



SOLIDWORKS ADVANCED

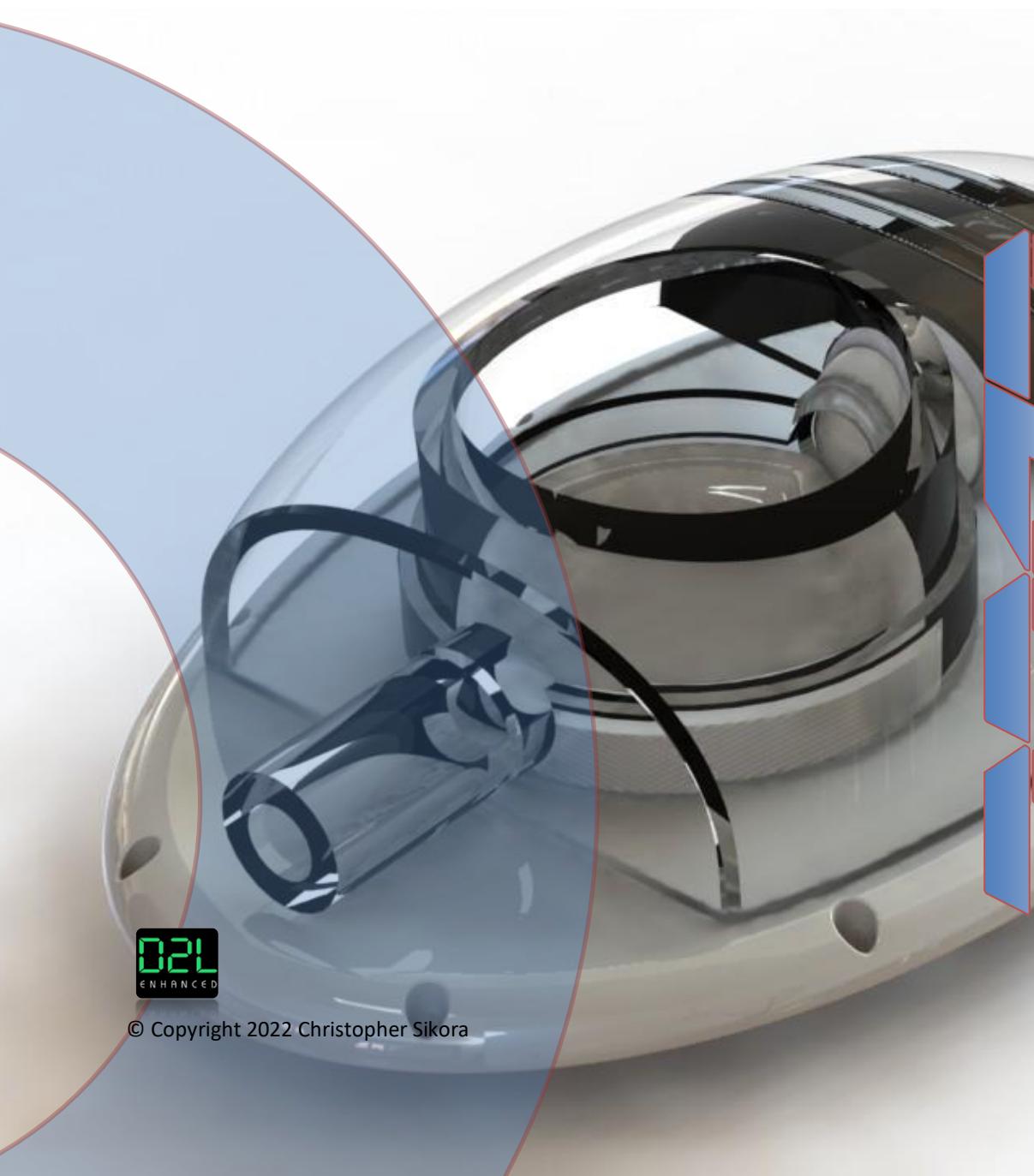
BY C.F. SIKORA

COMPUTER AIDED DESIGN

D2L



© Copyright 2022 Christopher Sikora



This manual is for educational purposes only. It may be printed, but not resold for profit for its content.

SolidWorks® is a registered trademark of Dassault Systèmes SolidWorks Corp.

SolidWorks® is a product name of Dassault Systèmes SolidWorks Corp.

ACIS® is a registered trademark of Dassault Systèmes SolidWorks Corp.

IGES™ Access Library is a trademark of IGES Data Analysis, Inc.

Other brand or product names are trademarks or registered trademarks of their respective holders.

The information discussed in this document is subject to change without notice and should not be considered commitments by Christopher F. Sikora.

The software discussed in this document is furnished under a license and may be used or copied only in accordance with the terms of this license.

CAD 121 COURSE SYLLABUS

SolidWorks Advanced

Course Description:

SolidWorks Advanced

3 credit hours

Exploration of the theory and application of solid modeling techniques for product design and manufacturing. Prerequisite: SolidWorks Basics CAD 120 or consent of instructor.

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 Advanced (Free/pdf. provided)

Instructional videos of lecture provided at www.vertanux1.com



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
Attendance & Participation	<u>100 pts</u>
Total	1000 pts

General Course Outline

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

1. Import and Export – 2D and 3D Translation,
2. Advanced lofting exercise
3. Mold Tools, Cast part, complex draft, setback fillets, and draft analysis.
4. Design Tables
5. SolidWorks Administration
6. Equations
7. Modeling complex parts using sweeps
8. Advanced Sheet Metal. Review Mid-term
9. Mid Term Exam
10. LOFTED SHEET METAL
11. Cylindrical and Conical Sheet Metal Parts, Palette forming tools.
12. Assembly Automation methods
13. Freeform Surfacing
14. Weldments & Structural Steel
15. IGES Translation Repair. Review for Final
16. Final Exam

Required Hardware

16+ Gigabyte USB Flash / Thumb Drive

Software

SolidWorks Educational Edition

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 8:00AM 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.

EXAMS

Midterm and Final exams are to be taken on-site with the teacher or proctor present.

All exams are closed book, note, and video.

Absolutely no cell/smart phones or tables are permitted while taking the exams.

Headphones and music are not permitted during the exams.

LABS

Labs are there to help challenge and sharpen your skills, and are a great resource for additional training. It is recommended you try to complete the labs on your own. "Self-discovery is the best method of educational retention." ~ ~unknown

EXERCISES

All exercises must be completed before the end of the semester as a portfolio.

1. To create a portfolio at the end of each exercise, take a screen capture using 'ctrl-print screen' keys on the keyboard.
2. Then open a word document and paste the image using 'ctrl-v'. Type in the Exercise number next to the image.
3. Send the completed portfolio with your name on the front cover to me via email or hard copy. No more than two exercise images per page.

COMPLETE THE FOLLOWING EXERCISES



1. **EI2 SolidWorks 2012**
Exercise I2 - DWG and DXF conversion to 3D model
-



2. **EI3 SolidWorks 2012**
Exercise I3 - Lofting with guide curves, Threads, Curvature Continous Face Blend Fillets
-



3. **EI4 SolidWorks 2012**
Exercise I4 - Part editing, Molding prep, Draft



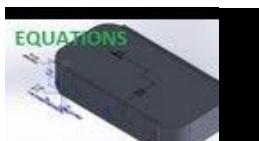
4. **E15 SolidWorks 2012** 11:33

Exercise 15 - Mold Design, Cavity and Core creation, using cavity feature, creating drawings with layers.



5. **E16 SolidWorks 2012** 11:02

Exercise 16 - Design Tables



6. **E17 SolidWorks 2012** 8:59

Exercise 17 - Introduction to Equations inside SolidWorks



7. **E18 SolidWorks 2012** 15:18

Exercise 18 - Sweeps with guide curves, embossed text, multi - thickness shelling



8. **E19 SolidWorks 2012** 14:50

Exercise 19 - Sheet Metal in Top-Down assembly approach



9. **E20 SolidWorks 2012** 24:21

Exercise 20 - Assembly Library Component Automation, Rendering



10. **E21 SolidWorks 2012**

Exercise 21 - Introduction to Surfacing, importing a jpeg image file and tracing around it



II. **E22 SolidWorks 2012**

Exercise 22 - IGES - IGS - STEP import repair. FeatureWorks



I2. **E23 SolidWorks 2012**

Exercise 23 - Cylindrical Sheet Metal parts, Linear sketch patterns



I3. **E24 SolidWorks 2012**

Exercise 24 - Conical Sheet Metal parts using the old 90's method, before the loft option became available. It still works.



