).



Enter a new **Part Number** (if applicable).

Enter **Custom Bolt** as Description.

Save and close your documents.

Sheet Metal Assembly

Designing Parts in-context of an Assembly

Top-Down Assembly, also referred to as In-Context Assembly, lets you see the components in their correct locations in the assembly while you are creating the new parts or editing existing features of others. Additionally, you can use geometry of the surrounding components to define the size or shape of the new ones.

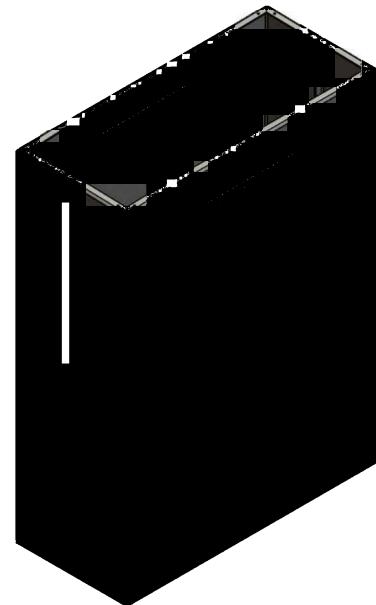
1. Opening an assembly document:

Select **File, Open**.

Browse to the Training Folder and open an assembly document named: **SM Enclosure Assembly.sldasm**

This assembly document contains 3 components, the **SM_Part1**, **SM_Part2**, and **SM_Part3**.

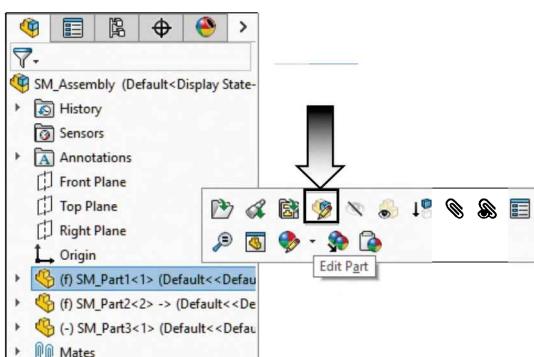
We will need to create the **SM_Part4** (the Lid), using the Top-Down Assembly approach.



2. Editing a component:

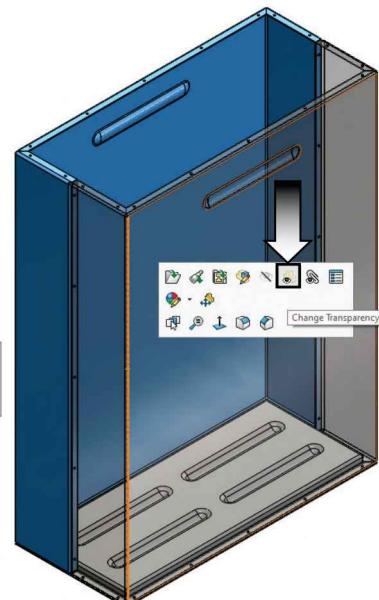
The rib feature in the component **SM_Part1** needs to be replicated a few more times and the vent holes must be added to its side.

Click the name **SM_Part1** from the Feature tree and select:
Edit Part.



Change the **SM_Part2** to Transparent (2nd arrow).

(The blue color means the component is being edited.)



3. Patterning the rib feature:

Switch to the **Features** tab and click **Linear Pattern**.

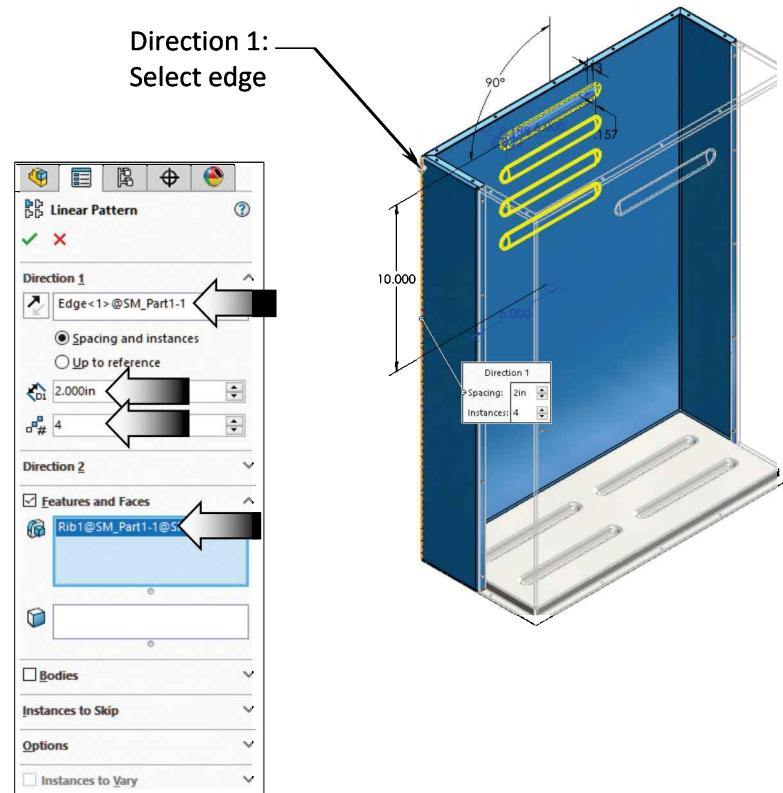
For Direction 1, select the **vertical edge** as indicated.

For Spacing, enter **2.00in**.

For Number of Instances, enter **4**.

For Features to Pattern, select the **Rib1** feature.

Click **OK**.



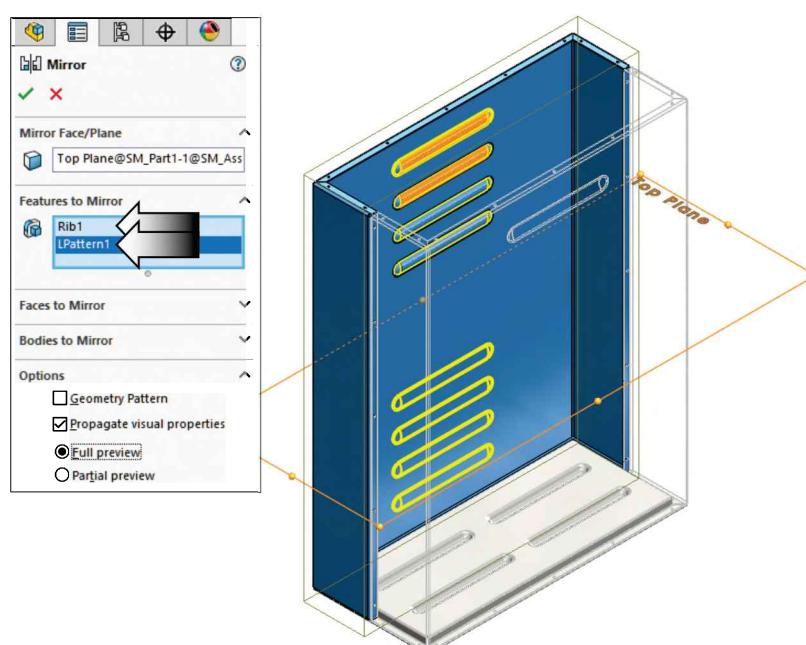
4. Mirroring the ribs:

Click **Mirror**.

For Mirror Plane, select the **Top** plane from the same component.

For Features to Mirror, select **Rib1** and **LPattern1**.

Click **OK**.



5. Adding the vent holes:

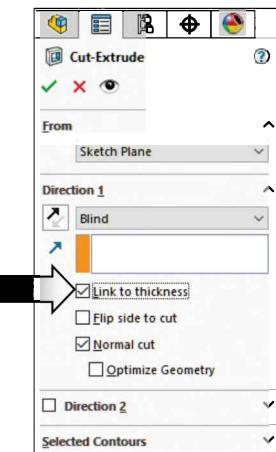
Select **Sketch10** (the sketch of the holes) from the FeatureManager tree.

Click **Extruded Cut**.

For Direction 1, use the default **Blind** type.

Enable the **Link To-Thickness** checkbox.

Enable **Normal Cut**.



Click **OK** and click **Rebuild**.

6. Exiting the Edit Component mode:

Click off the **Edit Component** command from the Assembly tab (or click the part's name and de-select **Edit Part**).

The modifications to the **SM_Part1** are completed. We can now move forward to creating another component, but first, let us turn-off the transparent mode for the **SM_Part2** component.

From the graphics area, click the component **SM_Part2** and de-select the **Change Transparency** button (arrow).



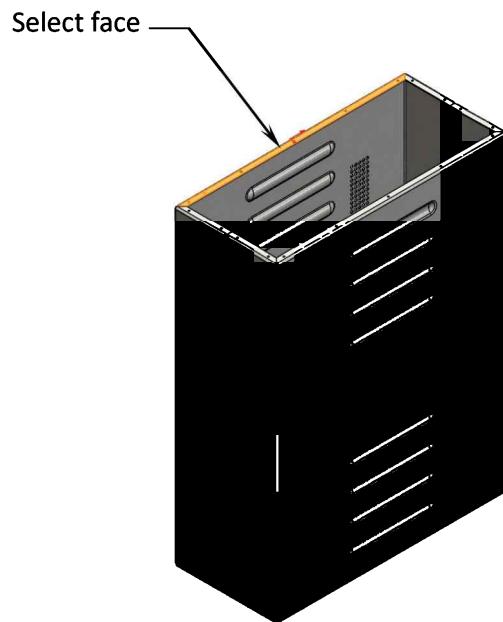
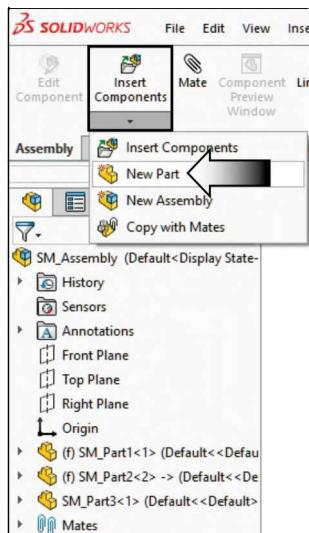
The component **SM_Part2** is an instance of the **SM-Part1**; changes done to 1 component will populate to the instances when pressing **Rebuild**.

7. Inserting a New Part:

Switch to the **Assembly** tab and click:
Insert Component, New Part (arrow).

Select the upper
face of the **SM_**
Part1 to reference
the new part.

A new component
is added to the
FeatureManager
tree. We will
change the name
of this component
within the next
few steps.



8. Sketching the Lid profile:

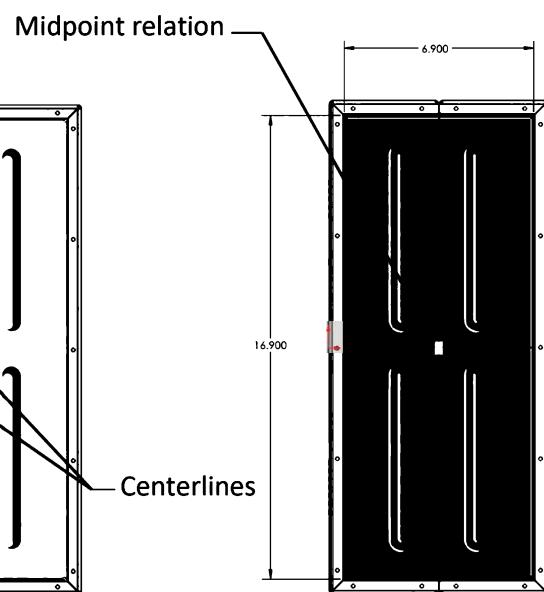
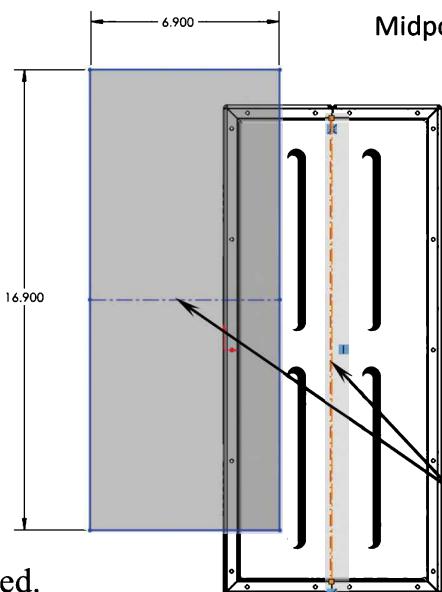
A **new sketch** is opened automatically on the upper face of the SM-Part1.

(X = 6.900in. Y = 16.900in.)

Sketch a **Rectangle**,
a **Horizontal**, and
a **Vertical**
Centerline
as shown in the
image.

Add the height
and width
dimensions

Add a **Mid-Point**
relation to center
the rectangle as noted.



9. Making the Base Flange:

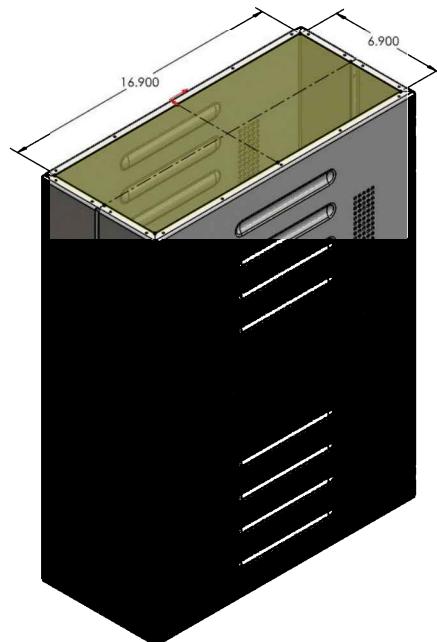
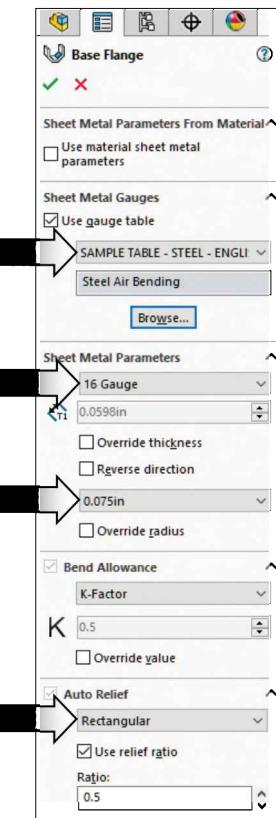
Switch to the **Sheet Metal** tab and click **Base Flange**.

Click **Use Gauge Table**.

Select the following:

Sample Table: Steel.
 Gauge: 16
 Bend Radius: .075in
 K-Factor: 0.5
 Auto Relief: Rectangular
 Use Relief Ratio: Enabled
 Ratio: 0.5

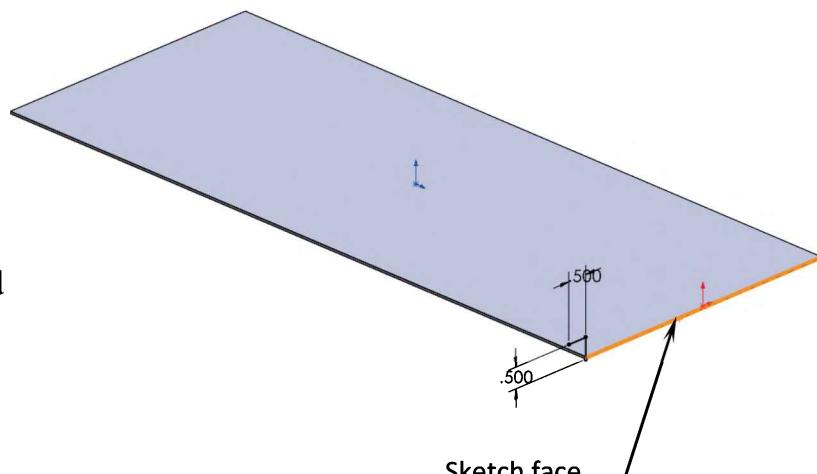
Click **OK**.



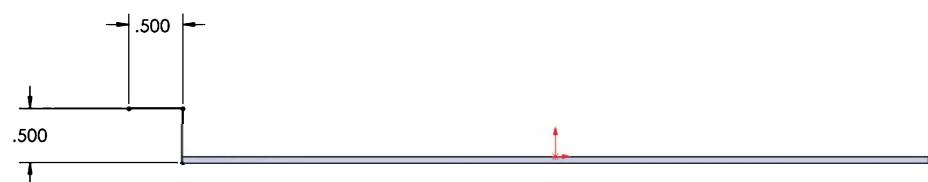
10. Sketch the Miter profile:

Select the side face
 and open a new
 sketch.

Sketch a **vertical line** and
 a **horizontal line** starting
 at the bottom left corner
 of the part.



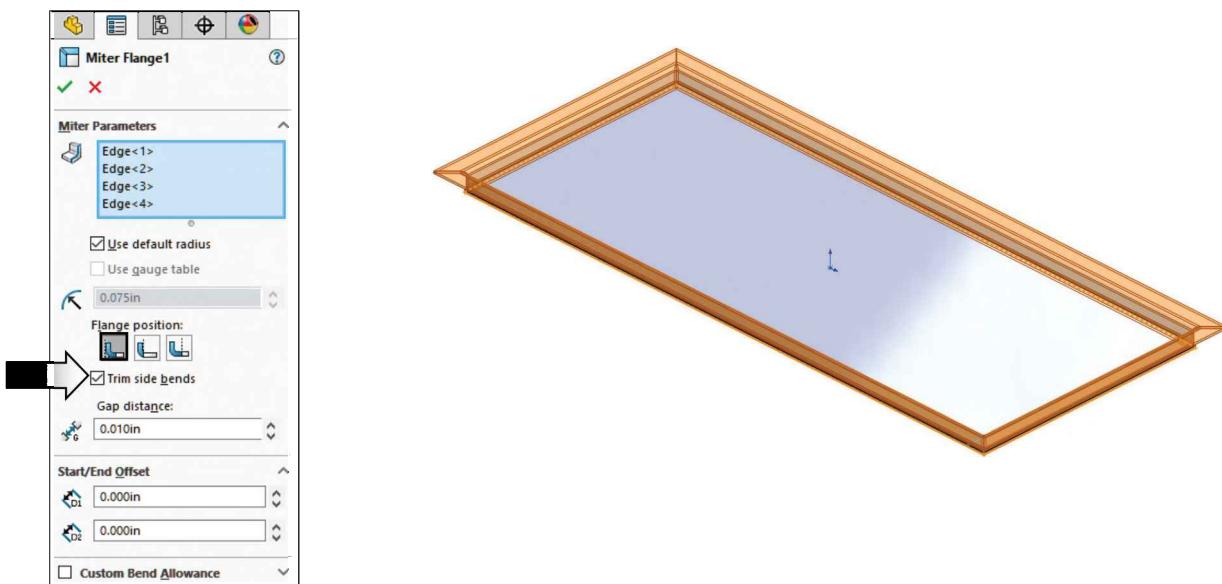
Add dimensions
 to fully define
 the sketch.



11. Creating the Miter Flanges:

Switch to the **Sheet Metal** tab and click **Miter Flange**.

The sketch is automatically swept to the end of the 1st edge of the model.
Select the other **3 bottom edges** to bring the Miter Flange all the way around the perimeter of the part.



For Flange Position, click **Material Inside**.

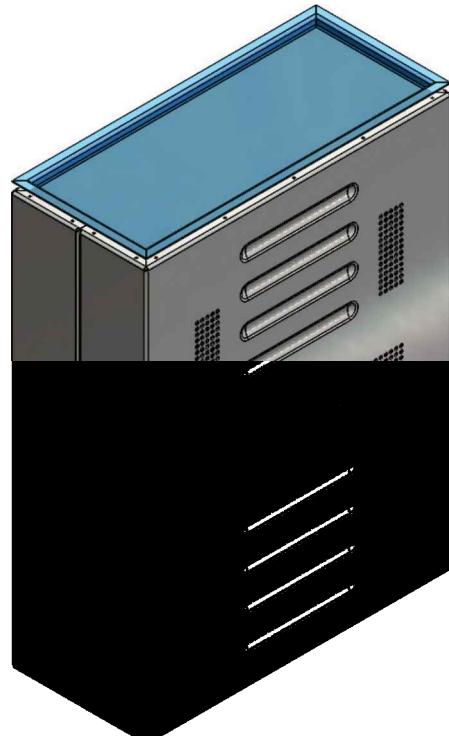
Enable the **Trim Side Bends** checkbox (arrow).

Enter **.010in** for Gap Distance.

Keep all other parameters at their default values.

Click **OK**.

Inspect your model against the image shown here.



12. Making the cuts:

Select the upper face as noted and open a **new sketch**.

Sketch the profiles of the cut feature.

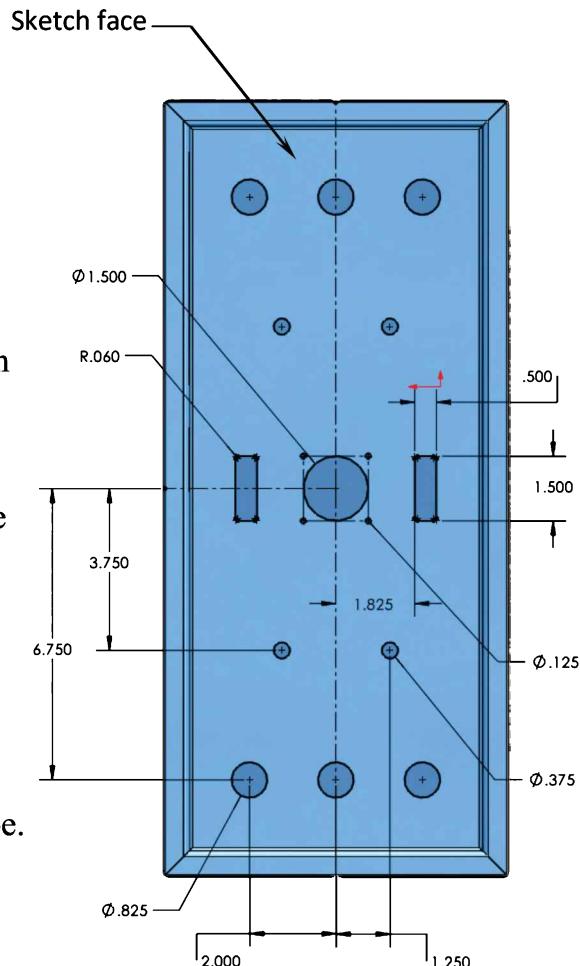
Use the mirror function to keep the sketch entities **symmetric** with one another.

Add the dimensions shown to fully define the sketch. (All fillets are R.060.)

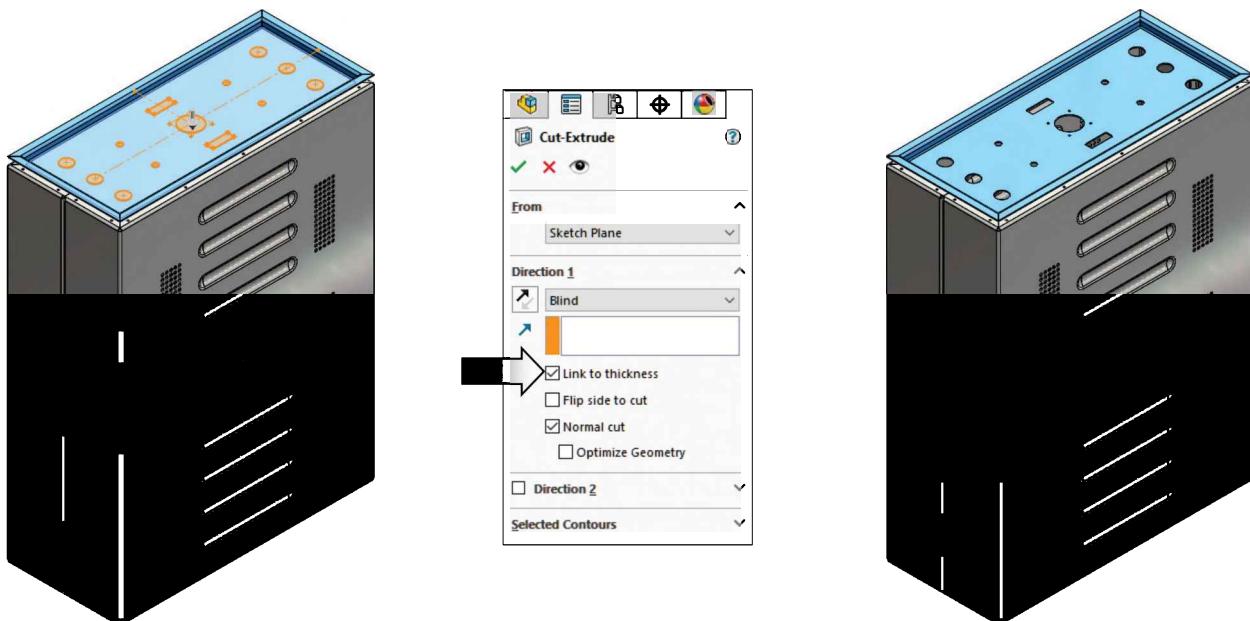
Switch to the **Sheet Metal** tab and click **Extruded Cut**.

For Direction 1, use the default **Blind** type.

Enable the **Link to Thickness** checkbox.



Click **OK**.



13. Adding the Countersink holes:

Switch to the **Features** tab.

Select the upper face of the part and click **Hole Wizard**.

For Hole Type, select **Countersink**.

Enter/select the following:

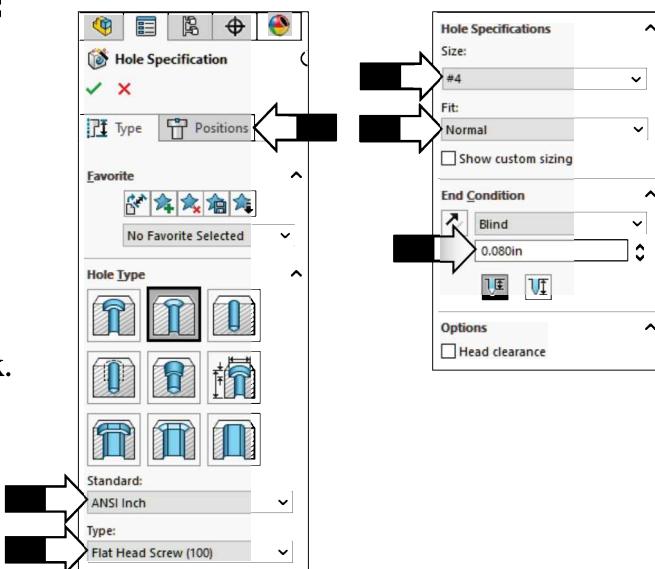
Standard: Ansi Inch

Type: Flat head Screw (100)

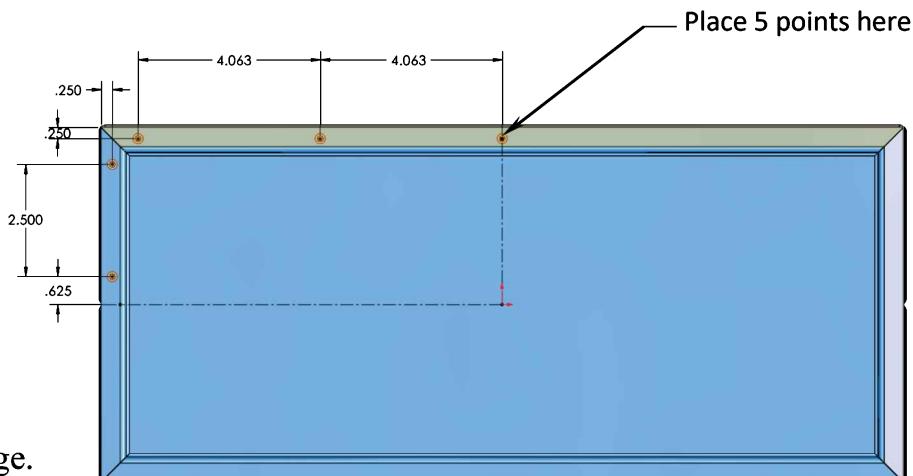
Size: #4

Fit: Normal

Blind: .080"



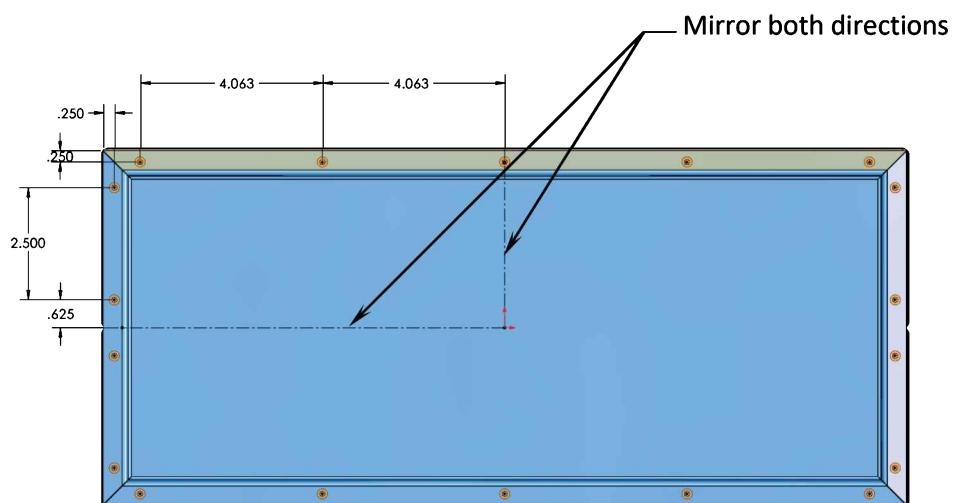
Click the **Position** tab.



Place **5 points** approximately as shown in the image.

Sketch a vertical and horizontal **centerline** and **mirror** the 5 sketch points.

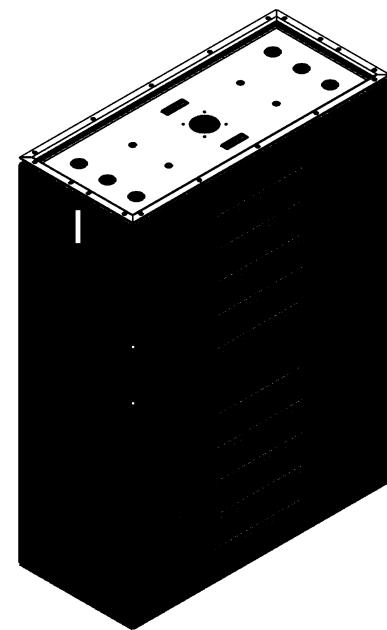
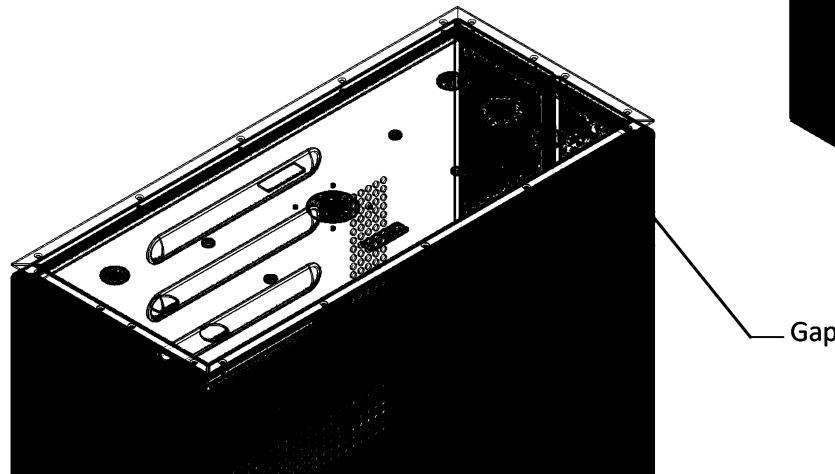
Add horizontal, vertical relations, and the location dimensions to fully define the sketch.



Click **OK** to exit the Hole Wizard mode.

Switch to the **Assembly** tab and click-off the **Edit Component** command.

The countersink holes should match up with the #4-40 holes on top of the SM-Part1 and SM_Part2 components.



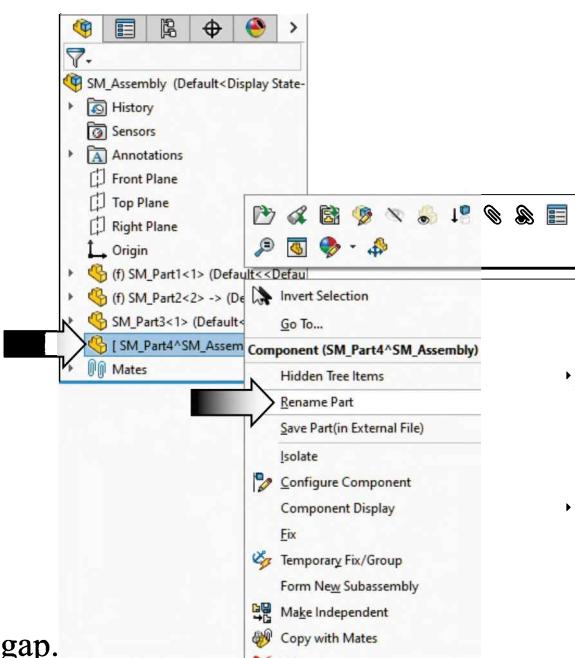
14. Renaming a component:

Right-click the name of the new part (Part1) and select **Rename**.

Enter **SM_Part4** for the name of the new part.

There is a gap between the new part and the SM_Part1.

We will add some mates to close off the gap.

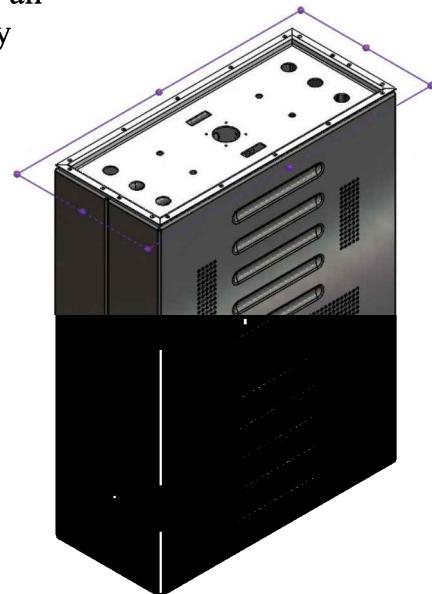
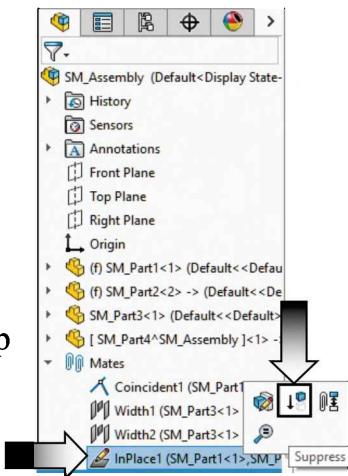


15. Suppressing the InPlace mate:

When new components are created in the context of an assembly, the InPlace mates are added automatically to reference the location of each new part.

The InPlace mate that was added to the new component must be suppressed before adding any new mates.

Expand the **Mates** group near the bottom of the FeatureManager tree.



Click the **Inplace1** mate and select **Suppress**.

16. Adding new mates:

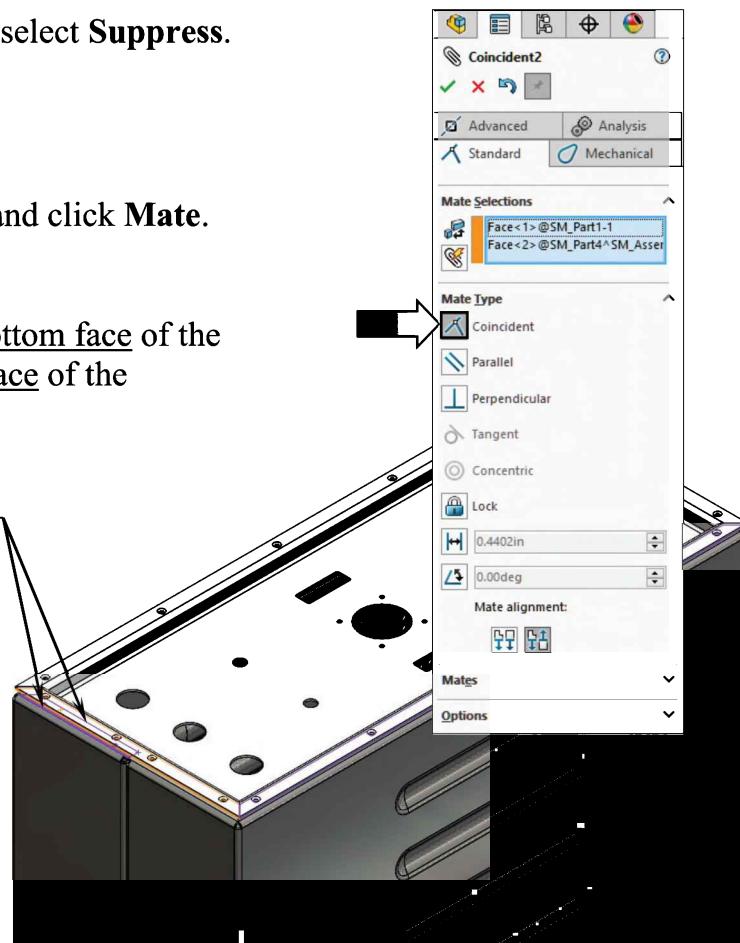
Switch to the **Assembly** tab and click **Mate**.

For the 1st mate, select the bottom face of the Miter Flange and the upper face of the SM_Part1 component.

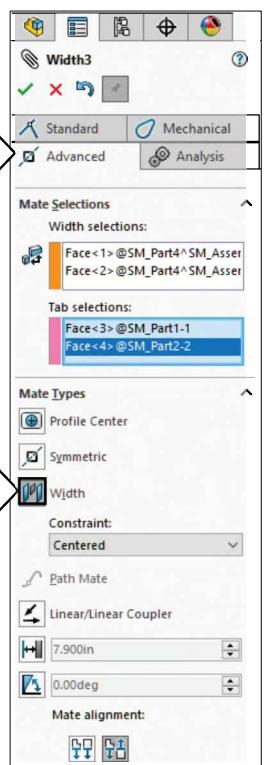
Top face of SM_Part1
and bottom face of
Miter Flange

Click **Coincident**.

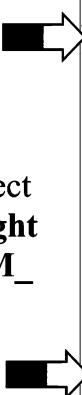
The SM-Part4 should move downward and touch the upper face of the SM_Part1.



For the 2nd mate,
switch to the
Advanced tab
(arrow) and
click **Width**.



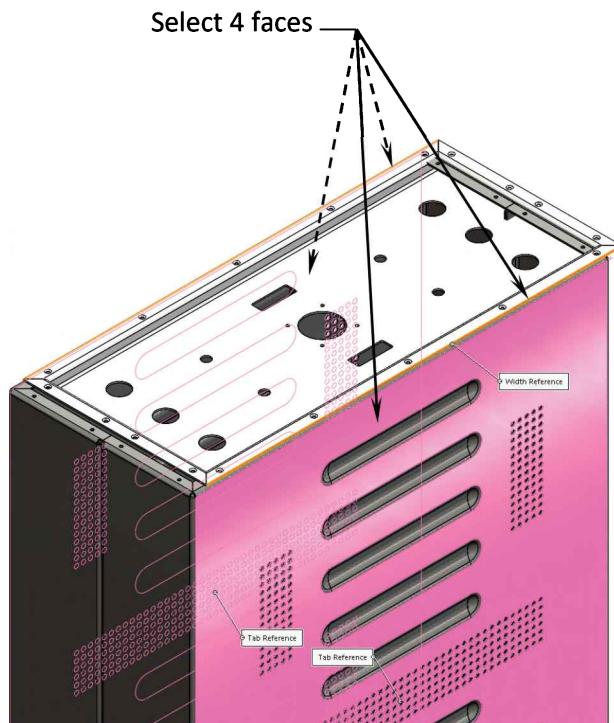
For Width-
Selections, select
the **left and right**
faces of the **SM_**
Part1.



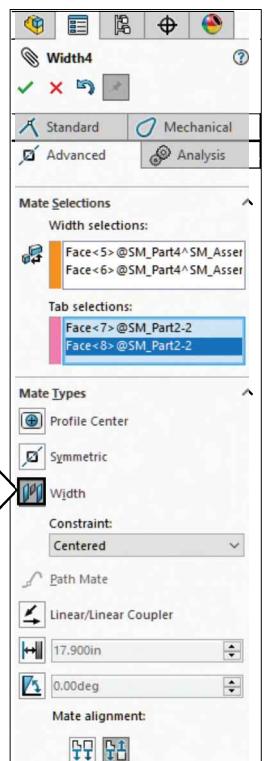
For the Tab-
Selections, select
the **left and right**
faces of the **SM_**
Part4.



Click **OK**.



For the 3rd mate,
stay in the
Advanced mate
mode and click
Width again.



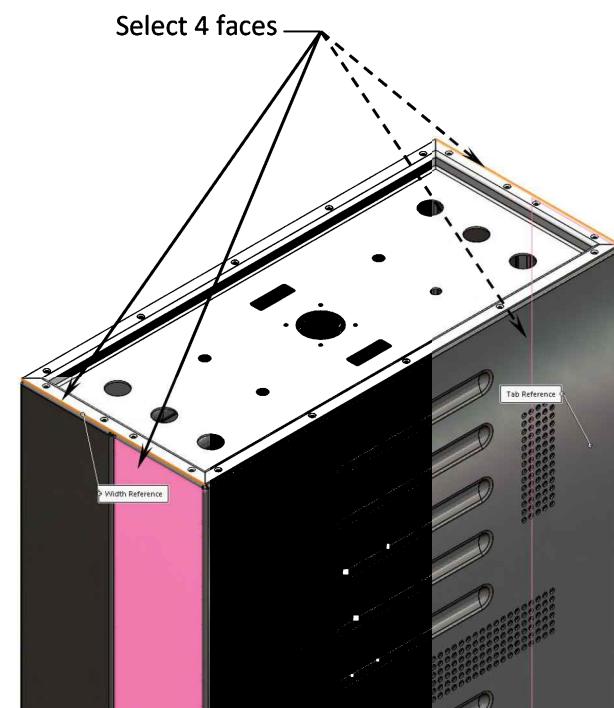
For Width-
Selections, select
the **left and right**
faces of the **SM_**
Part1.



For the Tab-
Selections, select
the **left and right**
faces of the **SM_**
Part4.



Click **OK**.

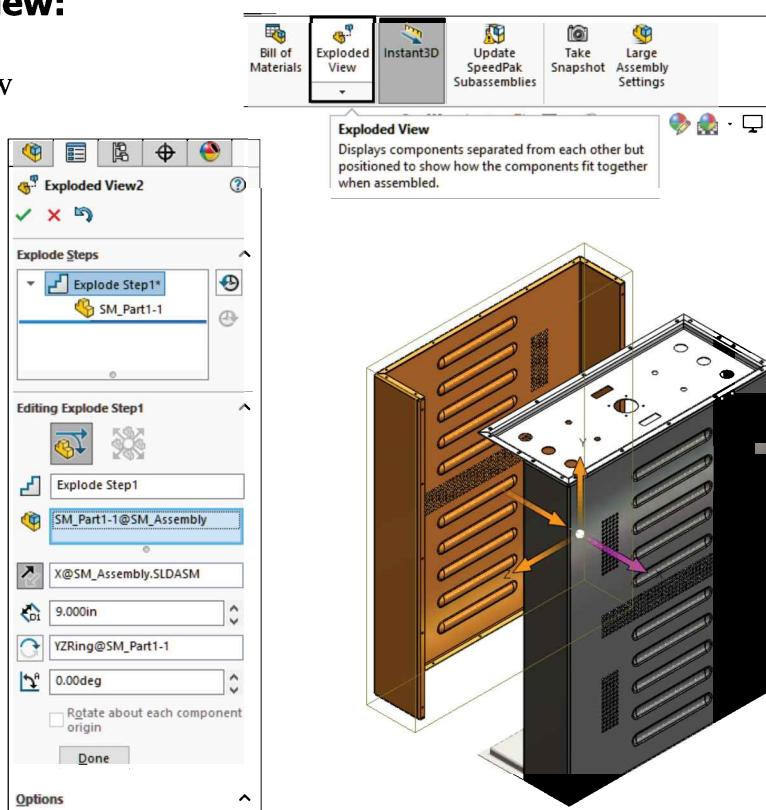


17. Creating an exploded view:

An assembly exploded view is usually created when interior components are not visible in the assembled view.

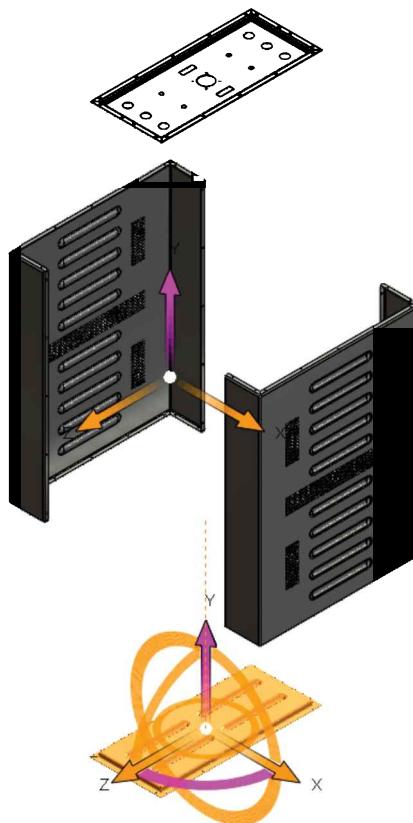
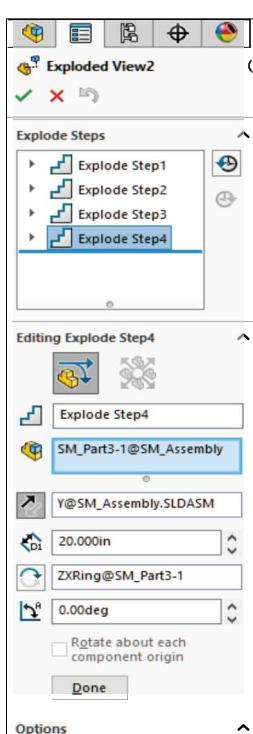
The exploded view is also used in a drawing to place balloons that identify the components in the Bill of Materials.

Switch to the Assembly tab and click Exploded View.



For Explode Step1, select the component **SM_Part1** either from the graphics area or from the FeatureManager tree.

Drag the X-Direction arrow to the left, approximately 9.00in.



Drag the SM_Part2 towards the X Positive, approximately 11.00in.

Drag the SM_Part3 towards the Y positive, approximately 19.00in.

Drag the SM_Part4 towards the Y negative, approximately 20.00in.

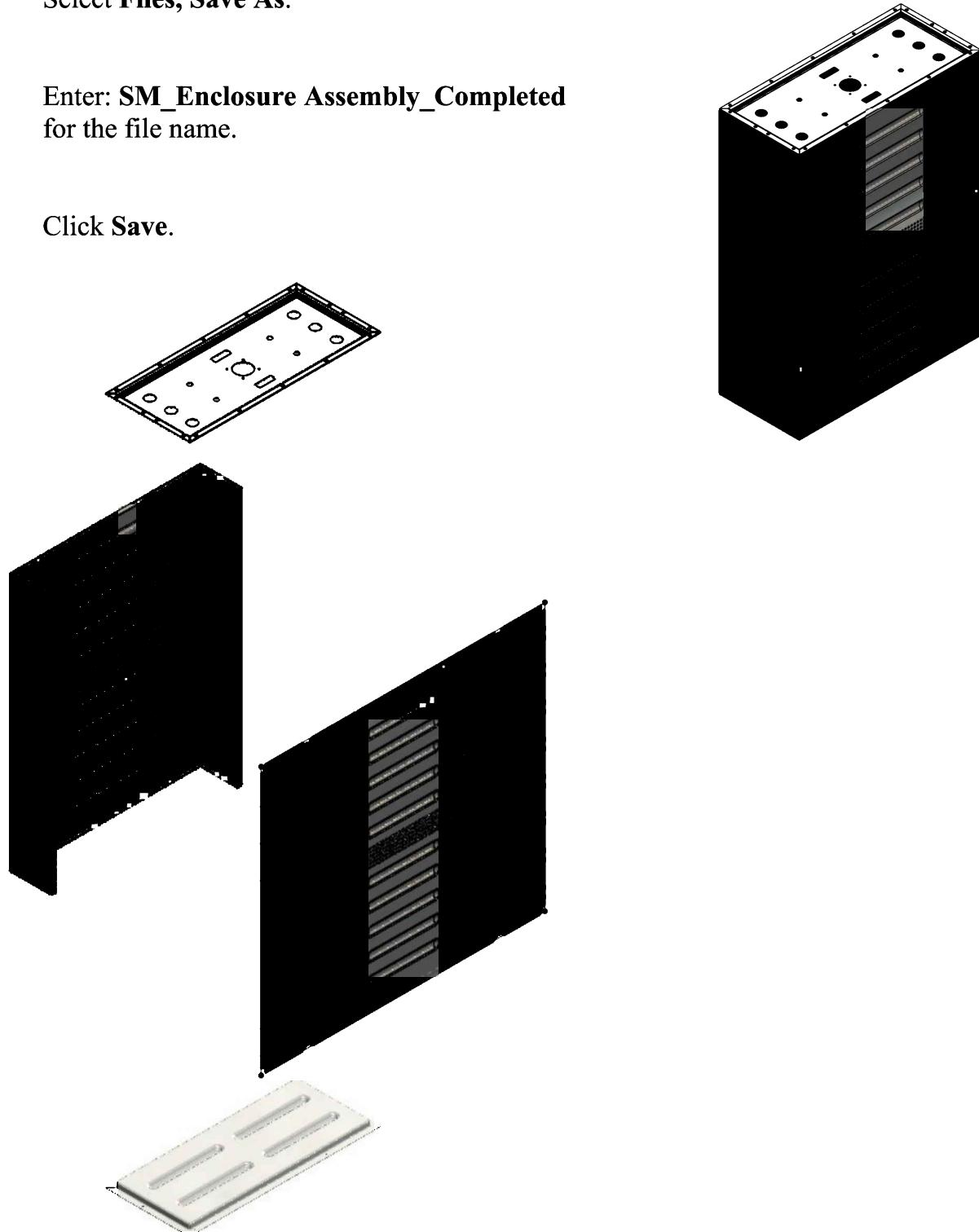
Click **OK** to accept the exploded view and to exit out of the command.

18. Saving your work:

Select **Files, Save As.**

Enter: **SM_Enclosure Assembly_Completed**
for the file name.

Click **Save.**



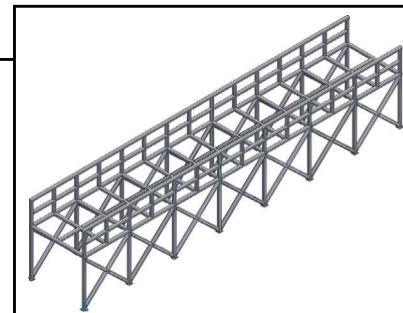
CHAPTER 17

Advanced Weldments

Advanced Weldments

Weldments Platform

Weldments can be used to create a structure as a single or multibody part. Either a 2D or 3D sketch can be used to define the framework, then create structural members containing groups of sketch segments.



Other Weldments tools can be used to add objects such as gussets and end caps. Weld gaps and beads can also be added to the model. Drawings can be made to document the design, including tables of cut materials, cut length, and weld bead totals.

Weldments are normally created in groups. A group is a collection of related segments in a structural member. A group is configured to affect all its segments without affecting other segments or groups in the structural member.

The 2 types of groups are:

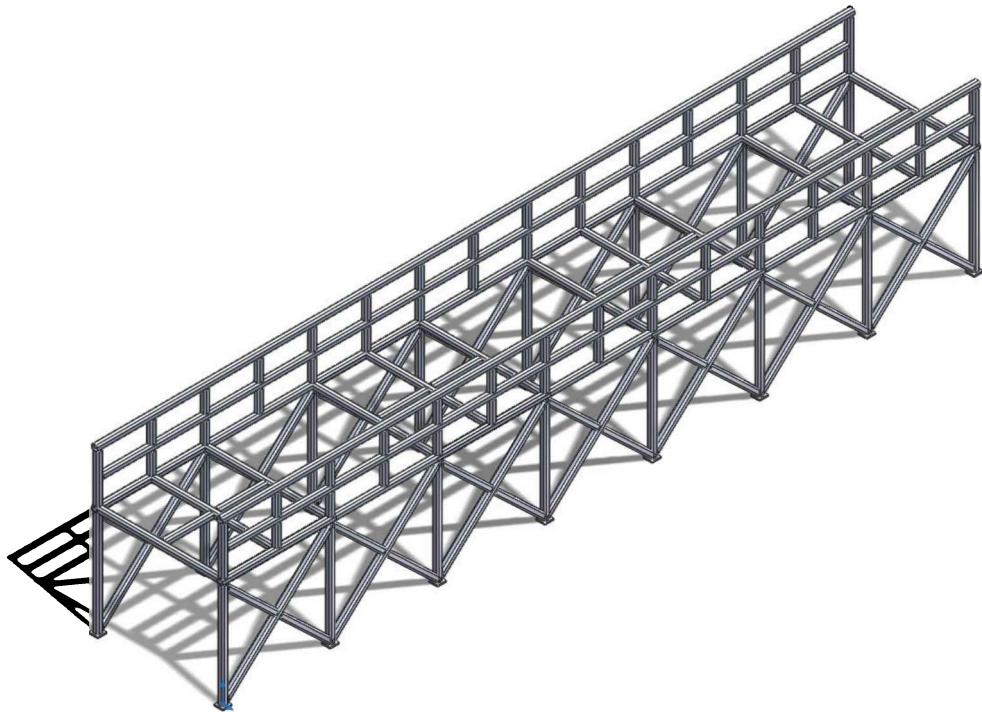
Contiguous: A continuous contour of segments joined end-to-end. You can control how the segments join to each other. The end point of the group can optionally connect to its beginning point.

Parallel: A discontinuous collection of parallel segments. Segments in the group cannot touch each other.

- * You can define a group in a single plane or in multiple planes.
- * A group can contain one or more segments.
- * A structural member can contain one or more groups.
- * After you define a group, you can operate on it as a single unit. Use the Structural Member PropertyManager to specify the corner treatment for the segments in the group.

Advanced Weldments

Weldments Platform



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI
Units: INCHES – 3 Decimals

Tools Needed:



3D Sketch



Extruded Boss-Base



Linear Pattern



Structural Member



Trim/Extend

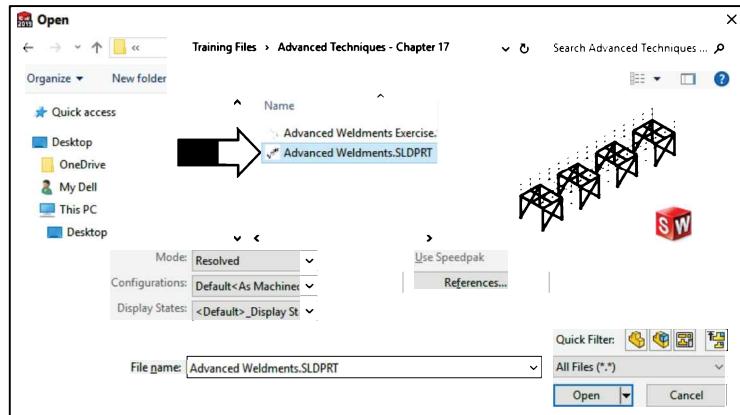


End Trim Type

1. Opening a part document:

Click File / Open.

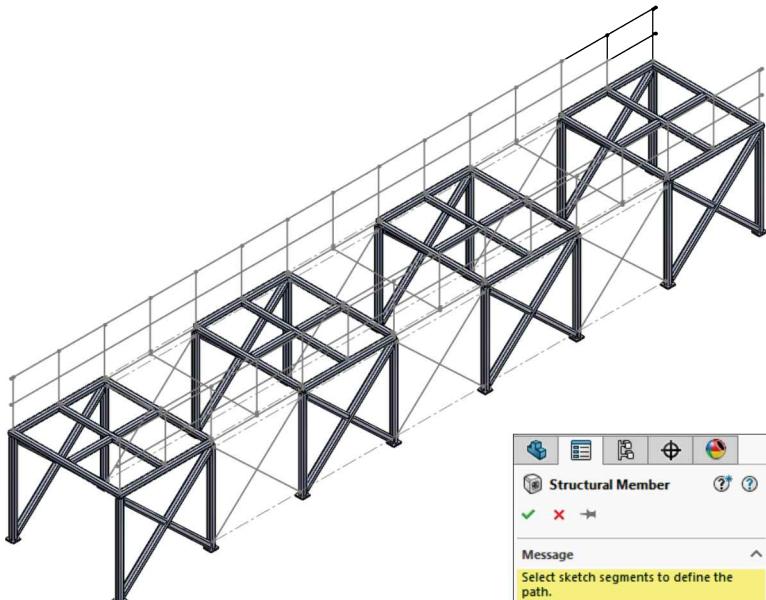
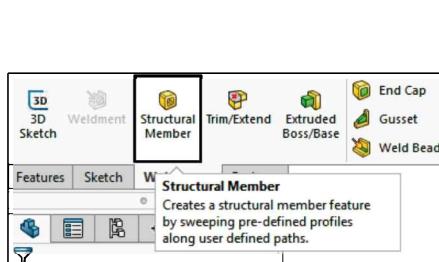
Browse to the Training Folder and open a part document named:
Advanced Weldments.



2. Adding the Weldments toolbar:

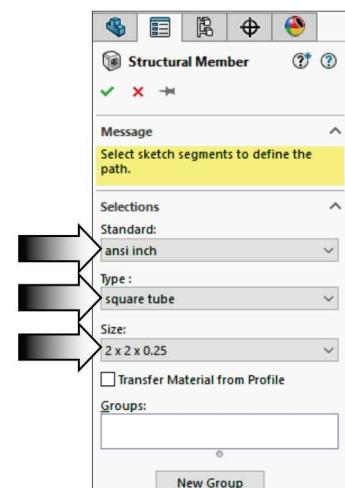
Right-click one of the tabs and enable the **Weldments** toolbar.

Select the **Structural Member** command . This adds a **Weldments** feature on the FeatureManager tree.



Select the following:

- * Standard:
ANSI Inch
- * Type:
Square Tube
- * Size:
2 x 2 x 0.25

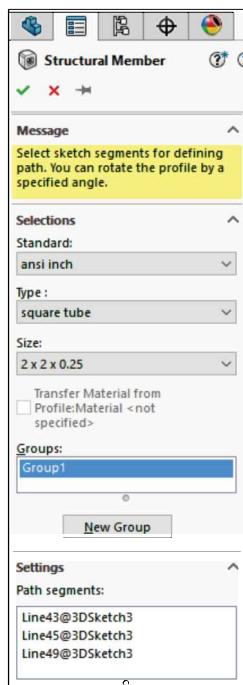


You can define a group in a single plane or in multiple planes. After you define a group, you can operate on it as a single unit.

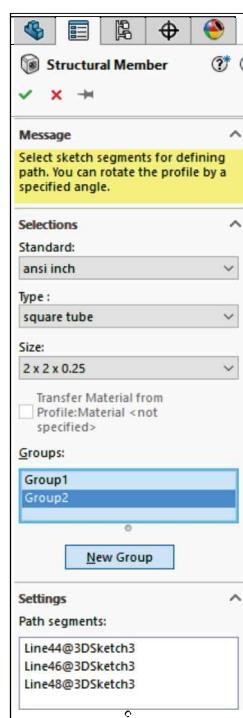
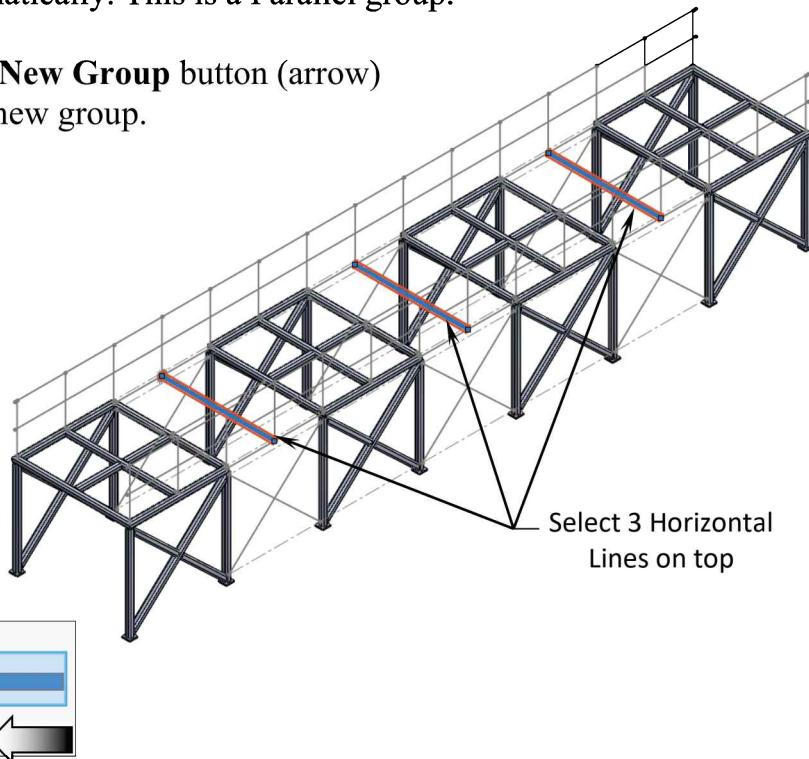
3. Adding the Structural Members:

Start by selecting the **3 horizontal lines** on the top portion as indicated.

Group 1 is created automatically. This is a Parallel group.



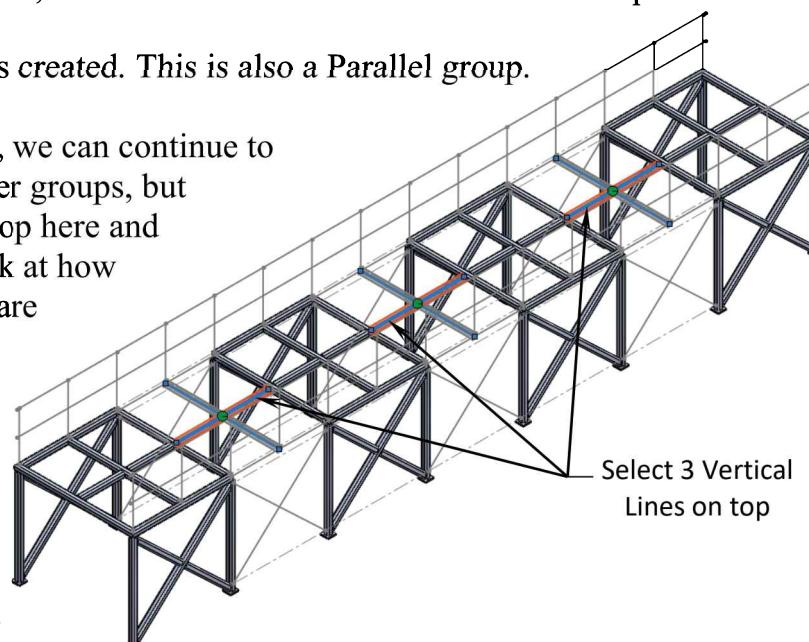
Click the **New Group** button (arrow) to start a new group.



For Group 2, select the **3 vertical lines** also from the top area.

Group 2 is created. This is also a Parallel group.

Typically, we can continue to select other groups, but we will stop here and take a look at how the tubes are trimmed.

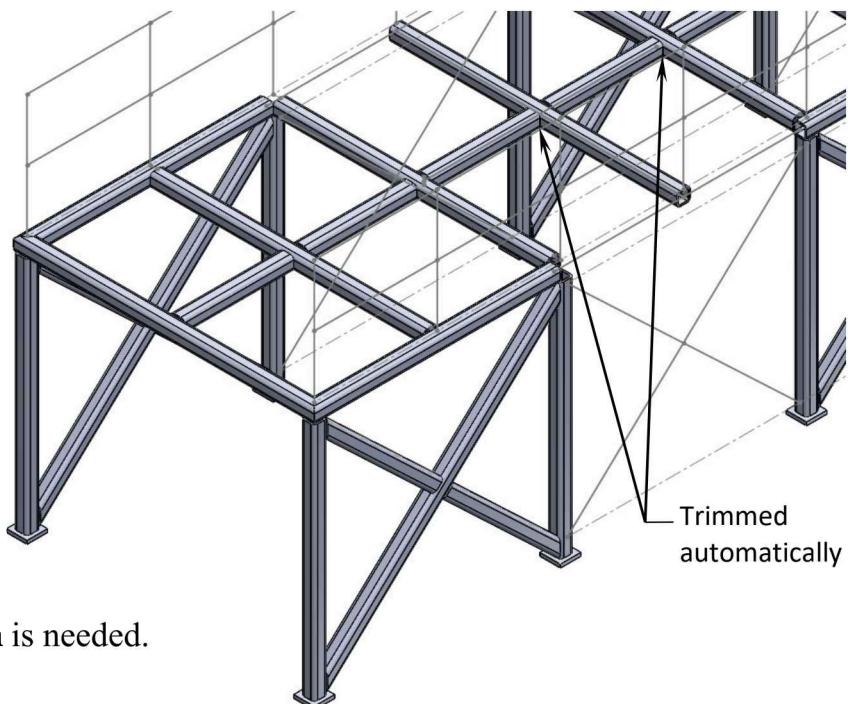


Click **OK**.

Zoom in on the first two sections of the platform frame.

Notice the new tubes were trimmed automatically.

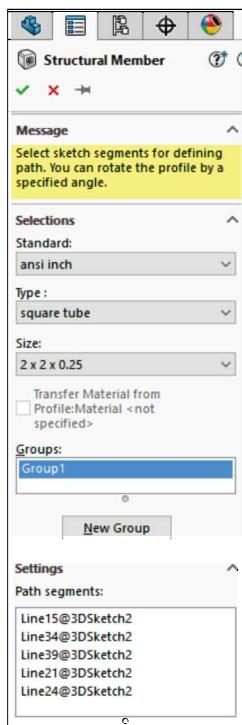
If the Structural Members are created separately, manual trim is needed.



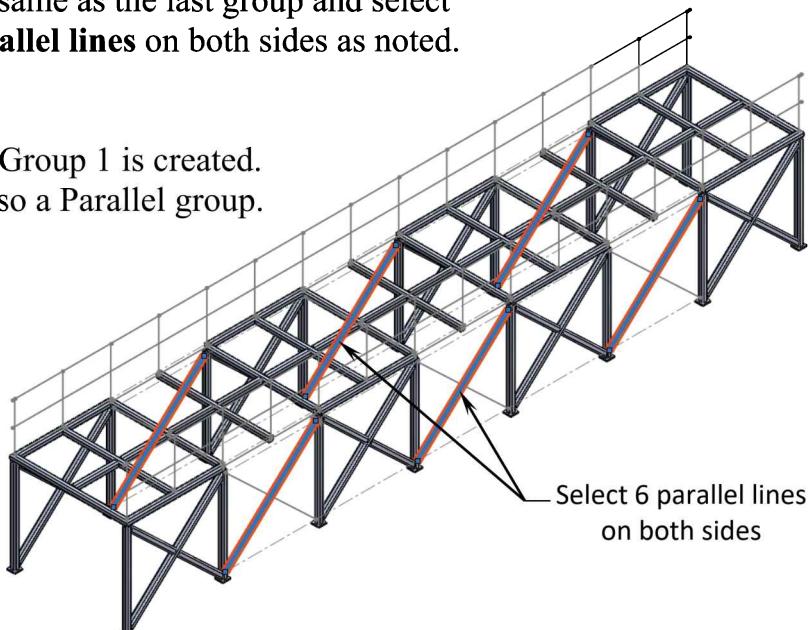
4. Adding more Structural Members:

Click the **Structural Member** command again.

Keep all parameters the same as the last group and select the **6 parallel lines** on both sides as noted.

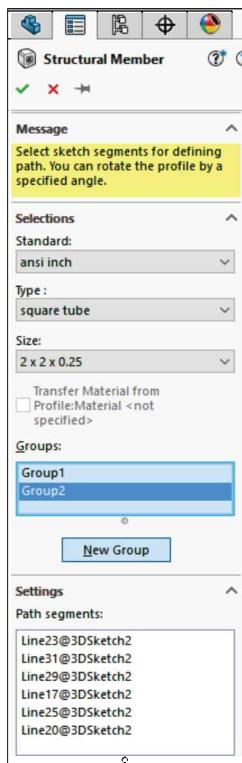


The new Group 1 is created. This is also a Parallel group.

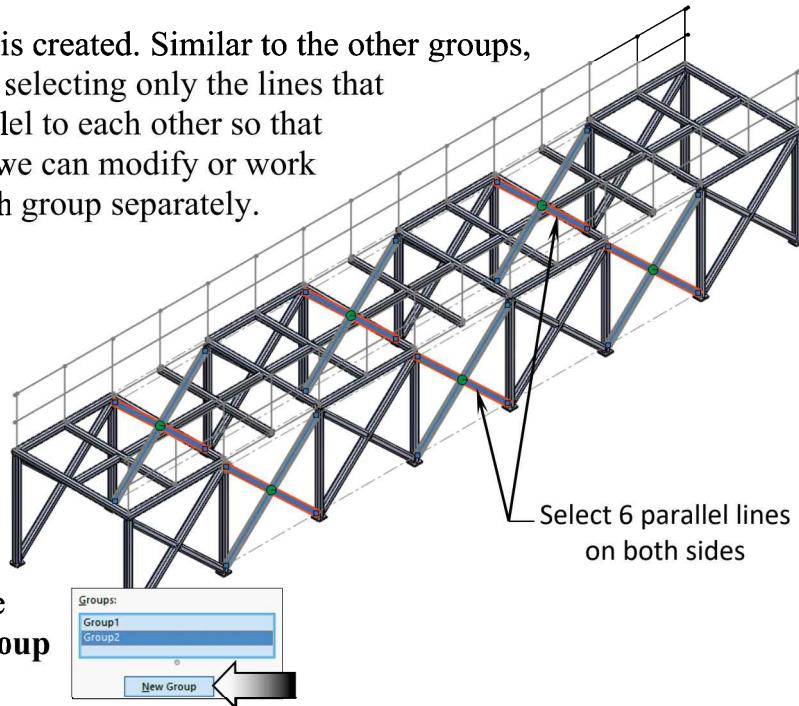


Click the **New Group** button to start a new group.

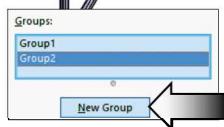
For Group 2, leave all settings the same as the last group and select the **6 parallel lines** on both sides as indicated in the image.



Group 2 is created. Similar to the other groups, we were selecting only the lines that are parallel to each other so that later on we can modify or work with each group separately.



Click the **New Group** button.

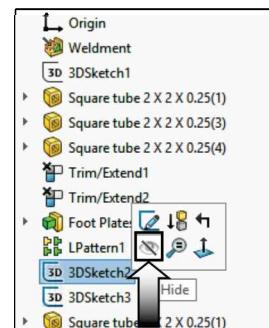
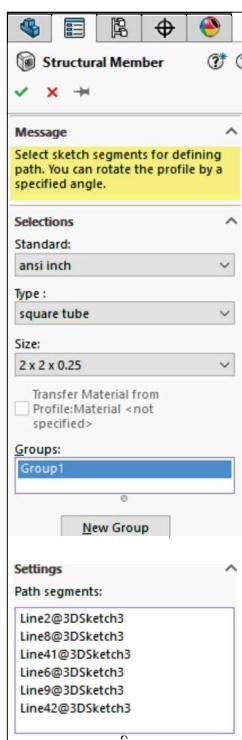


For the new Group 2, select the **10 horizontal lines** for the rails.

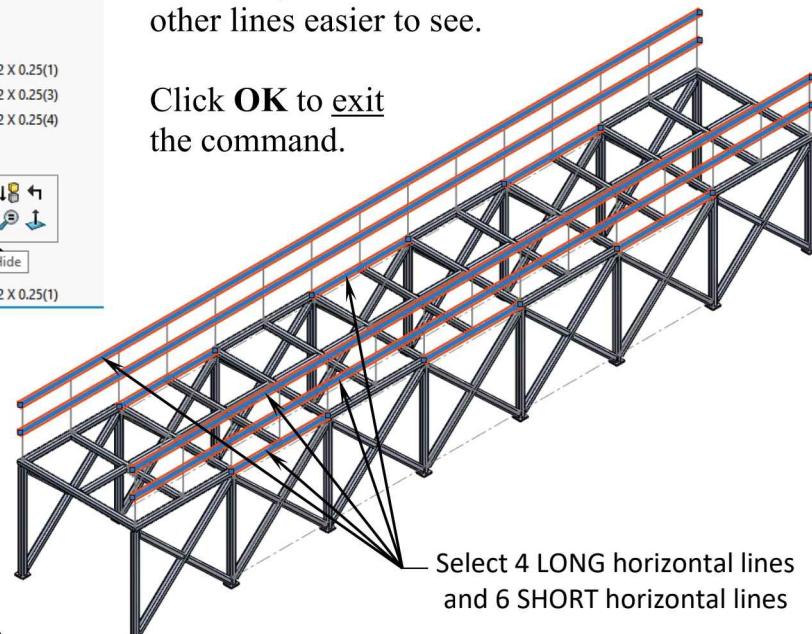
Group 2 is created but the weldment platform is becoming very busy, making it hard to see the other lines.

For clarity, we will hide Sketch2 to make the other lines easier to see.

Click **OK** to exit the command.



Click **Sketch2** on the Feature tree and select **Hide**.



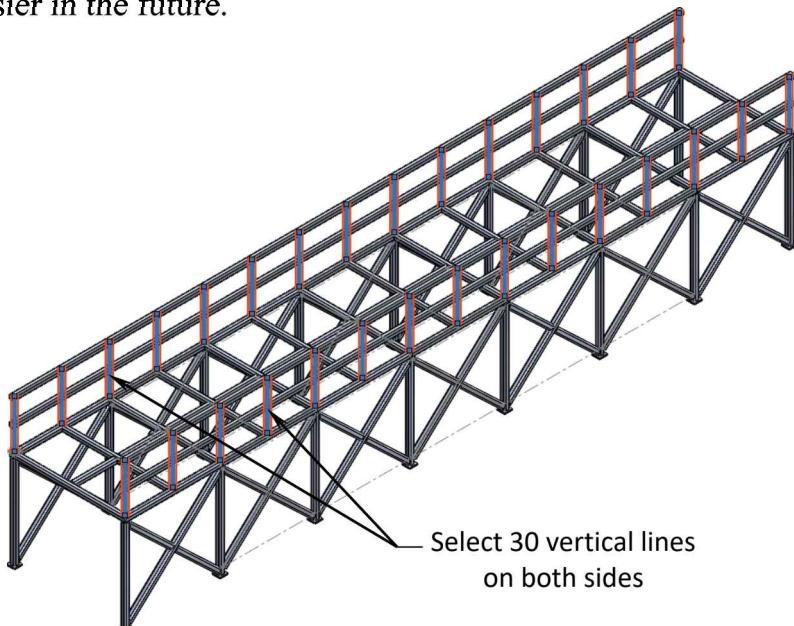
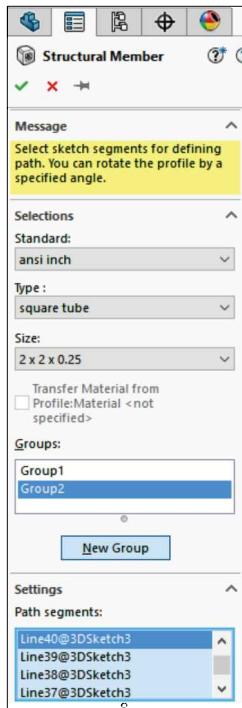
Select 4 LONG horizontal lines and 6 SHORT horizontal lines

5. Adding the vertical Structural Members to the rails:

Click the **Structural Member** command  again.

For the new Group 1, select the **30 vertical lines** as indicated.

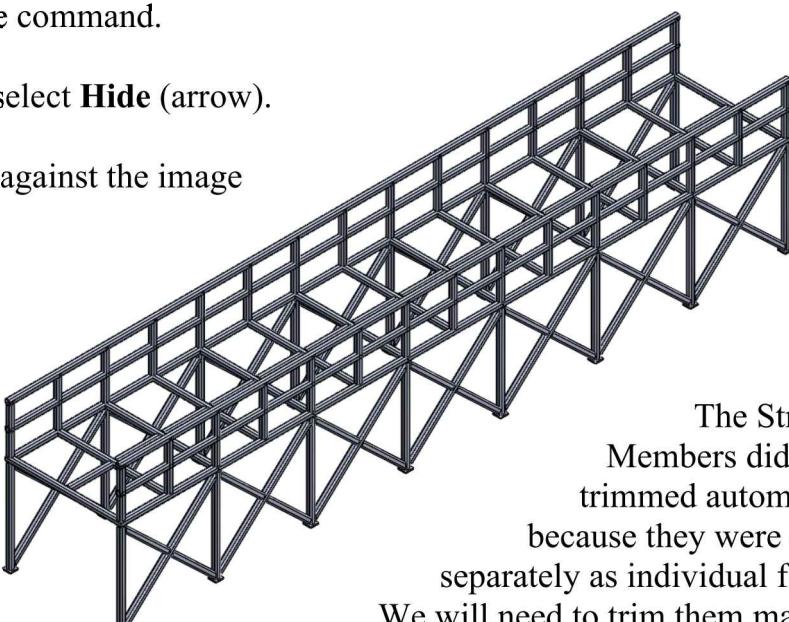
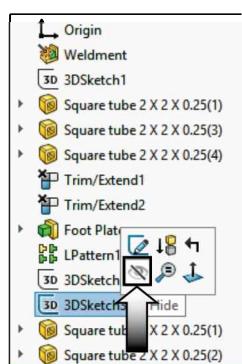
The new Group 1 is created with all parallel lines in it. Again, by grouping the lines, they become one unit. Editing will be much easier in the future.



Click **OK** to exit the command.

Click **Sketch3** and select **Hide** (arrow).

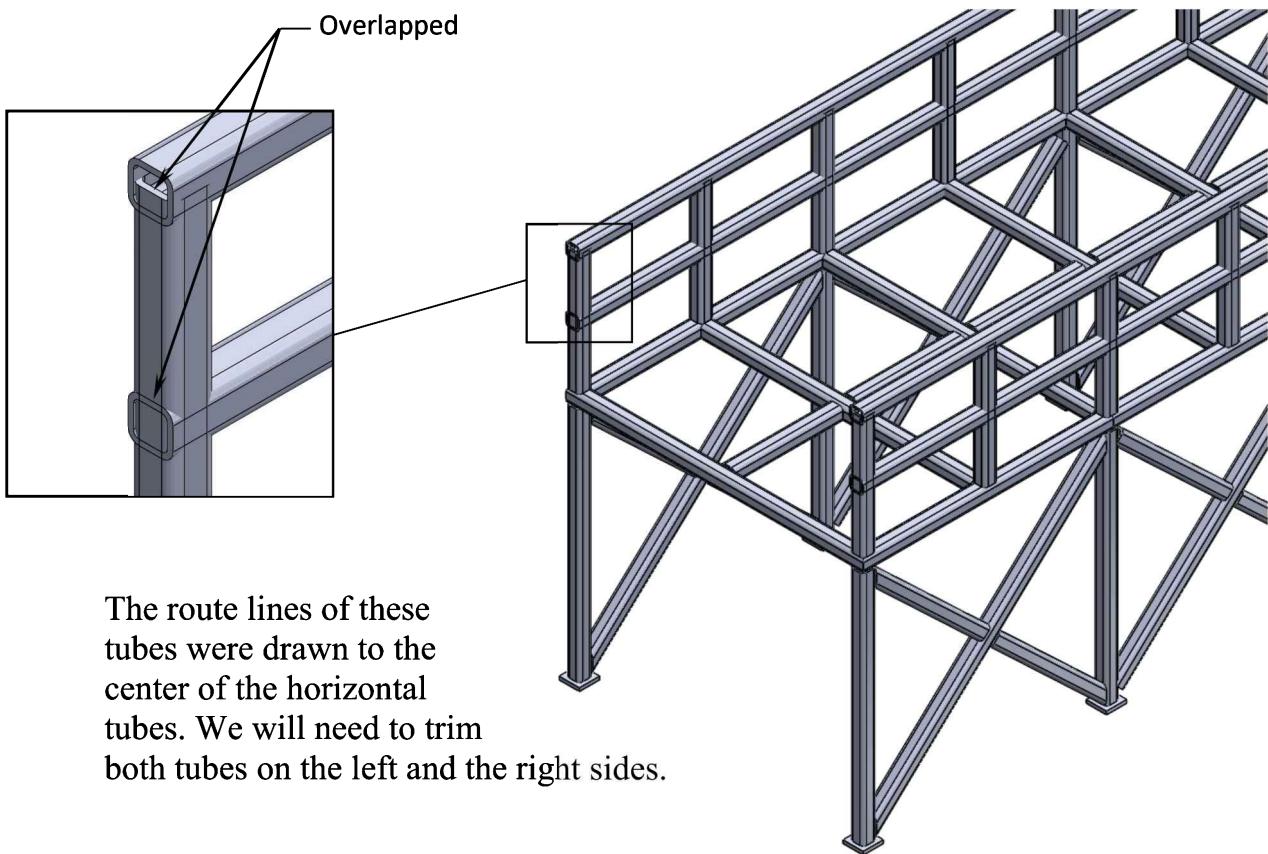
Inspect your model against the image shown here.



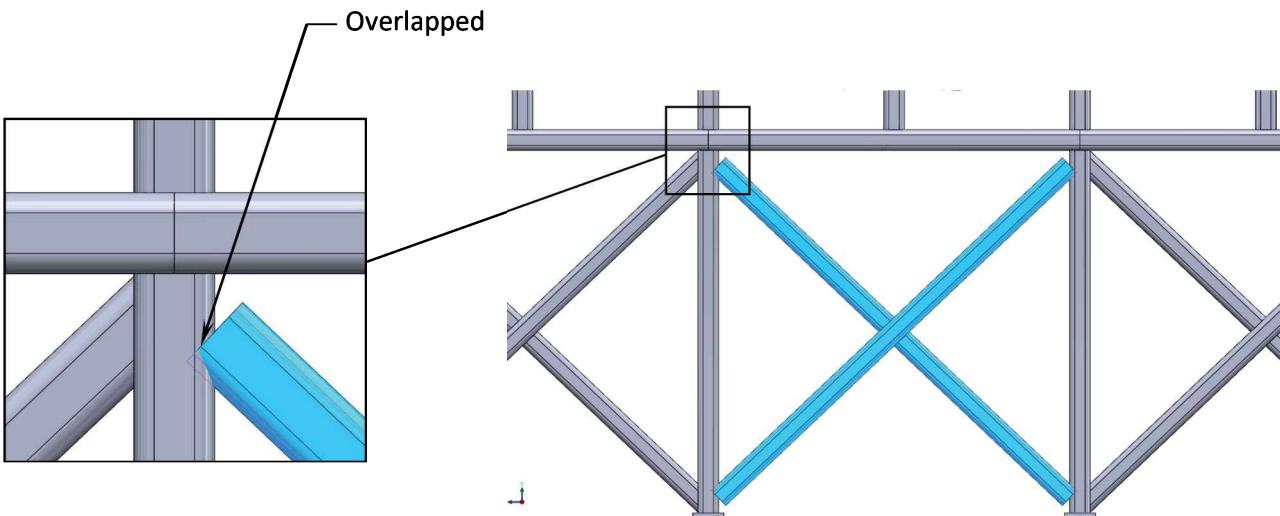
The Structural Members did not get trimmed automatically because they were created separately as individual features. We will need to trim them manually.

6. Viewing the overlapped areas:

Zoom in on the area shown in the enlarged view to see the overlapped issues.



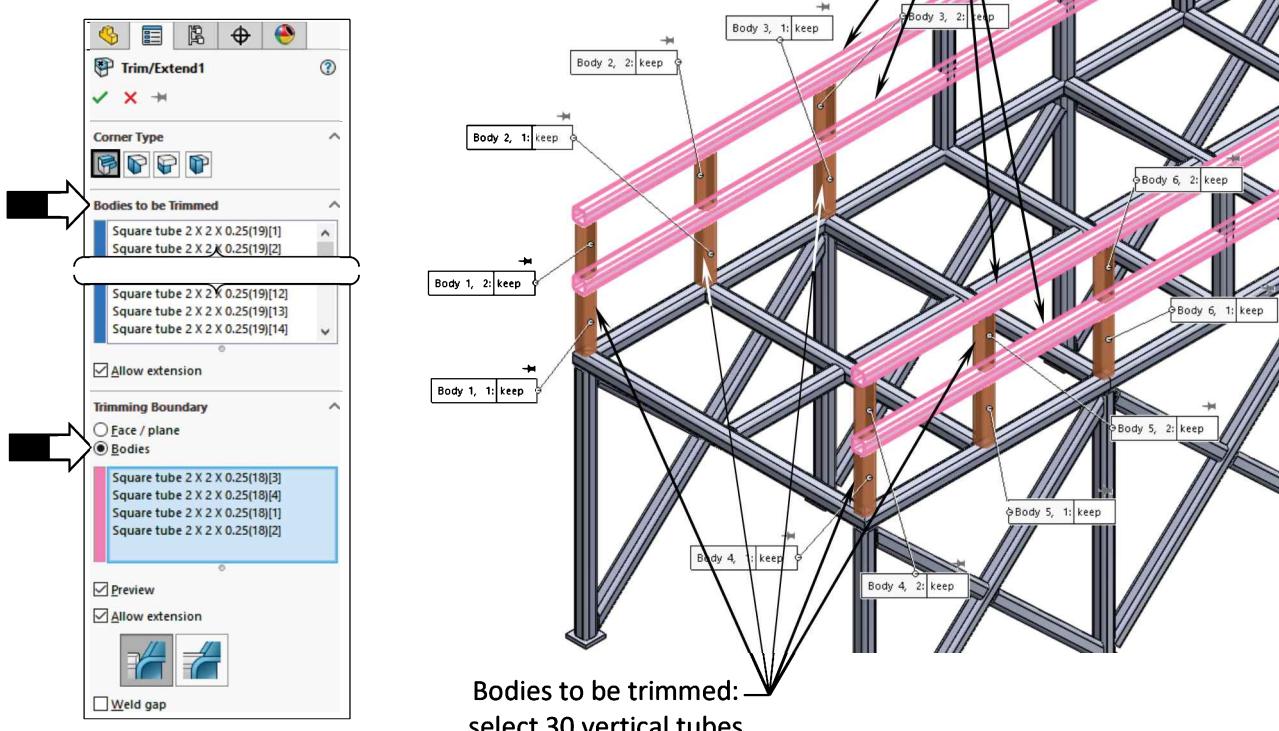
The cross-tubes are also overlapped with the vertical members. We will need to trim/extend all of them, front and back sides.



7. Trimming the overlapped:

Click the Trim/Extent command .

Use the default End Trim option .

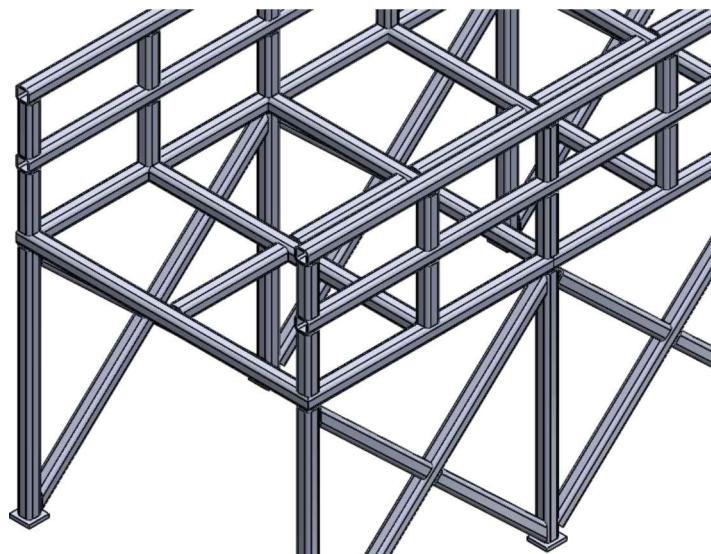


For Bodies to be Trimmed, select the **30 vertical members** as indicated.

For Trimming Boundary, click the **Bodies** button and select the **4 horizontal tubes** as indicated.

The preview graphics shows the vertical members are trimmed to match the faces of the horizontal bodies.

Click **OK** to exit the Trim.



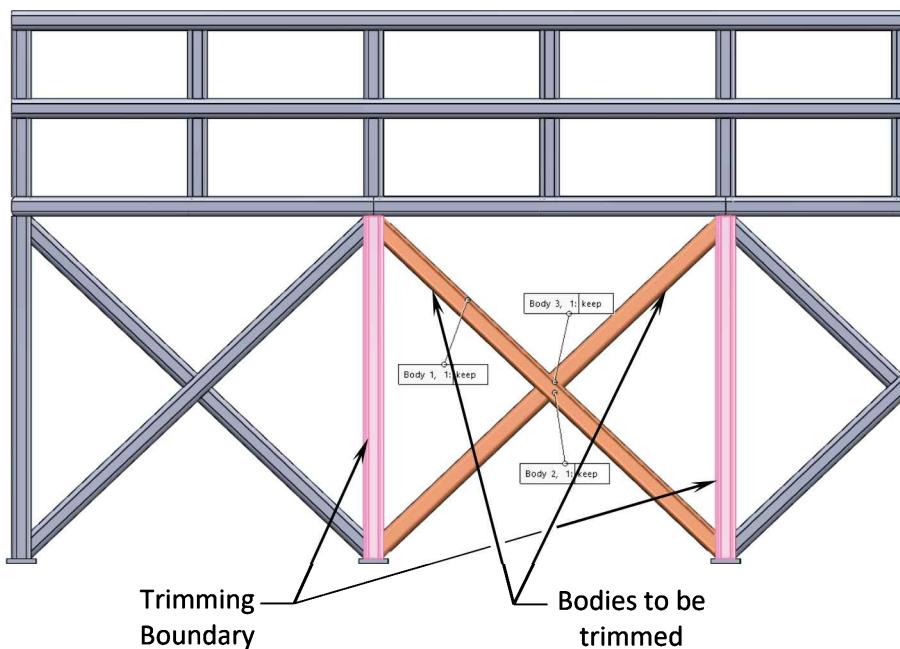
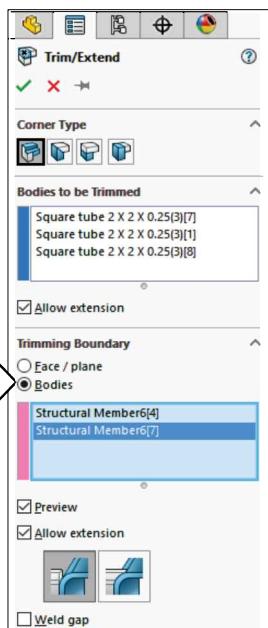
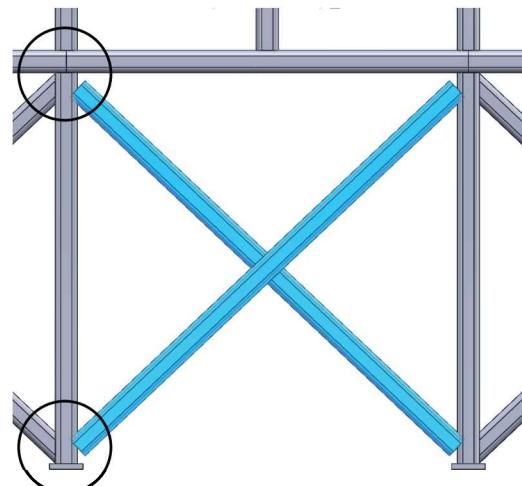
8. Trimming the cross-members:

Click the Trim/Extent command .

Use the default End Trim option .

Change to the Right orientation
(Control + 4).

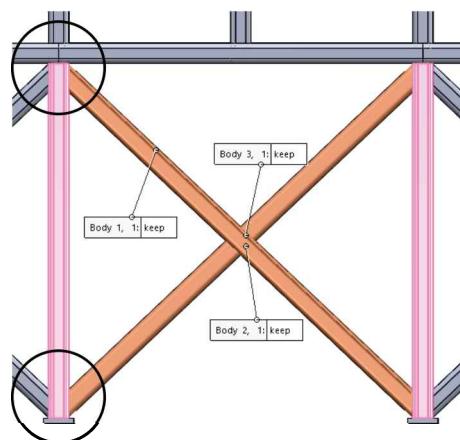
For Bodies to be Trimmed, select the
Cross-member indicated.



For Trimming Boundary, select the **vertical member** as noted.

The preview graphic shows the cross-member is trimmed to the right face of the vertical tube (circled).

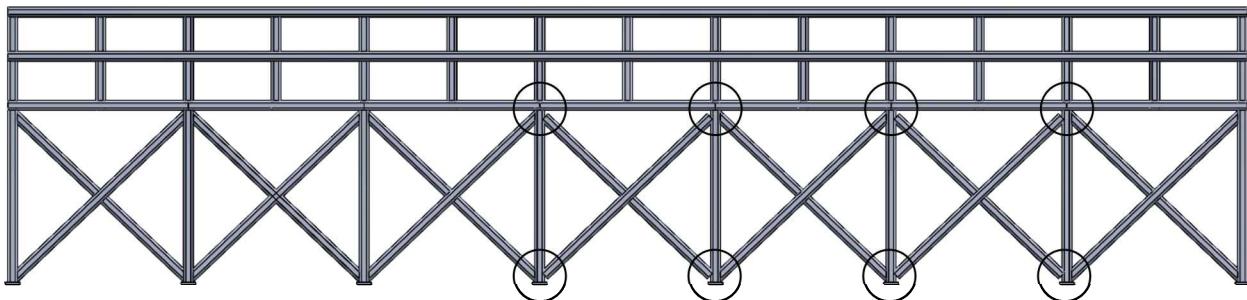
Click **OK** to exit the trim command.



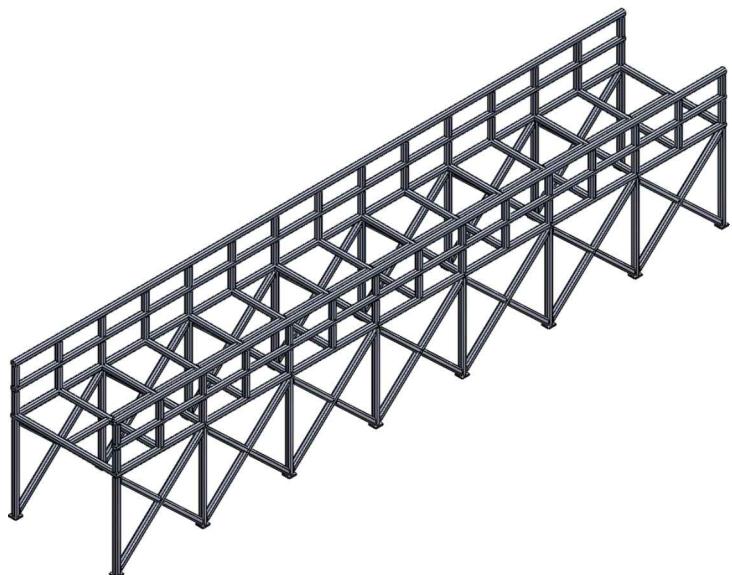
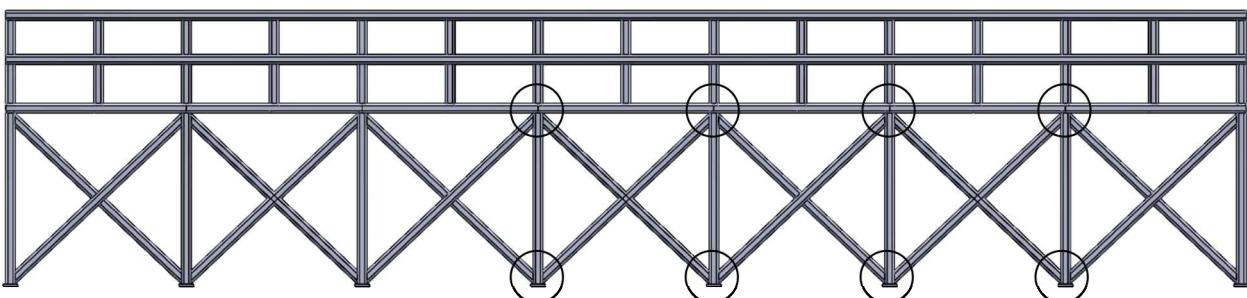
9. Trimming the other structural members:

There are several other structural members throughout the platform that still need to be trimmed (circles).

Change to the **Right** orientation (Control + 4).



Click the **Trim/Extend** command and trim the tubes shown in the circles.



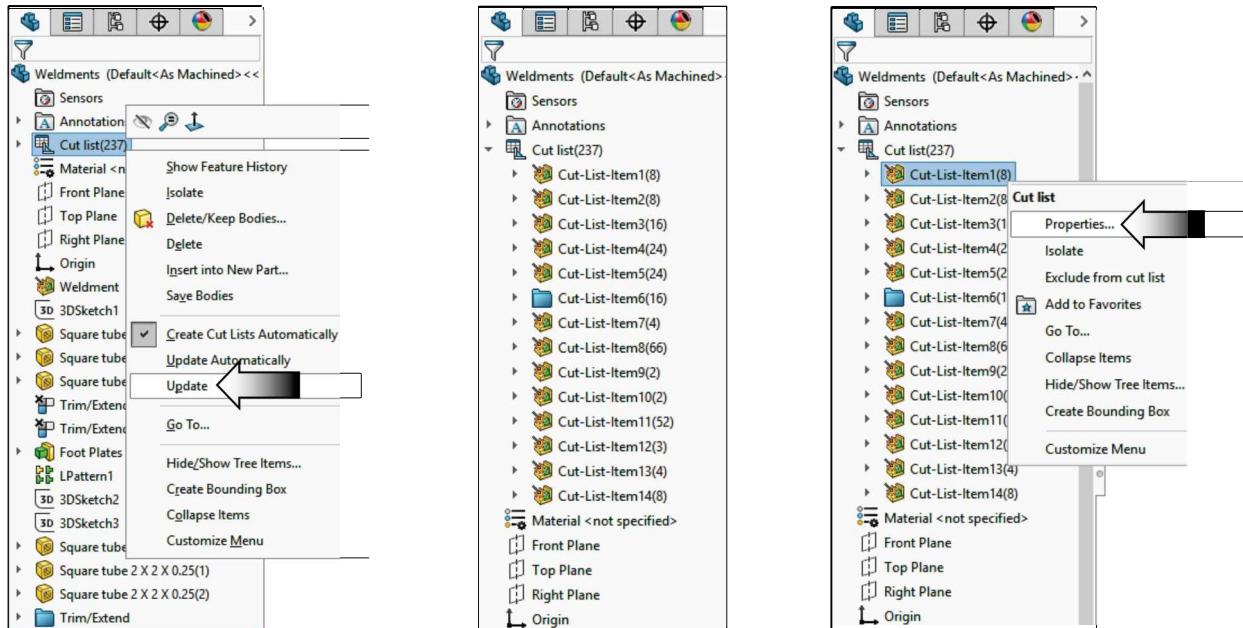
10. Updating the Cut list:

SOLIDWORKS updates the cut list automatically when new features are added, such as: extruded foot plates or 3D bounding boxes, edit existing features, or rebuilds the model. The model's custom properties and internal supporting data are also updated, preventing custom property errors.

This Icon in front of the Cut List  indicates the Cut List needs updating.

This Icon in front of the Cut List  indicates the Cut List is up to date.

Right-click the **Cut List** icon and select **Update** (arrow). The Cut List items that were created as groups are automatically sorted and placed in separate folders.



To view the Cut List details, right-click on one of the Cut List Items, and select: **Properties** (arrow).

Select one of the items such as Angle, Length, Description, etc. to see the details of the item.

Cut-List Properties

Cut List Summary				
Properties Summary				
Cut List Table				
ANGLE1 ANGLE2 DESCRIPTION LENGTH	Type	Value / Text Expression	Evaluated Value	
MATERIAL	Text	<Not Specified>	<Not Specified>	
QUANTITY	Text	<Not Specified>	<Not Specified>	
TOTAL LENGTH	Text	<Not Specified>	<Not Specified>	
4 Cut-List-Item5	Text	<Not Specified>	<Not Specified>	
5 Cut-List-Item6	Text	<Not Specified>	<Not Specified>	
6 Cut-List-Item7	Text	<Not Specified>	<Not Specified>	
7 Cut-List-Item8	Text	<Not Specified>	<Not Specified>	
8 Cut-List-Item10	Text	<Not Specified>	<Not Specified>	
9 TUBE, SQUARE 2.00	Text	'LENGTH@@@TUBE, SQUARE 2.00 X 2.00 X .2	35	
10 TUBE, SQUARE 2.00	Text	'LENGTH@@@TUBE, SQUARE 2.00 X 2.00 X .2	42	
11 TUBE, SQUARE 2.00	Text	'LENGTH@@@TUBE, SQUARE 2.00 X 2.00 X .2	16	
12 TUBE, SQUARE 2.00	Text	'LENGTH@@@TUBE, SQUARE 2.00 X 2.00 X .2	48.66	
13 TUBE, SQUARE 2.00	Text	'LENGTH@@@TUBE, SQUARE 2.00 X 2.00 X .2	48.66	
14 TUBE, SQUARE 2.00	Text	'LENGTH@@@TUBE, SQUARE 2.00 X 2.00 X .2	254	
15 TUBE, SQUARE 2.00	Text	'LENGTH@@@TUBE, SQUARE 2.00 X 2.00 X .2	36	
16 TUBE, SQUARE 2.00	Text	'LENGTH@@@TUBE, SQUARE 2.00 X 2.00 X .2	8	
17 TUBE, SQUARE 2.00	Text	'LENGTH@@@TUBE, SQUARE 2.00 X 2.00 X .2	7	

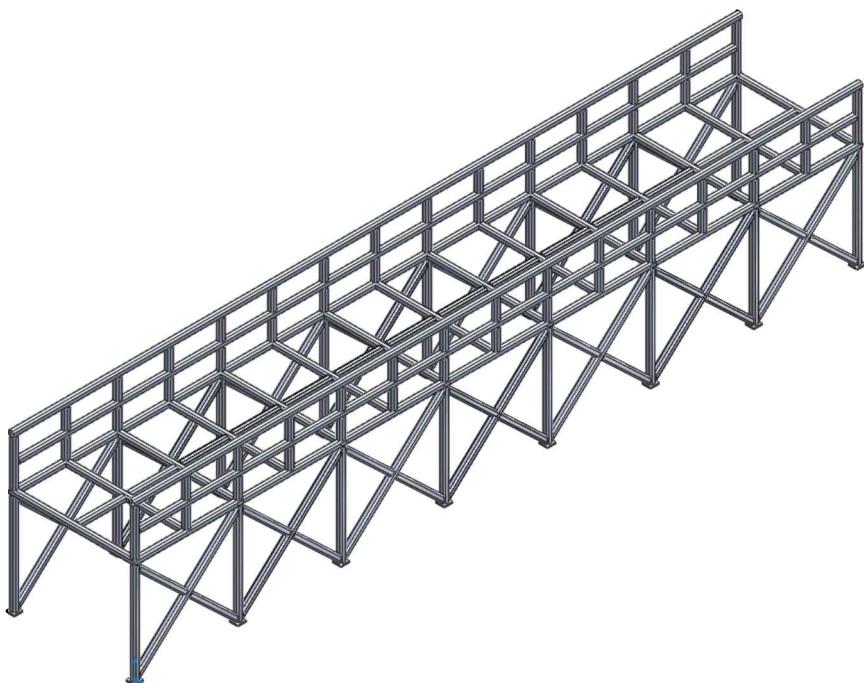
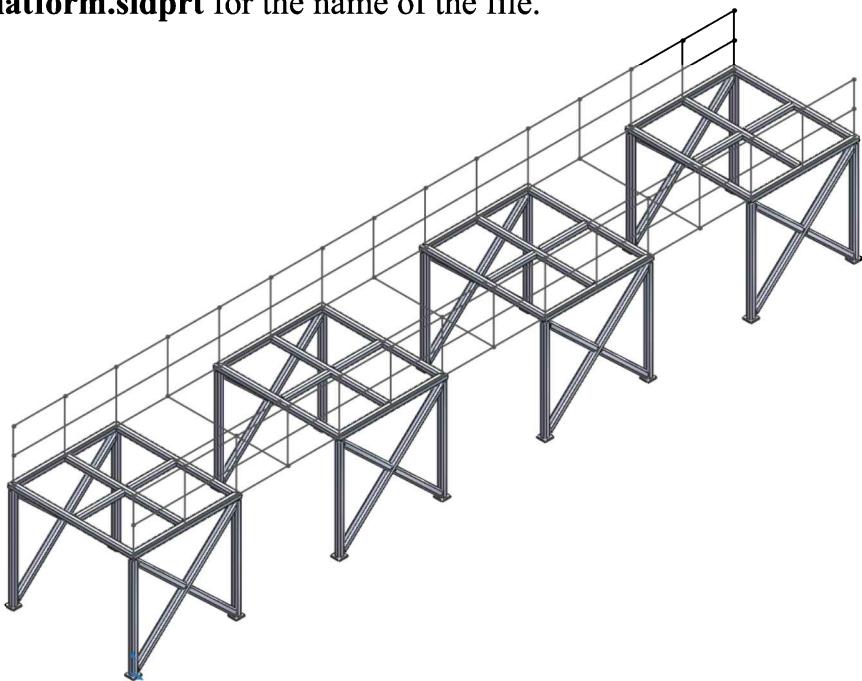
Click **OK** to exit the Cut List Properties.

11. Saving your work:

Select File / Save As.

Enter **Weldments Platform.sldprt** for the name of the file.

Click **Save**.

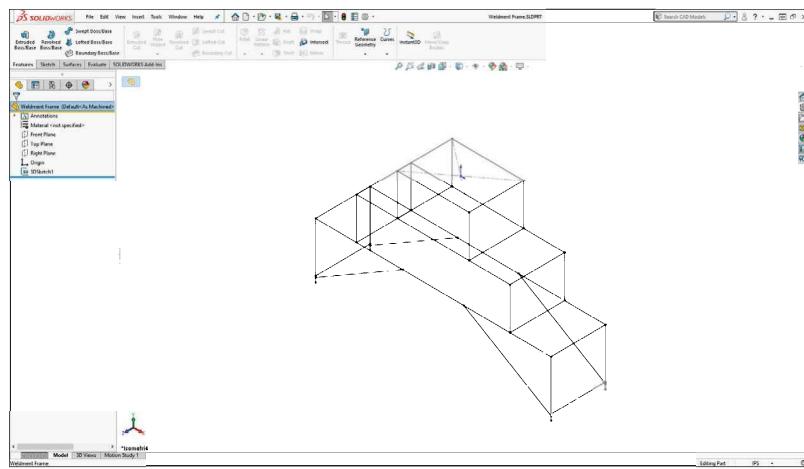


Using Weldments – Structural Members

The options in Weldments allow you to develop a weldment structure as a single multibody part. The basic framework is defined using 2D or a 3D sketch, and then structural members like square or round Tubes are added by sweeping the tube profile along the framework. Gussets, end caps, and weld-beads can also be added using the tools on the **Weldments** toolbar.

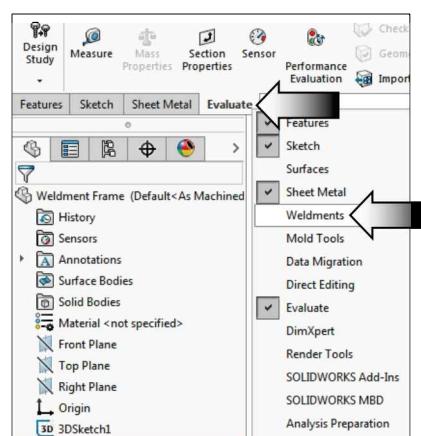
1. Opening an existing document:

Open a file named **Weldment Frame** from the Training Files folder.



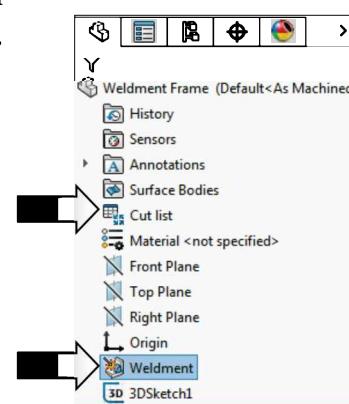
2. Enabling the Weldments toolbar:

Right-click one of the tool tabs and select the **Weldments** toolbar from the list (arrow).



Click the **Weldments** button from the Weldments toolbar.

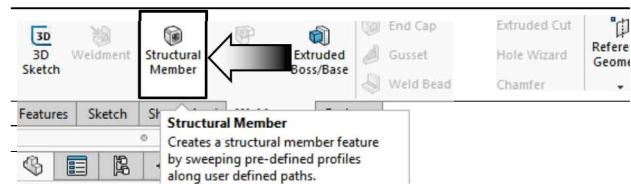
A **Weldment Feature** appears on the feature tree with a **Weldment Cut List** (arrows), which indicates the items from the model to include in this cut list.



A single 3D Sketch is created for the purpose of this exercise. Multiple sketches (2D & 3D) can be used to design the weldments structural members.

3. Adding Structural Members:

Click the **Structural Member** button from the Weldments toolbar.



Select the following:

- * **Ansi Inch.**
- * **Square Tube**
- * **4 X 4 X 0.25**

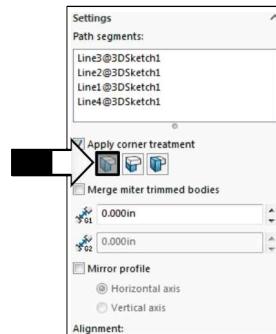
Click the **4 lines** on the top of the frame.



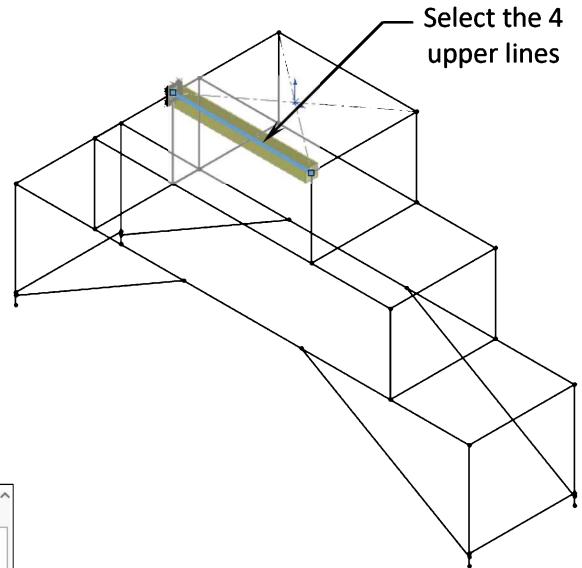
Select the **MITER** under Apply Corner-Treatment.

Click **OK**.

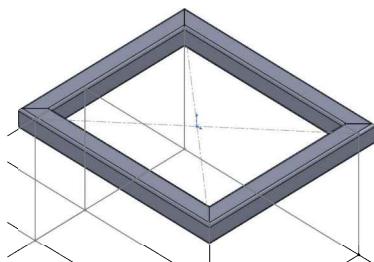
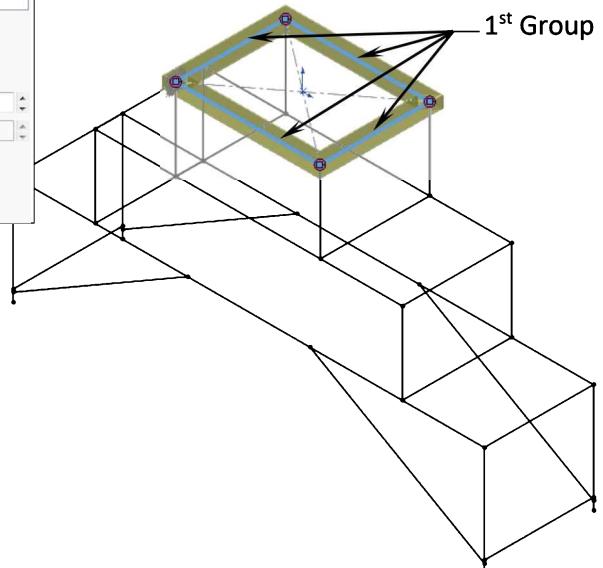
By default, the profile of the tube is automatically centered on the end of each line.



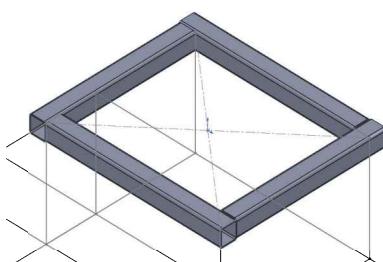
Try out all 3 options for corner treatments: End Miter, End Butt1, and End Butt2.



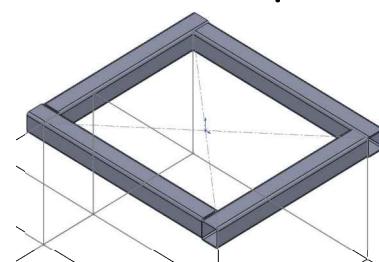
Switch back to the **End Miter** option when finished.



End Miter



End Butt1
(Overlapped)



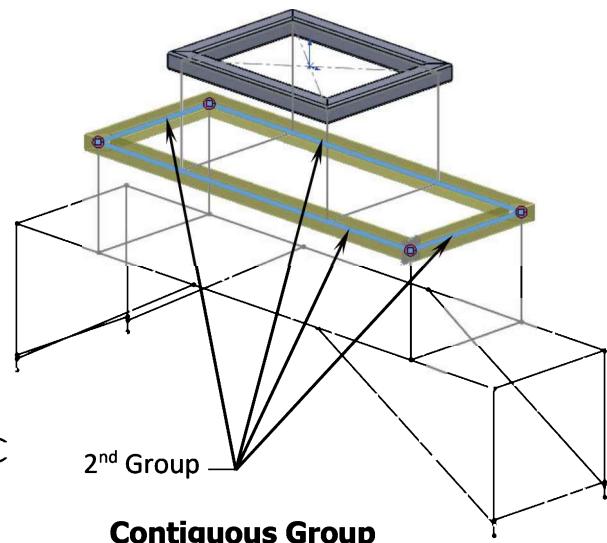
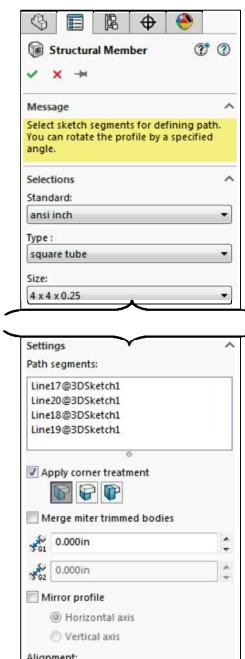
End Butt2
(Under-lapped)

4. Adding Structural Members to Contiguous Groups*:

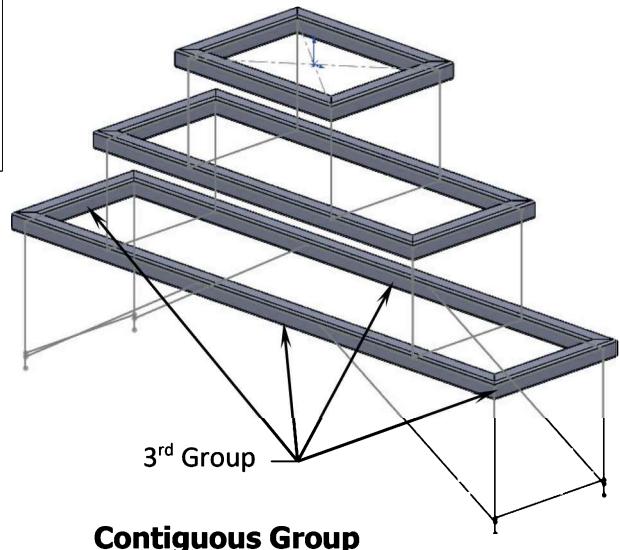
Repeat the previous step and add another **4 square tubes** to the **2nd group** as shown.

Use these same settings:

- * **Ansi Inch.**
- * **Square Tube**
- * **4 X 4 X 0.25**



* *A group is a collection of related segments in a structural member. There are 2 types of groups, one is called Contiguous, where a continuous contour of segments is joined end-to-end. The other is called Parallel, which includes a discontinuous collection of parallel segments. Segments in the group cannot touch each other.*

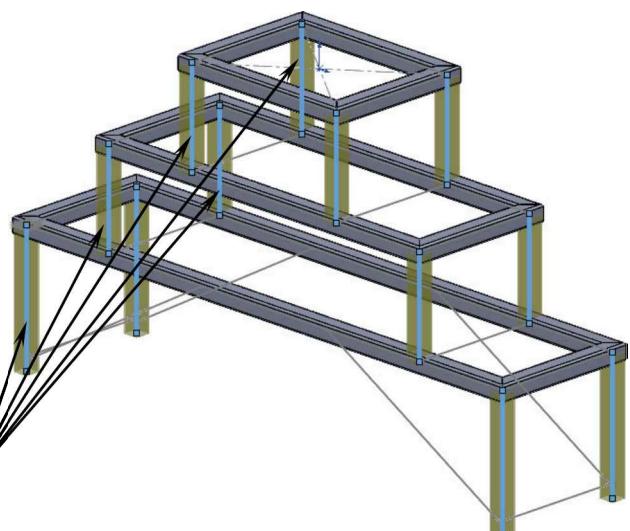


Repeat the same step for the 3rd group.

Follow the same procedure and add the same size tubing to the vertical members as noted for the 4th group.

Note: Select the exact same vertical tubes on both sides (total of 12).

4th Group
(both sides)



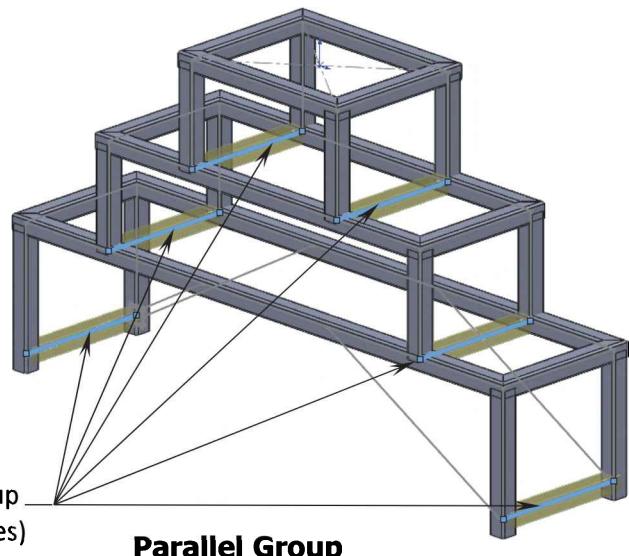
5. Adding Structural Members to the Parallel Groups:

Repeat the previous step and add the same structural members to the 5th group as indicated.

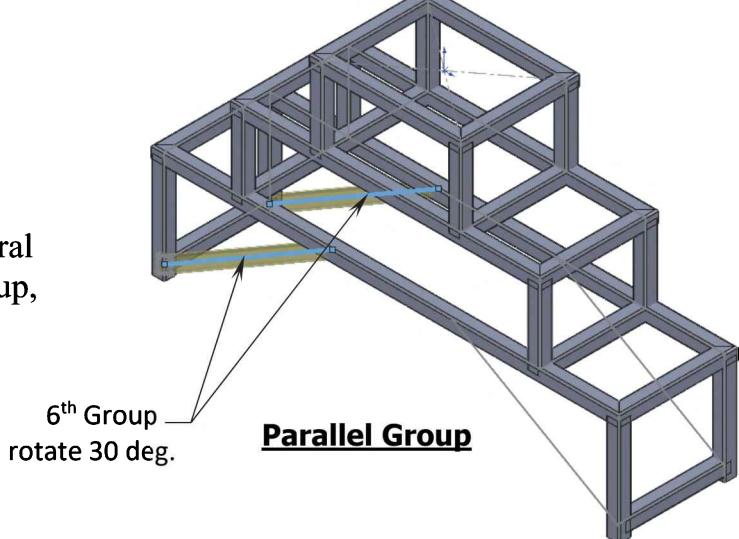


Groups

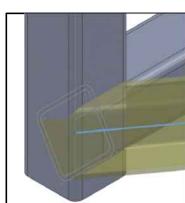
You can define a group in a single plane or in multiple planes. A 3D sketch is best suited for weldment designs since all entities can be drawn and controlled in the same sketch.



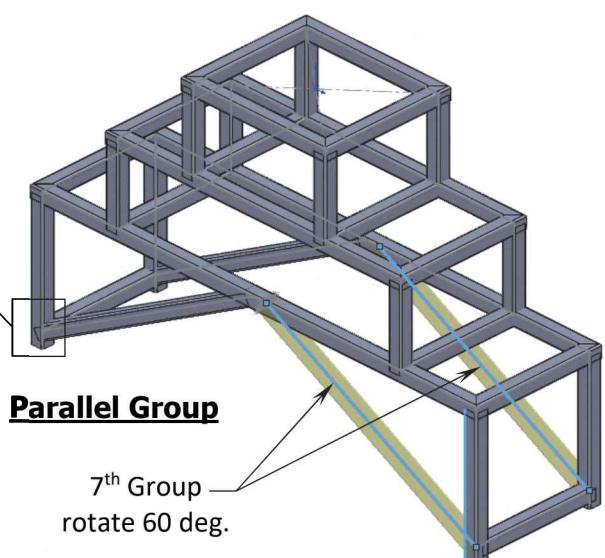
Create the same type of structural members for the 6th and 7th group, which has only 2 lines in each group...



Rotate the profile to **30 deg.** for the 6th group and **60 deg.** for the 7th group.

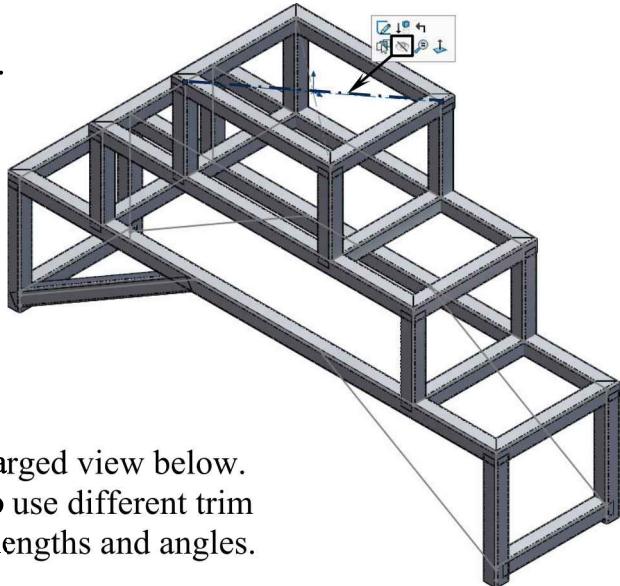


There are several over-lapped areas that need trimming; we will look into that in the next steps.

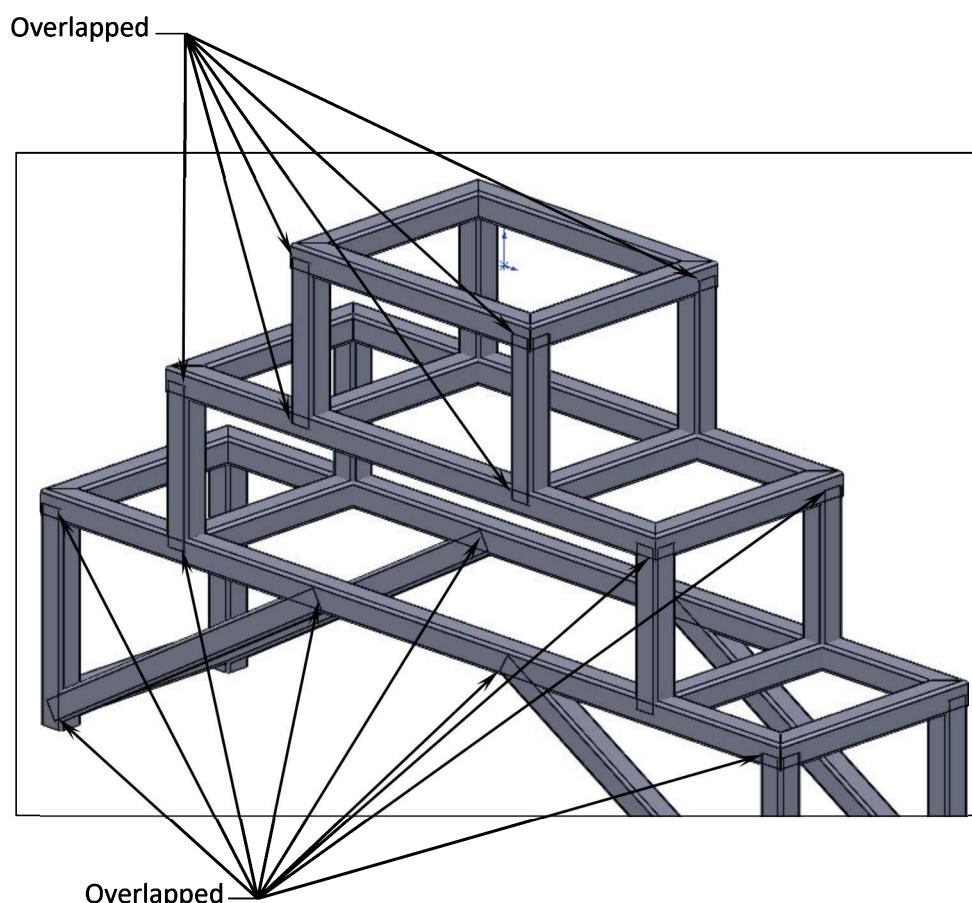


6. Hiding the 3D sketch:

Right-click on one of the lines in the 3D-sketch and select **Hide** .



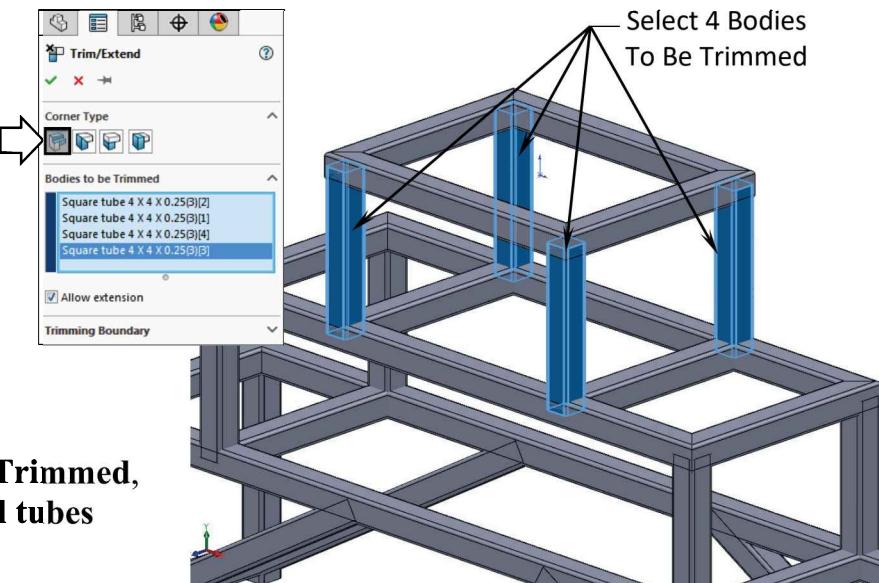
Notice the intersected areas in the enlarged view below. For practice purposes, we will learn to use different trim options to cut the tubes to their exact lengths and angles.



7. Trimming the Structural Members:

Click Trim/Extend  on the Weldments toolbar.

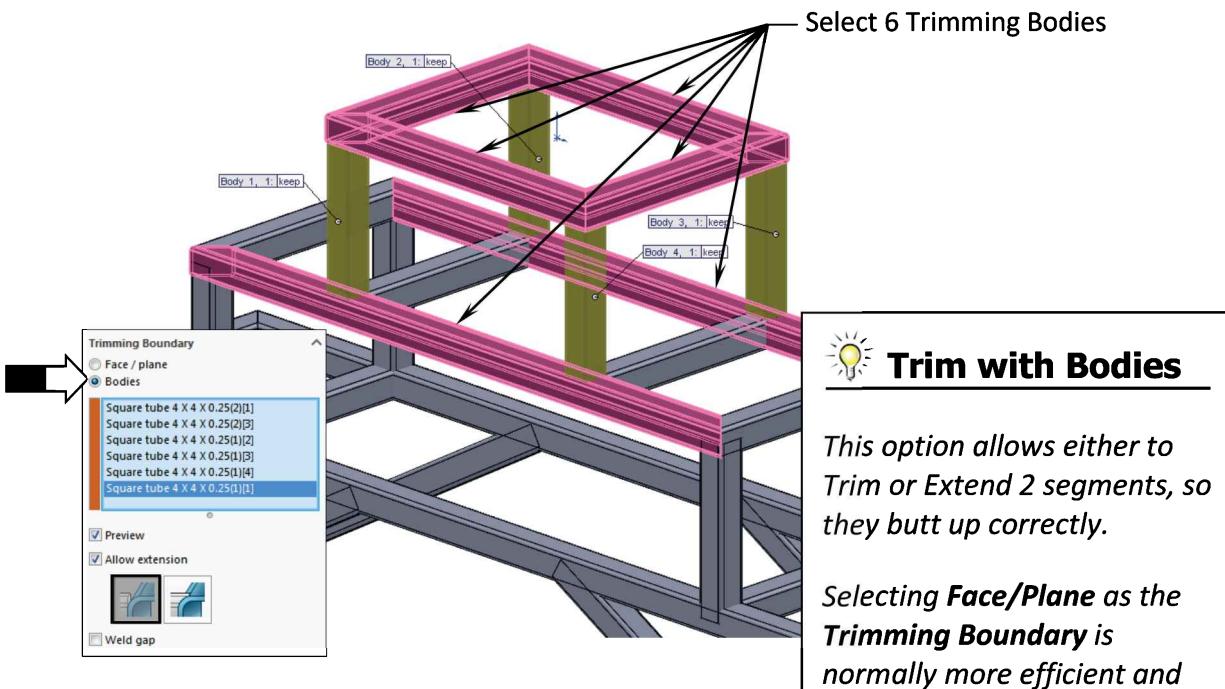
For Corner Type, use the default End Trim (arrow).



For Bodies To Be Trimmed, select the 4 vertical tubes as noted.

For Trimming Boundary, select the Body option (arrow).

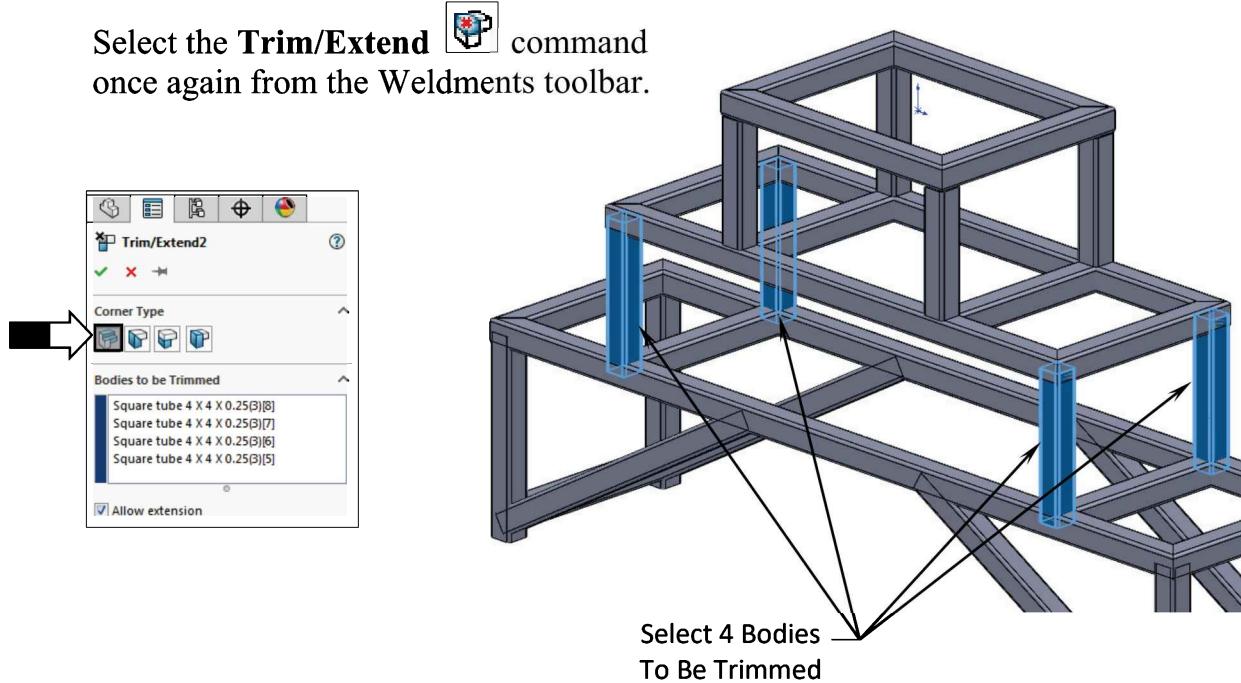
For Trimming Bodies, select the 6 horizontal tubes as indicated.



Click OK.

8. Trimming the Parallel Groups:

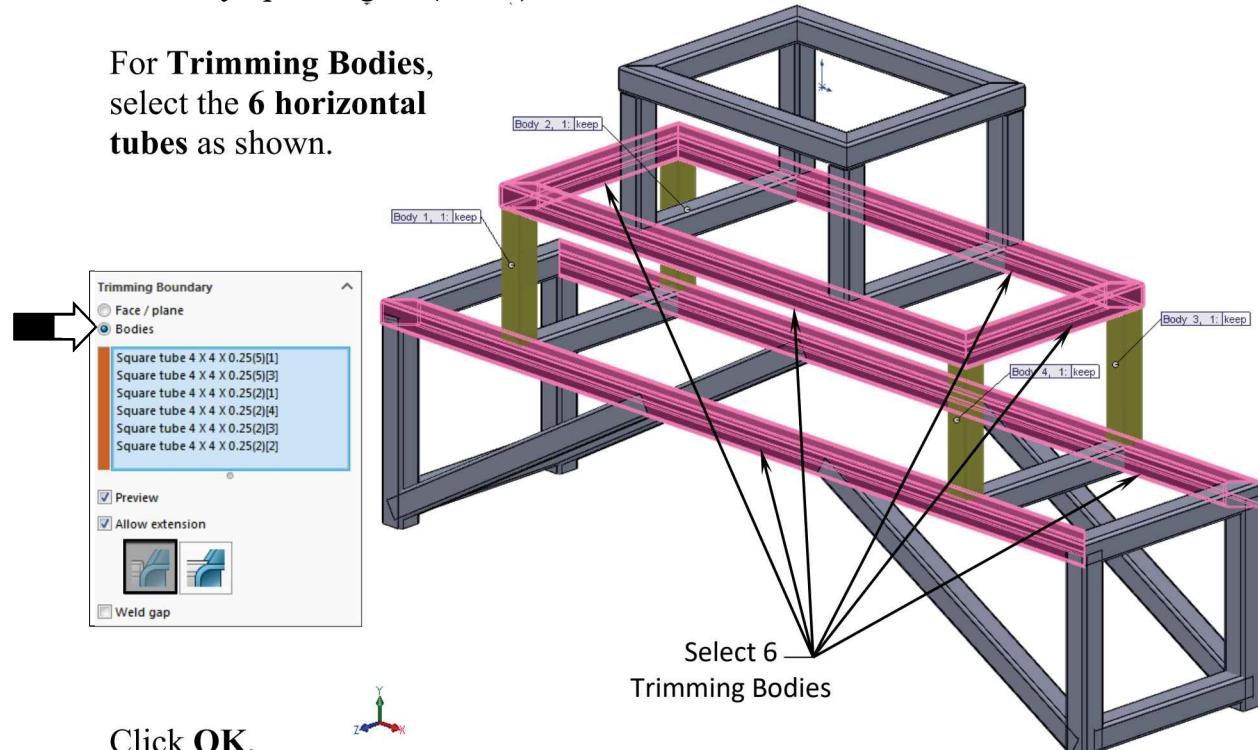
Select the Trim/Extend  command once again from the Weldments toolbar.



For **Bodies To Be Trimmed**, select the next 4 vertical tubes as noted.

For **Trimming Boundary**, select the **Body** option again (arrow).

For **Trimming Bodies**, select the **6 horizontal tubes** as shown.



