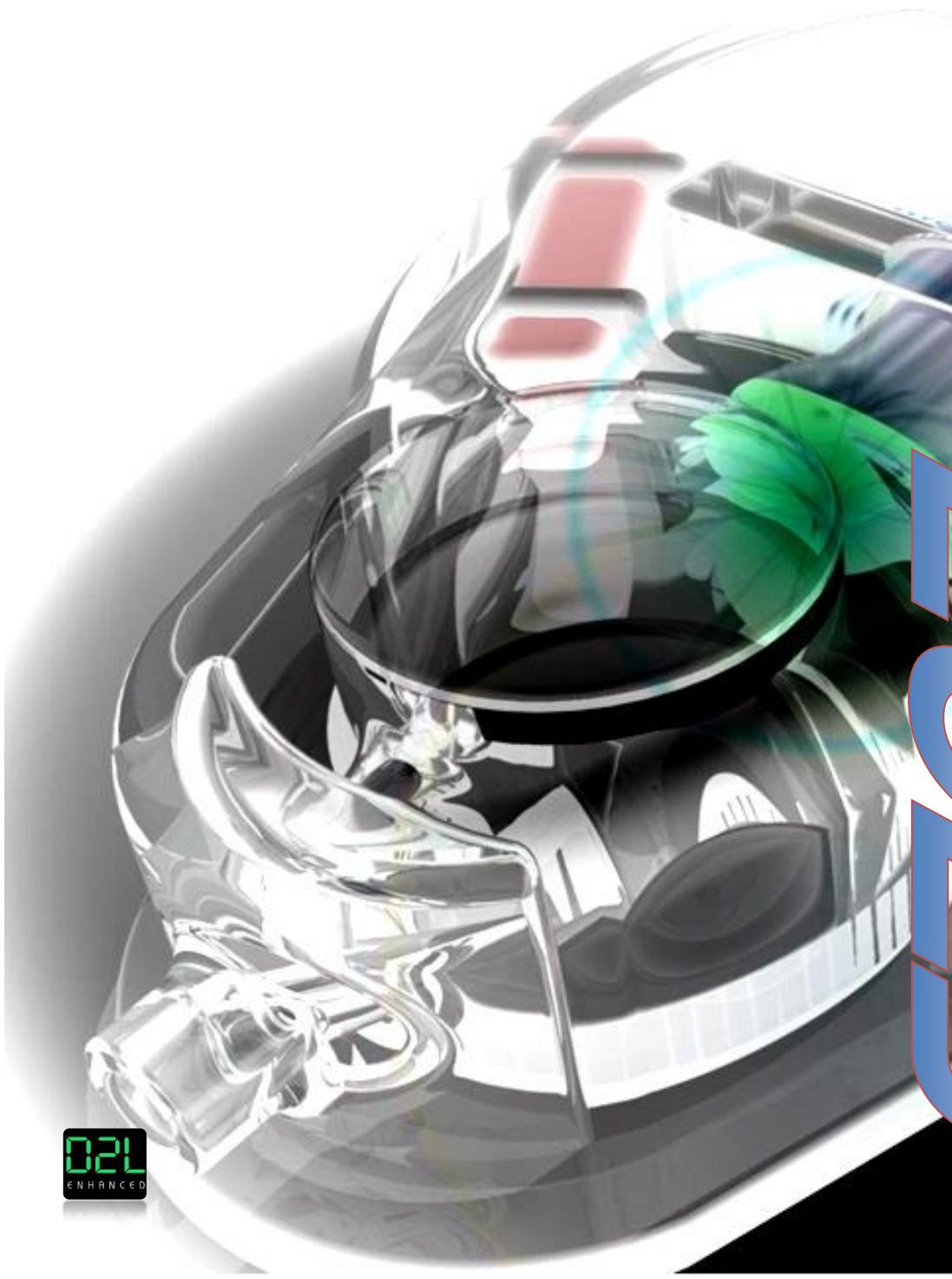




SOLIDWORKS BASICS

BY CHRISTOPHER F. SIKORA



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2023

COMPUTER AIDED DESIGN

COURSE SYLLABUS

SolidWorks Basics 120

Course Description:

SolidWorks Basics

3 credit hours

Exploration of the theory and application of solid modeling techniques for product design and manufacturing. Prerequisite: Intro to Engineering Drawings 101 or consent of instructor.

(1 lecture hours, 3 lab hours)

Course Objectives:

Provide the student with the knowledge and practical experience in the areas of 3D CAD modeling of parts, assemblies, and the creation of mechanical drawings from the models.

Textbook

SolidWorks Basics free/pdf. and videos provided on YouTube

<http://www.youtube.com/user/vertanux1>

Or simply search the exercise number (example: E5 SolidWorks)

Evaluation Scale:

- A 90% to 100%
- B 80% to 89%
- C 70% to 79%
- D 60% to 69%
- F Below 60%



Points:

Exercises	300 pts
Mid Term	300 pts
Final	300 pts
Participation/Attendance	<u>100 pts</u>
<u>Total</u>	1000 pts

General Course Outline

Date	Week	Topic
------	------	-------

1. Introduction to the Interface Lecture
Sketching and Base Feature Geometry Creation.
2. Revolved Features and Mirroring
3. Part Modeling - Draft, Patterns, Mirroring.
4. Sweeps, and Circular Patterns
5. Modeling Quiz and CAD Administration
6. Building Assemblies (Bottom-Up method "BU")
7. Creating Drawings. Review for Mid Term
8. Mid Term Exam
9. 3D Curves and Sweeps
10. Swept Blends/Lofting
11. Assemblies Creation (Top-Down Method "TD")
12. Assembly/Part Editing ("TD" & "BU" Methods)
13. Q2 Assembly Project
14. Q2 Assembly Project (continued)
15. Lab time, Review for Final Exam
16. Final Exam

Recommended Hardware

Flash Drive (>8 GB)

STUDENTS WITH DISABILITIES

We welcome students with disabilities and are committed to supporting them as they attend college. If a student has a disability (visual, aural, speech, emotional/psychiatric, orthopedic, health, or learning), s/he may be entitled to some accommodation, service, or support. While the College will not compromise or waive essential skill requirements in any course or degree, students with disabilities may be supported with accommodations to help meet these requirements.

The laws in effect at college level state that a person does not have to reveal a disability, but if support is needed, documentation of the disability must be provided. If none is provided, the college does not have to make any exceptions to standard procedures.

All students are expected to comply with the Student Code of Conduct and all other college procedures as stated in the current College Catalog.

PROCEDURE FOR REQUESTING ACCOMMODATIONS:

1. Go to SRC108 and sign release to have documentation sent to the college, or bring in documentation.
2. Attend an appointment that will be arranged for you with the ADA coordinator or designee.

CLASSROOM PROCEDURES:

1. Attendance of each scheduled class meeting is required unless otherwise specified by the instructor.
2. Daily work problems and hand-outs will be maintained in a notebook and turned in upon the instructor's request.
3. Reading assignments will be made prior to discussing the material.
4. Keep your drafting workstation clean and free of miscellaneous materials.
5. Please report any malfunctioning equipment to the instructor.

LABORATORY UTILIZATION:

1. Regular daytime hours. The room is open for your use starting at 9:00AM daily. Even though classes are being held, you are encouraged to find an open area and work in the laboratory.
2. There are evening classes, but you may use the lab up to 10:00PM.
3. On weekends, the lab will be available on Saturdays from 9:00AM to 4:00PM. The lab will be closed on Sundays.

INSTRUCTOR'S RESPONSIBILITY:

1. Present material in a manner that can be understood by each student.
2. Respect each student as an individual, to be of assistance in any way possible, and to help solve problems, but not to solve problems for the student.
3. Keep records of your progress and to summarize your learning experiences with a final

Attendance and Cheating Policies

Introduction: Drafting is a technical profession in our society; consequently, presentations in this course are factual and technical, and final grades represent the student's accomplishment of the learning activities.

Attendance: Attendance at each class meeting is required. Attendance may be a factor when determining the final grade. Your instructor will specify his/her policy concerning the relationship of attendance and the final grade.

Each instructor has the option of taking attendance for his/her personal use. If a student misses class because of illness, a field trip, or any other AUTHORIZED reason, the student is obligated to determine what was missed, and will be held responsible for that work. If a student is absent without an excused absence, he/she will also be held responsible, and must obtain all information from some source other than the class instructor. Instructors DO NOT have to accept any make-up work, do individual tutoring, or make special test arrangements for any UNEXCUSED ABSENCE.

Cheating: Cheating in this department is interpreted to mean the copying, tracing, or use of another person's work for the purpose of completing an assignment.

Individual initiative and personal performance in completing all assignments is required of all students. This course may seem to offer situations that are conducive to cheating. However, evidence of cheating on the part of any student will be sufficient cause for an assignment of an "F" for the course.

Instructors reserve the right to change a grade after the end of the semester if there is evidence to warrants.

Exercises



E1 Basic Modeling 1 Tutorial

1:12:40



E2 Basic Modeling 2 Tutorial

39:46



E3 Basic Modeling 3 Tutorial

39:20



E4 Basic Modeling 4 Tutorial

44:08



E5 Assembly Basics 1

25:37



E6 Detailing Basics 1 Tutorial

16:19



Midterm Exam Review

14:30



E7

Basic Modeling 5 Tutorial

34:10



E8

Basic Modeling 6 Tutorial

46:08



E9

Assembly Basics 2 Tutorial

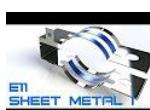
18:08



E10

Assemblies 3 Tutorial

26:07



E11

Sheet Metal I Tutorial

27:25



Final

Exam Review

17:08



HOW TO MAKE A PORTFOLIO

CAD 120 TOTALS (*E – Exercise, L-Lab, Q-Quiz*)

- E1 - 10pts
 - L1 – 5pts
 - L1b – 5pts
- E2 – 30pts
 - L2 – 5pts
 - Q1 - 10pts
- E3 – 30pts
 - L3 - 5pts
 - L3b – 5pts
- E4– 30pts
 - L4 – 5pts
- E5– 30pts
 - L5b - 10pts
- E6– 30pts
 - L6 - 10pts
- E7– 30pts
 - L7 - 5pts
- E8– 30pts
 - L8 – 5pts
- E9– 30pts
 - L10 – 5pts
- E10– 30pts
 - L11 – 5pts
 - L11c - 5pts
- E11– 30pts
 - L11d – 5pts

MIDTERM – 300pts

FINAL – 300pts

TOTAL - 1000pts

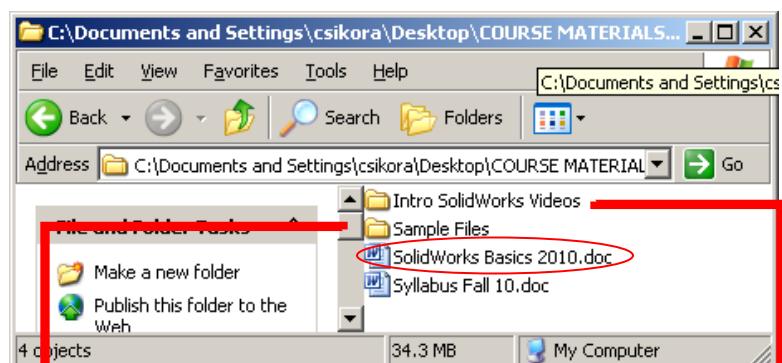
CAD 120 - COURSE MATERIALS and FILE CONTENTS

Every student is provided with a Training Manual.doc, Lecture Videos, and Sample Files.

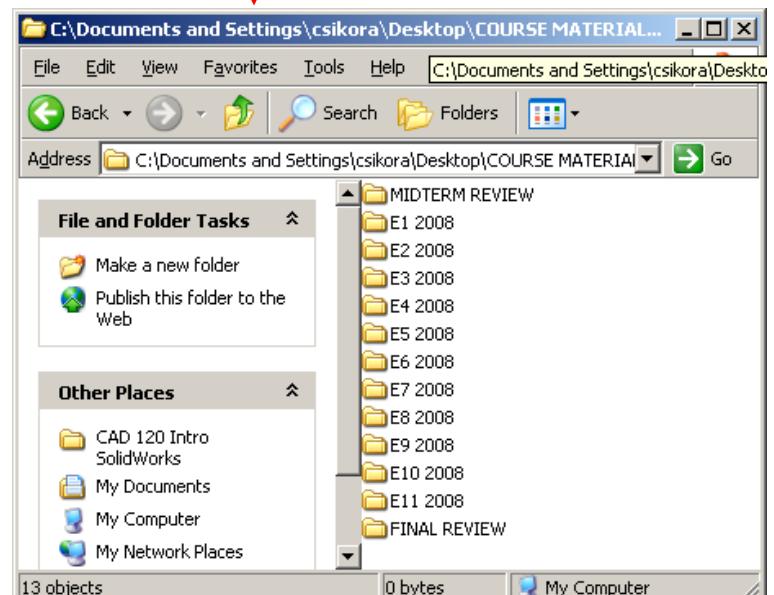
These files are located on www.vertanux1.com or

W:CAD Classes/Sikora/SolidWorks/CAD120 folder.

You can copy and paste the entire folder onto your USB Flash Drive.



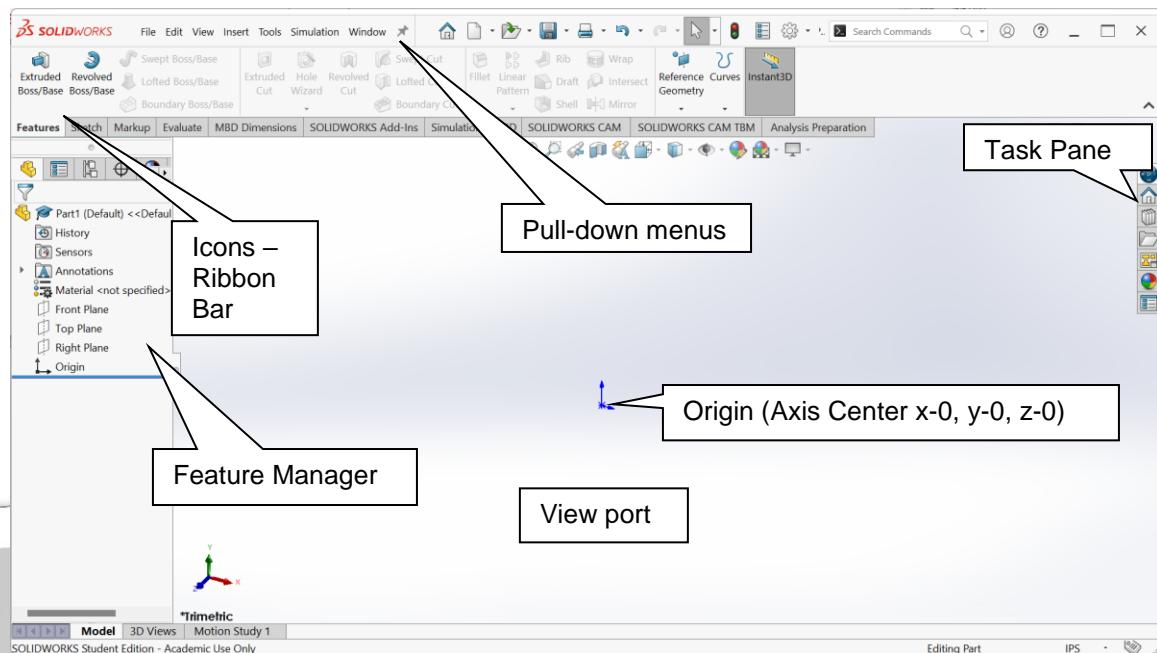
The first folder contains the training manual and syllabus and subfolders containing the training videos and work files needed to complete some of the exercise, as well as practice labs.
The Training Manual is in MS Word format so students have the ability to edit and add notes to it.



The Work Files folder contains all the part and assembly files needed to construct the exercises, labs and Midterm and Final exams.

Introduction to SolidWorks

2020-21 Educational Interface



Mouse Buttons

Left Button - Most commonly used for **selecting** objects on the screen or sketching.

Right Button – Used for activating pop-up **menu** items, typically used when editing.

Center Button – (option) Used for model **Rotation**, **Pan** when holding Ctrl key, and **Zoom** when holding Shift key.

Center Scroll Wheel – (option) same as Center Button when depressed, only it activates **Zoom** feature when scrolling wheel.

Options menu “*The heart of SolidWorks*”...

Selecting the Grid Icon opens the Options Menu or the Tools/Options pull down can be used.

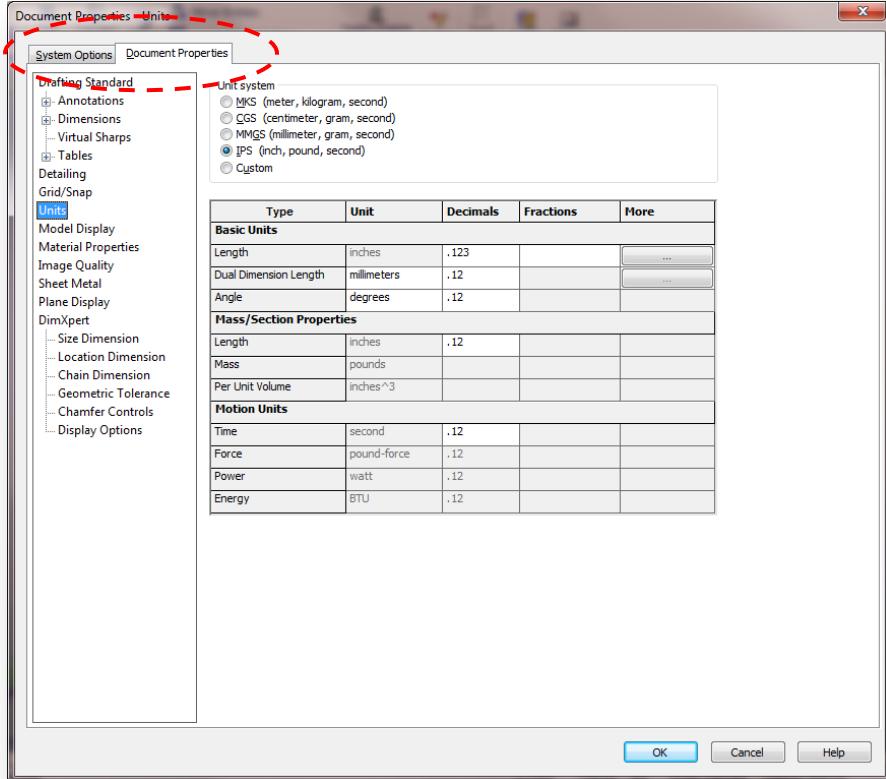


System Options – (Global Settings) are setting that affect all documents.

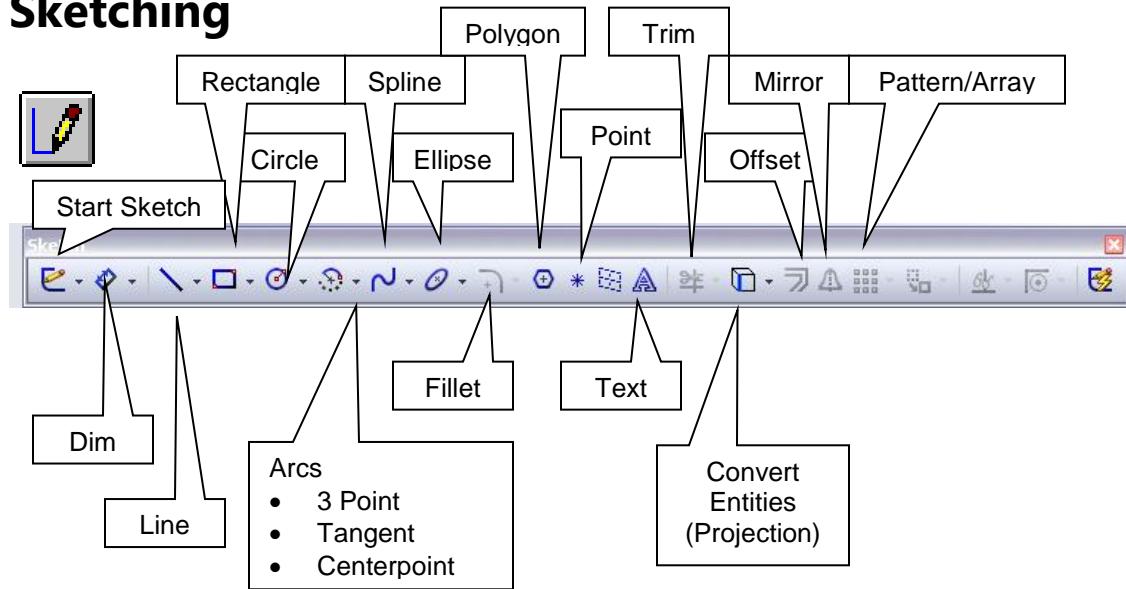
- Screen Background Color
- Performance Settings
- Interface Preferences

Document Options – (Individual Settings) are setting that adjust only the currently active document.

- Units (metric or inches)
- Note and Dimension Fonts
- Part Color



Sketching

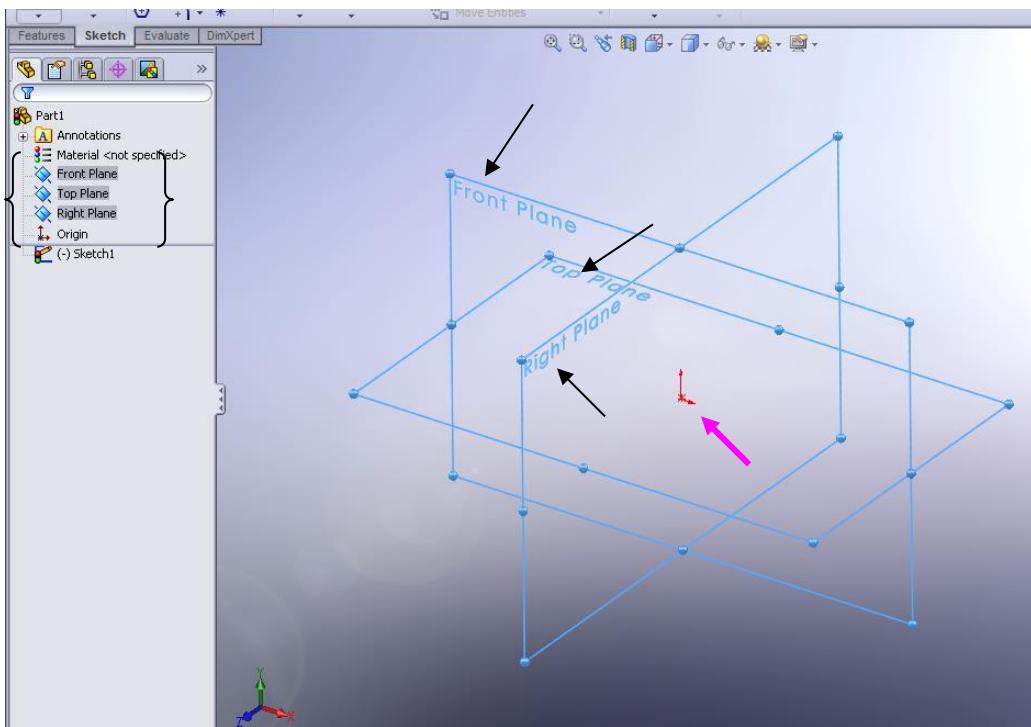


NOTE: If you do not see all of these icons on your interface you can customize the toolbars to bring them up. Right mouse button click on the top grey frame of the window and locate the "customize" option.

Where do you start a sketch?

Sketches can be created on any Plane or Planar Face or Surface. SolidWorks provides you with three planes centralized at the **Origin** (your zero marker in space)

NOTE: Planes can also be created and will be discussed in more detail in the future.



To start a sketch Pre-select the plane or face you desire to sketch on and then select the Sketch Icon.  **NOTE:** You can select the planes from the "Feature Manager".

Special Note: You can identify what plane you are sketching on by the X and Y arrows displayed on the red origin.



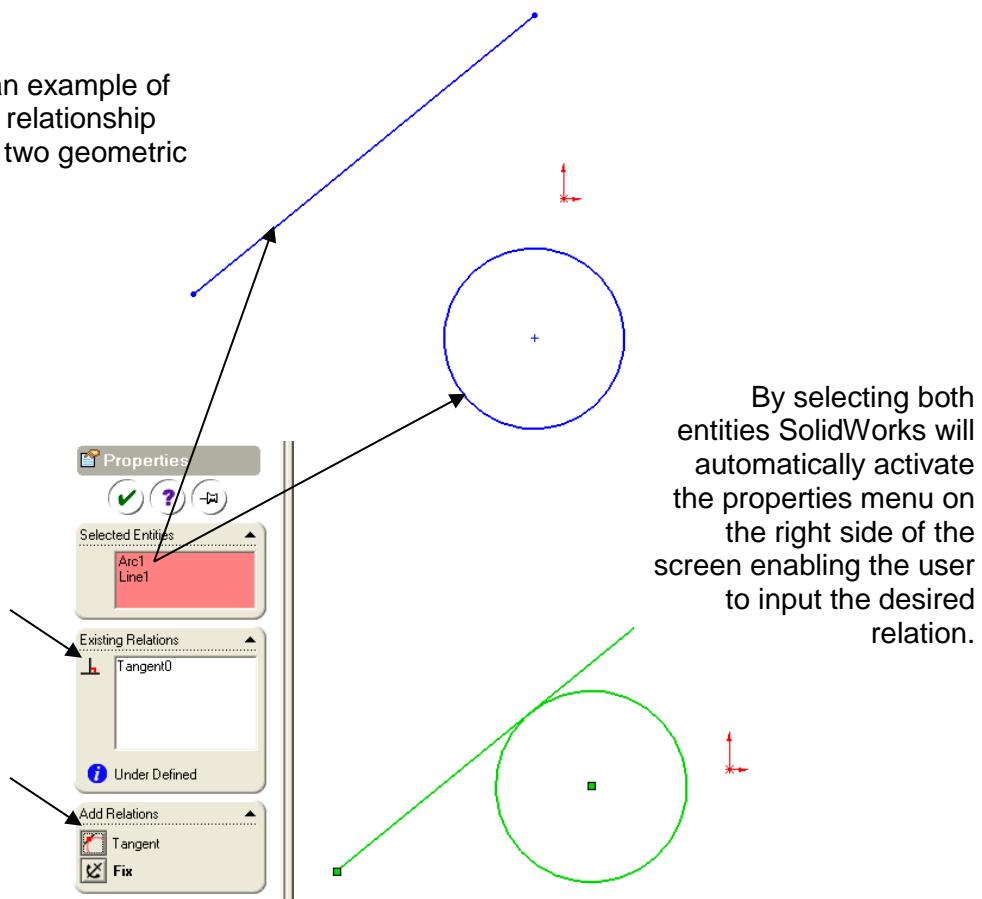
Controlling your geometry...

SolidWorks uses two methods for constraining geometric entities.

Relations and **Dimensions**

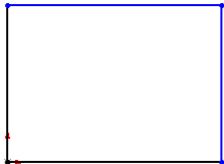
Relations can be referred to as common elements of geometry such as Tangency, Parallelism, and Concentricity. These elements can be added to geometric entities automatically or manually during the design process.

Here is an example of adding a relationship between two geometric entities.



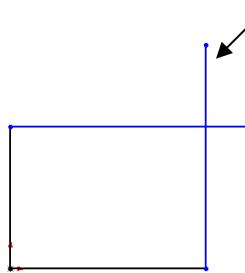
Cautious sketching can save time

Here are some tips to avoid modeling errors.



Good

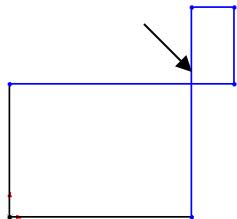
This is acceptable because it is a closed contour.



Bad

Use the "Trim" tool to cut and extend.

This is unacceptable because it has untrimmed geometry.



Ugly

This is unacceptable because it has multiple contours share a common entity.

There are 3 primary file types in SolidWorks, which include...

1. *Part* (.sldprt)

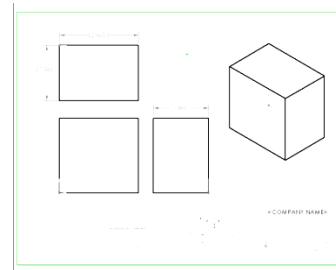
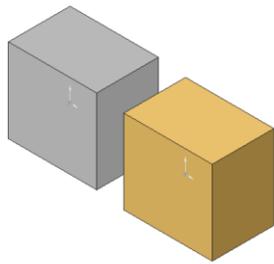
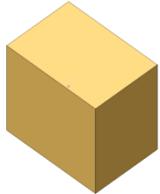
Single part or volume.

2. *Assembly* (.sldasm)

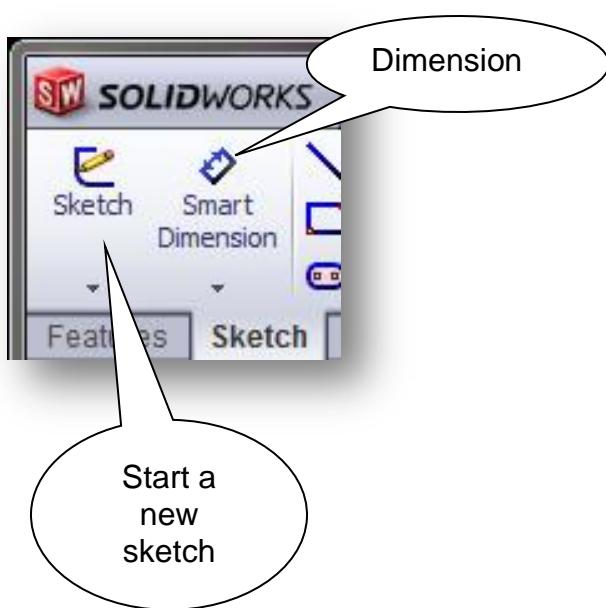
Multiple parts in one file assembled.

3. *Drawing* (.slddrw)

The 2D layout containing views, dimensions, and annotations.



Sketch Relations



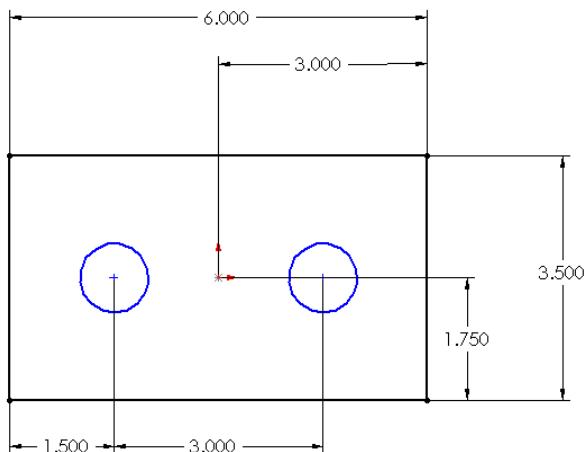
SKETCH COLORS

BLUE - Under Defined

BLACK - Fully Defined

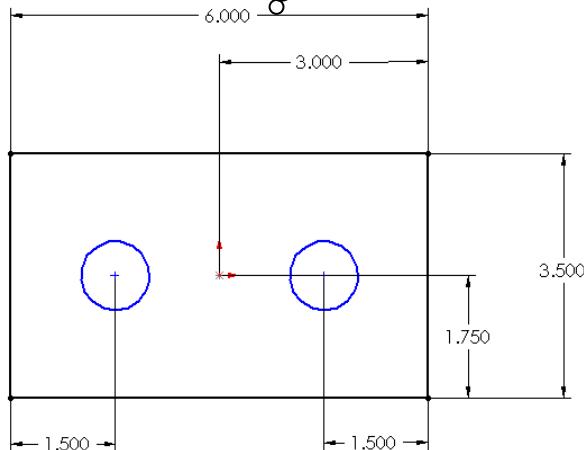
YELLOW - Over Defined

Controlling your geometry with dimensions



Dimensioning this way will enable the length of the bracket to change but the holes will always remain positioned to

Dimensioning this way will enable the length of the bracket to change but the holes will always remain positioned to the left side.



Solid Modeling Basics

Layer Cake method



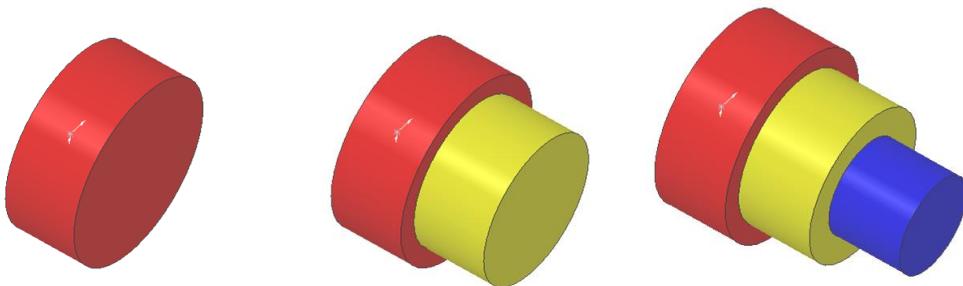
Extruded Boss/Base (Creates/Adds material)



Extruded Cut (Removes material)

Ingredients:

- Profile



Revolve method



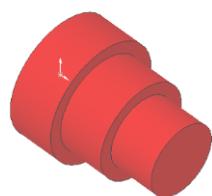
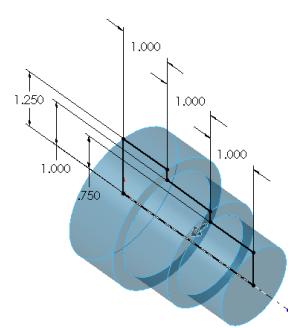
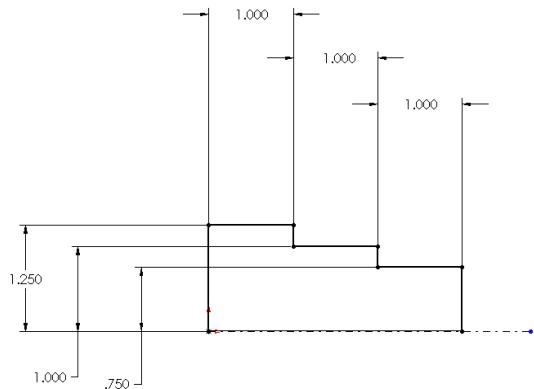
Revolve Boss/Base (Creates/Adds material)



Revolve Cut (Removes material)

Ingredients:

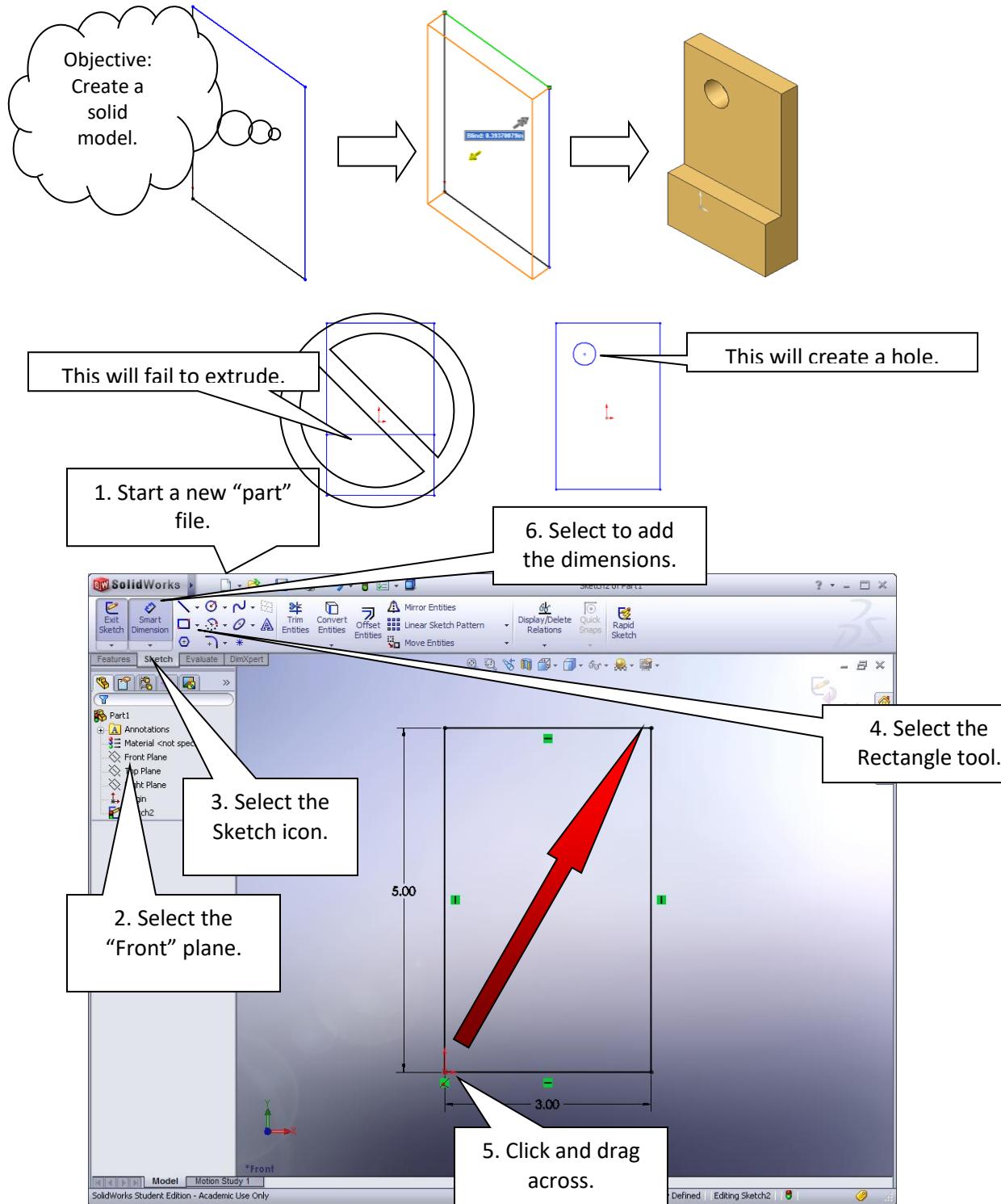
- Profile
- Center Line (*Note: The profile cannot cross over the center line!*)

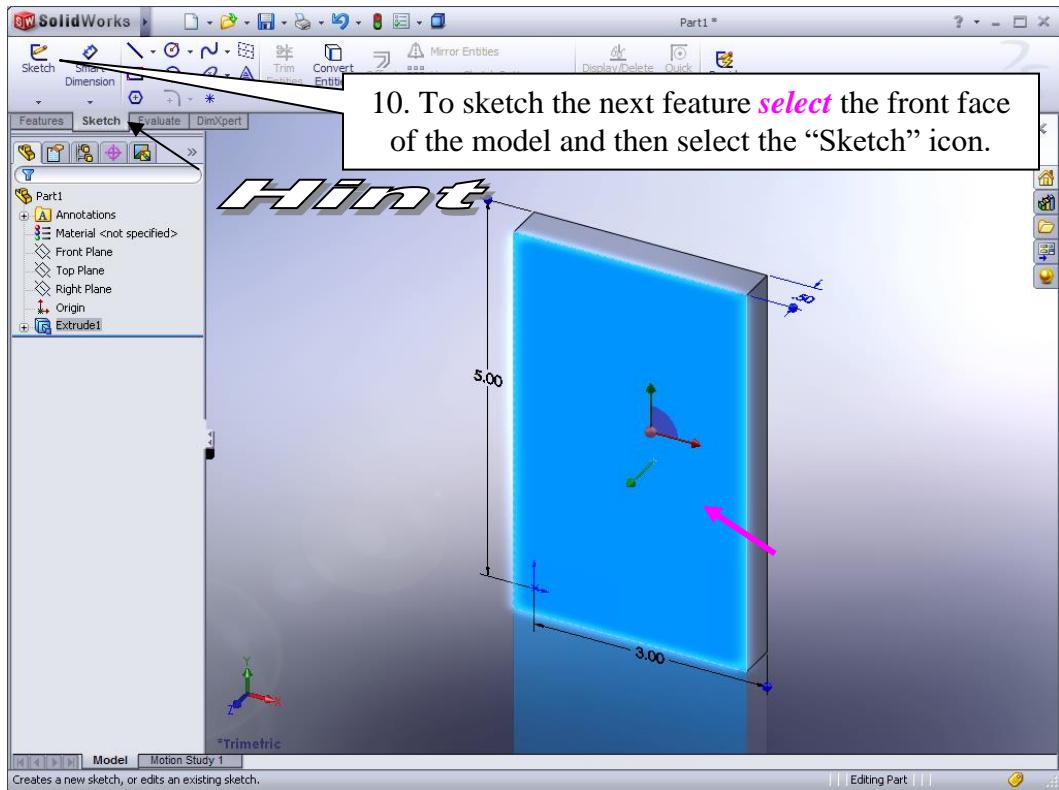
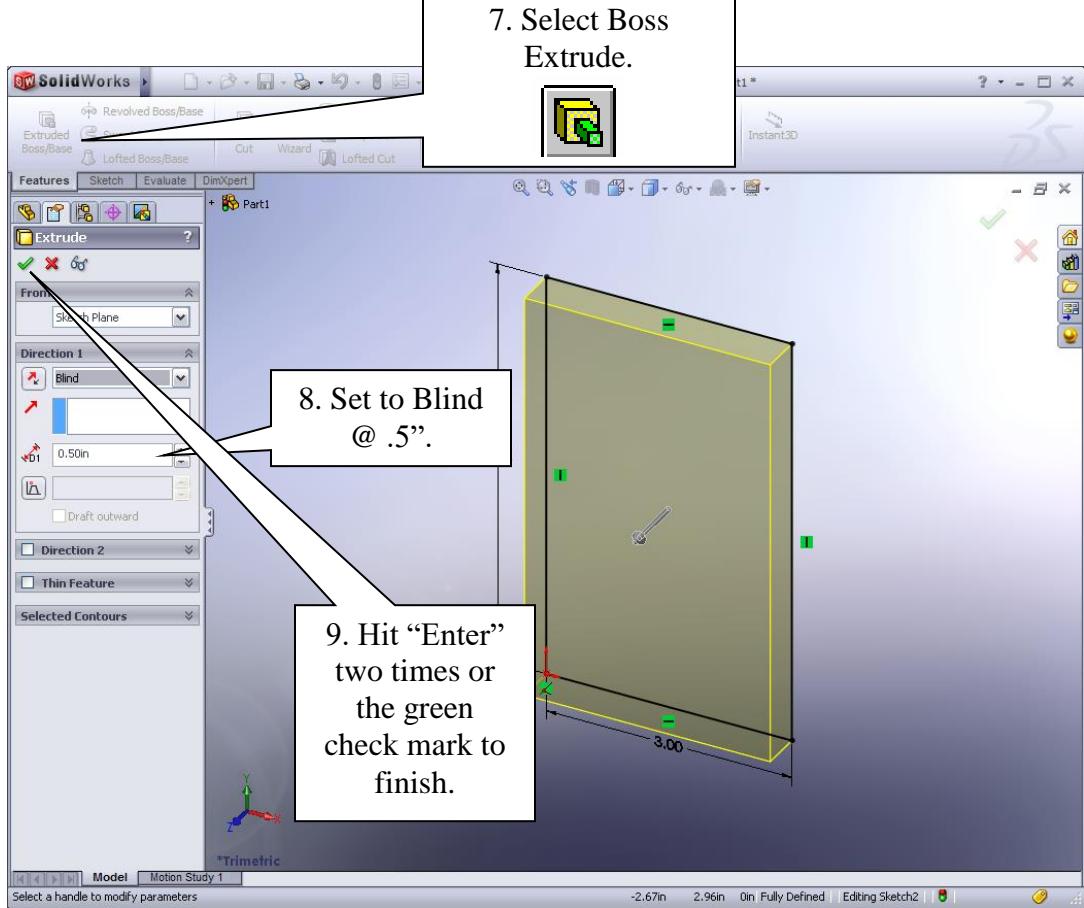


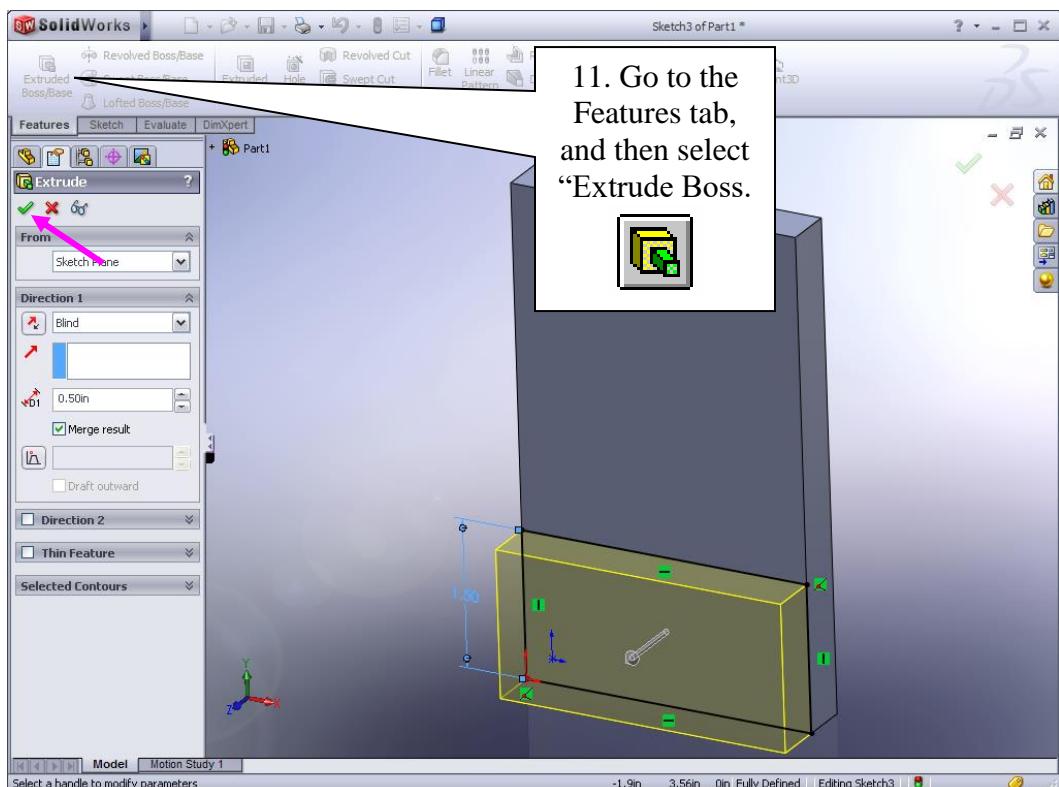
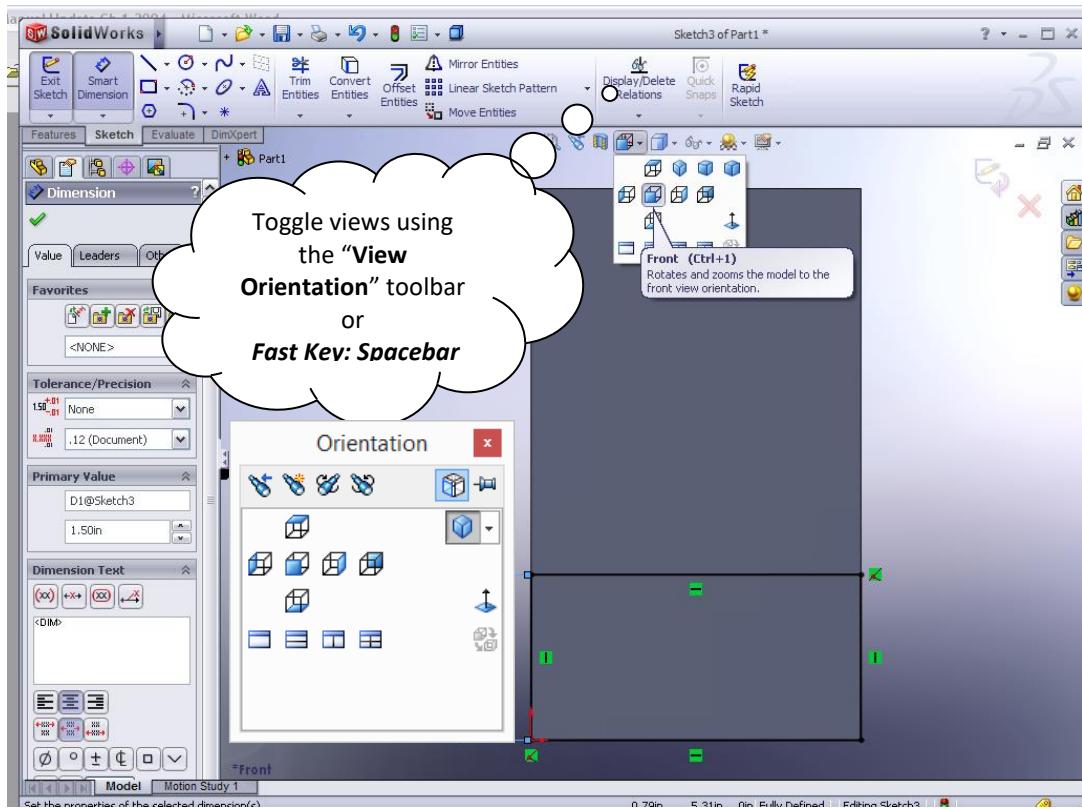
EXERCISE 1

Introduction to basic part modeling

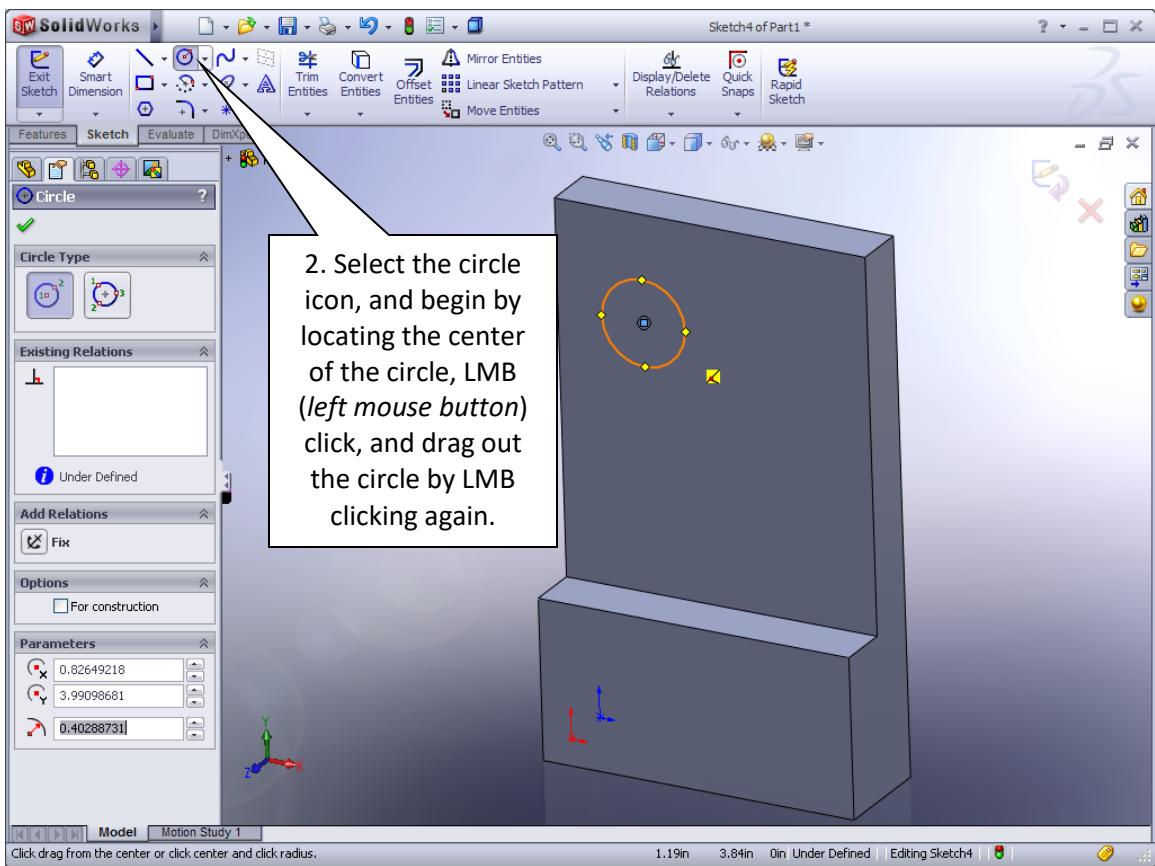
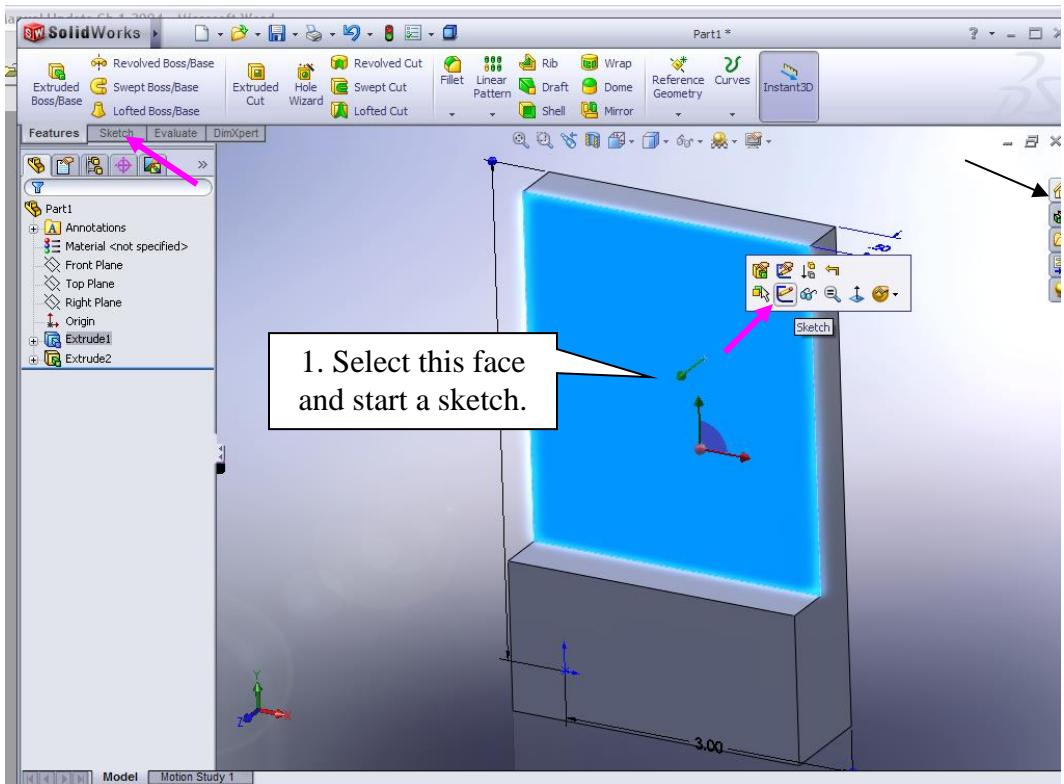
Base Extrude Features create a 3D solid representation by extruding a 2 dimensional profile of the entity.

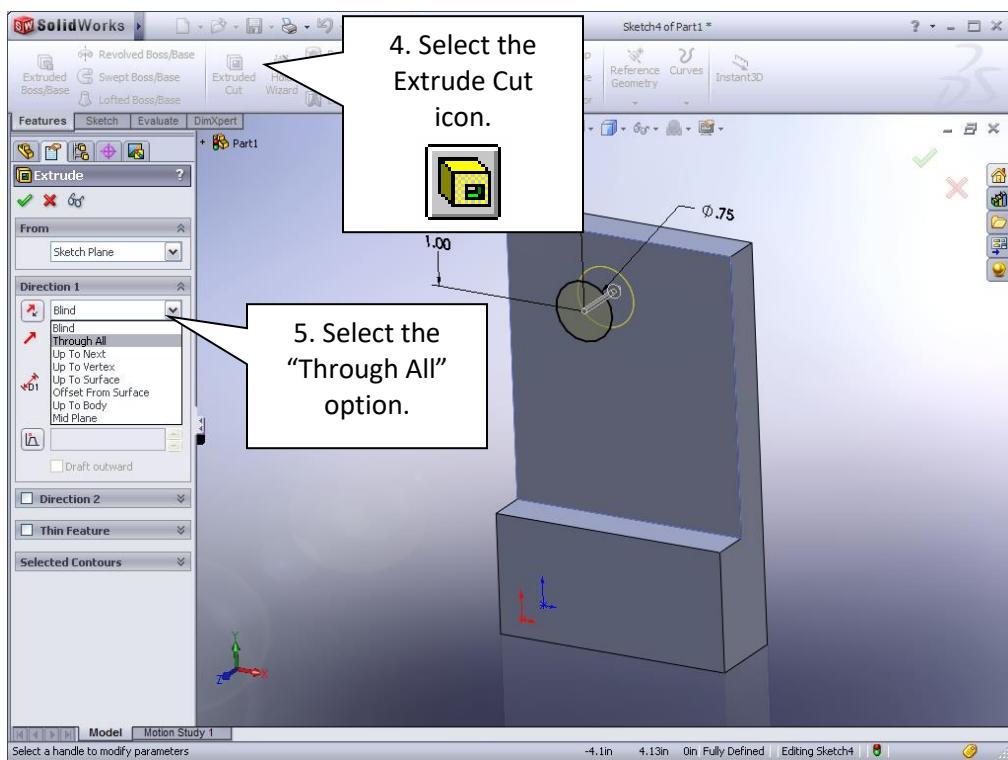
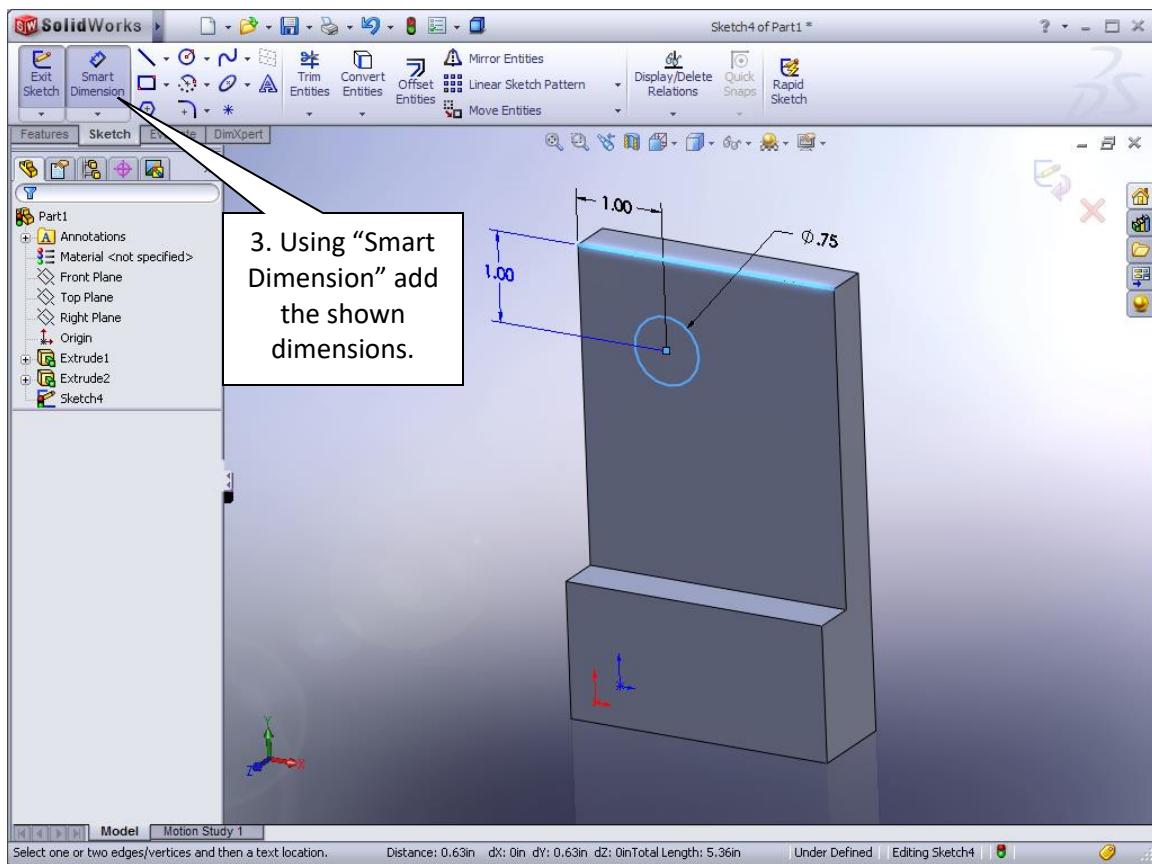


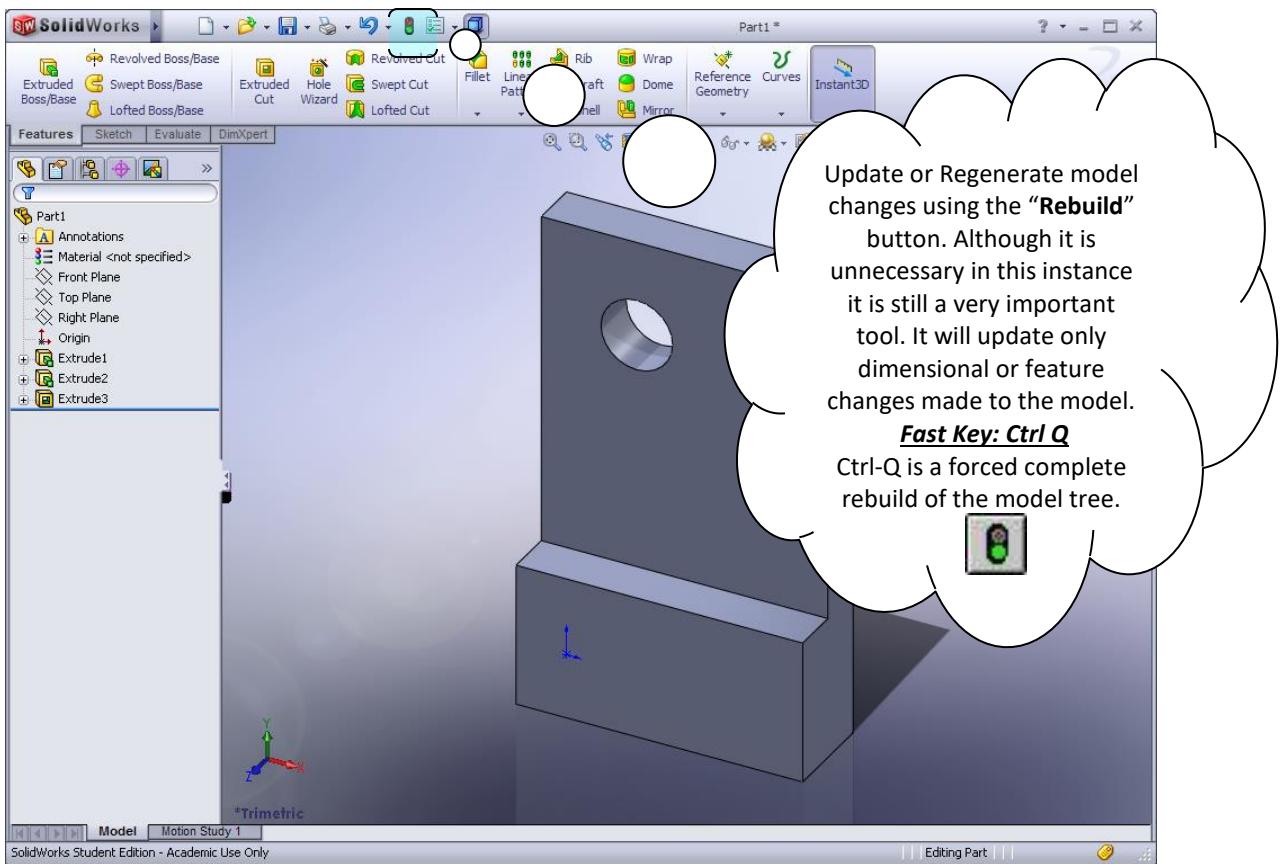




Adding the hole







Go to file save and save-as "E1"

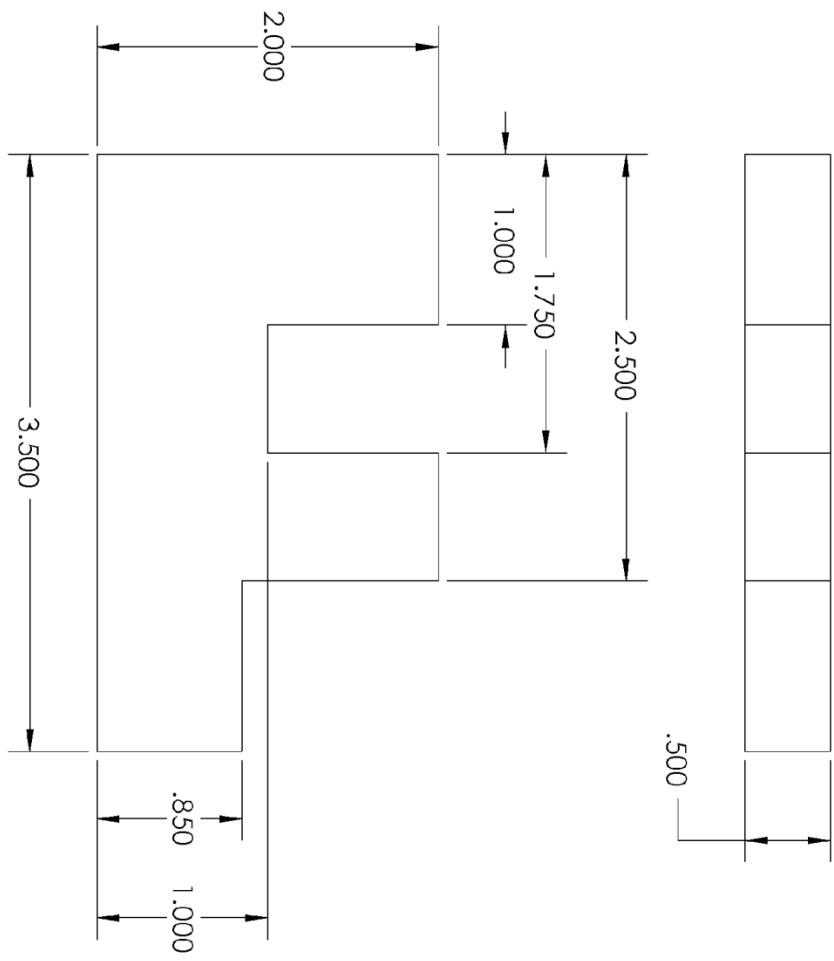
FINISHED

Now try LAB1...