I4. **E25 SolidWorks 2012**

Exercise 25 - Lofted Sheet Metal parts



I5. **E26 SolidWorks 2012**

Exercise 26 - Weldments, Structural Steel

CAD 121 TOTALS

E12 – 20pts

L12 - 10

E13 – 20pts

L13 - 10

E14 – 20pts

L14 - 10

E15 – 20pts

L15 - 10

E16 – 20pts

L16 - 10

E17 – 20pts

L17 - 10

E18 – 20pts

L18 - 10

E19 – 20pts

L19 - 10

E20 – 20pts

E21 – 20pts

L21 - 10

E22 – 20pts

E23 – 20pts

E24 – 20pts

E25 – 20pts

E26 – 20pts

L26 - 10

EXERCISES TOTAL 300pts

LAB TOTAL 100pts

MIDTERM 300pts

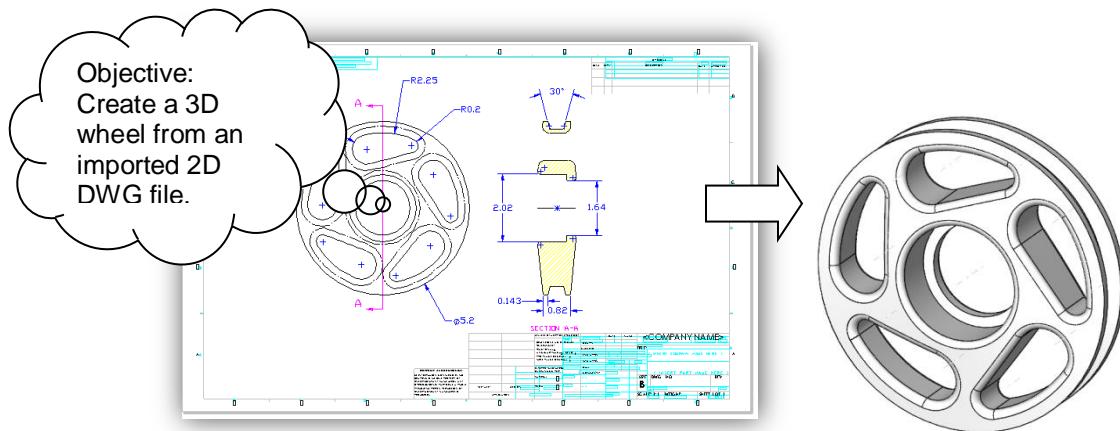
FINAL 300pts

TOTAL 1000pts

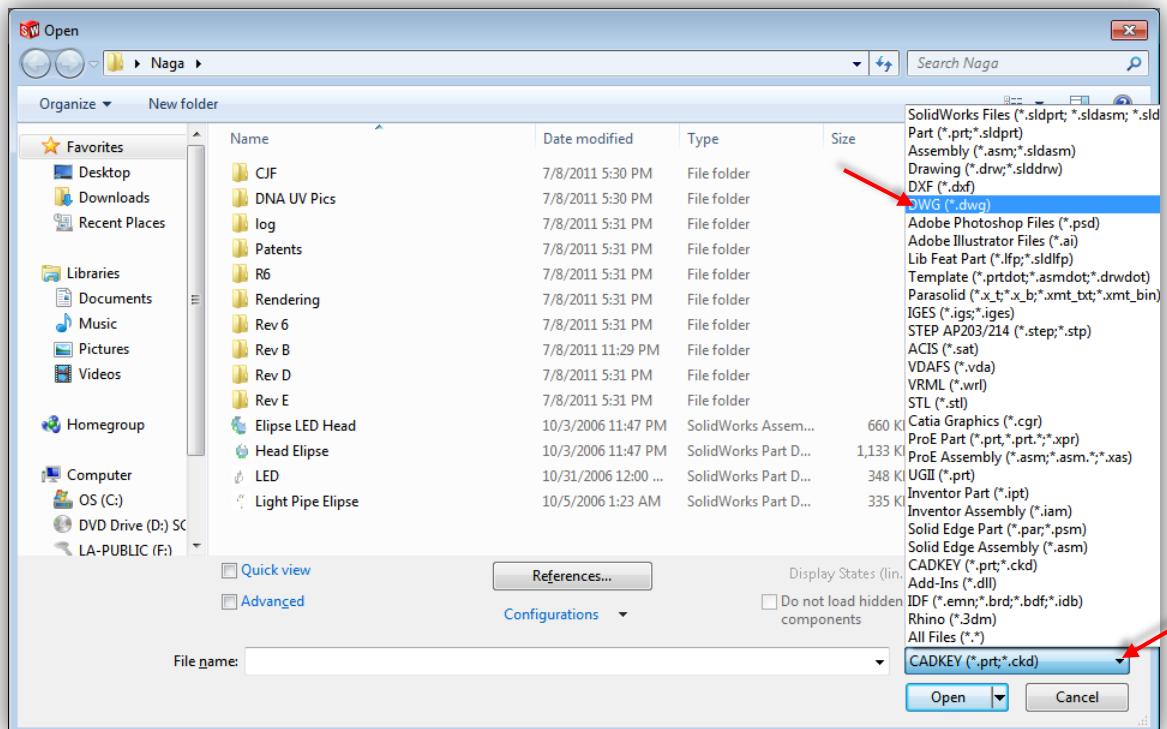
EXERCISE 12

Importing 2D DXF/DWG files

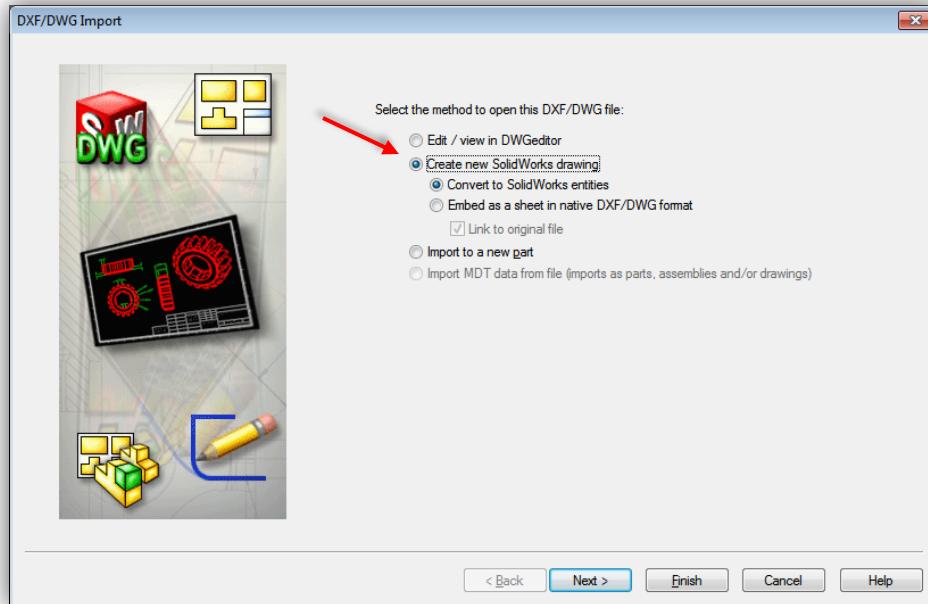
DWG and DXF files can be very useful if imported into SolidWorks.



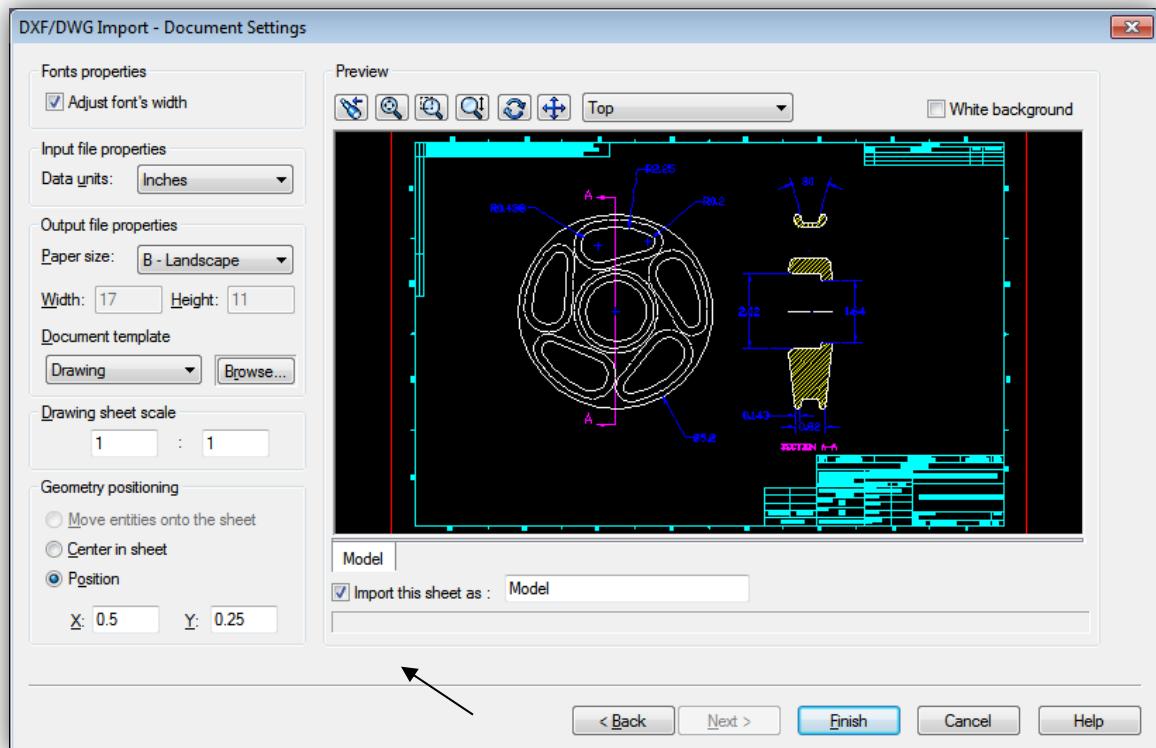
1. Go to file/open and select DWG from the options. Locate the Wheel Hub file.



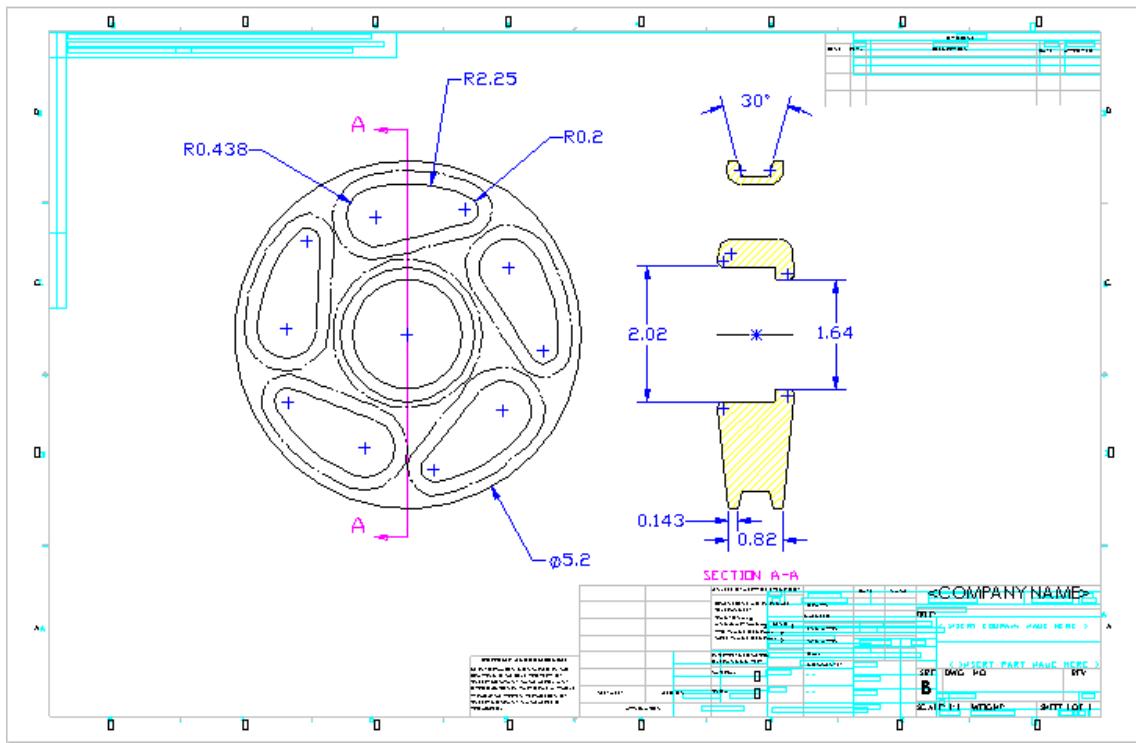
1. Import to a new drawing.



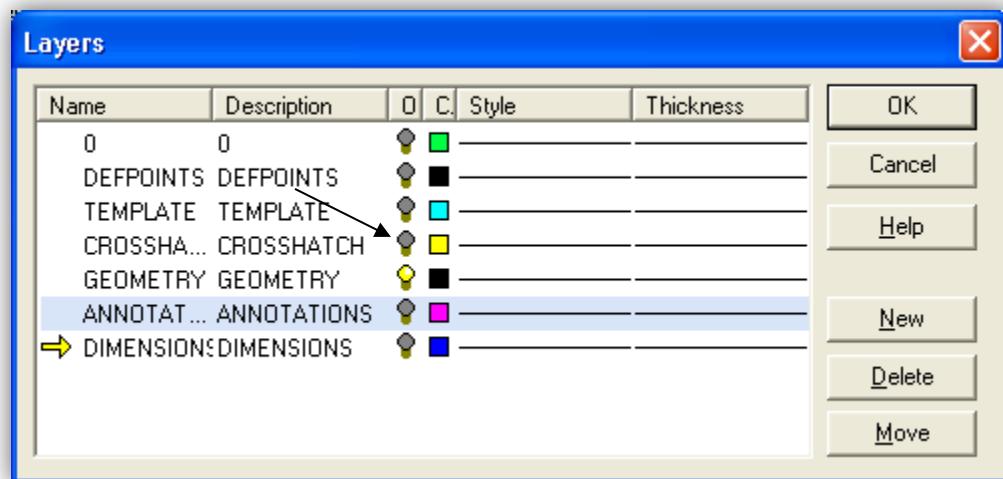
2. The next screen should look like this... Select the show preview box.



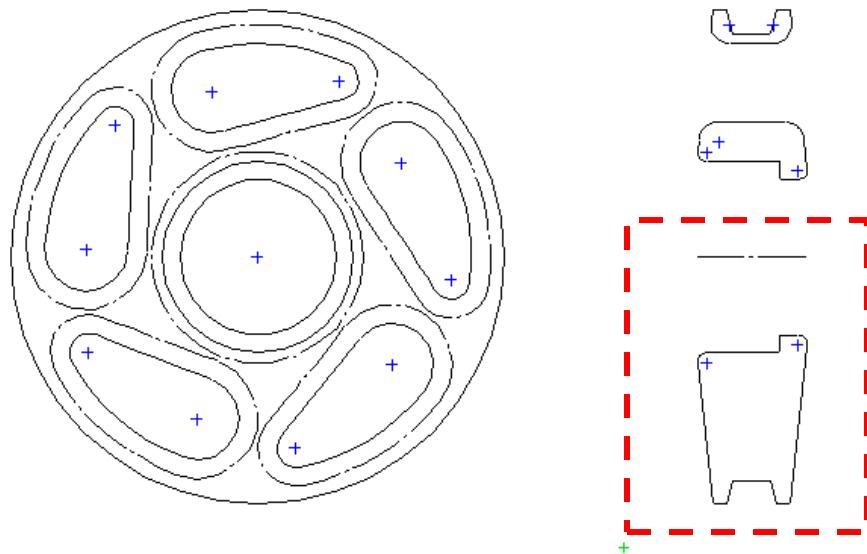
3. Once open the drawing should look like this...



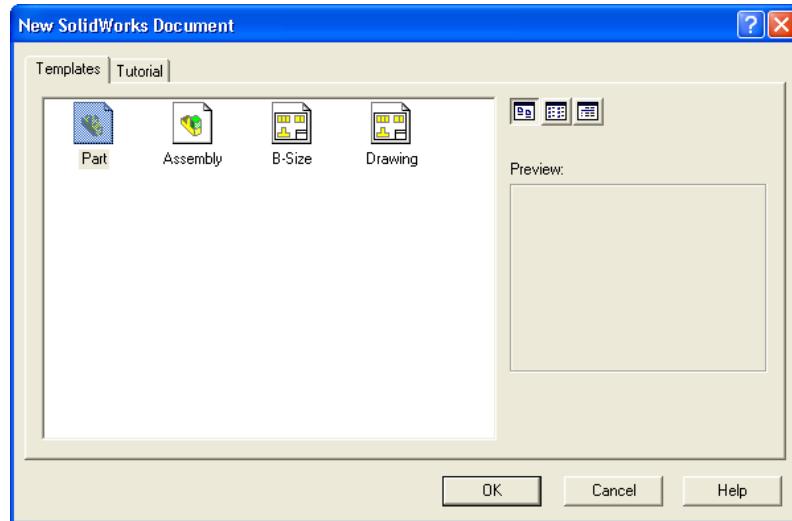
4. Go to the layer icon located on the Line Format toolbar, to disable all but the geometry layer.