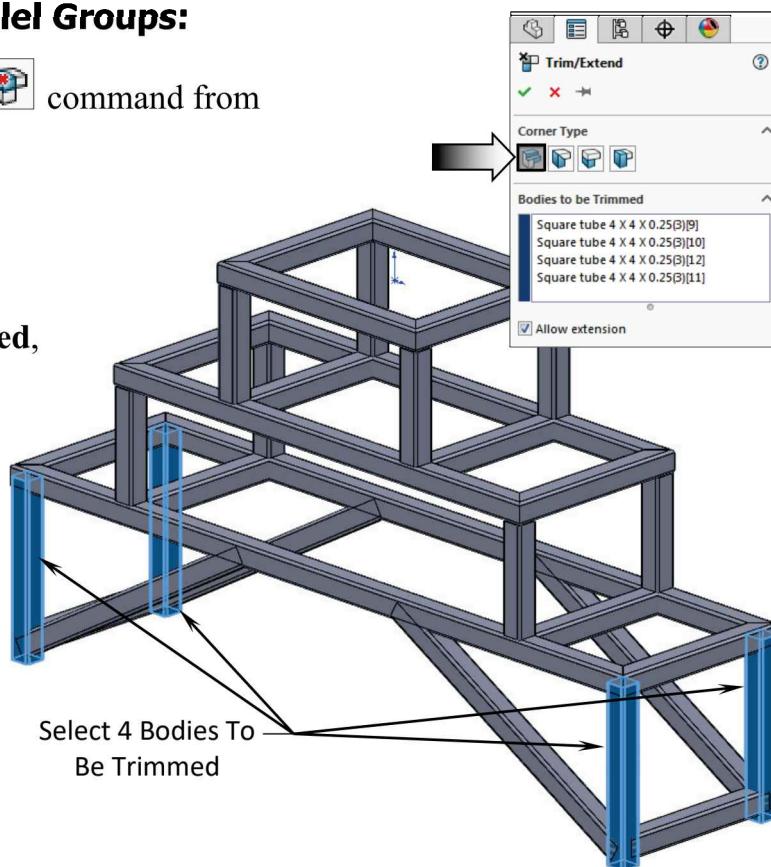
Click **OK**.

9. Trimming the next Parallel Groups:

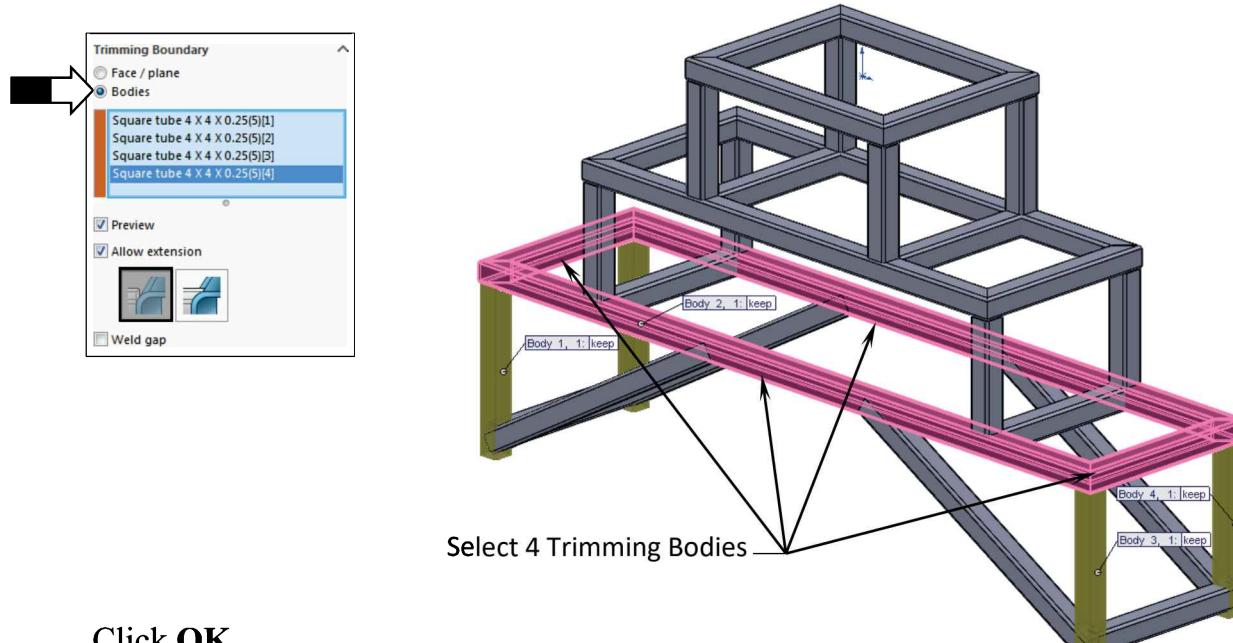
Select the Trim/Extend  command from the Weldments toolbar.

For Bodies To Be Trimmed, select the 4 vertical tubes on the bottom as noted.

For Trimming Boundary, click the Body option (arrow).



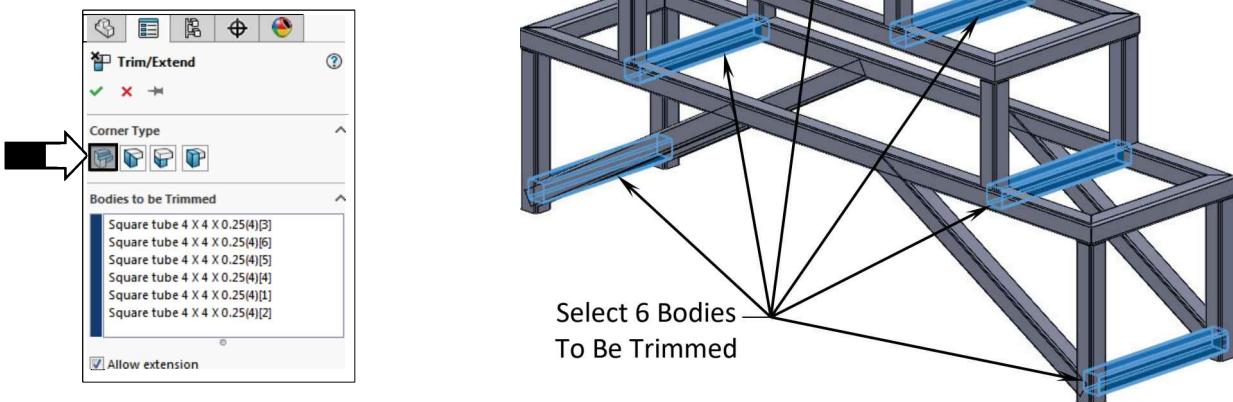
For Trimming Bodies, select the 4 horizontal tubes as shown.



Click OK.

10. More Trimming:

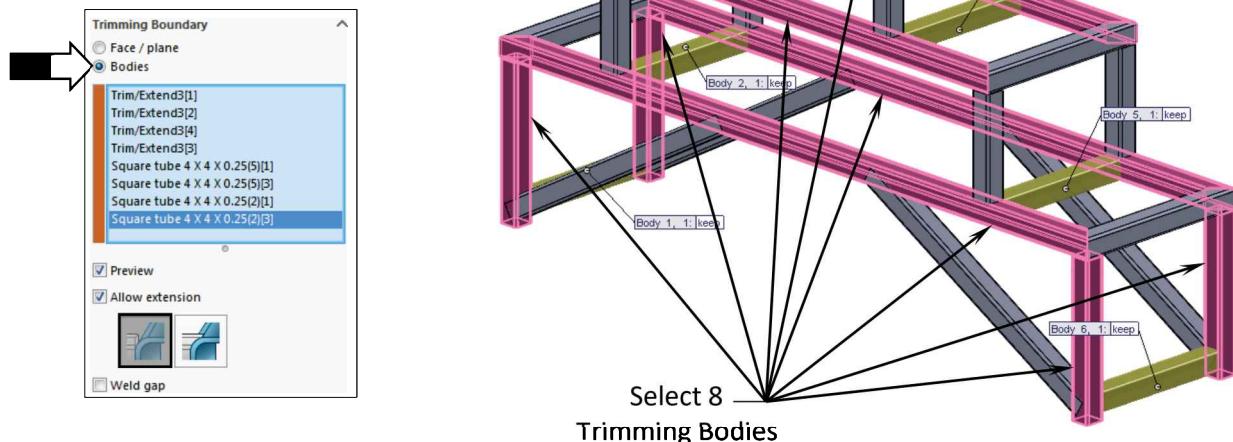
Select the **Trim/Extend**  command from the Weldments toolbar.



For **Bodies To Be Trimmed** select the **6 short horizontal tubes** as indicated.

For **Trimming Boundary**, click the **Body** option (Arrow).

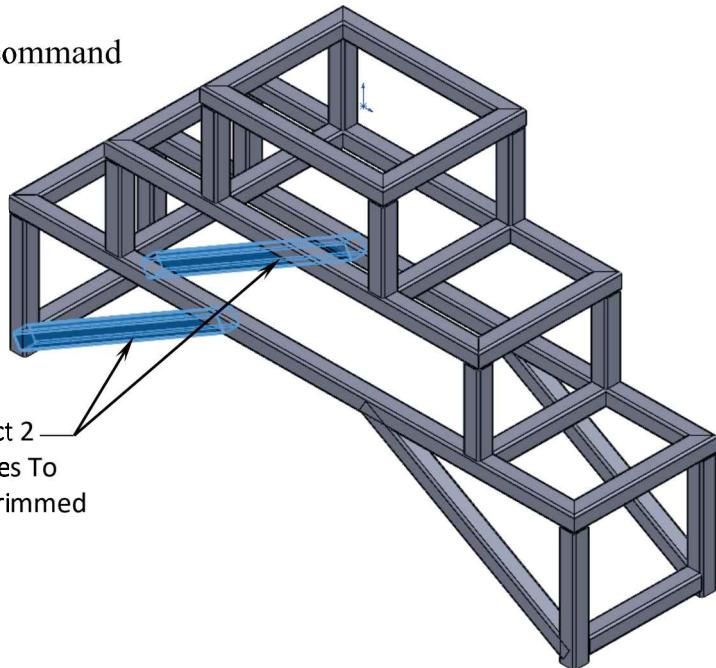
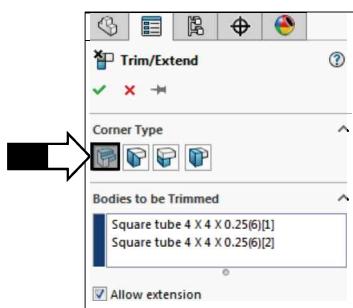
For **Trimming Bodies**, select the **8 structural members** as indicated.



Click **OK**.

11. Trimming with Face/Plane:

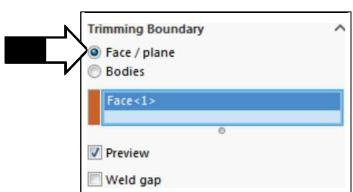
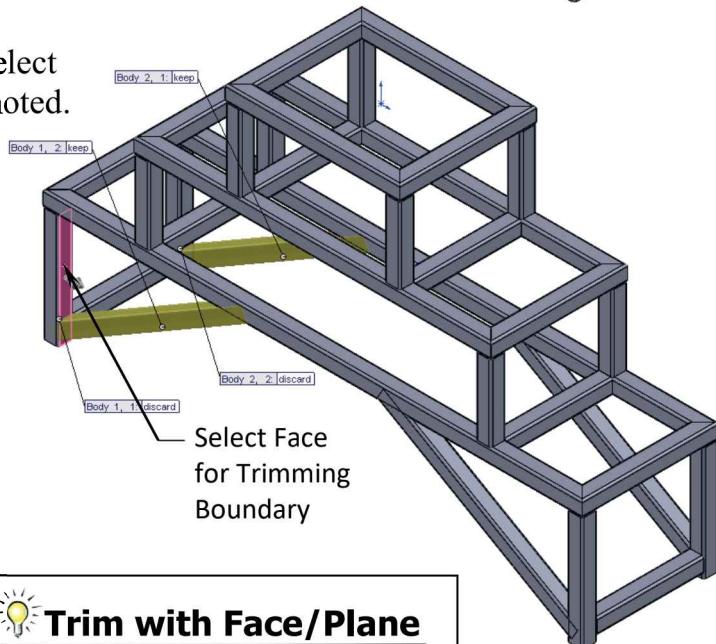
Select the **Trim/Extend**  command from the Weldments toolbar.



For **Bodies To Be Trimmed** select the 2 structural members as noted.

For **Trimming Boundary**, click the **Face / Plane** option (arrow).

Select the **planar surface** as noted, for **Trimming Bodies**.



Click **OK**.



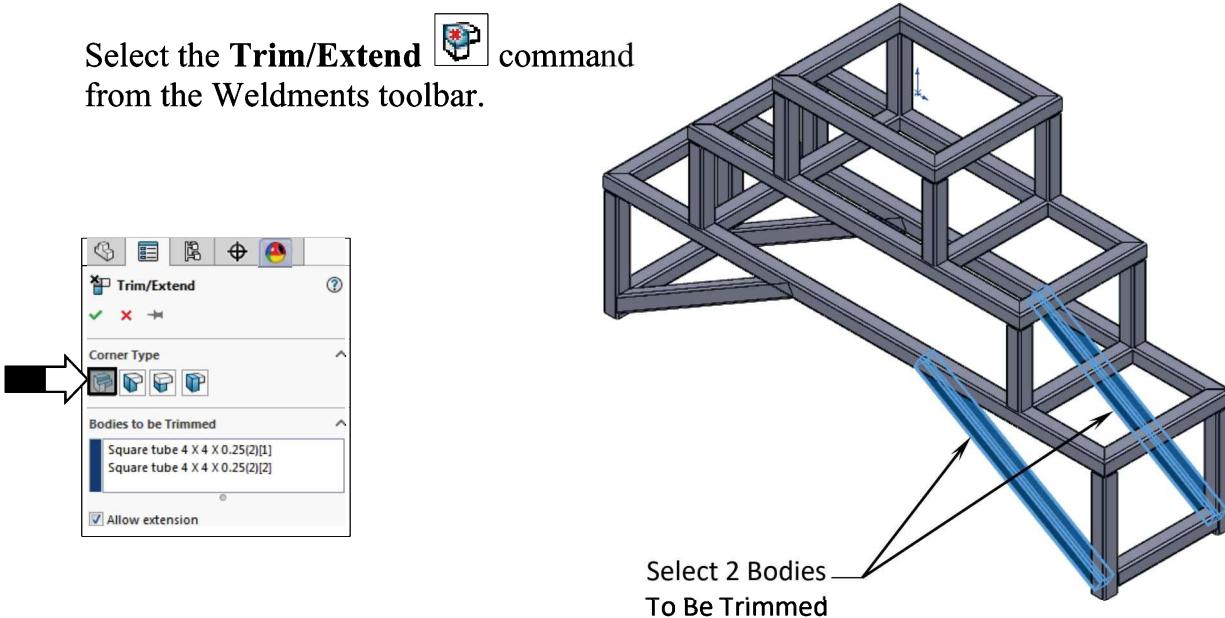
Trim with Face/Plane

This option allows a planar face(s) as a trimming boundary to trim one or more solid bodies.

*Selecting **Face/Plane** as the **Trimming Boundary** is normally more efficient and offers better performance.*

12. More Trimming with Face/Plane:

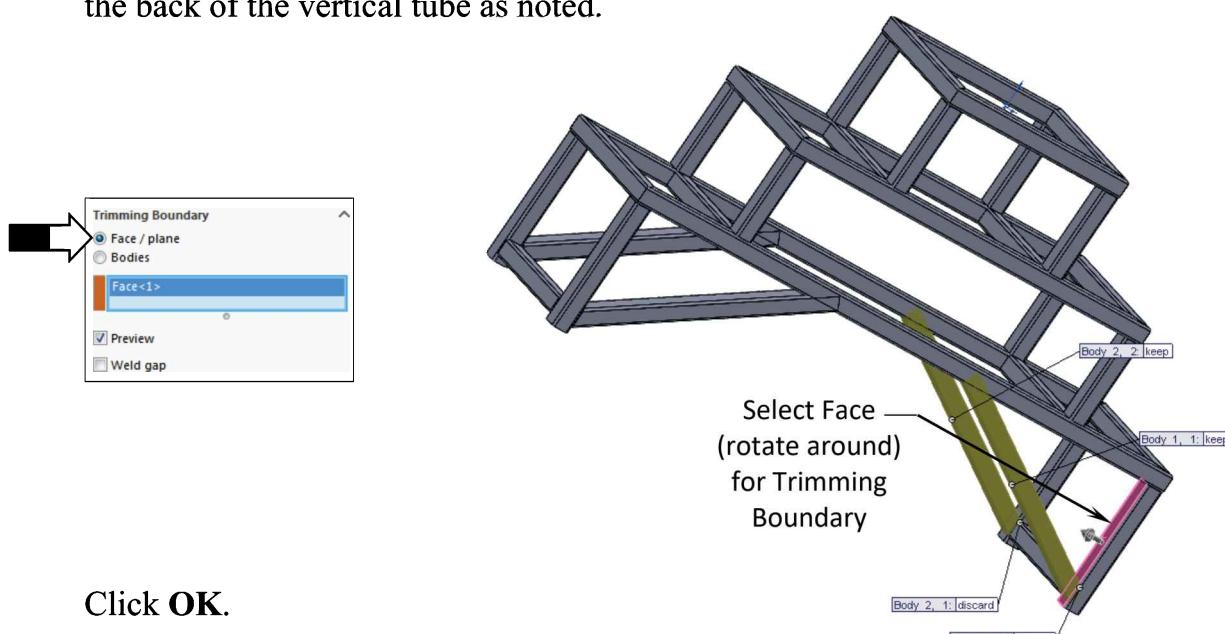
Select the **Trim/Extend**  command from the Weldments toolbar.



For **Bodies To Be Trimmed** select the **2 structural members** as indicated.

For **Trimming Boundary**, click the **Face / Plane** option (Arrow).

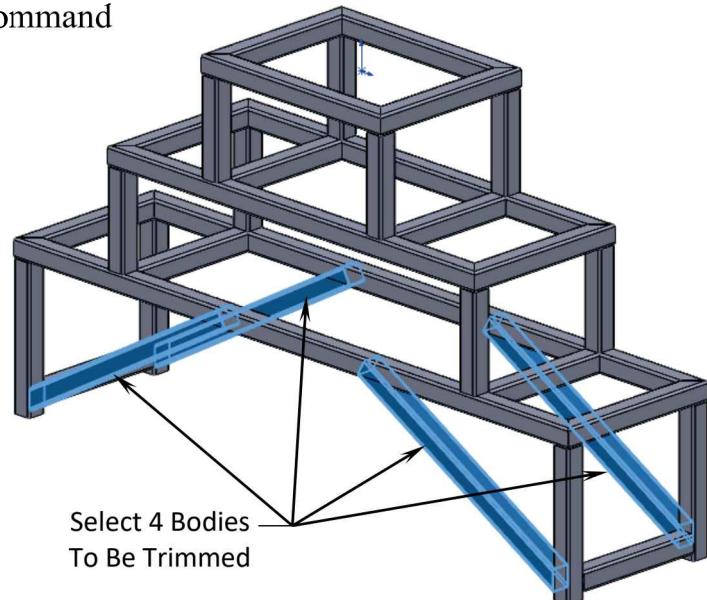
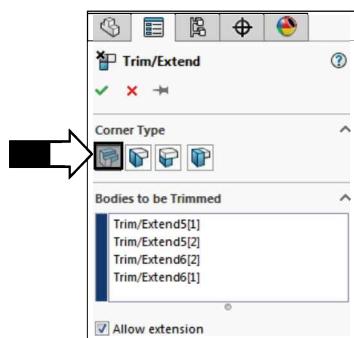
For **Trimming Bodies**, select the **planar surface** on the back of the vertical tube as noted.



Click **OK**.

13. Trimming the last 4 structural members:

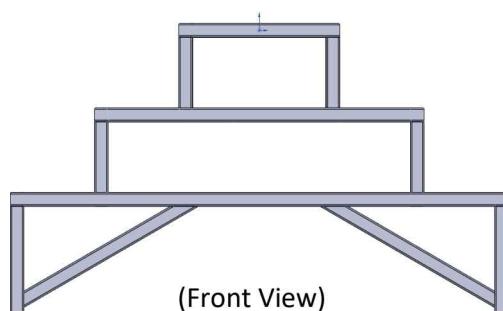
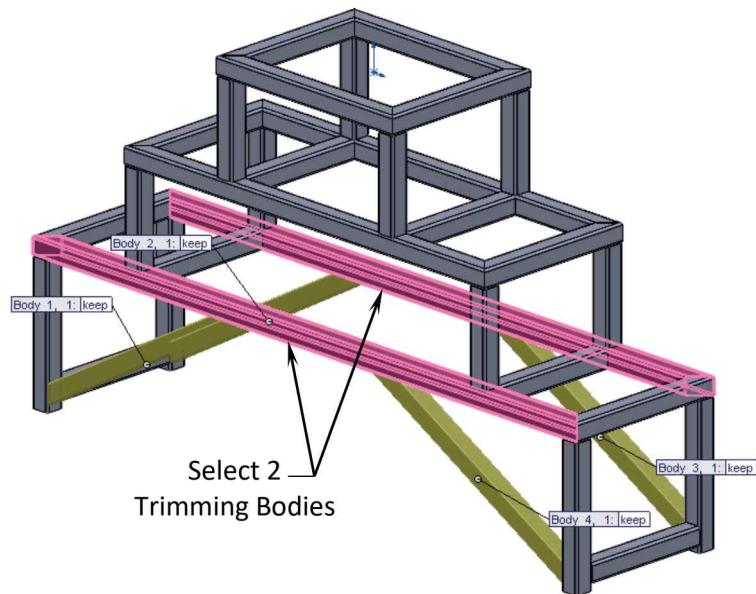
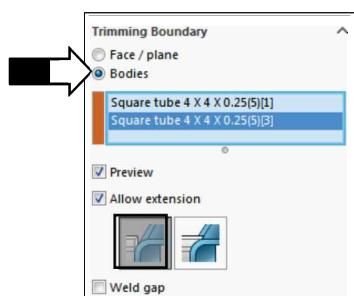
Select the Trim/Extend  command from the weldment toolbar.



For **Bodies To Be Trimmed** select the **4 structural members** as shown.

For **Trimming Boundary**, click the **Body** option (arrow).

For **Trimming Bodies**, select the **2 horizontal tubes** as noted.



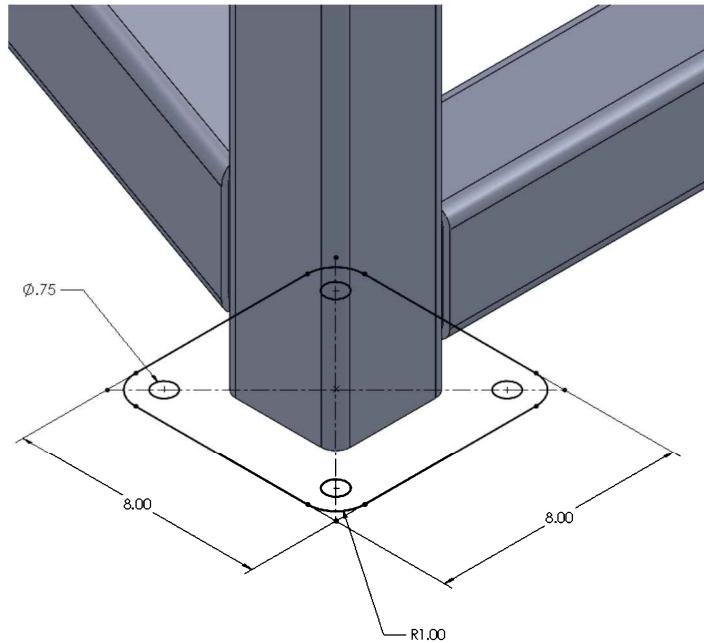
Click **OK**.

14. Adding the foot pads:

Insert a new sketch on the **bottom surface** of one of the 4 legs.

Sketch the profile as shown.

The 4 circles are **concentric** with the corner radii.

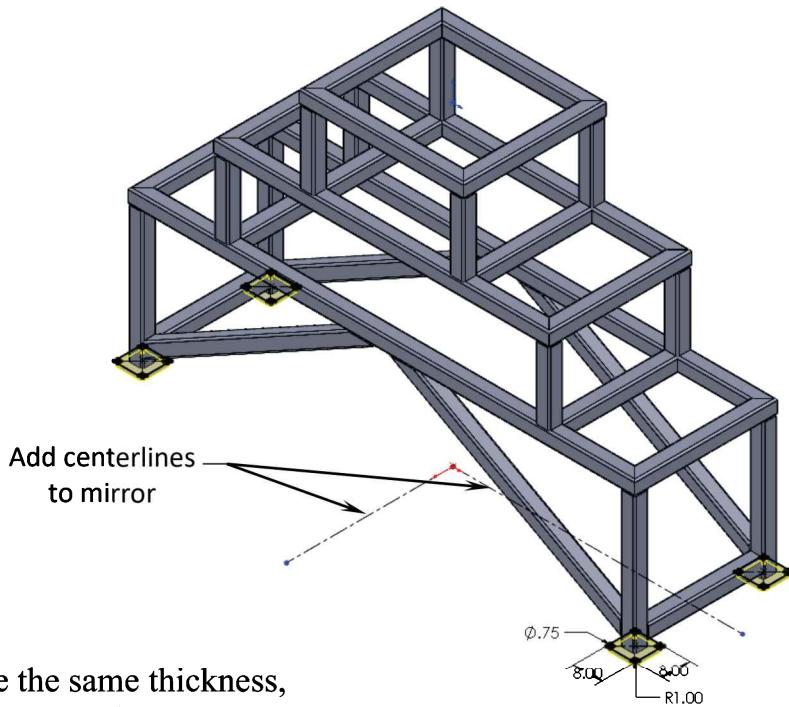


Add the dimensions and relations needed to fully define the sketch.

Mirror the sketch to make a total of 4 foot pads.

Note:

Add a couple of *centerlines* as shown prior to making the mirror.



Since the 4 foot pads have the same thickness, we can extrude them at the same time.

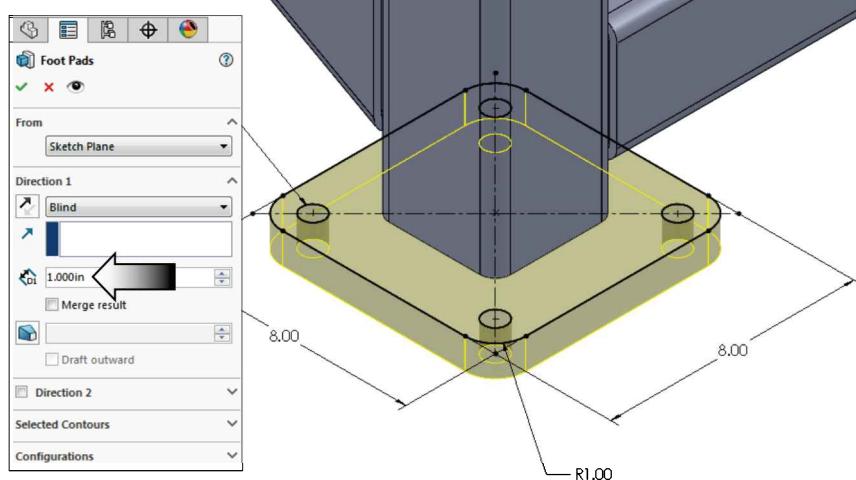
Click **Extruded Boss/Base**.

Enter the following:

- * Type: **Blind**

- * Depth: **1.000**

Click **OK**.



15. Adding the Gussets:

Rotate and zoom to an orientation that looks similar to the view below.

Click the **Gusset**  command.

Enter the following:

For **Supporting Faces**, select the **2 faces** as noted.

- * Distance1: **4.00 in.**

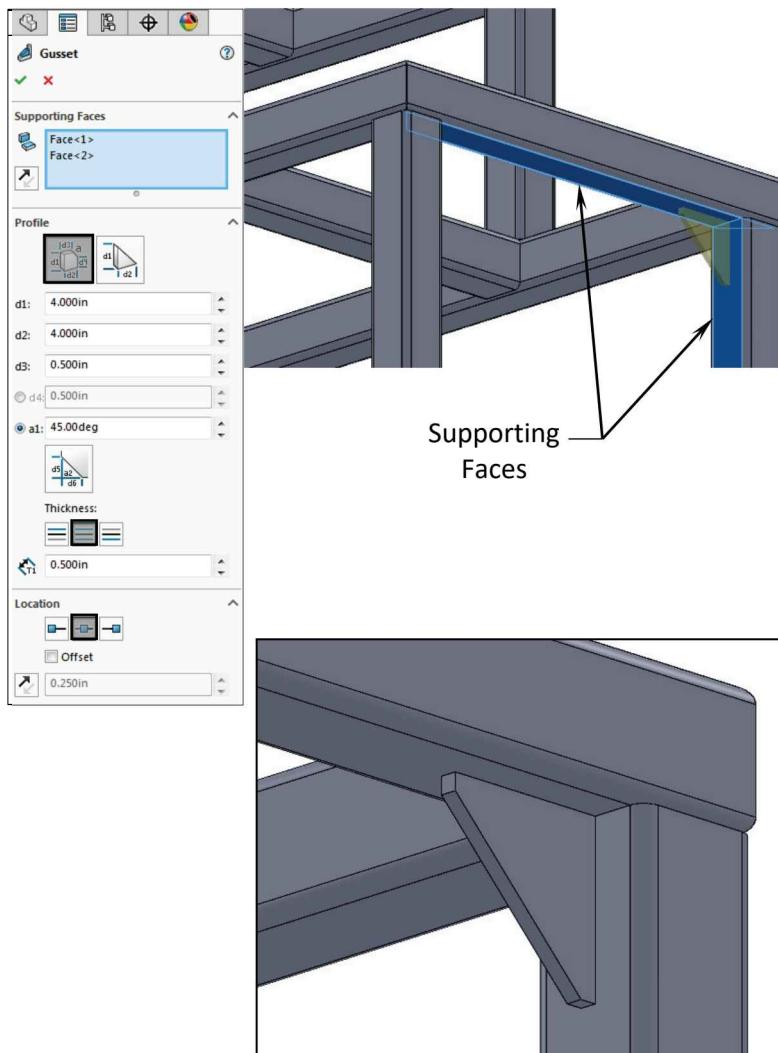
- * Distance2: **4.00 in.**

- * Distance3: **.500 in.**

- * Thickness: **.500 in.**
(Both Sides)

- * Location: **Midpoint**

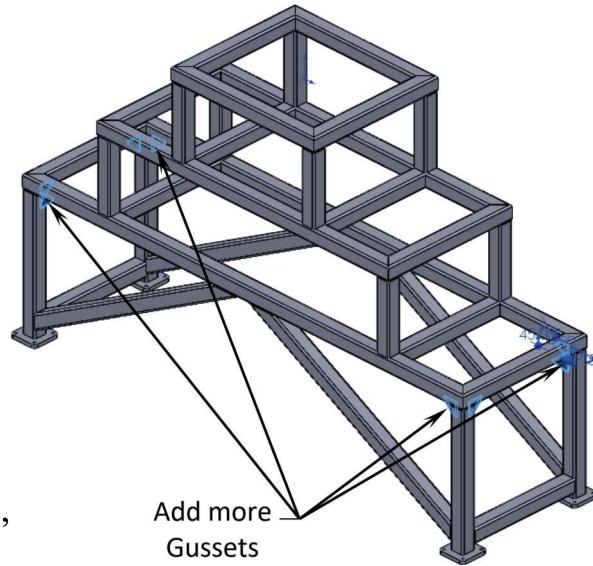
Click **OK**.



16. Adding more Gussets:

Repeat the step 15 and add a gusset to each corner of the frame.

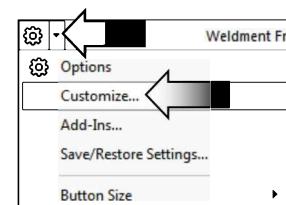
Next, we are going to add the weld beads around the gussets. Weld beads can be added as full length, intermittent, or staggered fillet weld beads between any intersecting weldment entities such as structural members, plate weldments, or gussets.



17. Adding the Fillet-Bead icon to the Weldments toolbar:

The **Fillet Bead** icon  needs to be added to the Weldments toolbar.

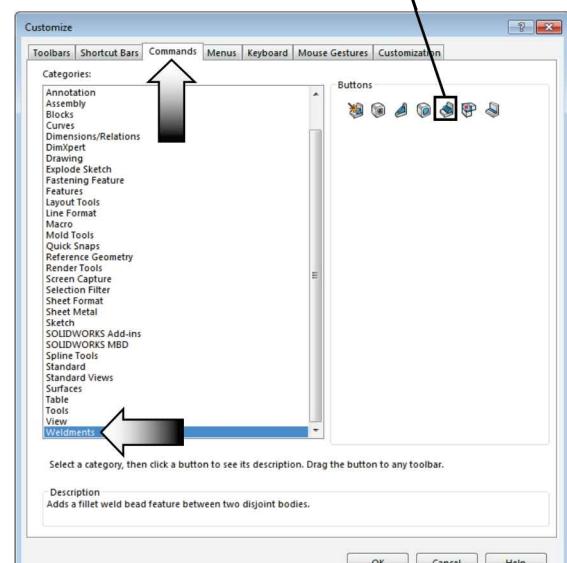
To add the missing icon:



Select **Options / Customize** (arrows).



Click the **Commands** tab (arrow).



Select the **Weldments** option under **Categories** (arrow).

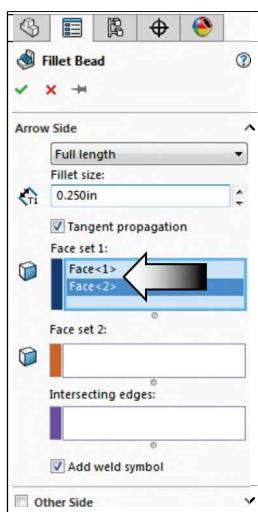
Drag the **Fillet Bead** icon and drop it onto the Weldments toolbar as noted.

Click **OK** to close the Customize dialog.

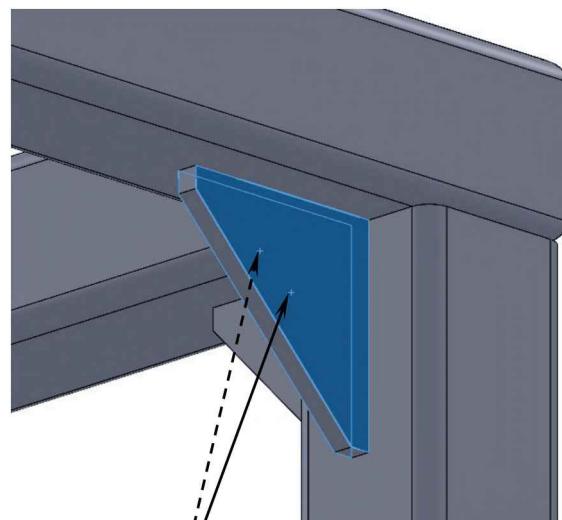
18. Adding the Fillet Beads:

Click **Fillet Bead**  on the Weldments toolbar.

From the Weld Bead properties tree, enter the following:

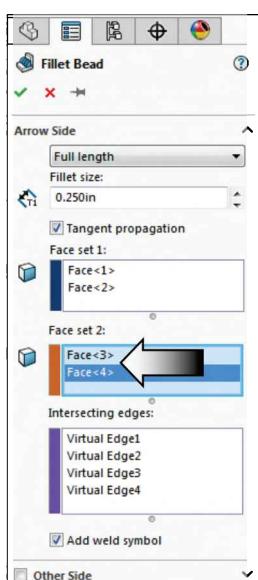


- * **Bead Type: Full Length**
- * **Fillet Size: .250 in.**
- * **Tangent Prop: Enabled**
- * **Face Set1: Select the 2 faces as noted.**



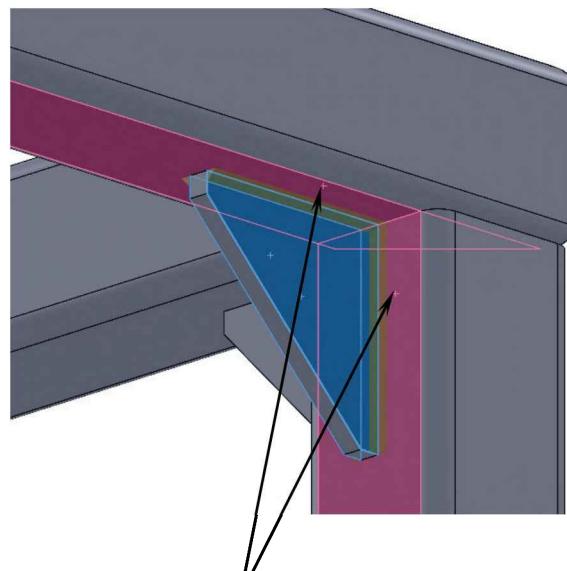
Select 2 Faces (front & back) for Face Set1

For **Face Set2**, select the next **2 faces** as indicated.



Intersecting Edges:
Highlights edges where Face Set1 and Face-Set2 intersects.

(You can also right-click an edge and select **Delete** to remove from the weld bead.)

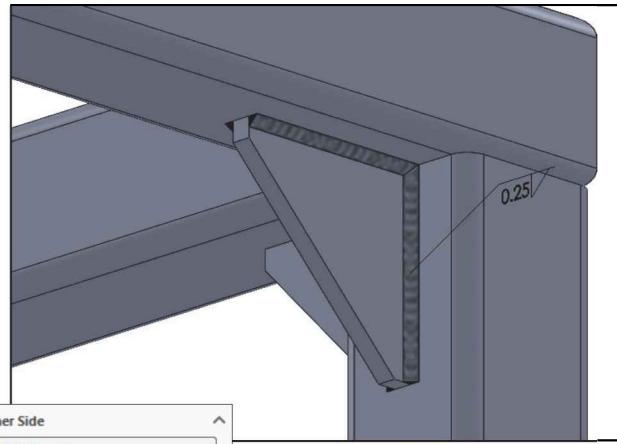
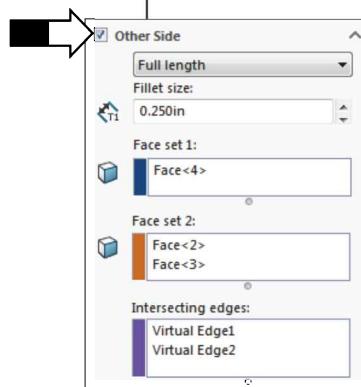


Select 2 Faces

Enable the **Other Side** check-box and apply the same settings to the back end of the gusset.

Different bead type or fillet size can be added to the other side, but we are going to use the same settings as the first side.

Click **OK**.

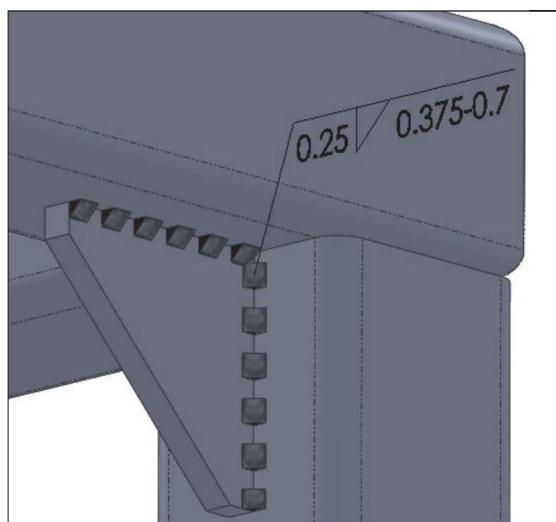


A bead call out is added automatically (see example below).

Example: 0.25 = Length of the leg of the fillet bead.

0.375 = Length of each bead segment.

0.7 = Distance between the start of each bead.

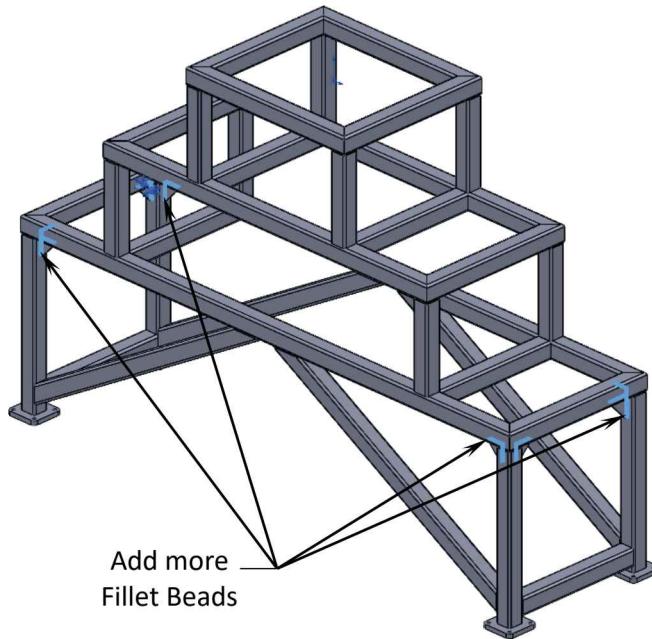


Intermittent or Staggered

19. Adding more Fillet Beads:

Repeat step 17 and add a set of fillet beads to each gusset that was created earlier.

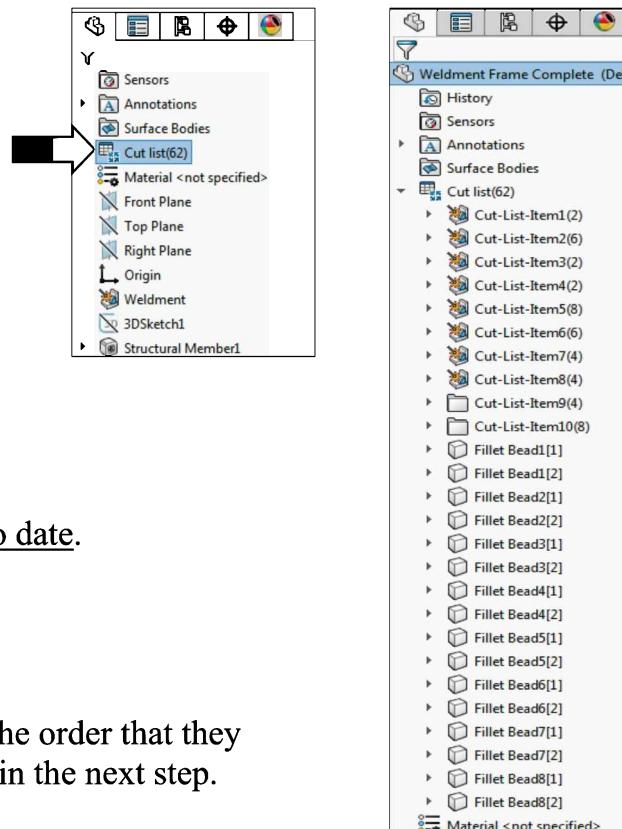
When adding the fillet beads, try using the different types of beads: Full Length, Intermittent, and Staggered to see the different results and callouts.



20. Viewing the Weldment Cut List:

Locate the **Cut List** on the FeatureManager tree and click the Plus (+) sign to expand it.

The Cut List needs to be updated every time something is added to the model.



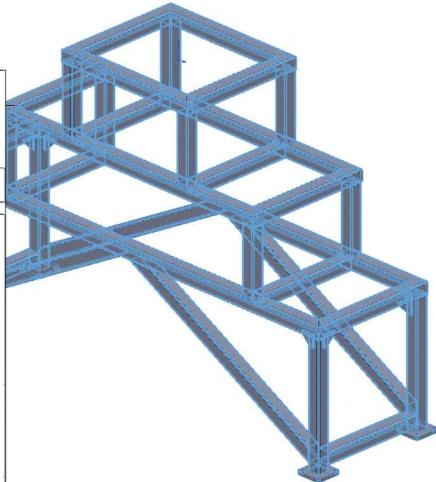
The icon in front of the cut list indicates that it needs updating and the icon indicates the list is up to date.

The current list displays all items in the order that they were created. We will update the list in the next step.

21. Updating the Cut List:

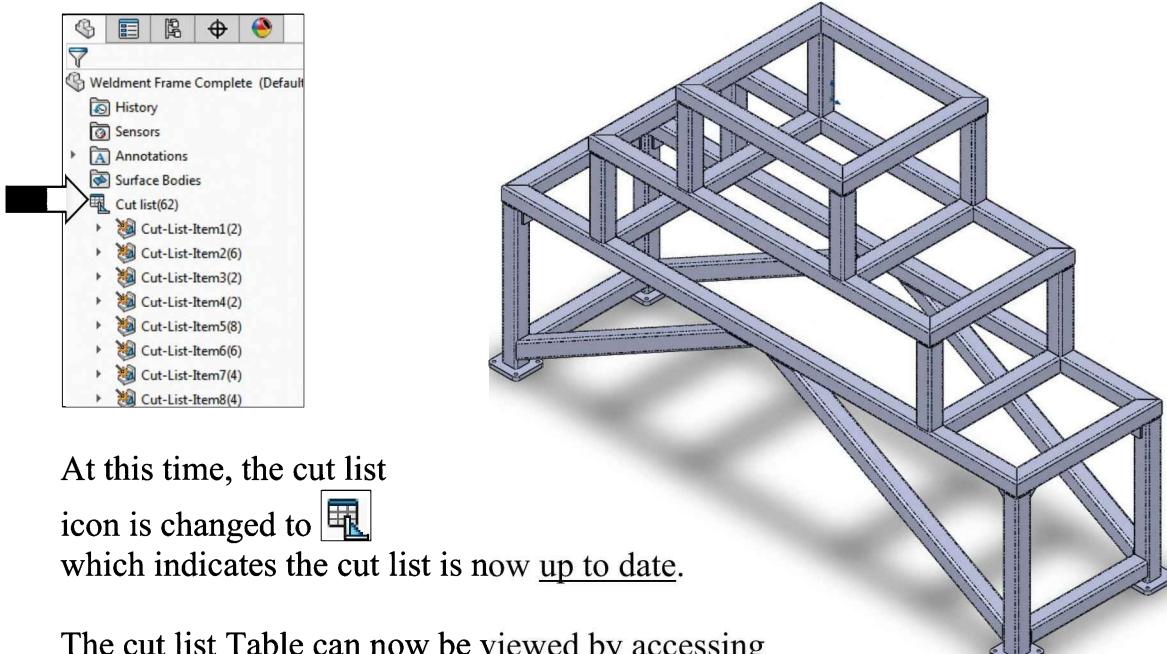
Right-click on the cut list and select **Update** (arrow).

Notice the option **Automatic** is on by default. This option organizes all of the weldment entities in the cut list for the new weldment parts.



Although the cut list is generated automatically, you can manually specify when to update the cut list in a weldment part document.

This enables you to make many changes, and then update the cut list once. However, the cut list updates automatically when you open a drawing that references the list.

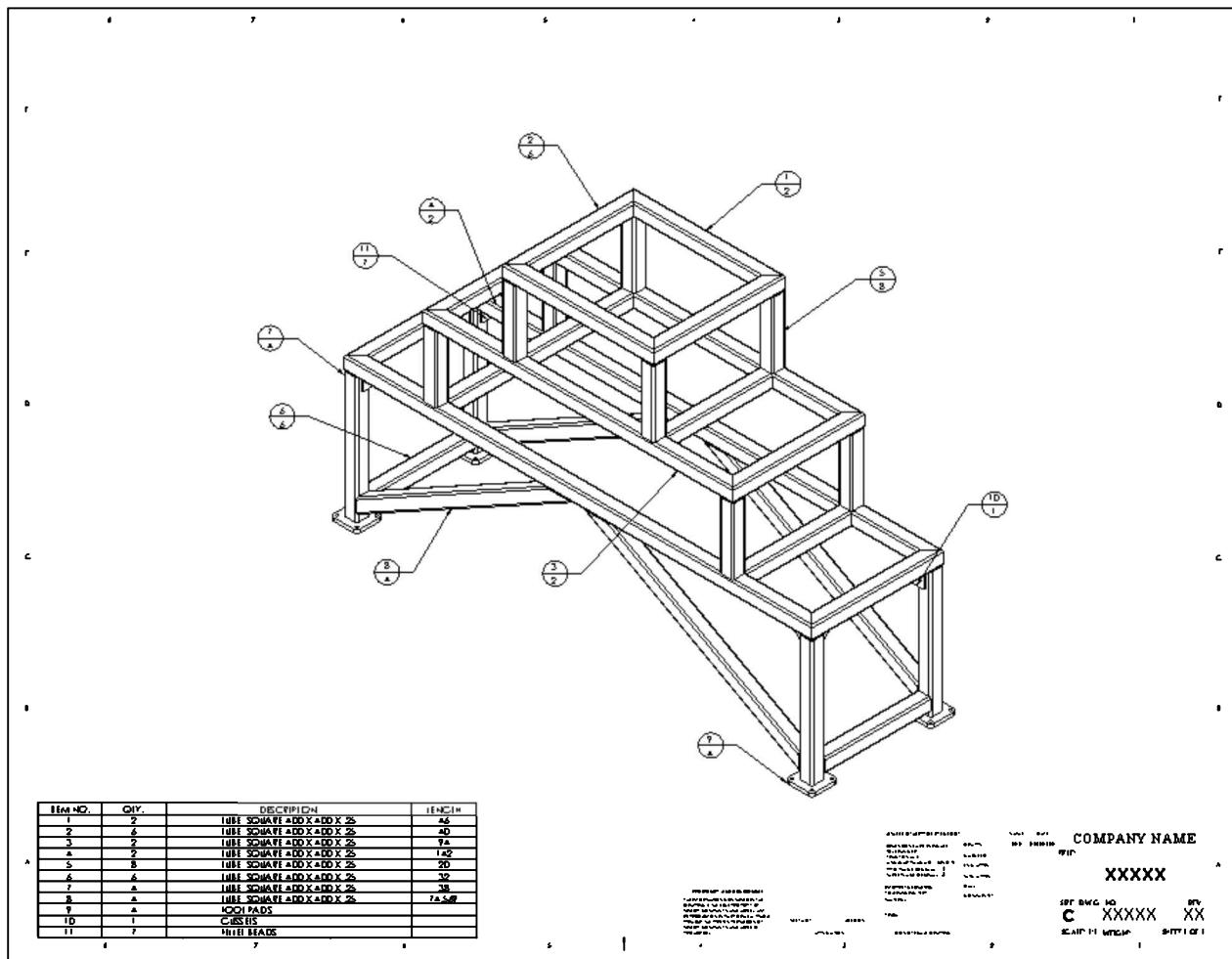


At this time, the cut list icon is changed to  which indicates the cut list is now up to date.

The cut list Table can now be viewed by accessing the Properties of one of the item folders.

22. Creating a drawing (OPTIONAL):

A drawing that includes the cut list can be generated. (Refer to the Part-1 Basic-Tools textbook for more information on how to create a detail drawing using SOLIDWORKS 2024.)



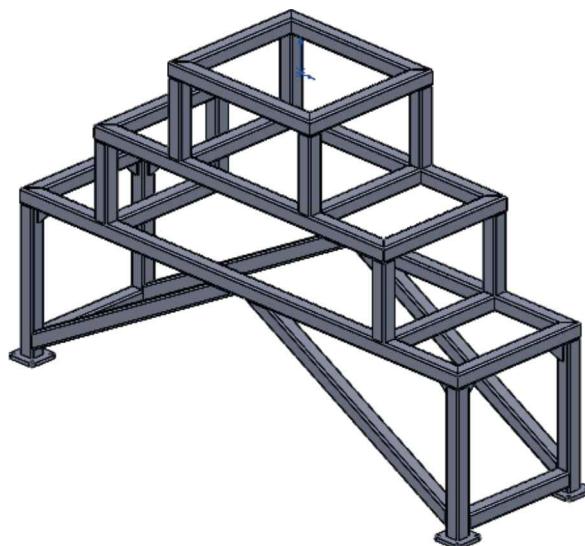
23. Saving your work:

Click **File / Save As**.

Enter **Weldment Frame** for the name of the file.

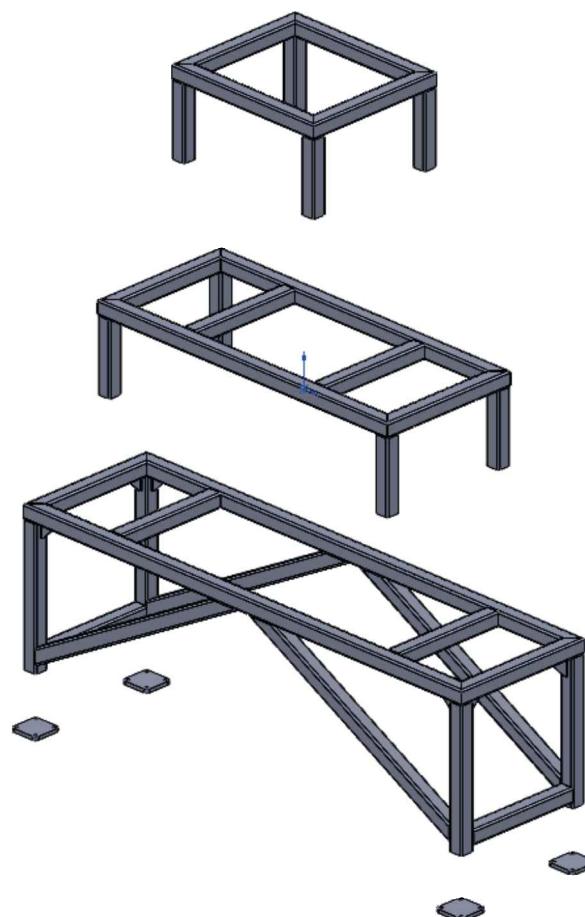
Click **Save**.

Replace the existing file when prompted.



Optional:

To create the exploded view similar to the one shown here:
Either use the Move/Copy command (Insert / Features/Move-Copy)
or use the Exploded View command (Insert /Exploded View).

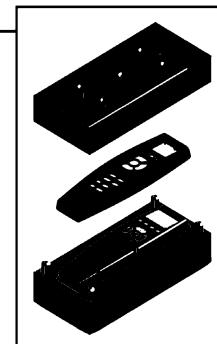


CHAPTER 18

Creating a Core and Cavity

Creating a Core and Cavity Linear parting Lines

In SOLIDWORKS a mold is usually designed using a sequence of intergraded tools that control the mold creation process. Using the finished model, these mold tools can be used to analyze and correct deficiencies in the part.



The process usually follows these steps: Draft analysis, Undercut Detection, Parting Lines, Shut-Off Surfaces, Parting Surfaces, Interlock Surfaces (Ruled Surfaces), and Tooling Split.

The Parting Lines  lie along the edge of the molded part, between the core and the cavity surfaces. They are used to create the Parting Surfaces and to separate the surfaces.

The Shut-Off Surfaces  are created after the Parting Lines. A shut-off surface closes up a through hole by creating a surface patch along the Edges that form a continuous loop, or a parting line you previously created, to define a loop.

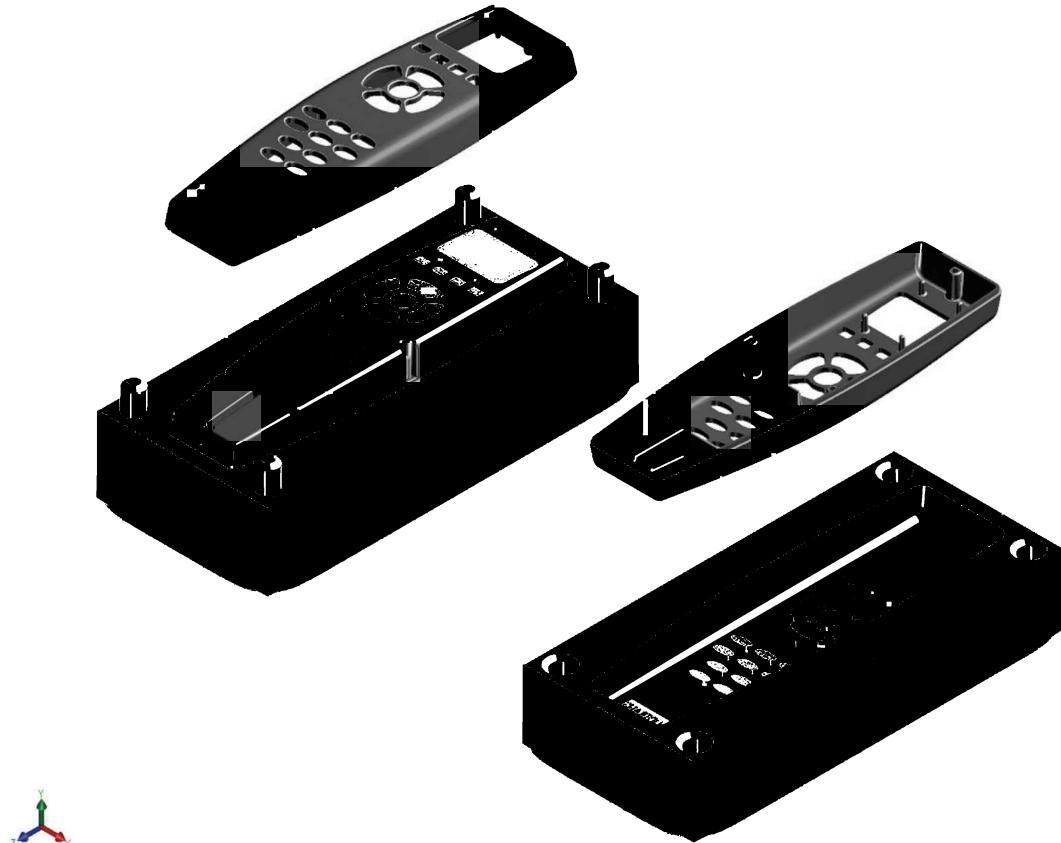
After the Parting Lines and the Shut-Off Surfaces are determined, the Parting Surfaces  are created. The Parting Surfaces extrude from the parting lines and are used to separate the mold cavity from the core.

After a parting surface is defined, the Tooling Split tool  is used to create the core and cavity blocks from the model. To create a tooling split, at least three surface bodies are needed in the Surface Bodies folder, a Core, a Cavity and a Parting surface.

With most mold parts, the interlock surfaces  need to be created. The interlock surfaces help prevent the core and cavity blocks from shifting, and are located along the perimeter of the parting surfaces. Usually they have a 5-degree taper.

Creating a Core and Cavity

Linear Parting Lines



Dimensioning Standards: ANSI
Units: INCHES – 3 Decimals

Tools Needed:



Parting Lines



Parting Surfaces



Shut-Off
Surfaces



Tooling Split



Planes



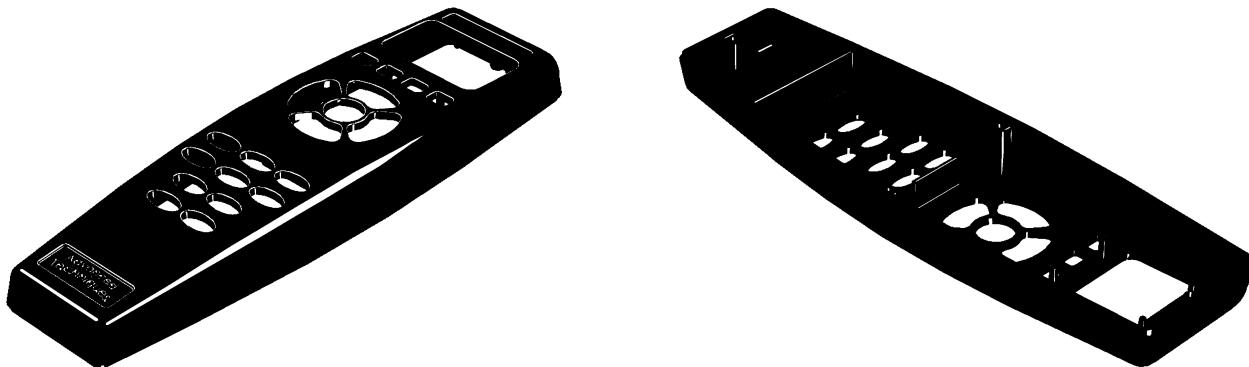
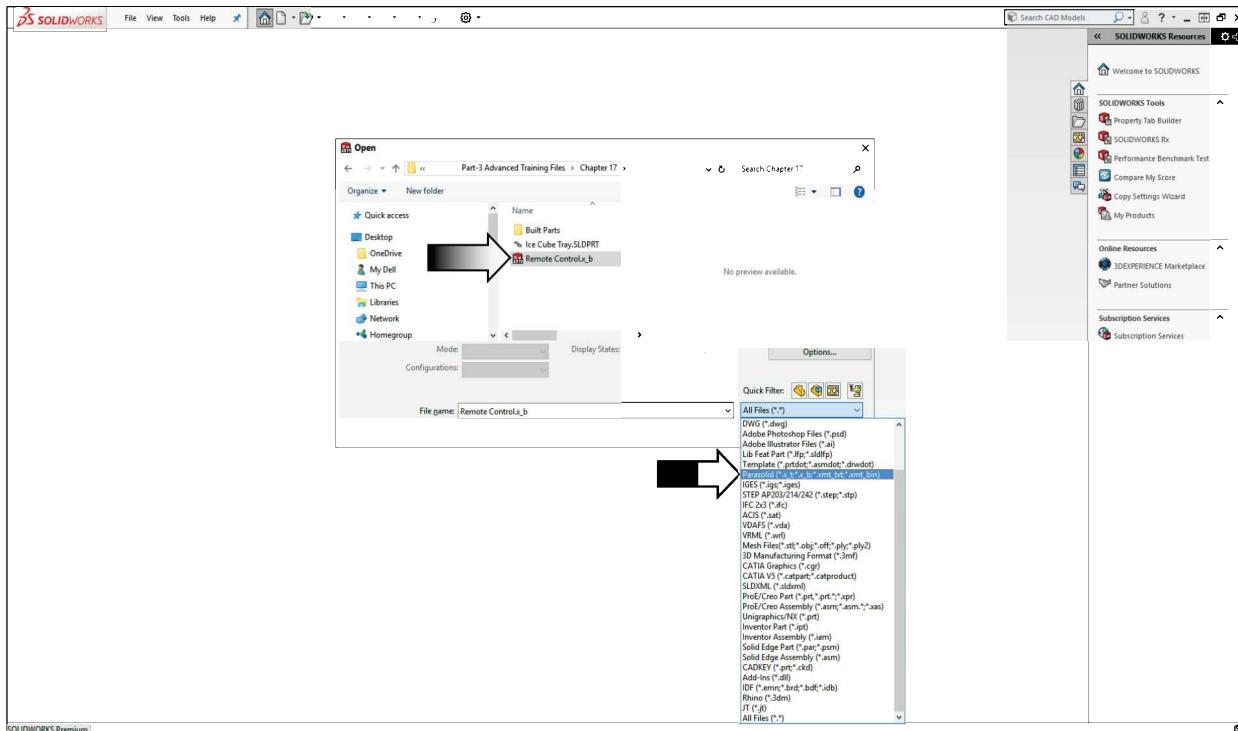
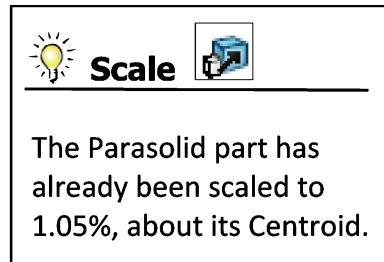
2D Sketch

1. Opening an existing Parasolid document:

Select File / Open.

Browse to the Training Files folder, change the Files of Type to **Parasolid**.

Select **Remote Control.x_b** and click **Open**.

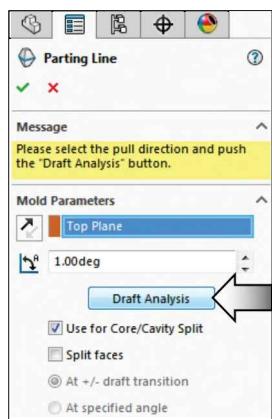
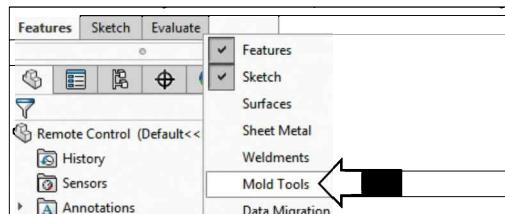


Click **NO** on the Import Diagnostics dialog and close it.

2. Creating the Parting Lines:

Right-click one of the tool tabs and enable the **Mold Tools**.

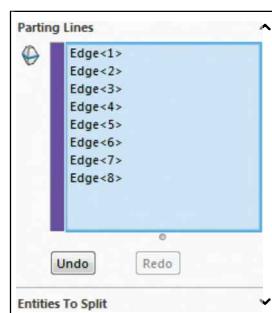
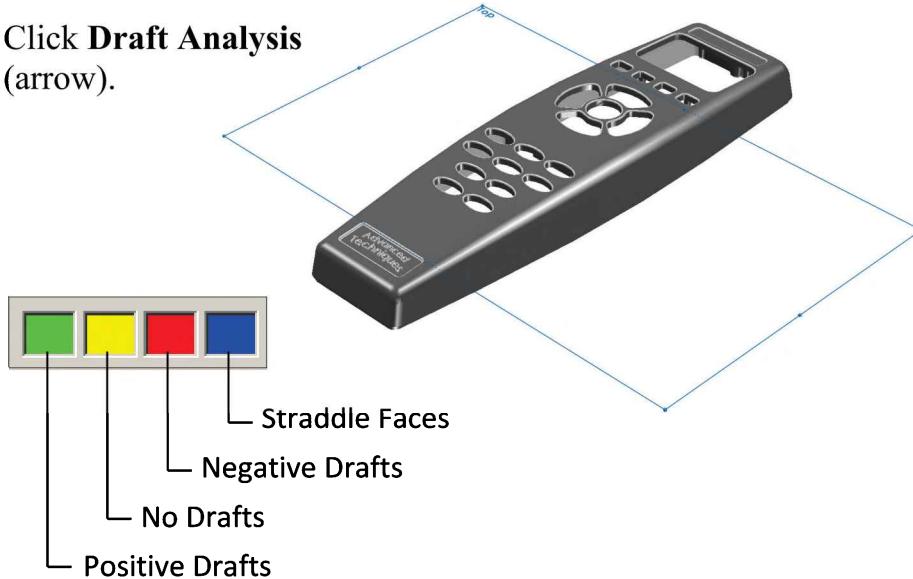
Click **Parting Line**  on the Mold Tools toolbar.



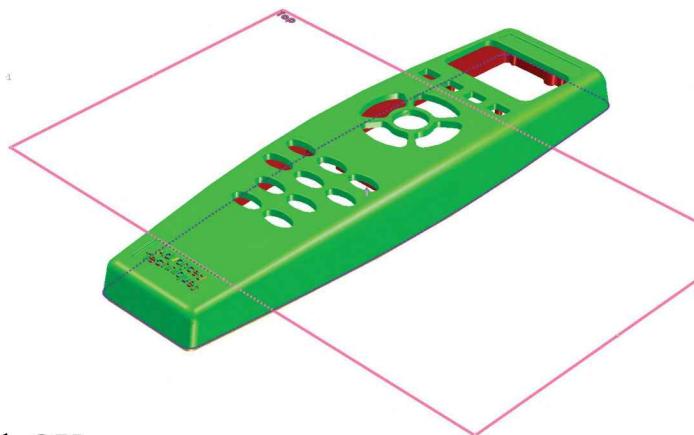
For Direction of Pull, select the **Top plane** from the Feature tree.

For Draft Angle, enter **1deg**.

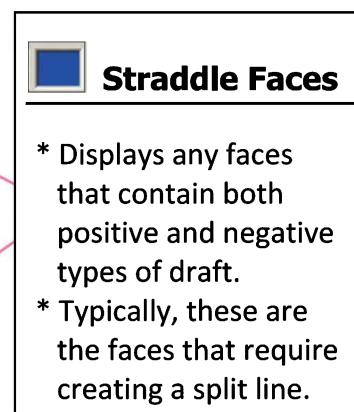
Click **Draft Analysis** (arrow).



SOLIDWORKS automatically selects the lower edges of the part where the red surfaces meet the green, and places them in the Parting Lines section.

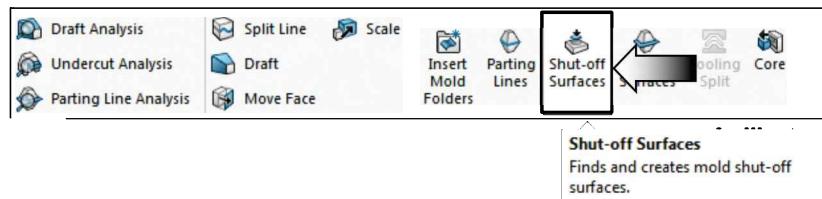
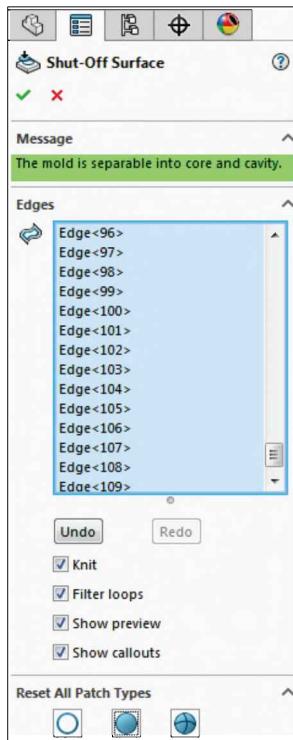


Click **OK**.



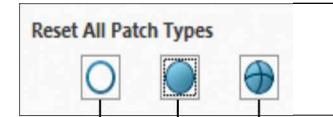
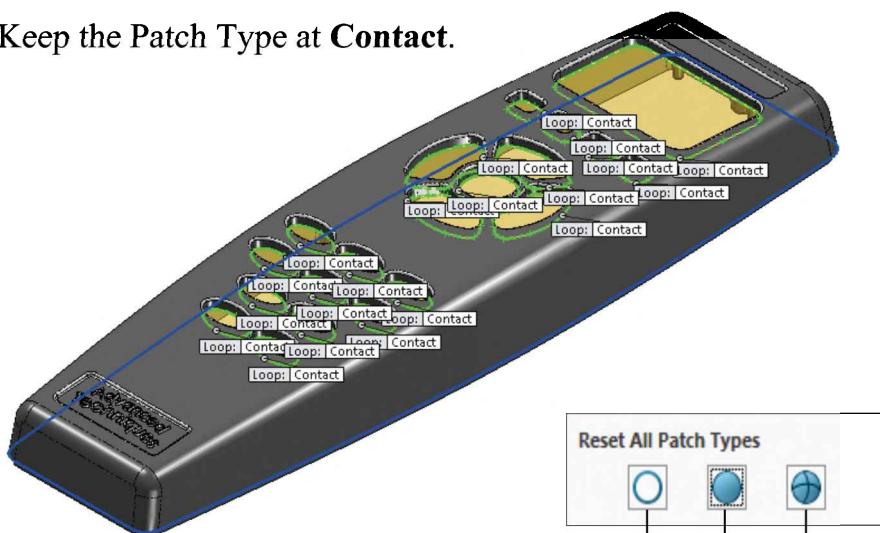
3. Creating the Shut-Off Surfaces:

Click  or select Insert / Molds / Shut-Off Surfaces.



SOLIDWORKS automatically selects the edges of all through openings and labels them as Loop/Contacts.

Keep the Patch Type at **Contact**.



Patch Types

Only one Shut-Off Surface feature is allowed in a model. Therefore, within the one feature, you must assign a fill type of **Contact**, **Tangent**, or **No Fill** to every hole.

Resets all through hole surface patches to one of the 3 settings.

All No-Fill

All Contact

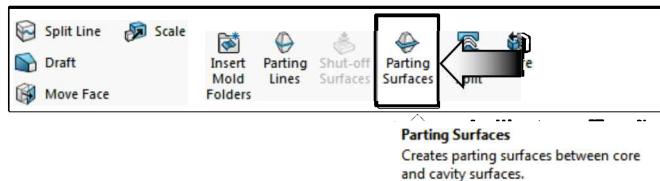
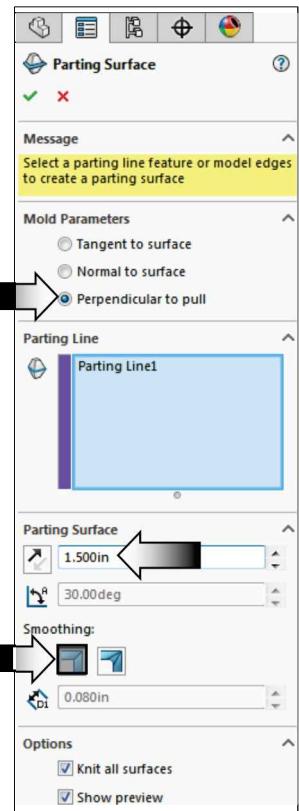
All Tangent



Click **OK**.

4. Creating the Parting Surfaces:

Click  or select Insert / Molds / Parting Surfaces.



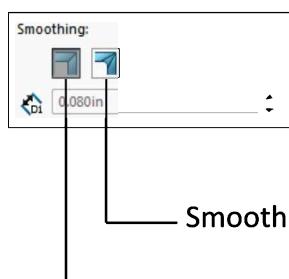
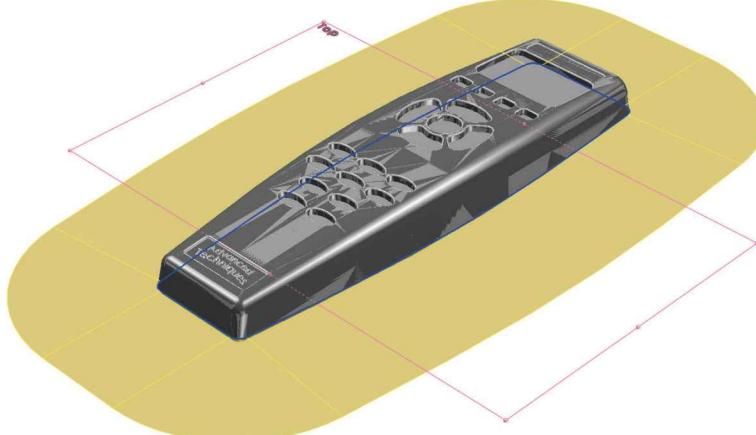
The Parting surfaces split the mold cavity from the core. It gets created after the parting lines and shut off surfaces.

For Mold Parameters, select **Perpendicular to Pull**.

For Parting Line, select the **Parting Line1** from the FeatureManager Tree.

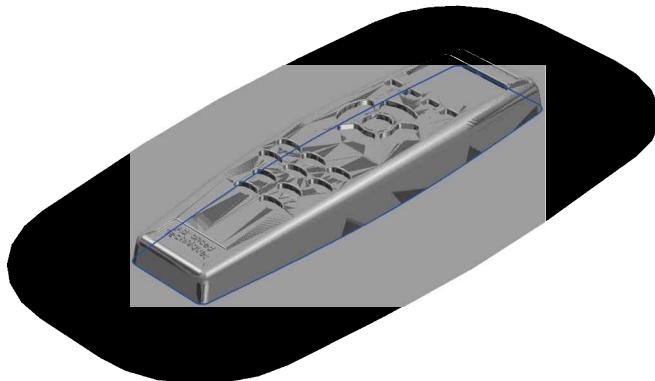
For Parting Surface Distance, enter **1.500 in**.

Select **Sharp Edges** under the Smoothing section.



Smooth Edges

Sharp Edges
(A higher value creates a smoother transition between adjacent edges)



Click **OK**.

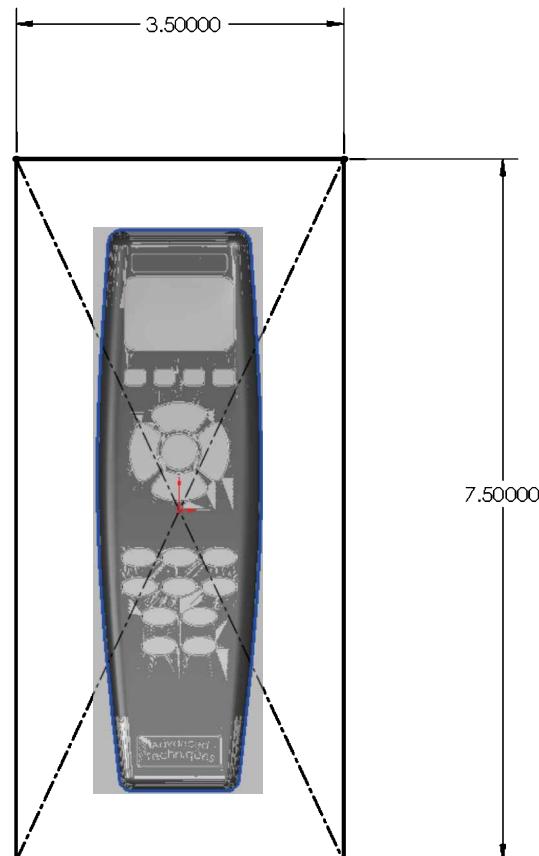
5. Sketching the profile of the mold-blocks:

Select the Top plane and open a new sketch .

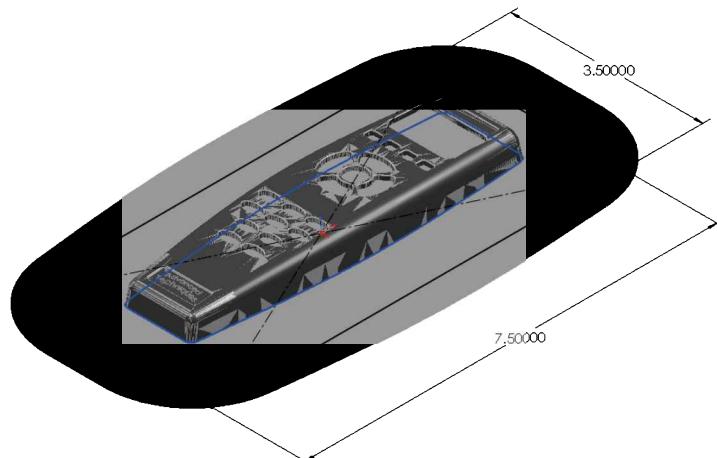
Sketch a **Center Rectangle** 

centered on Origin.

Add the width and the height dimensions.



The sketch should be fully defined at this point.



Exit the Sketch .

(The next step is to create the upper and lower blocks using the Tooling Split command. This command is only available when the Sketch is off.)

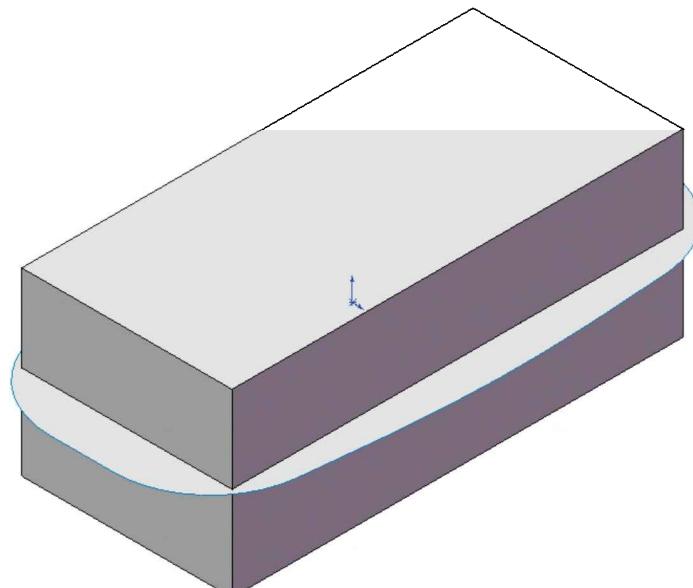
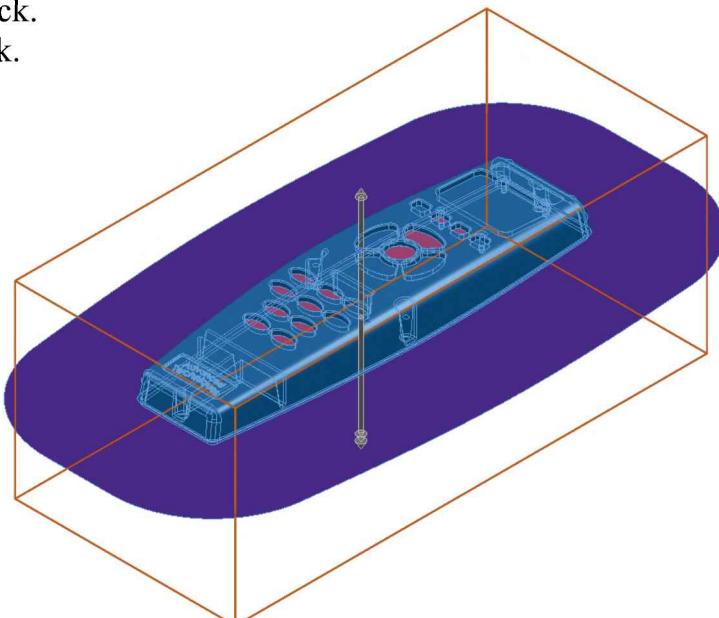
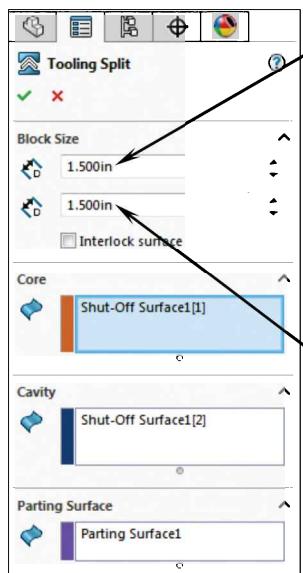
6. Creating the Tooling Split:

Click  or select:
Insert / Molds / Tooling Split.



In the Block Size selection, enter:

1.500 in for upper block.
1.50 in for lower block.



The Cavity and the parting surfaces options should already be filled.

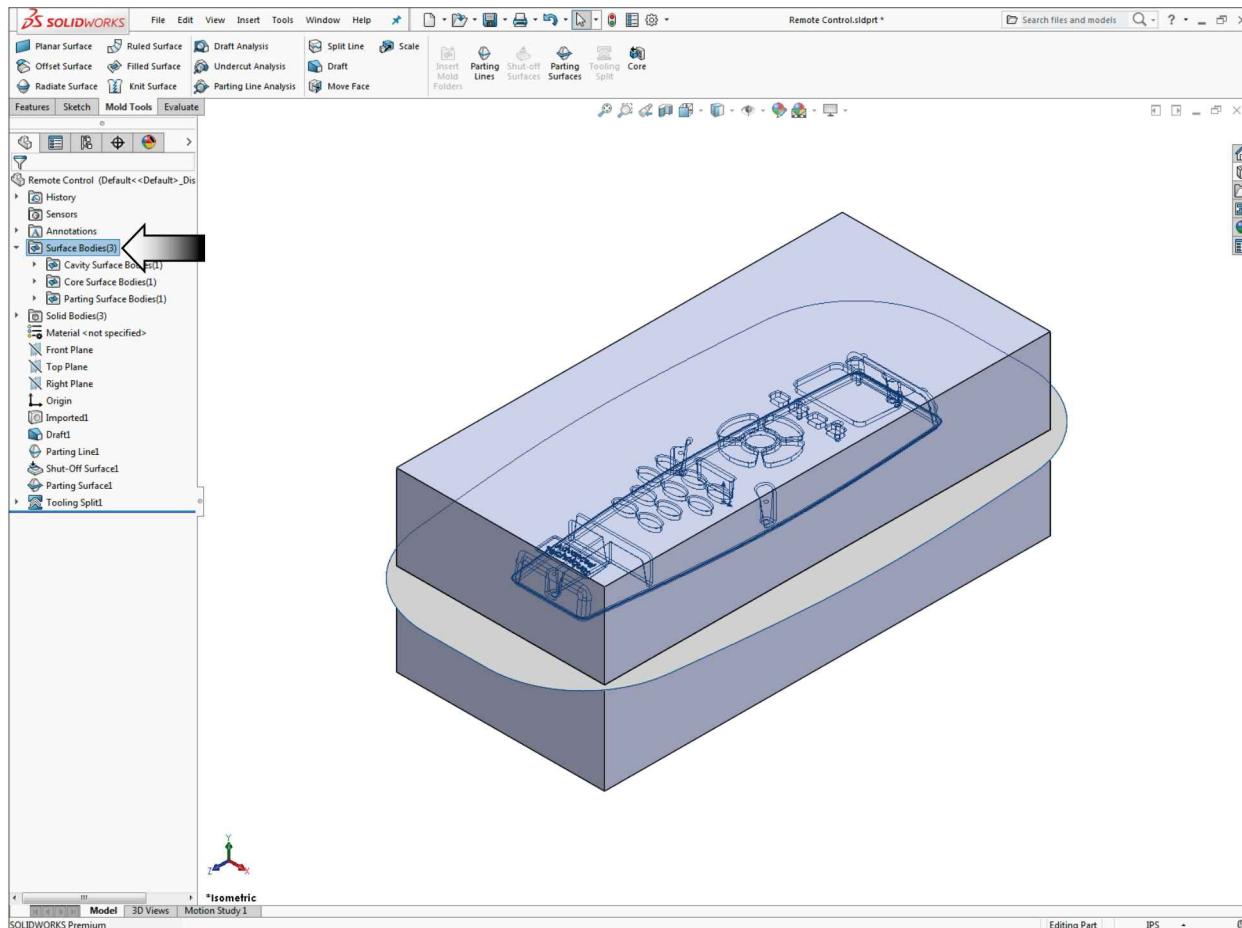
Click **OK**.

* The Interlock Surface surrounds the perimeter of the parting lines in a slight tapered direction. It helps seal the mold to prevent resins from leaking, prevents shifts, and maintains alignment between the tooling entities.

7. Hiding the Solid Bodies:

From the FeatureManager Tree, expand the **Surface Bodies** folder. There are 3 groups of surfaces in this folder.

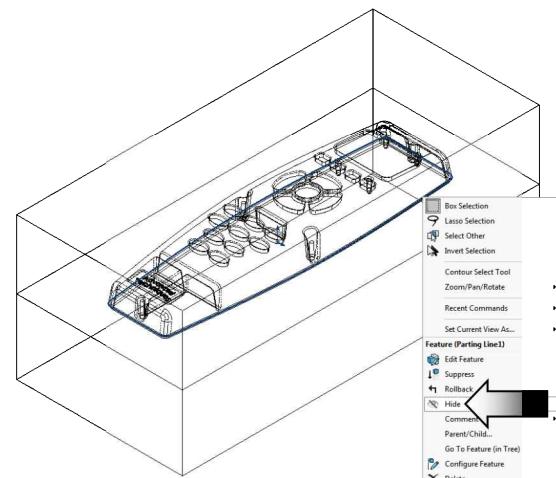
Click the **Surface Bodies** folder and select **Hide** .



The 3 surfaces that were created in the previous steps are temporarily removed from the graphics display.

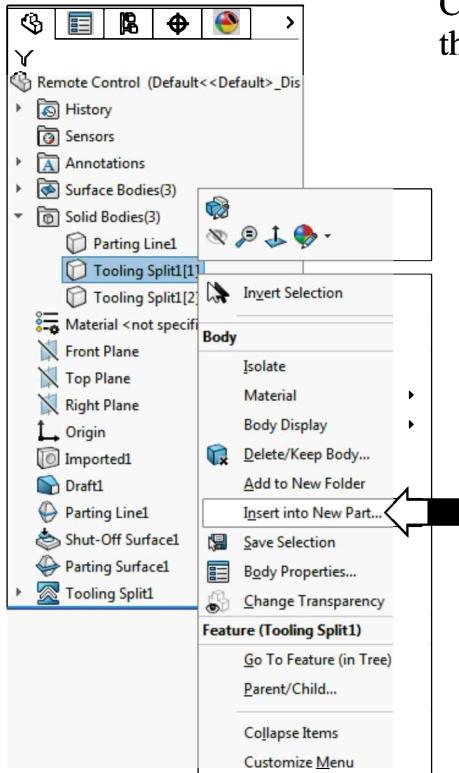
Change to the **Wireframe** mode to see the inside details of the blocks.

Right-click on the blue parting lines and Hide it (arrow).

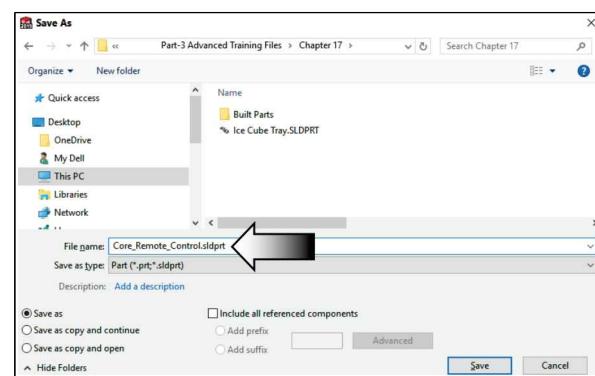
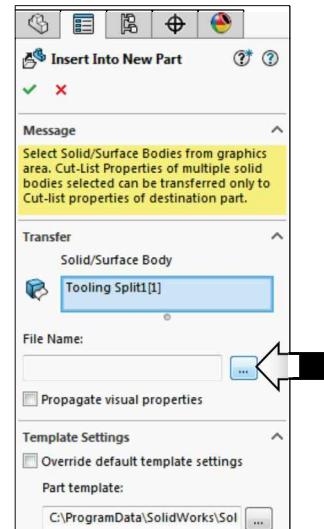


8. Saving the bodies as part files:

Expand the Solid Bodies folder, right-click on Tooling Split [1] and select: Insert into New Part.



Click the Browse button and select a location to save the document.

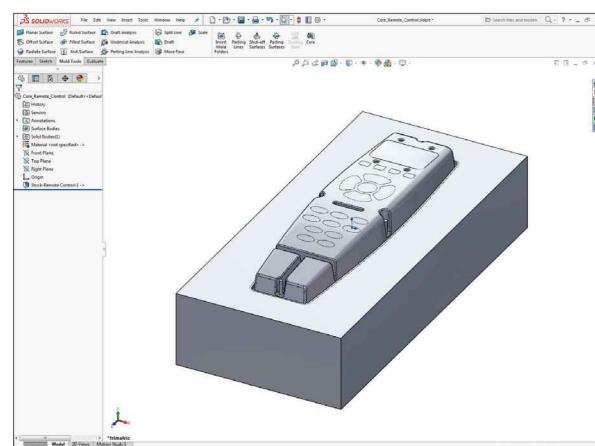


Enter: **Core_Remote_Control**, for the name of the file.

Click Save.

Hold the **Control** key and push the **Tab** key to switch back to the main part.

Repeat the last step to save the **Tooling Split[2]**, enter: **Cavity_Remote_Control** for the name of the 2nd block.



9. Separating the 2 blocks:

Select **Insert / Feature / Move-Copy**.

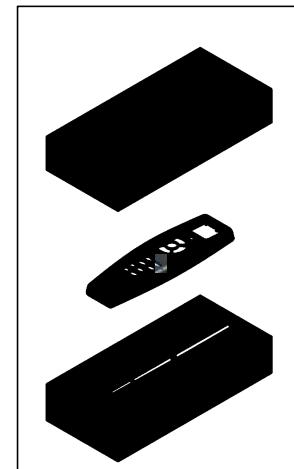
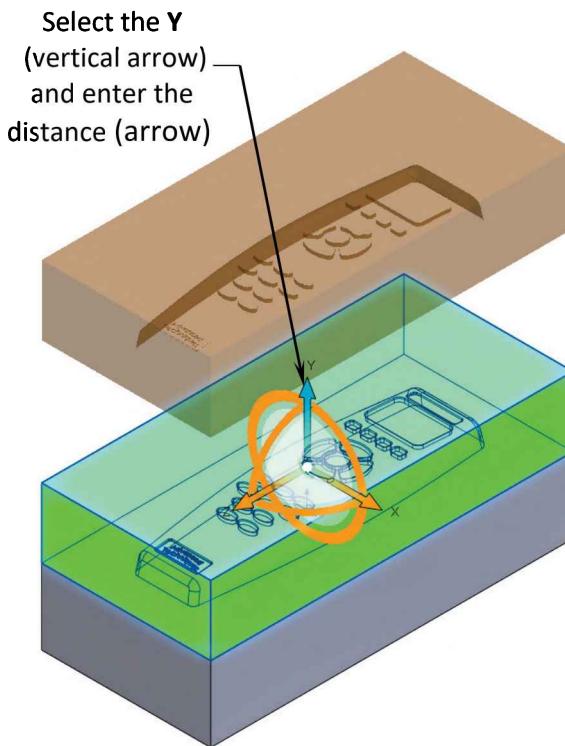
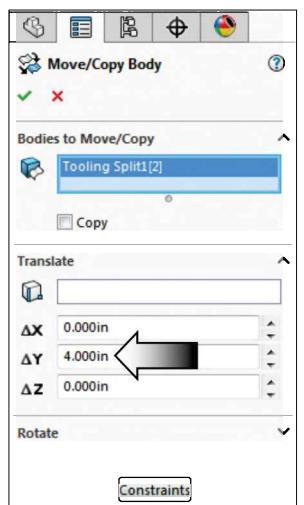
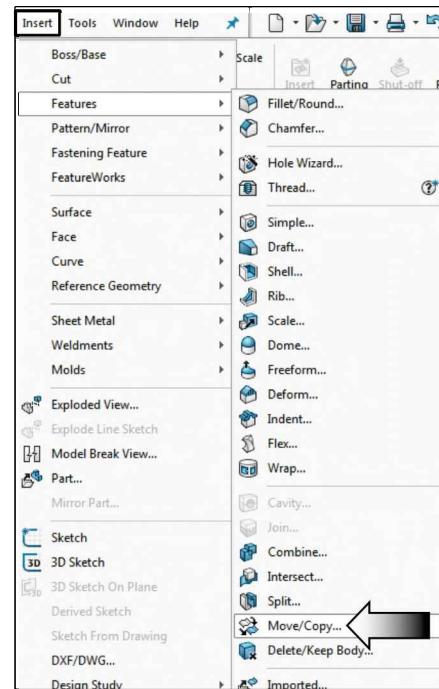
Select the **upper block** in the graphics area.

Click the **vertical arrow** to define the explode direction.

Under the Translate section, enter **4.00in** and press **Enter**.

Click **OK**.

The upper block moves 4 inches upward from its original position.



Repeat the same step to move the lower block downward (use -4.00" for distance).

10. Saving your work:

Save a copy of your work as **Remote Control Tooling**.

11. Optional:

Create a new Assembly document and assemble the 3 components.

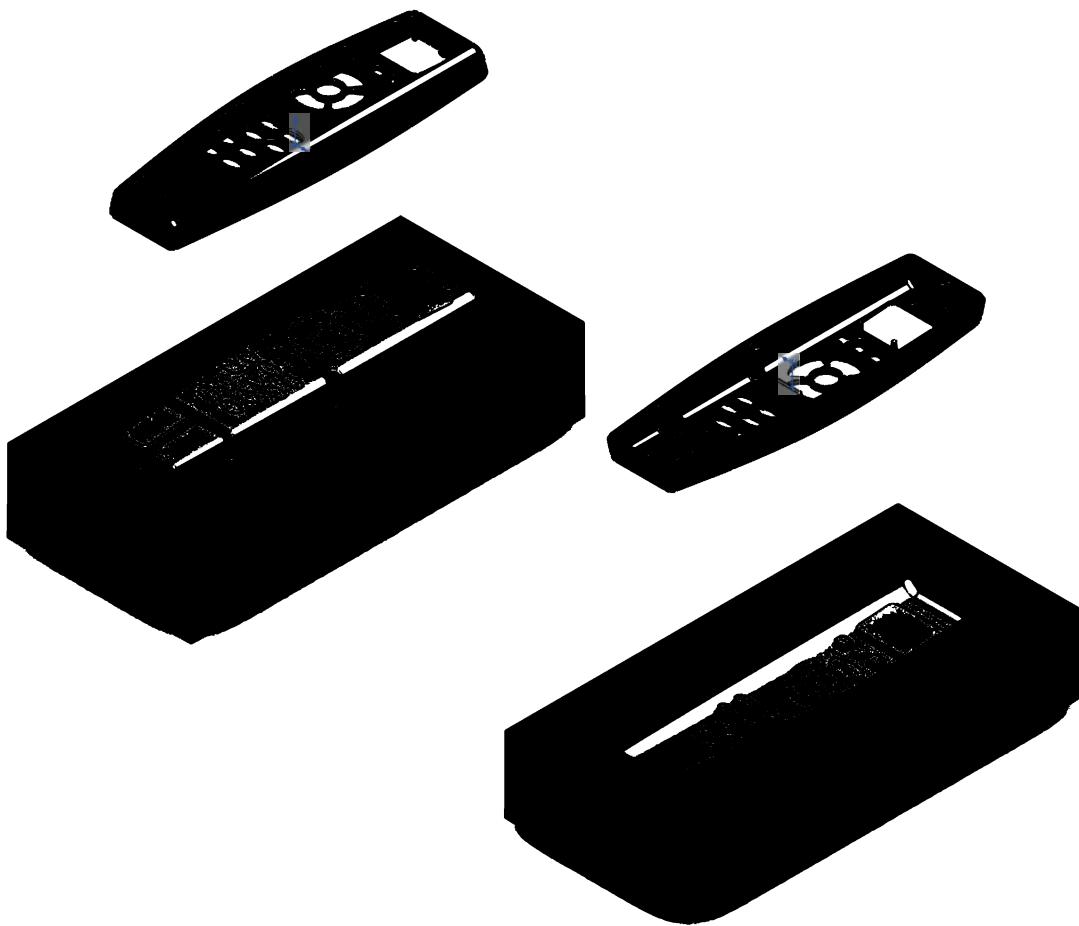
Create an Assembly Exploded View as a separate configuration.

Add Injector hole.

Ejector holes.

Alignment Pins.

Make copies of the components and create an exploded view as shown.

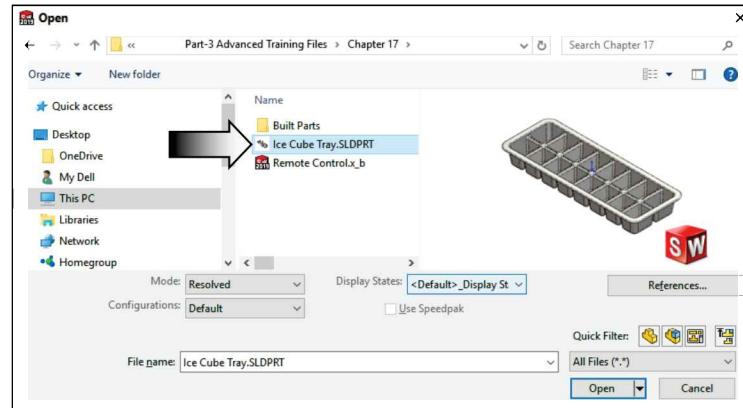


Exercise: Linear Parting Lines

1. Opening a part document:

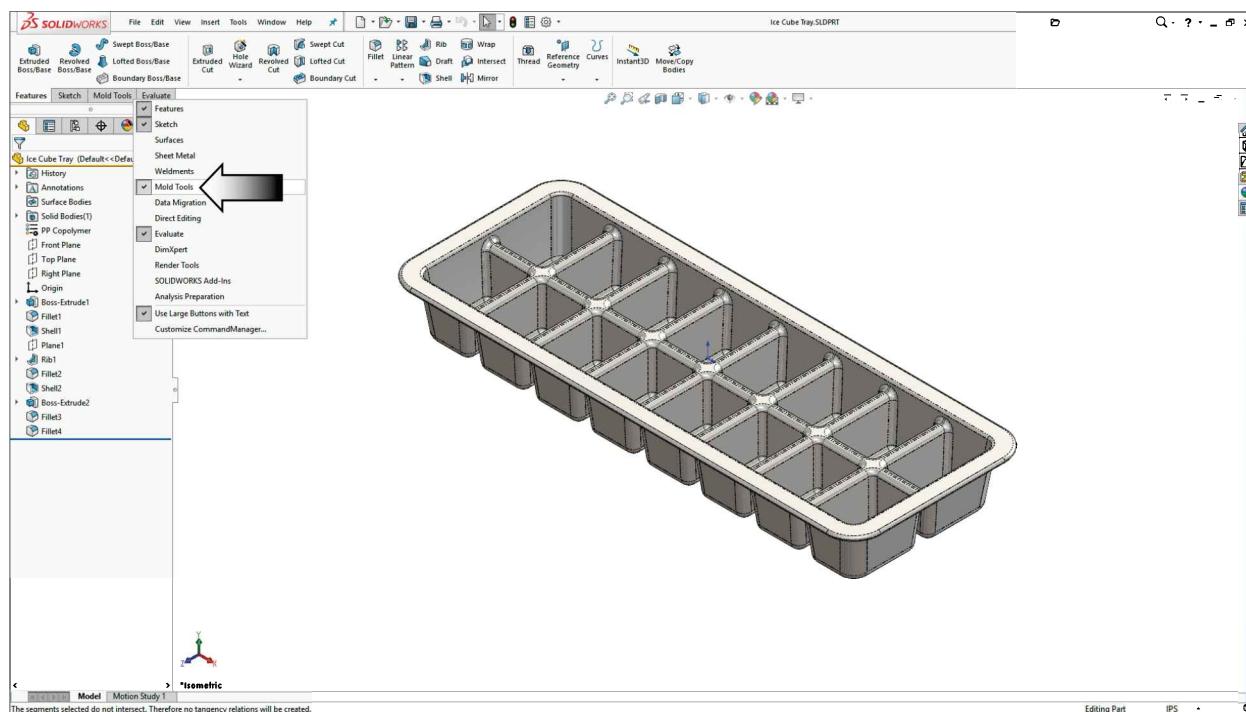
Open a part document named:

Ice Cube Tray.sldprt
from the training files folder.



2. Enabling the Mold Tools:

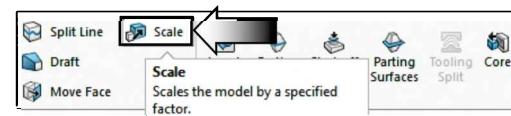
Right-click one of the tool tabs and select the **Mold Tools** (arrow).



A material has already been assigned to the part (**PP Copolymer**), but new material will be assigned to the mold blocks after they are created.

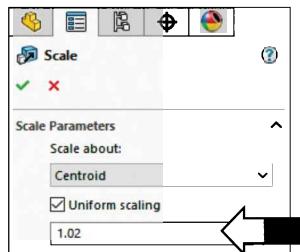
3. Applying Scale:

Switch to the **Mold Tools** tab.



Click the **Scale** command and enter **1.02** (2% larger).

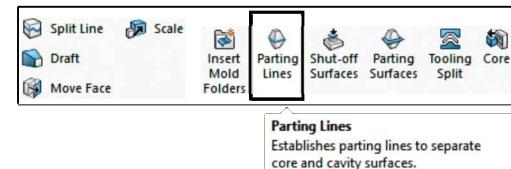
Use the default **Centroid** option.



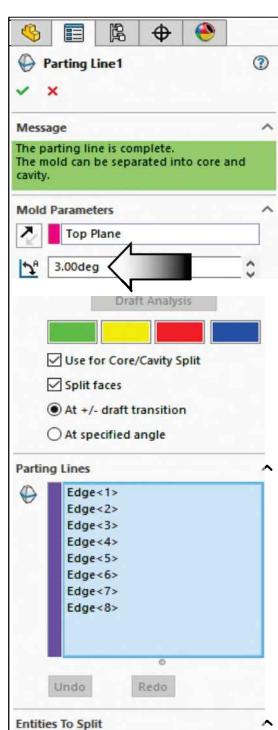
Click **OK**.

4. Creating the parting line:

Click the **Parting Lines** command.



For Direction of Pull, select the **Top** plane.

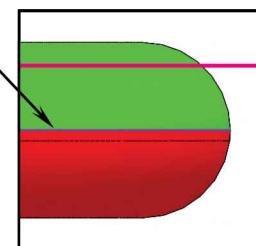


For Draft Angle, enter **3.00deg**.

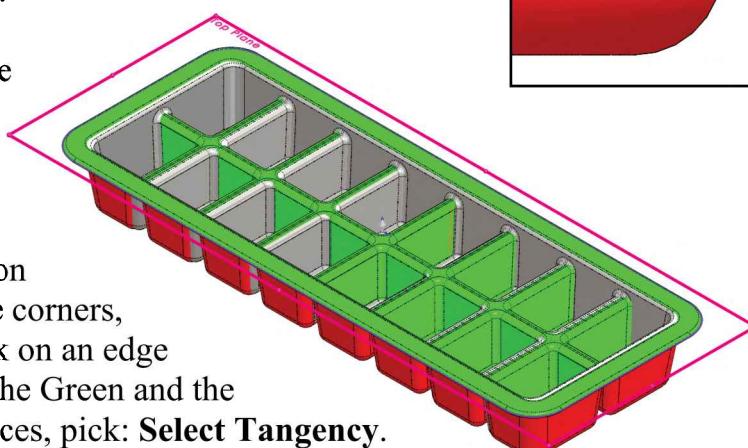
Right-click an edge between the Green & Red surfaces

Enable the **Use for Core/Cavity Split** checkbox.

Enable the **Split-Faces** box.



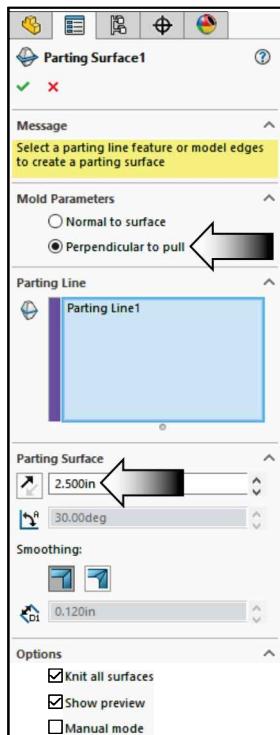
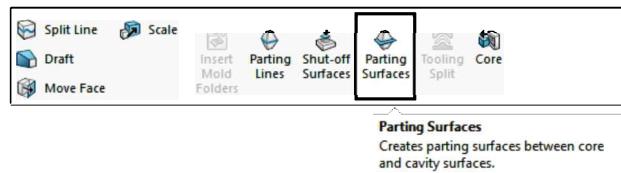
Zoom in on one of the corners, right-click on an edge between the Green and the Red surfaces, pick: **Select Tangency**.



Click **OK**.

5. Creating the parting surface:

Click the **Parting Surfaces** command.

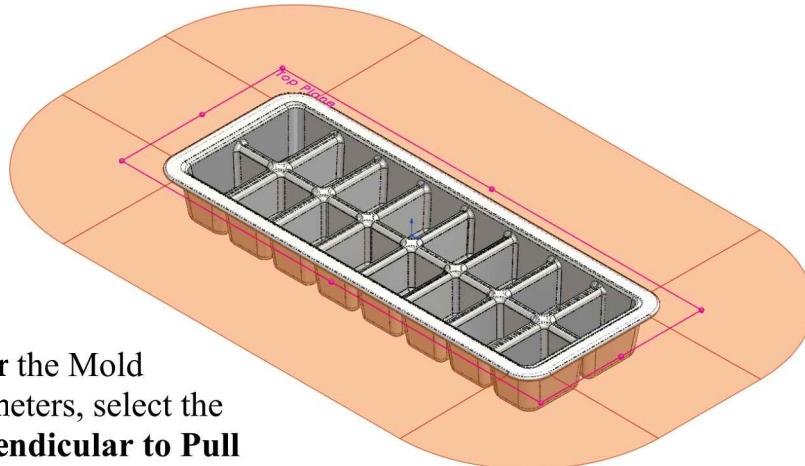


Under the Mold Parameters, select the **Perpendicular to Pull** option (arrow).

For Parting Surface, enter **2.500in** for Distance (arrow).

Enable the **Knit All Surfaces** checkbox.

Click **OK**.

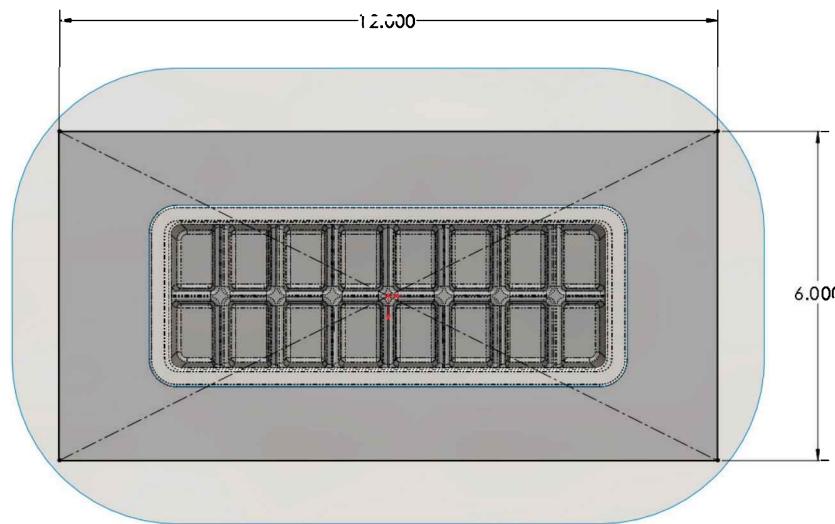


6. Sketching the mold block:

Open a new sketch on the Parting Surface.

Sketch a **Center-Rectangle** that is centered on the Origin.

Add the dimensions shown to fully define the sketch.

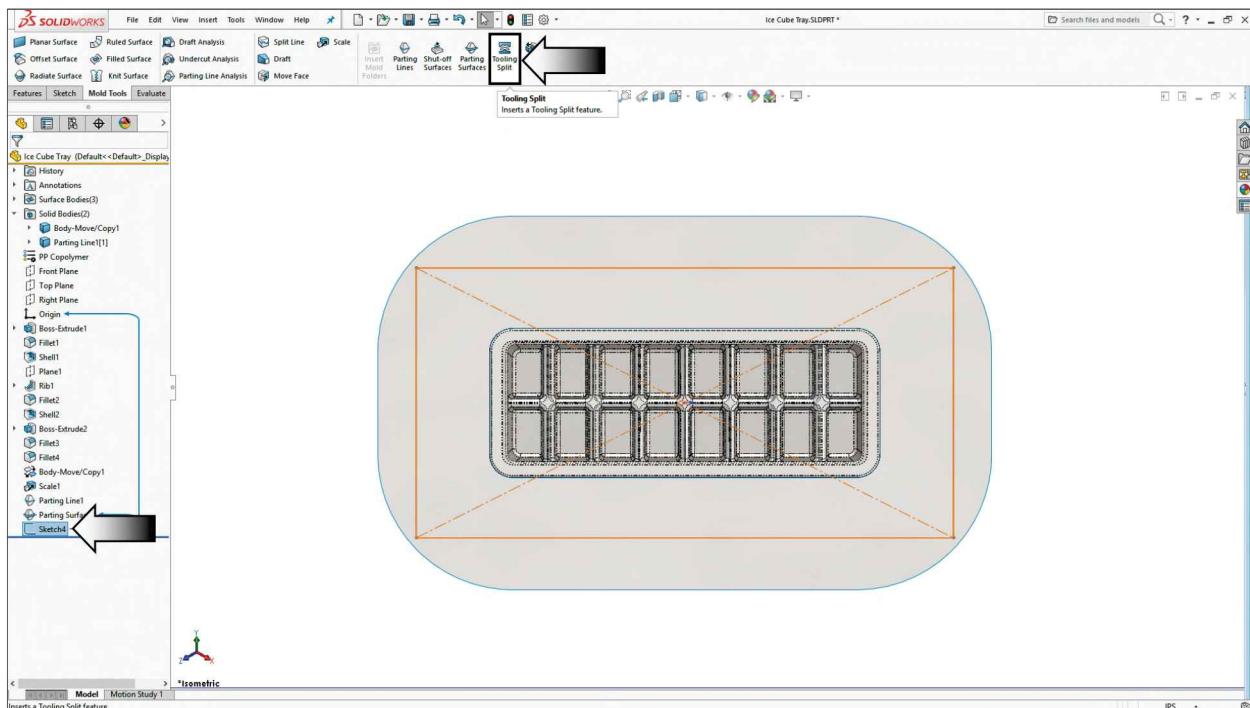


Exit the sketch.

7. Creating the tooling split:

Switch back to the **Mold Tools** tab.

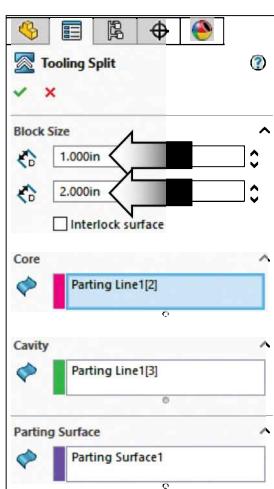
Select the **new sketch** from the FeatureManager tree and click **Tooling Split**.



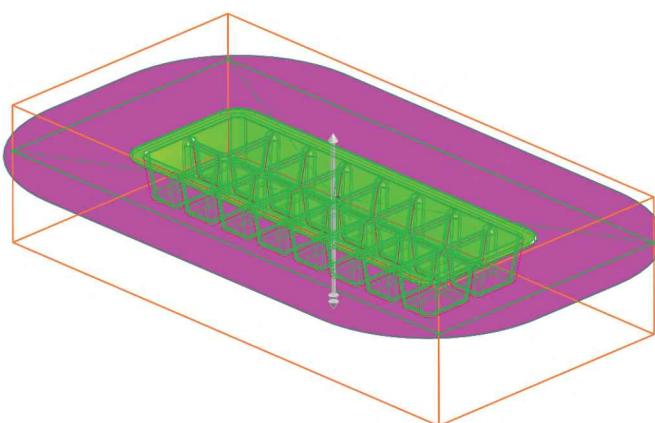
Select the rectangular sketch, if prompted.

For Upper Block Size, enter **1.000in**.

For Lower Block Size, enter **2.000in**.



The reference surfaces are automatically placed in their correct locations.

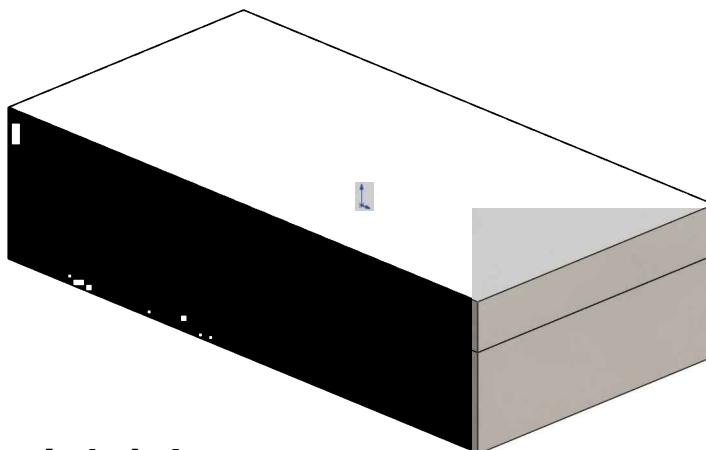
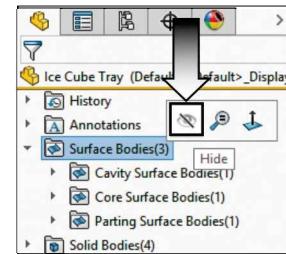


Click **OK**.

8. Hiding the reference surfaces:

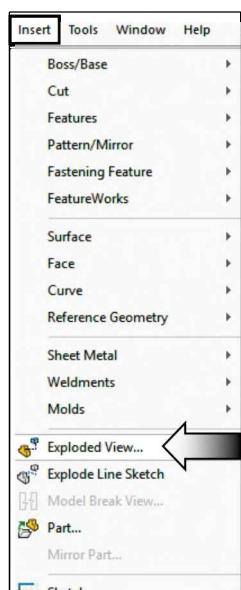
Locate the **Surface Bodies** folder near the top of the FeatureManager tree.

Click the Surface Bodies folder and select **Hide** (arrow).

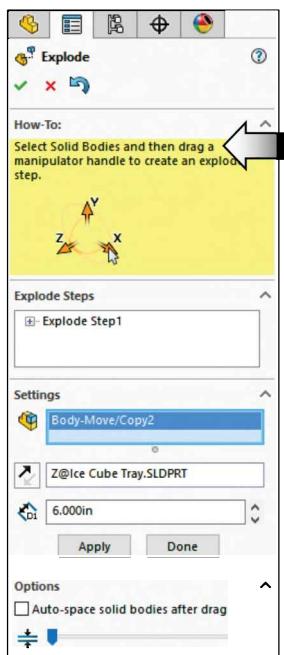


9. Creating an exploded view:

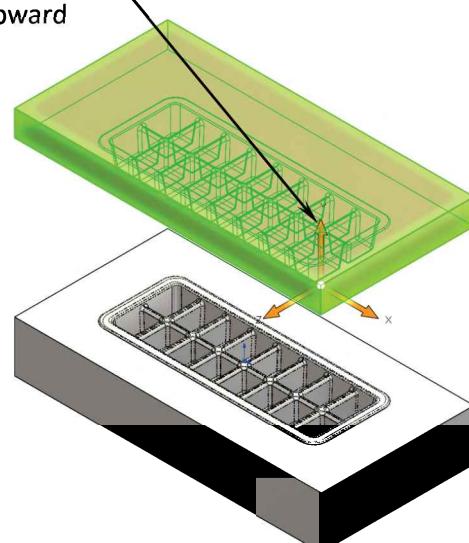
Select **Insert / Exploded View** (arrow).



Select the **upper mold block** and drag the **Y arrowhead** upwards approximately **6 inches**.



Drag the Y
arrowhead
upward

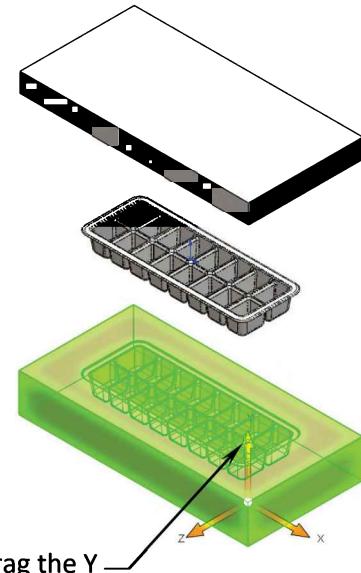


Explode Step1 is created and stored under the Explode Steps section.

Select the **lower mold block** and drag the Y arrowhead downwards, approximately **6.5 inches**.

The Explode Step2 is created.

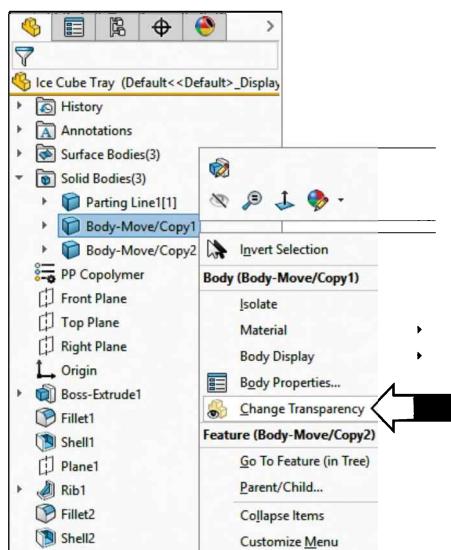
Click **OK**.



Drag the Y arrowhead downward

10. Assigning materials:

Expand the **Solid Bodies** folder.

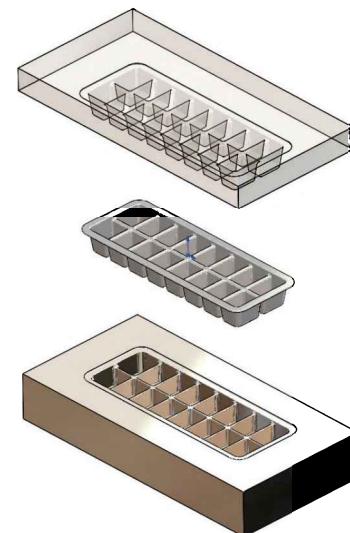
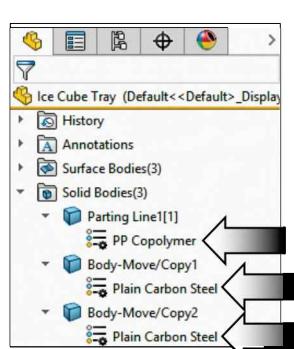


Right-click the **Body-Move/Copy1** (the upper mold block) and select: **Change Transparency**.

Right-click the same body and change the material to **Plain Carbon Steel**.

Assign the same material to the lower mold block.

Assign **PP Copolymer** to the plastic part.



11. Saving your work:

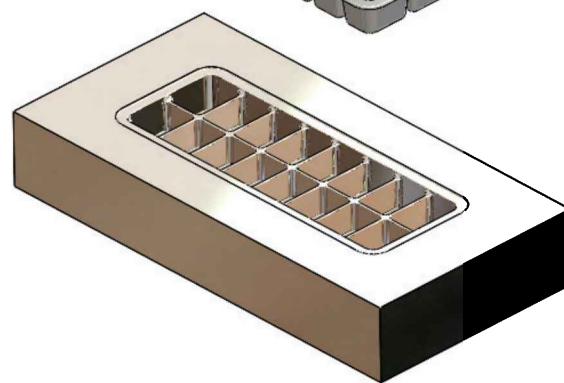
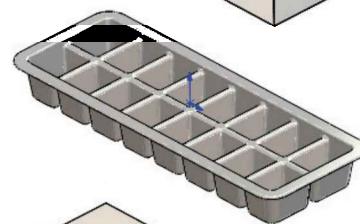
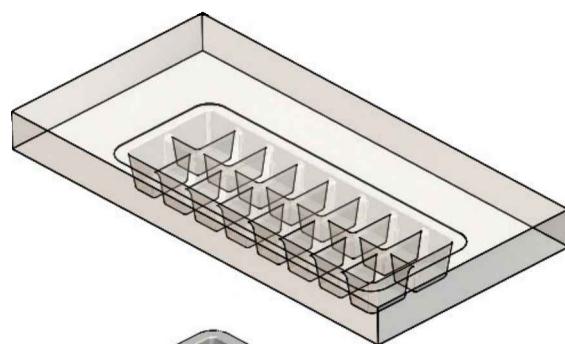
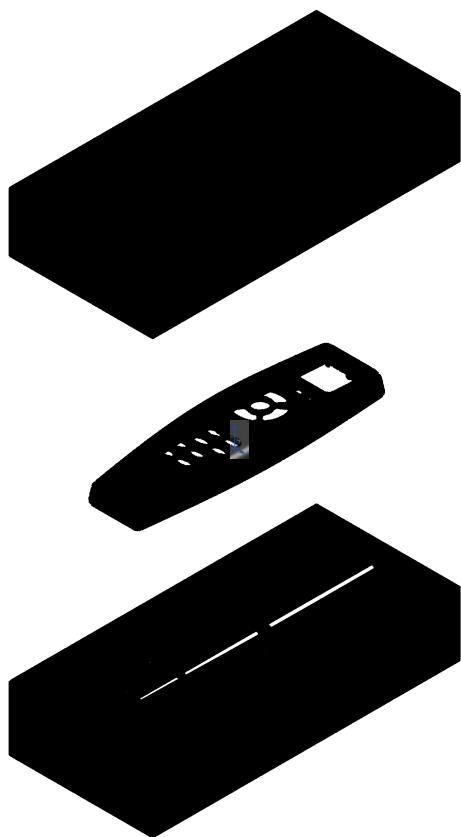
Click **File / Save As**.

Enter **Ice Cube Tray (Completed).sldprt** for the file name and click **Save**.

Questions for Review

1. Using the finished model, the mold tools can be used to analyze and correct the deficiencies such as undercuts, draft angles, shut-off surfaces, etc.
 - a. True
 - b. False
2. The Parting Lines are used to create the Parting Surfaces and to separate the surfaces.
 - a. True
 - b. False
3. A shut-off surface closes up a through hole by creating a surface patch along the edges that form a continuous loop.
 - a. True
 - b. False
4. The Parting Surfaces extrude from the parting lines and are used to separate the mold cavity from the core.
 - a. True
 - b. False
5. To create a tooling split, what surface bodies are needed for this operation?
 - a. The Core
 - b. The Cavity
 - c. The Parting Surface
 - d. All of the above
6. The Interlock surfaces help prevent the core and cavity blocks from shifting and are located along the perimeter of the parting surfaces.
 - a. True
 - b. False
7. The solid bodies can be hidden or shown just like any other features in SOLIDWORKS.
 - a. True
 - b. False

1. TRUE
2. TRUE
3. TRUE
4. TRUE
5. D
6. TRUE
7. TRUE

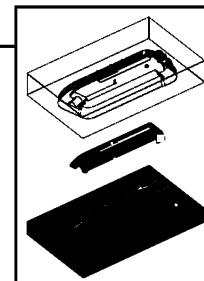


CHAPTER 19

Non-Planar Parting Lines

Non-Planar Parting Lines Mold-Tooling Design

Using SOLIDWORKS a mold is created by following a sequence of integrated tools that control the mold creation process.



The mold tools are used to analyze and correct deficiencies such as draft angles or undercuts with the plastic models to be molded.

The mold tools span from initial analysis to creating the tooling split. The result of the tooling split is a multibody part containing separate bodies for the molded part, the core, and the cavity, plus other optional bodies such as side cores.

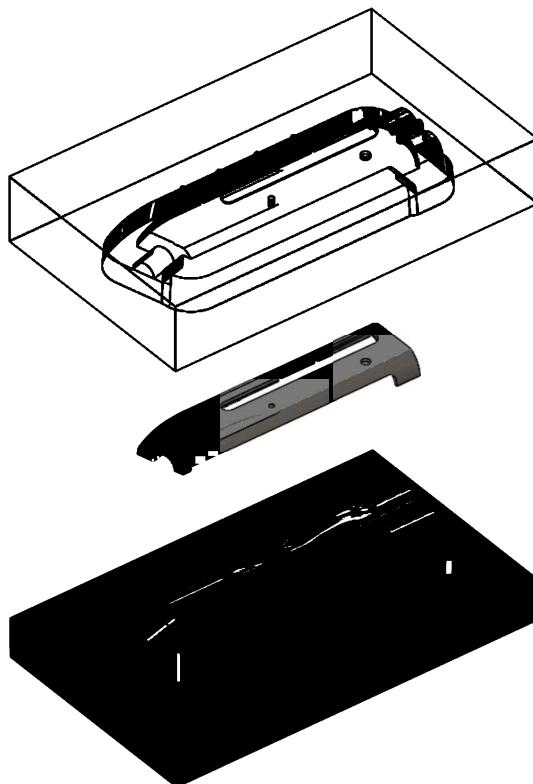
The multibody part file maintains your design intent in one convenient location. Changes to the molded part are automatically reflected in the tooling bodies.

The mold design process is as follows:

- * **Draft Analysis:** Examines the faces of the model for sufficient draft, to ensure that the part ejects properly from the tooling.
- * **Undercut Analysis:** Identifies trapped areas that prevent the part from ejecting.
- * **Parting Line Analysis:** Analyzes transitions between positive and negative draft to visualize and optimize possible parting lines.
- * **Parting Lines:** Creates a parting line from which you create a parting surface.
- * **Shut-off Surfaces:** Creates surface patches to close up through holes in the molded part.
- * **Parting Surfaces:** Extrude from the parting line to separate mold cavity from core. You can also use a parting surface to create an interlock surface.
- * **Ruled Surface:** Adds draft to surfaces on imported models. You can also use the Ruled Surface tool to create an interlock surface.
- * **Tooling Split:** Creates the core and cavity bodies, based on the steps followed earlier.

Non-Planar Parting Lines

Mold-Tooling Design



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: **ANSI**
Units: **INCHES – 3 Decimals**

Tools Needed:



Parting Lines



Tooling Split



Planar Surface



Shut-Off Surfaces



Ruled Surface



Knit Surface



Parting Surfaces



Filled Surface

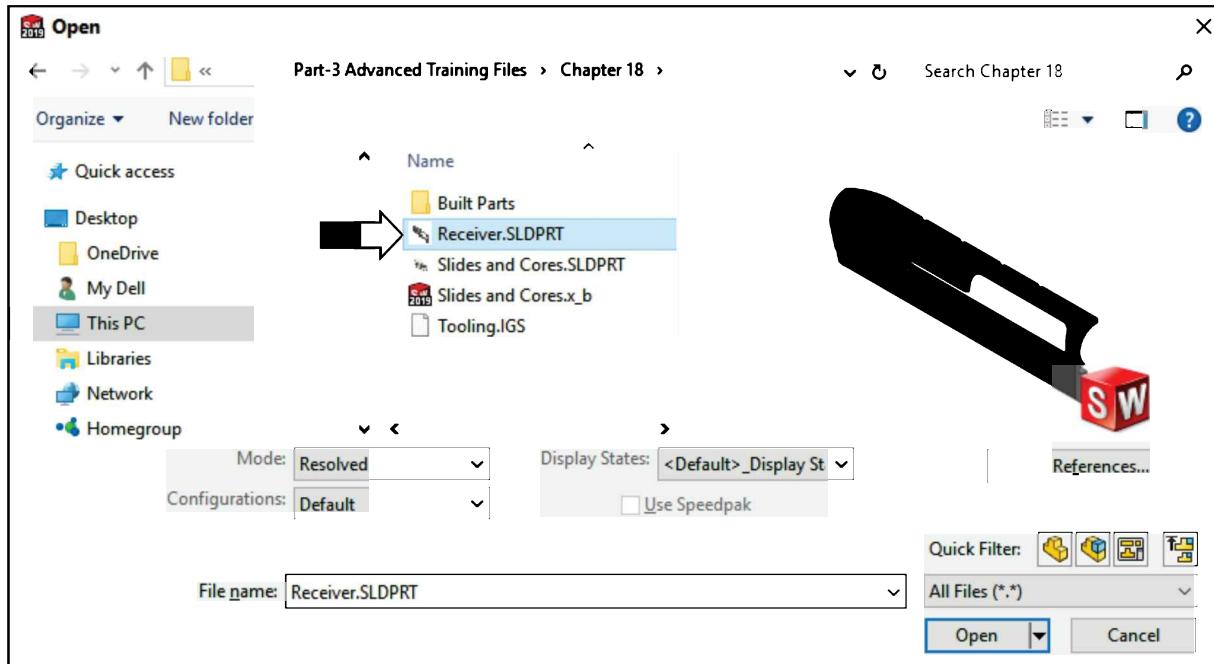


Trim Surface

1. Opening an existing part document:

Click **File / Open**.

Browse to the Training Files folder and open a part document named: **Receiver.sldprt**.

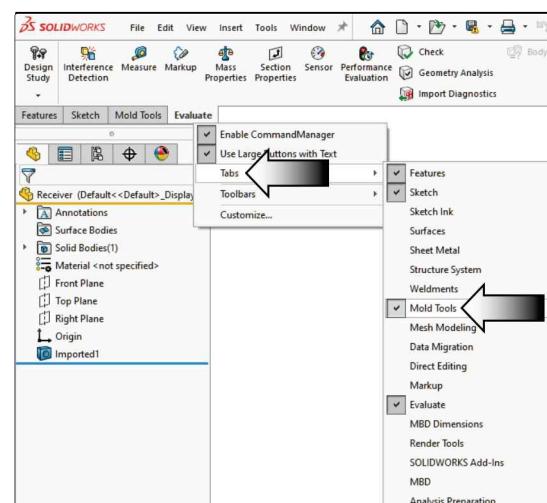


2. Enabling the Mold Tools toolbar:

Right-click one of the existing tool tabs and enable the **Mold Tools** checkbox (arrows).

For clarity, only keep the following toolbars enabled:

- * **Features**
- * **Sketch**
- * **Mold Tools**
- * **Evaluate**

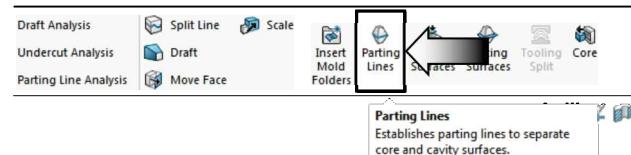


Disable the other toolbars.

NOTE: This model has already been scaled to accommodate the mold shrinkage. The draft angles also have been added to all features. The next step is to add the parting lines.

3. Creating the Parting Lines:

Switch to the new Mold Tools tab (arrow).

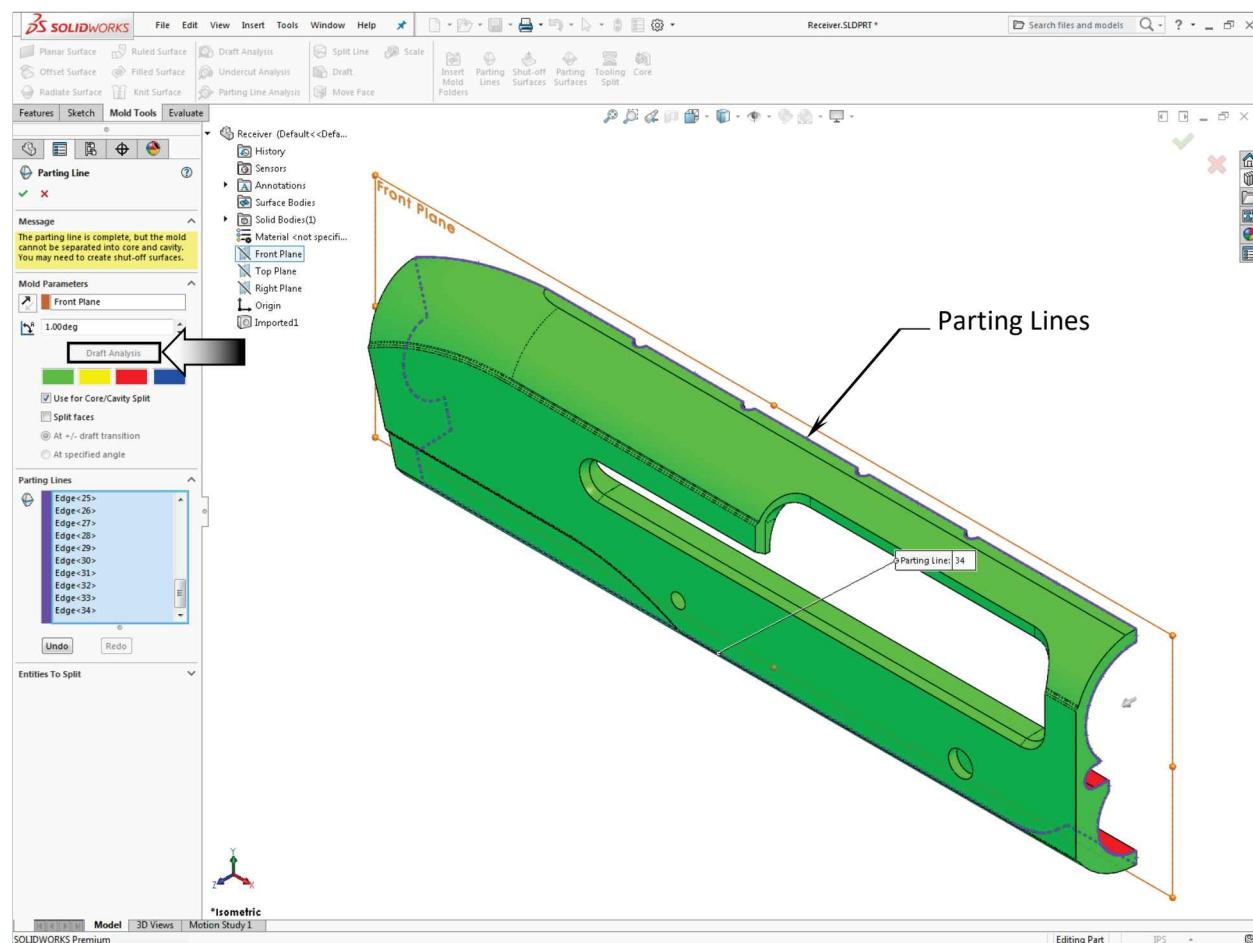


Select the Parting Lines command from the Mold Tools tab (arrow).

Parting lines lie along the edge of the molded part, between the core and the cavity surfaces. They are used to create the parting surfaces and to separate the two mold halves.

Select the Front plane from the FeatureManager tree for Direction of Pull.

Enter 1.00deg for Draft Angle and click the Draft Analysis button (arrow).



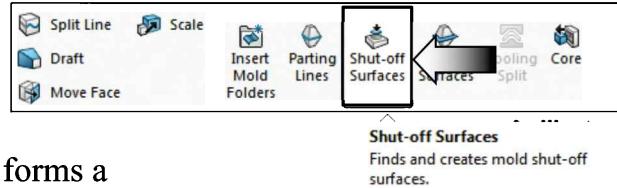
SOLIDWORKS automatically selects the edges of the model that border the two halves of the mold. The parting lines will be used to separate the surfaces between the core and the cavity.

The **Green** surfaces on the model represent the positive draft surfaces on the Cavity half, and the **Red** surfaces are negative draft surfaces on the Core half.