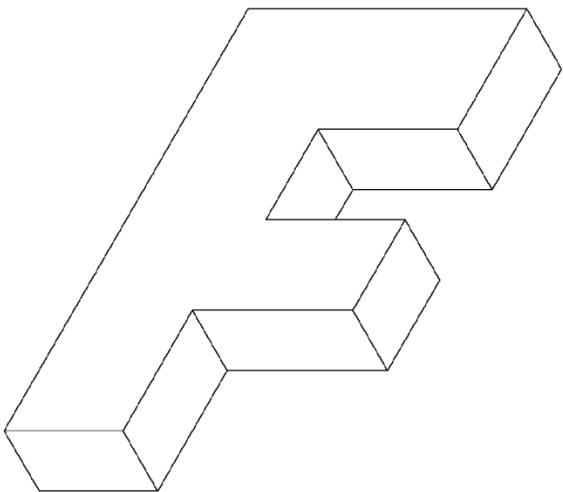
NOTE: Patterns/Arrays and Mirroring will be covered in the next three chapters.

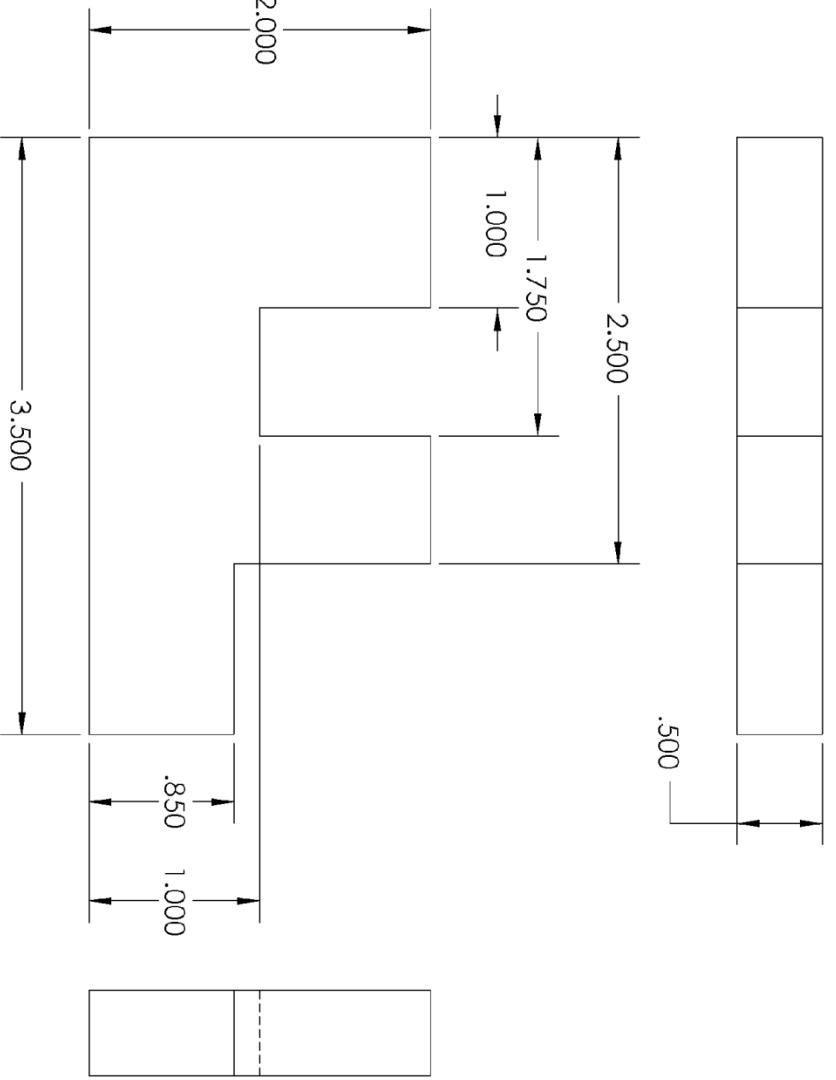
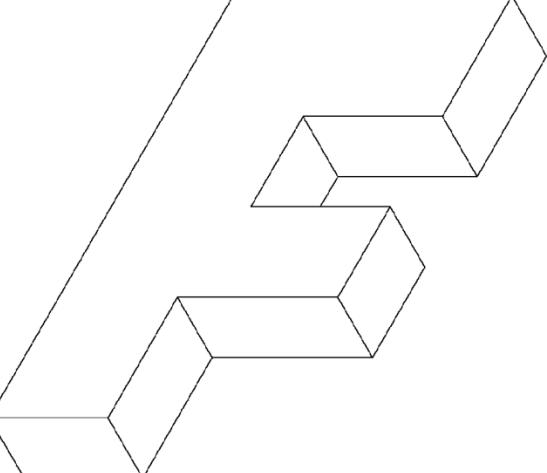
Please try to model LAB 1 without using them. It's good practice to just dimension and sketch all geometry when first starting out learning this software.

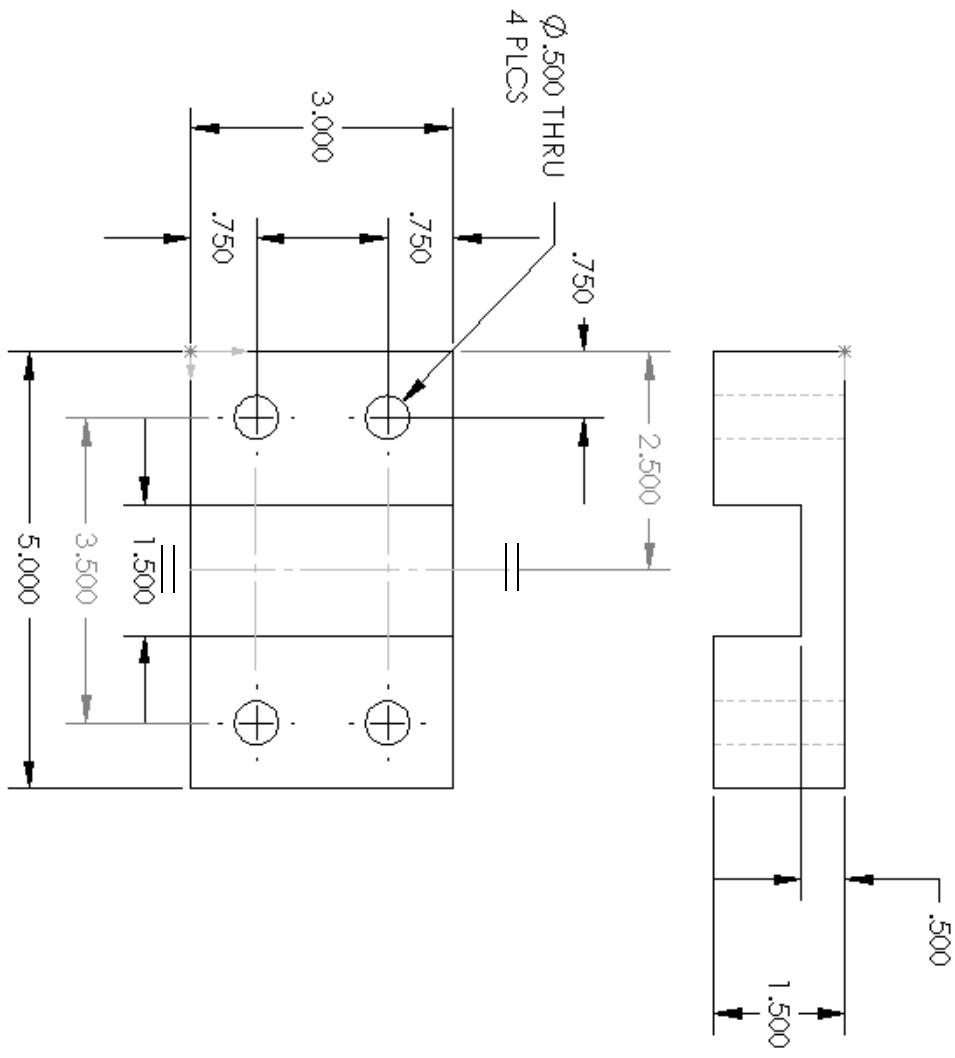
Please understand that I don't want to overwhelm you with too much information the first day. It is my goal to help you succeed, not to fail.



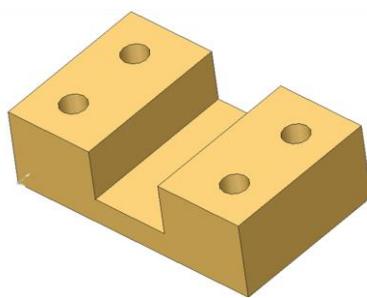
1. MODEL THE PART
2. Position in an Isometric View
3. Save-as PDF
4. Submit in Assignments section



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<p>L1</p> <p>1. MODEL THE PART</p> <p>2. Position it in an Isometric View</p> <p>3. Save as PDF</p> <p>4. Submit in Assignments section of D2L</p>																																																											
																																																											
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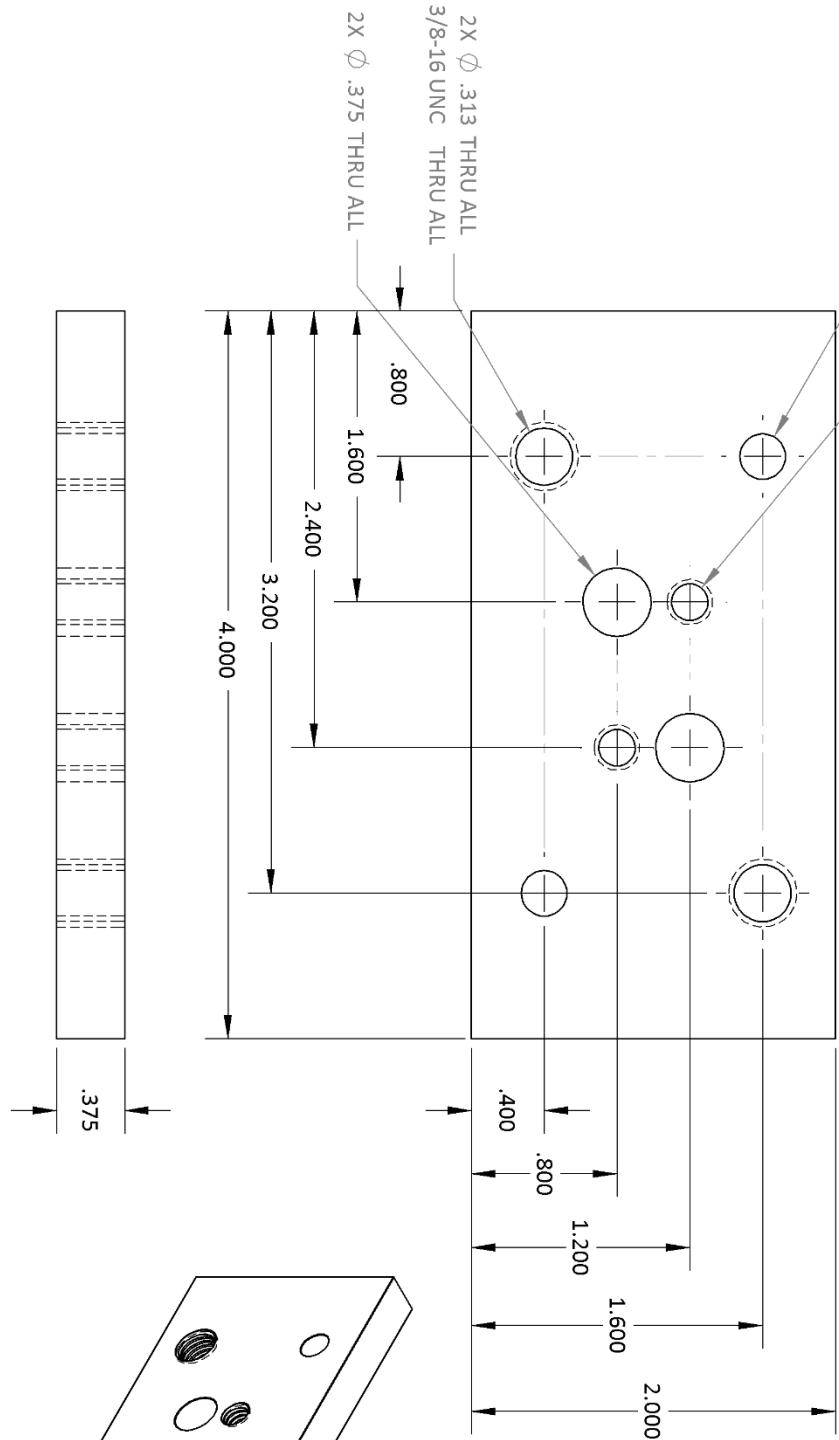
- **MODEL THE PART**
- **Position In an Isometric View**
- **Save-as PDF**
- **Submit In Assignments section of D2L**



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DIMENSIONS ARE IN INCHES	INCHES	
TO LEADER:	INCHES	
FRACTIONAL ⁺	DECIMAL	
ANGULAR: MACH ⁺	BEND [±]	
TWO PLACED DECIMAL [±]	INC APP.	
THREE PLACED DECIMAL [±]	INC APP.	
WALLS	D.A.	
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1/4-20 UNC THRU ALL

2X Ø .250 THRU ALL



OPTIONAL



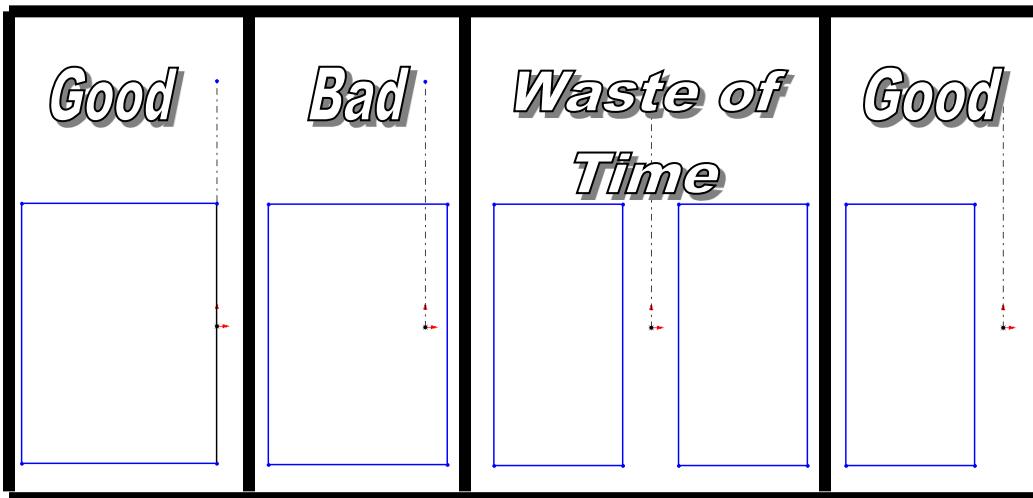
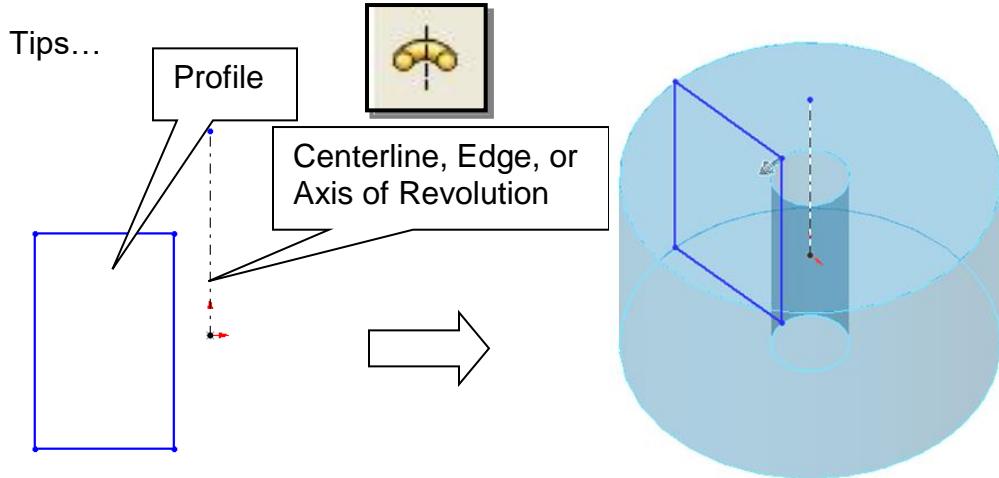
INT 103 - I

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		CHECKED ENG APPR.		TITLE:	
		INTERPRET GEOMETRIC TOLERANCING PER:		MFG APPR.	
		MATERIAL		Q.A.	
		COMMENTS:			
APPLICATION	NEXT ASSY	USED ON	FINISH	SIZE A	DWG. NO.
				SCALE: 1:1	WEIGHT: SHEET 1 OF 1
					REV:

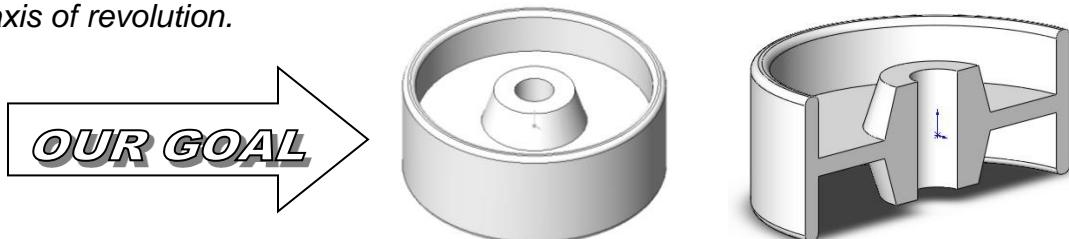
INT 103_I

EXERCISE 2
Revolved Features

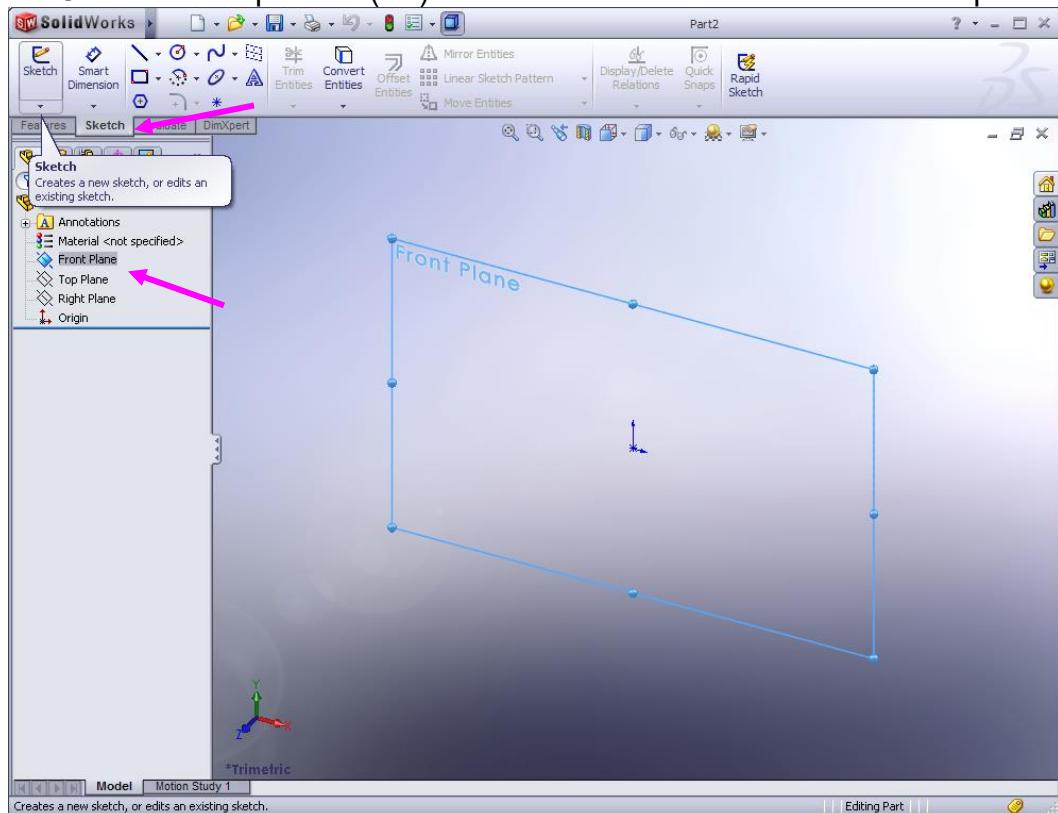
Revolved Feature - creates features that add or remove material by revolving one or more profiles around a centerline. The feature can be a solid, a thin feature, or a surface.



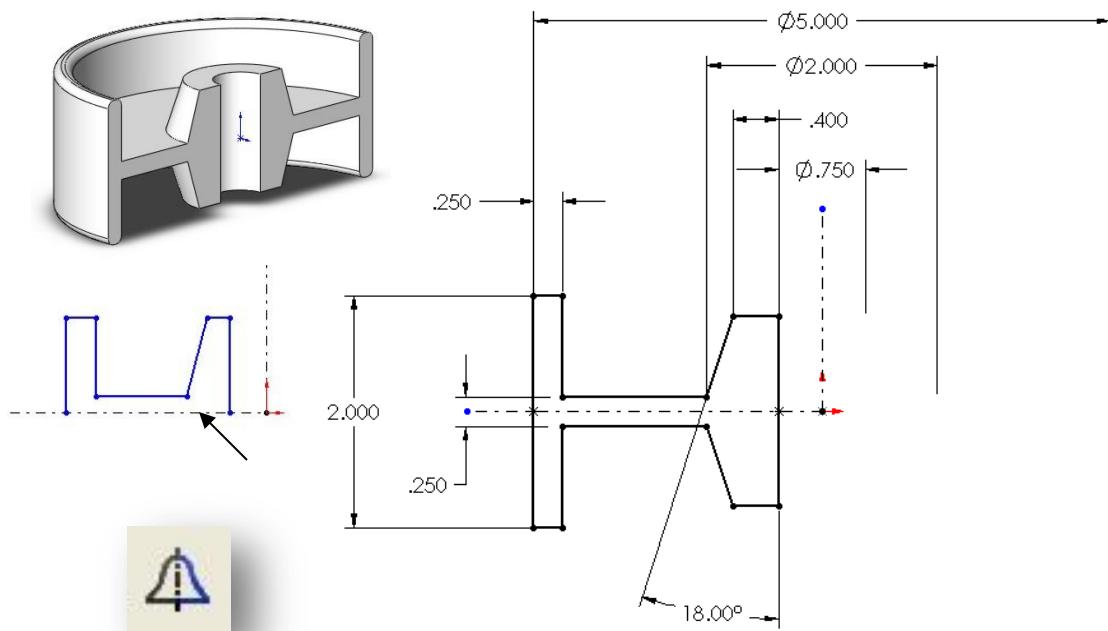
The profile should never cross over the centerline, nor should there be profiles on both sides of the centerline. Note: Since SolidWorks 2004 Centerlines are not always necessary if there is an available sketch edge which can be used as an axis of revolution.



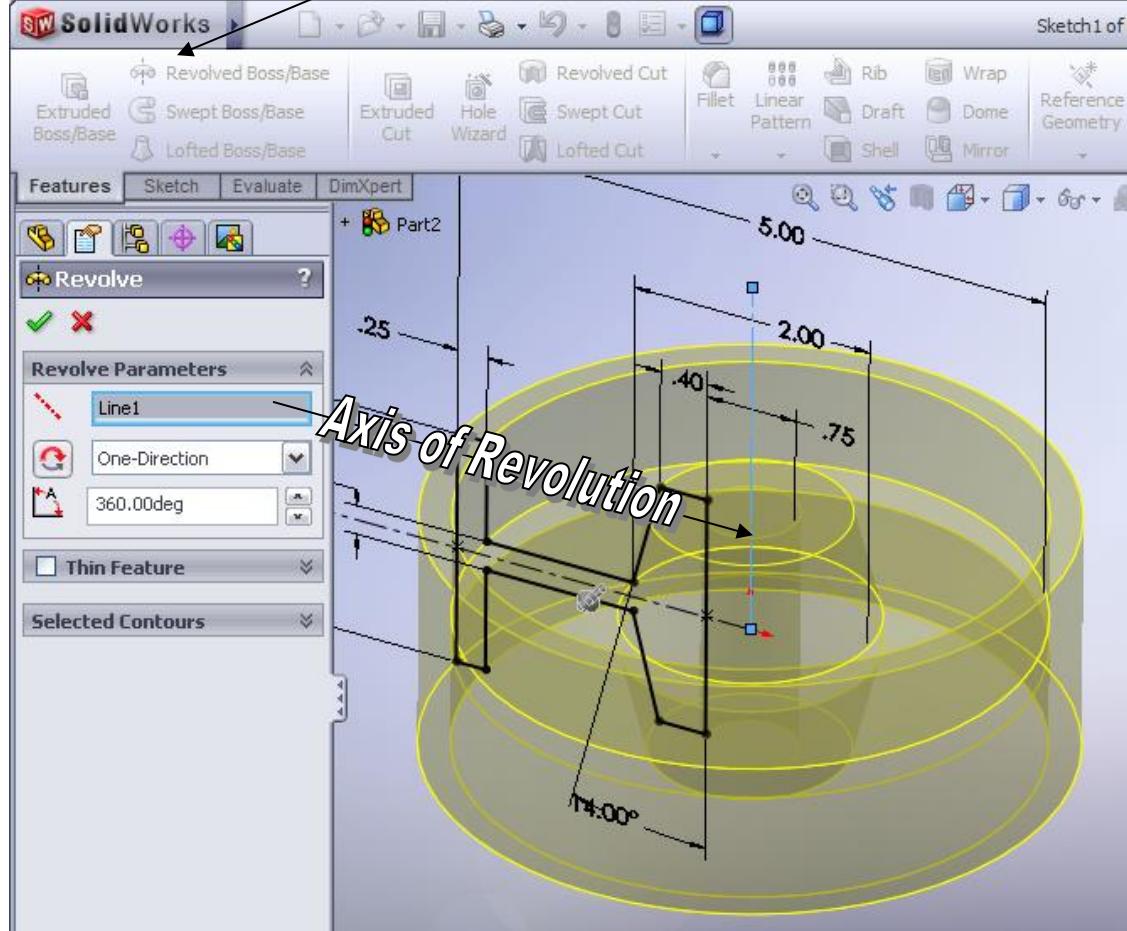
1. Create a new part file (E2) and then start a sketch on the “Front” plane.



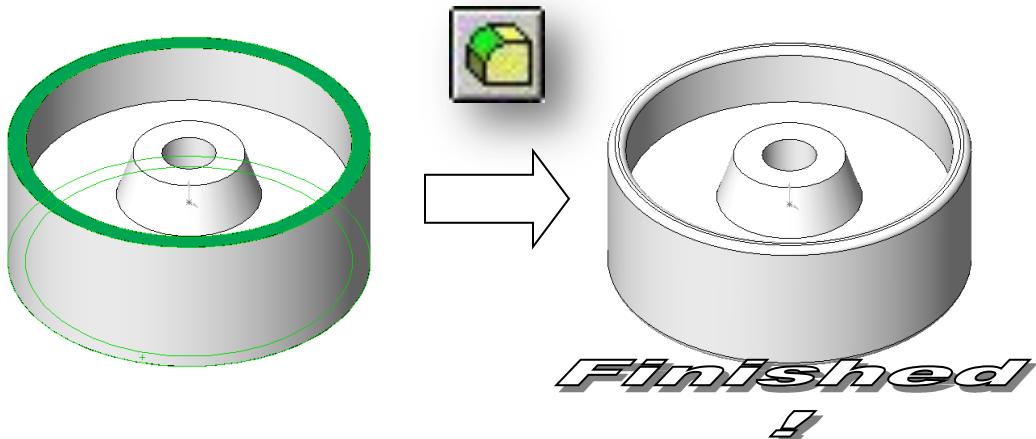
2. Sketch the following. Ctrl select the profile and the horizontal centerline, then using the “Mirror” tool to create a $\frac{1}{4}$ of the geometry and then mirror it to the other side. Make sure you finish adding the dimensions.

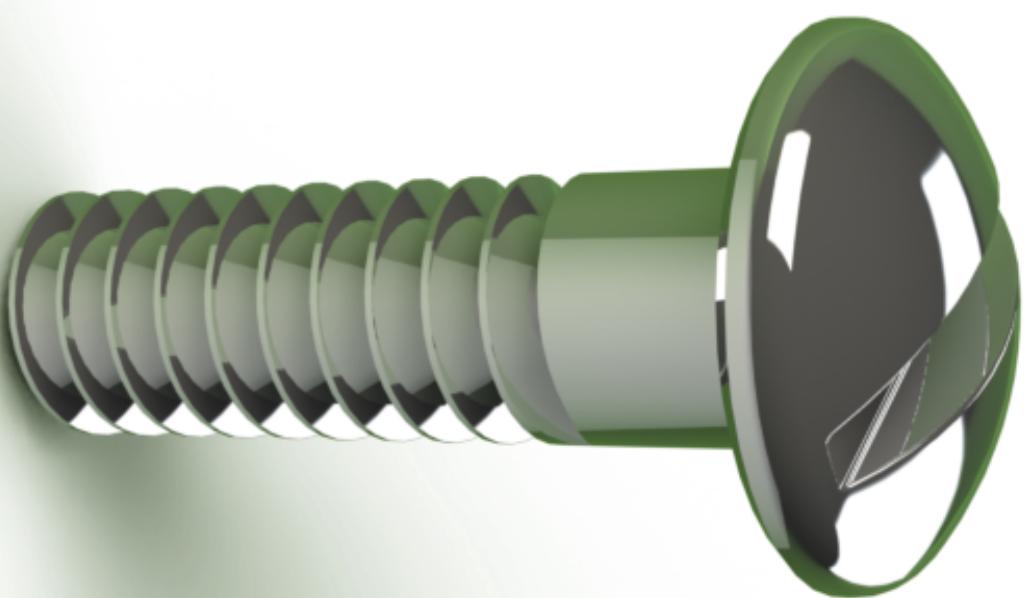


3. Select the Revolve feature icon. Then select the axis/centerline.

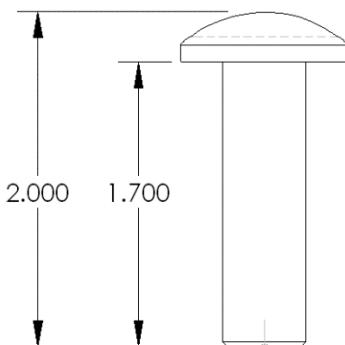
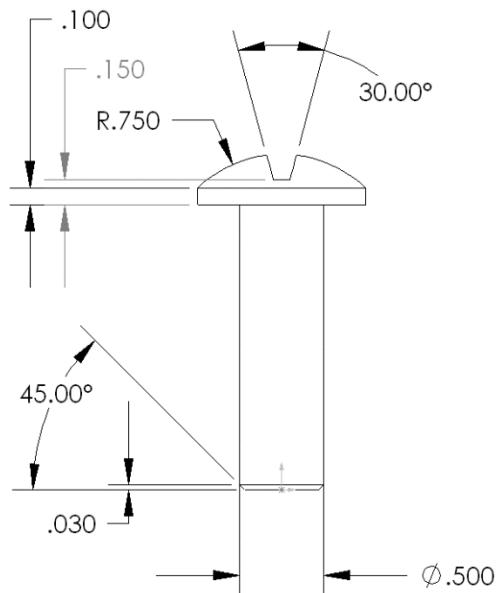
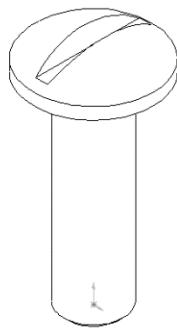
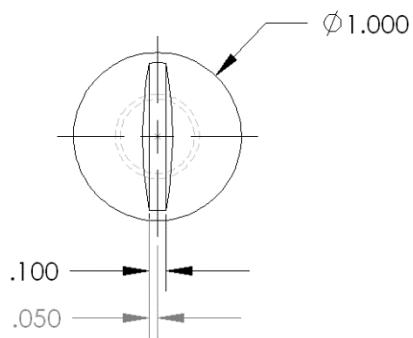


4. Select the top and bottom faces and add a .100" fillet.





REVISIONS			
ZONE	REV.	DESCRIPTION	DATE

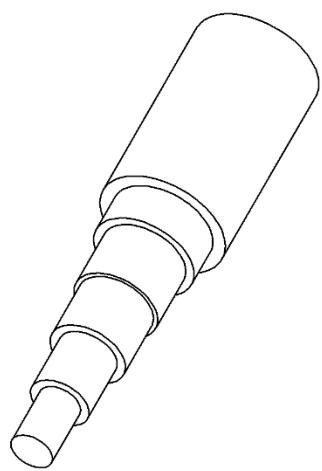
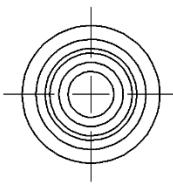
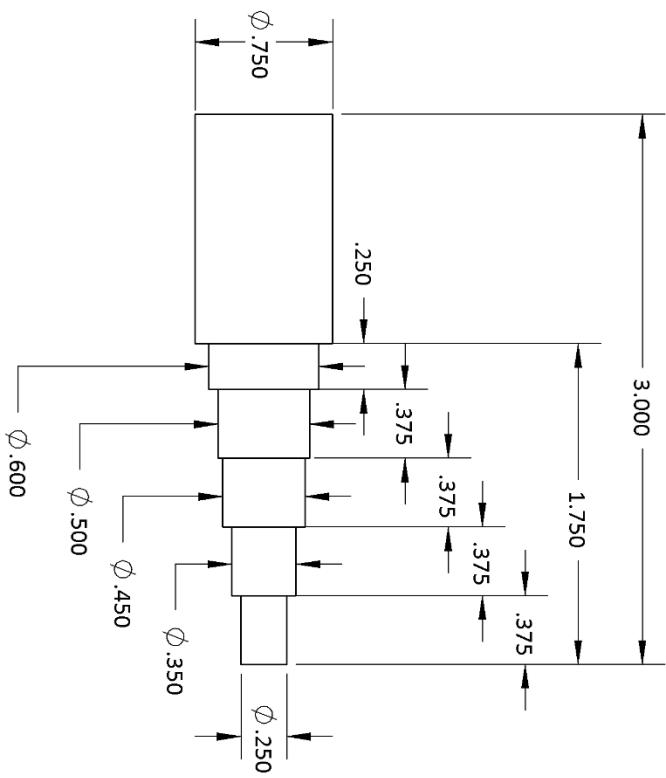


1. MODEL THE PART
2. Position in an Isometric View
3. Save-as PDF
4. Submit in Assignments section of D2L

		DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL \pm ANGULAR: MACH \pm BEND \pm TWO PLACE DECIMAL \pm THREE PLACE DECIMAL \pm		DRAWN CHECKED ENG APPR. MFG APPR. Q.A. COMMENTS:	LAB 2				
		MATERIAL	—						
NEXT ASSY		USED ON	FINISH				SIZE	DWG. NO.	REV.
		APPLICATION	DO NOT SCALE DRAWING				A		
							SCALE:1:1	WEIGHT:	SHEET 1 OF 1

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DIMENSIONS ARE IN INCHES	NAME	DATE	Elgin Community College Brighten Your Future.
TOLERANCES: FRACTIONAL ⁺	DRAWN		
ANGULAR: MACH ⁺	CHECKED		
TWO PLACE DECIMAL ±	ENG APPR.		
THREE PLACE DECIMAL ±	MFG APPR.		
INTERPRET GEOMETRIC TOLERANCING PER: MATERIAL	Q.A.		
COMMENTS:			
NEXT ASSY	USED ON	FINISH	
APPLICATION	DO NOT SCALE DRAWING		



IMT 103_2

OPTIONAL

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R.250
TYP.

QUIZ 1

QUIZ 1		DIMENSIONS ARE IN INCHES		
		TOLERANCE:	0.004	0.001
		FRACTIONAL ¹	0.004	0.001
		ANGULAR: MACH ² END ³	MACH APP.	MACH APP.
		TWO PLACE DECIMAL ⁴	MACH APP.	MACH APP.
		THREE PLACE DECIMAL ⁵	MACH APP.	MACH APP.
AND CONFIDENTIAL CONTAINED IN THIS PROPERTY OR BY NAME HEREIN, ANY PART OR AS A WHOLE OR IN PERMISSION OF MY NAME HEREIN		APPROX.	---	---
		COMMAS.		
APPLICATION	DO NOT SCALE DRAWING	SUP. DR. NO.	A	REV.
		SEARCHED	SEARCHED	SEARCHED

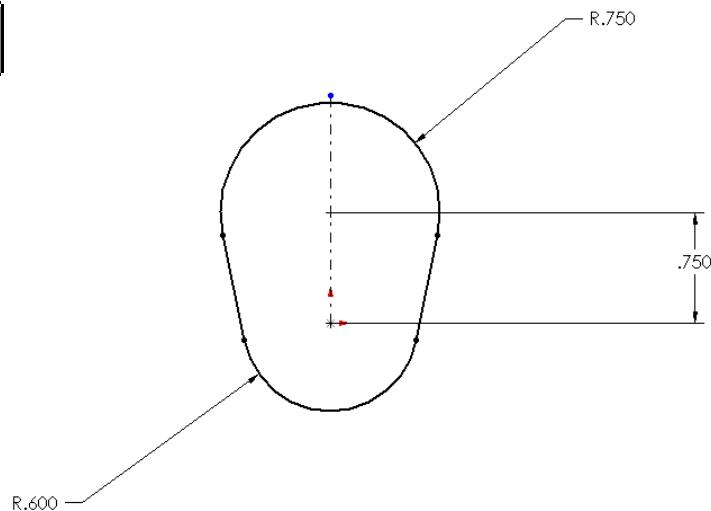
This technical drawing shows a cross-sectional view of a mechanical part. The outer dimensions are 6.000 wide and 1.000 high. A central hole has a diameter of ϕ 1.250 THRU. The top surface features four PLCS with a diameter of ϕ .500 THRU. A chamfer of .125 x 45° is present at the bottom. The part includes a base with a width of 3.000 and a height of .750, which is angled at 45°. There are also two small holes with a diameter of ϕ .250 located on the top surface.

1. MODEL THE PART
 2. Position in an Isometric View
 3. Save as PDF
 4. Submit in Assignments section of D2L

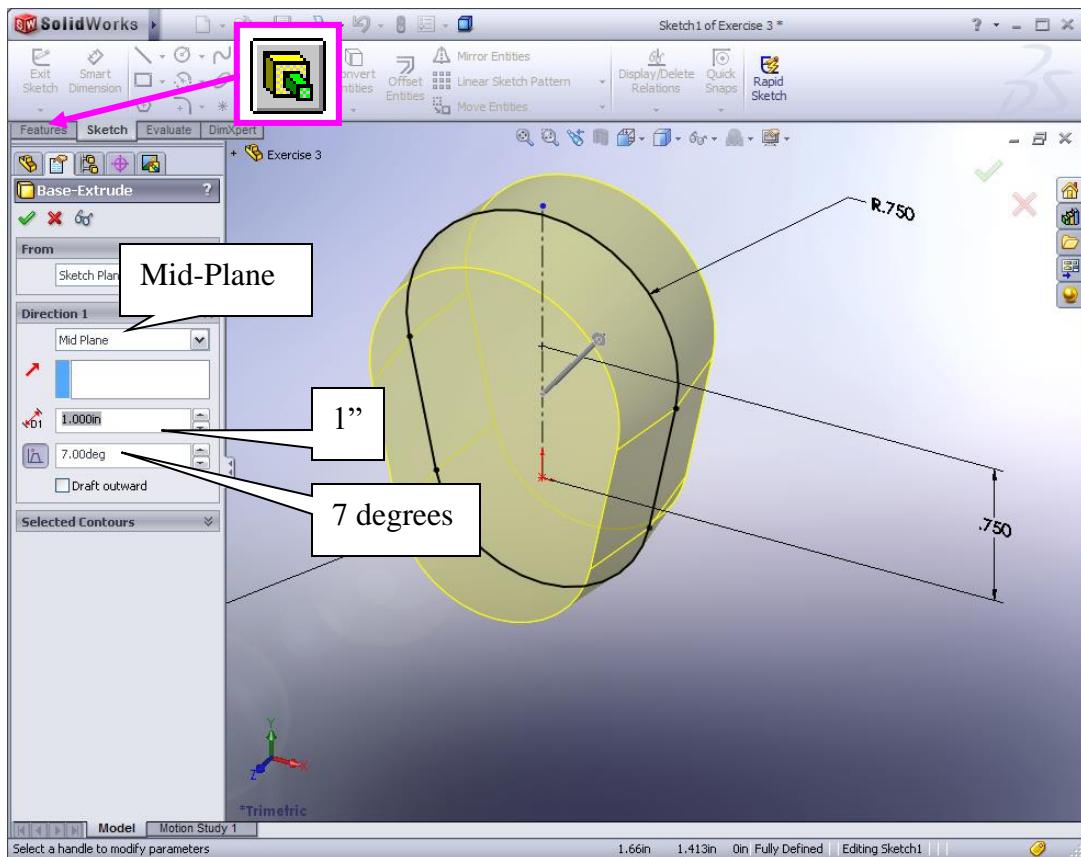
EXERCISE 3

Secondary Feature Modeling

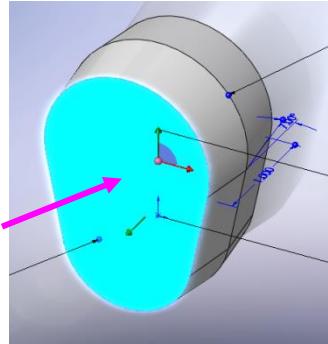
1. Sketch the geometry as show below on the “Front” plane.



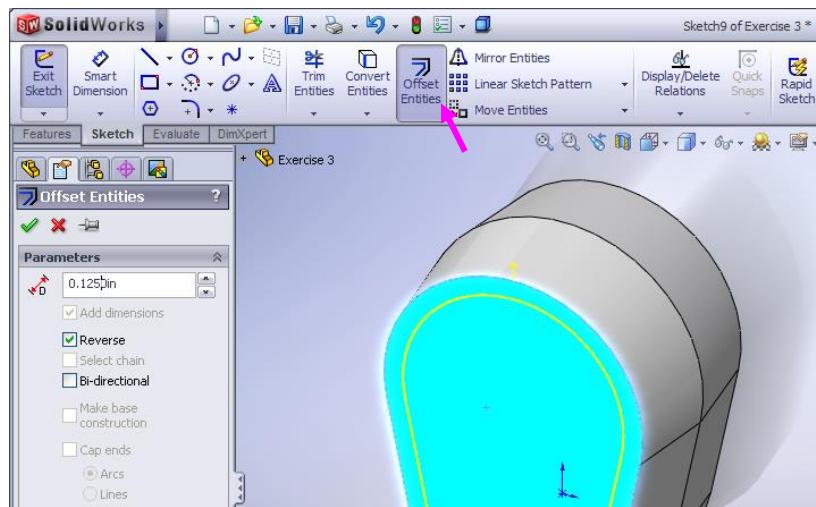
2. Extrude Boss/Base. Select Mid-Plane, 1" and add 7° of draft.



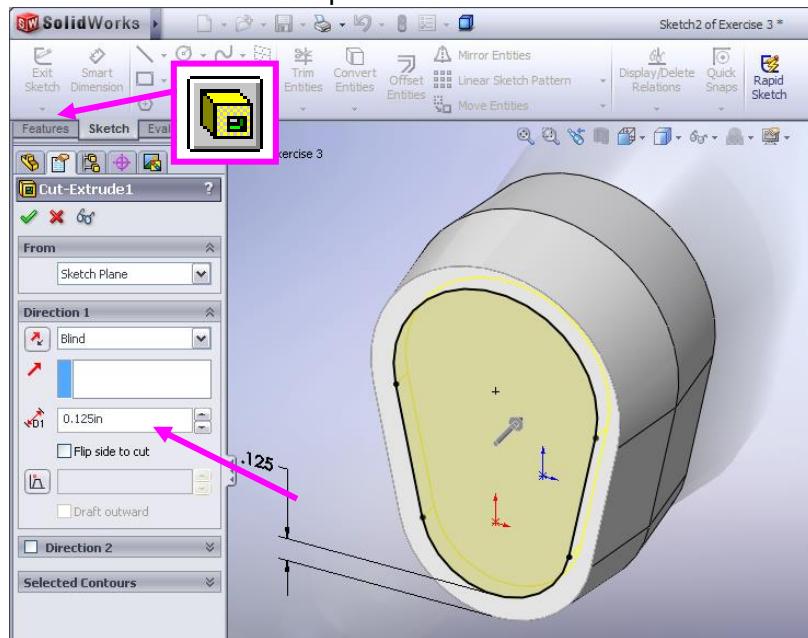
3. Select the front face of the model and start a sketch on it.



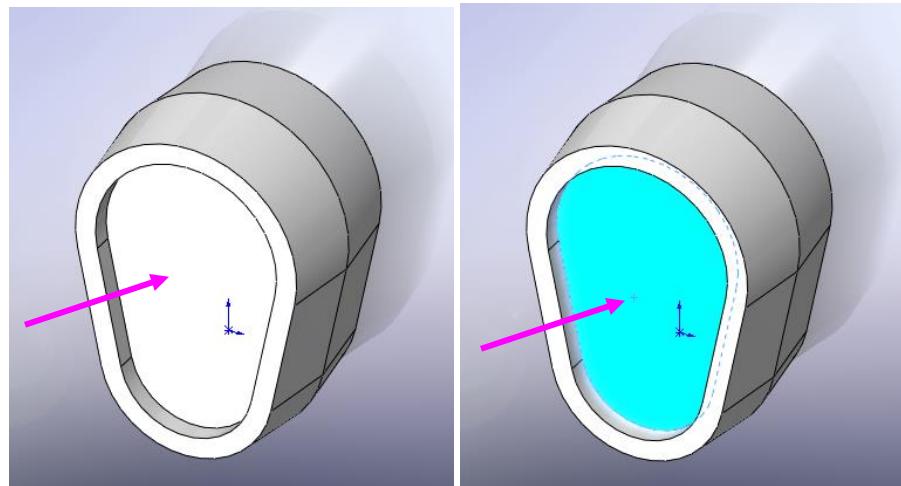
4. Use the “Offset Entities” icon to offset sketch geometry .125” from the outside edges. *NOTE: The face or edges must be selected to see dynamic offsetting.*



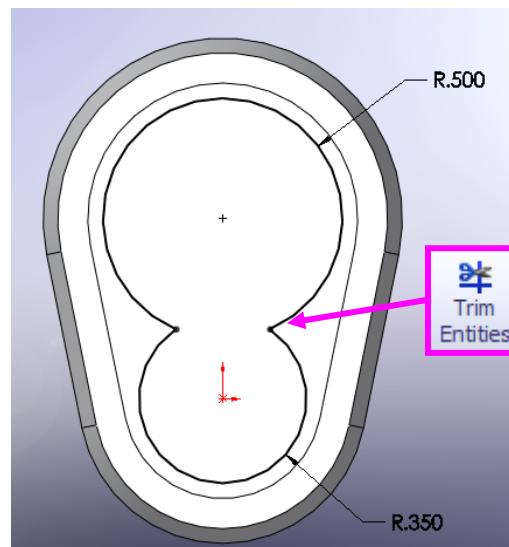
5. Cut Extrude at .125” deep.



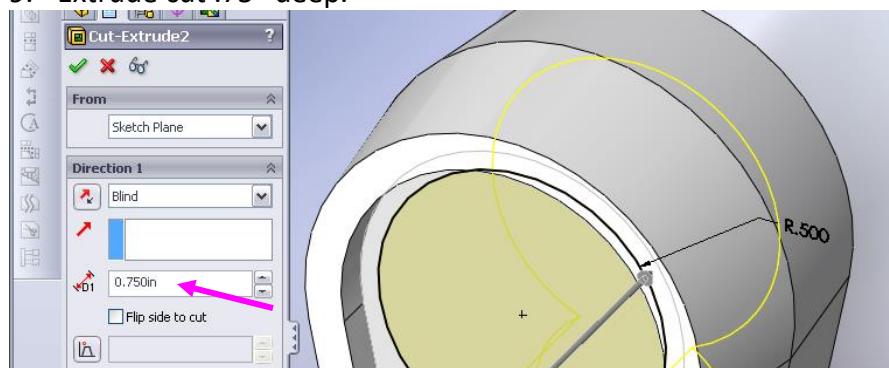
6. Select the base of the pocket and start a sketch on it.



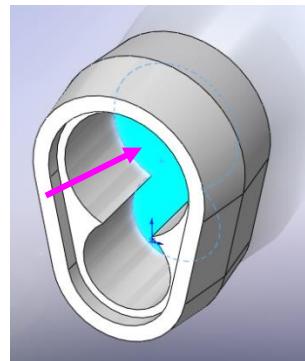
7. Go "normal to" and sketch the following. Draw 2 circles on the center points of the outside arcs (Note: You can wake the center points up by gliding the tip of the pointer over the edge of the arc before sketching) Use the trim tool (trim to closest check option) to remove intersections.



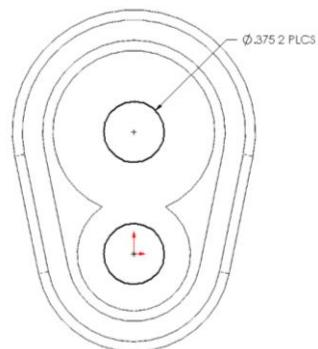
9. Extrude cut .75" deep.



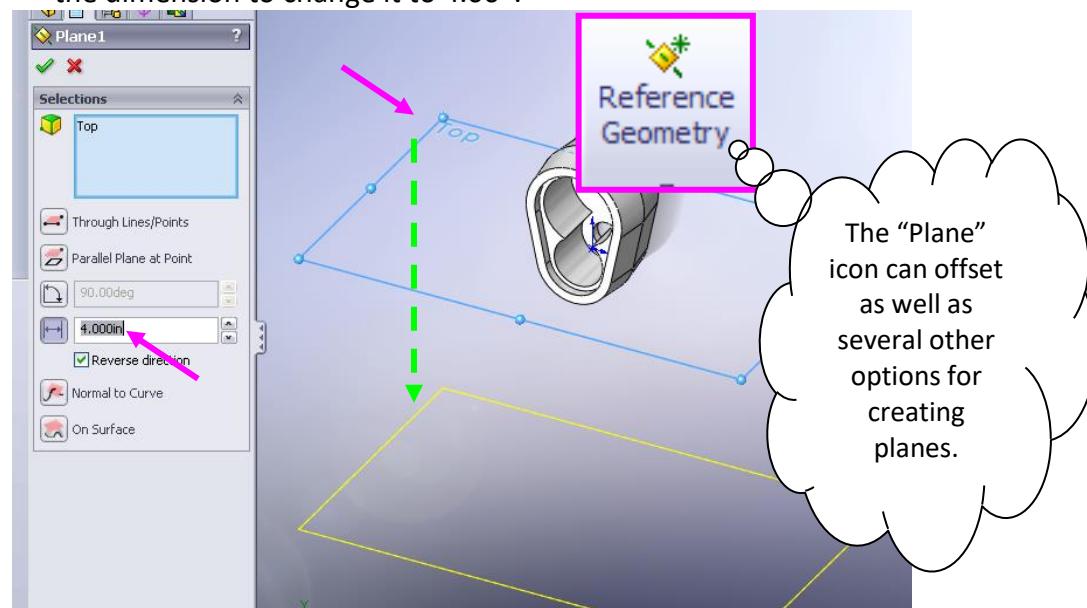
10. Select the base of the new pocket and start a sketch on it.



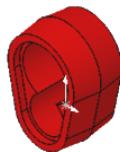
11. Sketch the following, and extrude cut “Through-all”.



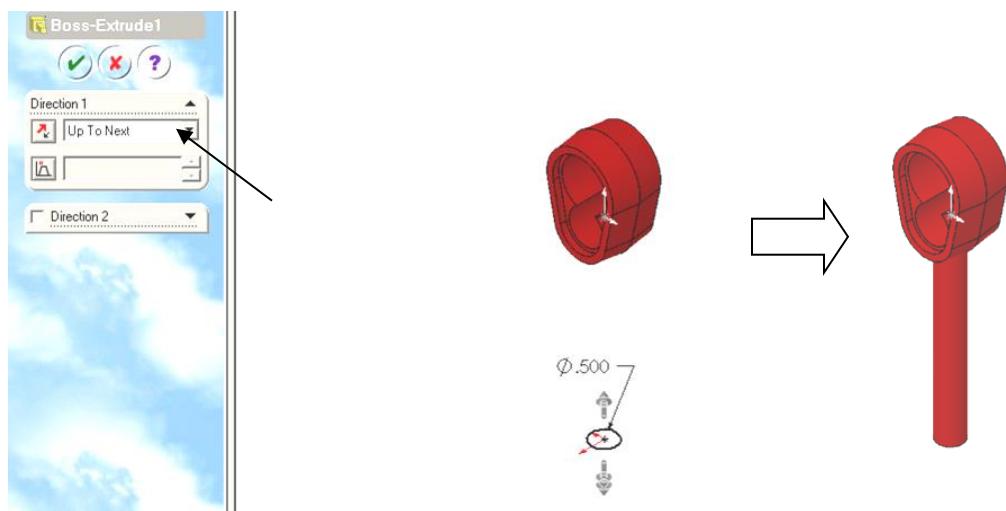
12. Hold the “CTRL” key and drag a plane below the “Top” plane. Release control after you have released the mouse button. Double click on the new plane to see dimension it has been dragged. Then double click on the dimension to change it to 4.00”.



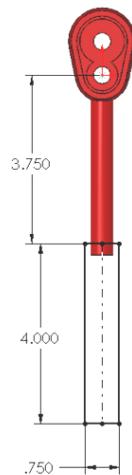
13. Start a sketch on “Plane 1” and draw a .5” dia. circle centered on the origin.



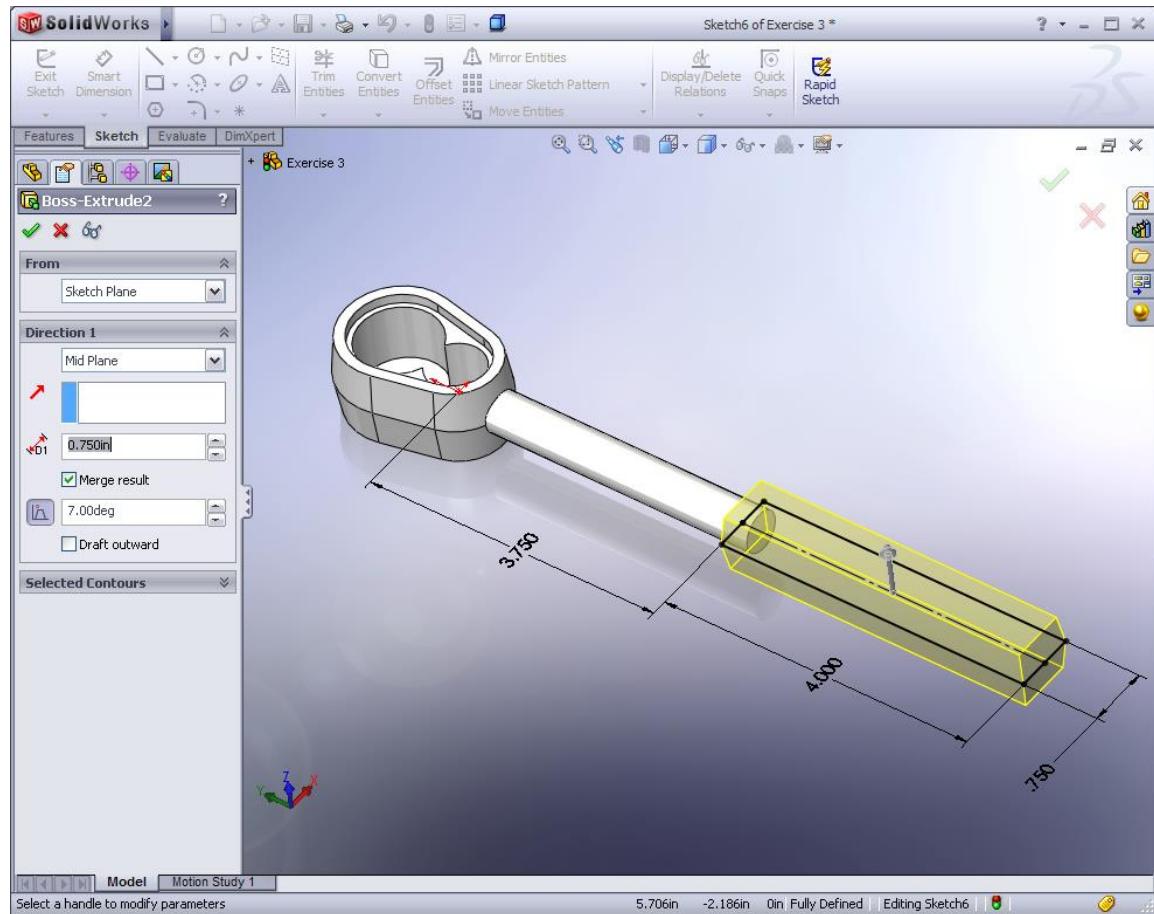
14. Extrude boss and use the “Up to next” option.



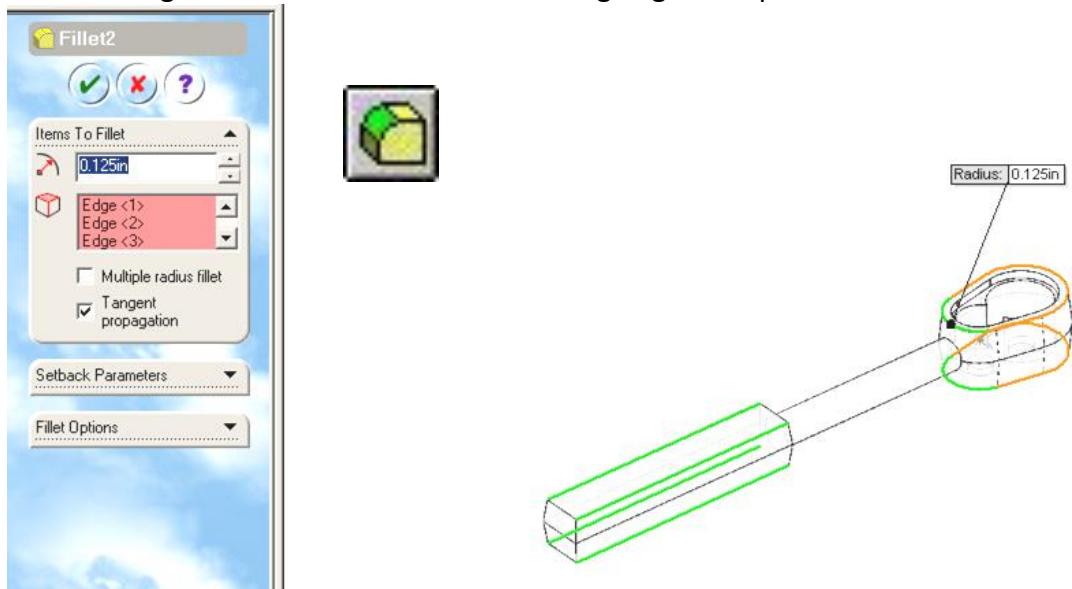
15. Start a sketch on the “Front” plane and sketch the following rectangle.



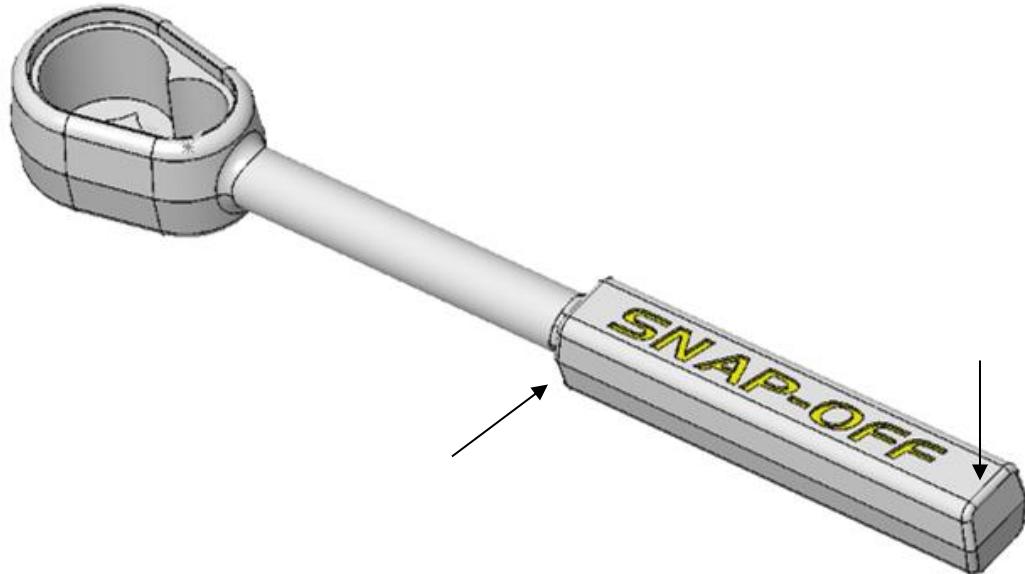
16. Extrude boss using the mid-plane option and .750 thick with 7° draft.



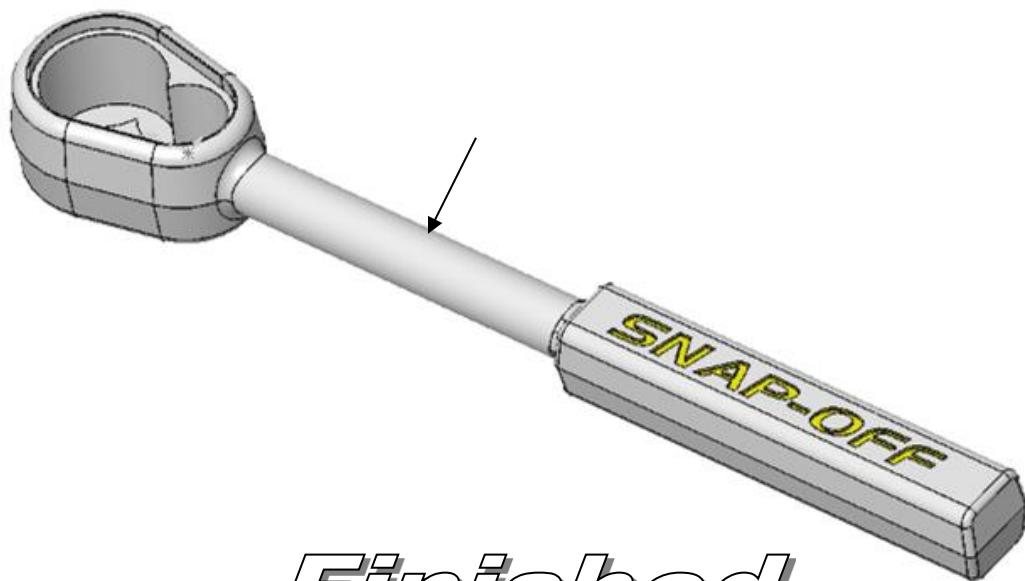
17. Using the fillet tool select the following edges and put a .125" radius on them.



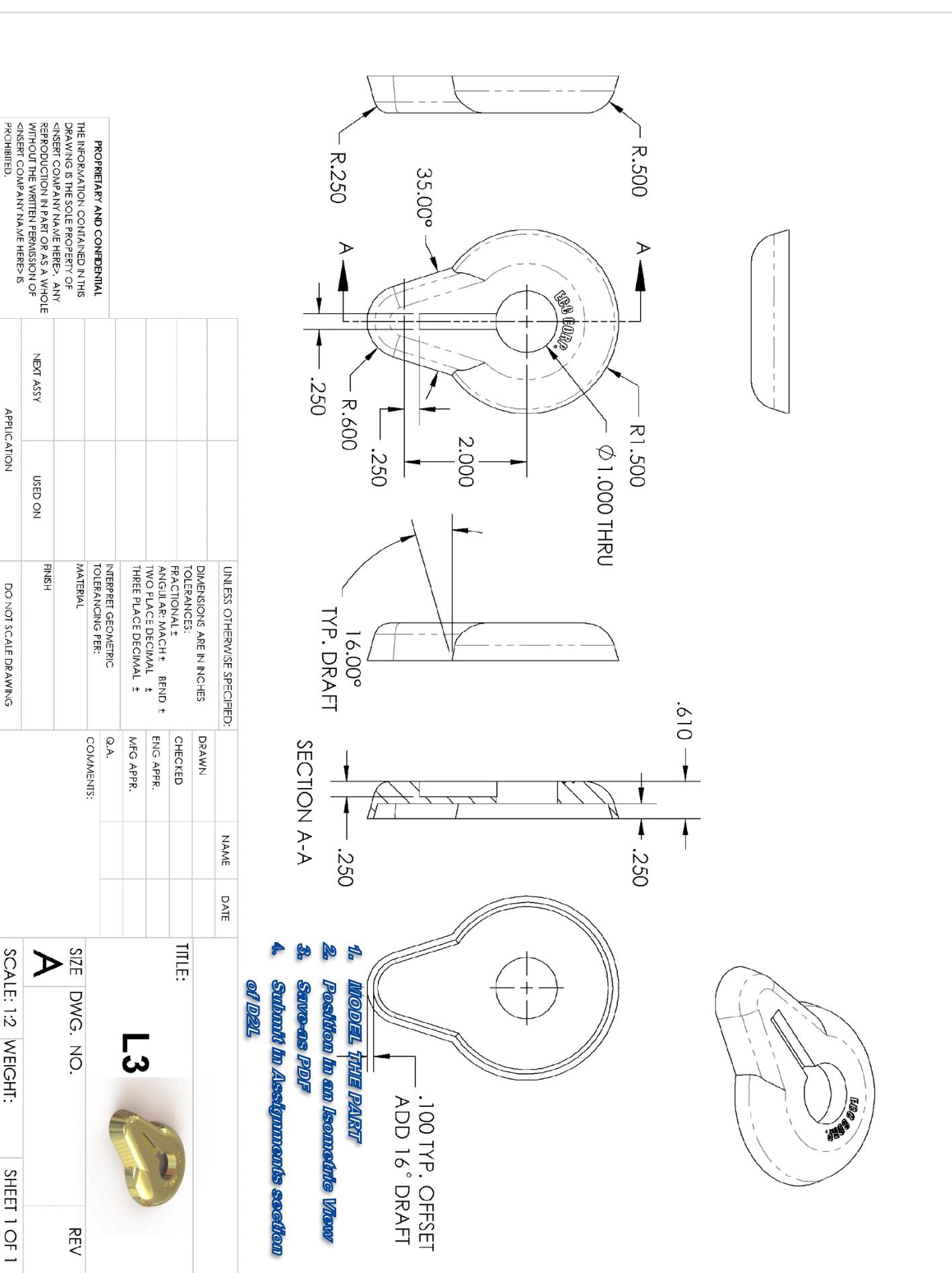
18. Add additional fillets of .06" on the following edges.