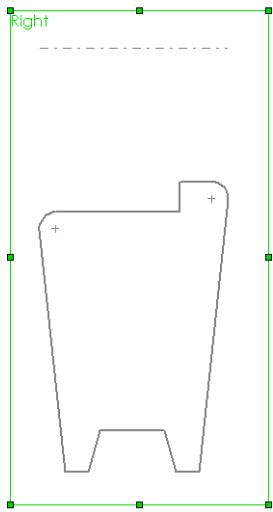
- Once the other layers are disabled your drawing should look like this... Drag a fence around the bottom section view and be sure to include the centerline. Hit CTRL-C to copy the geometry.



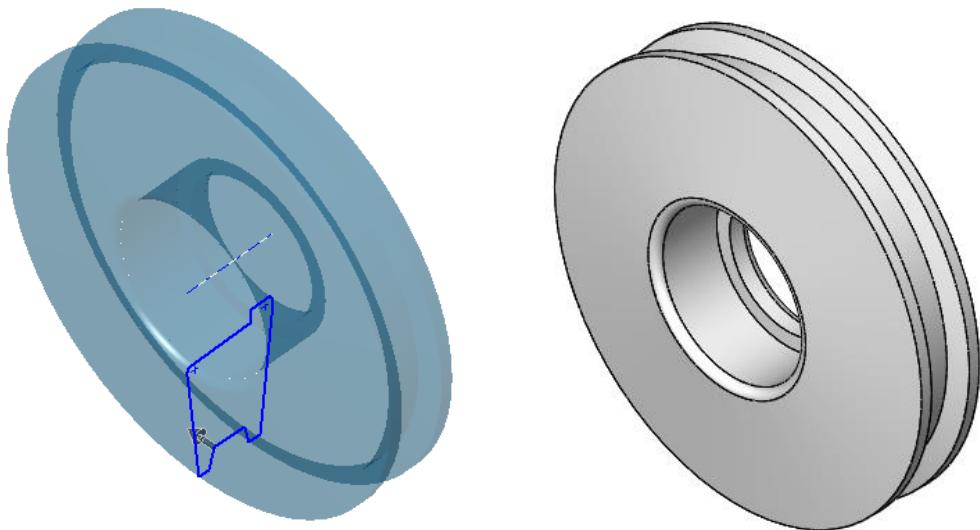
- Go to File/New part. Click on the “Right” plane and hit CTRL-V to paste the geometry into the new part file.



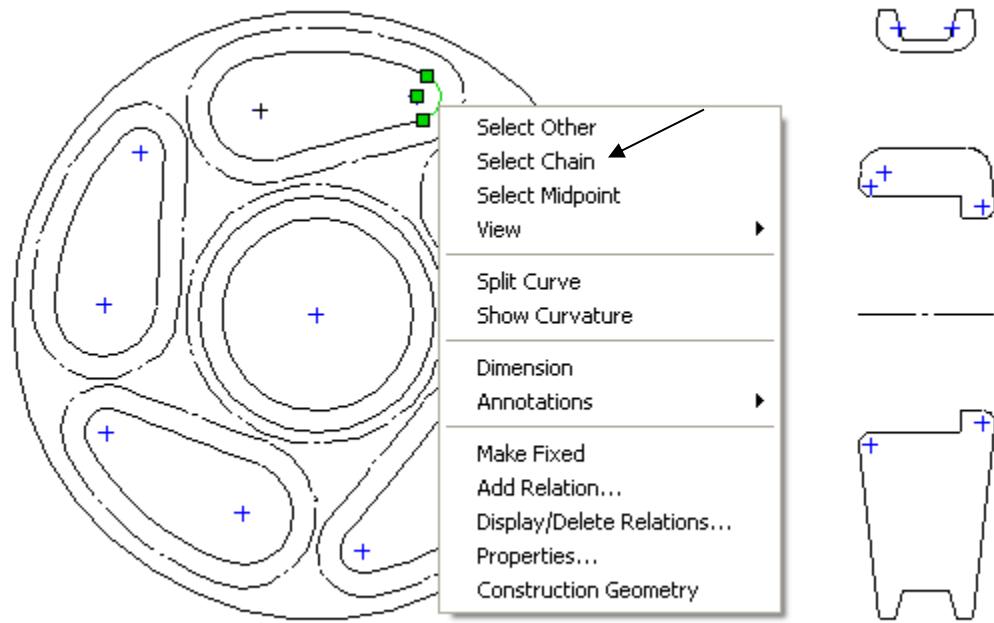
7. Go to the right view orientation and you should have this section view...



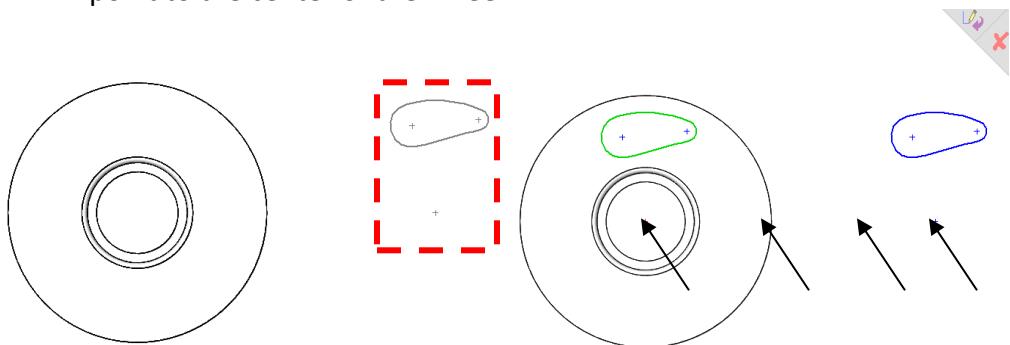
8. Click on the centerline and select the revolve feature. Hit okay.



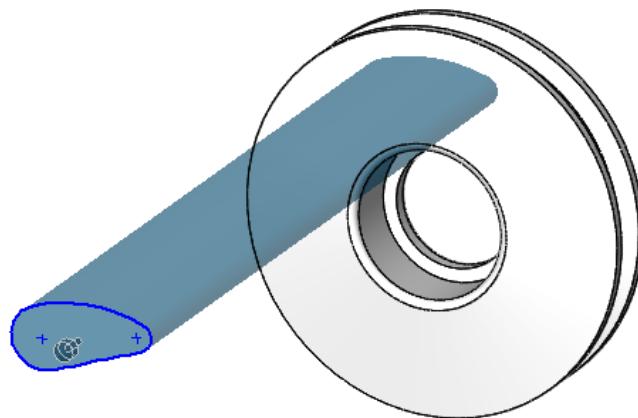
9. Right mouse button click on any portion of one of the slots and “select chain”. Hit the CTRL-C buttons to copy.



10. Open the part file and select the “Front” plane. Hit CTRL-V to paste the slot. It may appear to the right of the wheel. Just edit the sketch and window around the slot sketch. Hold down the CTRL key and drag the main center point to the center of the wheel.



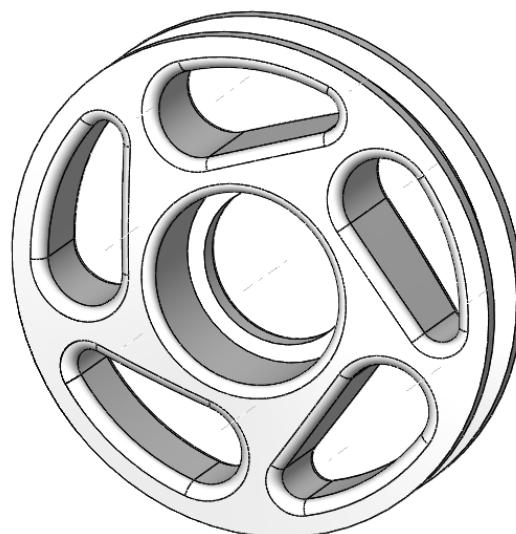
12. Extrude cut through all.