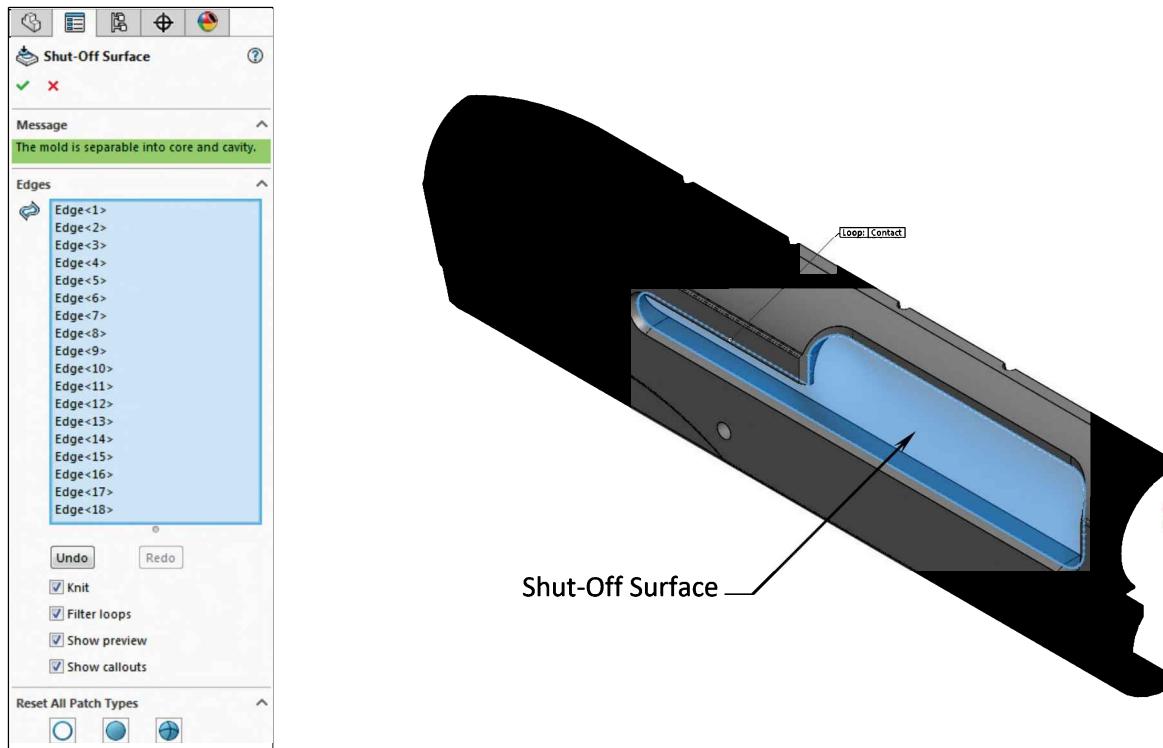
Click **OK**.

4. Creating the Shut-Off Surfaces:

A shut-off surface closes up a through hole by creating a surface patch along the edges that forms a continuous loop, or a parting line you previously created to define a loop.

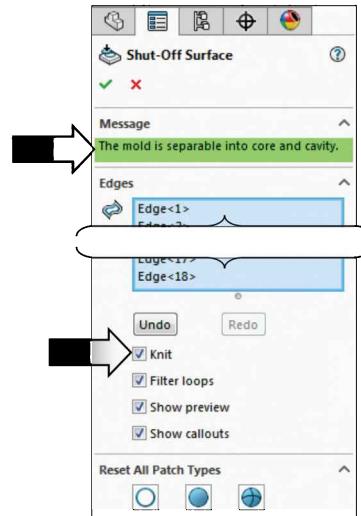


Click the **Shut-Off Surfaces** command on the **Mold Tools** tab (arrow).



SOLIDWORKS searches for any through holes and automatically creates a surface patch along the edges that form a continuous loop.

A “Green Message” appears on the Feature tree indicating the mold is separable into core and cavity.



Enable the **Knit** checkbox (arrow).

Click **OK**.

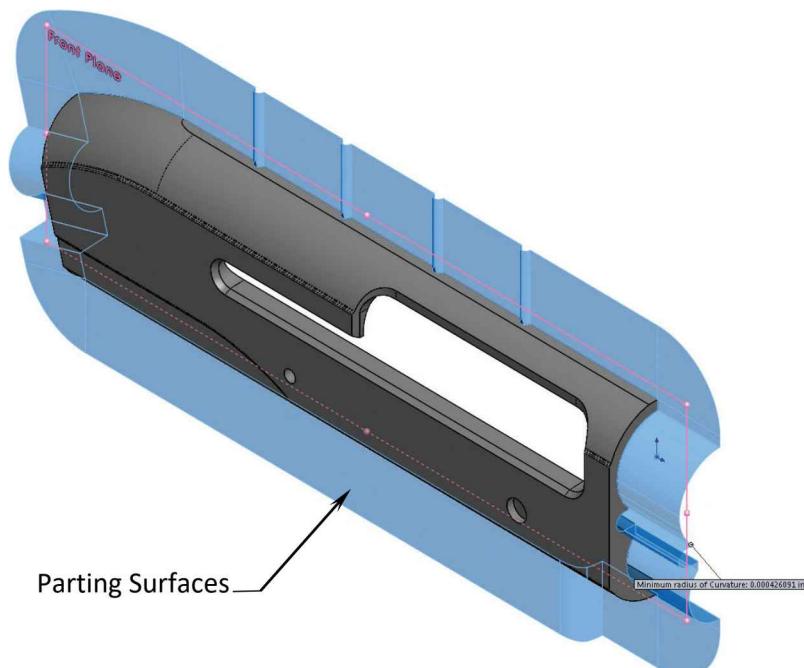
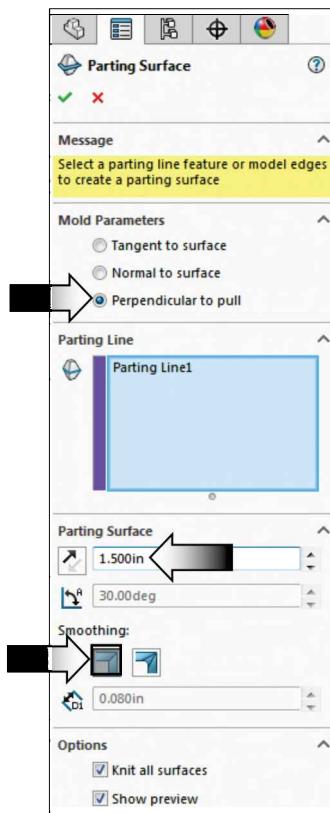
5. Creating the Parting Surfaces:



Parting surfaces are used to separate the mold cavity from the core. They must be created right after the Shut-Off Surfaces. (The Shut-Off Surface is not shown for clarity.)

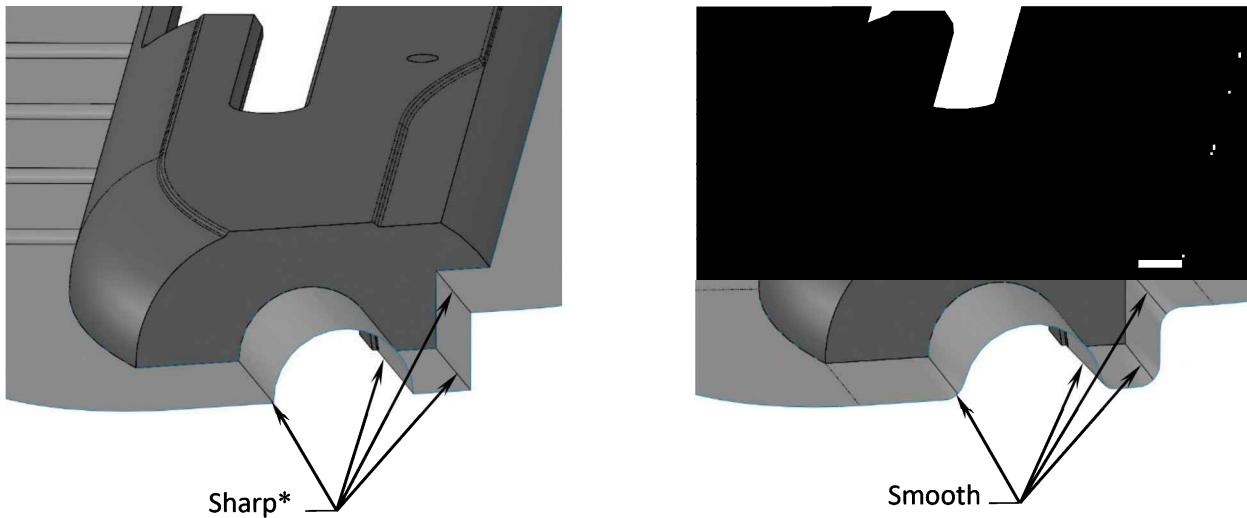
Click **Parting Surfaces** from the **Mold Tools** tab.

Select the **Perpendicular to Pull** option (arrow).



Enter **1.50in** for **Distance** (arrow).
(more on next page...)

Use the default **Sharp*** option for smoothing.



Click **OK**.

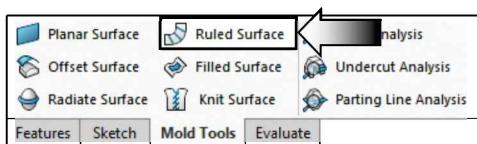
6. Creating a Ruled Surface:

To help prevent the core and cavity blocks from shifting, you can add an interlock surface. It is created along the perimeter of parting surfaces prior to inserting a tooling split in a mold part. The interlock surface can be created manually or automatically.

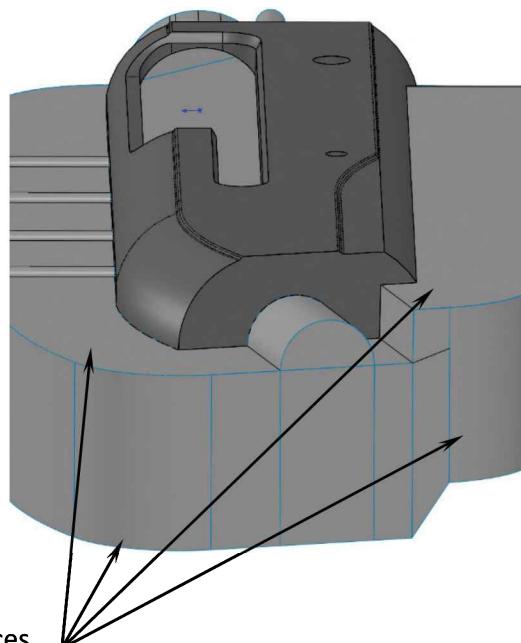
Because of the sudden changes in the parting surface geometry, the interlock surfaces will need to be created manually.

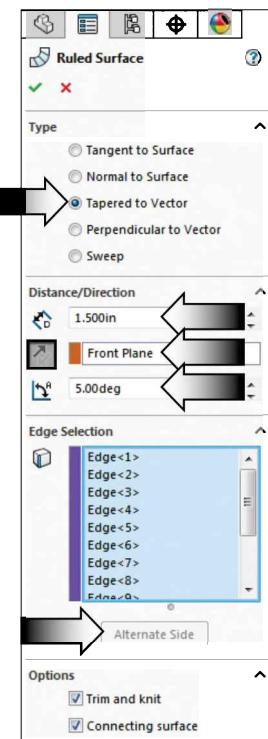
The Ruled Surface command is used to create the tapered surfaces that form the interlocks.

Select the **Ruled Surface** command from the **Mold Tools** tab (arrow).



Interlock Surfaces



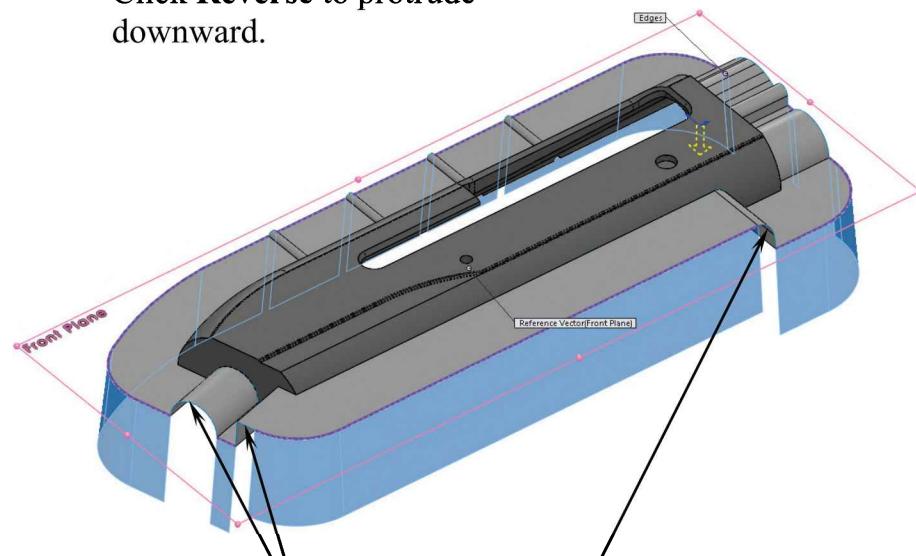


Click the **Taper to Vector** button (arrow).

Enter **1.50in** for **Distance**.

Select the **Front** plane for **Reference Vector**.

Click **Reverse** to protrude downward.



For Draft Angle,
enter **5.00deg**.

Skip these edges

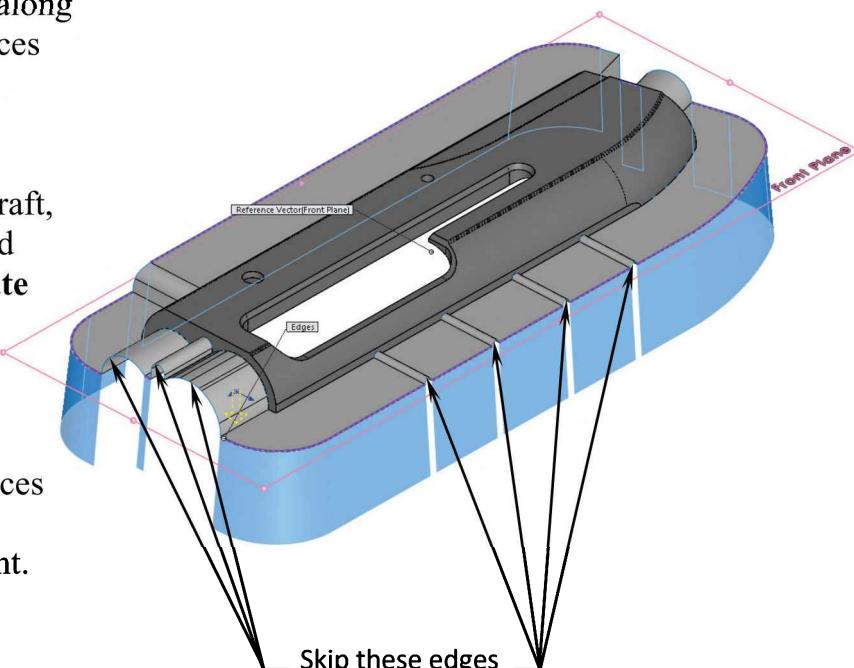
Select the **edges** along
the Parting Surfaces
but skip the ones
as indicated.

(To reverse the draft,
select an edge and
click the **Alternate
Side** button.)

Double check
your Ruled Surfaces
against the ones
shown on the right.

Skip these edges

Click **OK**.



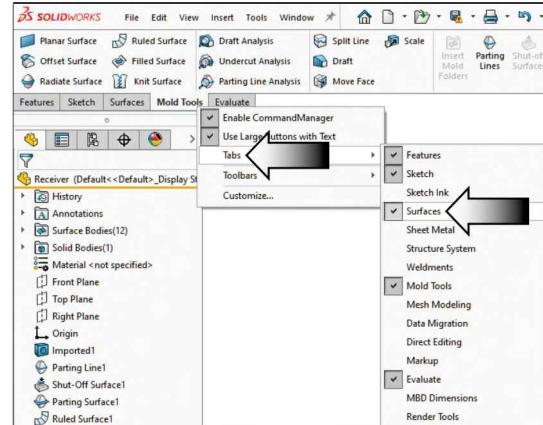
7. Creating the patches:

We will use a combination of Lofted Surface and Filled Surface commands to create the patches and close-off the openings that were left in the ruled surfaces.

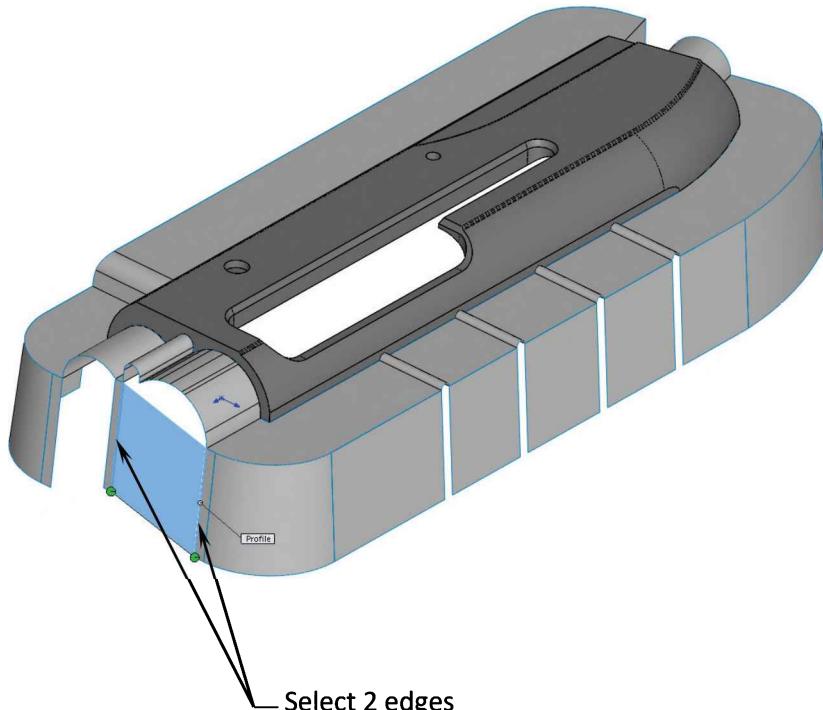
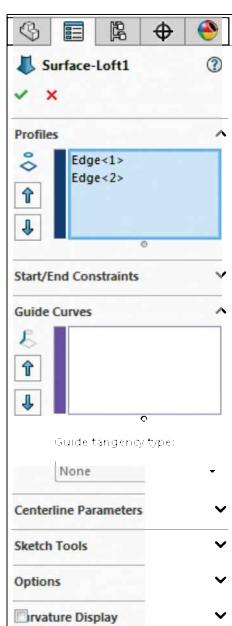
Right-click one of the tool tabs and enable the **Surfaces** toolbar (arrow).

Select **Lofted Surface**  from the **Surfaces** tool tab.

Rotate the model to a position similar to the one shown below.



Select the **2 edges** of the opening as noted.



A preview of a lofted surface appears filling the lower portion of the opening.

Click **OK**.

8. Patching with Filled Surface:

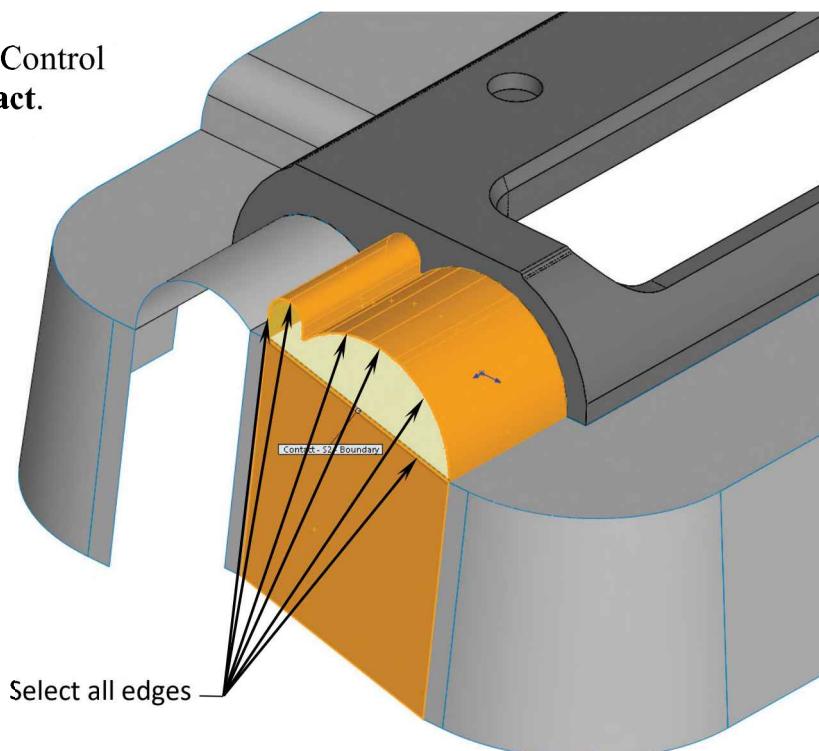
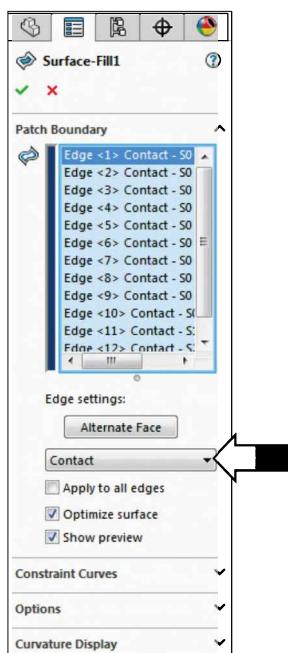
The Filled Surface command creates a surface patch with any number of sides, within a boundary defined by existing model edges, sketches, or curves, including composite curves.

Zoom in on the upper portion of the opening. We will patch it up with the Filled Surface command.

Click the **Filled Surface** command from the Surfaces toolbar.

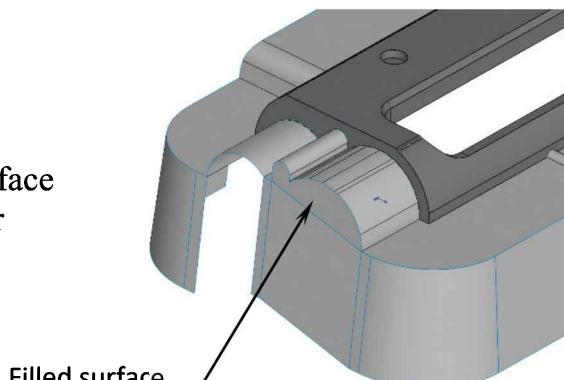
Keep the Curvature Control at the default: **Contact**.

Select all remaining edges on the upper portion of the same opening.



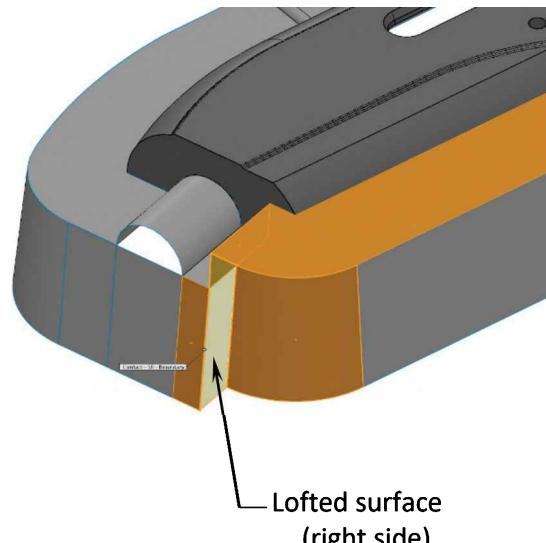
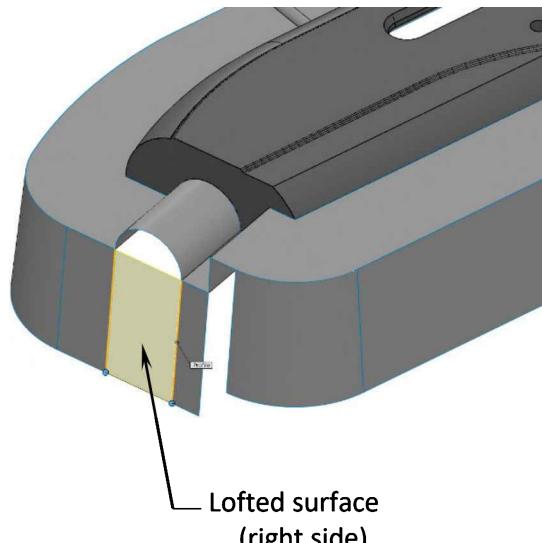
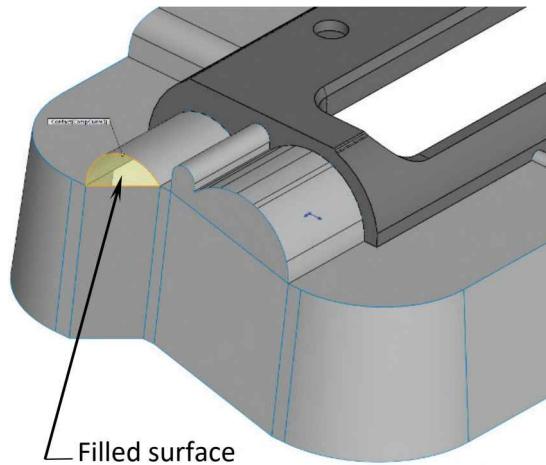
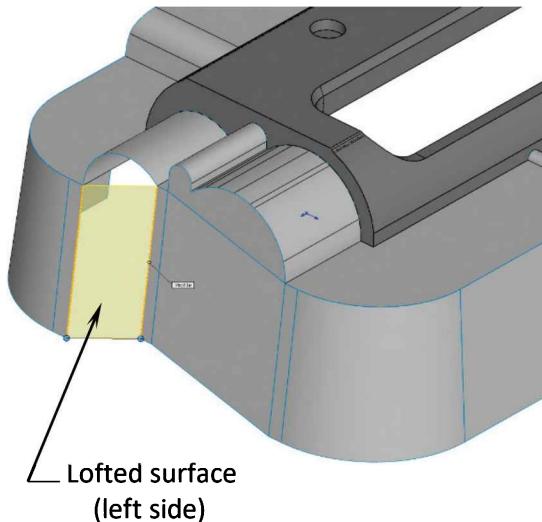
A preview of a filled surface appears, filling the upper portion of the opening.

Click **OK**.

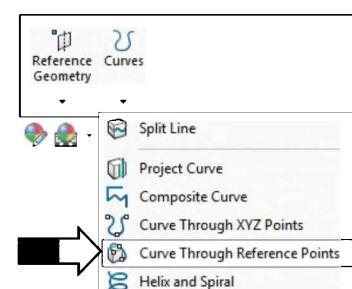
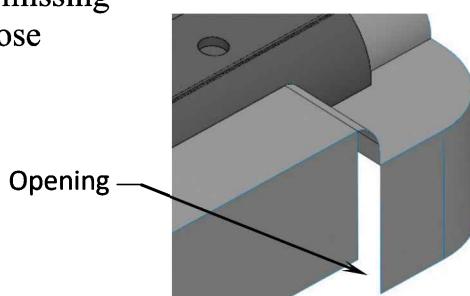


9. Patching all openings:

Repeat steps 7 and 8 and patch / fill the rest of the openings in the model.



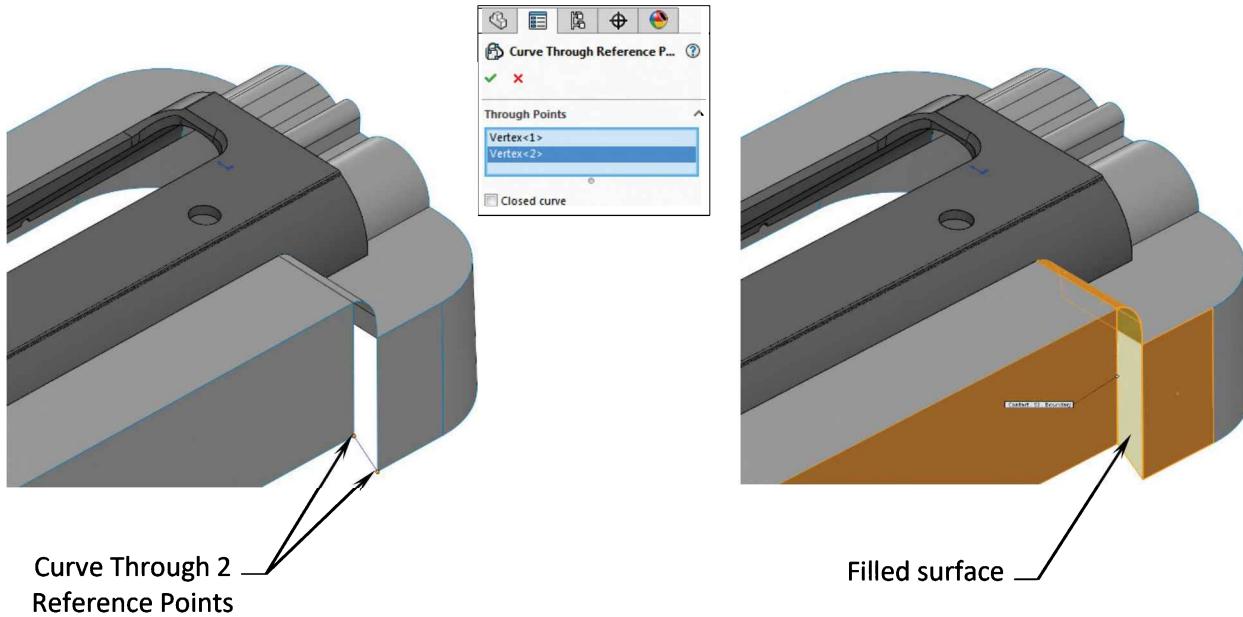
The Filled Surface command requires the boundary to be closed in order to fill or patch that area. The Curve Through Reference Points command can be used to create the missing entities in those openings.



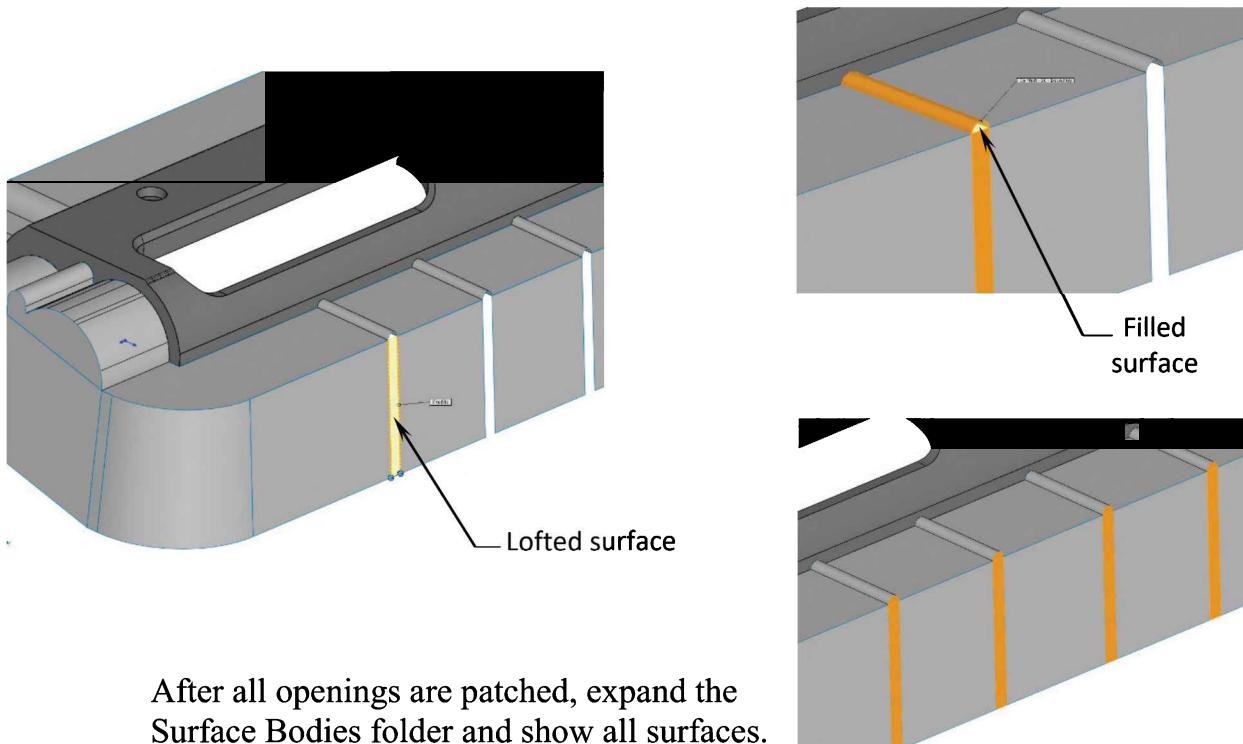
From the **Surfaces** tab, select **Curves / Curve Through Reference Points**.

Select the **2 vertices** as indicated and the preview of a curve appears.

Click **OK**.



Continue with filling / patching the rest of the openings.

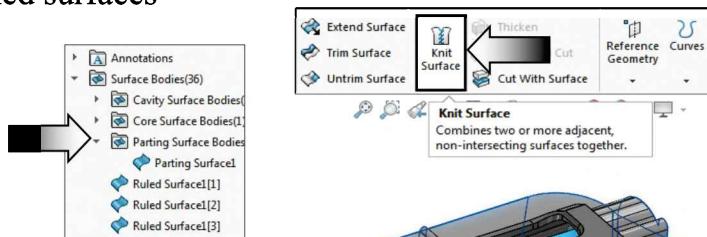


After all openings are patched, expand the Surface Bodies folder and show all surfaces.

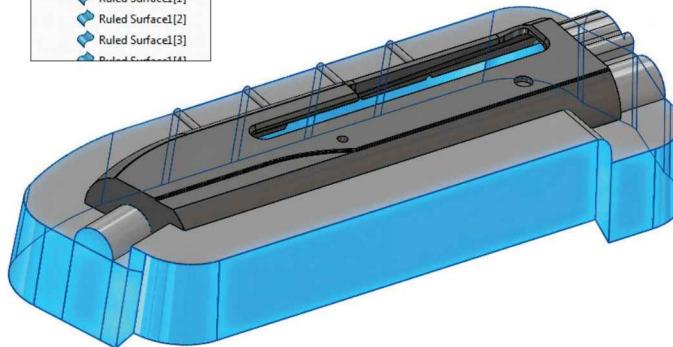
10. Knitting the surfaces:

To help select multiple surfaces more easily, we will need to knit all of the ruled surfaces and the patched surfaces into a single surface.

Click the **Knit Surface** command (arrow).



Expand the **Surface Bodies** folder and select all surfaces inside this folder (arrow).



Click **OK**.

11. Creating a new plane:

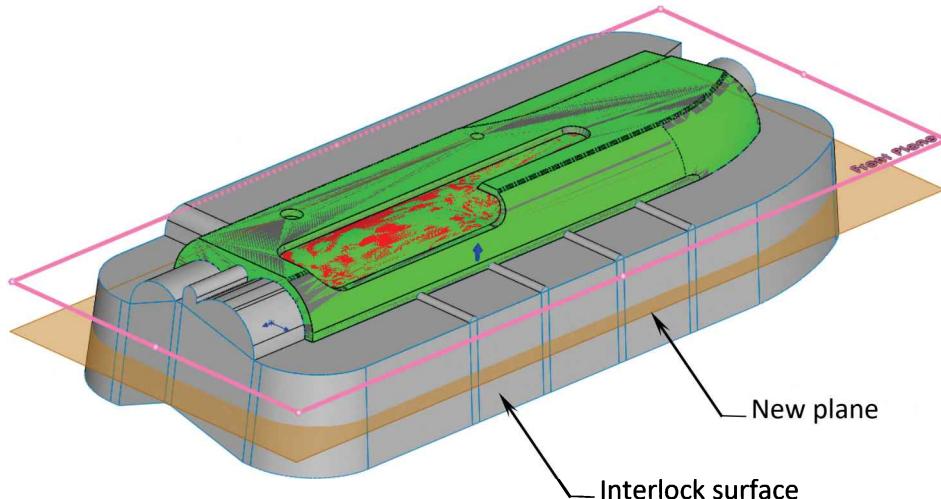
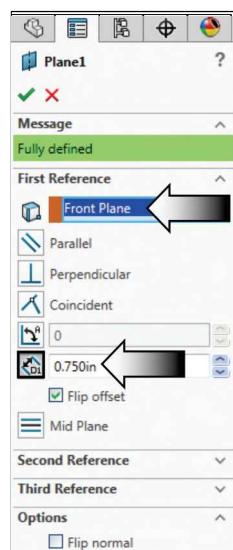
The bottom of the interlock surface is not flat. We will need to create a plane and a planar surface and use them to trim the bottom.

Click **Reference Geometry / Plane**, from the **Surfaces** tab.

Select the **Front** plane for **First Reference**.

Click the **Offset Distance** button and enter **.750in** for **Distance**.

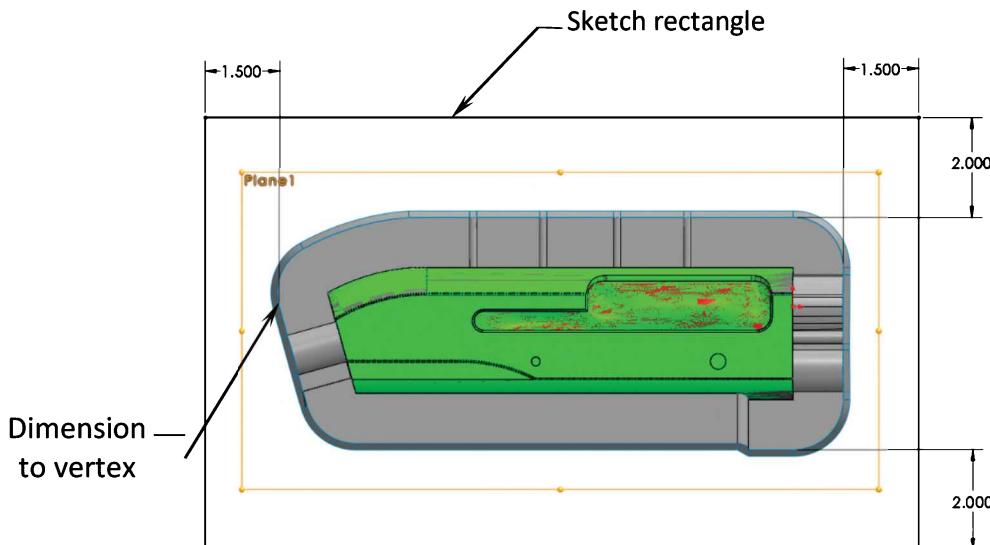
Place the new plane below the Front plane and click **OK**.



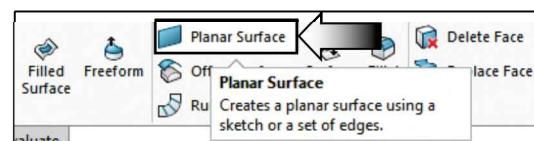
12. Creating a new sketch:

Select the new plane1 and open a **new sketch**.

Sketch a **Rectangle** around the model and add the dimensions shown to fully position it.

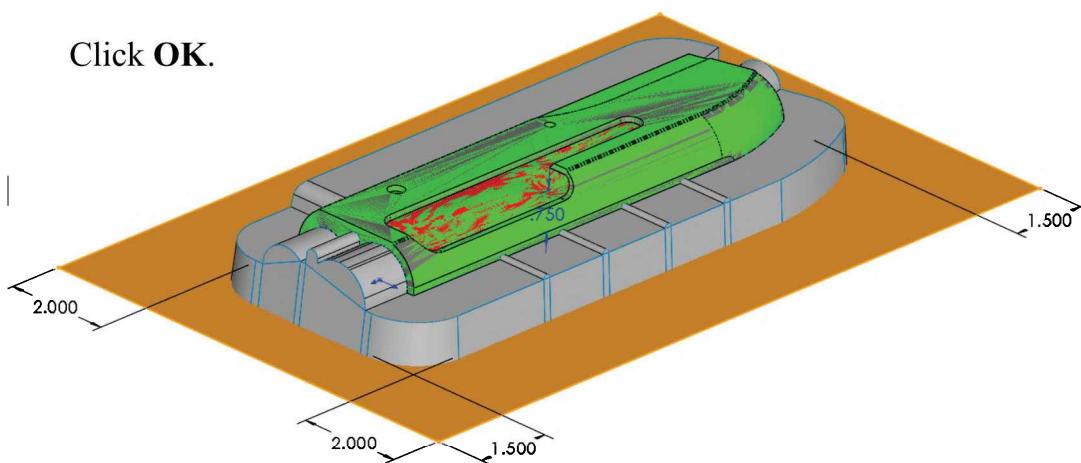


While the sketch is still active click the **Planar Surface** command from the **Surfaces** tab (arrow).

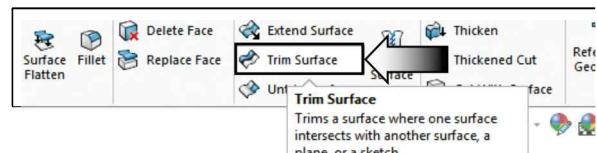


The rectangular sketch is converted into a planar surface.

Click **OK**.



13. Trimming the bottom of the ruled surface:

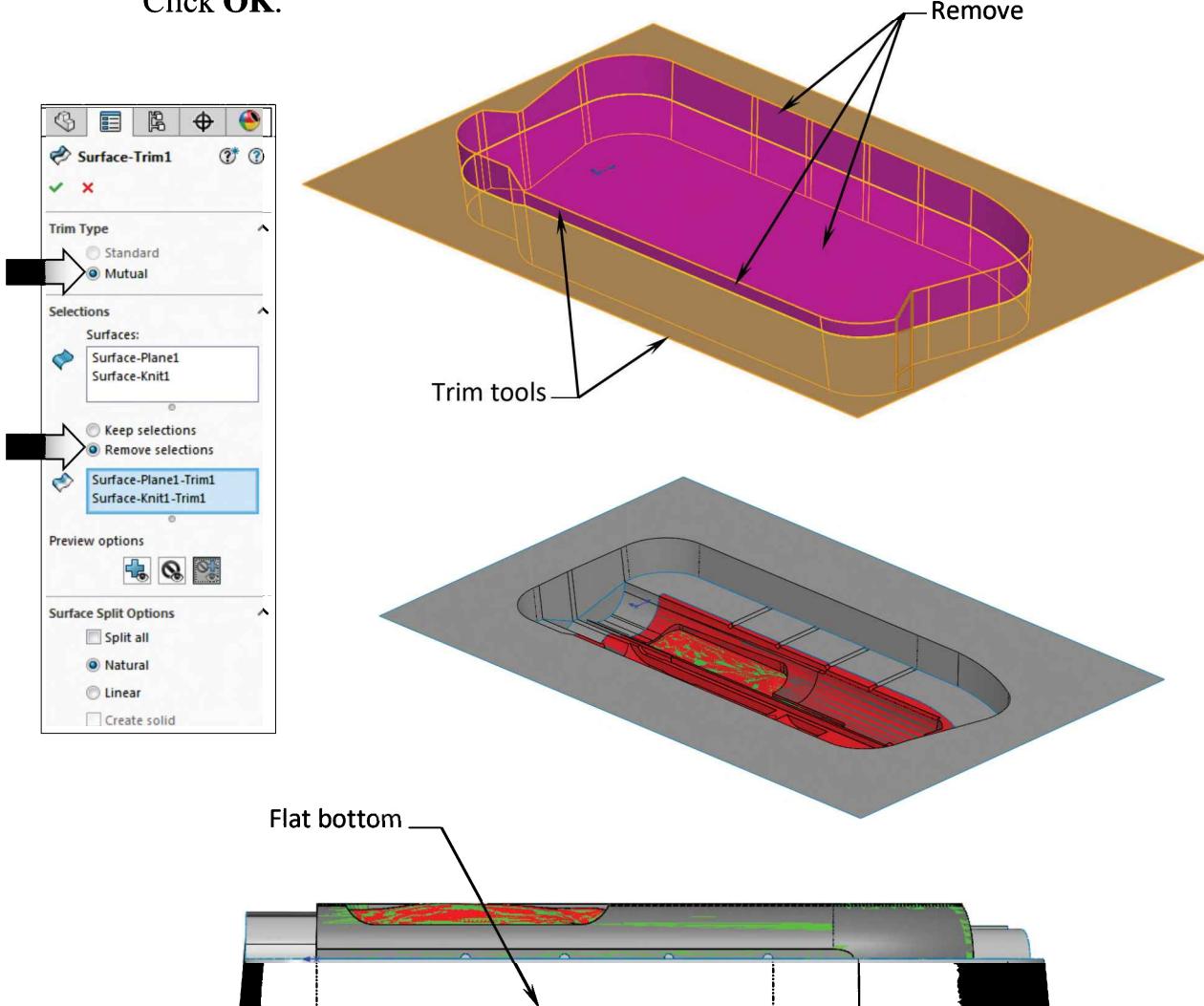


Select the **Trim Surface** command from the **Surfaces** tab.

Click the **Mutual** trim option (arrow). Select the **Planar Surface** and all of the surfaces along the perimeter of the Interlock Surface.

Click the **Remove Selections** options and select the inside of the Interlock Surface plus all of its surfaces along the perimeter as indicated.

Click **OK**.



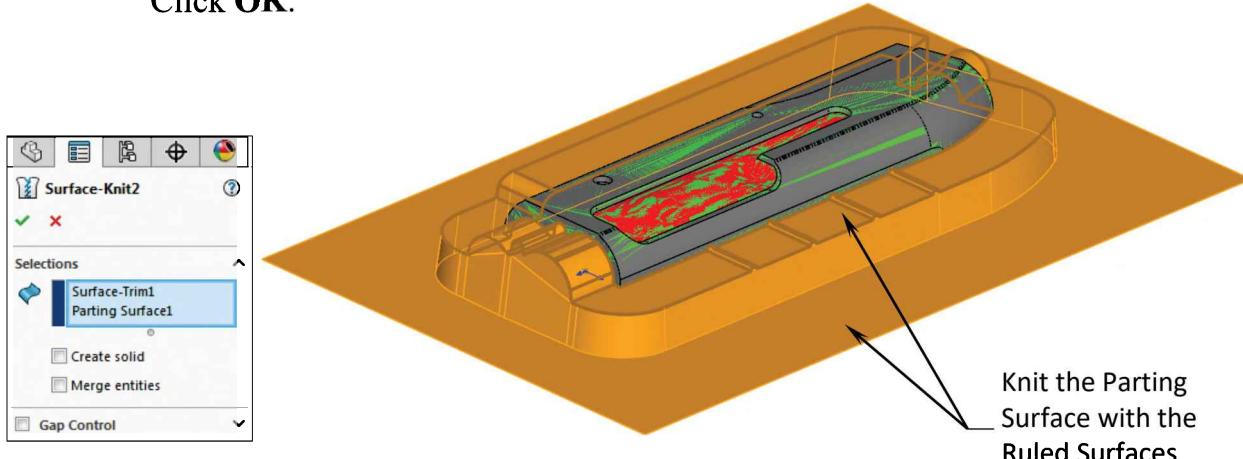
The resulting Trimmed Surface. Change to the side view to verify the trim. The bottom of the interlock surface should be flat at this point.

14. Knitting the surfaces:

Select the **Knit Surface** command from the **Surfaces** tab.

Expand the **Surfaces Bodies** folders. Select the **Parting Surface** and the **Surface-Trim1** from the Surface Bodies folder.

Click **OK**.



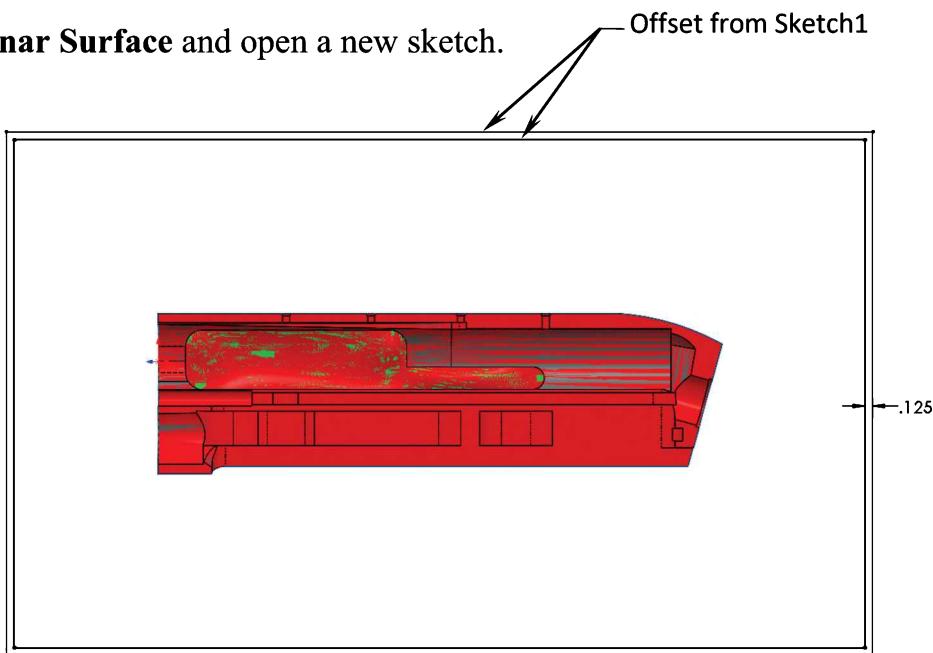
(Skip this step if SOLIDWORKS knits the surfaces automatically.)

15. Creating the tooling split sketch:

Select the **Planar Surface** and open a new sketch.

Locate the **Sketch1** under the Planar-Surface and Show it.

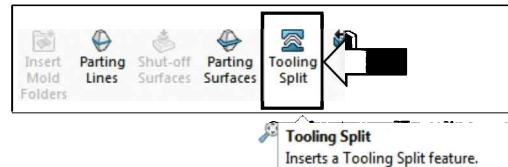
Create an offset of **.125in** (inside) from Sketch1.



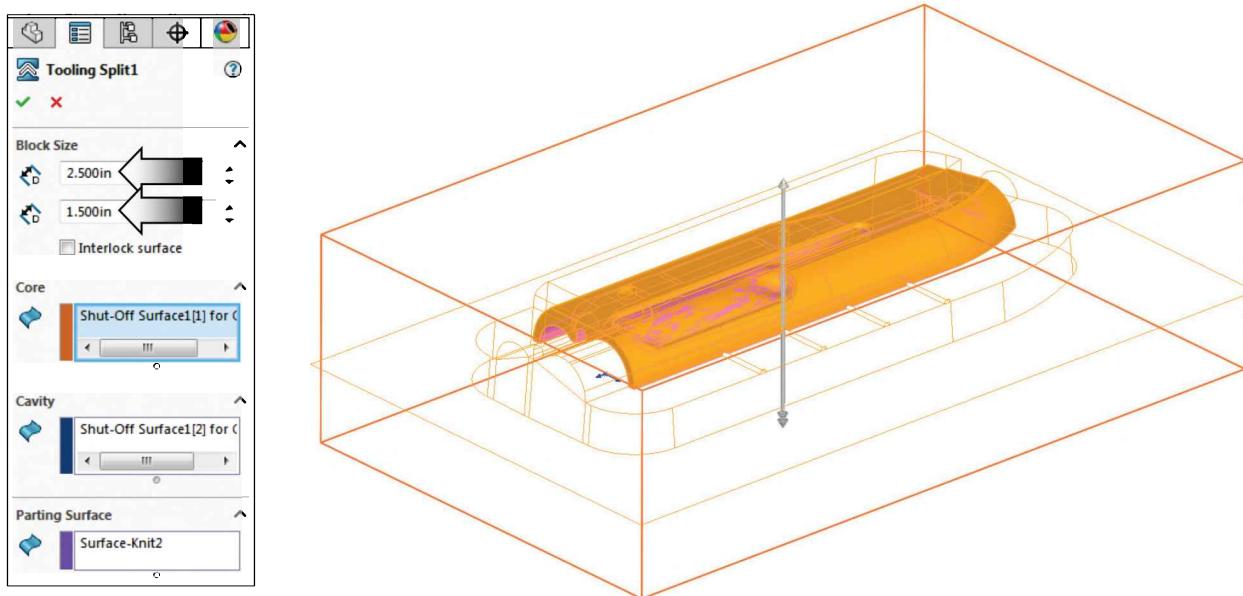
Exit the sketch and click **Tooling Split**. The Tooling Split properties appears. Select the rectangular sketch, if prompted.

For Block Size, enter the following:

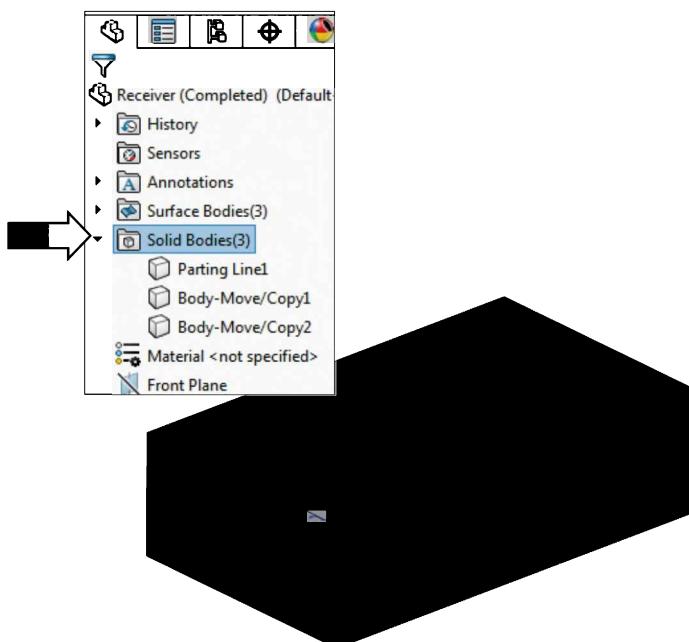
Upper block: 2.500in
Lower block: 1.500in



Click **OK**.



There are 3 solid bodies on the FeatureManager tree: the Original part, the Upper Mold Block, and the Lower Mold Block.



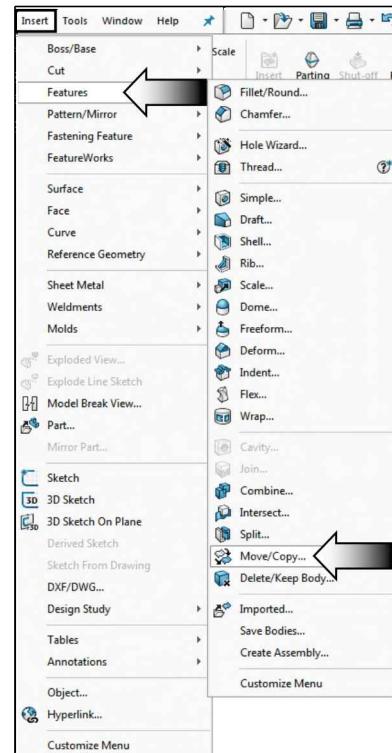
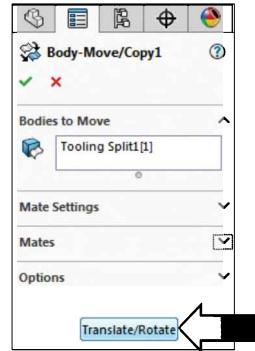
We will separate them in the next step.

16. Separating the solid bodies:

Select Insert / Features / Move-Copy.

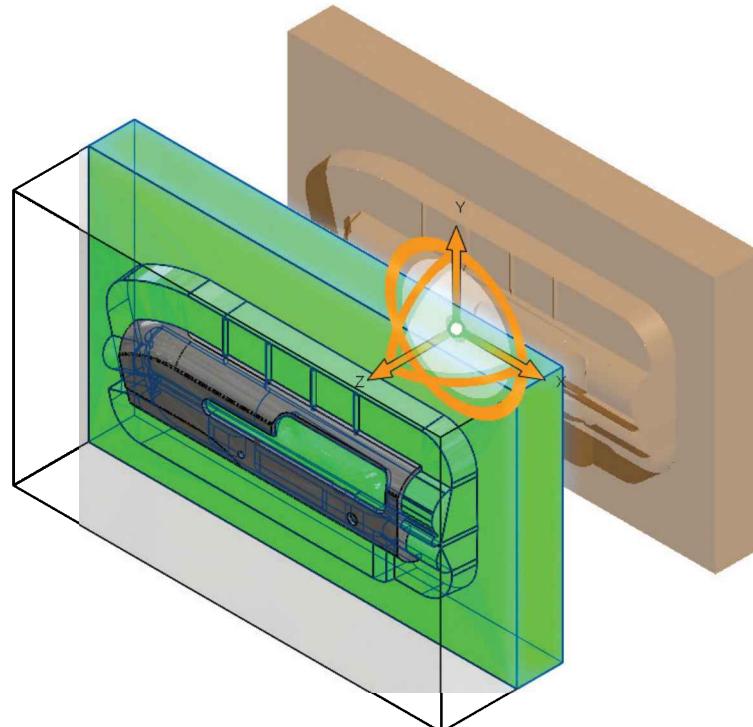
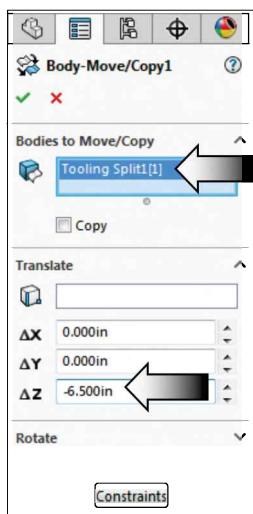
Click the Translate / Rotate button in the Options section.

For Bodies to Move,
Select the right half
of the mold (the core).



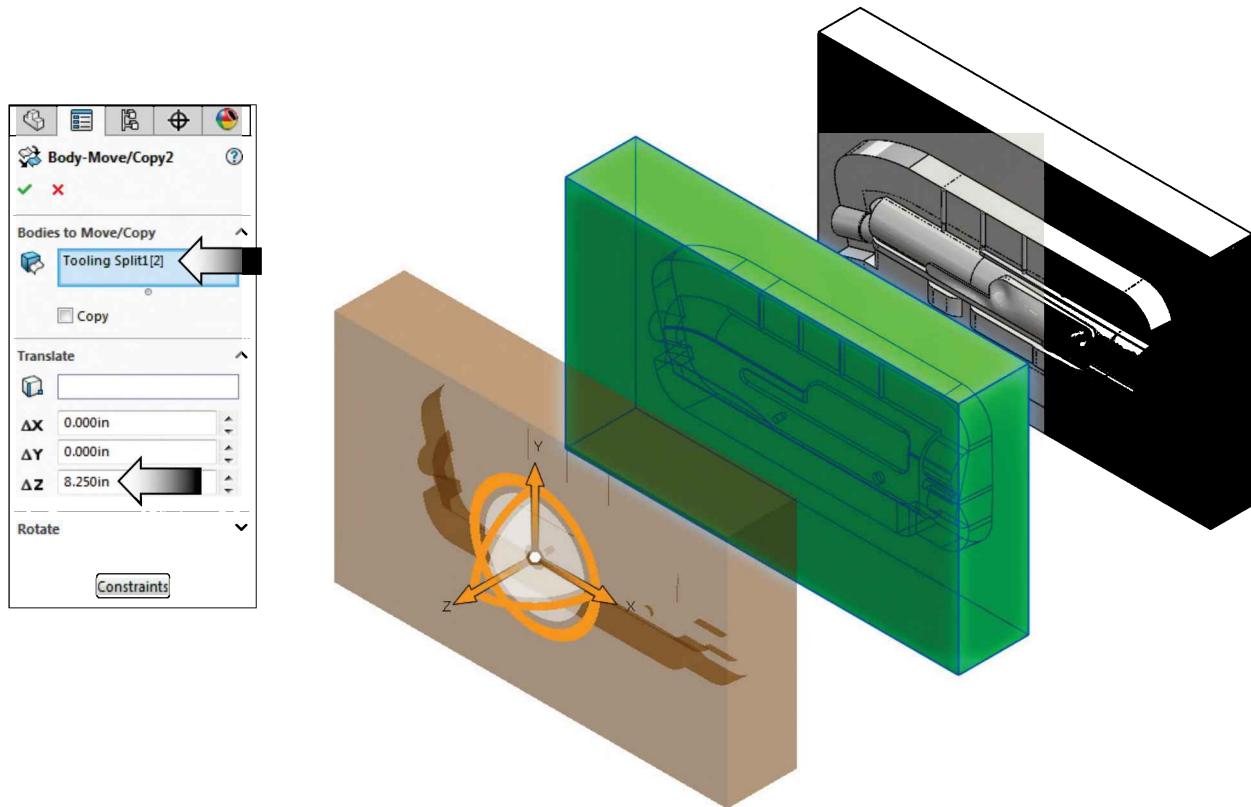
Enter **-6.500in** for Delta Z distance.
The core half moves outward.

Click **OK**.



Repeat step number 16 and move the left half of the mold (the cavity).

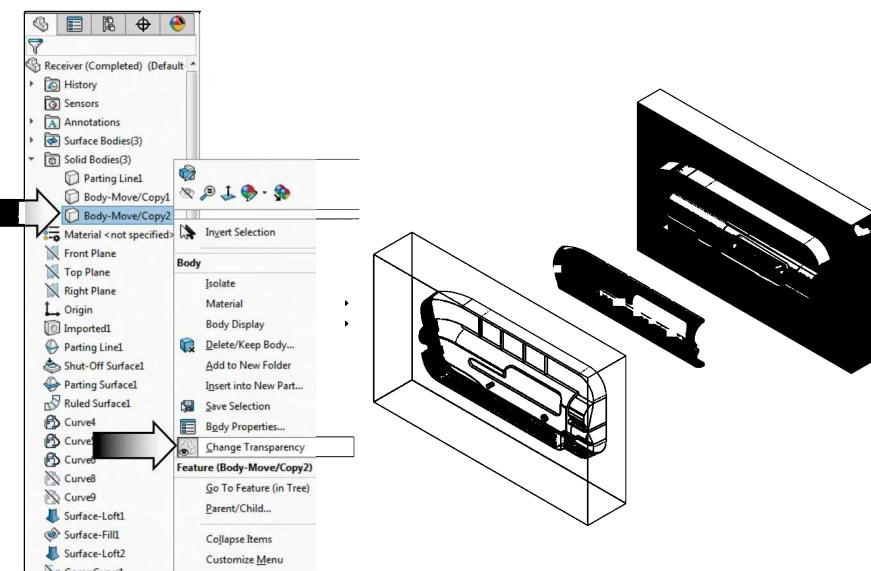
Move the cavity block about the **Delta Z** direction; use a distance of **8.250in**.



17. Making the body transparent:

From the FeatureManager tree, Expand the Solid Bodies folder.

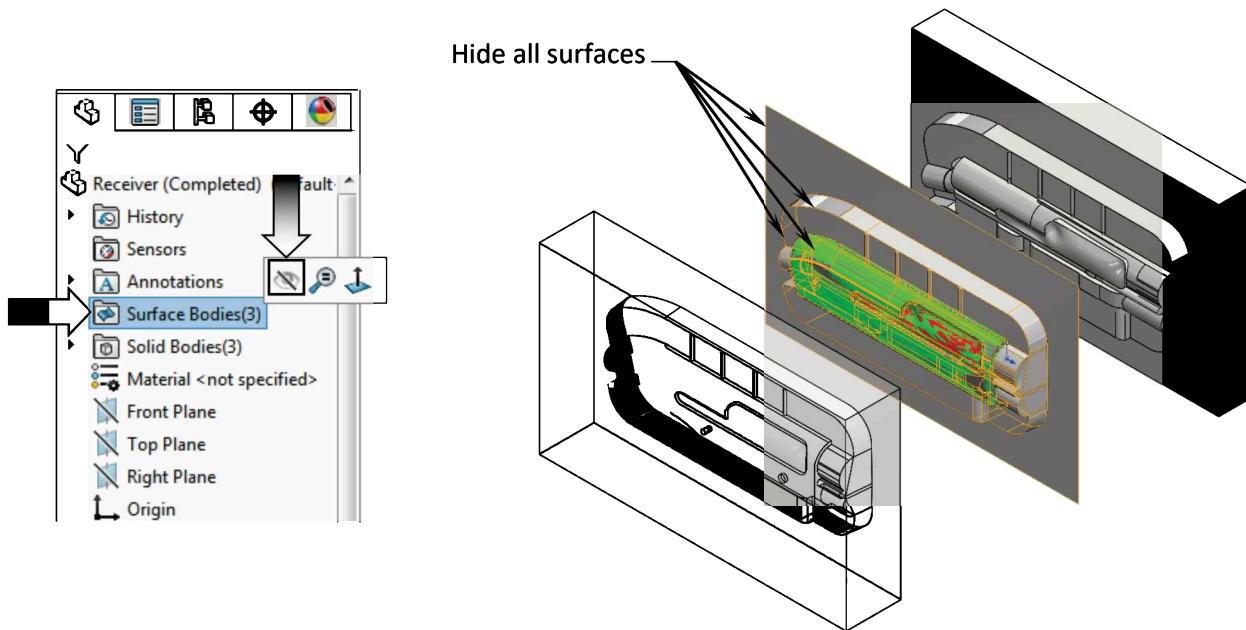
Right-click on the Cavity body and select:
Change Transparency (arrow).



18. Hiding the surface bodies:

The Core, Cavity, and other surfaces are still visible in the graphics making it difficult to see the molded part. We will need to hide them.

From the FeatureManager tree, right-click the **Surface Bodies** folder and select: **Hide**.

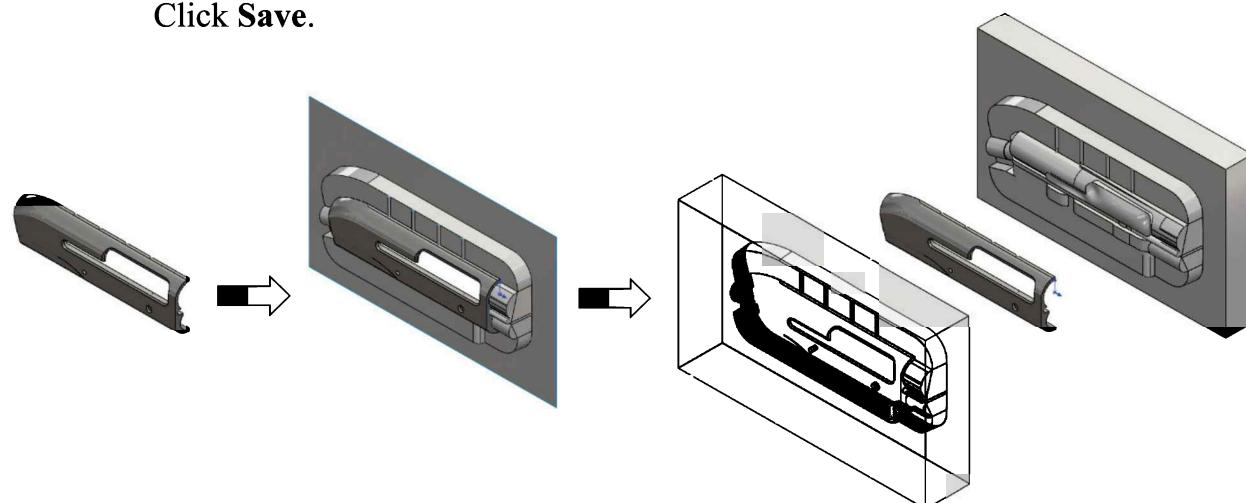


19. Saving your work:

Select **File / Save As**.

Enter **Mold_Manual Creation.sldprt** for the file name.

Click **Save**.



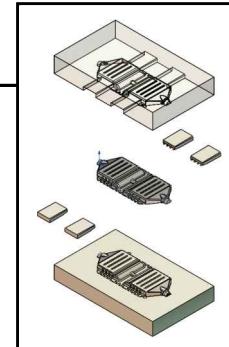
Mold Tooling Design

Creating Slides and Cores

Mold Tooling Design

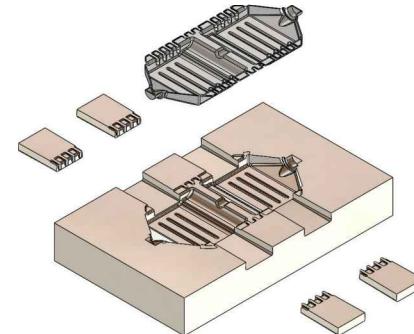
Creating Slides and Cores

Many times undercuts cannot be avoided in plastic part designs. Features such as slots, triggers, locks, and latches, etc. are often seen in plastic parts even though they may be more expensive to design and manufacture.



The Undercut Analysis tool can assist you with finding and visualizing trapped areas that may prevent the part from ejecting from the mold. These areas will need additional tooling such as lifters and cores to allow the part to be released from the tooling using the primary direction of pull.

A Slide or Core is a piece of tooling that slides out of the mold from the side, perpendicular to the direction that the part is ejected from the mold.



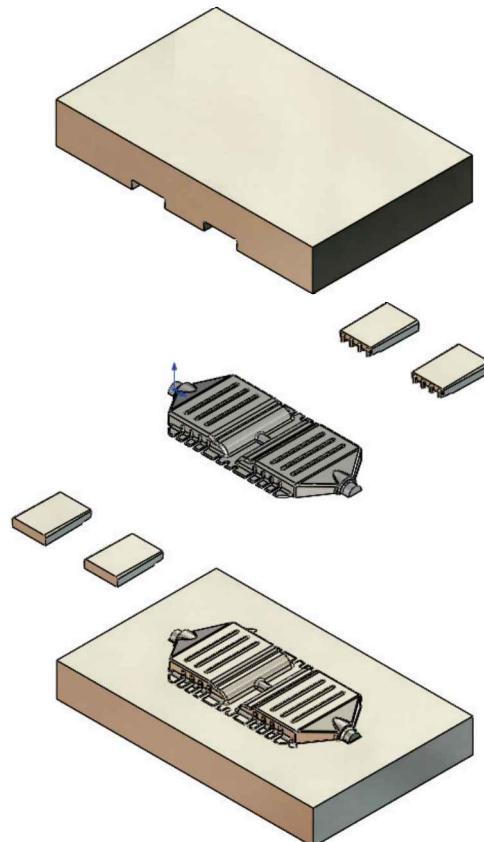
A sketch is used to define the shape and location of the core and it is normally drawn directly on the sides of the core or cavity block. Generally the sketch plane is parallel or perpendicular to the direction in which the side core travels away from the plastic part.

The Core command is used to extrude the sketch into a separate solid body and at the same time, it is subtracted from the core or cavity body. The Core command works like the Split command; they both divide a body into two or more bodies.

Additional tooling like Lifters, Core Pins, and Ejector Pins may be required to complete the mold design, but this lesson will discuss the use of the standard mold tools to automate the creation of the Core and Cavity as well as the use of the Core command.

Mold Tooling Design

Creating Slides and Cores



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI
Units: INCHES – 3 Decimals

Tools Needed:



Scale



Parting Lines



Parting Surfaces



Tooling Split



Extruded Boss/Base



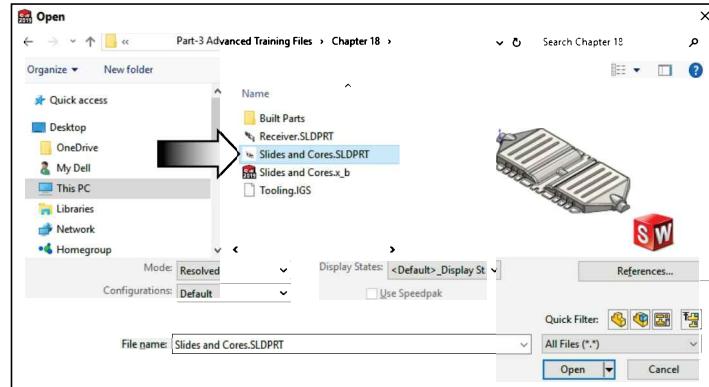
Core

1. Opening a part document:

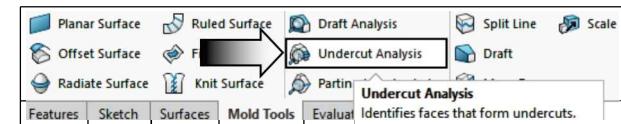
Select File / Open.

Browse to the Training Files folder and open a part document named:
Slides and Cores.sldprt

One of the first things to do is to examine the model for potential problems that might prevent the core and cavity from separating.



2. Analyzing the undercuts:

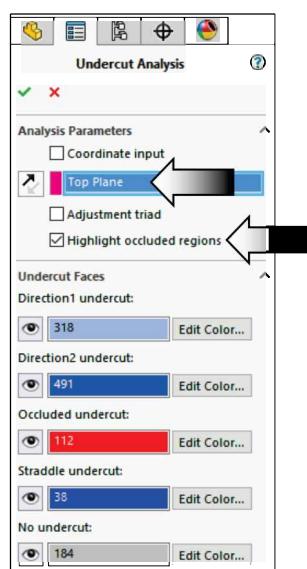


The Undercut Analysis tool finds trapped areas in a model that cannot be ejected from the mold. These areas require a side core. When the main core and cavity are separated, the side core slides in a direction perpendicular to the motion of the main core and cavity, enabling the part to be ejected.

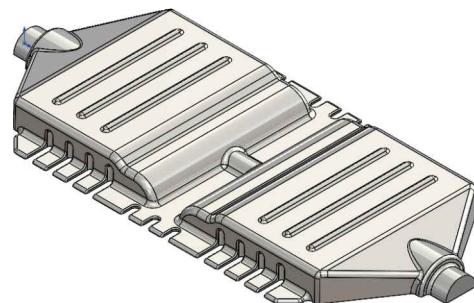
Undercut Analysis works only on solid bodies, not surface bodies.

Right-click on one of the tabs and enable the **Mold Tools**.

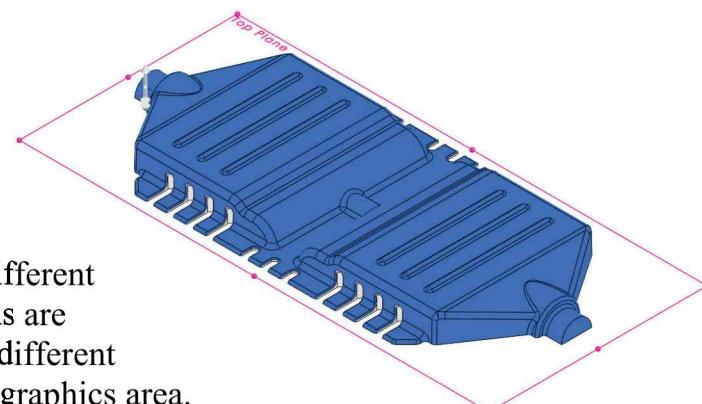
Select the **Undercut Analysis** command.



For Direction of Pull, select the **Top** plane.



Enable the **Highlight Occluded Regions** checkbox.



Faces with different classifications are displayed in different colors in the graphics area.

The faces are classified as follows:

Analysis Parameters explained:

- * **Direction1 Undercut:** Faces that are not visible from above the parting line.
 - * **Direction2 Undercut:** Faces that are not visible from below the parting line.
 - * **Occluded Undercut:** Faces that are not visible from above or below the parting line.
 - * **Straddle Undercut:** Faces that have undercuts in both directions.
 - * **No Undercut:** No Undercut.
-

- * **Direction of Pull:** (Not required if you select a Parting Line below.) Select a planar face, a linear edge, or an axis to define the draw direction, or select Coordinate input and set conditions along the X, Y, and Z axes.
- * **Parting Line:** Faces above the parting line are evaluated to determine if they are visible from above the parting line. Faces below the parting line are evaluated to determine if they are visible from below the parting line. This identifies depressions in the wall of the part that require a side core, and also helps you to identify sections of the parting line that you can modify to avoid the need for side cores.
- * **Adjustment Triad:** Manipulates the direction of pull to help you visualize ways to avoid or minimize problems with undercut regions. When you drag the rings of the triad in the graphics area, the direction of pull changes, face colors update dynamically, and the following read-only values appear in the PropertyManager:

Angle with X axis
Angle with Y axis
Angle with Z axis

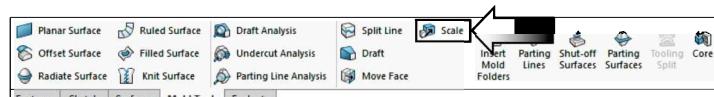
- * **Highlight Occluded Regions:** For faces that are only partially occluded, the analysis identifies those regions of the face that are occluded and those that are not. With this option cleared, the

analysis identifies the entire face as being occluded.

- * **Undercut Faces:** Faces with different classifications are displayed in different colors in the graphics area. The results update in real time when you change the direction of pull.

Click **Cancel**  to close the Undercut Analysis. A side core will be created to release the trapped areas in the mold.

3. Scaling the part:

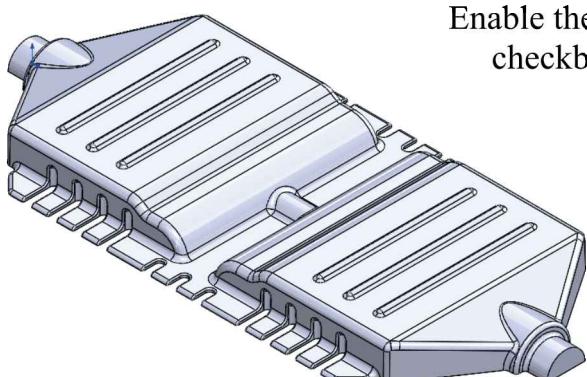


The Scale feature scales only the geometry of the model. It does not scale dimensions, sketches, or reference geometry. To temporarily restore the model to its un-scaled size, you can roll-back or suppress the Scale feature.

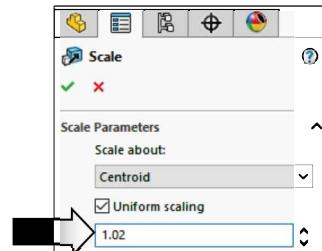
Use the Scale tool to account for the shrink factor when plastic cools. For irregular shaped parts and glass filled plastics, you can specify nonlinear values.

Click the **Scale** command on the **Mold Tools** toolbar.

For Scale About, use the default **Centroid** option.

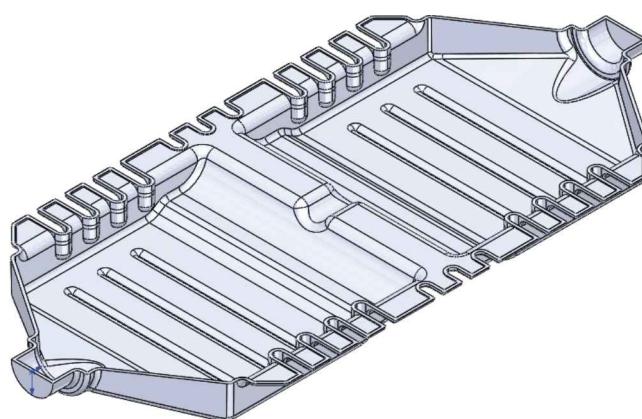


Enable the **Uniform Scaling** checkbox.



For Scale Factor, enter **1.02%** (2% larger).

Click **OK**.

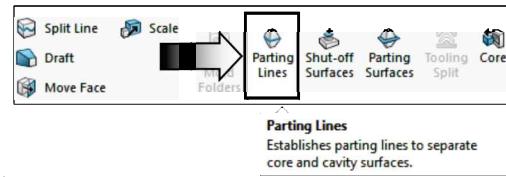


A Scale feature is like any other features listed in the FeatureManager design tree: it manipulates the geometry, but it does not change the definitions of features created before it was added.

4. Creating the parting lines:

The parting lines lie along the edge of the molded part, between the core and the cavity surfaces. They are used to create the parting surfaces and to separate the surfaces.

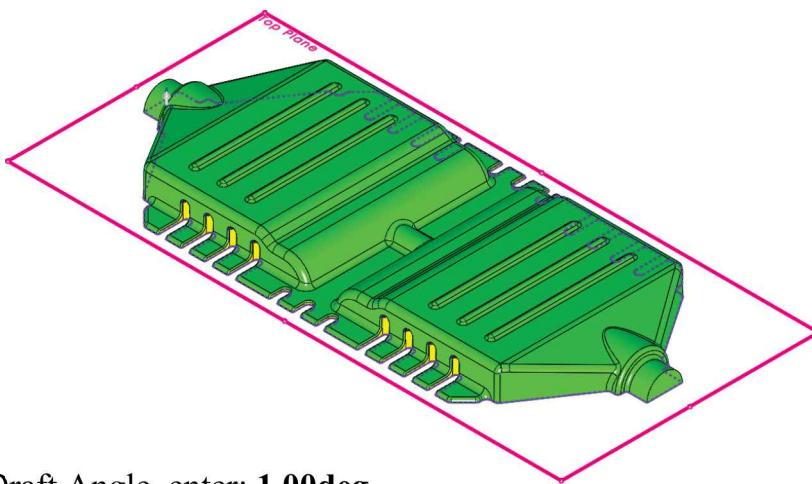
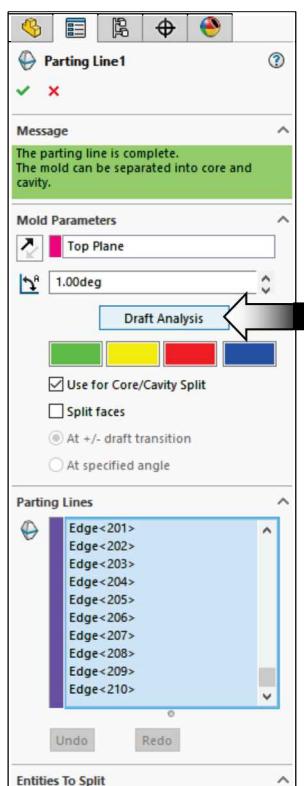
The parting line is created after the model is scaled and proper draft is applied.



Parting Lines
Establishes parting lines to separate core and cavity surfaces.

Select the **Parting Lines** command from the Mold Tools toolbar (arrow).

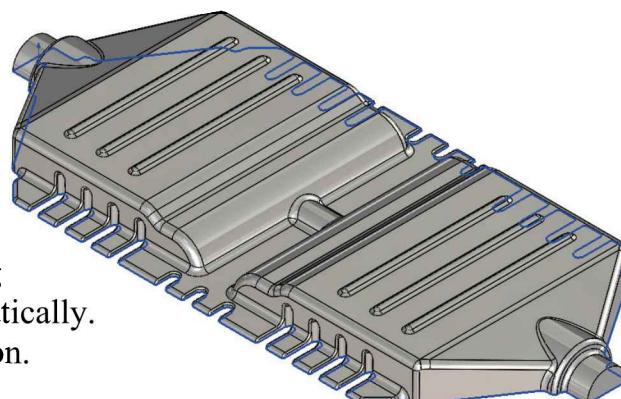
For Direction of Pull, select the **Top** plane (arrow).



For Draft Angle, enter: **1.00deg**.

Click the **Draft Analysis** button (arrow).

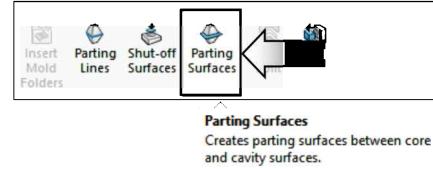
The model includes a chain of edges that runs between positive and negative faces and the parting line segments are selected automatically. They are listed in the Edges section.



Click **OK**.

5. Creating the parting surfaces:

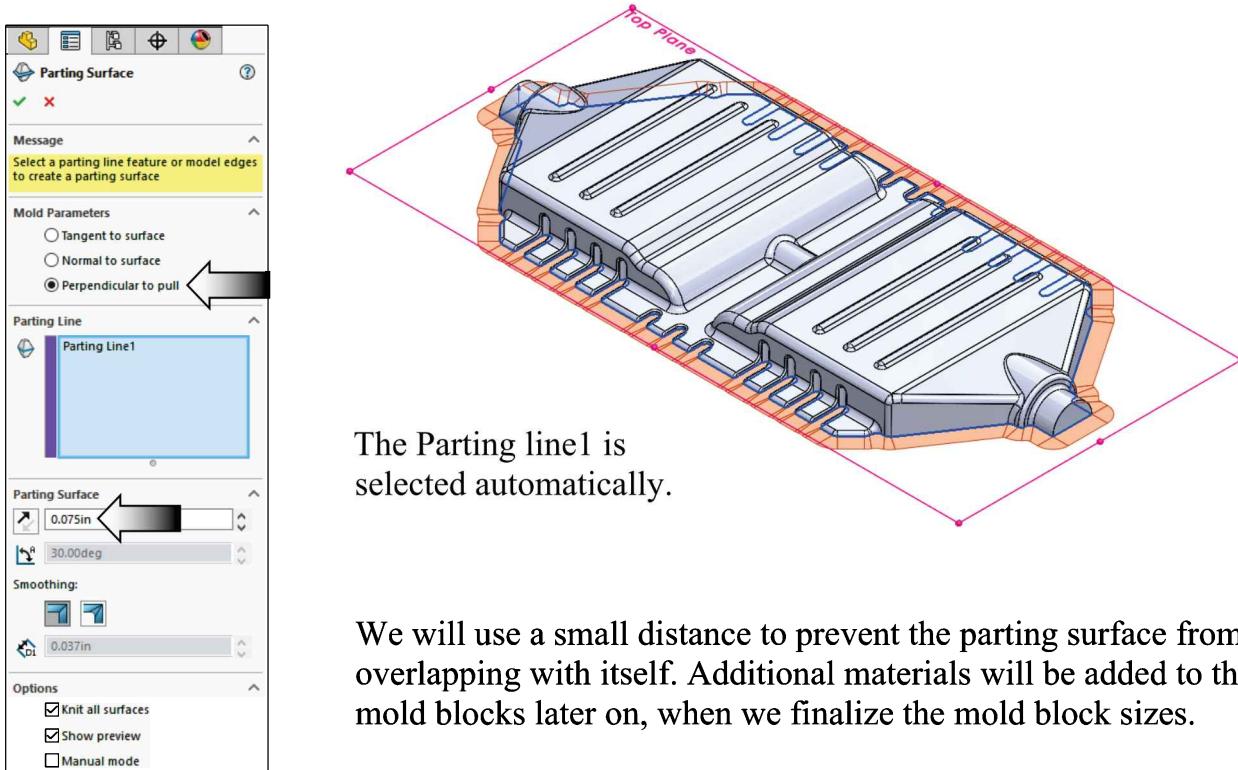
The Parting Surfaces split the mold cavity from the core. Create the parting lines and shut off surfaces before creating parting surfaces.



The Shut-Off surfaces is used to shut-off the openings of the through holes in the plastic parts. Since this model does not have any through holes, we can skip this step and move on to creating the parting surfaces.

Select the **Parting Surfaces** command (arrow).

For Mold Parameters, select: **Perpendicular to Pull** (arrow).



For Distance, enter **.075in.**; also enable the **Knit All Surfaces** checkbox.

Click **OK**.

6. Creating a new sketch:

Open a **new sketch** on the parting surfaces as noted. This sketch will be used in the next step to create the tooling split.

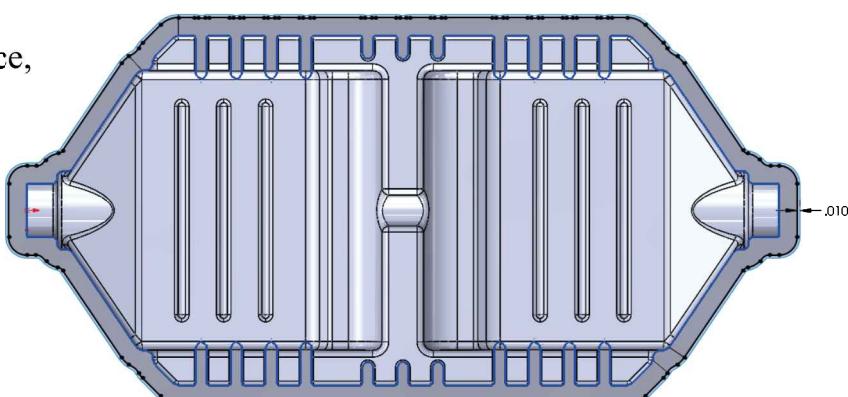
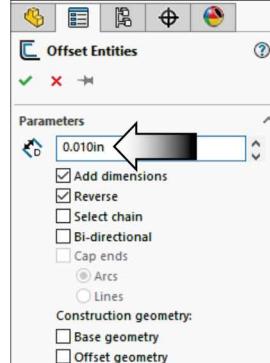
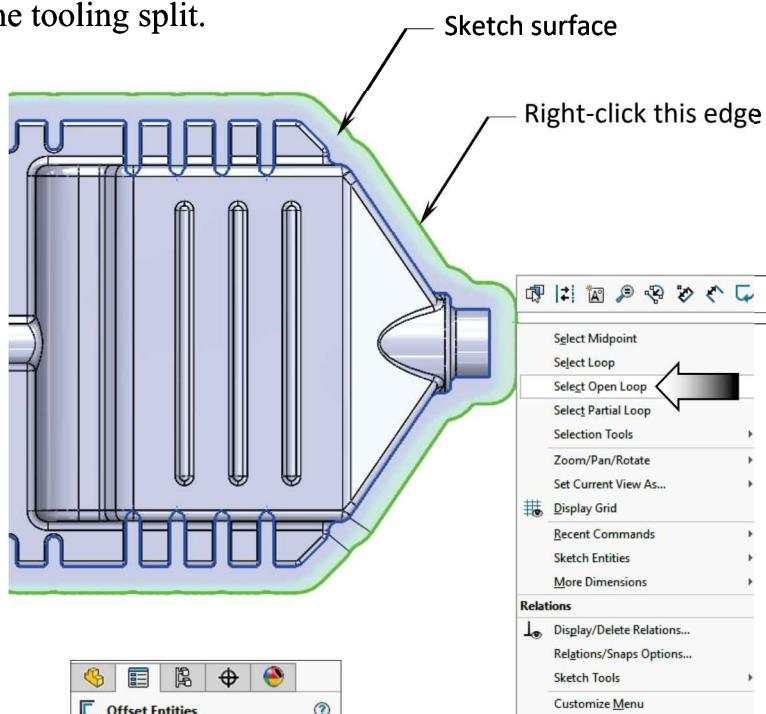
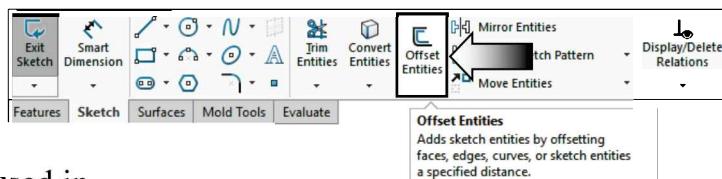
Right-click on one of the outer edges of the parting surfaces and pick: **Select Open Loop** (arrow).

The sketch that is used for tooling split should be smaller than the parting surfaces. We will use the **Offset Entities** command to create a new sketch that is slightly smaller than the parting surfaces.

Make sure all outer edges of the parting surfaces are still highlighted; click the **Offset Entities** command.

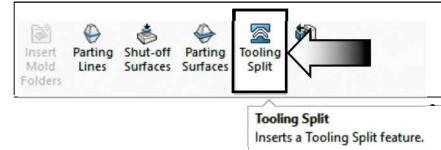
For Offset Distance, enter **.010in**. and click **Reverse** to place the entities on the inside of the parting surfaces.

Click **OK**.



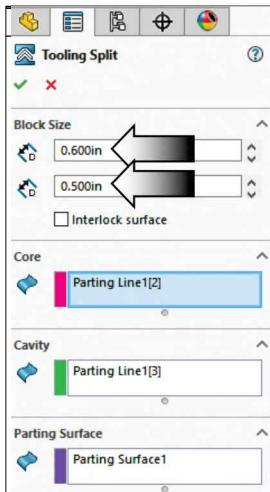
7. Creating the tooling split:

The Tooling Split tool uses the parting line, the shut off surfaces, and the parting surfaces information to create the core and cavity and allows the user to specify the block sizes.



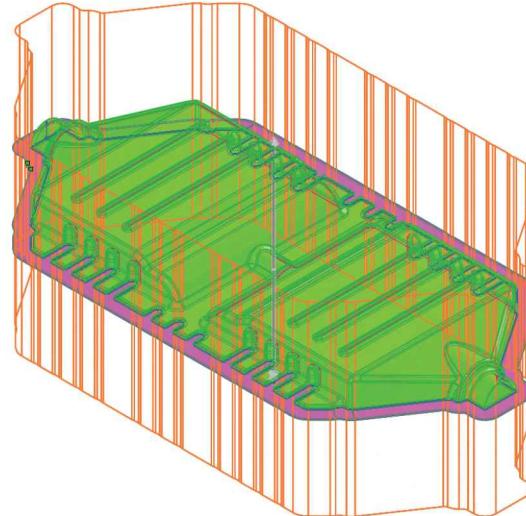
Exit the sketch and select **Tooling Split** (arrow).

For Upper Block size, enter: **.600in**



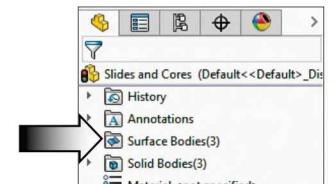
For Lower Block size, enter: **.500in**

Click **OK**.



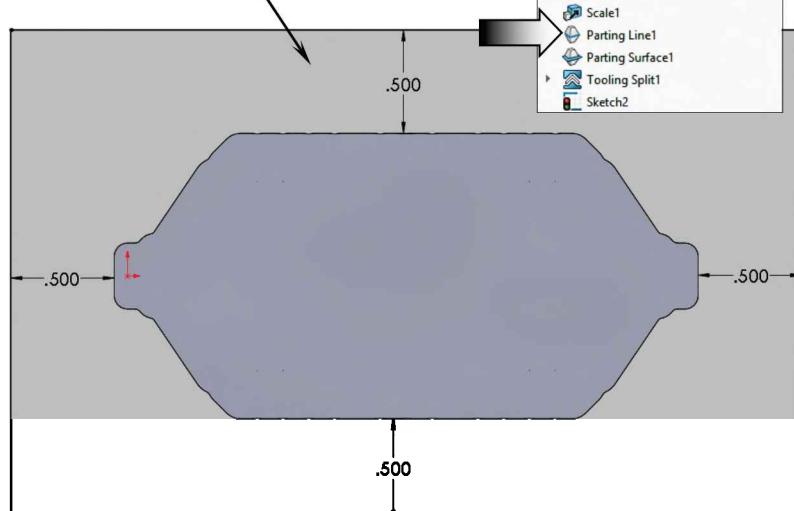
8. Finalizing the upper block size:

For clarity, right-click the **Surface Bodies** folder and select **Hide**. Also right-click and hide **Parting Line1**.



Select the uppermost surface of the upper block and open a new sketch.

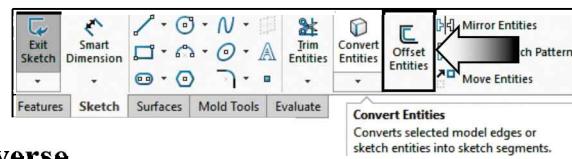
Sketch face



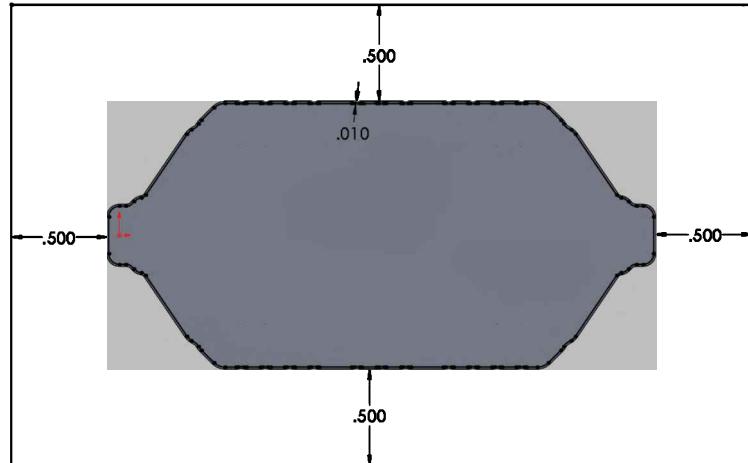
Sketch a **Corner Rectangle** as shown on the right.

Add the **dimensions** as indicated to fully define the sketch.

Expand the **Tooling Split1** feature on the FeatureManager tree, select **Sketch1** and click **Offset Entities**. Enter **.010"** for distance and click **Reverse**.

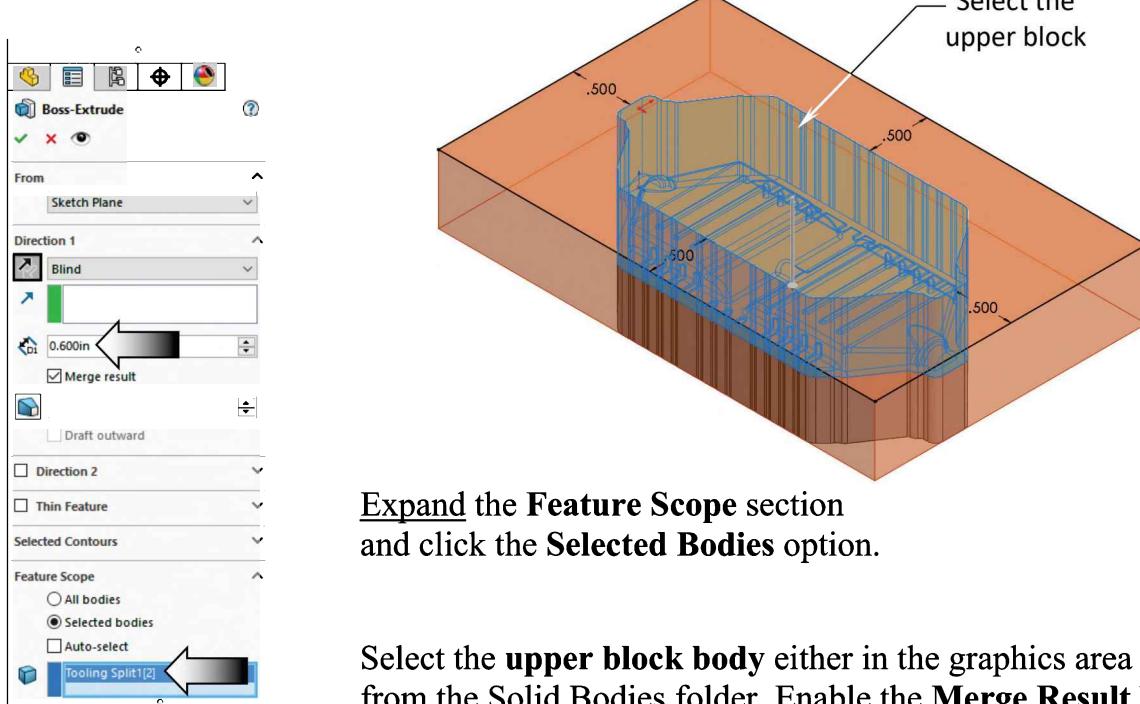


The Corner Rectangle and the Offset Entities formed a closed, nested contour and will be used to add the additional material around the outside of the upper block.



Switch to the Features tab and select:
Extruded Boss-Base.

Use the default **Blind** type and a depth of **.600in**. Click **Reverse** direction.



Expand the **Feature Scope** section and click the **Selected Bodies** option.

Select the **upper block body** either in the graphics area or from the Solid Bodies folder. Enable the **Merge Result** box.

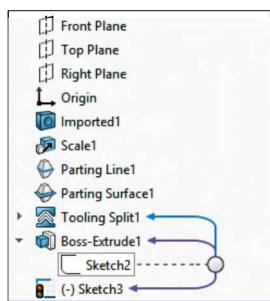
Click **OK**.

9. Finalizing the lower block size:

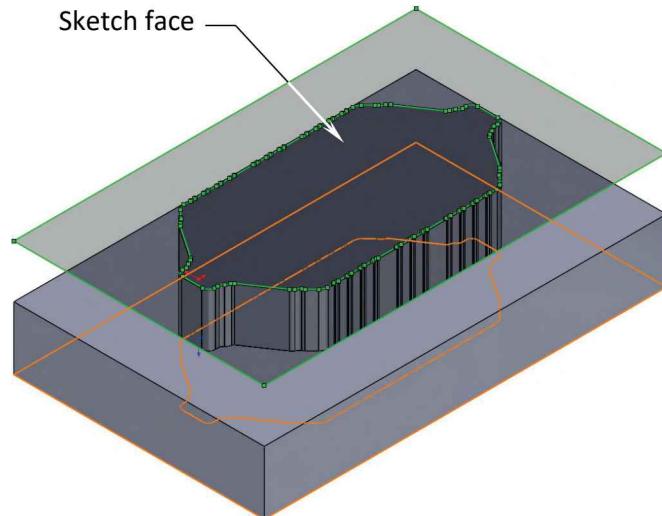
Switch to the **Isometric** view (Control+7).

Hold the **Shift** key and press the **Up Arrow** twice to rotate isometric view **180°** (the default angle is **90°** when rotating with the Shift and Arrow keys).

Open a **new sketch** on the top surface of the lower block.



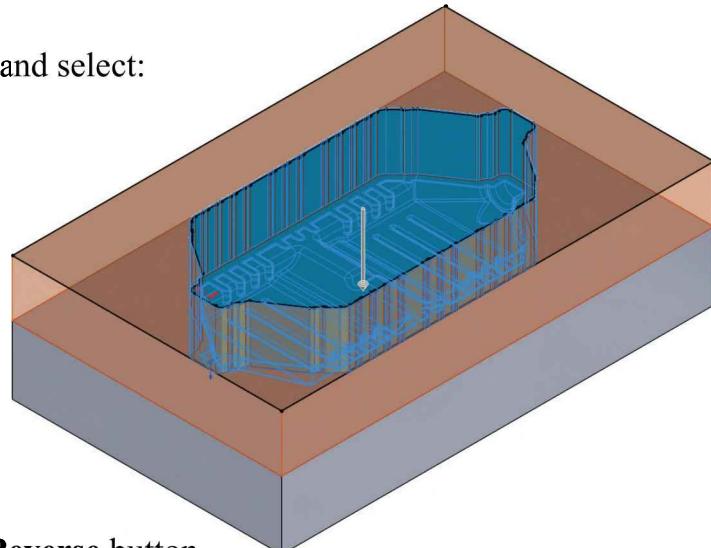
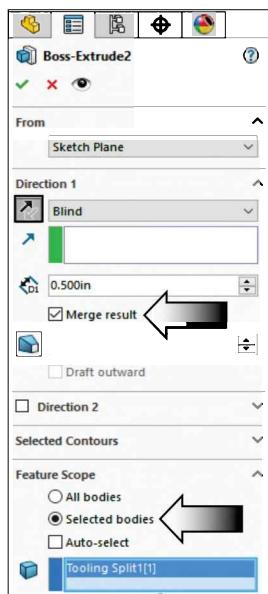
Expand the **Boss-Extrude1** and select **Sketch2**.



Click **Convert-Entities** to copy the entire sketch.

Switch to the **Features** tab and select: **Extruded Boss-Base**.

Use the default **Blind** type and a depth of **.500in**.



Enable the **Reverse** button and the **Merge Result** checkbox.

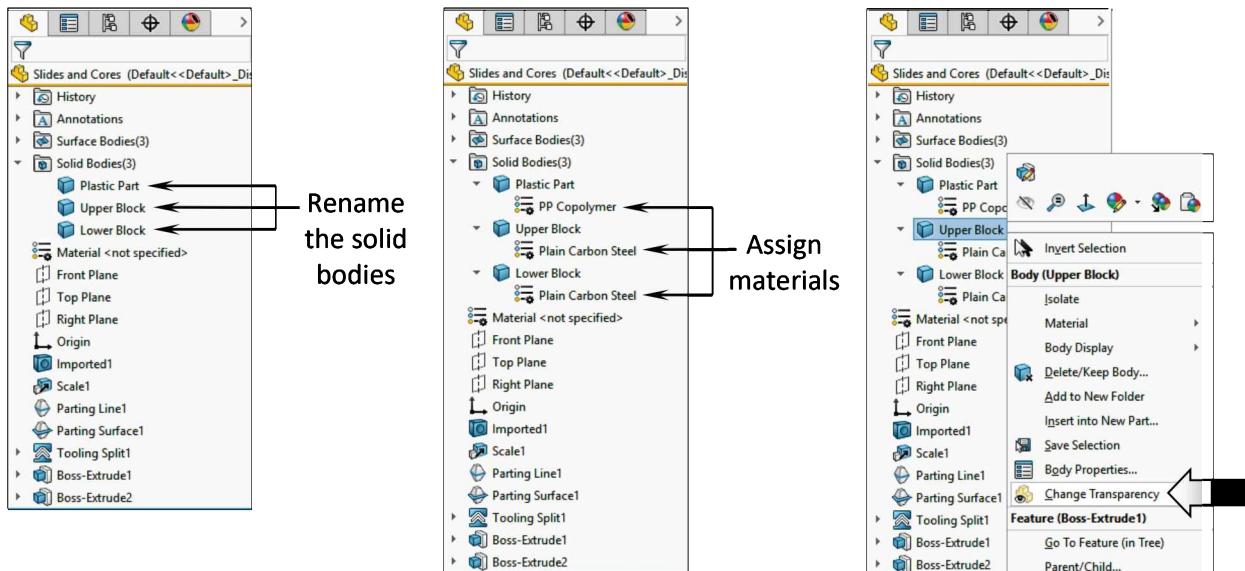
Expand the **Feature Scope** section and click the **Selected Bodies** option. Select the **lower block** body from the graphics area.

Click **OK**.

10. Renaming the bodies and assigning materials:

Expand the **Solid Bodies** folder and rename the solid bodies as noted below.

Right-click **each solid body** and assign the material as indicated.



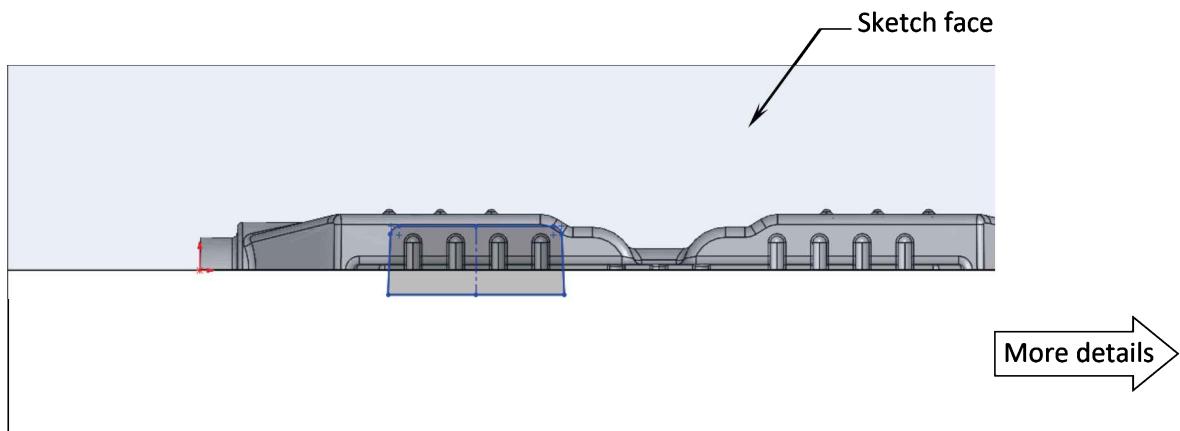
Right-click the Upper Block body and select: **Change Transparency** (arrow).

11. Creating the front slide core:

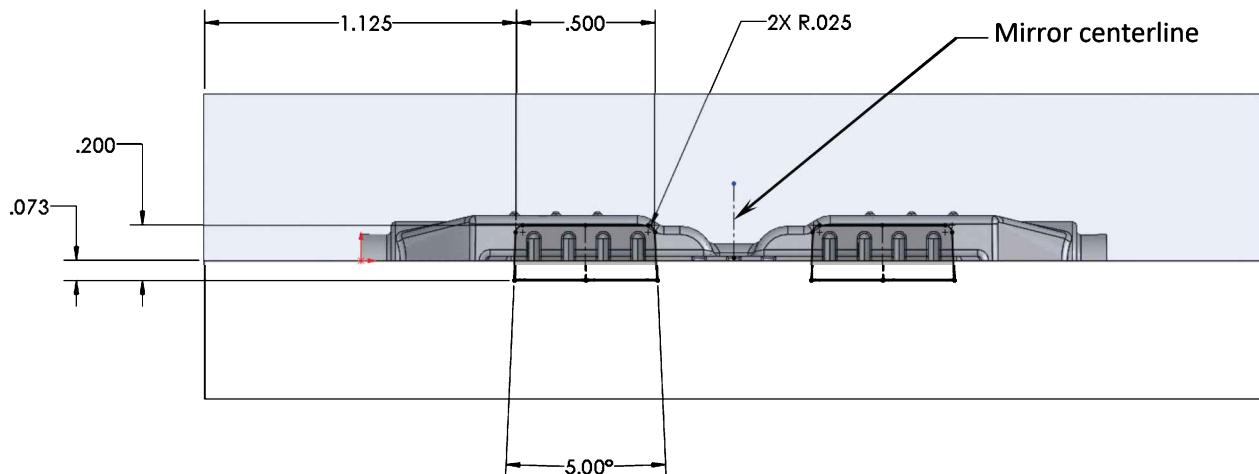
We will create a couple of slide cores to capture the undercut features in the part.

Open a **new sketch** on the face indicated.

Sketch the profile shown. Use the mirror function to maintain the symmetric relations between the sketch entities.

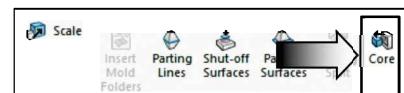


Notice the sketch is overlapped with the lower block by .073in.
We will select only the upper block when making the core.

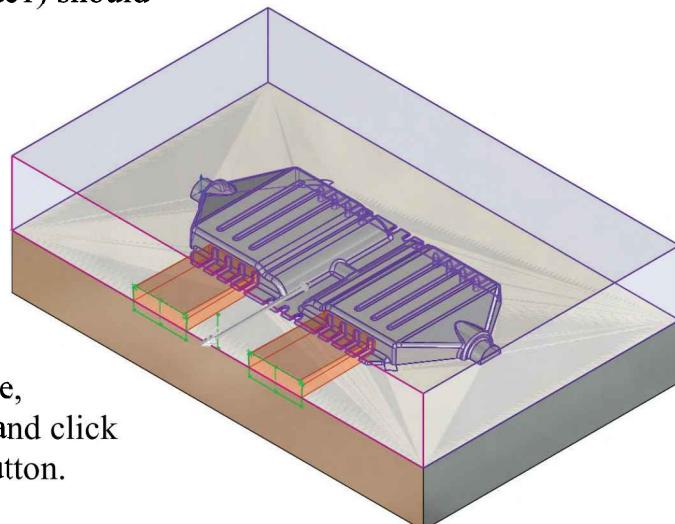
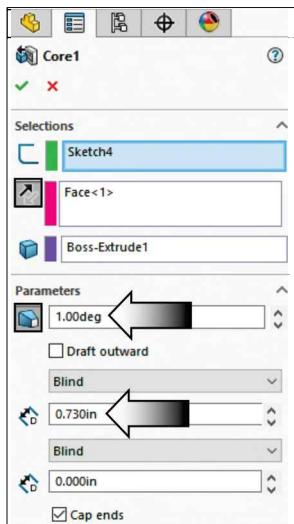


Mirror the sketch and add the dimensions shown to fully define it.

Exit the sketch and select the **Core** command on the **Mold Tools** toolbar.

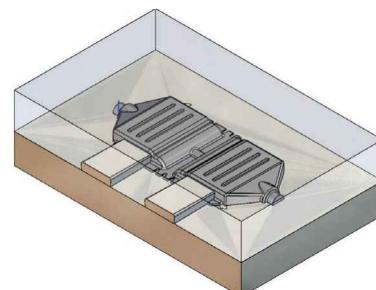


The upper block (Boss-Extrude1) should be selected automatically.



For Draft Angle,
enter **1.00deg** and click
the **Reverse** button.

Use the default **Blind** type
and enter **.730in** for depth.



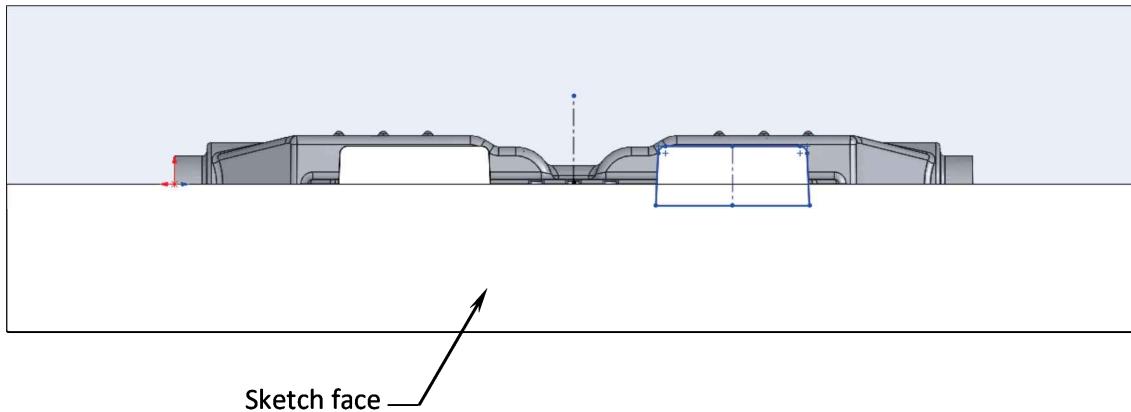
The extrude direction is towards the plastic part.

Click **OK**.

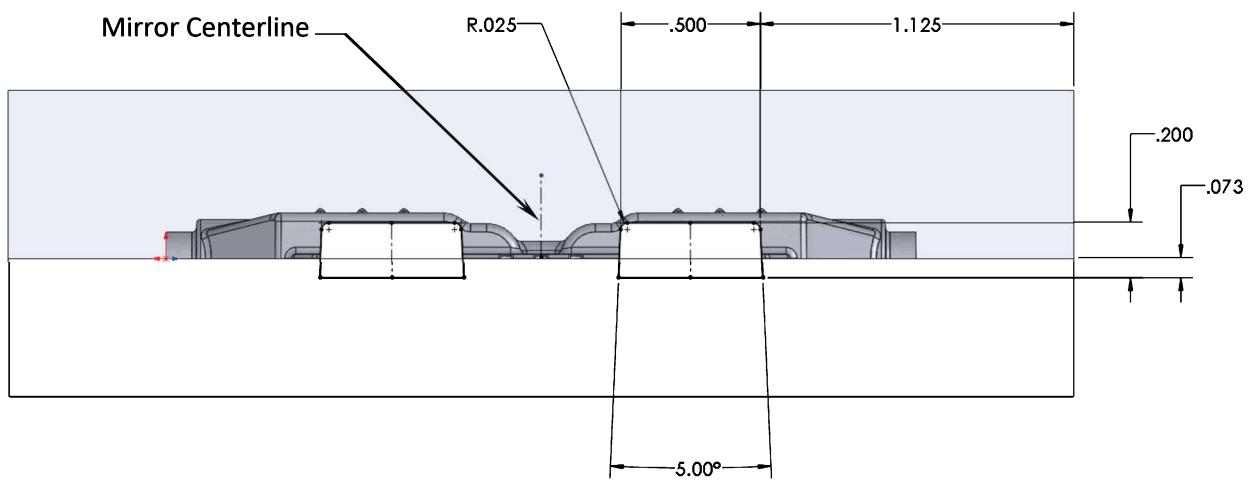
12. Creating the back slide core:

We will create an exploded view after both slide cores are created.

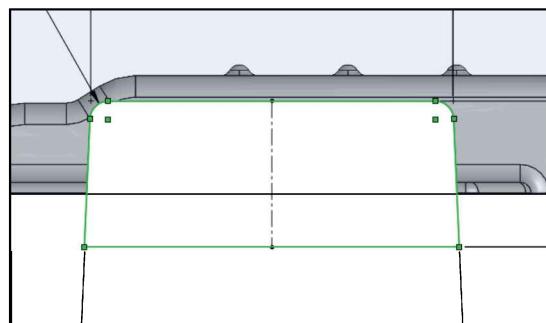
Select the opposite face of the lower block and open a **new sketch**.



Either convert, copy & paste the last sketch, or re-create it once again.

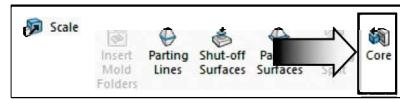


Avoid snapping any endpoints of the profile as it may cause the sketch to become over defined.

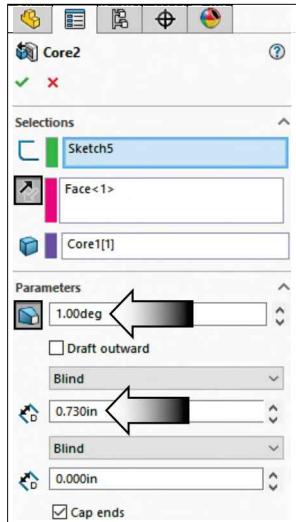


Verify that the sketch is fully defined and positioned correctly.

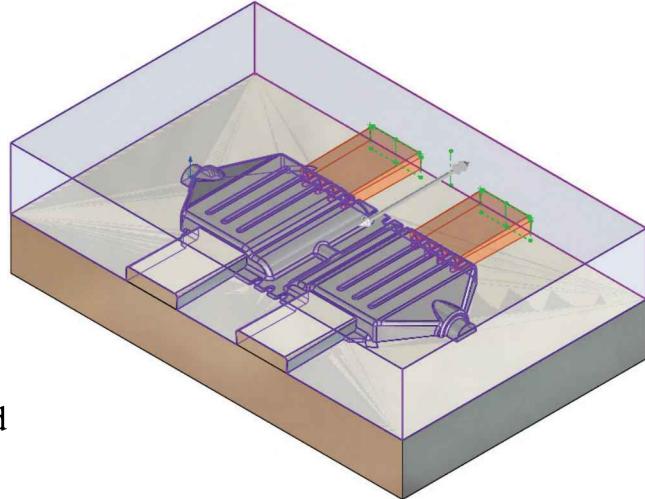
Exit the sketch and select the **Core** command on the **Mold Tools** tab.



The lower block (Boss-Extrude2) should be selected automatically.



For Draft Angle, enter **1.00deg** and click **Reverse**.



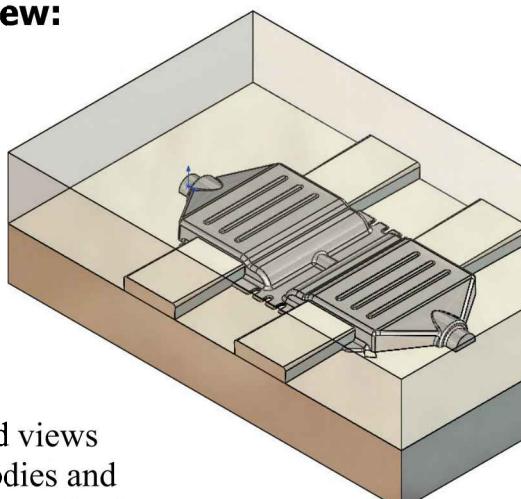
Use the default **Blind** type and enter **.730in** for depth.

Verify the extrude direction is, in fact, towards the plastic part.

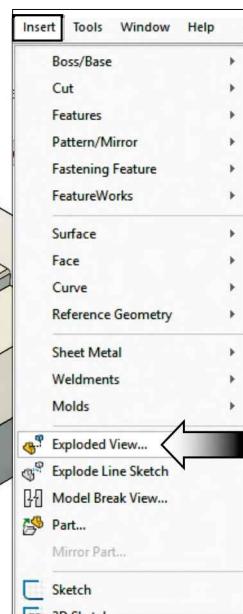
Click **OK**.

13. Creating an exploded view:

An exploded view in a multibody part shows the solid bodies spread out but positioned to show how they fit together.



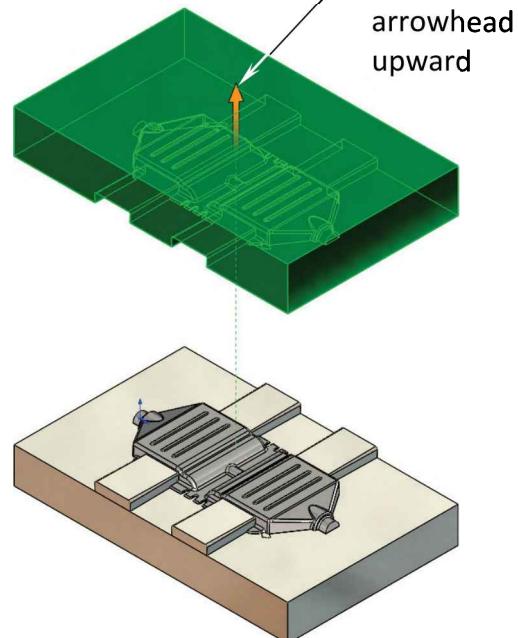
You can create exploded views by selecting the solid bodies and dragging the direction arrow in the graphics area, creating one or more explode steps.



Click  or select **Insert, Exploded View**.
 Select the **upper block** from the graphics area.

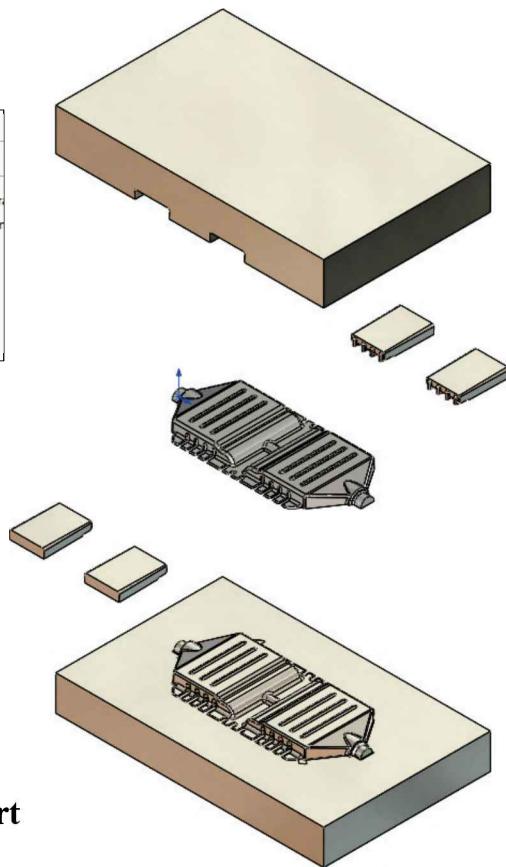
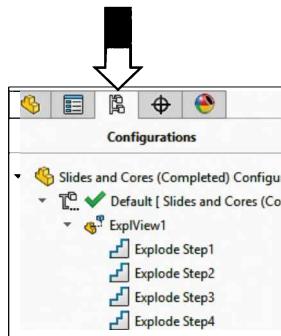
Drag the **Y arrowhead** upward,
 towards the vertical
 direction approx.
3.500 inches.

Repeat the step
 above and explode
 the lower block and
 the 4 core blocks to
 the positions
 approximately as
 shown.



Click **OK** when
 the exploded view
 is completed.

Click the
Configuration-
Manager tab
 (arrow).



Expand the default and double-click
ExplView1 to collapse it (and right-
 click to edit).

14. Saving your work:

Select **File, Save As**.

Enter **Slides and Cores_Completed.sldprt**
 for the file name and press **Save**.

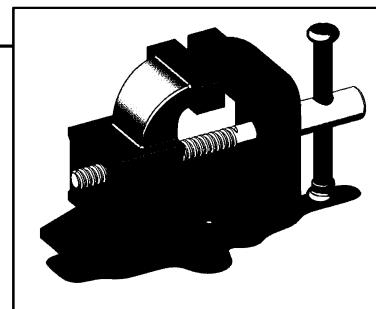
CHAPTER 20

Top Down Assembly - Part 1

Top Down Assembly - Part 1

Miniature Vise

This chapter will guide us through the use of a special technique for creating new parts in the context of an assembly or Top Down assembly design.



The method of using existing geometry of other components such as their locations, sketch entities, and model edges to construct the geometry of the new components is called In-Context Assembly, or in SOLIDWORKS, it is called Top-Down Assembly. This option greatly helps you capture your design intent and reduce the time it takes to do a design change, and having the parts update within themselves based on the way they were created.

While working in the top down assembly mode, every time a face or a plane is selected for a new sketch, the system automatically creates an INPLACE mate to reference the new part.

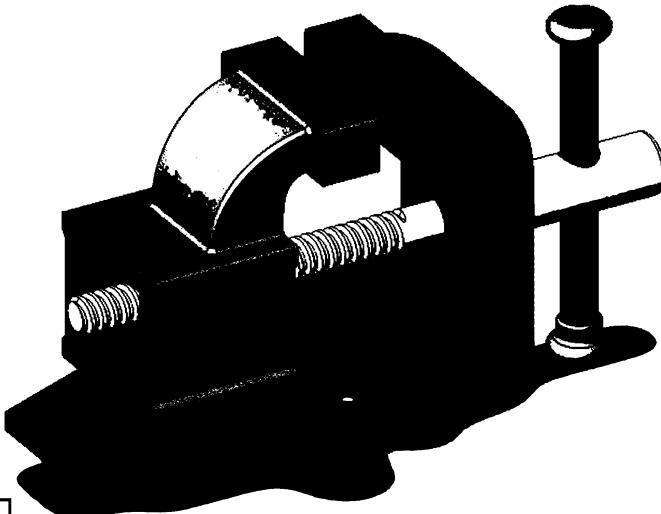
The Inplace mates can be suppressed so that components can be moved or re-positioned and the Inplace mates can also be deleted as well; new mates can be added to establish new relationships with other components.

When a part is being edited in the Top Down Assembly mode, the Edit Component command is selected, and the component's color changes to Blue (or Magenta depending on the color settings in the system options).

Upon the successful completion of this lesson, you will have a better understanding of the 2 assembly methods used in SOLIDWORKS: the traditional Bottom Up Assembly, where parts are created separately, and then inserted into an assembly and mated together; and the dynamic Top-Down Assembly, where multiple parts can be created together in the context of an assembly.

Top Down Assembly – Part 1

Miniature Vise



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI

Units: INCHES – 3 Decimals

Tools Needed:



Insert Sketch



Rectangle



Circle



Dimension



Add Geometric Relations



Sketch Mirror



Offset Entities



Planes



Fillet/Round



Base/Boss Extrude



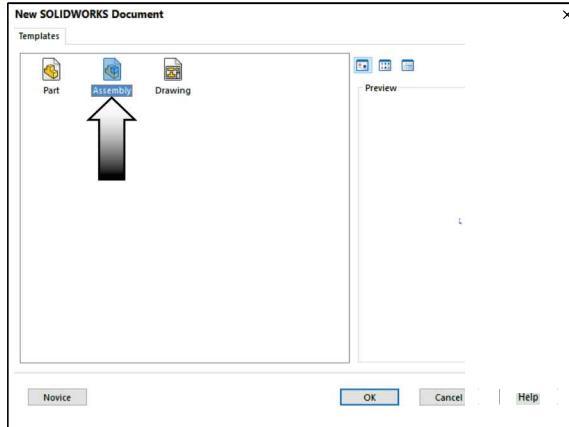
Loft



Edit Component

1. Starting a new assembly template:

Select File / New / Assembly.



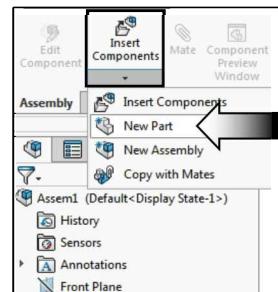
Click **Cancel** to exit the **Begin Assembly** mode.



Save the new assembly document as **Mini Vise.sldasm**.

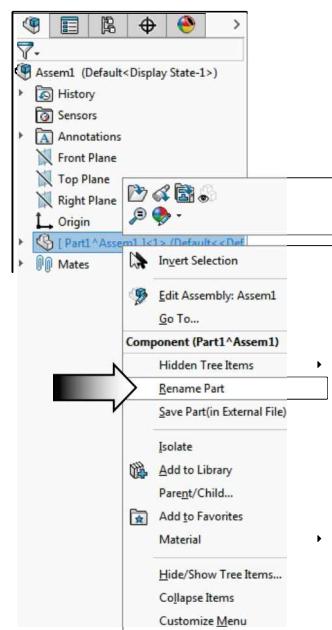
2. Creating the Base part:

Select **Insert**, **Component**, **New Part**.



Select the Front plane from the FeatureManager tree.
An Inplace1 mate is created to reference the new part.

A new component is created with a default name
[Part1^Assembly]<1> and displayed on the tree.



To rename the part, right-click on the default name
and select **Rename Part**.

Enter: **Base** for the name of the first component.



The color of the new component
changes to the default Blue color.

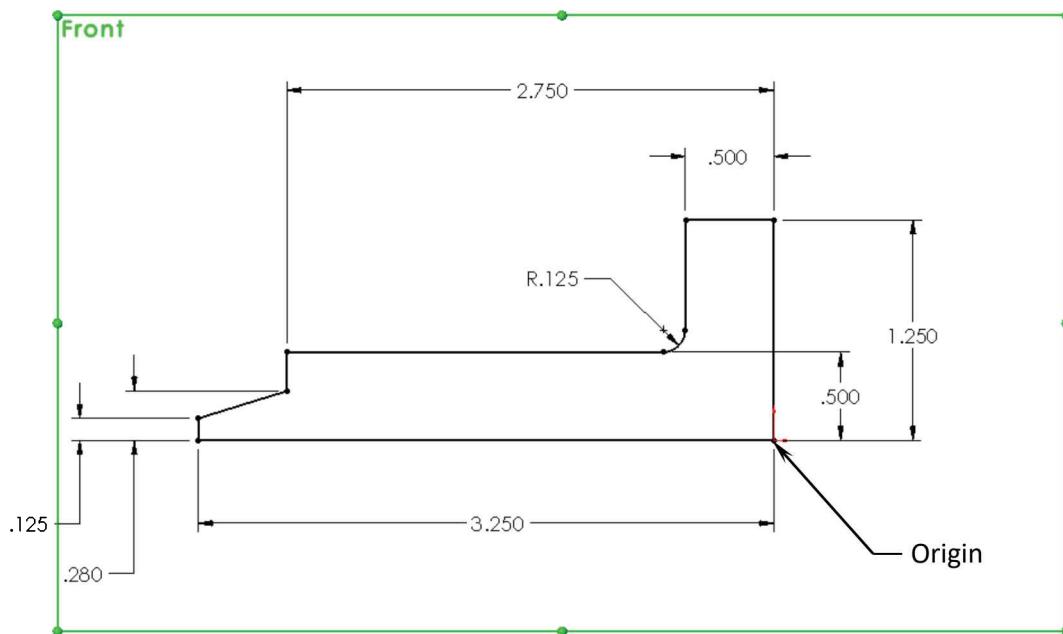
To change the part's color, go to:
Tools/ Options / System Options / Colors / Assembly / Edit Part.

Blue color

A **new sketch** is created automatically when a new component is inserted.

Sketch the profile shown below; keep the **Origin** at the lower right corner.

Add the dimensions shown below to fully define the sketch.

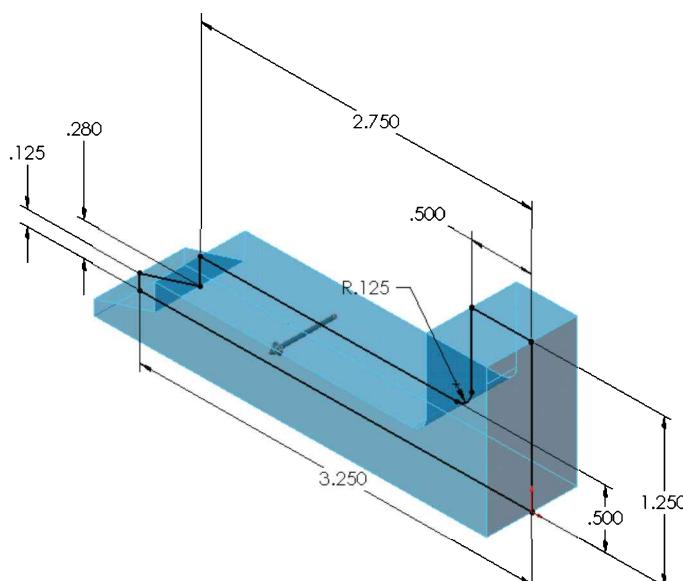
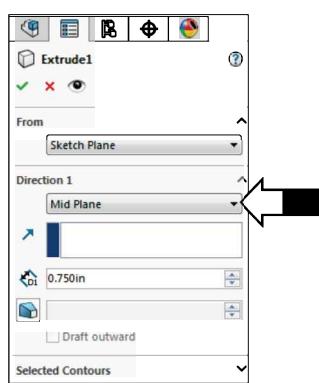


3. Extruding the Base:

Switch to the **Features** tab and click or select **Insert, Boss-Base, Extrude**.

Direction 1: Mid-Plane.

Extrude Depth: .750 in.



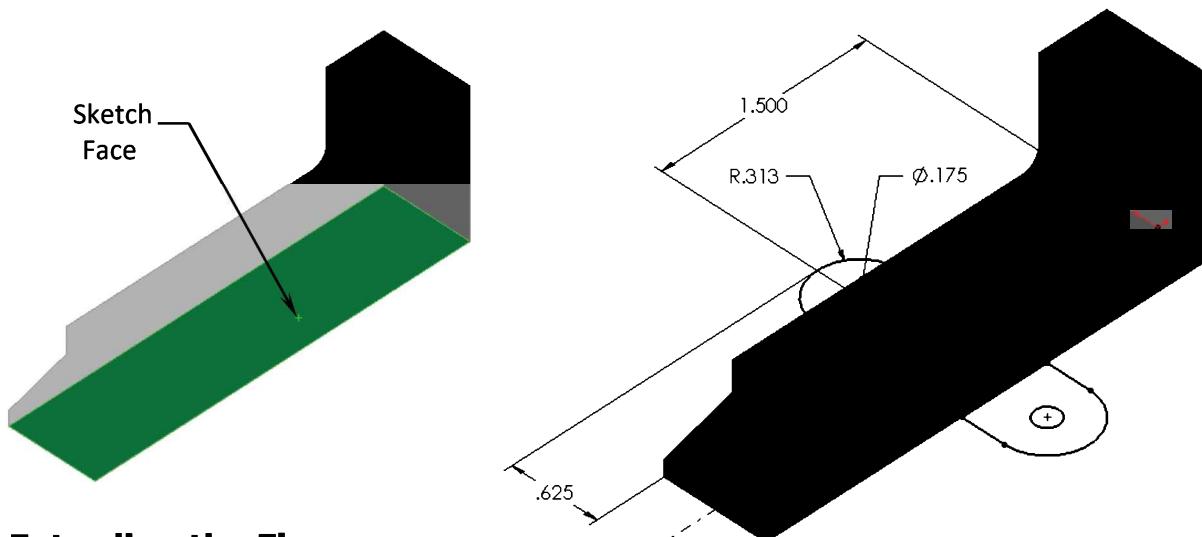
Click OK.

4. Adding the side flanges:

Select the bottom face of the base and open a **new sketch** .

Sketch the profile below; use the **Mirror**  option when sketching to keep the sketch entities symmetrical about the Centerline.

Add the dimensions shown. (Hold the **Shift** key when adding the **.625** dim.)

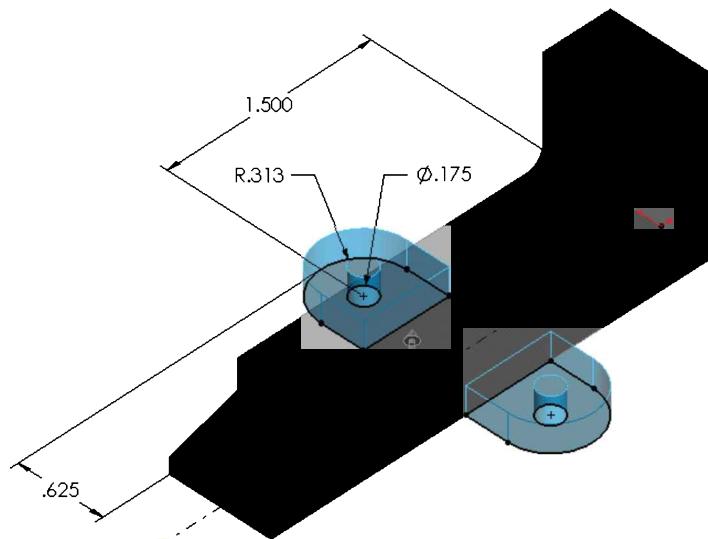
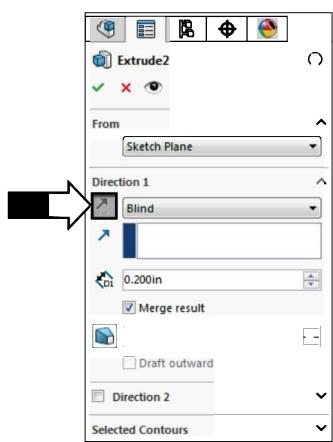


5. Extruding the Flanges:

Switch to the **Features** tab and click  or select **Insert, Boss-Base, Extrude**.

Direction 1: Blind (Reverse) **Depth: .200 in.**

Enable Reverse Direction.

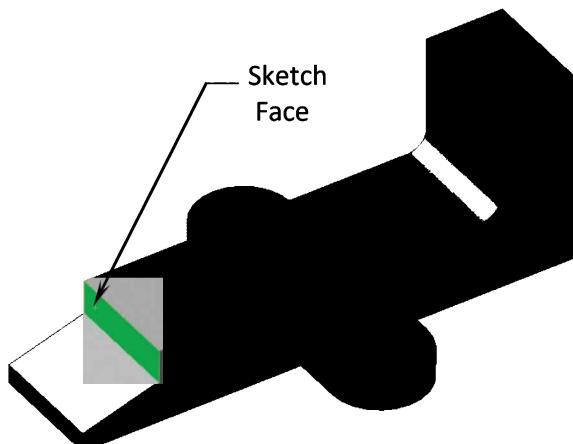


Click OK.