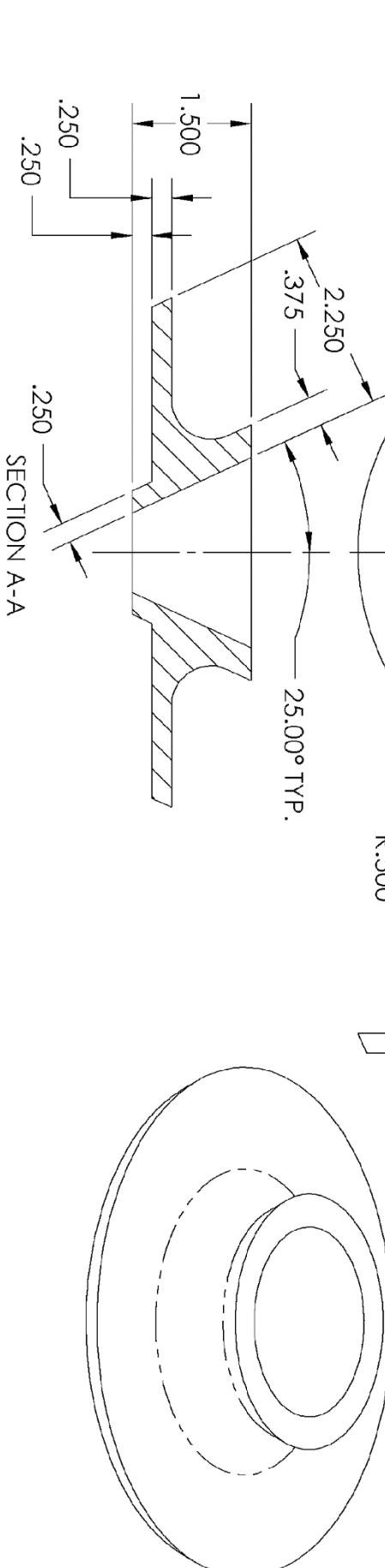
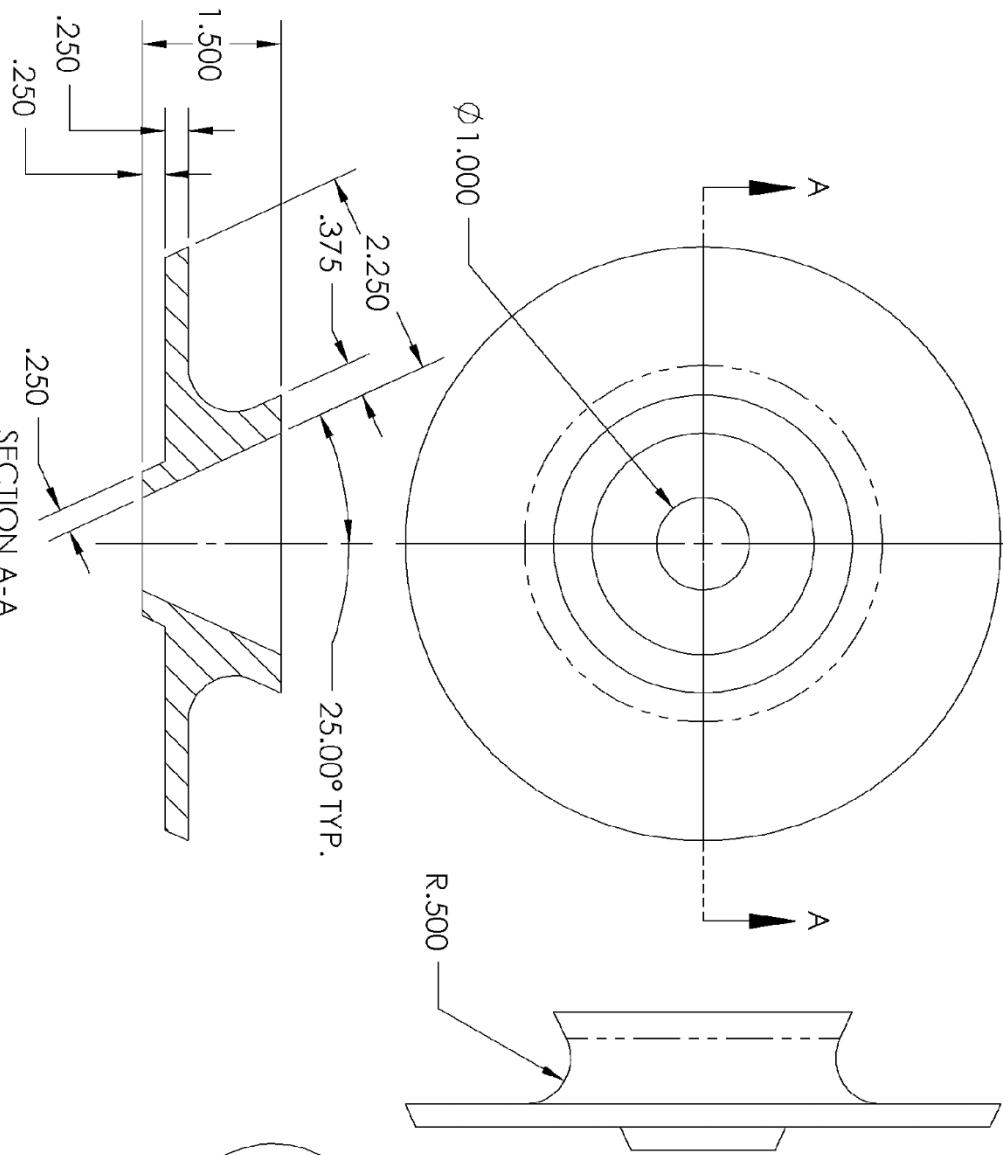
19. Short-Cut fillet is applied by selecting the entire face of the transition. SolidWorks then fillets any and all edges that face comes in contact with.



Finished



1. MODEL THE PART
2. Position in an Isometric View
3. Save-as PDF
4. Submit in Assignments section of D2L



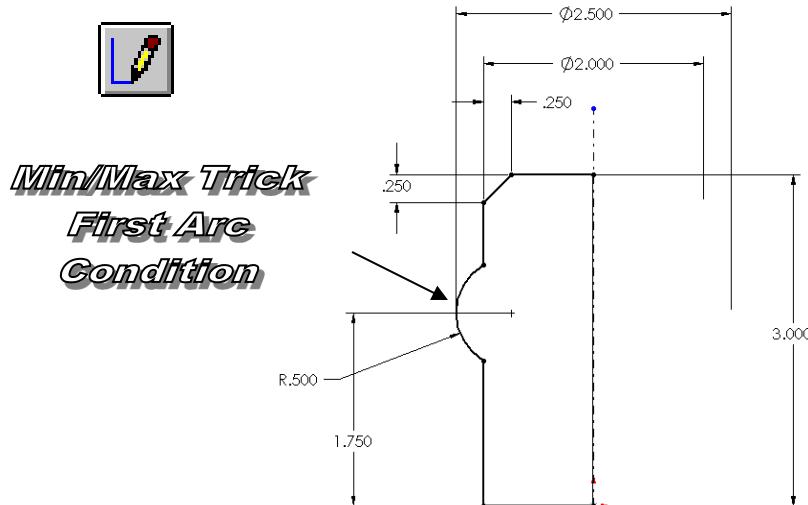
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DIMENSIONS ARE IN INCHES	DRAWN	NAME
TOLERANCES: FRACTIONAL \pm ANGULAR: MACH \pm BEND \pm TWO PLACE DECIMAL \pm THREE PLACE DECIMAL \pm	CHECKED	DATE
INTERPRET GEOMETRIC TOLERANCING PER: MATERIAL	ENG APPR.	TITLE:
Q.A.	MFG APPR.	
COMMENTS:		
NEXT ASSY	USED ON	
APPLICATION	FINISH	DO NOT SCALE DRAWING
SCALE: 1/2 WEIGHT: SHEET 1 OF 1		

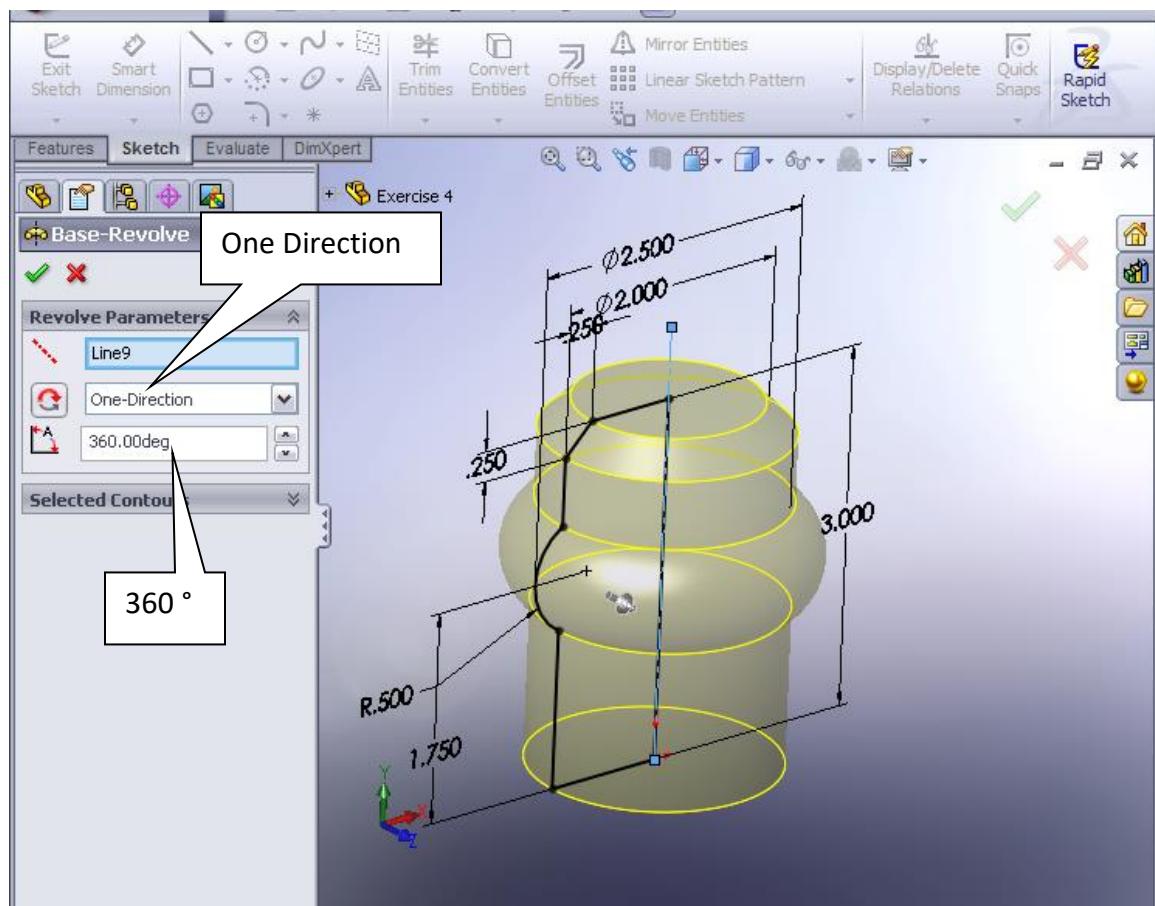
EXERCISE 4

Secondary Feature Modeling

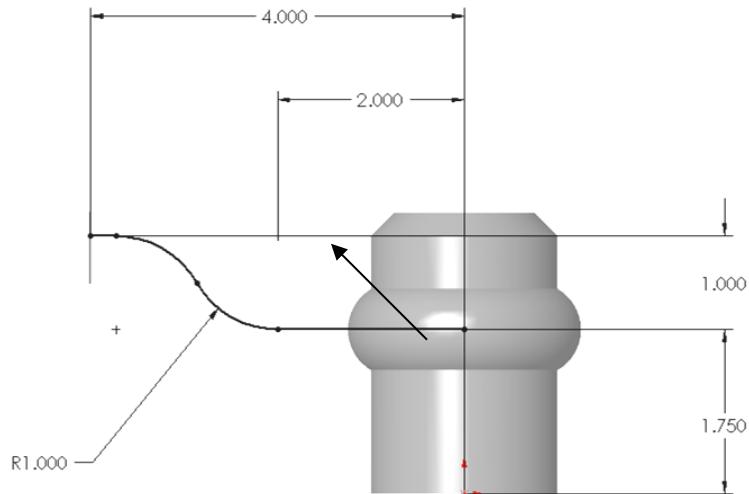
- Sketch the geometry as show below on the "Front" plane.



- Base-Revolve.

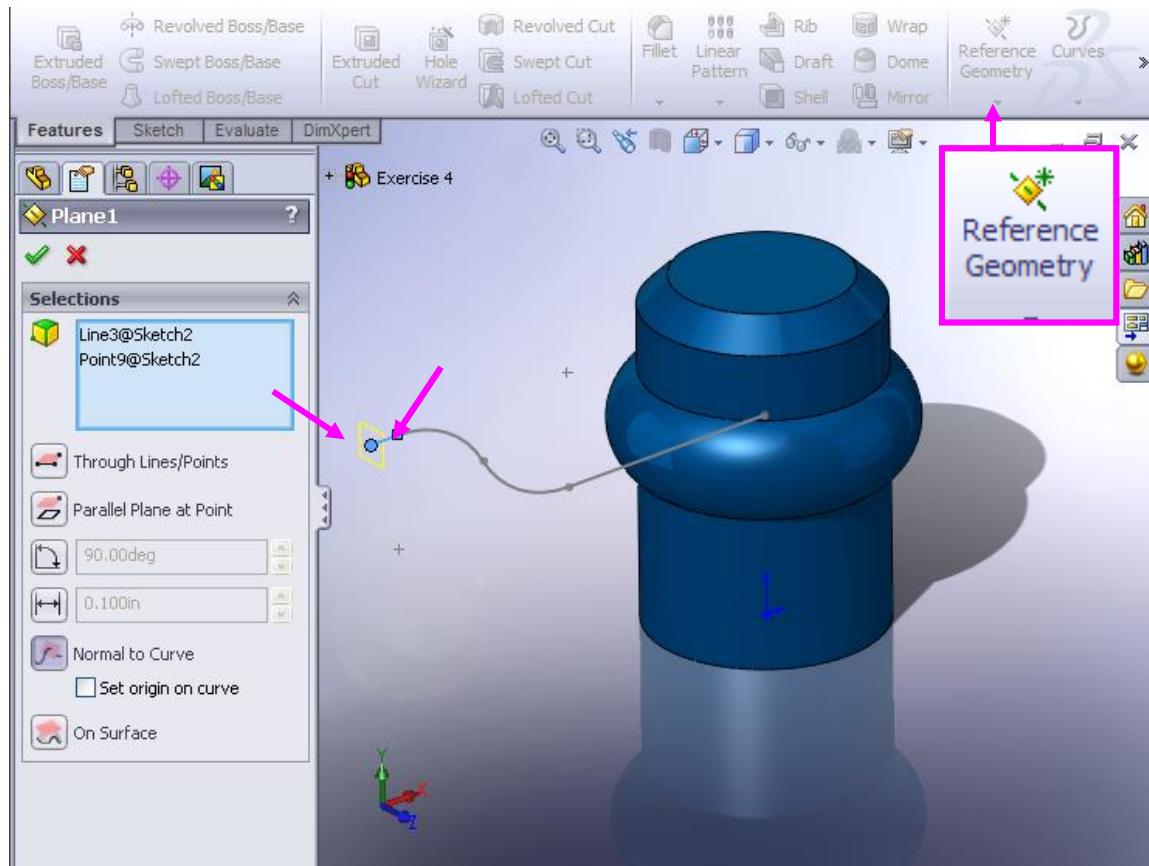


8. Select the “Front” plane and start a sketch on it. Rebuild after completion.

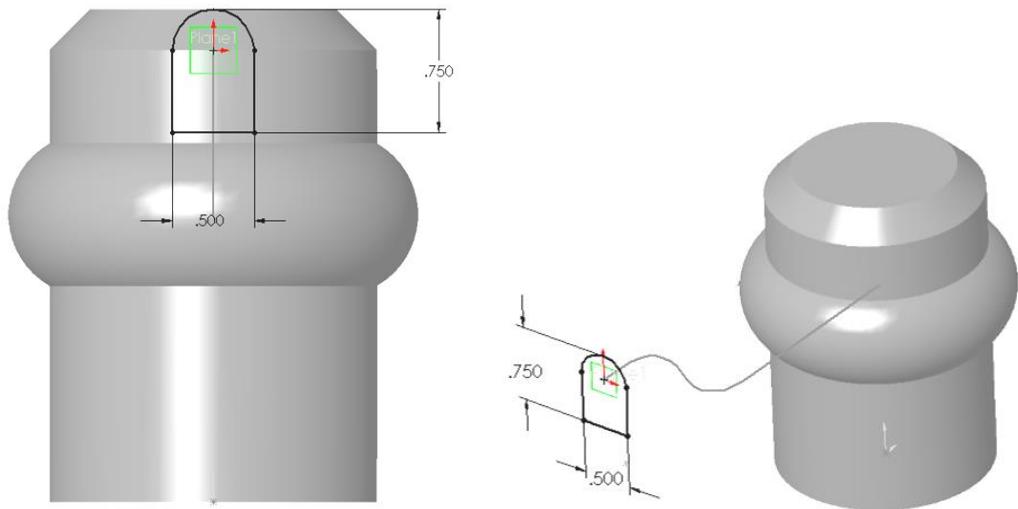


9. Create a plane using the “Plane” tool located under “Reference Geometry”.

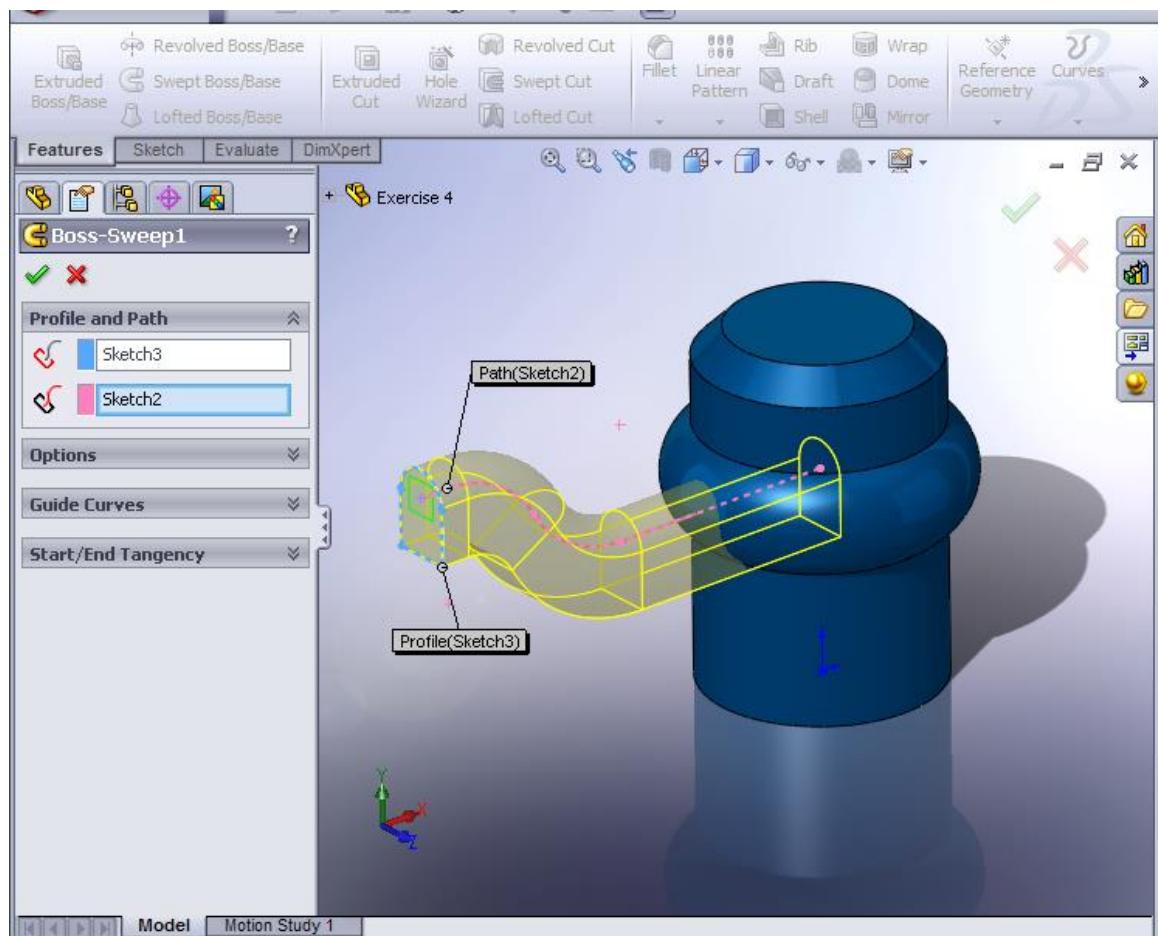
Note: The 2 ingredients for creating a plane perpendicular to a curve are the Curve and a Point (in this case the end point of the curve). Rebuild when completed.



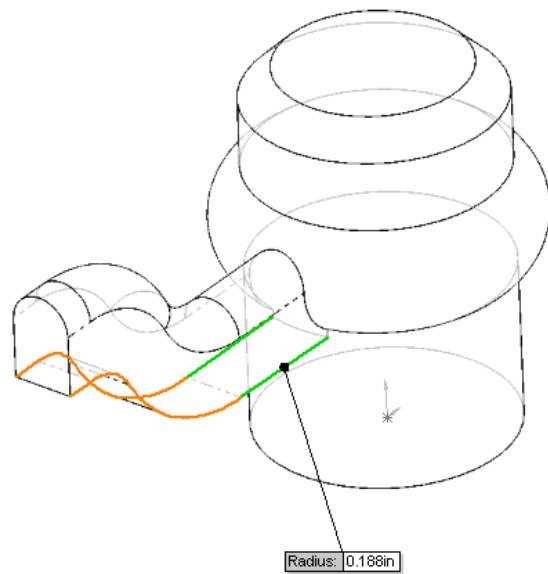
10. Select the new plane and sketch the following on it. Rebuild.



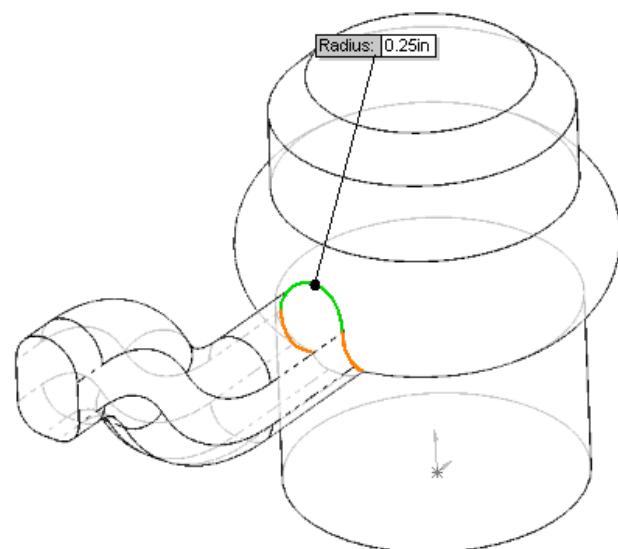
11. Select the Sweep Icon. Then select the Path and Profile.



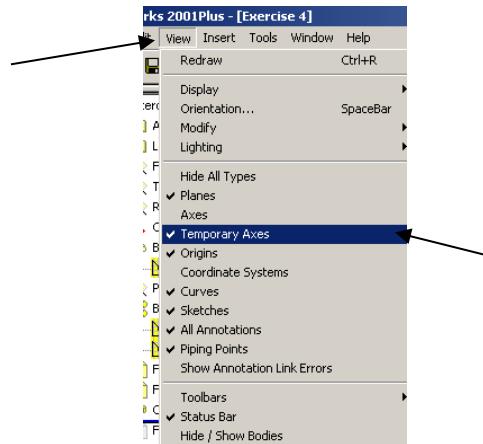
12. Add .188" Fillets on the bottom edges of the spoke.



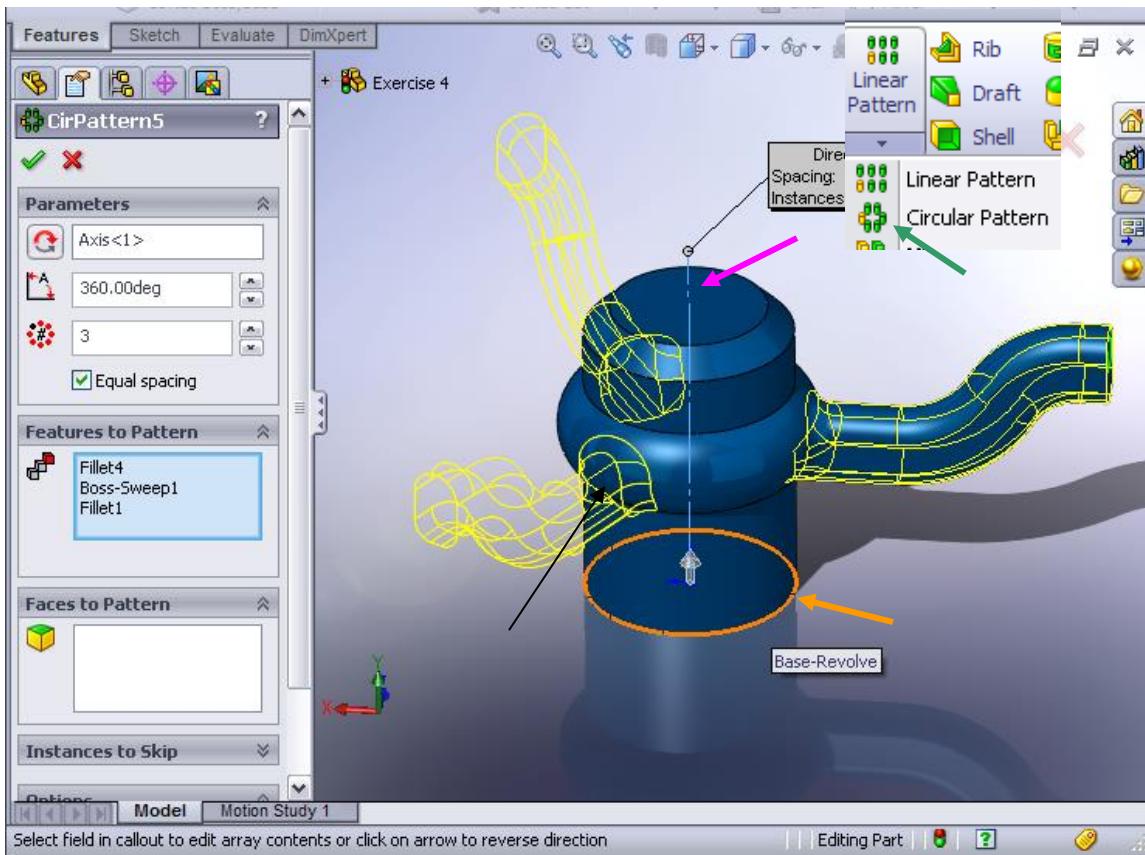
13. Add .25" fillets to the intersection of the Spoke and Center.



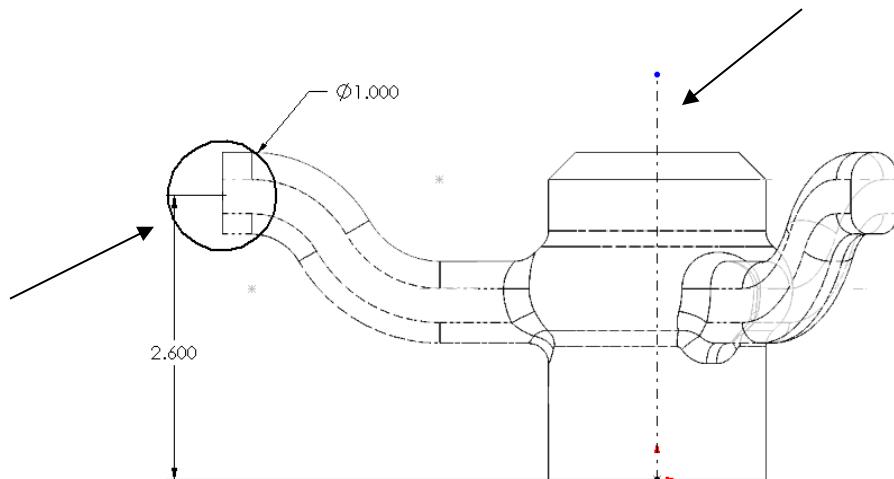
14. **Creating the Circular Pattern/Array:** Go to View/and Check on Temporary Axis. Note: This is not necessary in SolidWorks 2008 as new functionality allows you to select the edge of the cylinder as the pattern axis.



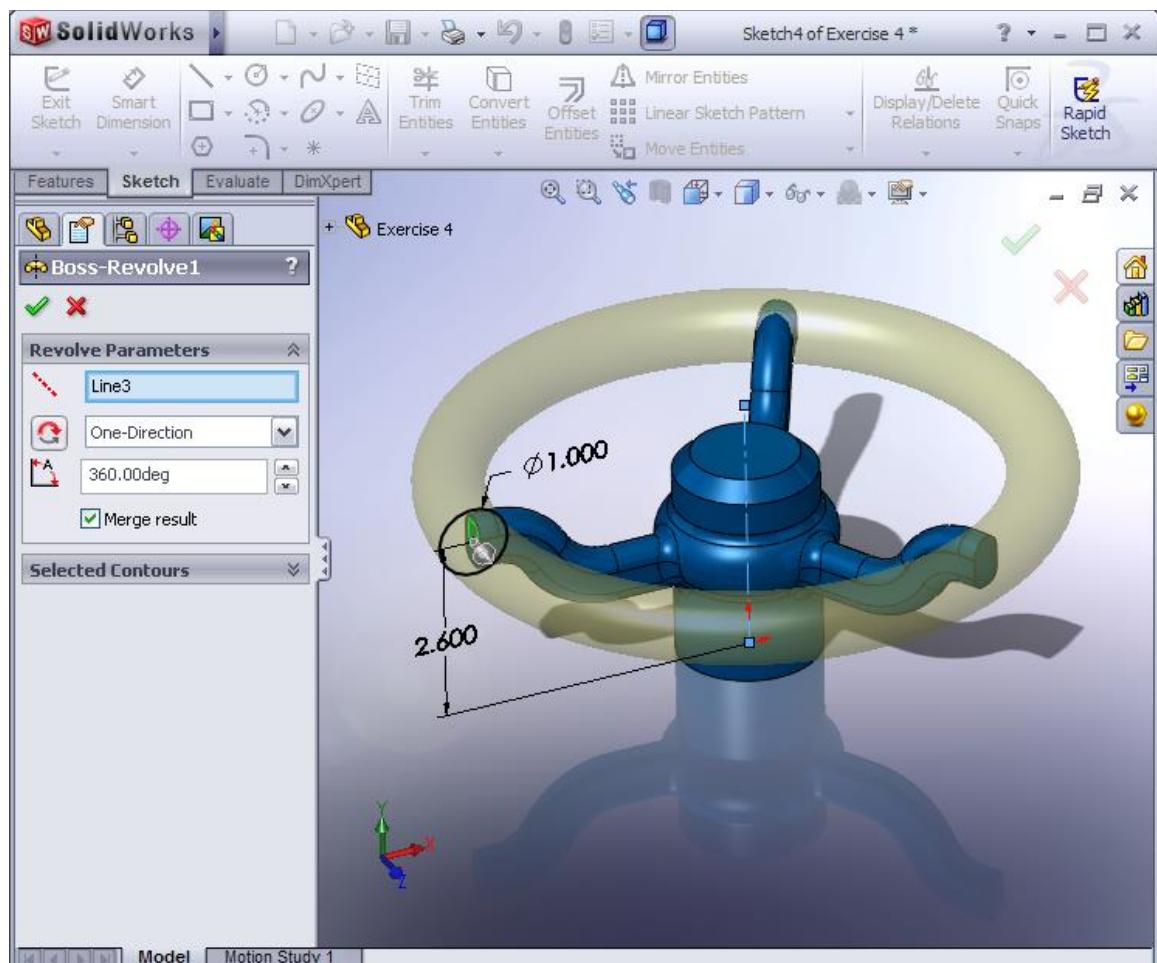
15. Select the ***Circular Pattern Icon***. Then select the ***Axis*** or the ***cylinder edge*** located at the center and the Spoke. Enter 3 for the number of spokes. Note: be sure to select the fillets as well, or they will not show up on the instances.



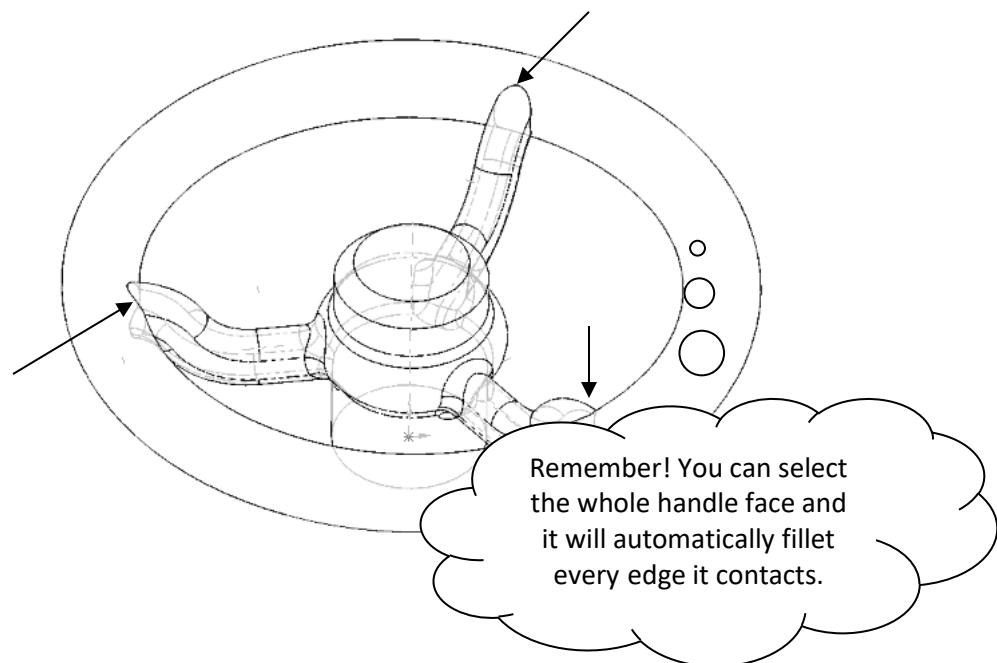
11. Select the “Front” plane and start a sketch on it. Draw the following and don’t forget the Centerline.



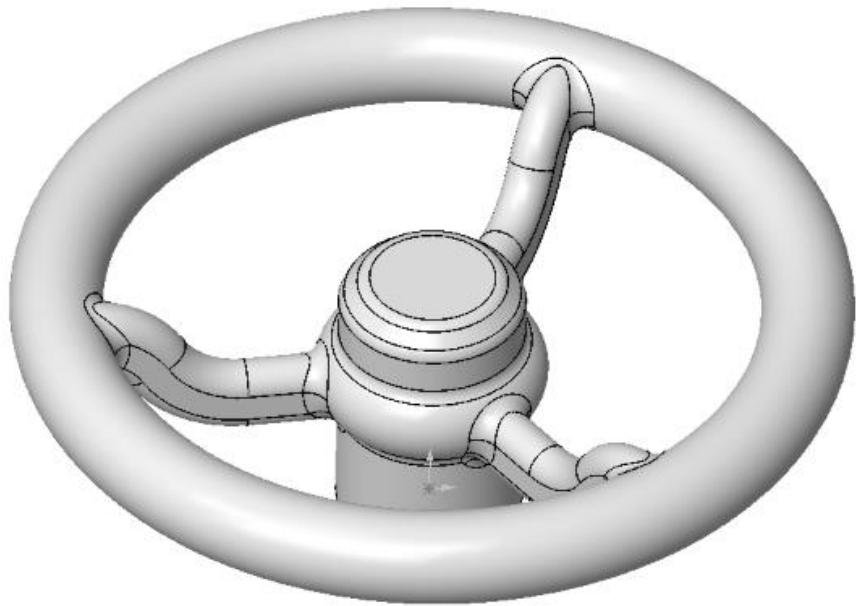
16. Select the Boss-Revolve Icon, and revolve 360° “One-Direction”.



17. Add .188" fillets around the intersections of the handle and spokes".

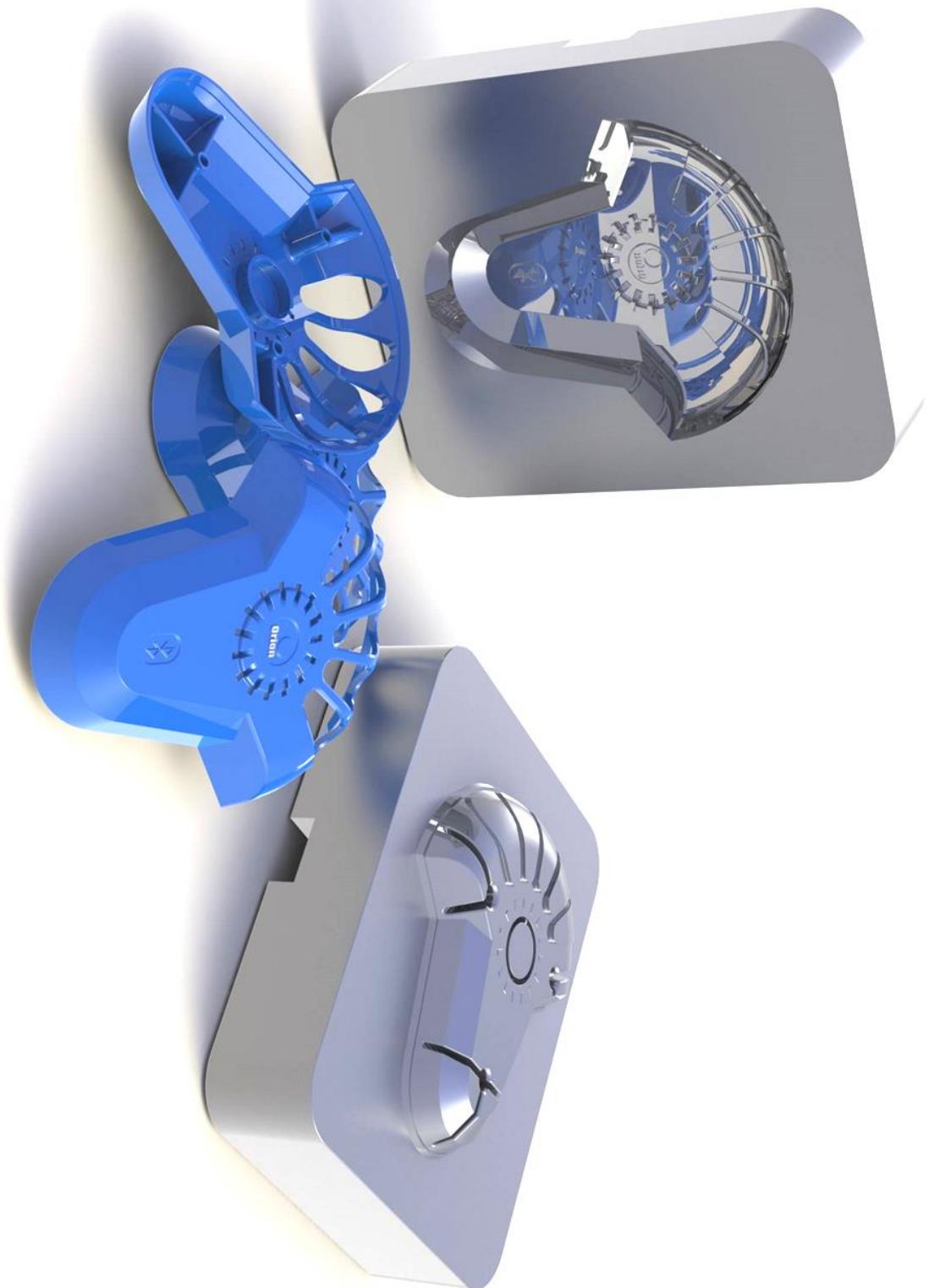


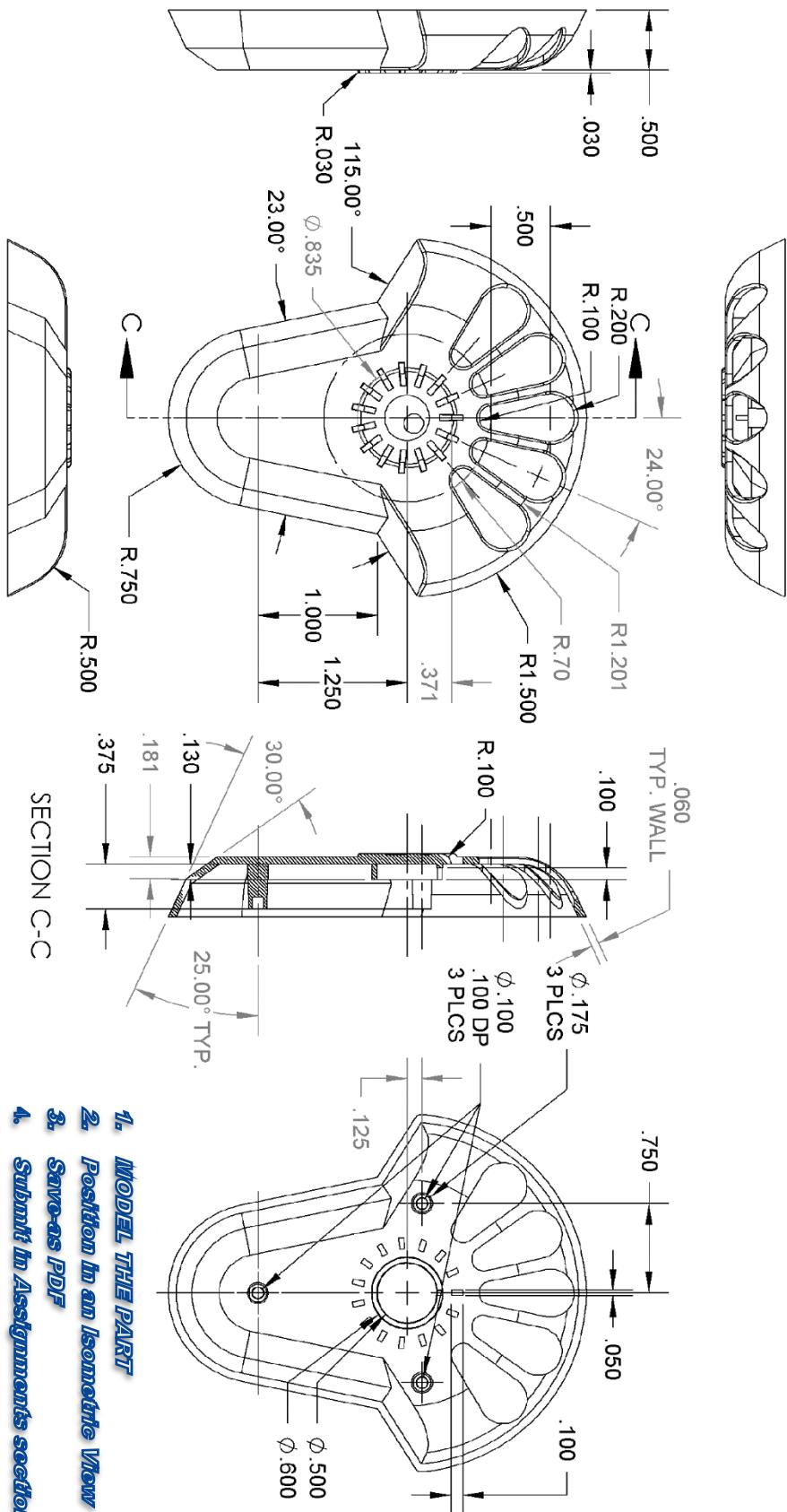
18. You are finished.



- 19.

Finished

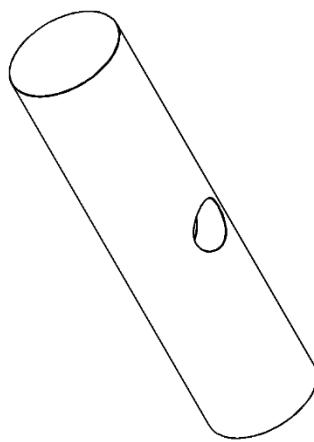
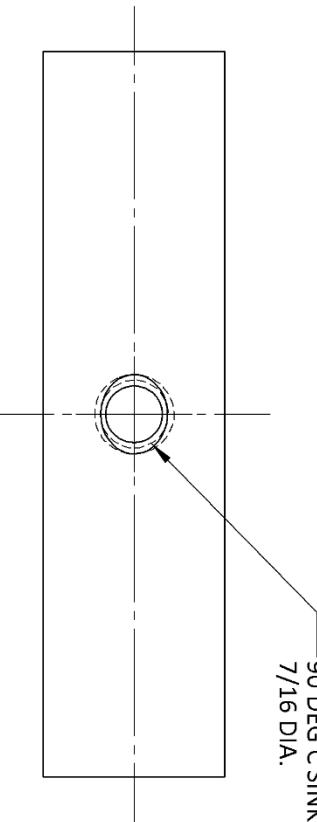
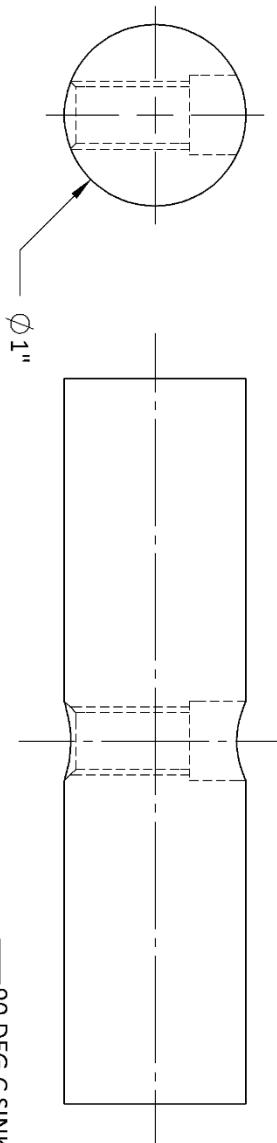
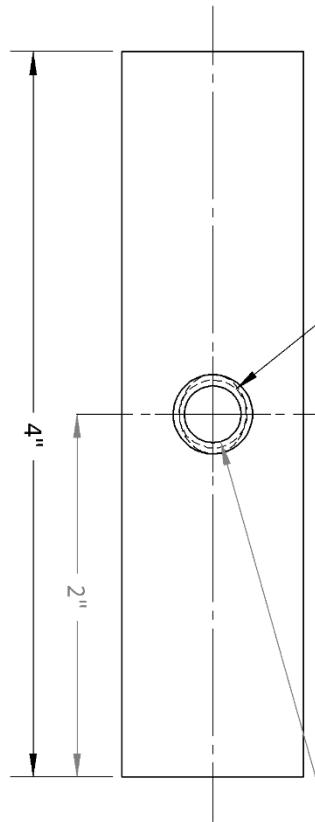




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<p style="text-align: center;">DRAWN CHECKED ENG APPR. MFG APPR. Q.A.</p> <p style="text-align: center;">INTERPRET GEOMETRIC TOLERANCING PER: MATERIAL</p> <p style="text-align: center;">COMMENTS:</p>		
<p style="text-align: center;">UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL[±] ANGULAR; MACH⁺ BEND [±] TWO PLACE DECIMAL [±] THREE PLACE DECIMAL [±]</p>		
NEXT ASSY	USED ON	FINISH
APPLICATION	DO NOT SCALE DRAWING	
SCALE: 3:4		WEIGHT:
A		DWG. NO. L4
		REV
SHEET 1 OF 1		

C BORE 3/8 DIA. x 5/16 DP

ϕ .31 THRU ALL
3/8-16 UNC - 2B THRU ALL

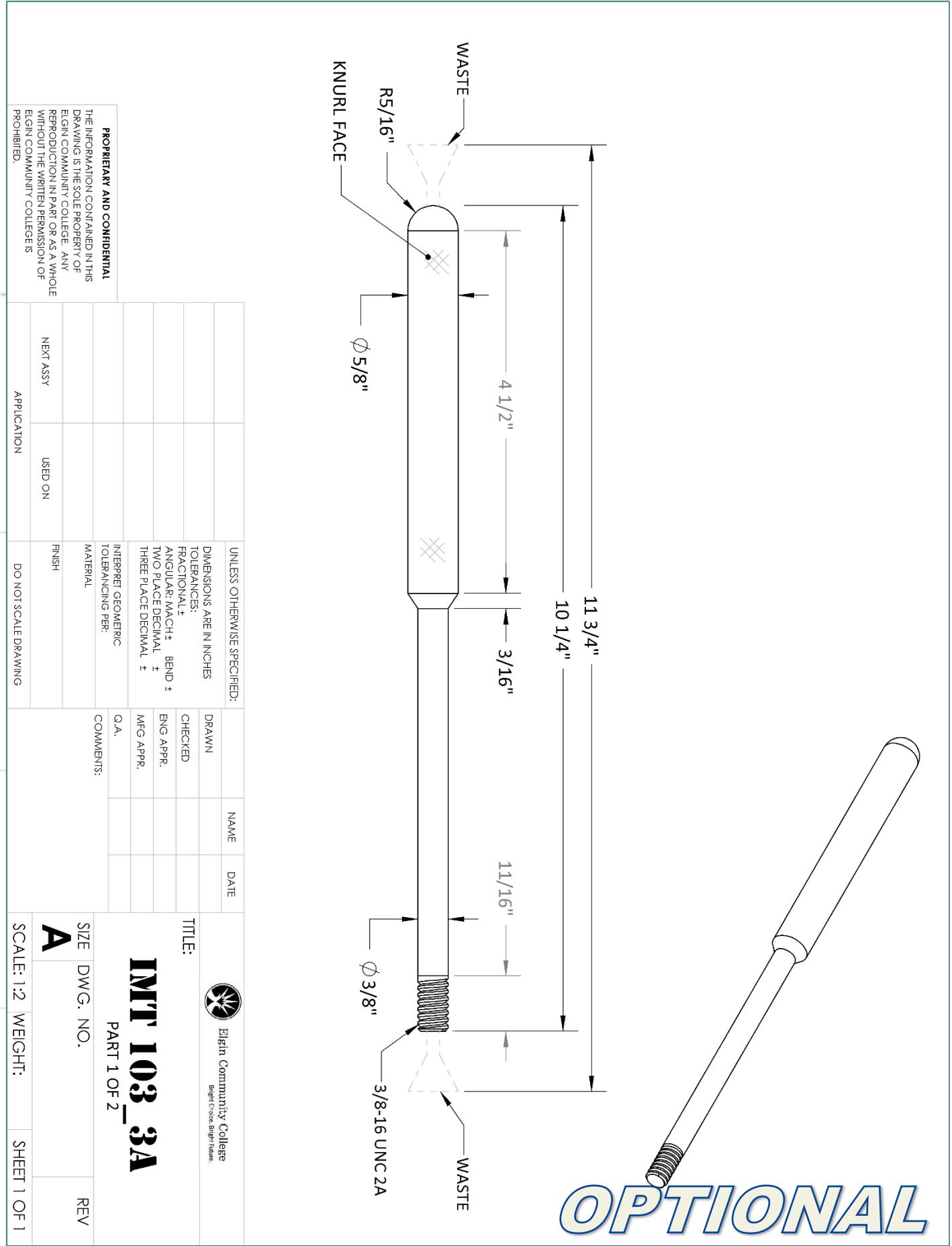


OPTIONAL

INT 103_3B

PART 2 OF 2

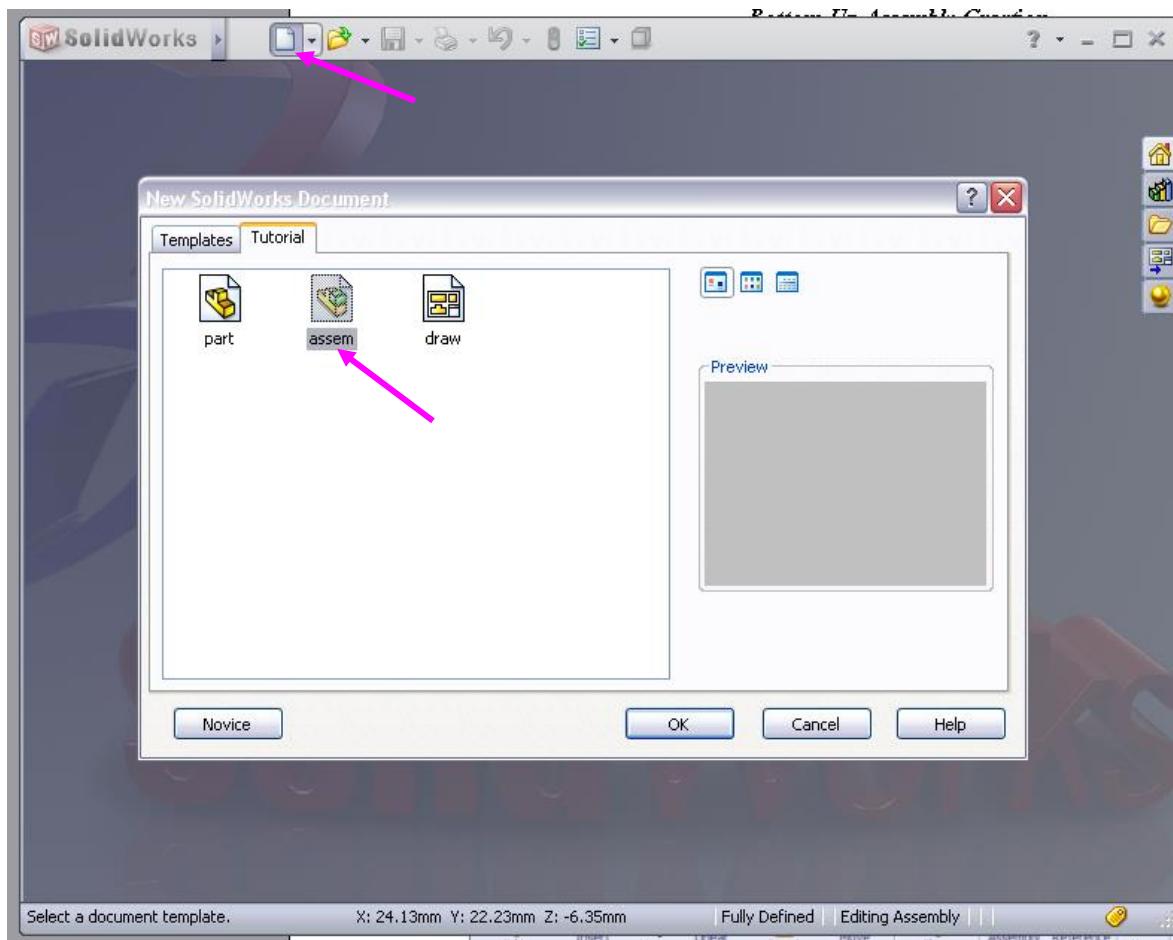
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NEXT ASSY	USED ON
FINISH	DO NOT SCALE DRAWING.
APPLICATION	
UNLESS OTHERWISE SPECIFIED:	
DIMENSIONS ARE IN INCHES	
TOLERANCES:	
FRACTIONAL \pm	
ANGULAR: MACH \pm	
TWO PLACE DECIMAL \pm	BEND \pm
THREE PLACE DECIMAL \pm	
INTERPRET GEOMETRIC	
TOLERANCING PER:	
MATERIAL	COMMENTS:
DRAWN	
CHECKED	
ENG APPR.	
MFG APPR.	
Q.A.	
TITLE:	
Elgin Community College <small>Big-Block Big-Time</small>	
A	
SIZE	DWG. NO.
A	
SCALE: 1:1	WEIGHT:
	SHEET 1 OF 1



EXERCISE 5

Bottom-Up Assembly Creation

1. Go to “File/New and select the Assembly Template”.



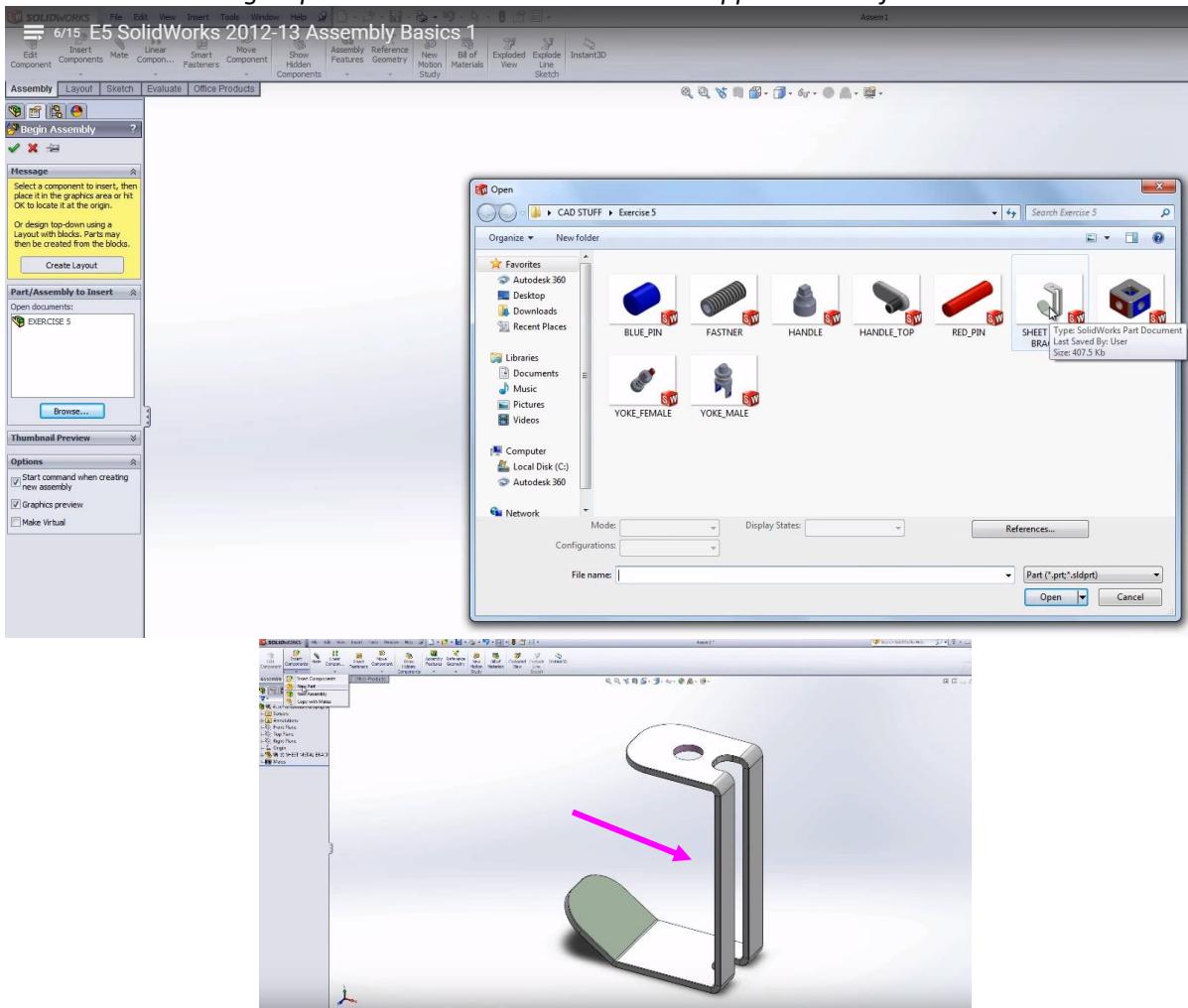
2. Assemblies Toolbar.



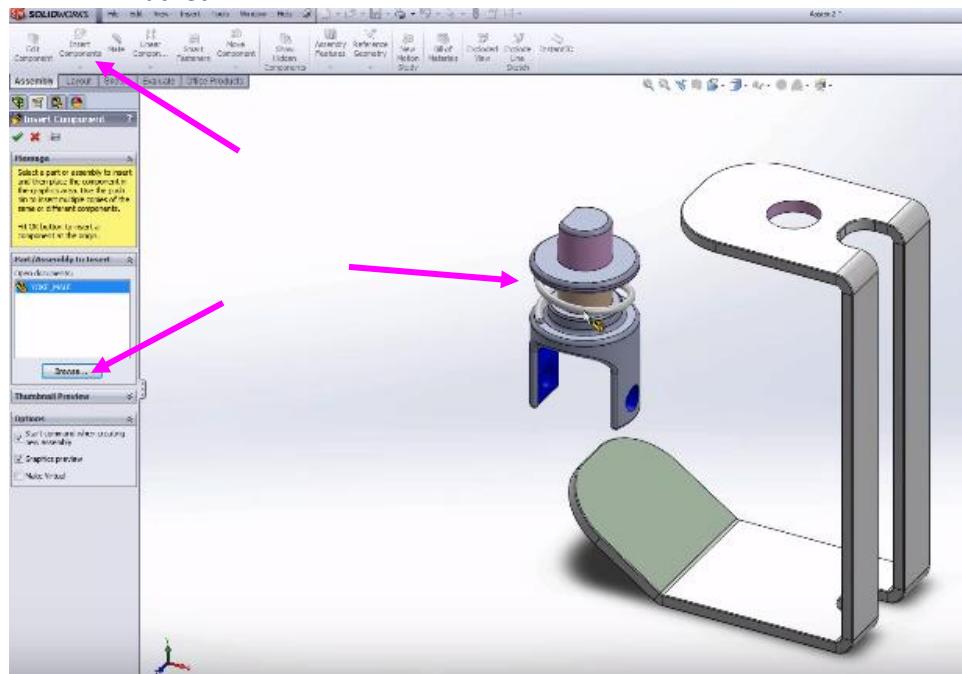
3. To insert a part into the assembly go to Insert Components.



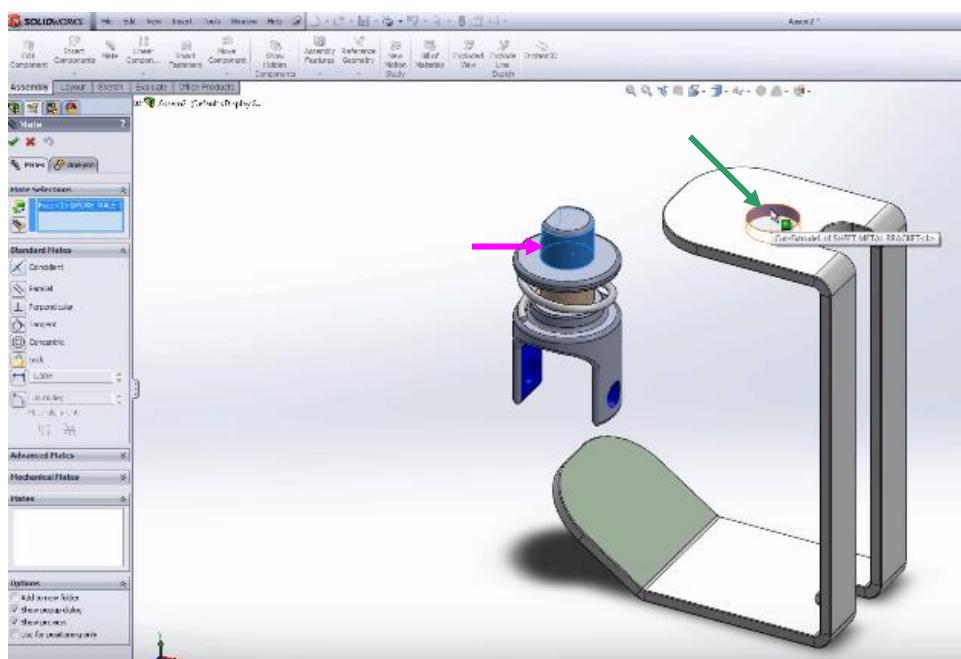
4. Select “**Browse**” then search for the “Sheet Metal Bracket.sldprt” in the Exercise 5 folder provided to you. Then move the pointer to the Origin and click the LMB to insert. *NOTE: This will drop the origin of the part coincident to the assemblies’ origin matching all planes. Notice the Part name now appears in the feature tree.*



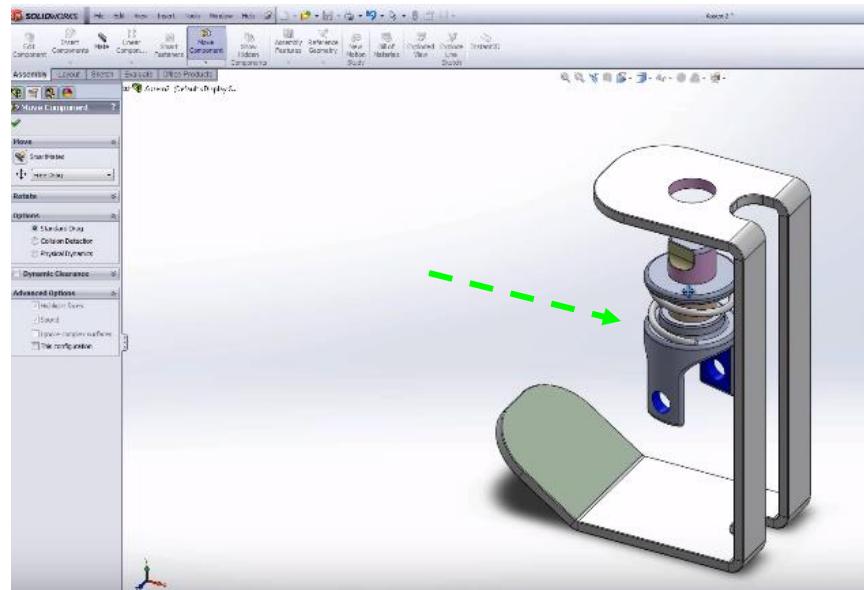
5. Go thru the insert steps again to bring in the "Yoke Male", drop it to the left of the "Bracket".