13. Add .125" fillets around the edges of the cutout. Then create a circular pattern from the one cutout.

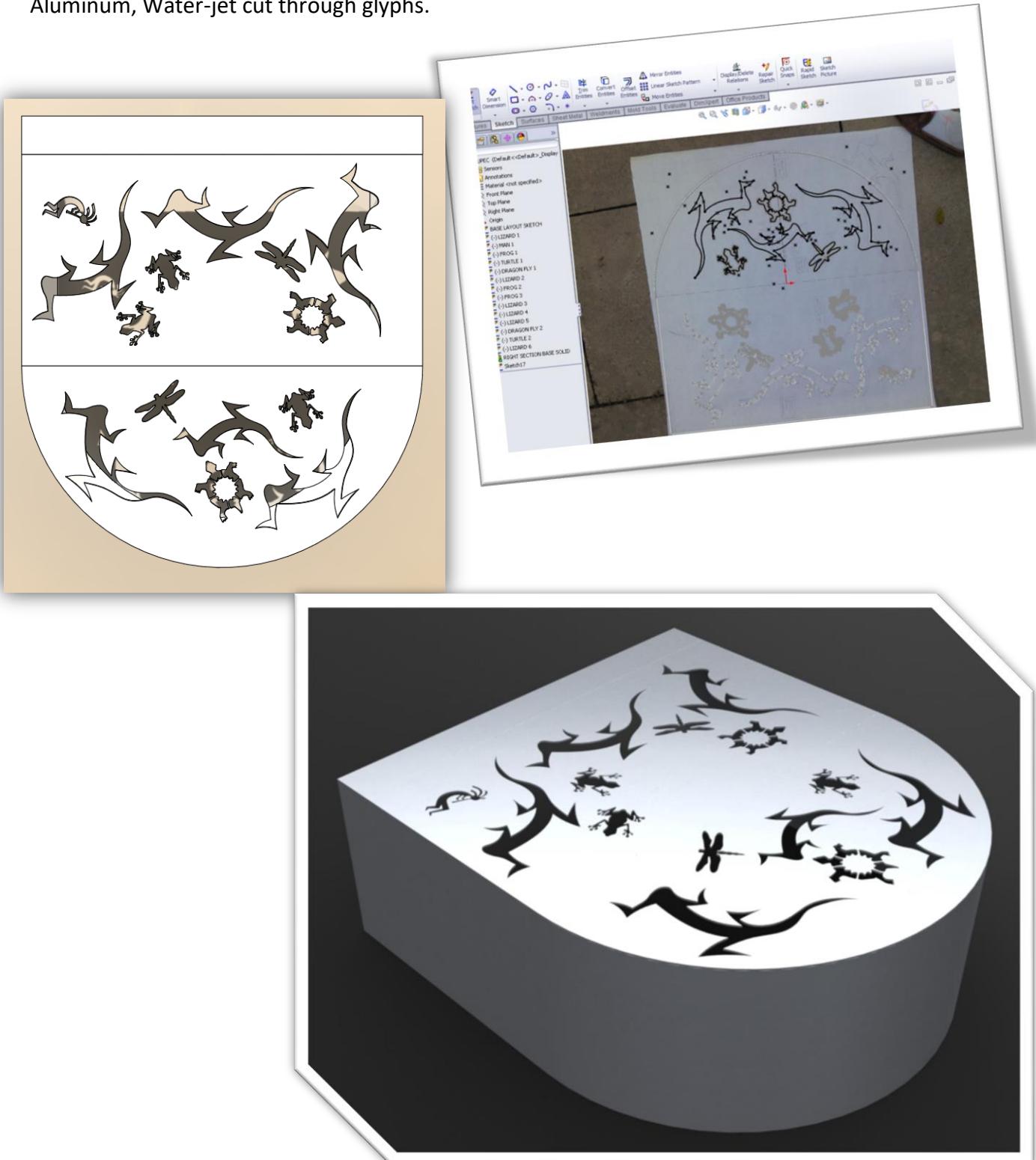


14. The wheel is now completed.

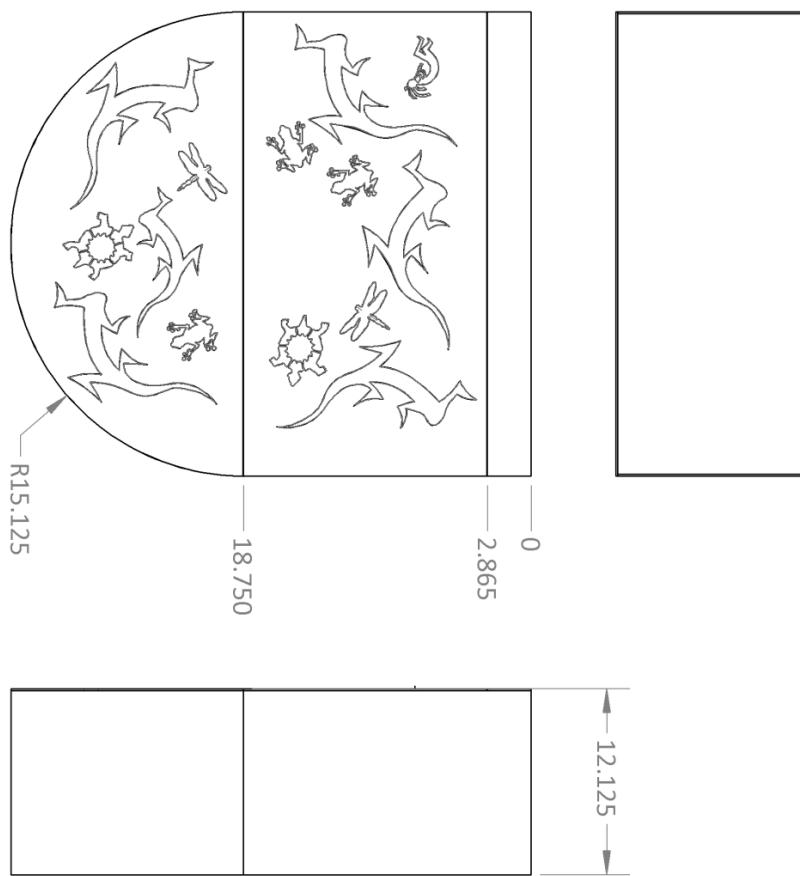


L12 HOSE BASKET PROJECT KUPEC METALS INC.

Import the KUPEC DXF File and create a sheet metal garden hose enclosure. .125" Thick.
Aluminum, Water-jet cut through glyphs.



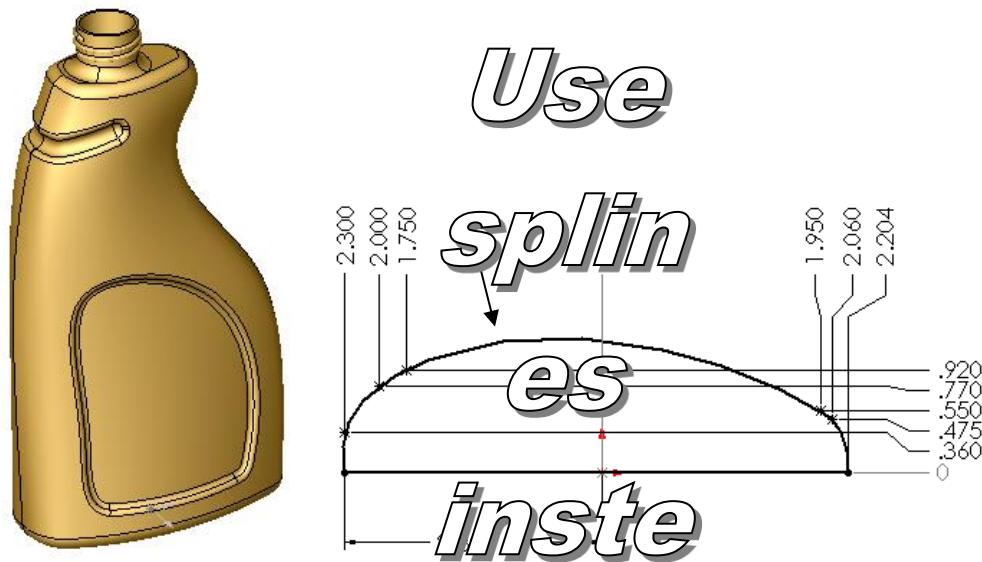
<p>PROPRIETARY AND CONFIDENTIAL</p> <p>THE INFORMATION CONTAINED IN THIS DRAWING IS THE SOLE PROPERTY OF <INSERT COMPANY NAME HERE>. ANY REPRODUCTION IN PART OR AS A WHOLE WITHOUT THE WRITTEN PERMISSION OF <INSERT COMPANY NAME HERE> IS PROHIBITED.</p>			
<p>UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL: ± ANGULAR: MACH: ± TWO PLACE DECIMAL: ± THREE PLACE DECIMAL: ± INTERPRET GEOMETRIC TOLERANCING PER: MATERIAL</p> <p>DRAWN CHECKED ENG APPR. MFG APPR. Q.A.</p> <p>COMMENT: ARN KUPEC</p>			
<p>NEXT ASSY USED ON FINISH</p> <p>APPLICATION DO NOT SCALE DRAWING</p>			
<p>SCALE: 1:12 WEIGHT: A HOSE BASKET</p>		<p>SIZE DWG. NO. A REV</p>	
<p>SCALE: 1:12 WEIGHT: SHEET 1 OF 1</p>			
5	4	3	2



EXERCISE 13

Lofting with Guide Curves

- Sketch the geometry as show below on the “Front” plane. Rebuild. (5 spline points) Use ordinate dimensions for simplicity.



- Offset a plane 8" from the front plane

ad of

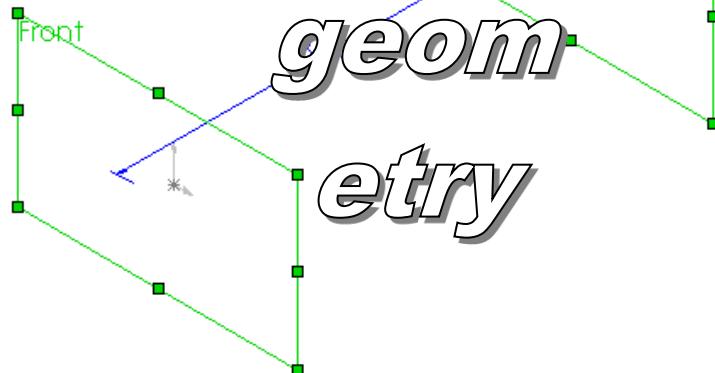


analy

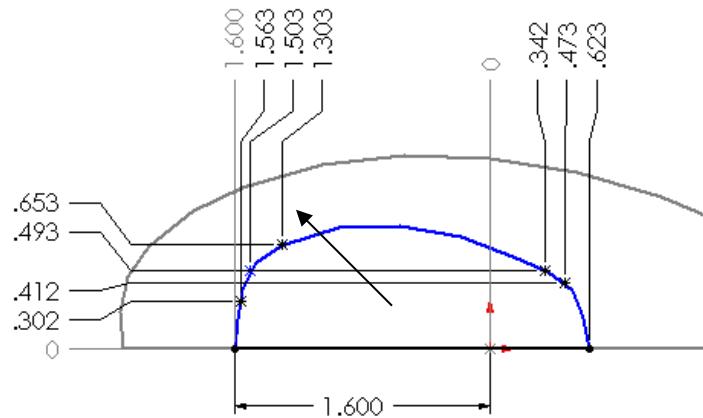
tical

geom

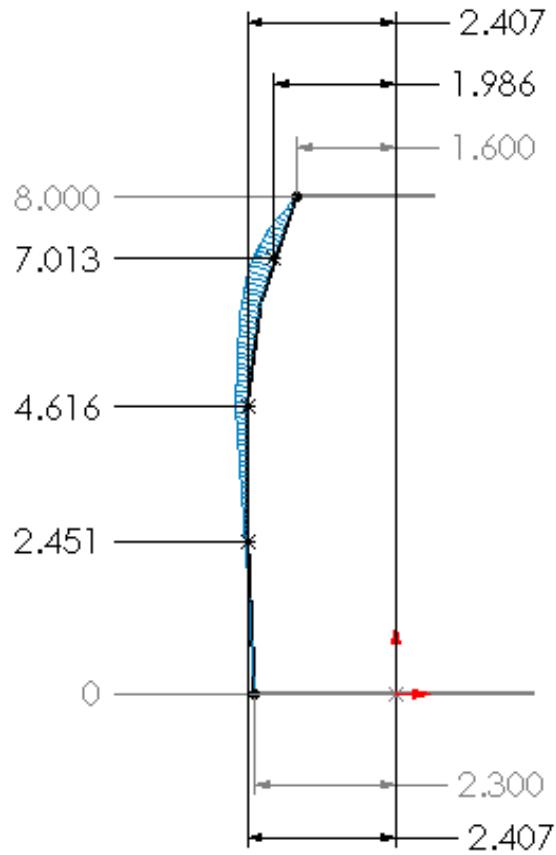
etry



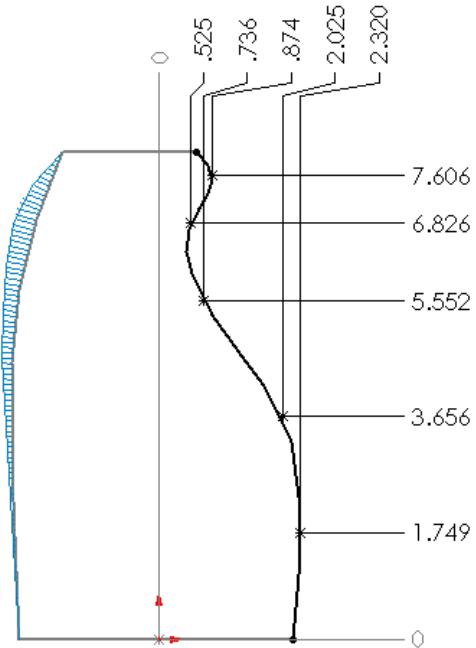
3. Sketch the following on Plane 1. (5 spline points)



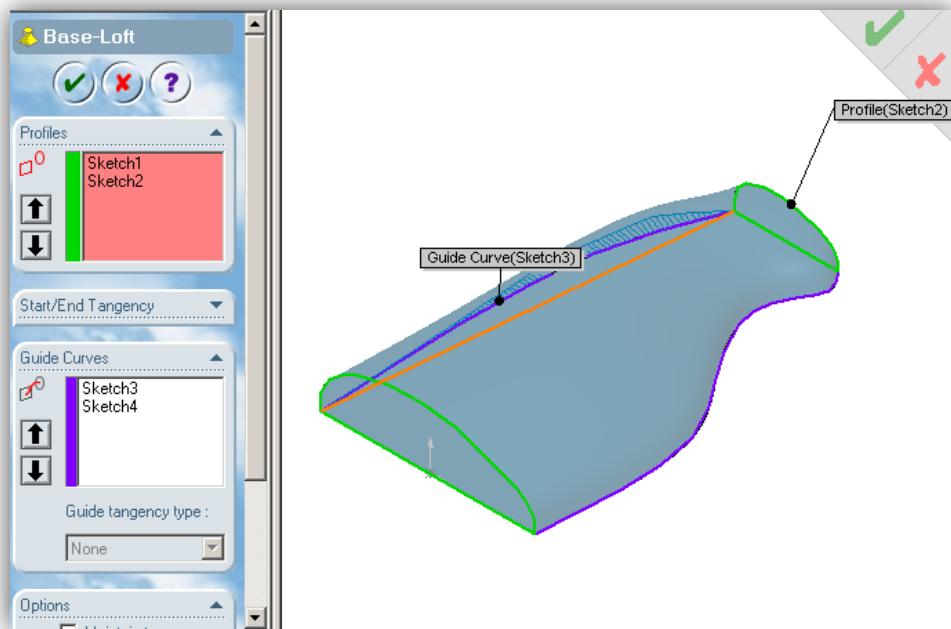
4. Start a sketch on the top plane, and draw the following. (3 spline points)



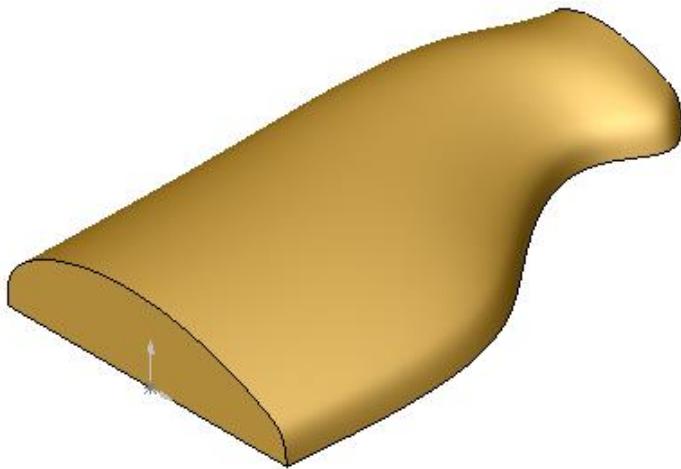
5. Start a new sketch on the top plane and draw the following. (5 spline points)