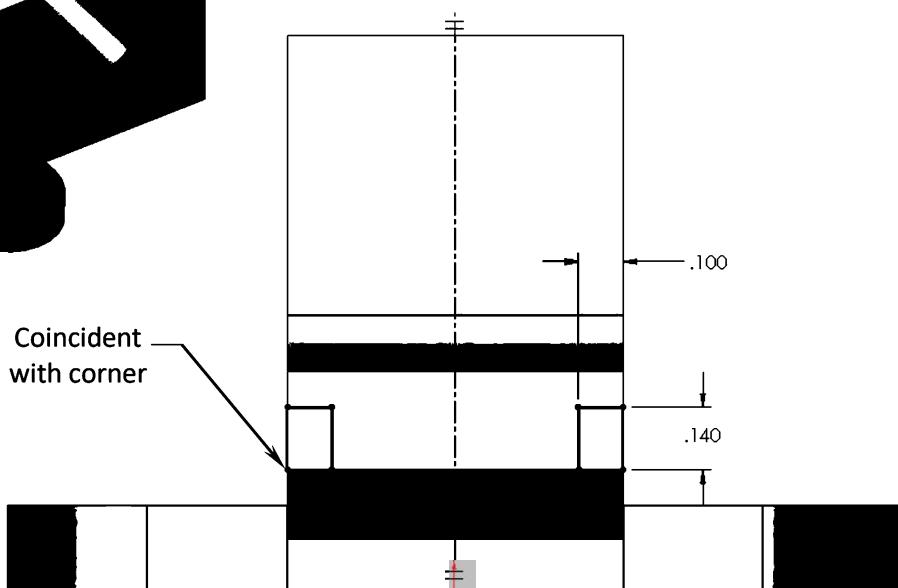
6. Adding the side cuts:

Select the face as indicated and click  or select **Insert, Sketch**.

Sketch a **Centerline**  starting at the Origin and click **Dynamic Mirror** .



Sketch a **Rectangle** and add the dimensions/relations shown below.



7. Extruding the side cuts:

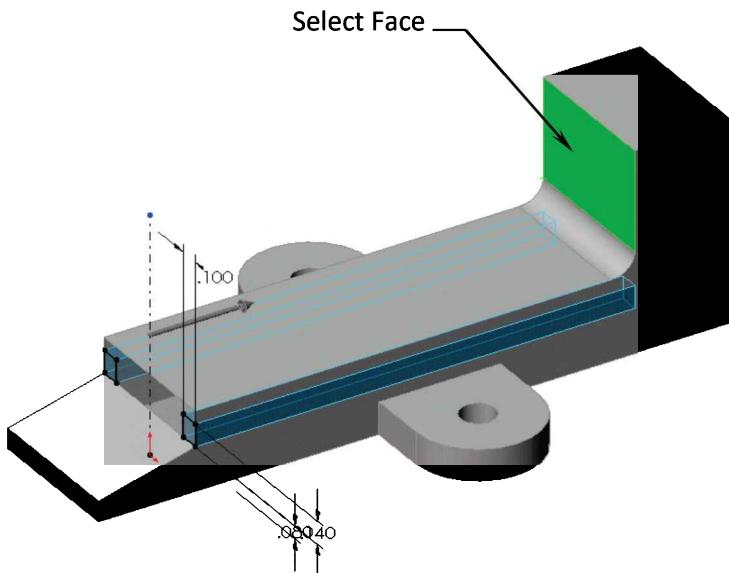
Click  or select **Insert, Cut, Extrude**.

Direction 1: **Up-To-Surface**.

Select the face as indicated.

The depth of the cut is linked to the selected face.

Click **OK**.



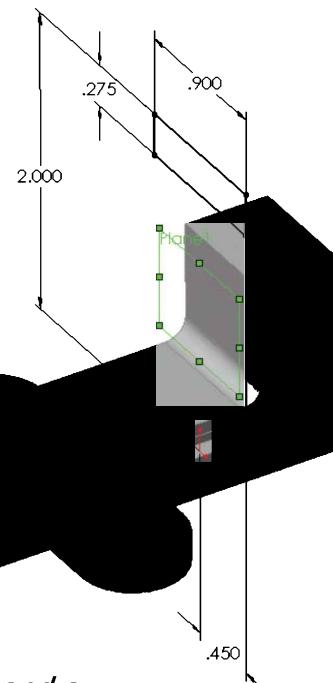
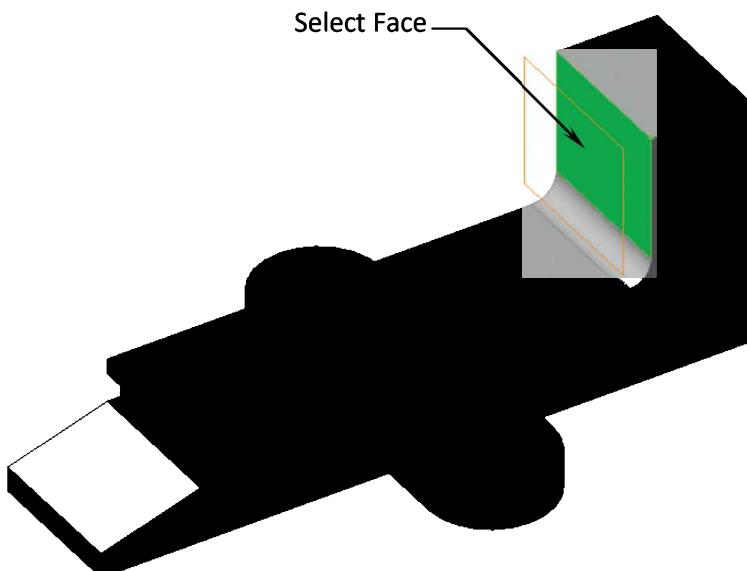
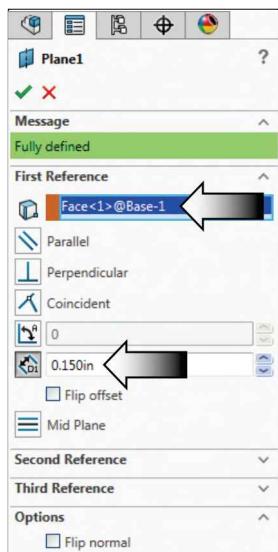
8. Creating an offset distance plane:

Select the face as indicated and click  or select **Insert, Reference Geometry, Plane**.

Click **Offset Distance** option.

Enter **.150 in.** (the new plane is created away from the face).

Click **OK**.

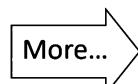


9. Creating the Fixed Jaw, sketch 1 of 4:

Select the new plane and click  or select **Insert, Sketch**.

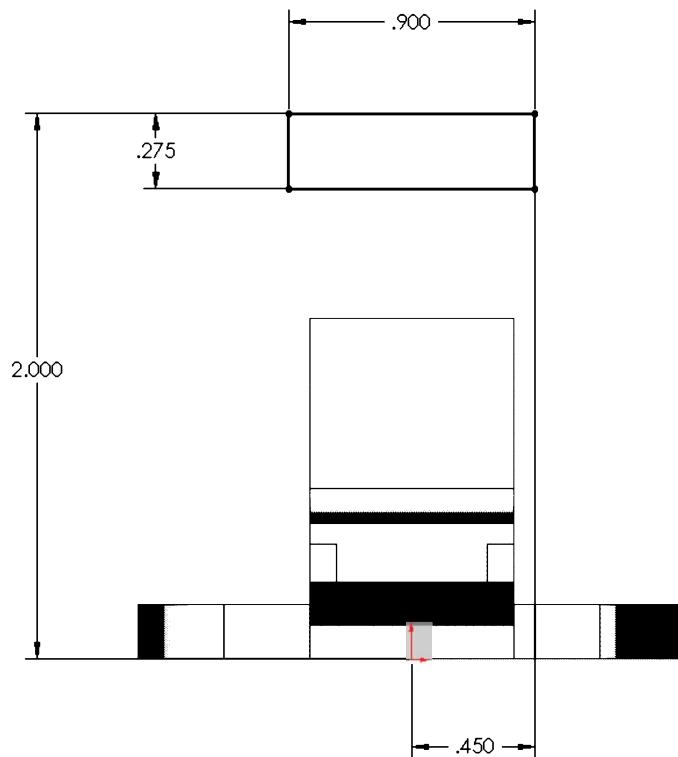
Sketch a **Rectangle** approximately 2 inches above the origin.

NOTE: The dimension **.450** can be replaced with a centerline and a symmetric relation.



Add the dimensions shown below to fully position the sketch.

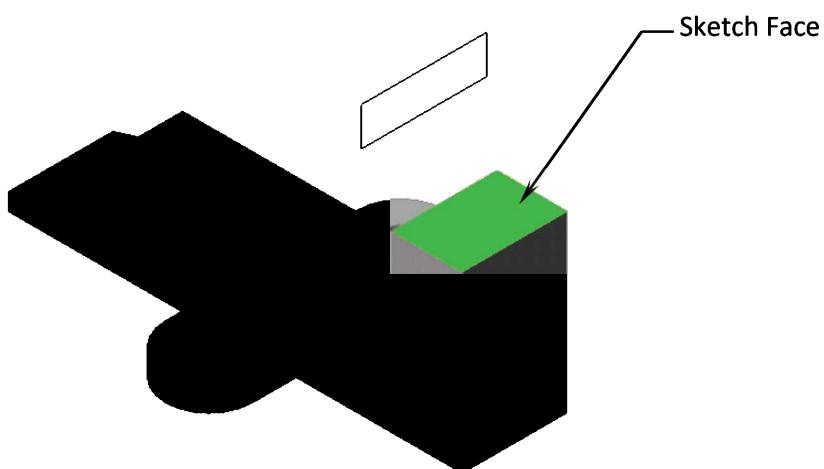
(Sketch a vertical centerline and use Symmetric relation to center the rectangle also works well.)



Exit the sketch or select **Insert, Sketch**.

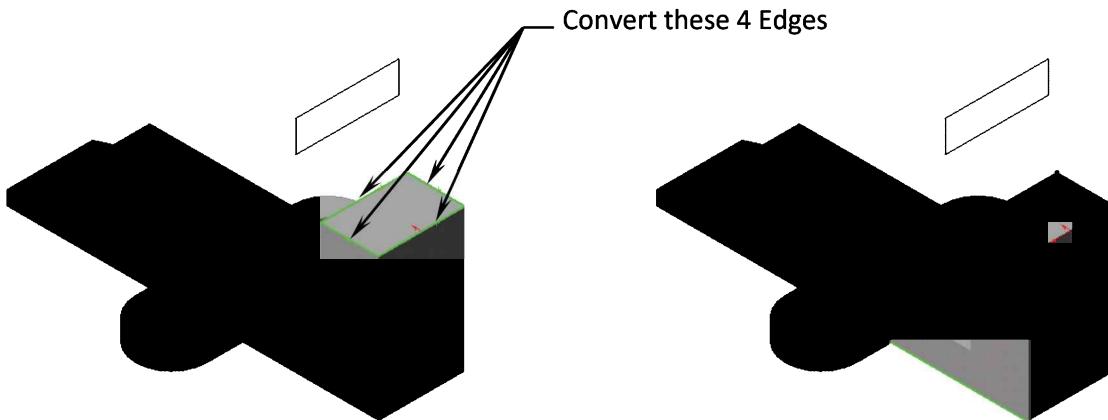
10. Creating the 2nd profile, sketch 2 of 4:

Select the face as indicated and click or select **Insert, Sketch**.



Hold the **Control** key and select the **4 edges** as noted below (or simply select the rectangular face and click the Convert Entities command).

Click **Convert Entities**  on the Sketch-Tools tab.



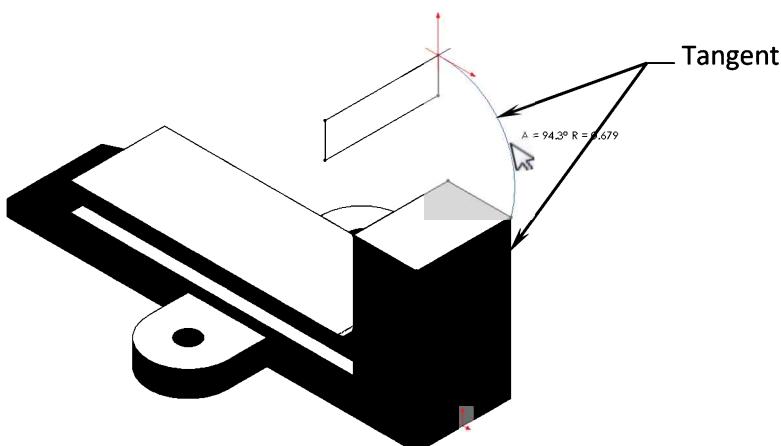
The 4 selected edges are converted into a rectangle.

Exit the sketch  or select **Insert, Sketch**.

11. Creating the 3D Guide Curves:

Select **3D Sketch** from the Sketch tab  or select **Insert, 3D Sketch**.

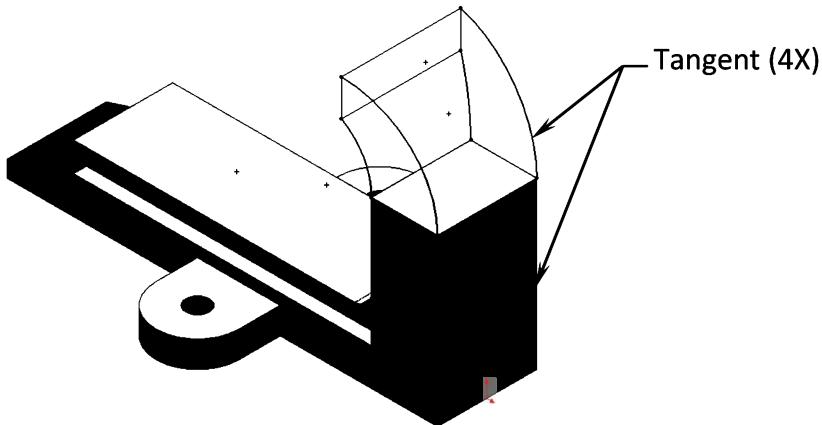
Sketch a **3-Point-Arc**  approximately as shown below and add a **Tangent** relation between the arc and the vertical edge of the base.



Remain in the Isometric view while sketching the 3-Point-Arc.

Repeat the last step and create the **other 3 arcs** the same way.

A **Perpendicular** relation between the endpoint of the arc and the upper face of the part can also be used to constrain the 4 arcs.

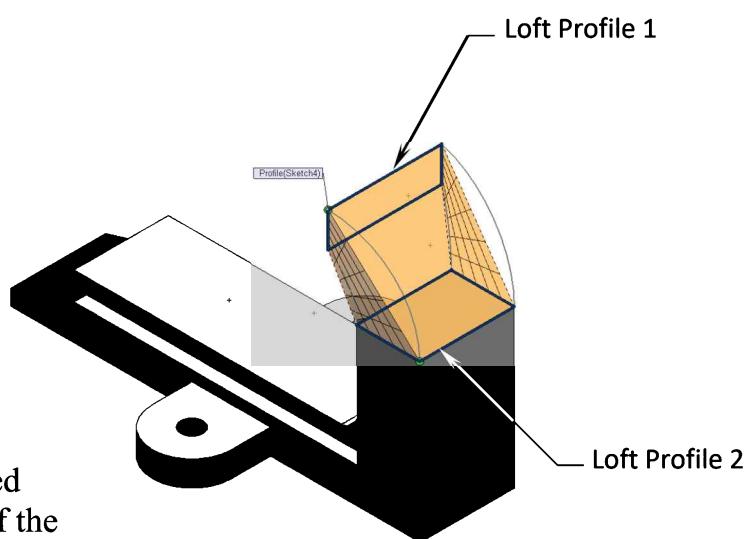
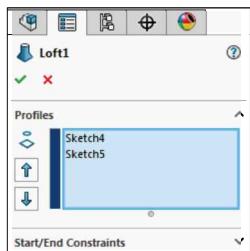


Exit the 3D Sketch or press Control + Q.

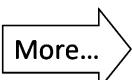
12. Creating the Fixed Jaw loft:

Click or select **Insert, Boss-Base, Loft**.

Select the **2 sketch profiles** as labeled (Profile 1 and Profile 2).
(Click at or near the ends of the rectangles. SOLIDWORKS will select the nearest endpoints automatically.)

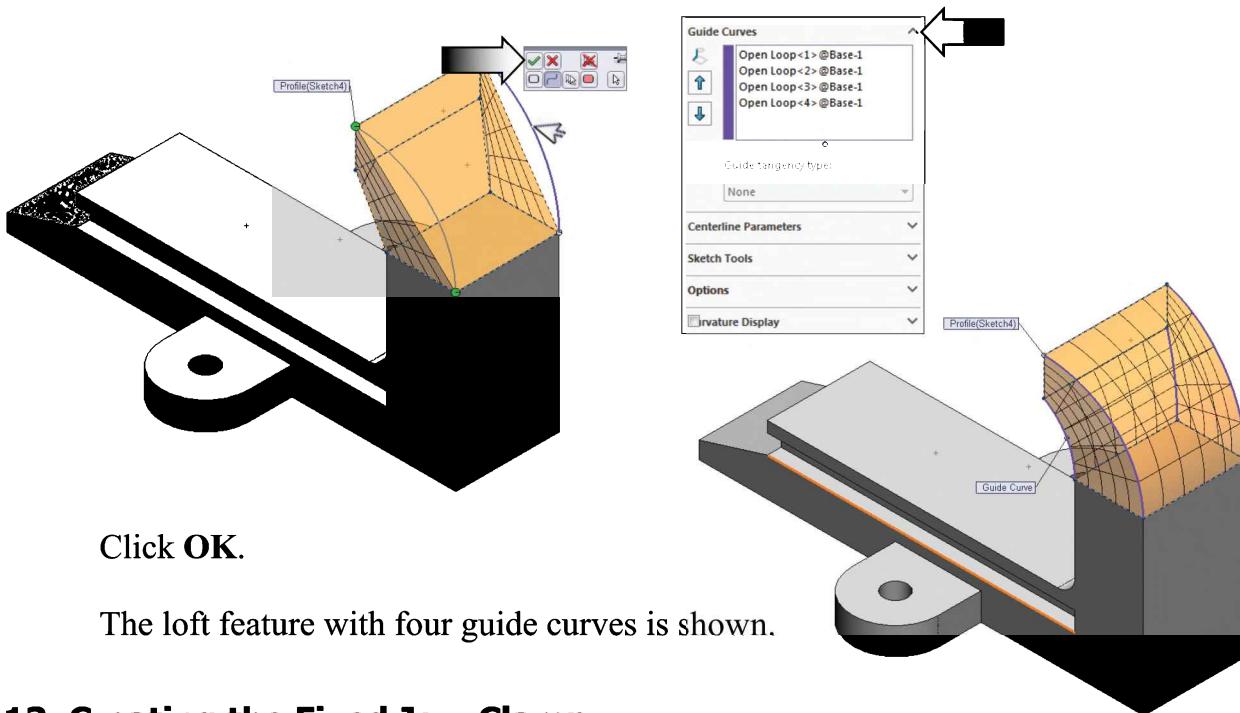


If the preview shows a twisted transition, simply drag one of the connectors to the right corner to correct the problem.



Expand the **Guide Curve** section and select **one of the guide curves** in the 3D-Sketch.

Because this sketch has multiple entities that are not connected with one another, you will have to click the OK button (the check mark) on the SelectionManager after selecting each arc (arrow).



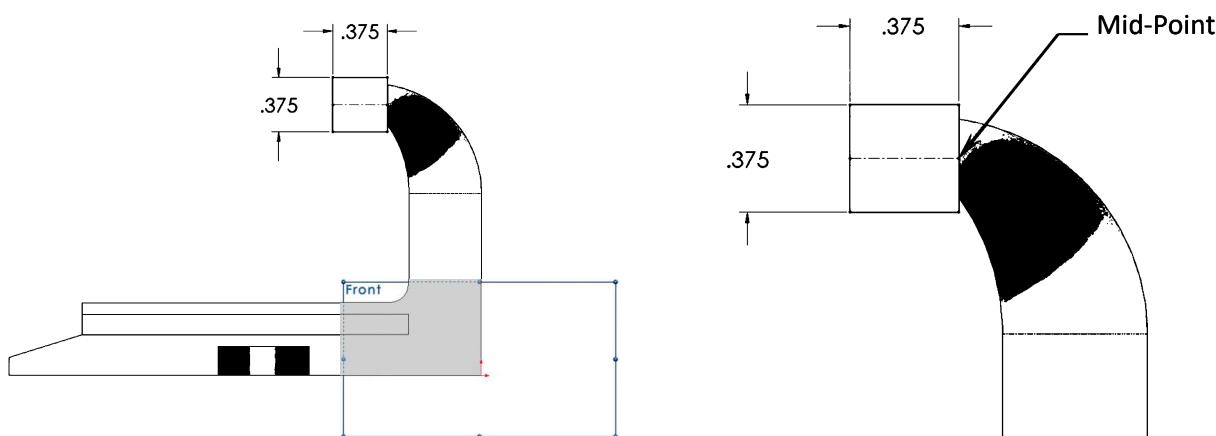
Click **OK**.

The loft feature with four guide curves is shown.

13. Creating the Fixed Jaw Clamp:

Select the Front plane from the FeatureManager tree and click or select: **Insert, Sketch**.

Sketch a **Rectangle** and add the dimensions and relations as indicated.
(Add a horizontal centerline and a midpoint relation between the right endpoint and the model edge.)



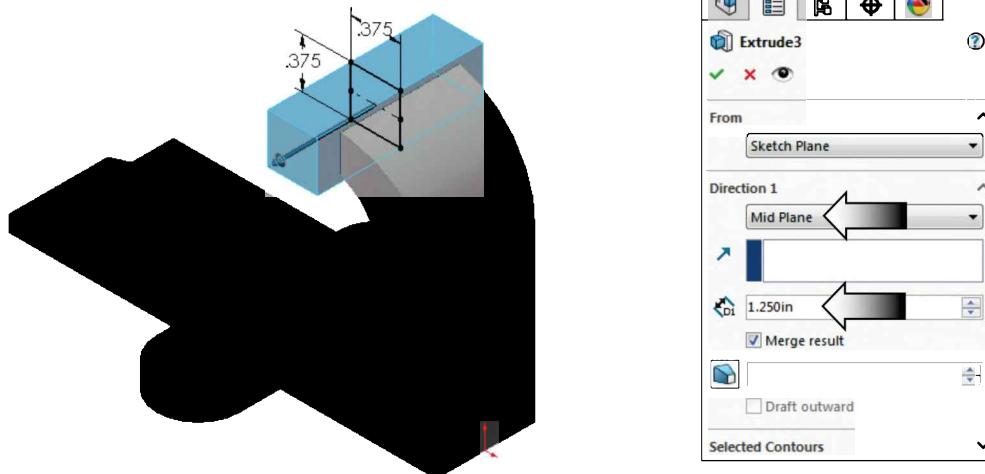
14. Extruding the Fixed Jaw Clamp:

Click  or select **Insert, Boss-Base, Extrude**.

Direction 1: Mid-Plane.

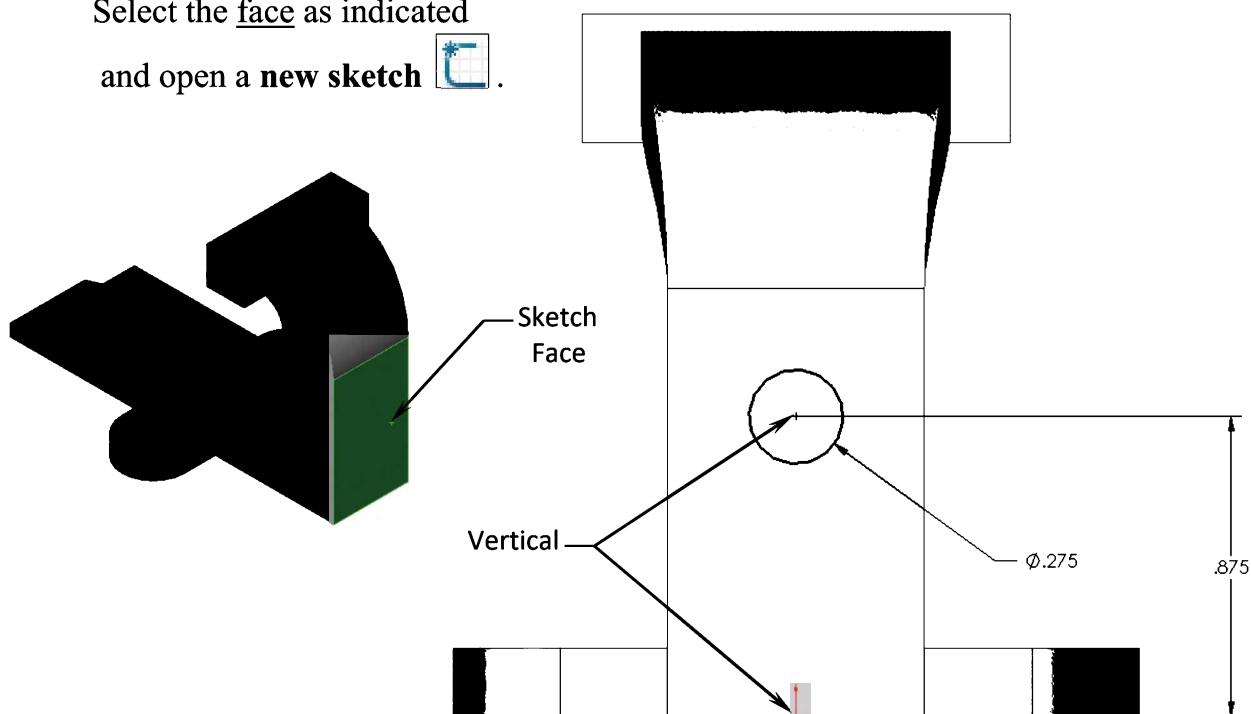
Extrude Depth: 1.250 in.

Click OK.



15. Creating the Lead Screw Hole:

Select the face as indicated
and open a new sketch .



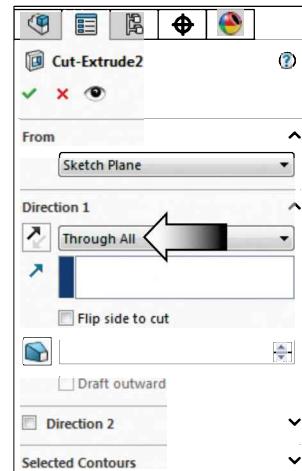
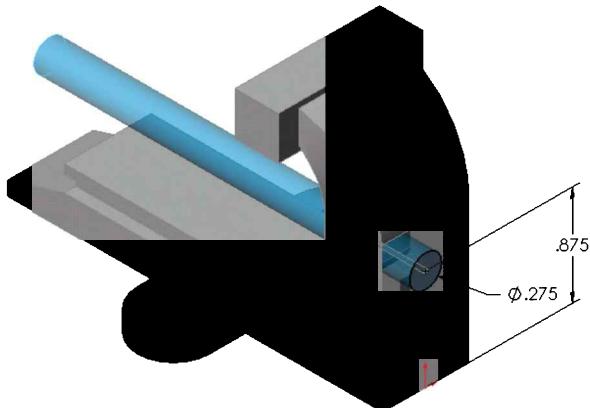
Sketch a **Circle** and add the dimensions and relations as shown in the image.

16. Extruding the Hole:

Click  or select **Insert, Cut, Extrude**.

Direction 1: **Through All**.

Click **OK**.



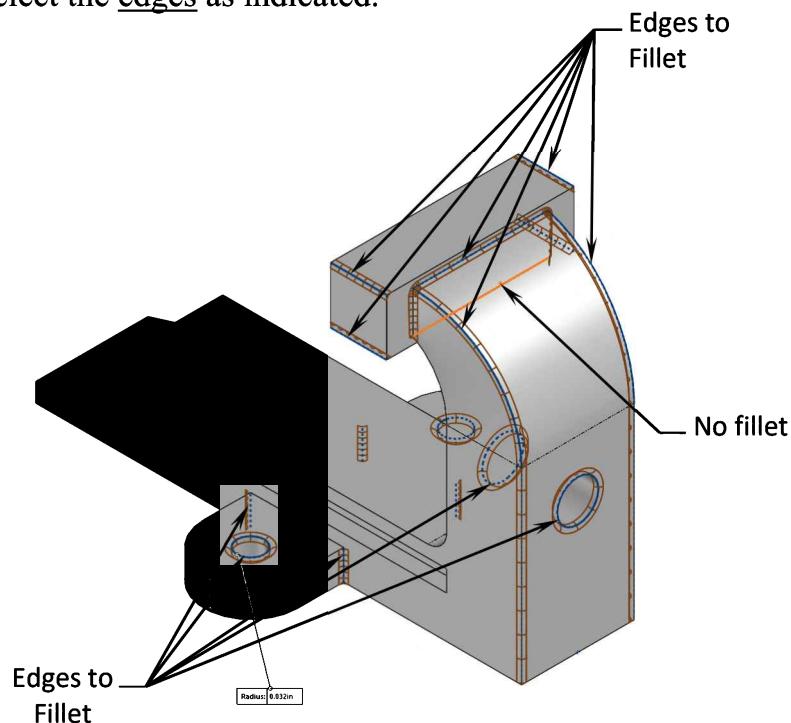
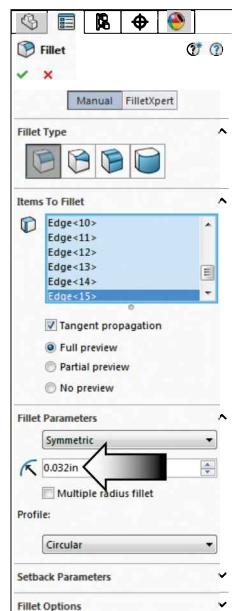
17. Adding Fillets:

Click  or select **Insert, Features, Fillets-Rounds**.

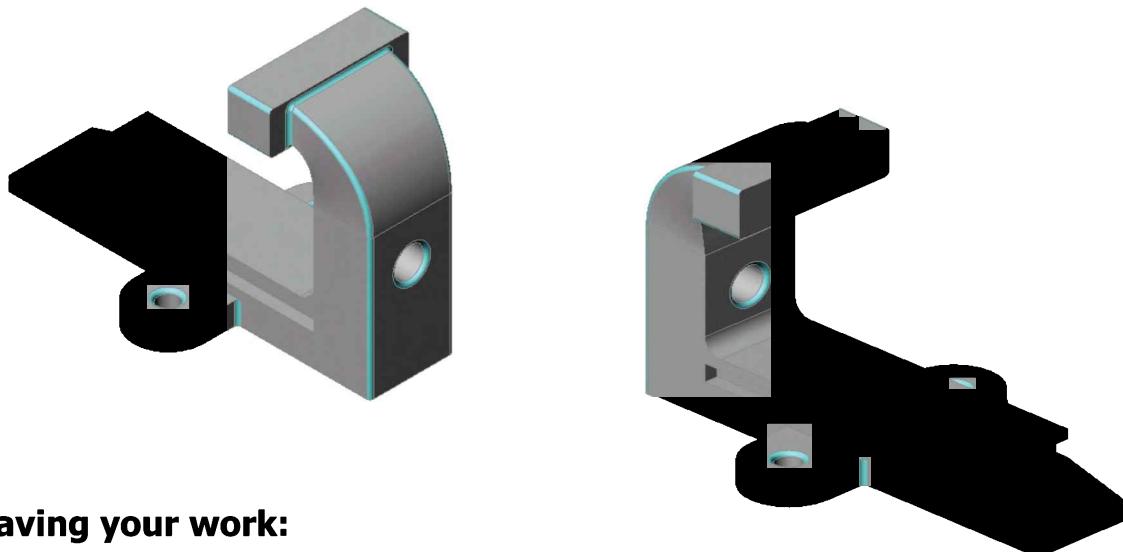
Enter **.032 in.** for Radius.

For Edges to Fillet, select the edges as indicated.

Click **OK**.



The Base component is shown in the Front and Back Isometric views.



18. Saving your work:

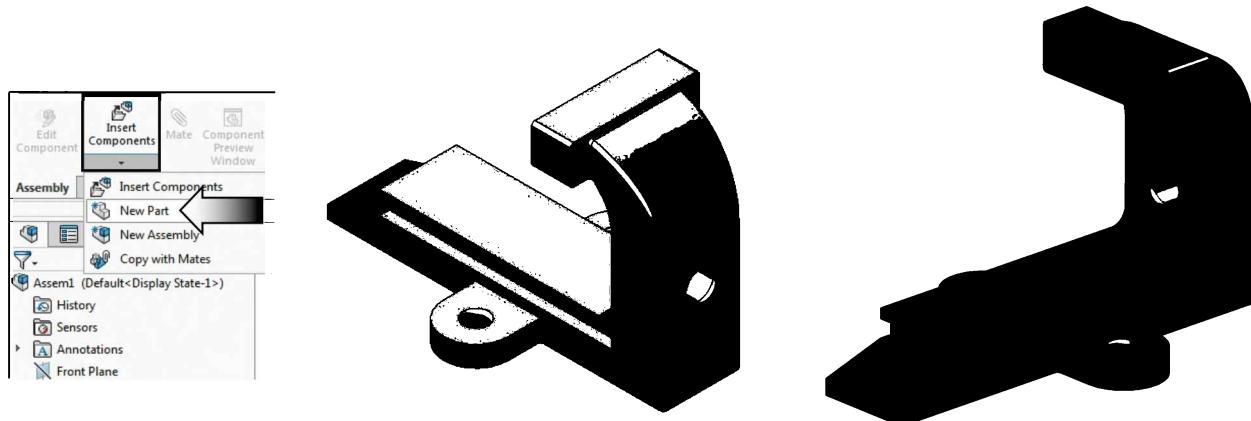
Select File, Save As.

Enter **Base.sldprt** for the file name and press **Save**.

Click  to exit the **Edit Component** mode.

19. Creating a new component: The Slide Jaw

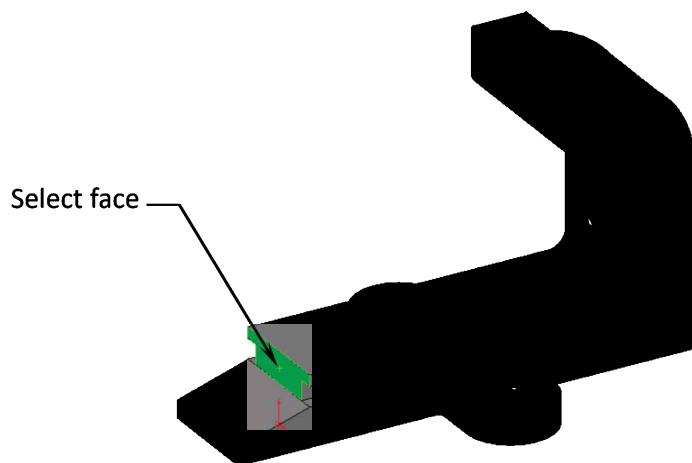
Select **Insert, Component, New Part**.



Rotate the model to a similar position as shown above; the planar surface on the left side will be used as the sketch plane in the next step.

Select the face indicated as sketch plane for the new component (Inplace2). A new part and a new sketch are created and displayed on the FeatureManager tree.

Rename the new component to **Slide Jaw**.



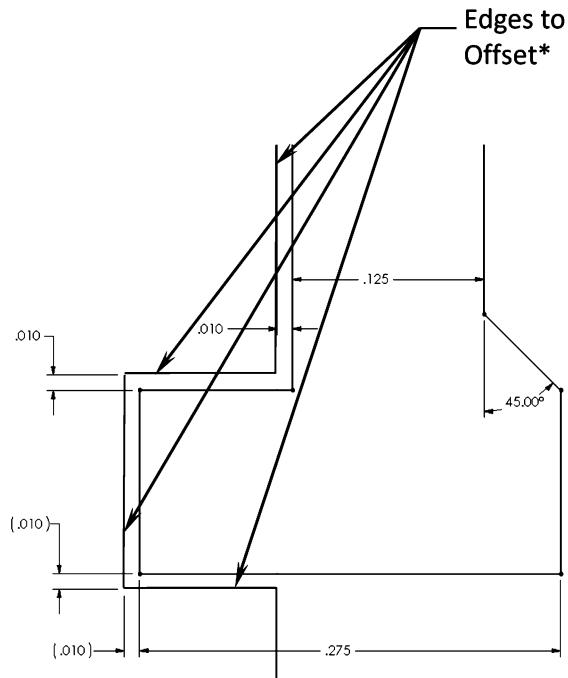
A new part is created in the FeatureManager tree and a **new sketch** is activated.

An **Inplace mate** **InPlace1 (Base<1>)** is also created for the new component to reference its location.

20. Using the Offset Entities command:

Select the **4 edges** of the model (as shown) and click **Offset Entities** .

Enter **.010 in.** for offset value. This offset distance between 4 the lines and the model edges will remain locked and get updated at the same time when the offset value is changed.



Offset Entities

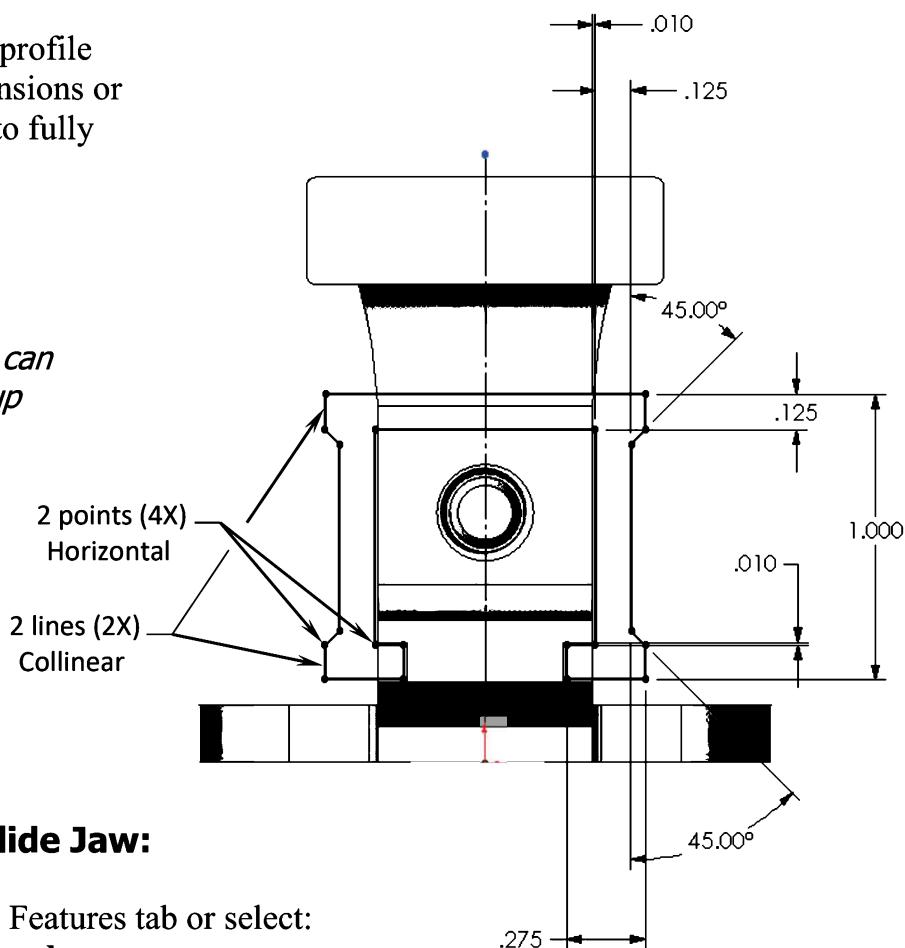
- * The geometry of a model such as edges, faces, and other sketch entities can be offset or converted to create the new geometry for the new part.
- * The offset entities can be set to one direction or bidirectional.
- * An On-Edge relation is created for each converted sketch entity.

More on next page

Sketch the entire profile and add the dimensions or relations needed to fully define the sketch.

Note:

The mirror option  can be used to help speed up the sketching process and keep the profile symmetrical at the same time.



21. Extruding the Slide Jaw:

Click  on the Features tab or select: Insert, Base, Extrude.

Direction 1: **Blind** and reverse direction.

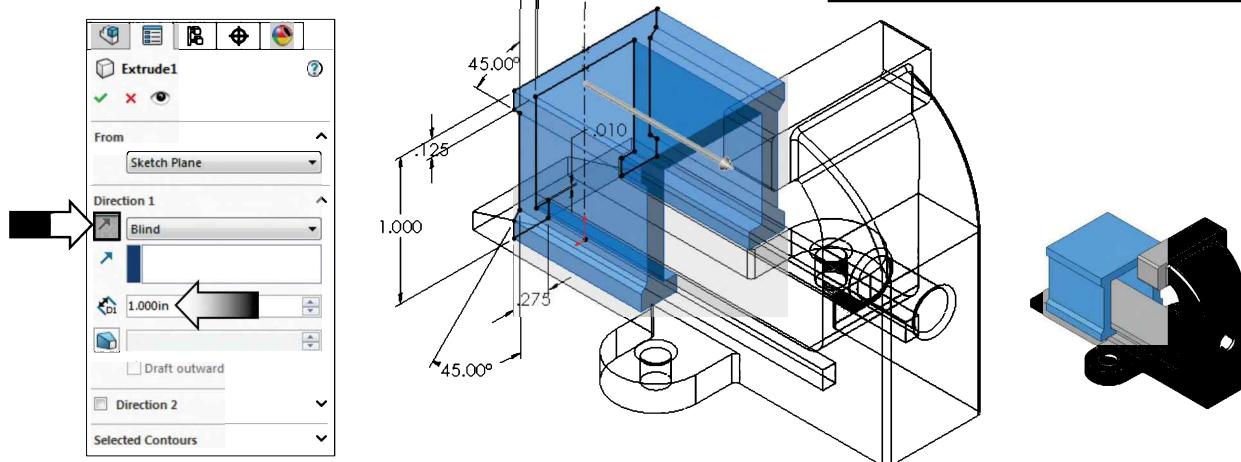
Extrude Depth: **1.000 in.**

Click **OK**.



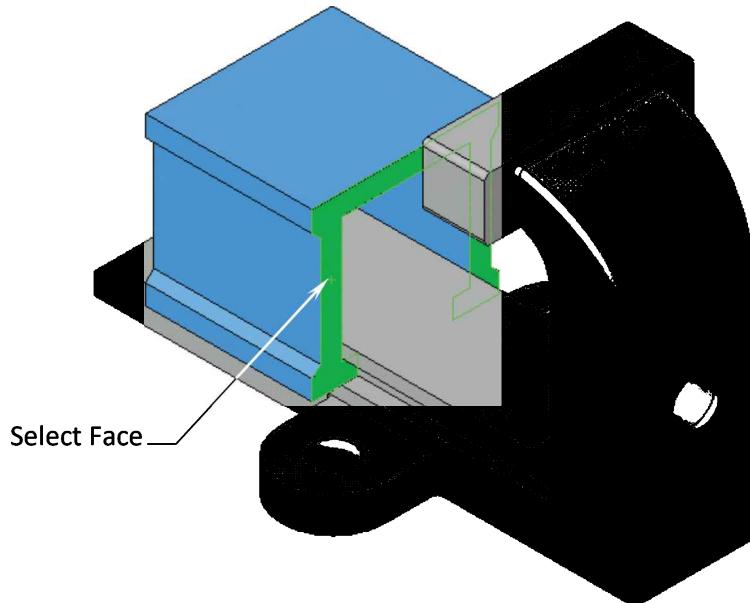
Transparency

* The transparent images are sometimes toggled on/off for clarity purposes.

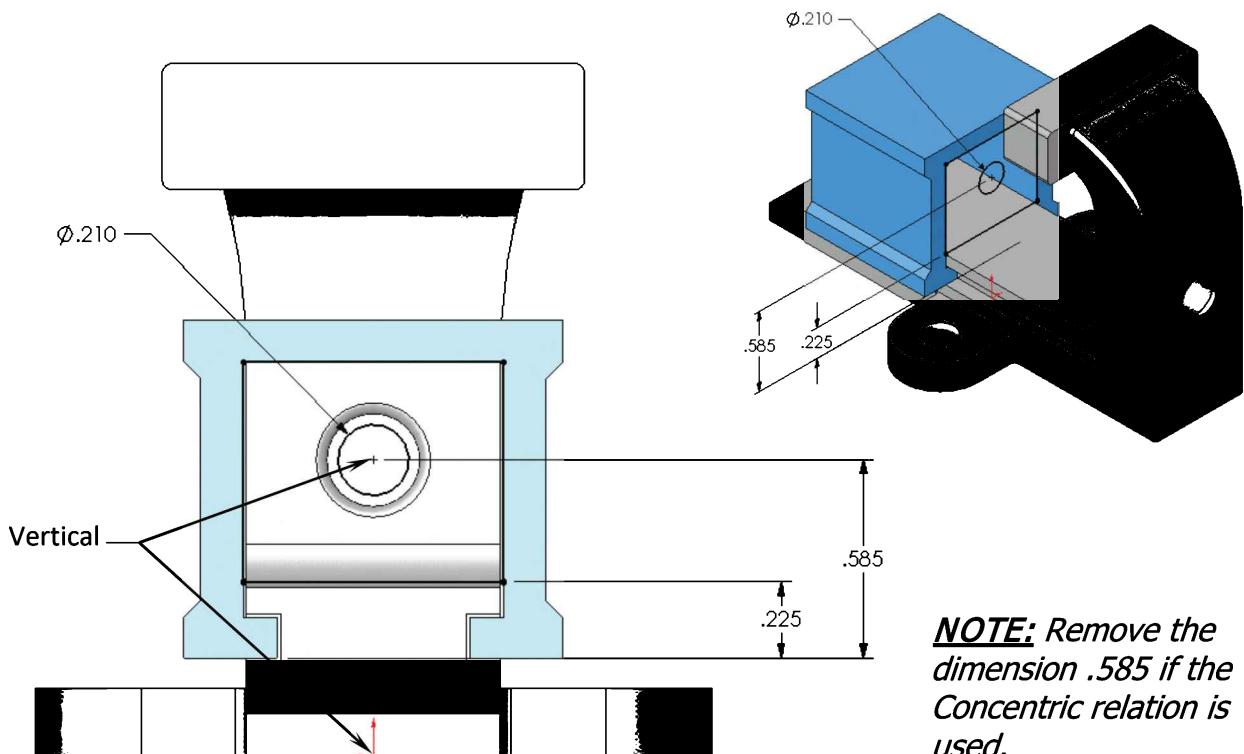


22. Adding the support wall:

Select the face indicated and open a new sketch  or select **Insert, Sketch**.



Sketch the profile and add the dimensions and relation as shown below to fully define the sketch.



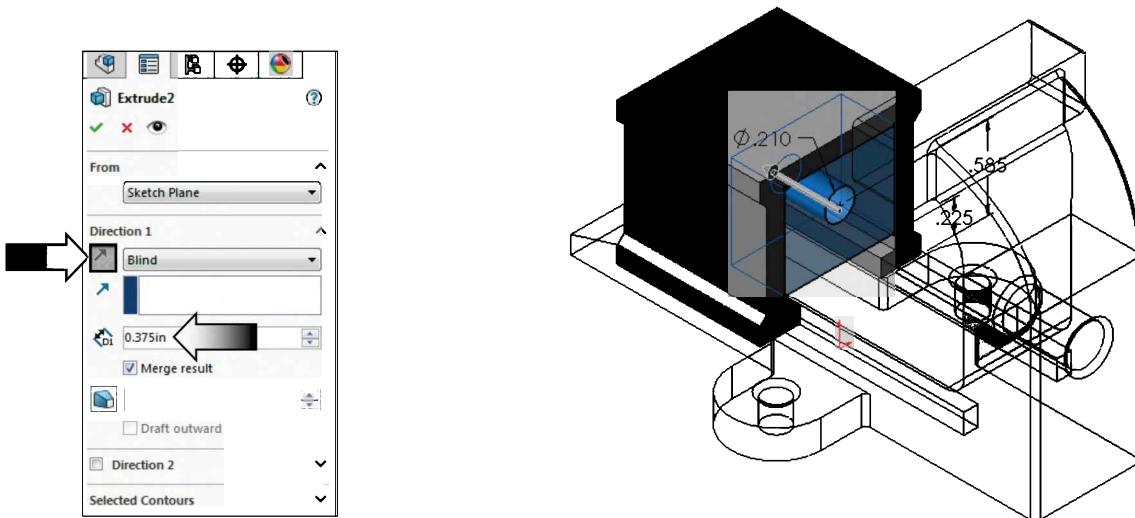
23. Extruding the Support Wall:

Click  on the Features tab or select **Insert, Base, Extrude**.

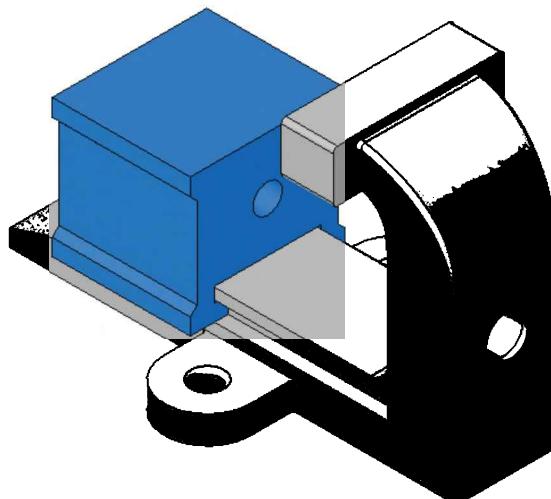
Direction 1: **Blind** and click the reverse direction arrow.

Extrude Depth: **.375 in.**

Click **OK**.



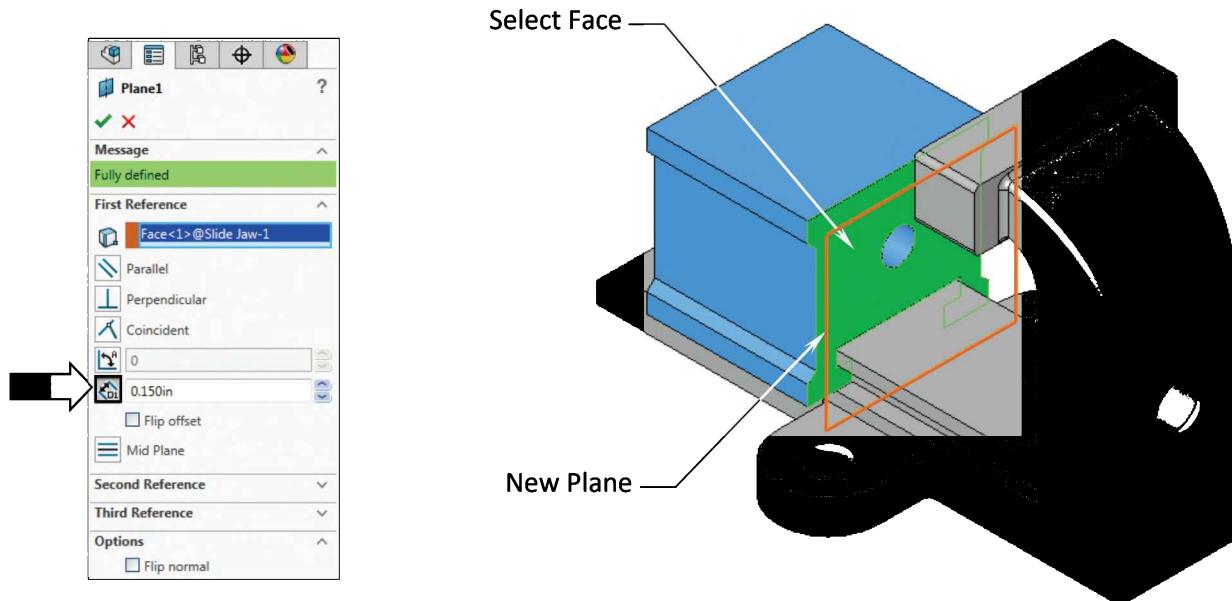
The Support Wall is built with a guide hole.



24. Creating a new work plane:

Select the face as indicated and click  or select: **Insert, Reference Geometry, Plane.**

Enter **.150 in.** for Offset Distance and place the new plane on the right side.



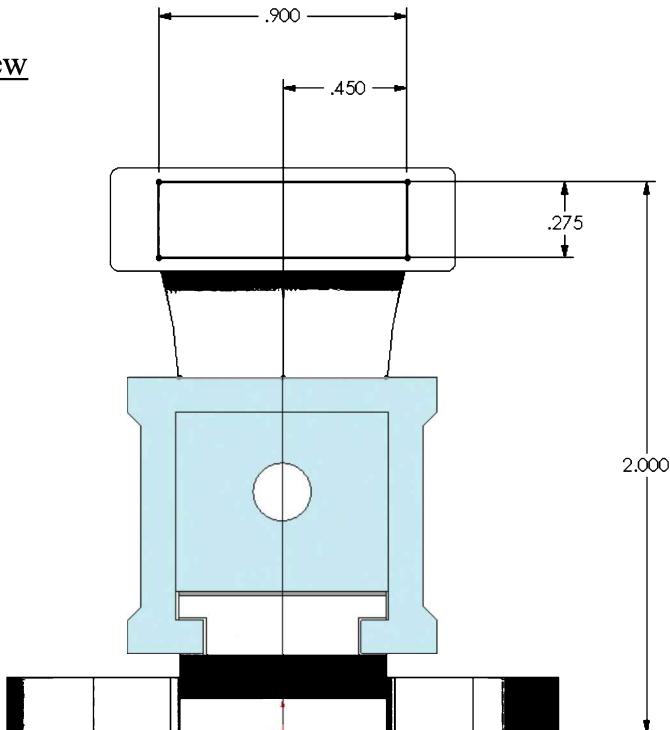
25. Creating the Slide Jaw 1st sketch:

Open a **new sketch**  on the new plane or select **Insert, Sketch.**

Sketch a **Rectangle** and add the dimensions shown to fully define the sketch.

(Sketch a vertical centerline and add a Symmetric relation to center the rectangle will also work fine.)

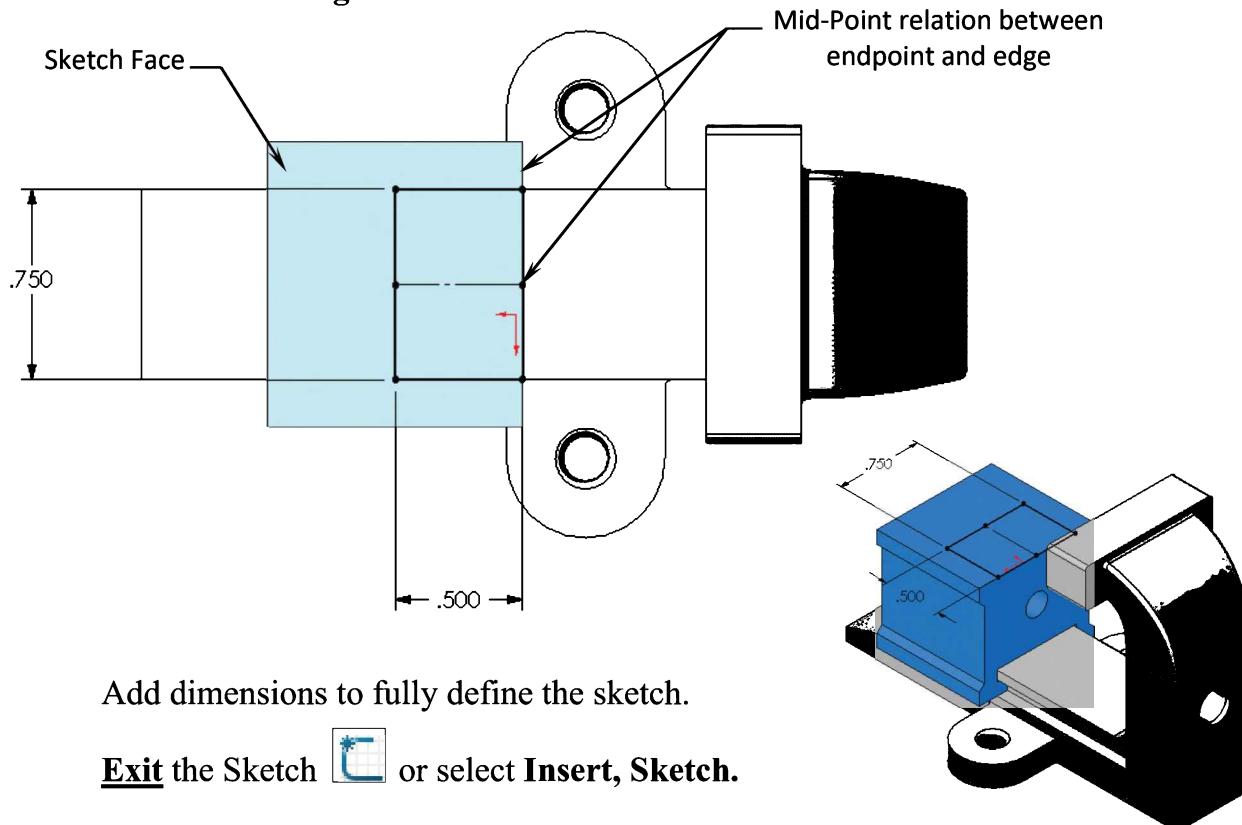
Exit the sketch  or select: **Insert, Sketch.**



26. Creating the Slide Jaw 2nd sketch:

Select the face indicated and open a new sketch  or select: **Insert, Sketch**.

Sketch a **Rectangle** as shown below.



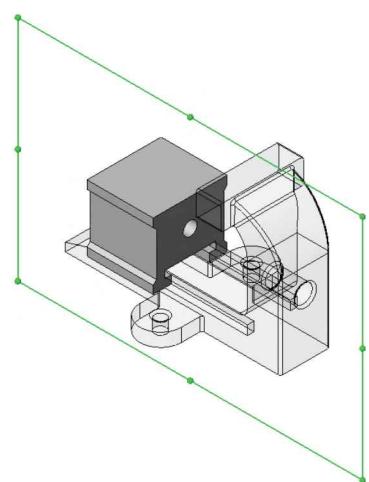
Add dimensions to fully define the sketch.

Exit the Sketch  or select **Insert, Sketch**.

27. Sketching the Guide Curve:

Select the Right plane of the part from the FeatureManager tree.

Click  to open a new sketch or select: **Insert, Sketch**.

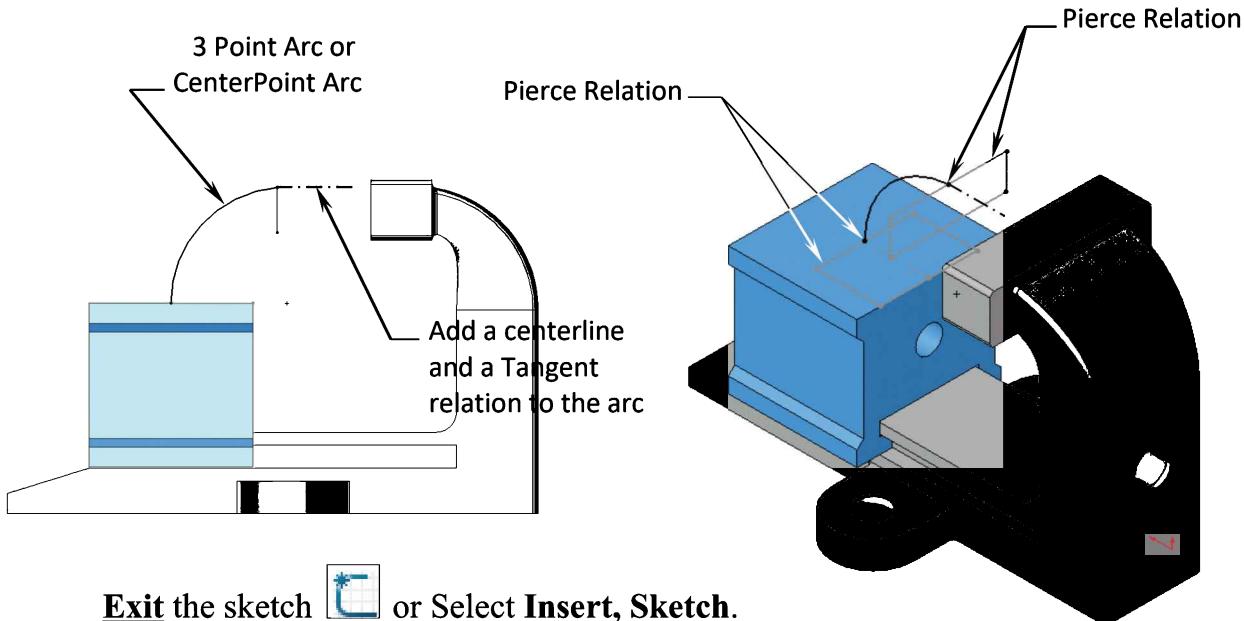


Guide Curves

- * Guide curves are used to control the profile from twisting as the sketch is swept along the path.
- * Guide curves are also used in Loft to control the transitions between the profiles.

Sketch either a **CenterPoint Arc**  or a **3-Point Arc**  that connects the two sketches.

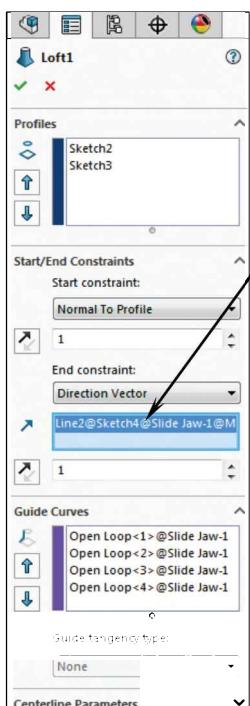
Add the relations shown below to fully define the sketch.



Exit the sketch  or Select **Insert, Sketch**.

28. Creating the Slide Jaw Loft:

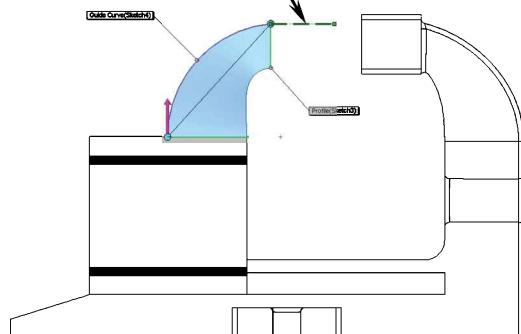
Click  on the Features tab or select **Insert, Boss-Base, Loft**.



For Loft Profiles, select the upper corners of the **2 rectangular sketches**.

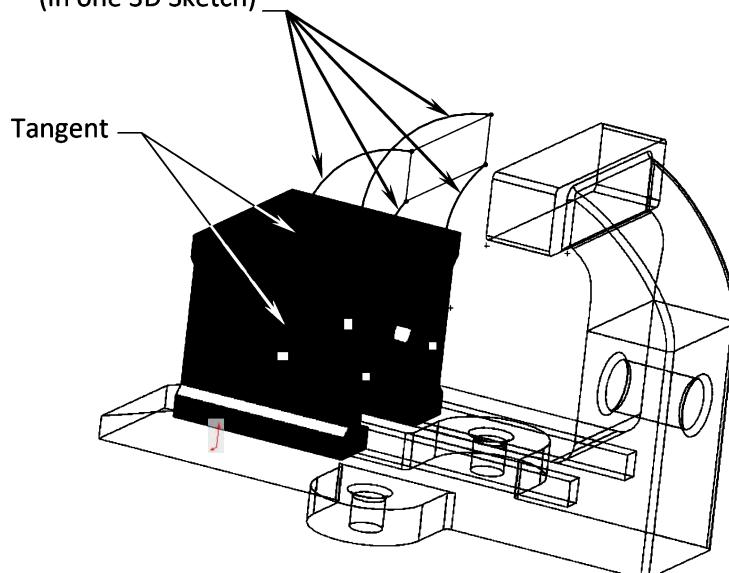
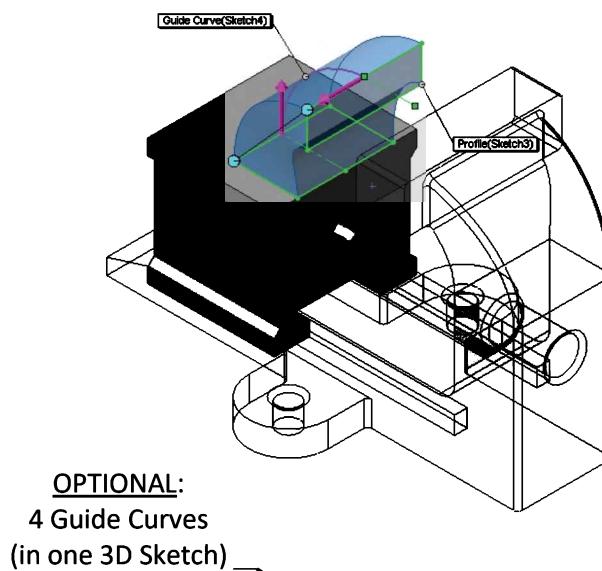
Expand the **Start/End Constraints** section and set the following:

- **Select line for Direction Vector**
- * **Start Constraint: Normal to Profile**
- * **End Constraint: Direction Vector** and select the centerline as noted.



Expand the **Guide Curves** section and select the **Arc** (sketch4) to use as a Guide Curve.

(See next page for more details.)



(Optional: Four guide curves can be used to increase accuracy of the loft if needed.)

Click **OK**.

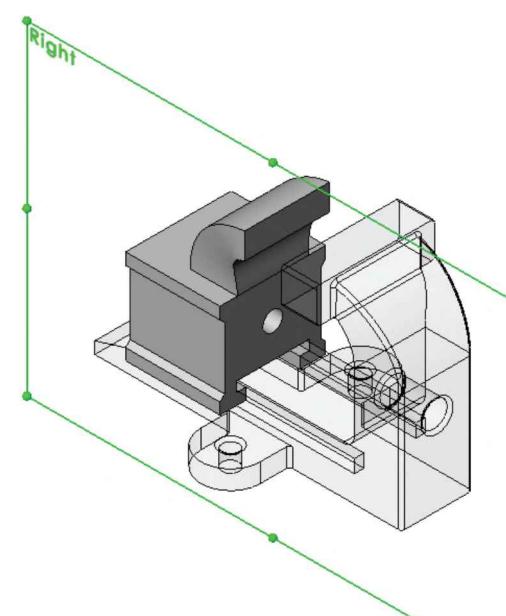
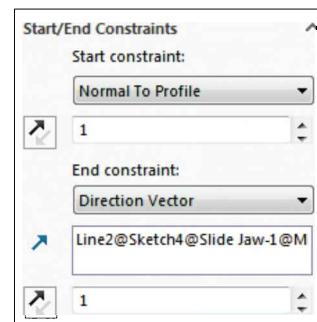
29. Creating the Clamp block:

Select the part's Right plane from the FeatureManager tree.

Click  to open a new sketch or select:
Insert, Sketch.

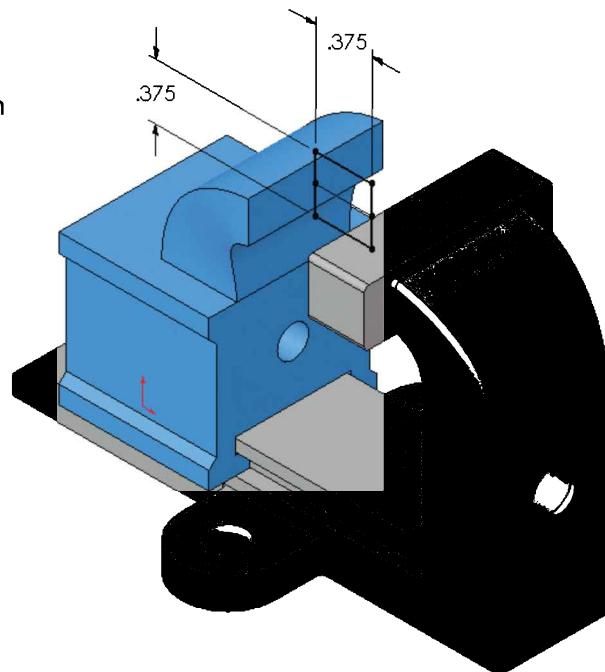
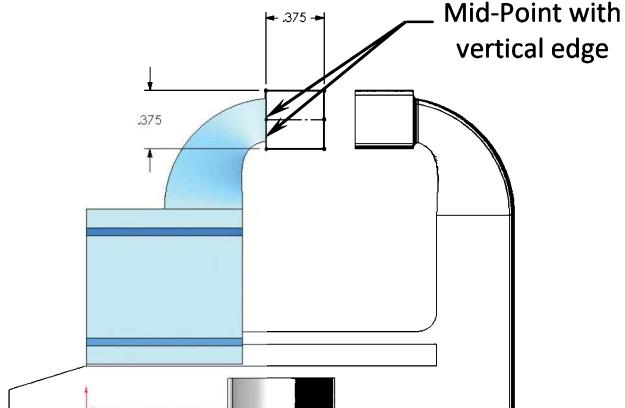
 **Start/End Constraints**

- * The Start constraint and End constraint option applies a constraint to control tangency to the start and end profiles.
- * The Direction Vector option applies a tangency constraint based on a selected entity used as a direction vector.



Sketch a Rectangle and add the dimensions shown.

Add a Centerline  in the center of the rectangle and position it on the Mid-Point of the vertical edge.



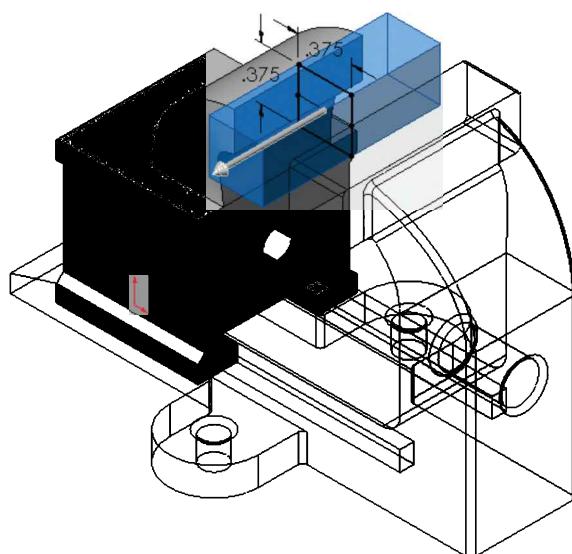
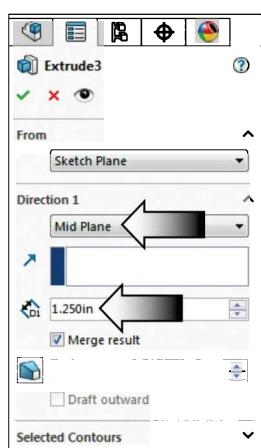
30. Extruding the Clamp block:

 Click  or select Insert, Boss-Base, Extrude.

Direction 1: Mid-Plane.

Extrude Depth: 1.250in.

Merge Result: Enabled.



Click OK.

31. Which option is better?

Instead of using the Mid-Plane extrude type, the **Up-To-Surface** option can be used to link the length dimensions of the 2 Clamp Jaws together.

Right-click on the last Extruded feature and select **Edit Feature**.

Change **Direction 1** from Mid-Plane to **Up-To-Surface** and select the **face** on the left side.

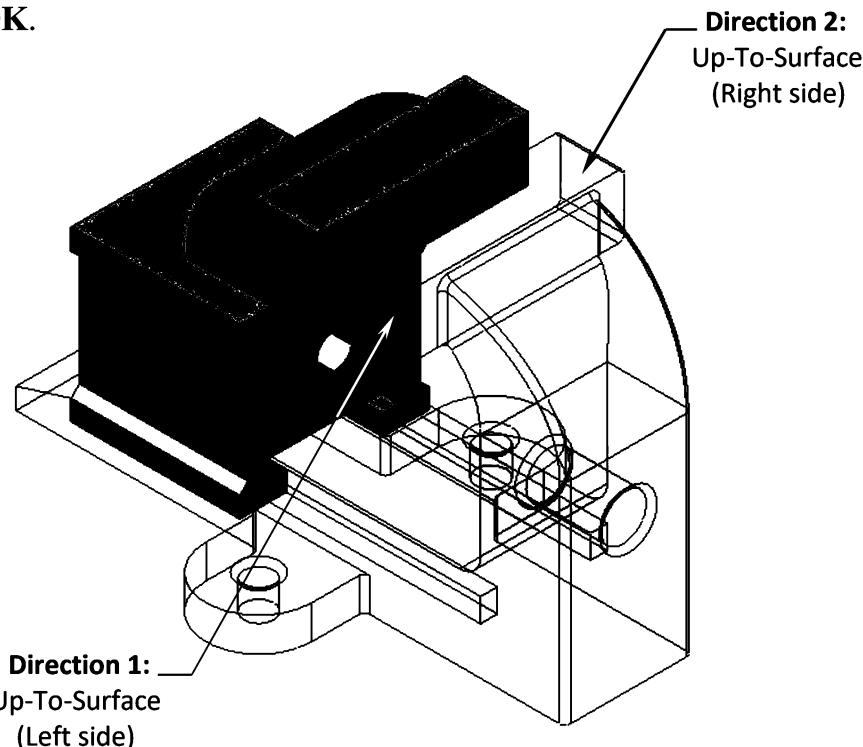
Change **Direction 2** to **Up-To-Surface** and select the **face** on the right side as indicated.

Click **OK**.



Up-To-Surface

- * Extends the feature from the sketch plane to the selected surface.
- * When the driving surface is changed in length, the referenced extruded feature will also be reflected.



External References:

SOLIDWORKS creates an external relation every time a face of another component is used as an extrude end-condition.

If the reference face is moved or rotated, the related feature will also move or rotate with it.

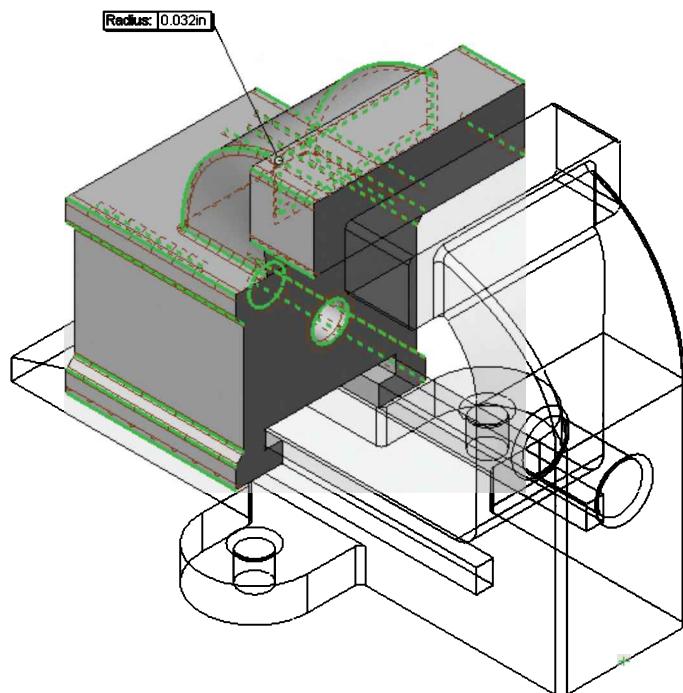
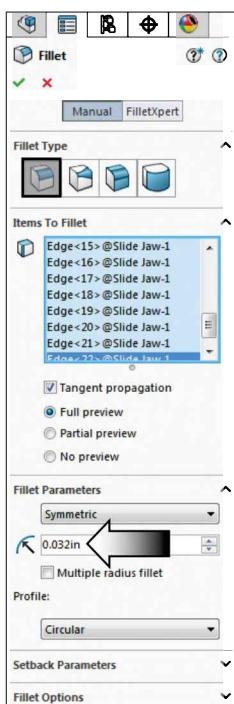
32. Adding fillets:

Click  or select **Insert, Features, Fillet-Round**.

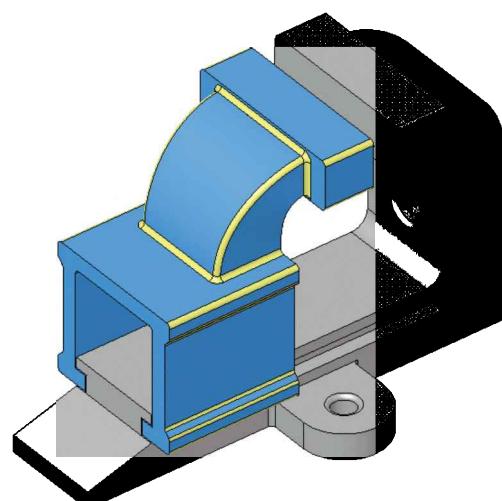
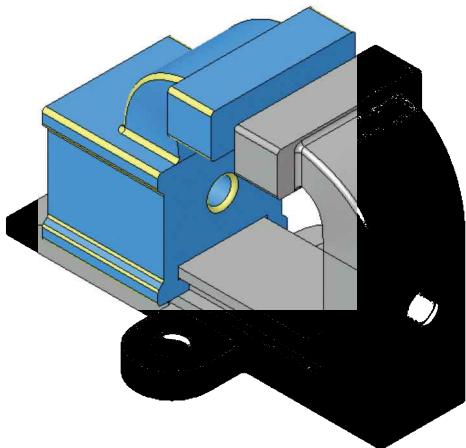
Enter **.032** for radius value.

Select the **edges** as shown.

Click **OK**.



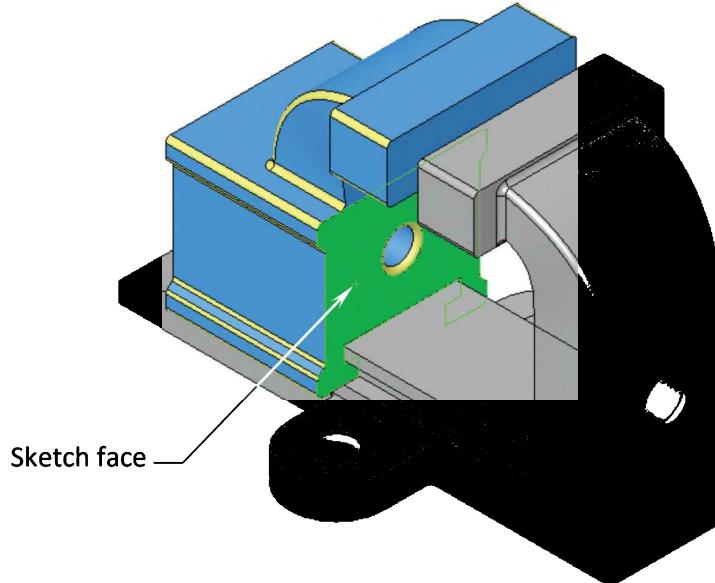
Inspect the fillets from the Front and Back Isometric views. Make any corrections if needed.



33. Creating the internal threads:

Starting with the sweep path.

Select the face indicated and open a new sketch  or select: **Insert, Sketch**.

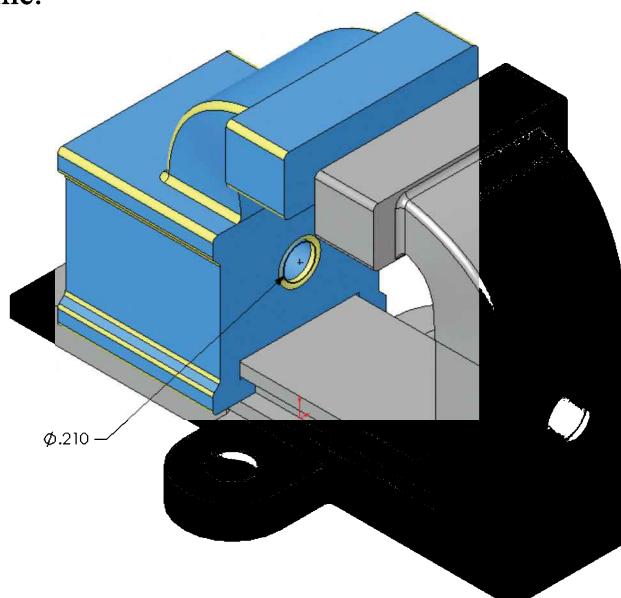


Sketch a **Circle** that is **Concentric** with the hole. (Converting the ID of the hole is another good way to link the diameter of the circle to the hole's diameter.)

Add a **$\text{Ø}.210$** dimension to fully define.

Wake up Center Points

- * With the Circle tool selected, hover the mouse cursor over the circumference of the hole, the 4 quadrant points appear, and the center-point of the circle is visible for snapping.



Select **Insert, Curve, Helix-Spiral.**

Enter the following parameters:

Defined by: **Pitch and Revolution.**

Pitch: **.080in.**

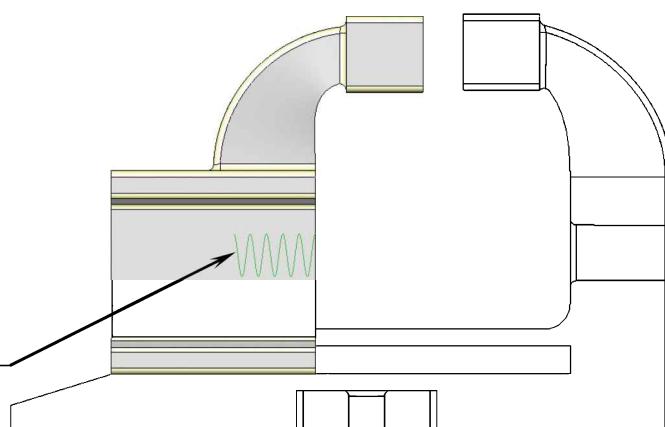
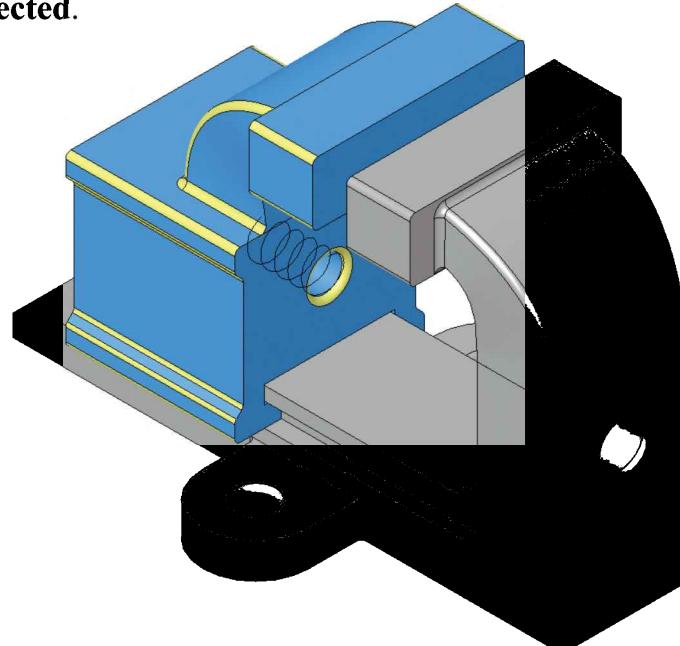
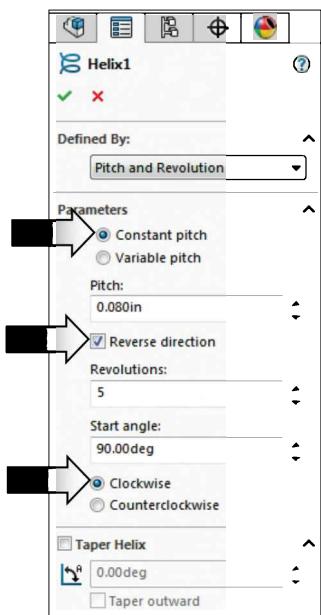
Revolution: **5.000.**

Starting Angle: **90.00 deg.**

Reverse Direction: **Enabled.**

Clockwise: **Selected.**

Click **OK.**



Sketching the Sweep Profile:

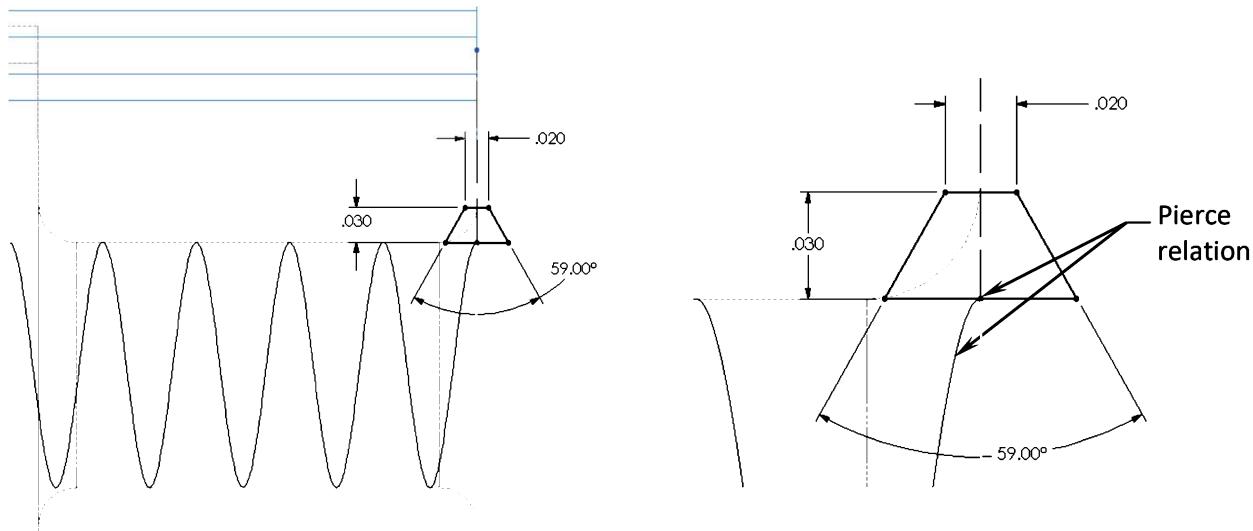
Select the Right plane of the part and open a **new sketch**  or select: **Insert, Sketch**.

Sketch the profile shown below.

Note: Use the **Mirror** option to keep the entities symmetrical.

Add dimensions and relations to fully define the sketch.

Add a **Pierce** relation between the endpoint of the Centerline and the 1st revolution of the helix.



Change to the **Front** view  and **Hidden Lines Visible** option .

Exit the sketch  or Select **Insert / Sketch**.

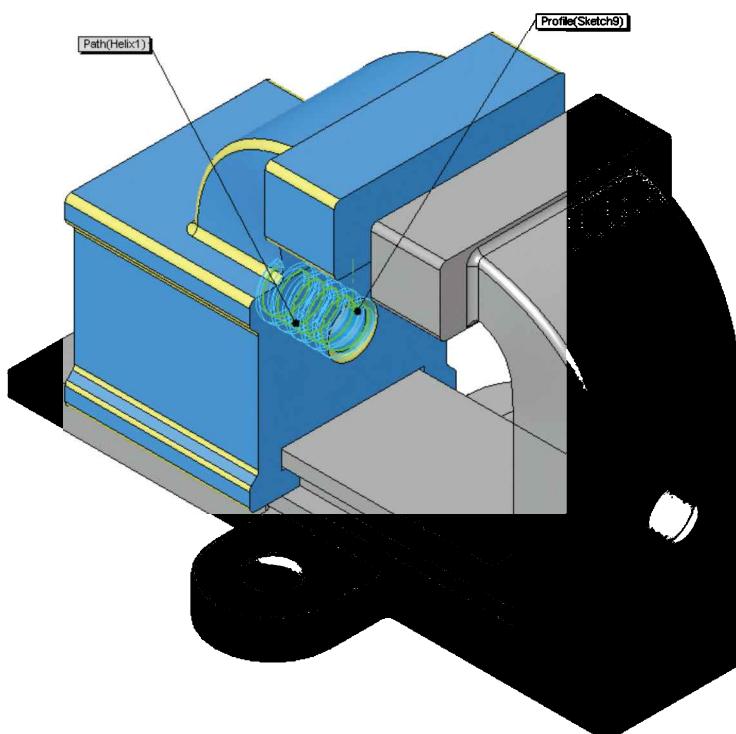
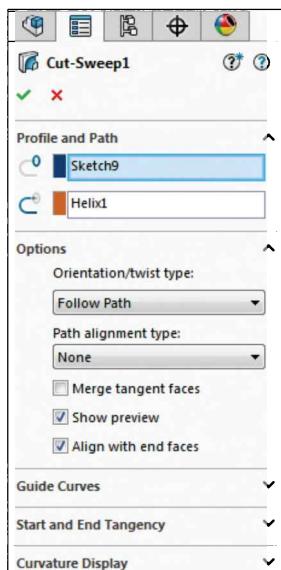
34. Sweeping the thread Profile along the Helix:

Click  or select **Insert, Cut, Sweep**.

For Sweep Profile, select the **Thread Profile**.

For Sweep Path, select the **Helix**.

Click **OK**.



35. Creating a Section View:

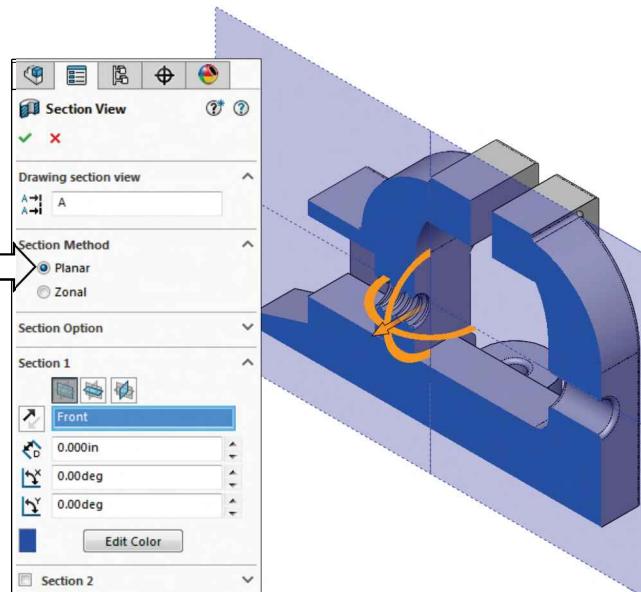


Click the **Section View** command or select **View, Display, Section View**.

Select the **Front** plane of the assembly for cutting plane.

Verify the details of the threads.

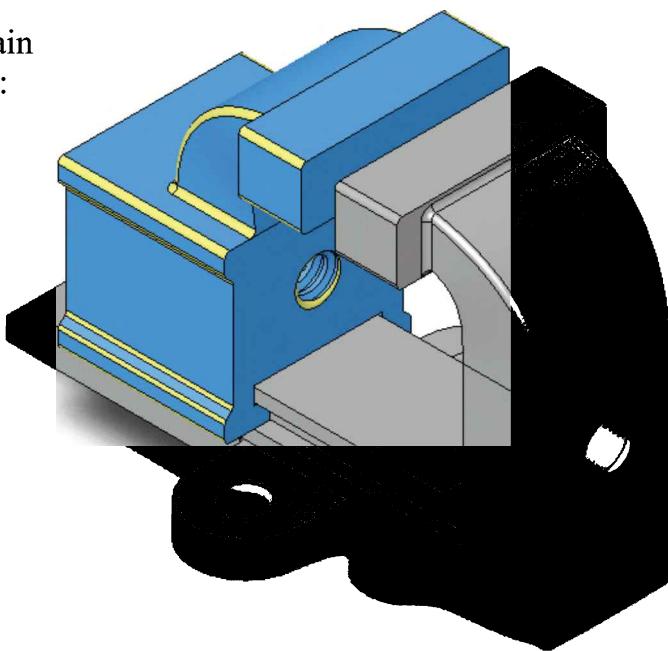
Click the **Section View** icon again to turn it off.



36. Saving your work.

Save your work once again using the same file name:
Mini-Vise.sldasm

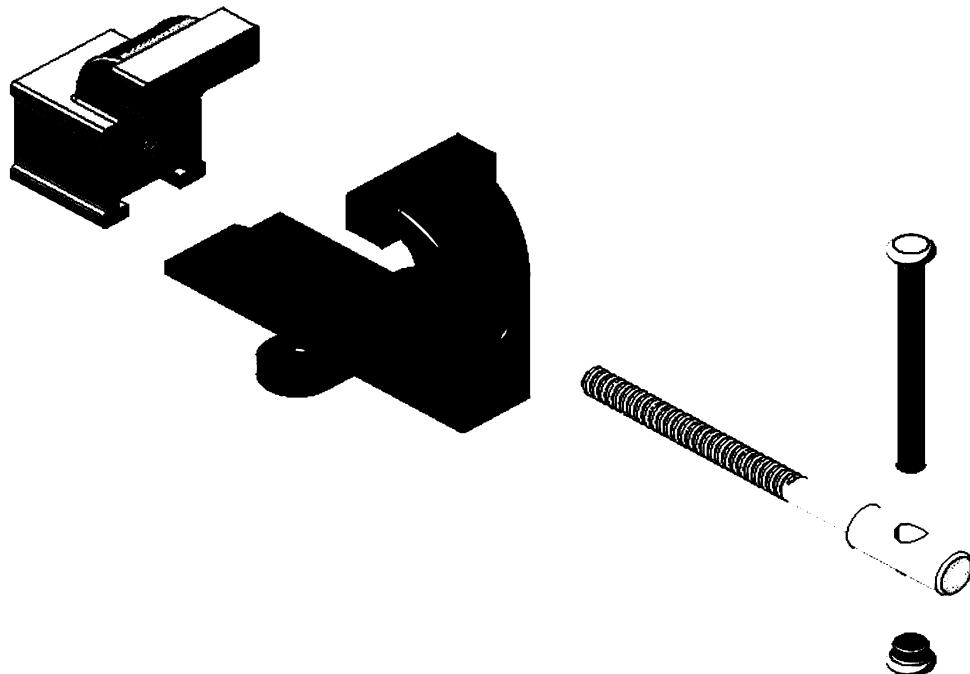
Overwrite the old file when prompted.

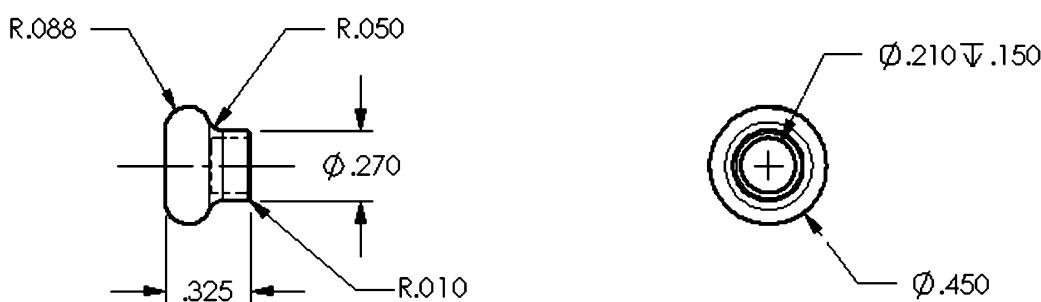
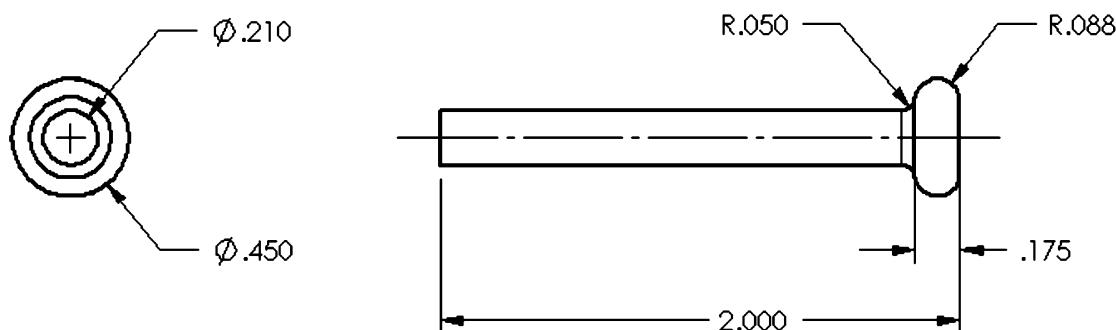
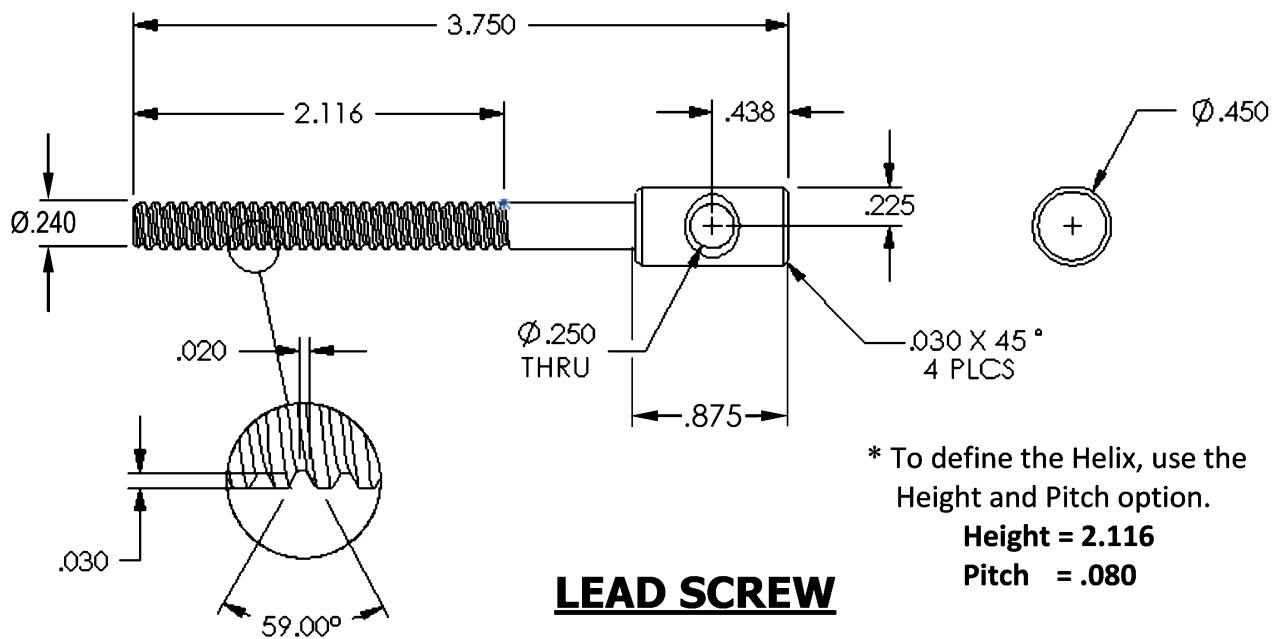


37. Assembly Exploded view (Optional):

Create the additional components: Lead Screw, Crank Handle, and Crank Knob using the Top Down Assembly method.

Create an assembly exploded view as shown (details on next page).





Questions for Review

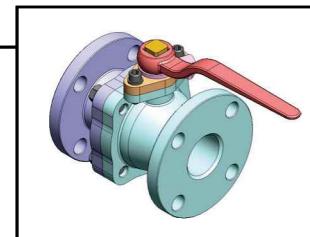
1. New parts can be created in the context of an assembly.
 - a. True
 - b. False
2. Geometry of other components such as sketch entities, model edges, and locations etc., can be used to construct the geometry of the new part.
 - a. True
 - b. False
3. Part documents can be inserted into an assembly using:
 - a. Insert menu
 - b. Windows Explorer
 - c. Drag and drop from an open window
 - d. All of the above
4. The suffix (f) next to the first part's name in the FeatureManager tree stands for:
 - a. Fail
 - b. Fixed
 - c. Float
5. When inserting new components into an assembly, the Inplace mates are created by the user.
 - a. True
 - b. False
6. Either in the part or assembly mode, the guide curves are used to help control the profiles from twisting, as they are swept along the path.
 - a. True
 - b. False
7. CenterPoint Arcs are drawn from its center, then radius, and angle.
 - a. True
 - b. False
8. The Link Values option allows a user to link only two dimensions at a time.
 - a. True
 - b. False

1. TRUE
2. TRUE
3. D
4. B
5. FALSE
6. TRUE
7. TRUE
8. FALSE

CHAPTER 21

Top Down Assembly – Part 2

Top Down Assembly – Part 2 Water Control Valve



When a component is built in the context of an assembly, external references are created to reference how it was constructed, and which plane or surface was used to create it with. Starting from the very Top level assembly, information regarding the new component is added and flows Down to the component level and gets repeated every time a new component is added.

For example: The mounting holes in the second part can be converted from the first, so that the hole diameters and the location dimensions are the same for both parts. When the holes in the 1st part are changed, the holes in the 2nd part would also change. Thus the sketch of the holes in the 2nd part is defined in the assembly, not by sketching and dimensioning them as in the part mode.

Using the Top Down assembly design, one of the better approaches is to use the geometry of the existing parts to create the new. This way several parts can be controlled and changed at the same time.

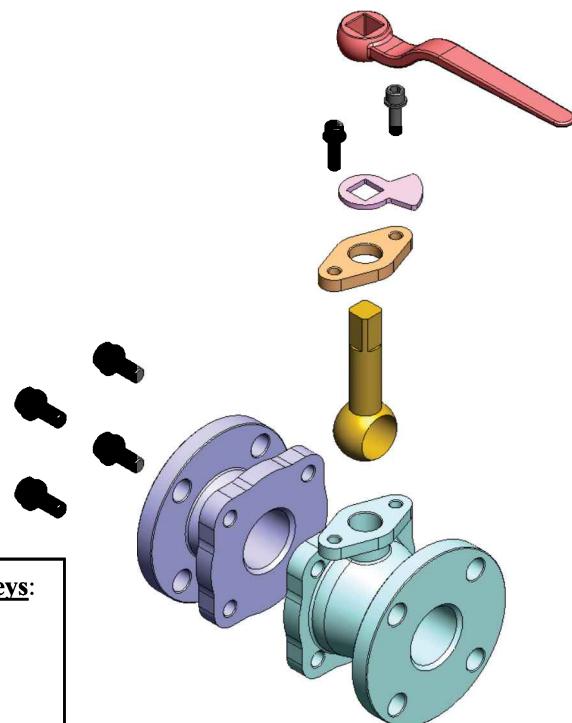
There are many advantages for creating parts in Top Down mode, and just to mention a few: not only is this method much quicker than the others due to the ability to use existing geometry to reference the new parts, but because all the parts are always visible in the assembly to help develop the Form of the new part and how it is supposed to Fit with other parts, it is more predictable how it is going to Function. Interference, friction and or clearance fits can be created and controlled within the very same screen.

However, there are a few things to consider when designing in Top Down mode:

- * External references are created to the geometry that the new part is referenced to.
- * When changes occur, the assembly updates all of its internal parts, and if drawings were made from these parts earlier, they will get updated as well.

Top Down Assembly – Part 2

Water Control Valve



View Orientation Hot Keys:

Ctrl + 1 = Front View
Ctrl + 2 = Back View
Ctrl + 3 = Left View
Ctrl + 4 = Right View
Ctrl + 5 = Top View
Ctrl + 6 = Bottom View
Ctrl + 7 = Isometric View
Ctrl + 8 = Normal To Selection

Dimensioning Standards: **ANSI**

Units: **INCHES** – 3 Decimals

Tools Needed:



Insert Sketch



Line



Circle



Add Geometric Relations



Sketch Fillet



Trim



Dimension



Centerline



Fillet/Round



Base/Boss Revolve



Extruded Boss/Base



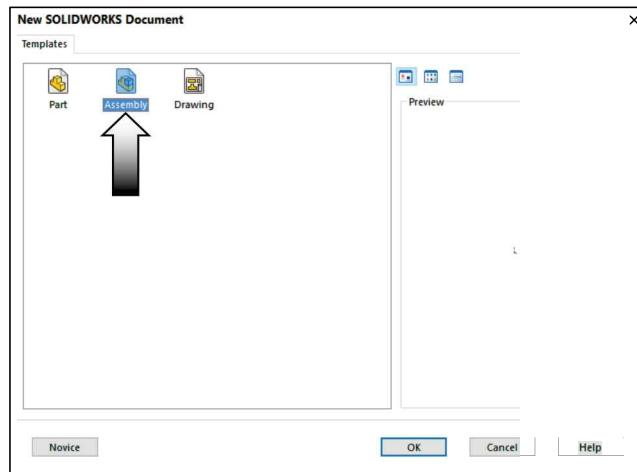
Edit Component

1. Starting with a new Assembly Template:

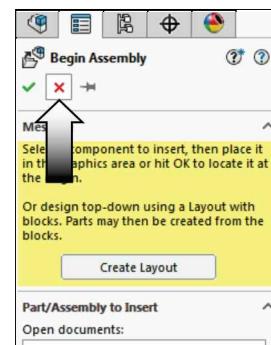
Click File / New.

Select an **Assembly** template either from the Template or the Tutorial tab.

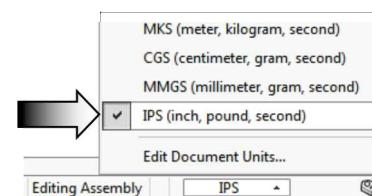
(Click the **Advance** button at the lower left corner of this dialog box if you do not see the similar templates.)



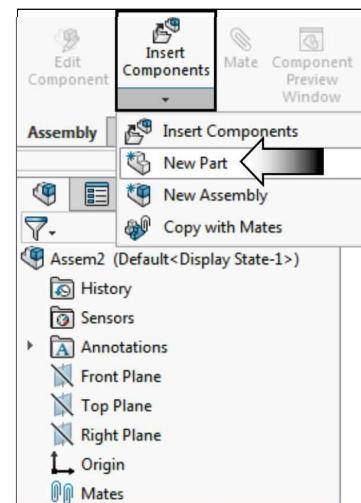
The **Begin Assembly** dialog appears on the left side; click **Cancel**. We are going to use a different approach to create the new components.



At the bottom right of the screen, set the Units to **IPS** (Inch, Pound, Second) 3 decimal places.



From the **Assembly** tab, expand the Insert-Components command and select: **New Part**.



Creating components in context of an assembly will require a few additional steps:

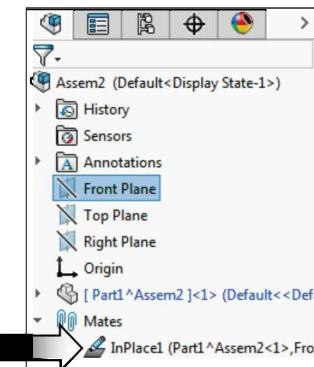
- A new part is inserted into an Assembly and the Part's name is entered.
- A plane is selected at this time to reference the new part.
- The Edit Component command is activated and the Sketch mode is enabled for the plane selected in step b.
- The active part will change to the blue color by default.

2. Creating the 1st component:

When the symbol appears next to your mouse cursor, select the **Front** plane on the Feature tree. An Inplace mate is created to reference the new part.

Press **Control + 1** to switch to the **Front** orientation.

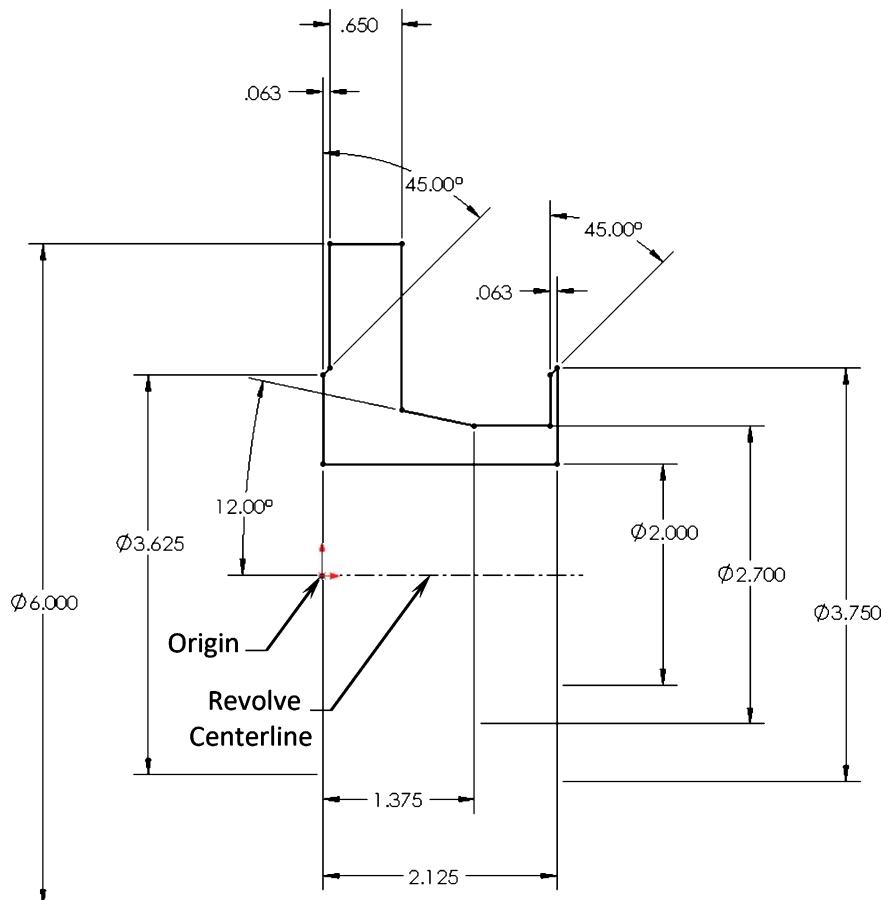
NOTE: To automatically rotate the sketch, go to:
System Options / Sketch, and enable the checkbox:
Auto Rotate View Normal to Sketch Plane...



3. Sketching the Base Profile:

Sketch the profile above the origin.

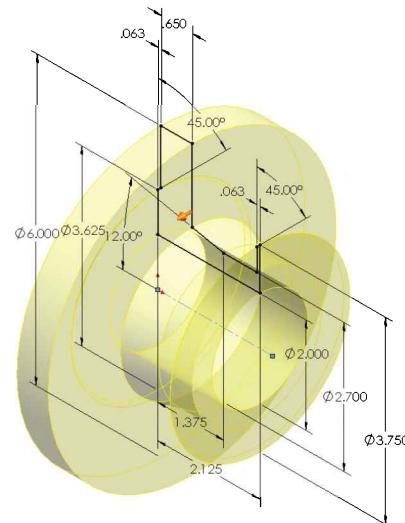
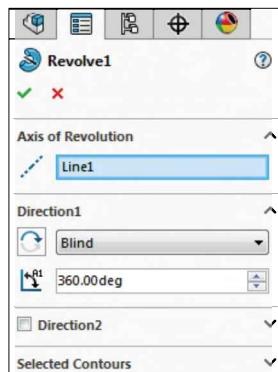
Add the dimensions shown below. The diameter dimensions are shown as Virtual Diameters.



Switch to the **Features** tab and click **Revolve Boss-Base** .

The centerline should be selected automatically.

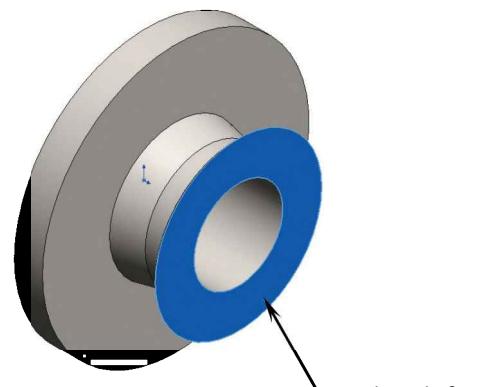
Use the default **Blind** option and revolve the sketch one complete revolution.



Click **OK**.

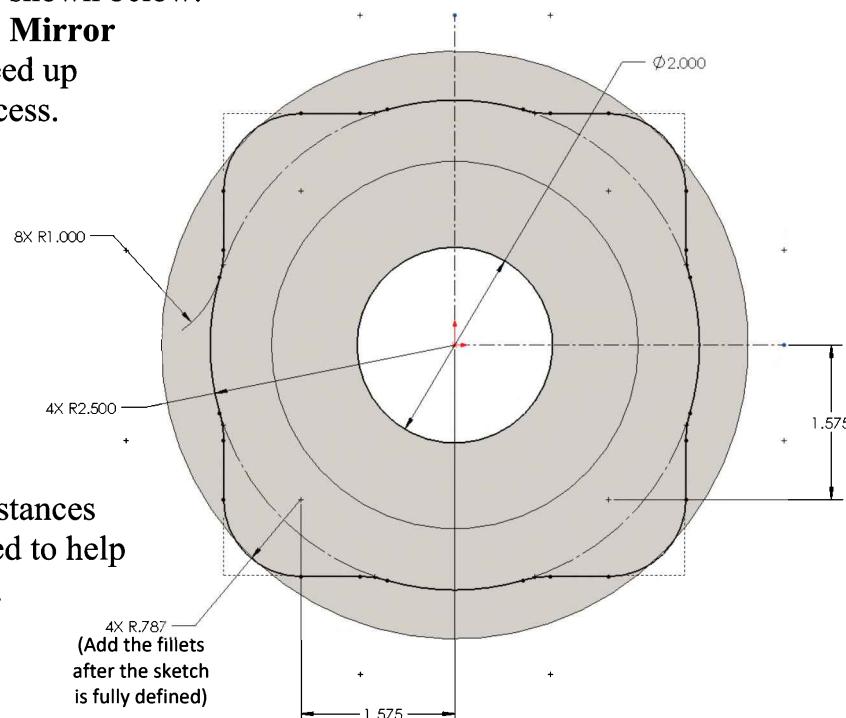
4. Adding the Inlet Flange:

Select the face indicated and open a new sketch.



Sketch the profile shown below. Use the **Dynamic Mirror** option to help speed up the sketching process.

Add the dimensions and relations needed to fully define the sketch.



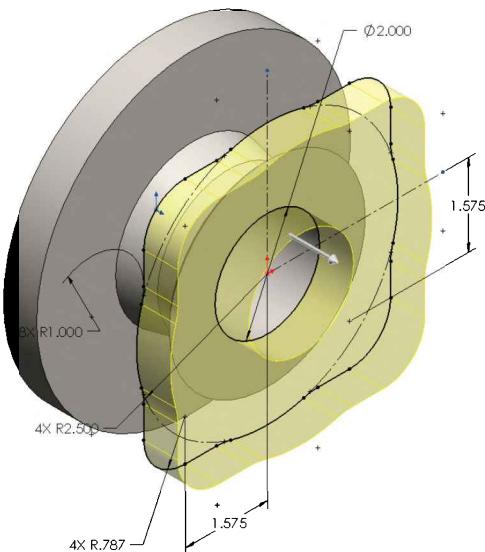
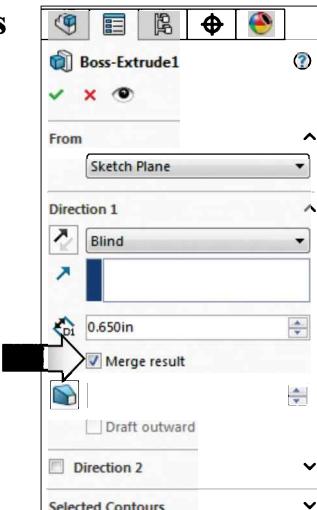
The number of instances (4X, 8X) are added to help clarify the sketch; you do not have to add them.

Click the **Features** tab and select:
Extruded Boss-Base.

Use the **Blind** extrude option.

Enter **.650"** for depth.

Enable the **Merge Result** checkbox.



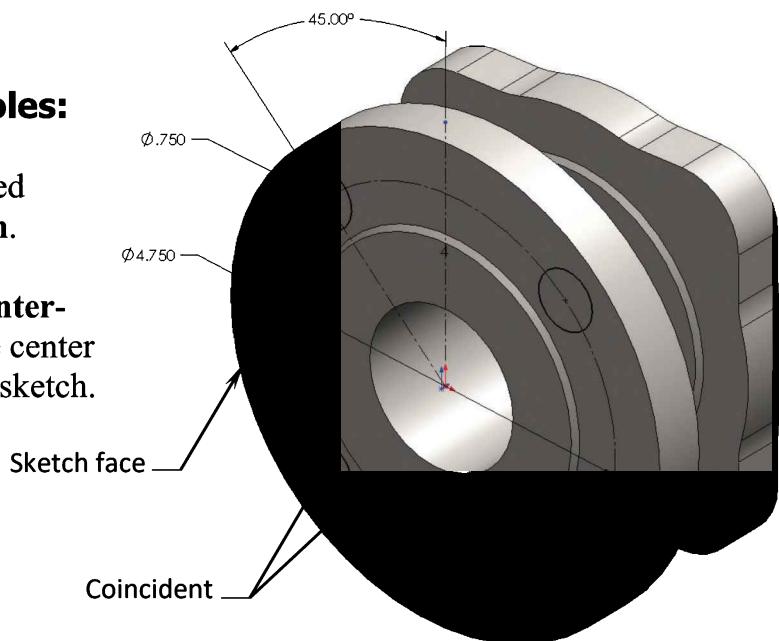
Click **OK**.

5. Adding the mounting holes:

Select the face indicated and open a **new sketch**.

Sketch a couple of **Center-lines** to help locate the center and directions for this sketch.

Add a **Circle** and either mirror it or circular pattern it 4 times around.

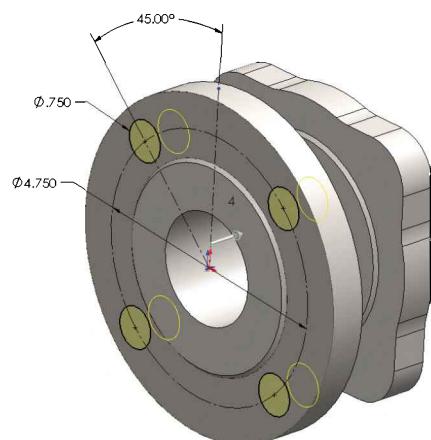
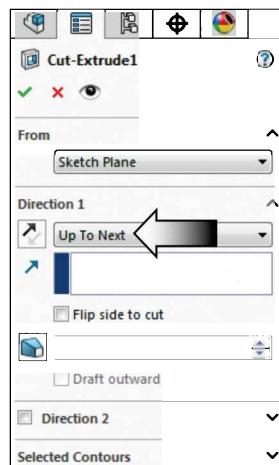


Add the dimensions and relations needed to fully define the sketch.

Click the Feature tab and select **Extruded Cut**.

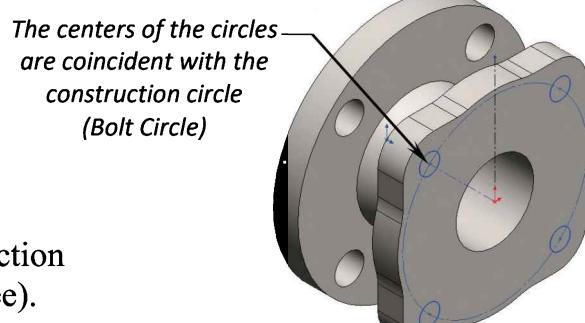
Select the **Up-To-Next** extrude option.

Click **OK**.



6. Adding other mounting holes:

Select the face as noted and open a new sketch.



Sketch a **Circle** and **convert** it to **construction** (click the For-Construction checkbox on the FeatureManager tree).

Add a couple of **Centerlines** as shown.

Sketch a smaller **Circle** and add the **Coincident** relation as noted.

Use the **Circular-Sketch-Pattern** option to array the small circle 4 times around.

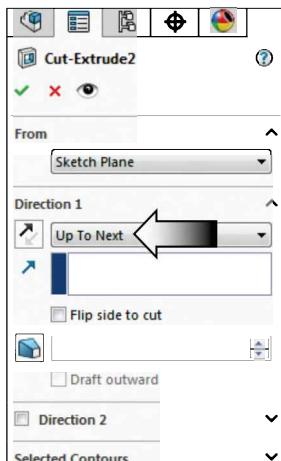
Add the dimensions and relations needed to fully define this sketch.

Coincident

Horizontal

NOTE: An additional Vertical or Horizontal relation between the centers of the small circles is needed when using the 2D sketch pattern.

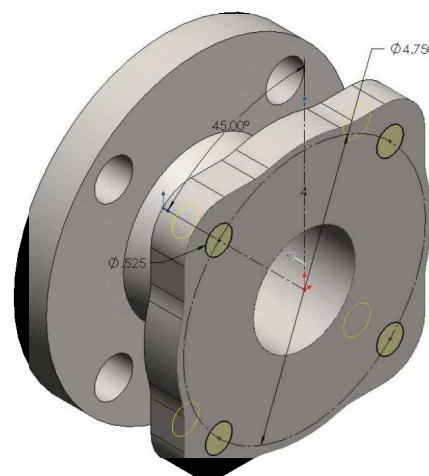
Switch to the Features tab and click **Extruded Cut**.



Use the **Up-To-Next** extrude option to ensure the cut only goes through the thickness of the flange.

Click **OK**.

Rotate the view to inspect the cut result.



7. Adding the .032" chamfers:

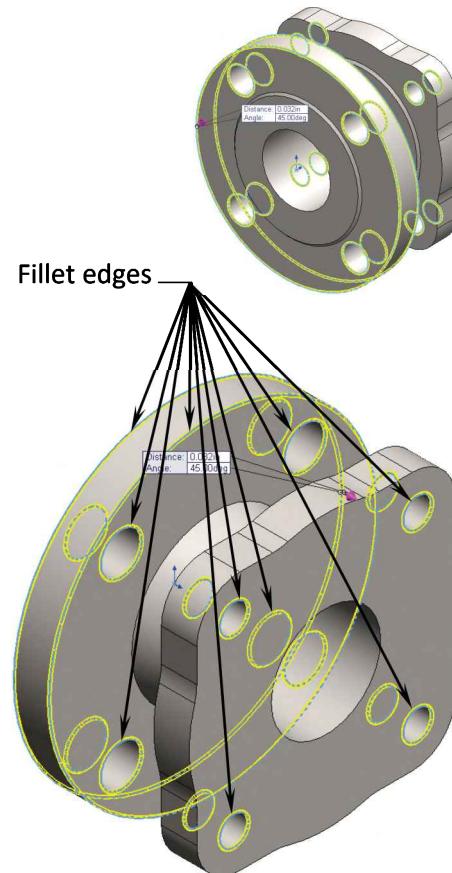
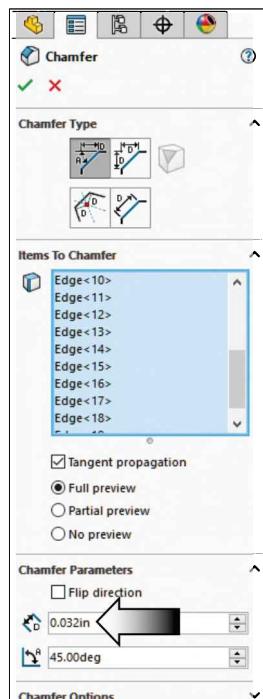
Click Chamfer
(under the Fillet drop-down).

Enter **.032"** for Depth and use the default **45°** angle.

Select the edges of the **8 holes** and the 2 edges of the round flange.

To deselect an edge simply click it once again.

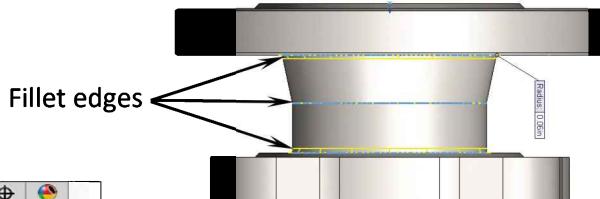
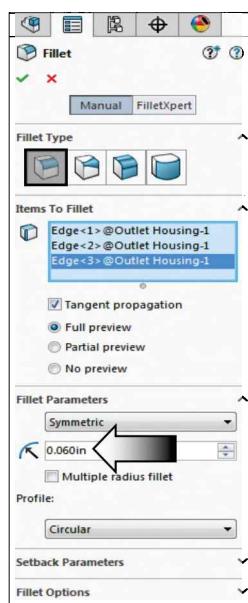
Click **OK**.



8. Adding the .060" fillets:

Click the **Fillet** command.

Enter **.060"** for radius.

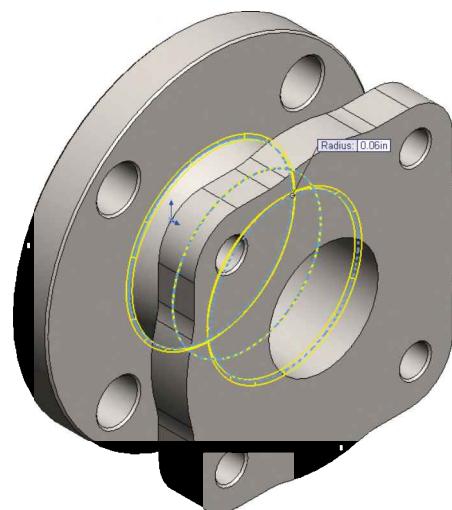


Select the **3 edges** of the transition body.

Enable the **Full Preview** checkbox.

Change to the **Top** orientation (Control+5) to inspect the selection.

Click **OK**.



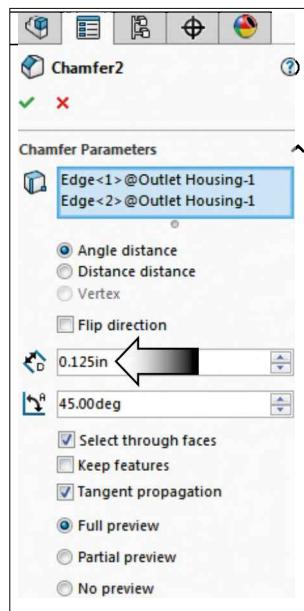
9. Adding the .125" chamfers:

Click Chamfer once again.

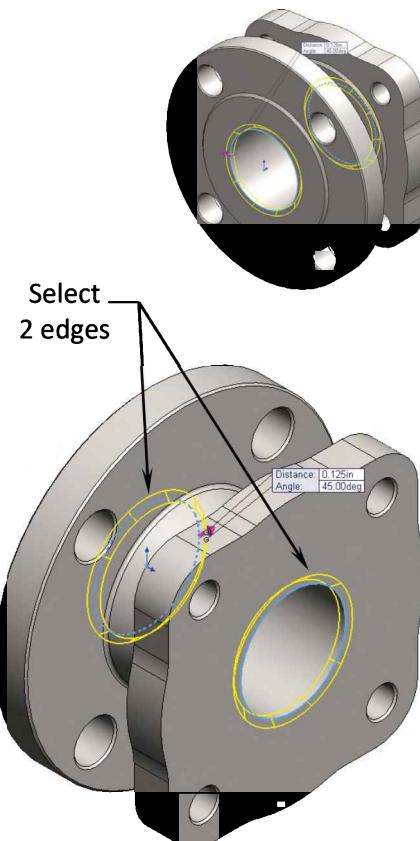
Enter **.125"** for Depth and use the default **45°** angle.

Select the **2 edges** of the center hole.

Selecting the face of the hole would get the same result as selecting the **2 edges**.

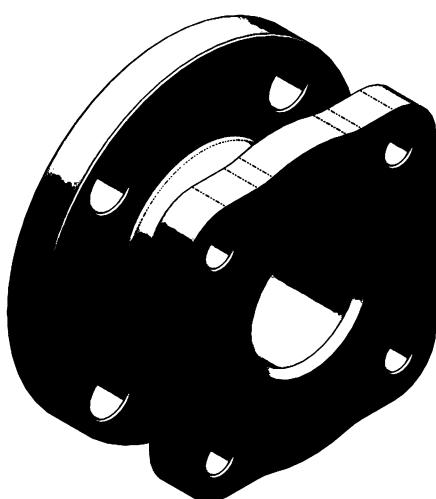
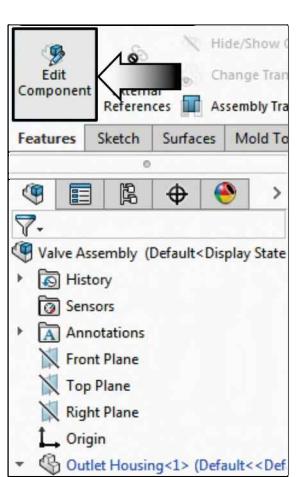


Click **OK**.



10. Exiting the Edit Component mode:

Click-off the **Edit Component** button to return to the Edit Assembly mode.

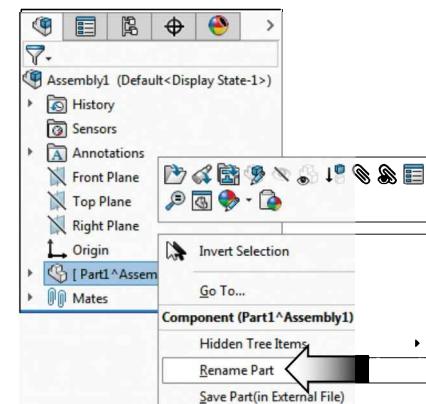


When the Edit Component command is not active, the part's color changes back to its default color (gray).

11. Renaming the component:

Right-click the name of the part (Part1) from the FeatureManager and select: **Rename Part** (arrow).

Enter **Outlet Housing** and press **enter**.

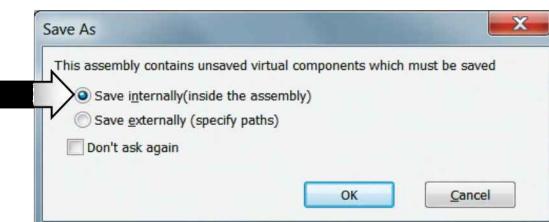


12. Saving as Virtual Component:

Virtual components are quite useful in the Top-Down Assembly mode. These components are saved internally, or embedded in the assembly document, instead of as separate part or sub-assembly documents.

Click **File, Save As**.

Enter: **Water Control Valve** for the file name and press **Save**.



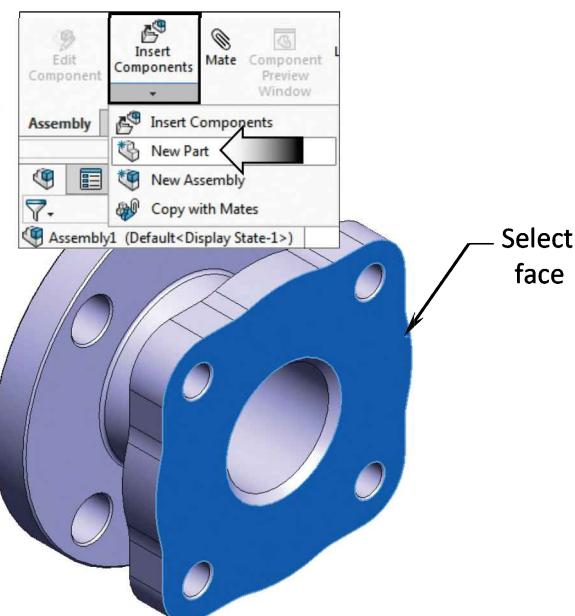
The Save As Virtual Component dialog appears; click the **Save Internally** option (Inside the Assembly) and click **OK**.

When the parent assembly is opened, all virtual components are also loaded into RAM. The virtual components can then be opened so that the detail drawings can be generated from them, or they can simply be saved as external part documents to share with others.

13. Creating the 2nd component:

Click the **New Part** command under the Insert Components drop down.

When the symbol appears next to your mouse cursor, click the Face of the flange as indicated.



At this point, another Inplace mate is created for the new part.

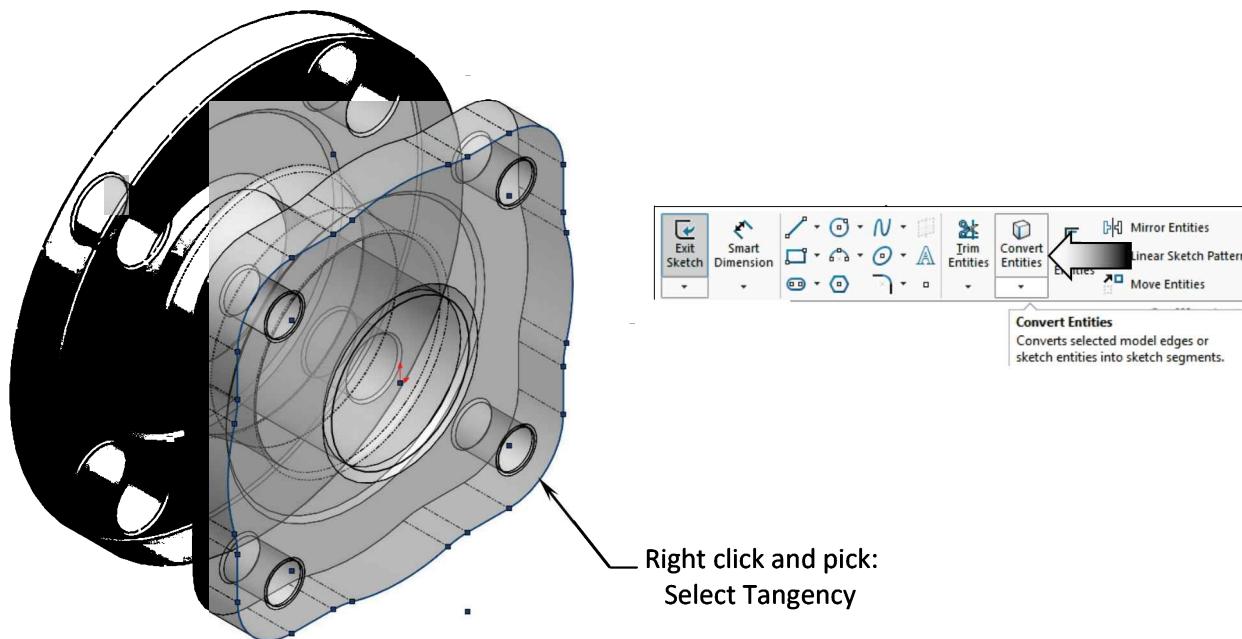
A new (Part2) component is created on the FeatureManager tree.

The Outlet Housing changes to transparent (inactive).

The **Edit Component**  command is activated.

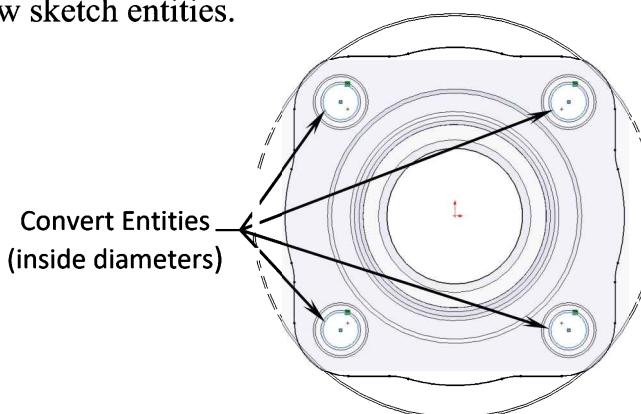
A new **Sketch** is also enabled automatically.

Right-click one of the **outer edges** and pick **Select Tangency**.



Press **Convert Entities** .

The selected edges are
converted to new sketch entities.



Also convert the **circular edges** of the 4 holes.

Switch to the **Features** tab.

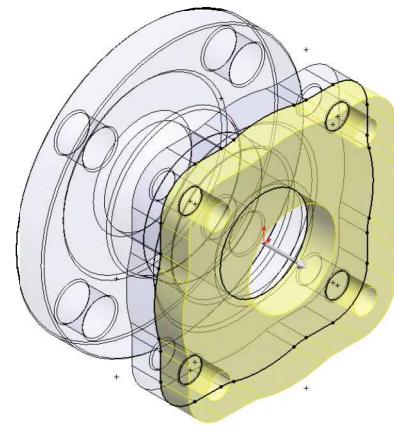
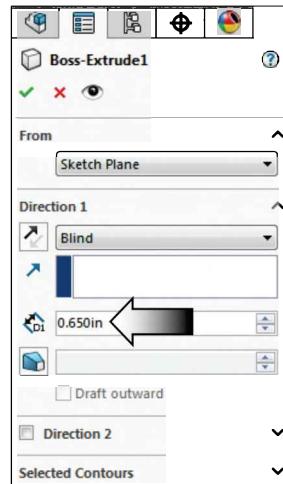
Click Extruded Boss-Base.

Use the default **Blind** extrude option.

Enter **.650"** for extrude depth.

Click **OK**.

The new flange is created by converting the geometry of the 1st component.

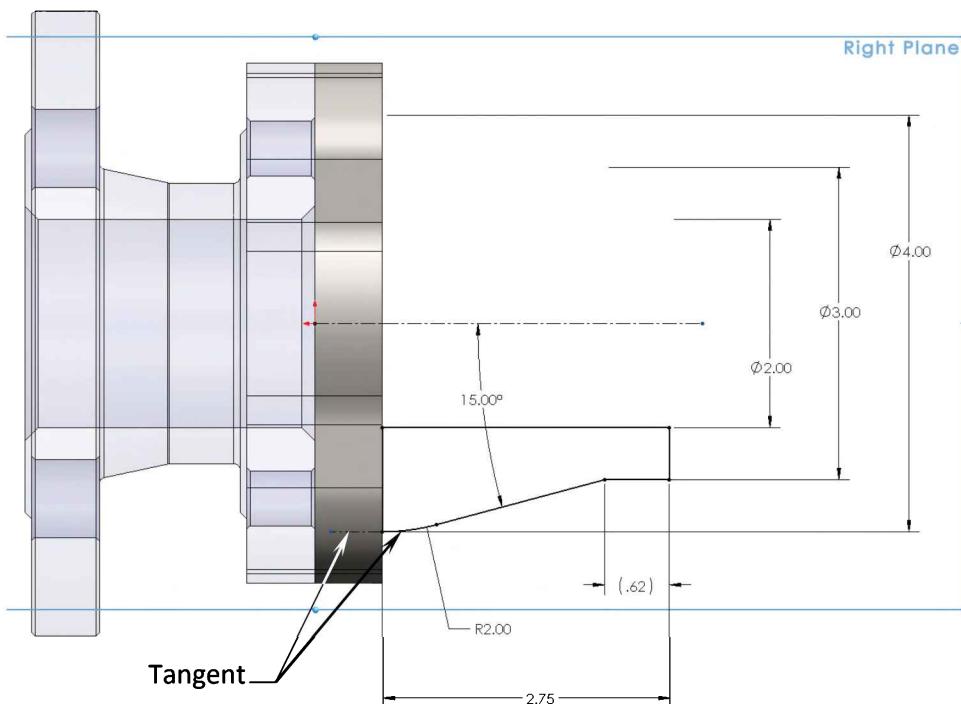


If the 1st component is changed the 2nd component will also change. We will take a look at some of the changes toward the end of this chapter.