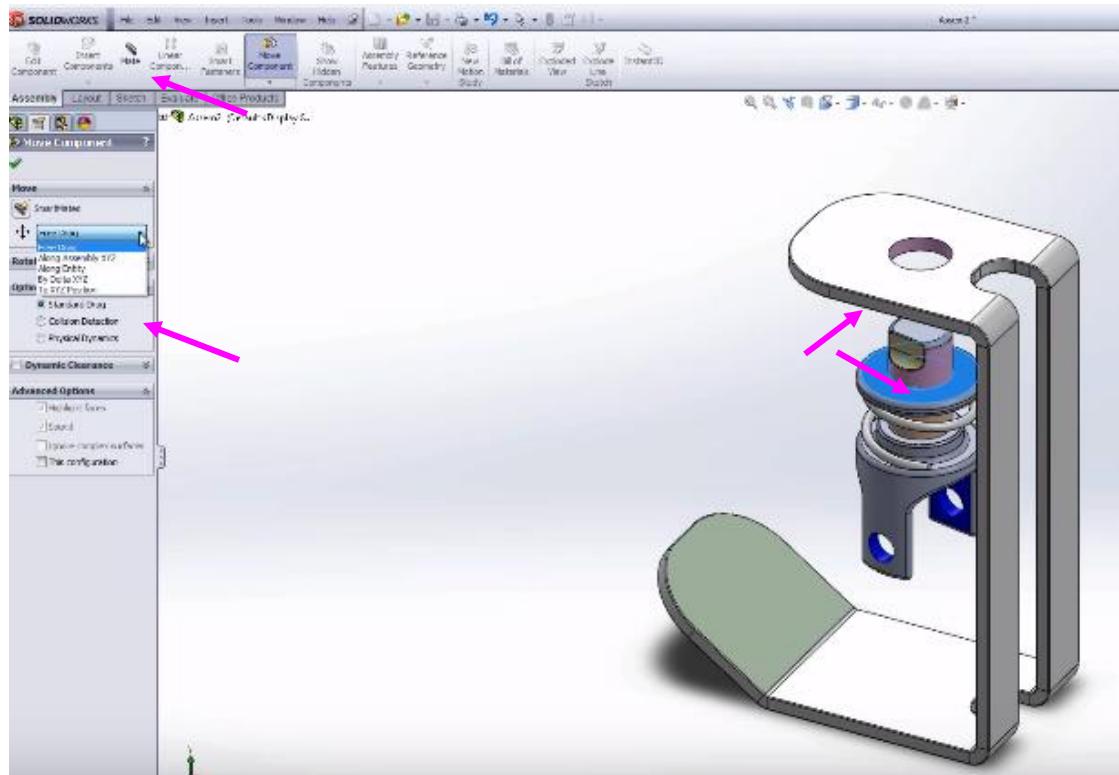
6. Select the "mate" icon and then select the side face of the boss sticking out at the top of the Yoke. Then select the inside face of the hole on the bracket.



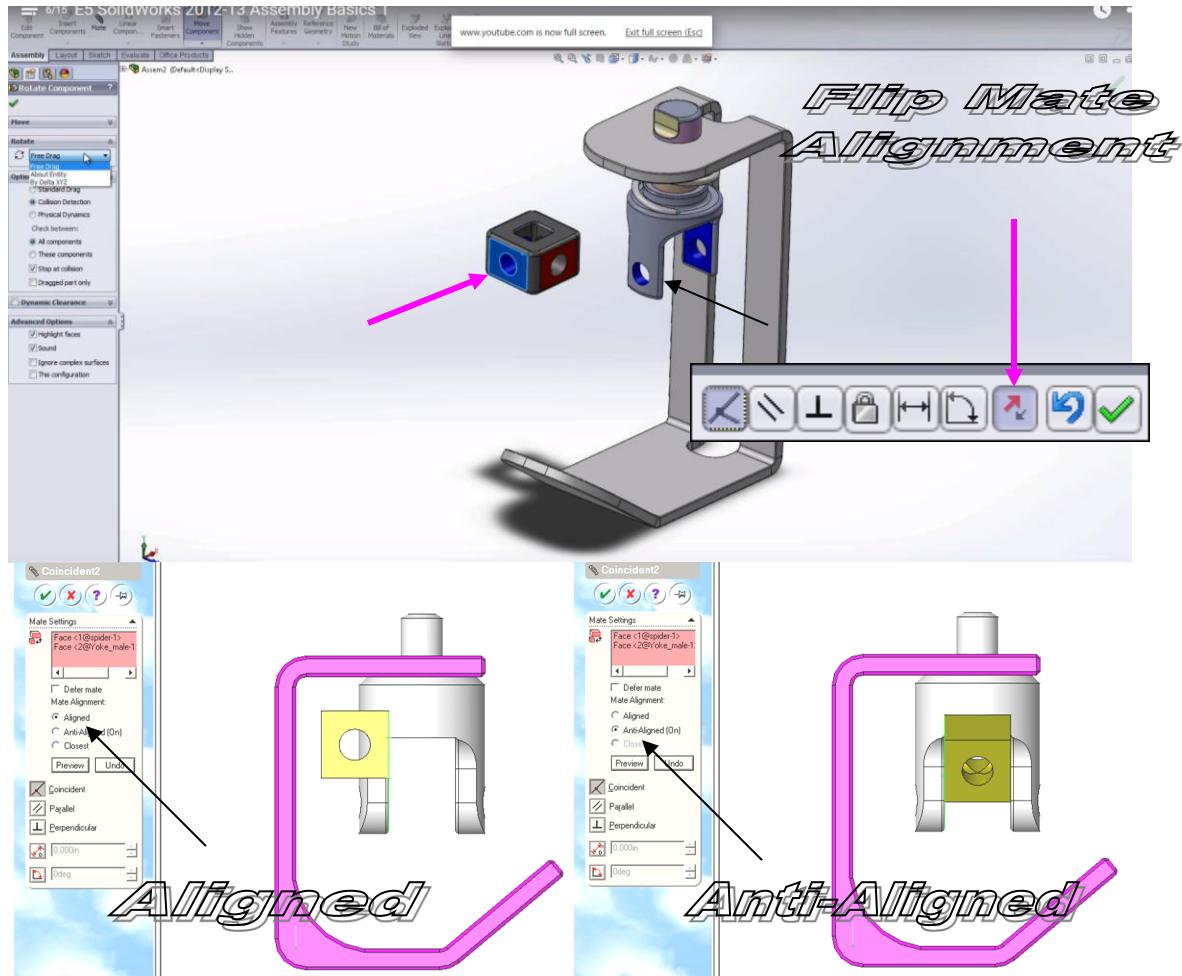
Select the concentric option and apply. The part should now move into place.



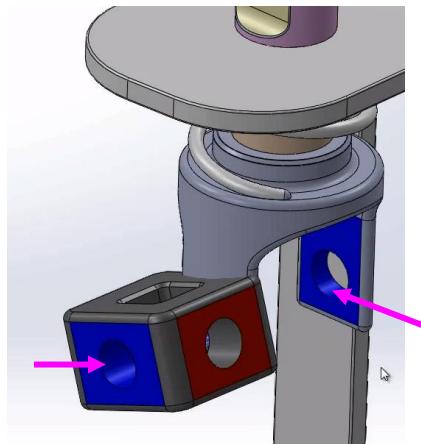
7. Select the mate icon again and attach the top face of the Yoke with the underside face of the bracket. *Note: To select thru a body RMB click over the surface and select the "Select Other" option. LMB click to toggle faces RMB to accept face.*



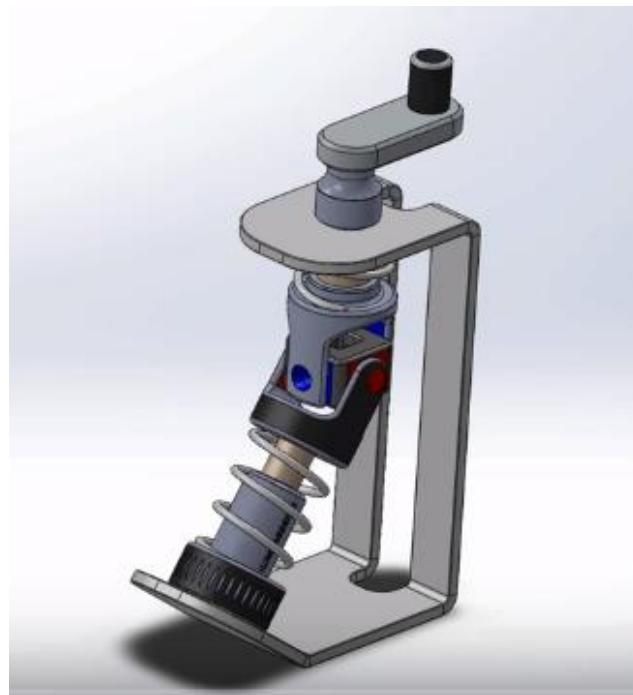
8. Insert the “Spider” part next and mate it between the legs on the yoke, you may need to select ***flip mate alignment*** in order to rotate the part 180 degrees. Note: *The aligned /anti-aligned option can reverse the direction to the model face.*



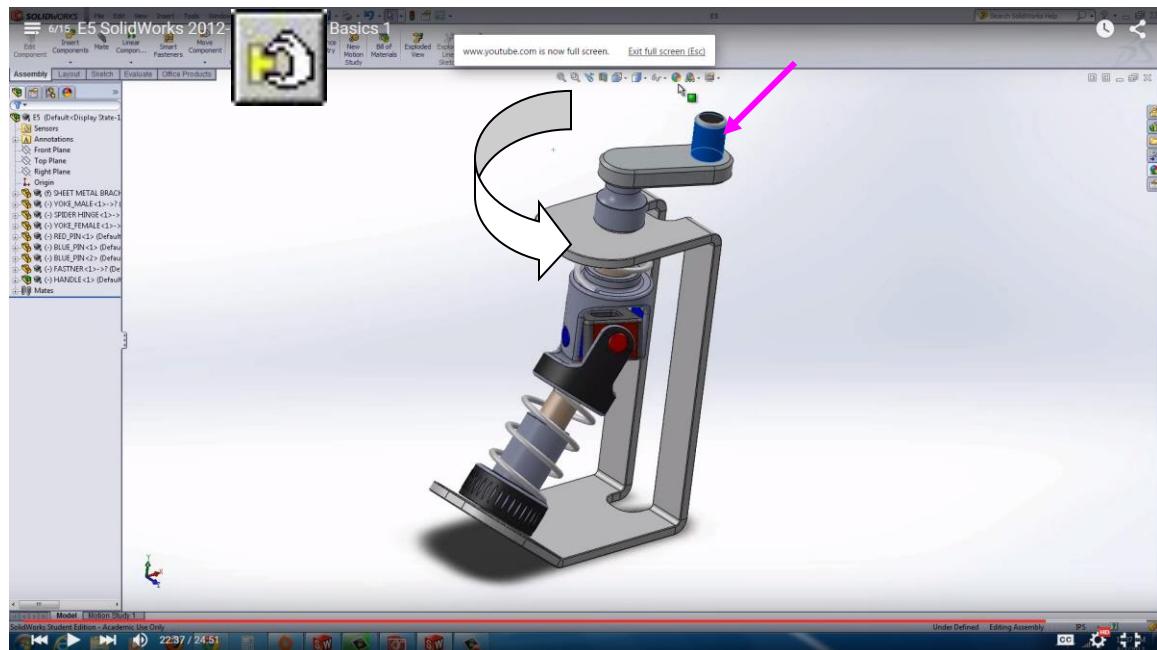
9. Use a concentric mate to align the center hole with the holes on the yoke.



12. Attach the remainder of the components.

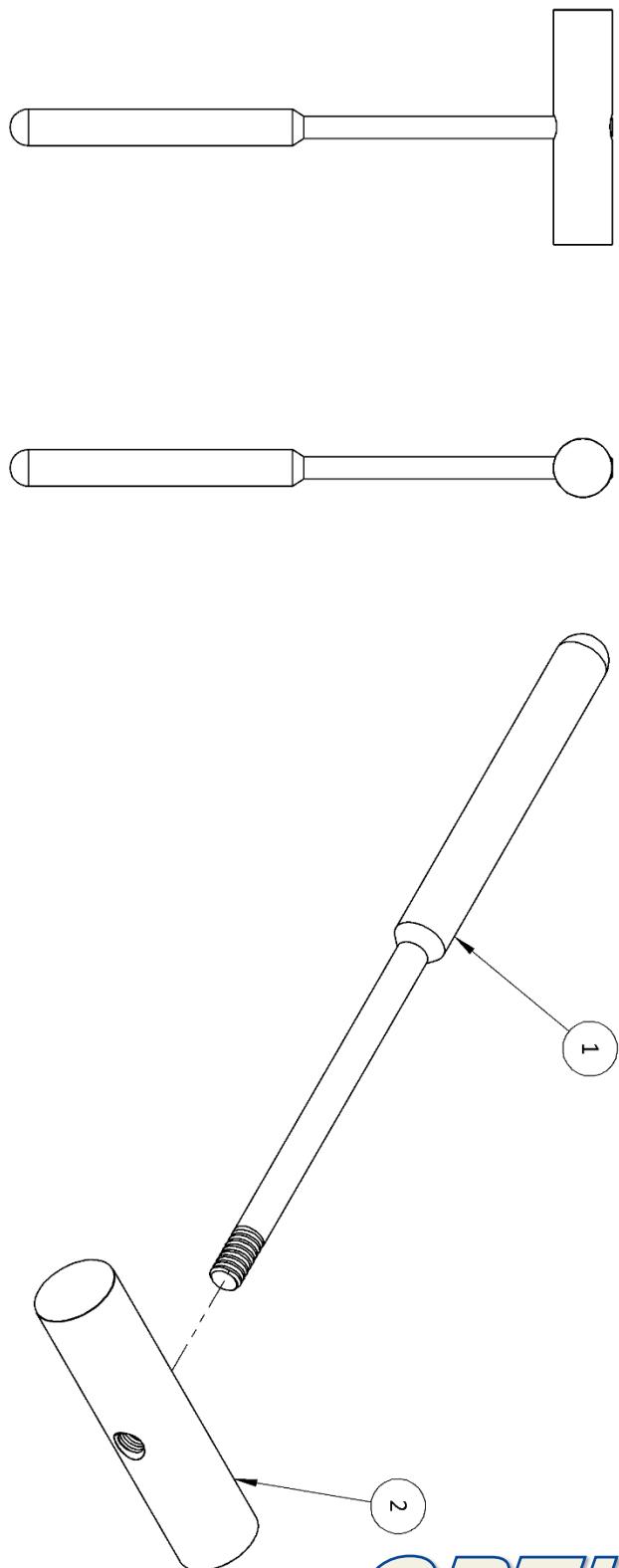


20. After completion you should be able to use the Move Component icon to dynamically rotate the assembly.



Finished

ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	IMT 103_3a		1
2	IMT 103_3b		1



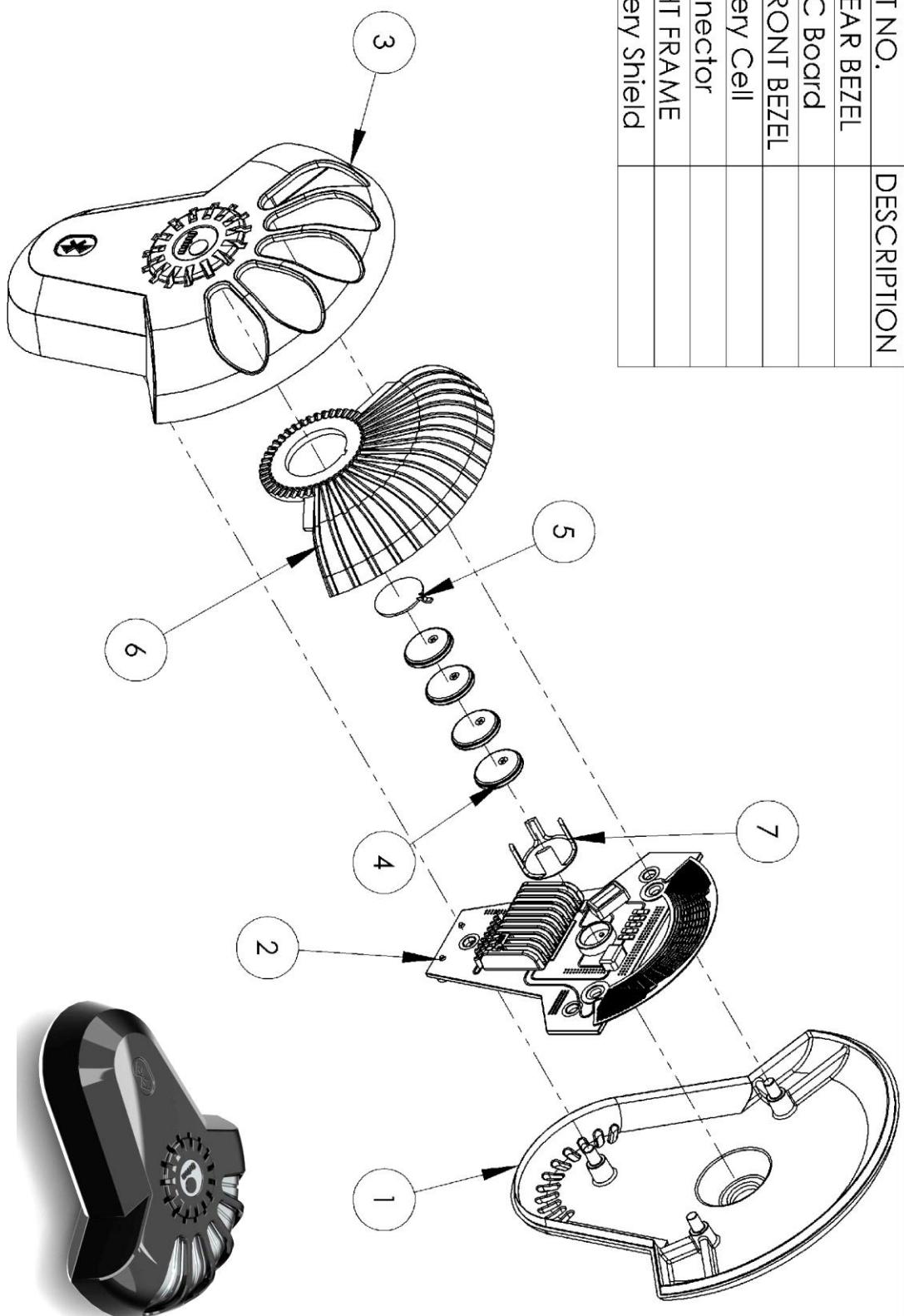
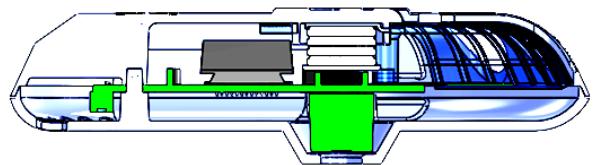
OPTIONAL

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5	4																		
3	2																		
1	1																		
<table border="1"> <tr> <td>APPLICATION</td> <td>DO NOT SCALE DRAWING</td> </tr> <tr> <td>NEXT ASSY</td> <td>USED ON</td> </tr> <tr> <td>FINISH</td> <td></td> </tr> <tr> <td colspan="2">COMMENTS:</td> </tr> <tr> <td colspan="2">INTERPRET GEOMETRIC TOLERANCING PER: MATERIAL</td> </tr> <tr> <td colspan="2">DRAWN BY: Elgin Community College Title: <i>[Signature]</i></td> </tr> <tr> <td colspan="2">CHECKED BY: <i>[Signature]</i></td> </tr> <tr> <td colspan="2">ENG APPR. MFG APPR.</td> </tr> <tr> <td colspan="2">Q.A.</td> </tr> </table>		APPLICATION	DO NOT SCALE DRAWING	NEXT ASSY	USED ON	FINISH		COMMENTS:		INTERPRET GEOMETRIC TOLERANCING PER: MATERIAL		DRAWN BY: Elgin Community College Title: <i>[Signature]</i>		CHECKED BY: <i>[Signature]</i>		ENG APPR. MFG APPR.		Q.A.	
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<table border="1"> <tr> <td>SIZE</td> <td>DWG. NO.</td> <td>REV</td> </tr> <tr> <td>A</td> <td></td> <td></td> </tr> <tr> <td>SCALE: 1:2</td> <td>WEIGHT:</td> <td>SHEET 1 OF 1</td> </tr> </table>		SIZE	DWG. NO.	REV	A			SCALE: 1:2	WEIGHT:	SHEET 1 OF 1									
SIZE	DWG. NO.	REV																	
A																			
SCALE: 1:2	WEIGHT:	SHEET 1 OF 1																	

Lab 5b
Bluetooth Headgear



ITEM NO.	QTY.	PART NO.	DESCRIPTION
1	1	L4 REAR BEZEL	
2	1	L4 PC Board	
3	1	L4 FRONT BEZEL	
4	4	Battery Cell	
5	1	Connector	
6	1	LIGHT FRAME	
7	1	Battery Shield	



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TOLERANCES:
FRACTIONAL: ±
ANGULAR: MACH†
TWO PLACE DECIMAL: †
THREE PLACE DECIMAL: ±
INTERPRET GEOMETRIC
TOLERANCING PER:

MATERIAL

COMMENTS:

REV

SIZE DWG. NO.
A **L5b**

SCALE: 1:5

WEIGHT:

REV

NEXT ASSY

USED ON

FINISH

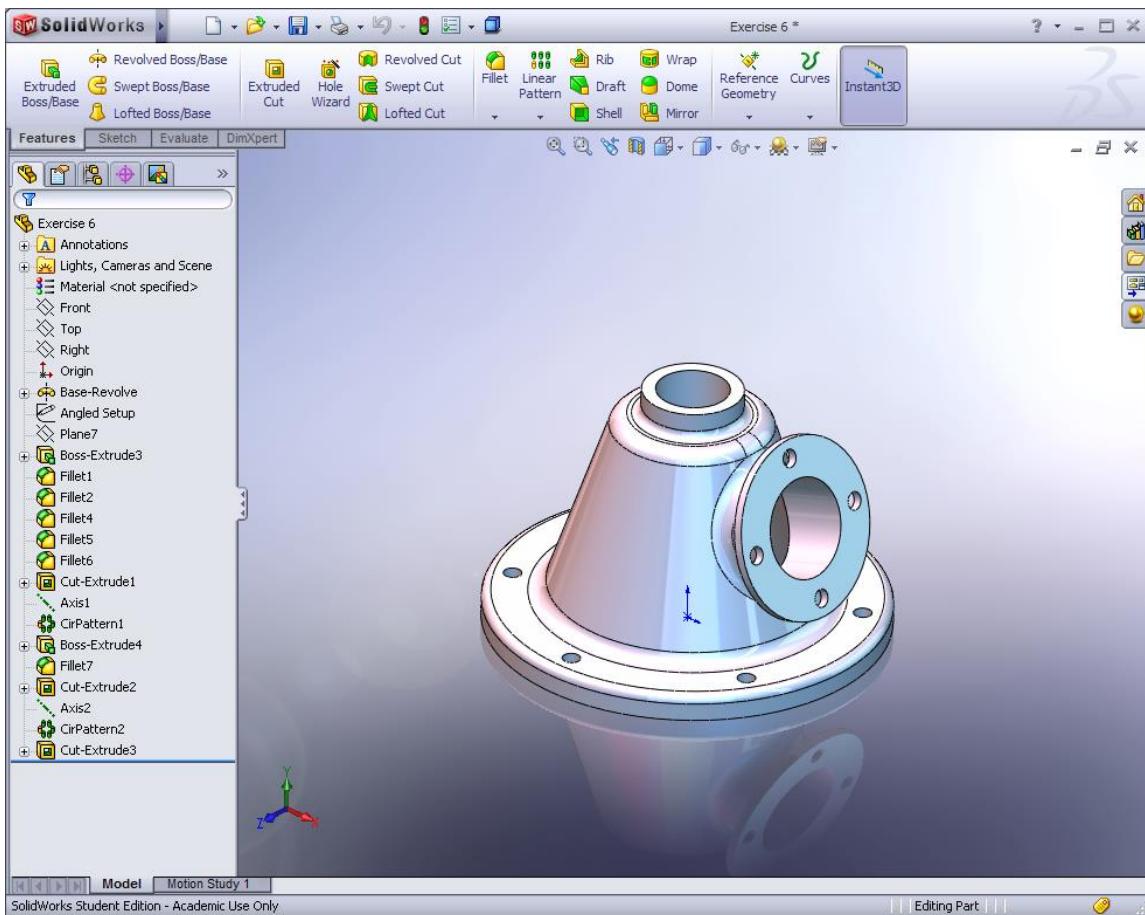
APPLICATION

DO NOT SCALE DRAWING

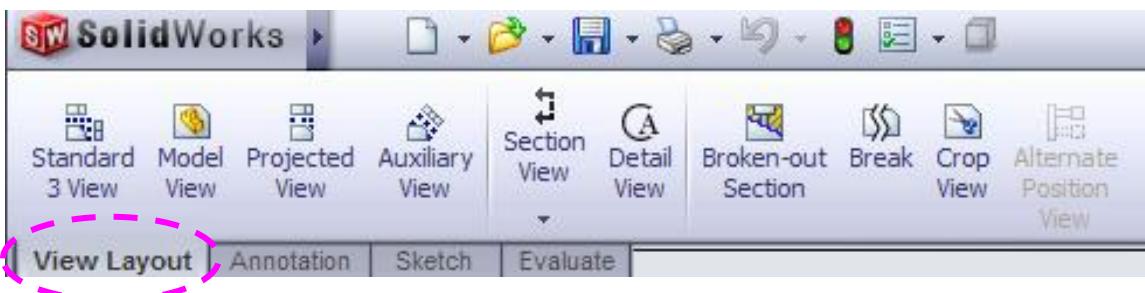
EXERCISE 6

Fundamental 2D Drawing Creation

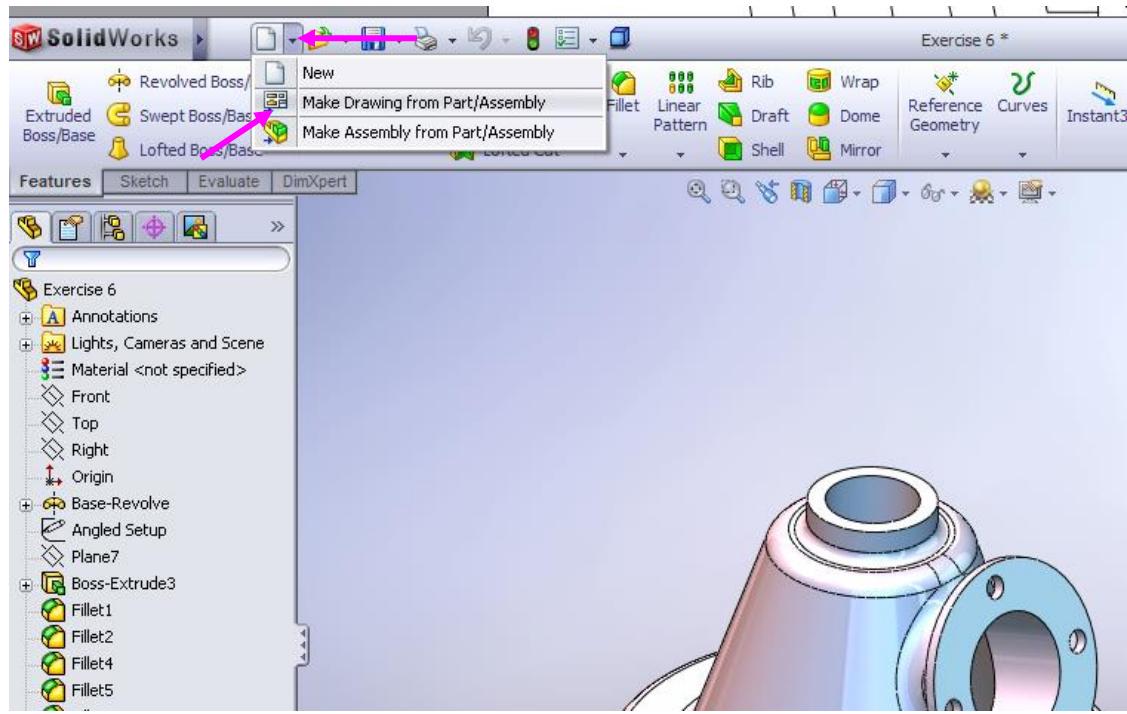
1. Open the “Exercise 6” part file.



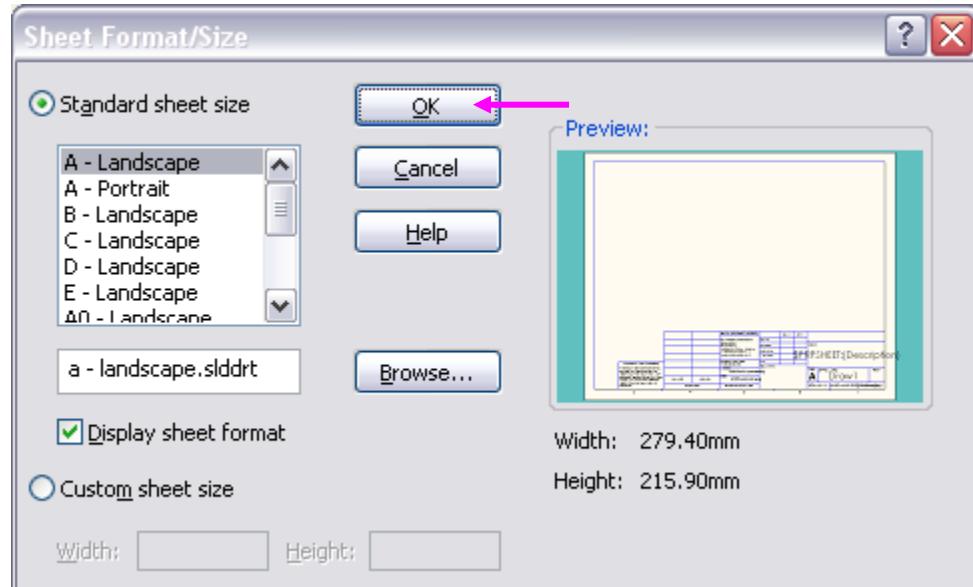
2. View Layout/Drawing Toolbar.



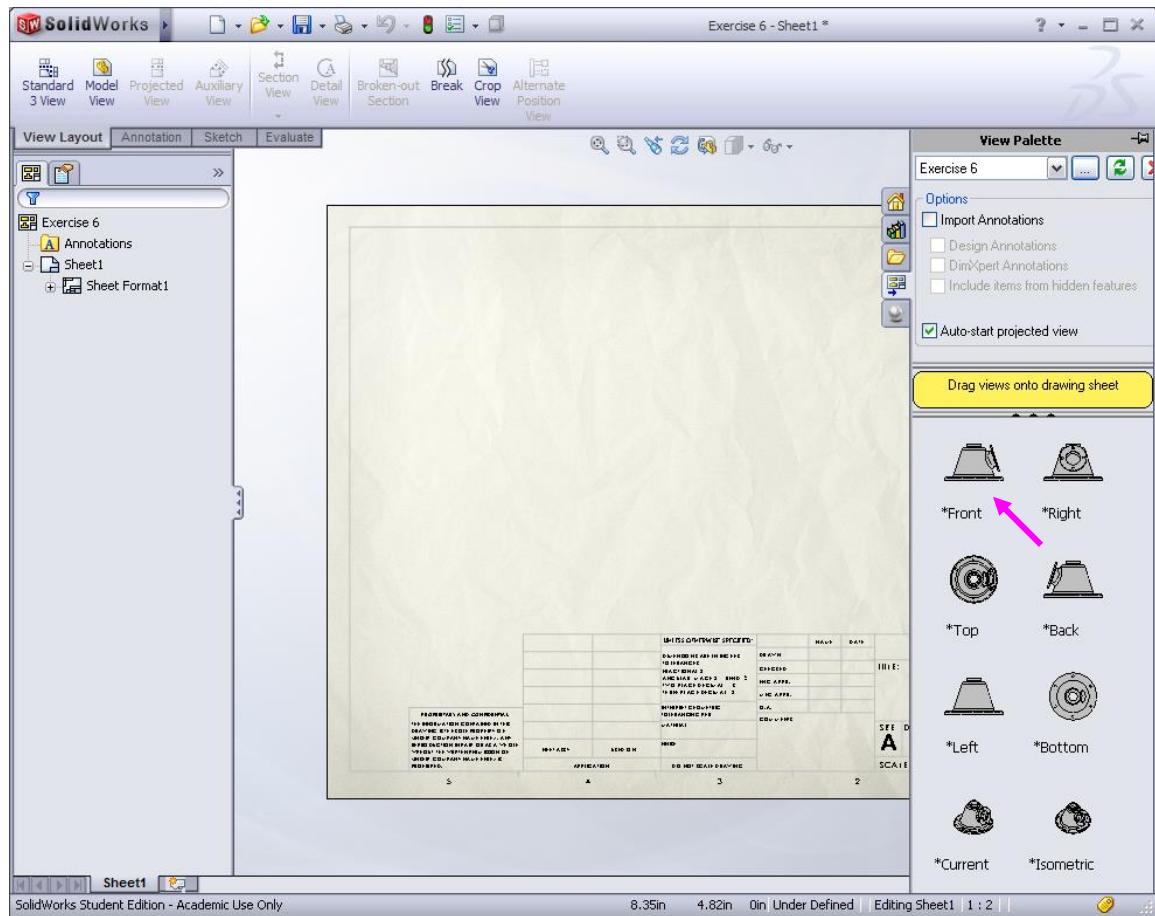
3. One method of inserting a part (E6) into a drawing is to first open the desired part file. Next, go to the new document icon and find and click on the arrow to its right. Select the Make Drawing from Part/Assembly option.



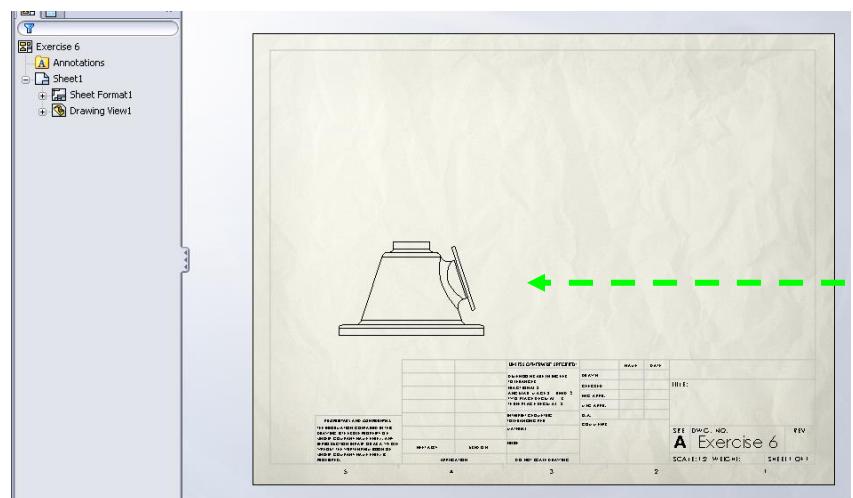
4. Notice the options for Sheet Format/Size will pop up. Select A- Landscape and hit OK.



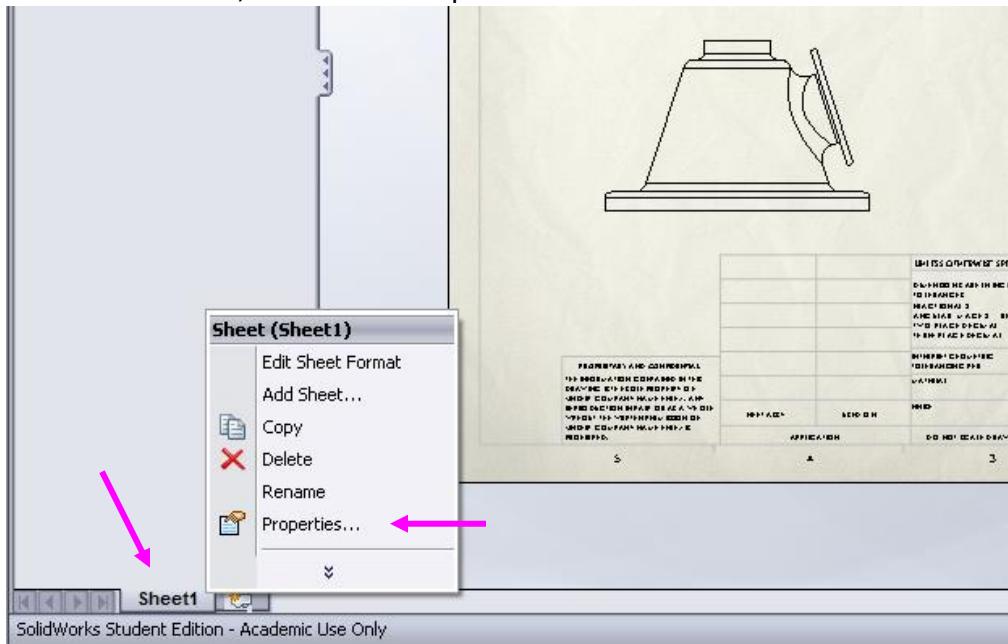
5. **Inserting a View:** You should now see the paper border and the “View Palette” to the right.



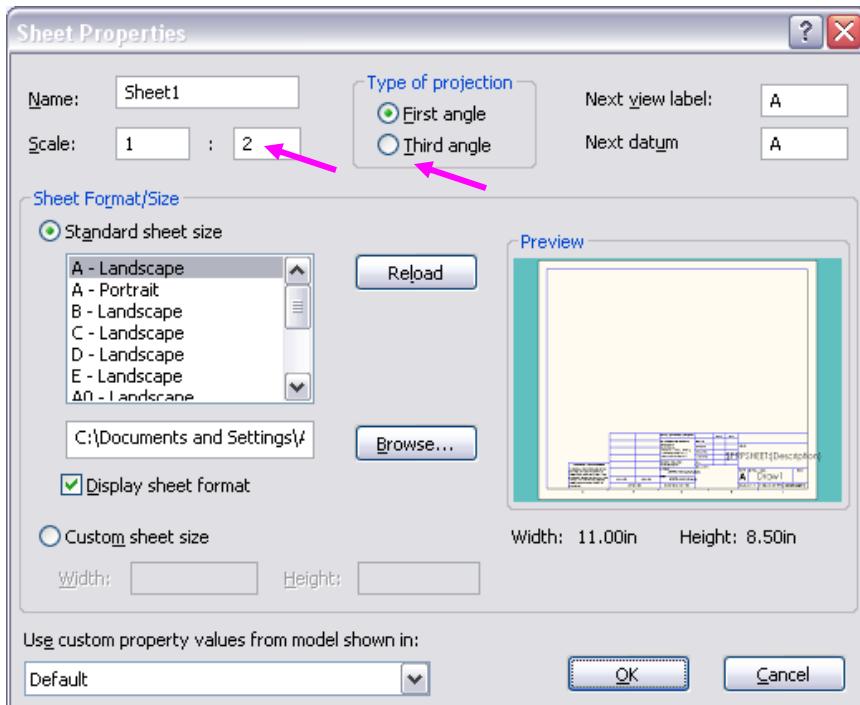
6. Grab the “Front” view by moving the pointer over it and hold the left mouse button down. Drag it onto the sheet and release the LMB. Hit “ESC”.



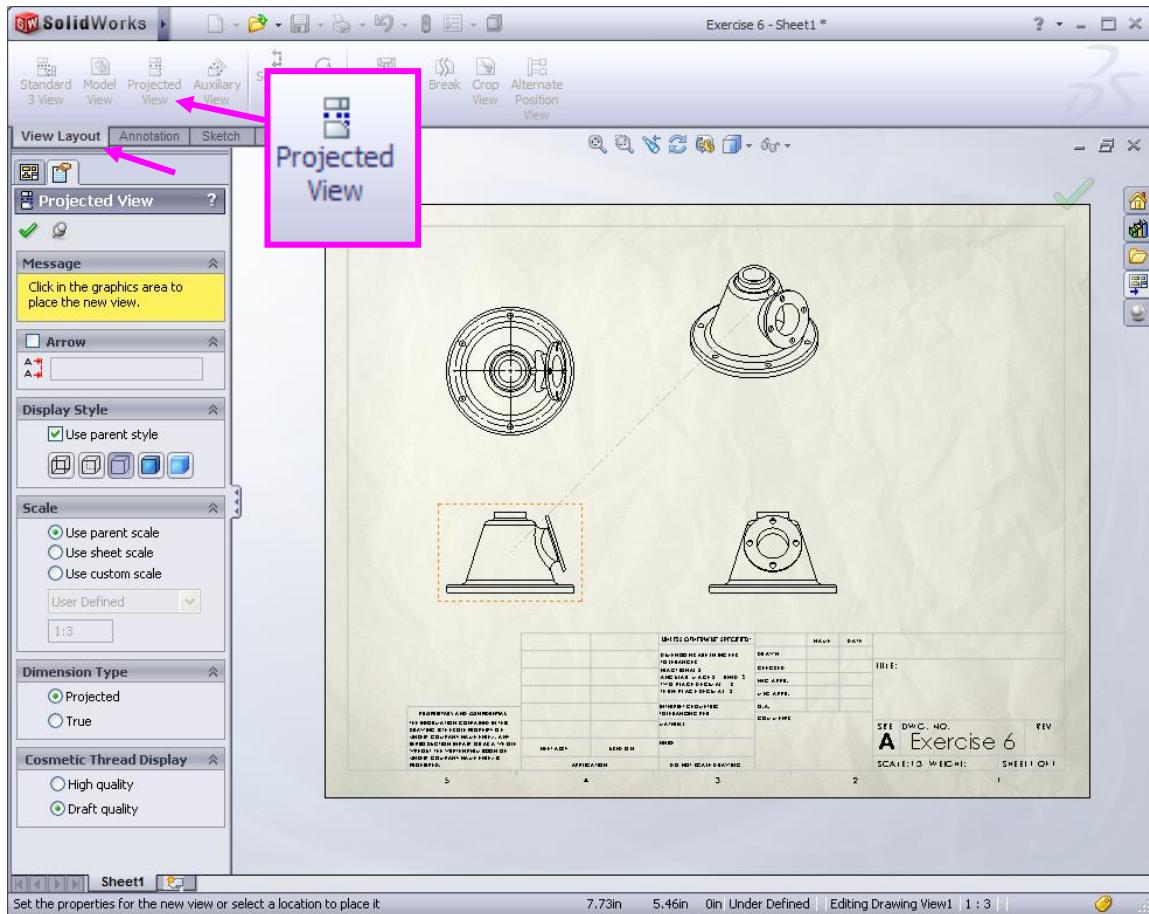
7. **Editing the Scale/Paper size:** RMB click on the “Sheet1” tab at the bottom left corner, and select “Properties”



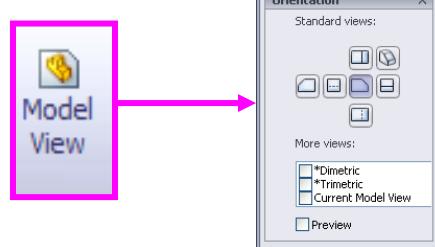
8. This dialog box contains many useful options for editing the drawing. Change the 1:2 scale to 1:3 and hit ok. Also select the “Third Angle” option.



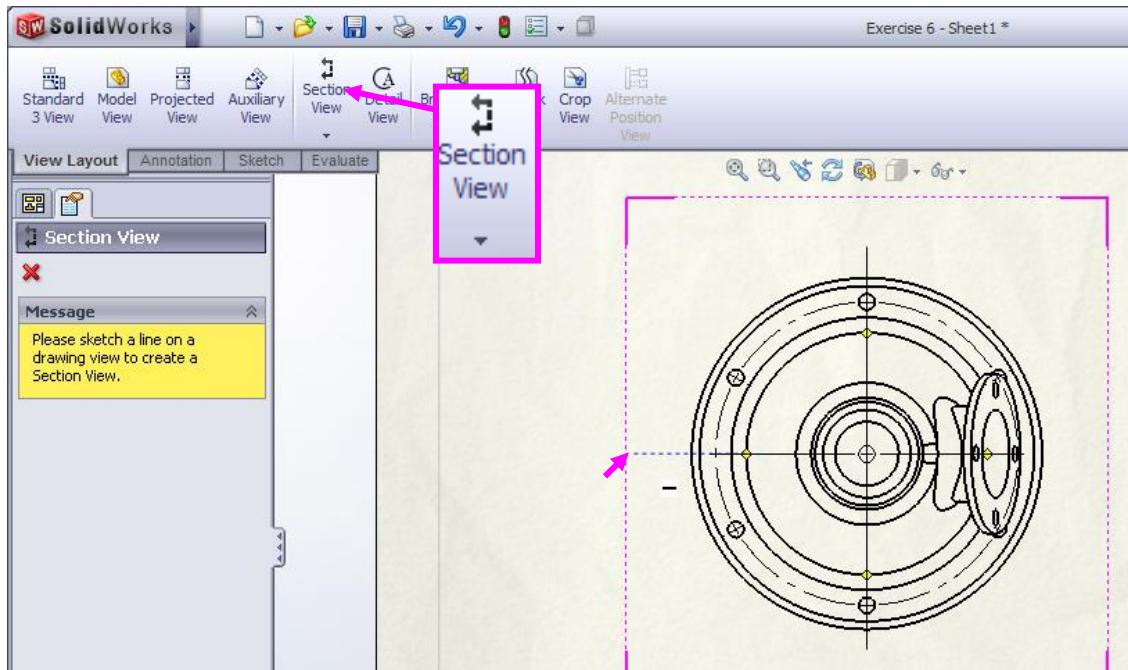
9. **Projected (Unfolding) Views:** Select the “View Layout” tab and then the “Projected view” option. Then select the front view and move your pointer up, LMB click, then move your pointer right and LMB click and then 45 Degrees from the front view and LMB click to get your isometric view unfolded.



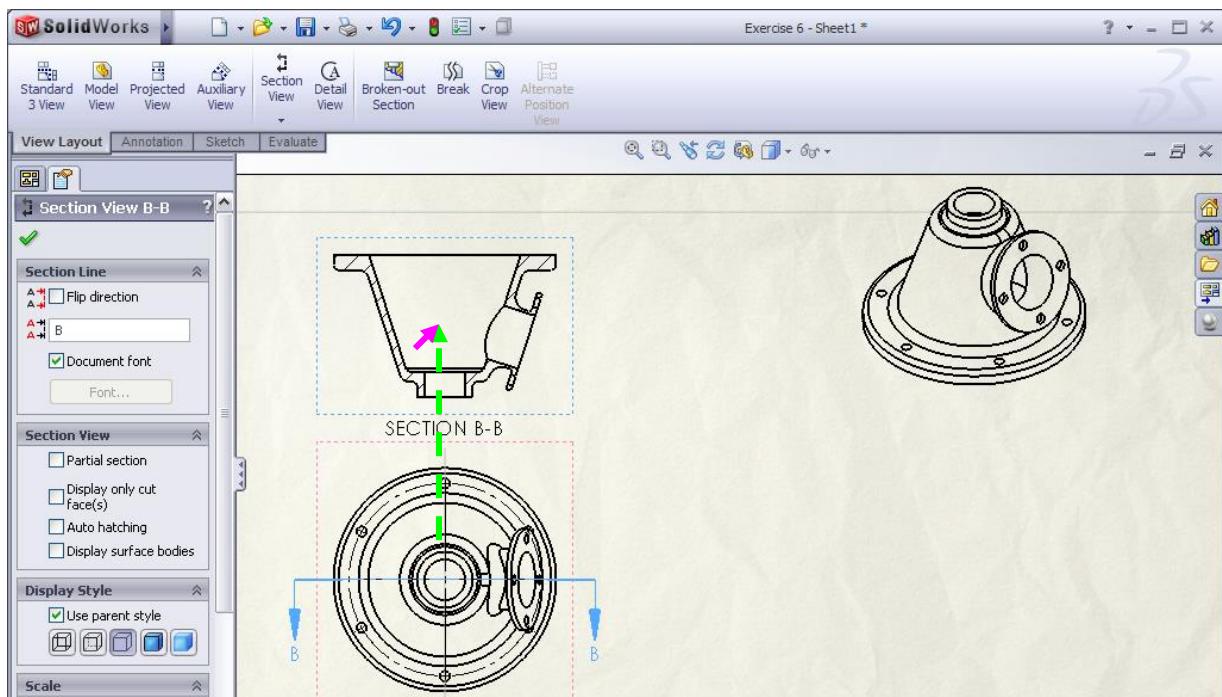
10. **How to move views:** Move pointer over a view border and once the border highlights in red LMB click and hold on the border line. Drag pointer to desired location. Most projected views are locked into the original view they are projected from, and their movement is limited to either horizontal or vertical to the source view. If you wish to disconnect a view from its source then simply RMB click on the desired view and find the “Alignment”/“Break alignment” option to disconnect the view.
11. **Model View:** Can also be used to create standard views individually in a drawing by selecting the icon.



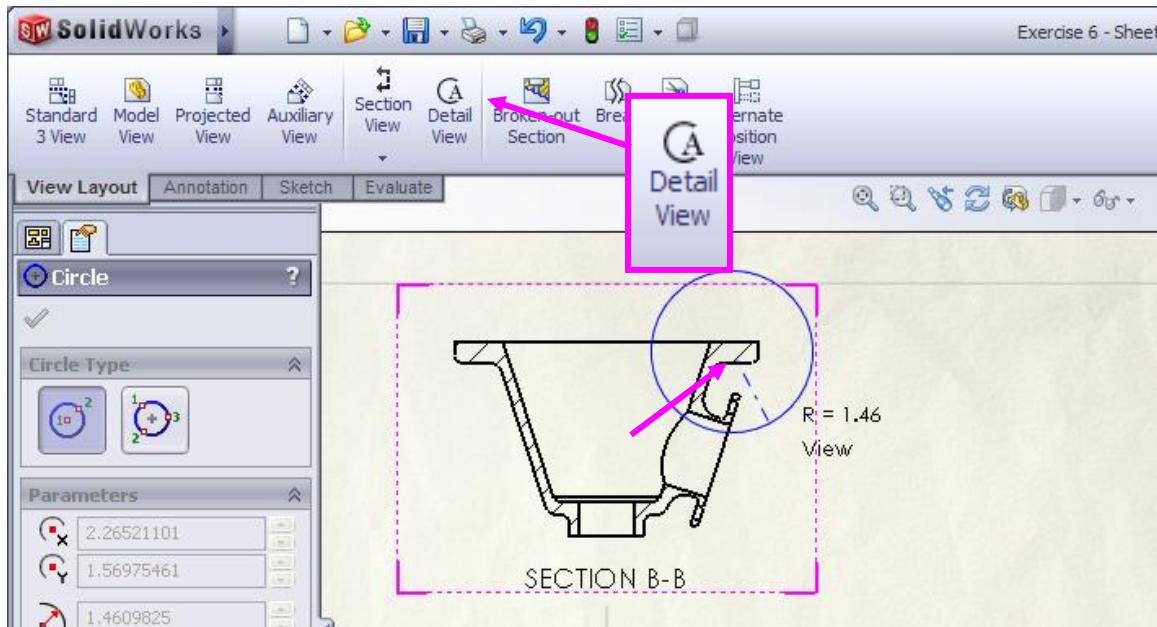
12. **Section Views:** Select the “Section View” icon and then locate the left side quadrant edge of the top view. While a short distance away from contacting the edge LMB click and drag a line horizontally through the entire view. (Don’t stop in the middle).



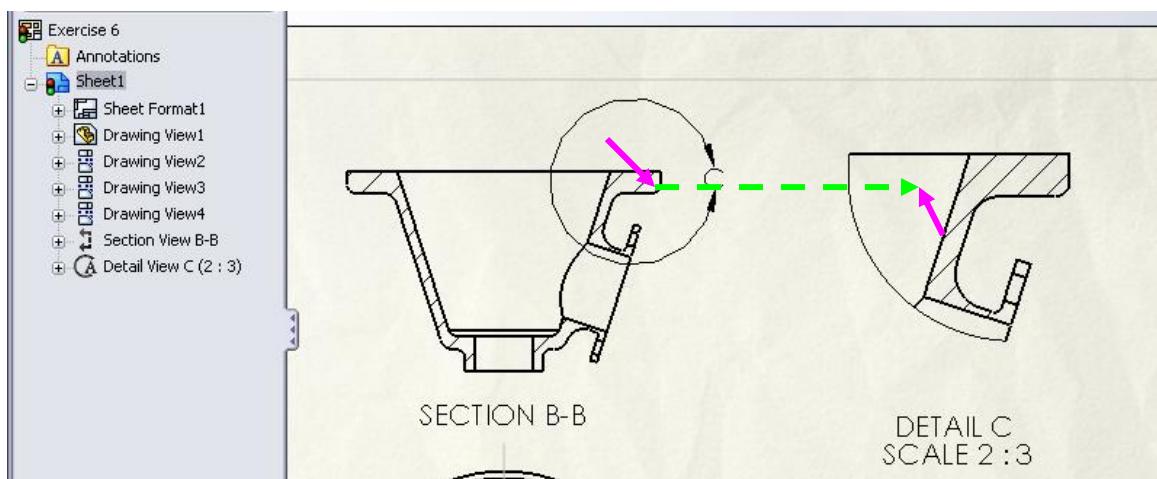
13. Once through release the LMB and you should be able to now drag off a section view and drop it just above the “Top” view by LMB clicking again.



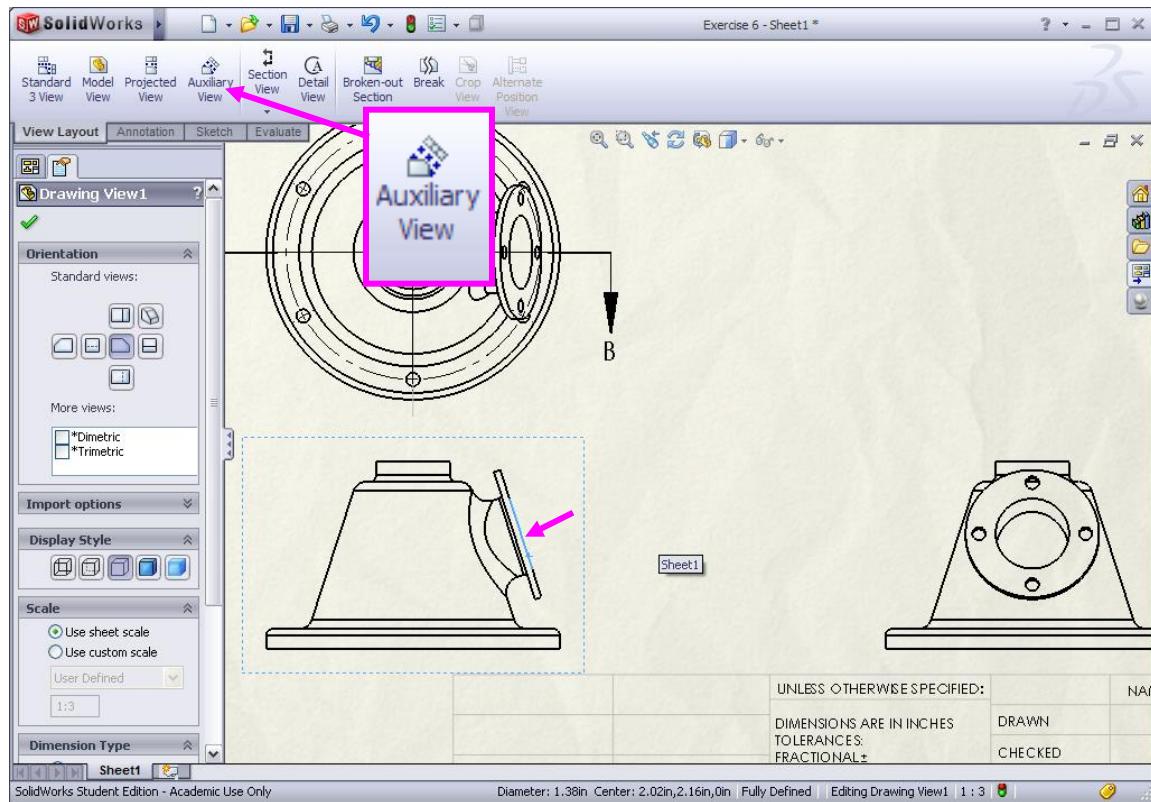
14. **Detail Views:** Select the “Detail View” icon. The circle tool is automatically activated so then you can draw a circle surrounding the region you wish to create a detail view from.



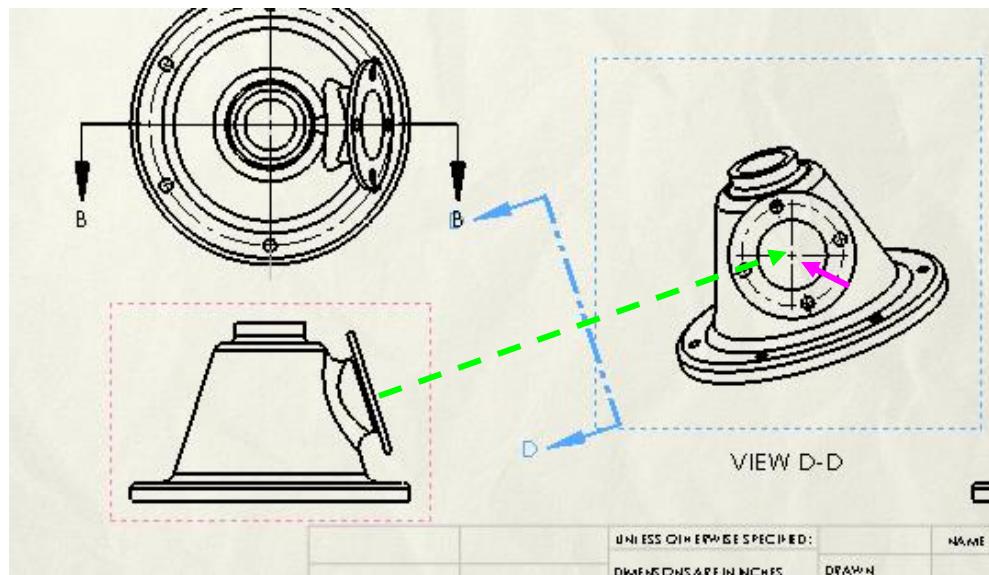
15. Move the view to the desired location and LMB click to release/drop it.
Note: the view scale can be changed by simply double clicking on the “scale” text and typing in a new value, and the position and diameter of the circle can be changed dynamically by LMB clicking and dragging its center or diameter.



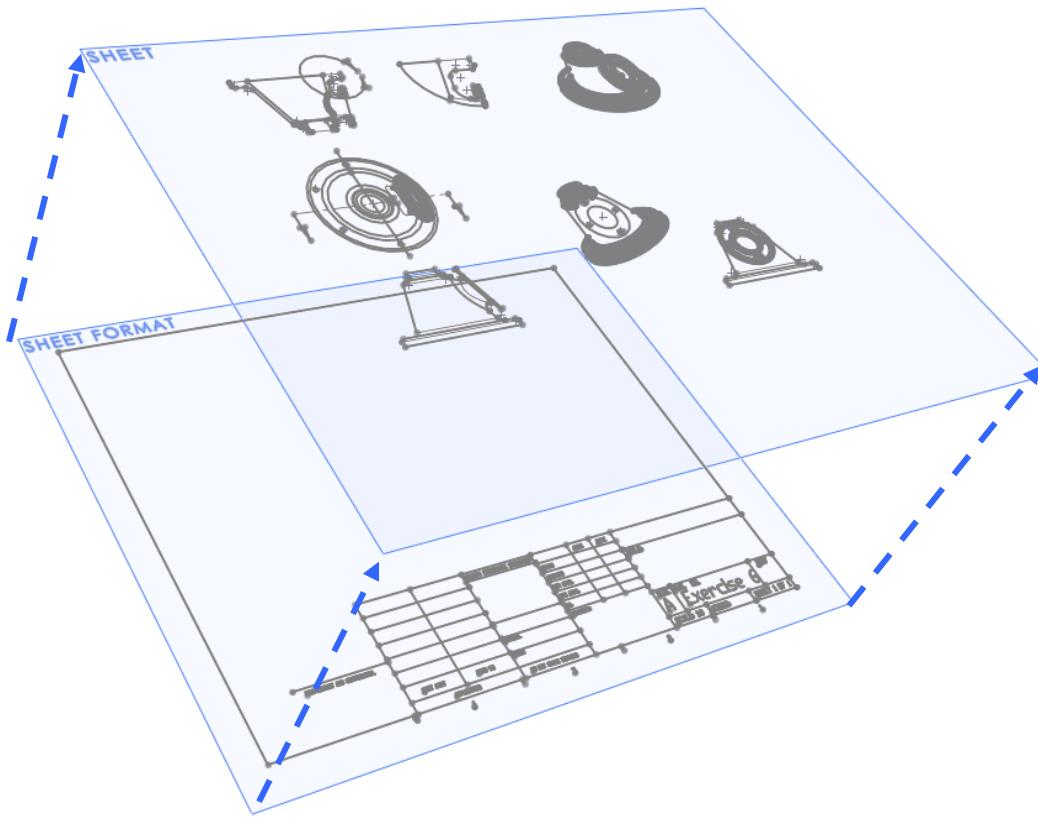
16. **Auxiliary Views:** Can be created by selecting an edge of the part and then select the Auxiliary view icon.



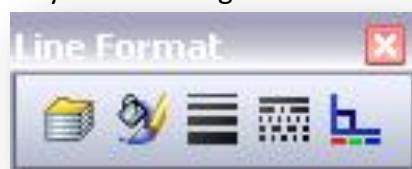
17. Then drag out the view and LMB click to drop it into position.



18. **Editing the Sheet and Sheet Format:** The “**Sheet**” is the transparent top level containing the model views, dimension and notes. The “**Sheet Format**” is the lower level that contains the border, title block, and notes.

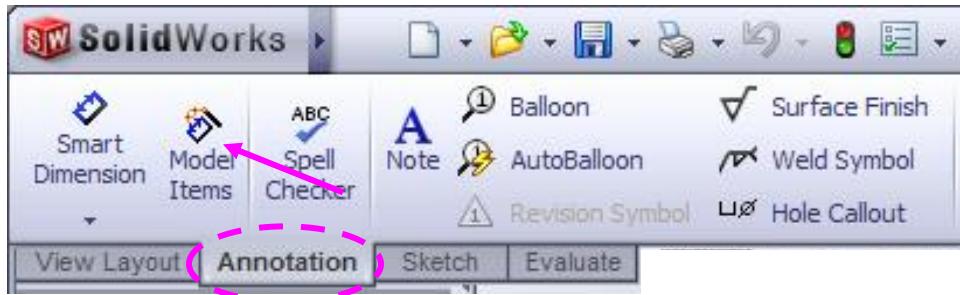


19. RMB click on the “SHEET1” tab located at the bottom left corner of the screen, and select “edit sheet format”.
20. Once in the sheet format you can draw lines, add or edit notes and with the line format toolbar you can change individual line colors, styles, and thicknesses.

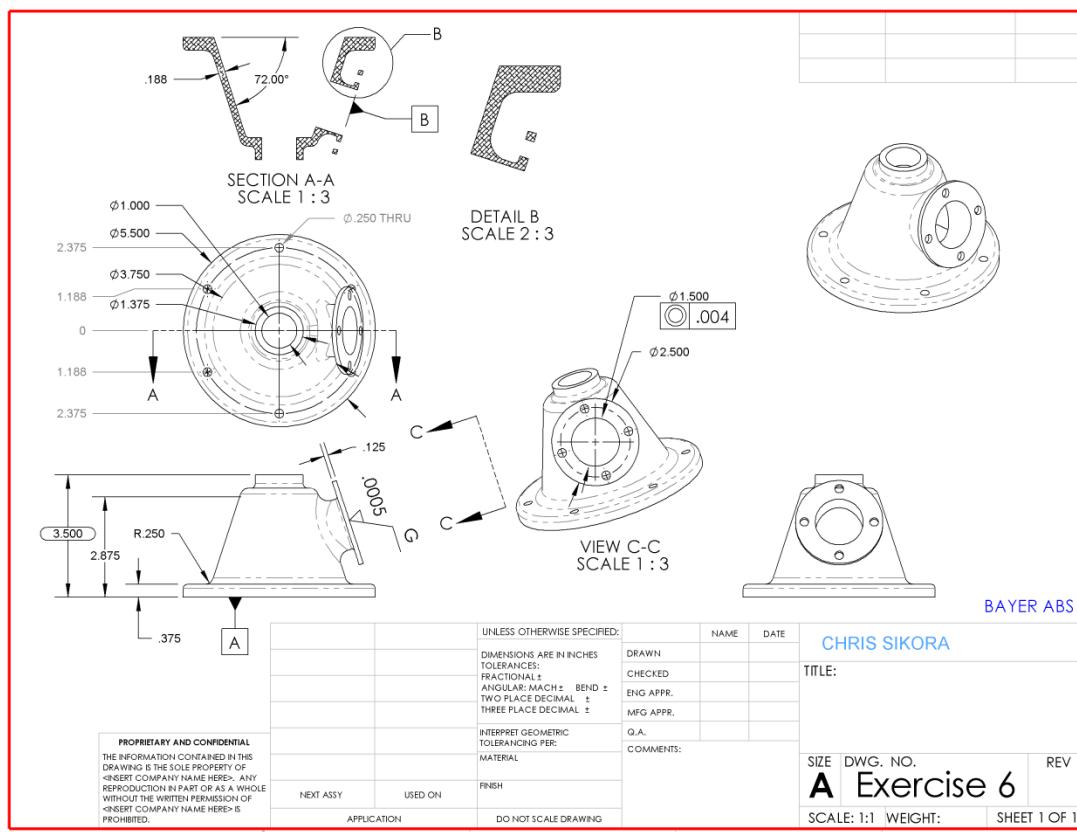


Note: The line Format toolbar can be opened by RMB clicking on the top grey border of the SolidWorks screen and locating it on the pull down list.