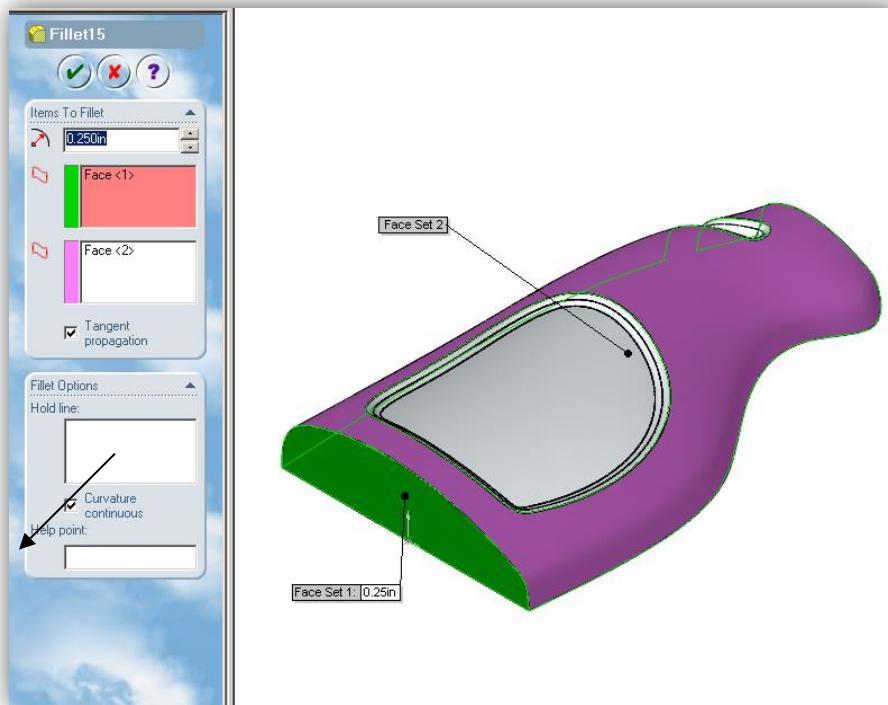
6. Loft



7. Loft completed.



8. **Creating Curvature Continuous Face Blend Fillets.** Select the bottom face and side face of the bottle. Select the fillets icon. Input $.250"$ radius and check curvature continuous.



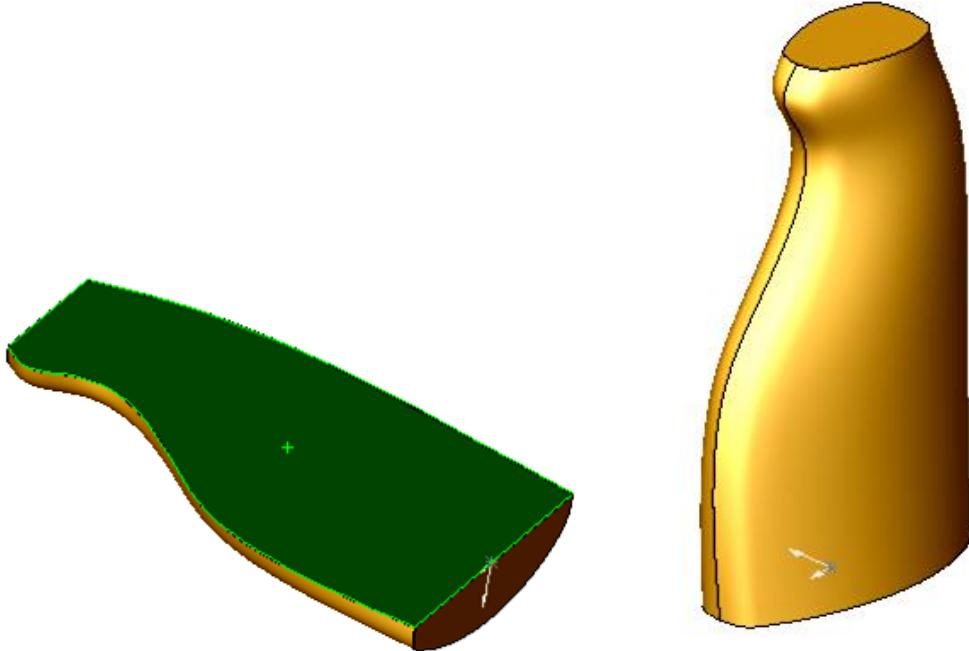
What are Curvature Continuous fillets?



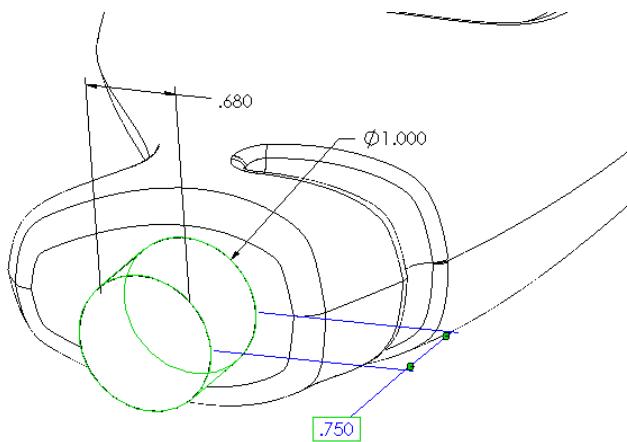
Click [Curvature continuous](#) to resolve discontinuity problems and create a smoother curvature between adjacent surfaces. To verify the effect of the curvature continuity, you can display [Zebra Stripes](#). You can also analyze the curvature using the [curvature](#) tool.

Curvature continuous fillets differ from standard fillets in the following ways. They have a spline cross-section as opposed to a circular cross-section. Curvature continuous fillets are smoother than standard fillets because there is no jump in curvature at the boundary. Standard fillets include a jump at the boundary because they are tangent continuous at the boundary.

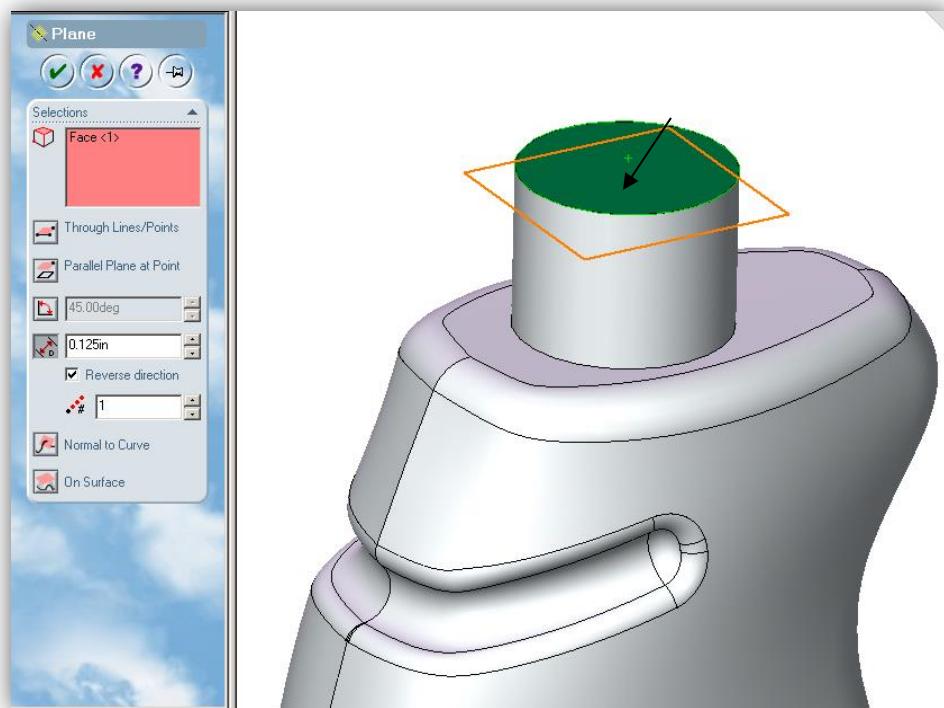
9. Select the back flat face and go to Insert/Mirror all.



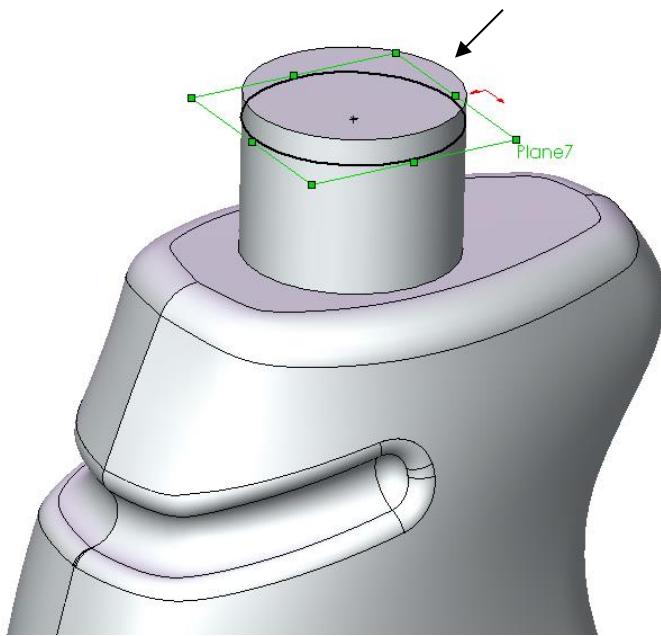
10. Insert the neck of the bottle as shown below.



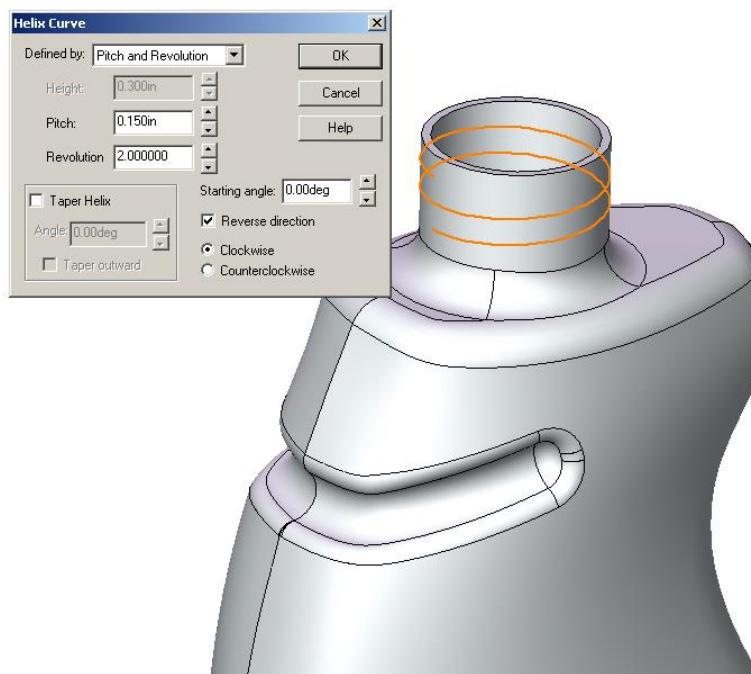
11. **Creating a Thread** - Select the top face of the neck and go to the plane wizard. Offset a plane .125" from the top.



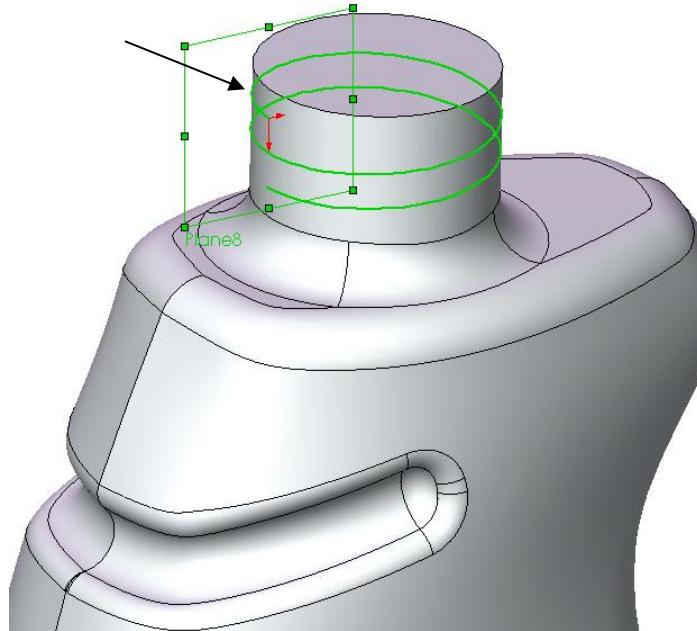
12. Start a sketch on the new offset plane. Select the top face of the neck and convert entities.