14. Creating the transition body:

Select the component's Right plane and open a **new sketch**.

Sketch the profile shown below and add the dimensions and any relations needed to fully define this sketch. (Add the R2.00 fillet after fully defined.)

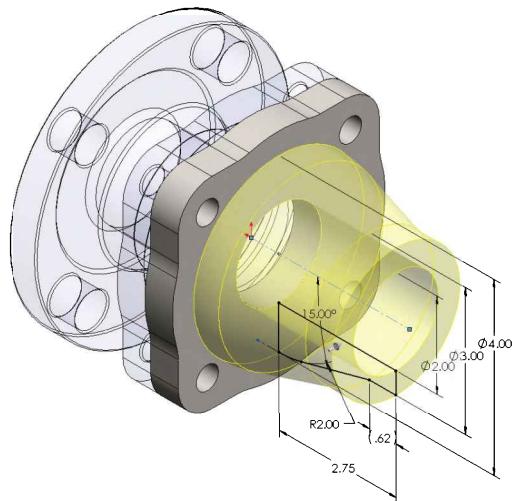
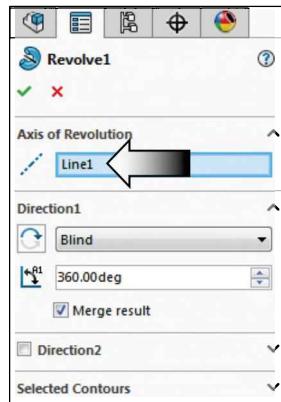


Switch to the **Features** tab and click **Revolve Boss-Base**.

The revolve centerline is selected by default.

Use the default settings:

- * **Blind**
- * **360deg**

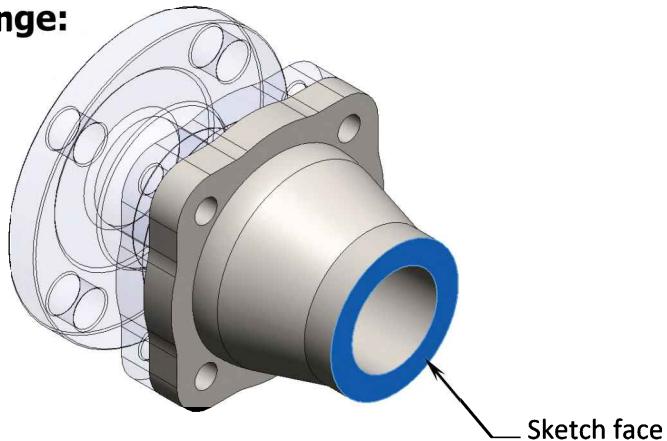


Click **OK**.

15. Adding another mounting flange:

Select the face as indicated and open a **new sketch**.

Hold the control key and select the **circular edge** of the round flange and the **4 edges** of its mounting holes (arrow).

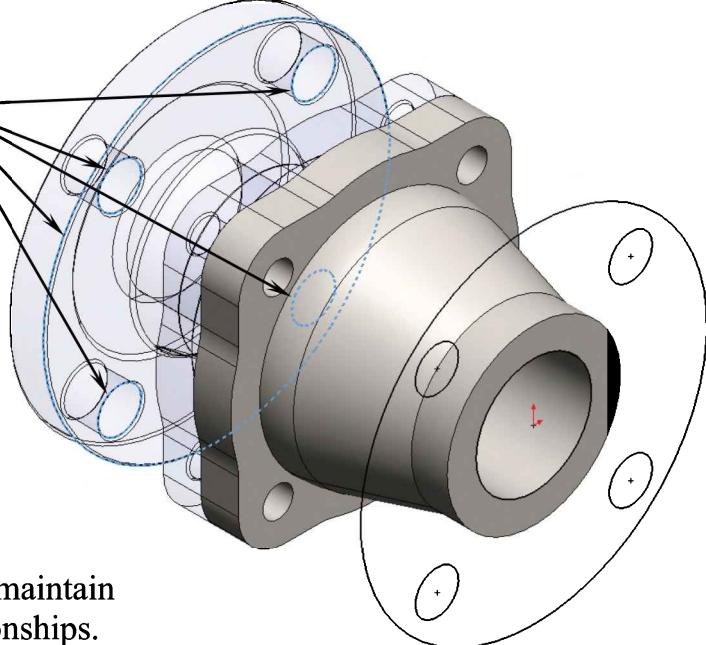


Click **Convert Entities**.

Convert 5 edges
(4 inside diameters
and 1 outer flange)

The selected edges are converted and projected to the sketch face.

Each sketch entity is linked to the original geometry where it was converted from. A relation called **On-Edge** is added to maintain their parent and child relationships.



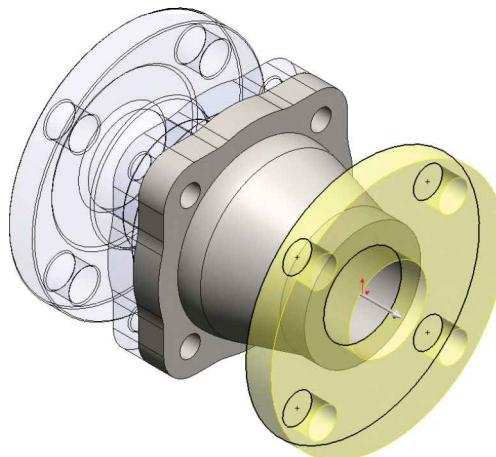
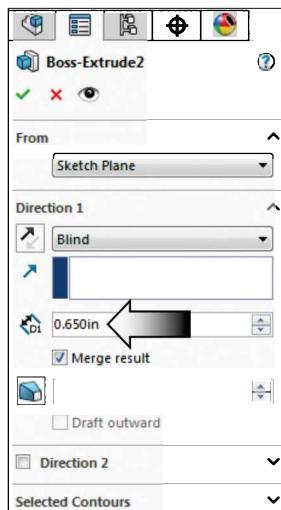
Click Extruded Boss-Base.

Use the default **Blind** option.

Enter **.650"** for depth.

Extrude direction is outward.

Click **OK**.

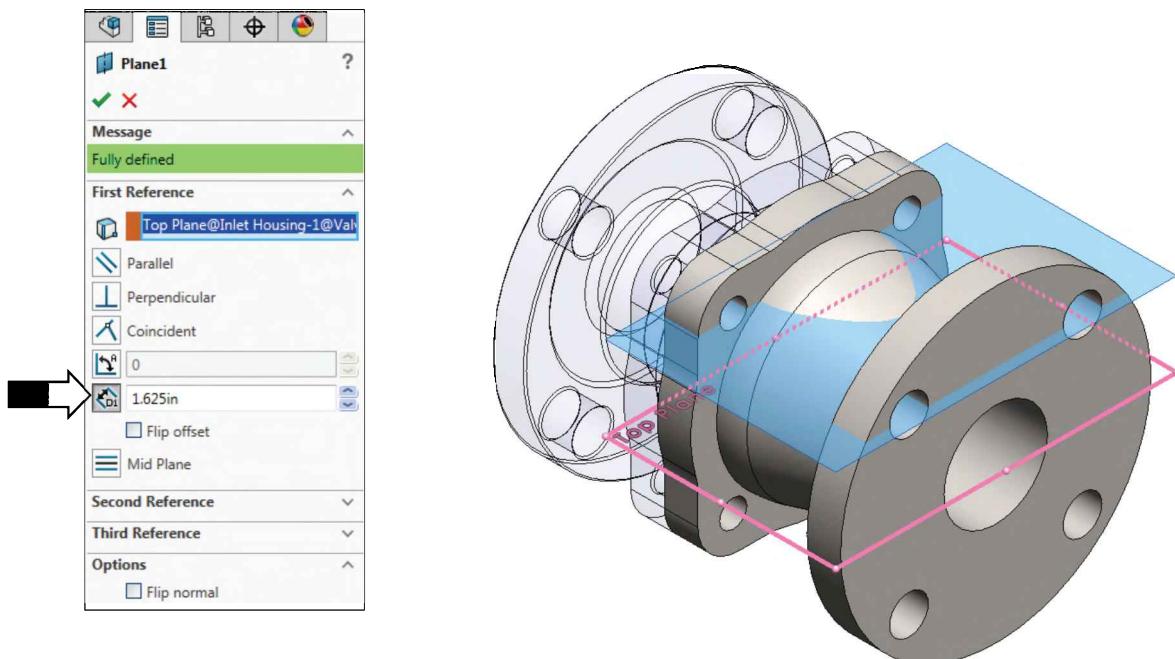


16. Adding an Offset-Distance plane:

Select the Top plane of the part from the FeatureManager tree.

From the Features tab, click **Reference Geometry / Plane**.

The **Offset Distance** option should be selected by default; enter **1.625"** for distance, and place the new plane above the Top plane.



Click **OK**.

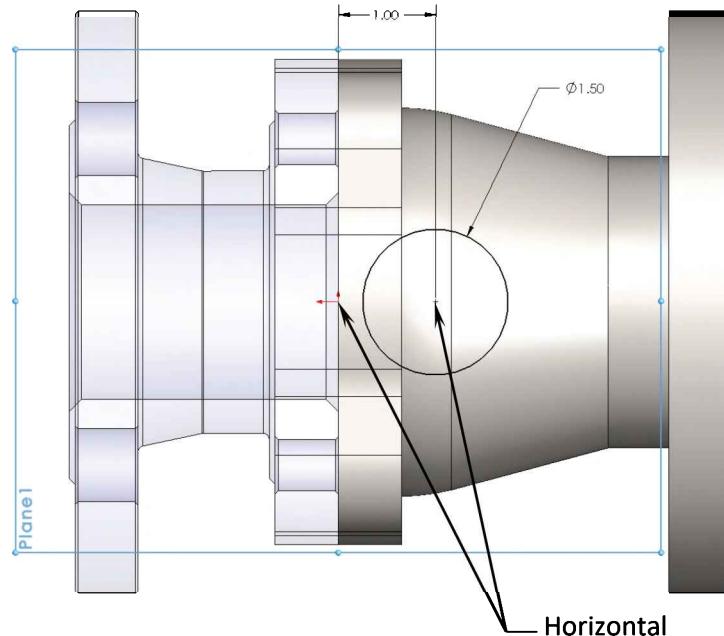
17. Adding a circular boss:

Select the new Plane1 and open a **new sketch**.

Sketch a **Circle** approx. as shown.

Add the **1.50"** diameter dimension and the **1.00"** locating dimension.

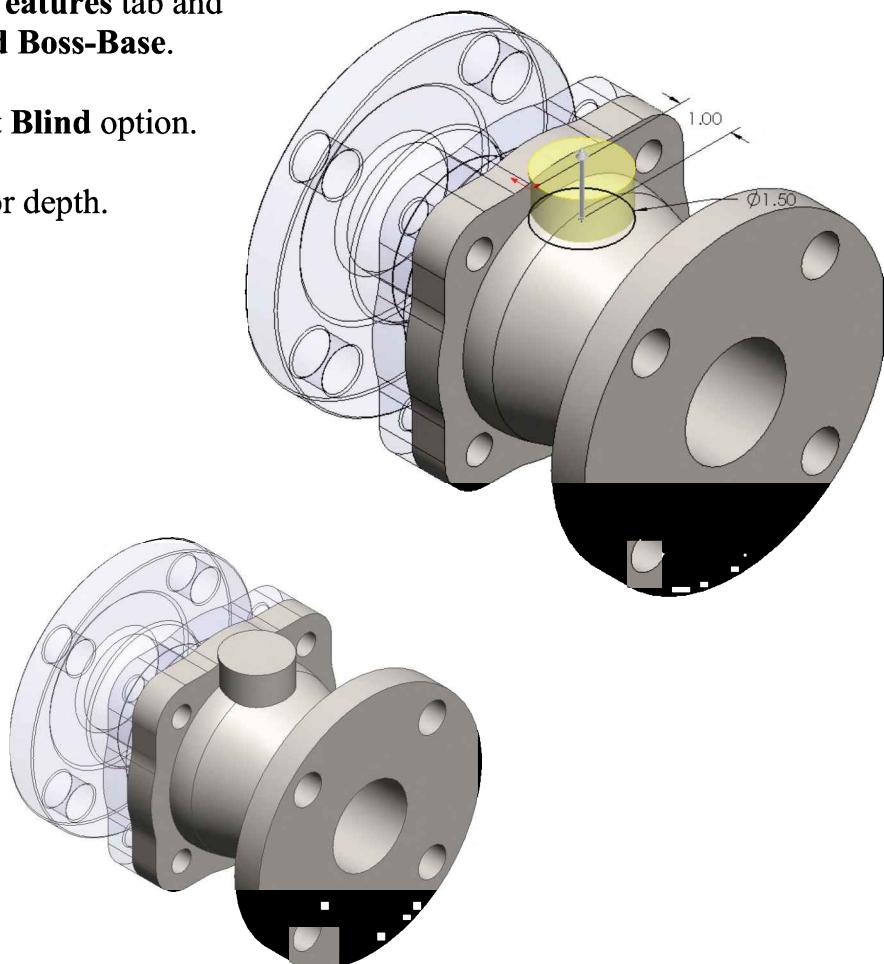
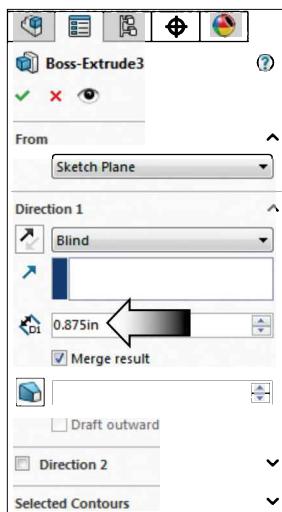
Add the **horizontal** relation as noted to fully define the sketch.



Switch to the **Features** tab and click **Extruded Boss-Base**.

Use the default **Blind** option.

Enter: **.875"** for depth.



Click **OK**.

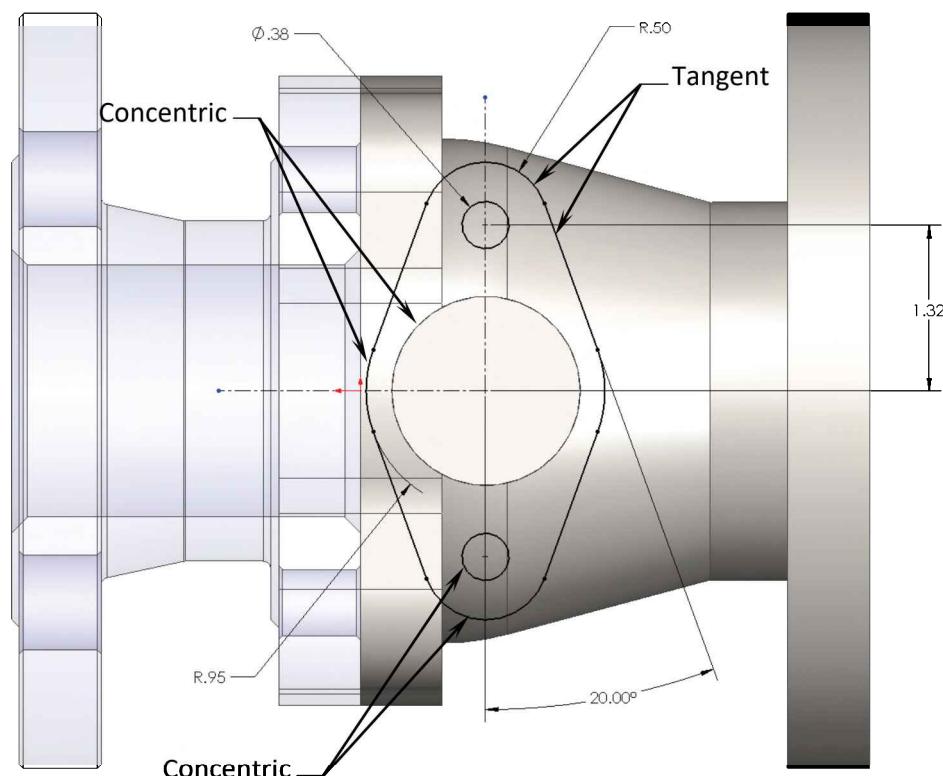
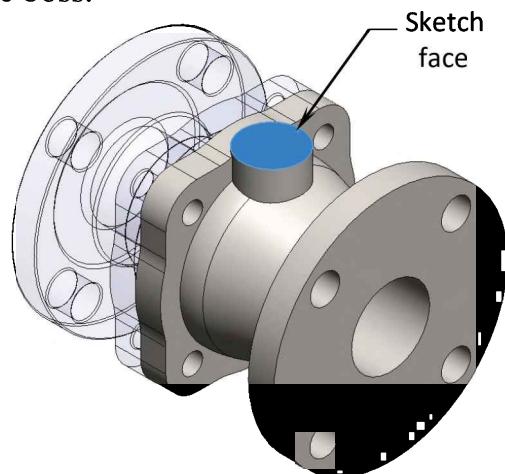
18. Adding a thermostat valve mount:

Open a **new sketch** on the upper face of the boss.

Sketch the profile of the thermostat shown below.

Use the **mirror** function to help maintain the symmetrical relationships between the sketch entities.

Add the dimensions and the relations as noted in the image below.

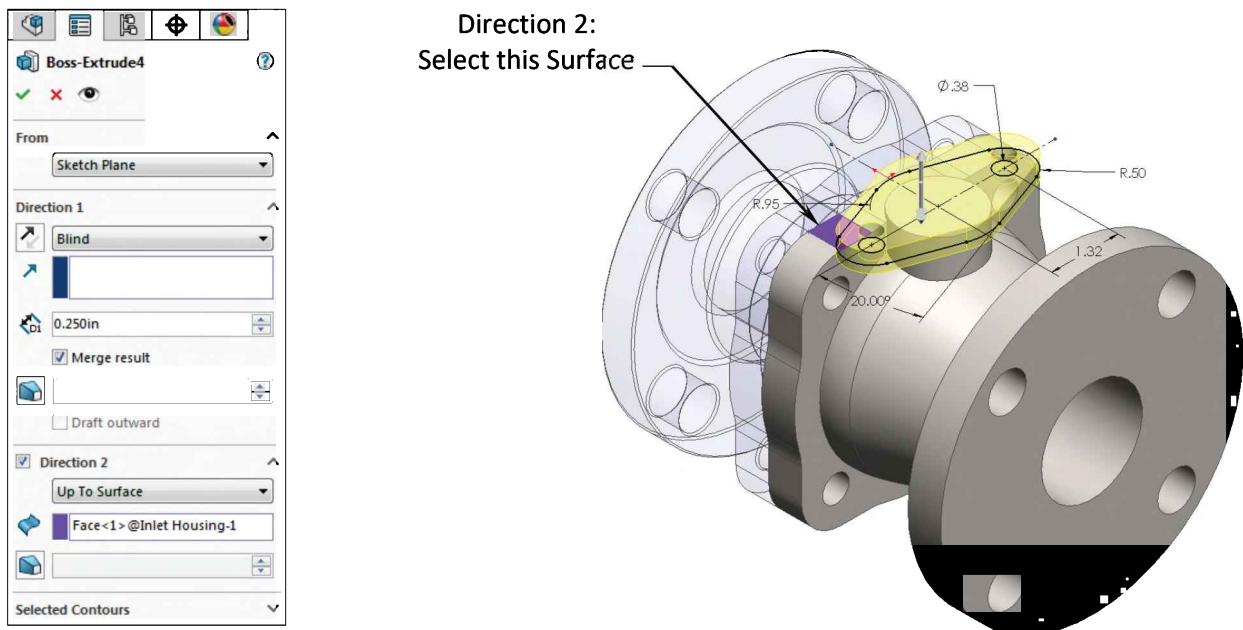


The centers of the 2 circles should be vertical with each other.

If the mirror feature was not used, then be sure to add the symmetric relations to fully define this sketch.

Switch to the **Features** tab and click **Extruded Boss-Base**.

For **Direction 1**: Use **Blind** and the depth of **.250"**.



For **Direction 2**: Use **Up-To-Surface** and select the planar face as indicated.

Click **OK**.

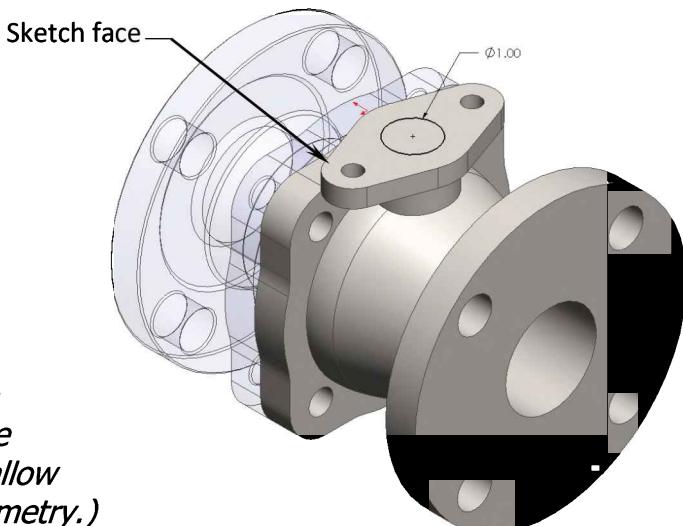
19. Adding a hole:

Open a **new sketch** on the upper face of the thermostat.

Sketch a **Ø1.00** circle and add a **concentric** relation to center it.

Make use of the "Wake-up the Entities Snap Mode".

(With the Circle command selected, hover the mouse cursor over the circular edge of the radius; the appropriate snap-entities will appear to allow snapping to the existing geometry.)

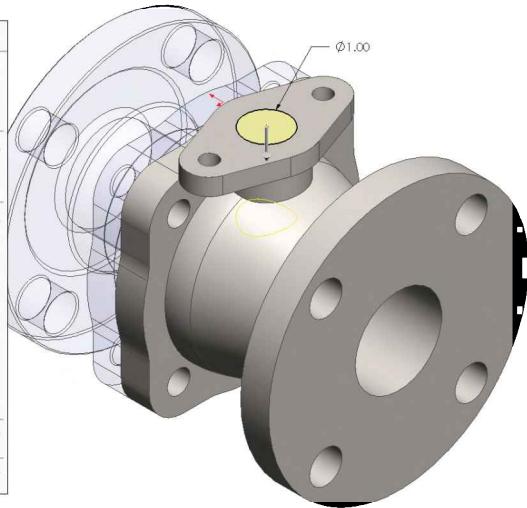
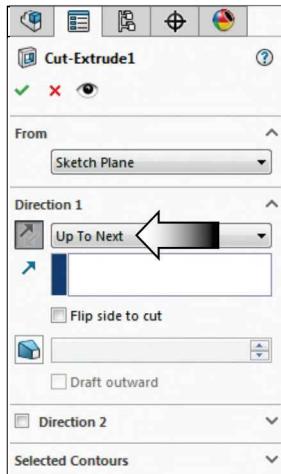


Switch to the **Features** tab and click **Extruded Cut**.

Click the **Reverse** direction button.

Select the option **Up-To-Next** from the list.

Click **OK**.



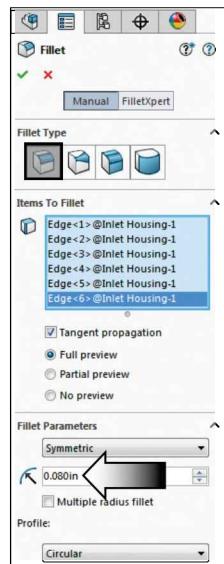
20. Adding the .080" fillets:

From the **Features** tab, click **Fillet**.

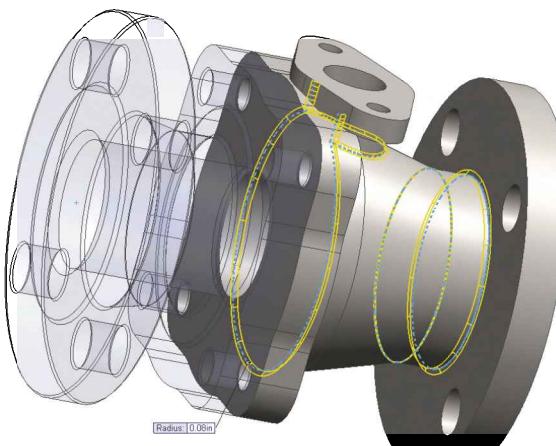
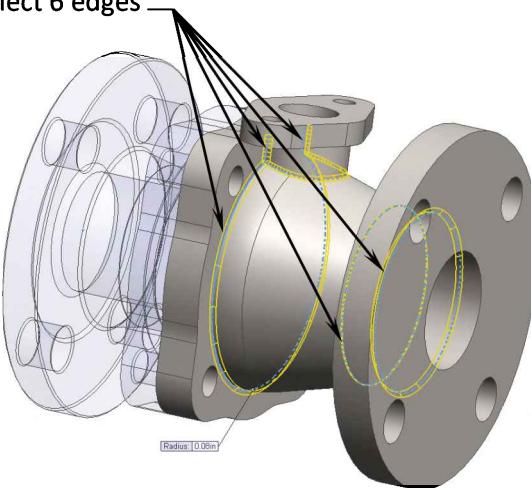
Enter **.080"** for radius.

Select the **6 edges** as indicated.

The **Tangent Propagation** checkbox should be selected.



Select 6 edges



Click **OK**.

21. Adding the .032" chamfers:

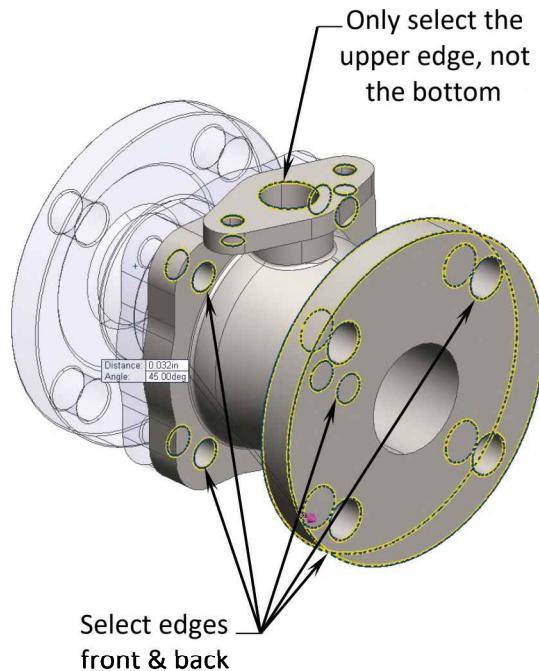
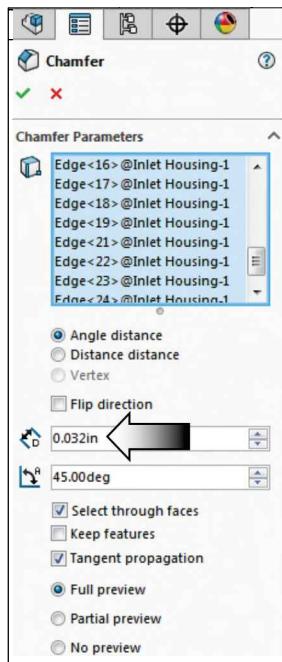
Expand the Fillet command and select Chamfer.

Enter **.032"** for depth.

Use the default **45deg** angle.

Select the **edges** of the holes as shown in the preview image (23 edges total).

The same result can be achieved by selecting the inner faces of the holes.



Click **OK**.

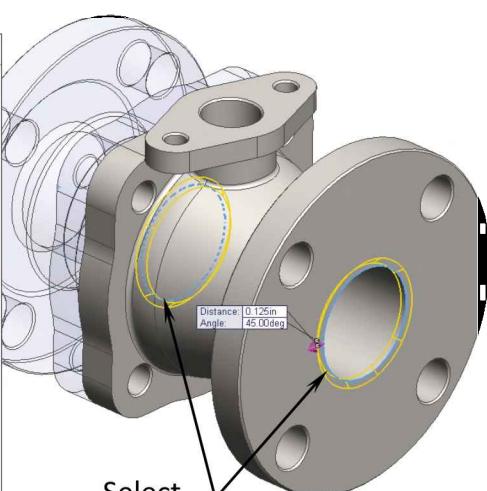
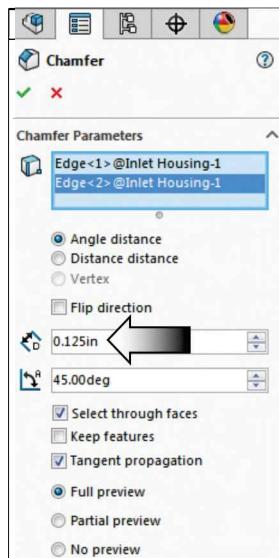
22. Adding the .125" chamfers:

Click Chamfer once again.

Enter **.125"** for depth.

Use the same **45deg** angle.

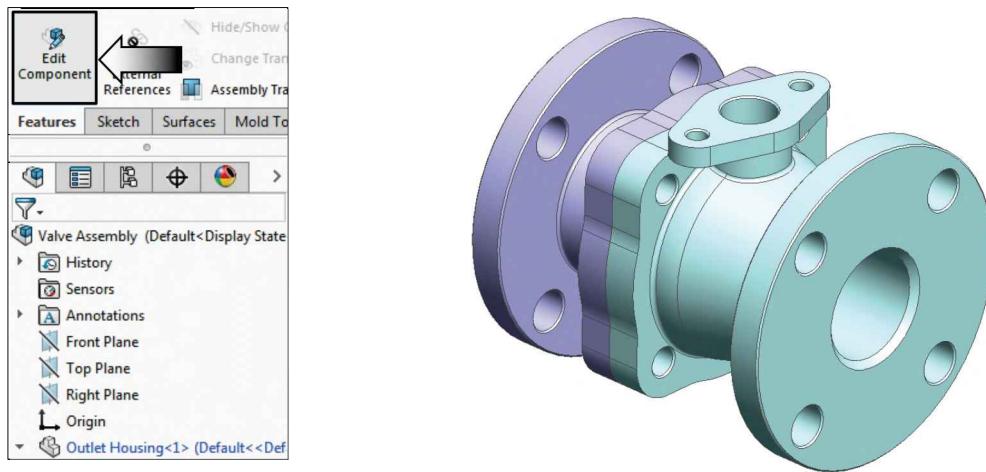
Select the **2 edges** of the center hole.
(Selecting the face of the hole would get the same result.)



Click **OK**.

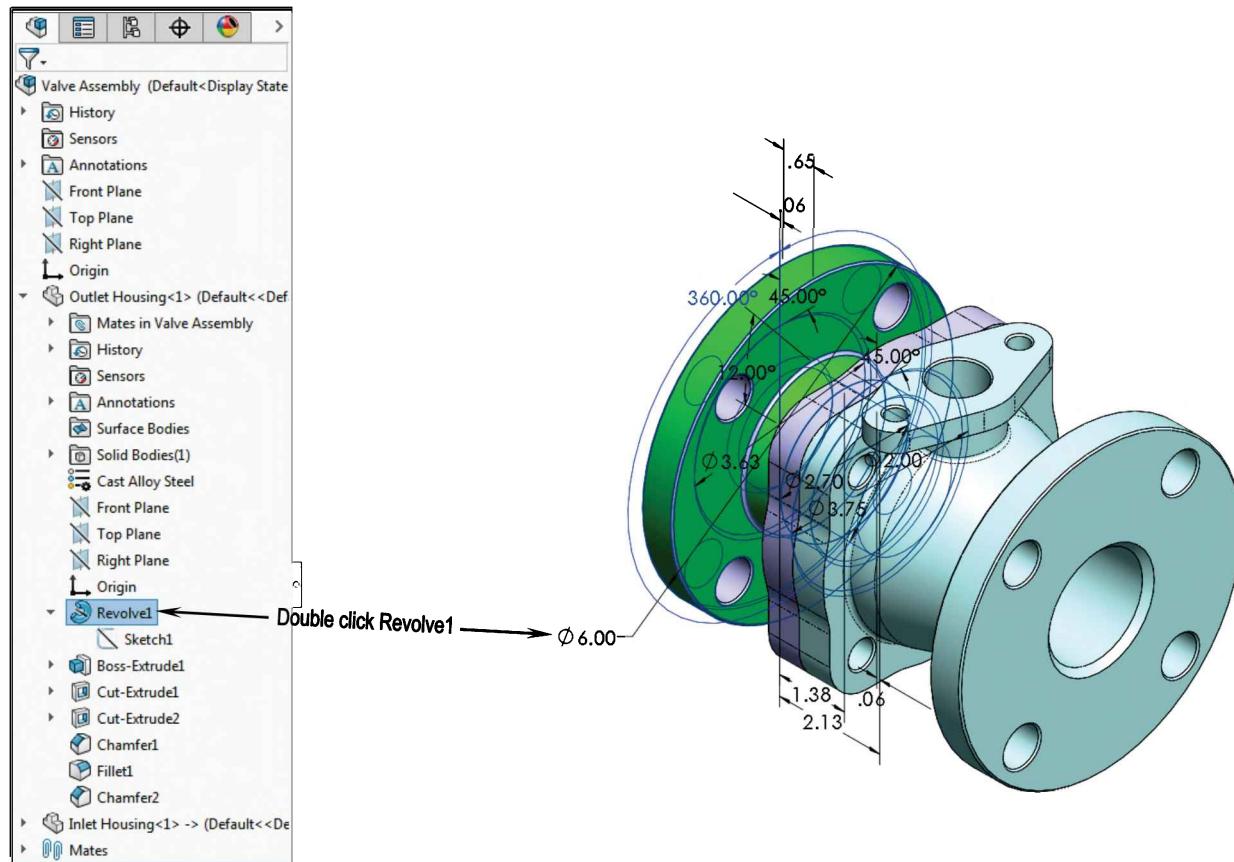
23. Exiting the Edit Component mode:

On the Assembly tab, click off the **Edit Component** command (arrow).



24. Applying dimension changes:

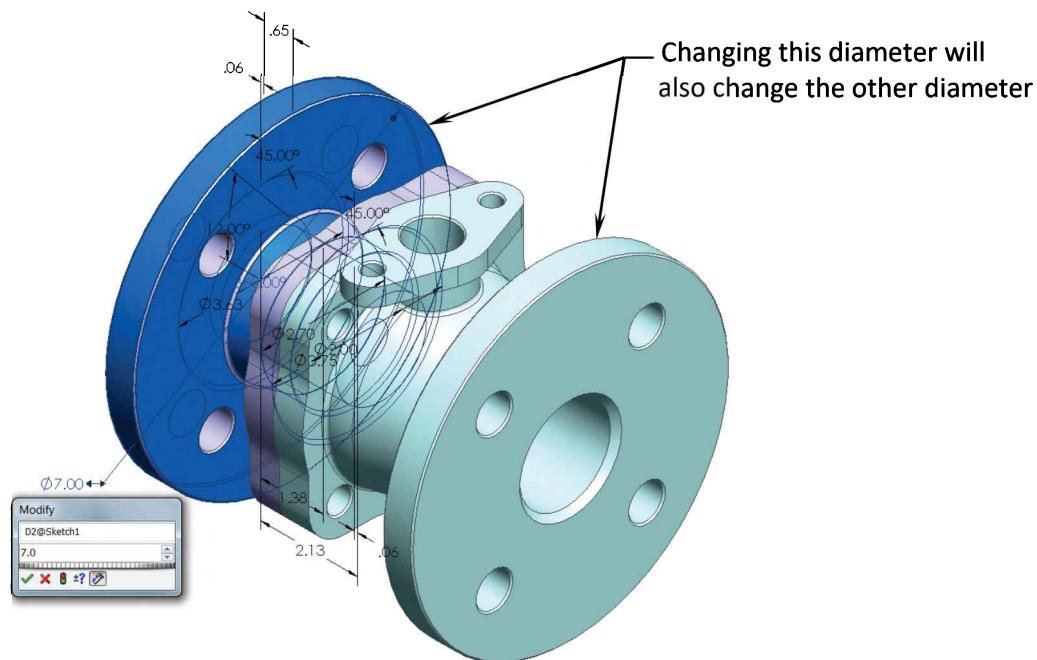
Expand the component **Outlet Housing** and double-click on the feature **Revolve1**.



Change the flange diameter from **6.00"** to **7.00"**

Click the **Rebuild** (the stop-light) to execute the change*.

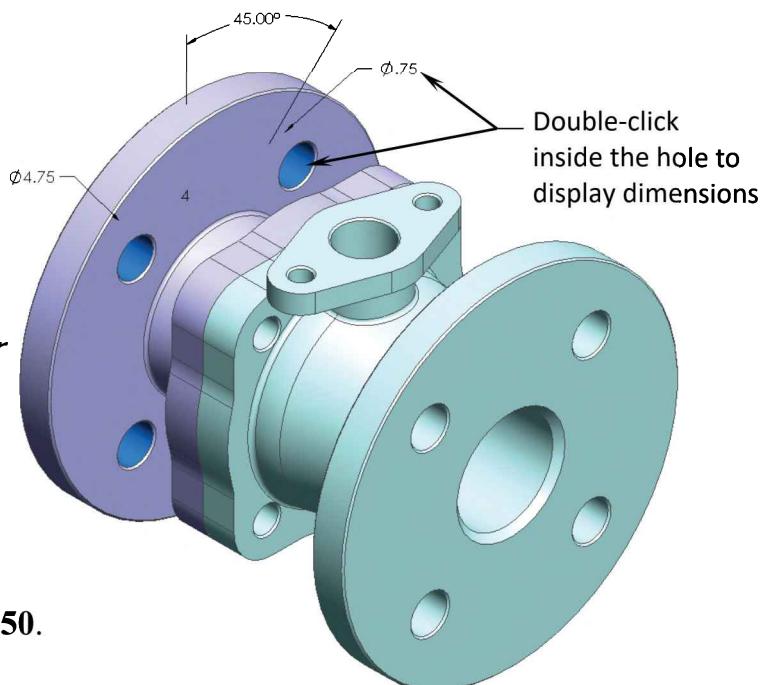
Notice the dimension change also updates the flange diameter on the right.



* Press undo to switch the dimension back to its original value, or double-click the same dimension, re-enter the previous value and click Rebuild again. Both cylindrical flanges should be **6.00"** in diameter.

Double-click the feature **Cut-Extrude1** of the same part.

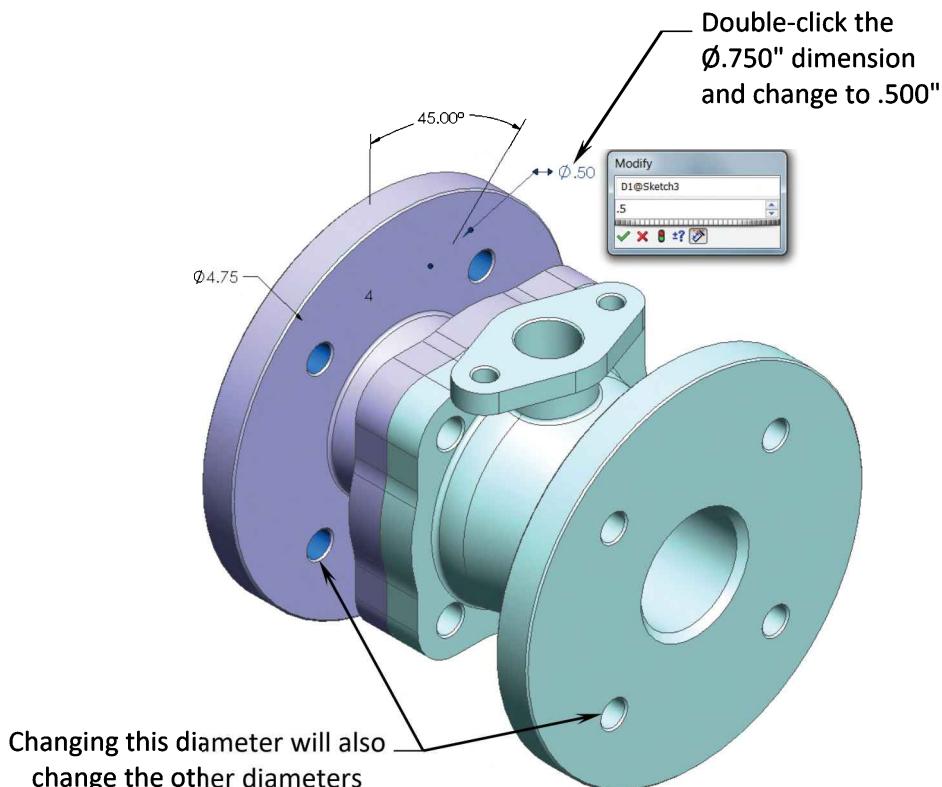
(Double-clicking the inner face of the small hole would also display its dimensions.)



Locate the dimension **Ø.750**.

Change the hole diameter from .750" to .500".

Click the **Rebuild** (the stop-light) to execute the change*.



Notice the dimension change also updates the hole diameters on the right.

- * Press undo to switch the dimension back to its original value, or double-click the same dimension, re-enter the previous value and click Rebuild again.

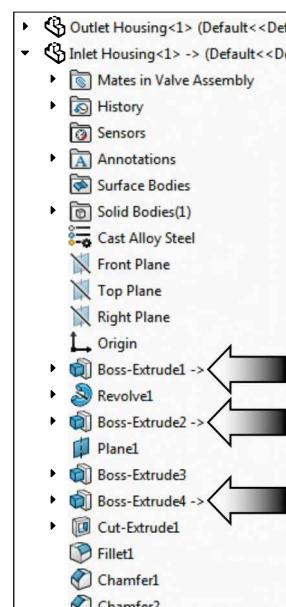
25. Viewing the External Reference Symbols:

Expand the 2nd part, the **Inlet Housing**.

Some of the features have the External (->) reference symbols next to their names. These references were created automatically when we converted the entities of Part1 to create the new sketch for Part2.

They are called On-Edge relations.

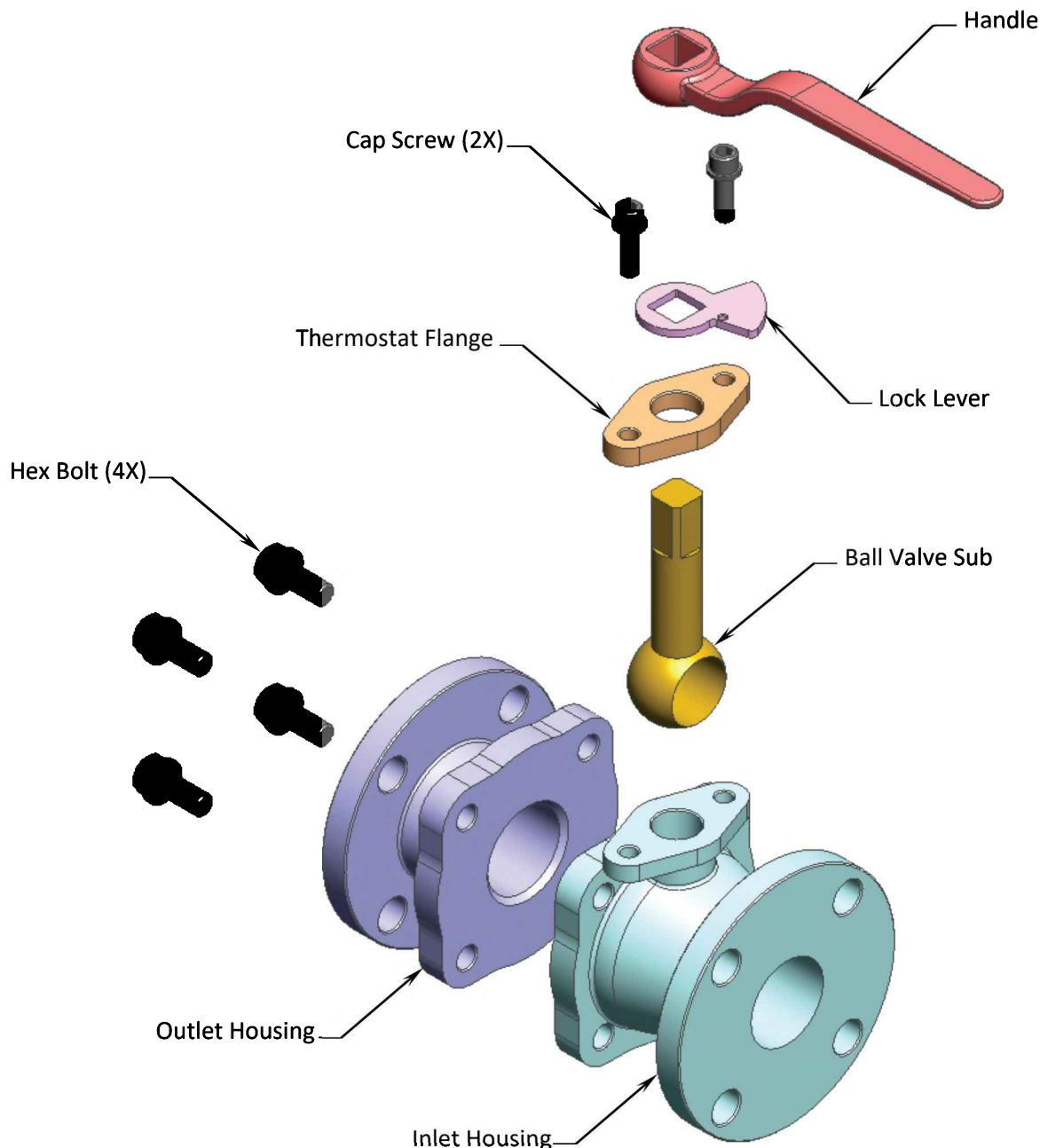
An external reference is also created when we add a dimension or a relation between Part1 and Part2.



26. Inserting other components:

Due to the length of this lesson, we are going to insert and assemble the rest of the components that belong to this assembly.

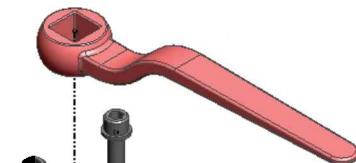
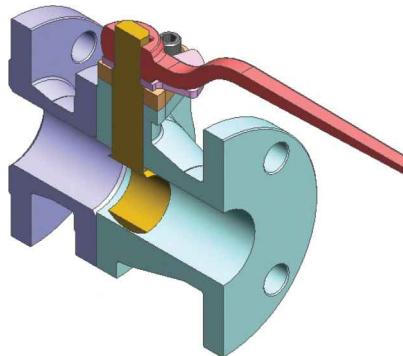
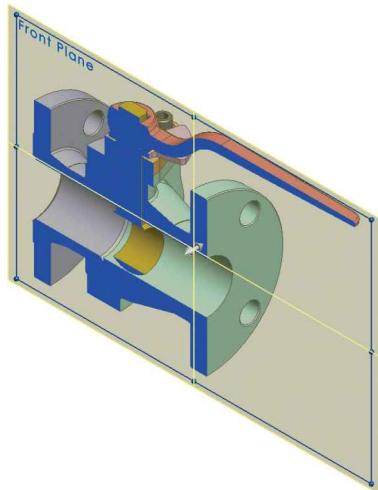
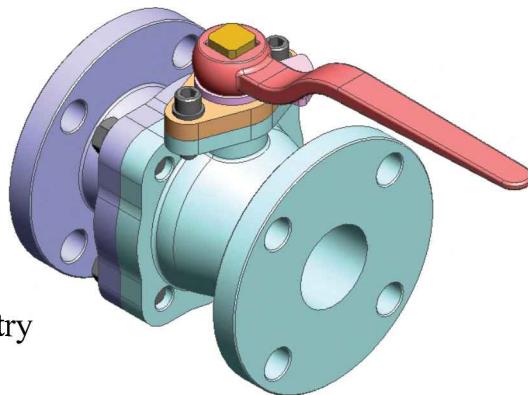
Click **Insert Components** and browse to the Training Files folder and select the components as labeled below.



Create the mates that are appropriate for each component. The non-moving components will get 3 mates, and the moving components will need only two.

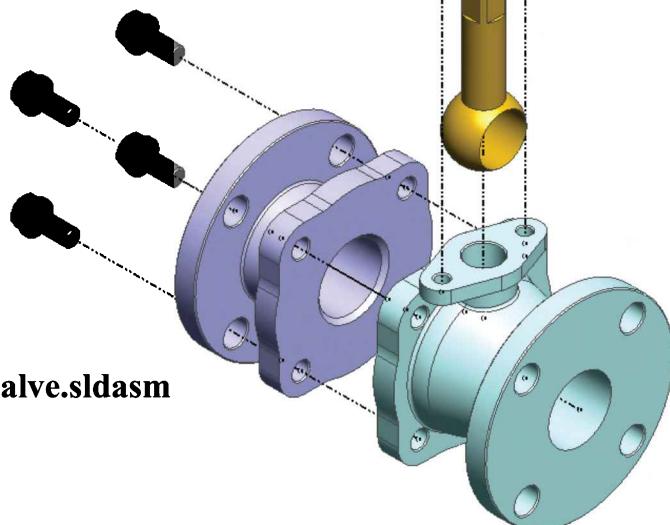
27. Optional:

- a/. Create a section view to verify how the components were mated.
- b/. To center the 2 components, it is best to use the Width mate option.
- c/. Change / correct any mates or geometry that would cause interferences.



Add the Explode-Line Sketch as shown.

When adding the explode lines, pay attention to the direction arrows, flip or reverse them before completing each line.



28. Saving your work:

Click File / Save As.

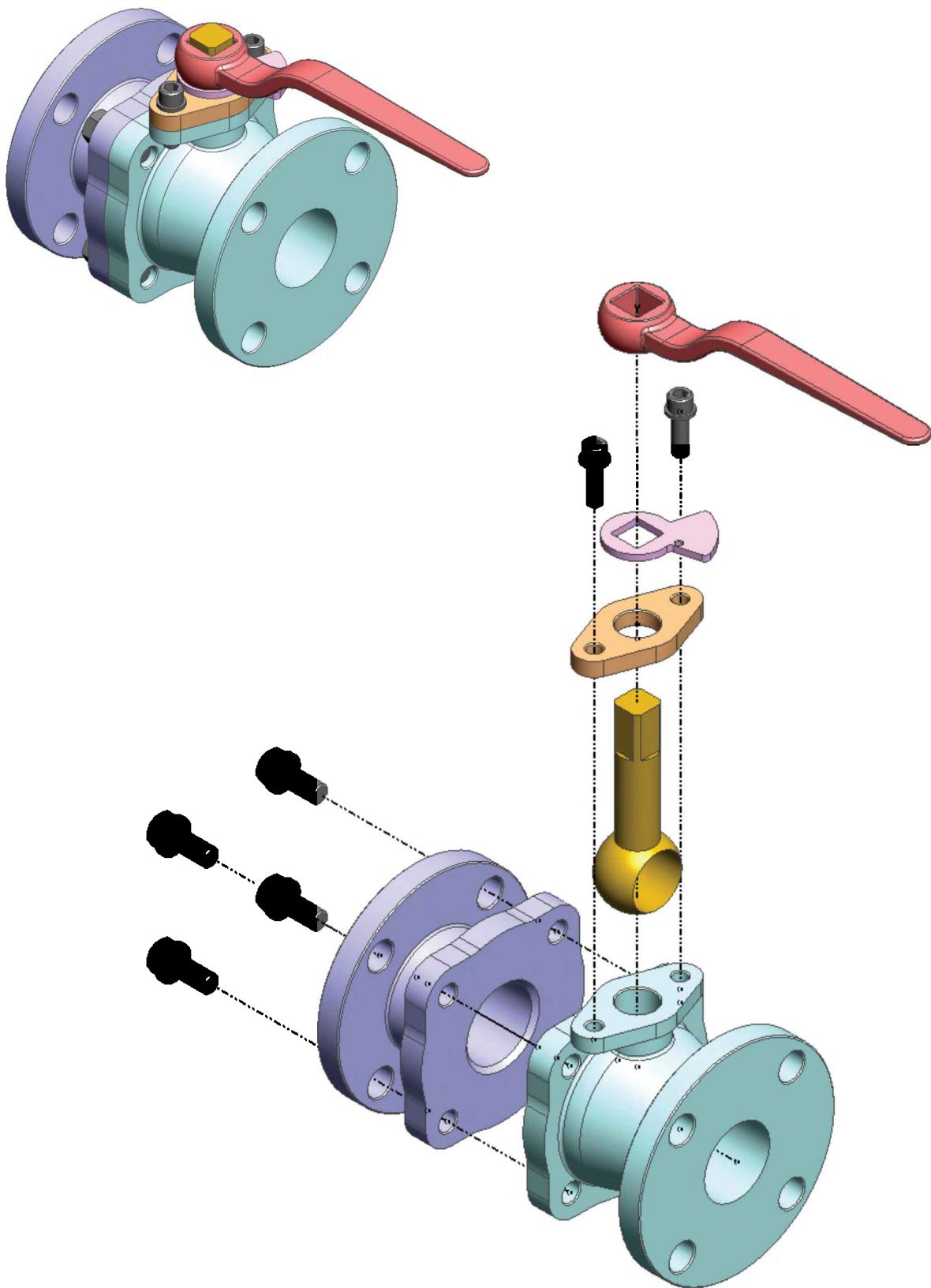
Enter **Water Control Valve.sldasm** for the name of the file.

Press Save.

Questions for Review

1. In Top Down mode, when a plane is selected to sketch the new part, SOLIDWORKS will create an Inplace mate to reference the new part.
 - a. True
 - b. False
2. After the plane is selected, SOLIDWORKS will also activate the Edit Component command and the Sketch mode at the same time.
 - a. True
 - b. False
3. The option Auto-Rotate Normal to the Sketch Plane is not available to set as the default.
 - a. True
 - b. False
4. The Convert Entities command can only be used when the Sketch is not active.
 - a. True
 - b. False
5. The Virtual part is saved / embedded inside an assembly document.
 - a. True
 - b. False
6. When editing a part in Top Down mode, both of the active part and non-active parts can be filleted at the same time.
 - a. True
 - b. False
7. The Edit Component command should be left active prior to inserting a new part.
 - a. True
 - b. False
8. When a dimension is changed, any reference geometry of other parts should also change.
 - a. True
 - b. False

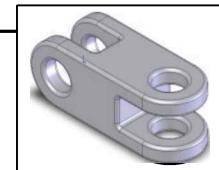
1. TRUE	2. TRUE	3. FALSE	4. FALSE	5. TRUE	6. FALSE	7. FALSE	8. TRUE
---------	---------	----------	----------	---------	----------	----------	---------



CHAPTER 22

External References & Repair Errors

External References & Repair Errors



An *external reference* is created when one component is dependent on another component for its solution. If the original document is changed, the dependent document will also change.

In an assembly, you can create an *in-context* feature on one component that references the geometry of another component. This in-context feature has an external reference to the other component. If you change the geometry on the referenced component, the associated in-context feature changes accordingly.

The External Symbols:

->	External Reference	?	Out Of Context
(+)	Over Defined	*	Reference Locked
X	Reference Broken		

-> External Reference:

The part itself or some of its entities are depending on the geometry of other parts for their solutions.

? Out Of Context:

The part or its features are not solved, not up-to-date or disconnected from its assembly.

(+) Over Defined:

The Dimensions or Relations of the sketch are conflicting, redundant dimensions or wrong relations were used.

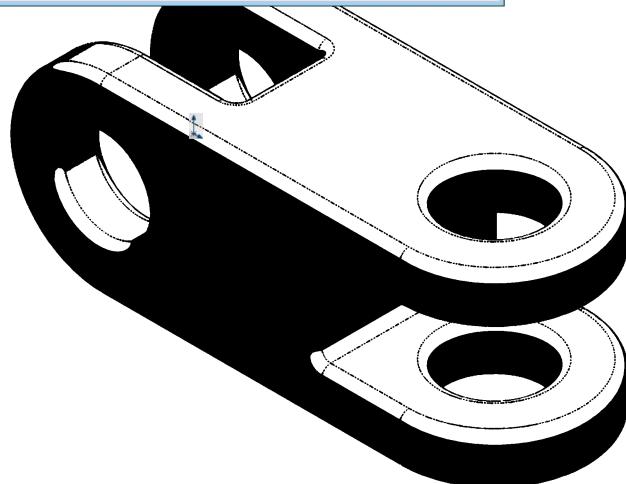
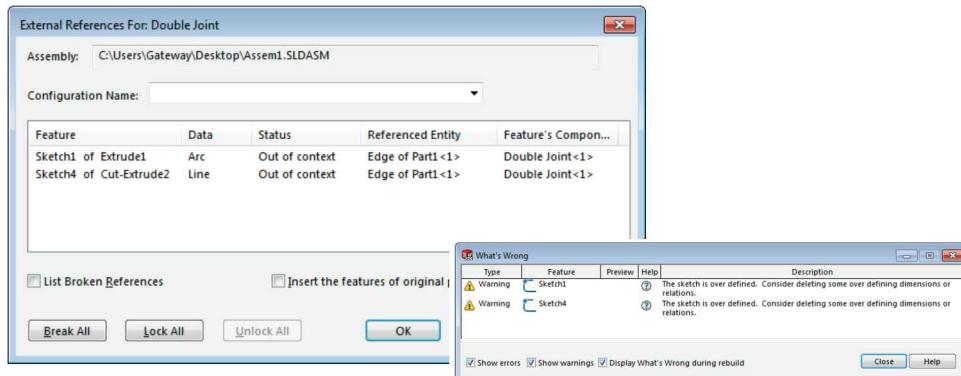
* Reference Locked:

Lock the external references on a part, the existing references no longer update and the part will not accept any new references from that point.

X Reference Broken:

The references between the part and the others are broken. Changes done to the Part will not affect the others.

External References & Repair Errors



View Orientation Hot Keys:

Ctrl + 1 = Front View
 Ctrl + 2 = Back View
 Ctrl + 3 = Left View
 Ctrl + 4 = Right View
 Ctrl + 5 = Top View
 Ctrl + 6 = Bottom View
 Ctrl + 7 = Isometric View
 Ctrl + 8 = Normal To Selection

Dimensioning Standards: ANSI
Units: INCHES – 3 Decimals

External Reference Symbols:



External Reference



Out of Context



Over Defined



External Reference Locked



External Reference Broken



Display/Delete Relations

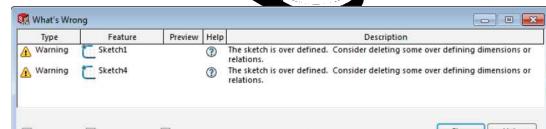
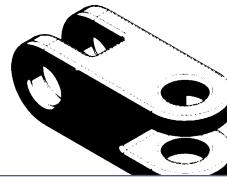
Understanding & Removing External References

1. Opening a part document:

Go to: Training Files folder

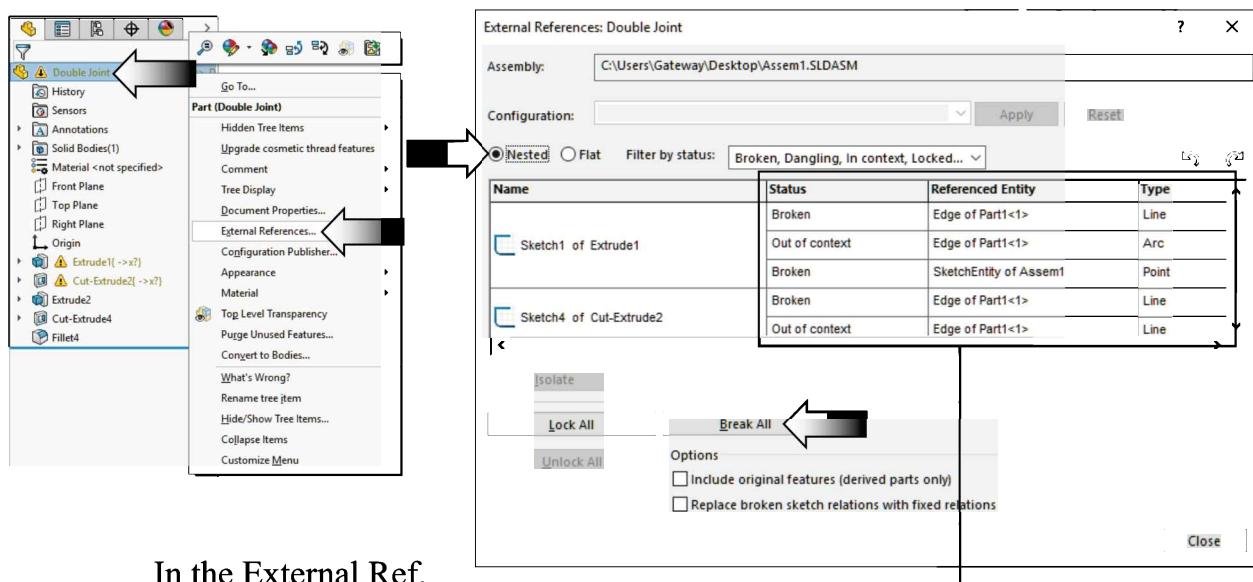
Open a part named: **Double Joint**.

The **What's Wrong** dialog appears displaying the current errors. Close it. We need to look at the external references first.



2. Listing the External References:

Right-click on the part's name and select **External References**.



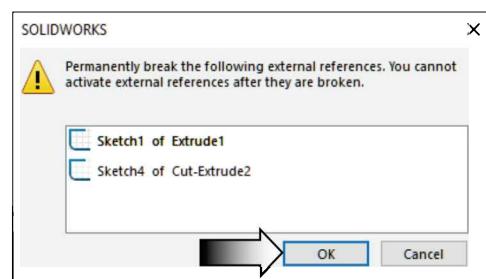
In the External Ref. dialog box, enable the option: **Nested** (arrow).

Several Lines and Arcs were converted from other parts in an assembly; deleting these references will cause those entities to become Dangling

3. Removing External References:

Click **Break All** to remove the external references.

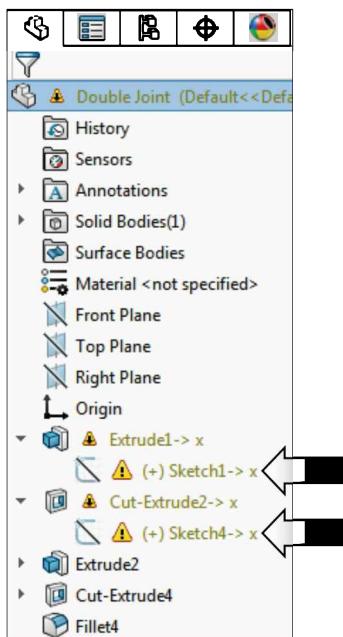
Click **Close** and **OK**.
(Click Continue Ignore Error to close the message.)



4. Understanding the External Symbols:

- > External Reference
- ? Out Of Context
- (+) Over Defined
- * Reference Locked
- X Reference Broken

(From the FeatureManager tree, expand the first two features to see their sketches.)



Error Colors

- Olive Green:** Dangling (Missing references/detached).
- Red:** Over Defined (Wrong relations/dimension).
- Yellow:** Not Solved (A relation is conflicting with a dimension).

a. External Reference ->:

The model itself or some of its entities are depending on the geometry of other parts for their solutions.

b. Out Of Context ?:

The model or its features are not solved, not up-to-date, or disconnected from its assembly.

c. Over Defined (+):

The Dimensions or Relations of the sketch are conflicting, redundant dimensions or wrong relations were used.

d. Reference Locked *:

Lock the external references on a model; the existing references no longer update - and - the model will accept any new references from that point.

e. Reference Broken X:

The references between the part and the others are broken. Changes done to the part will not affect the others.

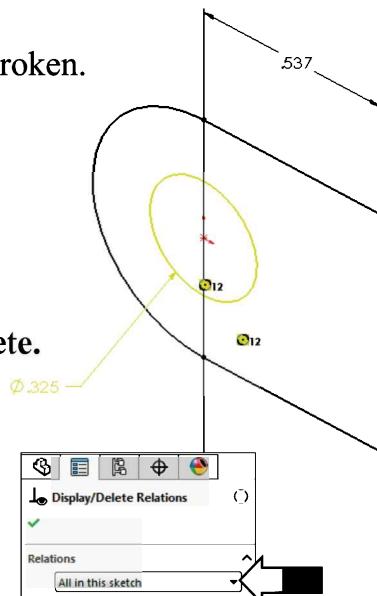
5. Viewing the existing Relations:

Right-click on Sketch1 and select **Edit-Sketch**.

Click or select **Tools, Relations, Display-Delete**.

The Ø.325 is shown in **Olive Green** color; this indicates either an entity is missing or the sketch is over dimensioned.

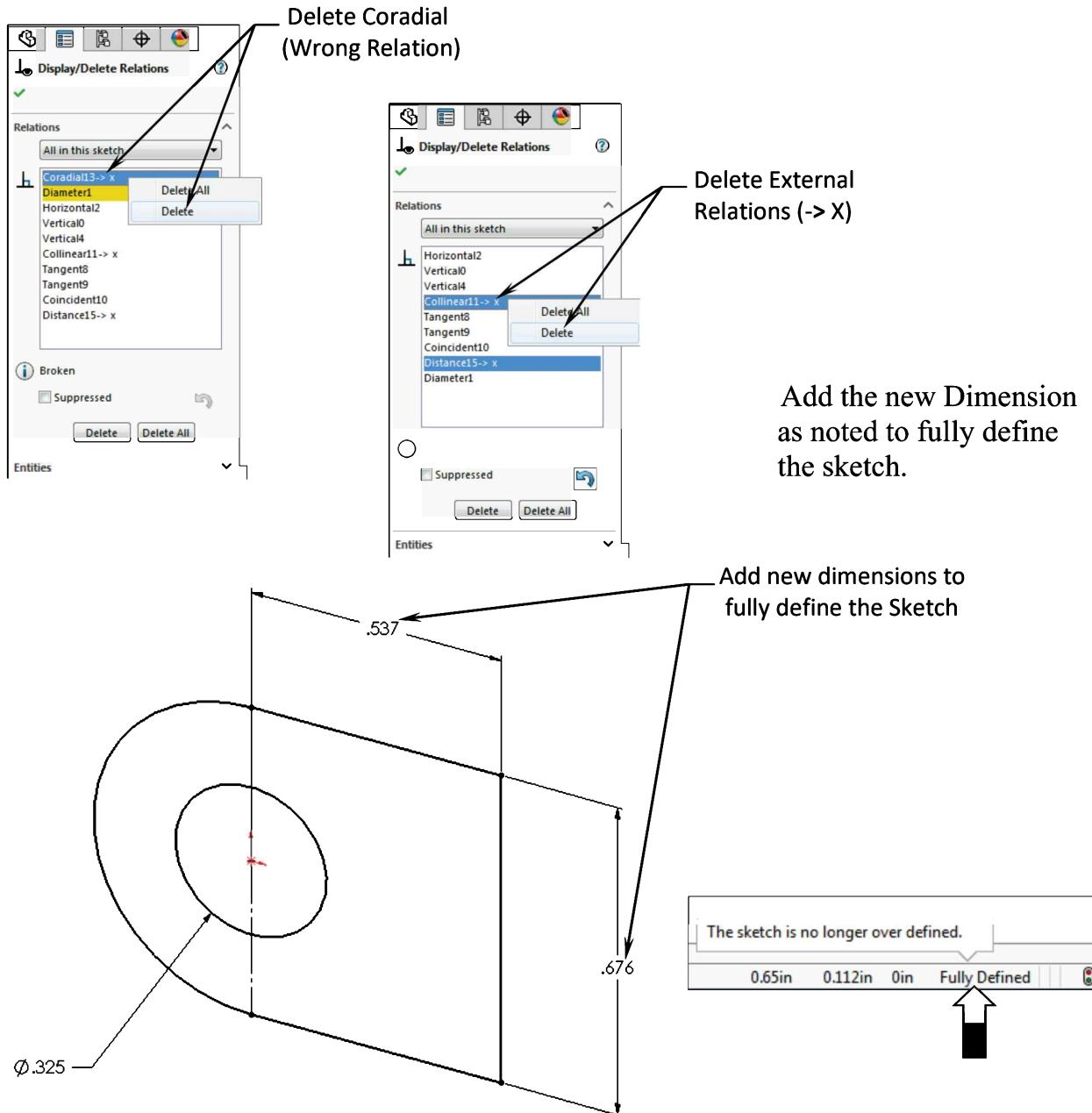
Change the Relations Filter to **All In This Sketch**.



6. Repairing the 1st sketch:

Click **Display / Delete Relations** and delete the Coradial relation (the Circle is Coradial with an entity that no longer exists).

Delete all relations that have the External Relations (->X) next to their names.



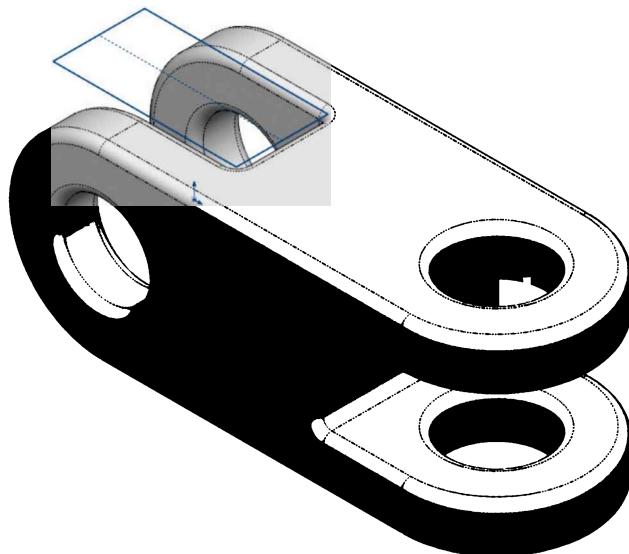
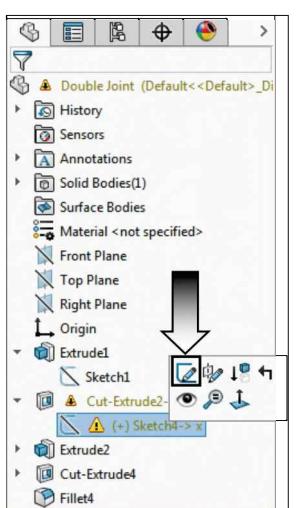
A message on the lower right indicates the sketch is no longer over defined.

Click **OK** and Exit the sketch. There are still some other errors in the part.

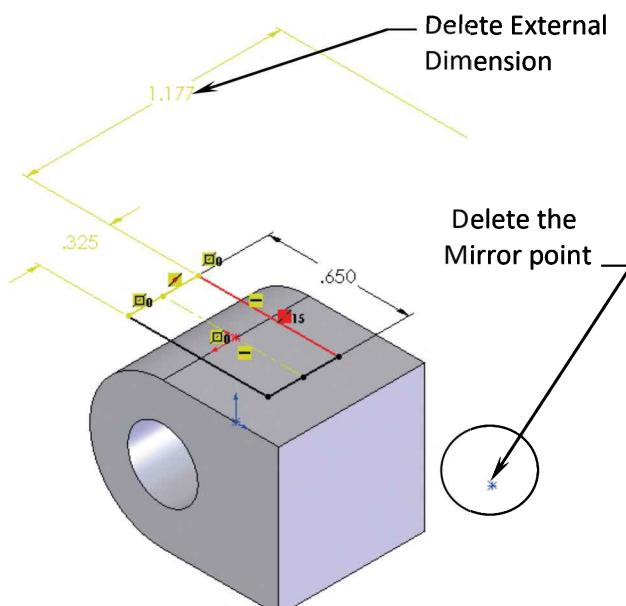
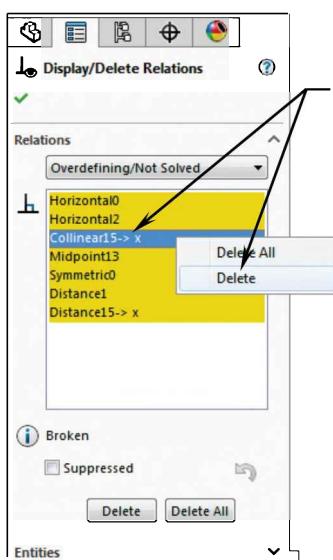
7. Repairing the 2nd sketch:

Right-click on the 2nd sketch and select **Edit-Sketch** .

Select the **Display / Delete Relations** command  once again.



Delete the External dimensions and relations that were created in context of other parts.



Delete also the two “Mirror-Points” as noted.

Exit the sketch when the message:

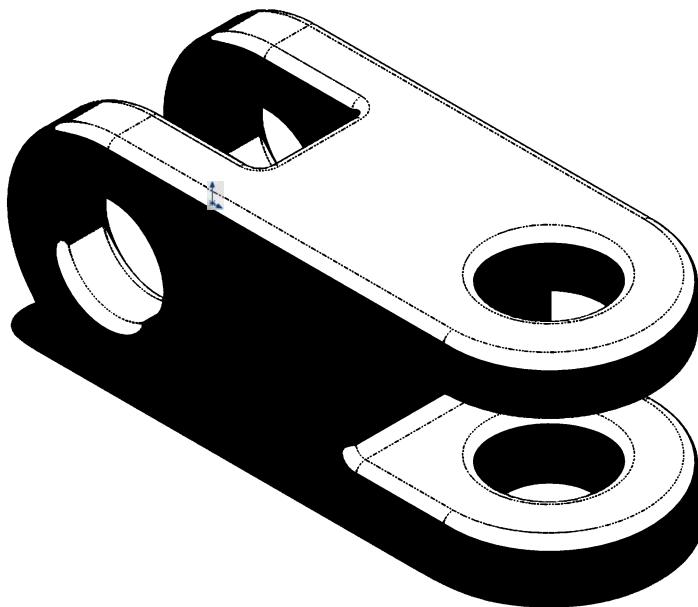
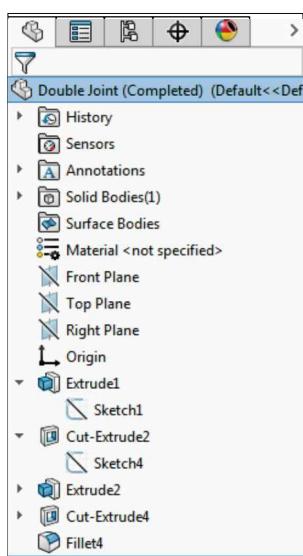
Fully Defined status is displayed on the lower right.



8. Rebuilding the model:

Press **Rebuild**  to re-generate the model.

Verify that the part has no rebuild errors and there should not be any external reference symbols in the FeatureManager tree.



9. Saving your work:

Select **File / Save As**.

Enter **Breaking External References** for the file name.

Click **Save**.

Questions for Review

1. The symbol **->** next to a file name means:
 - a. Dangling dimension
 - b. External reference
 - c. Not solved
2. The symbol **?** next to a file name means:
 - a. The part cannot be found
 - b. Wrong mates
 - c. Wrong relations
 - d. Out of context
3. The symbol **X** next to a file or a feature name means:
 - a. The part or feature is wrong
 - b. The part or feature is deleted
 - c. The external references are broken
4. The symbol ***** next to a file name means:
 - a. External references are locked
 - b. Select all references
 - c. Deselect all references
5. The symbol ***X** next to a feature name means:
 - a. The feature is fully defined
 - b. The feature is over defined
 - c. The feature is under defined
 - d. None of the above
6. The Olive-Green color in a sketch means:
 - a. The sketch entity is selected
 - b. The sketch entity is being copied
 - c. The sketch has dangling entities, relations, or dimensions
7. The dangling dimensions can be “re-attached” simply by dragging its handle point to a sketch line or a model edge.
 - a. True
 - b. False

1. B
2. D
3. C
4. A
5. D
6. C
7. TRUE

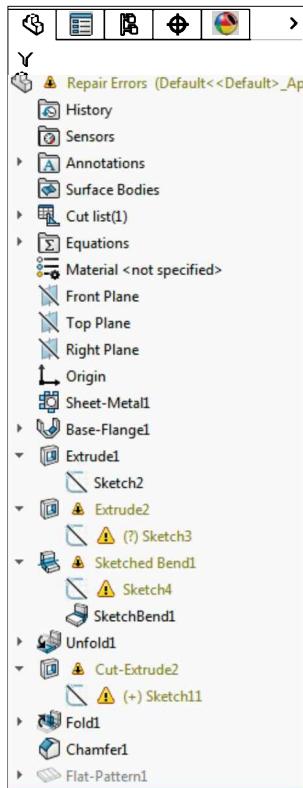
Understanding and Repairing Part Errors

When an error occurs SOLIDWORKS will try and solve it based on the settings below:

To pre-set the rebuild action:

- A. Click **Options**  or **Tools / Options / General**.
- B. Select **Stop**, **Continue**, or **Prompt** for **When rebuild error occurs**, and then click **OK**.

With **Stop** or **Prompt**, the rebuild action stops for each error so you can fix feature failures one at a time.

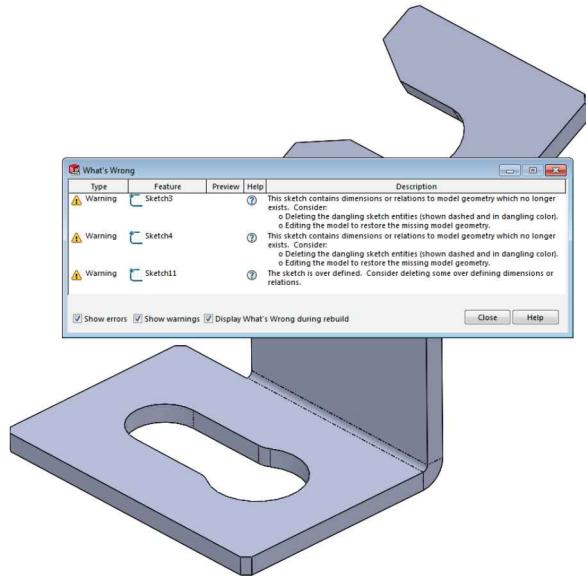
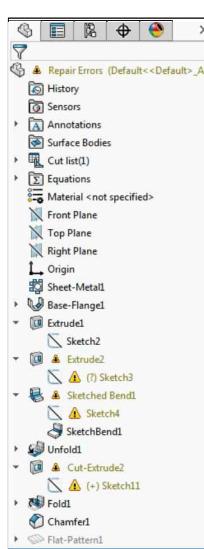


-  Indicates an error with the model. This icon appears on the document name at the top of the FeatureManager design tree, and on the feature that contains the error. The text of the part or feature is displayed in **red** color.
-  Indicates an error with a feature. This icon appears on the feature name in the FeatureManager design tree. The text of the feature is displayed in **red** color.
-  Indicates a warning underneath the node indicated. This icon appears on the document name at the top of the FeatureManager design tree and on the parent feature in the FeatureManager design tree whose child feature issued the error. The text of the feature is displayed in **olive green** color.
-  Indicates a warning with a **feature** or **sketch**. This icon appears on the specific feature in the FeatureManager design tree that issued the warning. The text of the feature or sketch is displayed in **olive green** color.

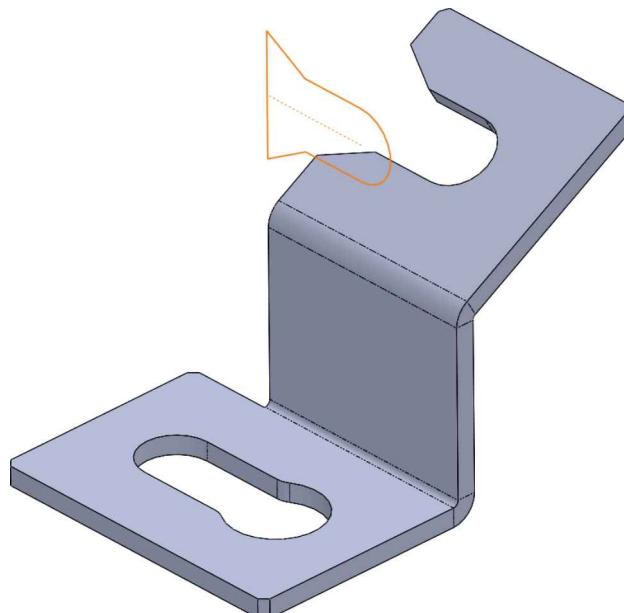
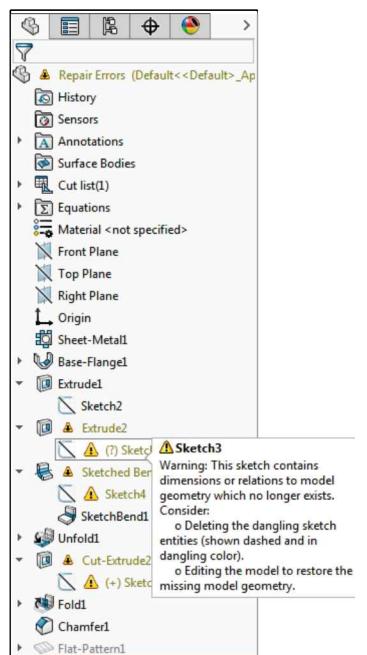
1. Opening a part document:

In the Training File folder, browse to the **Repair Errors** folder, and open a part document named: **Repair Errors**.

When opening a document that contains errors, the What's Wrong dialog box will appear and display where the errors are located and suggest some solutions in solving them.



Expand each feature on the FeatureManager tree and hover the pointer over the **Sketch3** (Control+T is the hot key to expand the FeatureManager tree).



A description about the error or the warning is displayed in the tooltip. This is the same as right clicking on the error and selecting the **What's Wrong** option.

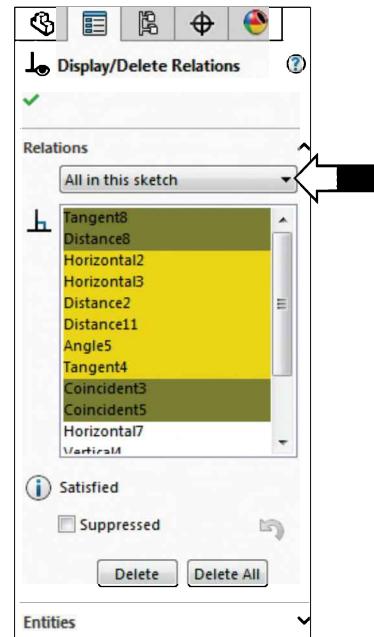
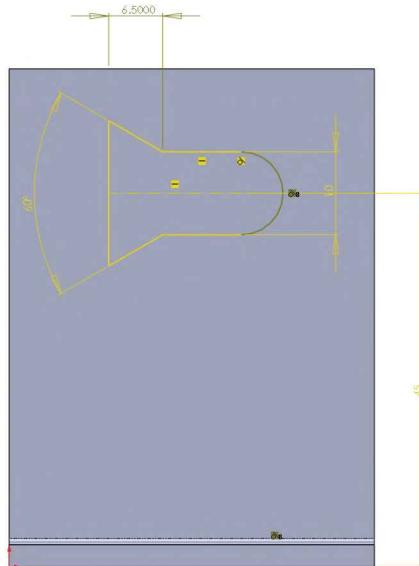
2. Repairing the 1st error:

Select Sketch3 and click **Edit Sketch**.

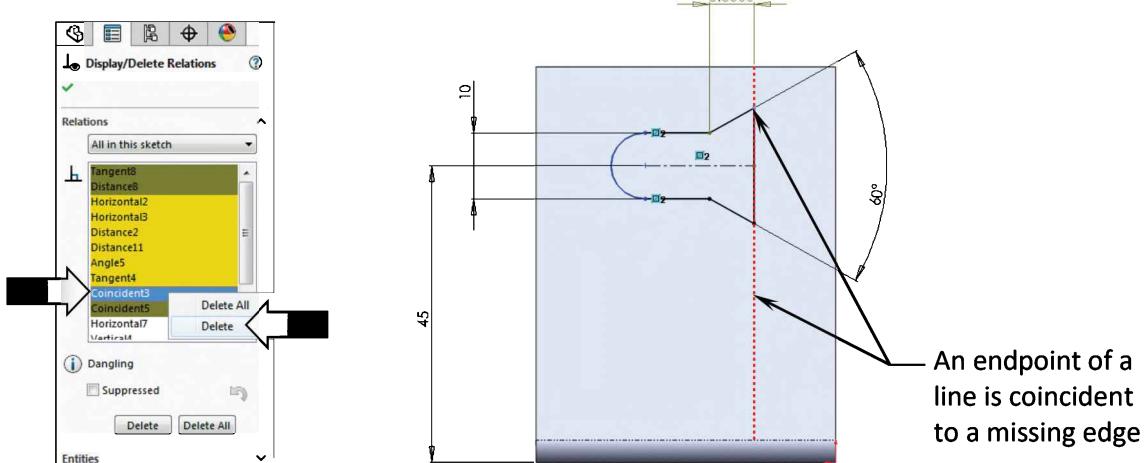
Click the **Display/Delete Relation** command .

Change the Filter to **All In This Sketch** (arrow).

There are some dangling coincident relations in this sketch; they are displayed with the **olive green** color.



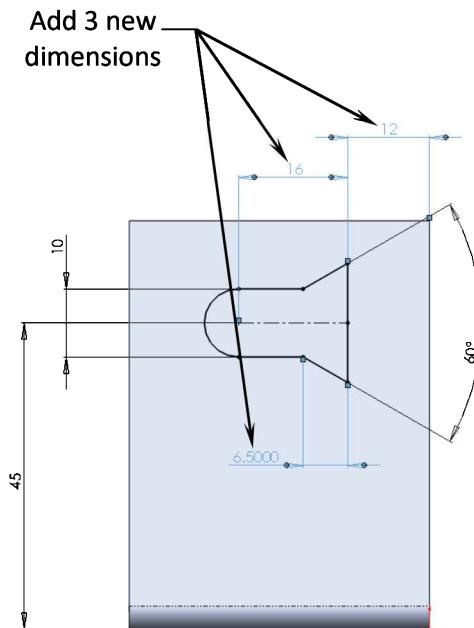
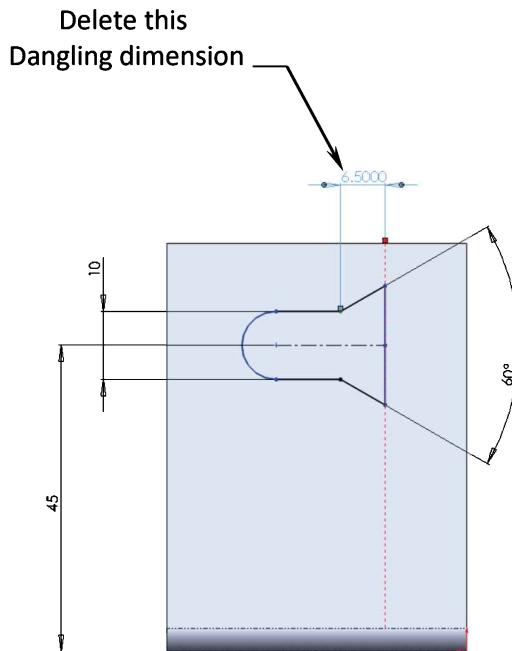
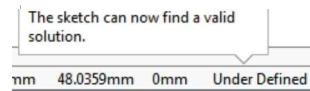
Select **Coincident3**. One endpoint of a line is coincident to a missing edge.
Delete this **Coincident3**.



Delete also **Coincident5**. It is missing the same edge as the previous relation.

The dimension **6.500** is also dangling. It was measured to a missing entity; delete dimension 6.500.

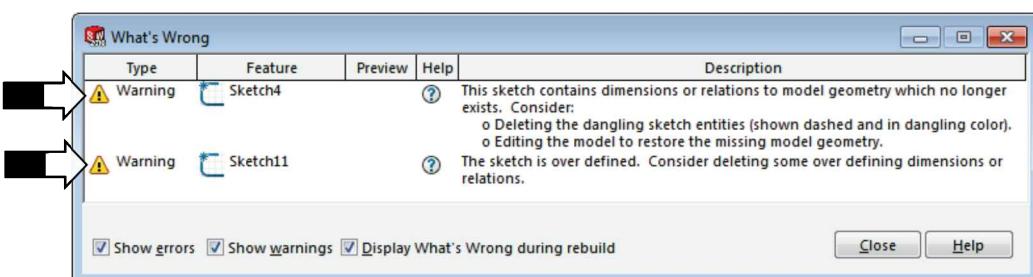
A message on the bottom right of the screen appears.
It indicates: **The sketch can now find a valid solution.**



Add 3 new dimensions shown above to fully define the sketch.
Exit the sketch when completed.

After exiting the sketch, SOLIDWORKS continues to report other errors still remaining in the part. The **Sketch4** and **Sketch11** still need to be repaired.

The What's Wrong dialog box pops up displaying the same description about the selected error. Enable the **Show Warnings** checkbox (arrow) to see the warnings after each rebuild.



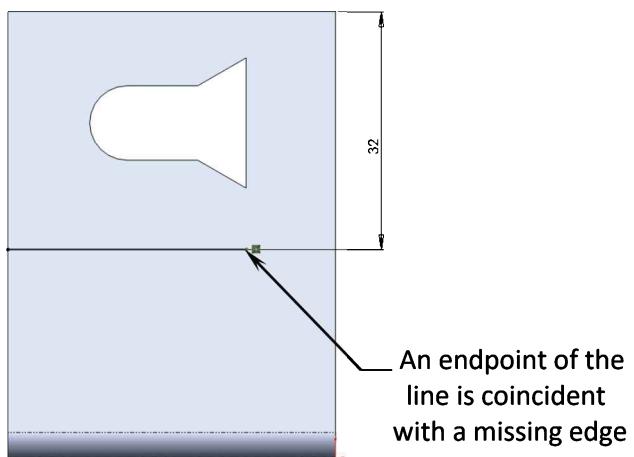
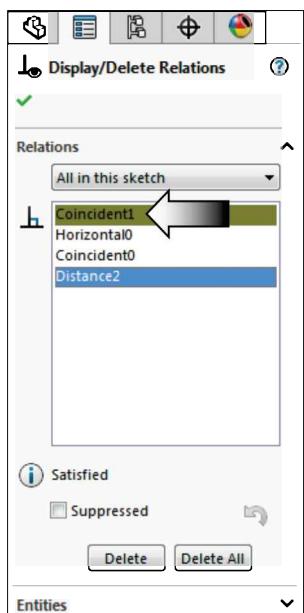
Close the What's Wrong dialog box.

3. Repairing the 2nd error:

Click **Sketch4** (under the Sketched Bend1 feature) and select: **Edit Sketch**.

Select the **Display / Delete Relations** command  once again.

Coincident1 is dangling and has the **Olive Green** color. Delete it.

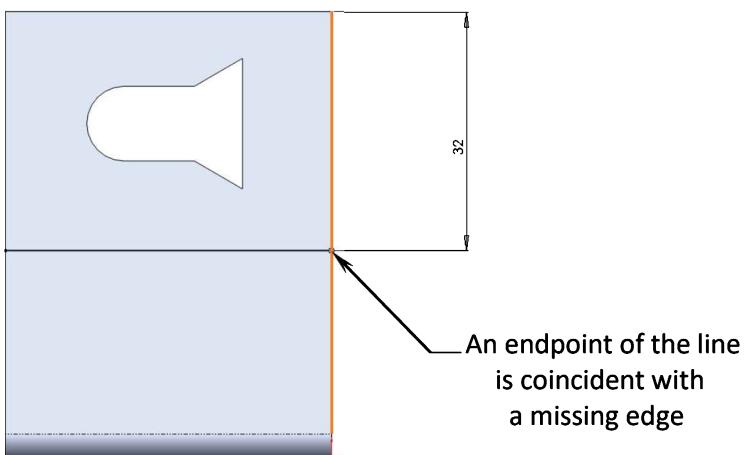


An endpoint of the line is coincident with a missing edge

The endpoint of the line changes to **Blue** color.
This indicates the sketch is now under defined.

Drag/drop the endpoint of the line until it touches the vertical right edge of the part.

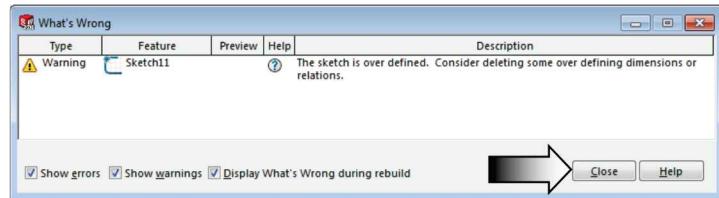
A coincident relation is added automatically.



An endpoint of the line is coincident with a missing edge

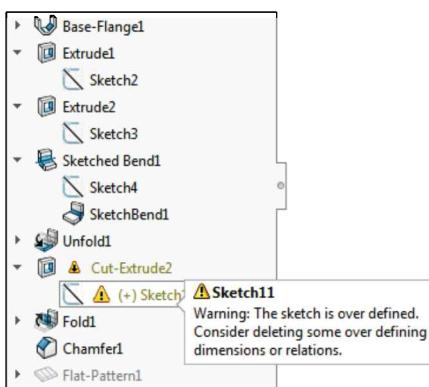
Exit the sketch or press **Control + B** (Rebuild).

SOLIDWORKS continues to report the last error in the model. Click the **Close** button and continue with repairing the last error.



4. Repairing the 3rd error:

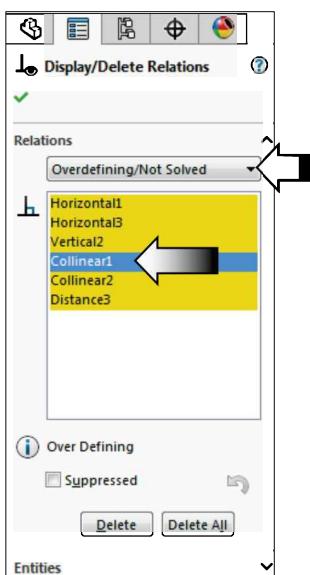
The **Sketch11** has a plus sign next to its name. This indicates that the sketch is **Over Defined**.



Edit the **Sketch11** and change to the **Top** orientation (Control + 5).

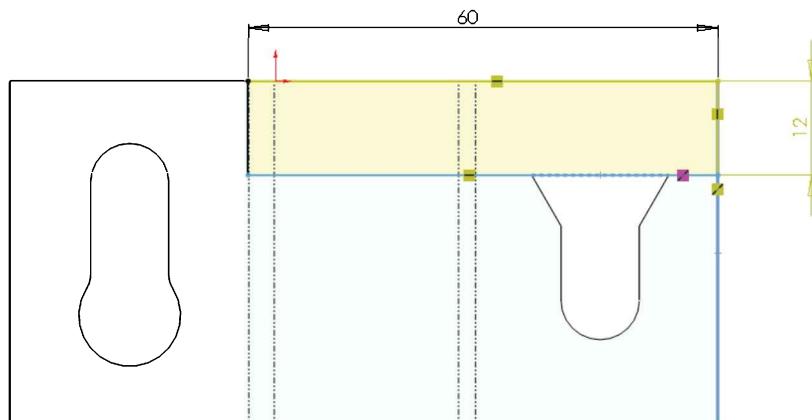
Click the **Display / Delete Relations** button

Set the display relations filter to **Over Defining / Not Solved**.



Select Collinear1 from the list. This relation shows a **Magenta** color next to its Collinear symbol.

Delete the **Collinear1** from the list.



This last step should bring the sketch back to its Fully Defined status.

Exit the sketch or press **Control + B**.

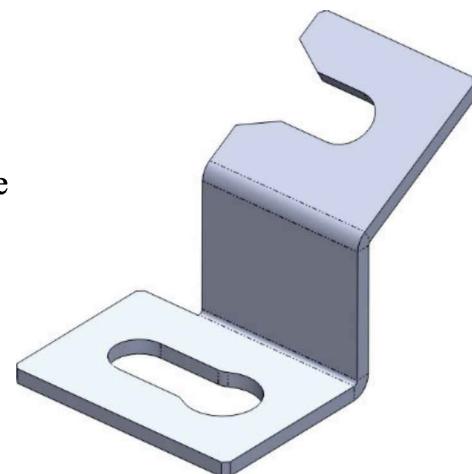
5. Saving your work:

Click File / Save As.

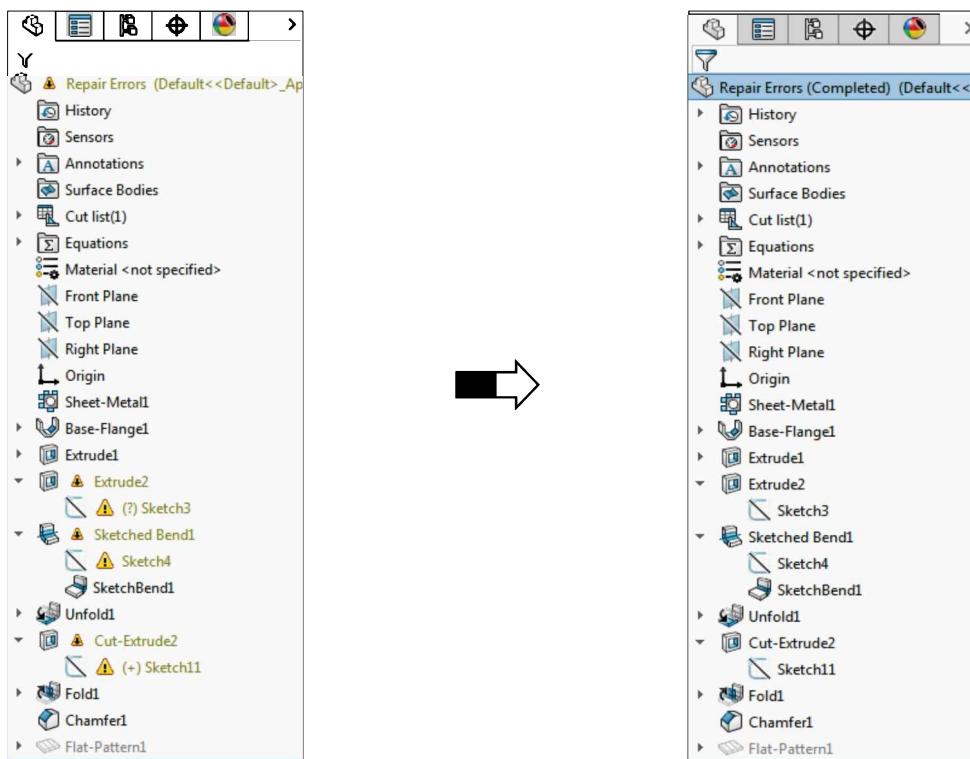
Enter **Repair Errors (Completed)** for the name of the file.

Click Save.

All errors have been repaired. The FeatureManager tree is now free of errors.



Press **Control+T** to collapse the FeatureManager tree.



Close all documents.

Repair Errors & External References

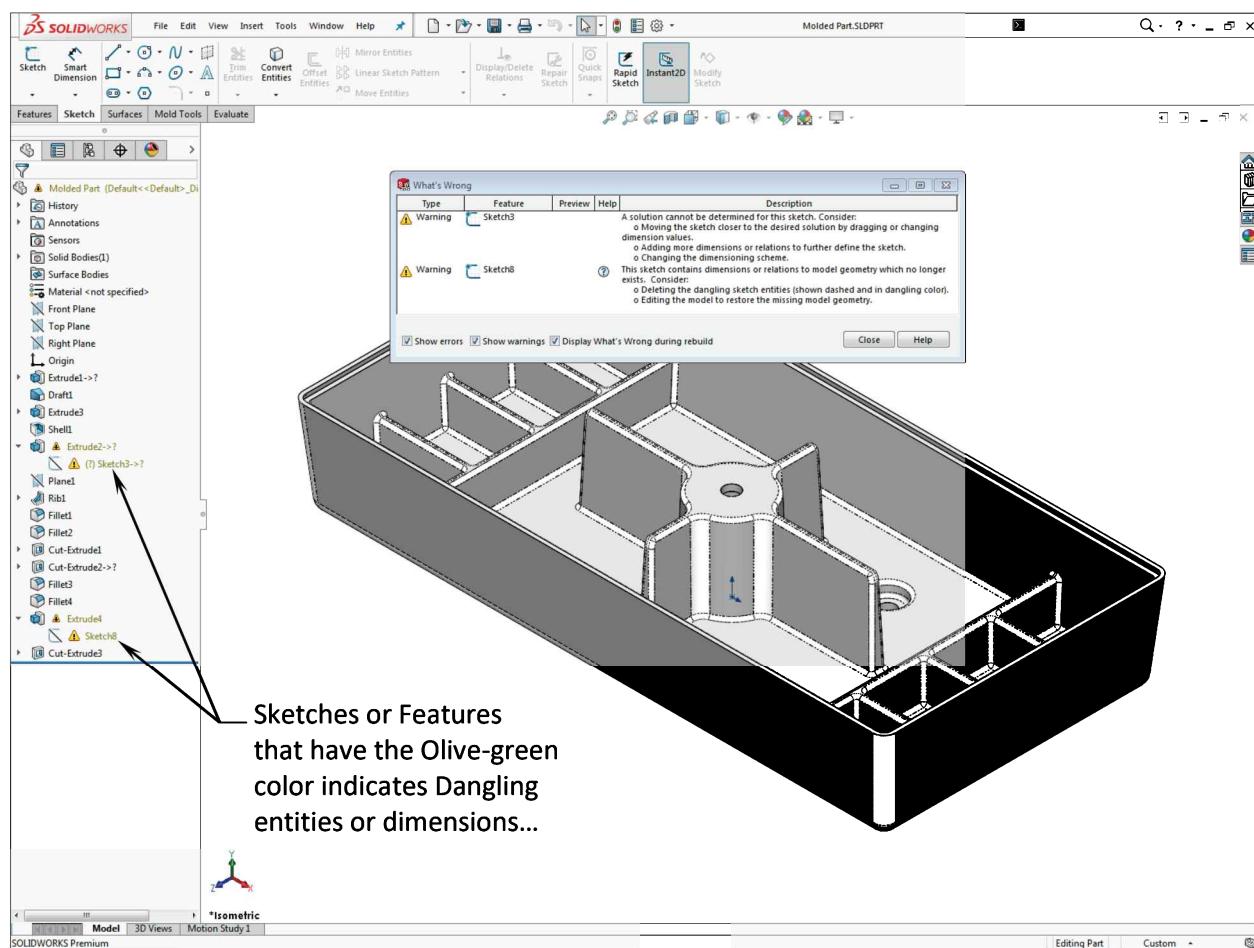
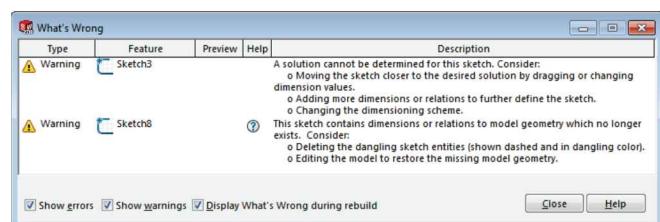
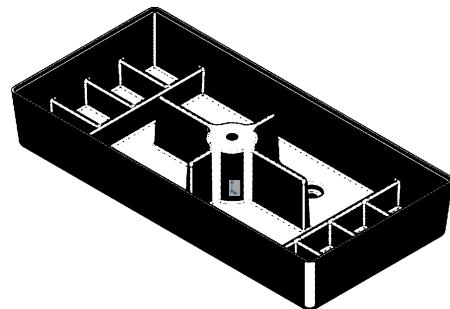
1. Opening a part document:

Browse to the Training Files folder.

Open a document named:
Molded Part.

The **What's Wrong** dialog appears displaying the current errors in the model.

Click **Close**. We are going to take a look at breaking the external references first.

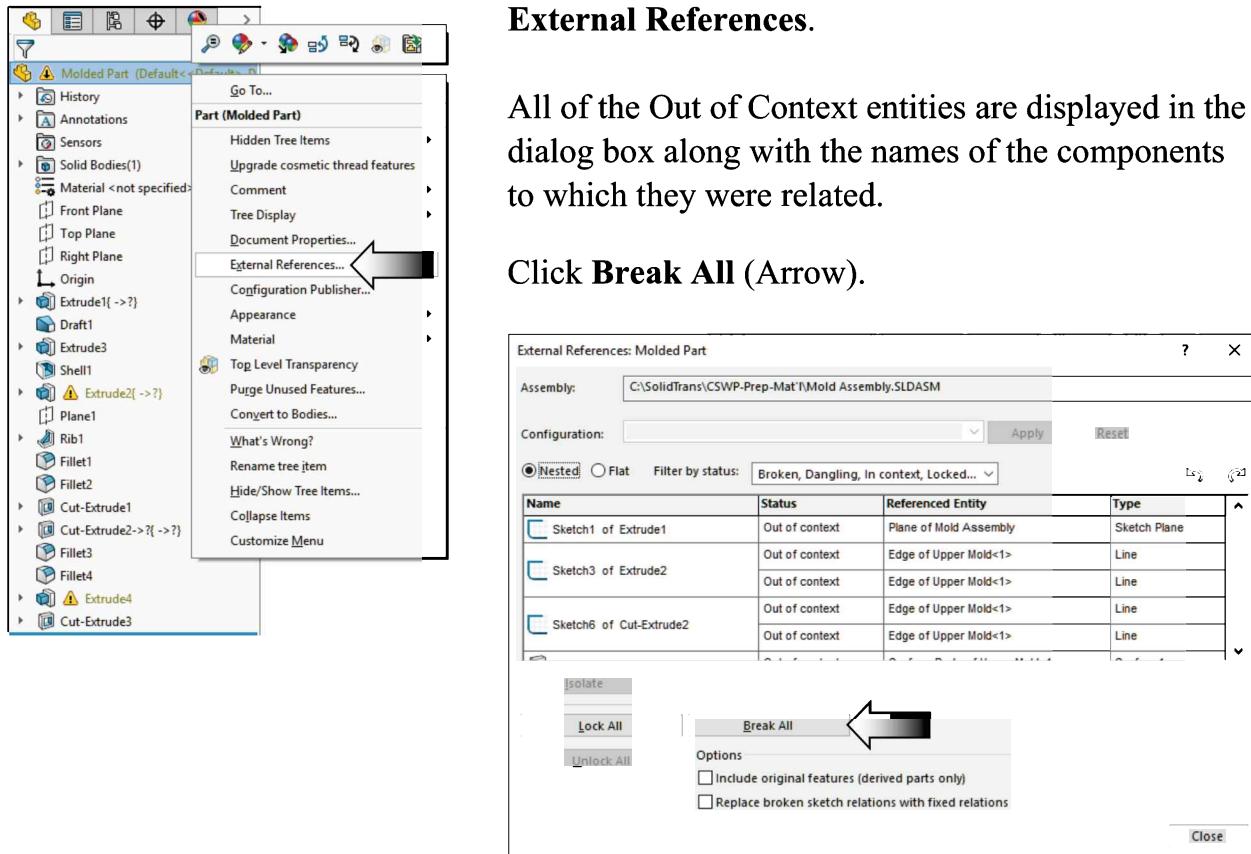


2. Breaking all External References:

Right-click the name of the part and select:
External References.

All of the Out of Context entities are displayed in the dialog box along with the names of the components to which they were related.

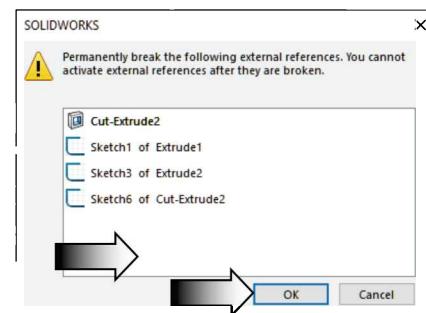
Click Break All (Arrow).



Click OK to confirm the delete of all External References.

Click **Continue** and close the External dialog box.

TIPS:



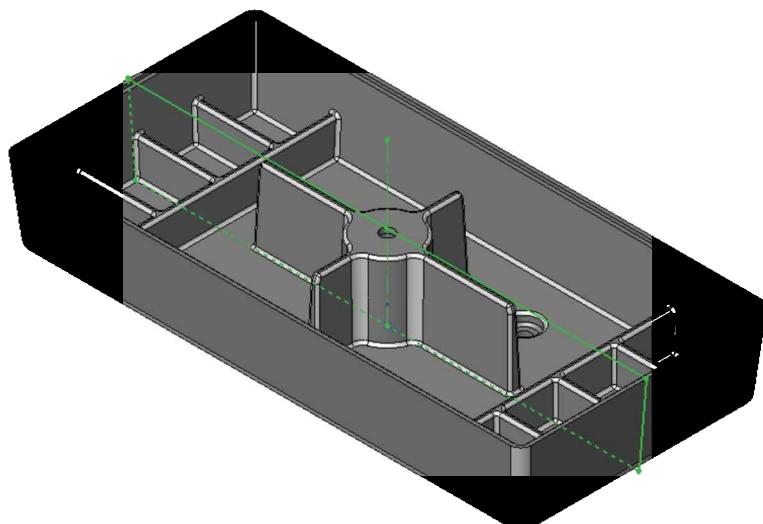
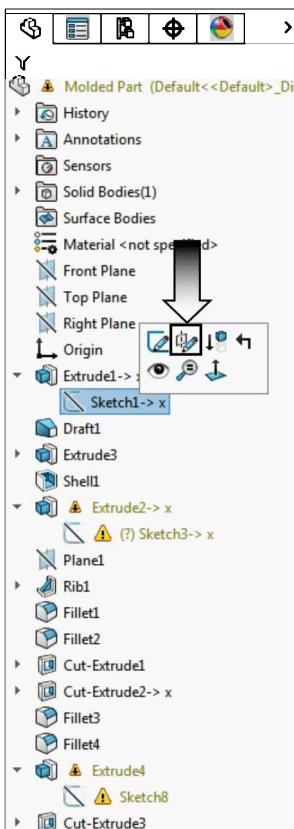
4 basic steps should be done, in most cases, to repair or replace External References:

1. *Break all External references (Right click on the part's name and select External Refs.).*
2. *Replace the sketch Plane or Face (if missing).*
3. *Delete or replace any Relation with an External Reference symbol next to it (Display/Delete Relations).*
4. *Repair or replace the extrude type.*

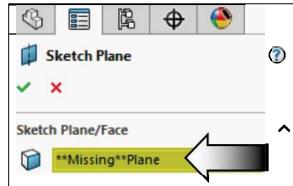
3. Replacing the Sketch Plane:

Expand the **Extrude1** feature to see **Sketch1** under it.

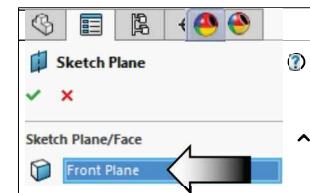
Right-click **Sketch1** and select **Edit Sketch Plane** .



The Sketch Plane is missing; a new plane or face must be selected to replace it.



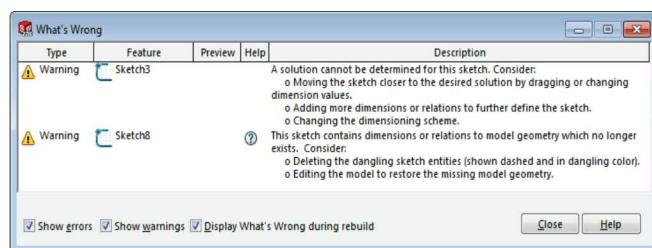
Select the **Front** plane from the FeatureManager tree to replace the missing plane.



After replacing the plane, click **OK**.

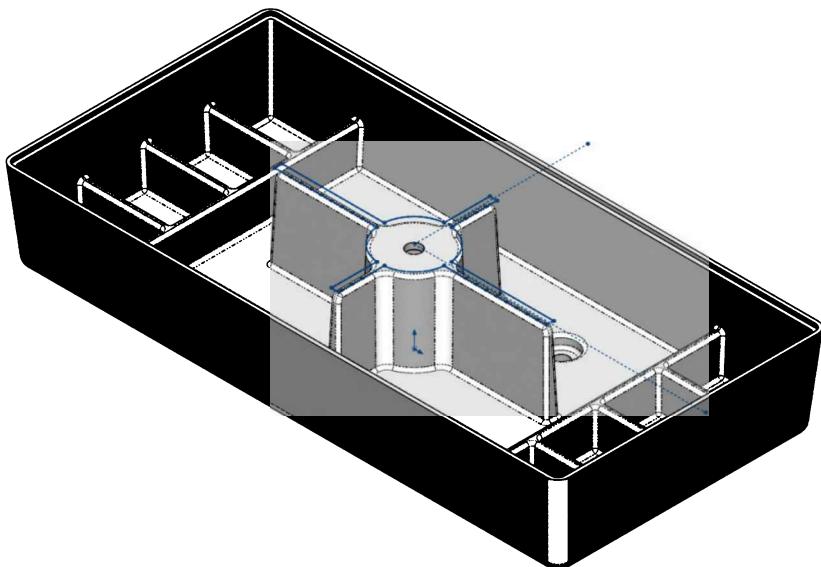
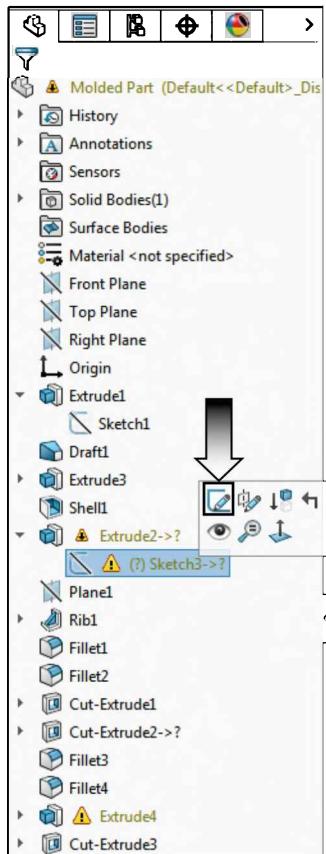
The system displays the warnings on other errors along with the solutions for repairing them.

Click **Close** .



4. Repairing the sketch Relations and Dimensions:

Right-click Sketch3 and select Edit Sketch.



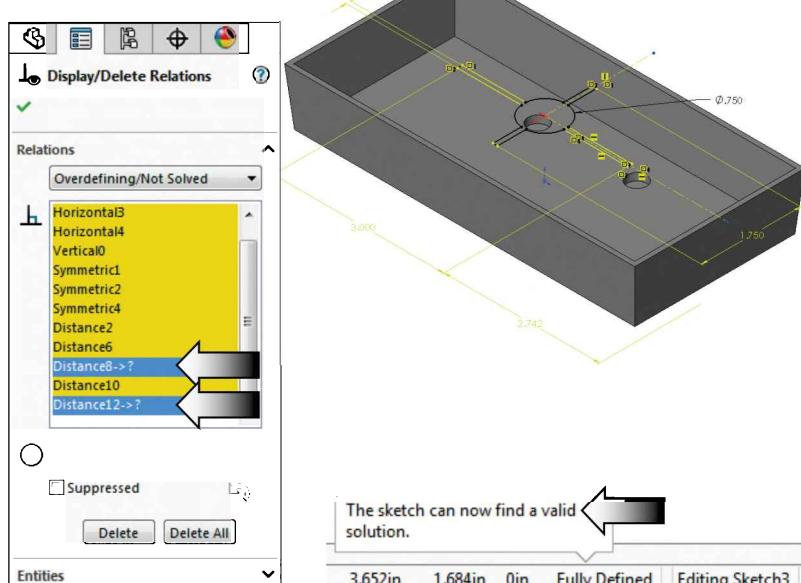
Click the **Display/Delete Relations** command or select: Tools / Relations / Display-Delete.

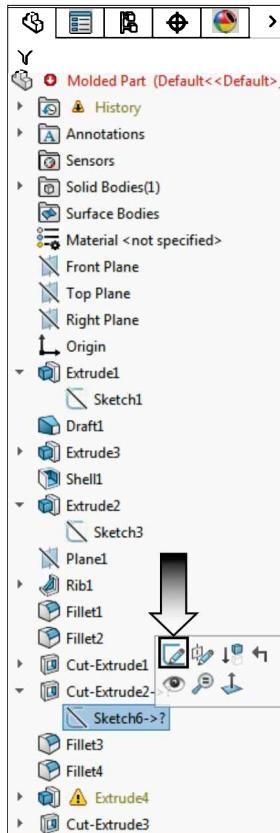
Delete the 2 geometric relations that have the external symbols next to their names (arrows).

The sketch becomes fully defined.

Click **OK**.

Exit the sketch .

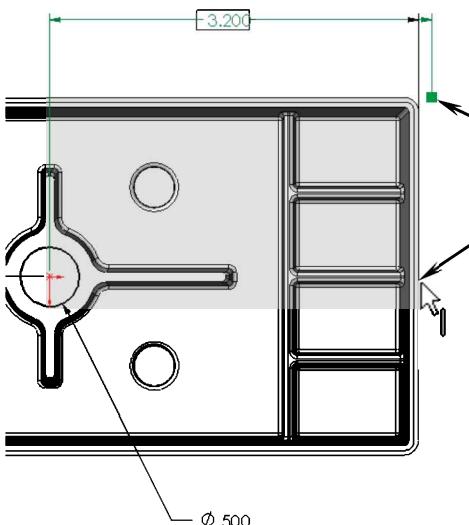
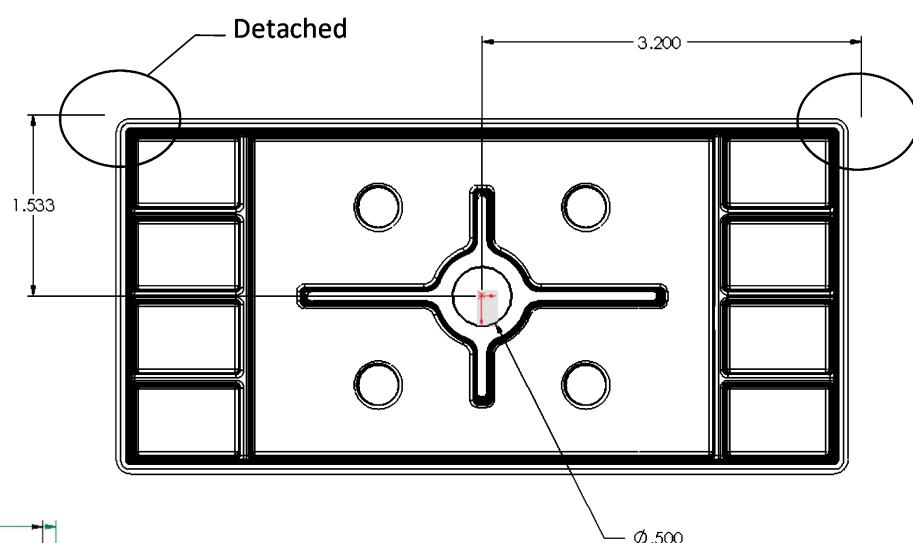




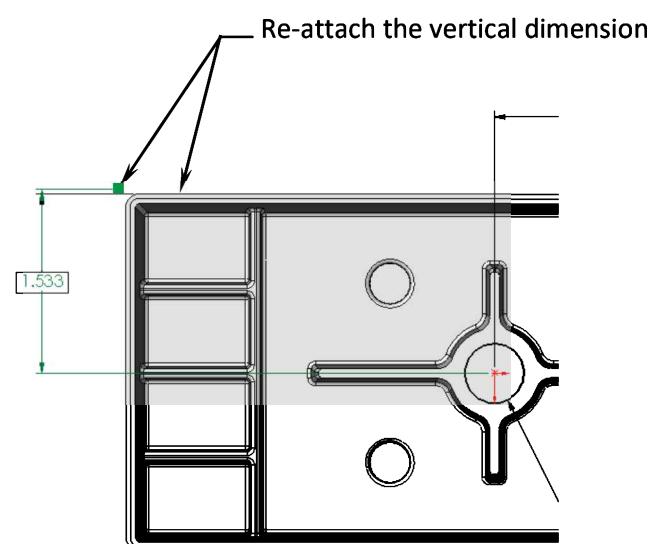
5. Repairing the next sketch errors:

Right-click **Sketch6** (under Cut-Extrude2) and select: **Edit Sketch**

Two dimensions are no longer attached to the part (circled).



Select the dimension to see the handle points. Drag the handle point of the dimension line (the red dot) to the model edge as shown, to re-attach it.

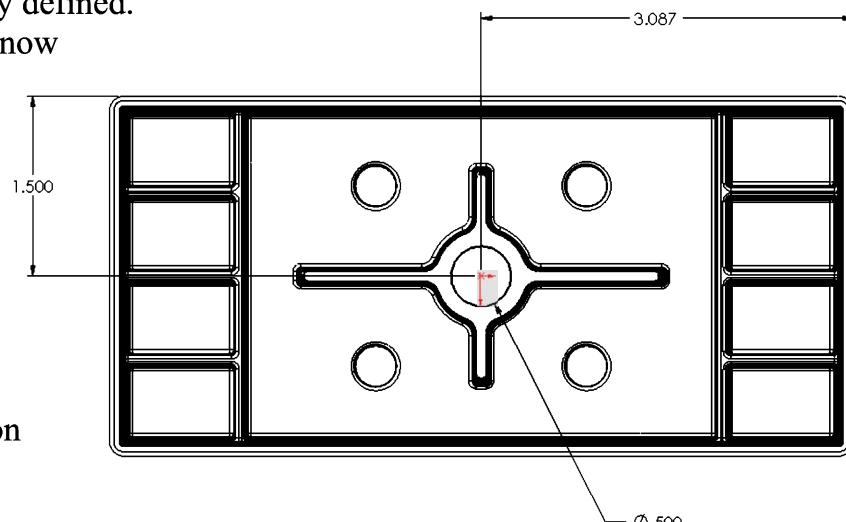


The sketch becomes fully defined.

The two dimensions are now

attached to the edges
of the model and also
locate the center
of the circle.

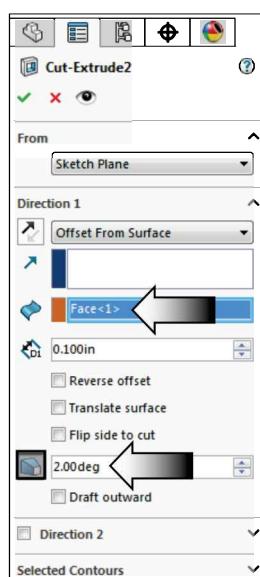
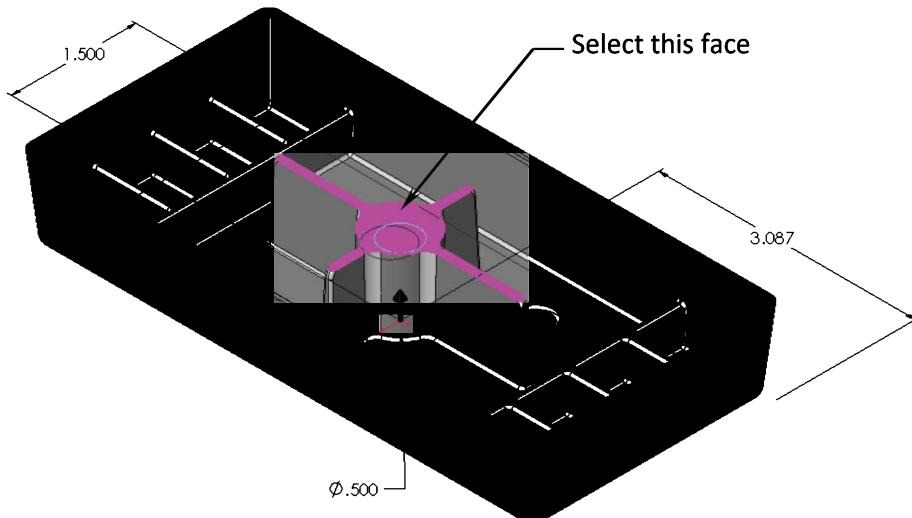
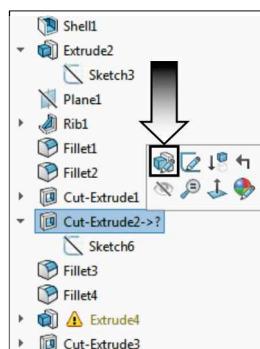
Exit the sketch .



There is still a warning on
the Cut-Extrude2.

6. Correcting the extrude type:

Right-click **Cut-Extrude2** and select **Edit Feature**.



The face that was used as the end condition option is no longer recognized; a new face must be selected to replace the missing one.

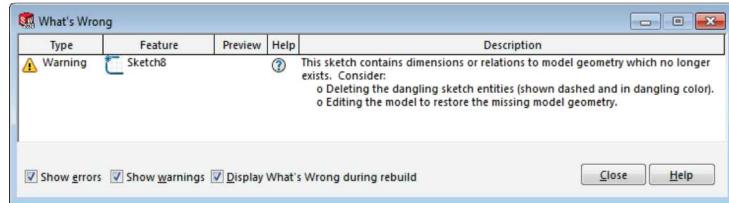
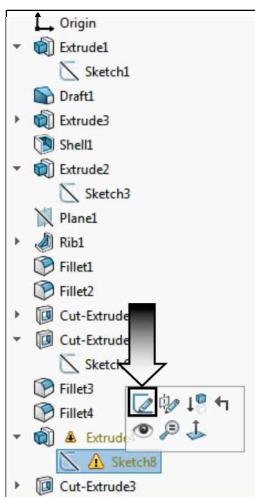
Select the **face** as indicated.

Leave extrude depth as **.100in** and the draft angle at **2deg**.

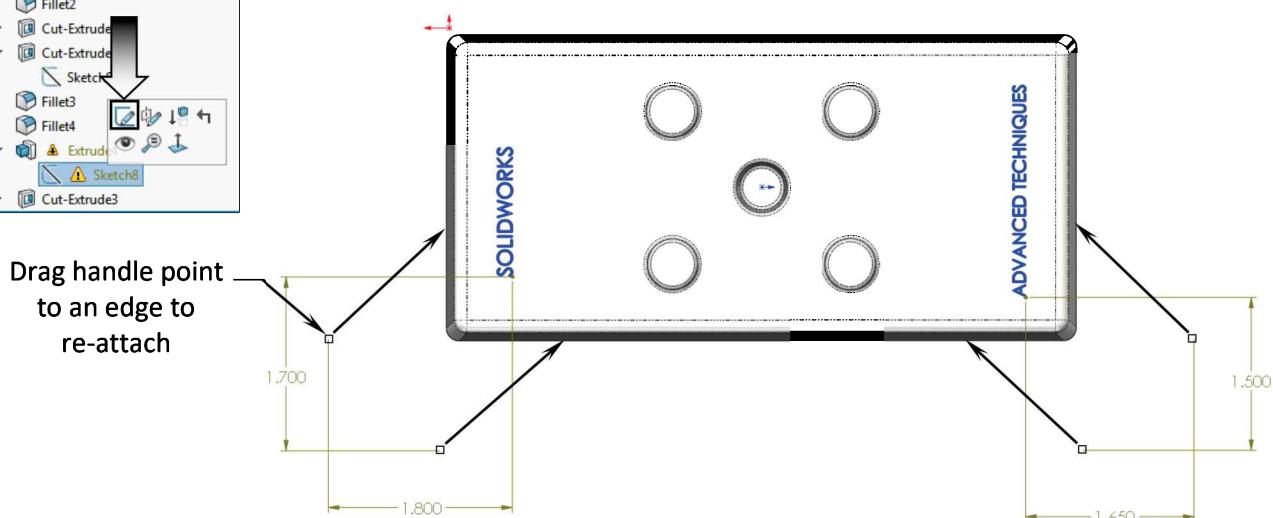
Click **OK**.

7. Repairing the errors in the last sketch:

Right-click Sketch8 and select **Edit Sketch**.

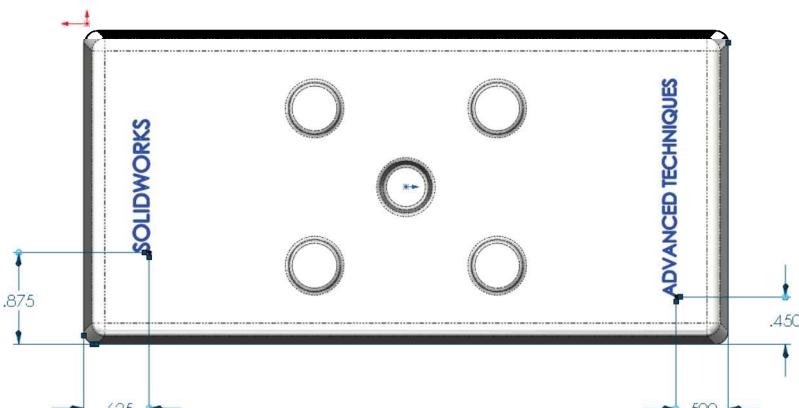


Change to the **Bottom** view orientation or press **Control + 6** to switch to the bottom view.



The dimensions that were created from or to other parts became Dangling (Olive-green color) because they are no longer attached to the models.

Select one of the dimensions and **drag its handle point** (the red dot) to a model edge to reattach it.

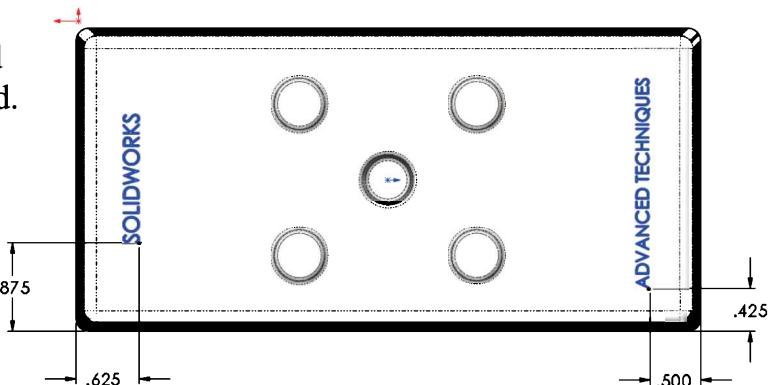


Repeat the same step to re-attach all other dimensions.

The sketch becomes fully defined after all dimensions are reattached.

Change the dimension values as shown to reposition the text.

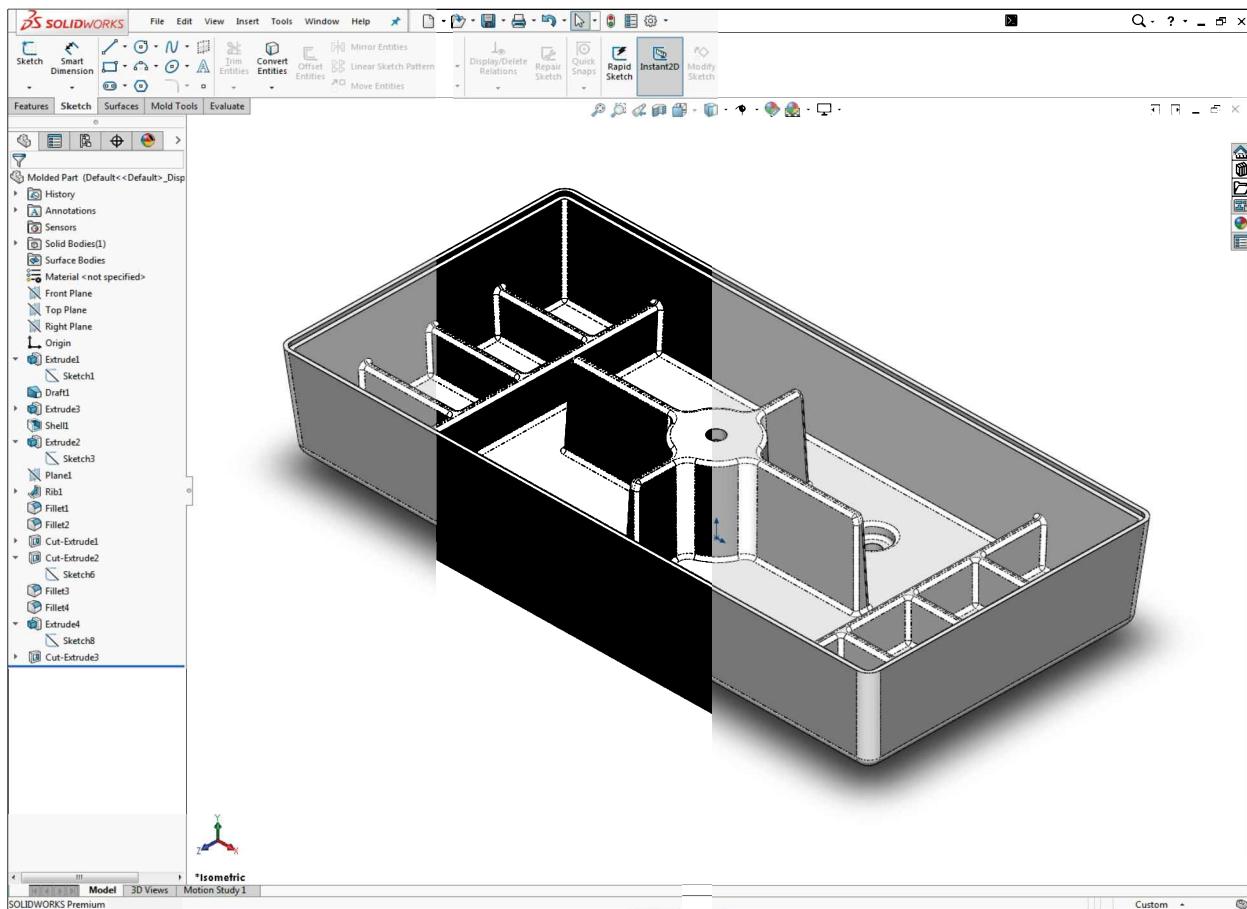
Exit the sketch.



The reference symbols and the error colors on the FeatureManager tree should now be all removed.

8. Saving your work:

Select File / Save As / Repair Errors / Save.



Final Exam 1 of 2

(The instructions will be limited so that you can try out your own approaches.)

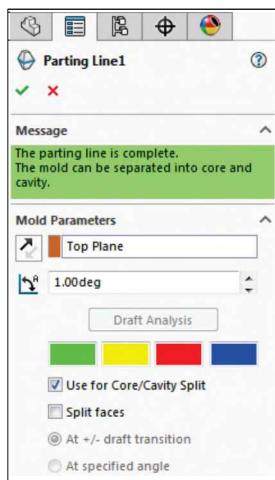
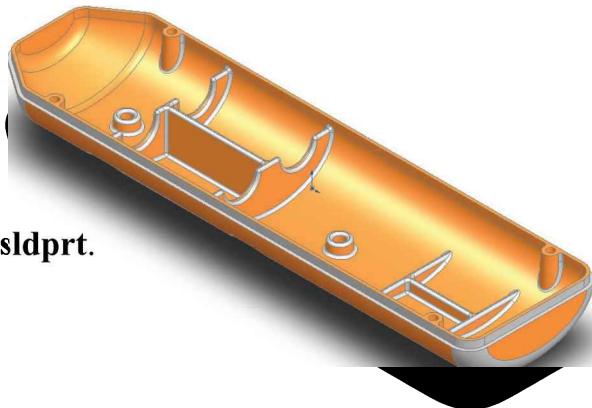
1. Opening a part document:

Go to:

Training Files folder

Tooling Design folder

Open: **L4 Final Exam_Handle Mold.sldprt.**

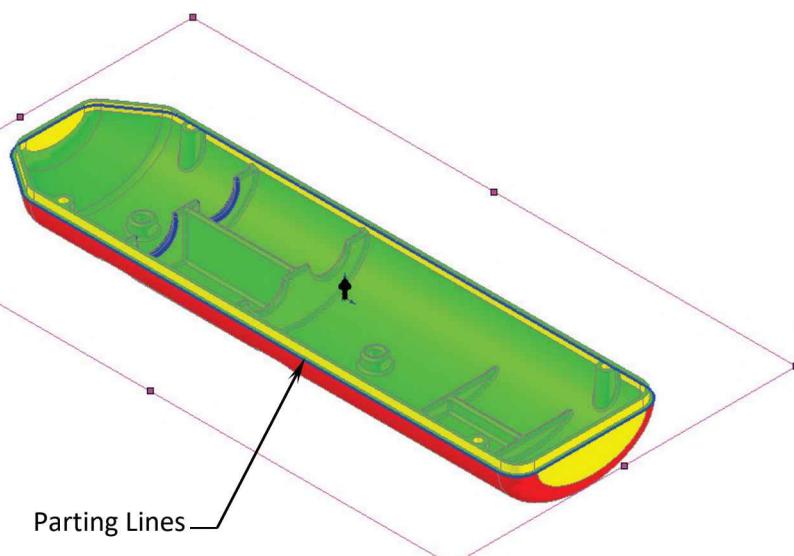
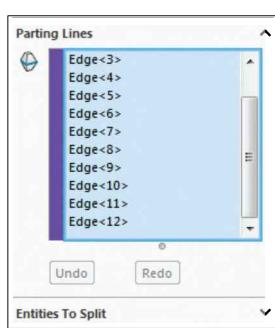


2. Creating a Parting Line:

Direction of Pull = **TOP plane.**

Draft Angle = **1deg.****

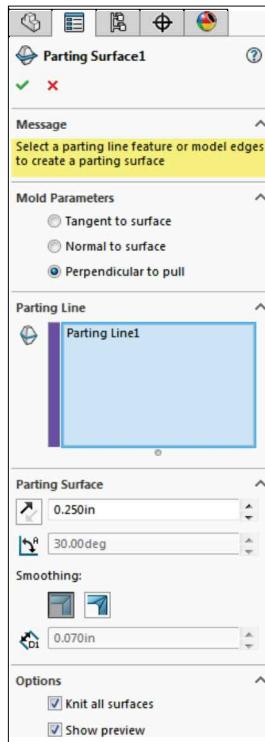
Parting Lines = **All outer edges as noted.**



**** Important - Roll back under the Loft feature and:**

a/ Add 1° drafts to all Yellow faces, including the 4 holes; this change will create some errors in the part.