21. **Dimensions and Annotations:** To add independent and non-parametric dimensions you can select the “Smart Dimension” icon. Then just add dimensions the way you would in the sketcher. Otherwise you can use the “Model Items” option in the “Annotation” tab.



22. The “Model Items” are dimensions and annotations added to the model during its construction by the designer. These annotations can be inserted automatically into the drawing. These are true, editable, parametric dimensions.
23. Try using some of the other annotations like Note, Surface Finish, Welds, and GD&T.

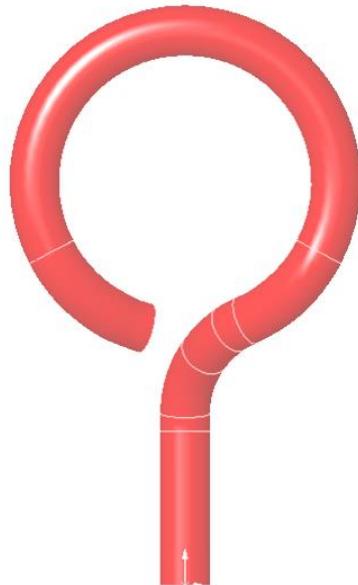
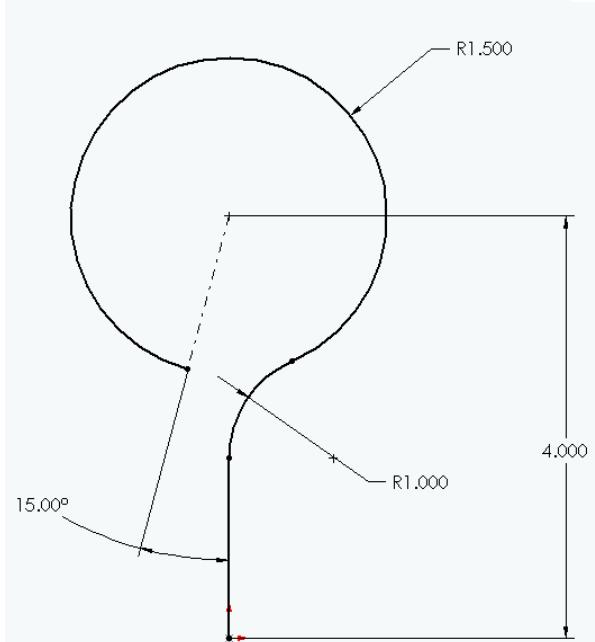


Finished

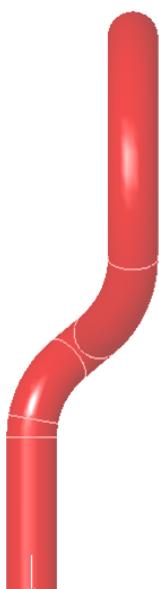
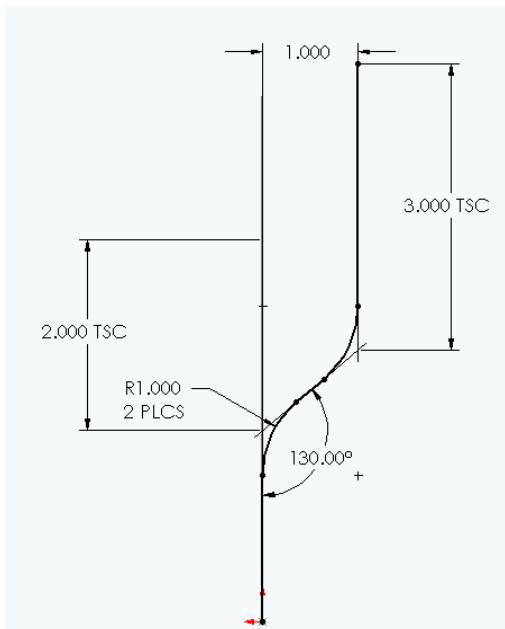
EXERCISE 7

Projected Curves and Sweeping

1. Sketch this on the “Front” plane.

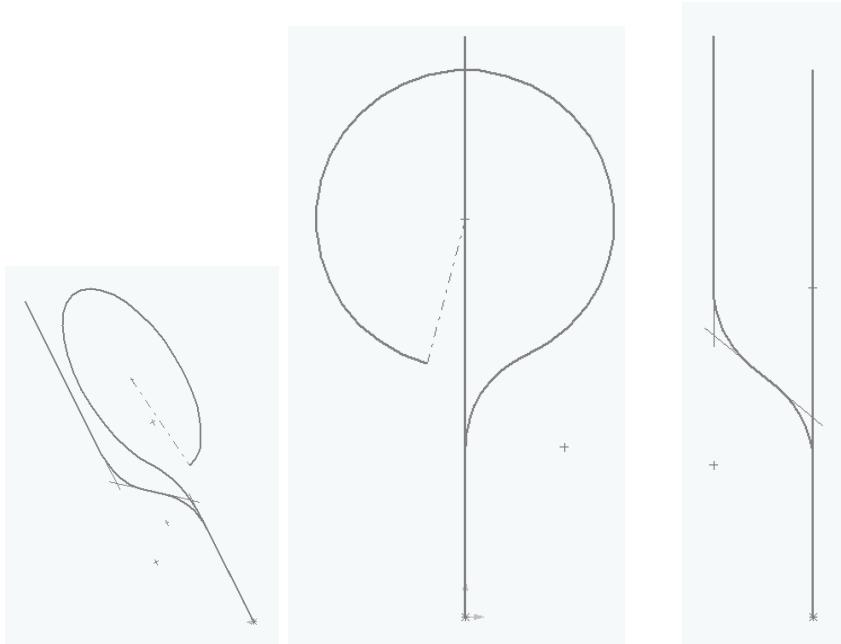


24. Hit Rebuild to exit the sketch.
25. Select the “Right” plane and start a sketch on it.
26. Draw the following.

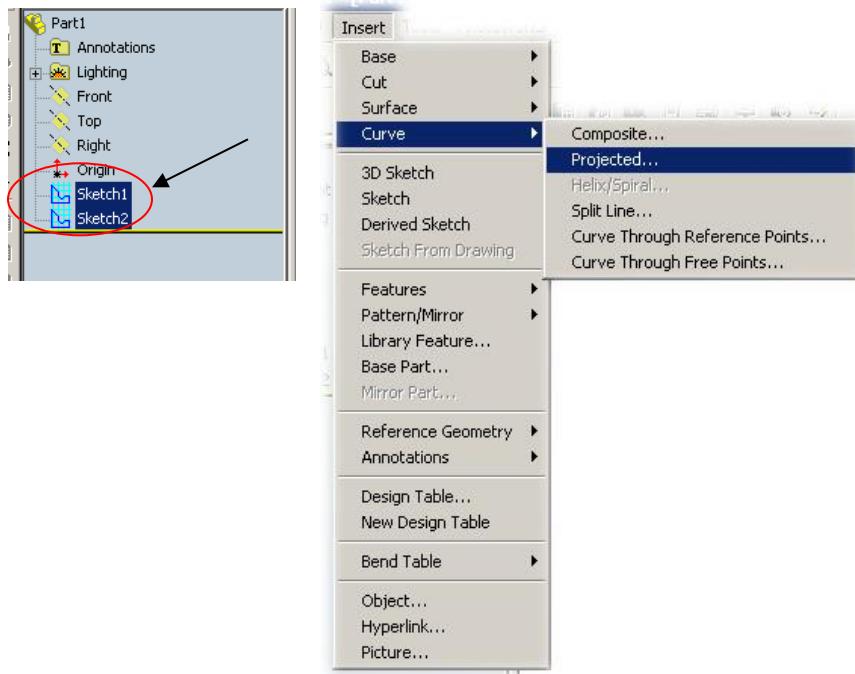


27. Rebuild to exit sketch.

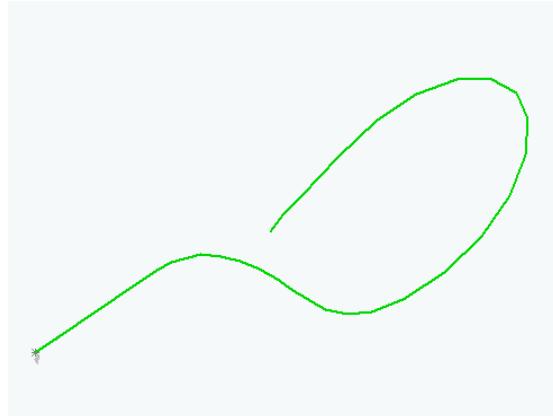
28. If you rotate the view you should see 2 gray sketched on top of each other. Our intent is to combine the profiles from both the "Front" and "Right" views to make 1 standalone 3 dimensional profile.



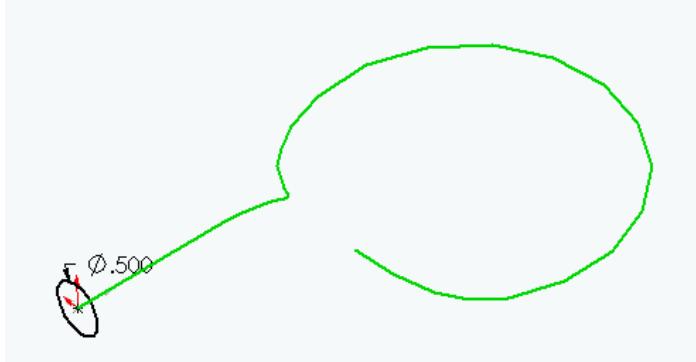
29. From the feature manager select both sketches. Then go to Insert/Curve/Projected.



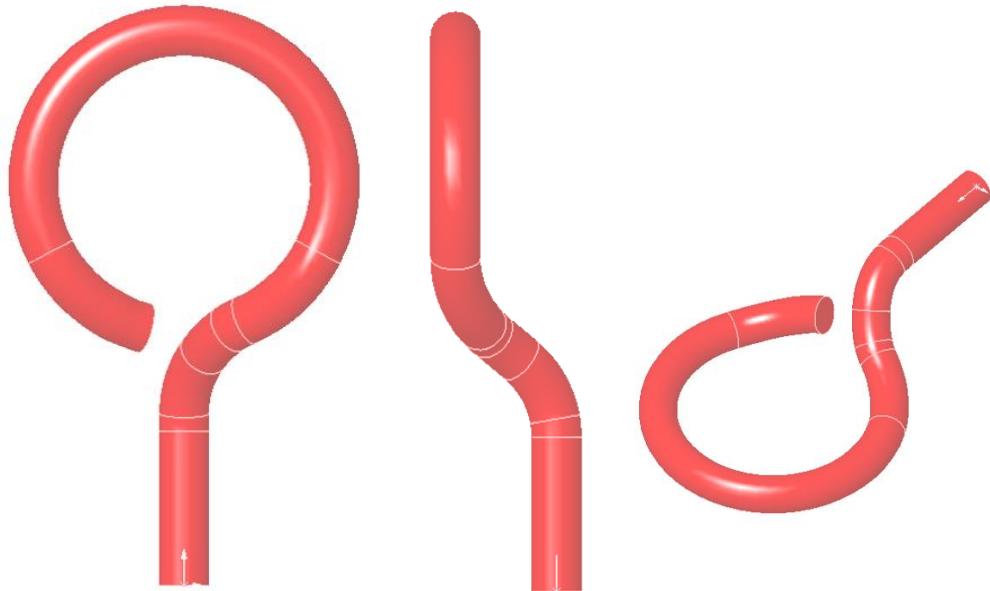
30. You should now have a single 3 Dimensional curve.



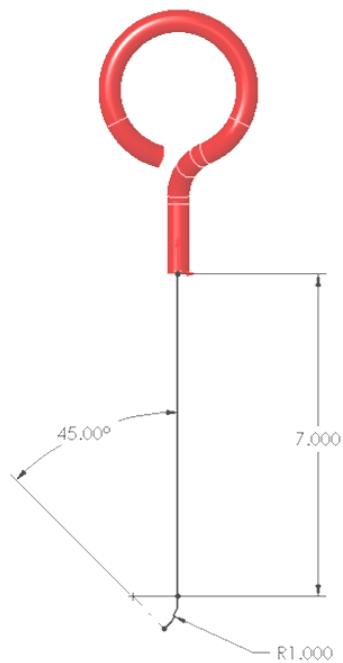
9. Create a $\frac{1}{2}$ " circle on the "Top" plane.



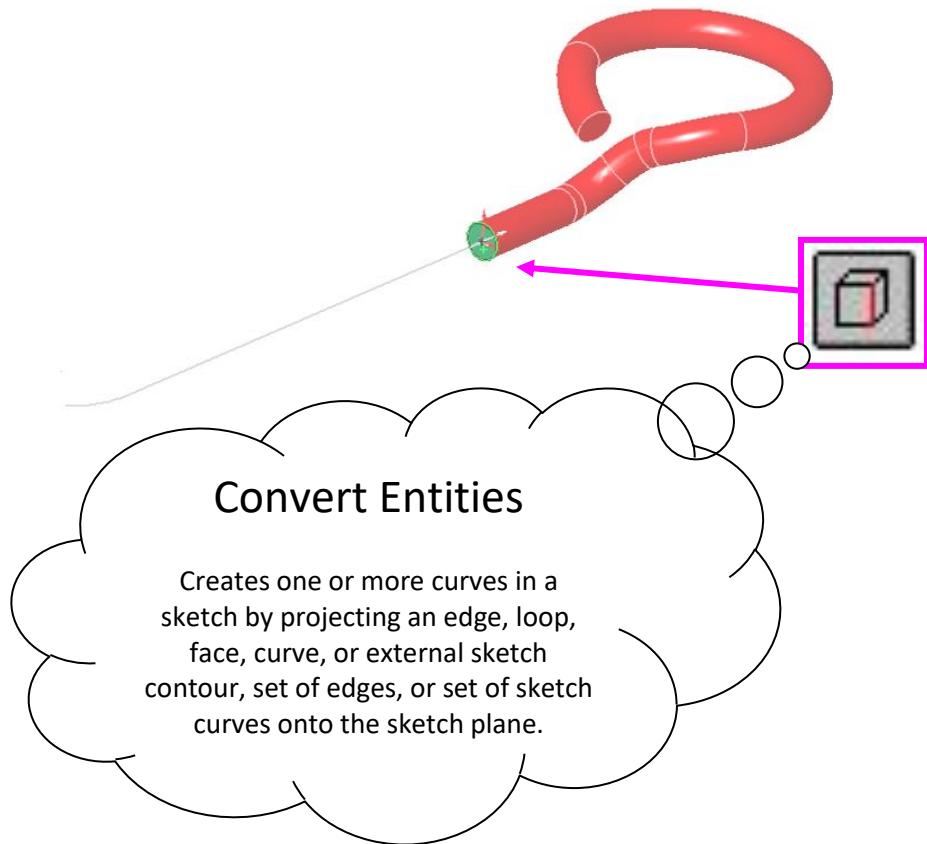
10. Rebuild and Sweep using the curve as the Path and the circle as the Profile.



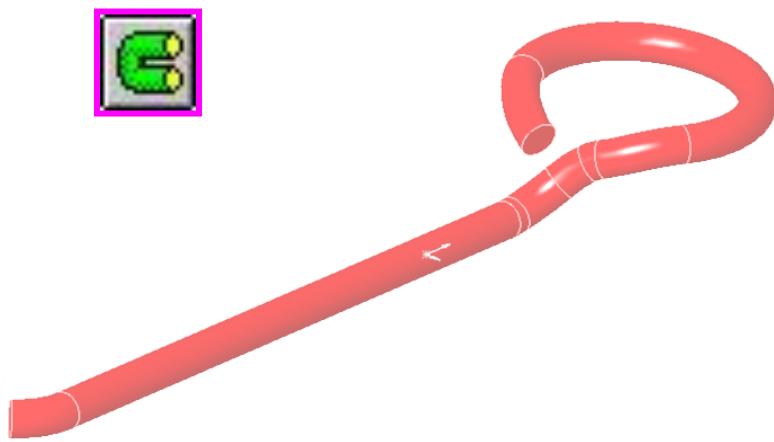
21. Start a sketch on the “Front” plane. Draw the following.



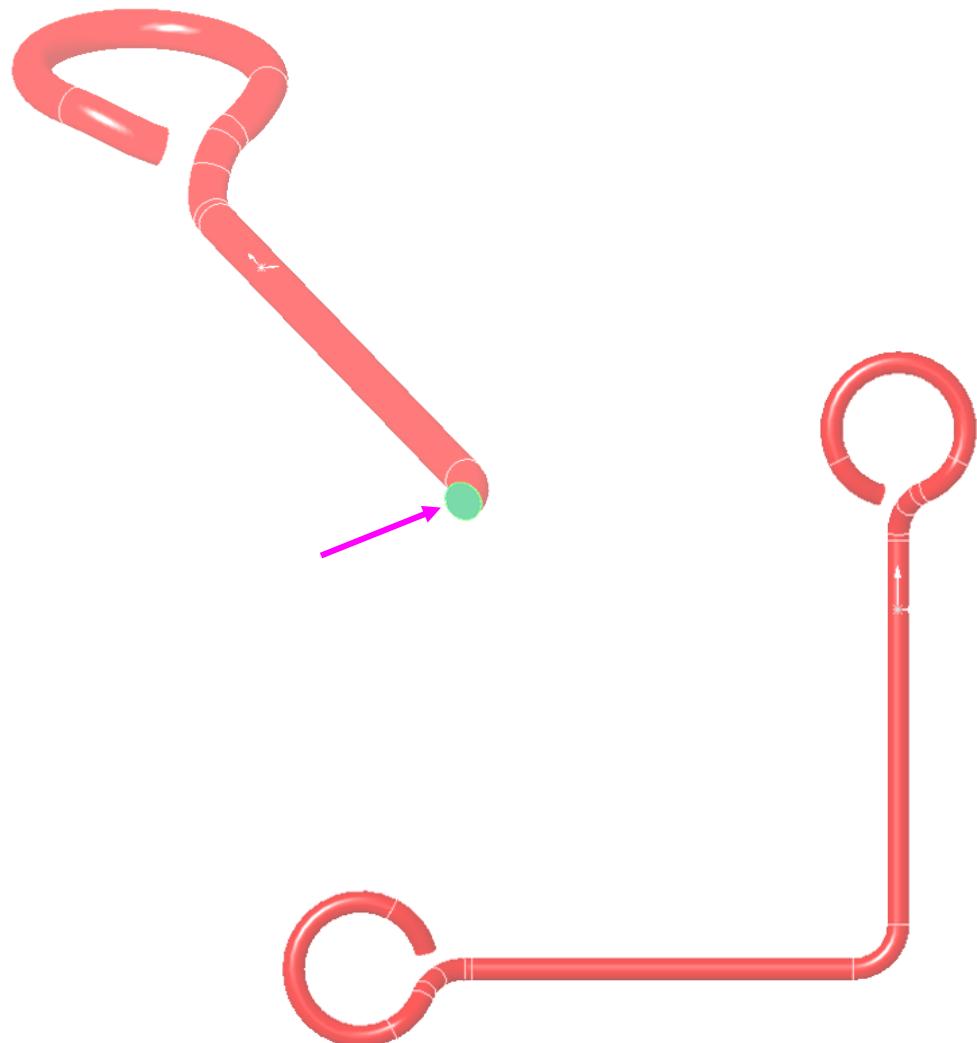
22. Rebuild to exit the sketch. Select the end face and start a sketch. While the end face is still highlighted green select the Convert Entities icon. Rebuild.



23. Sweep using the new path and converted entity as the profile.

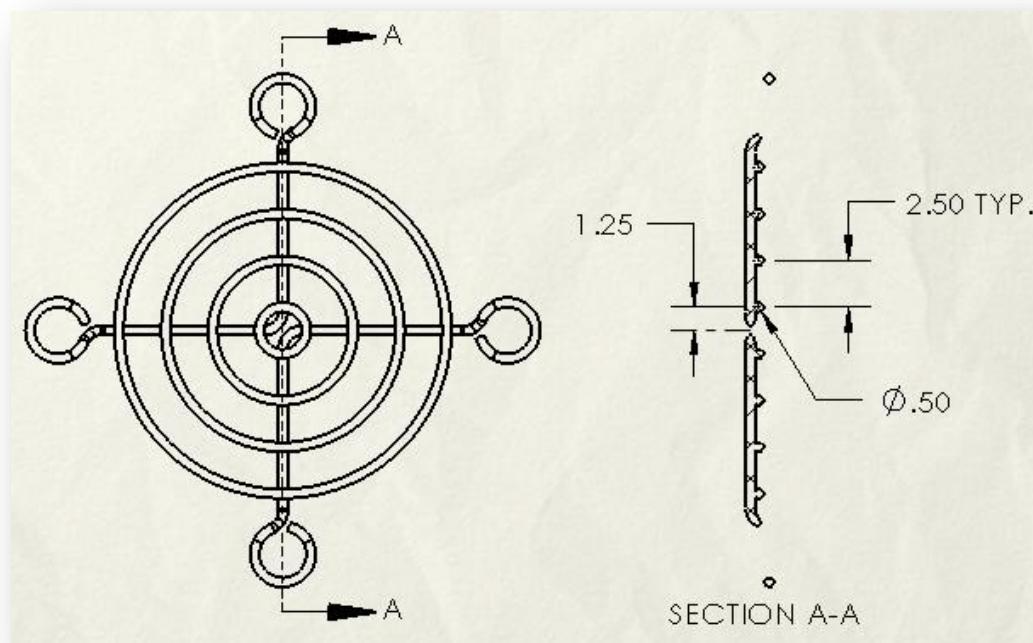


14. Select the end face and go to Insert/Pattern/Mirror/Mirror Body.

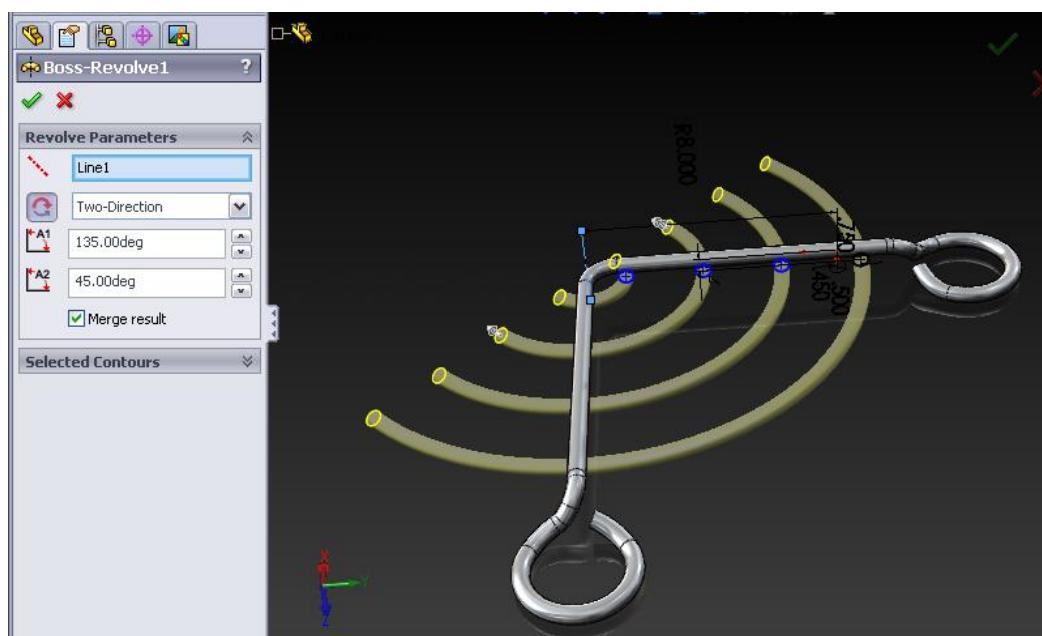


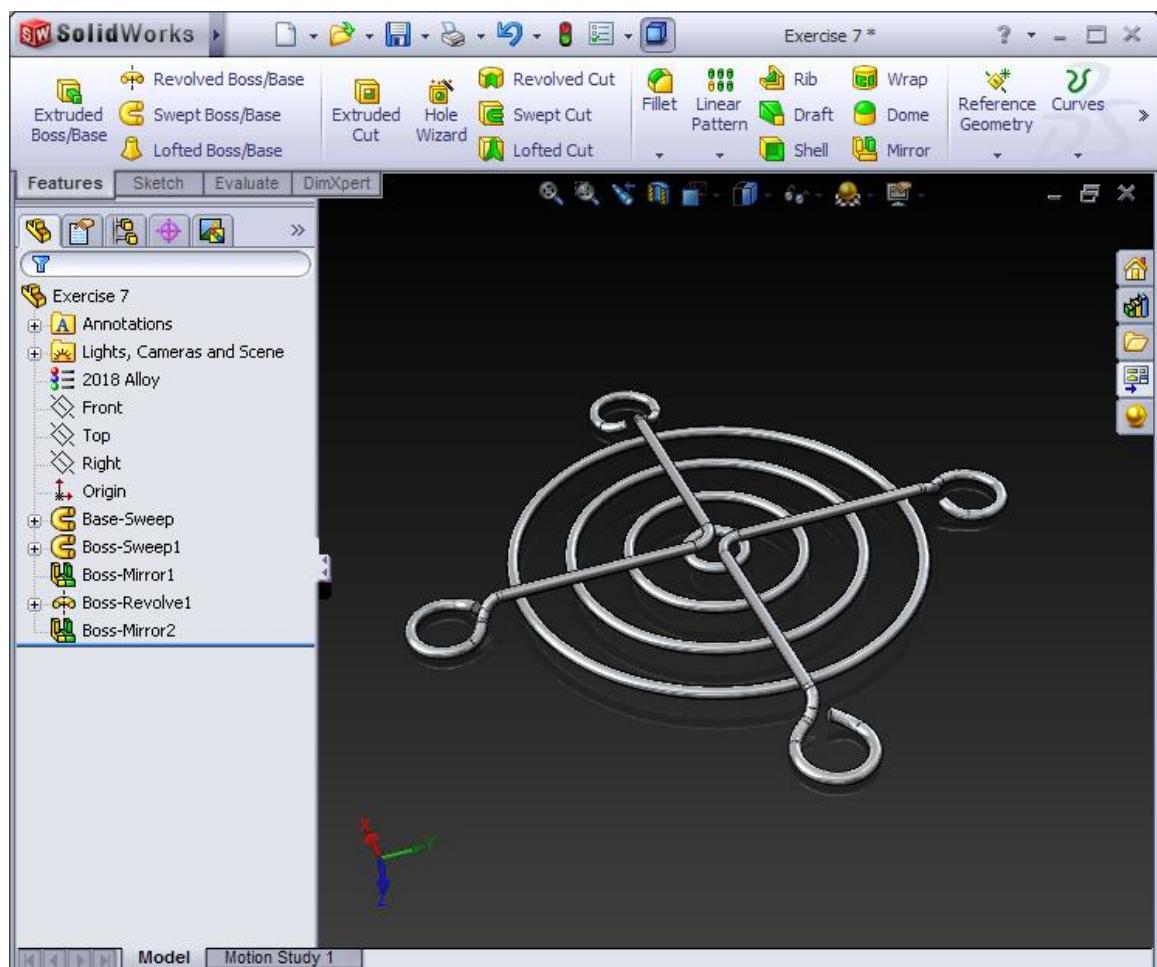
Now using the tools you have learned over the past 5 weeks finish the remainder of the model.

Hints to complete the model...

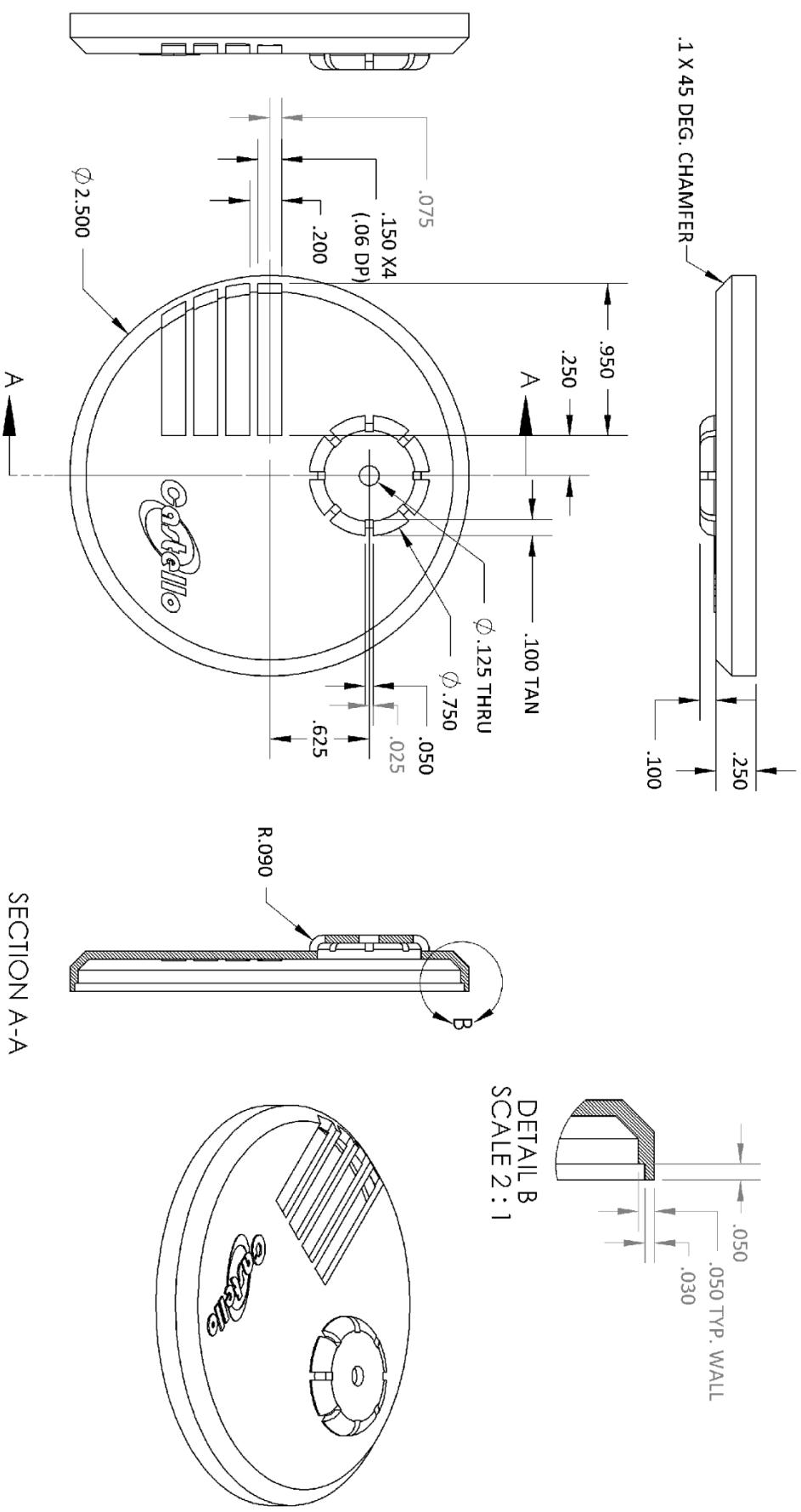


Boss Revolve “Two Directions”





The completed part; check to see if your feature tree looks the same as this one.



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4

3

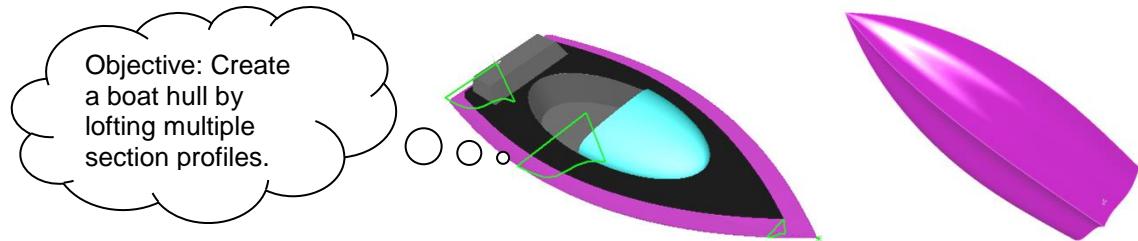
2

1

EXERCISE 8

Lofting

Loft creates a feature by making transitions between profiles. A loft can be a base, boss, cut, or surface.

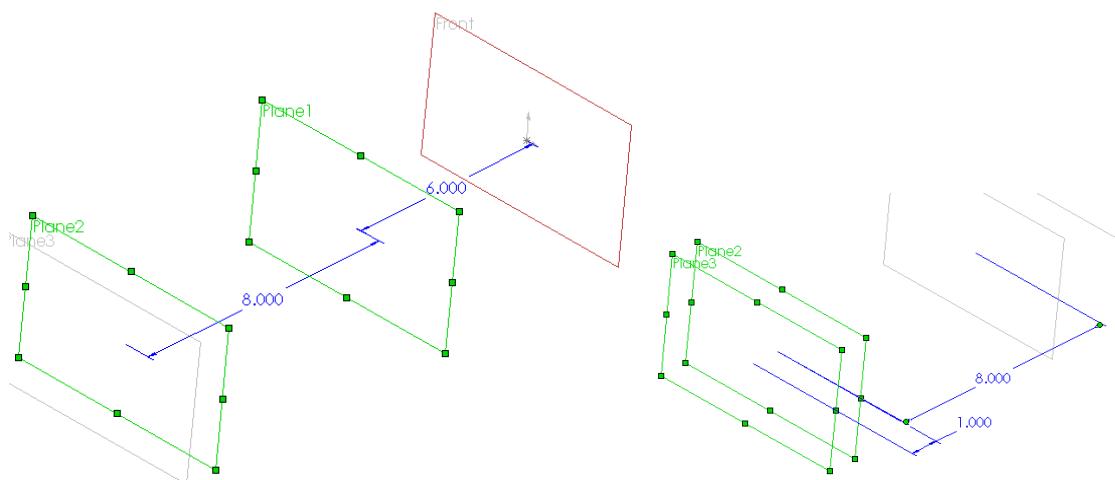


1. Create 4 planes beginning from the "Front" plane and offset from each other as shown.

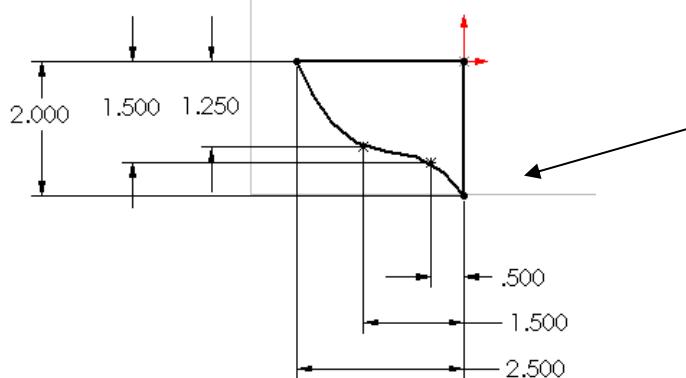
Plane 1 – 6.00"

Plane 2 – 8.00"

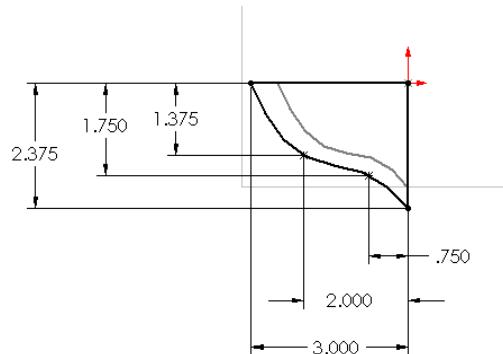
Plane 3 – 1.00"



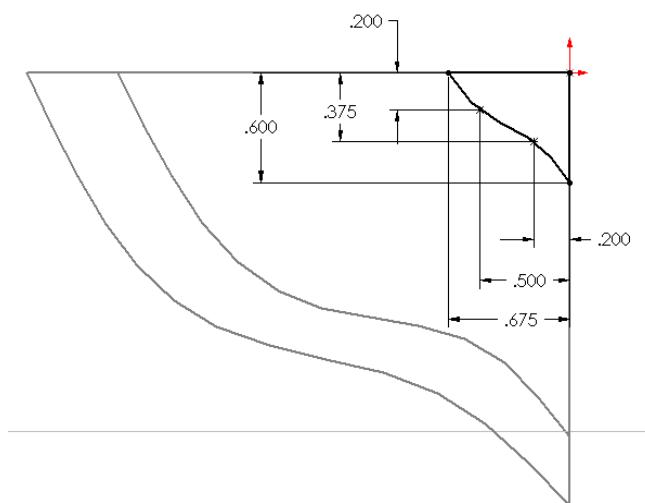
2. Sketch 1 on the "Front" plane should look like this... use the Spline tool.



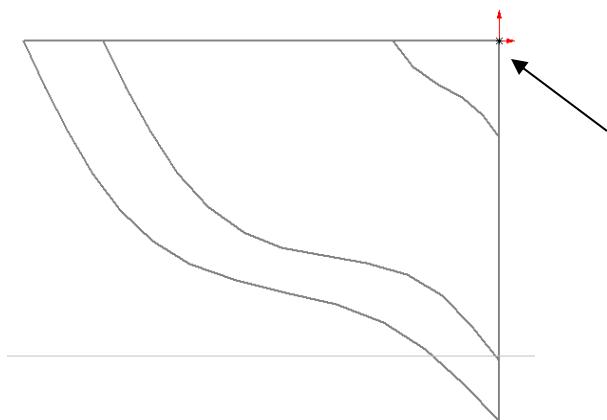
3. Sketch 2 on “Plane 1” should look like this...



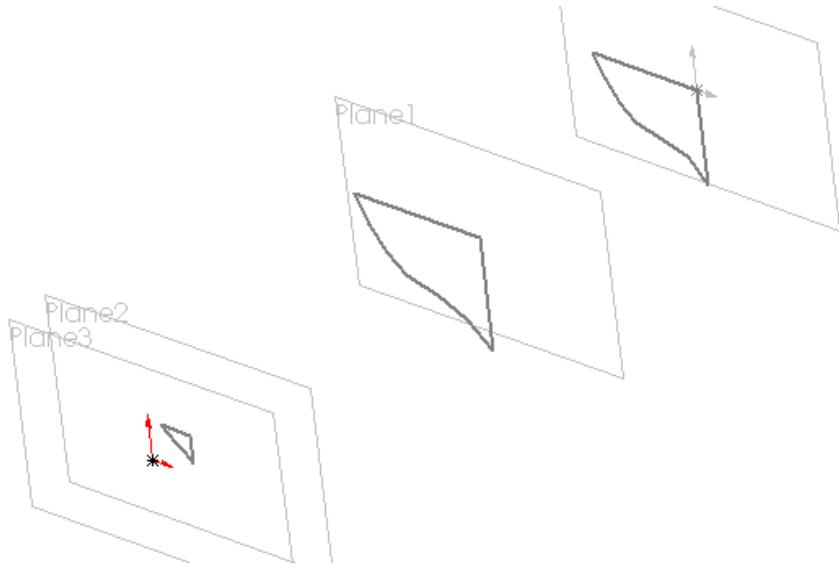
4. Sketch 3 on “Plane 2” should look like this...



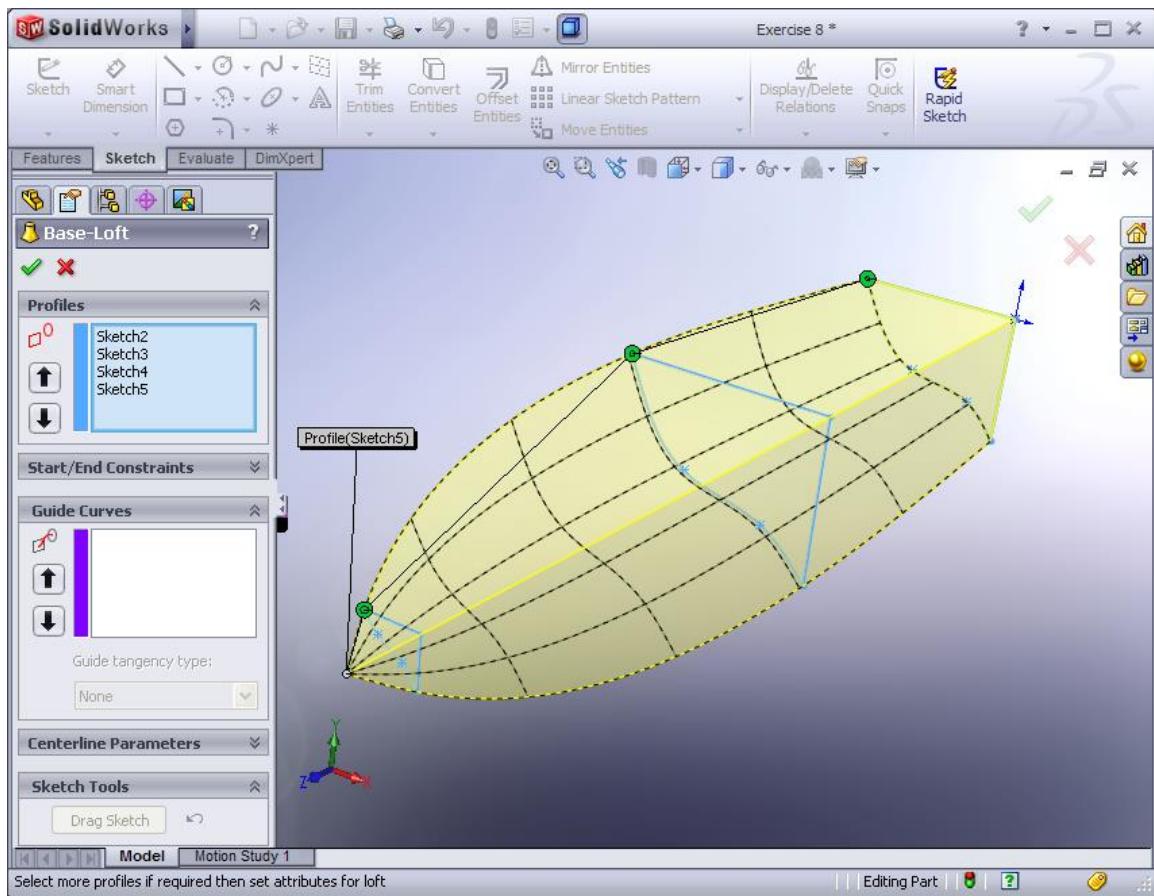
5. Sketch 4 on “Plane 3” should look like this... A single point at the origin.



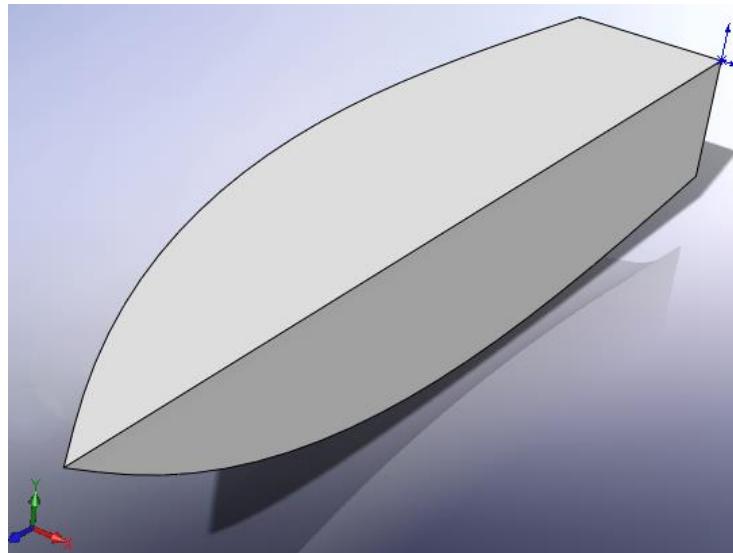
6. If you rotate the view it should appear like this...



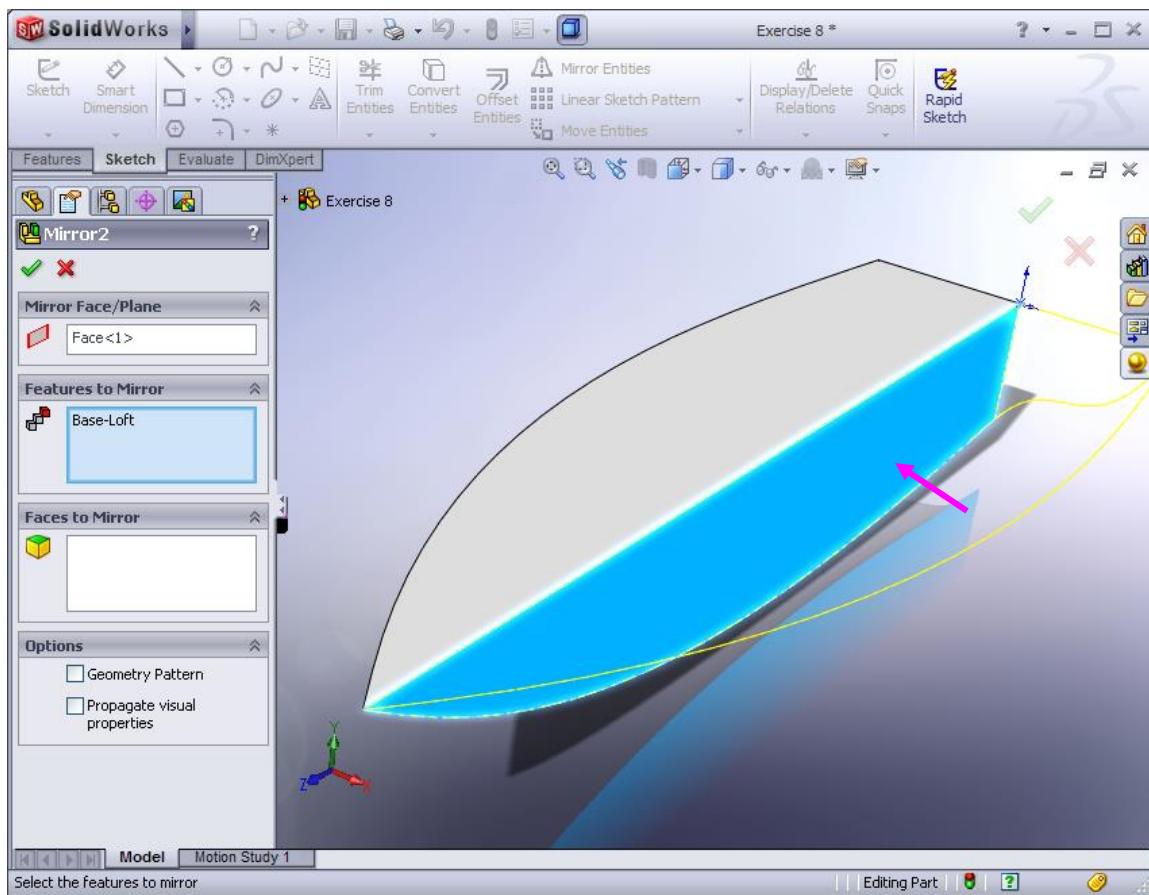
7. Select the Loft icon and begin to select the top left corner of each profile.



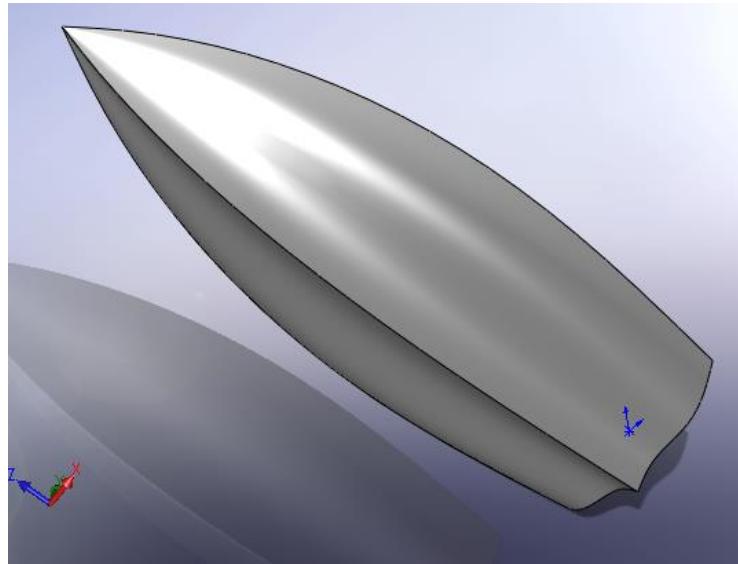
8. You should have $\frac{1}{2}$ a boat now...



9. Use the Mirror-Body feature and select the flat side face as the plane to mirror from.

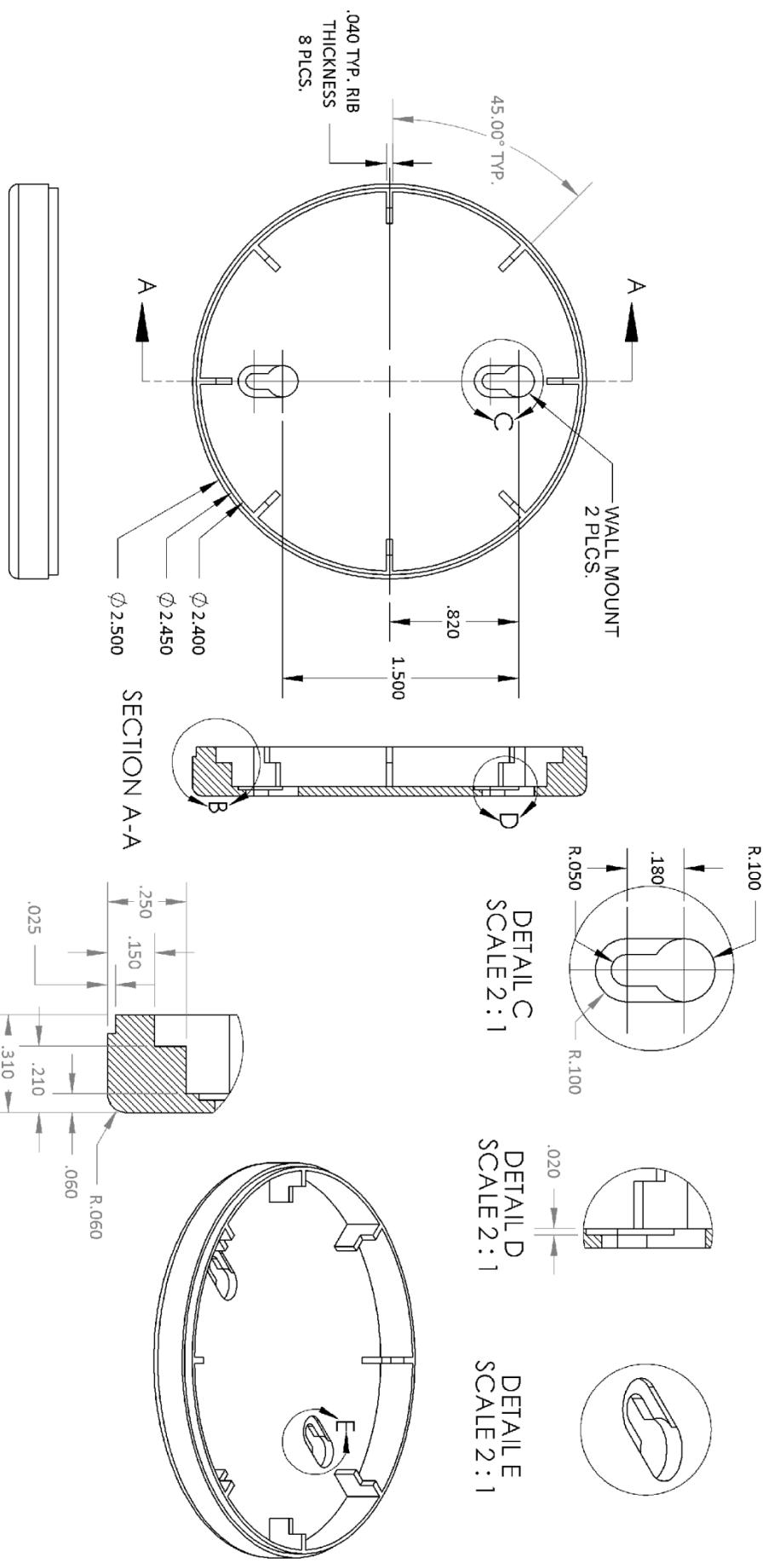


10. You are finished with the boat Hull.



11. (Optional) Now, dress it up for the contest...



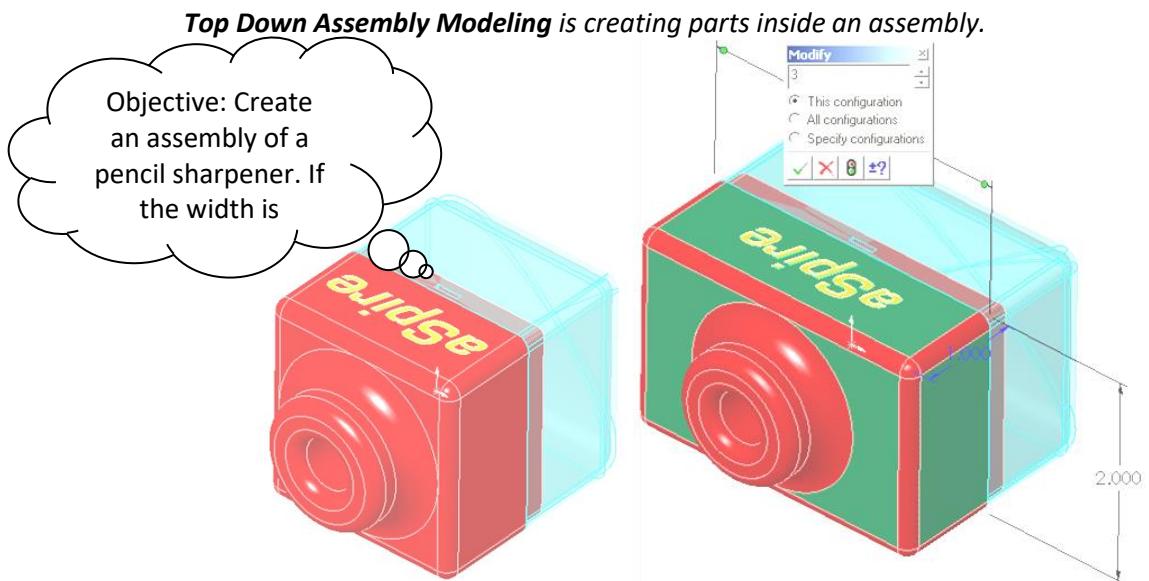


		UNLESS OTHERWISE SPECIFIED:	
		NAME	DATE
DIMENSIONS ARE IN INCHES		DRAWN	
TOLERANCES:		CHECKED	
FRACTIONAL [±]		ENG. APPR.	
ANGULAR: MACH [±]		MFG. APPR.	
BEND [±]			
TWO PLACE DECIMAL [±]			
THREE PLACE DECIMAL [±]			
INTERPRET GEOMETRIC TOLERANCING PER:		Q.A.	
MATERIAL		COMMENTS:	
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NEXT ASSY	USED ON		
	FINISH		
APPLICATION		DO NOT SCALE DRAWING	
SCALE: 1:1		SIZE	DWG. NO.
		A	L8 SMOKE DETECTOR
			REV
			REAR BEZEL
		SHEET 1 OF 1	

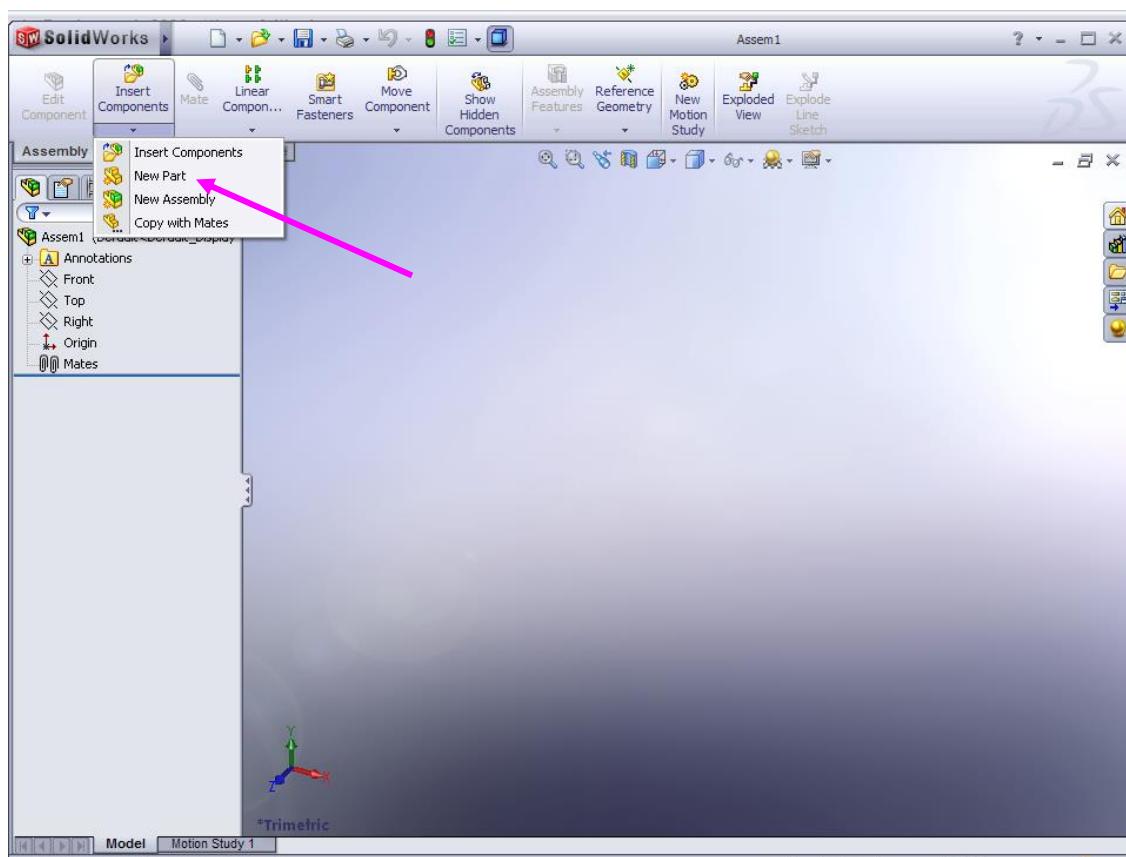
 Elgin Community College
Bright Choice. Bright Future.

EXERCISE 9

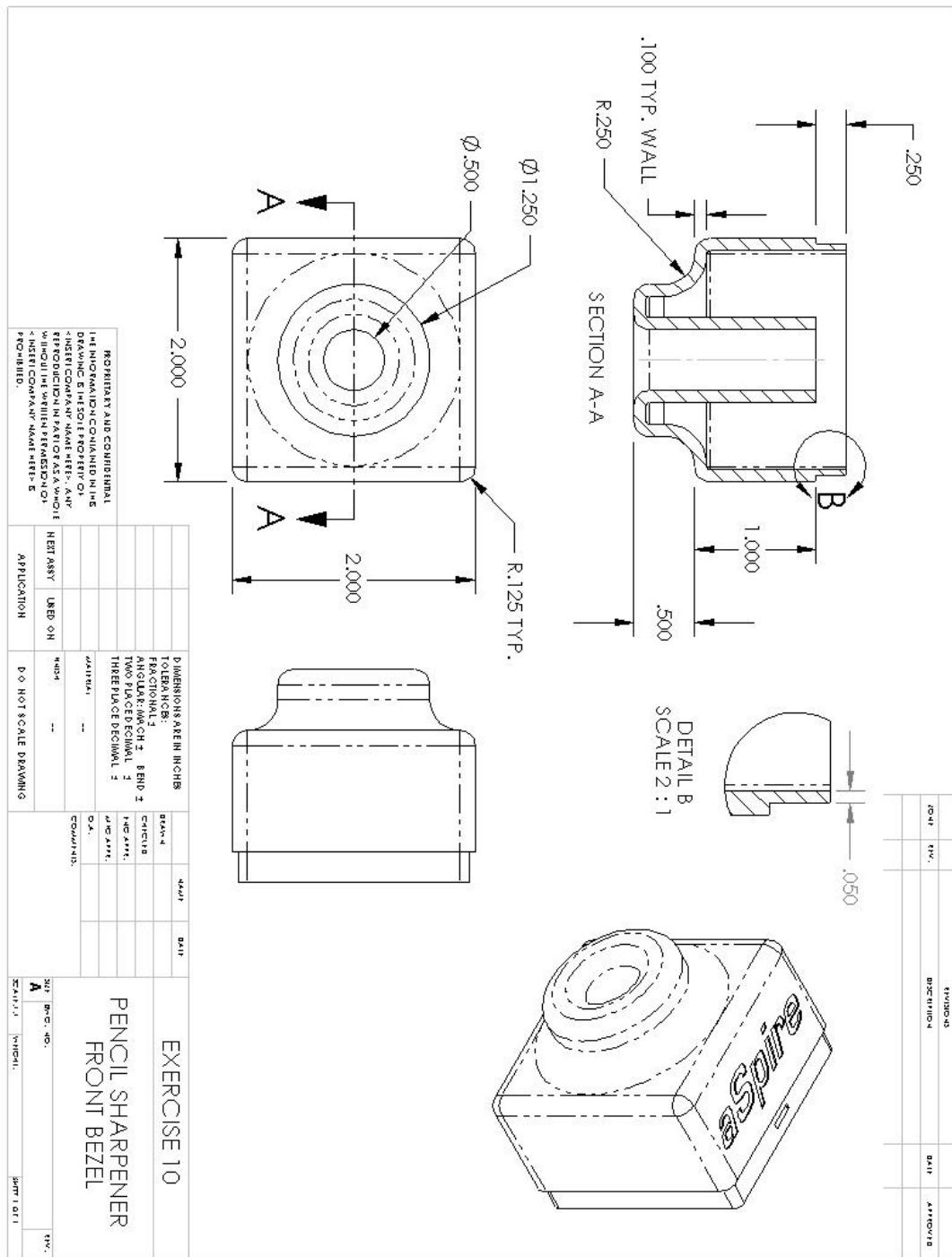
Top-Down Assembly Modeling



1. Create a new assembly file. Save it as E9.
2. Go to insert/components and select new part.



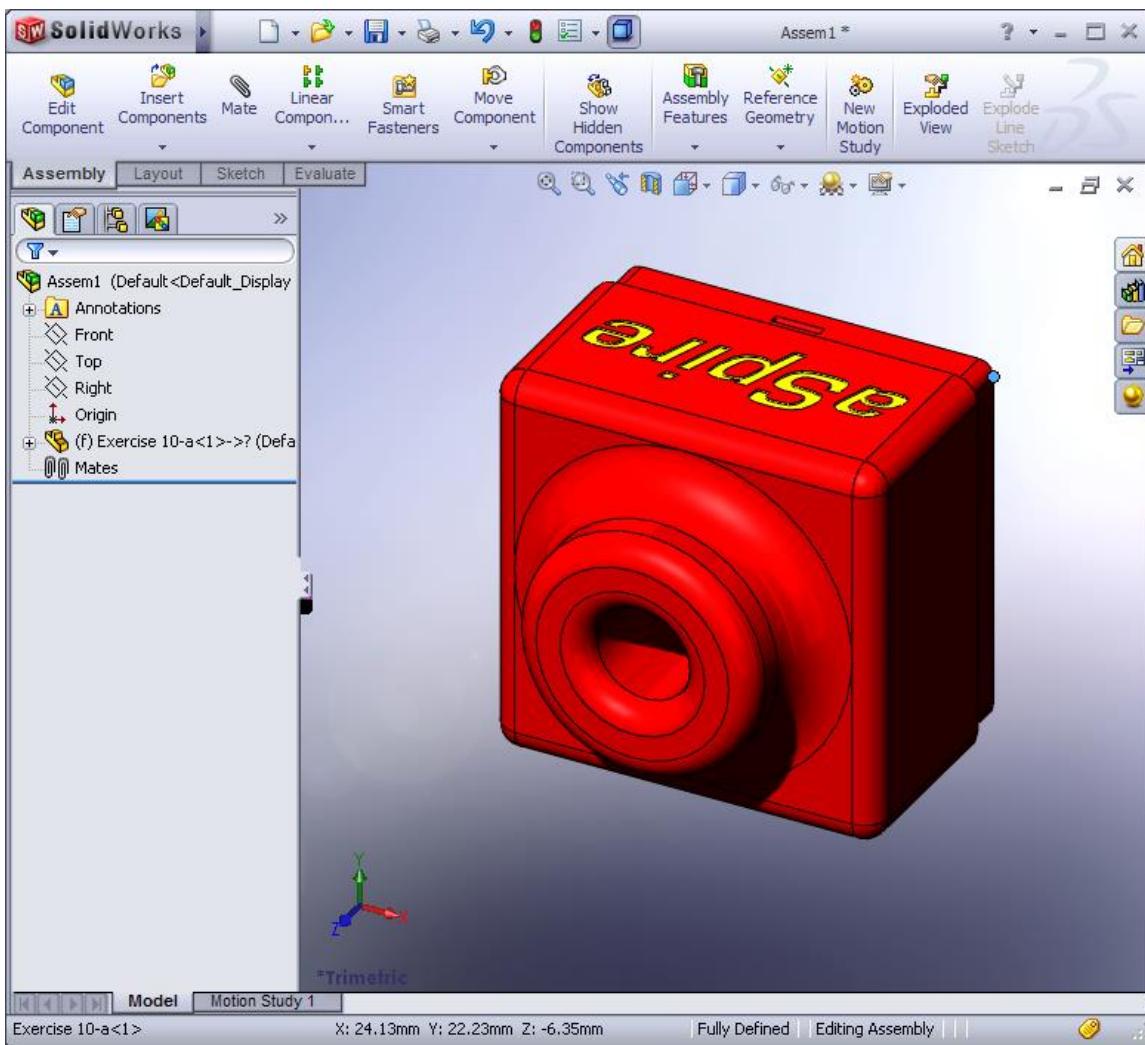
3. Save it as E9 Front and drop it on the “Front” plane. Create the following part from the drawing.