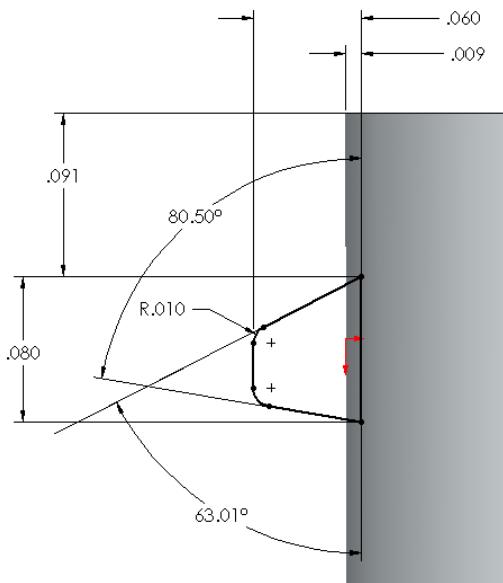
13. Go to Insert, Curve, Helix/Spiral. Apply the following parameters.



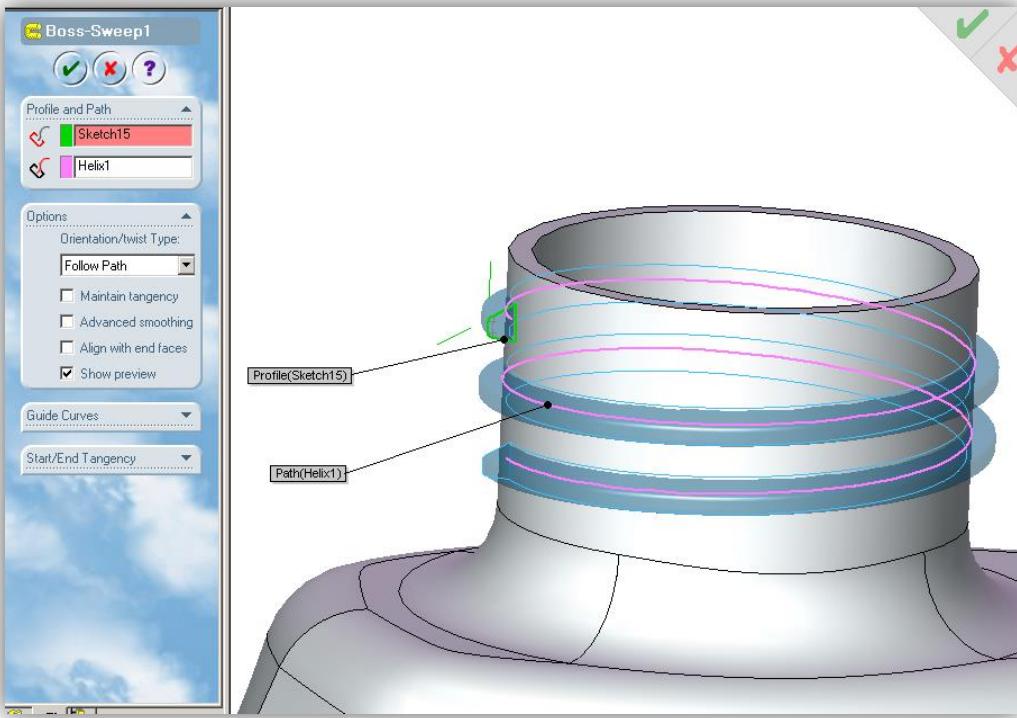
14. Now select the sketch tool and click on the portion of the helix closest to the top end point. This automatically creates a new plane perpendicular to the helix and starts a sketch on it.



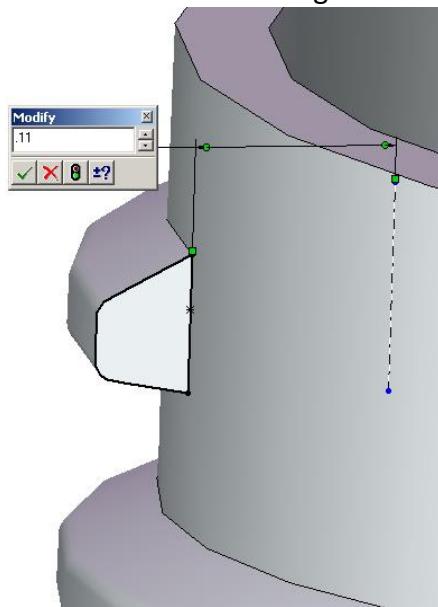
15. Now just draw the geometry of the thread.



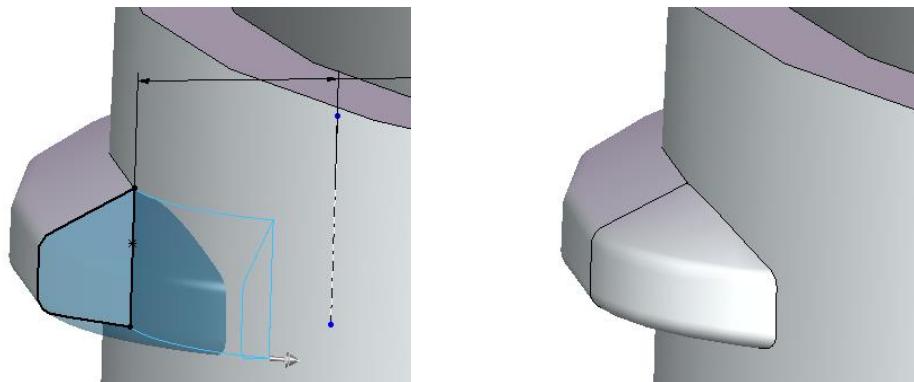
16. Rebuild, and go to the Sweep feature. Select Path and Profile. Finish.



17. Select the end face of the thread, start a sketch and Convert Entities. Draw a vertical centerline .110" offset from the edge.



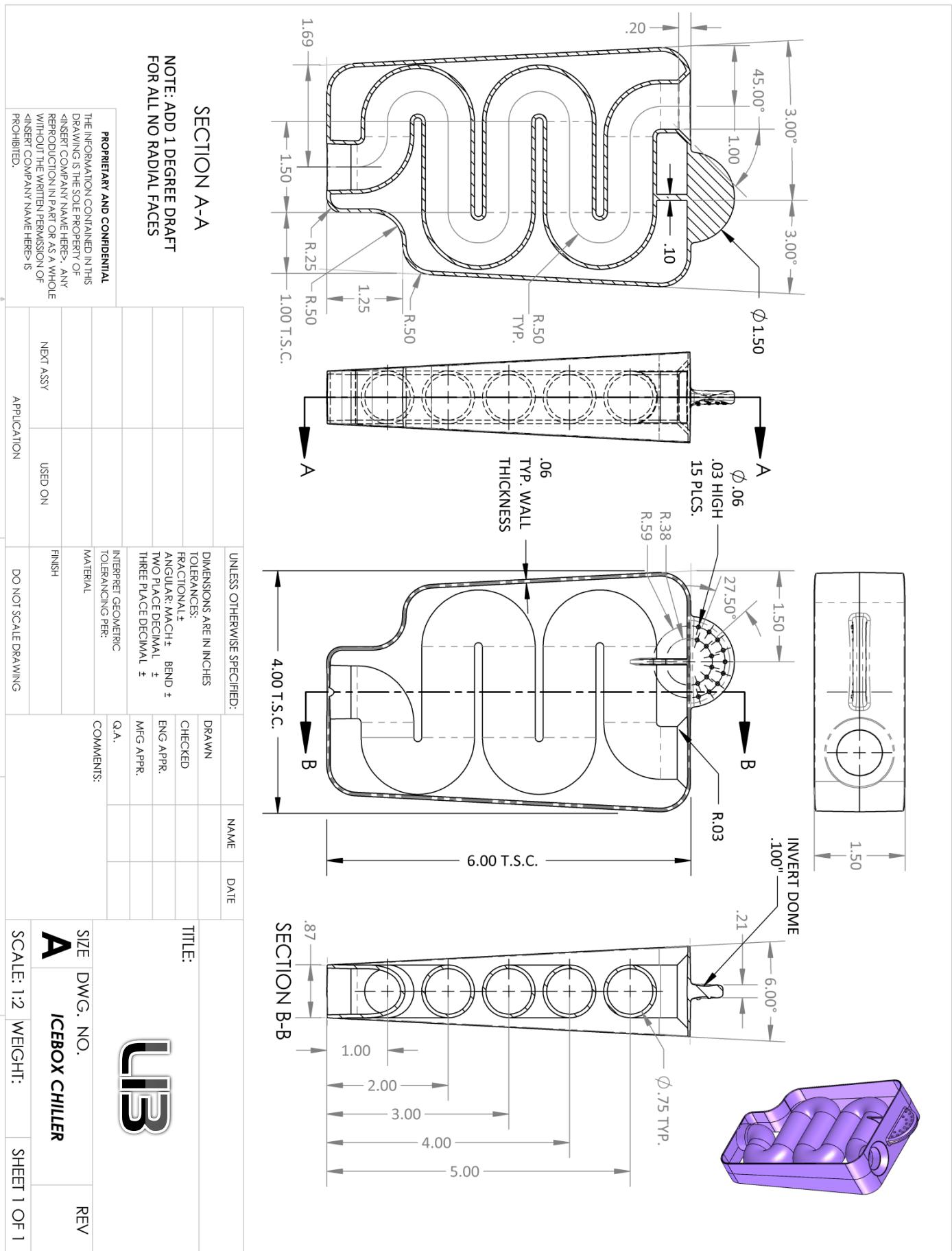
18. Revolve 56°.



19. Complete the other side the same way. Add additional features to finish bottle. Shell at .050".



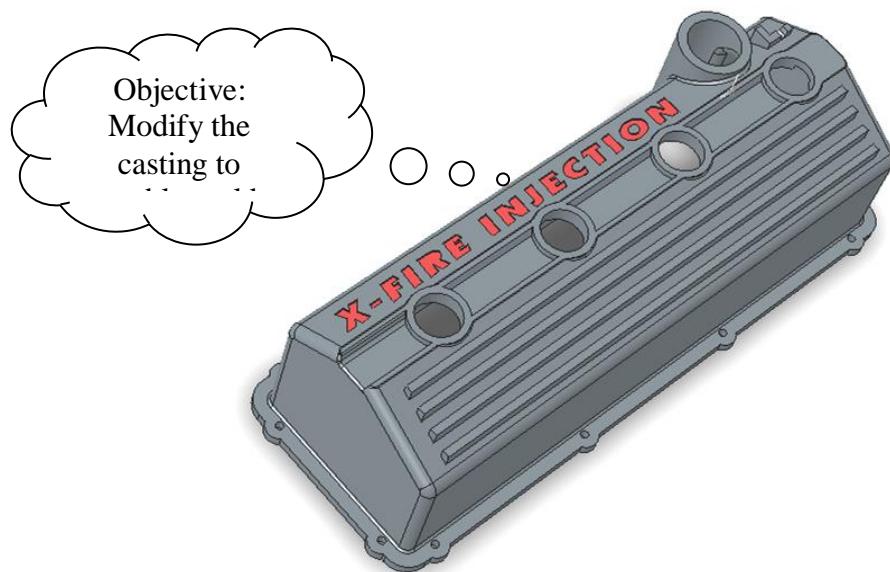
Finished



EXERCISE 14

Draft and Filleting

Cast parts can be some of the most difficult to modify if draft and complex fillets have not been inserted early on with design intent.



11. Using the Mold Tools toolbar.





12. Draft Angle Analysis.

Face Classification (source: SolidWorks help)

In the graphics area, each face displays a color based on the **Draft angle** you selected.

Draft analysis results listed under **Color Settings** are grouped into four categories,

when you specify **Face classification**:

Positive draft. Displays any faces with a positive draft, based on the reference draft angle you specified.

A positive draft means the angle of the face, with respect to the direction of pull, is more than the reference angle.

Negative draft. Displays any faces with a negative draft, based on the reference draft angle you specified.