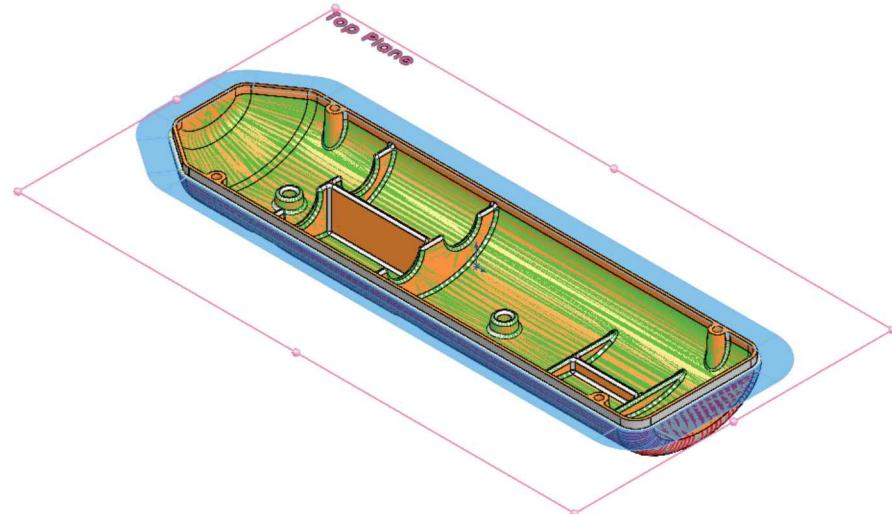
b/ Reorder or recreate the fillets, if necessary, after adding the drafts.



3. Creating a Parting Surface:

Select the **Perpendicular to Pull** option.

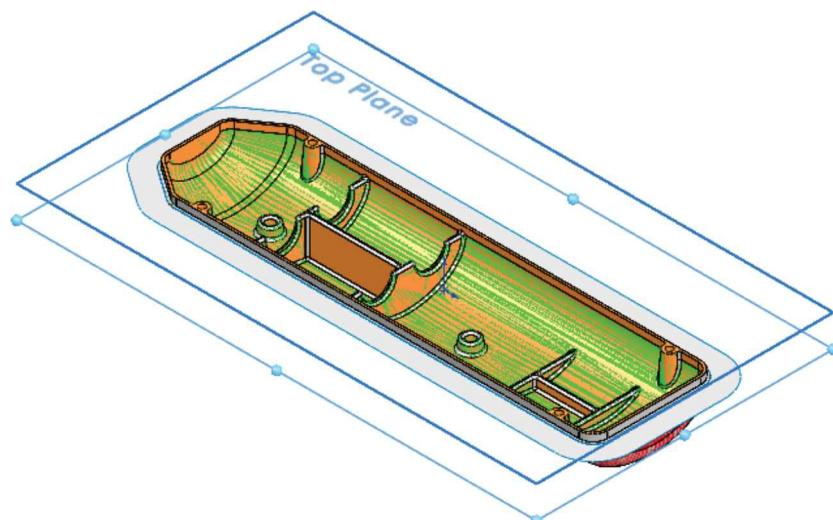
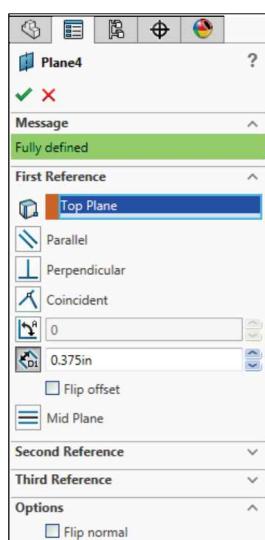
For Parting Line: select the **Parting Line1** from the FeatureManager tree.



4. Adding an Offset Distance plane:

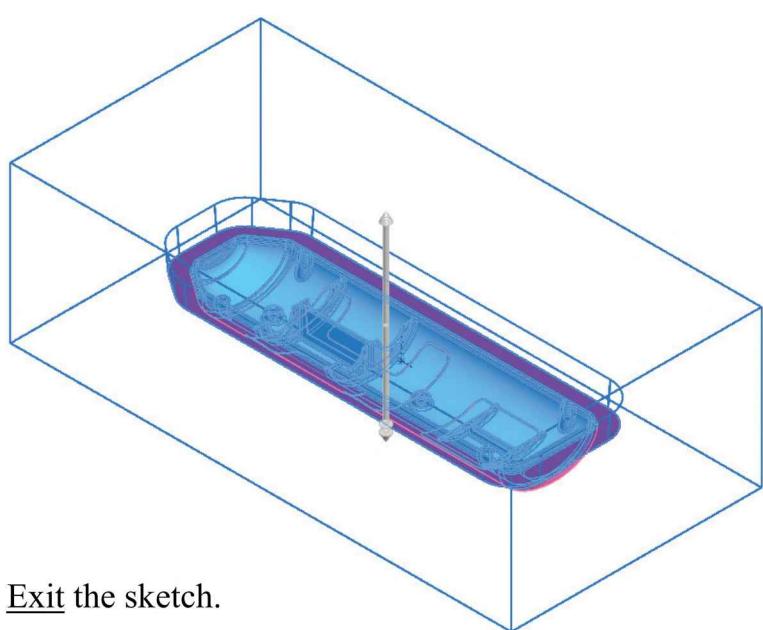
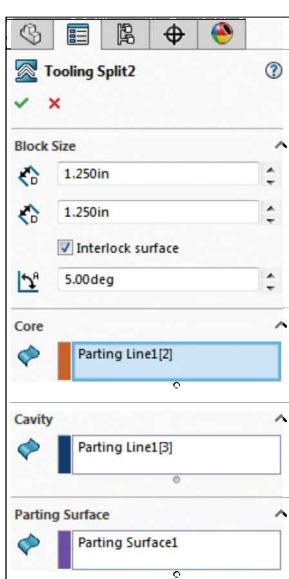
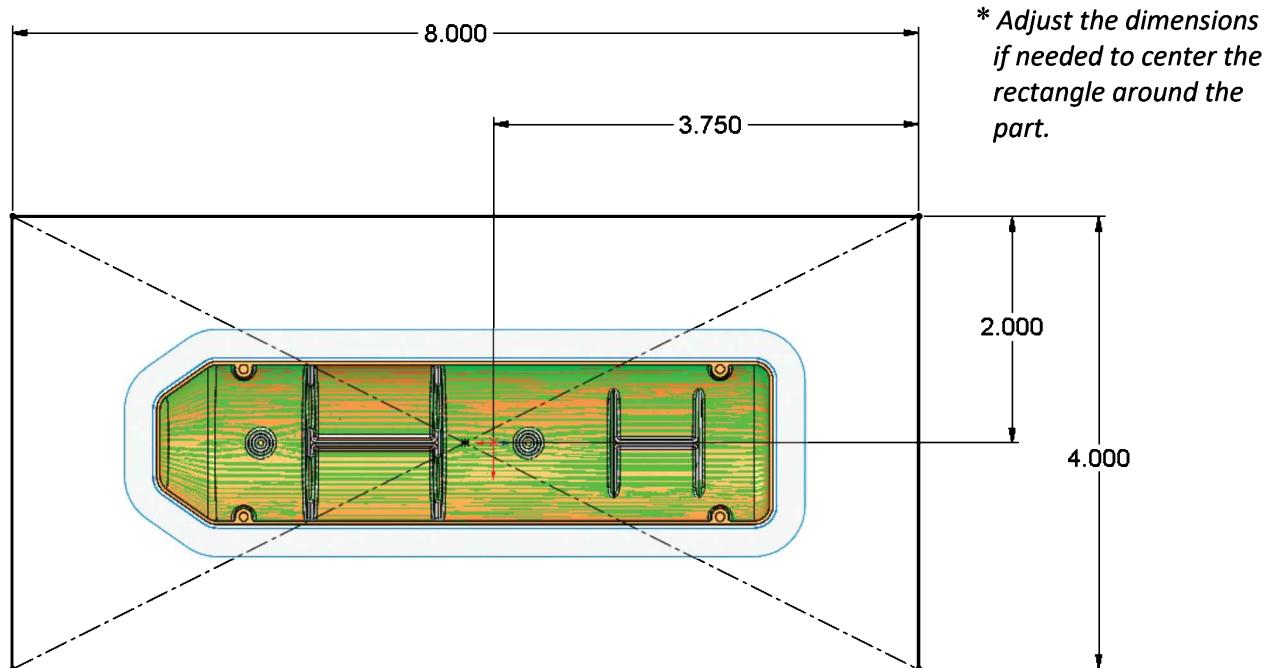
Use the **Top** reference plane and **.375 in.** distance.

The new plane is placed **above** the Top plane.
(This new plane will also be used to create the Interlock Surfaces.)



5. Sketching the mold block profile:

Sketch a **Rectangle** on the new plane (Plane1) and add dimensions* shown.



6. Making a Tooling split:

1.250in. (upper block)

1.250in. (lower block)

Use **Interlock Surface** with **5° Draft**.

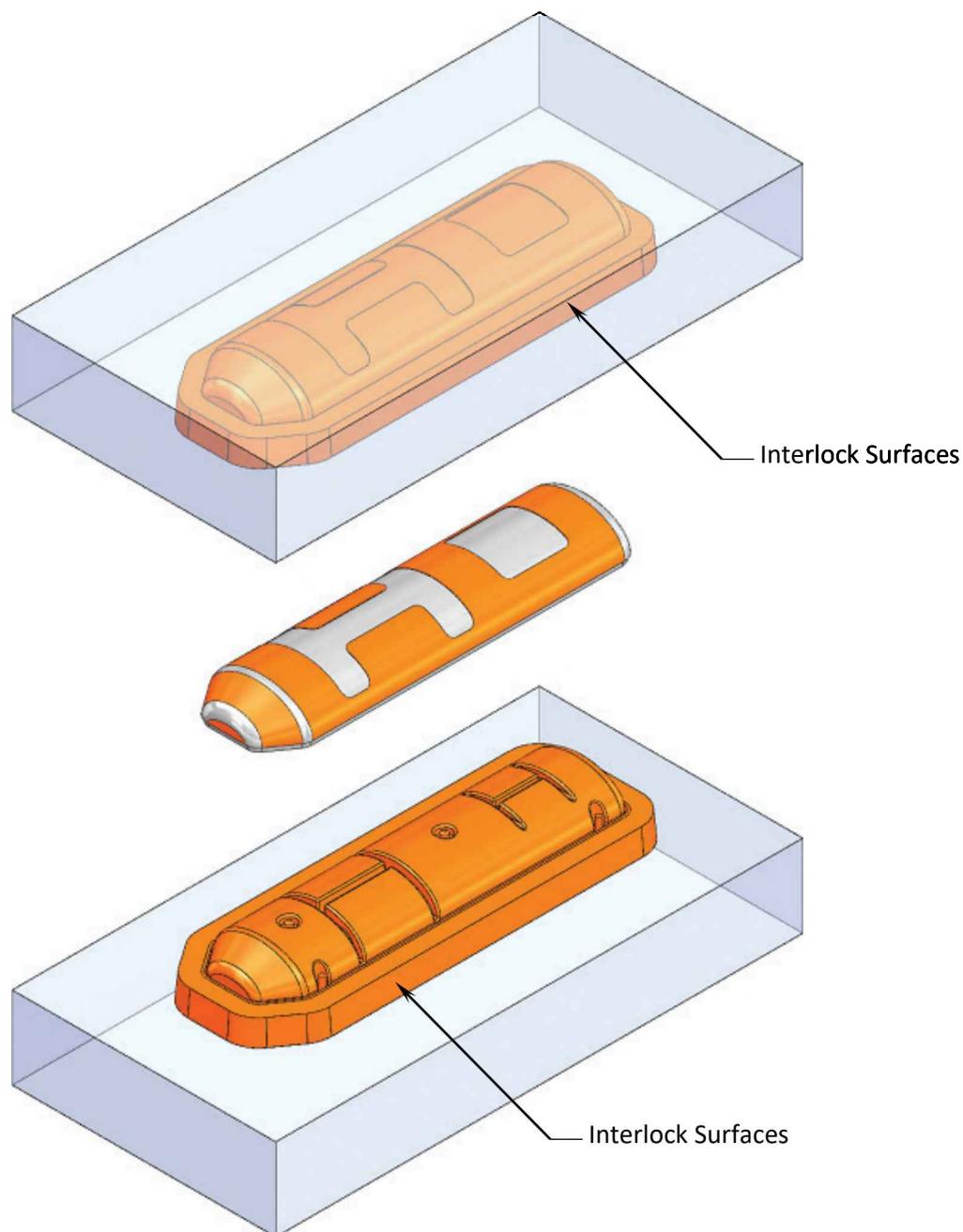
7. Separating the mold blocks:

Use the **Move/Copy** command to separate the two halves.

**** OPTIONAL:** Make the upper and lower solid bodies transparent for clarity.

8. Saving your work as:

L4 Final Exam_Handle Mold_Completed.



Final Exam 2 of 2

(The instructions will be limited so that you can try out your own approaches.)

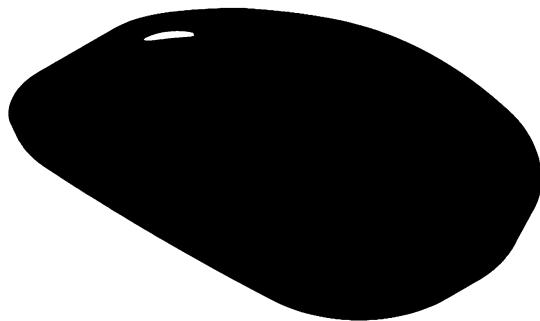
1. Opening a part document:

Go to:

Training Files folder

Tooling Design folder

Open: **Final Exam_Mouse Mold.sldprt.**



The intent is to create a small wall (Lip & Groove) that runs around the perimeter of the part to prevent the upper and lower halves from shifting and also to help align them more easily during assembly stage.

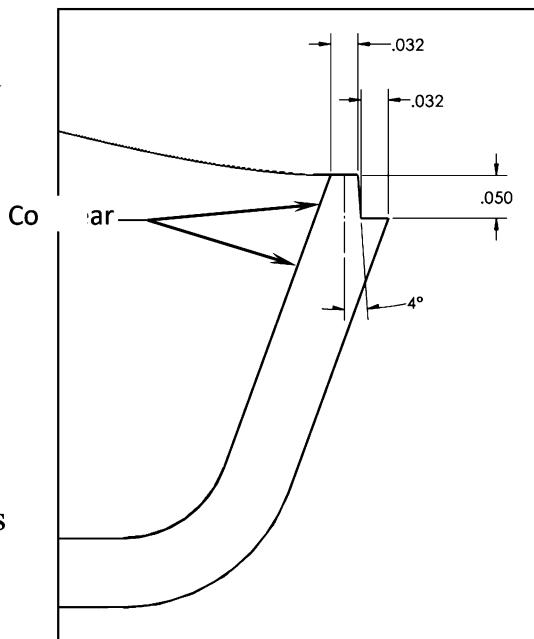
After the Lip feature is created, a pair of Core & Cavity molds needs to be made from the part. All surfaces in the model must have a minimum of 3° draft angle.

2. Creating a Lip feature*:

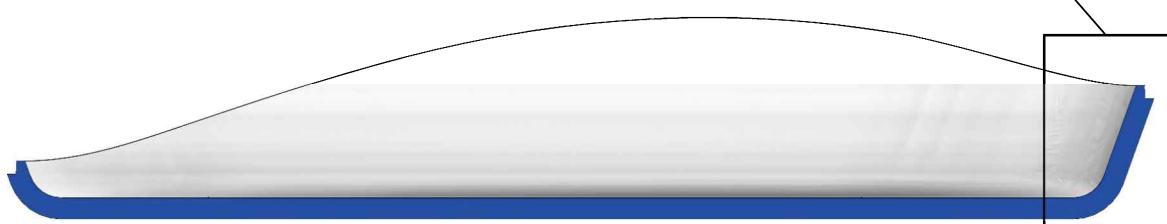
There are several ways to create the Lip feature. You can use any methods that you like as long as it has all of the required size and location dimensions.

The section and detail views show the details of the Lip feature added to the part.

It is .050" tall and .032" wide and has a 4° draft outside.



* Create the Lip feature before making the mold in the next step.



3. Analyzing the Draft Angles:

Switch to the **Evaluate** tab and click **Draft Analysis**.

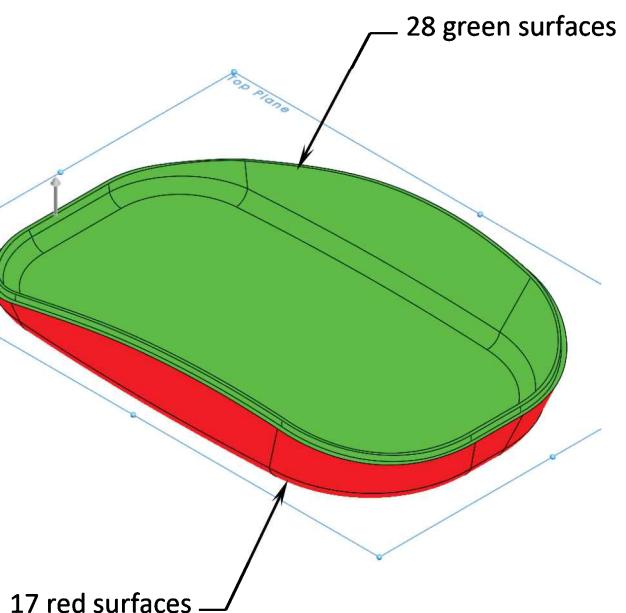
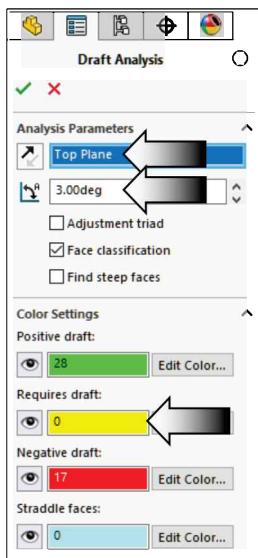
For Direction of Pull, select the **Top** plane.

For Draft Angle enter **3.00deg**.

Click the **Face Classification** checkbox.

The Required Draft section (yellow) must show **0** (zero).

Click **OK**.



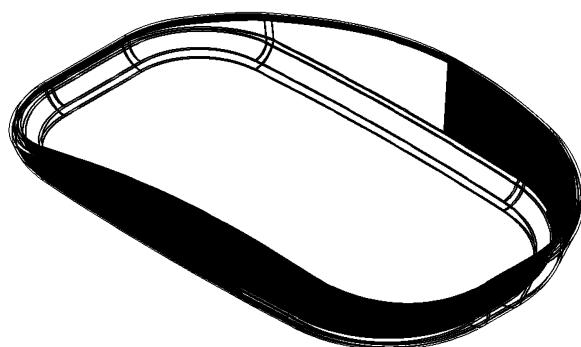
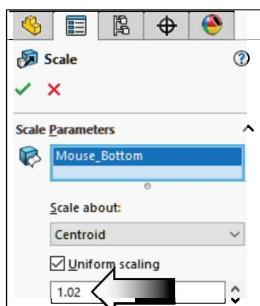
4. Applying the Scale Factor:

Switch to the **Mold Tools** tab.

Click **Scale**.

For Scale Parameter, select the part.

For Scale About, use the default **Centroid**.



For Scale Factor, enter **1.02** (2% larger) and enable the **Uniform Scaling** checkbox.

Click **OK**.

Set the material to **ABS**.

5. Creating a Parting Line:

Click **Parting Line** on the **Mold Tools** tab.

For Pull Direction, select the **Top** plane from the FeatureManager tree.



For Draft Angle, enter **3.00deg**.

Click the **Draft Analysis** button to analyze the drafts in the model.

Click **OK**.

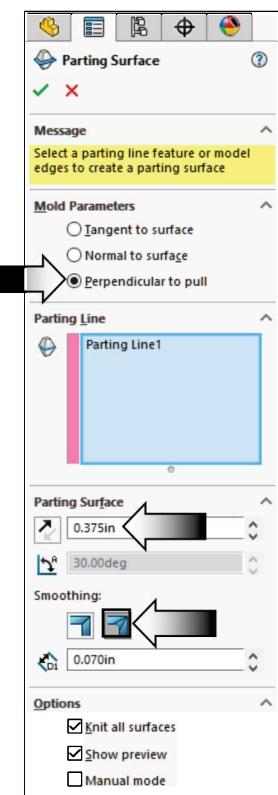
Ensure that there are no yellow surfaces in the model at this point before moving forward to the next step. Stop and make any corrections if needed.



A Parting Line is created where the green surfaces (Core) meet the red surfaces (Cavity).

6. Adding a Parting Surface:

A parting surface is used to split the mold into the Core and Cavity halves.



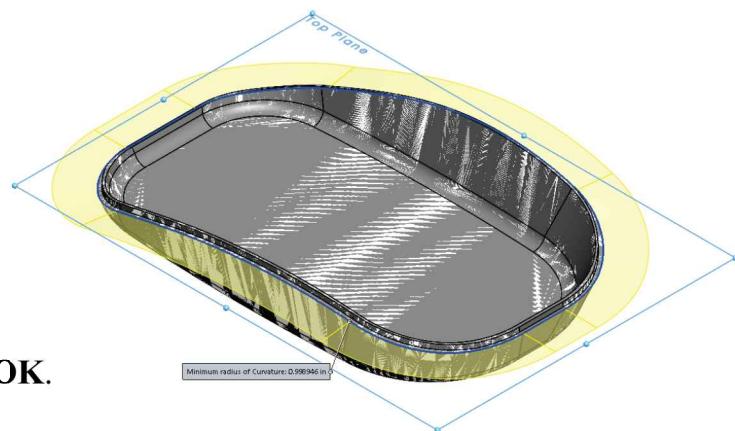
Click Parting Surface on the **Mold Tools** tab.

For Mold Parameters, select **Perpendicular to Pull**.

The **Parting Line** should be selected automatically.

For Parting Surface Distance, enter **.375in**.

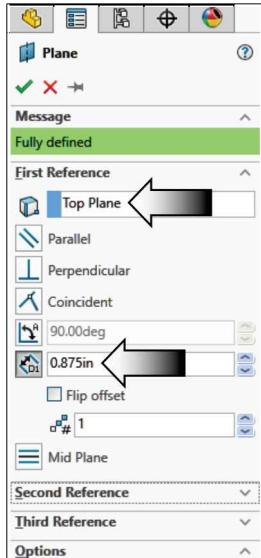
For Smoothing between adjacent surfaces, select **Smooth**.



Click OK.

7. Creating a new plane:

A plane is used to sketch the profile of the mold block and also to define the height of the Interlock Surface.

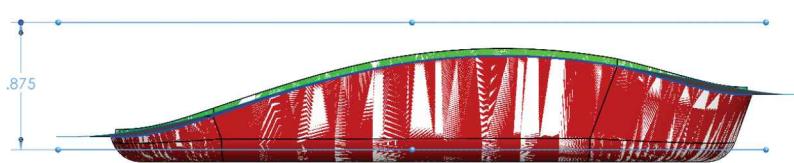


Switch to the **Features** tab and click: **Reference Geometry, Plane**.

For First Reference, select the **Top** plane from the tree.

For Offset Distance, enter **.875in**.

Click OK. The new plane is placed above the Top plane.



8. Creating a Tooling Split:

Select Plane1 and open a **new sketch**.

Sketch a **Corner Rectangle** around the model.

Add the location dimensions shown to fully define the sketch.

Exit the sketch.

Switch to the **Mold Tools** tab.

Select the sketch of the **Rectangle** and click:
Tooling Split.

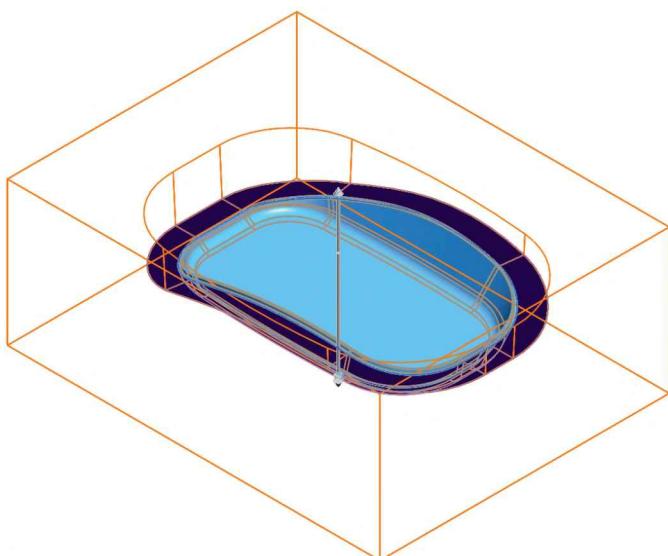
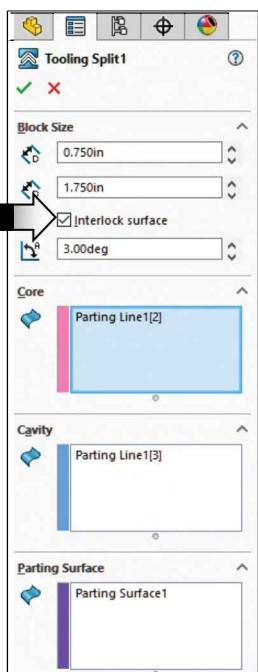
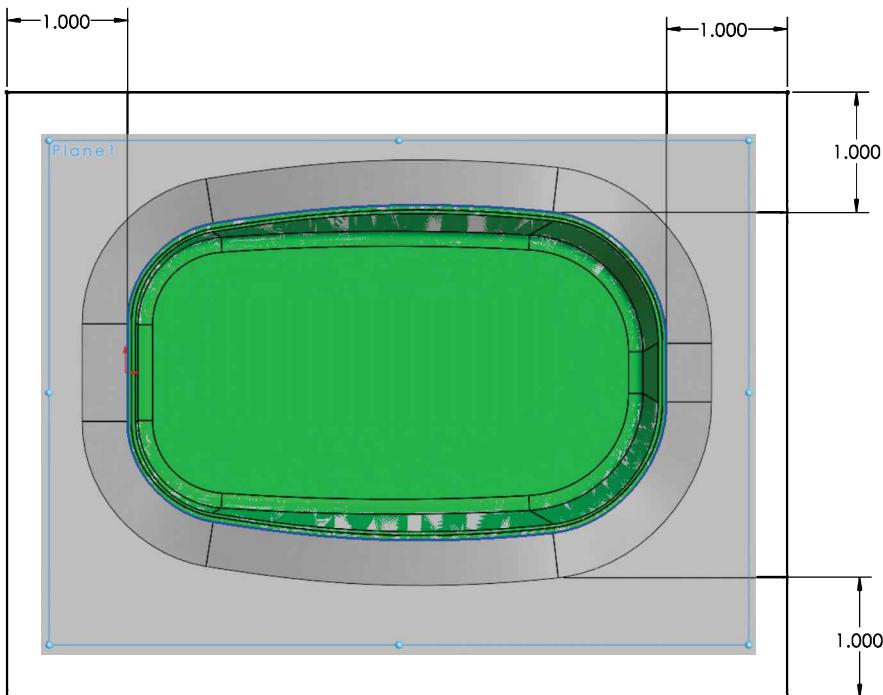
For Block Size enter:

.750in.
1.750in.

Enable the **Interlock-Surface** checkbox.

Enter 3° for Draft Angle.

Click **OK**.



Hide all surfaces in the Surface Bodies folder.
Also hide the Parting Line.

9. Separating the mold blocks:

Use the **Move/Copy Bodies** command to separate the 2 mold blocks.

Move the Core Block along the Y direction **4.50in**.

Also move the Cavity Block along the Y direction **-5.50in**.

Rename the 3 solid bodies to:

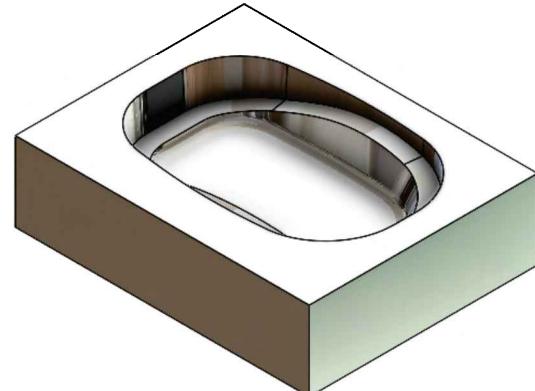
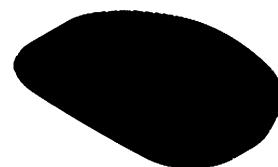
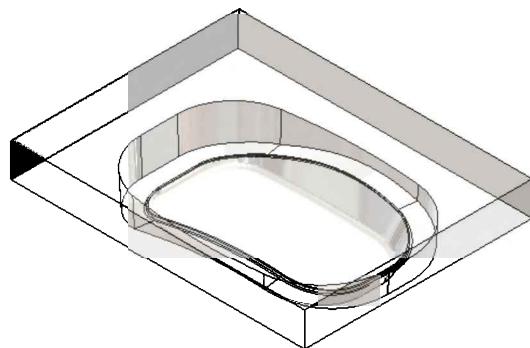
Core Block

Plastic Part

Cavity Block

Change the material of the 2 mold blocks to
Plain Carbon Steel

The material **ABS** was assigned to the plastic part in step number 4.

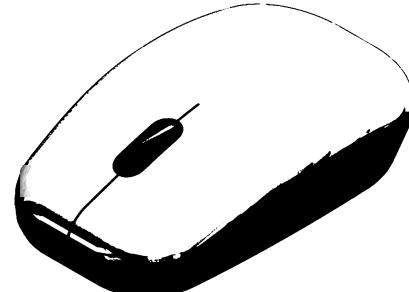
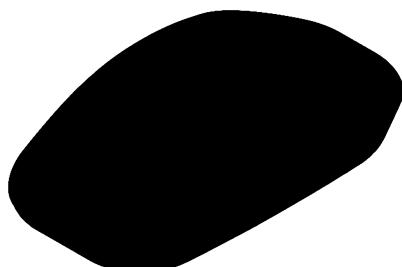
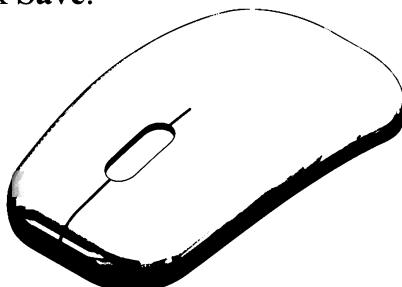


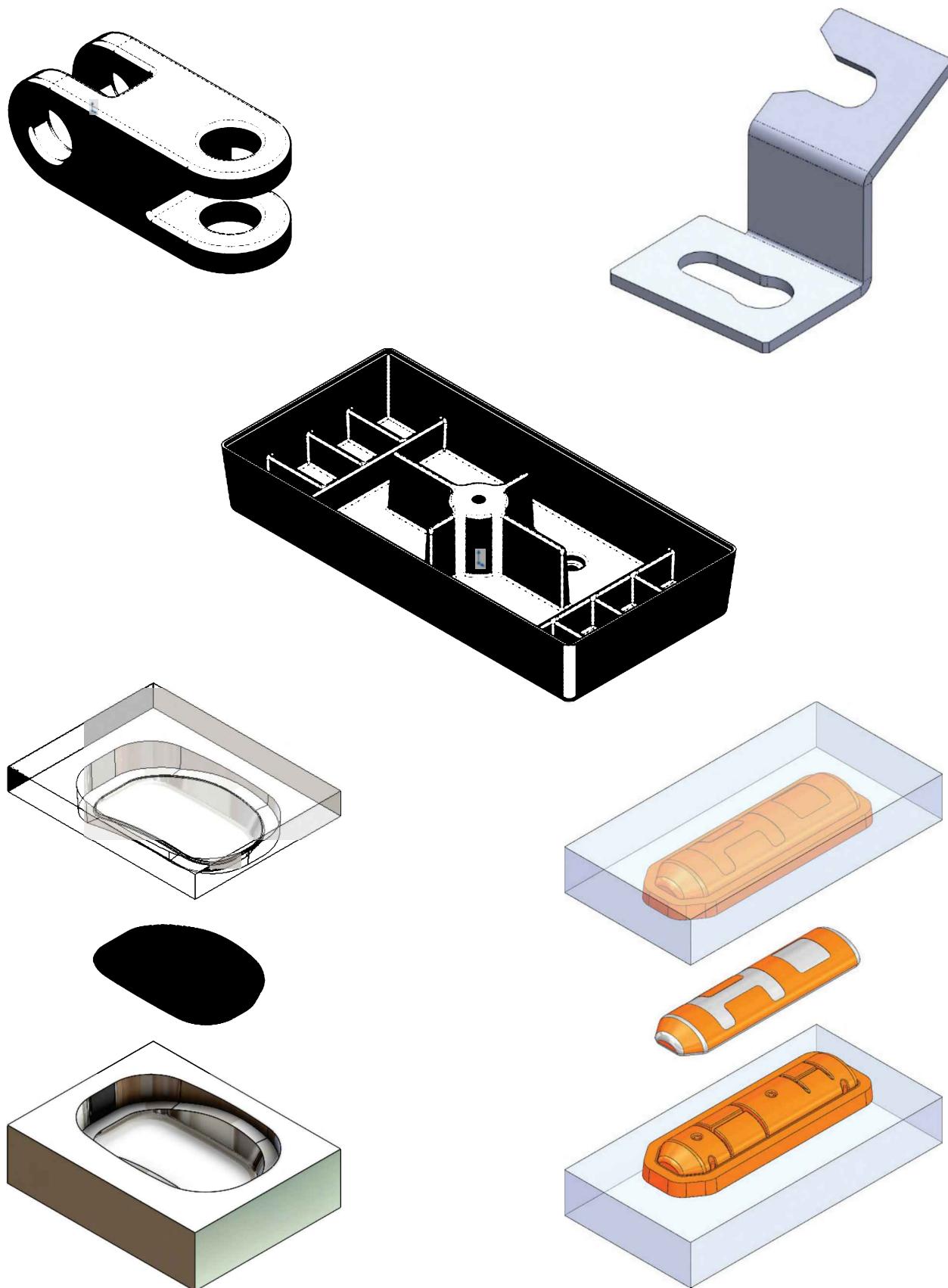
10. Saving your work:

Select **File, Save As**.

Enter: **Final Exam_Mouse Mold_2of2_Completed**

Click **Save**.





CHAPTER 23

Using Appearances

A knurled feature will create hundreds of extra faces in the model; it will increase the file size and bog down graphical performance. But if it is important to show a knurl feature in a model, then there are a couple of techniques that we can use to accommodate it.

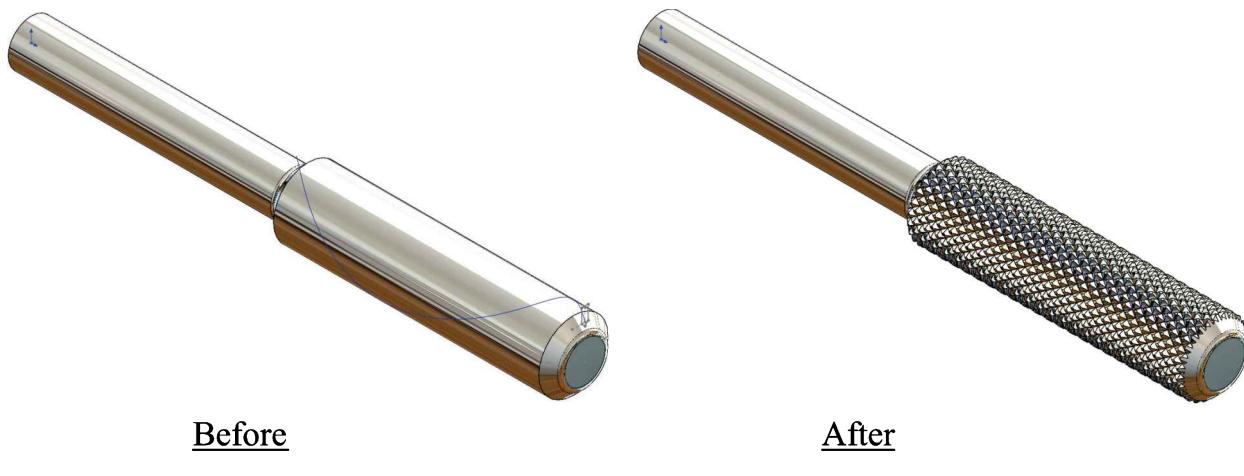
This first half of the chapter will guide you through the method of modeling the Raised Diamond Knurls where a sketch profile is swept along a path to create a spiral cut. The spiral cut is then patterned and mirrored to repeat the raised diamond shapes of the knurl.

Modeling Diamond Knurls

1. Opening a part document:

Browse to the Training Files folder and open a part document named:
Knurled Handle.

This model has a single solid body, a sweep profile, and a sweep path that was previously created.

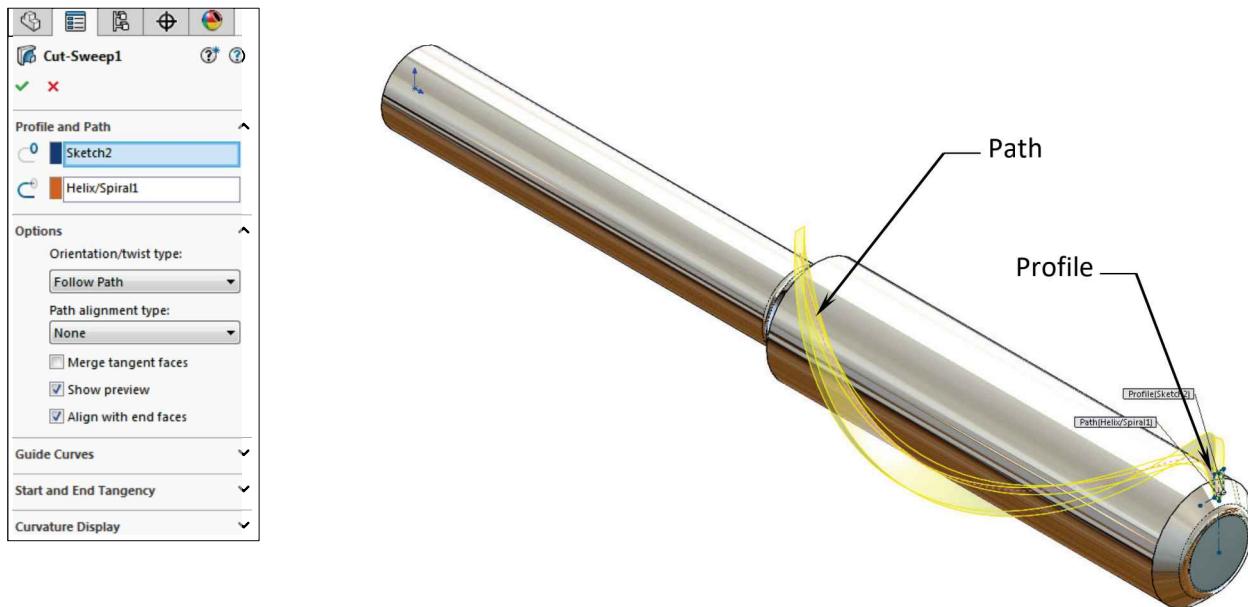


2. Creating a swept cut:

Switch to the **Features** tab and click the **Swept Cut** command.

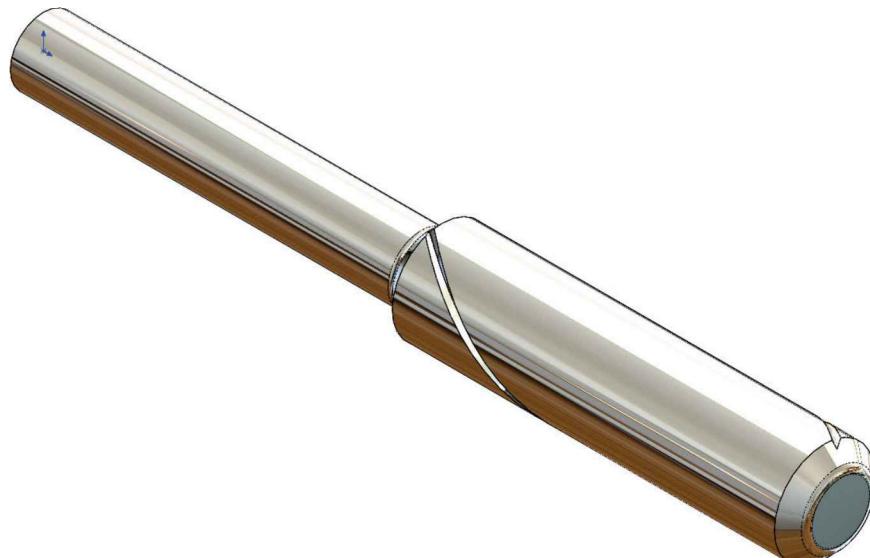
For Sweep profile, select **Sketch2** from the FeatureManager tree.

For Sweep path, select the **Helix**.



Click **OK**.

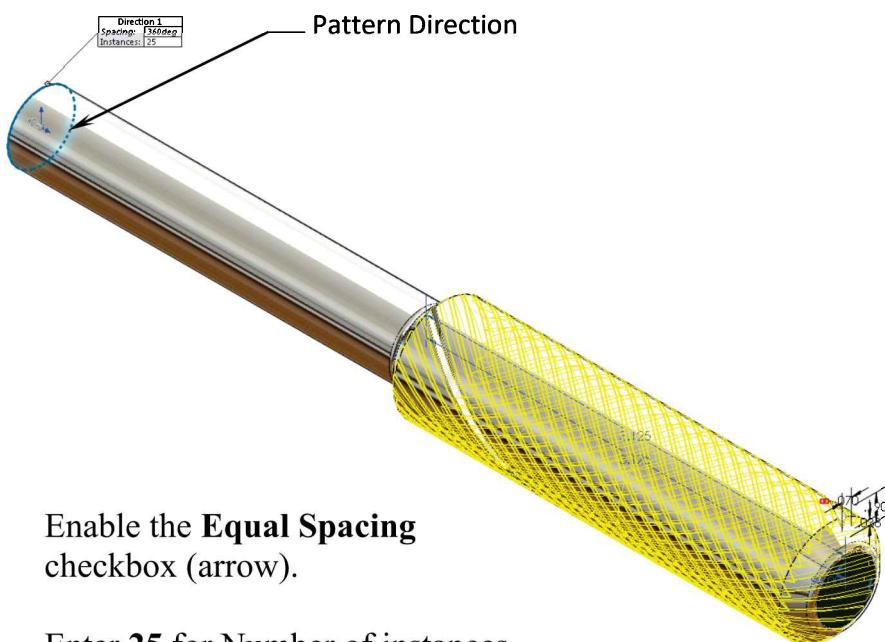
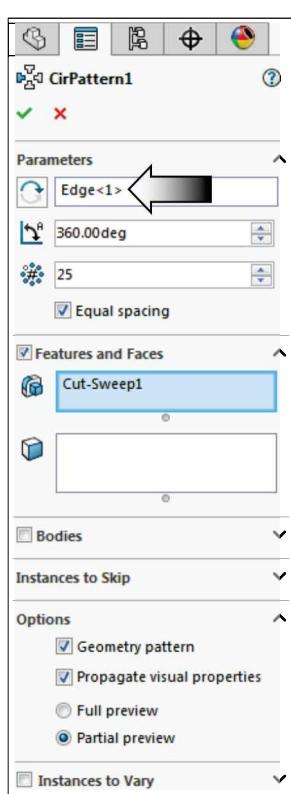
Zoom closer to inspect the model and to examine the swept cut feature.



3. Creating a circular pattern:

Click the **Circular Pattern** command below the Linear Pattern drop-down.

For Pattern Direction select the circular edge on the left end of the model.



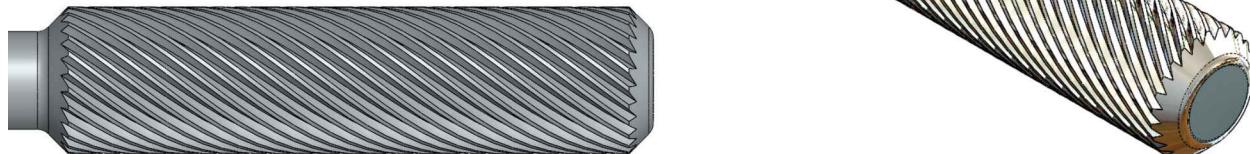
Enable the **Equal Spacing** checkbox (arrow).

Enter **25** for Number of instances.

Select the **Swept Cut** feature either from the FeatureManager tree or directly from the graphics area.

Click **OK**.

Change to the **Top** orientation (Control+5) to verify the pattern of the swept cut feature.

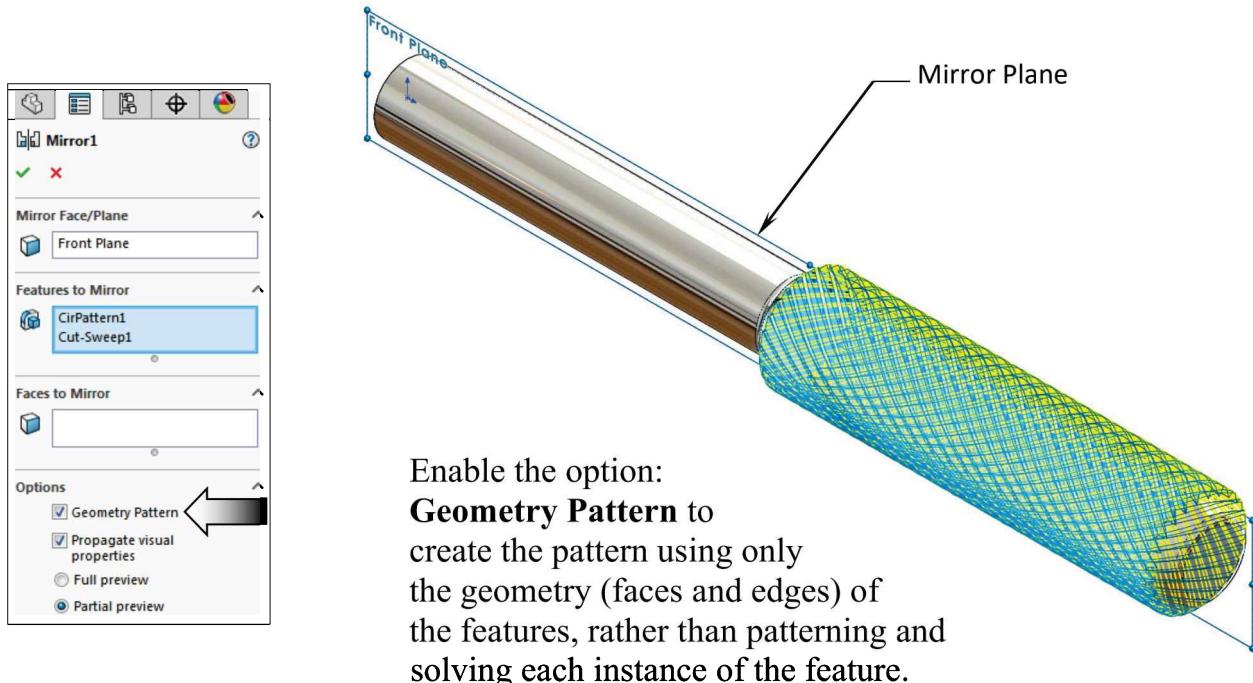


4. Creating a mirror pattern:

Click the **Mirror** command on the **Features** tab.

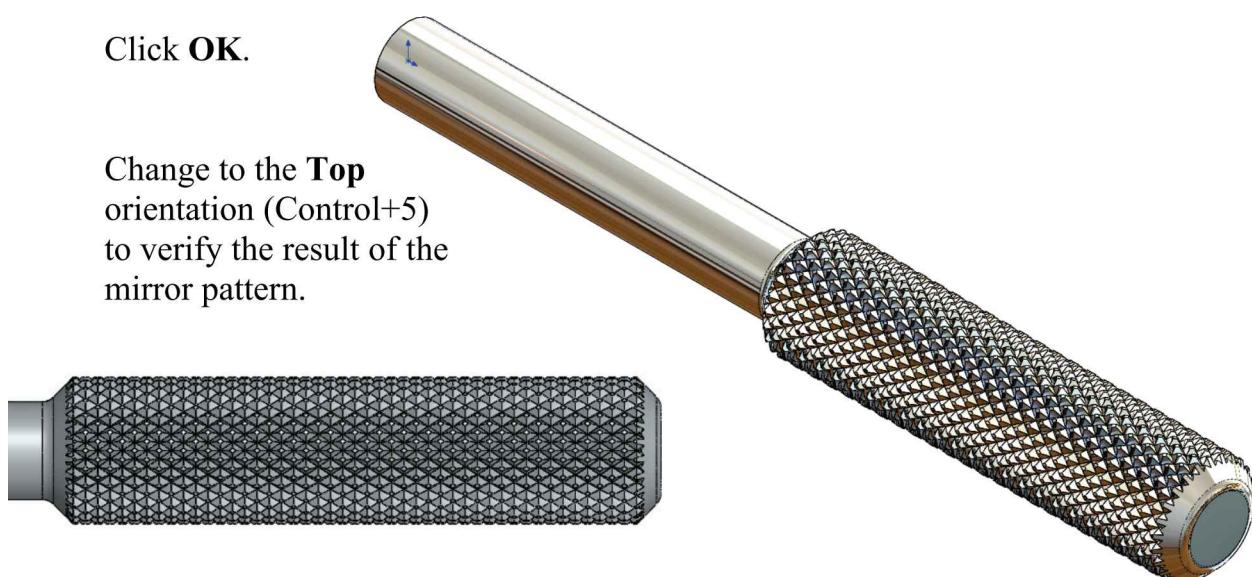
Expand the FeatureManager tree and select the **Front** plane to use as Mirror Plane.

For Features to Mirror select both the **Cut-Swept1** and the **CirPattern1** features.



Click **OK**.

Change to the **Top**
orientation (Control+5)
to verify the result of the
mirror pattern.



Save and close the document.

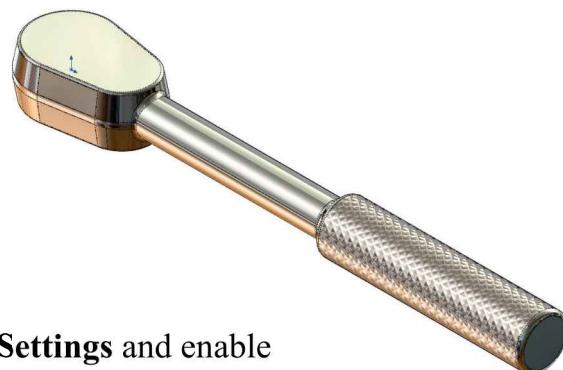
Applying the Knurl Appearance

Certain appearances require the use of RealView Graphics for more realistic representation; knurled, dimpled, or sandblasted are some examples.

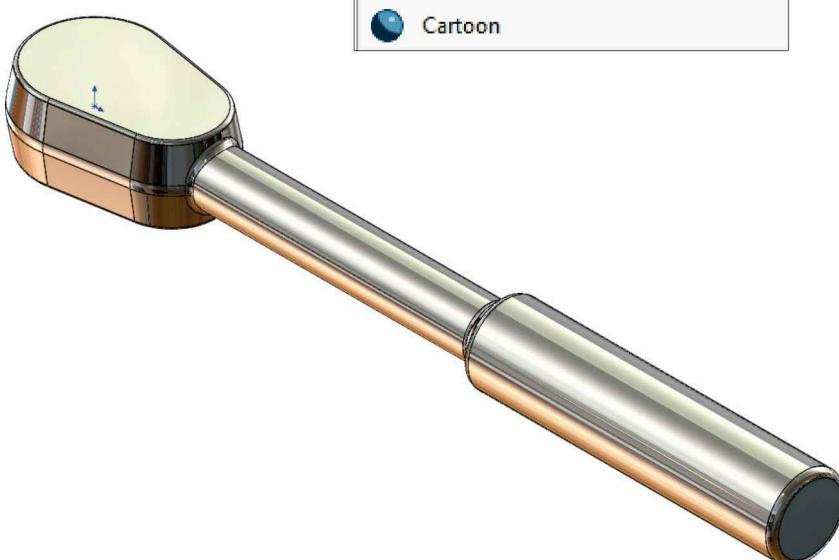
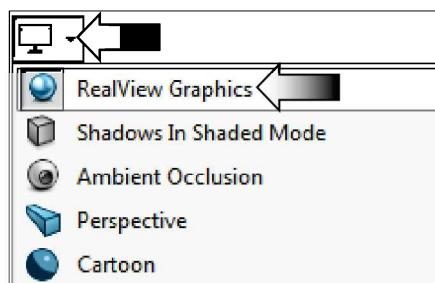
RealView gives models a realistic and dynamic representation without the need to render. If your graphics card is RealView-compatible, RealView is enabled by default.

1. Opening a part document:

Open a part document named:
Knurl Appearance.



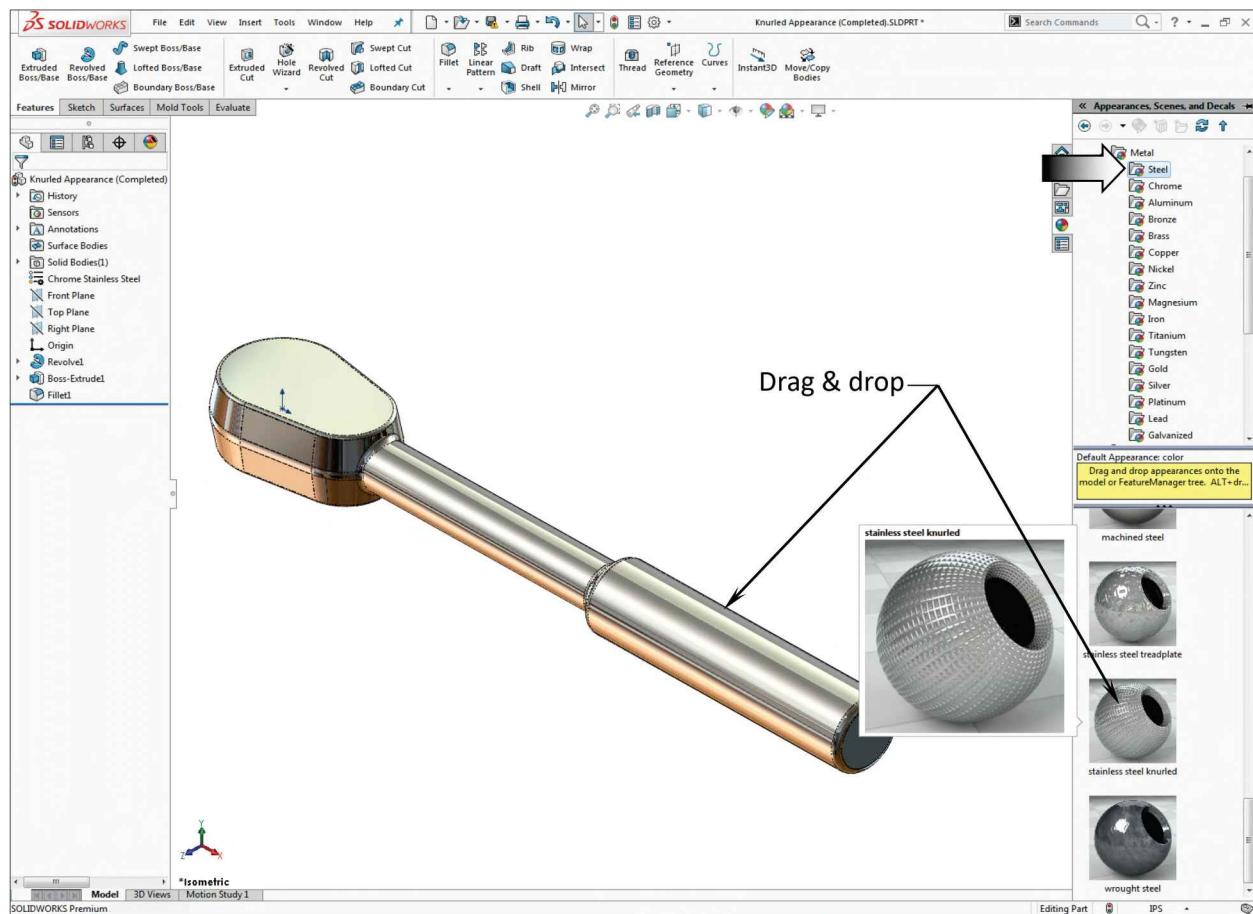
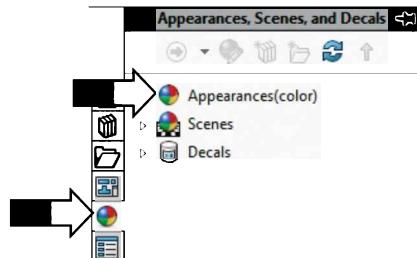
Click the drop-down arrow next to **View Settings** and enable the **RealView Graphics** option (arrow).



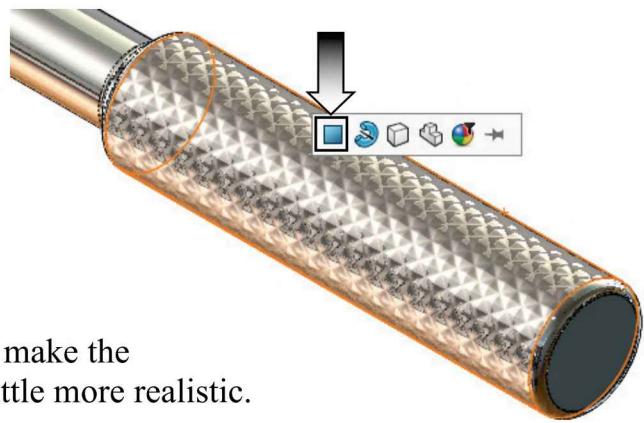
2. Applying the knurl appearance:

Expand the **Task pane** on the right side of the screen and pin it.

Expand the **Metal** and **Steel** folders. Drag and drop the **Stainless Steel Knurled** appearance onto the surface of the handle.



Select the **Apply to Face** option in the pop-up menu.

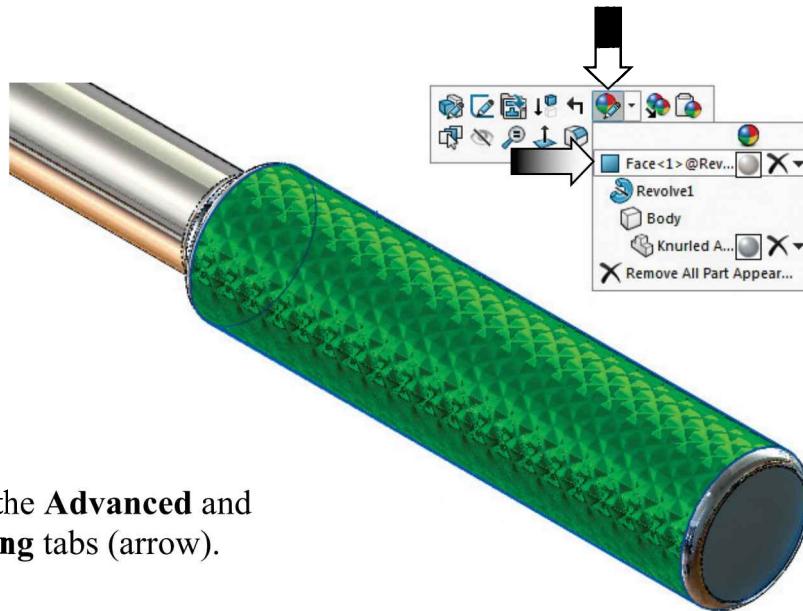


The default knurled appearance is applied to the selected face.

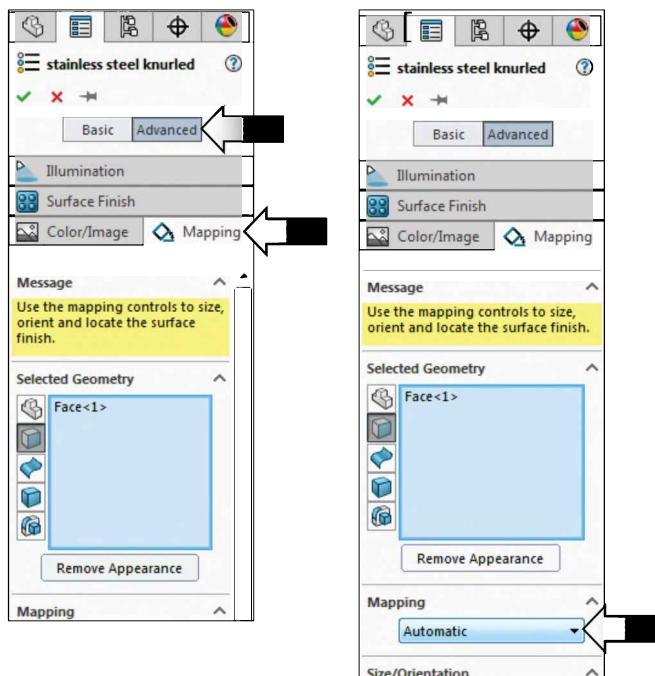
Next, we will modify the settings to make the diamond knurl appearance looks a little more realistic.

3. Modifying the knurl appearance:

Right-click the surface of the handle where the knurled appearance was applied and select: **Appearance > Face edit option** (arrow).



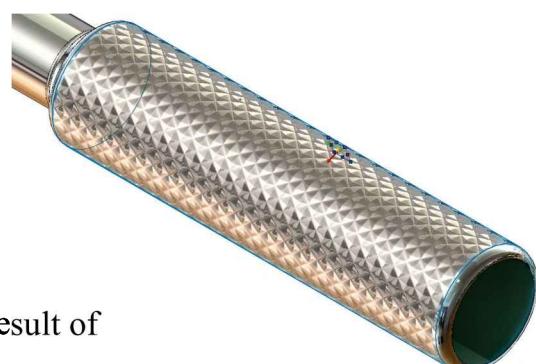
Select the **Advanced** and **Mapping** tabs (arrow).



Select **Automatic** under the mapping drop down selection.

Enter **.125in** for both **Width** and **Height** of the diamonds.

Click **OK**.

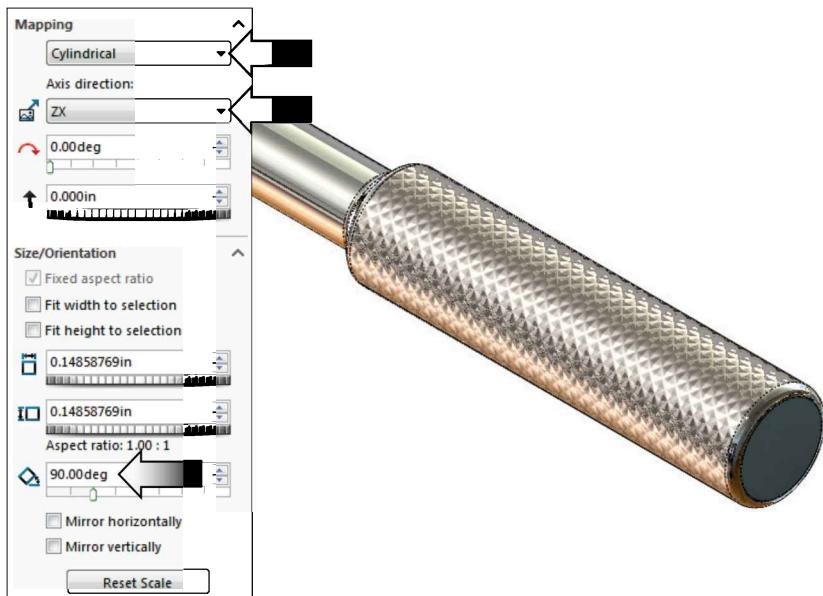


Zoom closer to the handle and inspect the result of the knurls appearance.

Additionally, change the following:

- * Mapping Type: **Cylindrical**
- * Axis Direction: **ZX**
- * Rotation: **90deg.**

Click **OK**.

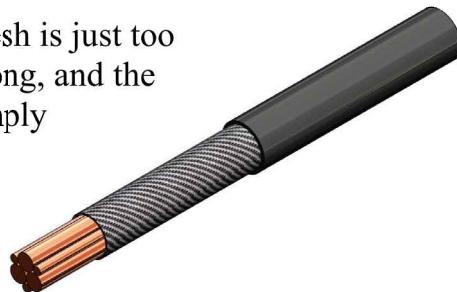


Save and close the document.

Applying Wire Mesh Appearance

Similar to diamond knurls, modeling 3D wire mesh is just too time consuming; the rebuild time would be too long, and the amount of memory needed for the task is just simply not practical.

Appearance once again can offer some acceptable results without sacrificing your computer performance, or the time it takes to create one.

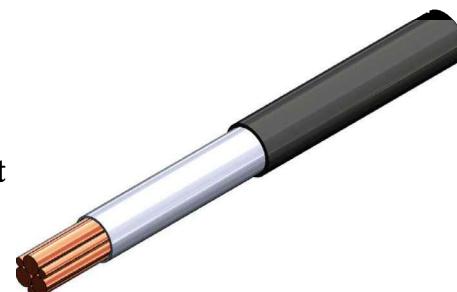


We will take a look at modifying an existing appearance and make it look like a wire meshed cable (pictured)

1. Opening a part document:

Open a part document named: **Wire Mesh.sldprt**

This model has 8 solid bodies and the sleeve in the middle will be used to apply the wire mesh.

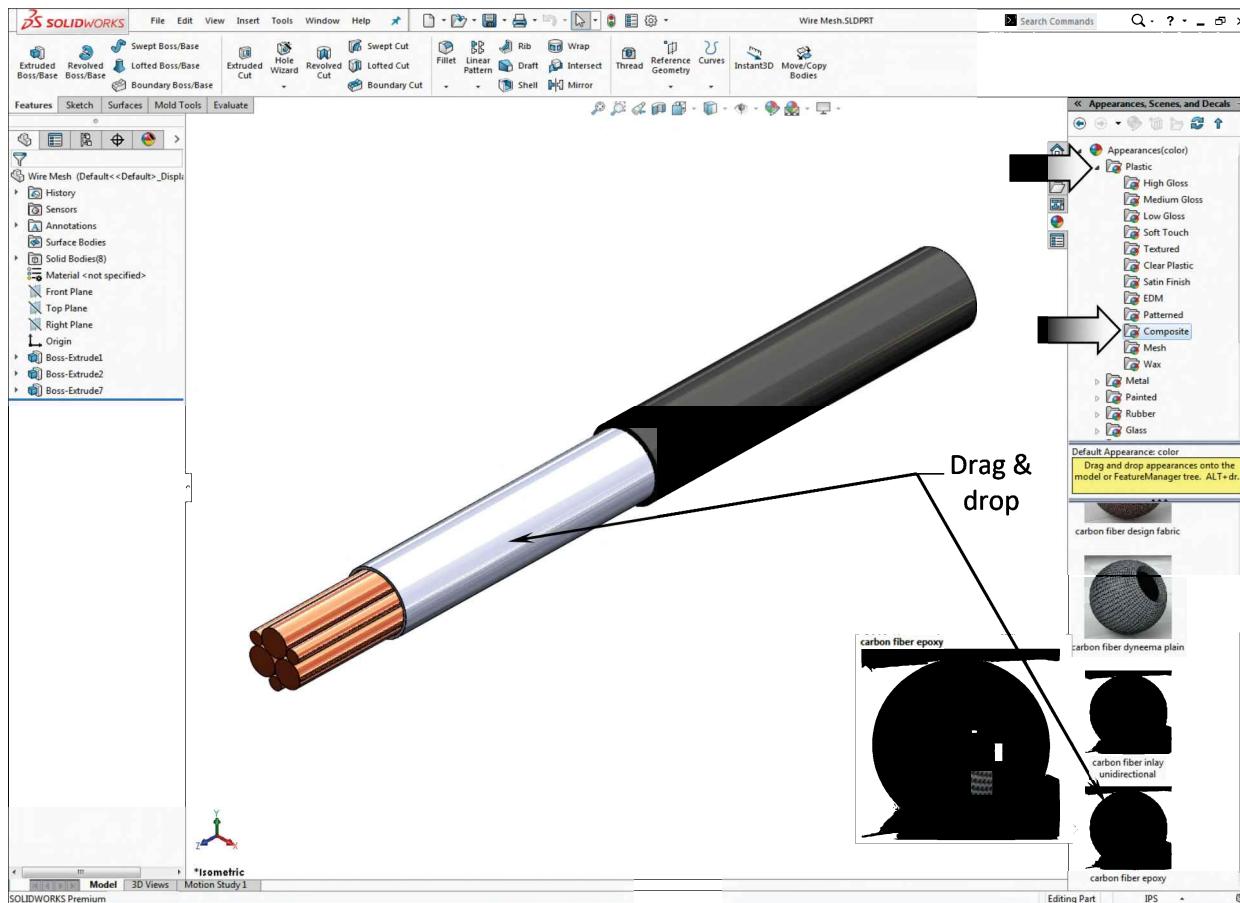


2. Applying the wire mesh:

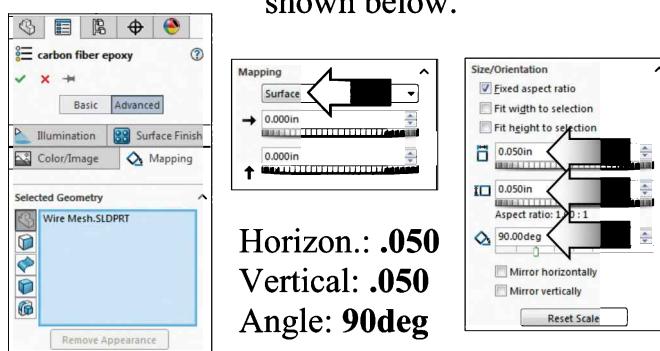
Expand the **Task Pane** on the right side of the screen and click the push pin to lock.

Expand the following 2 folders: **Plastic > Composite** (arrows).

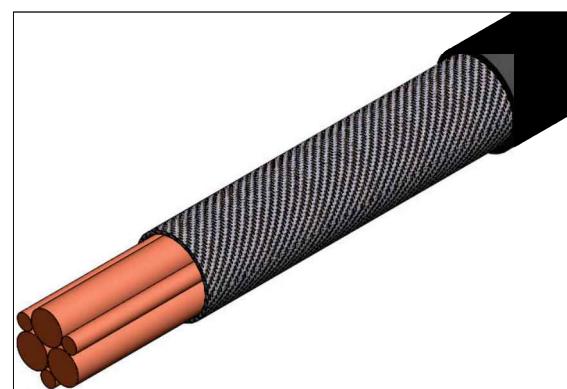
Drag and drop the **Carbon Fiber Epoxy** appearance to the middle sleeve as noted.



Select the **Advanced** and the **Mapping** tabs and set the parameters shown below.



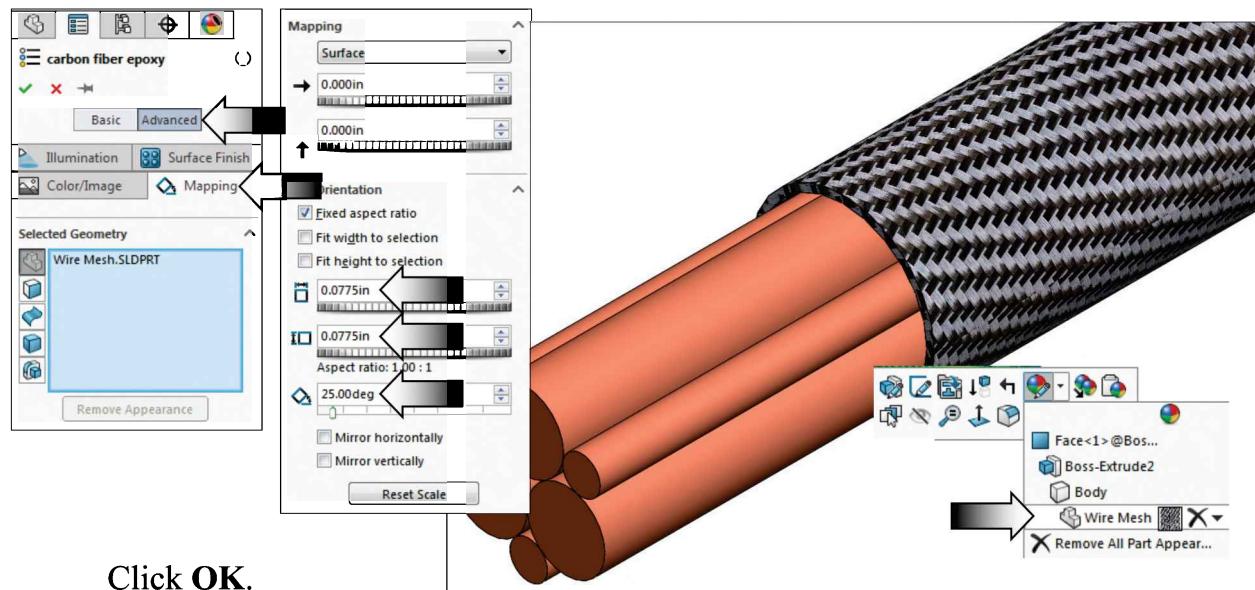
Horizontal: .050
Vertical: .050
Angle: 90deg



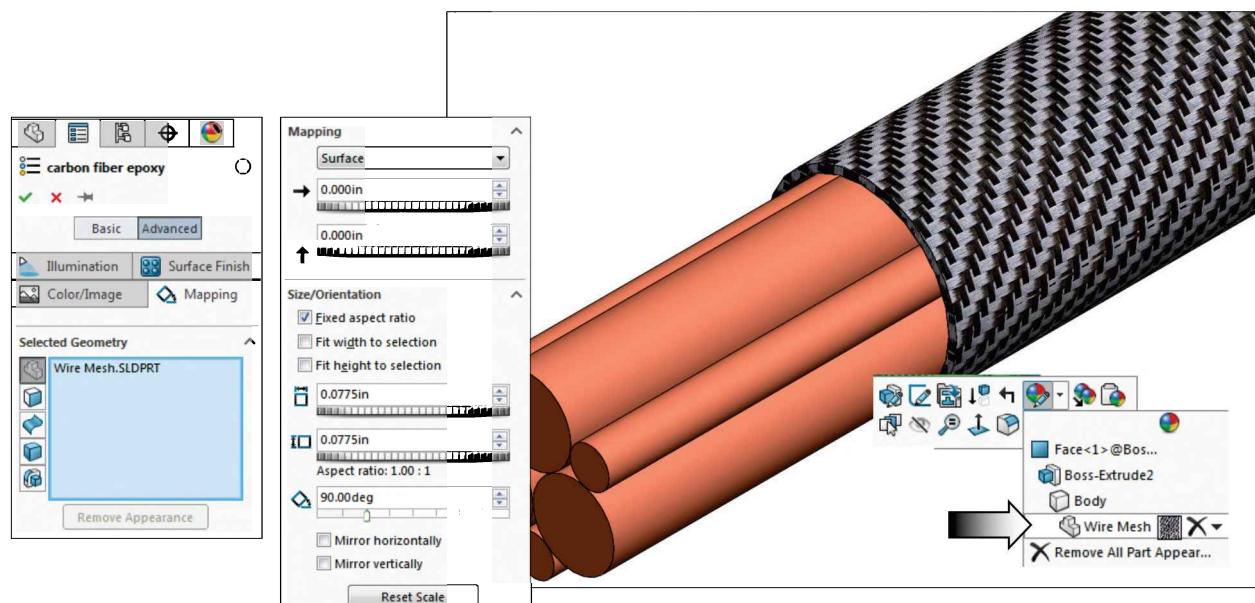
3. Modifying the wire mesh appearance:

Right-click the middle sleeve and select: **Appearance > Edit** (arrow).

Select the **Advanced** and **Mapping** tabs. Set the Mapping type to **Surface**, and set the **Width** and **Height** to **.075in**, and the **Angle** to **25deg**.



Alternatively, change the rotation to **90deg** (arrow) to change the angle of the mesh if desired.



Save and close the document.

Applying the Car-Paint Appearance

An appearance defines the visual properties of a model, including color and texture.

Appearances do not affect physical properties, which are defined by materials.



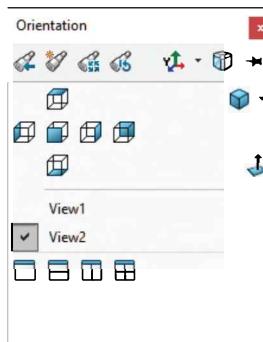
1. Opening an assembly document:

Open an assembly document named:
Concept Helicopter Assembly.sldasm

There are two
Named Views
saved under
the Orientation
dialog box.

Press the **Space-Bar** to see them.

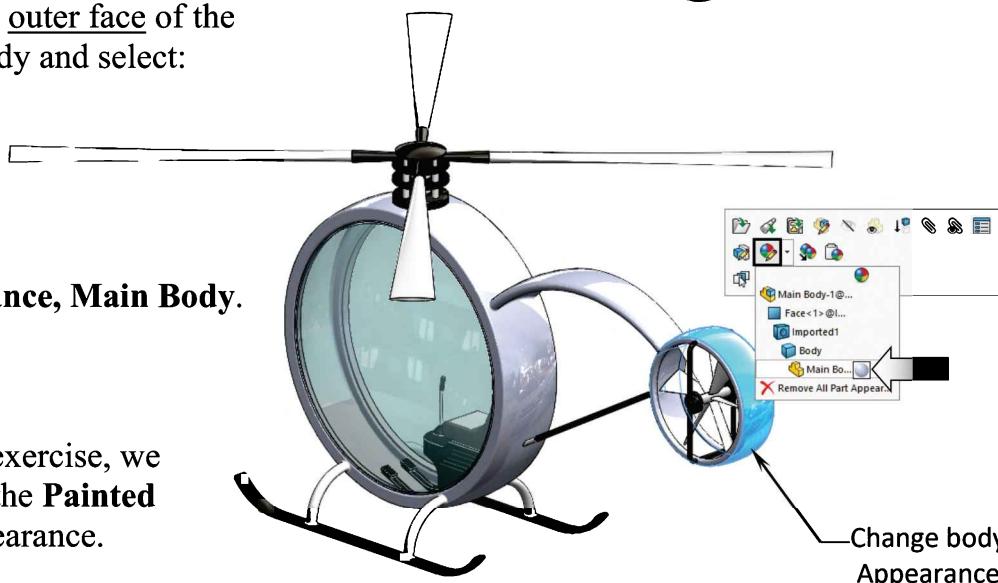
View1 is shown.



2. Changing the appearance of a body:

Click the outer face of the
Main Body and select:

Appearance, Main Body.



For this exercise, we
will use the **Painted
Car** appearance.

You can drag/drop an appearance from the Task Pane directly onto the component as shown in the previous chapter.

This exercise shows us an alternative method to see step by step how an appearance is applied to a model.

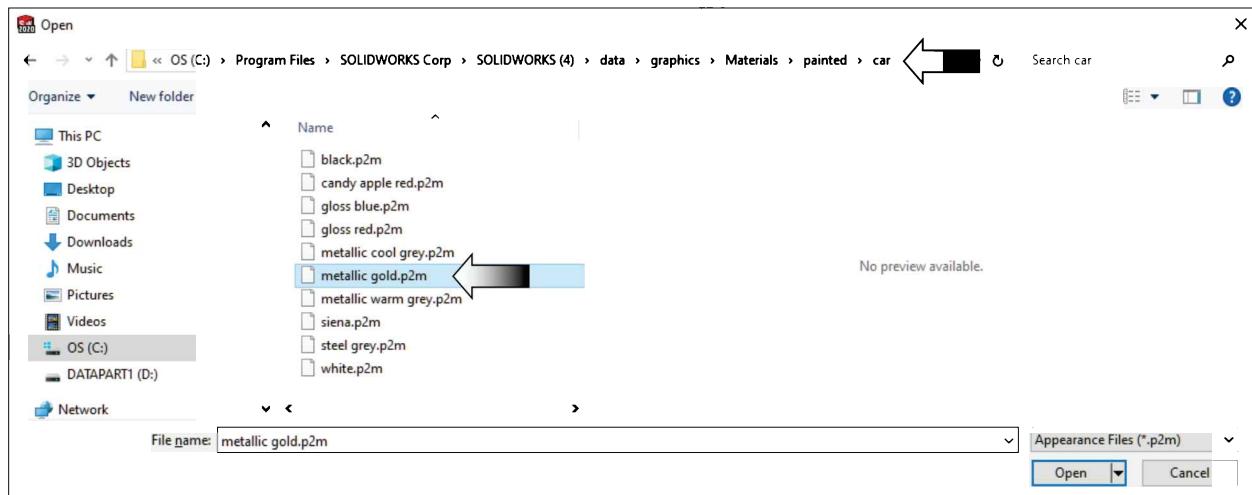
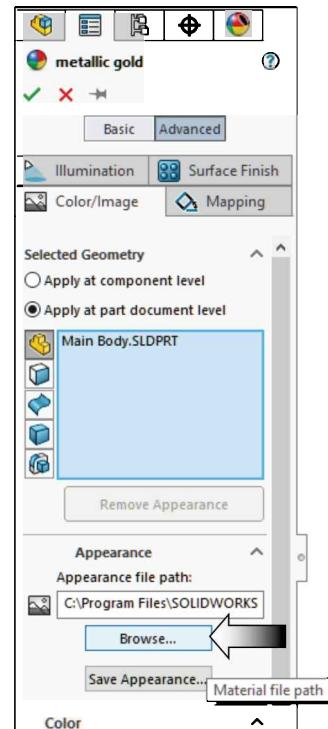
Click the **Browse** button and select the following directories:

C:\Program Files\SOLIDWORKS Corp\SOLIDWORKS (version)\Data\Graphics\Materials\Painted\Car.

Select the **Metallic Gold.p2m** appearance

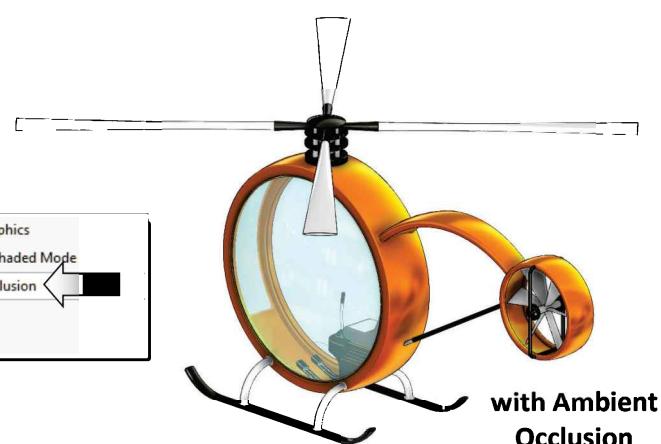
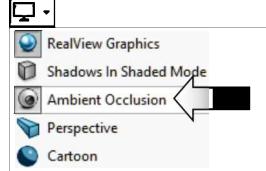


Click **Open**.



The selected appearance is applied to the entire body of the Concept Helicopter.

Ambient Occlusion is a global lighting method that adds realism to models by controlling the attenuation of ambient light due to occluded areas.





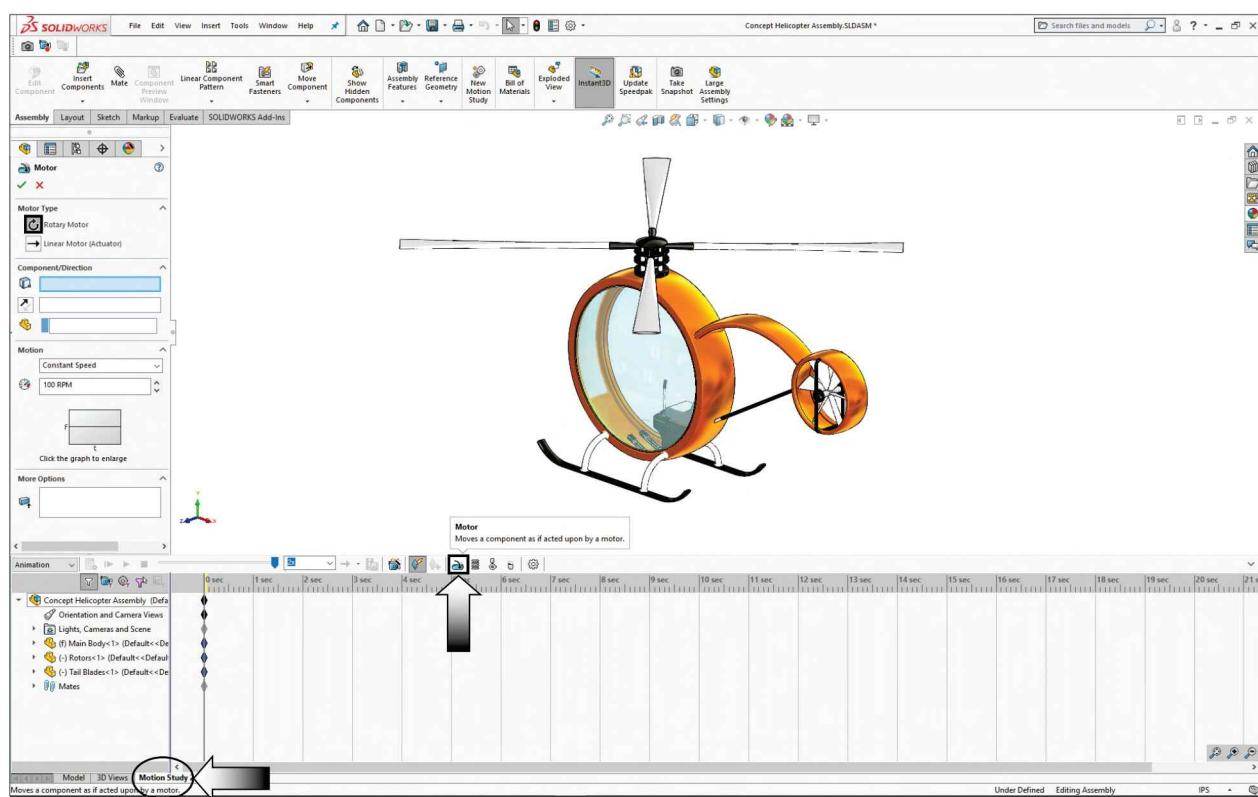
3. Creating an animation:

Select the **Motion Study** tab at the lower left side of the screen.

Click the **Motor Icon** (arrow).

Select the **Rotary Motor** option.

Change face Appearance

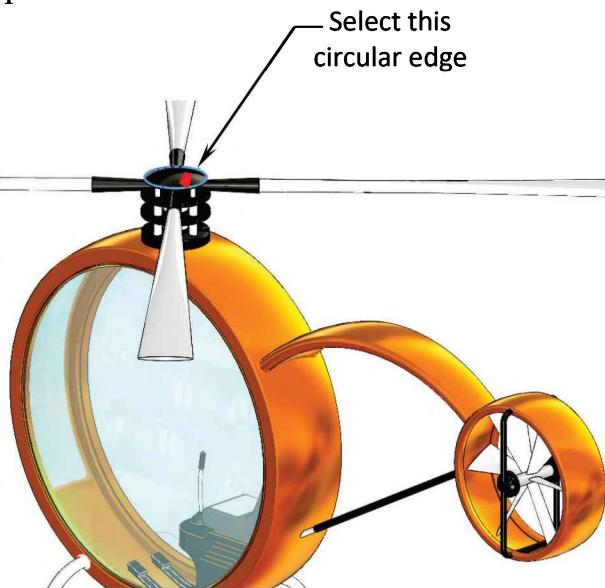
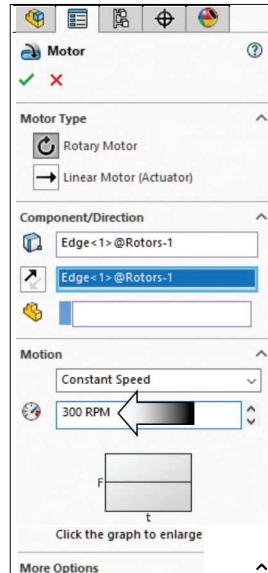


4. Placing the virtual motors:

For virtual motor location, select the upper circular edge as indicated.

A red arrow appears showing the rotate direction; click reverse if needed.

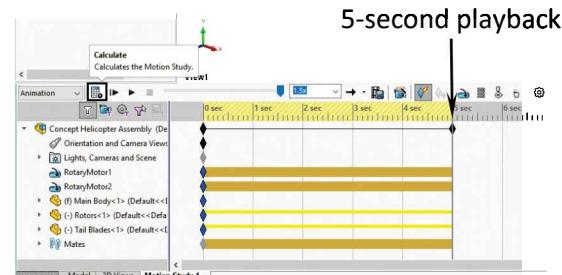
Set the Constant Speed to:
300RPM



Click OK.

A 5-second playback time is automatically applied to the 1st motor. (The time can be changed by dragging the diamond key.)

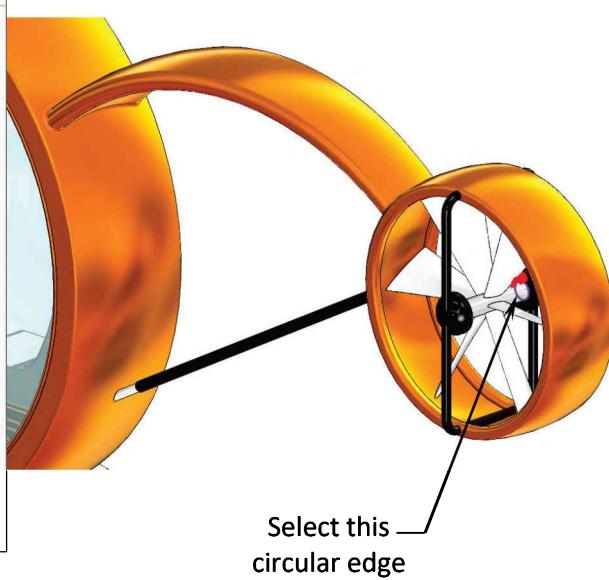
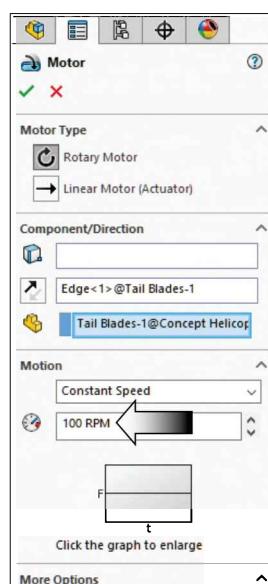
Click the **Calculate** button to view the animation.



Click the **Motor** button again.

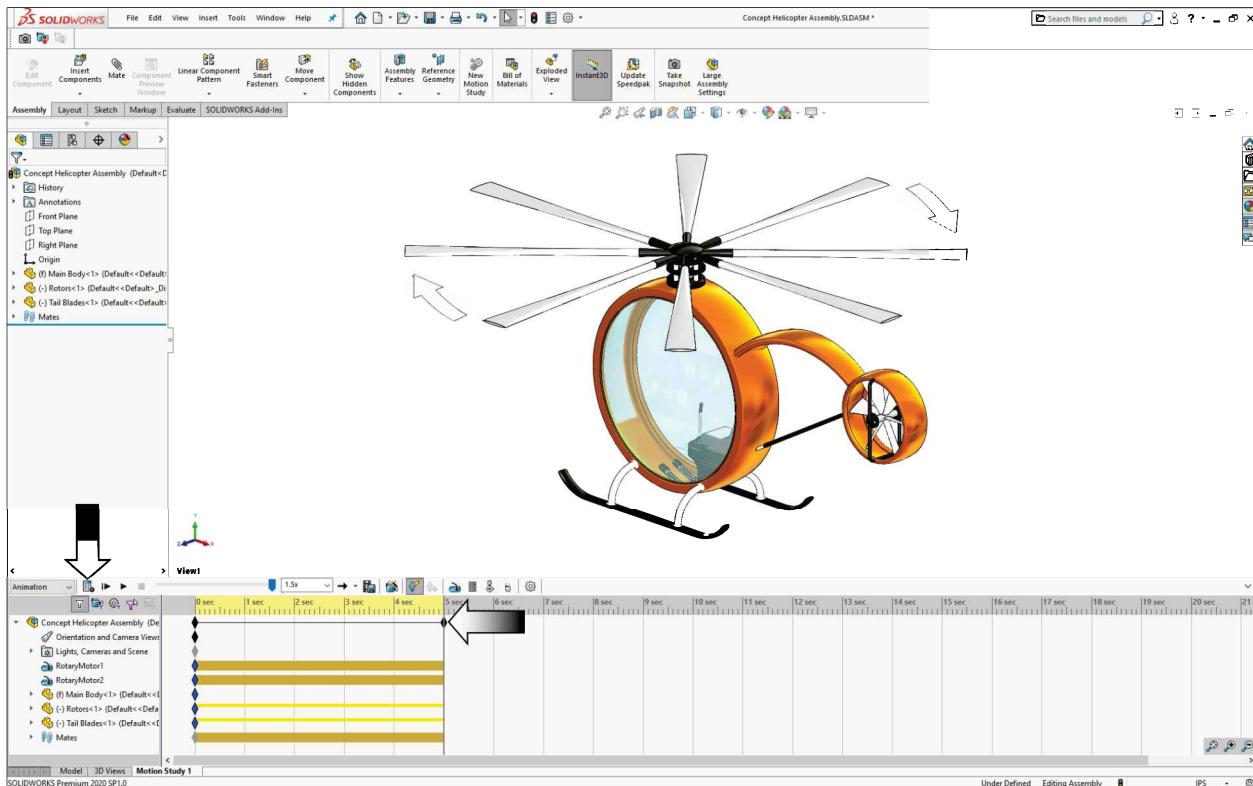
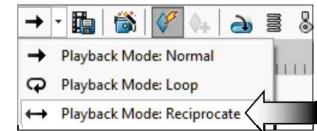
Zoom in on the Tail Blades and select the circular edge as noted to place the 2nd motor.

Click OK.

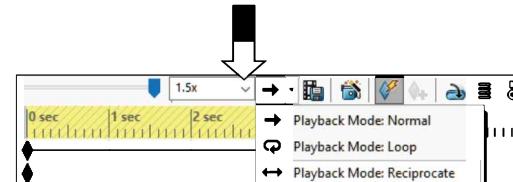


Click **Calculate** to view the rotary motions of the Rotors and the Tail Blades.

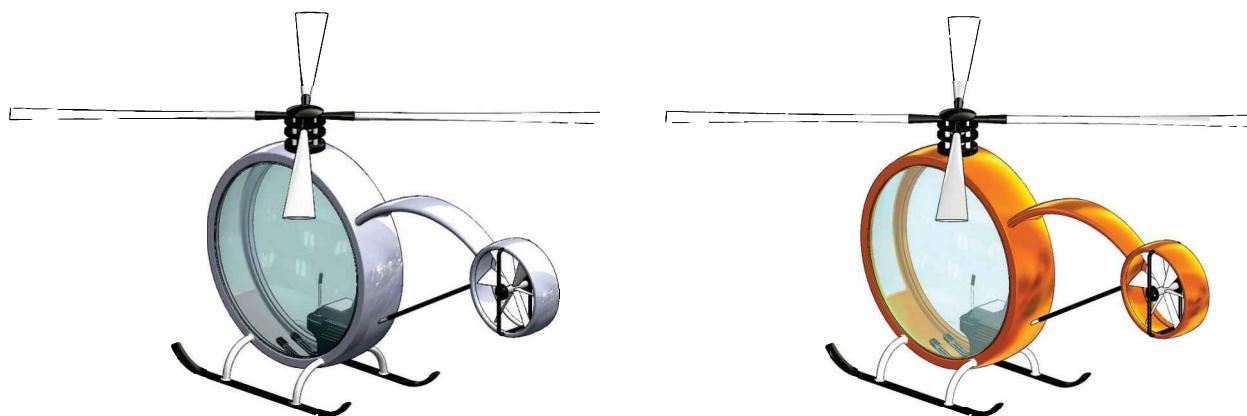
To play back continuously either select the **Loop** or **Reciprocate** options from the drop-down list.



Change the play back speed (arrow) to speed up or to slow down the animation.



Save and close all documents.



Flatten Surfaces

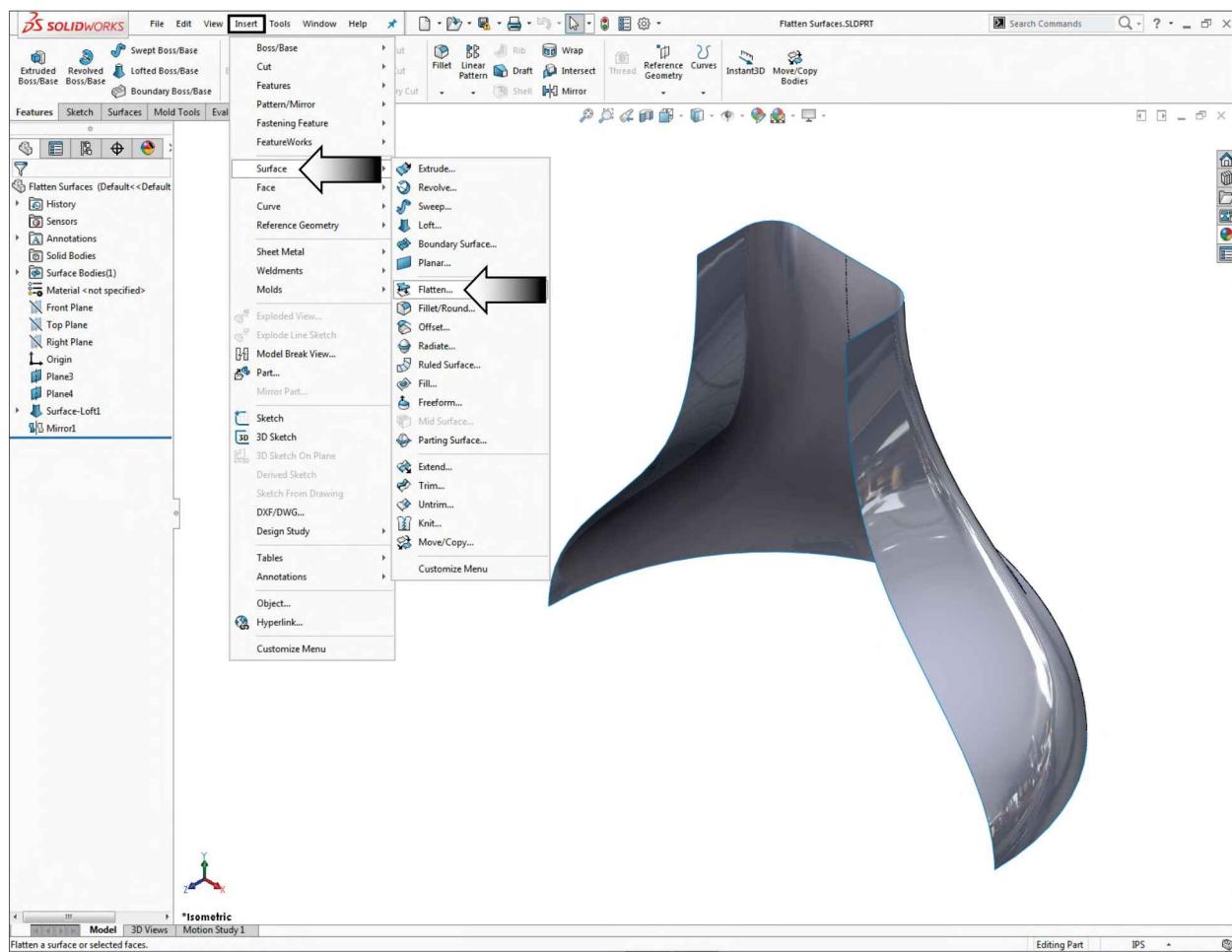
This feature is only available in SOLIDWORKS Premium.

You can flatten surfaces and multi-faced surfaces (such as surfaces that are split into multiple faces). Surfaces that have holes or other internal geometries cut out of the middle cannot be flattened.

1. Opening a part document:

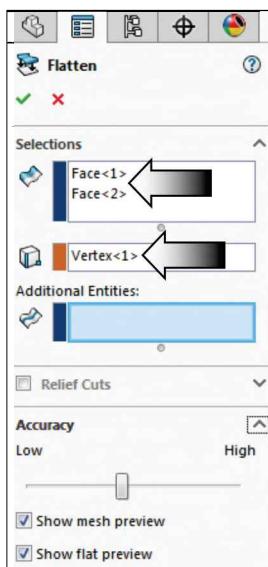
Open a part document named:
Flatten Surfaces.sldprt

Click **Insert > Surface > Flatten** .



2. Flattening a surface:

Click in the Selections box, select the left and the right surfaces of the model.



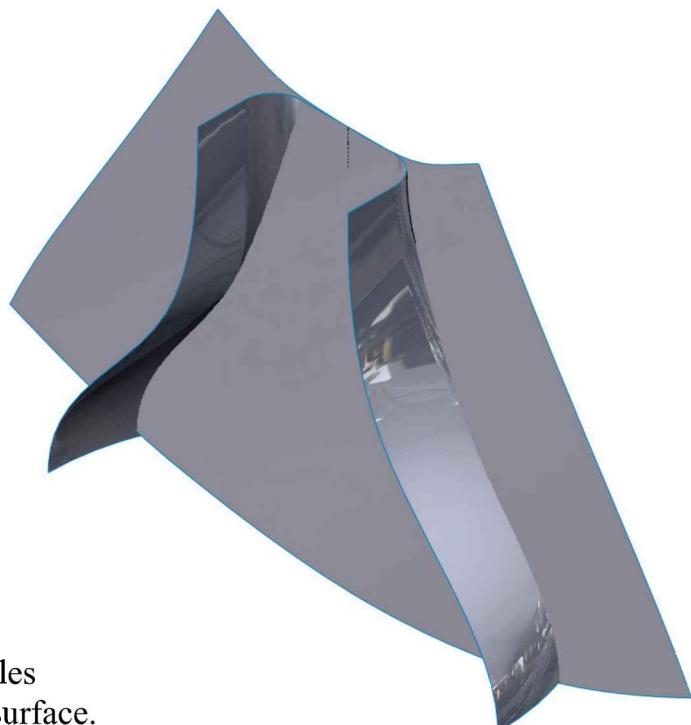
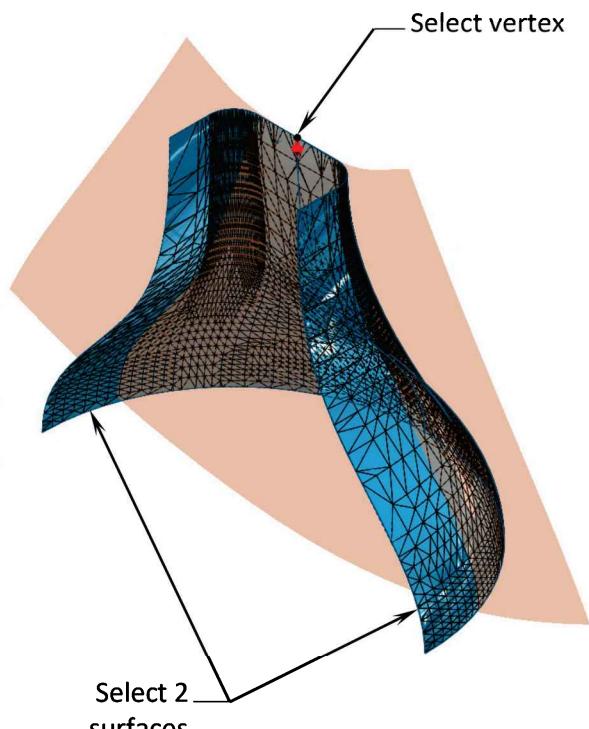
Click in the Vertex to Flatten From box and select the vertex as indicated.

Leave the Accuracy slider at its default location.

Click OK.

The new flatten surface is created over the original surface. These surfaces can be toggled to show or hide.

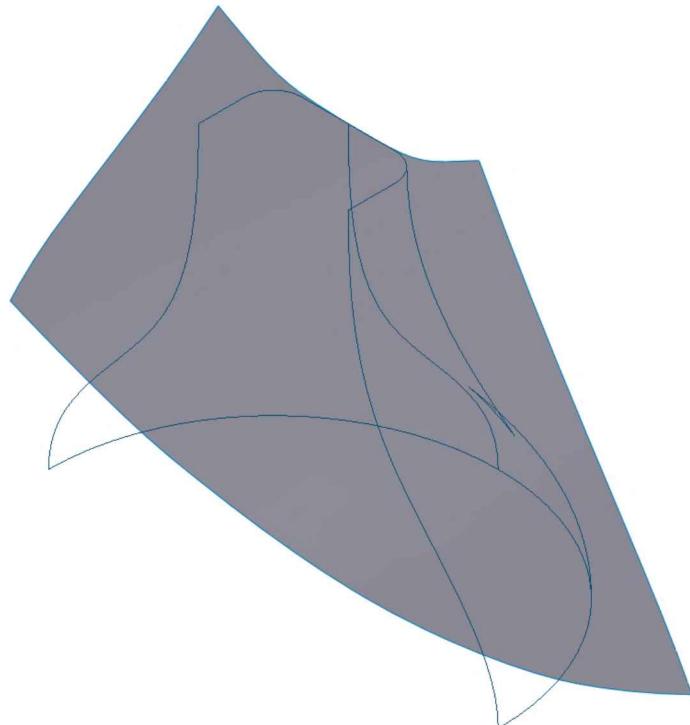
Rotate the model to different angles to verify the result of the flatten surface.



3. Viewing the deformation plot:

To view a deformation plot of the flattened surface, **right-click** the surface and click **Deformation Plot**.

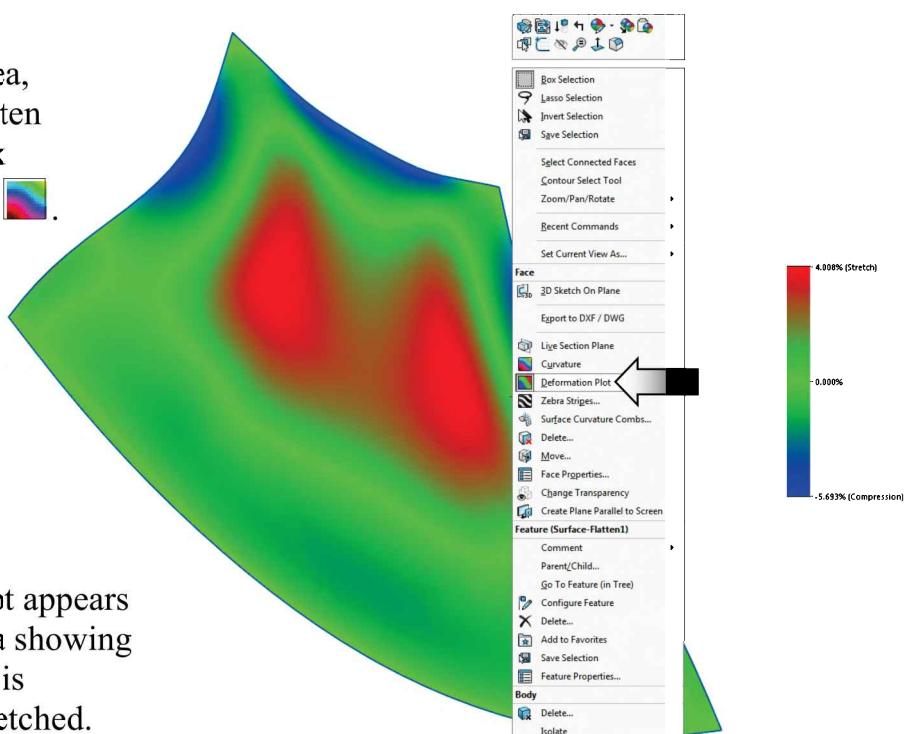
The deformation plot shows the areas on the flattened surface with the highest levels of stretch and compression. You can mouse over the surface to see the percent of deviation at any given point.



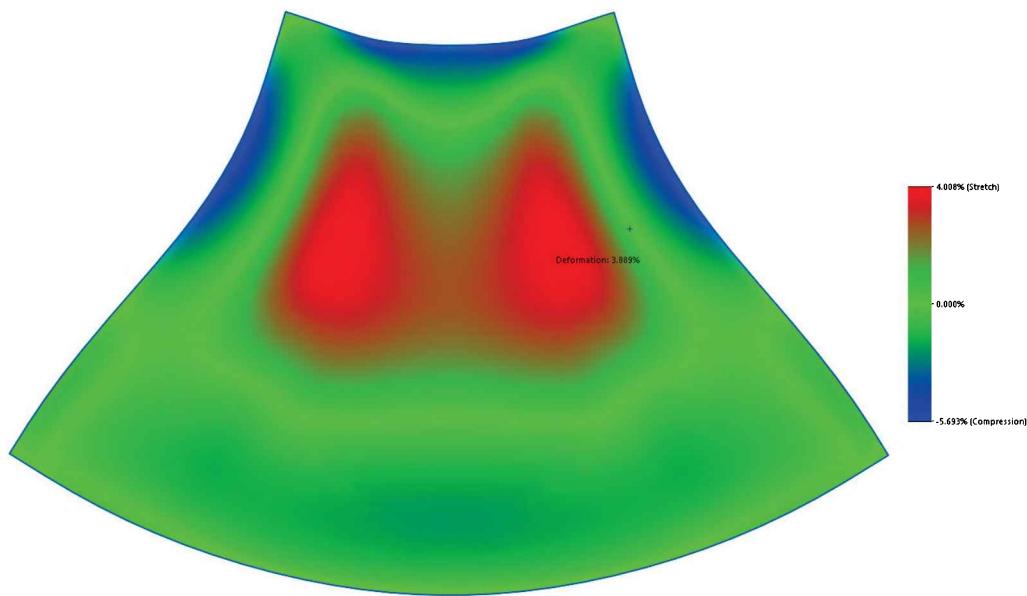
In the graphics area, right-click the flatten surfaces, and click

Deformation Plot .

A deformation plot appears in the graphic area showing where the surface is compressed or stretched.



Change to the front orientation and mouse over one of the red areas to see the percentage of the deformation (stretch).



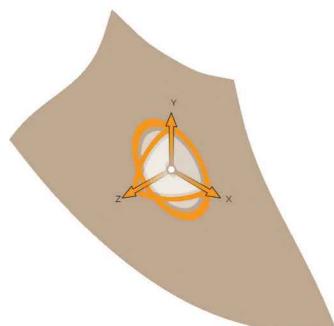
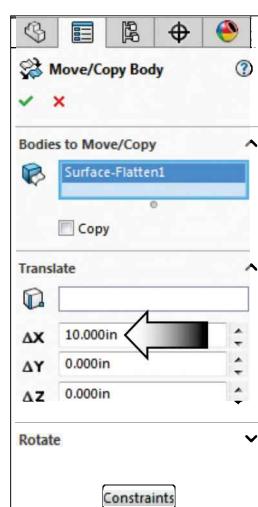
4. Moving the surfaces:

Click **Insert, Surface, Move/Copy**.

In the Translate section, select the **Flatten Surface** to move.

Enter **10.00in** in the Delta X box.

Click **OK**.

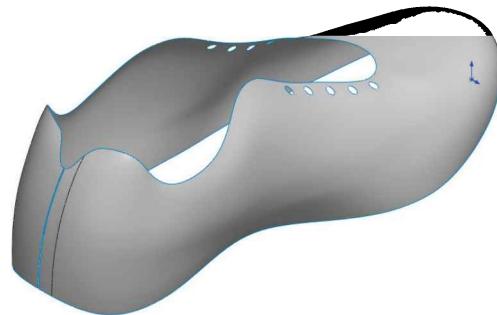


Save and close the document.

Exercise: Flattening a Shoe Sole

1. Opening a part document:

Open a Parasolid document named:
Shoe Sole.x_b



Click **NO** to close the Import Diagnostic dialog box.



2. Flattening a surface:

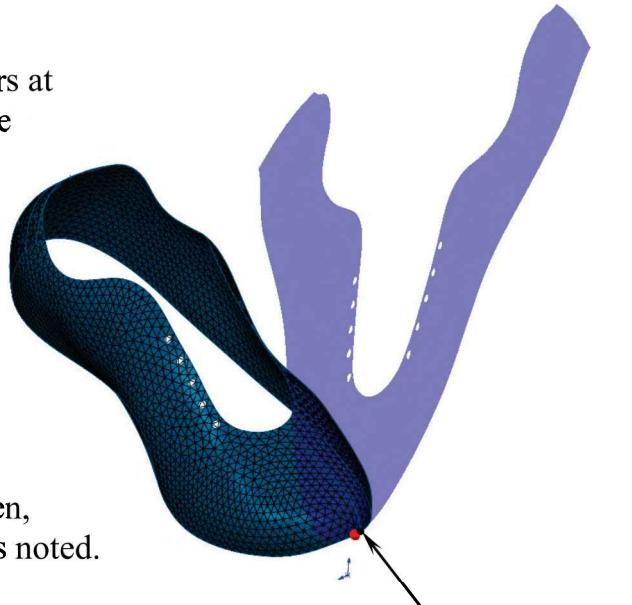
Click **Insert, Surface, Flatten**

For Surface to Flatten, select the **3 surfaces** of the entire sole.

Press **F5** to display the Selection Filters at the bottom left of the screen. Select the **Filter Vertices** button (arrow).



For Vertex to Flatten, select the **Vertex** as noted.

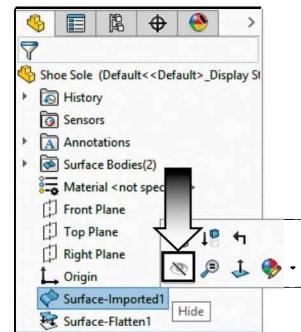


Enable the **Show Mesh Preview** and **Show Flat Preview** checkboxes.

Leave the Accuracy slider at its default value and click **OK**.

Click-off the **Filter Vertices** button and press **F5** again to hide the Selection Filters toolbar.

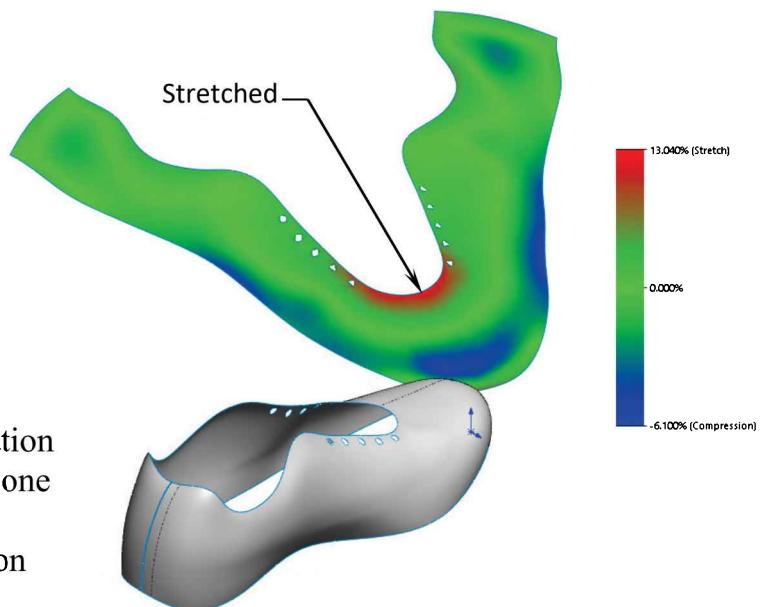
Click the **Surface-Imported1** on the FeatureManager tree and select **Hide** (arrow).



3. Viewing the deformation plot:

Right-click the surface and select **Deformation Plot**.

A deformation plot appears in the graphic area showing where the surface is compressed or stretched.



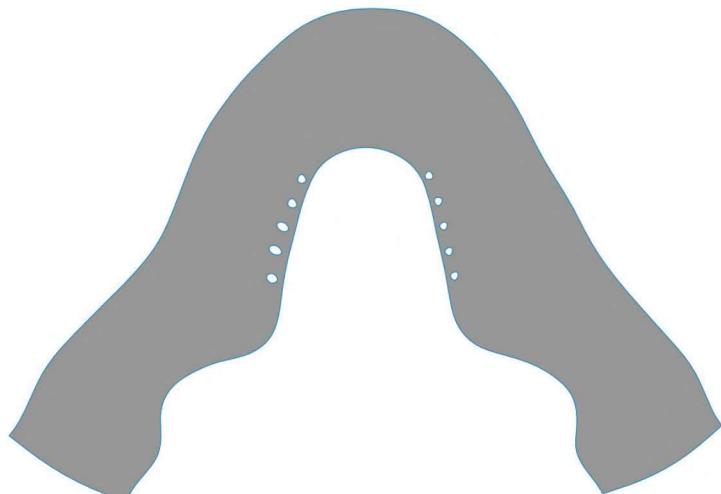
Change to the **Front** orientation (Control+1) and hover over one of the red areas to see the percentage of the deformation (stretch).

4. Saving your work:

Click **File > Save As**.

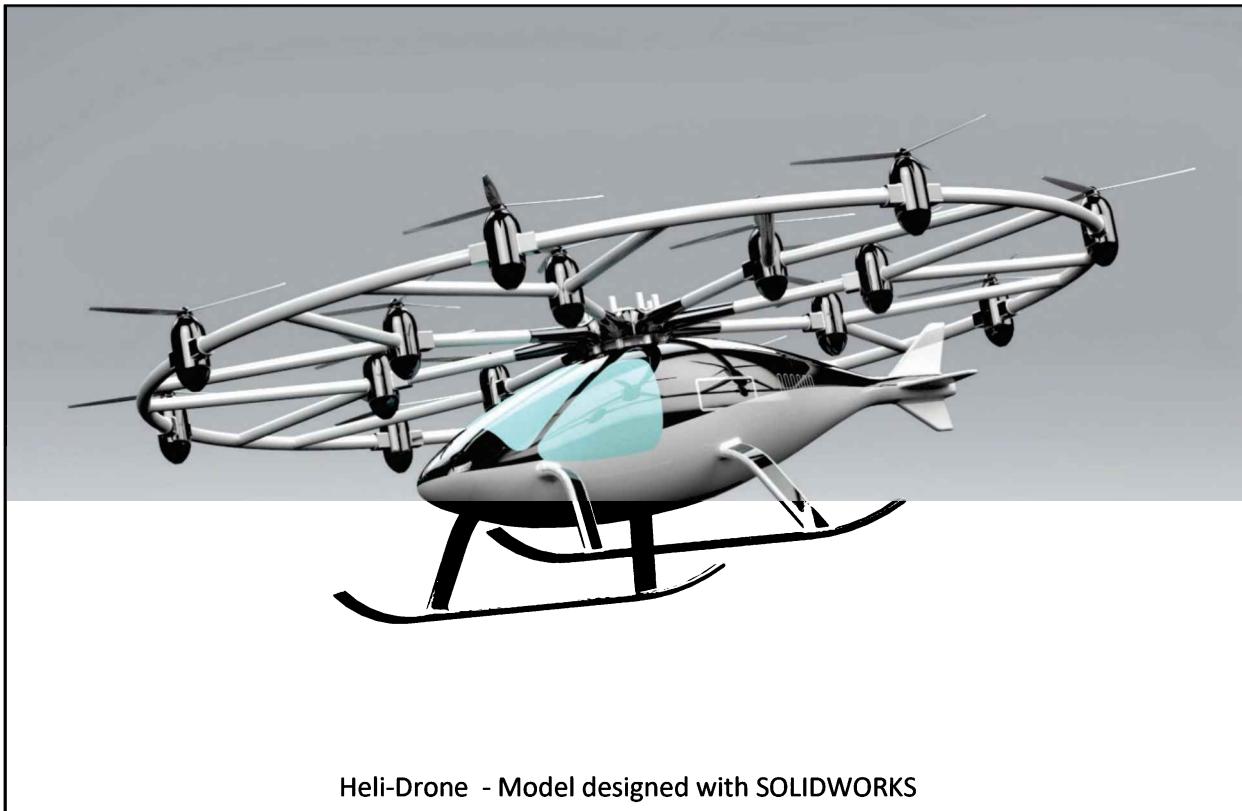
Enter **Shoe Sole Completed** for the file name.

Press **Save**.