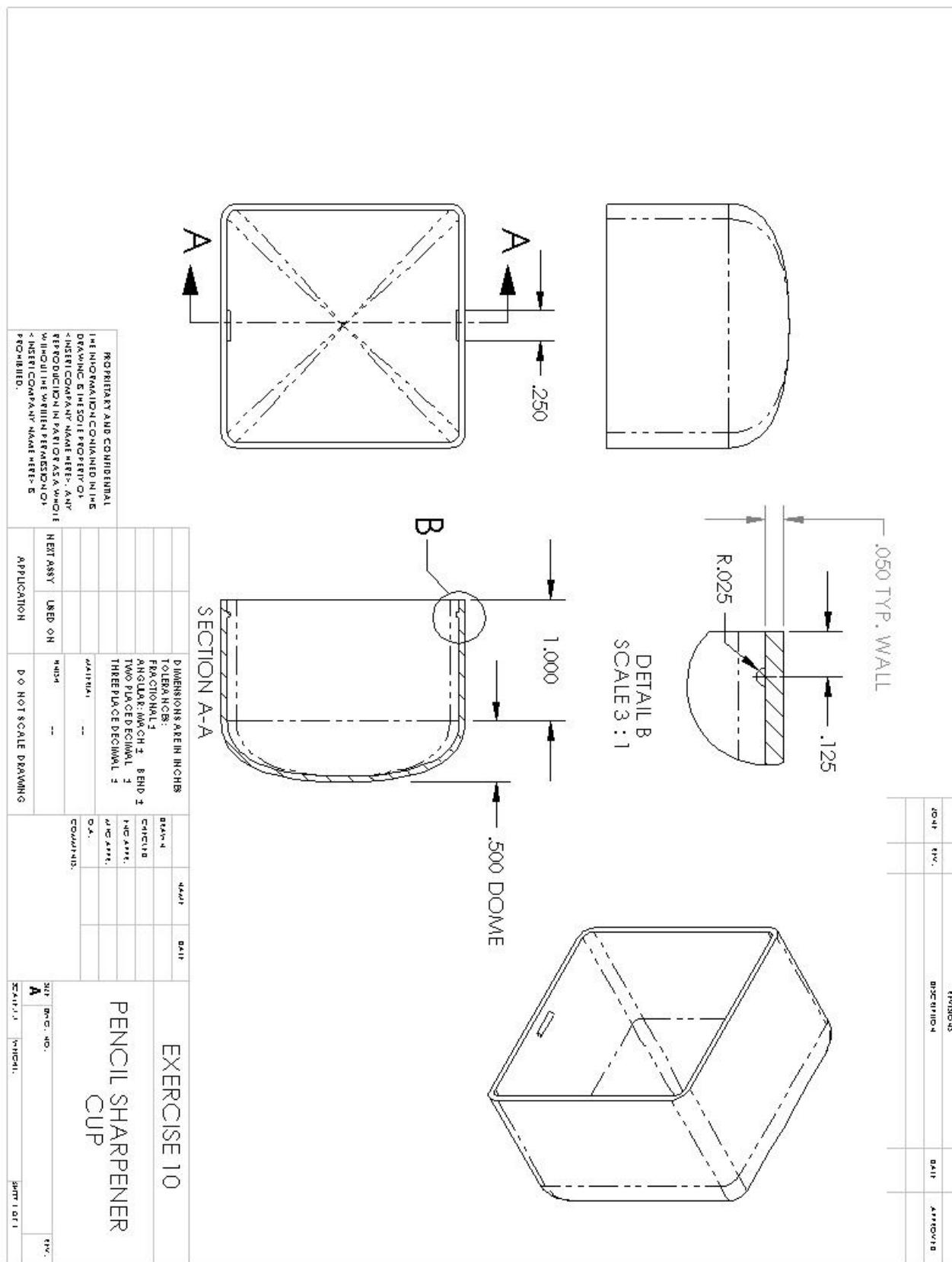
4. When finished select the **Edit Component** icon *to exit part editing mode.*

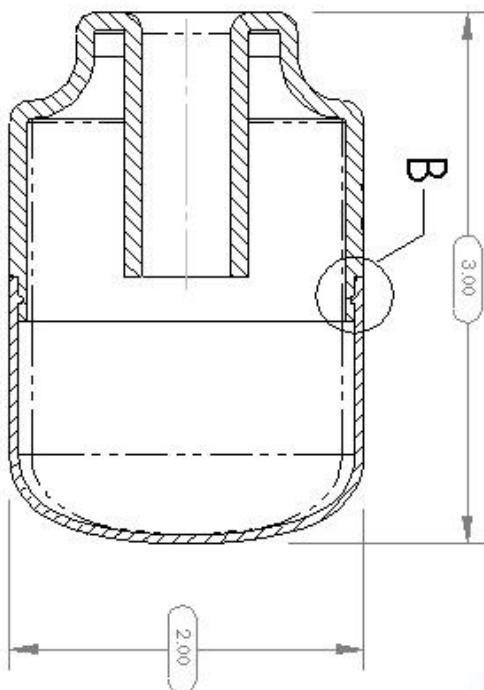
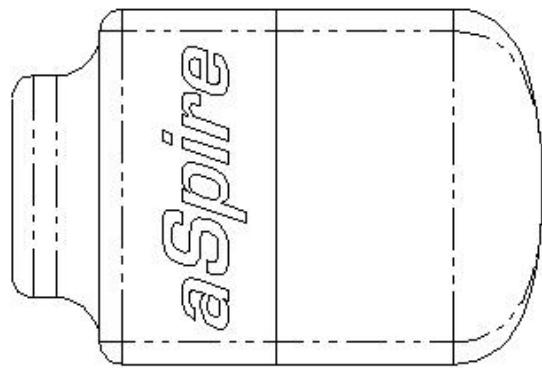
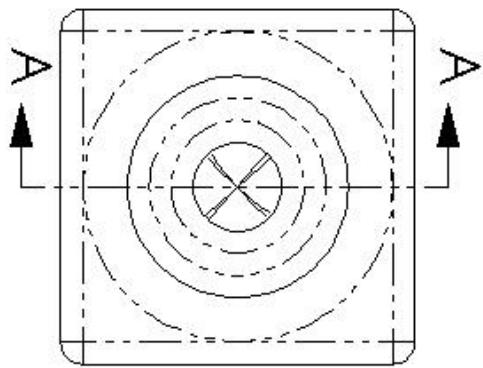


5. Insert another new component and save it as E9 Reservoir.

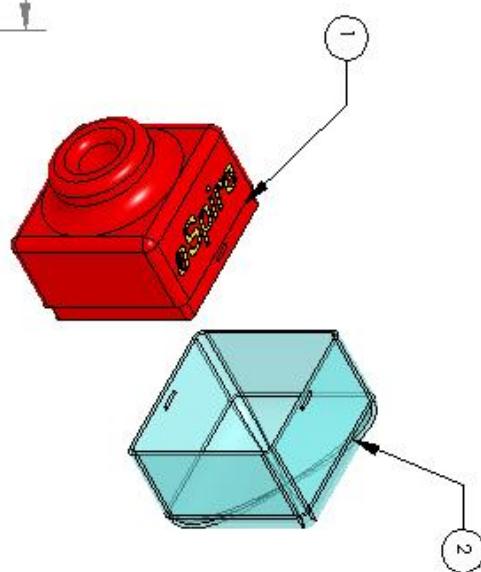


6. Create the following model in the context of the assembly-using offset or convert entities from the E9 Front model.





DETAIL B
SCALE 2 : 1



INSTRUCTIONS: RECREATE THIS DRAWING

ITEM NO.	QTY.	PART NO.	DESCRIPTION
1	1	Berdice 10-a	
2	1	E10 CUP	

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INKED FOR USE DRAFTING			
APPLICATION	USED IN	REVISION	DATE
HEX ASSY	4-4134	--	

WILSON APP.

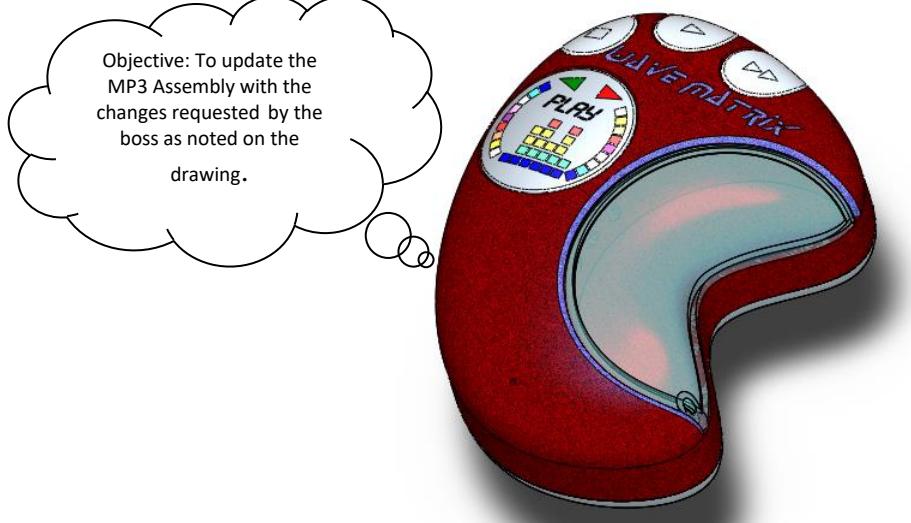
J

6

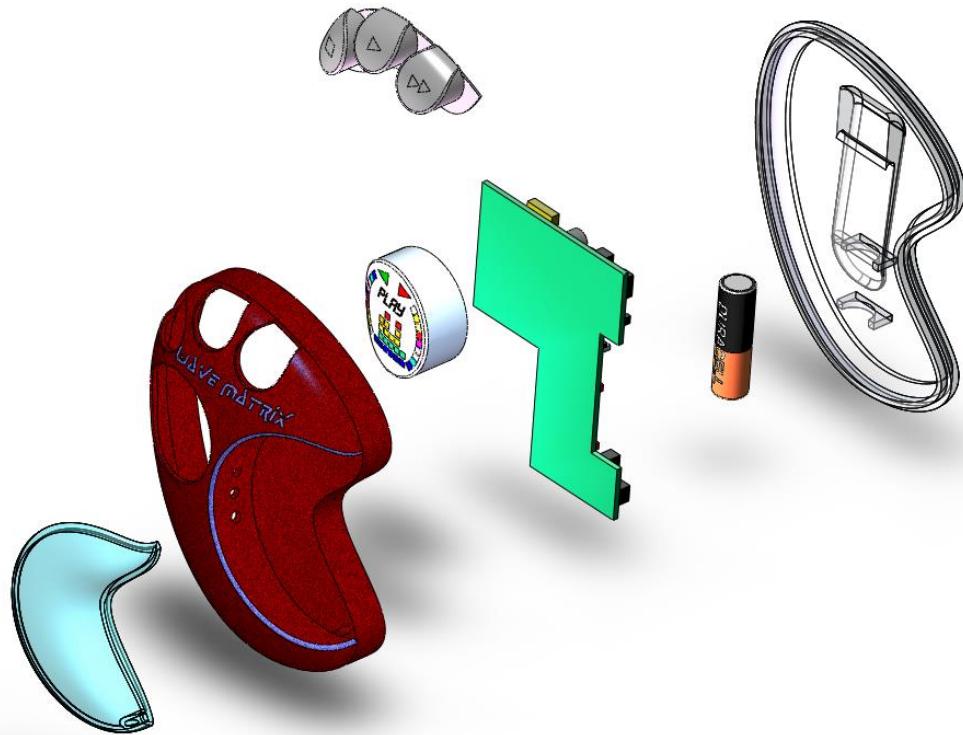
EXERCISE 10

Assembly Editing

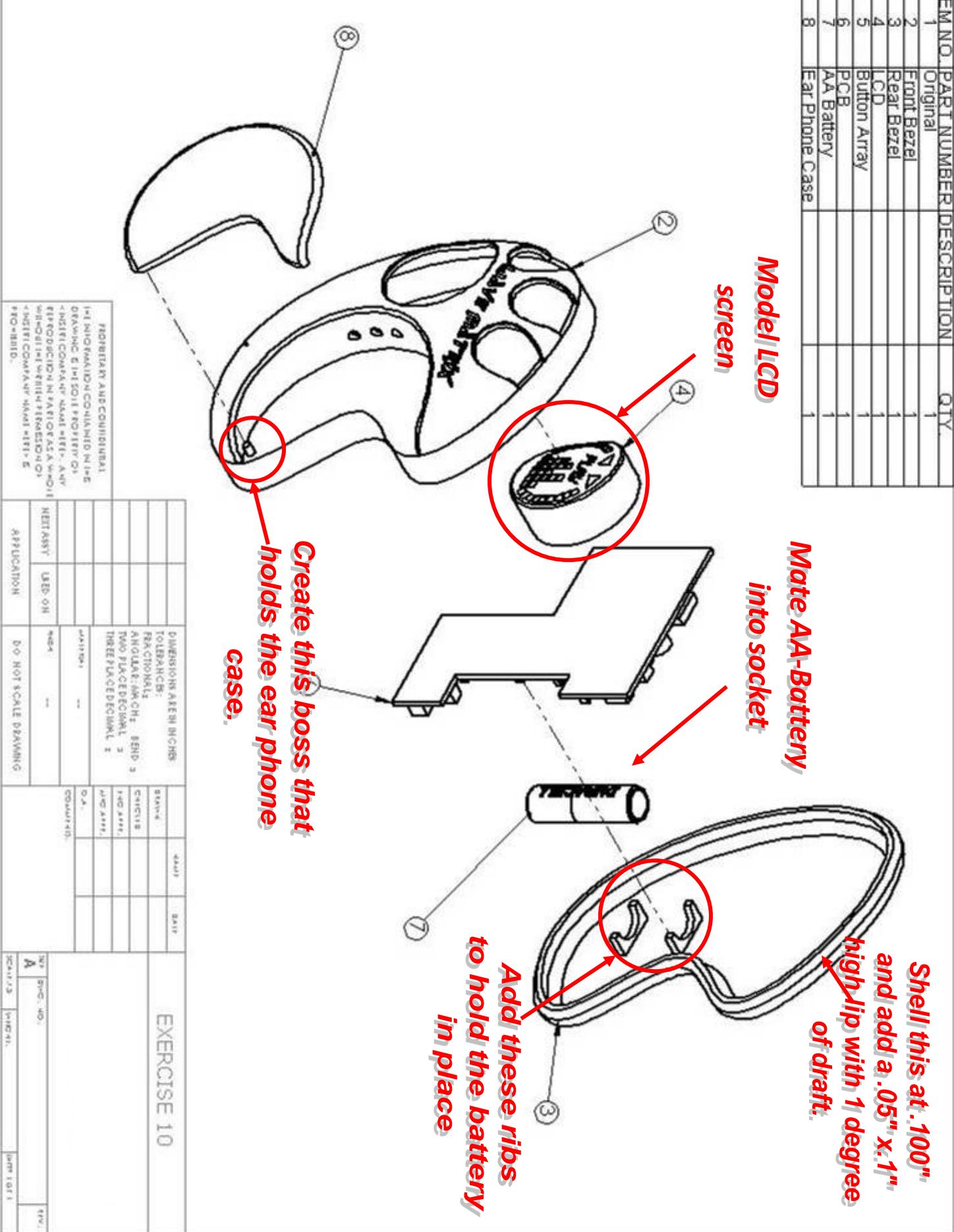
This exercise will include both Bottom-Up and Top-Down Assembly Modeling.

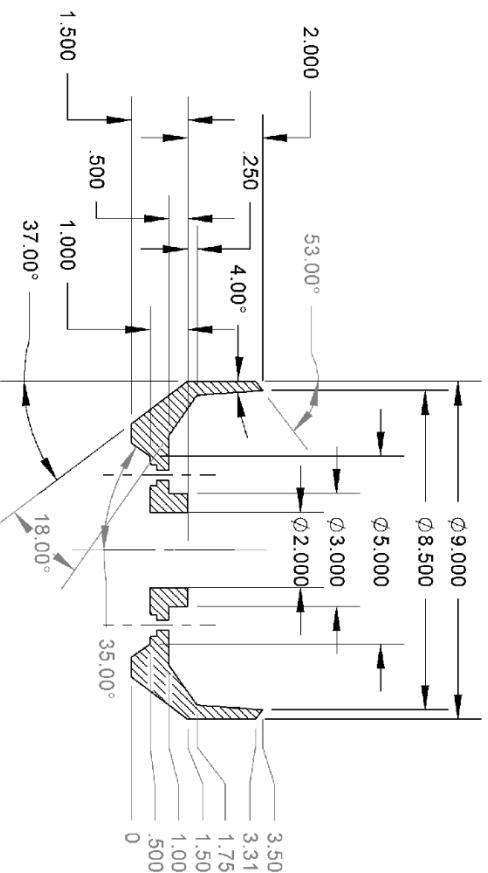


1. Open the E10 assembly file and modify according to the instructions noted on the drawing provided. You will have to mate the Battery part file.

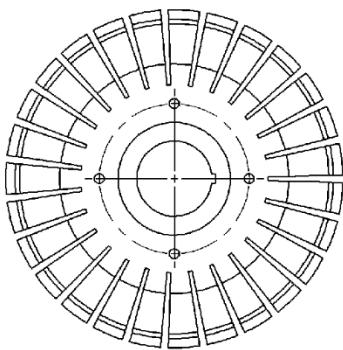
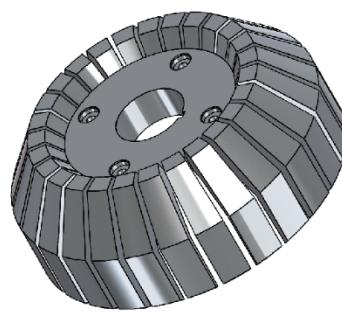
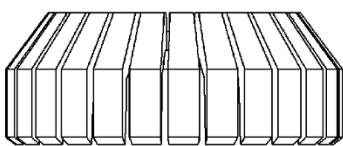
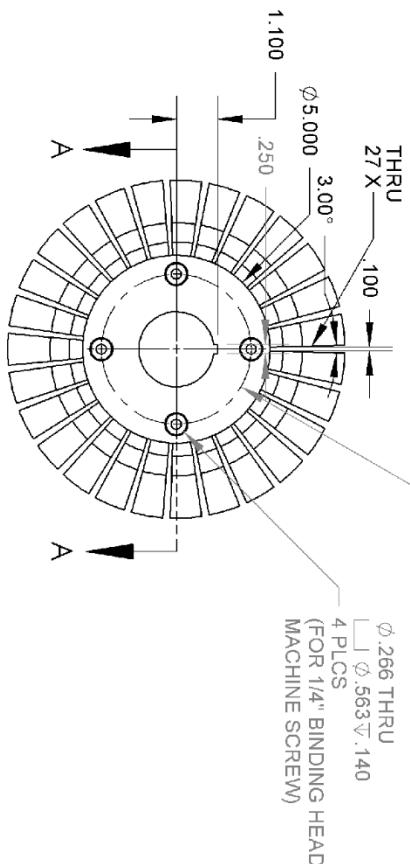


ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Original		1
2	Front Bezel		1
3	Rear Bezel		1
4	LCD		1
5	Button Array		1
6	PCB		1
7	AA Battery		1
8	Ear Phone Case		1





SECTION A-A



Quantum Aviation

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INTERPRET GEOMETRIC TOLERANCING PER:		Q.A.
MATERIAL		COMMENTS:
FINISH		
APPLICATION	DO NOT SCALE DRAWING	

A SIZE DWG. NO.
L10TURBINE FILTER

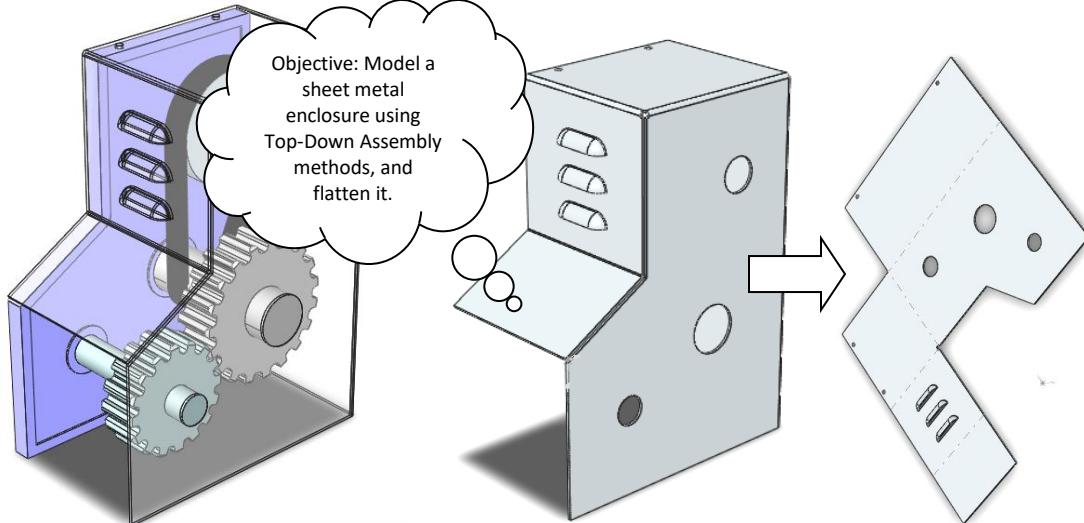
REV

SCALE: 1:5 WEIGHT: SHEET 1 OF 1

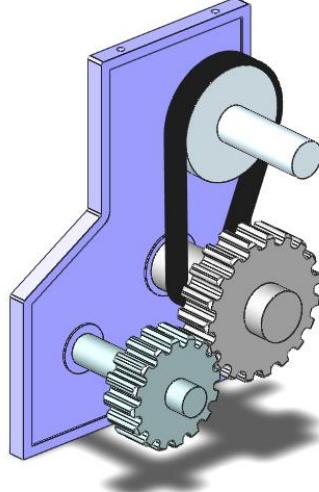
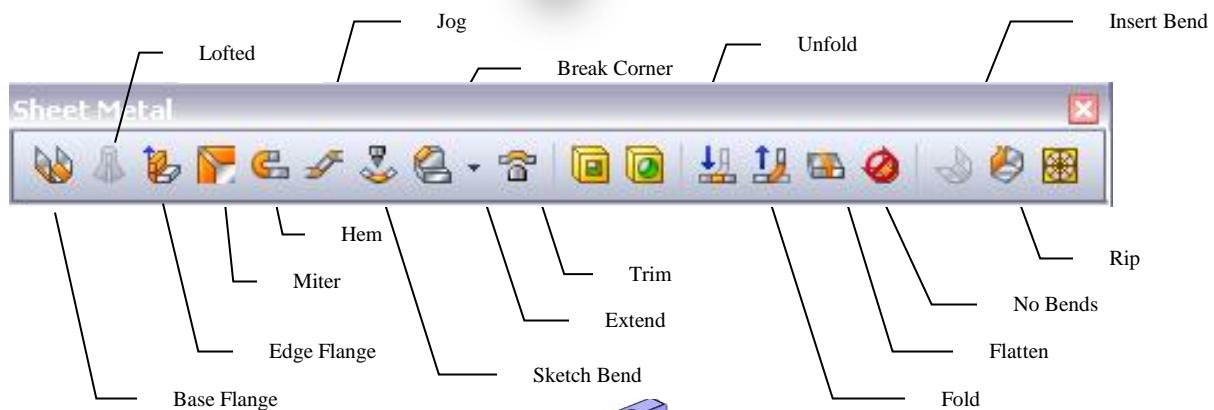
EXERCISE 11

Sheet Metal Design

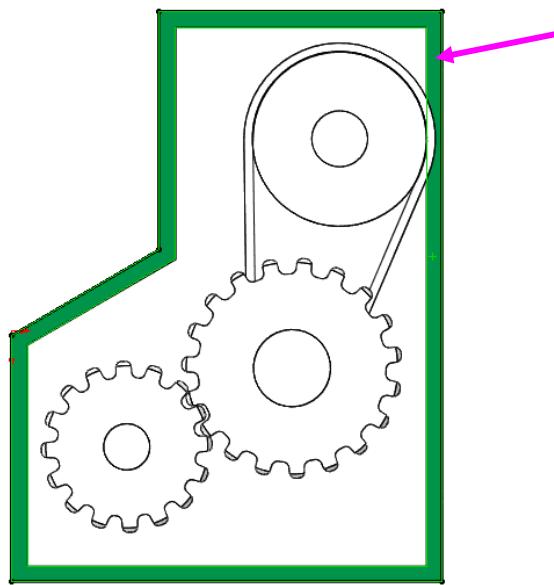
Sheet Metal part files can be very useful for extracting a flat pattern.



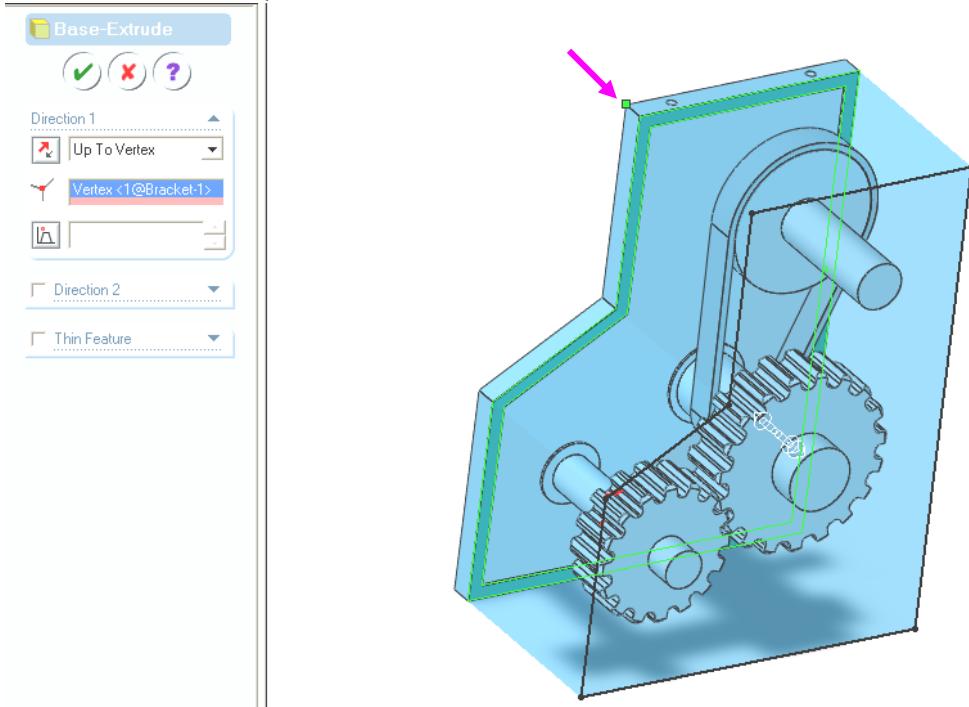
2. Open the Exercise 11.sldasm and hide the cover (RMB click on the cover and select the Hide icon (eye glasses)).



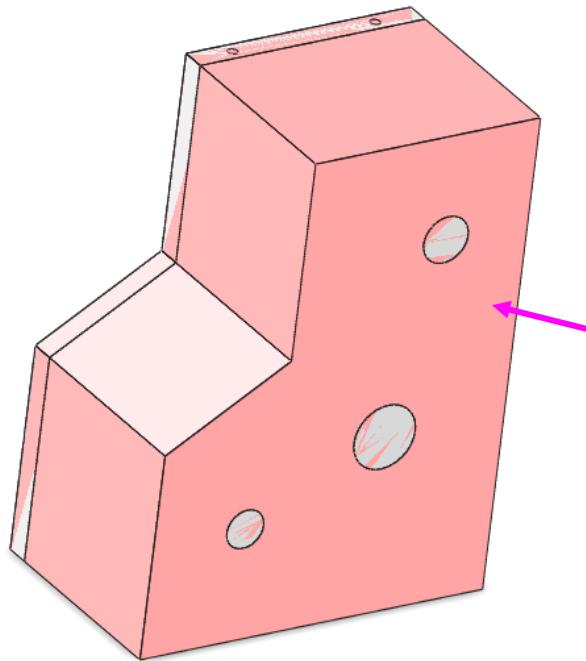
3. Insert a new part into the assembly; drop it on the “Front” plane of the assembly. Name it “E11 Cover” (This will be the enclosure) then select the front outside face. Convert Entities.



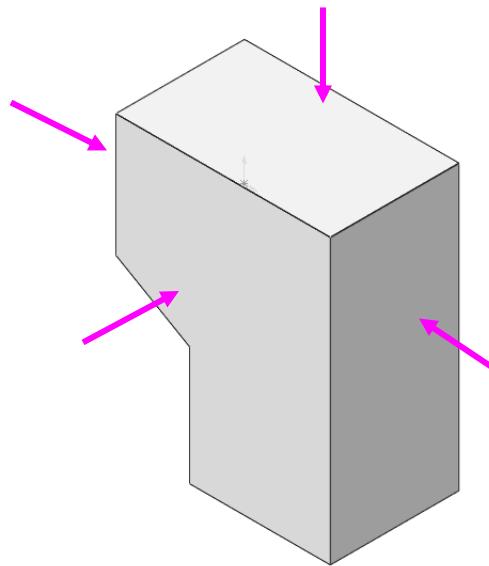
4. Extrude up to vertex.

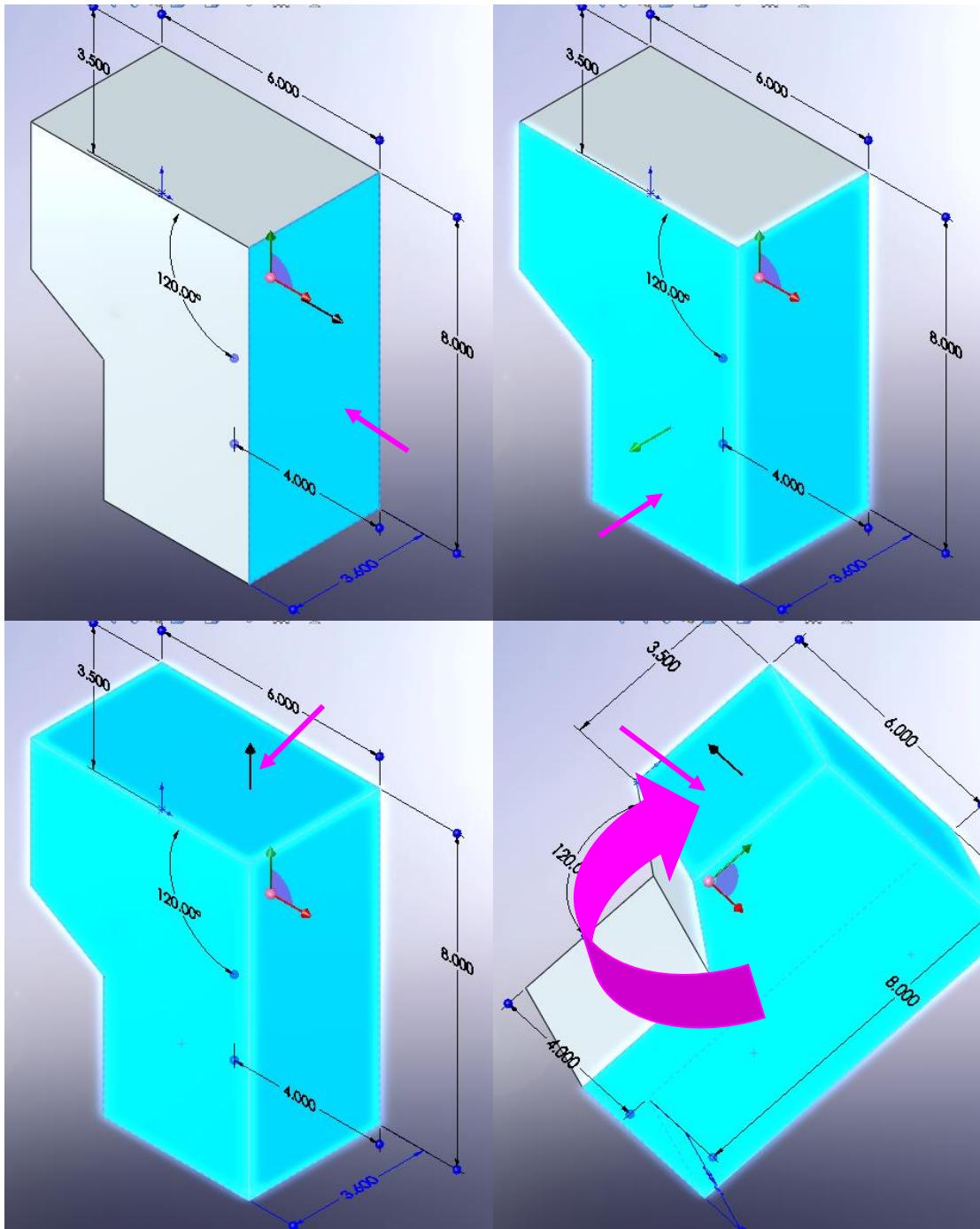


- Once open the assembly should look like this. Right Mouse click on the surface of the enclosure and select “open E12.sldprt”.



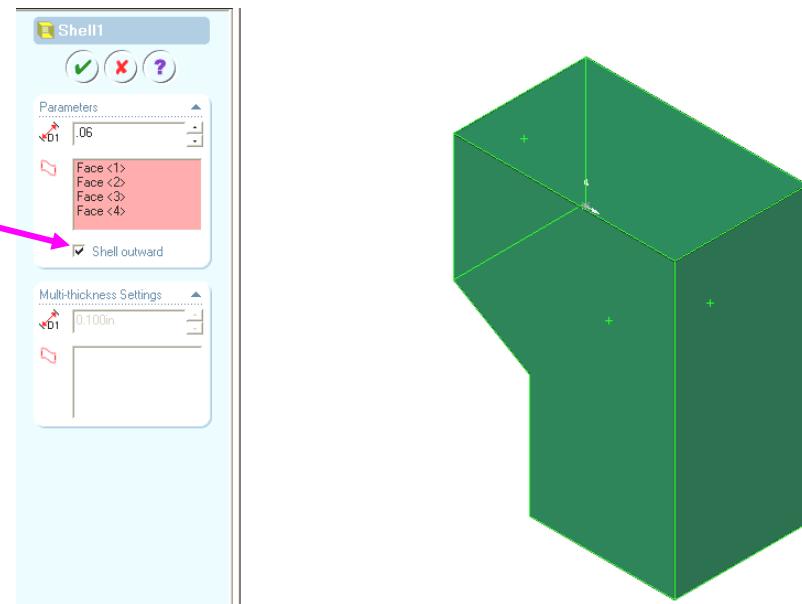
- Go to an isometric view and “ctrl” select the four faces as shown.



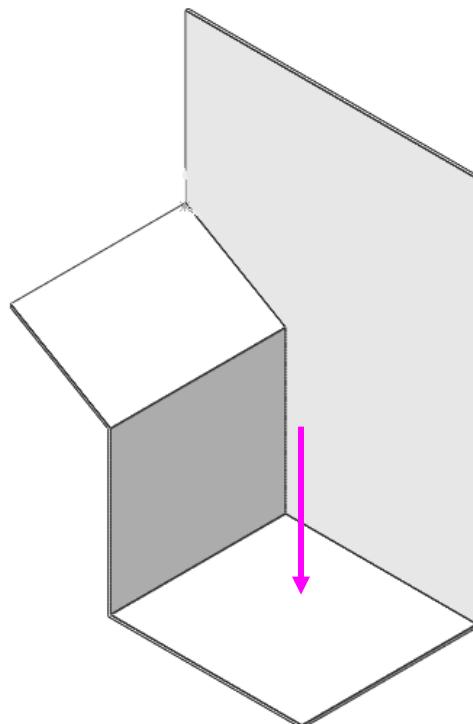


Rotate the view to select the fourth face.

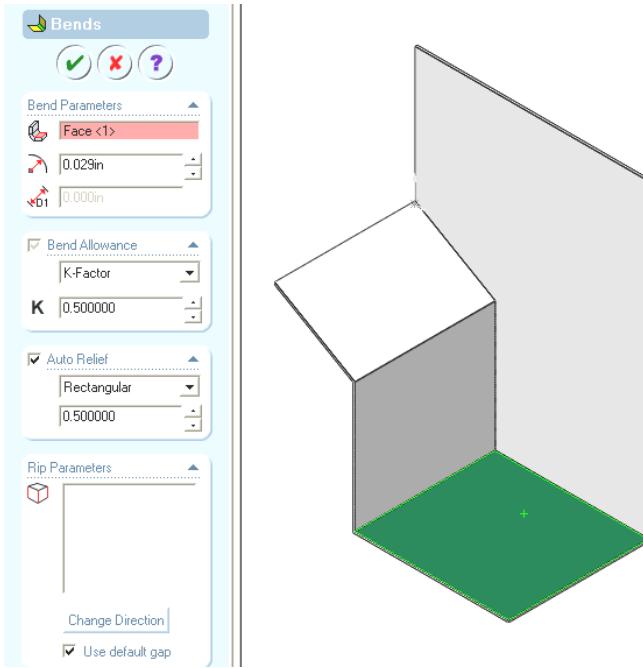
Select the shell command. Set it to .06", and select "Shell outward".



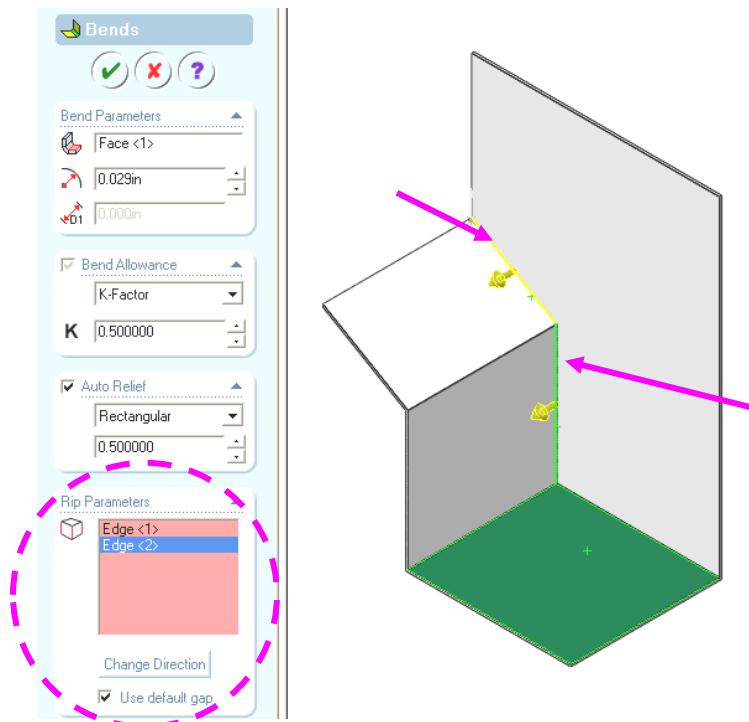
7. Select the bottom face and select the "insert bends" icon.



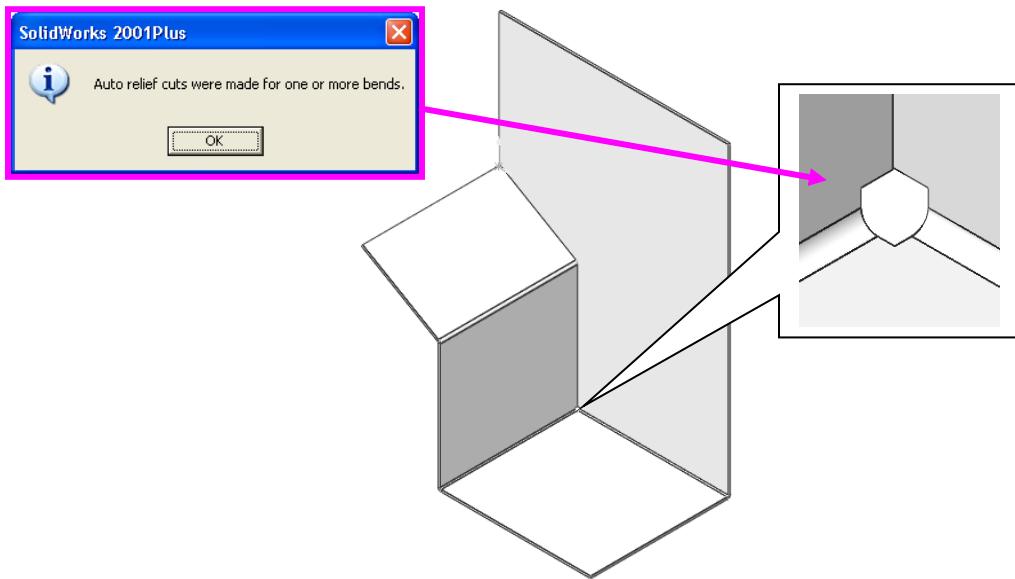
8. Go to the right view orientation and you should have this section view...



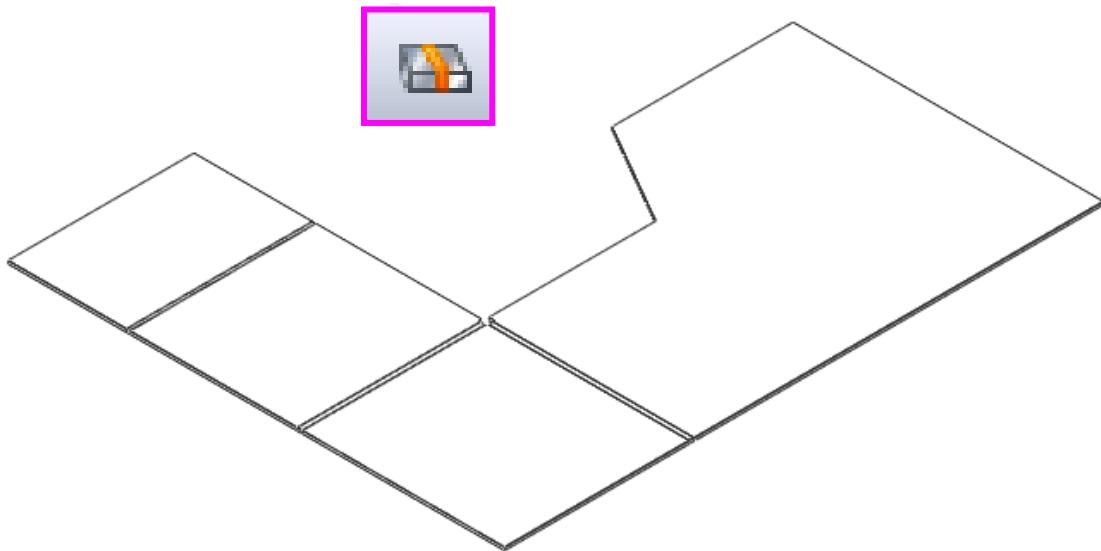
9. Click on the Rip parameters and select the two inside edges. Use the arrows to control rip face direction. Hit apply.



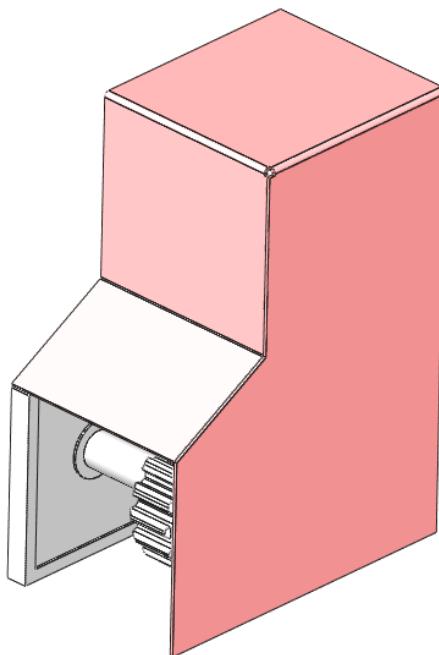
10. Right mouse button click on any portion of one of the slots and “select chain”. Hit the CTRL-C buttons to copy.



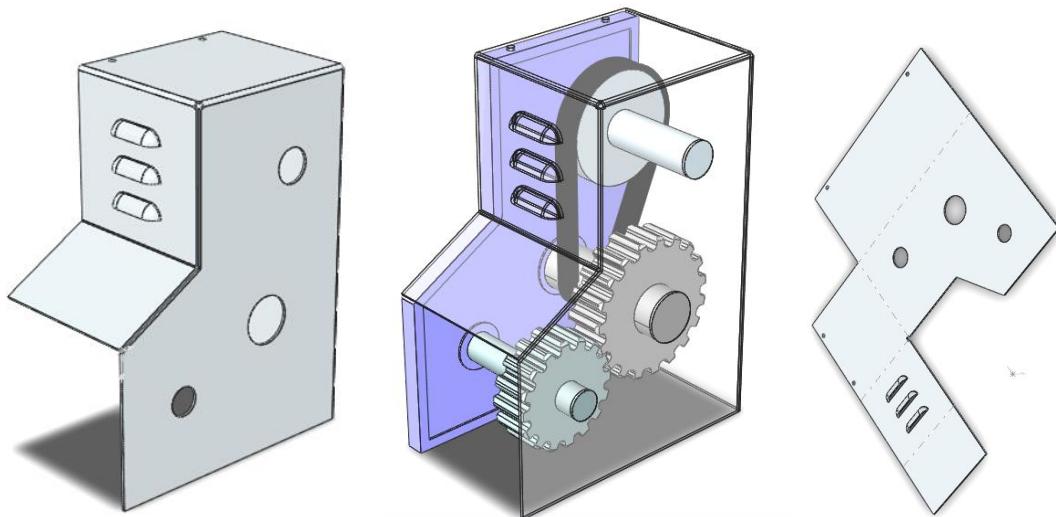
11. Select the flatten icon.



12. Return to the assembly.

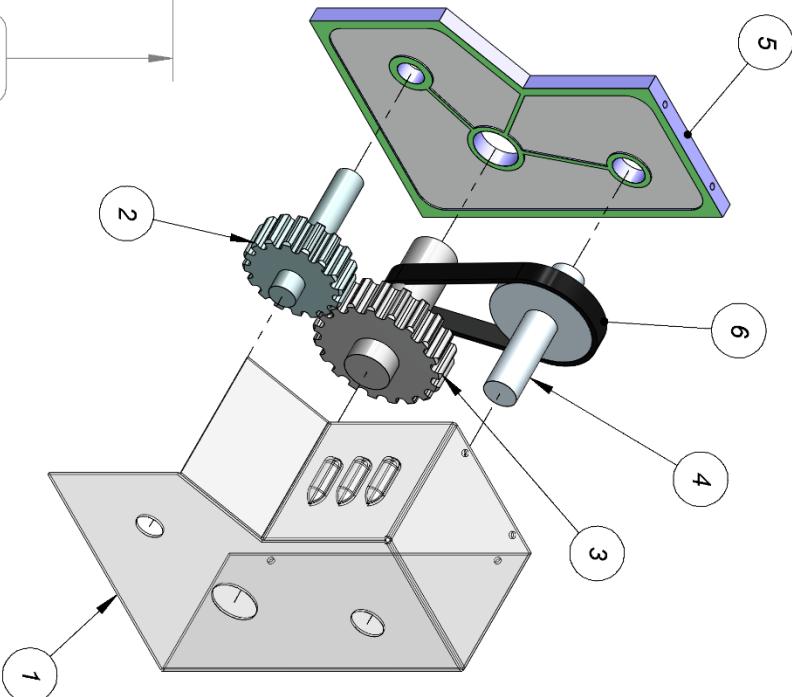
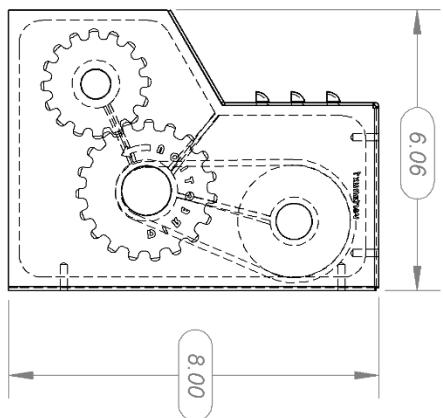
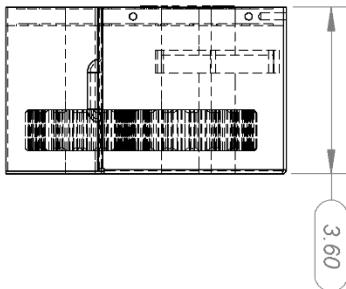
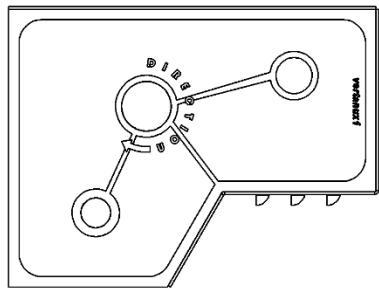


13. Add holes and additional features.



14. The enclosure is now completed. Recreate the attached drawing for your Lab 11.

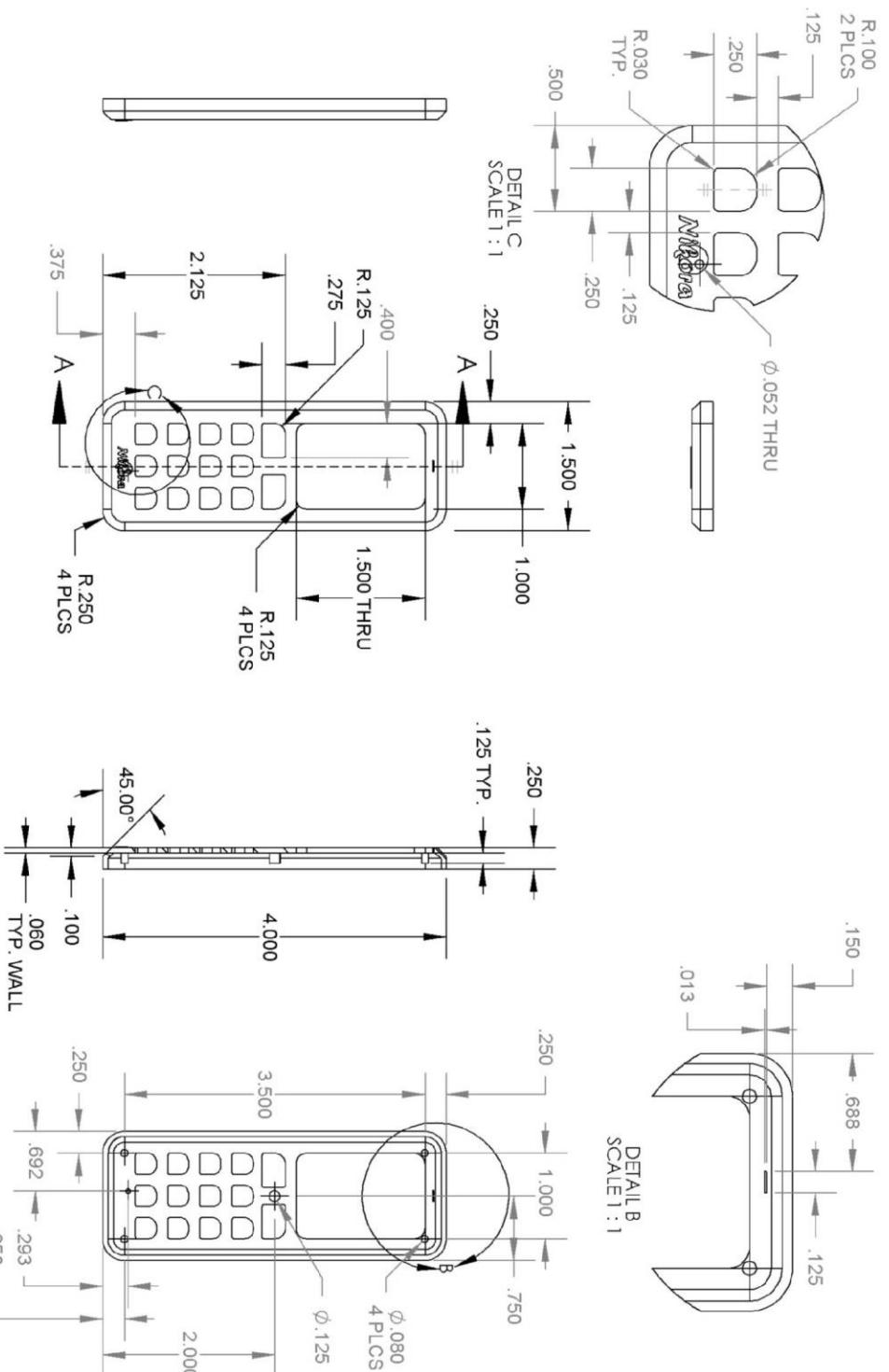
ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Cover		1
2	Gear 1		2
3	Gear 2		2
4	Pulley		1
5	Bracket		1
6	Belt		1



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DRAWN	NAME	DATE
CHECKED		
ENG APPR.		
MFG APPR.		
INTERPRET GEOMETRIC TOLERANCING PER:	COMMENTS:	
MATERIAL	Q.A.	
FINISH		
APPLICATION	DO NOT SCALE DRAWING	

A	SIZE DWG. NO. L11	SCALE: 1:4	WEIGHT:	SHEET 1 OF 1	REV
					UNITED STATES METAL FABRICATION USMF
INSTRUCTIONS: RECREATE THIS DRAWING					



106

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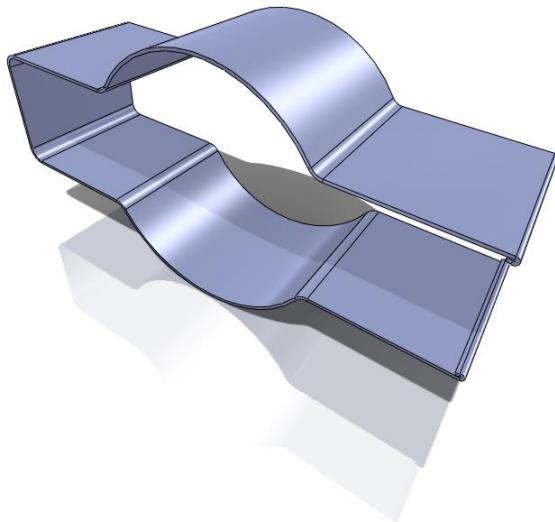
UNLESS OTHERWISE SPECIFIED:			
DIMENSIONS ARE IN INCHES		NAME _____ DATE _____	
TOLERANCES:		DRAWN _____	
FRACTIONAL [±]		CHECKED _____	
ANGULAR; MACH ⁺		ENG APPR. _____	
TWO PLACE DECIMAL [±]		MFG APPR. _____	
THREE PLACE DECIMAL [±]		Q.A. _____	
INTERPRET GEOMETRIC TOLERANCING PER: MATERIAL		COMMENTS: _____	
NEXT ASSY	USED ON	SIZE DWG. NO. REV	
APPLICATION	FINISH	A	
DO NOT SCALE DRAWING		SCALE: 1:2 WEIGHT: SHEET 1 OF 1	

L10b

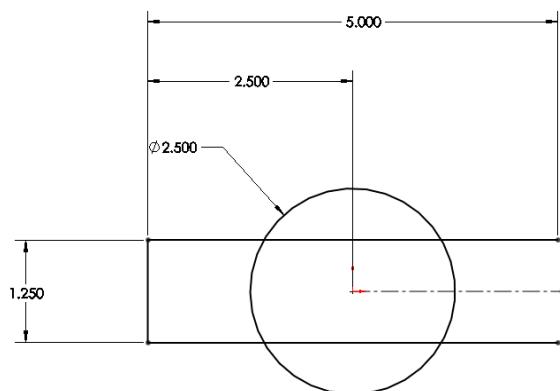
Nifora
TECHNOLOGIES

EXERCISE 11B

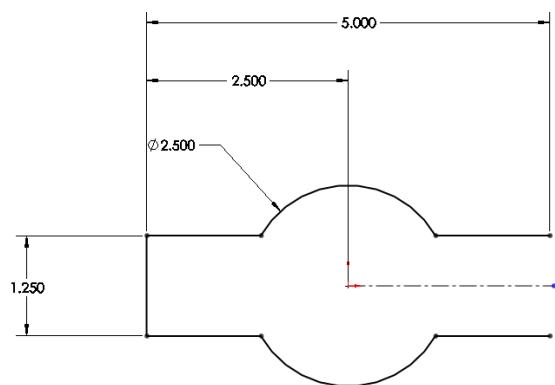
Alternatives, using base flange sheet metal tool



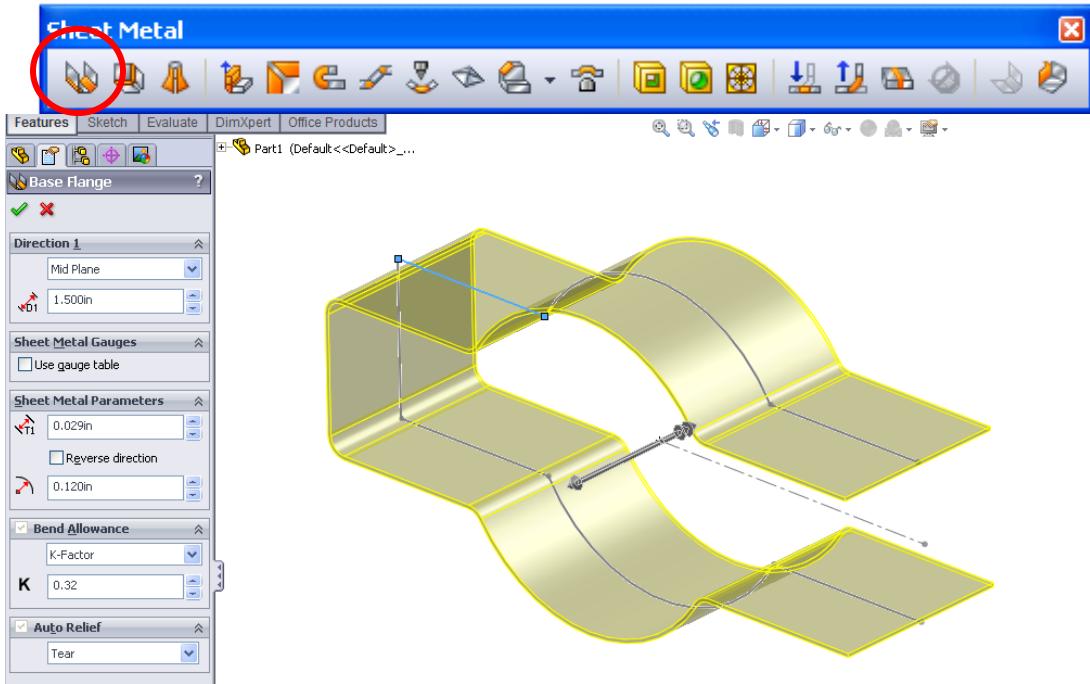
1. Draw the following sketch.



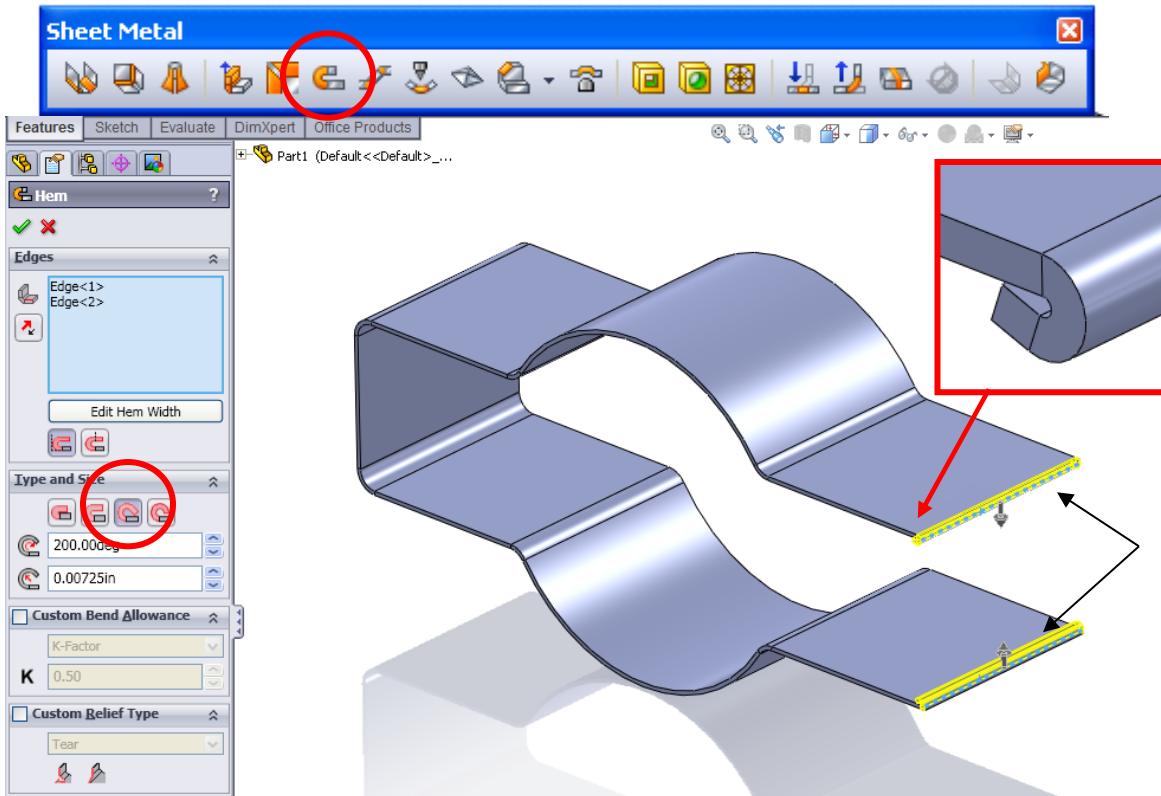
2. Trim the inside.



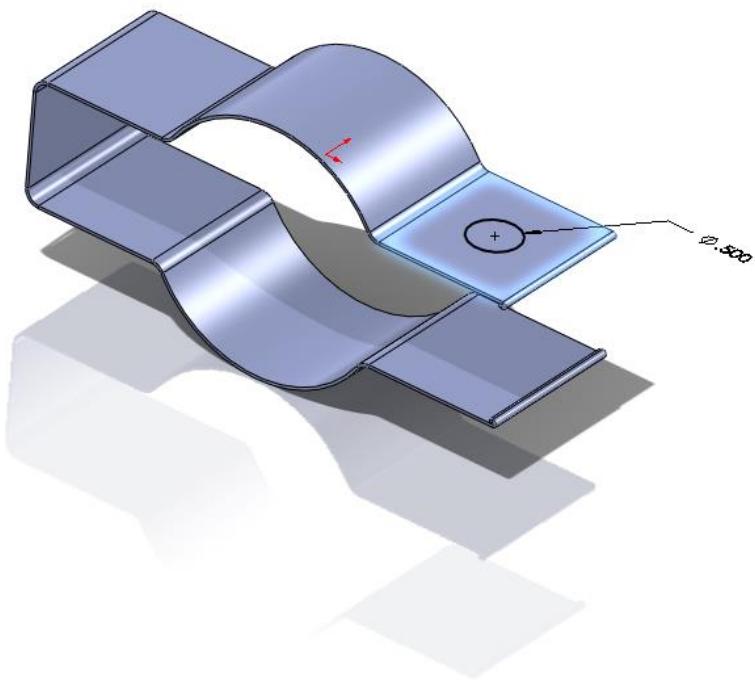
3. Use the base flange tool.



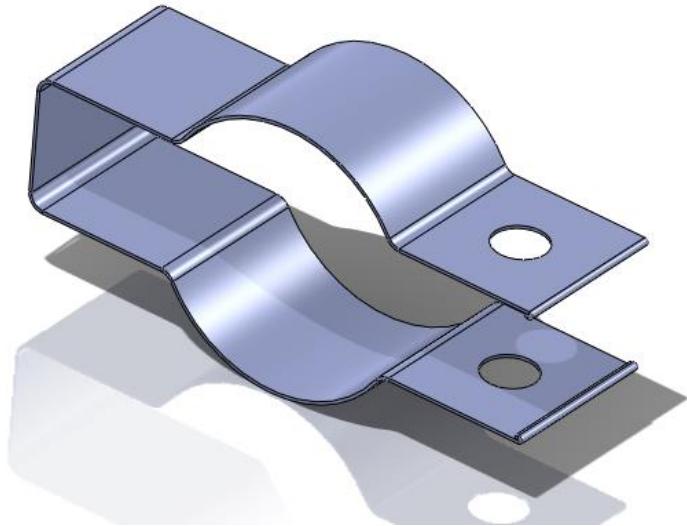
4. Insert a hem feature on both front edges.



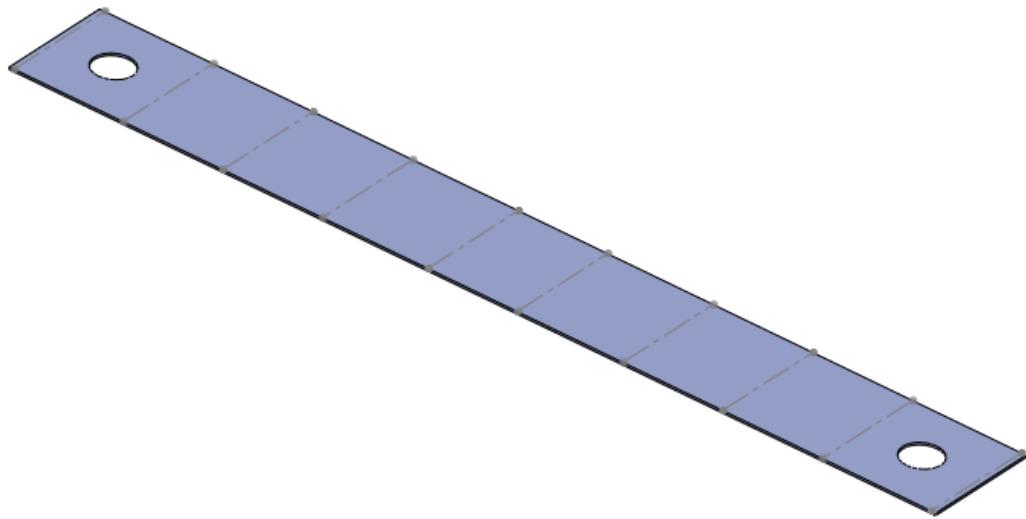
5. Sketch a .5" circle on the top flange.