A negative draft means the angle of the face, with respect to the direction of the pull, is less than the negative reference angle.

Draft required. Displays any faces that require correction.

These are faces with an angle greater than the negative reference angle, and less than the positive reference angle.

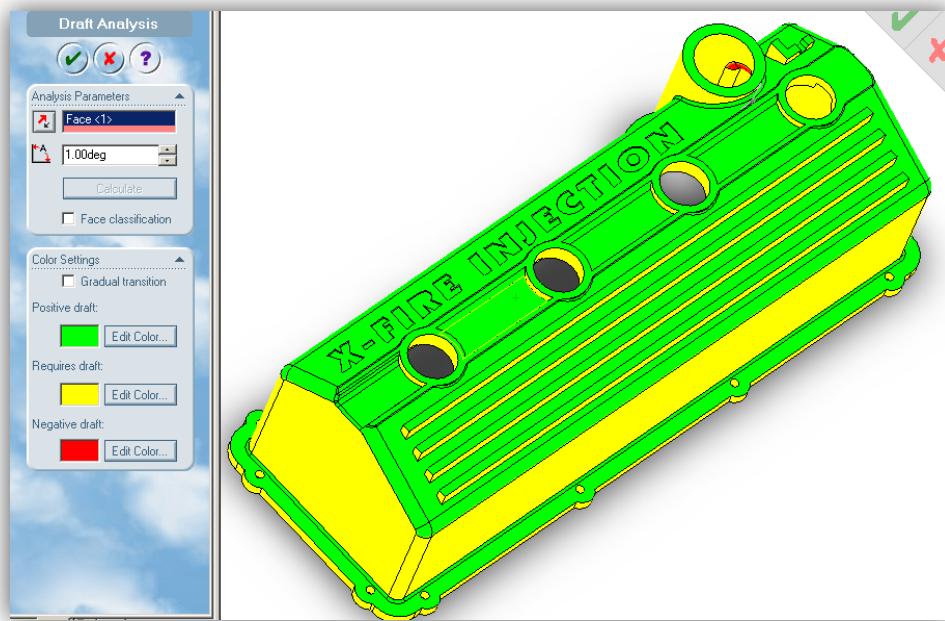
Straddle faces. Displays any faces that contain both positive and negative types of draft.

Typically, these are faces that require you to create a split line.

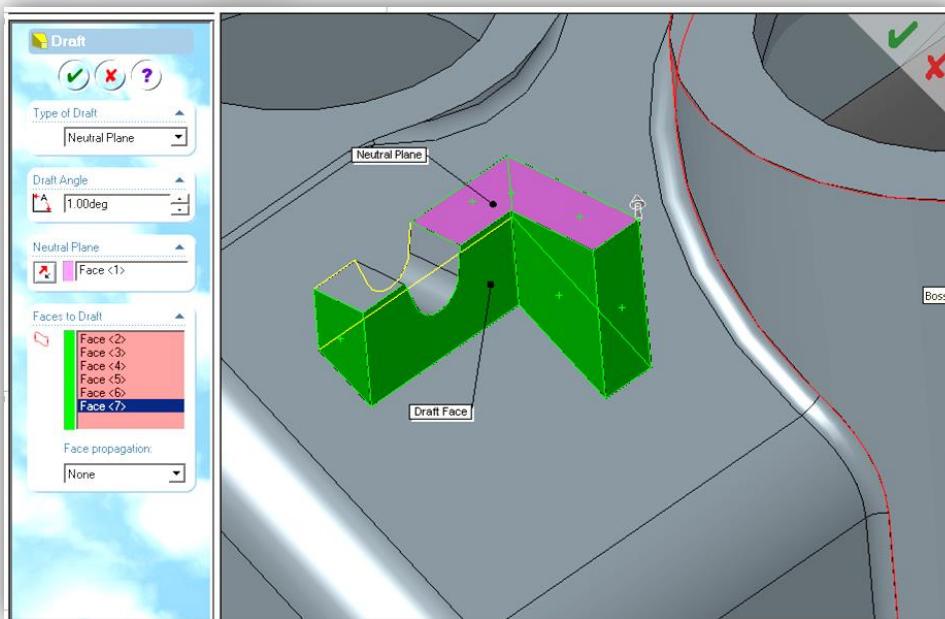
NOTE: Display colors may vary. The first time you use draft analysis, the system uses default colors.

If you modify the colors, the system will use the new colors you specified.

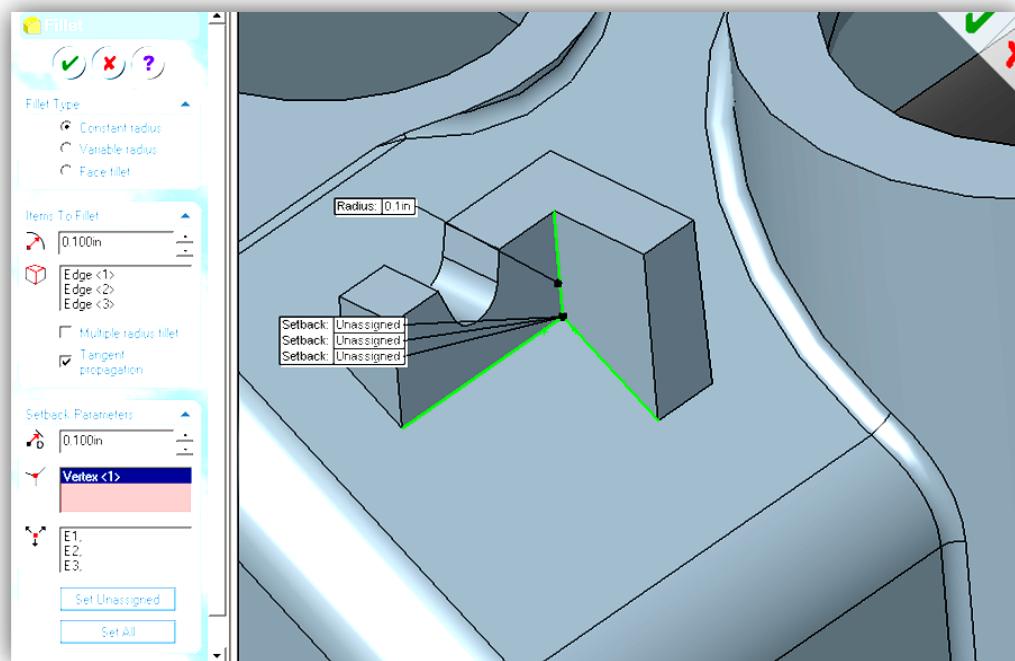
13. Here we can see that no draft is present on the current part.



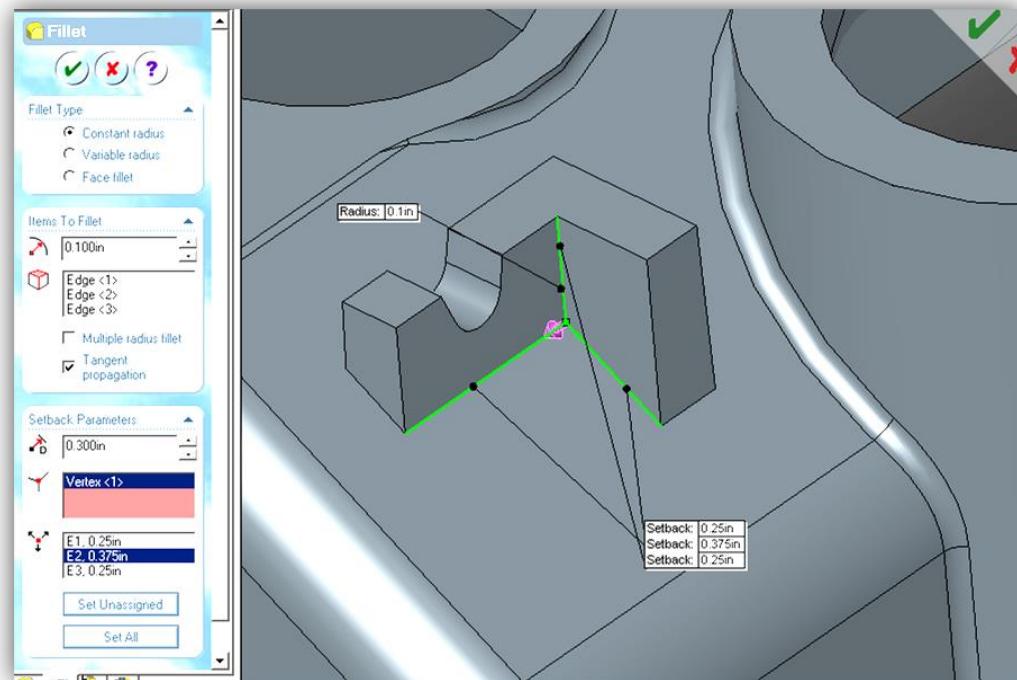
14. Applying Draft. Select the Draft icon, and then select the top planar face of the object.



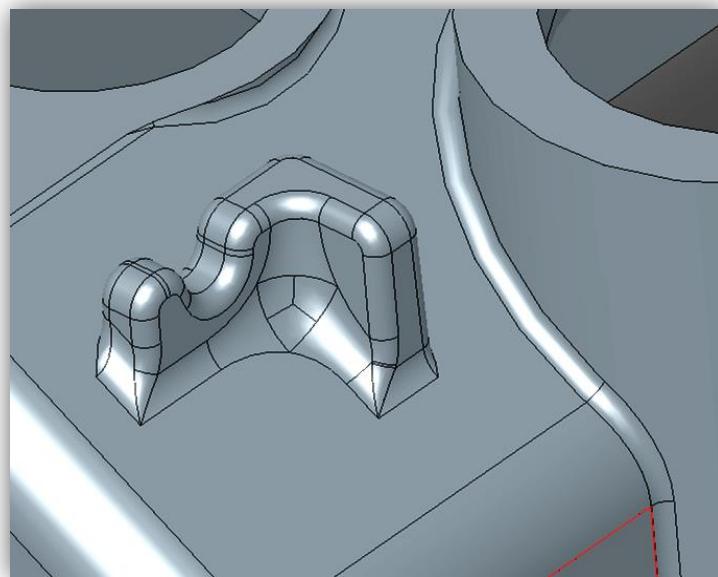
15. Applying “setback” fillets. Select the three intersecting edges, and then select the intersection vertex.



6. Apply the following parameters.

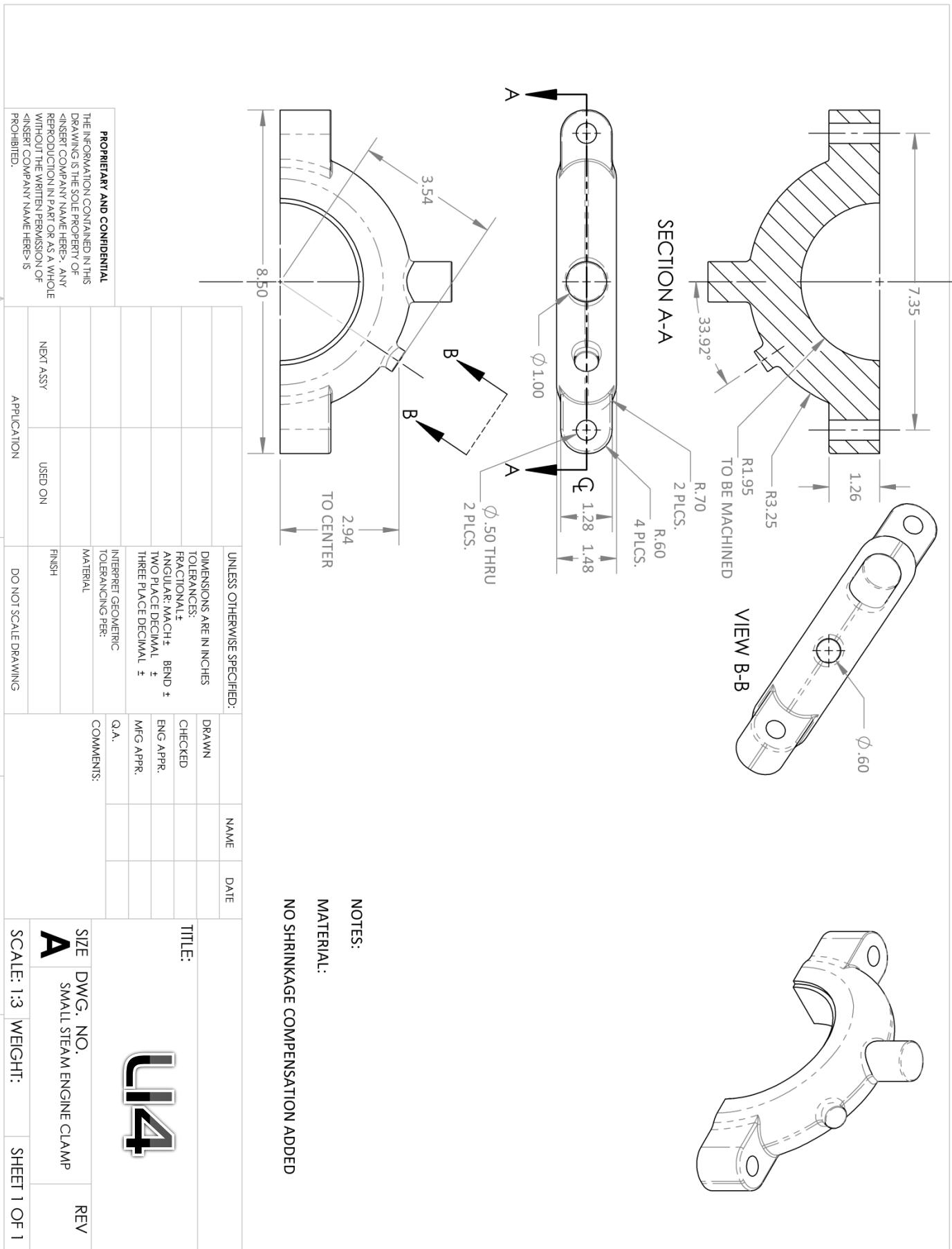


16. Add draft and fillets where required. Run a draft check to ensure all faces have draft. Typical Radii - .125", .060". Typical Draft 1°.