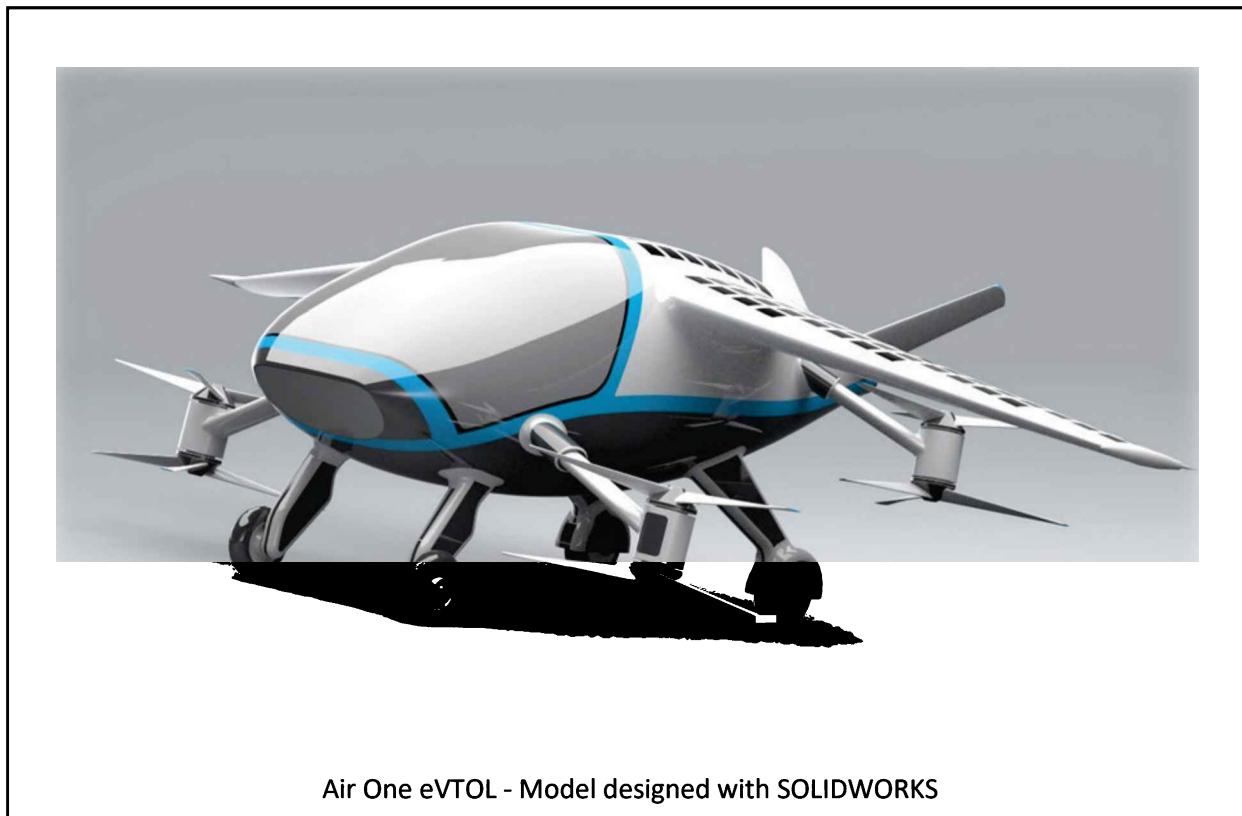


Model Library

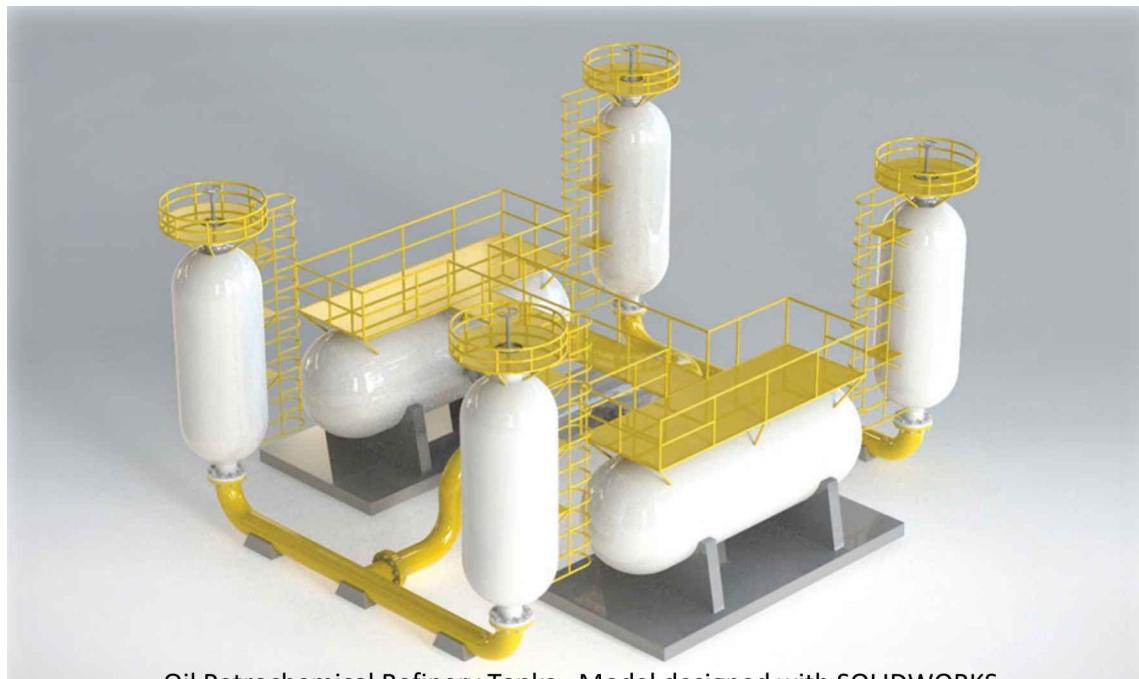
All models designed using SOLIDWORKS



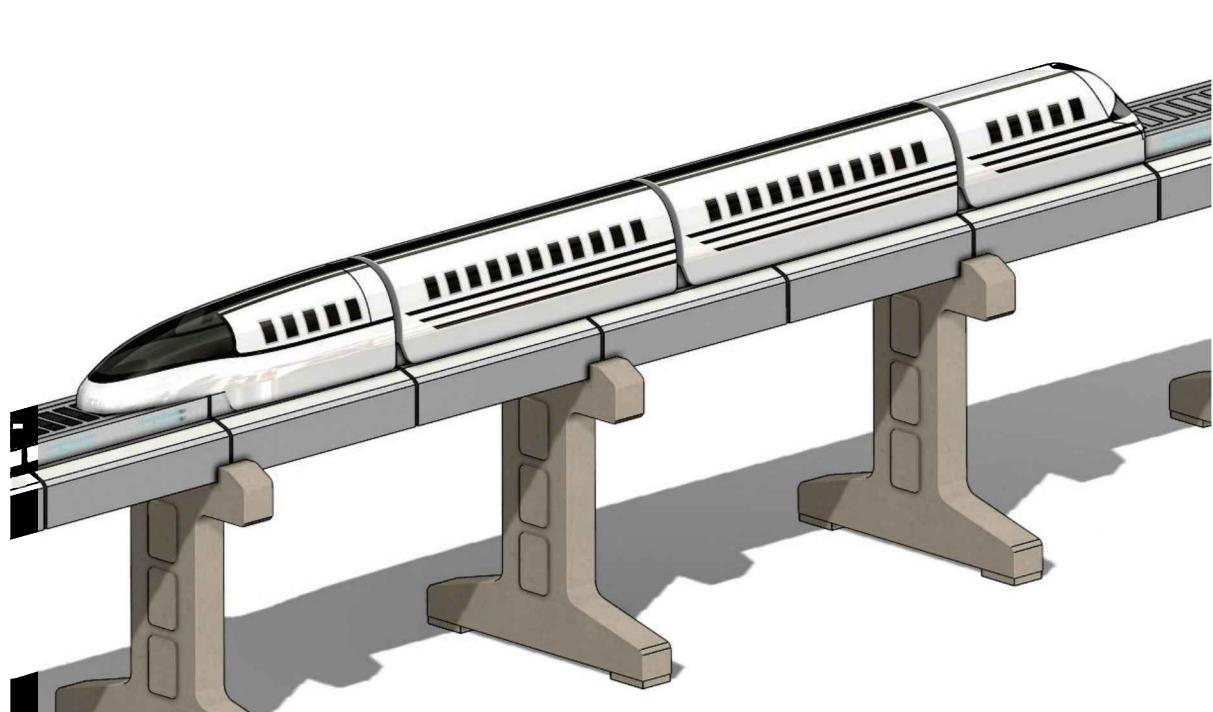
Heli-Drone - Model designed with SOLIDWORKS



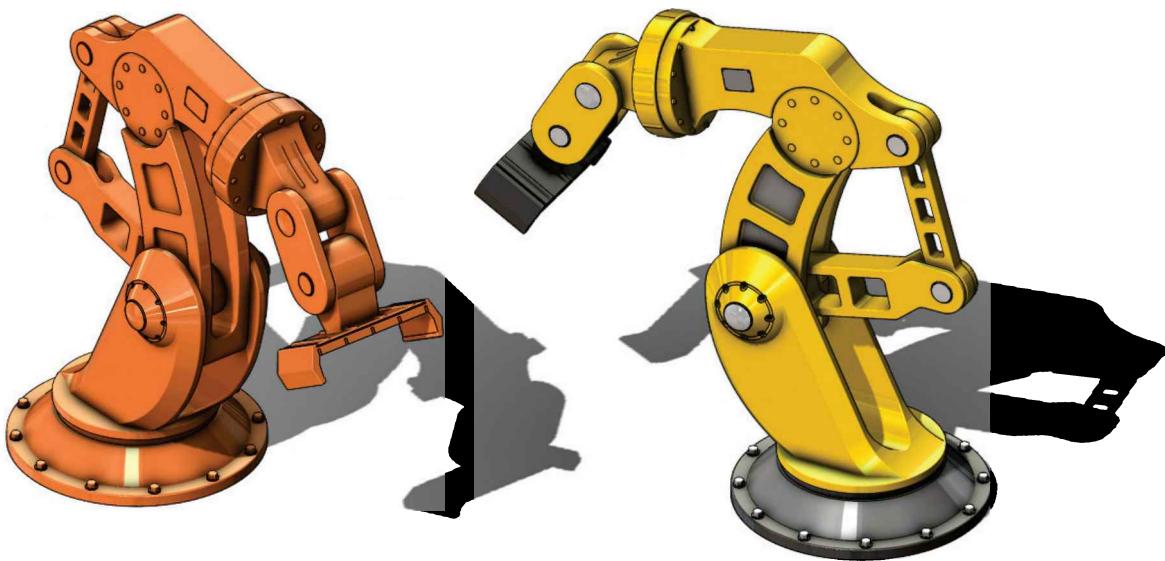
Air One eVTOL - Model designed with SOLIDWORKS



Oil Petrochemical Refinery Tanks - Model designed with SOLIDWORKS



Bullet Train - Model designed with SOLIDWORKS



Pick & Place Robots - Model designed with SOLIDWORKS



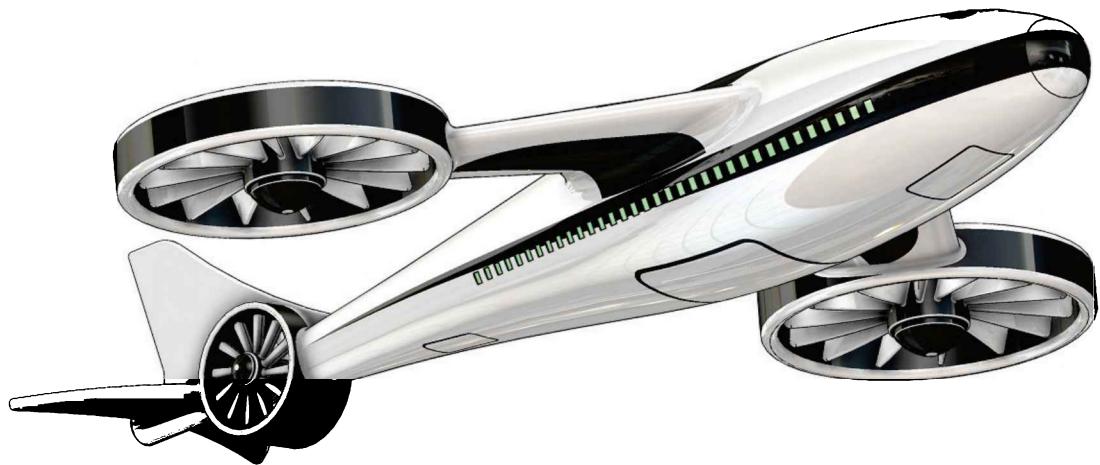
Spaceship - Model designed with SOLIDWORKS



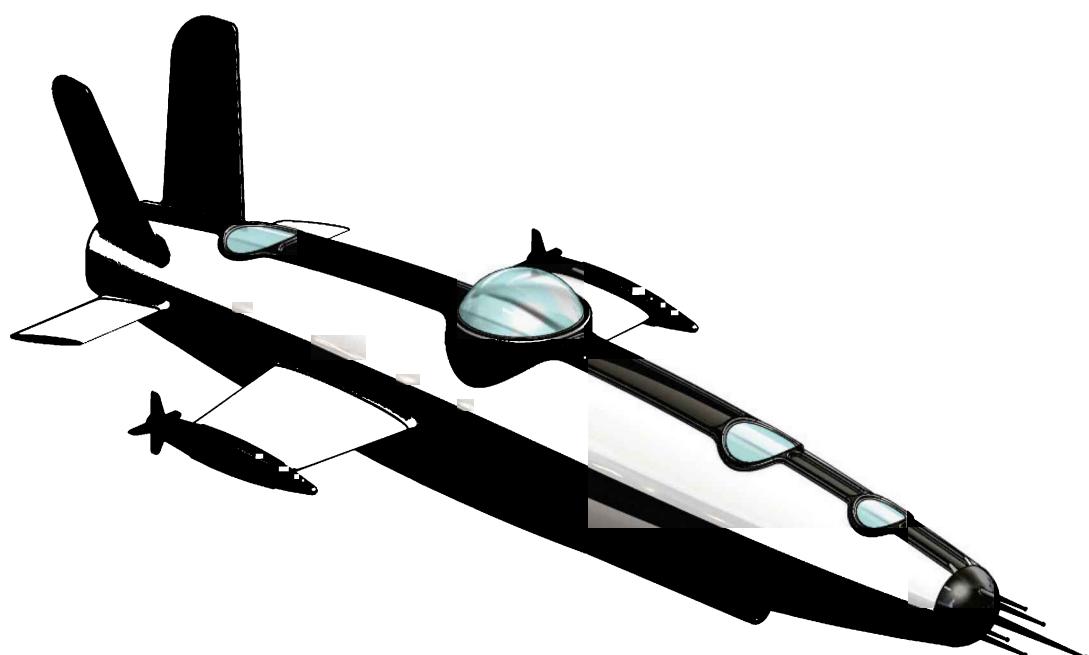
Airship - Model designed with SOLIDWORKS



Spaceship - Model designed with SOLIDWORKS



Futuristic Planes - Model designed with SOLIDWORKS



Modern Submarine - Model designed with SOLIDWORKS



Toy Car Designs - Model designed with SOLIDWORKS



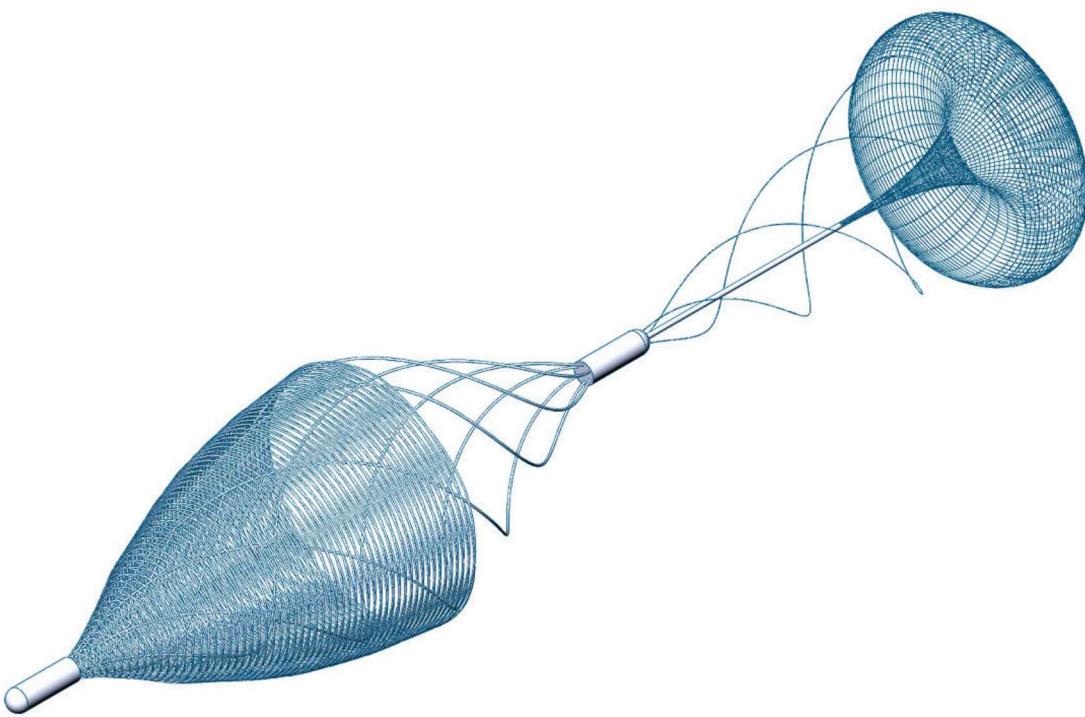
Water Bike - Model designed with SOLIDWORKS



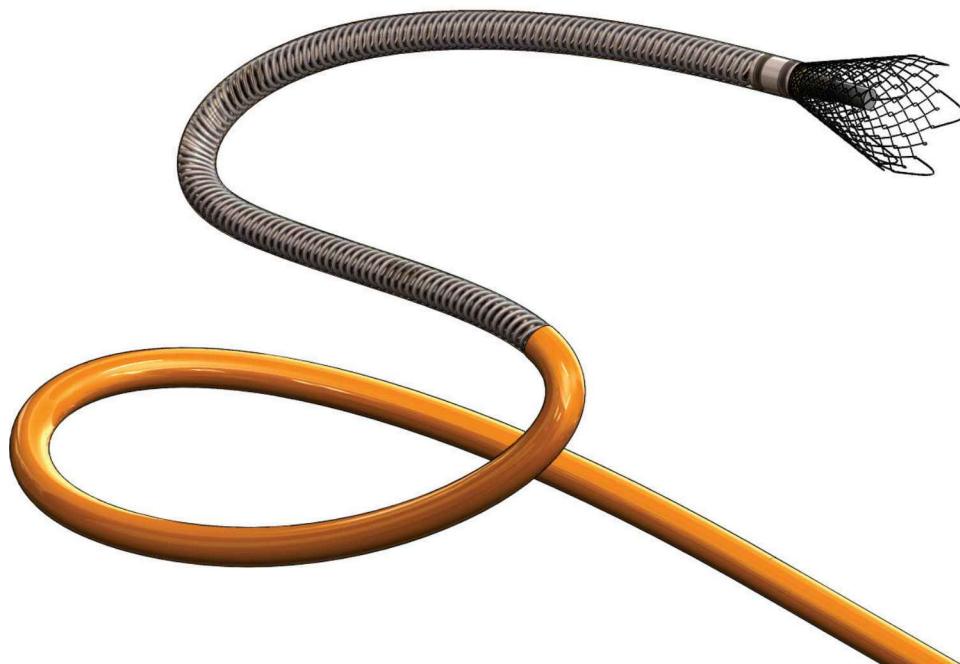
Airbus Popup - Model designed with SOLIDWORKS



Airbus Pod - Model designed with SOLIDWORKS



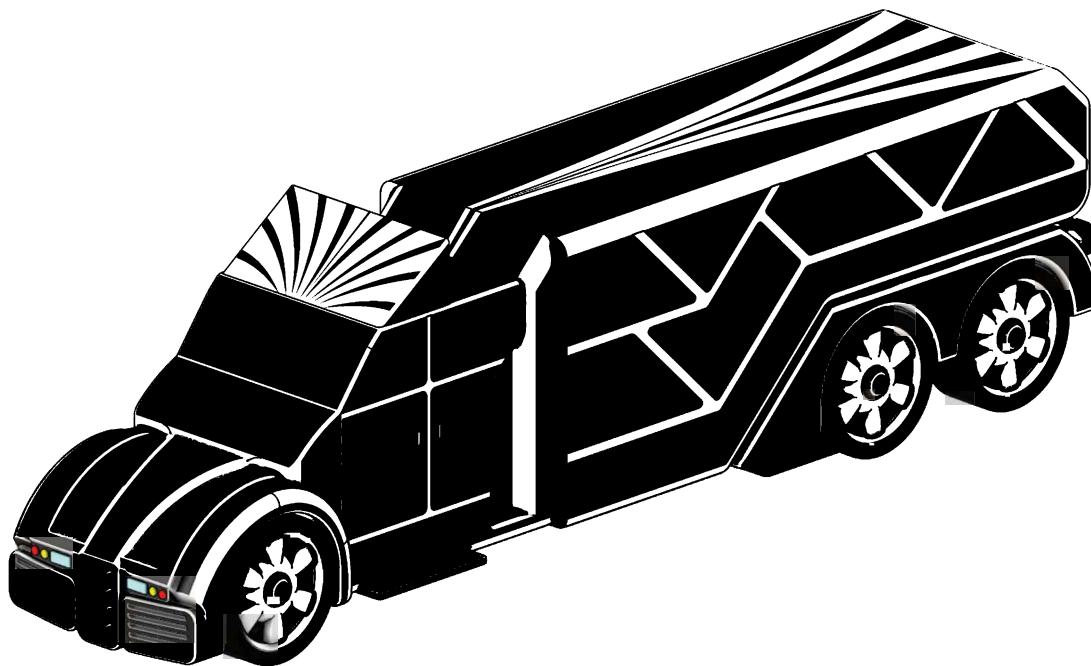
Artery Debris Collector - Model designed with SOLIDWORKS



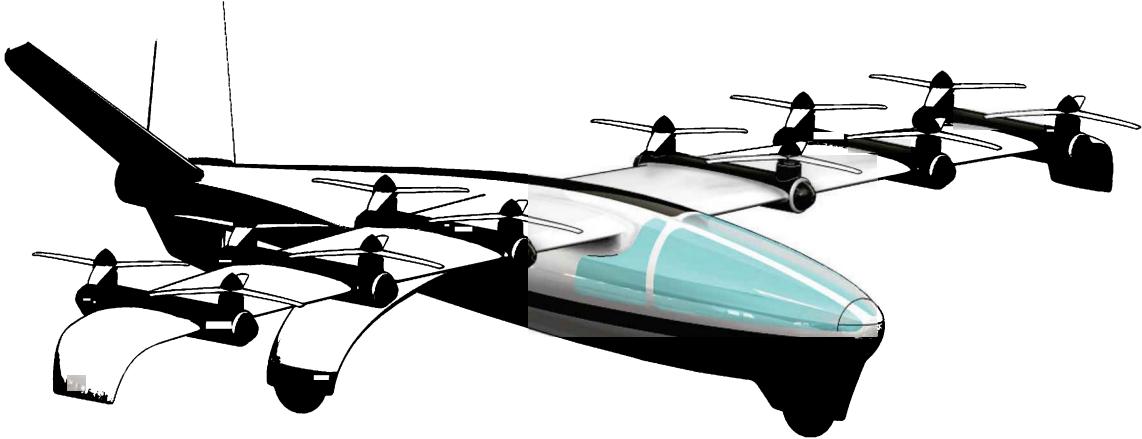
Stent Deployment - Model designed with SOLIDWORKS



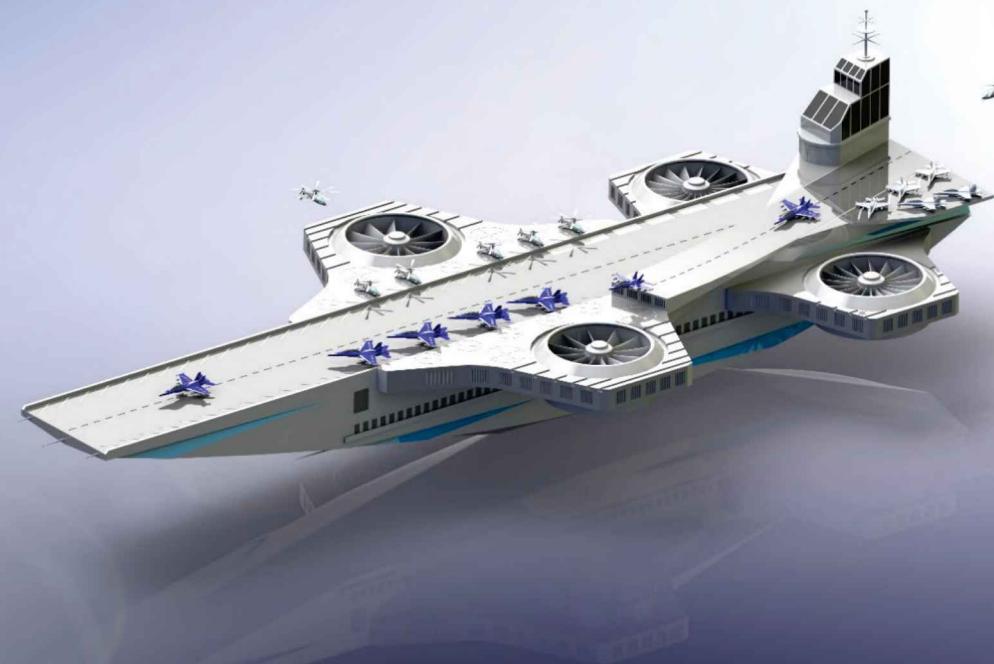
Non-Planar Parting Line Molds - Model designed with SOLIDWORKS 2024



Modern Big Rig - Model designed with SOLIDWORKS 2024



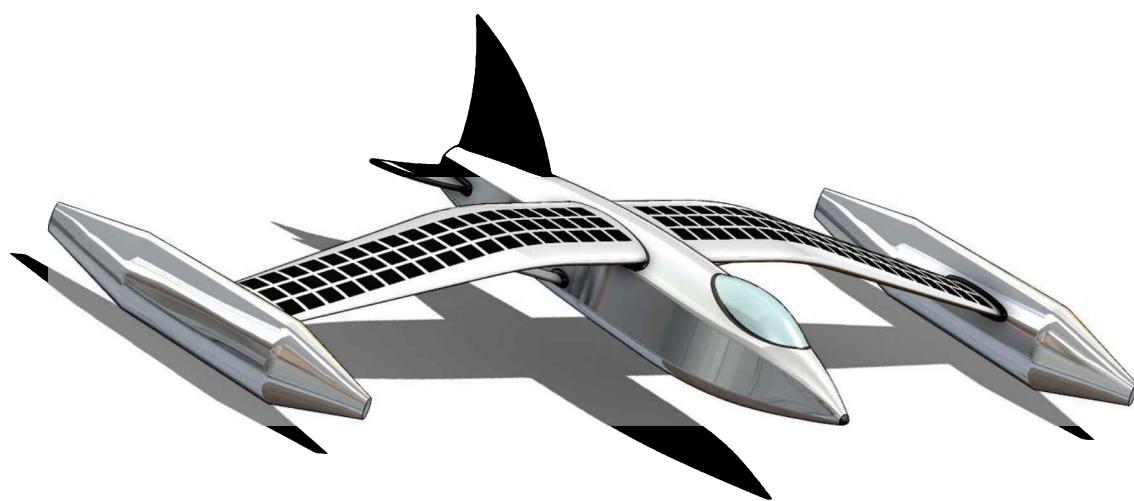
Archer Electric Taxi Aircraft - Model designed with SOLIDWORKS 2024



Futuristic Aircraft Carrier - Model designed with SOLIDWORKS 2024



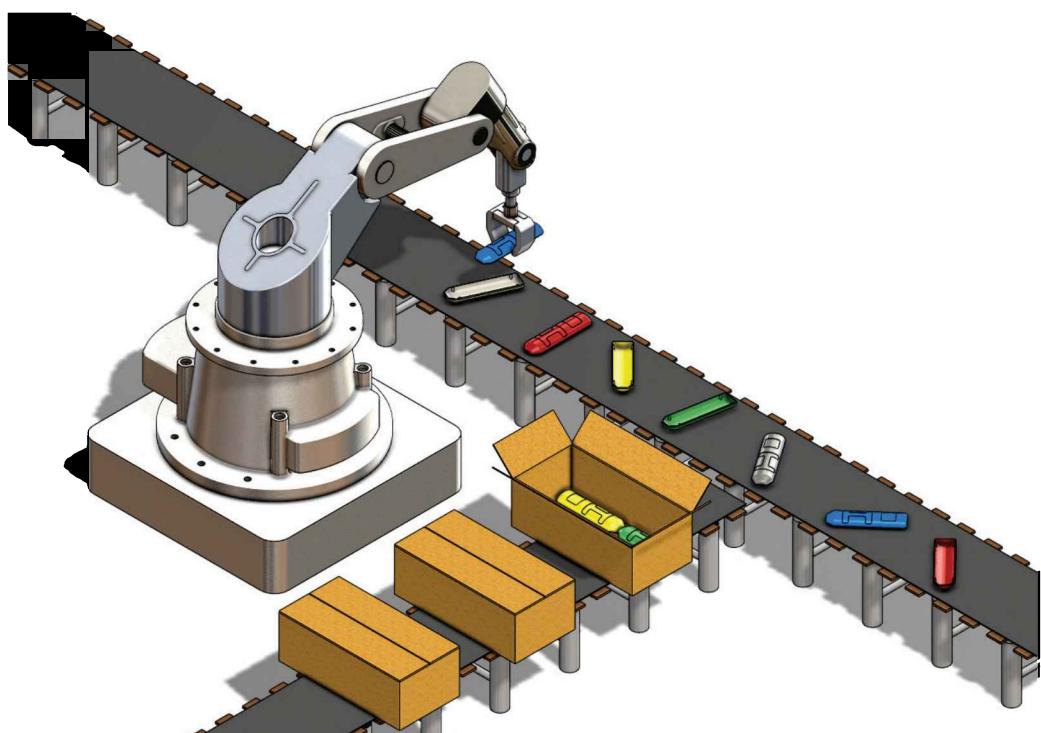
Helicopter Concept - Model designed with SOLIDWORKS 2024



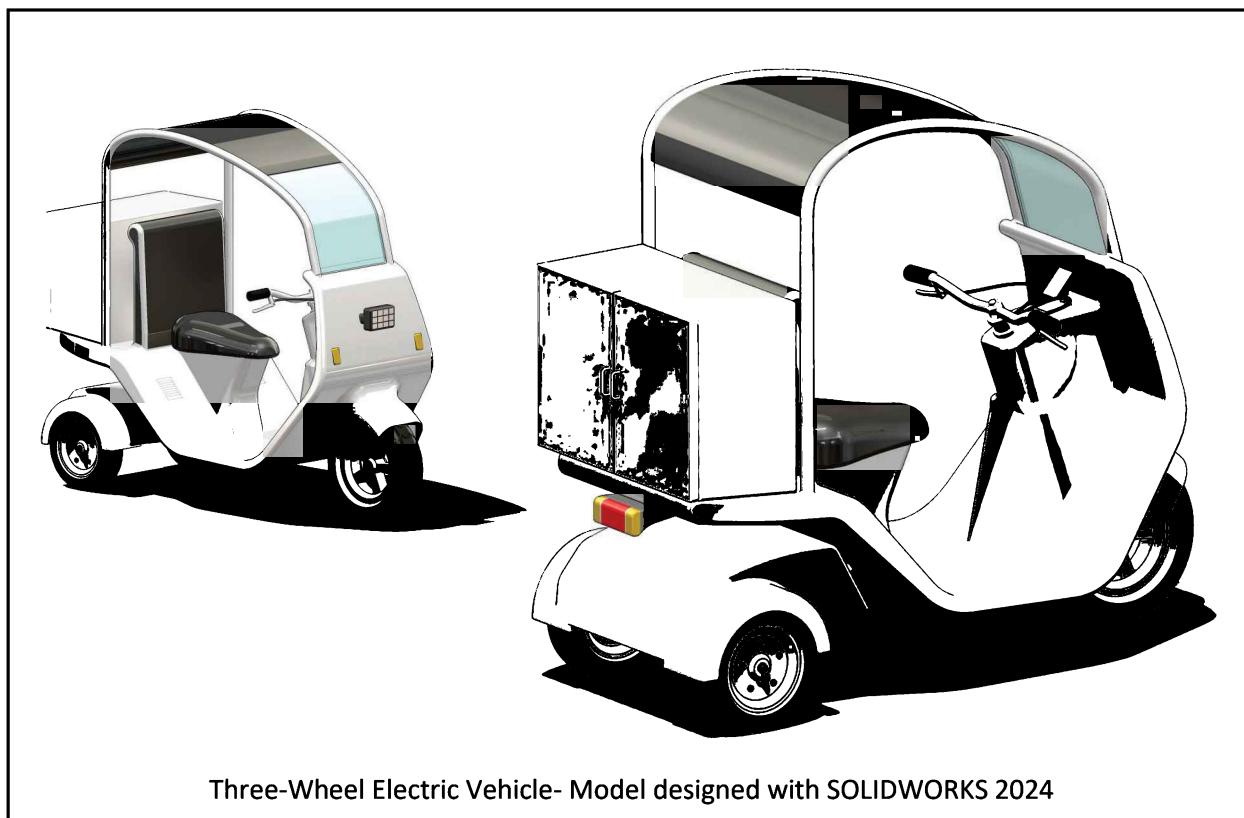
Solar Boat Concept - Model designed with SOLIDWORKS 2024

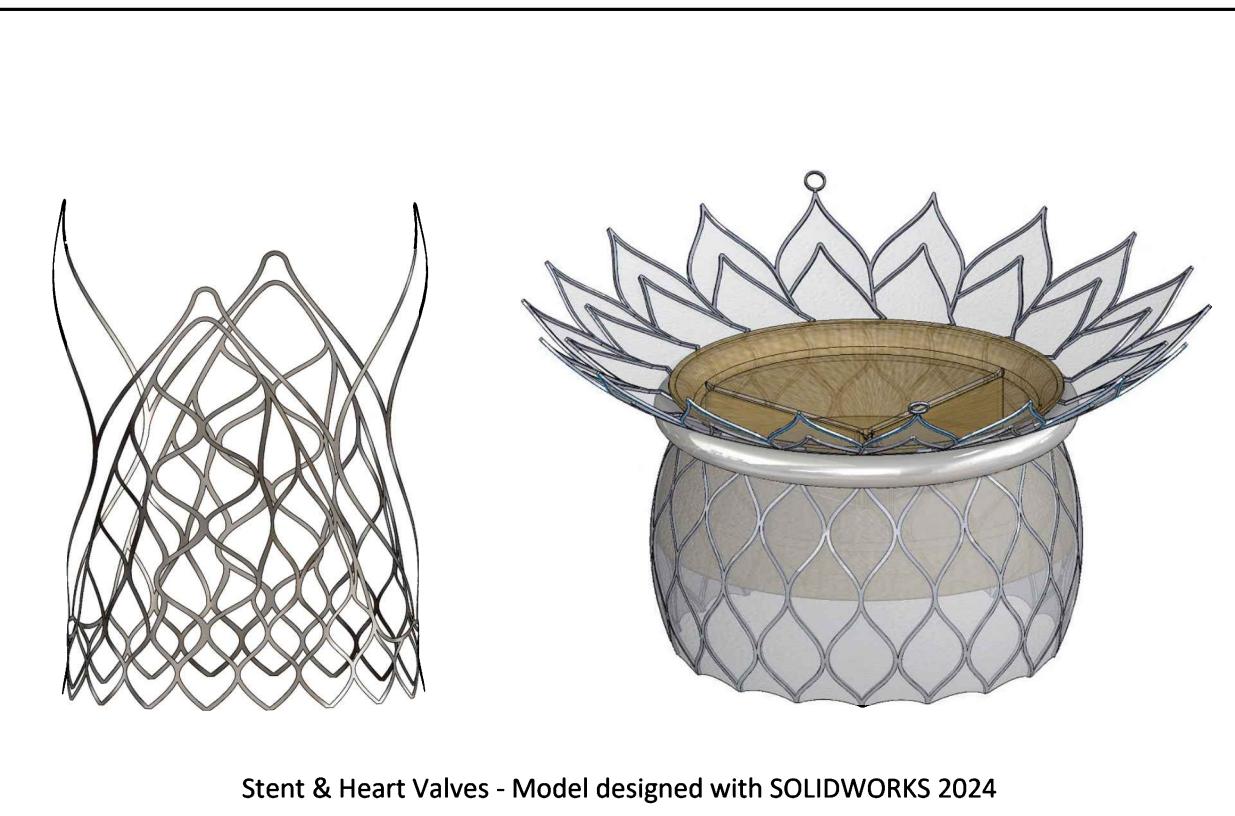
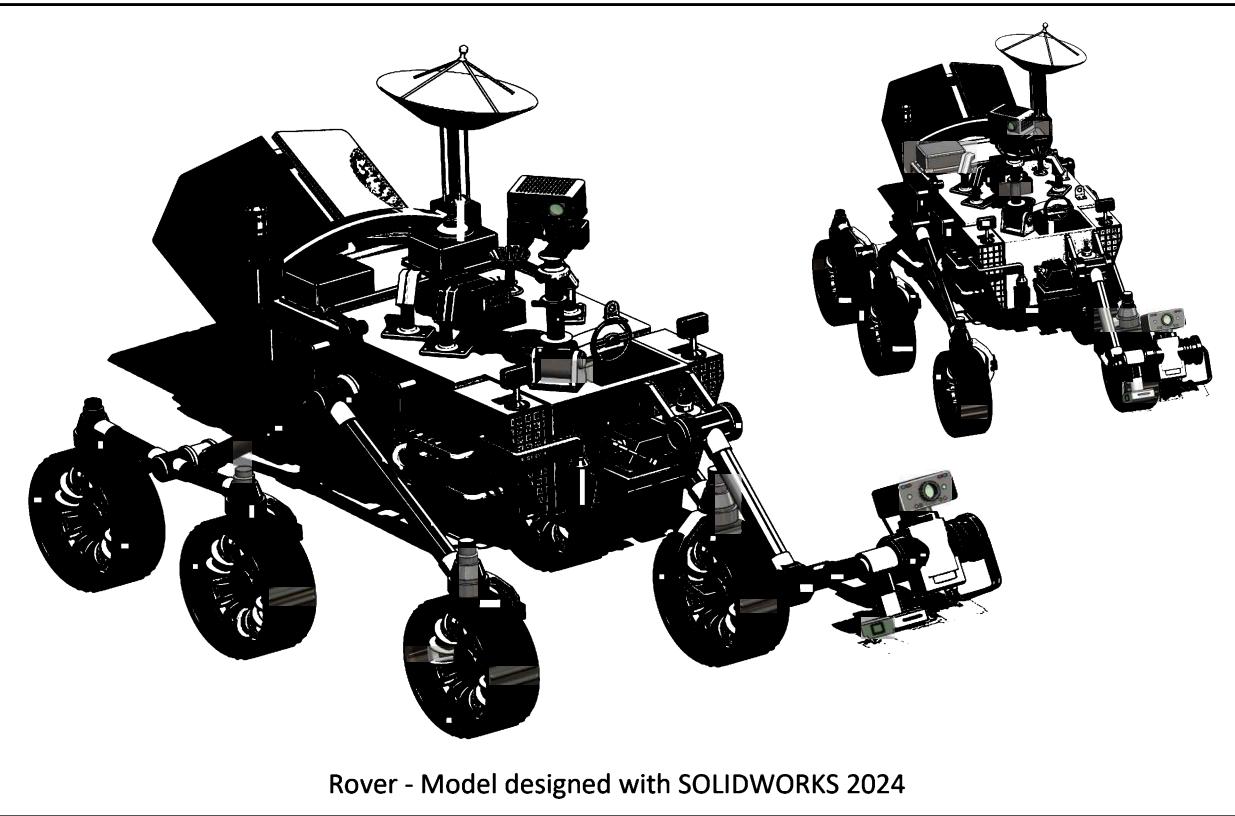


RC Robo-Snake - Model designed with SOLIDWORKS 2024



Pick & Place Robot - Model designed with SOLIDWORKS 2024







SOLIDWORKS 2024

Certified SOLIDWORKS Professional (CSWP)

Certification Practice for the Mechanical Design Exam

Courtesy of Paul Tran, Sr. Certified SOLIDWORKS Instructor

Certified-SOLIDWORKS-Professional (CSWP)

Certification Practice for the Mechanical Design Exam

Challenge I: Part Modeling & Modifications

Complete this challenge within 70 minutes (1 part)

(The following examples are intended to assist you in familiarizing yourself with the structures of the exams and the method in which the questions are asked.)

Create this part in SOLIDWORKS 2010 or newer

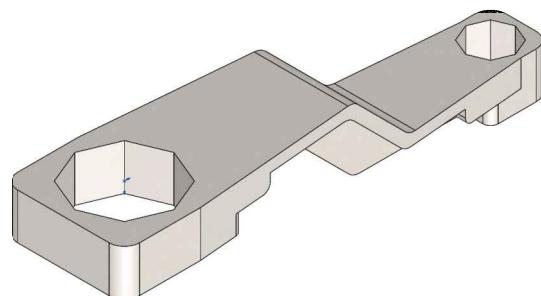
Unit: **Inches, 3 decimals**

Drafting Standards: **ANSI**

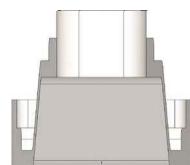
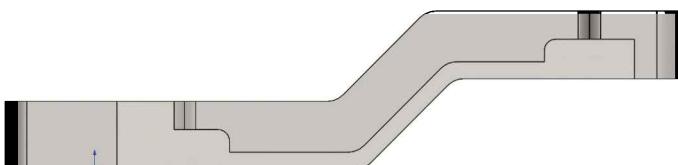
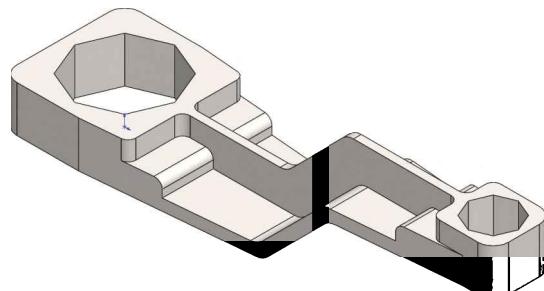
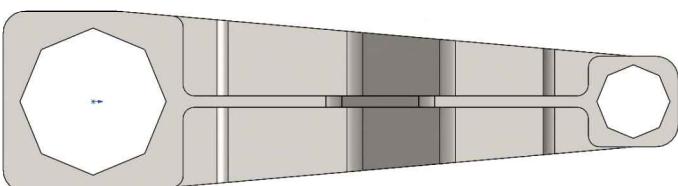
Origin: **Arbitrary**

Material: **Cast Alloy Steel**

Density: **0.264 lb/in³**



Save the model after each question as a different file in case it must be reviewed.



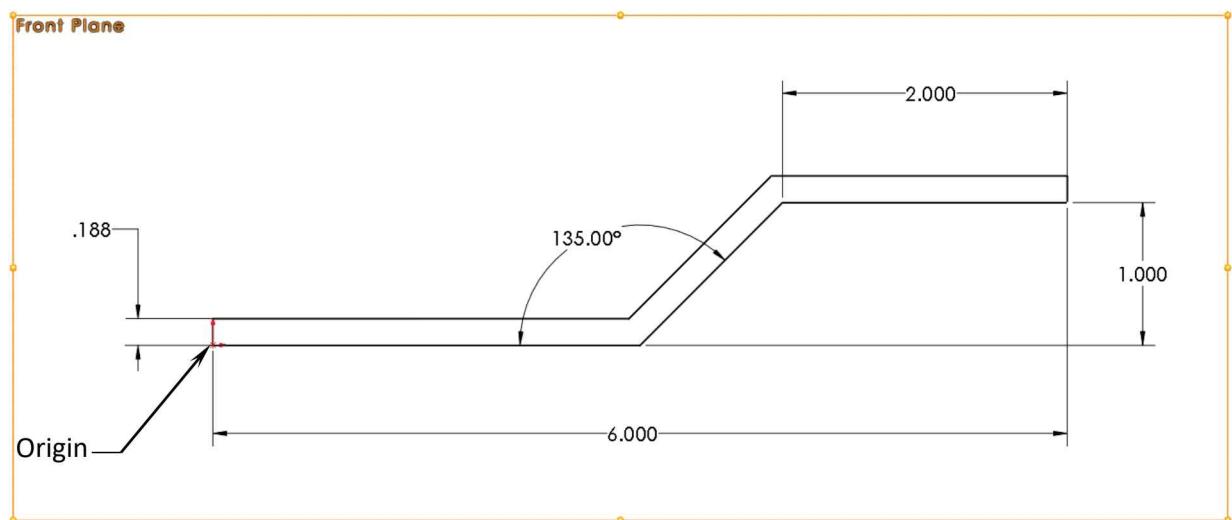
(The CSWP-Segment 1 examines your skills on creating a parametric model, where various methods will be used to create and constrain the geometry of each feature.)

1. Creating the base plate:

Select the Front plane and open a **new sketch**.

Sketch the profile shown below.

Keep the Origin at the lower left corner of the profile.



Add a **Parallel** relation to the two 135.00° lines.

The Vertical lines are also parallel with one another.

Add **dimensions** to fully define the sketch (dimensions are in Inches, three decimal places).

The .188" thickness dimension applies to the entire profile.

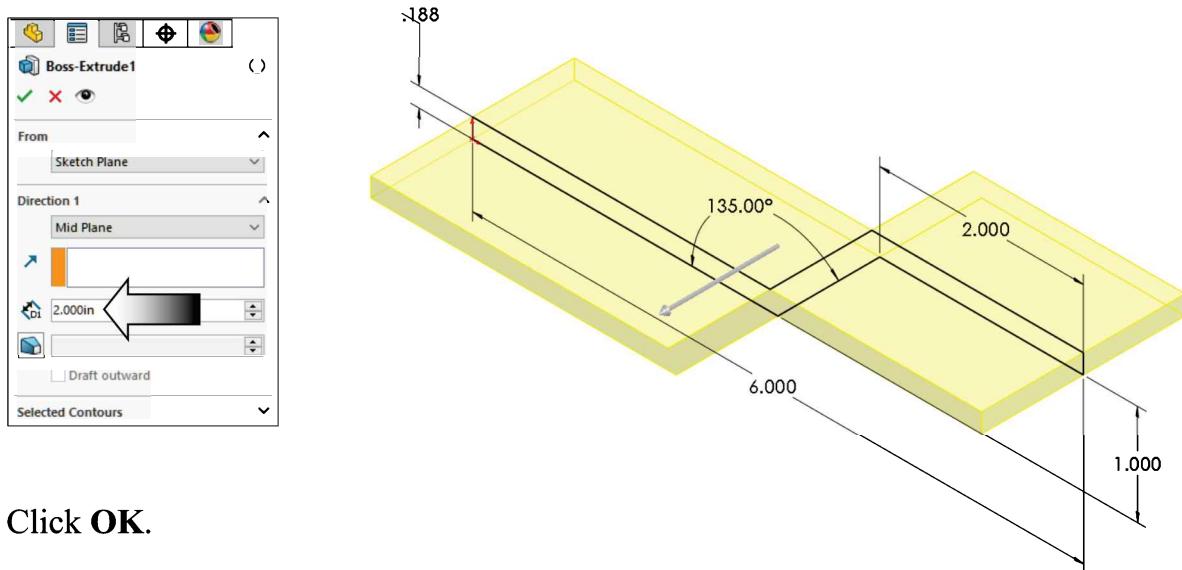
2. Extruding the sketch:

Switch to the **Features** tab.

Select the **Extruded Boss-Base** command.

For Direction 1, select the **Mid Plane** type.

For Extrude Depth, enter **2.000in**

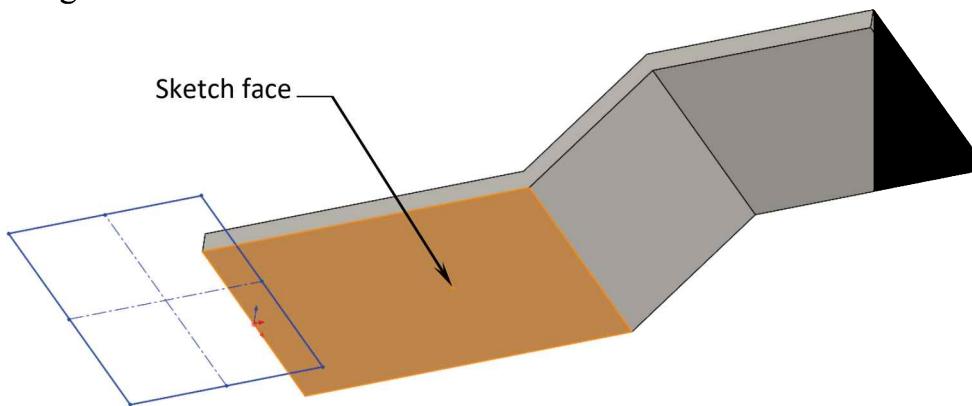


Click **OK**.

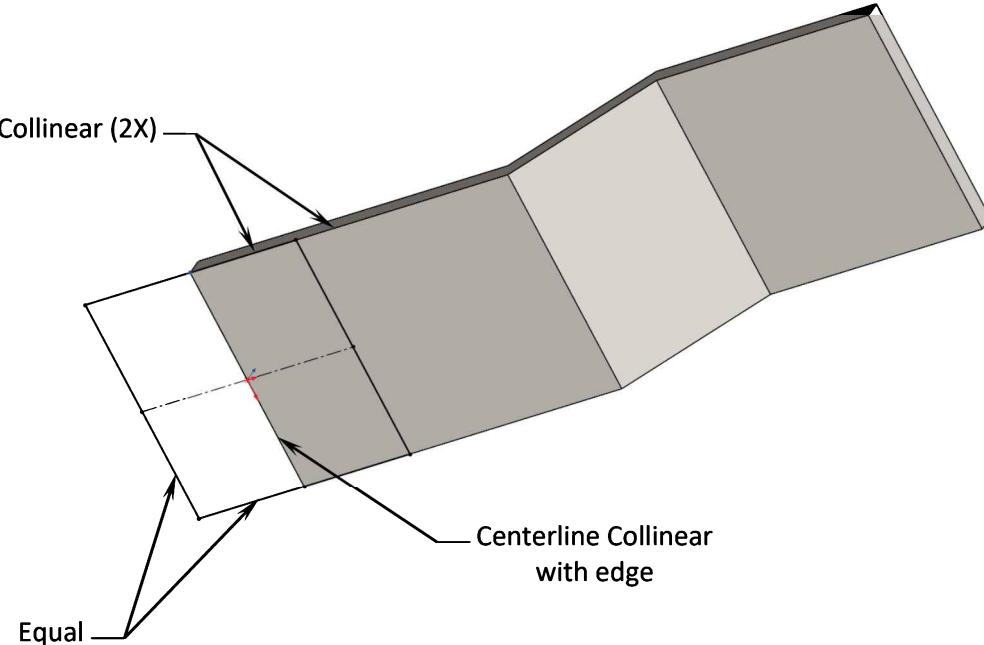
3. Making the 1st block:

Open a **new sketch** on the bottom face as indicated.

Sketch a **Center Rectangle** similar to the one shown in the image below.



Add the relations shown below to fully define the sketch.

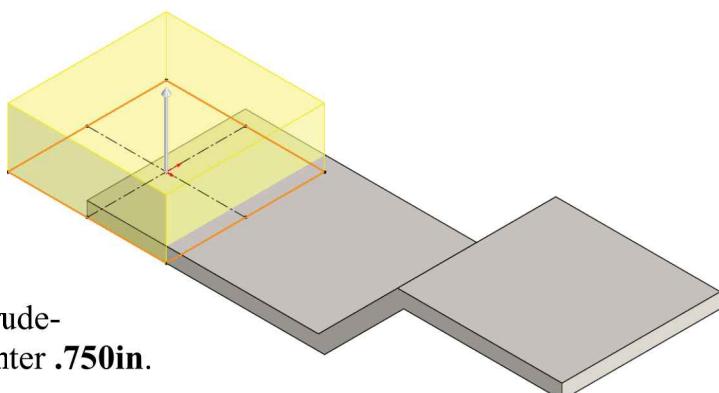
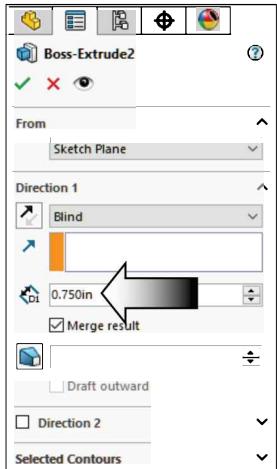


The relations above will make the rectangle share the same width as the base.

4. Extruding the sketch:

Switch to the **Features** tab and select the **Extruded Boss-Base** command.

For Direction 1, select **Blind**.



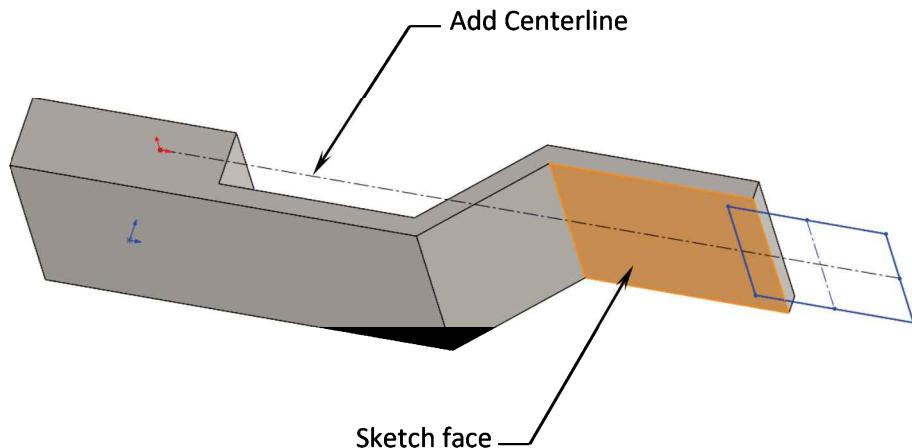
For Extrude-
Depth enter .750in.

Click **OK**.

5. Making the 2nd block:

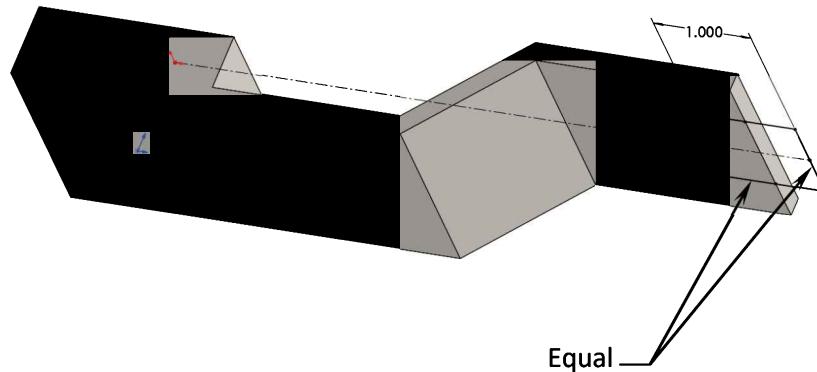
Select the bottom face on the right and open a **new sketch**.

Sketch a **Center Rectangle** approx. as shown.



Add a **Centerline** that starts from the Origin and connects to the midpoint of the line on the right side of the rectangle.

Add a dimension **1.000in.** to one of the lines.



Add an **Equal** relation to one horizontal and one vertical line to fully define the sketch.

6. Extruding the sketch:

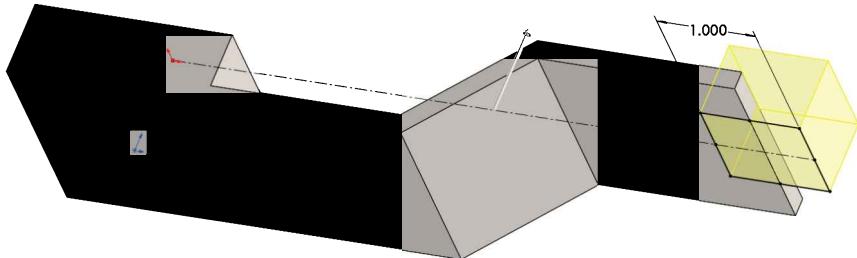
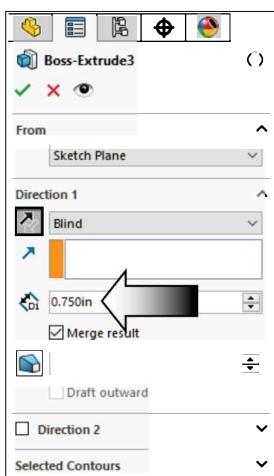
Switch to the **Features** tab.

Click the **Extruded Boss-Base** command.

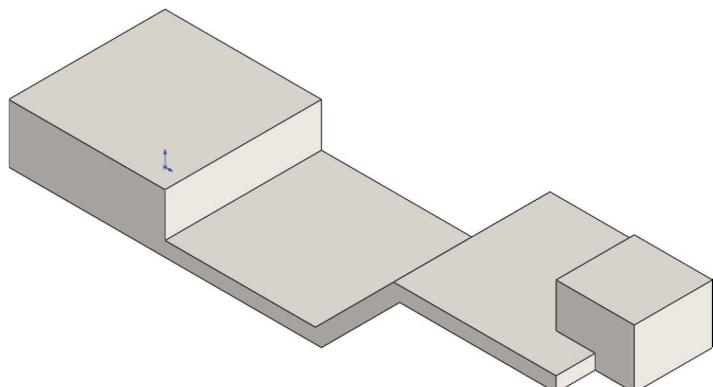
For Direction 1, select the **Blind** type.

For Extrude Depth, enter **.750in**

Enable the **Merge Result** checkbox.



Click the **Reverse Direction** button if needed to extrude the sketch upward.



Click **OK**.

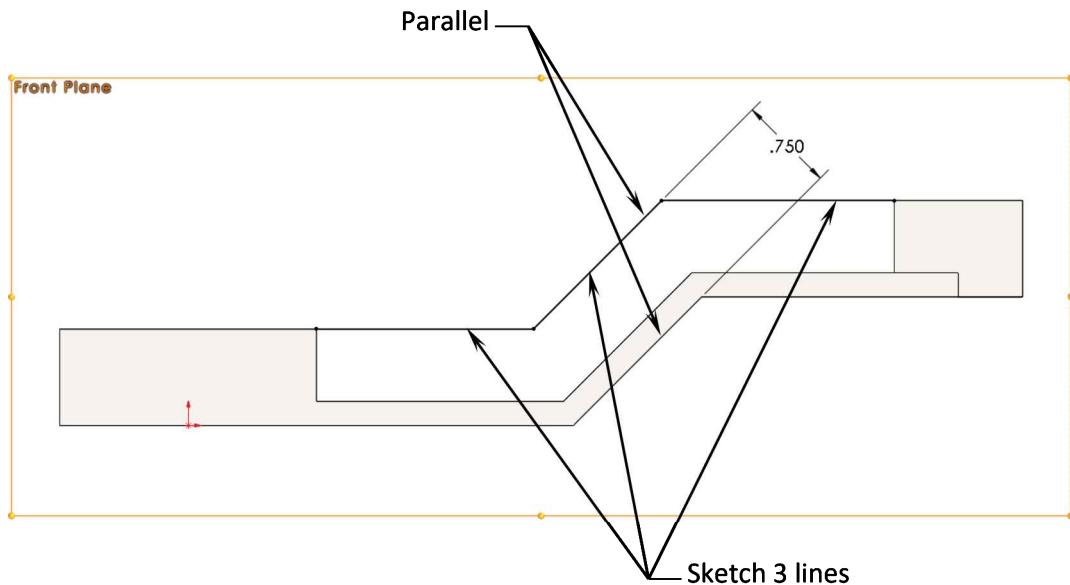
The 1st block and the 2nd block should have the same thickness (.750in.).

7. Adding a Rib feature:

The next step is to add the support rib across the length of the part.

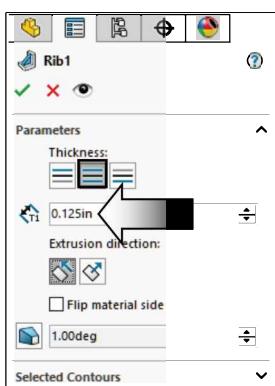
Select the Front plane and open a **new sketch**.

Sketch 3 lines shown below and add a **Parallel** relation as indicated.

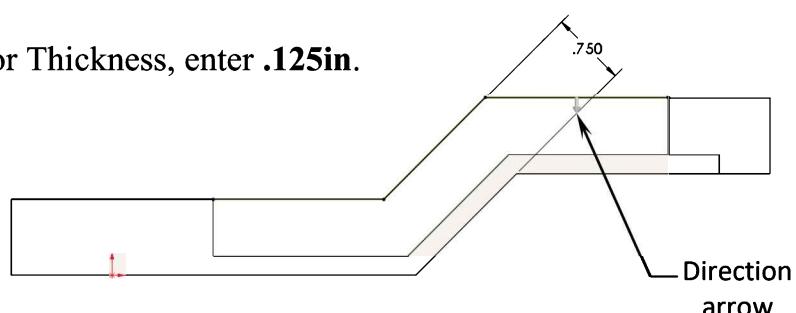


Switch to the **Features** tab and select the **Rib** command.

For Direction, select **Both Sides**.

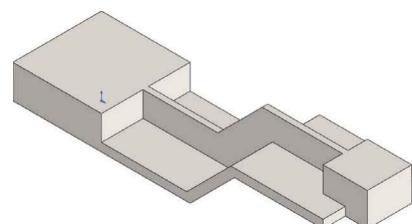


For Thickness, enter **.125in**.



The Direction arrow should be pointing downward.

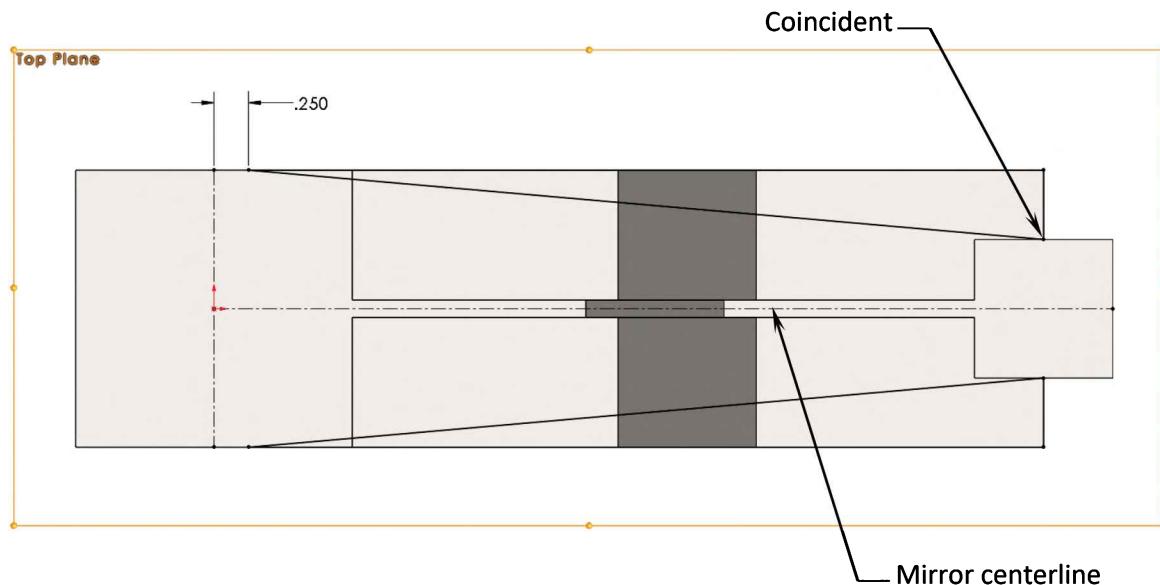
Click **OK**.



8. Trimming the base:

Open a **new sketch** on the Top plane.

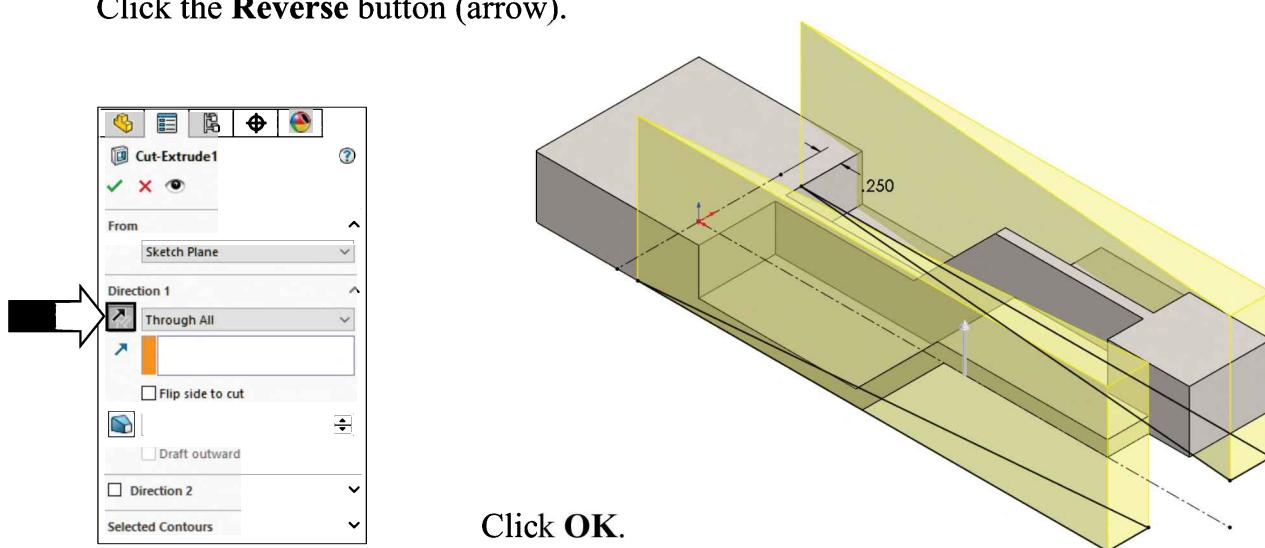
Sketch the profile shown below. Use the **Mirror Entities** option to keep the two halves symmetrical about the horizontal centerline.



Switch to the **Features** tab and select **Extruded Cut**.

For Direction 1, select **Through All**.

Click the **Reverse** button (arrow).

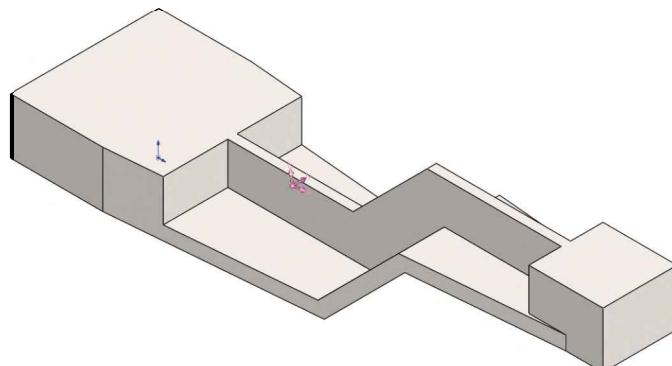
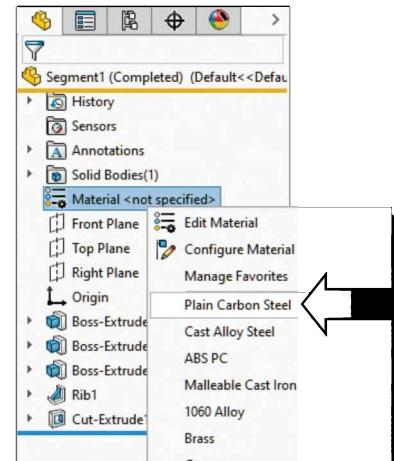
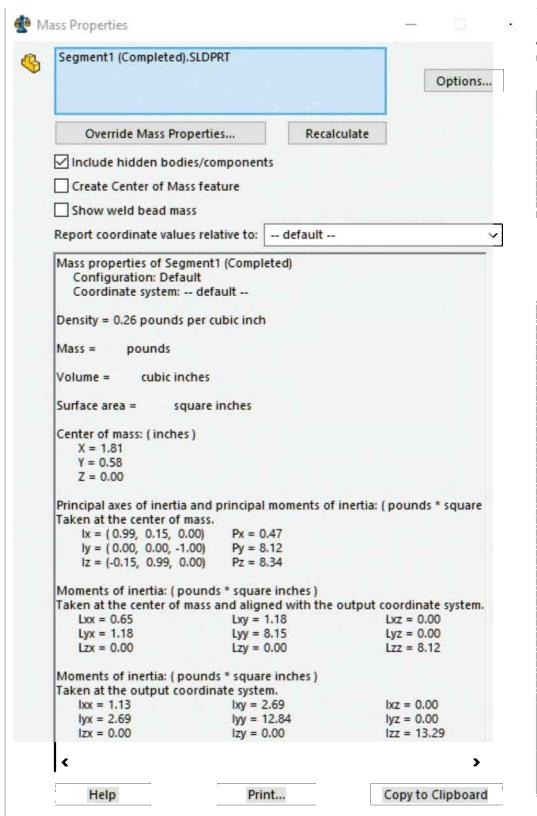


Click OK.

9. Calculating the mass properties:

Right-click the Material option and select:
Plain Carbon Steel.

Switch to the Evaluate tab and click **Mass Properties**.



Enter the mass of the part below:

_____ pounds.

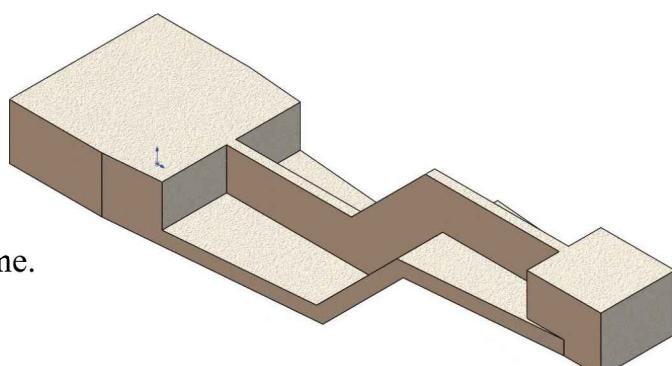
10. Saving the part:

Select **File, Save As.**

Enter **Segment1_Q1** for the file name.

Click **Save**.

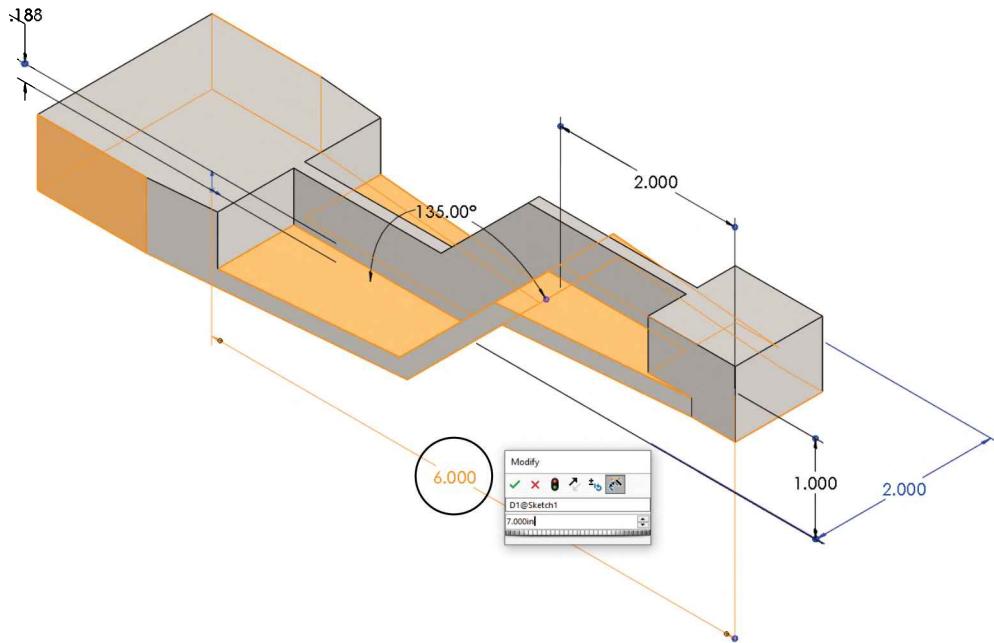
Keep the part open.



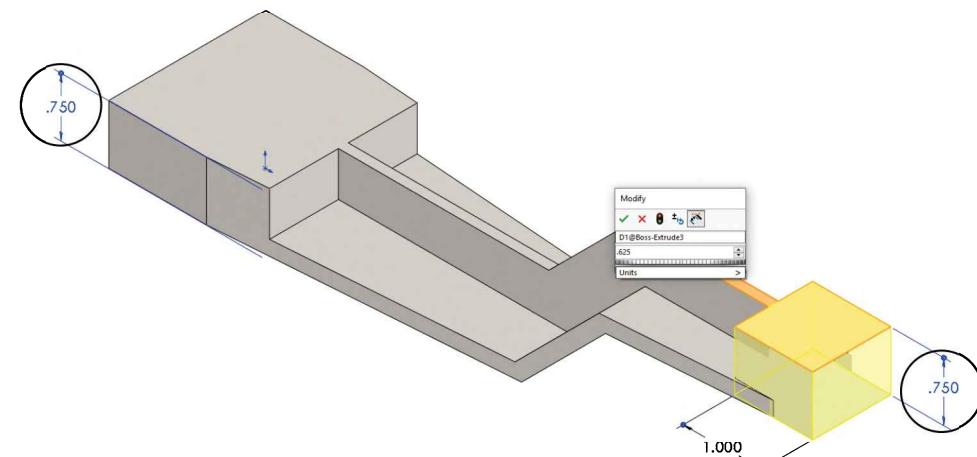
11. Modifying the dimensions:

Double-click **Boss-Extrude1** to display its dimensions.

Change the dimension **6.000in** (circled) to **7.000in**.



Double-click **Boss-Extrude2** and **Boss-Extrude3** and change both thickness dimensions from **.750in** (circled) to **.625in**.



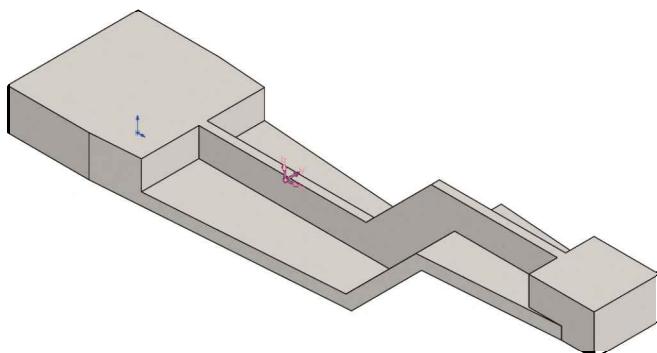
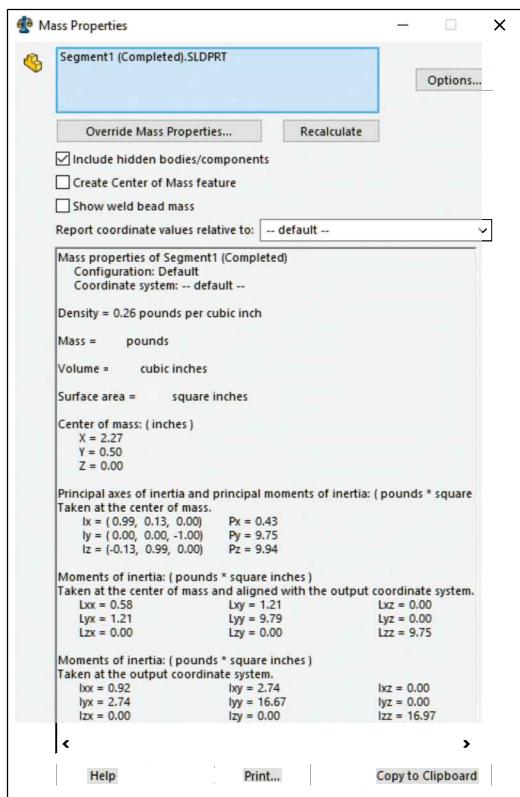
Press **Rebuild**  to update the model.

12. Calculating the mass:

There are several dimensions changes in the actual exam, but we will change only the 3 main ones in this exercise.

Switch to the **Evaluate** tab.

Click **Mass Properties**.



Enter the mass of the part below:

_____ pounds.

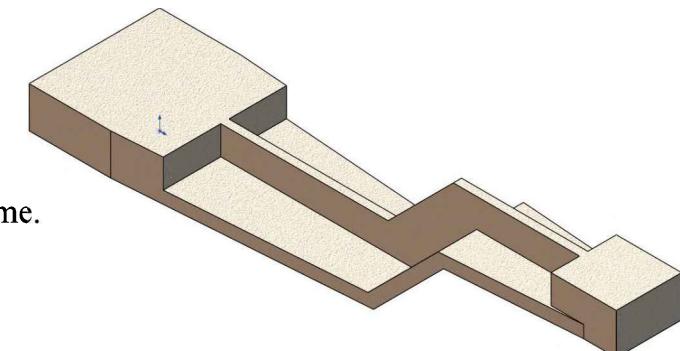
13. Saving the part:

Select **File, Save As**.

Enter **Segment1_Q2** for the file name.

Click **Save**.

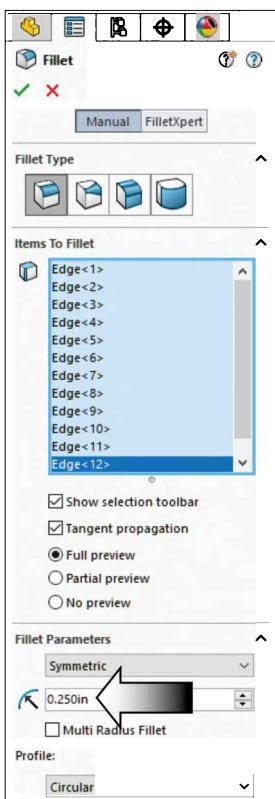
Keep the part open.



14. Adding fillets:

Switch to the **Features** tab.

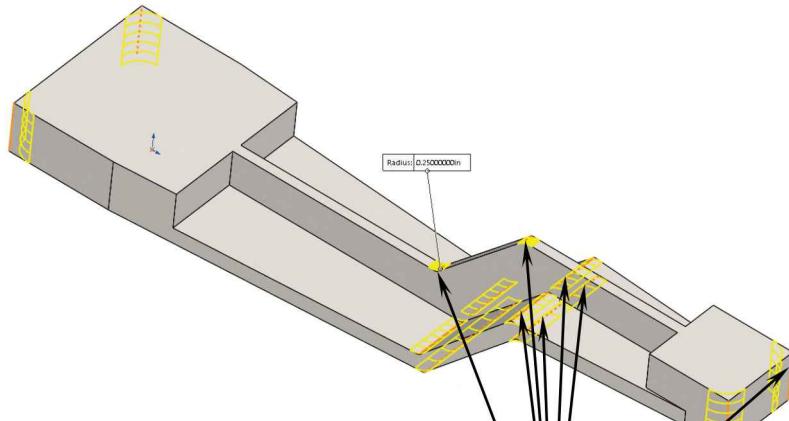
Click **Fillet** .



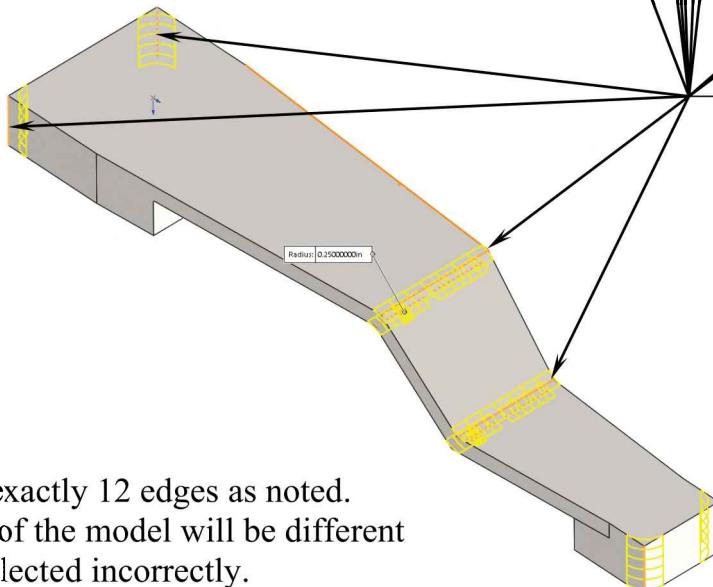
For Fillet Type, use the default **Constant Size** fillet.

Enter **.250in** for radius.

Select **12 edges** as indicated.



Click **OK**.

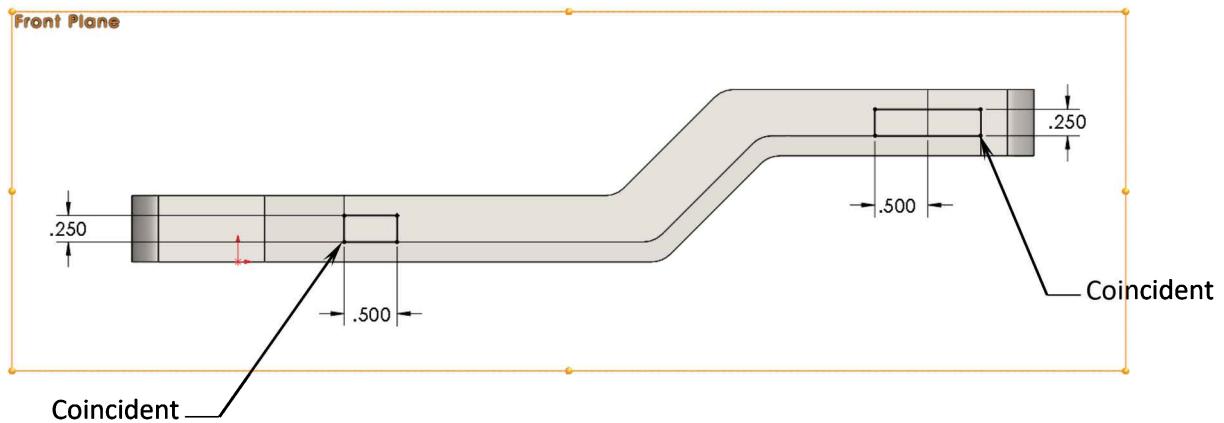


Be sure to select exactly 12 edges as noted.
The overall mass of the model will be different
if the edges are selected incorrectly.

15. Creating the side tabs:

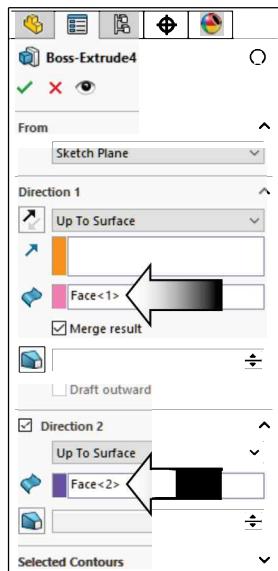
Select the Front plane and open a **new sketch**.

Sketch 2 Corner Rectangles. The left and the right corners of the 2 rectangles are **Coincident** to the vertical edges of the square blocks.

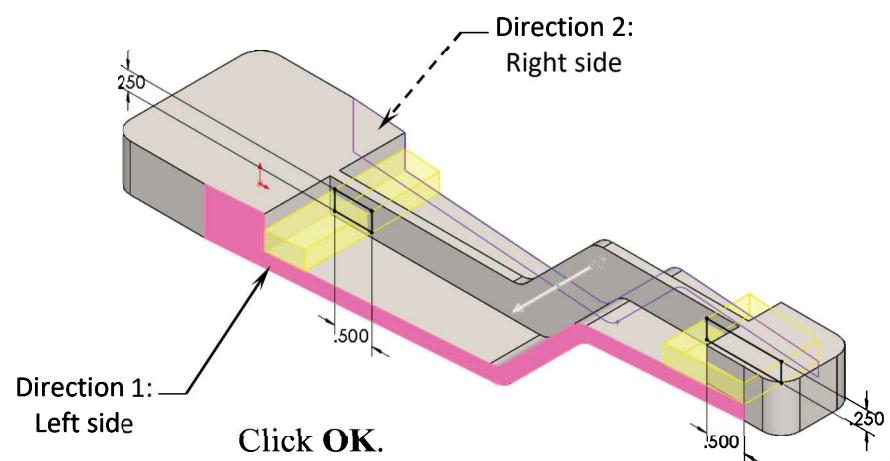


Switch to the **Features** tab and select the **Extruded Boss-Base** command.

For Direction 1, select the **Up-to-Surface** option and click the face on the left side of the part as indicated.



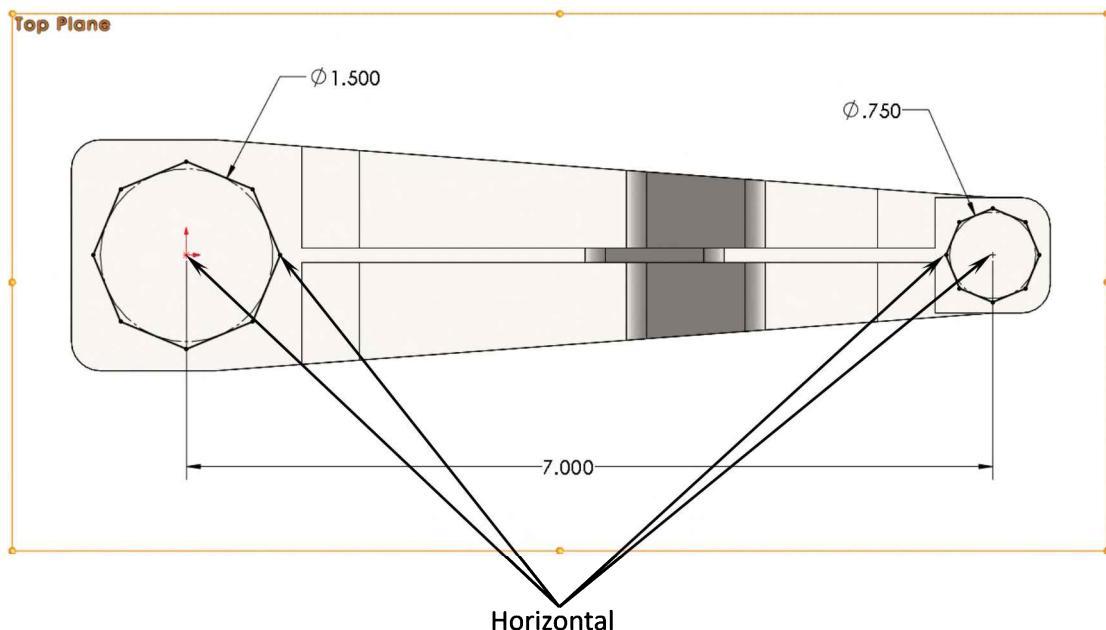
For Direction 2, also select the **Up-to-Surface** option and click the face on the right side of the part.



16. Adding the 8-sided Polygonal holes:

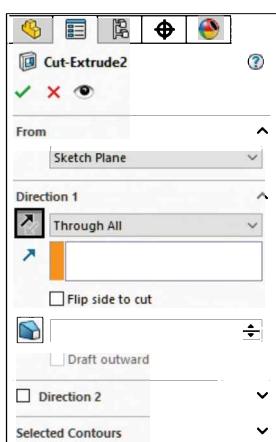
Select the **Top** plane and open a **new sketch**.

Sketch **two 8-sided Polygons** and add the **Horizontal** relations as indicated.



Add the dimensions shown above to fully define the sketch.

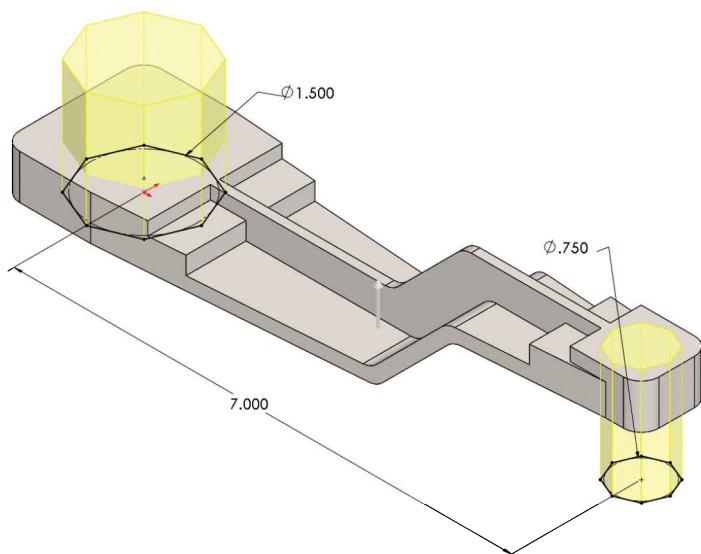
Switch to the **Features** tab and click **Extruded Cut**.



For Direction 1
select **Through All**.

**Click Reverse
Direction.**

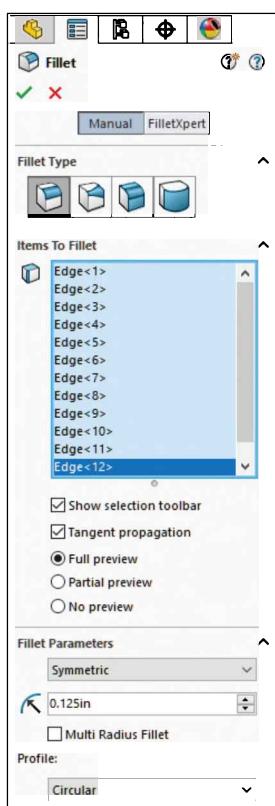
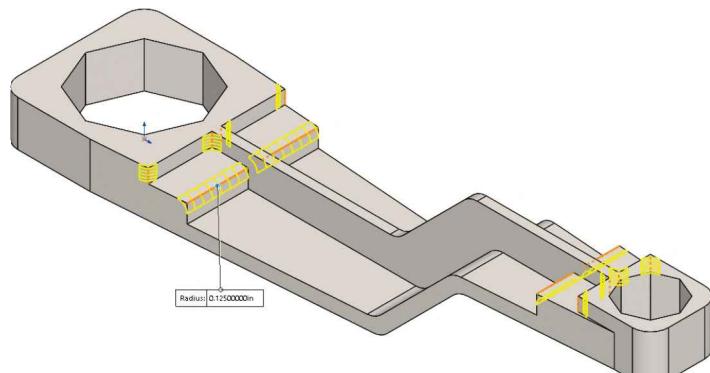
Click OK.



17. Adding fillets:

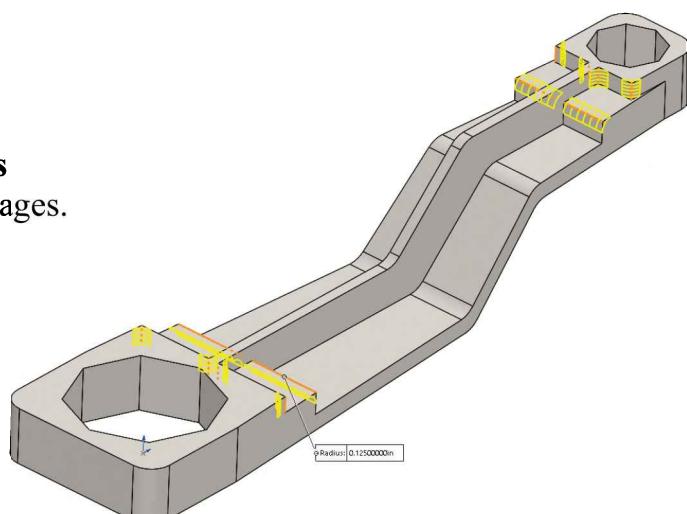
Click **Fillet** .

Use the default **Constant Size Radius** option.



Enter **.125in.** for radius.

Select the **12 edges** as shown in the images.



Click **OK**.

18. Saving the part:

Select **File, Save As**.

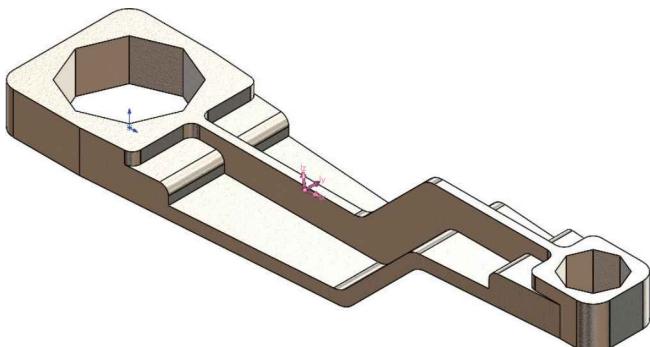
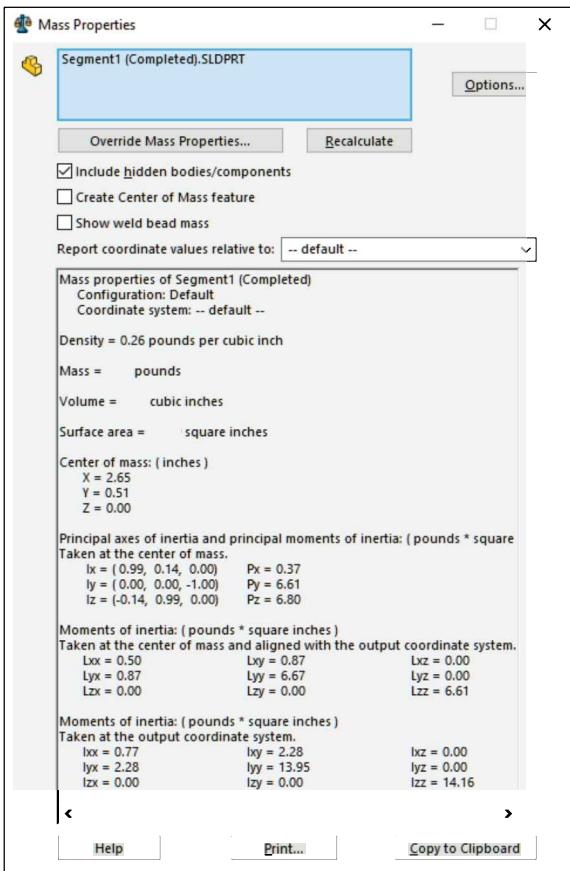
Enter **Segment1_Q3** for the file name.

Click **Save**.

19. Calculating the final mass:

Switch to the **Evaluate** tab.

Click **Mass Properties**.



Locate the mass of the part and enter it below:

_____ pounds

Resave the part and close all documents.

Certified-SOLIDWORKS-Professional (CSWP)

Certification Practice for the Mechanical Design Exam

Challenge II: Part Modifications & Configurations

Complete this challenge within 50 minutes (3 parts)

(The following examples are intended to assist you in familiarizing yourself with the structures of the exams and the method in which the questions are asked.)

Modify this part using SOLIDWORKS 2010 or newer.

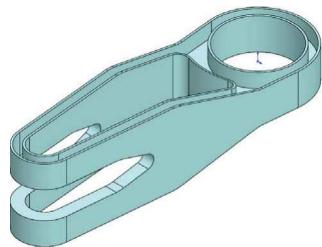
Unit: **Inches, 3 decimals**

Drafting Standards: **ANSI**

Origin: **Arbitrary**

Material: **ABS**

Density: **0.2037 lb/in³**

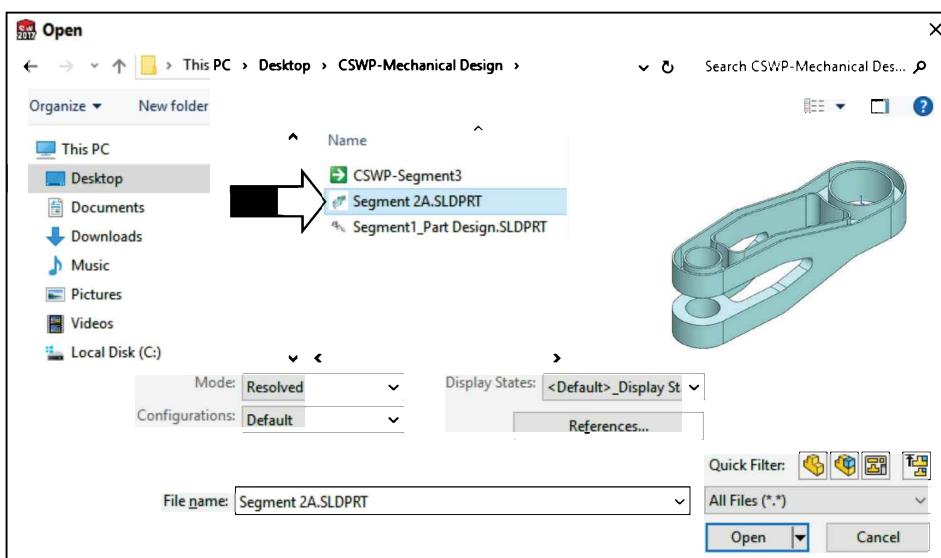


(This portion of the test will examine your skills on the modification of dimensions, geometry, and repair errors in a part.)

1. Opening a part document: (1 of 3)

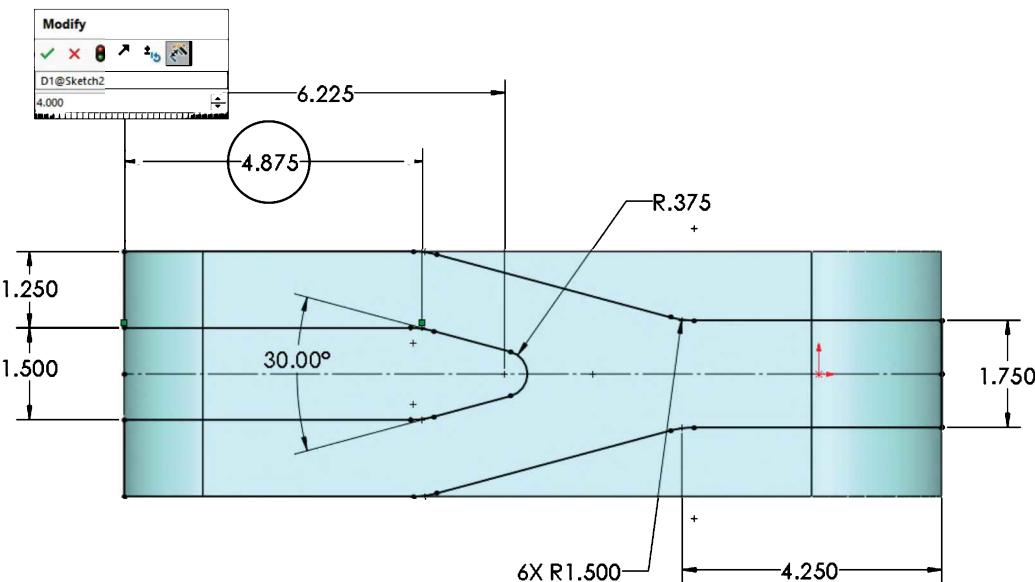
Click File / Open.

Locate and open the part document named: **Segment 2A.sldprt**



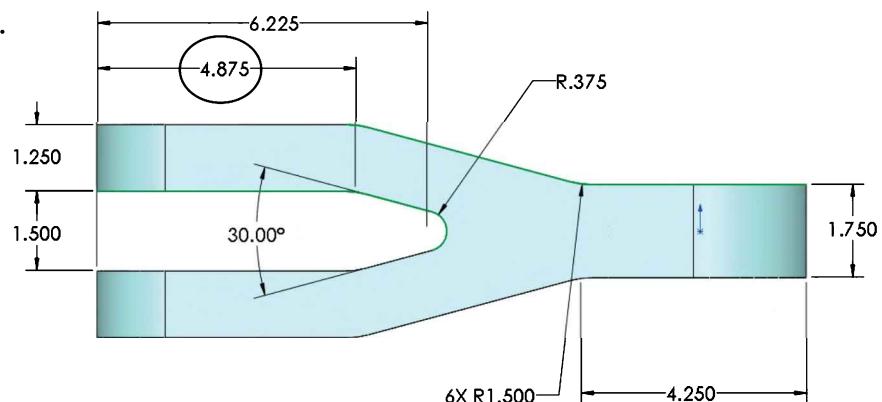
2. Modifying dimensions:

Double-click the feature named **Cut-Extrude1** and change the dimension **4.875** (circled) to **4.000**.

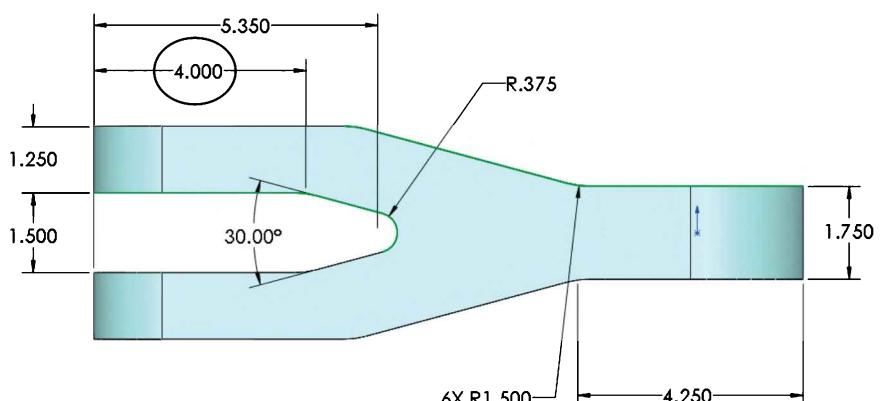


Click **Rebuild**  to update the model.

The model shown before the dimension change.

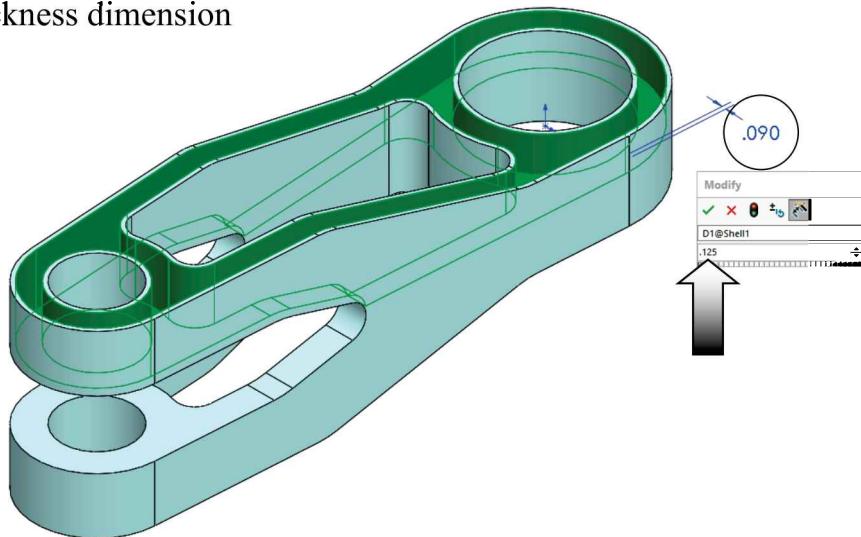


The model shown after the dimension change.



Double-click the **Shell** feature from the FeatureManager tree.

Change the wall thickness dimension from **.090** to **.125**.

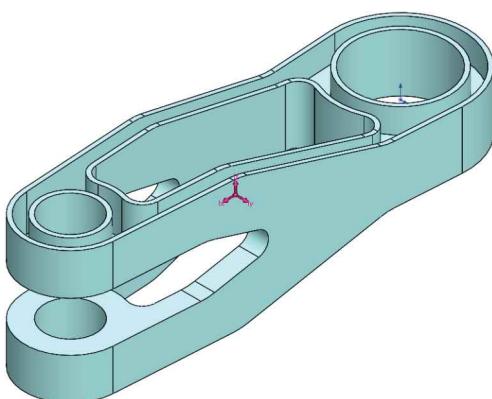


Click **Rebuild**  to update the model.

3. Calculating the mass:

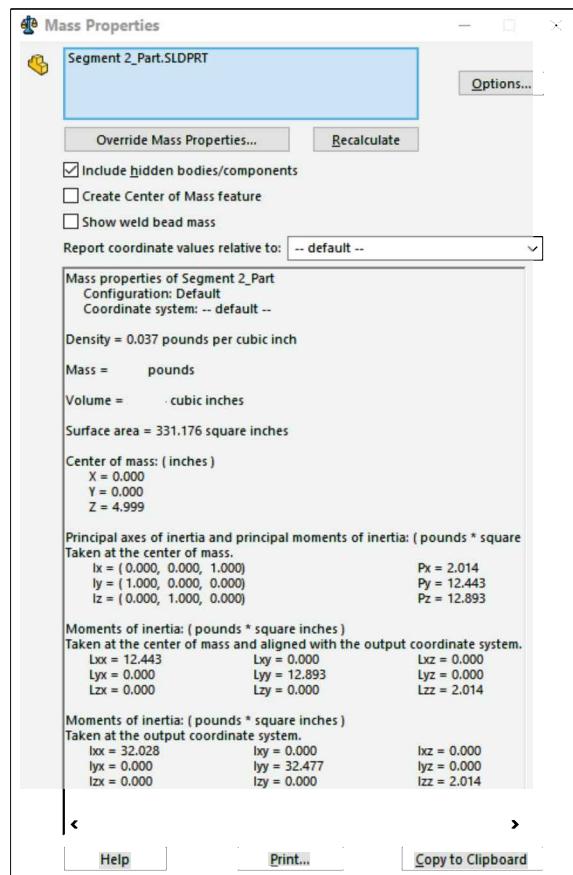
Switch to the **Evaluate** tab.

Click **Mass Properties**.



Enter the mass of the part here:

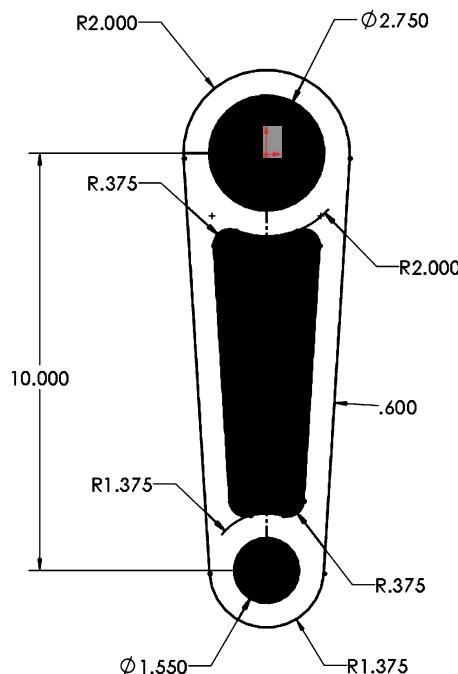
_____ pounds.



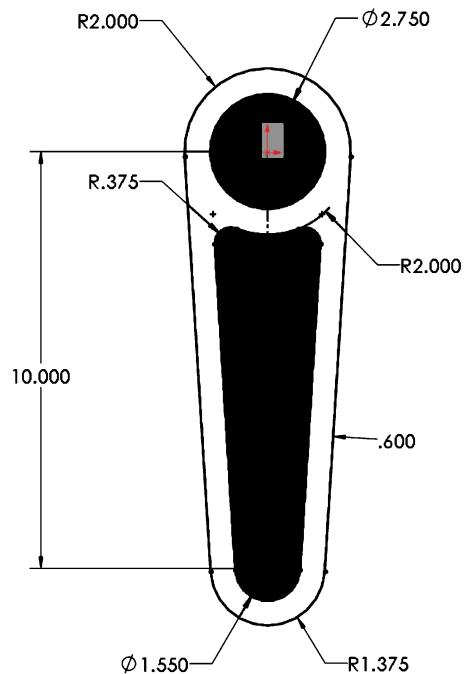
4. Modifying the geometry:

Edit Sketch1 under the Boss-Extrude1 feature.

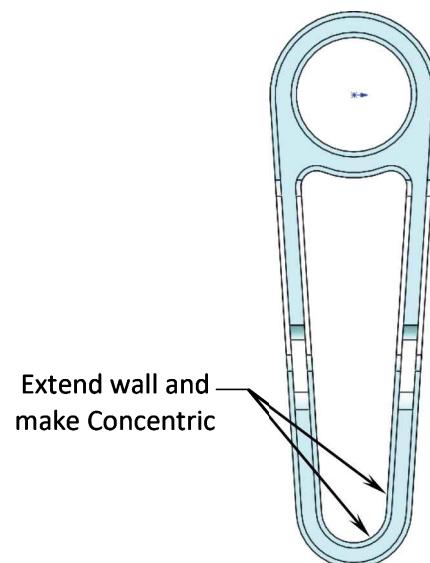
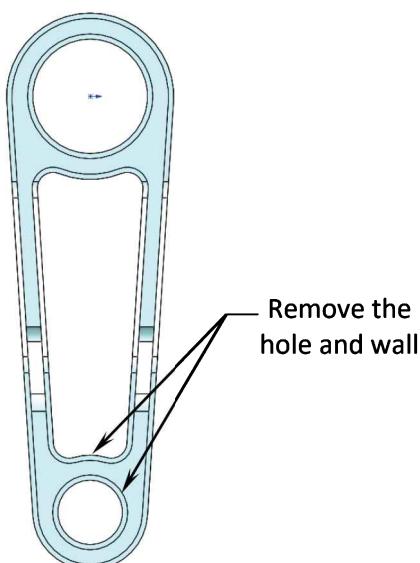
Modify the geometry of the sketch so that it would look like the images on the right.



Before the changes



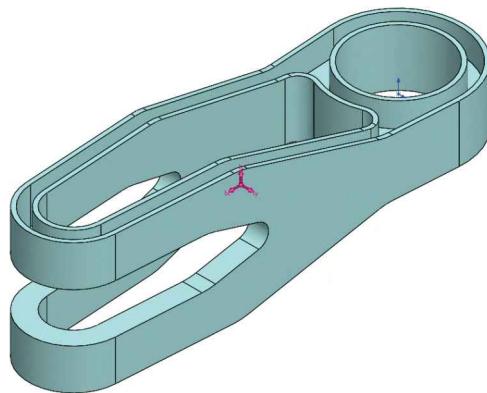
After the changes



5. Calculating the mass:

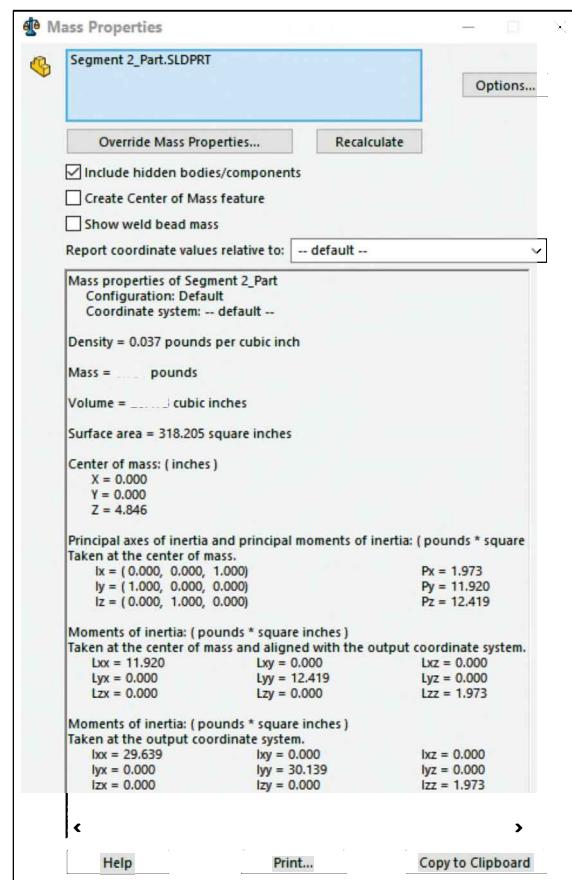
Switch to the **Evaluate** tab.

Click Mass Properties.



Enter the mass of the part here:

pounds



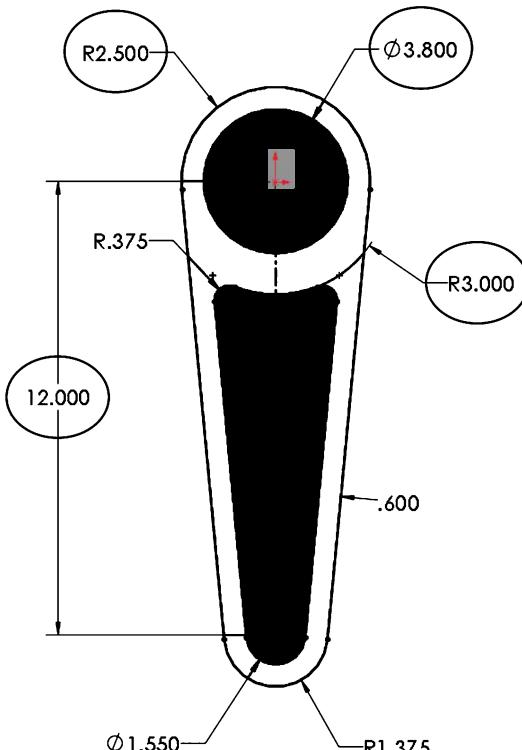
6. Making additional changes:

Edit the **Sketch1** under the Boss-Extrude1 feature.

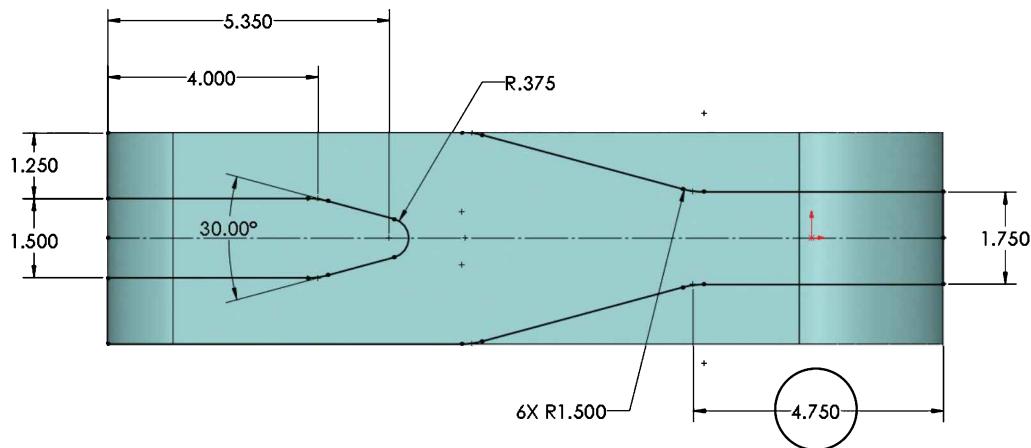
Change the dimensions to match the values in the circles.

Add any relations needed to fully define the sketch.

Exit the sketch and **Rebuild** the model.
The model should not have any errors.
Correct the errors when/where they occur.



Edit Sketch2 under the Cut-Extrude1 feature.

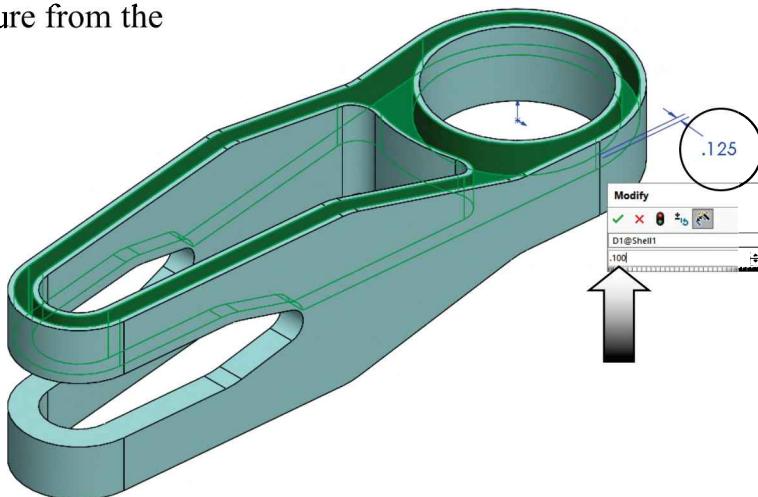


Change the dimension **4.250** to **4.750** (circled).

Click **Rebuild** to update the model.

Double-click the **Shell** feature from the FeatureManager tree.

Change the wall thickness dimension from **.125** to **.100**.



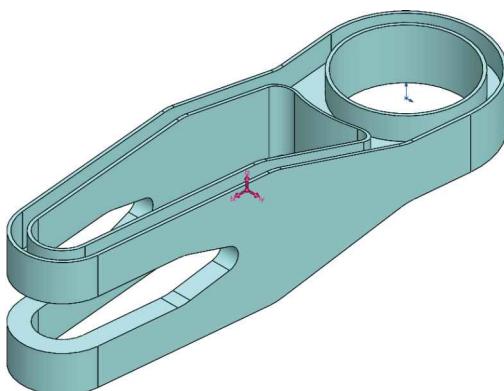
Click **Rebuild** to update the model.

The model should be free of errors. Correct/repair any errors when/where they occur.

7. Calculating the mass:

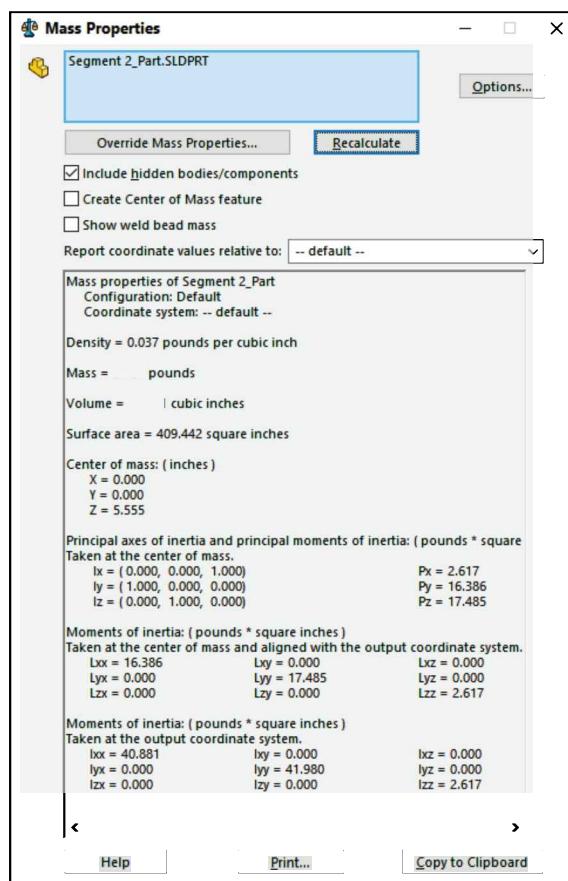
Switch to the **Evaluate** tab.

Click **Mass Properties**.



Enter the mass of the part here:

_____ pounds.



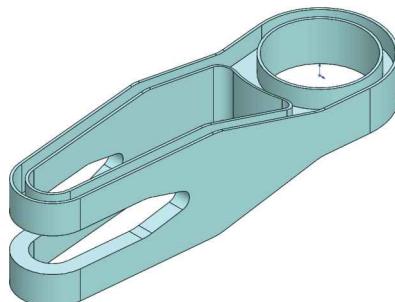
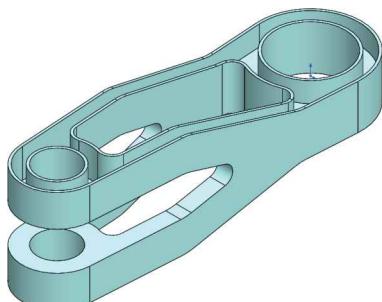
8. Saving the part:

Select **File, Save As**.

Enter **Segment 2A (Completed).sldprt** for the file name.

Click **Save**.

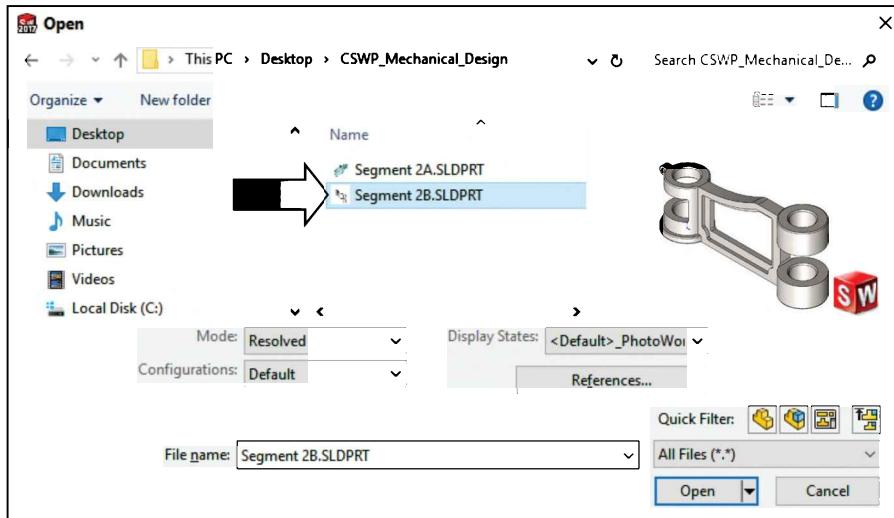
Close the part document.



1. Opening a part document: (2 of 3)

Click **File / Open**.

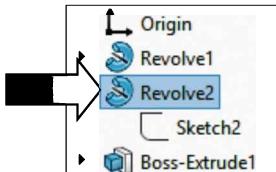
Locate and open the part document named: **Segment 2B.sldprt**



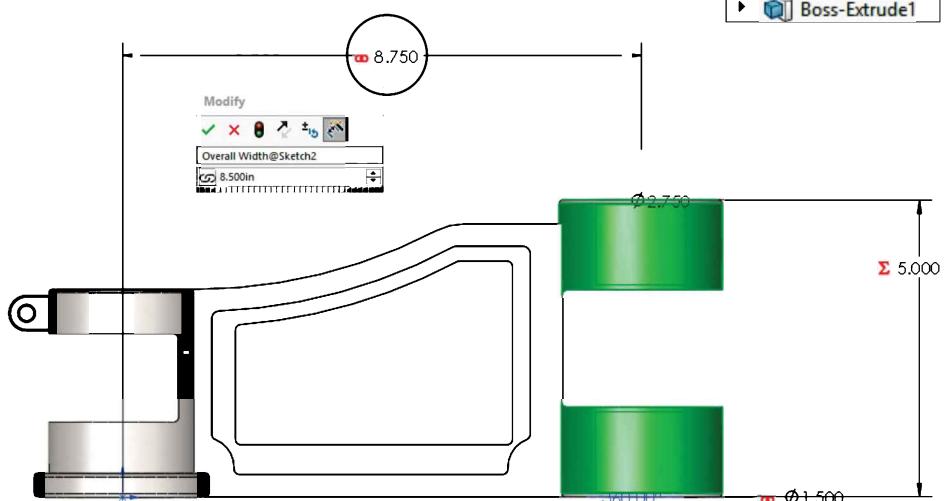
(This portion of the test will examine your skills on the modification of dimensions, geometry, and repair errors in a part.)

2. Modifying dimensions:

Double-click the feature **Revolve2** from the Feature-Manager tree.



Change the dimension **8.750** (circled) to **8.500**.

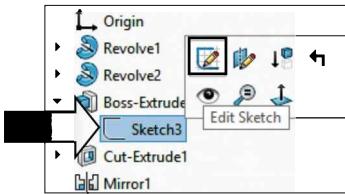


Click **Rebuild** to update the model.

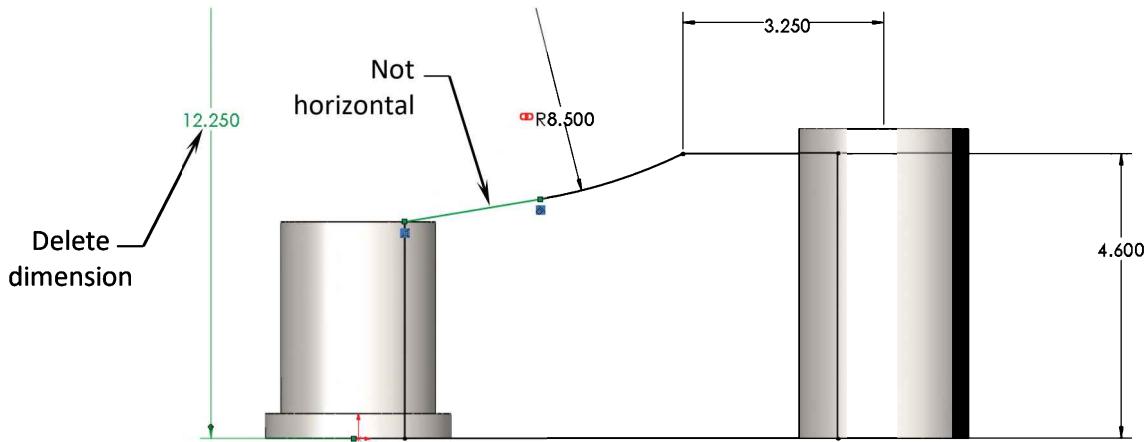
The dimension change triggers some errors in the model. These errors will be repaired after the geometry in the Sketch3 are corrected.



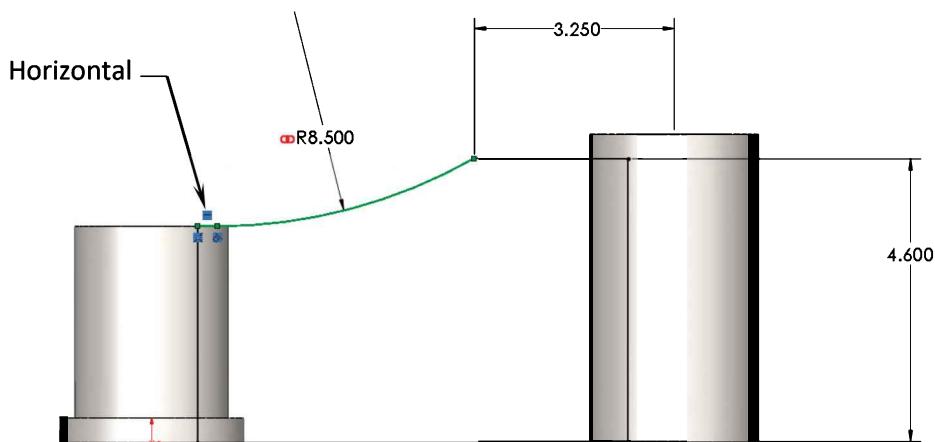
Edit Sketch3 under the Boss-Extrude1 feature (arrow).



The dimension 12.250 prevents the line from being tangent to the arc. Delete the dimension 12.250 as indicated.

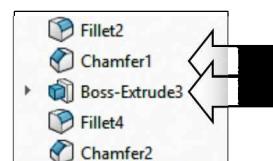


The sketch is no longer over defined.
Add a **Horizontal** relation to the line as noted.



Exit the sketch when it is fully defined.

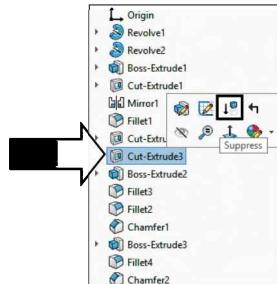
The errors are automatically corrected and removed.



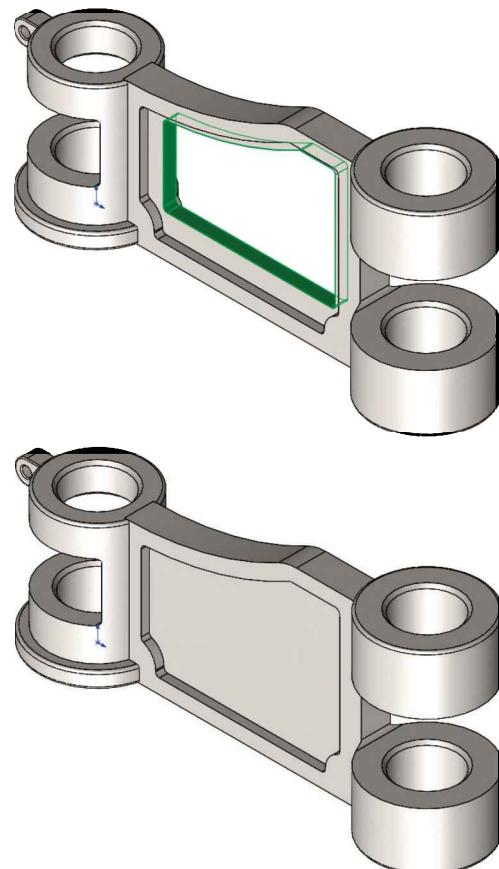
3. Suppressing features:

Locate the feature named **Cut-Extrude3** from the FeatureManager tree. It is the cutout feature in the middle of the part.

Click the **Cut-Extrude3** feature and select **Suppress**.



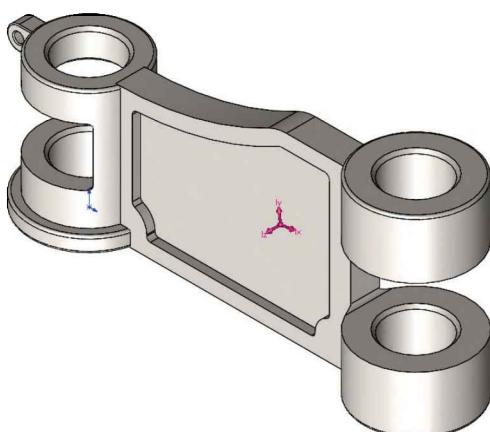
The cutout feature is suppressed.



4. Calculating the mass:

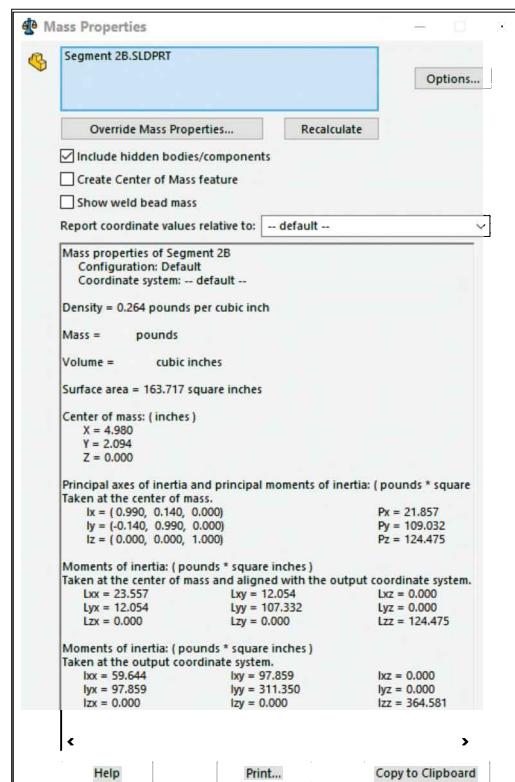
Switch to the **Evaluate** tab.

Click **Mass Properties**.



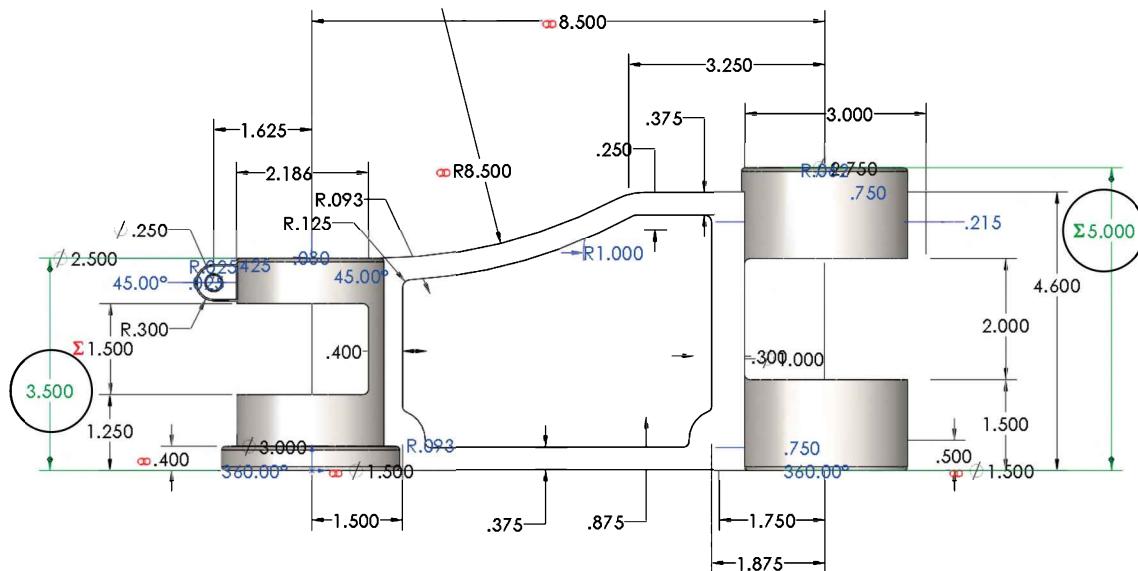
Enter the mass of the part here:

_____ pounds.



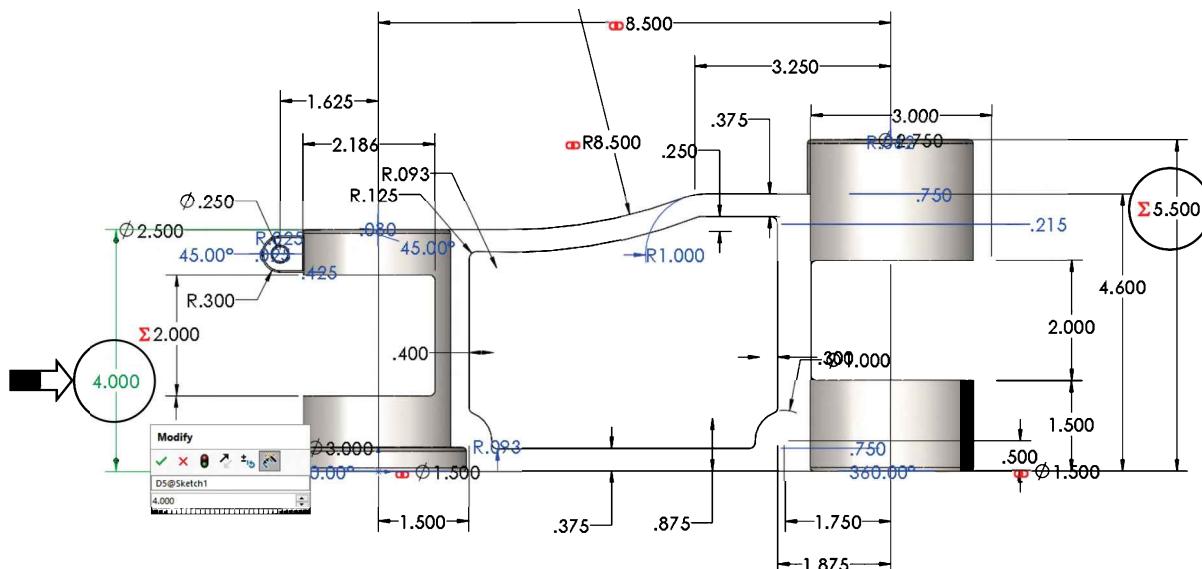
5. Changing a dimension linked to an equation:

Right-click the **Annotations folder** and select **Display Annotations**.



The dimension **3.500** (circled) is tied to the dimension **5.000** (circled) in an equation that was previously created. It is the driving dimension.

Double-click the dimension **3.500** and change it to **4.000**.

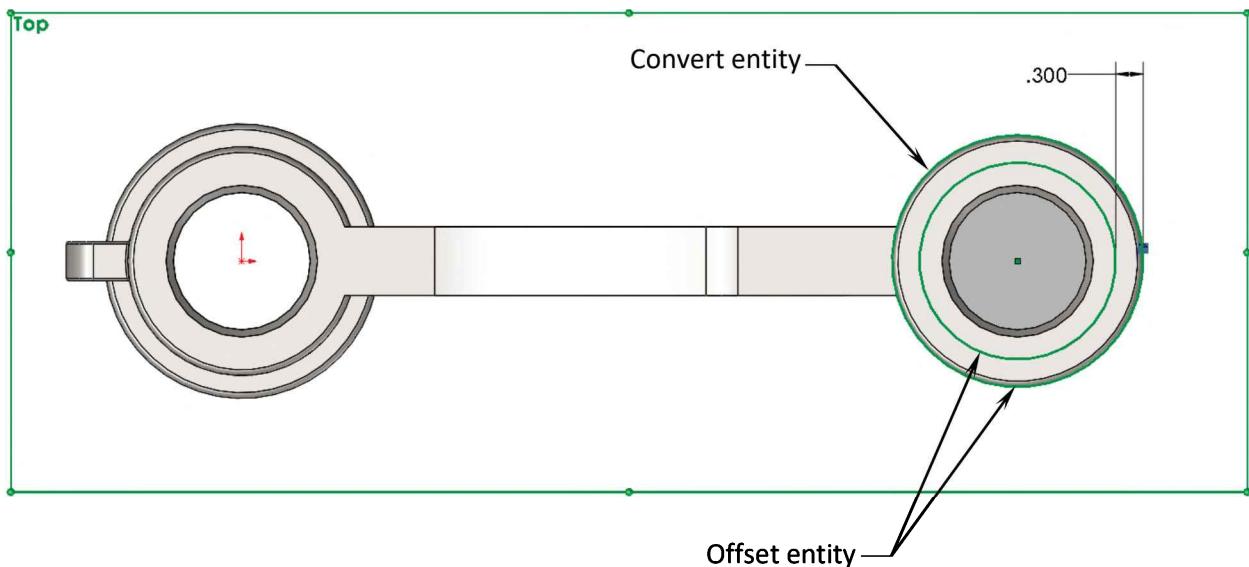


Click Rebuild to update the model. Hide all annotations.

6. Modifying the revolved feature:

Select the Top plane and open a **new sketch**.

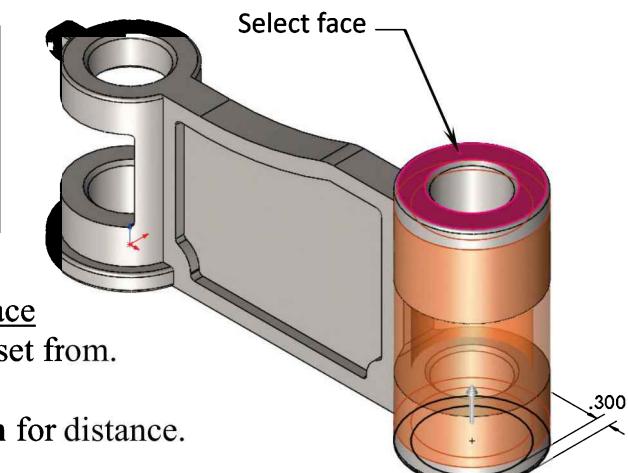
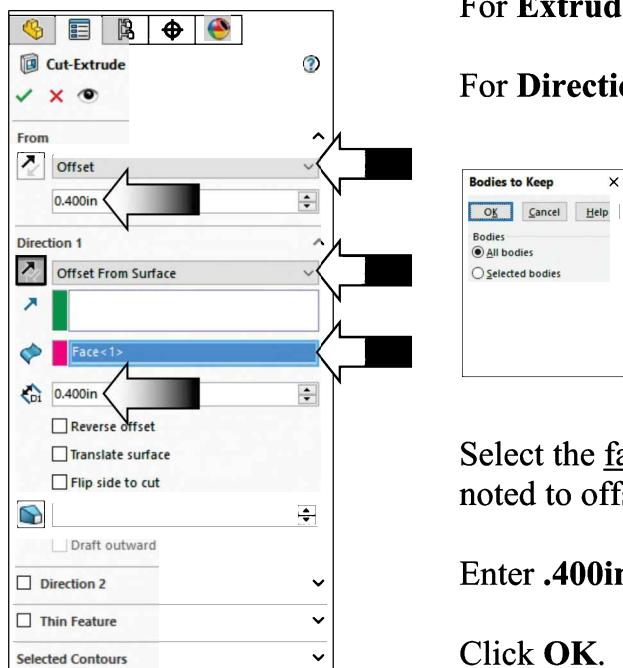
Select the outer edge of the revolved feature (on the right side) and create an **Offset Entity** using a distance of **.300in.** (smaller). Also **Convert** the outer edge of the same revolved feature into a circle as indicated.



Switch to the **Features** tab and click **Extruded Cut**.

For **Extrude From**, select **Offset** and enter **.400** (arrow).

For **Direction 1**, select **Offset From Surface** (arrow).

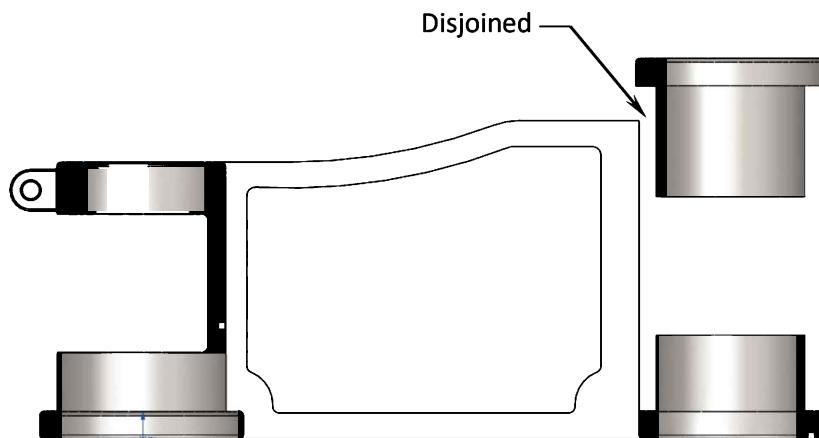


Select the face noted to offset from.

Enter **.400in** for distance.

Click OK.

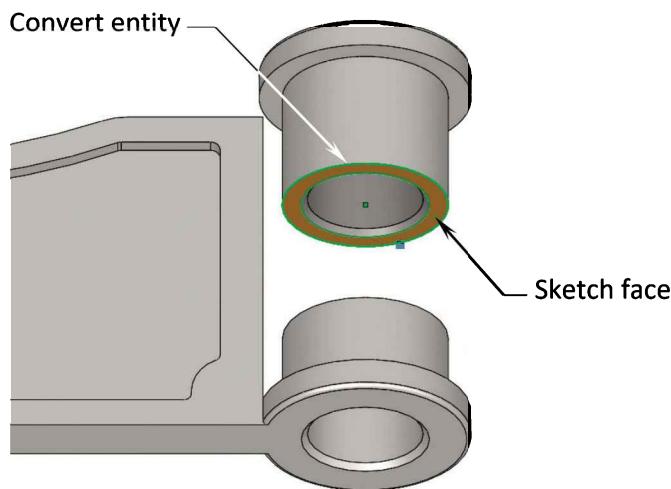
The cut feature causes the part to have 2 disjoined solid bodies. The gap between the 2 bodies must be filled in.



7. Filling the gap:

Select the **face** as indicated and open a new sketch.

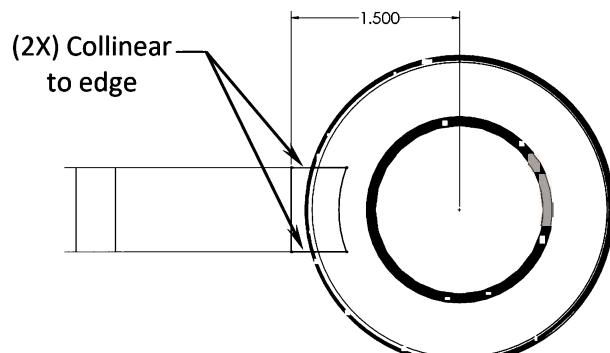
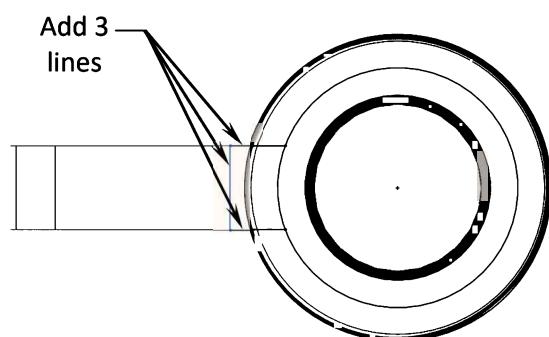
Select the outer edge of the cylinder and click **Convert Entities** as noted.



Add 3 additional **lines** as shown below.

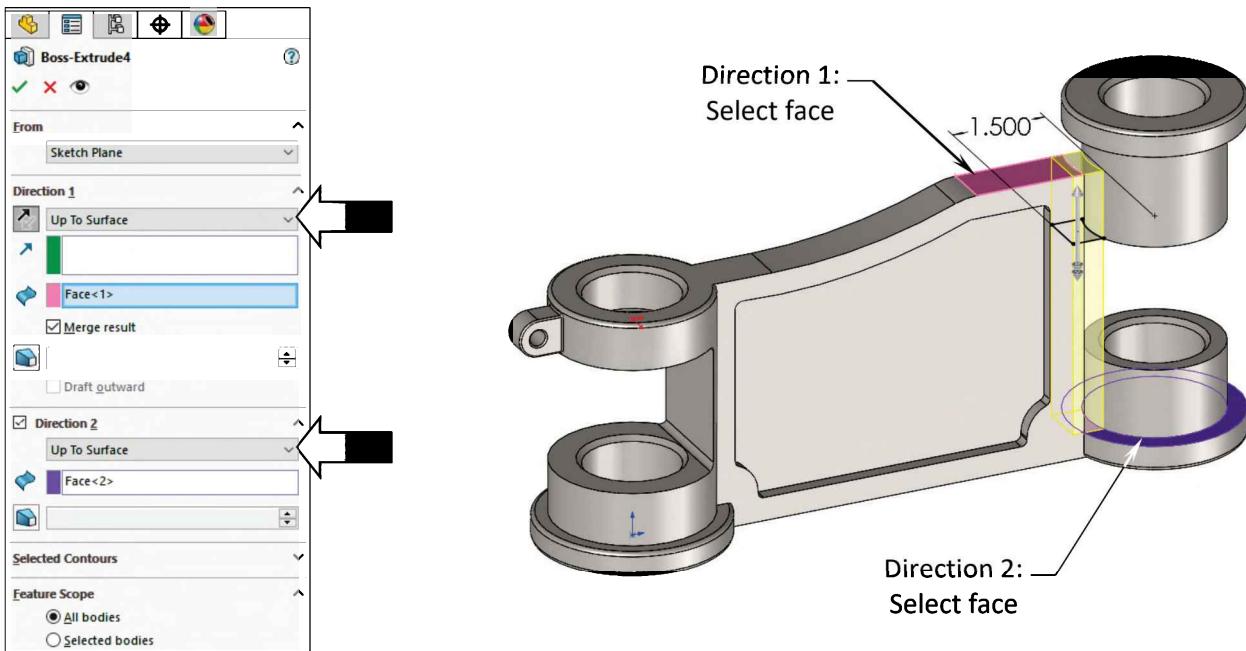
Trim the circle and add the **1.500** dimension.

Add **Collinear** relations between the lines and the model edges.



Switch to the **Features** tab and select **Extruded Boss-Base**.

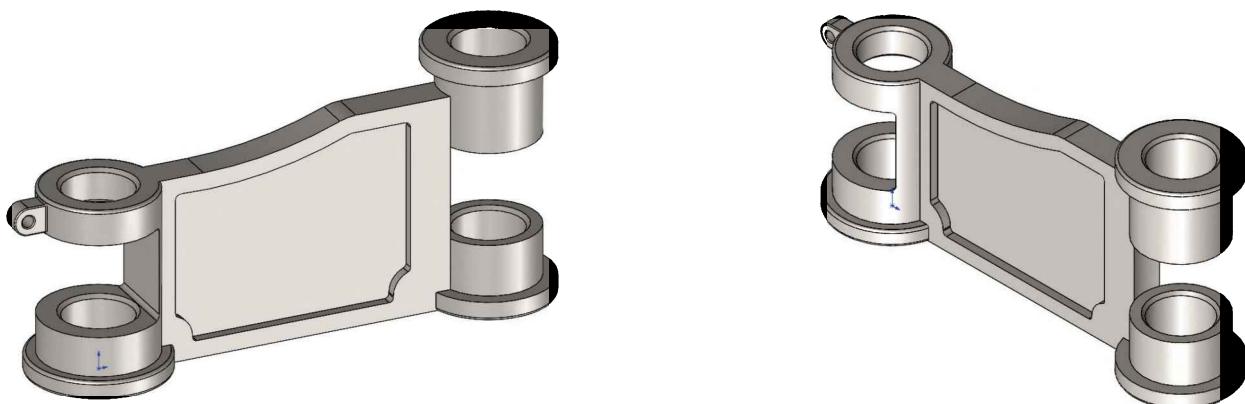
For Direction 1, select **Up-to-Surface** and click the face as noted.



For Direction 2, also select **Up-to-Surface** and click the face indicated.

Click **OK**.

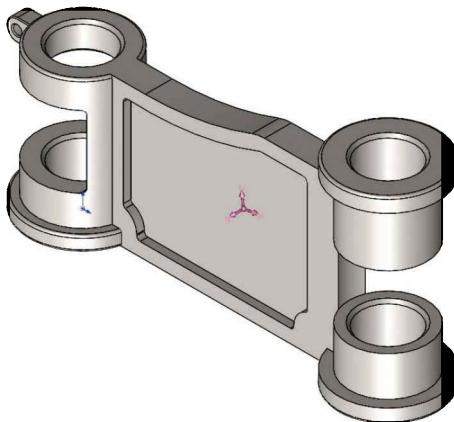
The extruded feature joins the 2 solid bodies into a single body.
Rotate the model and examine the model from different angles.



8. Calculating the mass:

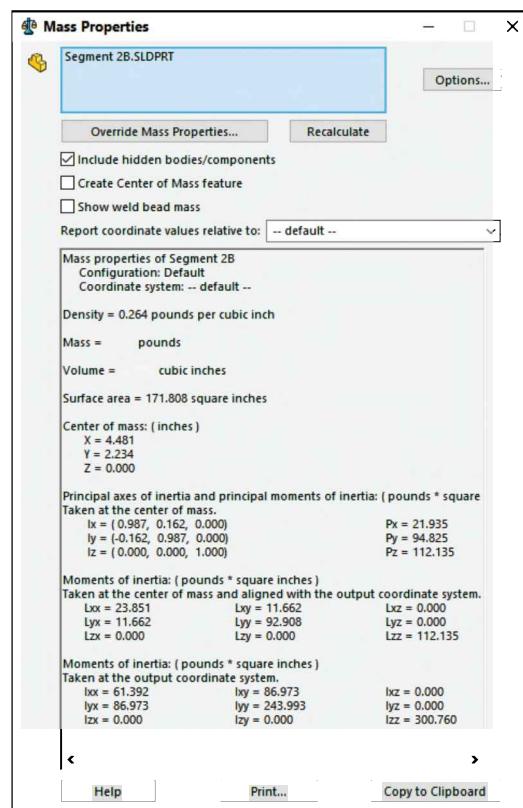
Switch to the **Evaluate** tab.

Click **Mass Properties**.



Enter the mass of the part here:

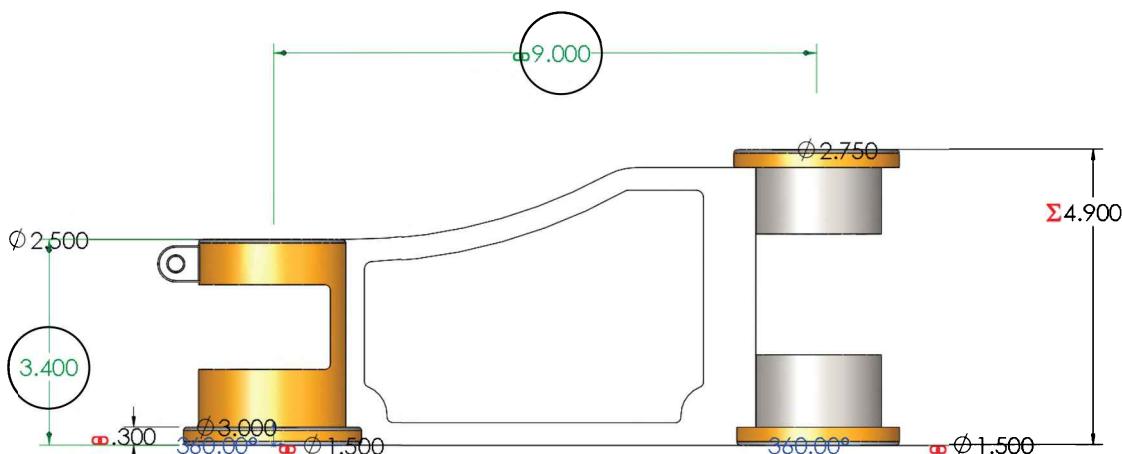
_____ pounds.



9. Making the final changes:

Change the dimension **8.500** in the **Revolve2** to **9.000** (circled).

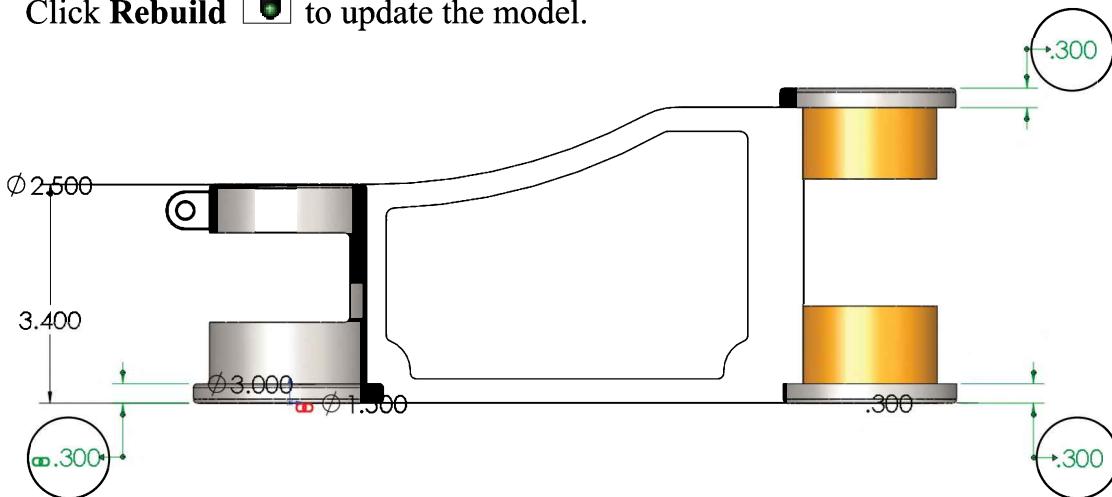
Change the dimension **4.000** in the **Revolved1** to **3.400** (circled).