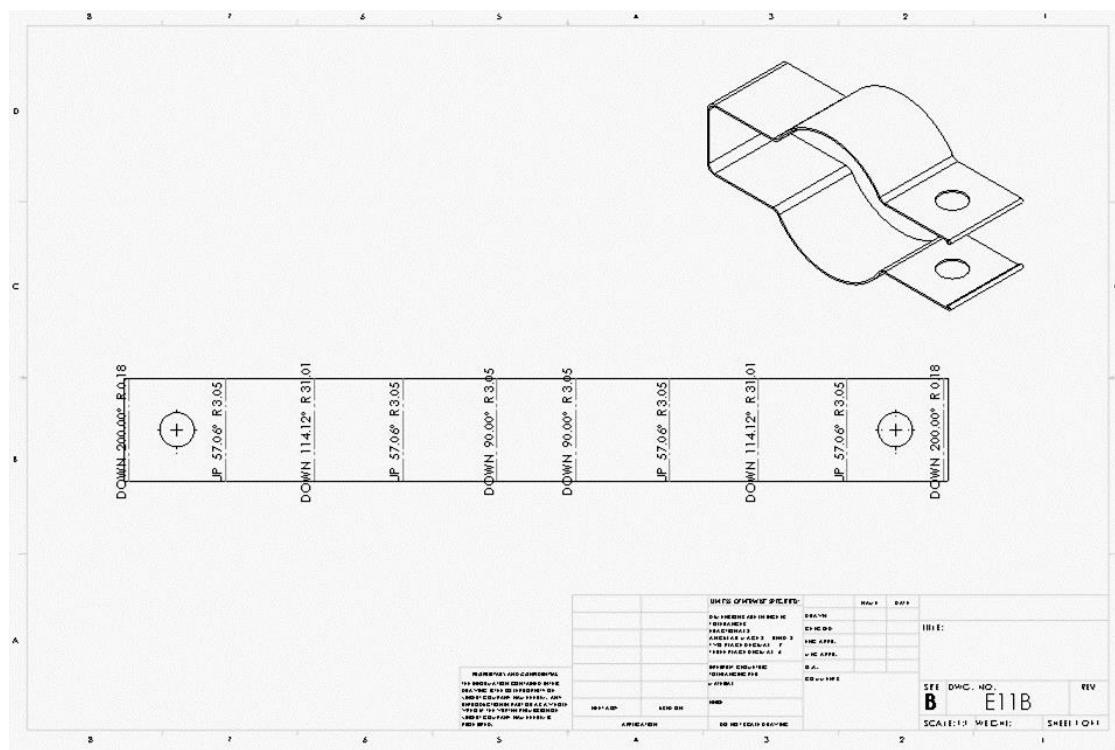
6. Extrude cut through-all.



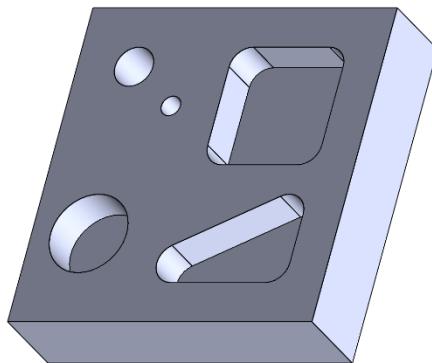
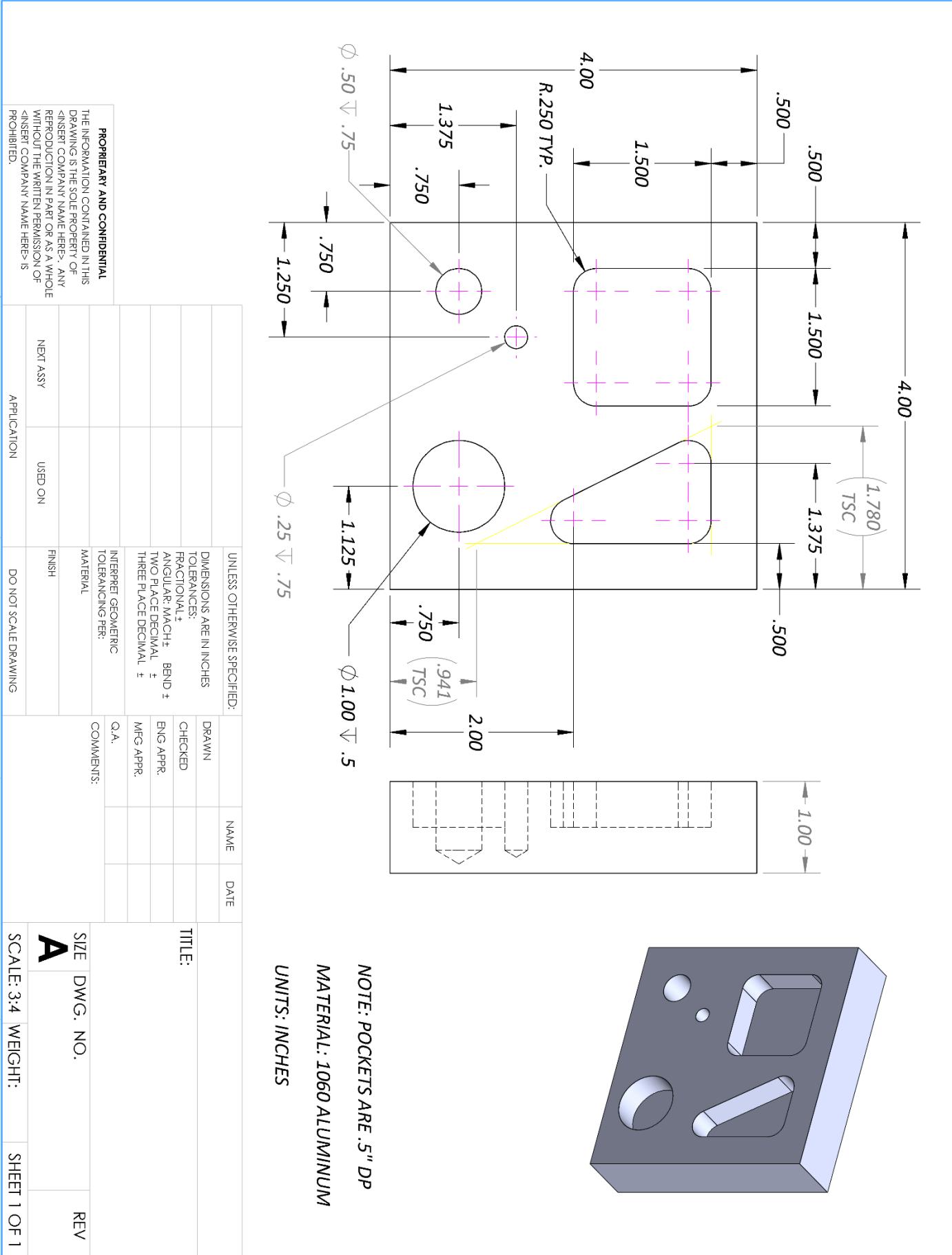
7. Flatten to verify. Refold.



8. Make a drawing from the part. Bring in the “Flat-Pattern” view. Rotate 90°.



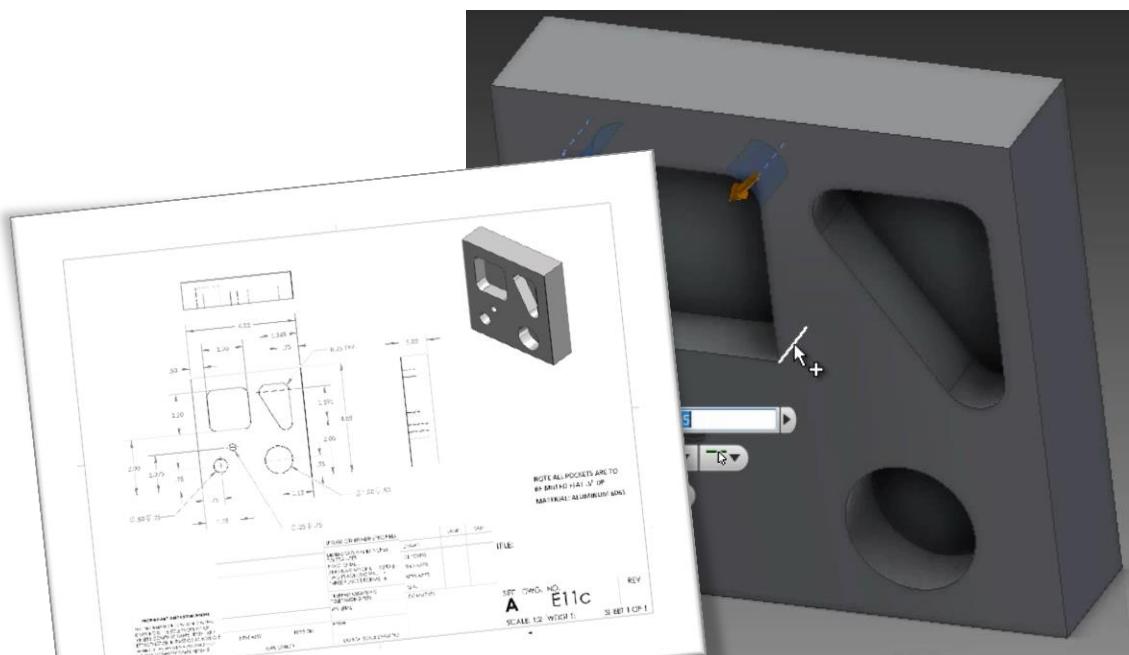
FINISHED



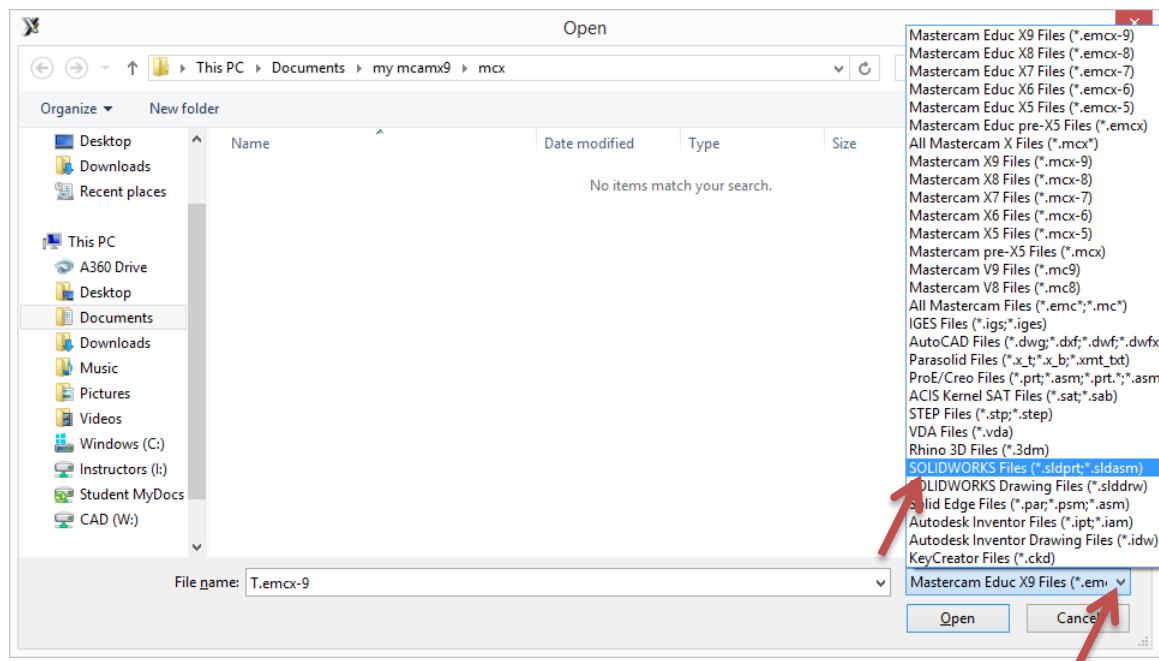
EXERCISE 11C

Computer Aided Manufacturing (CAD/CAM) with Mastercam and SolidWorks files.

1. Model the drawing on pg. 113

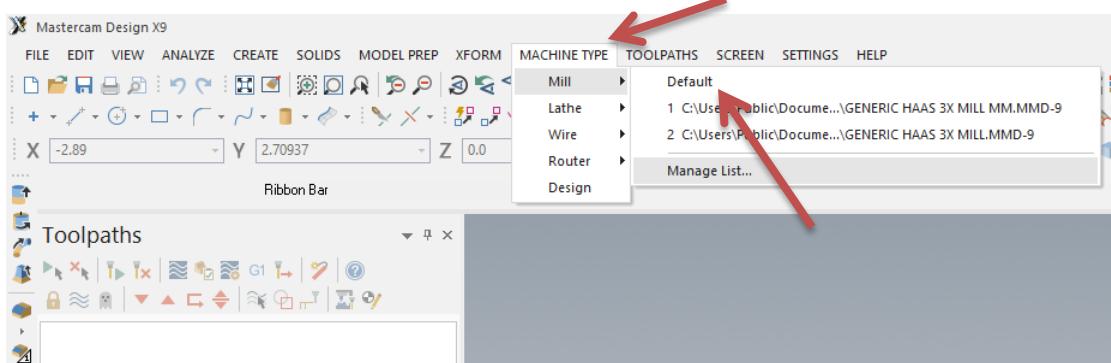


2. Save the part as E11c.
3. Start Matercam and import the SolidWorks E11c.sldprt. be sure to set your “Files of type” filter to SolidWorks part files.

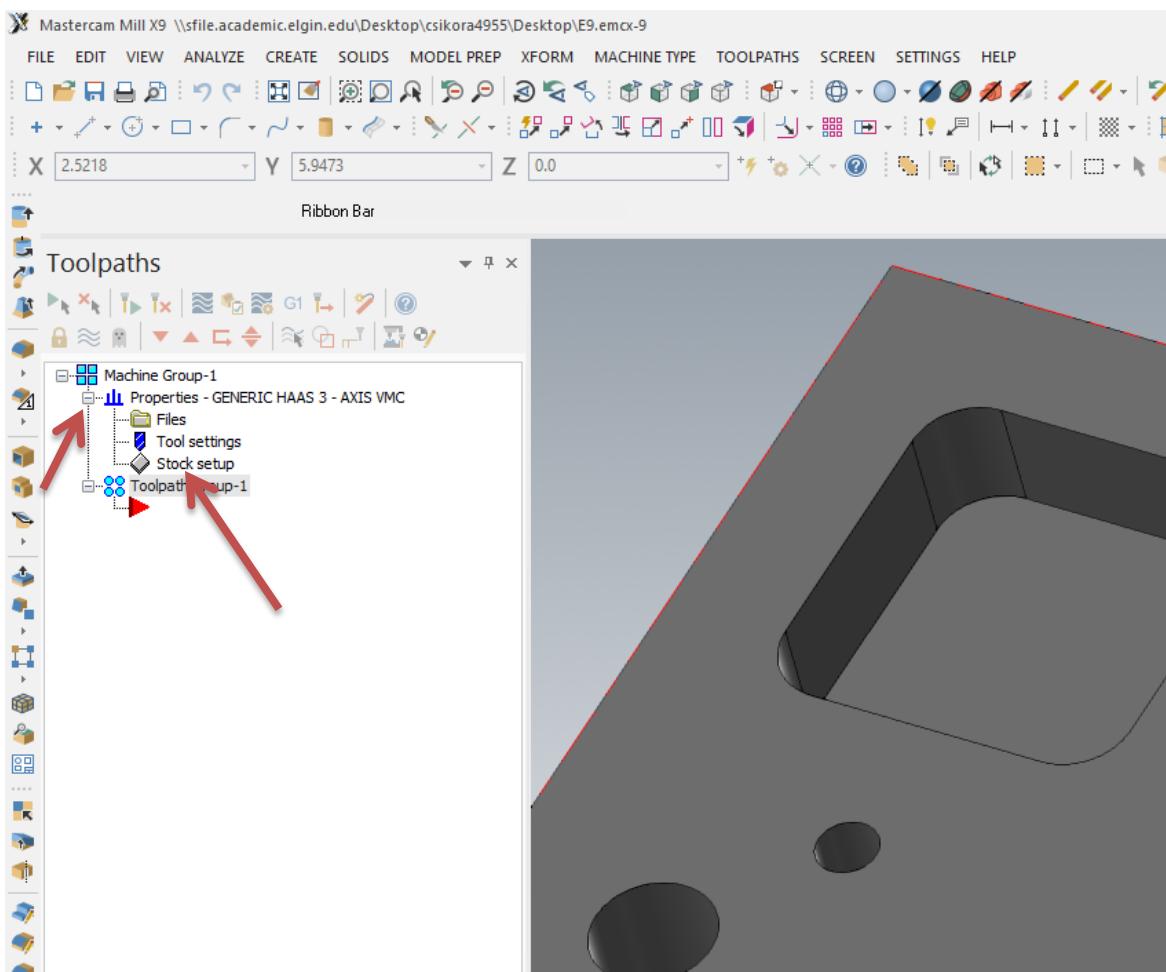


4. Select the Machine type (AKS: Post or Postprocessor)/ Mill/ “Default”

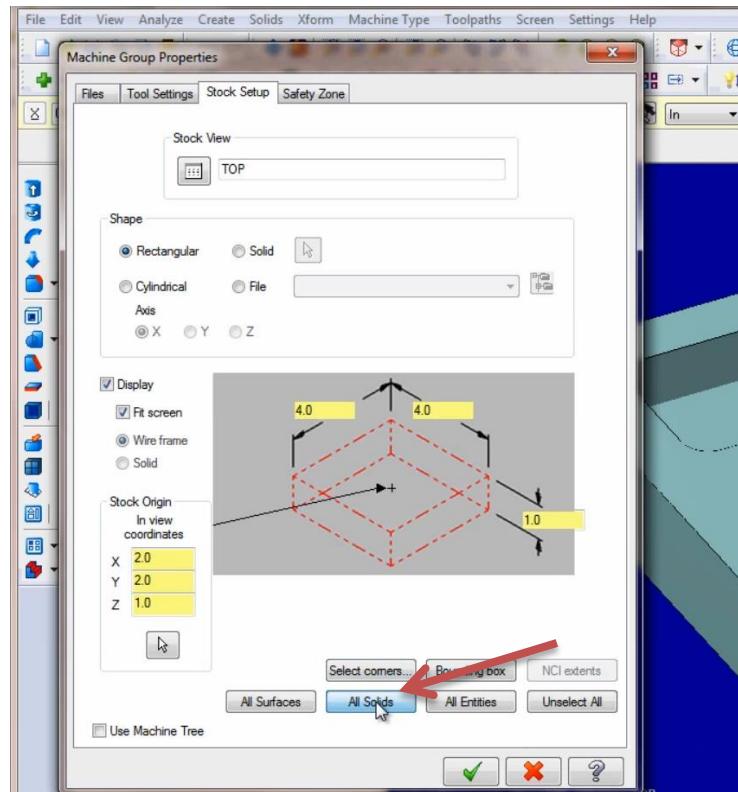
NOTE: You can select from a number of generic posts in the “Manage List” area.



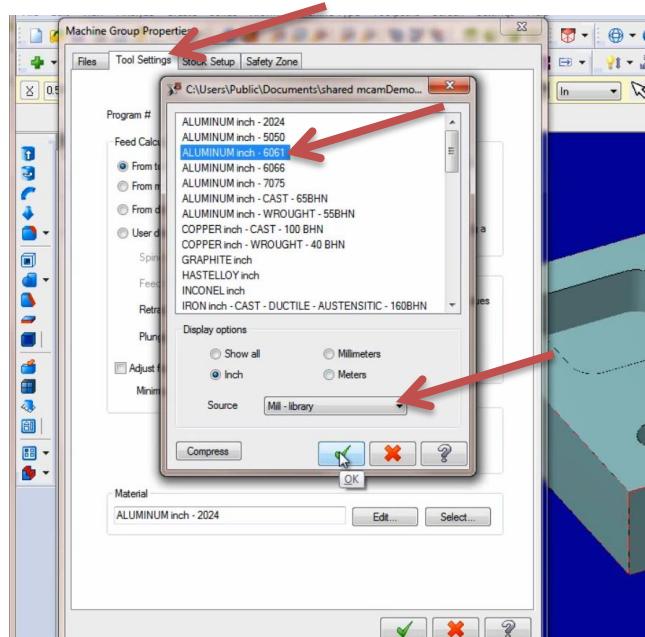
5. Select stock setup from the feature tree.



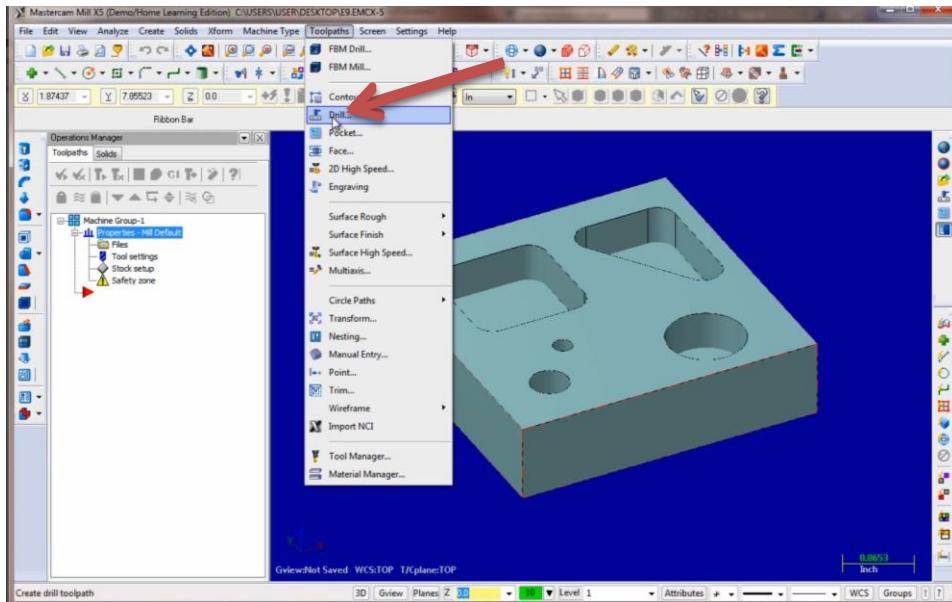
6. Select all solids.



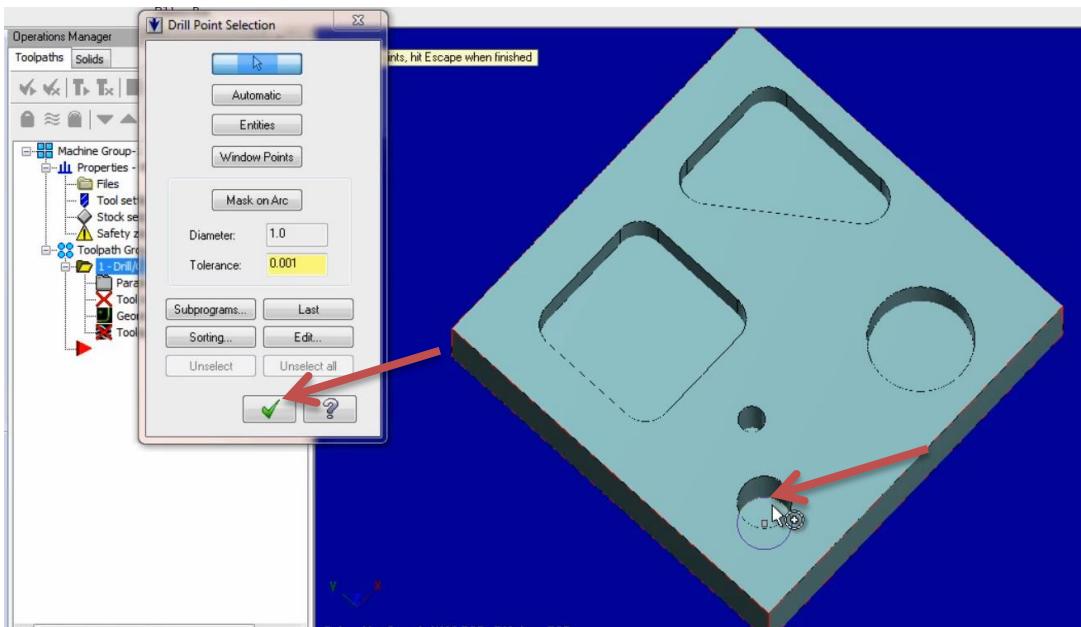
7. **Selecting Material:** Go to the Tool Settings tab and select Mill Library at the bottom and find Aluminum 6061.



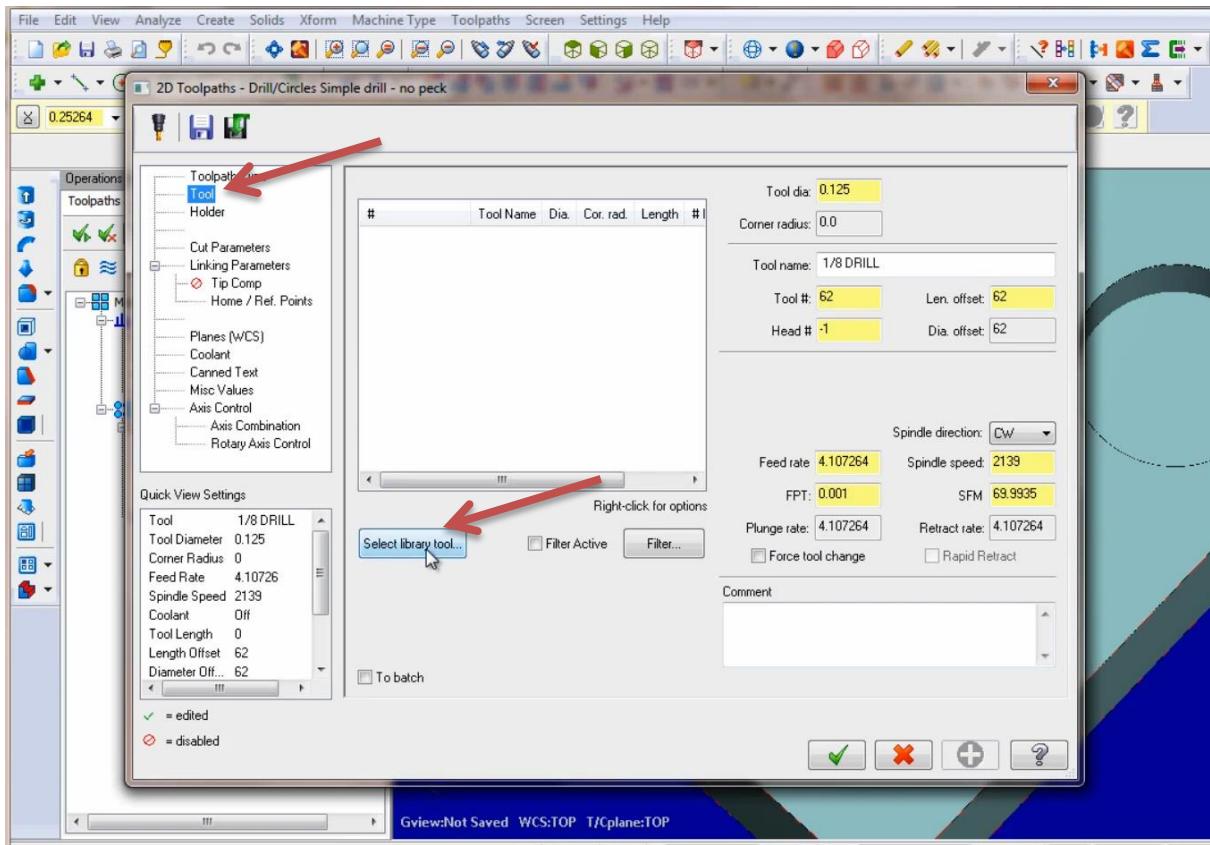
8. Select Properties and Mill Default in the Feature tree, and select Drill.



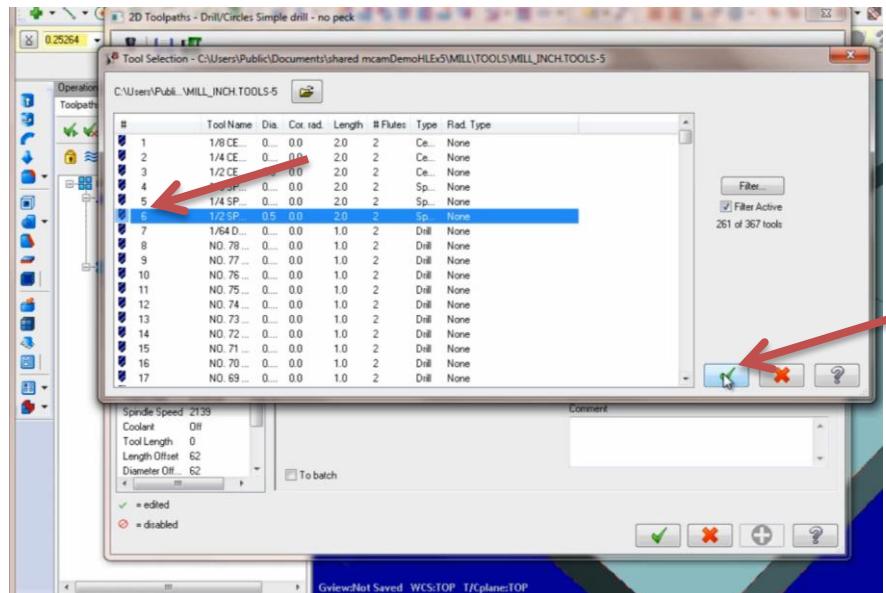
9. Select the bottom floor surfaces for the drill. Select the green checkbox.



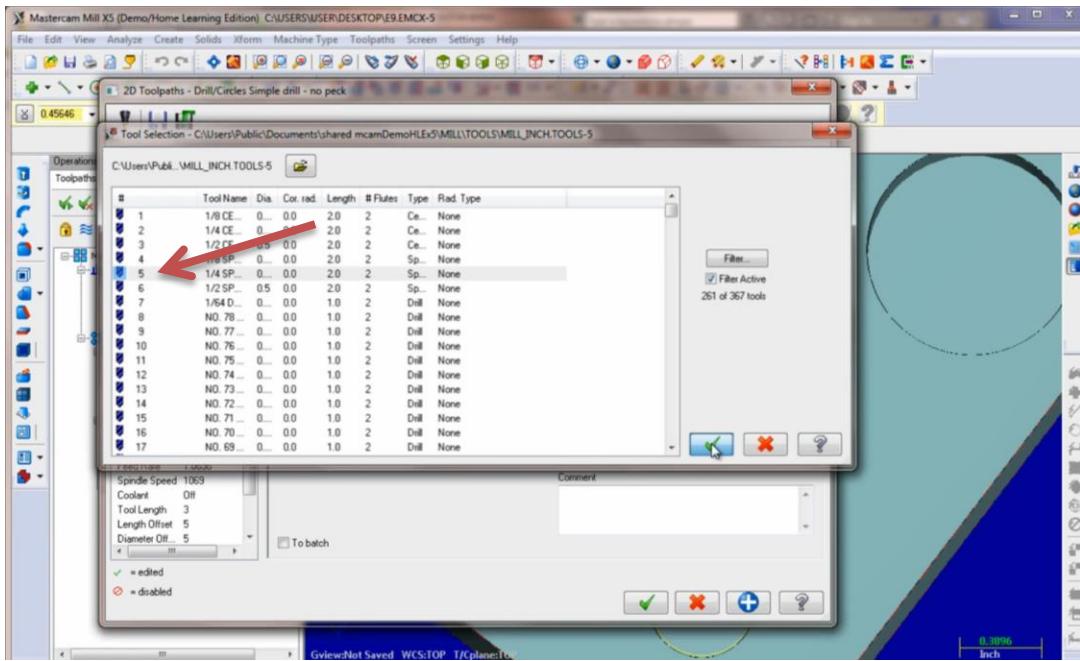
10. Select the Tool option, and then select the Library Tool



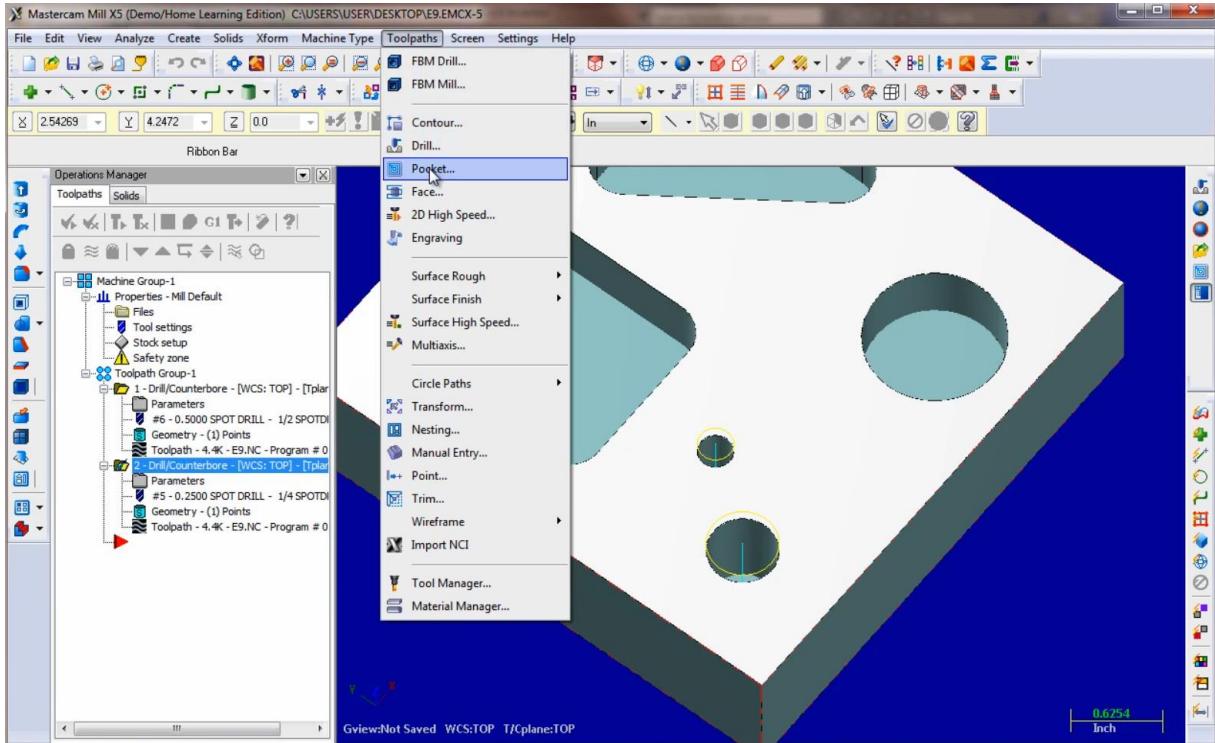
11. Select a 1/2" (#6) DIA. Drill. Hit the two green check boxes.



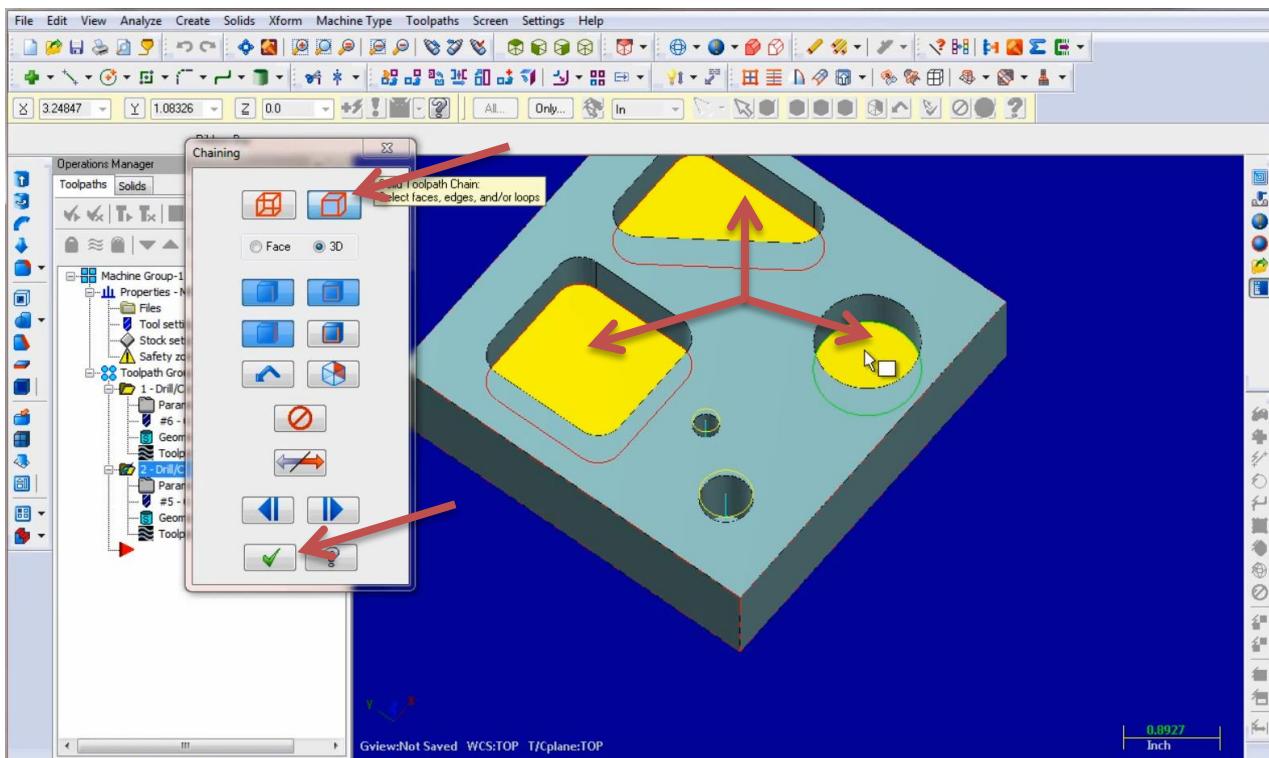
12. Repeat the same steps to add a $\frac{1}{4}$ " DIA (#5) drill for the second hole. Hit the two green check boxes.



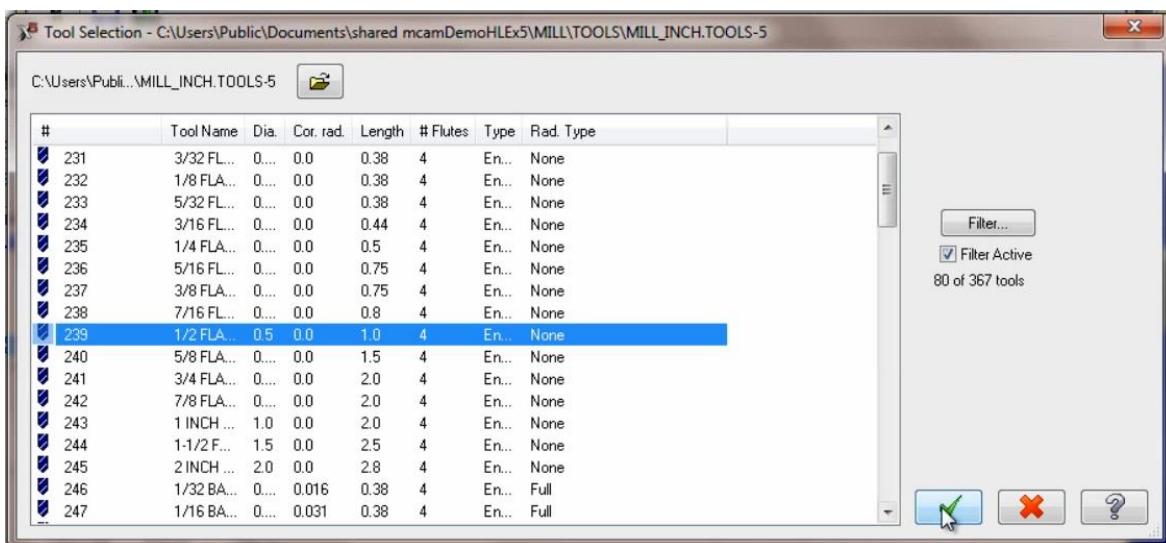
13. Pockets: Select the Pocket tool



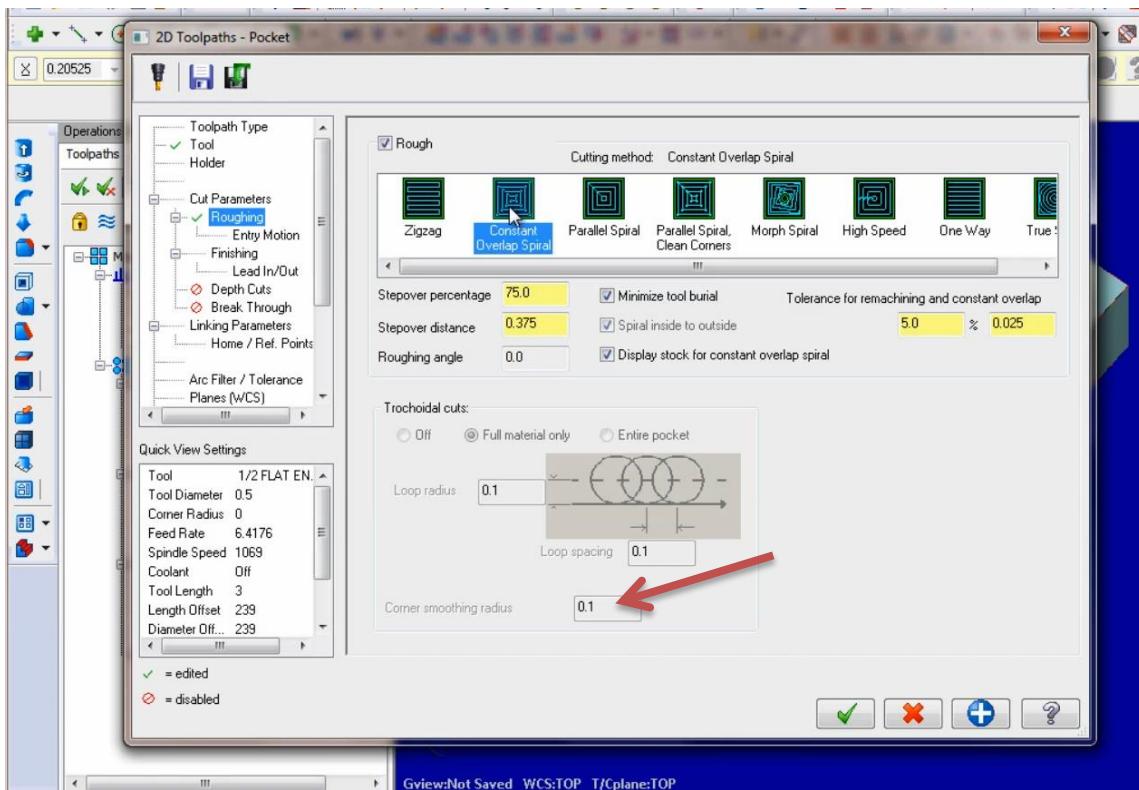
14. Select solids, and then select the floor of each pocket. Select the green check box.



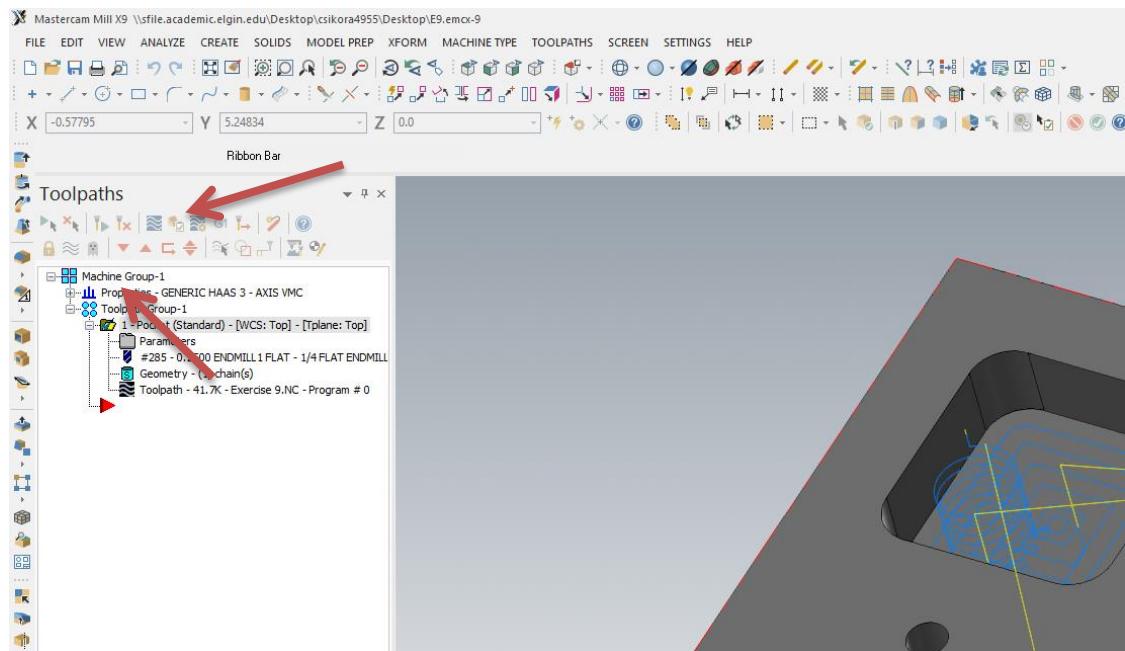
15. Select Tool, Select Tool. Select the $\frac{1}{2}$ " FLA (FLAT) (#239). Select the green check box.



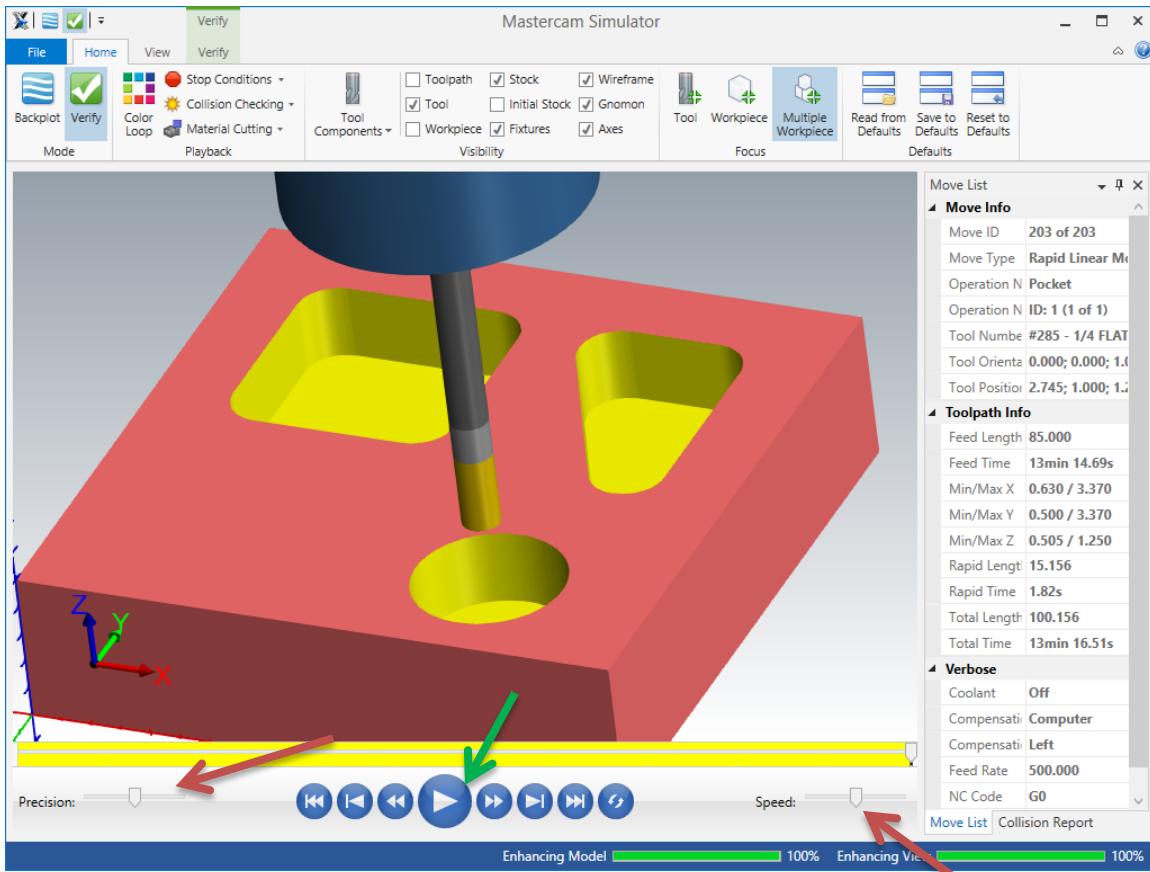
16. Select Roughing, and Constant overlap Spiral. Hit the green check box. Set Corner Smoothing Radius to .001



17. Select Machine Group 1 from the top of the feature tree, then select the Backplot icon.

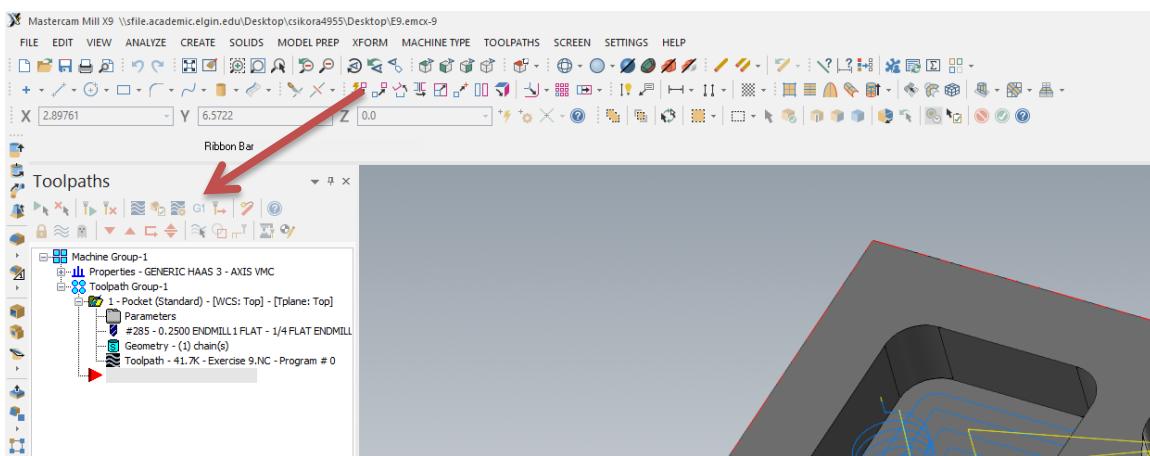


18. Adjust the setting as seen below and select the Machine icon to begin the toolpath simulation.



19. Watch the simulation.

20. G-Code Program Output – Select Post to have Mastercam generate the program.



E9.NC - Mastercam Code Expert

File Home View NC Functions

Go To Insert Block Numbers Remove Block Numbers Remove Spaces Insert Block Skip Remove Block Skip Remove Comments

Editing Communications

Send File Send Receive

First Previous Next Last First Previous Next

Syncs Tools

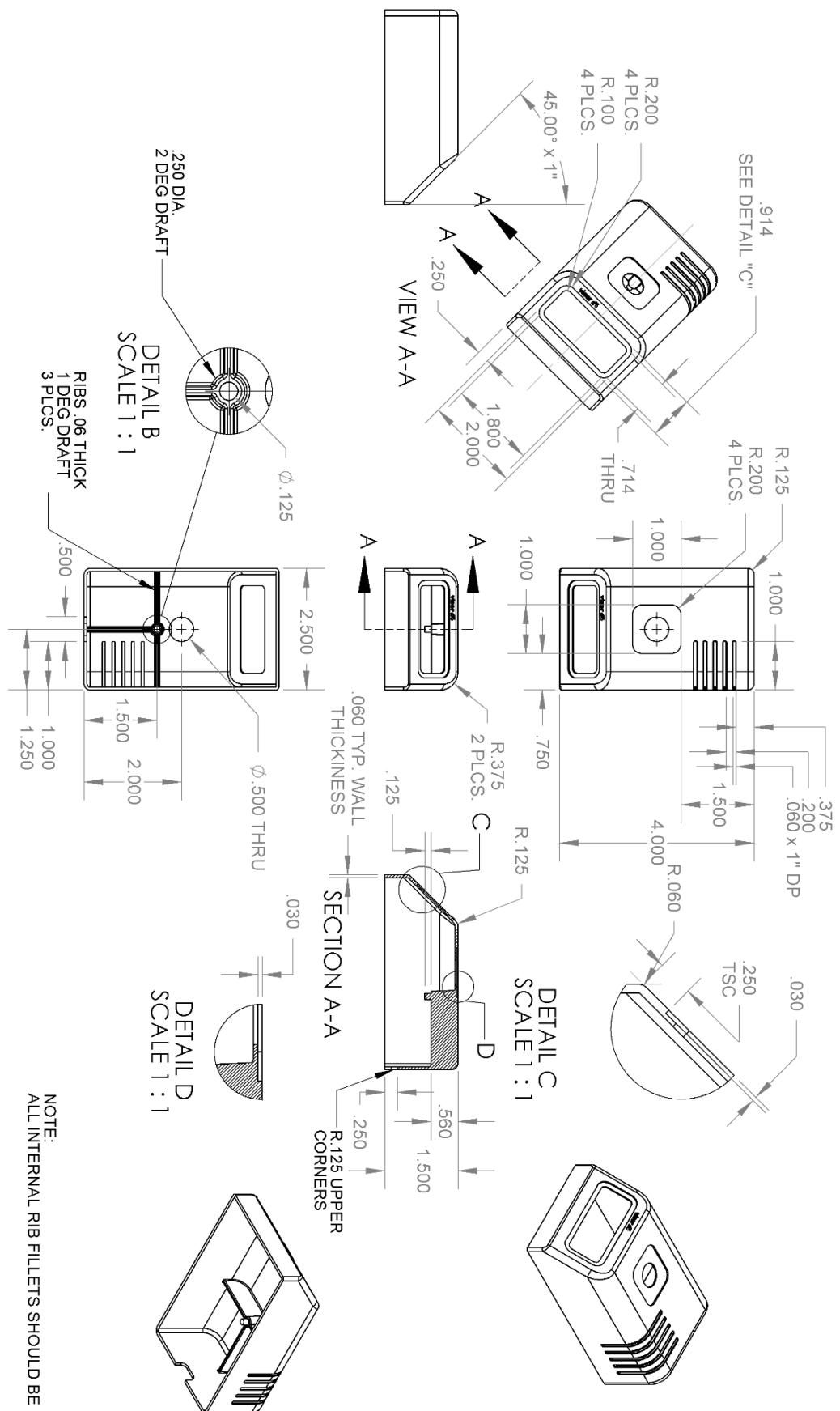
E9.NC X

```

1 %
2 00
3 (E9)
4 (DATE=DD-MM-YY - 02-03-16 TIME=HH:MM - 21:19)
5 (MCX FILE - \\SFILE.ACADEMIC.ELGIN.EDU\DESKTOP\CSIKORA4955\DESKTOP\E9.EMCX-9)
6 (NC FILE - \\SFILE.ACADEMIC.ELGIN.EDU\MYDOCS\CSIKORA4955\MY MCAMX9\MILL\NC\E9.N)
7 (MATERIAL - ALUMINUM INCH - 6061)
8 (T285|1/4 FLAT ENDMILL|H285|D285|TOOL DIA. - .25|XY STOCK - .005|Z STOCK - .005)
9 N100 G20
10 N110 G0 G17 G40 G49 G80 G90
11 N120 T285 M6
12 N130 G0 G90 G54 X.8352 Y2.6841 S2139 M3
13 N140 G43 H285 Z1.25
14 N150 Z1.2
15 N160 G1 Z1.1 F6.4
16 N170 X.857 Y2.6981 Z1.0986
17 N180 X.8802 Y2.7098 Z1.0973
18 N190 X.9044 Y2.719 Z1.0959
19 N200 X.9295 Y2.7256 Z1.0945
20 N210 X.9551 Y2.7297 Z1.0932
21 N220 X.981 Y2.731 Z1.0918
22 N230 X1.009 Y2.7294 Z1.0903
23 N240 X1.0366 Y2.7247 Z1.0889
24 N250 X1.0636 Y2.717 Z1.0874
25 N260 X1.0895 Y2.7063 Z1.0859
26 N270 X1.114 Y2.6927 Z1.0845
27 N280 X1.1369 Y2.6765 Z1.083
28 N290 X1.1578 Y2.6578 Z1.0816
29 N300 X1.1765 Y2.6369 Z1.0801
30 N310 X1.1927 Y2.614 Z1.0786
31 N320 X1.2063 Y2.5895 Z1.0772
32 N330 X1.217 Y2.5636 Z1.0757
33 N340 X1.2247 Y2.5366 Z1.0742
34 N350 X1.2294 Y2.509 Z1.0728
35 N360 X1.231 Y2.481 Z1.0713
36 N370 X1.2294 Y2.453 Z1.0698
37 N380 X1.2247 Y2.4254 Z1.0684
38 N390 X1.217 Y2.3984 Z1.0669
39 N400 X1.2063 Y2.3725 Z1.0654
40 N410 X1.1927 Y2.348 Z1.0639
41 N420 X1.1765 Y2.3251 Z1.0625
42 N430 X1.1578 Y2.3042 Z1.061
43 N440 X1.1369 Y2.2855 Z1.0595
44 N450 X1.114 Y2.2693 Z1.0581
45 N460 X1.0895 Y2.2557 Z1.0566
46 N470 X1.0636 Y2.245 Z1.0551

```

Finished



NOTE:
ALL INTERNAL RIB FILLETS SHOULD BE .020"

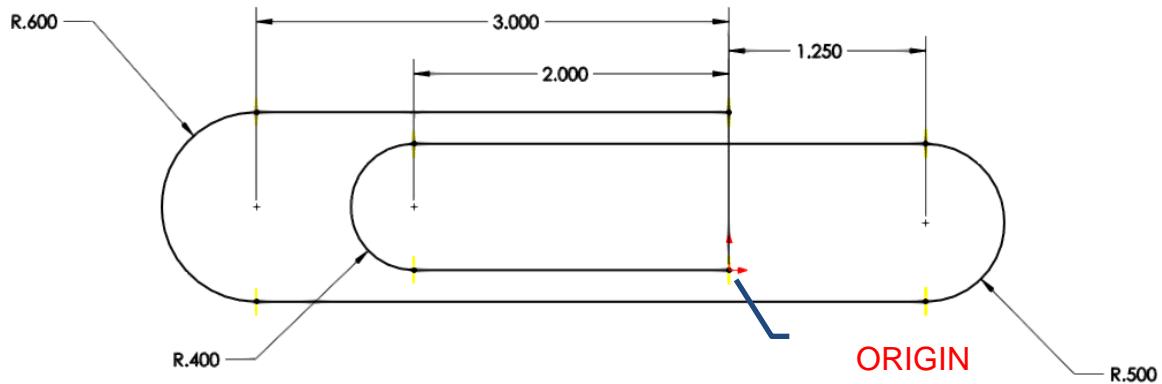
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NEXT ASSY	USED ON	INTERPRET GEOMETRIC TOLERANCING PER: MATERIAL	Q.A.	COMMENTS:	
APPLICATION	FINISH				
DO NOT SCALE DRAWING		SCALE: 1:3	SIZE A	DWG. NO. L11c COVER	REV
		SHEET 1 OF 1			

EXERCISE11D

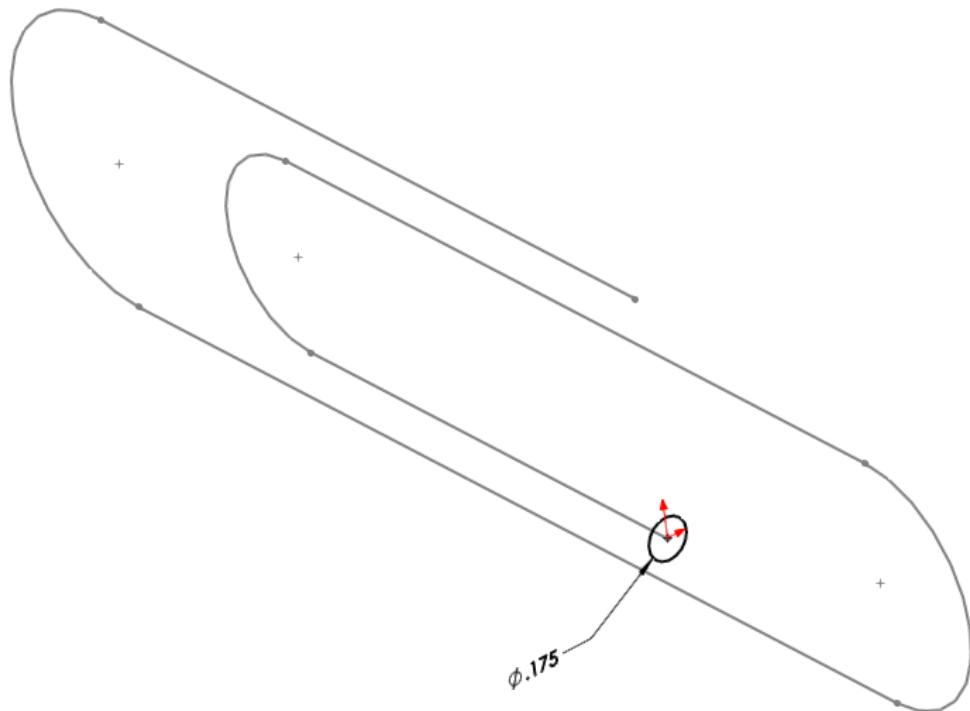
Finite Element Analysis (F.E.A.)

In this exercise we will look at a basic stress analysis using SolidWorks Simulation Express.

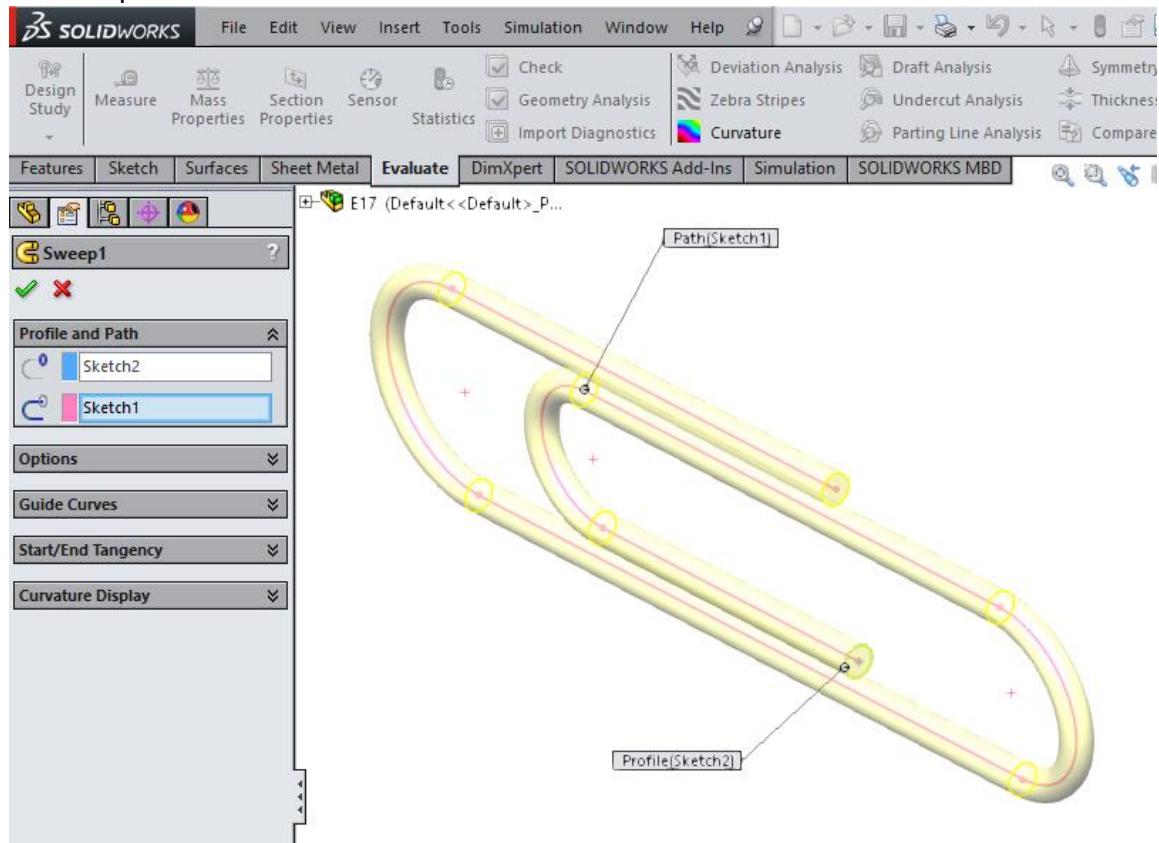
Model the paperclip using the sweep feature. Draw the following on the “Front” plane.
Use tangent arcs.



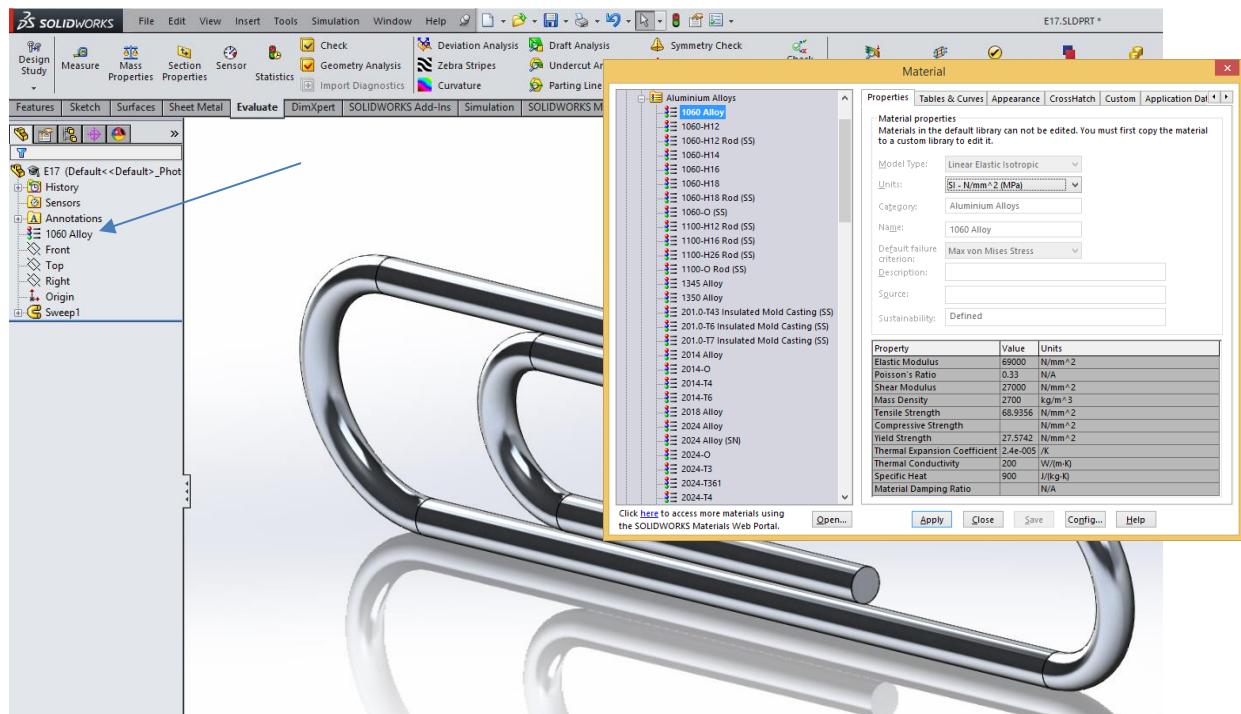
Sketch the .175 DIA. circle on the “Right” plane.



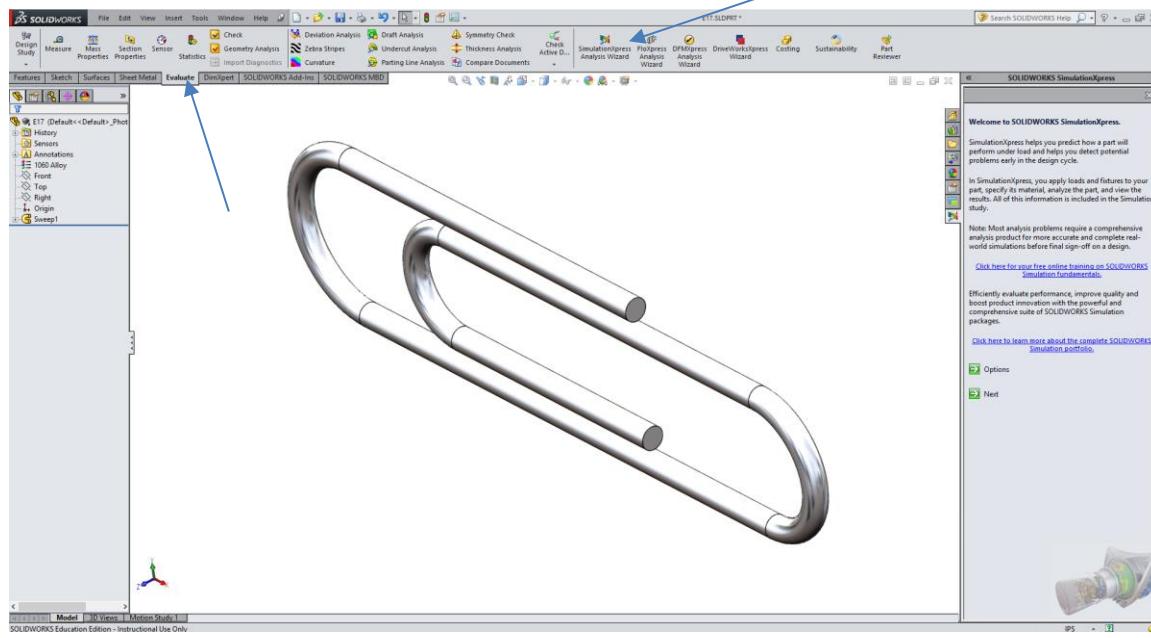
Use Swept Boss Base



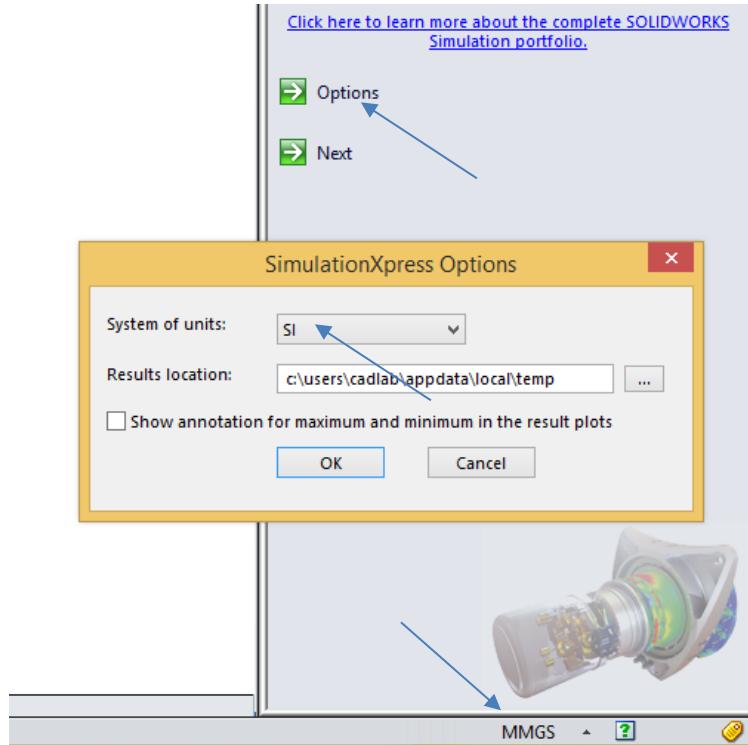
R.M.B. select Material/Edit/Aluminum Alloy/1060 Alloy



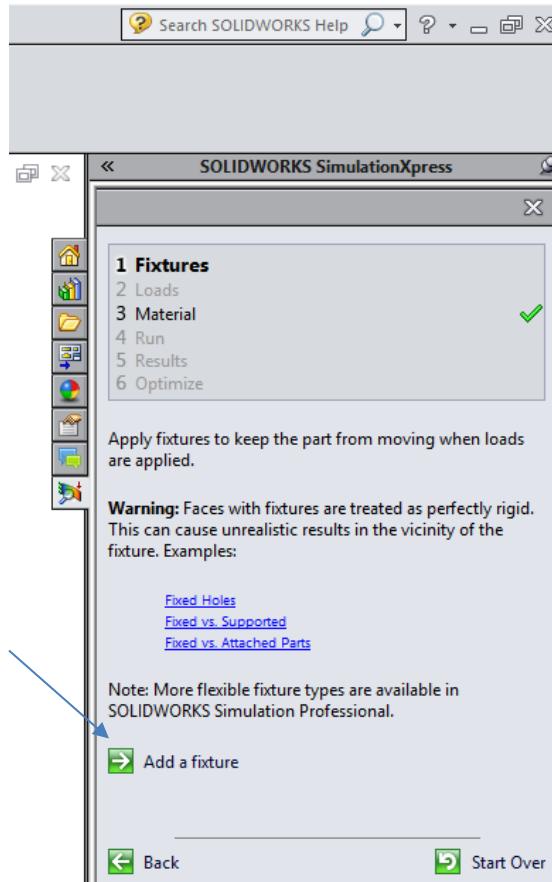
Select Evaluate tab, SolidWorks Simulation Express



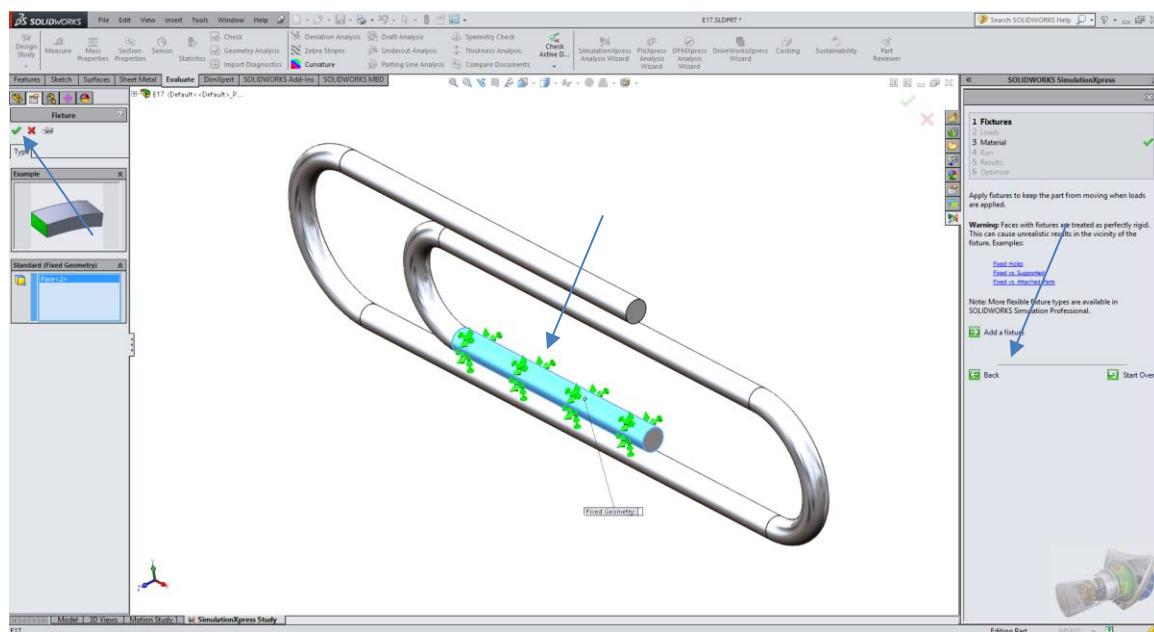
Select MMGS, and Options/SI (Lower right corner of screen).



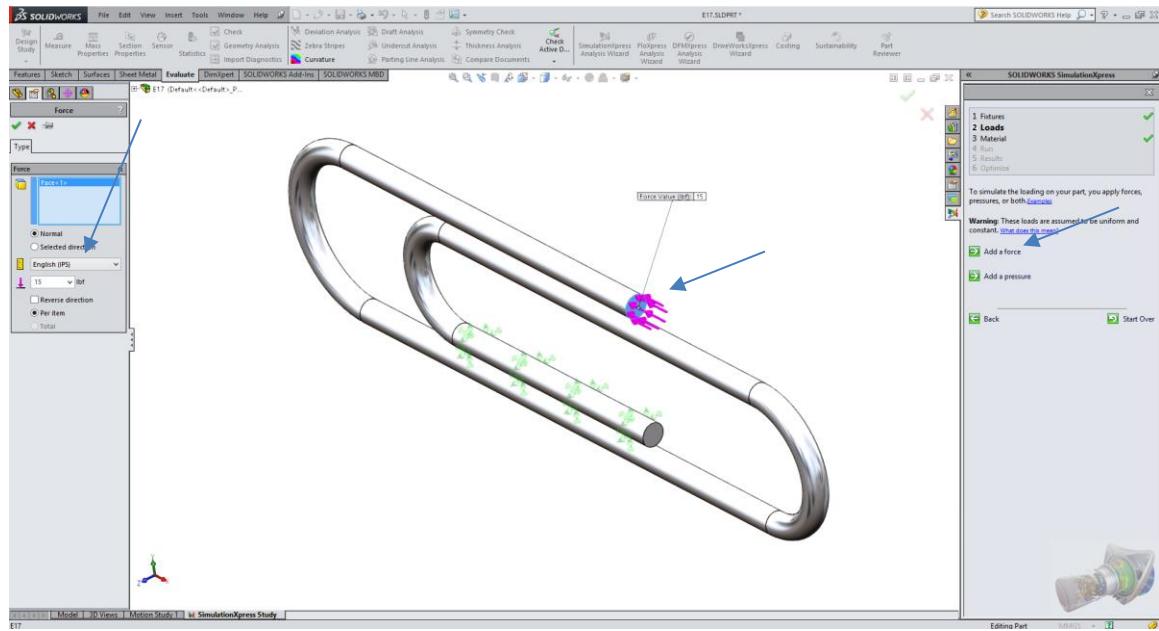
Select Next/Next/Add Fixtures



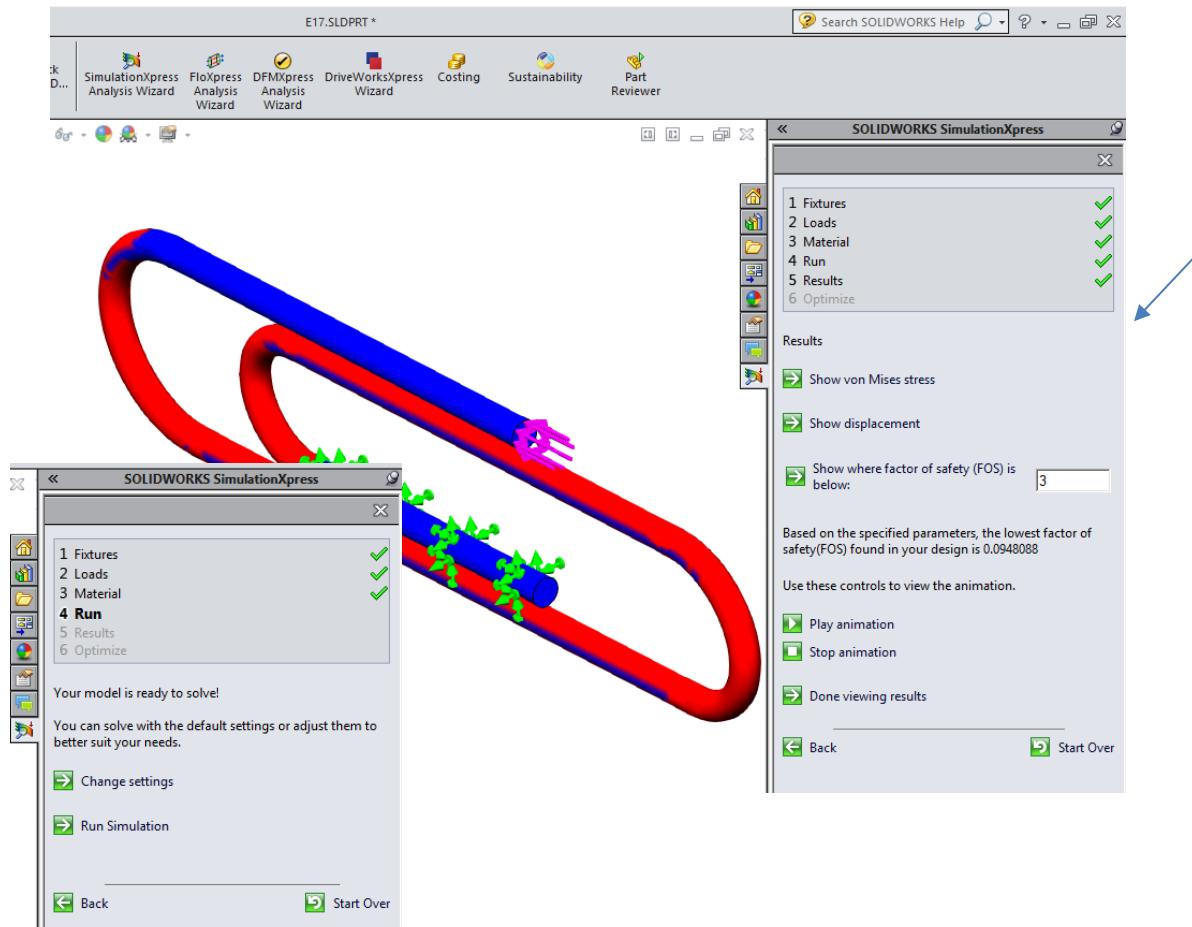
Select cylindrical face shown on picture. Select the green check.



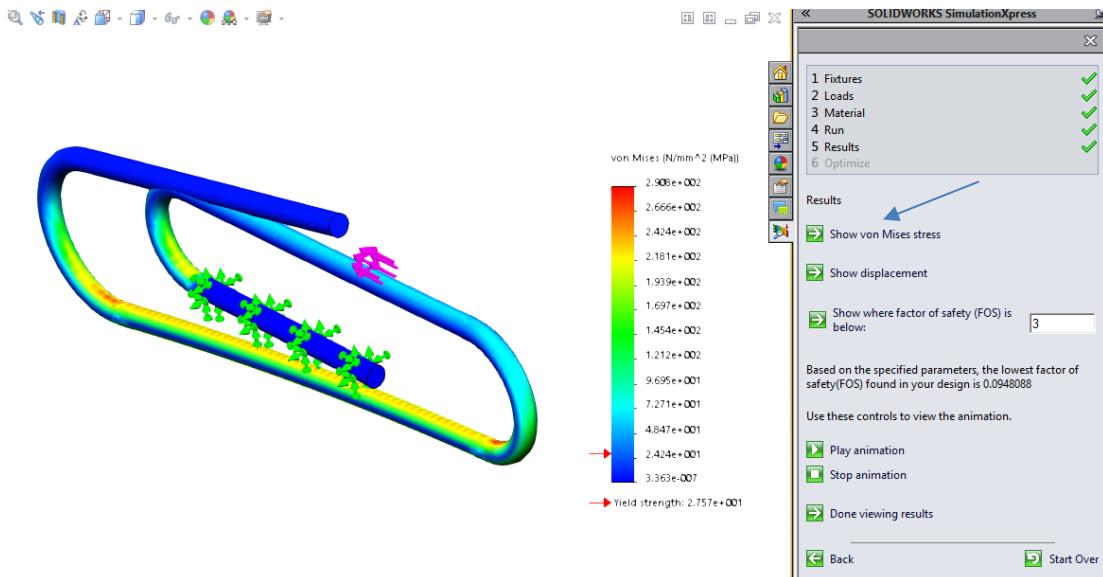
Select Next and Add a Force. Select the end face of the clip, Set to English IPS, 15 lbf.



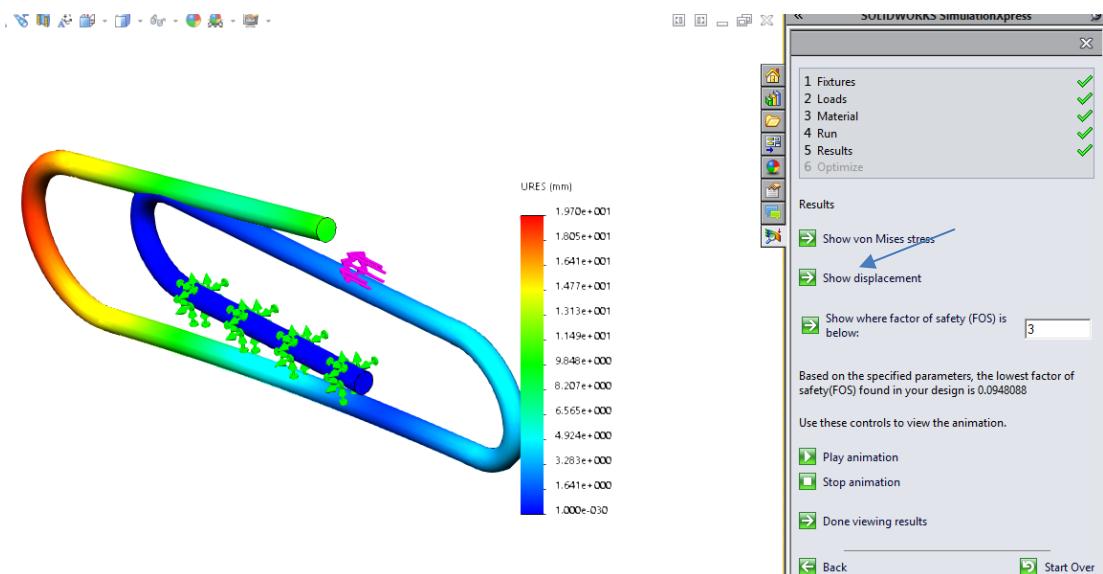
Select Next/Run Simulation/Yes/Set Factor of Safety to 3.0



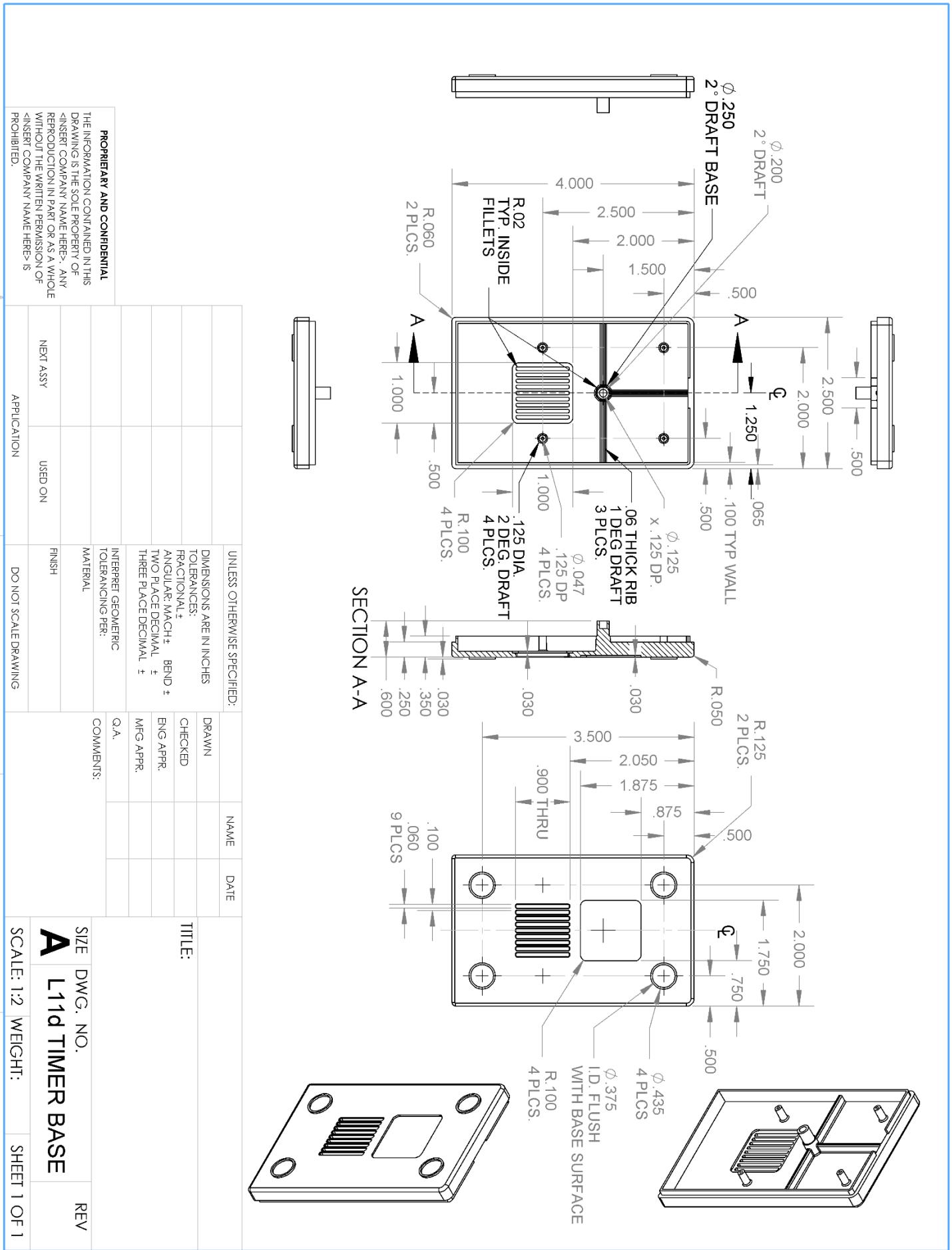
Select show von Mises Stress



Select show Displacement



Complete

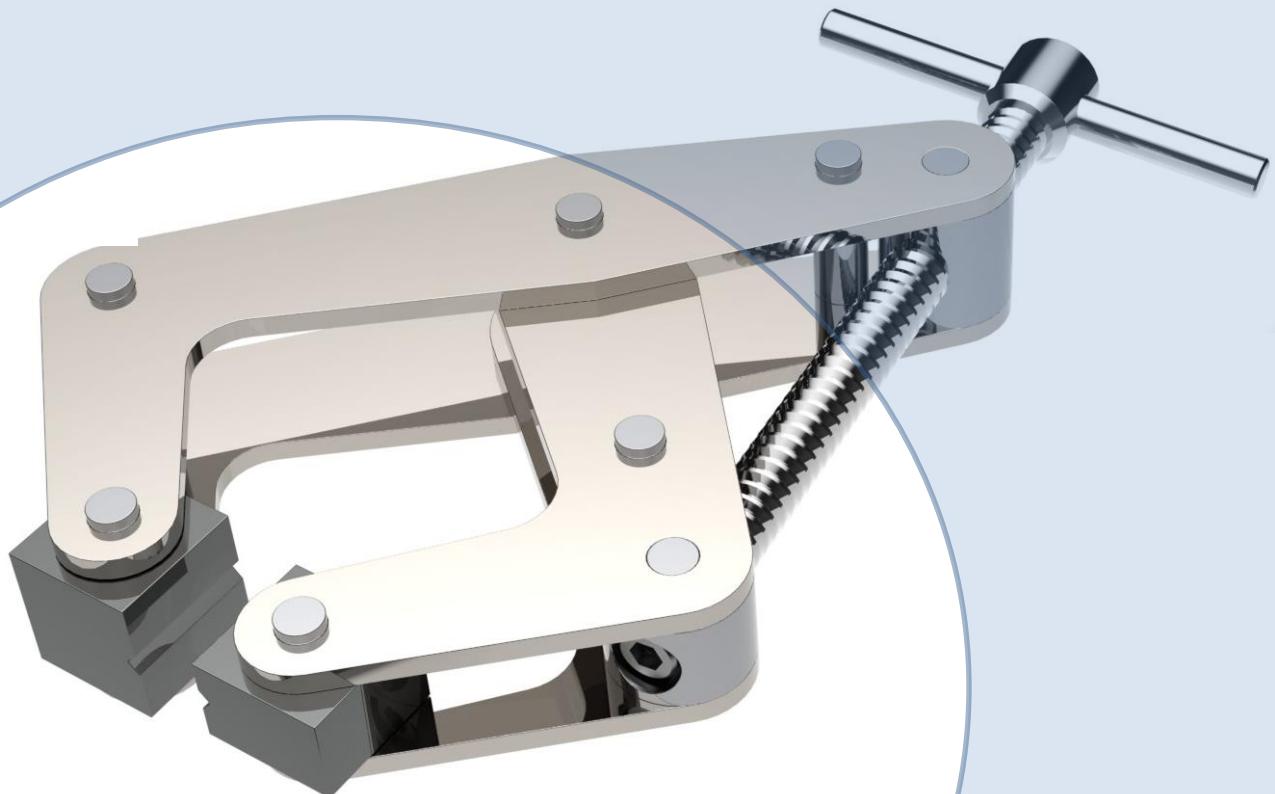


Quiz 2 *Special Project*

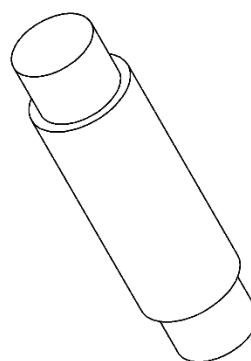
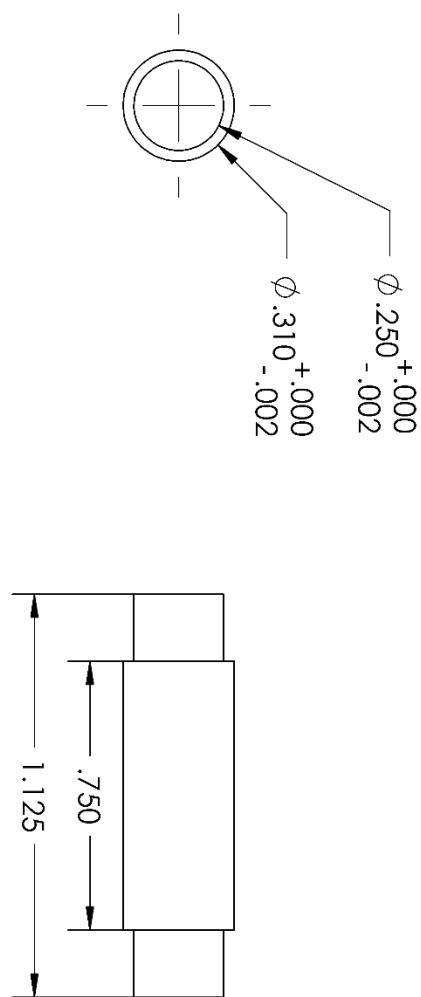
Q2

20 Points

1. MODEL THE PARTS
2. ASSEMBLE
3. RECREATE THE DRAWINGS



NOTE: SYMMETRIC MODEL



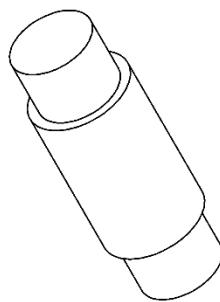
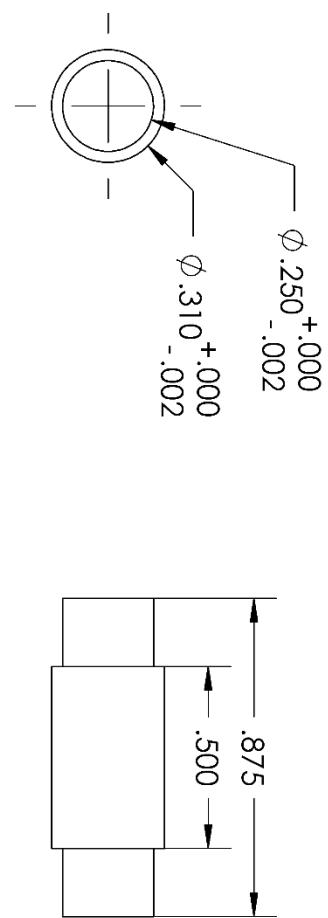
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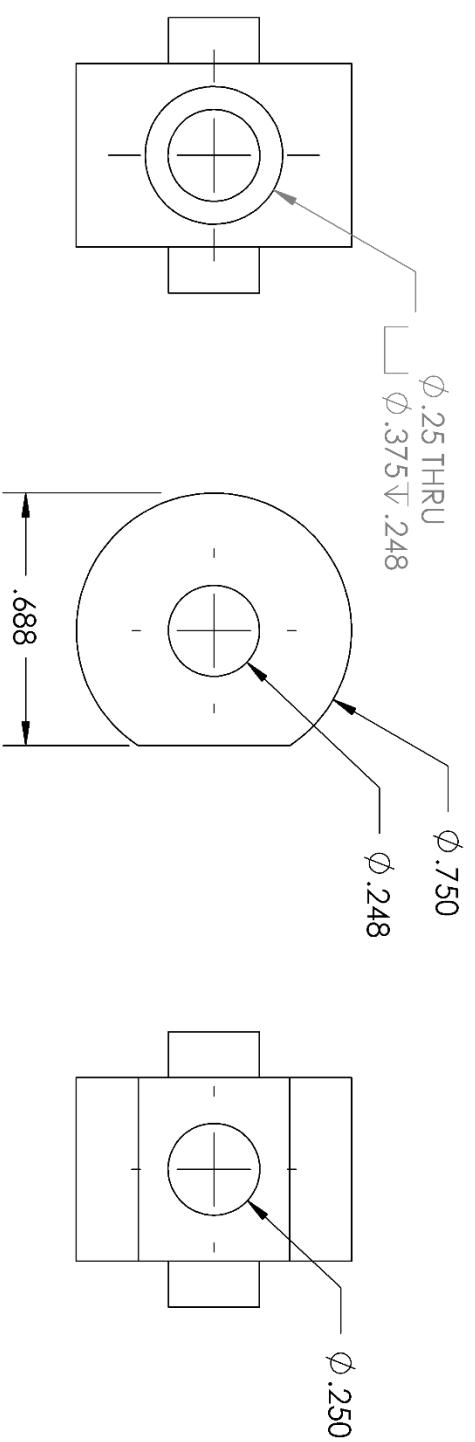
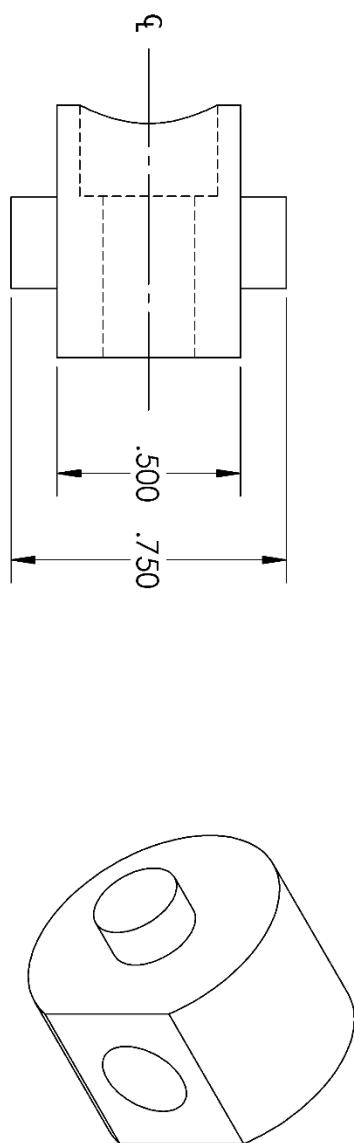
UNLESS OTHERWISE SPECIFIED:		
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INTERPRET GEOMETRIC TOLERANCING PER: MATERIAL	Q.A.	
FINISH		
NEXT ASSY	USED ON	COMMENTS:
APPLICATION	DO NOT SCALE DRAWING	

SIZE A	DWG. NO. EYE PIN	REV	SCALE: 2:1		
			WEIGHT:	SHEET 1 OF 1	

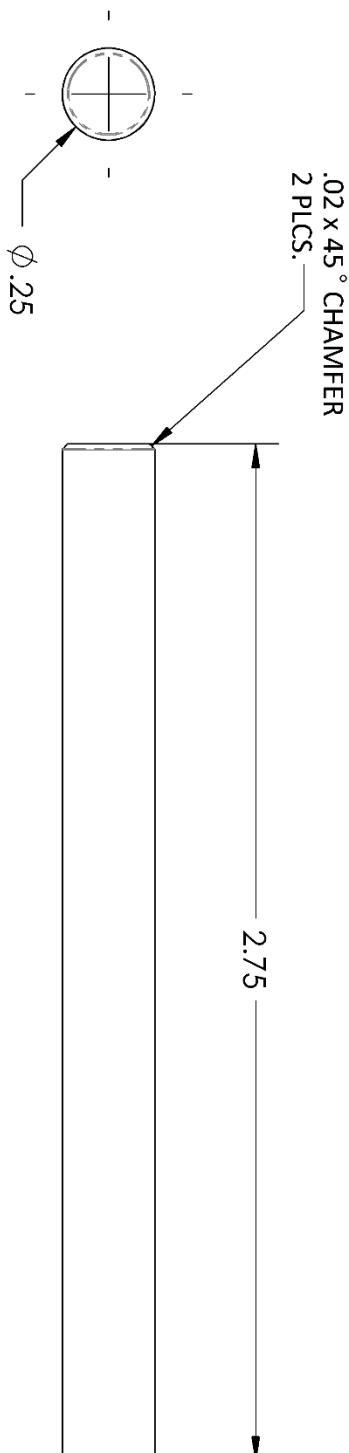
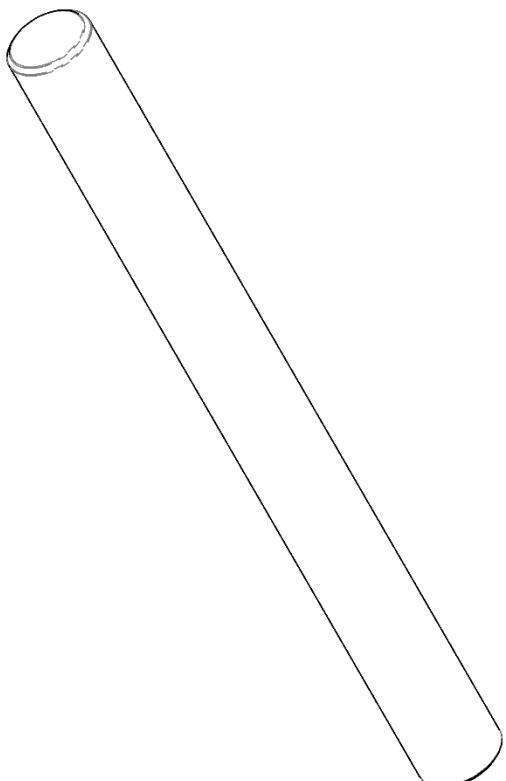
NOTE: SYMMETRIC MODEL



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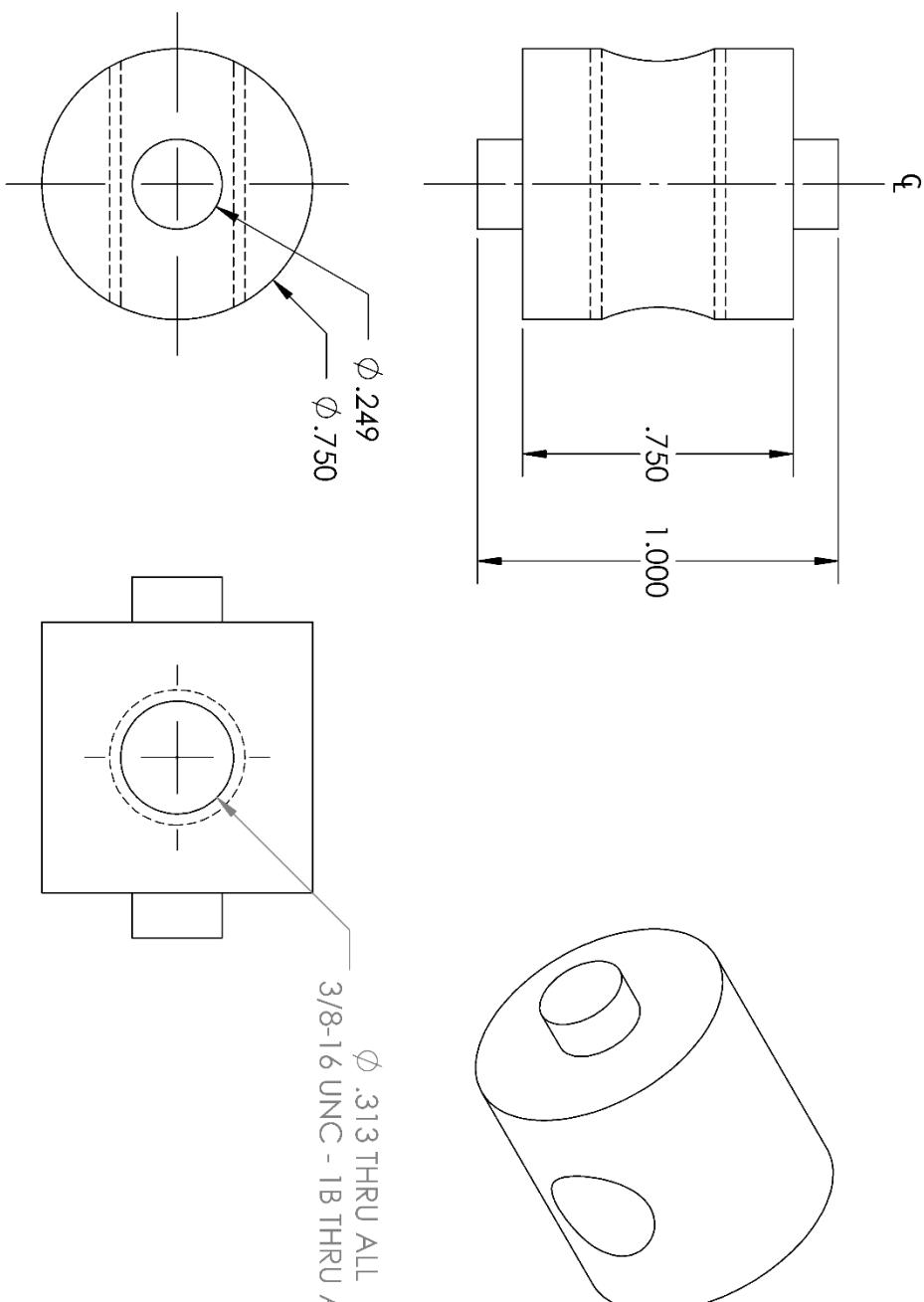
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INTERPRET GEOMETRIC TOLERANCING PER: MATERIAL	COMMENTS:
NEXT ASSY USED ON FINISH	SIZE A DWG. NO. HINGE BOLT SCALE: 2:1 WEIGHT: REV
APPLICATION DO NOT SCALE DRAWING	SHEET 1 OF 1



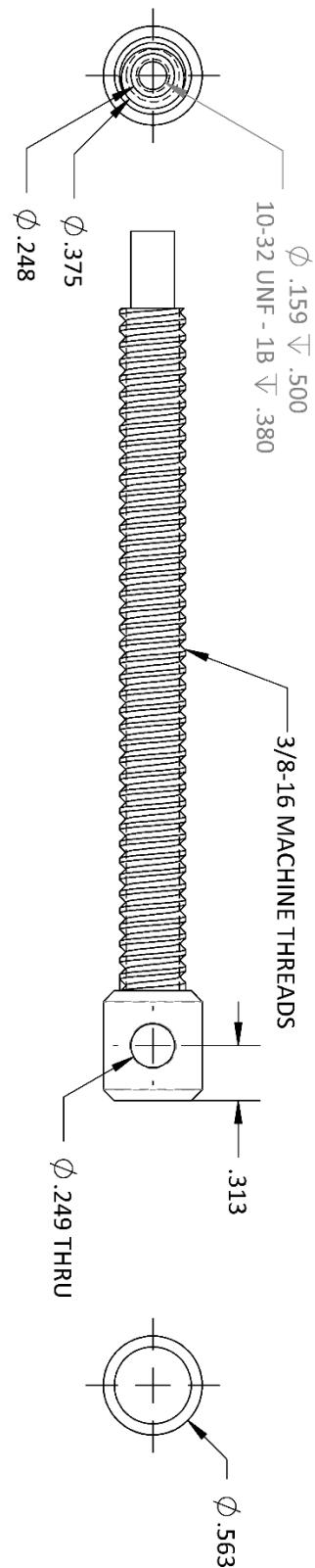
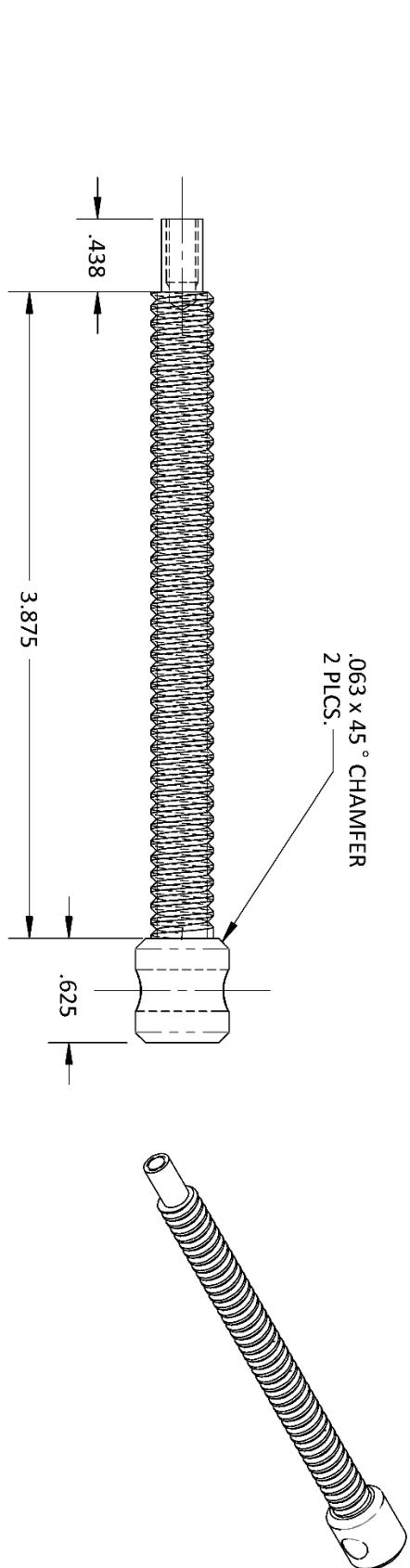
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DIMENSIONS ARE IN INCHES	DRAWN	NAME	DATE
TOLERANCES: FRACTIONAL \pm	CHECKED		TITLE:
ANGULAR: MACH \pm	BEND \pm		
TWO PLACE DECIMAL \pm	ENG APPR.		
THREE PLACE DECIMAL \pm	MFG APPR.		
INTERPRET GEOMETRIC TOLERANCING PER:	Q.A.		
MATERIAL	COMMENTS:		
NEXT ASSY	USED ON	SIZE	DWG. NO.
FINISH		A	PF SHAFT
	DO NOT SCALE DRAWING	SCALE: 2:1	WEIGHT: SHEET 1 OF 1
		REV	



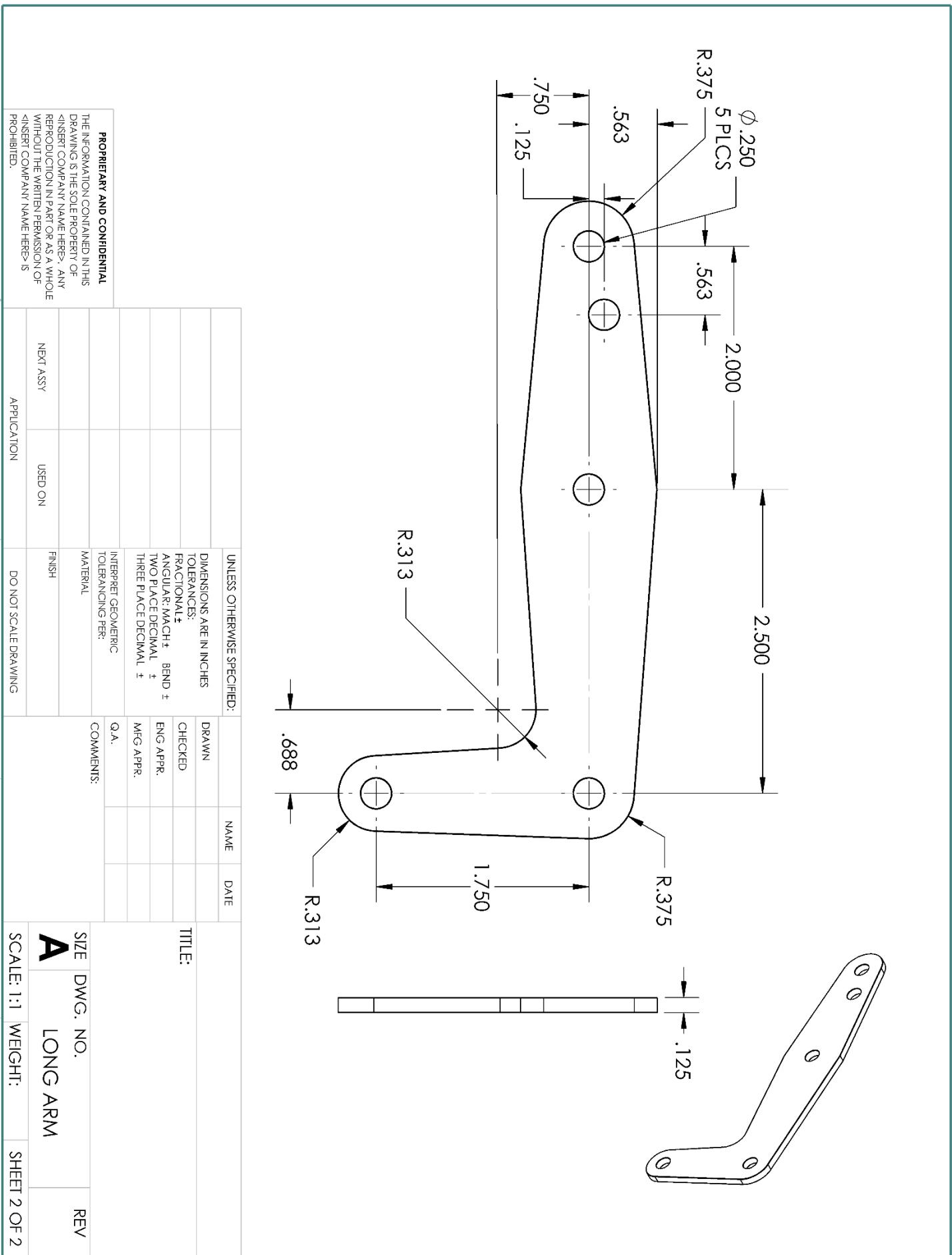
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TWO PLACE DECIMAL \pm THREE PLACE DECIMAL \pm		CHECKED	ENG APPR.	MFG APPR.	
INTERPRET GEOMETRIC TOLERANCING PER: MATERIAL		Q.A.			
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APPLICATION	NEXT ASSY	USED ON	FINISH		
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SCALE: 2:1		WEIGHT:	SHEET 1 OF 1		

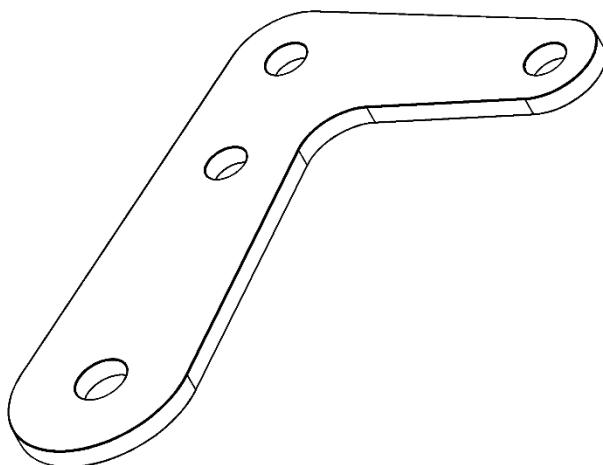
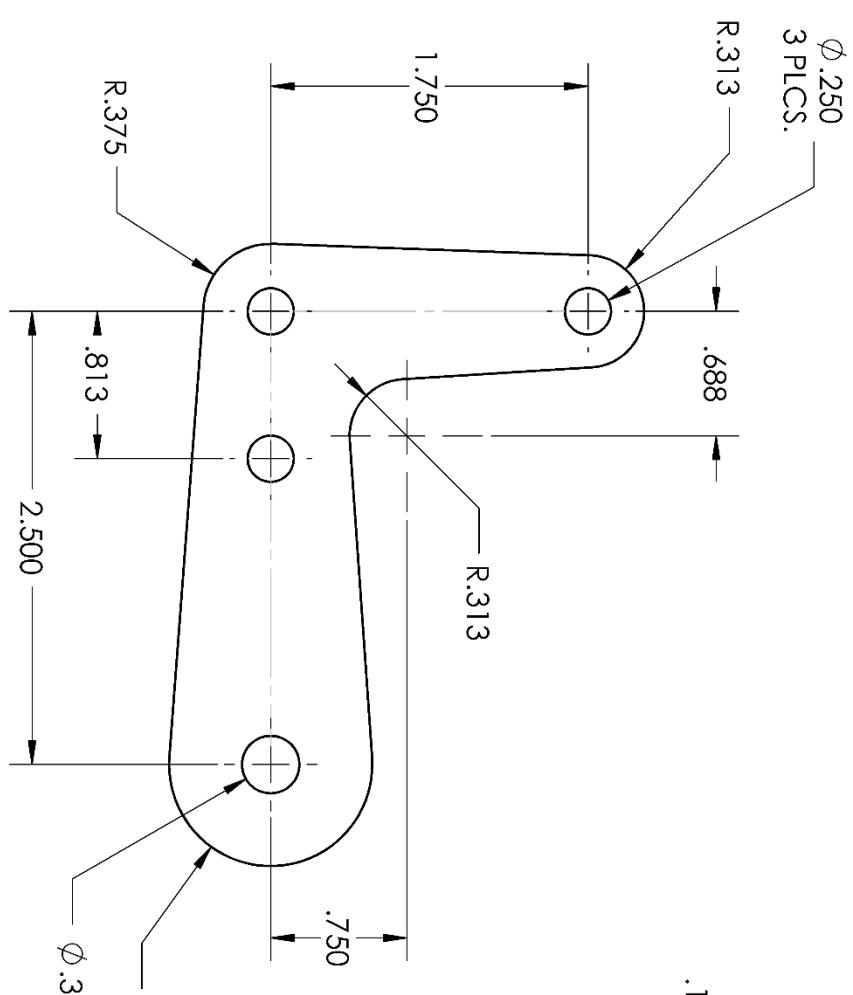


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ANGULAR: MACH \pm	BEND \pm	ENG APPR.		
TWO PLACE DECIMAL \pm		MFG APPR.		
THREE PLACE DECIMAL \pm		Q.A.		
INTERPRET GEOMETRIC TOLERANCING PER:	COMMENTS:			
MATERIAL				
FINISH				
NEXT ASSY	USED ON			
APPLICATION	DO NOT SCALE DRAWING			

SIZE	DWG. NO.	REV
A	THREADED SHAFT	
SCALE: 1:1	WEIGHT:	SHEET 1 OF 1





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NEXT ASSY: USED ON: FINISH:	
APPLICATION: DO NOT SCALE DRAWING	

UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN INCHES
TOLERANCES:
FRACTIONAL \pm
ANGULAR: MACH \pm
TWO PLACE DECIMAL \pm
THREE PLACE DECIMAL \pm
INTERPRET GEOMETRIC
TOLERANCING PER:
MATERIAL:

NAME: DATE: TITLE:
DRAWN: CHECKED: ENG APPR.
BEND \pm : MFG APPR.

Q.A.

COMMENTS:

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SIZE	DWG. NO.	REV
A	TWIST CLAMP SHORT ARM	
SCALE: 1:1	WEIGHT:	SHEET 1 OF 2

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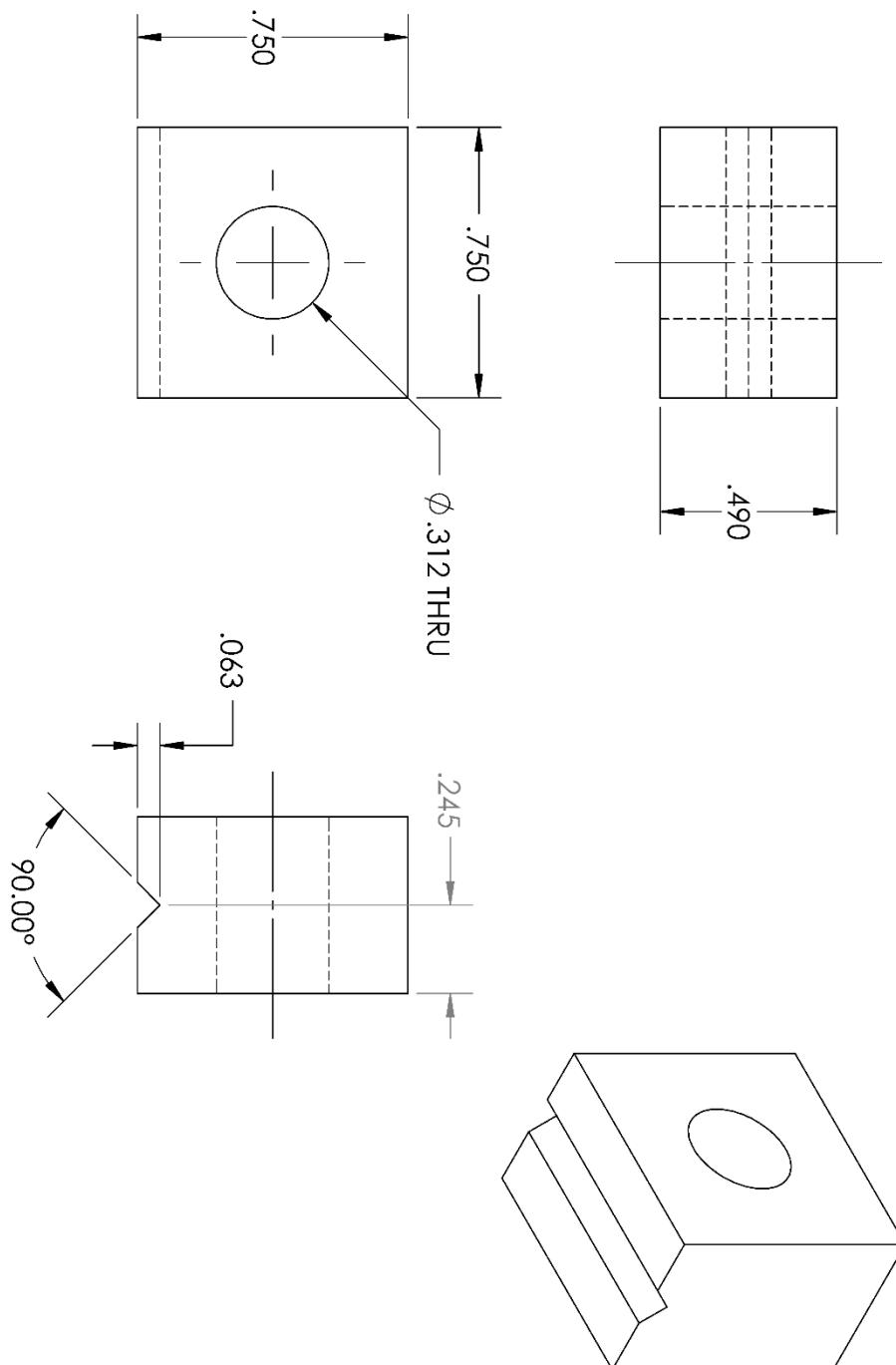
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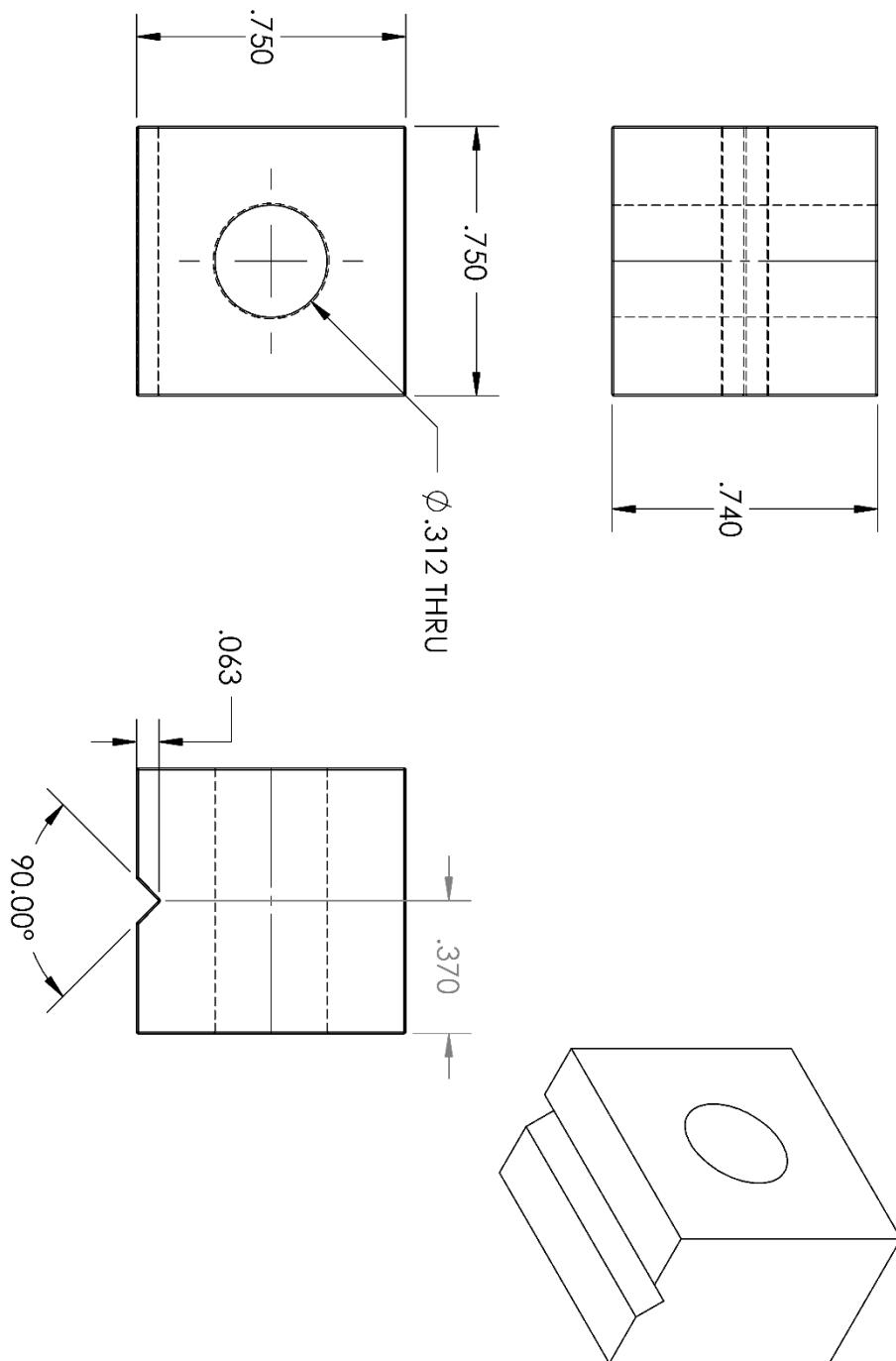
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ANGULAR: MACH \pm		BEND \pm				
TWO PLACE DECIMAL \pm						
THREE PLACE DECIMAL \pm						
INTERPRET GEOMETRIC TOLERANCING PER:		MFG APPR.				
MATERIAL		ENG APPR.				
COMMENTS:		Q.A.				
SIZE	DWG. NO.					REV
A	NARROW PAD					
SCALE: 2:1	WEIGHT:					SHEET 1 OF 1
APPLICATION	USED ON	FINISH	DO NOT SCALE DRAWING			
NEXT ASSY						

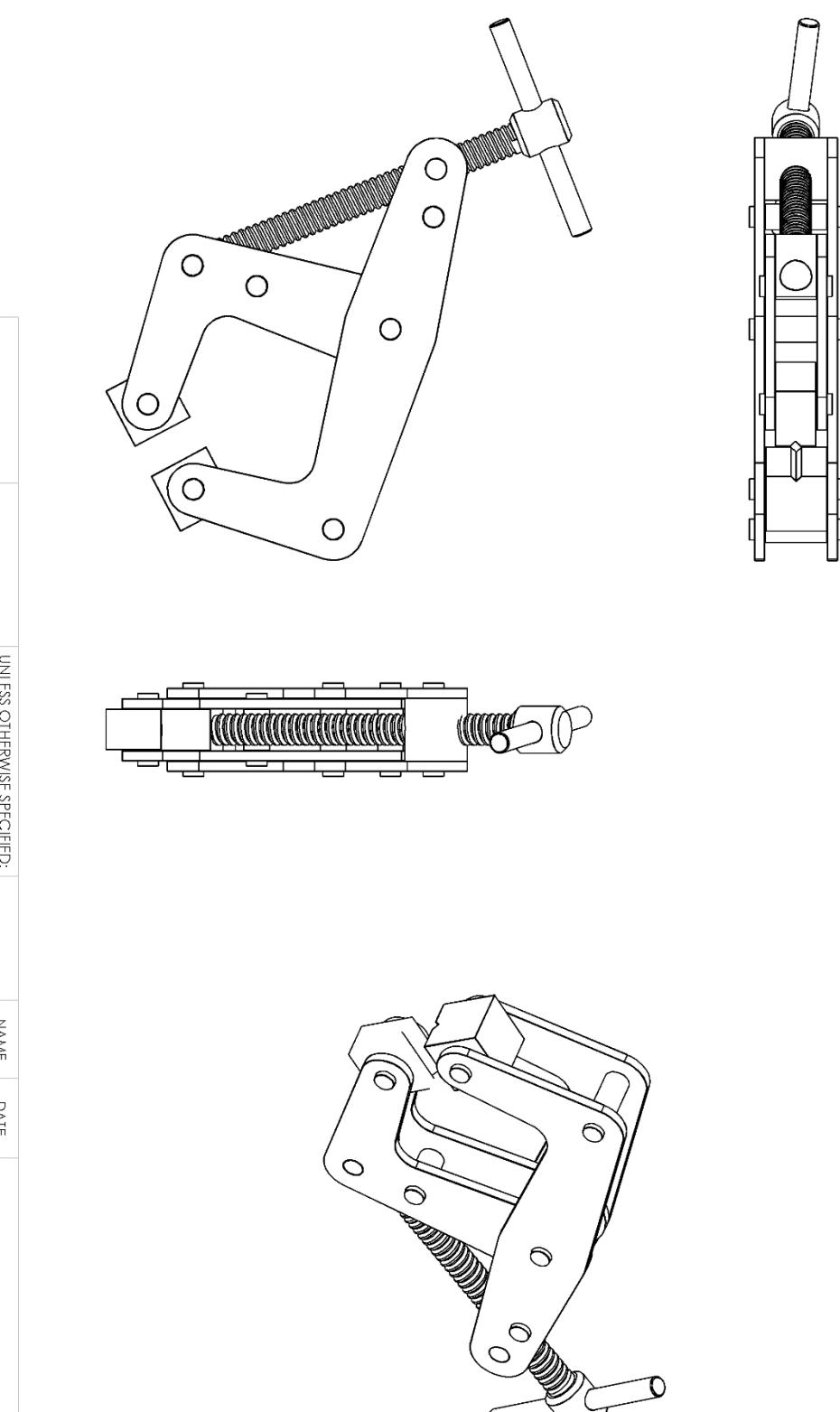


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INTERPRET GEOMETRIC TOLERANCING PER: MATERIAL		TITLE: INT 104_1
COMMENTS:		



ITEM NO.	QTY.	PART NO.	DESCRIPTION
1	4	EYE PIN	
2	2	TWIST CLAMP LARGE ARM	
3	1	THREAD PIN	
4	2	TWIST CLAMP SHORT ARM	
5	1	HINGE BOLT	
6	1	THREADED SHAFT	
7	1	PF SHAFT	
8	1	WIDE PAD	
9	1	NARROW PAD	
10	2	RIB PIN	