Change the dimension **.400** in the **Revolved1** to **.300** (circled).

Also change both of the dimension **.400** in the **Cut Extrude4** to **.300** (circled).

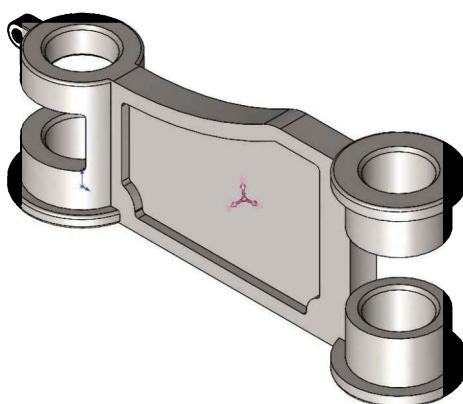
Click **Rebuild**  to update the model.



10. Calculating the mass:

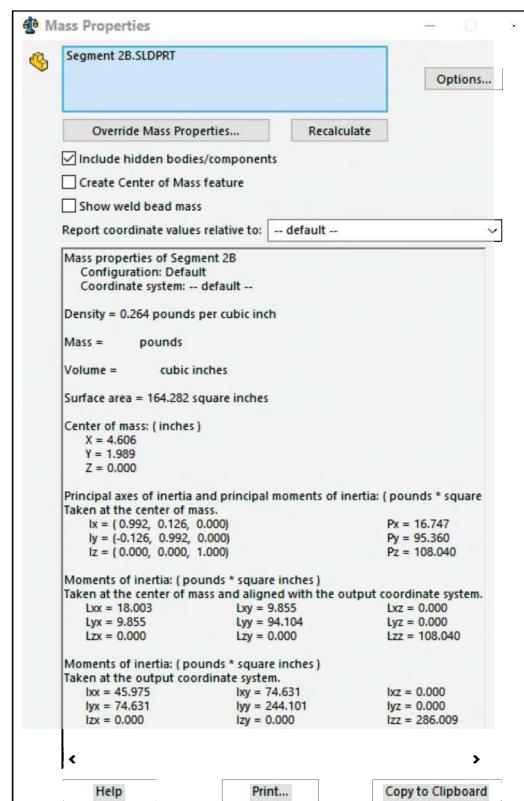
Switch to the **Evaluate** tab.

Click **Mass Properties**.



Enter the mass of the part here:

_____ pounds.

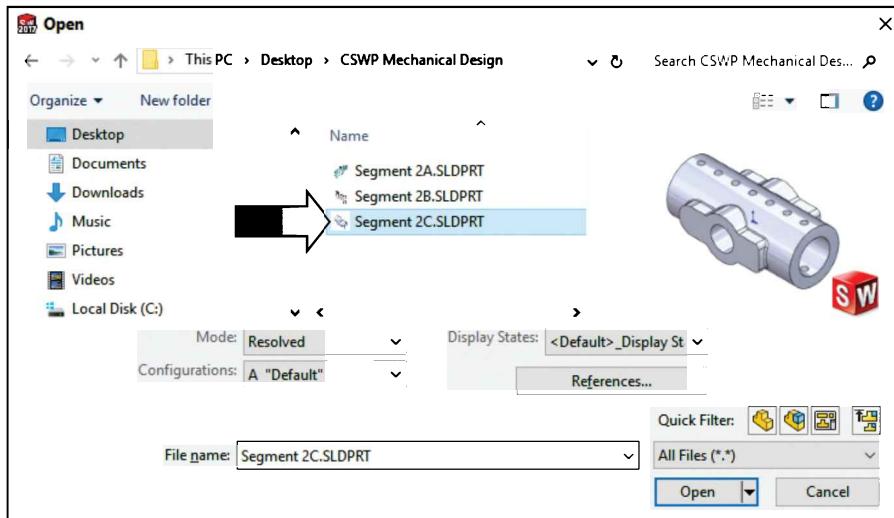


Save your model as **Segment 2B (Completed)** and close the part document.

1. Opening a part document: (3 of 3)

Click **File / Open**.

Locate and open the part document named: **Segment 2C.sldprt**

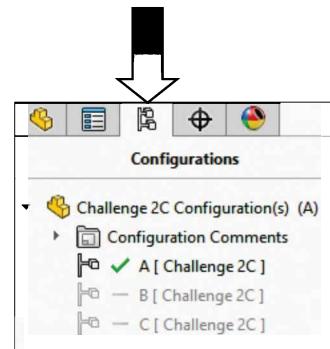


(This portion of the test will examine your skills on the modification of existing configurations, as well as creating new configurations in a part.)

2. Locating the configurations:

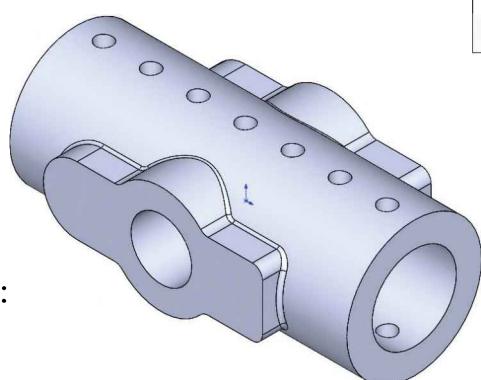
Switch to the **ConfigurationManager** (arrow).

How many configurations are there in this model?



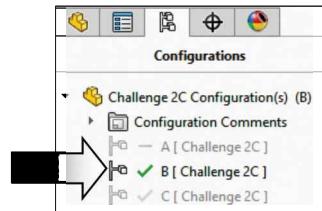
Enter the number of configurations found here:

_____ (1, 2, or 3).



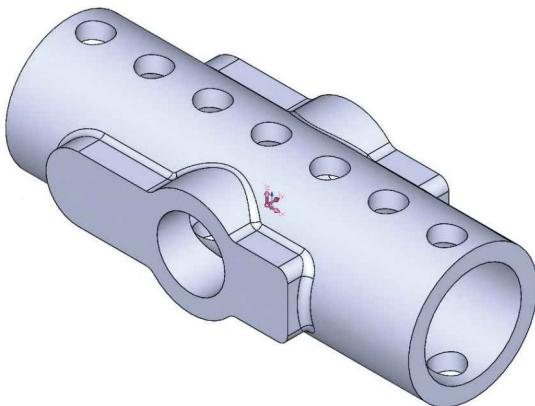
3. Calculating the mass:

Double-click configuration **B** to activate it.
Each configuration has its own material and different feature sizes.



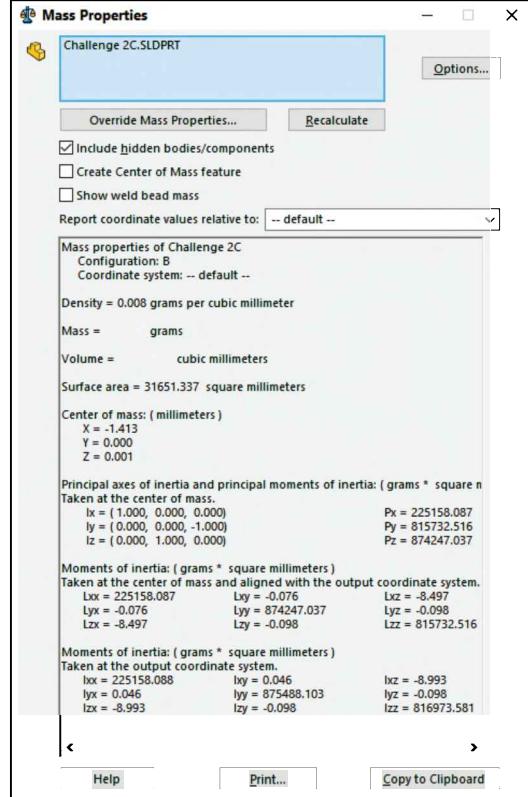
Switch to the **Evaluate** tab.

Click **Mass Properties**.



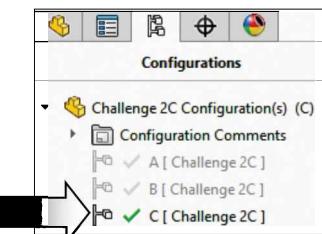
Enter the mass of **Configuration B** here:

_____ pounds.



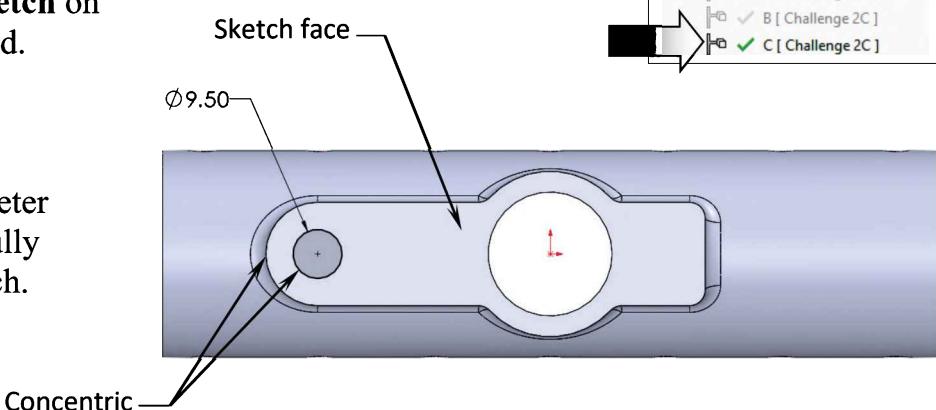
4. Adding a hole:

Double-click **Configuration C** to activate it.



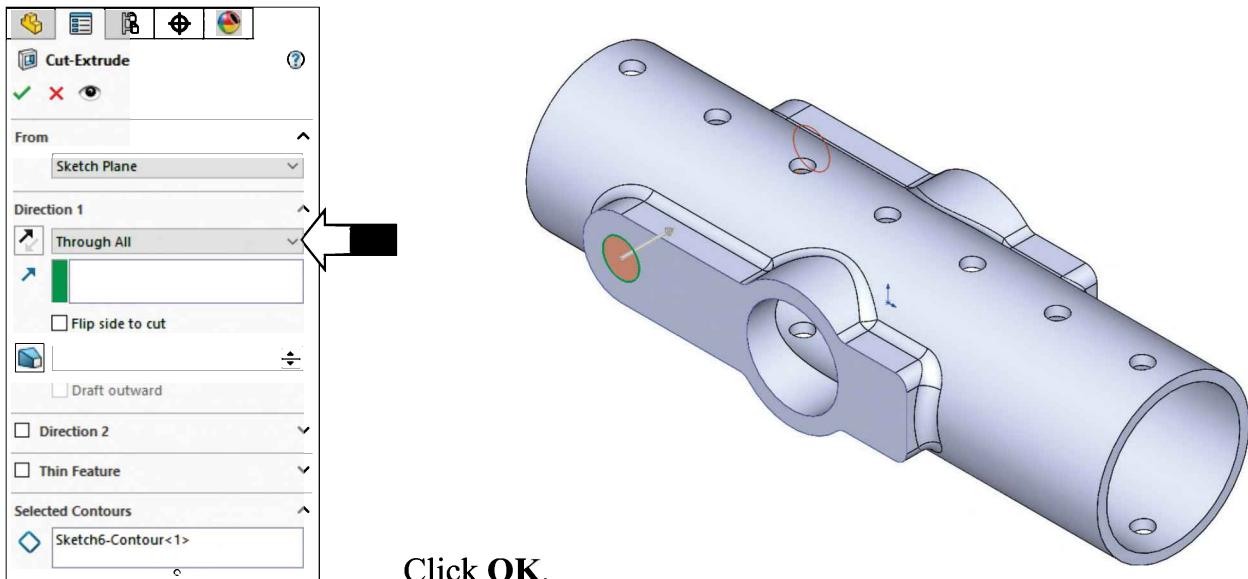
Open a new sketch on the face as noted.

Sketch a **circle** and add a diameter dimension to fully define the sketch.



Switch to the **Features** tab and click **Extruded Cut**.

For Direction 1, select **Through All**.

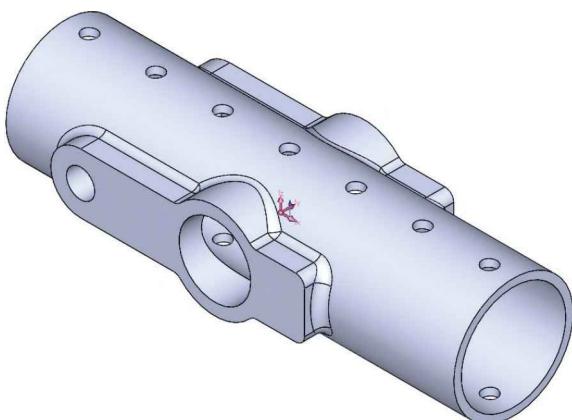


Click OK.

5. Calculating the mass:

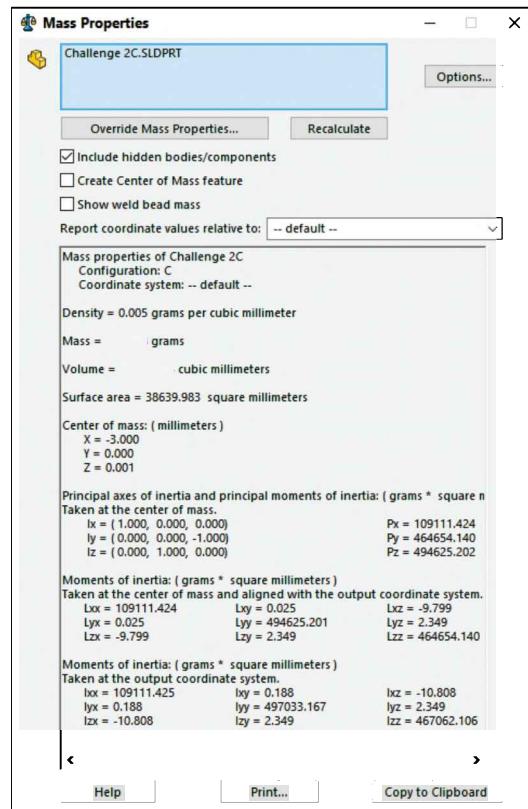
Switch to the **Evaluate** tab.

Click **Mass Properties**.



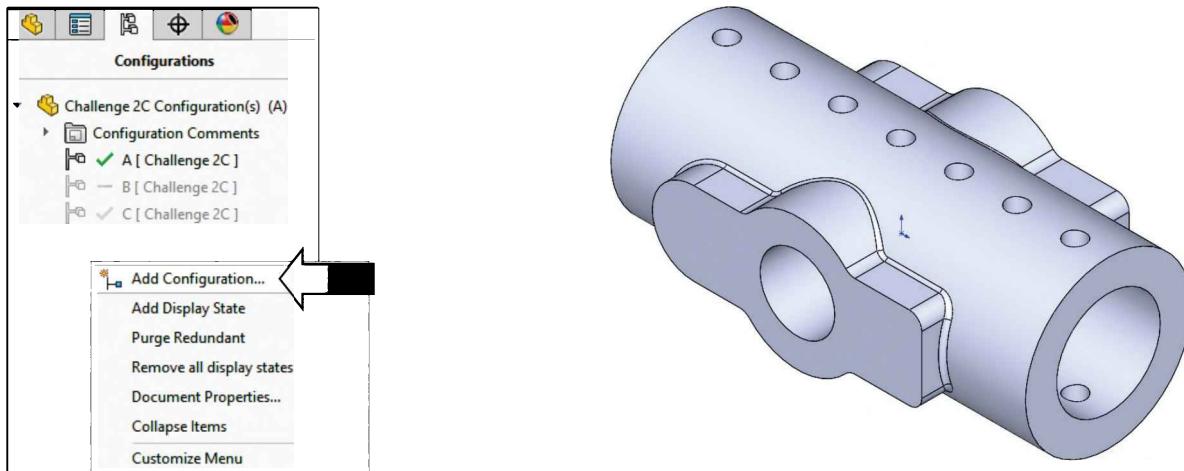
Enter the mass of **Configuration C** here:

_____ pounds.

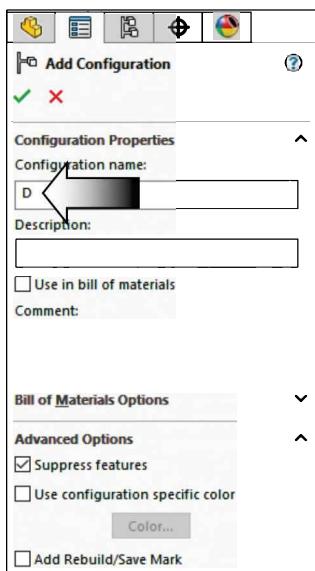


6. Creating a new configuration:

Double-click **Configuration A** to activate it.



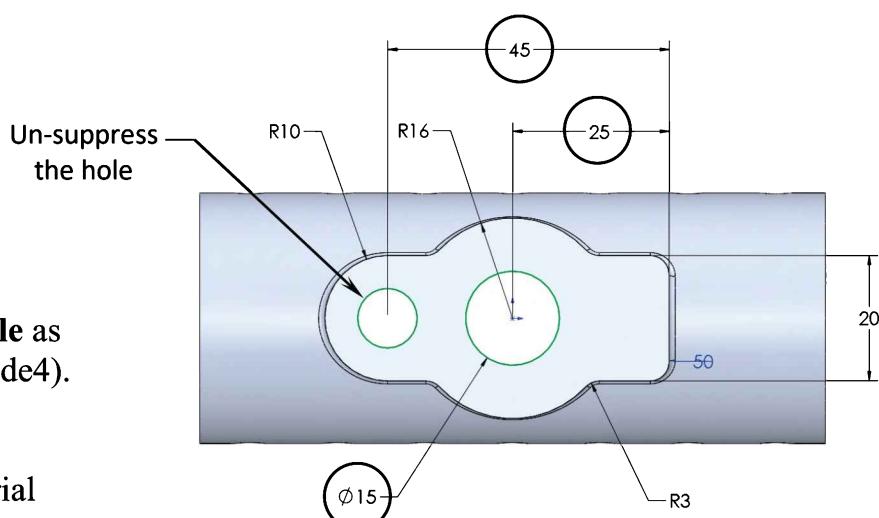
Right-click the name of the part and select:
Add Configuration.



Enter **D** for the name of the new configuration.

Change the 3 dimensions to match the values in the circles.

Un-suppress the **hole** as noted (in Cut-Extrude4).



Use the same material **1060 Alloy** for the new configuration.

The hole (Cut-Extrude4) should be unsuppressed in the configurations C and D.

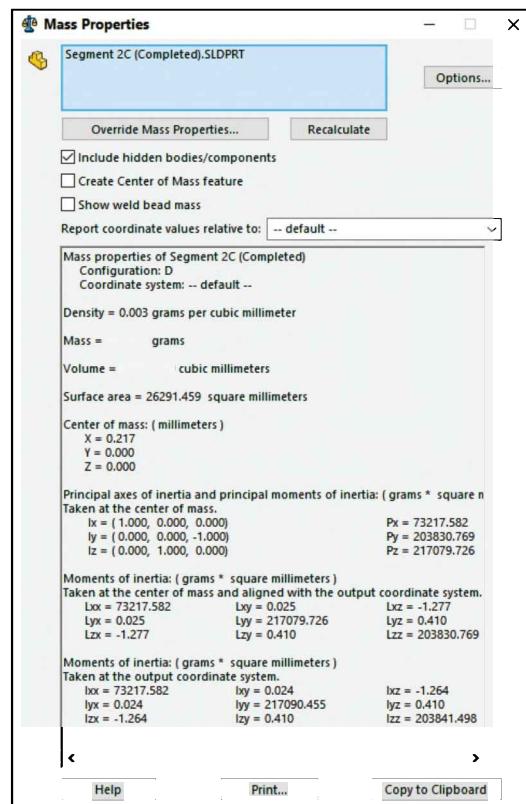
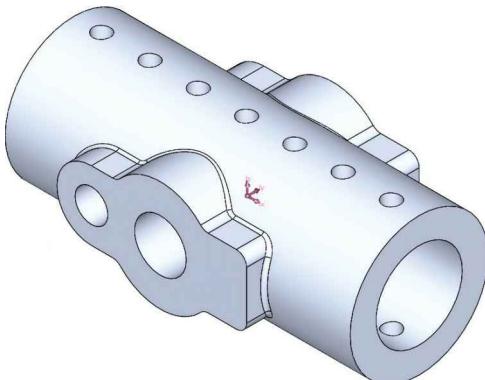
The hole (Cut-Extrude4) should be suppressed in the configurations A and B.

7. Calculating the mass:

Remain in Configuration D.

Switch to the Evaluate tab.

Click Mass Properties.



Enter the mass of configuration D here:

_____ pounds.

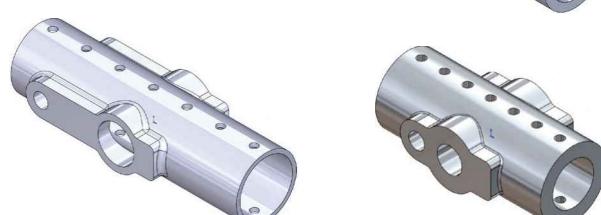
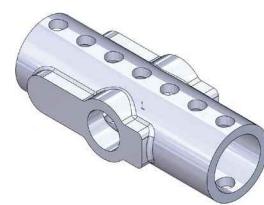
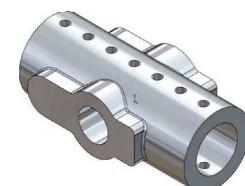
8. Saving your work:

Select File, Save As.

Enter: Segment 2C (Completed) for the file name.

Click Save.

Close all part documents.



Certified-SOLIDWORKS-Professional (CSWP)

Certification Practice for the Mechanical Design Exam

Challenge III: Bottom Up Assembly & Mates

Complete this challenge within 80 minutes

(The following examples are intended to assist you in familiarizing yourself with the structures of the exams and the method in which the questions are asked.)

SOLIDWORKS 2010 or newer is required for this exam.

Unit: **Inches, 3 decimals**

Drafting Standards: **ANSI**

Origin: **Arbitrary**

Material: **Already Specified for each part**

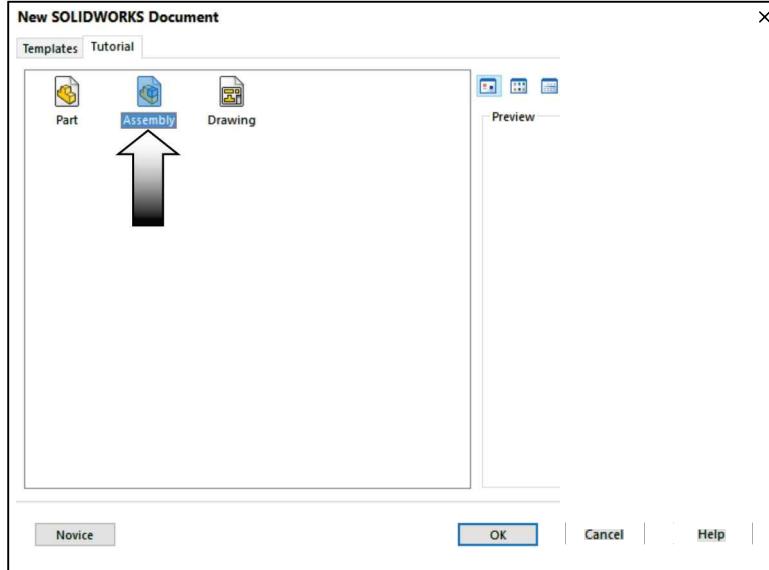
(This portion of the test will examine your skills on constraining components using the Bottom Up Assembly approach.)

1. Starting a new assembly document:

Click File / New.

Select an Assembly Template and click OK.

Change the Drafting Standard to **ANSI** and the Units to **IPS, 3 decimal places**.

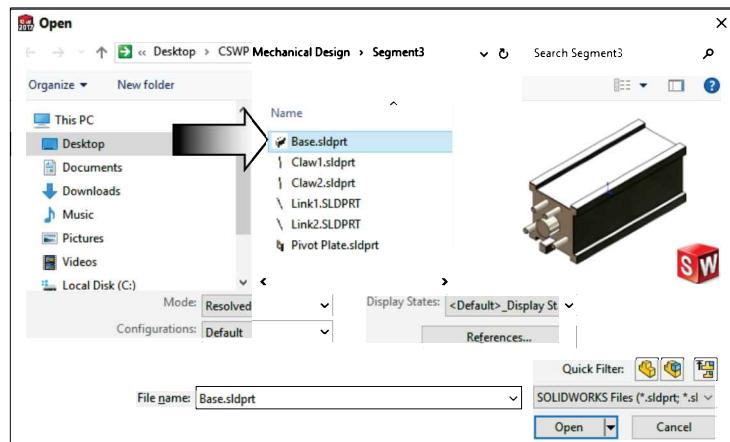


You will be asked to demonstrate the use of different mate types to constrain the component in an assembly.

2. Inserting the 1st component:

Change to the Assembly tab and select:
Insert Component.

Locate the component named **Base.sldprt** and click **Open**.

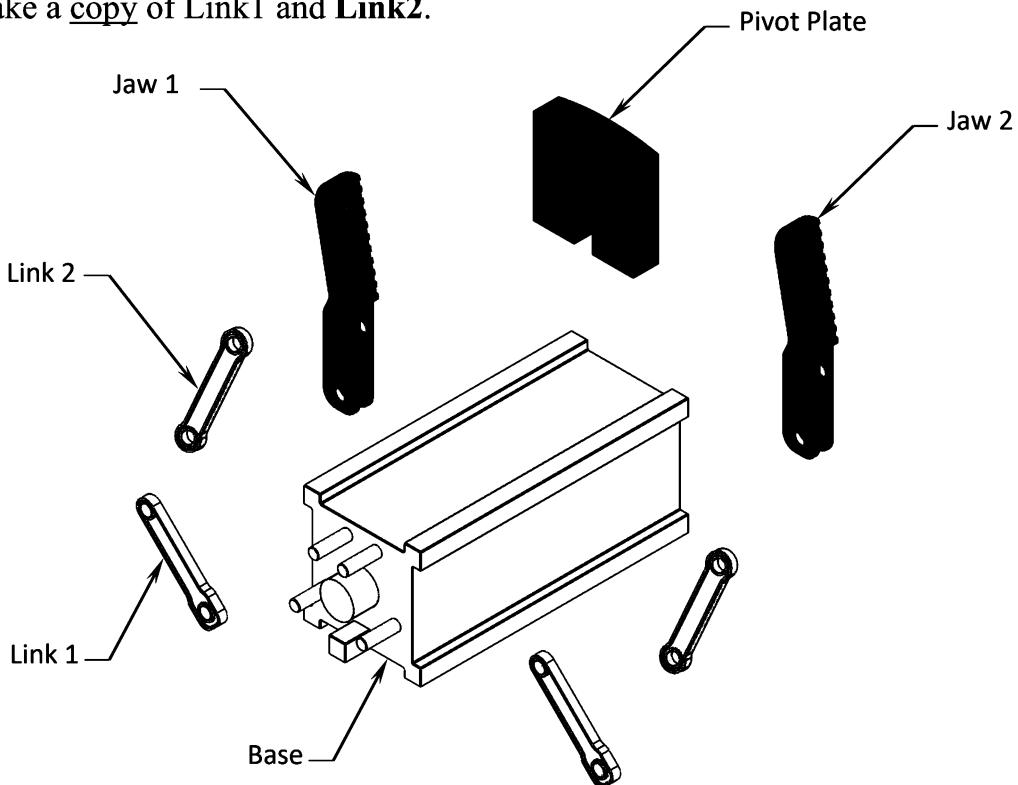


Click the **Green check** to place the component on the Origin.

3. Inserting other components:

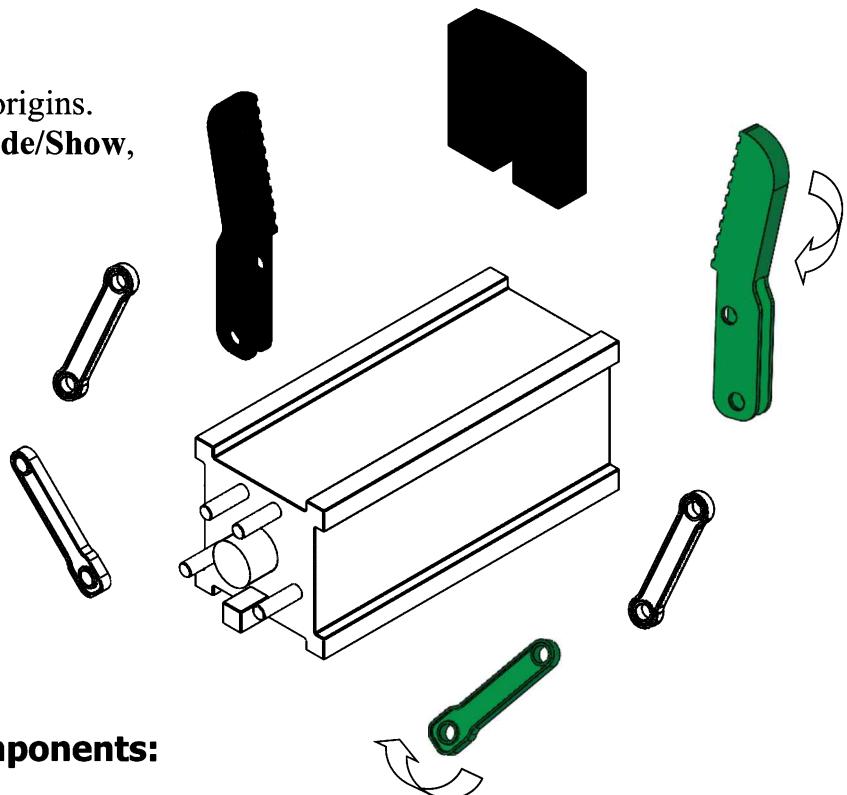
Insert all other components into the assembly.

Place the components approximately as shown.
Make a copy of **Link1** and **Link2**.



Rotate the **Jaw** and the **Link** ahead of time to make it easier to see and to mate them to the **Base**.

Also Hide the origins.
Click **View, Hide/Show, Origins**.



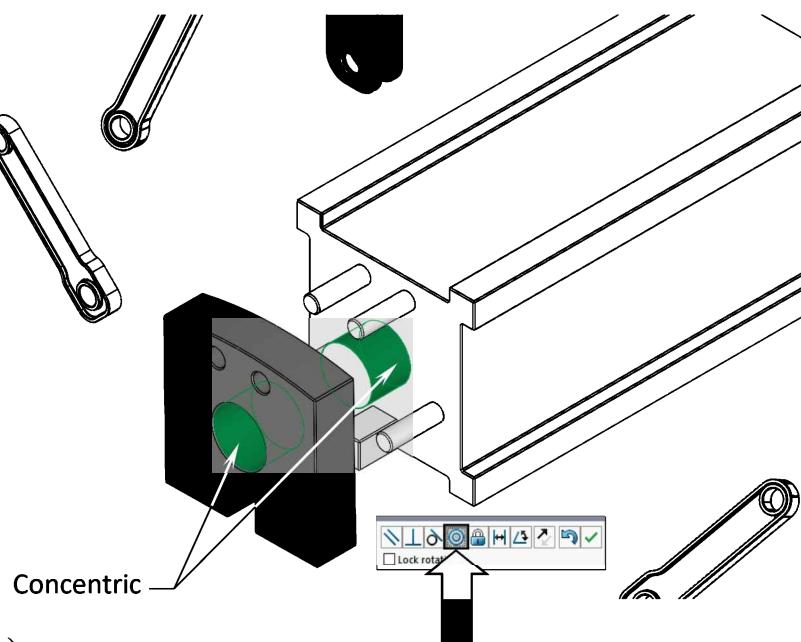
4. Mating the components:

The **F5** function key enables the **Selection Filters**. Use the **Filter Faces** (or press the letter **X** shortcut key) to help select the faces more easily.

Hold the **Control** key and select the hole and the shaft as noted.

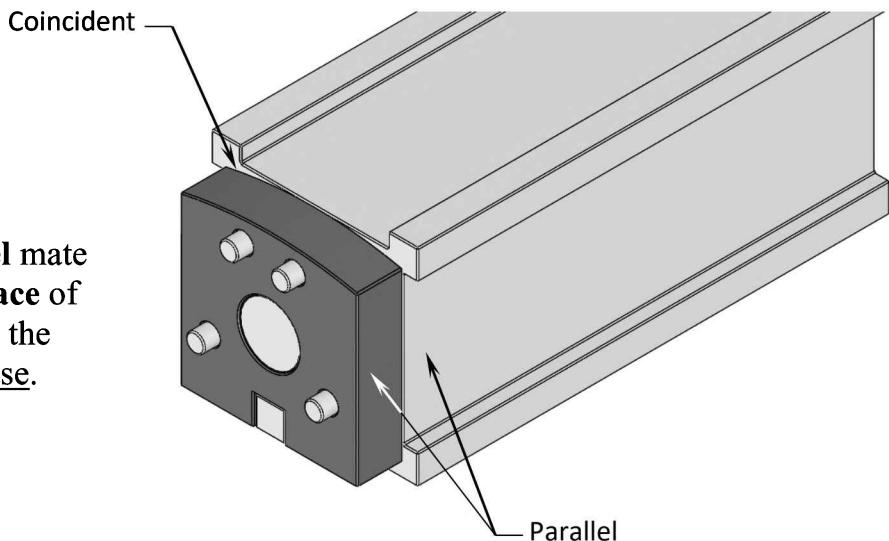
Release the control key to see the mate shortcuts popup options.

Select **Concentric** (arrow) and click the **Green check** (OK).



Add a **Coincident** mate between the **back face** of the Pivot Plate and the **front face** of the Base.

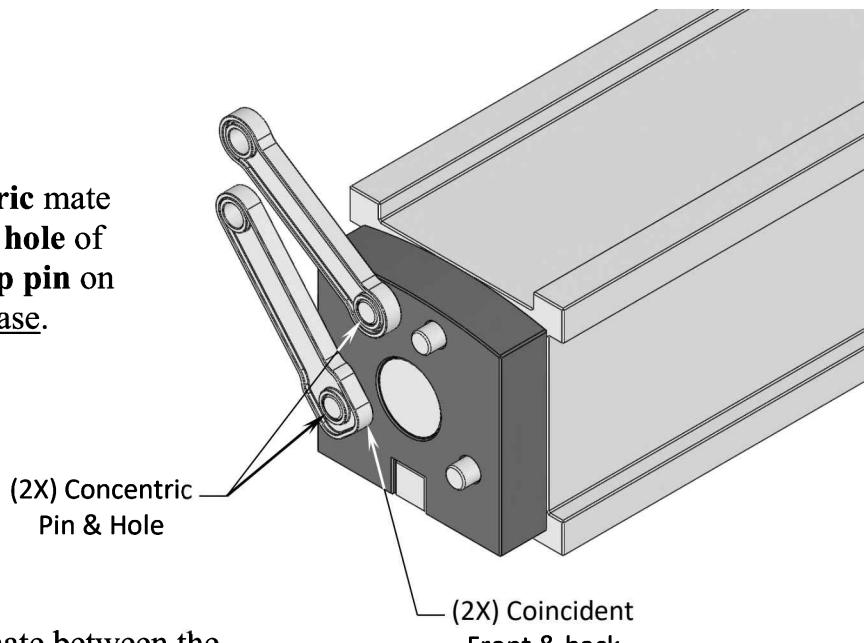
Also add a **Parallel** mate between the **side face** of the Pivot Plate and the **side face** of the Base.



Add a **Concentric** mate between the **bottom hole** of the Link1 and the **bottom pin** on the left side of the Base.

Also add a **Concentric** mate between the **bottom hole** of the Link2 and the **top pin** on the left side of the Base.

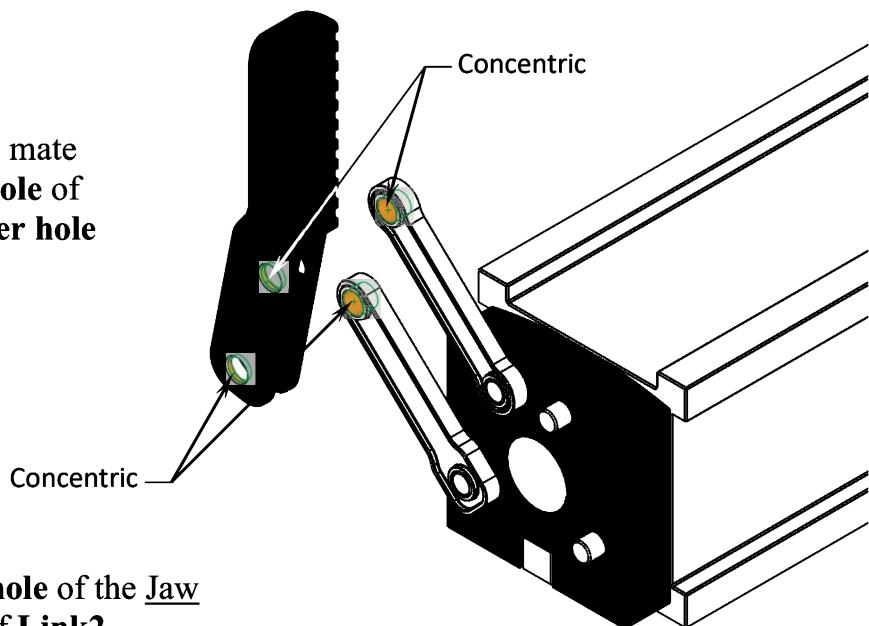
Add a **Coincident** mate between the **back face** of the Link1 and Link2 and the **front face** of the Base.



5. Assembling the 1st Claw:

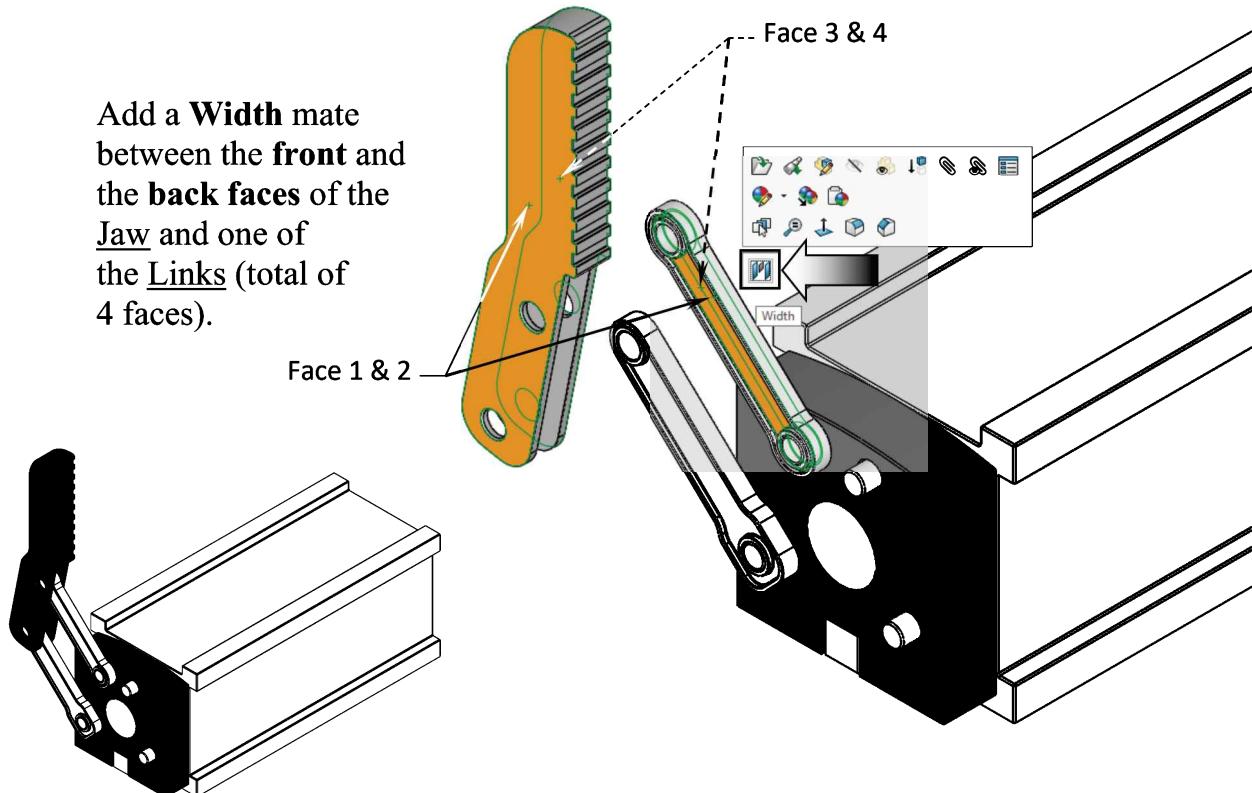
Zoom closer to the Jaw and the Links components.

Create a **Concentric** mate between the **lower hole** of the Jaw and the **upper hole** of **Link1**.



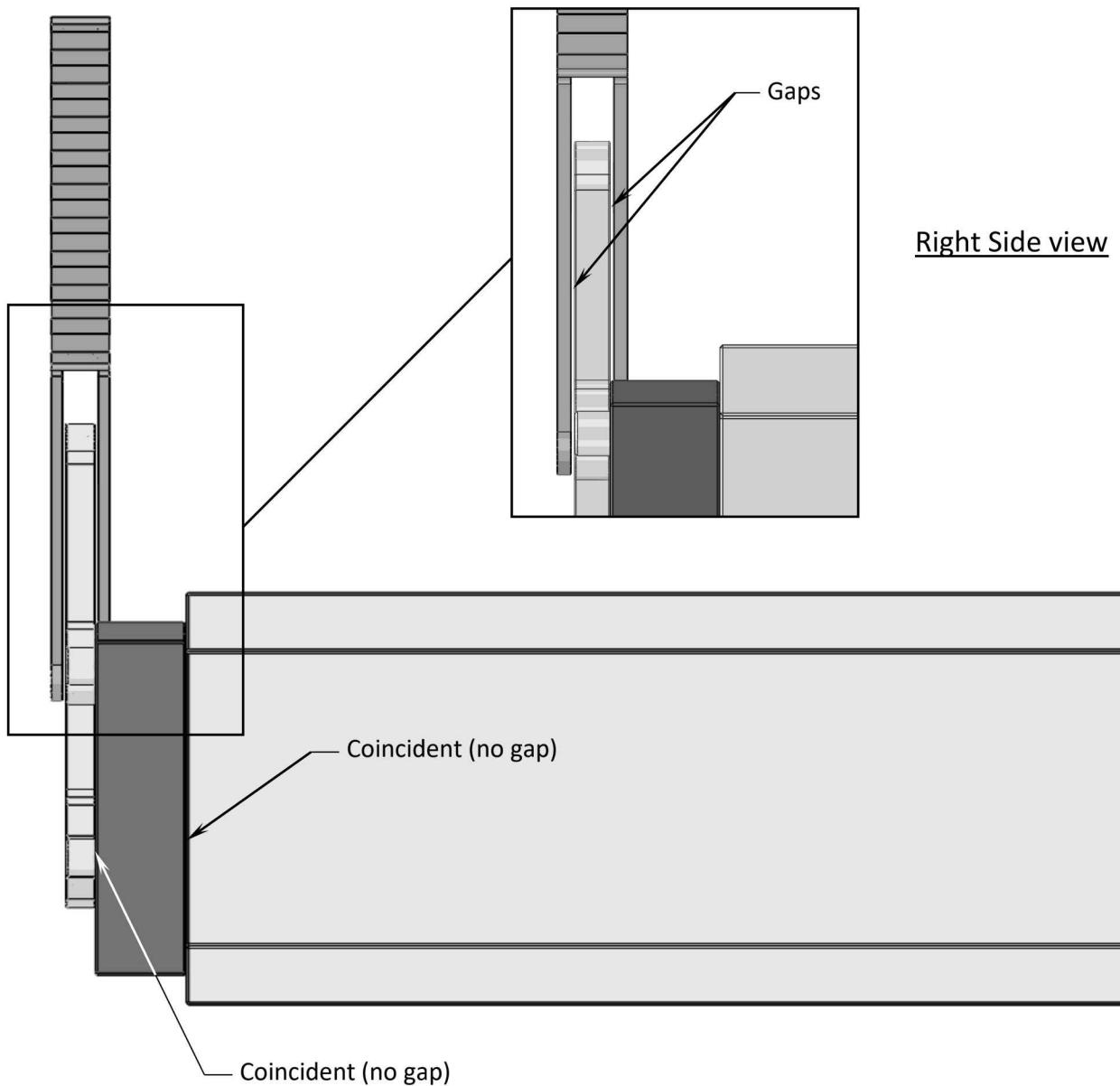
Create another **Concentric** mate between the **upper hole** of the Jaw and the upper hole of **Link2**.

Add a **Width** mate between the **front** and the **back faces** of the Jaw and one of the Links (total of 4 faces).



Change to the **Right** orientation (Control+4).

Zoom very close and examine the gaps between the Jaw and the 2 Links.
The gaps must be exactly the same on both sides.



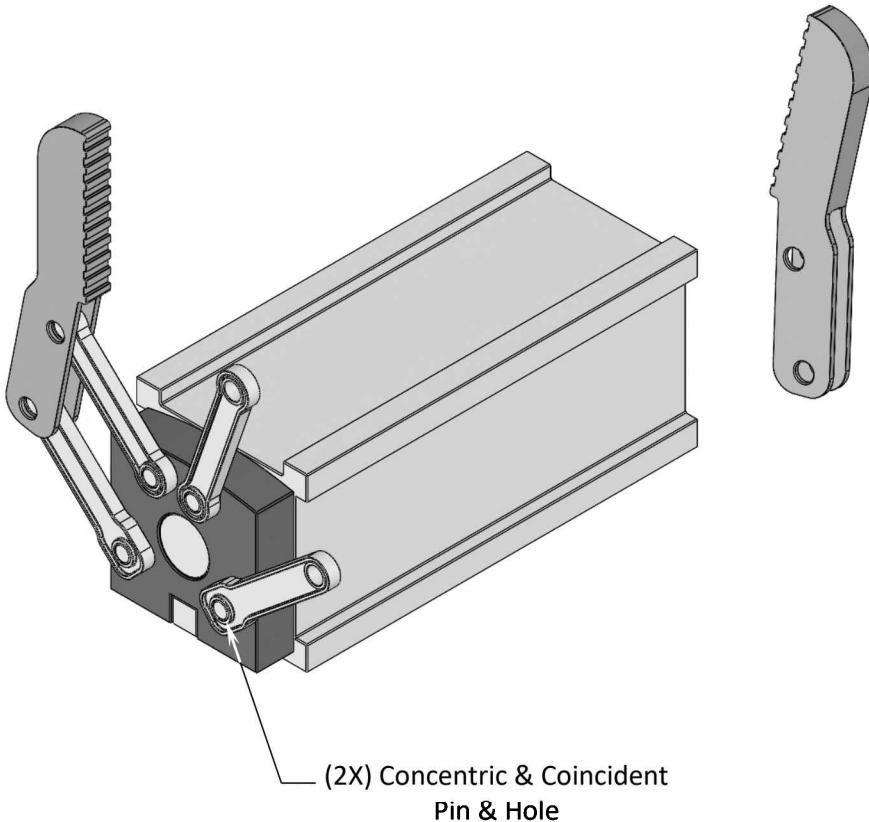
Check the back side of the 2 Links to ensure it is Coincident to the Pivot Plate.

The back side of the Pivot Plate should also be Coincident to the Base.

6. Assembling the 2nd Jaw:

Add a **Concentric** relation and a **Coincident** relation between the Link1 and the Lower Pin in the Base.

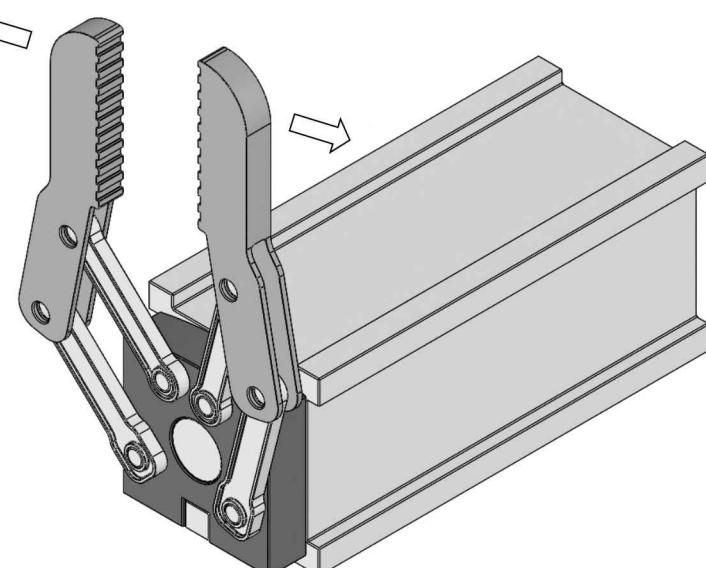
Add the same
Concentric
and **Coincident**
relations
between the
Link2 and
the Upper Pin
in the Base.



Also add a **Width** mate
between the **Jaw2** and
one of the **Links**.

(Use the front & back
faces of the **Jaw2**,
and the front & back
faces of the **Link**.)

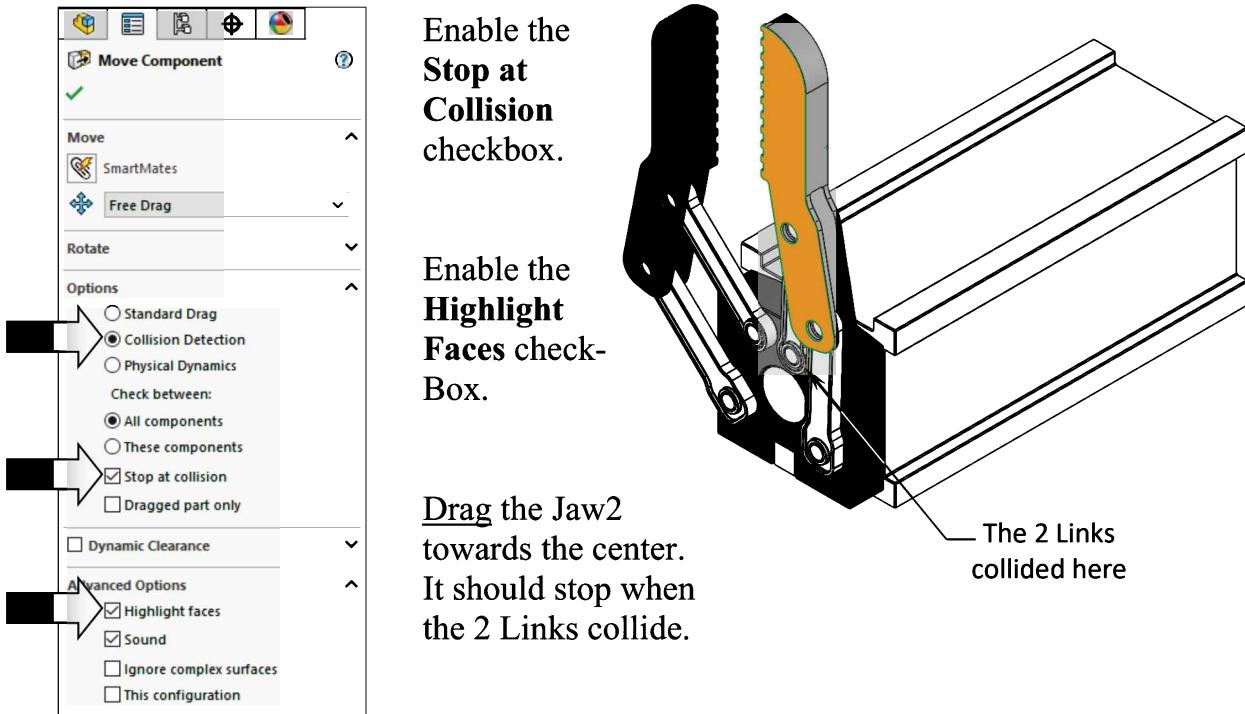
Move the 2 Jaws back and forth to ensure that they can be moved freely.



7. Detecting Collision:

Switch to the **Assembly** tab and click the **Move Component**  command.

Expand the **Options** section and select the **Collision Detection** option (arrow).



Click **OK** but do not move the Jaw2.

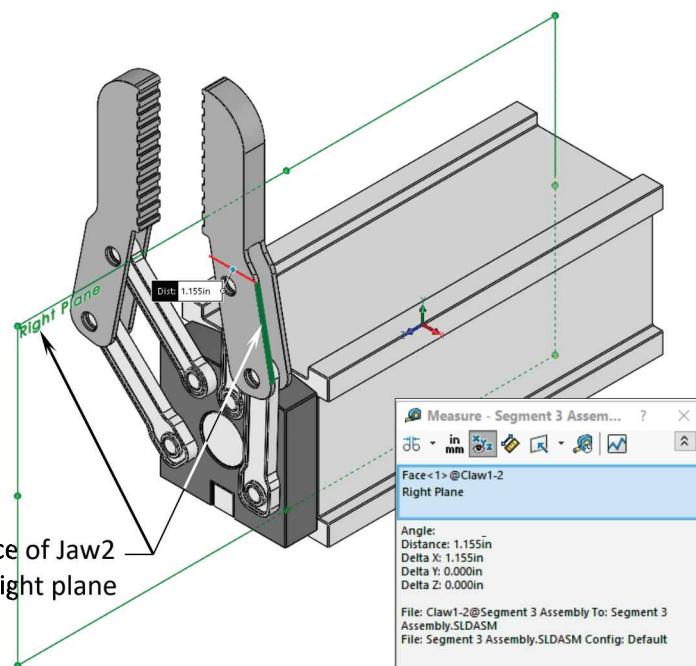
8. Measuring the angle:

Switch to the **Evaluate** tab.

Select the **narrow face** on the side of the Jaw2 and the **Right** plane of the Assembly.

Locate the **Angle Measurement** and enter it here:

_____ degrees.



9. Moving components with Collision Detection:

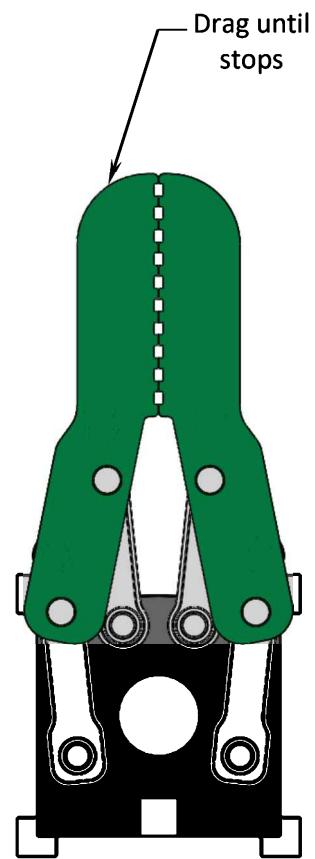
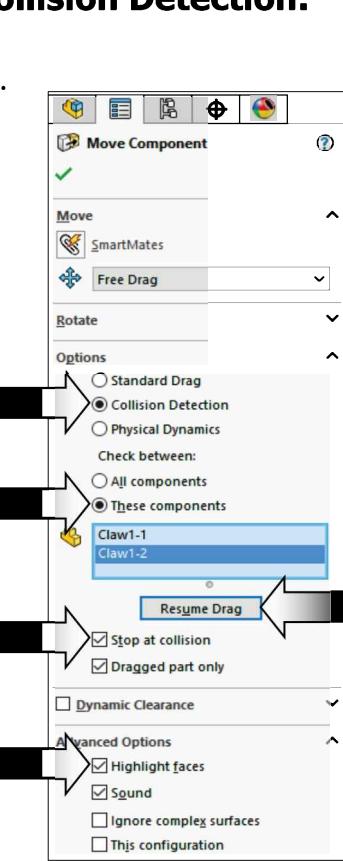
Click **Move Component** .

Select the following options:

- * **Collision Detection**
- * **These Components**
(select the 2 Jaws)
- * **Stop at Collision**
- * **Highlight Faces**

Click the **Resume Drag** button and drag one of the Jaws until it stops.

It is important to keep the Jaws at the collided position for measurement.



10. Calculating the Center of Mass:

Switch to the **Evaluate** tab.

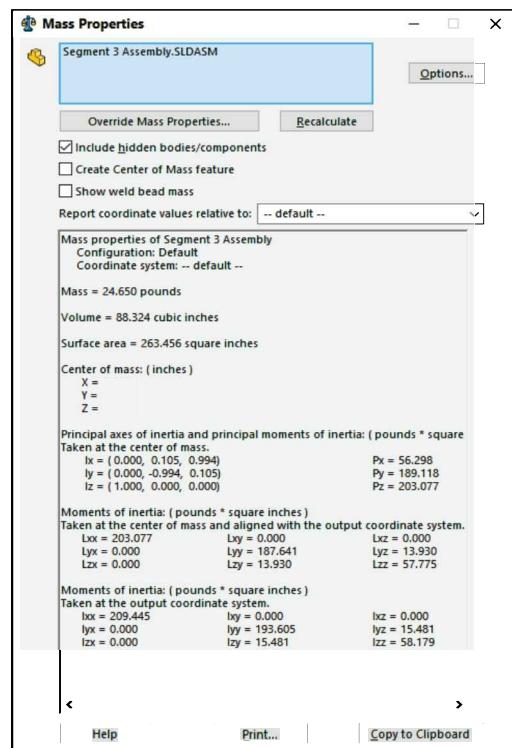
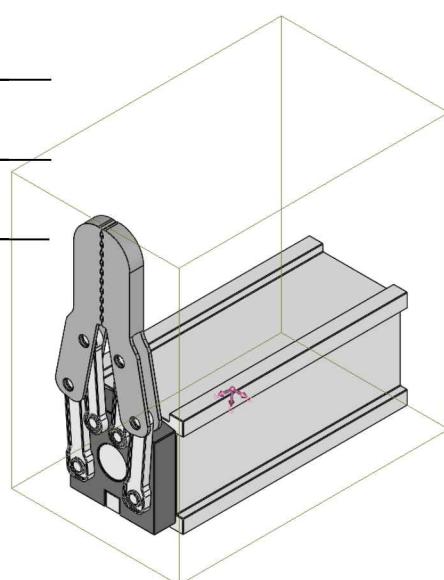
Click **Mass Properties**.

Enter the **Center of Mass** here:

X = _____

Y = _____

Z = _____



11. Adding a Gear mate:

Change to the **Front** view.

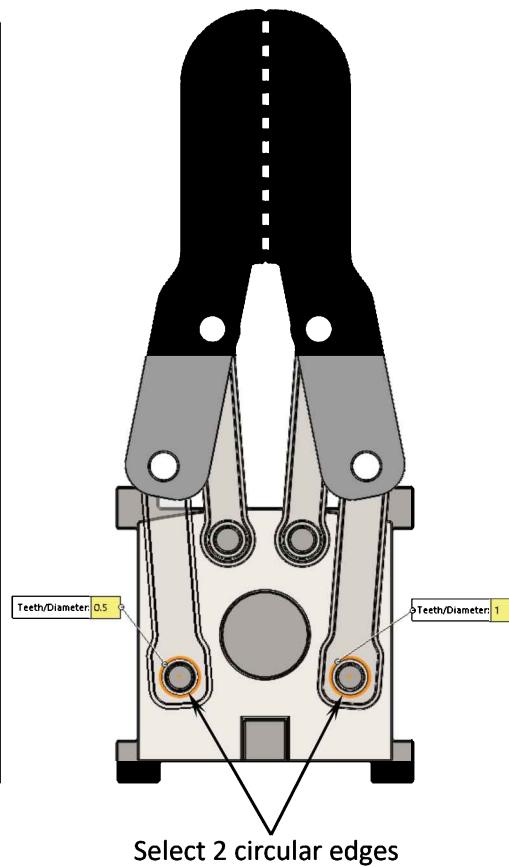
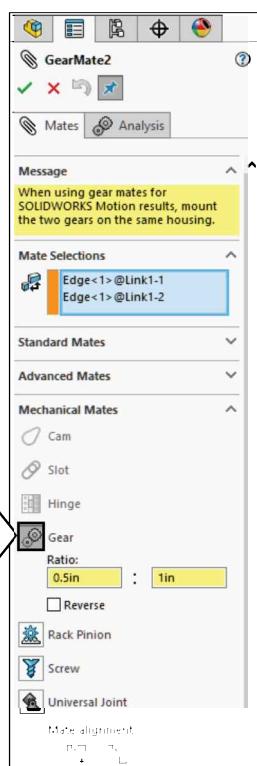
Click **Mate** and expand the Mechanical Mate option.

Click **Gear**.

Select the **2 circular edges** as indicated.

Enter the ratio **0.5in** for the left side and **1.0in** for the right side.

Click **OK**.



12. Adding an angle mate:

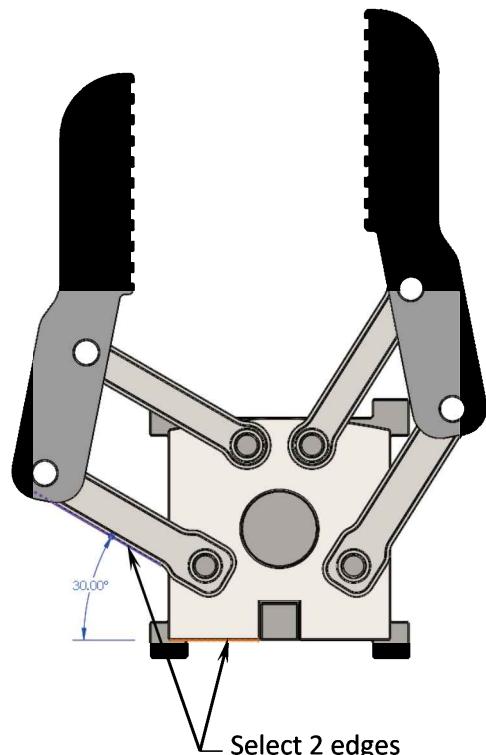
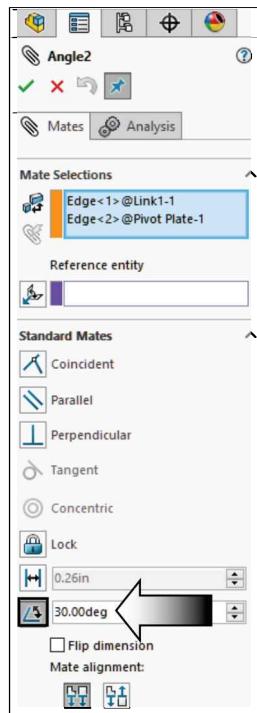
Remain in the **Front** view orientation.

Click **Mate** and select the Standard Mates option.

Select the **Angle** option and enter **30°** for the angle.

Select the 2 edges of the **Link1** and the **Base** as noted.

Click **OK**.



13. Measuring the angle:

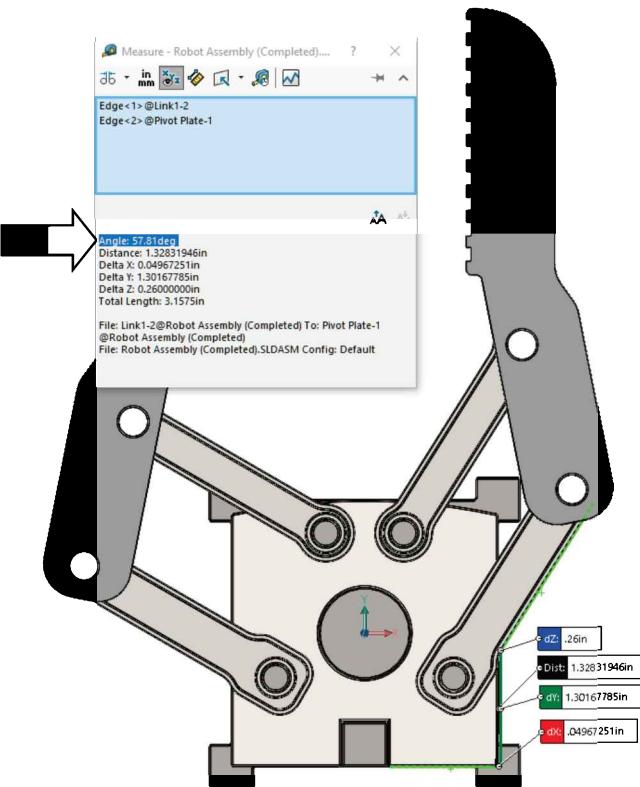
Switch to the **Evaluate** tab.

Click **Measure**.

Select the 2 edges of the **Link1** and the **Base** as indicated

Enter the angle below:

_____ deg.

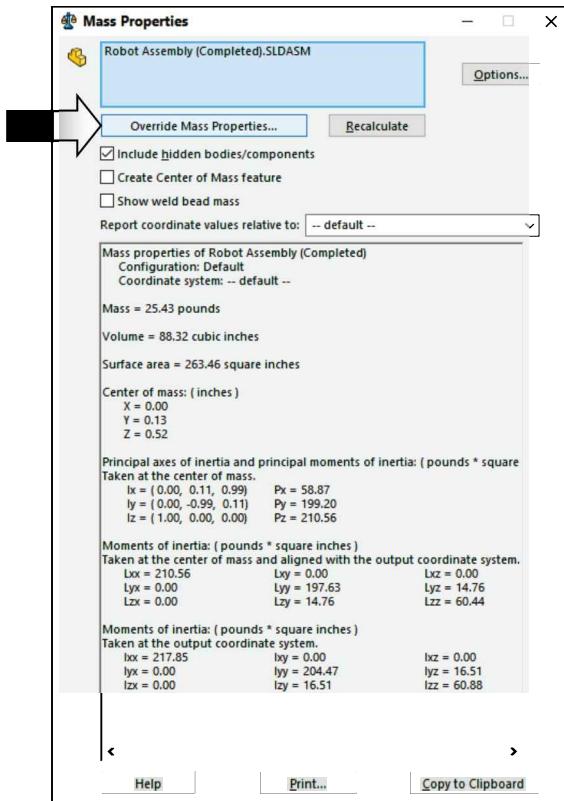


14. Modifying the Mass:

This step will modify the values of the Mass, Center of mass, and Moments of Inertia of the model.

Click **Mass Properties**.

Click the **Override Mass Properties** button (arrow).



Enable the **Override Mass** checkbox.

Enter **12.50lbs**.

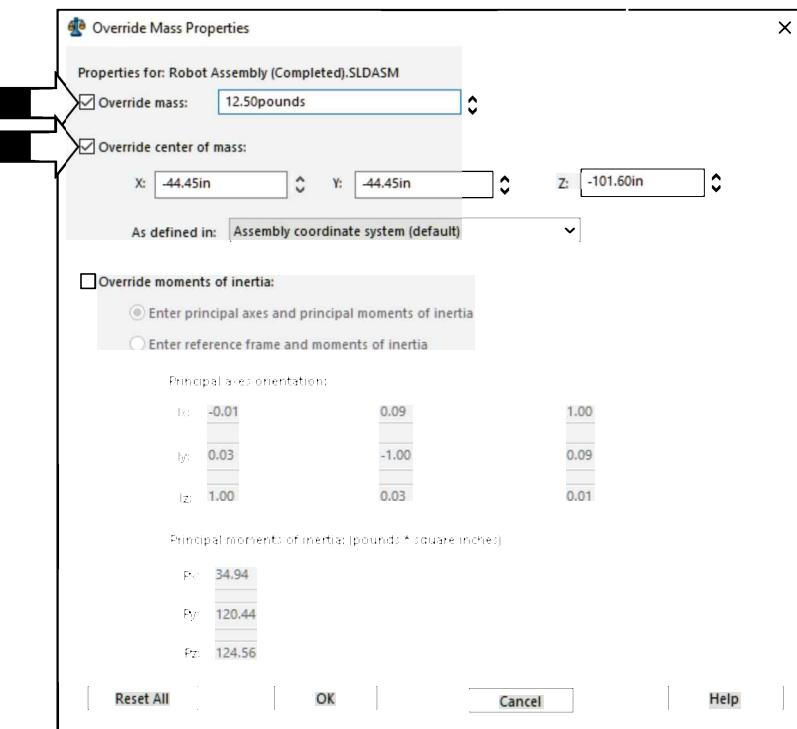
Enable the **Override Center of Mass** checkbox.

Enter the following:

X = -44.45in

Y = -44.50in

Z = -101.60in



Click **OK**.

15. Calculating the modified Center of Mass:

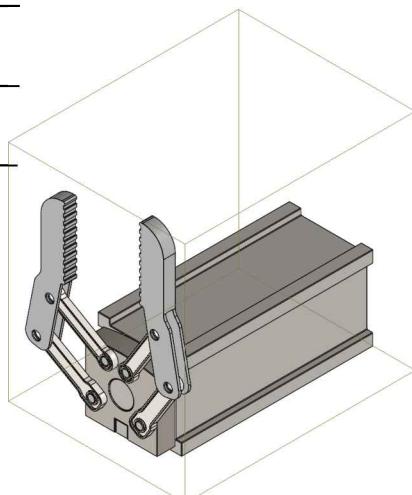
Click the **Recalculate** button.

Enter the modified Center of Mass below:

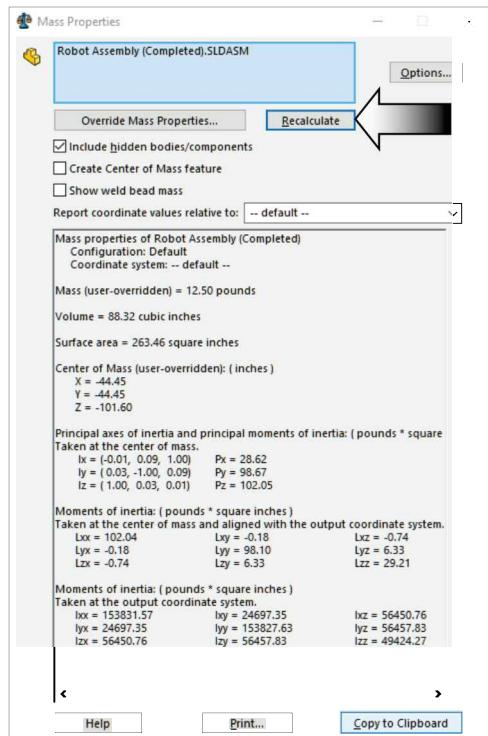
X = _____

Y = _____

Z = _____



Close the Mass Properties box.

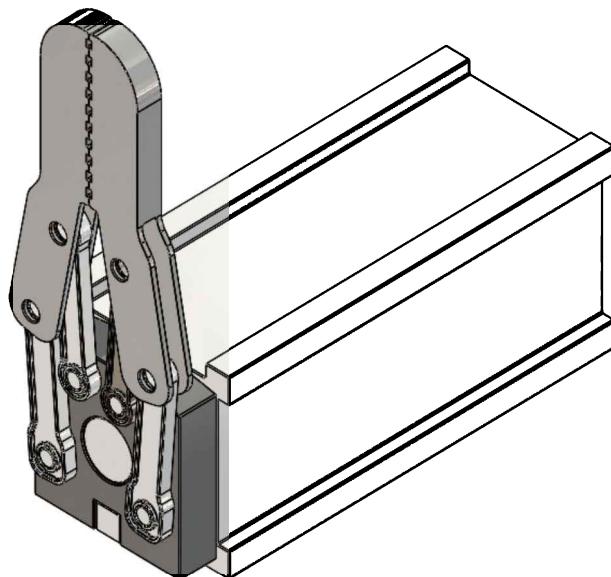
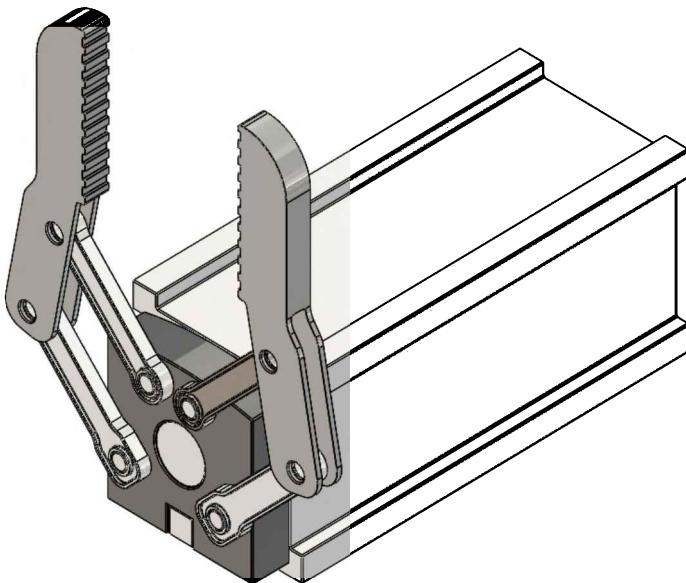


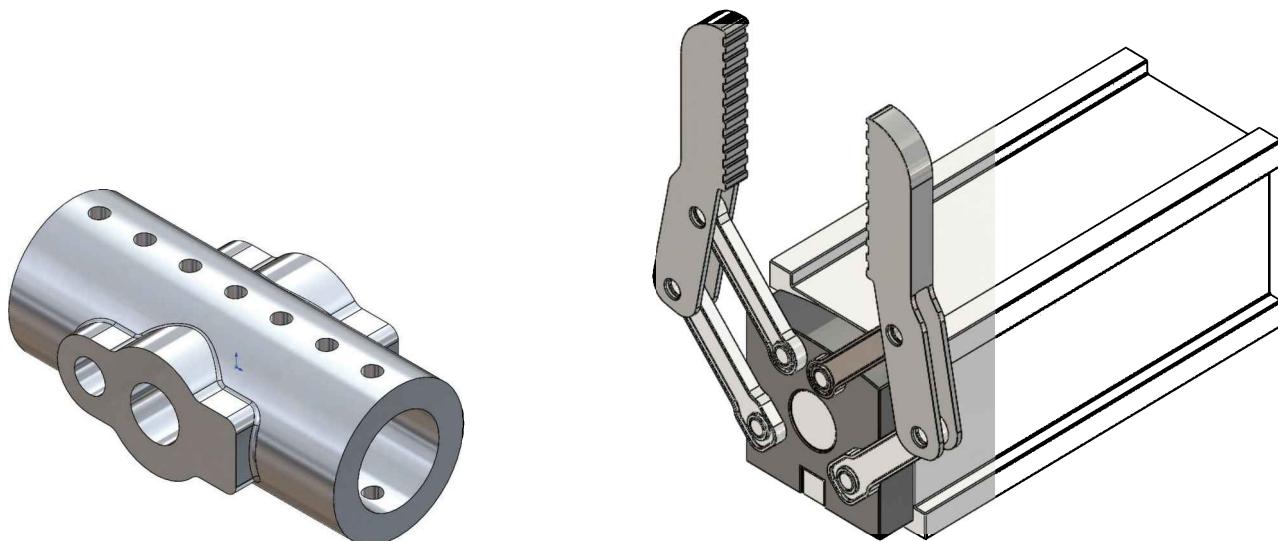
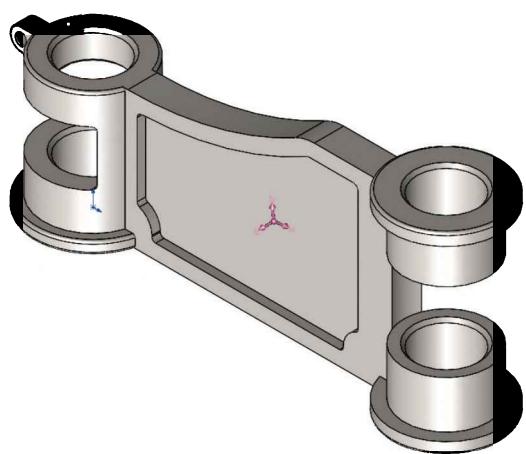
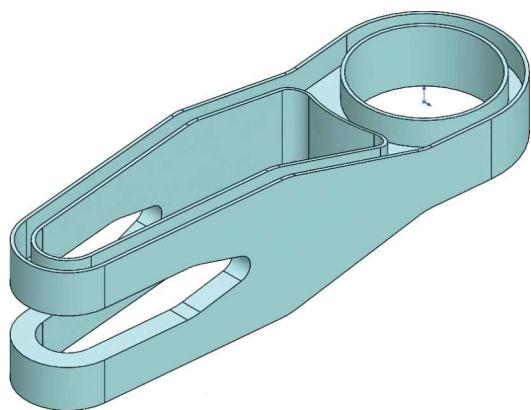
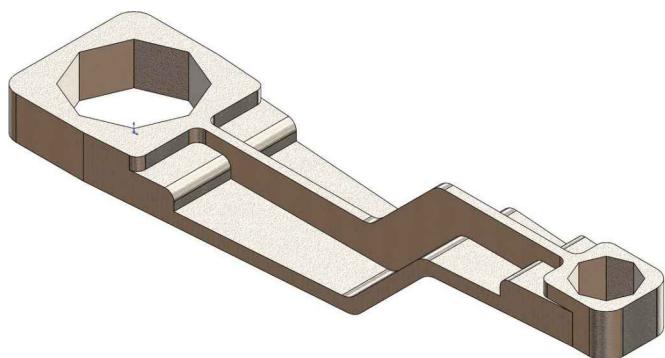
16. Saving your work:

Select **File, Save As.**

Enter: **Segment 3C (Completed)** for the file name.

Click **Save**. Close all part documents.





Glossary

Alloys:

An Alloy is a mixture of two or more metals (and sometimes a non-metal). The mixture is made by heating and melting the substances together.

Example of alloys are Bronze (Copper and Tin), Brass (Copper and Zinc), and Steel (Iron and Carbon).

Gravity and Mass:

Gravity is the force that pulls everything on earth toward the ground and makes things feel heavy. Gravity makes all falling bodies accelerate at a constant 32ft. per second (9.8 m/s). In the earth's atmosphere, air resistance slows acceleration. Only on airless Moon would a feather and a metal block fall to the ground together.

The mass of an object is the amount of material it contains.

A body with greater mass has more inertia; it needs a greater force to accelerate.

Weight depends on the force of gravity, but mass does not.

When an object spins around another (for example: a satellite orbiting the earth) it is pushed outward. Two forces are at work here: Centrifugal (pushing outward) and Centripetal (pulling inward). If you whirl a ball around you on a string, you pull it inward (Centripetal force). The ball seems to pull outward (Centrifugal force) and if released will fly off in a straight line.

Heat:

Heat is a form of energy and can move from one substance to another in one of three ways: by Convection, by Radiation, and by Conduction.

Convection takes place only in liquids like water (for example: water in a kettle) and gases (for example: air warmed by a heat source such as a fire or radiator).

When liquid or gas is heated, it expands and becomes less dense. Warm air above the radiator rises and cool air moves in to take its place, creating a convection current.

Radiation is the movement of heat through the air. Heat forms match set molecules of air moving and rays of heat spread out around the heat source.

Conduction occurs in solids such as metals. The handle of a metal spoon left in boiling liquid warms up as molecules at the heated end moves faster and collide with their neighbors, setting them moving. The heat travels through the metal, which is a good conductor of heat.

Inertia:

A body with a large mass is harder to start and also to stop. A heavy truck traveling at 50mph needs more power breaks to stop its motion than a smaller car traveling at the same speed.

Inertia is the tendency of an object either to stay still or to move steadily in a straight line, unless another force (such as a brick wall stopping the vehicle) makes it behave differently.

Joules:

Joules is the SI unit of work or energy.

One Joule of work is done when a force of one Newton moves through a distance of one meter. The Joule is named after the English scientist James Joule (1818-1889).

Materials:

Stainless steel is an alloy made of steel with chromium or nickel.

Steel is made by the basic oxygen process. The raw material is about three parts melted iron and one part scrap steel. Blowing oxygen into the melted iron raises the temperature and gets rid of impurities.

All plastics are chemical compounds called polymers.

Glass is made by mixing and heating sand, limestone, and soda ash. When these ingredients melt they turn into glass, which is hardened when it cools. Glass is in fact not a solid but a “supercooled” liquid; it can be shaped by blowing, pressing, drawing, casting into molds, rolling, and floating across molten tin to make large sheets.

Ceramic objects, such as pottery and porcelain, electrical insulators, bricks, and roof tiles are all made from clay. The clay is shaped or molded when wet and soft, and heated in a kiln until it hardens.

Machine Tools:

Are powered tools used for shaping metal or other materials, by drilling holes, chiseling, grinding, pressing, or cutting. Often the material (the work piece) is moved while the tool stays still (lathe), or vice versa, the work piece stays while the tool moves (mill).

Most common machine tools are Mill, Lathe, Saw, Broach, Punch press, Grind, Bore and Stamp break.

CNC

Computer Numerical Control is the automation of machine tools that are operated by precisely programmed commands encoded on a storage medium, as opposed to controlled manually via hand wheels or levers, or mechanically automated via cams alone. Most CNC today is computer numerical control in which computers play an integral part of the control.

3D Printing

All methods work by working in layers, adding material, etc. different to other techniques, which are subtractive. Support is needed because almost all methods could support multi material printing, but it is currently only available with certain top-tier machines.

A method of turning digital shapes into physical objects. Due to its nature, it allows us to accurately control the shape of the product. The drawbacks are size restraints and materials are often not durable.

While FDM does not seem like the best method for instrument manufacturing, it is one of the cheapest and most universally available methods.

EDM

Electric Discharge Machining.

FDM

Fused Deposition Modeling.

SLA

Stereo Lithography.

SLS

Selective Laser Sintering.

SLM

Selective Laser Melting.

J-P

Jetted Photopolymer (or Polyjet)

Newton's Law:

1. Every object remains stopped or goes on moving at a steady rate in a straight line unless acted upon by another force. This is the inertia principle.
2. The amount of force needed to make an object change its speed depends on the mass of the object and the amount of acceleration or deceleration required.
3. To every action there is an equal and opposite reaction. When a body is pushed on way by a force, another force pushes back with equal strength.

Polymers:

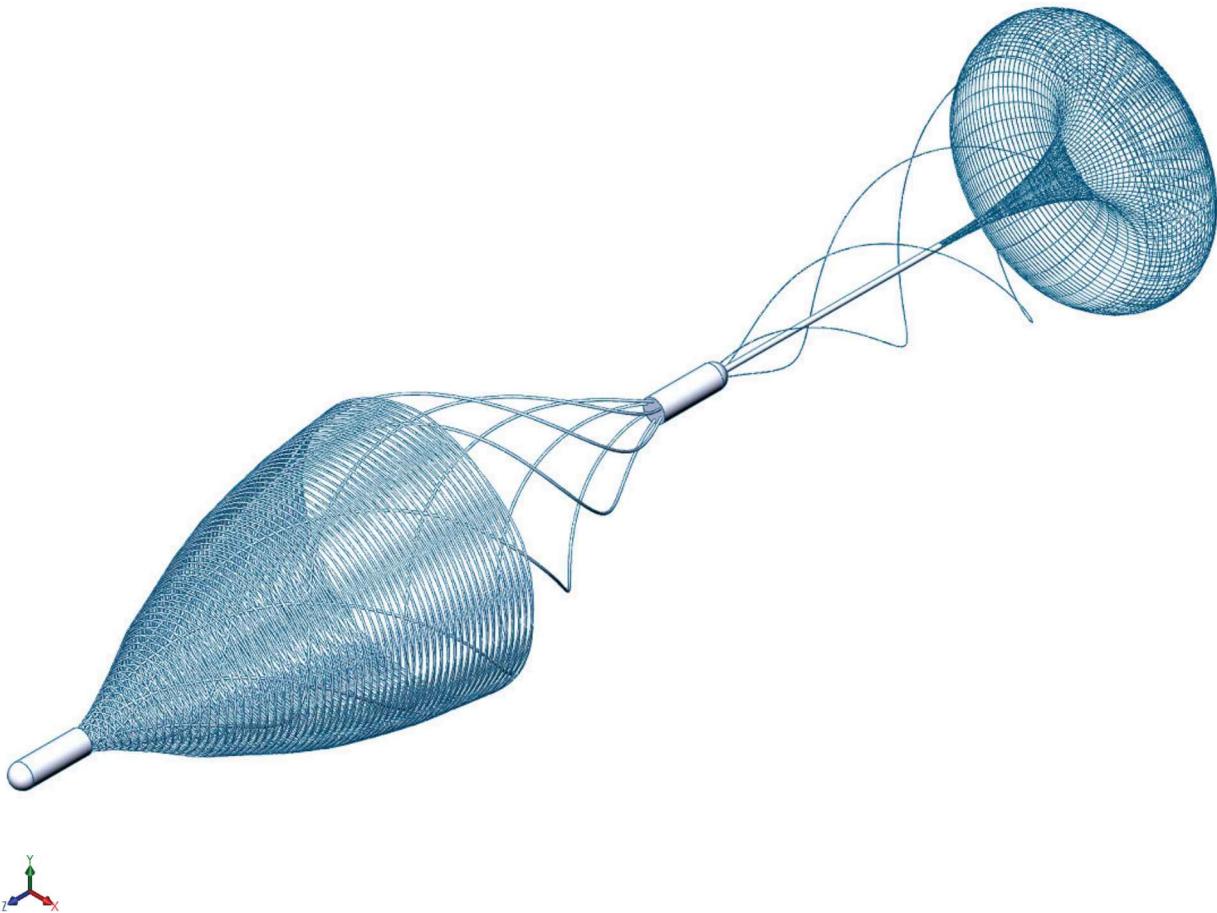
A polymer is made of one or more large molecules formed from thousands of smaller molecules. Rubber and Wood are natural polymers. Plastics are synthetic (artificially made) polymers.

Speed and Velocity:

Speed is the rate at which a moving object changes position (how far it moves in a fixed time).

Velocity is speed in a particular direction.

If either speed or direction is changed, velocity also changes.



Absorbed

A feature, sketch, or annotation that is contained in another item (usually a feature) in the FeatureManager design tree. Examples are the profile sketch and profile path in a base-sweep, or a cosmetic thread annotation in a hole.

Align

Tools that assist in lining up annotations and dimensions (left, right, top, bottom, and so on). For aligning parts in an assembly.

Alternate position view

A drawing view in which one or more views are superimposed in phantom lines on the original view. Alternate position views are often used to show the range of motion of an assembly.

Anchor point

The end of a leader that attaches to the note, block, or other annotation. Sheet formats contain anchor points for a bill of materials, a hole table, a revision table, and a weldment cut list.

Annotation

A text note or a symbol that adds specific design intent to a part, assembly, or drawing. Specific types of annotations include note, hole callout, surface finish symbol, datum feature symbol, datum target, geometric tolerance symbol, weld symbol, balloon, and stacked balloon. Annotations that apply only to drawings include center mark, annotation centerline, area hatch, and block.

Appearance callouts

Callouts that display the colors and textures of the face, feature, body, and part under the entity selected and are a shortcut to editing colors and textures.

Area hatch

A crosshatch pattern or fill applied to a selected face or to a closed sketch in a drawing.

Assembly

A document in which parts, features, and other assemblies (sub-assemblies) are mated together. The parts and sub-assemblies exist in documents separate from the assembly. For example, in an assembly, a piston can be mated to other parts, such as a connecting rod or cylinder. This new assembly can then be used as a sub-assembly in an assembly of an engine. The extension for a SOLIDWORKS assembly file name is .SLDASM.

Attachment point

The end of a leader that attaches to the model (to an edge, vertex, or face, for example) or to a drawing sheet.

Axis

A straight line that can be used to create model geometry, features, or patterns. An axis can be made in a number of different ways, including using the intersection of two planes.

Balloon

Labels parts in an assembly, typically including item numbers and quantity. In drawings, the item numbers are related to rows in a bill of materials.

Base

The first solid feature of a part.

Baseline dimensions

Sets of dimensions measured from the same edge or vertex in a drawing.

Bend

A feature in a sheet metal part. A bend generated from a filleted corner, cylindrical face, or conical face is a round bend; a bend generated from sketched straight lines is a sharp bend.

Bill of materials

A table inserted into a drawing to keep a record of the parts used in an assembly.

Block

A user-defined annotation that you can use in parts, assemblies, and drawings. A block can contain text, sketch entities (except points), and area hatch, and it can be saved in a file for later use as, for example, a custom callout or a company logo.

Bottom-up assembly

An assembly modeling technique where you create parts and then insert them into an assembly.

Broken-out section

A drawing view that exposes inner details of a drawing view by removing material from a closed profile, usually a spline.

Cavity

The mold half that holds the cavity feature of the design part.

Center mark

A cross that marks the center of a circle or arc.

Centerline

Centerline marks, in phantom font, an axis of symmetry in a sketch or drawing.

Chamfer

Bevels a selected edge or vertex. You can apply chamfers to both sketches and features.

Child

A dependent feature related to a previously built feature. For example, a chamfer on the edge of a hole is a child of the parent hole.

Click-release

As you sketch, if you click and then release the pointer, you are in click-release mode. Move the pointer and click again to define the next point in the sketch sequence.

Click-drag

As you sketch, if you click and drag the pointer, you are in click-drag mode. When you release the pointer, the sketch entity is complete.

Closed profile

Also called a closed contour, it is a sketch or sketch entity with no exposed endpoints, for example, a circle or polygon.

Collapse

The opposite of explode. The collapse action returns an exploded assembly's parts to their normal positions.

Collision Detection

An assembly function that detects collisions between components when components move or rotate. A collision occurs when an entity on one component coincides with any entity on another component.

Component

Any part or sub-assembly within an assembly

Configuration

A variation of a part or assembly within a single document. Variations can include different dimensions, features, and properties. For example, a single part such as a bolt can contain different configurations that vary the diameter and length.

ConfigurationManager

Located on the left side of the SOLIDWORKS window, it is a means to create, select, and view the configurations of parts and assemblies.

Constraint

The relations between sketch entities, or between sketch entities and planes, axes, edges, or vertices.

Construction geometry

The characteristic of a sketch entity that is used in creating other geometry but is not itself used in creating features.

Coordinate system

A system of planes used to assign Cartesian coordinates to features, parts, and assemblies. Part and assembly documents contain default coordinate systems; other coordinate systems can be defined with reference geometry. Coordinate systems can be used with measurement tools and for exporting documents to other file formats.

Cosmetic thread

An annotation that represents threads.

Crosshatch

A pattern (or fill) applied to drawing views such as section views and broken-out sections.

Curvature

Curvature is equal to the inverse of the radius of the curve. The curvature can be displayed in different colors according to the local radius (usually of a surface).

Cut

A feature that removes material from a part by such actions as extrude, revolve, loft, sweep, thicken, cavity, and so on.

Dangling

A dimension, relation, or drawing section view that is unresolved. For example, if a piece of geometry is dimensioned, and that geometry is later deleted, the dimension becomes dangling.

Degrees of freedom

Geometry that is not defined by dimensions or relations is free to move. In 2D sketches, there are three degrees of freedom: movement along the X and Y axes, and rotation about the Z axis (the axis normal to the sketch plane). In 3D sketches and in assemblies, there are six degrees of freedom: movement along the X, Y, and Z axes, and rotation about the X, Y, and Z axes.

Derived part

A derived part is a new base, mirror, or component part created directly from an existing part and linked to the original part such that changes to the original part are reflected in the derived part.

Derived sketch

A copy of a sketch, in either the same part or the same assembly that is connected to the original sketch. Changes in the original sketch are reflected in the derived sketch.

Design Library

Located in the Task Pane, the Design Library provides a central location for reusable elements such as parts, assemblies, and so on.

Design table

An Excel spreadsheet that is used to create multiple configurations in a part or assembly document.

Detached drawing

A drawing format that allows opening and working in a drawing without loading the corresponding models into memory. The models are loaded on an as-needed basis.

Detail view

A portion of a larger view, usually at a larger scale than the original view.

Dimension line

A linear dimension line references the dimension text to extension lines indicating the entity being measured. An angular dimension line references the dimension text directly to the measured object.

DimXpertManager

Located on the left side of the SOLIDWORKS window, it is a means to manage dimensions and tolerances created using DimXpert for parts according to the requirements of the ASME Y.14.41-2003 standard.

DisplayManager

The DisplayManager lists the appearances, decals, lights, scene, and cameras applied to the current model. From the DisplayManager, you can view applied content, and add, edit, or delete items.

Document

A file containing a part, assembly, or drawing.

Draft

The degree of taper or angle of a face usually applied to molds or castings.

Drawing

A 2D representation of a 3D part or assembly. The extension for a SOLIDWORKS drawing file name is .SLDDRW.

Drawing sheet

A page in a drawing document.

Driven dimension

Measurements of the model, but they do not drive the model and their values cannot be changed.

Driving dimension

Also referred to as a model dimension, it sets the value for a sketch entity. It can also control distance, thickness, and feature parameters.

Edge

A single outside boundary of a feature.

Edge flange

A sheet metal feature that combines a bend and a tab in a single operation.

Equation

Creates a mathematical relation between sketch dimensions, using dimension names as variables, or between feature parameters, such as the depth of an extruded feature or the instance count in a pattern.

Exploded view

Shows an assembly with its components separated from one another, usually to show how to assemble the mechanism.

Export

Save a SOLIDWORKS document in another format for use in other CAD/CAM, rapid prototyping, web, or graphics software applications.

Extension line

The line extending from the model indicating the point from which a dimension is measured.

Extrude

A feature that linearly projects a sketch to either add material to a part (in a base or boss) or remove material from a part (in a cut or hole).

Face

A selectable area (planar or otherwise) of a model or surface with boundaries that help define the shape of the model or surface. For example, a rectangular solid has six faces.

Fasteners

A SOLIDWORKS Toolbox library that adds fasteners automatically to holes in an assembly.

Feature

An individual shape that, combined with other features, makes up a part or assembly. Some features, such as bosses and cuts, originate as sketches. Other features, such as shells and fillets, modify a feature's geometry. However, not all features have associated geometry. Features are always listed in the FeatureManager design tree.

FeatureManager design tree

Located on the left side of the SOLIDWORKS window, it provides an outline view of the active part, assembly, or drawing.

Fill

A solid area hatch or crosshatch. Fill also applies to patches on surfaces.

Fillet

An internal rounding of a corner or edge in a sketch, or an edge on a surface or solid.

Forming tool

Dies that bend, stretch, or otherwise form sheet metal to create such form features as louvers, lances, flanges, and ribs.

Fully defined

A sketch where all lines and curves in the sketch, and their positions, are described by dimensions or relations, or both, and cannot be moved. Fully defined sketch entities are shown in black.

Geometric tolerance

A set of standard symbols that specify the geometric characteristics and dimensional requirements of a feature.

Graphics area

The area in the SOLIDWORKS window where the part, assembly, or drawing appears.

Guide curve

A 2D or 3D curve is used to guide a sweep or loft.

Handle

An arrow, square, or circle that you can drag to adjust the size or position of an entity (a feature, dimension, or sketch entity, for example).

Helix

A curve defined by pitch, revolutions, and height. A helix can be used, for example, as a path for a swept feature cutting threads in a bolt.

Hem

A sheet metal feature that folds back at the edge of a part. A hem can be open, closed, double, or teardrop.

HLR

(Hidden lines removed) a view mode in which all edges of the model that are not visible from the current view angle are removed from the display.

HLV

(Hidden lines visible) A view mode in which all edges of the model that are not visible from the current view angle are shown gray or dashed.

Import

Open files from other CAD software applications into a SOLIDWORKS document.

In-context feature

A feature with an external reference to the geometry of another component; the in-context feature changes automatically if the geometry of the referenced model or feature changes.

Inference

The system automatically creates (infers) relations between dragged entities (sketched entities, annotations, and components) and other entities and geometry. This is useful when positioning entities relative to one another.

Instance

An item in a pattern or a component in an assembly that occurs more than once. Blocks are inserted into drawings as instances of block definitions.

Interference detection

A tool that displays any interference between selected components in an assembly.

Jog

A sheet metal feature that adds material to a part by creating two bends from a sketched line.

Knit

A tool that combines two or more faces or surfaces into one. The edges of the surfaces must be adjacent and not overlapping, but they cannot ever be planar. There is no difference in the appearance of the face or the surface after knitting.

Layout sketch

A sketch that contains important sketch entities, dimensions, and relations. You reference the entities in the layout sketch when creating new sketches, building new geometry, or positioning components in an assembly. This allows for easier updating of your model because changes you make to the layout sketch propagate to the entire model.

Leader

A solid line from an annotation (note, dimension, and so on) to the referenced feature.

Library feature

A frequently used feature, or combination of features, that is created once and then saved for future use.

Lightweight

A part in an assembly or a drawing has only a subset of its model data loaded into memory. The remaining model data is loaded on an as-needed basis. This improves the performance of large and complex assemblies.

Line

A straight sketch entity with two endpoints. A line can be created by projecting an external entity such as an edge, plane, axis, or sketch curve into the sketch.

Loft

A base, boss, cut, or surface feature created by transitions between profiles.

Lofted bend

A sheet metal feature that produces a roll form or a transitional shape from two open profile sketches. Lofted bends often create funnels and chutes.

Mass properties

A tool that evaluates the characteristics of a part or an assembly such as volume, surface area, centroid, and so on.

Mate

A geometric relationship, such as coincident, perpendicular, tangent, and so on, between parts in an assembly.

Mate reference

Specifies one or more entities of a component to use for automatic mating. When you drag a component with a mate reference into an assembly, the software tries to find other combinations of the same mate reference name and mate type.

Mates folder

A collection of mates that are solved together. The order in which the mates appear within the Mates folder does not matter.

Mirror

- (a) A mirror feature is a copy of a selected feature, mirrored about a plane or planar face.
- (b) A mirror sketch entity is a copy of a selected sketch entity that is mirrored about a centerline.

Miter flange

A sheet metal feature that joins multiple edge flanges together and miters the corner.

Model

3D solid geometry in a part or assembly document. If a part or assembly document contains multiple configurations, each configuration is a separate model.

Model dimension

A dimension specified in a sketch or a feature in a part or assembly document that defines some entity in a 3D model.

Model item

A characteristic or dimension of feature geometry that can be used in detailing drawings.

Model view

A drawing view of a part or assembly.

Mold

A set of manufacturing tooling used to shape molten plastic or other material into a designed part. You design the mold using a sequence of integrated tools that result in cavity and core blocks that are derived parts of the part to be molded.

Motion Study

Motion Studies are graphical simulations of motion and visual properties with assembly models. Analogous to a configuration, they do not actually change the original assembly model or its properties. They display the model as it changes based on simulation elements you add.

Multibody part

A part with separate solid bodies within the same part document. Unlike the components in an assembly, multibody parts are not dynamic.

Native format

DXF and DWG files remain in their original format (are not converted into SOLIDWORKS format) when viewed in SOLIDWORKS drawing sheets (view only).

Open profile

Also called an open contour, it is a sketch or sketch entity with endpoints exposed. For example, a U-shaped profile is open.

Ordinate dimensions

A chain of dimensions measured from a zero ordinate in a drawing or sketch.

Origin

The model origin appears as three gray arrows and represents the (0,0,0) coordinate of the model. When a sketch is active, a sketch origin appears in red and represents the (0,0,0) coordinate of the sketch. Dimensions and relations can be added to the model origin, but not to a sketch origin.

Out-of-context feature

A feature with an external reference to the geometry of another component that is not open.

Over defined

A sketch is over defined when dimensions or relations are either in conflict or redundant.

Parameter

A value used to define a sketch or feature (often a dimension).

Parent

An existing feature upon which other features depend. For example, in a block with a hole, the block is the parent to the child hole feature.

Part

A single 3D object made up of features. A part can become a component in an assembly, and it can be represented in 2D in a drawing. Examples of parts are bolt, pin, plate, and so on. The extension for the SOLIDWORKS part file name is .SLDPRT.

Path

A sketch, edge, or curve used in creating a sweep or loft.

Pattern

A pattern repeats selected sketch entities, features, or components in an array, which can be linear, circular, or sketch driven. If the seed entity is changed, the other instances in the pattern update.

Physical Dynamics

An assembly tool that displays the motion of assembly components in a realistic way. When you drag a component, the component applies a force to other components it touches. Components move only within their degrees of freedom.

Pierce relation

Makes a sketch point coincident to the location at which an axis, edge, line, or spline pierces the sketch plane.

Planar

Entities that can lie on one plane. For example, a circle is planar, but a helix is not.

Plane

Flat construction geometry. Planes can be used for a 2D sketch, section view of a model, a neutral plane in a draft feature, and others.

Point

A singular location in a sketch, or a projection into a sketch at a single location of an external entity (origin, vertex, axis, or point in an external sketch).

Predefined view

A drawing view in which the view position, orientation, and so on can be specified before a model is inserted. You can save drawing documents with predefined views as templates.

Profile

A sketch entity is used to create a feature (such as a loft) or a drawing view (such as a detail view). A profile can be open (such as a U shape or open spline) or closed (such as a circle or closed spline).

Projected dimension

If you dimension entities in an isometric view, projected dimensions are the flat dimensions in 2D.

Projected view

A drawing view projected orthogonally from an existing view.

PropertyManager

Located on the left side of the SOLIDWORKS window, it is used for dynamic editing of sketch entities and most features.

RealView graphics

A hardware (graphics card) support of advanced shading in real time; the rendering applies to the model and is retained as you move or rotate a part.

Rebuild

Tool that updates (or regenerates) the document with any changes made since the last time the model was rebuilt. Rebuild is typically used after changing a model dimension.

Reference dimension

A dimension in a drawing that shows the measurement of an item but cannot drive the model and its value cannot be modified. When model dimensions change, reference dimensions update.

Reference geometry

Includes planes, axes, coordinate systems, and 3D curves. Reference geometry is used to assist in creating features such as lofts, sweeps, drafts, chamfers, and patterns.

Relation

A geometric constraint between sketch entities or between a sketch entity and a plane, axis, edge, or vertex. Relations can be added automatically or manually.

Relative view

A relative (or relative to model) drawing view is created relative to planar surfaces in a part or assembly.

Reload

Refreshes shared documents. For example, if you open a part file for read-only access while another user makes changes to the same part, you can reload the new version, including the changes.

Reorder

Reordering (changing the order of) items is possible in the FeatureManager design tree. In parts, you can change the order in which features are solved. In assemblies, you can control the order in which components appear in a bill of materials.

Replace

Substitutes one or more open instances of a component in an assembly with a different component.

Resolved

A state of an assembly component (in an assembly or drawing document) in which it is fully loaded in memory. All the component's model data is available, so its entities can be selected, referenced, edited, and used in mates, and so on.

Revolve

A feature that creates a base or boss, a revolved cut, or revolved surface by revolving one or more sketched profiles around a centerline.

Rip

A sheet metal feature that removes material at an edge to allow a bend.

Rollback

Suppresses all items below the rollback bar.

Section

Another term for profile in sweeps.

Section line

A line or centerline sketched in a drawing view to create a section view.

Section scope

Specifies the components to be left uncut when you create an assembly drawing section view.

Section view

A section view (or section cut) is (1) a part or assembly view cut by a plane, or (2) a drawing view created by cutting another drawing view with a section line.

Seed

A sketch or an entity (a feature, face, or body) that is the basis for a pattern. If you edit the seed, the other entities in the pattern are updated.

Shaded

Displays a model as a colored solid.

Shared values

Also called linked values, these are named variables that you assign to set the value of two or more dimensions to be equal.

Sheet format

Includes page size and orientation, standard text, borders, title blocks, and so on. Sheet formats can be customized and saved for future use. Each sheet of a drawing document can have a different format.

Shell

A feature that hollows out a part, leaving open the selected faces and thin walls on the remaining faces. A hollow part is created when no faces are selected to be open.

Sketch

A collection of lines and other 2D objects on a plane or face that forms the basis for a feature such as a base or a boss. A 3D sketch is non-planar and can be used to guide a sweep or loft, for example.

Smart Fasteners

Automatically adds fasteners (bolts and screws) to an assembly using the SOLIDWORKS Toolbox library of fasteners.

SmartMates

An assembly mating relation that is created automatically.

Solid sweep

A cut sweep created by moving a tool body along a path to cut out 3D material from a model.

Spiral

A flat or 2D helix, defined by a circle, pitch, and number of revolutions.

Spline

A sketched 2D or 3D curve defined by a set of control points.

Split line

Projects a sketched curve onto a selected model face, dividing the face into multiple faces so that each can be selected individually. A split line can be used to create draft features, to create face blend fillets, and to radiate surfaces to cut molds.

Stacked balloon

A set of balloons with only one leader. The balloons can be stacked vertically (up or down) or horizontally (left or right).

Standard 3 views

The three orthographic views (front, right, and top) that are often the basis of a drawing.

Stereolithography

The process of creating rapid prototype parts using a faceted mesh representation in STL files.

Sub-assembly

An assembly document that is part of a larger assembly. For example, the steering mechanism of a car is a sub-assembly of the car.

SUPPRESS

Removes an entity from the display and from any calculations in which it is involved. You can suppress features, assembly components, and so on. Suppressing an entity does not delete the entity; you can unsuppress the entity to restore it.

Surface

A zero-thickness planar or 3D entity with edge boundaries. Surfaces are often used to create solid features. Reference surfaces can be used to modify solid features.

Sweep

Creates a base, boss, cut, or surface feature by moving a profile (section) along a path. For cut sweeps, you can create solid sweeps by moving a tool body along a path.

Tangent arc

An arc that is tangent to another entity, such as a line.

Tangent edge

The transition edge between rounded or filleted faces in hidden lines visible or hidden lines removed modes in drawings.

Task Pane

Located on the right-side of the SOLIDWORKS window, the Task Pane contains SOLIDWORKS Resources, the Design Library, and the File Explorer.

Template

A document (part, assembly, or drawing) that forms the basis of a new document. It can include user-defined parameters, annotations, predefined views, geometry, and so on.

Temporary axis

An axis created implicitly for every conical or cylindrical face in a model.

Thin feature

An extruded or revolved feature with constant wall thickness. Sheet metal parts are typically created from thin features.

TolAnalyst

A tolerance analysis application that determines the effects that dimensions and tolerances have on parts and assemblies.

Top-down design

An assembly modeling technique where you create parts in the context of an assembly by referencing the geometry of other components. Changes to the referenced components propagate to the parts that you create in context.

Triad

Three axes with arrows defining the X, Y, and Z directions. A reference triad appears in part and assembly documents to assist in orienting the viewing of models. Triads also assist when moving or rotating components in assemblies.

Under defined

A sketch is under defined when there are not enough dimensions and relations to prevent entities from moving or changing size.

Vertex

A point at which two or more lines or edges intersect. Vertices can be selected for sketching, dimensioning, and many other operations.

Viewports

Windows that display views of models. You can specify one, two, or four viewports. Viewports with orthogonal views can be linked, which links orientation and rotation.

Virtual sharp

A sketch point at the intersection of two entities after the intersection itself has been removed by a feature such as a fillet or chamfer. Dimensions and relations to the virtual sharp are retained even though the actual intersection no longer exists.

Weldment

A multibody part with structural members.

Weldment cut list

A table that tabulates the bodies in a weldment along with descriptions and lengths.

Wireframe

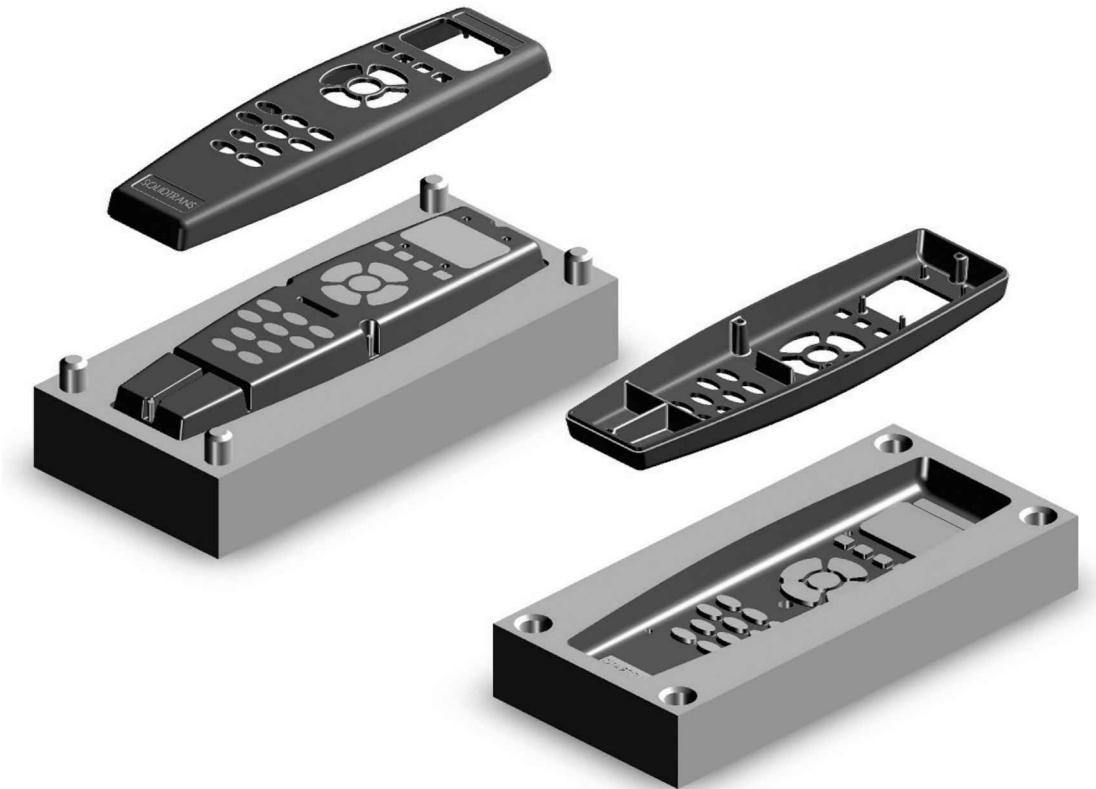
A view mode in which all edges of the part or assembly are displayed.

Zebra stripes

Simulate the reflection of long strips of light on a very shiny surface. They allow you to see small changes in a surface that may be hard to see with a standard display.

Zoom

To simulate movement toward or away from a part or an assembly.



Learn Mold-Tooling Designs with SOLIDWORKS Mold Making Textbook

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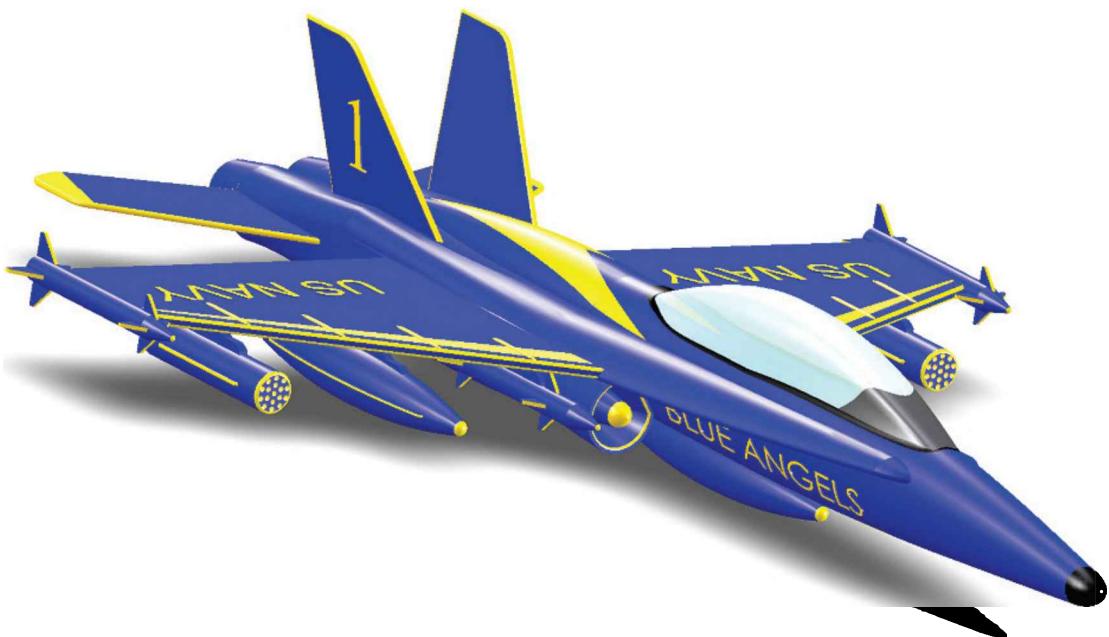
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Designed and Rendered with SOLIDWORKS

SOLIDWORKS Quick Guide

STANDARD Toolbar

	Creates a new document.		Loads or unloads the 3D instant website add-in.
	Opens an existing document.		Select tool.
	Saves an active document.		Select the entire document.
	Make Drawing from Part/Assembly.		Checks read-only files.
	Make Assembly from Part/Assembly.		Options.
	Prints the active document.		Help.
	Print preview.		Full screen view.
	Cuts the selection & puts it on the clipboard.		OK.
	Copies the selection & puts it on the clipboard.		Cancel.
	Inserts the clipboard contents.		Magnified selection.
	Deletes the selection.		
	Reverses the last action.		Select.
	Rebuilds the part / assembly / drawing.		Sketch.
	Redo the last action that was undone.		3D Sketch.
	Saves all documents.		Sketches a rectangle from the center.
	Edits material.		Sketches a CenterPoint arc slot.
	Closes an existing document.		Sketches a 3-point arc slot.
	Shows or hides the Selection Filter toolbar.		Sketches a straight slot.
	Shows or hides the Web toolbar.		Sketches a CenterPoint straight slot.
	Properties.		Sketches a 3-point arc.
	File properties.		Creates sketched ellipses.

Quick Reference Guide to SOLIDWORKS Command Icons & Toolbars

SKETCH TOOLS Toolbar

	3D sketch on plane.		Partial ellipses.
	Sets up Grid parameters.		Adds a Parabola.
	Creates a sketch on a selected plane or face.		Adds a spline.
	Equation driven curve.		Sketches a polygon.
	Modifies a sketch.		Sketches a corner rectangle.
	Copies sketch entities.		Sketches a parallelogram.
	Scales sketch entities.		Creates points.
	Rotates sketch entities.		Creates sketched centerlines.
	Sketches 3 point rectangle from the center.		Adds text to sketch.
	Sketches 3 point corner rectangle.		Converts selected model edges or sketch entities to sketch segments.
	Sketches a line.		Creates a sketch along the intersection of multiple bodies.
	Creates a center point arc: center, start, end.		Converts face curves on the selected face into 3D sketch entities.
	Creates an arc tangent to a line.		Mirrors selected segments about a centerline.
	Sketches splines on a surface or face.		Fillets the corner of two lines.
	Sketches a circle.		Creates a chamfer between two sketch entities.
	Sketches a circle by its perimeter.		Creates a sketch curve by offsetting model edges or sketch entities at a specified distance.
	Makes a path of sketch entities.		Trims a sketch segment.
	Mirrors entities dynamically about a centerline.		Extends a sketch segment.
	Insert a plane into the 3D sketch.		Splits a sketch segment.
	Instant 2D.		Construction Geometry.
	Sketch numeric input.		Creates linear steps and repeat of sketch entities.
	Detaches segment on drag.		Creates circular steps and repeat of sketch entities.
	Sketch picture.		

Quick Reference Guide to SOLIDWORKS Command Icons & Toolbars

SHEET METAL Toolbar

-  Add a bend from a selected sketch in a Sheet Metal part.
-  Shows flat pattern for this sheet metal part.
-  Shows part without inserting any bends.
-  Inserts a rip feature to a sheet metal part.
-  Create a Sheet Metal part or add material to existing Sheet Metal part.
-  Inserts a Sheet Metal Miter Flange feature.
-  Folds selected bends.
-  Unfolds selected bends.
-  Inserts bends using a sketch line.
-  Inserts a flange by pulling an edge.
-  Inserts a sheet metal corner feature.
-  Inserts a Hem feature by selecting edges.
-  Breaks a corner by filleting/chamfering it.
-  Inserts a Jog feature using a sketch line.
-  Inserts a lofted bend feature using 2 sketches.
-  Creates inverse dent on a sheet metal part.
-  Trims out material from a corner, in a sheet metal part.
-  Inserts a fillet weld bead.
-  Converts a solid/surface into a sheet metal part.
-  Adds a Cross Break feature into a selected face.
-  Sweeps an open profile along an open/closed path.
-  Adds a gusset/rib across a bend.
-  Corner relief.
-  Welds the selected corner.

SURFACES Toolbar

-  Creates mid surfaces between offset face pairs.
-  Patches surface holes and external edges.
-  Creates an extruded surface.
-  Creates a revolved surface.
-  Creates a swept surface.
-  Creates a lofted surface.
-  Creates an offset surface.
-  Radiates a surface originating from a curve, parallel to a plane.
-  Knits surfaces together.
-  Creates a planar surface from a sketch or a set of edges.
-  Creates a surface by importing data from a file.
-  Extends a surface.
-  Trims a surface.
-  Surface flatten.
-  Deletes Face(s).
-  Replaces Face with Surface.
-  Patches surface holes and external edges by extending the surfaces.
-  Creates parting surfaces between core & cavity surfaces.
-  Inserts ruled surfaces from edges.

WELDMENTS Toolbar

-  Creates a weldment feature.
-  Creates a structure member feature.
-  Adds a gusset feature between 2 planar adjoining faces.
-  Creates an end cap feature.
-  Adds a fillet weld bead feature.
-  Trims or extends structure members.
-  Weld bead.

Quick Reference Guide to SOLIDWORKS Command Icons & Toolbars

DIMENSIONS/RELATIONS Toolbar

-  Inserts dimension between two lines.
-  Creates a horizontal dimension between selected entities.
-  Creates a vertical dimension between selected entities.
-  Creates a reference dimension between selected entities.
-  Creates a set of ordinate dimensions.
-  Creates a set of Horizontal ordinate
-  Creates a set of Vertical ordinate dimensions.
-  Creates a chamfer dimension.
-  Adds a geometric relation.
-  Automatically Adds Dimensions to the current sketch.
-  Displays and deletes geometric relations.
-  Fully defines a sketch.
-  Scans a sketch for elements of equal length or radius.
-  Angular Running dimension.
-  Display / Delete dimension.
-  Isolate changed dimension.
-  Path length dimension.



Updates parent sketches affected by this block.



Saves the block to a file.



Explodes the selected block.



Inserts a belt.

STANDARD VIEWS Toolbar



Front view.



Back view.



Left view.



Right view.



Top view.



Bottom view.



Isometric view.



Trimetric view.



Dimetric view.



Normal to view.



Links all views in the viewport together.



Displays viewport with front & right



Displays a 4 view viewport with 1st or 3rd



Displays viewport with front & top.



Displays viewport with a single view.



View selector.



New view.

BLOCK Toolbar

-  Makes a new block.
-  Edits the selected block.
-  Inserts a new block to a sketch or drawing.
-  Adds/Removes sketch entities to/from blocks.



New view.

FEATURES Toolbar

	Creates a boss feature by extruding a sketched profile.		Intersect.
	Creates a revolved feature based on profile and angle parameter.		Variable Patterns.
	Creates a cut feature by extruding a sketched profile.		Live Section Plane.
	Creates a cut feature by revolving a sketched profile.		Mirrors.
	Thread.		Scale.
	Creates a cut by sweeping a closed profile along an open or closed path.		Creates a Sketch Driven pattern.
	Loft cut.		Creates a Table Driven Pattern.
	Creates a cut by thickening one or more adjacent surfaces.		Inserts a split Feature.
	Adds a deformed surface by push or pull on points.		Hole series.
	Creates a lofted feature between two or more profiles.		Joins bodies from one or more parts into a single part in the context of an assembly.
	Creates a solid feature by thickening one or more adjacent surfaces.		Deletes a solid or a surface.
	Creates a filled feature.		Instant 3D.
	Chamfers an edge or a chain of tangent edges.		Inserts a part from file into the active part document.
	Inserts a rib feature.		Moves/Copies solid and surface bodies or moves graphics bodies.
	Combine.		Merges short edges on faces.
	Creates a shell feature.		Pushes solid / surface model by another solid / surface model.
	Applies draft to a selected surface.		Moves face(s) of a solid.
	Creates a cylindrical hole.		FeatureWorks Options.
	Inserts a hole with a pre-defined cross section.		Linear Pattern.
	Puts a dome surface on a face.		Fill Pattern.
	Model break view.		Cuts a solid model with a
	Applies global deformation to solid or surface bodies.		Boundary Boss/Base.
	Wraps closed sketch contour(s) onto a face.		Boundary Cut.
	Curve Driven pattern.		Circular Pattern.
	Suppresses the selected feature or component.		Recognize Features.
	Un-suppresses the selected feature or component.		Grid System.
	Flexes solid and surface bodies.		

MOLD TOOLS Toolbar

	Extracts core(s) from existing tooling split.		Allows selection of edges only.
	Constructs a surface patch.		Allows selection filter for vertices only.
	Moves face(s) of a solid.		Allows selection of faces only.
	Creates offset surfaces.		Adds filter for Surface Bodies.
	Inserts cavity into a base part.		Adds filter for Solid Bodies.
	Scales a model by a specified factor.		Adds filter for Axes.
	Applies draft to a selected surface.		Adds filter for Planes.
	Inserts a split line feature.		Adds filter for Sketch Points.
	Creates parting lines to separate core & cavity surfaces.		Allows selection for sketch only.
	Finds & creates mold shut-off surfaces.		Adds filter for Sketch Segments.
	Creates a planar surface from a sketch or a set of edges.		Adds filter for Midpoints.
	Knits surfaces together.		Adds filter for Center Marks.
	Inserts ruled surfaces from edges.		Adds filter for Centerline.
	Creates parting surfaces between core & cavity surfaces.		Adds filter for Dimensions and Hole Callouts.
	Creates multiple bodies from a single body.		Adds filter for Surface Finish Symbols.
	Inserts a tooling split feature.		Adds filter for Geometric Tolerances.
	Creates parting surfaces between the core & cavity.		Adds filter for Notes / Balloons.
	Inserts surface body folders for mold operation.		Adds filter for Weld Symbols.

SELECTION FILTERS Toolbar

	Turns selection filters on and off.		Adds filter for Datum feature only.
	Clears all filters.		Adds filter for blocks.
	Selects all filters.		Adds filter for Cosmetic Threads.
	Inverts current selection.		



Adds filter for Dowel pin symbols.



Adds filter for connection points.



Adds filter for routing points.

SOLIDWORKS Add-Ins Toolbar



Loads/unloads CircuitWorks add-in.



Loads/unloads the Design Checker add-in.



Loads/unloads the Visualize add-in.



Loads/unloads the Scan-to-3D add-in.



Loads/unloads the SOLIDWORKS Motions add-in.



Loads/unloads the SOLIDWORKS Routing add-in.



Loads/unloads the SOLIDWORKS Simulation add-in.



Loads/unloads the SOLIDWORKS Toolbox add-in.



Loads/unloads the SOLIDWORKS TolAnalysis add-in.



Loads/unloads the SOLIDWORKS Flow Simulation add-in.



Loads/unloads the SOLIDWORKS Plastics add-in.



Loads/unloads the SOLIDWORKS MBD SNL license.

FASTENING FEATURES Toolbar



Creates a parameterized mounting boss.



Creates a parameterized snap hook.



Creates a groove to mate with a hook feature.



Uses sketch elements to create a vent for air flow.



Creates a lip/groove feature.

SCREEN CAPTURE Toolbar



Copies the current graphics window to the clipboard.



Records the current graphics window to an AVI file.



Stops recording the current graphics window to an AVI file.

EXPLODE LINE SKETCH Toolbar



Adds a route line that connects entities.



Adds a jog to the route lines.

LINE FORMAT Toolbar



Changes layer properties.



Changes the current document layer.



Changes line color.



Changes line thickness.



Changes line style.



Hides / Shows a hidden edge.



Changes line display mode.



* Ctrl+Q will force a rebuild on all features of a part.

Did you know?

* Ctrl+B will rebuild the feature being worked on and its dependents.

2D-To-3D Toolbar



Makes a Front sketch from the selected entities.



Makes a Top sketch from the selected entities.



Makes a Right sketch from the selected entities.

	Makes a Left sketch from the selected entities.
	Makes a Bottom sketch from the selected entities.
	Makes a Back sketch from the selected entities.
	Makes an Auxiliary sketch from the selected entities.
	Creates a new sketch from the selected entities.
	Repairs the selected sketch.
	Aligns a sketch to the selected point.
	Creates an extrusion from the selected sketch segments, starting at the selected sketch point.
	Creates a cut from the selected sketch segments, optionally starting at the selected sketch point.

	Evenly spaces selected dimensions.
	Aligns collinear selected dimensions.
	Aligns stagger selected dimensions.

SOLIDWORKS MBD Toolbar

	Captures 3D view.
	Manages 3D PDF templates.
	Creates shareable 3D PDF presentations.
	Toggles dynamic annotation views.

ALIGN Toolbar

	Aligns the left side of the selected annotations with the leftmost annotation.
	Aligns the right side of the selected annotations with the rightmost annotation.
	Aligns the top side of the selected annotations with the topmost annotation.
	Aligns the bottom side of the selected annotations with the lowermost annotation.
	Evenly spaces the selected annotations horizontally.
	Evenly spaces the selected annotations vertically.
	Centrally aligns the selected annotations horizontally.
	Centrally aligns the selected annotations vertically.
	Compacts the selected annotations horizontally.
	Compacts the selected annotations vertically.
	Creates a group from the selected items.
	Deletes the grouping between these items.
	Aligns & groups selected dimensions along a line or an arc.
	Aligns & groups dimensions at uniform distances.

MACRO Toolbar

	Runs a Macro.
	Stops Macro recorder.
	Records (or pauses recording of) actions to create a Macro.
	Launches the Macro Editor and begins editing a new macro.
	Opens a Macro file for editing.
	Creates a custom macro.

SMARTMATES icons

	Concentric & Coincident 2 circular edges.
	Concentric 2 cylindrical faces.
	Coincident 2 linear edges.
	Coincident 2 planar faces.
	Coincident 2 vertices.
	Coincident 2 origins or coordinate systems.

TABLE Toolbar

-  Adds a hole table of selected holes from a specified origin datum.
-  Adds a Bill of Materials.
-  Adds a revision table.
-  Displays a Design table in a drawing.
-  Adds a weldments cuts list table.
-  Adds a Excel based of Bill of Materials
-  Adds a weldment cut list table.

ANNOTATIONS Toolbar

-  Inserts a note.
-  Inserts a surface finish symbol.
-  Inserts a new geometric tolerancing symbol.
-  Attaches a balloon to the selected edge or face.
-  Adds balloons for all components in selected view.
-  Inserts a stacked balloon.
-  Attaches a datum feature symbol to a selected edge / detail.
-  Inserts a weld symbol on the selected edge / face / vertex.
-  Inserts a datum target symbol and / or point attached to a selected edge / line.
-  Selects and inserts block.
-  Inserts annotations & reference geometry from the part / assembly into the selected.
-  Adds center marks to circles on model.
-  Inserts a Centerline.
-  Inserts a hole callout.
-  Adds a cosmetic thread to the selected cylindrical feature.
-  Inserts a Multi-Jog leader.
-  Selects a circular edge or an arc for Dowel pin symbol insertion.
-  Adds a view location symbol.
-  Inserts latest version symbol.
-  Adds a cross hatch patterns or solid fill.
-  Adds a weld bead caterpillar on an edge.
-  Adds a weld symbol on a selected entity.
-  Inserts a revision cloud.
-  Inserts a magnetic line.
-  Hides/shows annotation.

REFERENCE GEOMETRY Toolbar

-  Adds a reference plane.
-  Creates an axis.
-  Creates a coordinate system.
-  Adds the center of mass.
-  Specifies entities to use as references using SmartMates.

SPLINE TOOLS Toolbar

-  Inserts a point to a spline.
-  Displays all points where the concavity of selected spline changes.
-  Displays minimum radius of selected spline.
-  Displays curvature combs of selected spline.
-  Reduces numbers of points in a selected spline.
-  Adds a tangency control.
-  Adds a curvature control.
-  Adds a spline based on selected sketch entities & edges.
-  Displays the spline control polygon.

DRAWINGS Toolbar

-  Updates the selected view to the model's current stage.
-  Creates a detail view.
-  Creates a section view.
-  Inserts an Alternate Position view.
-  Unfolds a new view from an existing view.
-  Generates a standard 3-view drawing (1st or 3rd angle).
-  Inserts an auxiliary view of an inclined surface.
-  Adds an Orthogonal or Named view based on an existing part or assembly.
-  Adds a Relative view by two orthogonal faces or planes.
-  Adds a Predefined orthogonal projected or Named view with a model.
-  Adds an empty view.
-  Adds vertical break lines to selected view.
-  Crops a view.
-  Creates a Broken-out section.

 Snap horizontally / vertically to points.

 Snap horizontally / vertically.

 Snap to discrete line lengths.

 Snap to angle.

LAYOUT Toolbar

-  Creates the assembly layout sketch.
-  Sketches a line.
-  Sketches a corner rectangle.
-  Sketches a circle.
-  Sketches a 3 point arc.
-  Rounds a corner.
-  Trims or extends a sketch.
-  Adds sketch entities by offsetting faces, edges curves.
-  Mirrors selected entities about a centerline.
-  Adds a relation.

 Creates a dimension.

 Displays / Deletes geometric relations.

 Makes a new block.

 Edits the selected block.

 Inserts a new block to the sketch or drawing.

 Adds / Removes sketch entities to / from a block.

 Saves the block to a file.

 Explodes the selected block.

 Creates a new part from a layout sketch block.

 Positions 2 components relative to one another.

QUICK SNAP Toolbar

-  Snap to points.
-  Snap to center points.
-  Snap to midpoints.
-  Snap to quadrant points.
-  Snap to intersection of 2 curves.
-  Snap to nearest curve.
-  Snap tangent to curve.
-  Snap perpendicular to curve.
-  Snap parallel to line.

CURVES Toolbar

-  Projects sketch onto selected surface.
-  Inserts a split line feature.
-  Creates a composite curve from selected edges, curves and sketches.
-  Creates a curve through free points.
-  Creates a 3D curve through reference points.
-  Helical curve defined by a base sketch and shape parameters.

VIEW Toolbar

-  Displays a view in the selected orientation.
-  Reverts to previous view.
-  Redraws the current window.
-  Zooms out to see entire model.
-  Zooms in by dragging a bounding box.
-  Zooms in or out by dragging up or down.
-  Zooms to fit all selected entities.
-  Dynamic view rotation.
-  Scrolls view by dragging.
-  Displays image in wireframe mode.
-  Displays hidden edges in gray.
-  Displays image with hidden lines removed.
-  Controls the visibility of planes.
-  Controls the visibility of axis.
-  Controls the visibility of parting lines.
-  Controls the visibility of temporary axis.
-  Controls the visibility of origins.



Controls the visibility of coordinate systems.



Controls the visibility of reference curves.



Controls the visibility of sketches.



Controls the visibility of 3D sketch planes.



Controls the visibility of 3D sketch.



Controls the visibility of all annotations.



Controls the visibility of reference points.



Controls the visibility of routing points.



Controls the visibility of lights.



Controls the visibility of cameras.



Controls the visibility of sketch relations.



Changes the display state for the current configuration.



Rolls the model view.



Turns the orientation of the model view.



Dynamically manipulate the model view in 3D to make selection.



Changes the display style for the active view.



Displays a shade view of the model with its edges.



Displays a shade view of the model.



Toggles between draft quality & high quality HLV.



Cycles through or applies a specific scene.



Views the models through one of the model's cameras.



Displays a part or assembly w/different colors according to the local radius of curvature.



Displays zebra stripes.



Displays a model with hardware accelerated shades.



Applies a cartoon affect to model edges & faces.



Views simulations symbols.

TOOLS Toolbar

	Calculates the distance between selected items.		Smart Fasteners.
	Adds or edits equation.		Positions two components relative to one another.
	Calculates the mass properties of the model.		External references will not be created.
	Checks the model for geometry errors.		Moves a component.
	Inserts or edits a Design Table.		Rotates an un-mated component around its center point.
	Evaluates section properties for faces and sketches that lie in parallel planes.		Replaces selected components.
	Reports Statistics for this Part/Assembly.		Replaces mate entities of mates of the selected components on the selected Mates group.
	Deviation Analysis.		Creates a New Exploded view.
	Runs the SimulationXpress analysis wizard powered by SOLIDWORKS Simulation.		Creates or edits explode line sketch.
	Checks the spelling.		Interference detection.
	Import diagnostics.		Shows or Hides the Simulation toolbar.
	Runs the DFMXpress analysis wizard.		Patterns components in one or two linear directions.
	Runs the SOLIDWORKS FloXpress analysis wizard.		Patterns components around an axis.
			Sets the transparency of the components other than the one being edited.
			Sketch driven component pattern.
			Pattern driven component pattern.
			Curve driven component pattern.
			Chain driven component pattern.
			SmartMates by dragging & dropping components.
			Checks assembly hole alignments.
			Mirrors subassemblies and parts.

ASSEMBLY Toolbar

	Creates a new part & inserts it into the assembly.		Curve driven component pattern.
	Adds an existing part or sub-assembly to the assembly.		Chain driven component pattern.
	Creates a new assembly & inserts it into the assembly.		SmartMates by dragging & dropping components.
	Turns on/off large assembly mode for this document.		Hides / shows model(s) associated with the selected model(s).
	Hides / shows model(s) associated with the selected model(s).		Checks assembly hole alignments.
	Toggles the transparency of components.		Mirrors subassemblies and parts.
	Changes the selected components to suppressed or resolved.		
	Inserts a belt.		
	Toggles between editing part and assembly.		

**To add or remove an icon
to or from the toolbar, first select:
Tools/Customize/Commands**
Next, select a **Category**, click a button to see its description and then drag/drop the command icon into any toolbar.

SOLIDWORKS Quick-Guide®

STANDARD Keyboard Shortcuts

Rotate the model

- * Horizontally or Vertically: _____ Arrow keys
- * Horizontally or Vertically 90°: _____ Shift + Arrow keys
- * Clockwise or Counterclockwise: _____ Alt + left or right Arrow
- * Pan the model: _____ Ctrl + Arrow keys
- * Zoom in: _____ Z (shift + Z or capital Z)
- * Zoom out: _____ z (lower case z)
- * Zoom to fit: _____ F
- * Previous view: _____ Ctrl+Shift+Z

View Orientation

- * View Orientation Menu: _____ Space bar
- * Front: _____ Ctrl+1
- * Back: _____ Ctrl+2
- * Left: _____ Ctrl+3
- * Right: _____ Ctrl+4
- * Top: _____ Ctrl+5
- * Bottom: _____ Ctrl+6
- * Isometric: _____ Ctrl+7

Selection Filter & Misc.

- * Filter Edges: _____ e
- * Filter Vertices: _____ v
- * Filter Faces: _____ x
- * Toggle Selection filter toolbar: _____ F5
- * Toggle Selection Filter toolbar (on/off): _____ F6
- * New SOLIDWORKS document: _____ F1
- * Open Document: _____ Ctrl+O
- * Open from Web folder: _____ Ctrl+W
- * Save: _____ Ctrl+S
- * Print: _____ Ctrl+P
- * Magnifying Glass Zoom: _____ g
- * Switch between the SOLIDWORKS documents: _____ Ctrl + Tab

SOLIDWORKS Quick-Guide[©]

Sample Customized Keyboard Shortcuts

SOLIDWORKS Sample Customized Hot Keys

Function Keys

F1	SW-Help
F2	2D Sketch
F3	3D Sketch
F4	Modify
F5	Selection Filters
F6	Move (2D Sketch)
F7	Rotate (2D Sketch)
F8	Measure
F9	Extrude
F10	Revolve
F11	Sweep
F12	Loft

Sketch

C	Circle
P	Polygon
E	Ellipse
O	Offset Entities
Alt + C	Convert Entities
M	Mirror
Alt + M	Dynamic Mirror
Alt + F	Sketch Fillet
T	Trim
Alt + X	Extend
D	Smart Dimension
Alt + R	Add Relation
Alt + P	Plane
Control + F	Fully Define Sketch
Control + Q	Exit Sketch

SOLIDWORKS® 2024 Advanced Techniques

Mastering Parts, Surfaces, Sheet Metal, SimulationXpress,
Top Down Assemblies, Core & Cavity Molds

- The perfect follow up to SOLIDWORKS Intermediate Skills
- Uses a step by step tutorial approach with real world projects
- Comprehensive coverage of advanced SOLIDWORKS tools and techniques
- Covers parts, surfaces, SimulationXpress, sheet metal, top-down assemblies and core and cavity molds
- Features a quick reference guide and a Certified SOLIDWORKS Professional practice exam

Description

SOLIDWORKS 2024 Advanced Techniques picks up where SOLIDWORKS 2024 Intermediate Skills leaves off. Its aim is to take you from an intermediate user with a basic understanding of SOLIDWORKS and modeling techniques to an advanced user capable of creating complex models and able to use the advanced tools provided by SOLIDWORKS. The text covers parts, surfaces, SimulationXpress, sheet metal, top-down assemblies and core and cavity molds.

Every lesson and exercise in this book was created based on real world projects. Each of these projects has been broken down and developed into easy and comprehensible steps for the reader. Furthermore, at the end of every chapter there are self test questionnaires to ensure that the reader has gained sufficient knowledge from each section before moving on to more advanced lessons. This book takes the approach that in order to understand SOLIDWORKS, inside and out, the reader should create everything from the beginning and take it step by step.

Who this book is for

This book is for the intermediate to advanced user who has already completed the SOLIDWORKS Basic Tools book and may have also completed the SOLIDWORKS Intermediate Skills book. People who are very familiar with SOLIDWORKS and its add ins will also find this book to be a valuable resource.

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ISBN 978-1-63057-635-6



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