17. This is how the finished model should appear after rendering.

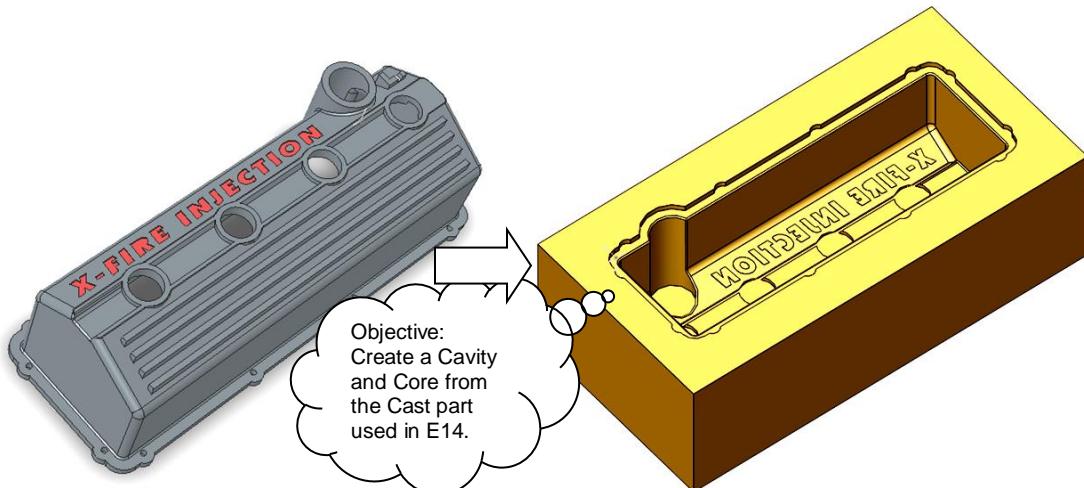




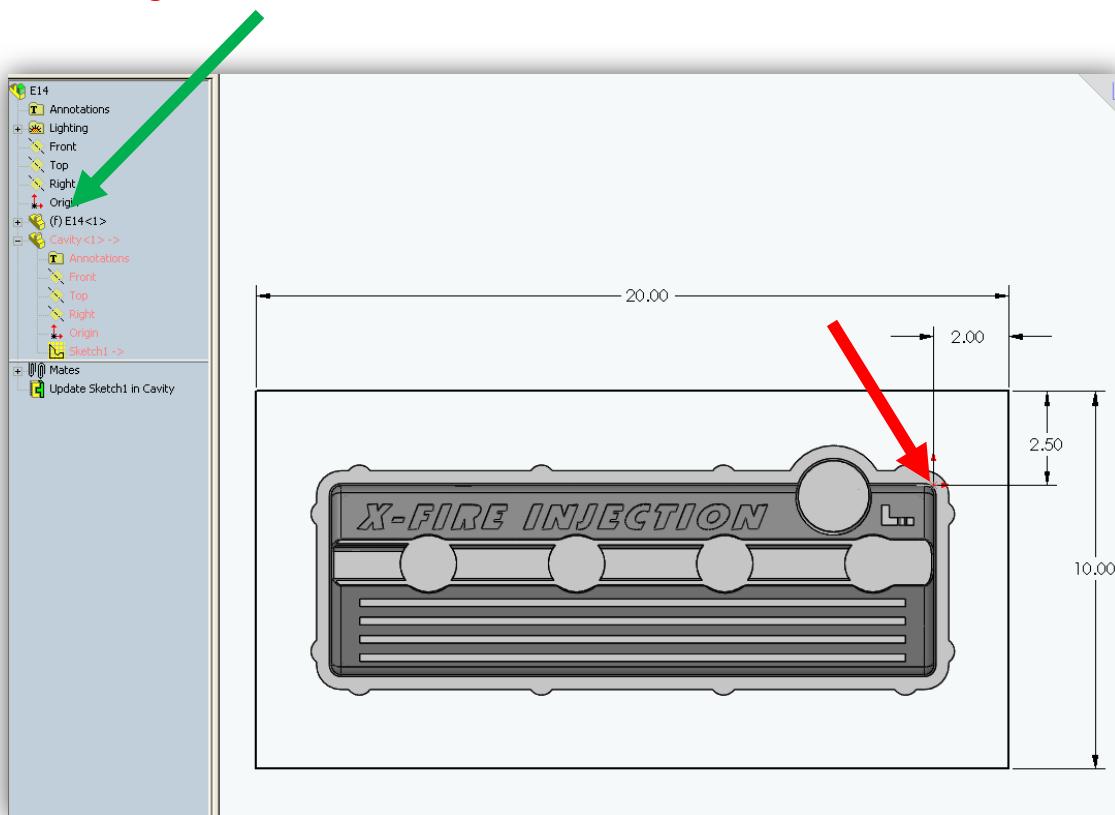
EXERCISE 15

Cavity & Core Creation

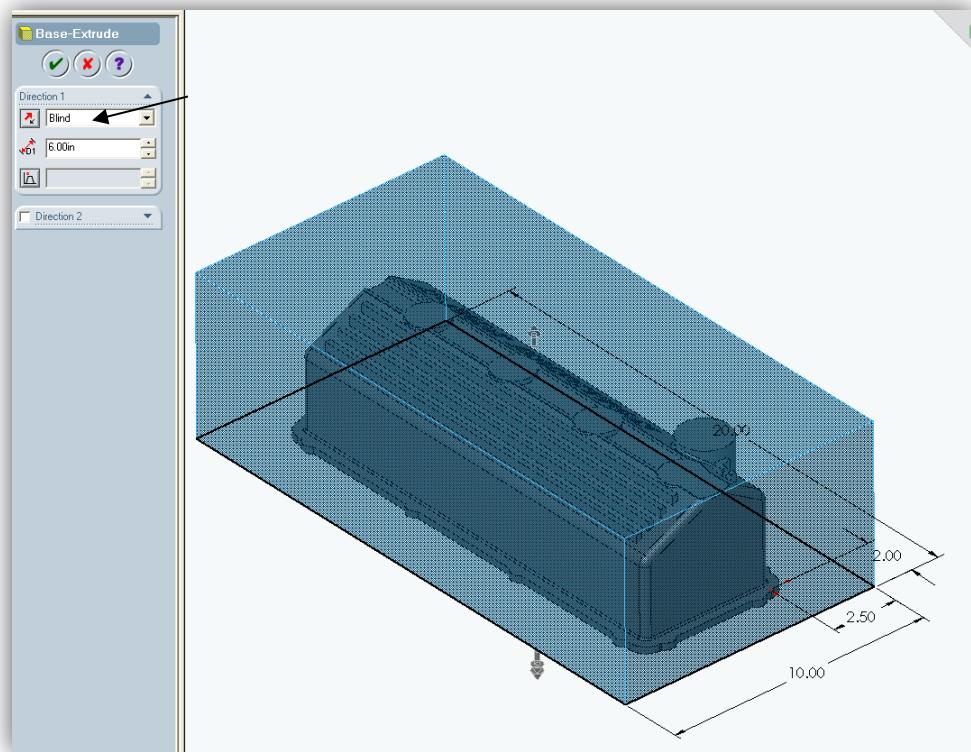
Cavities and Cores can be created using SolidWorks.



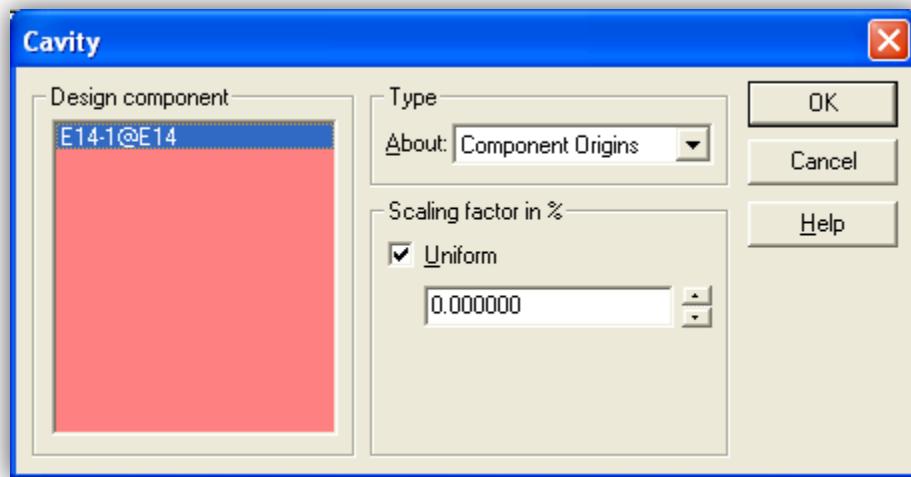
18. Start a new assembly and name it E15. Insert the E14 Finished part file into the new assembly at the origin. Insert a new part, name it Cavity, and drop it onto the “**Top**” plane of the E15 assembly. Draw the following. Note the **origin** location.



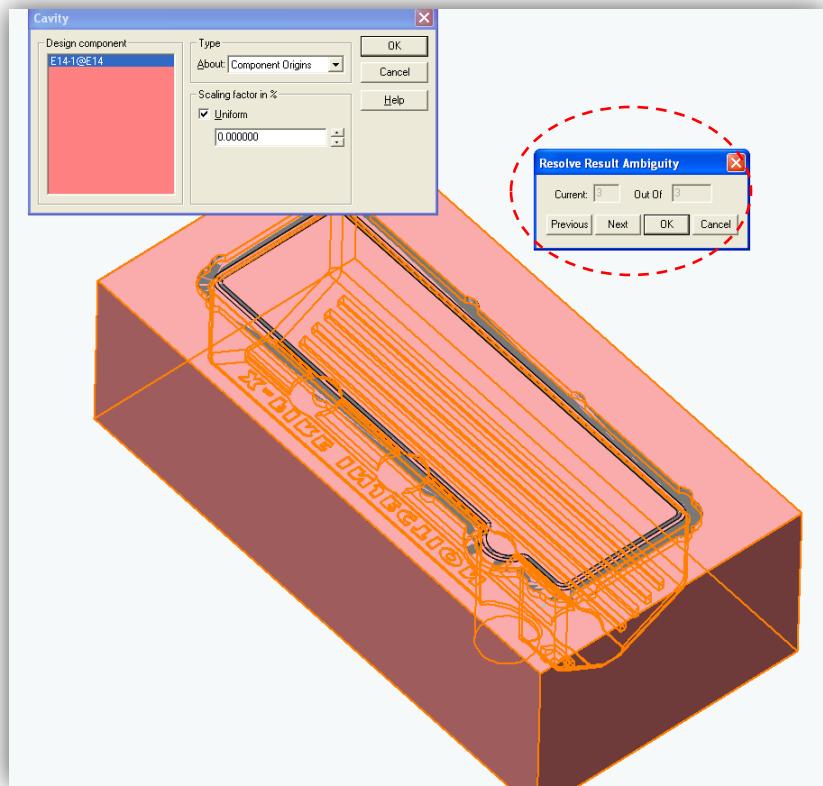
19. Extrude-boss 6".



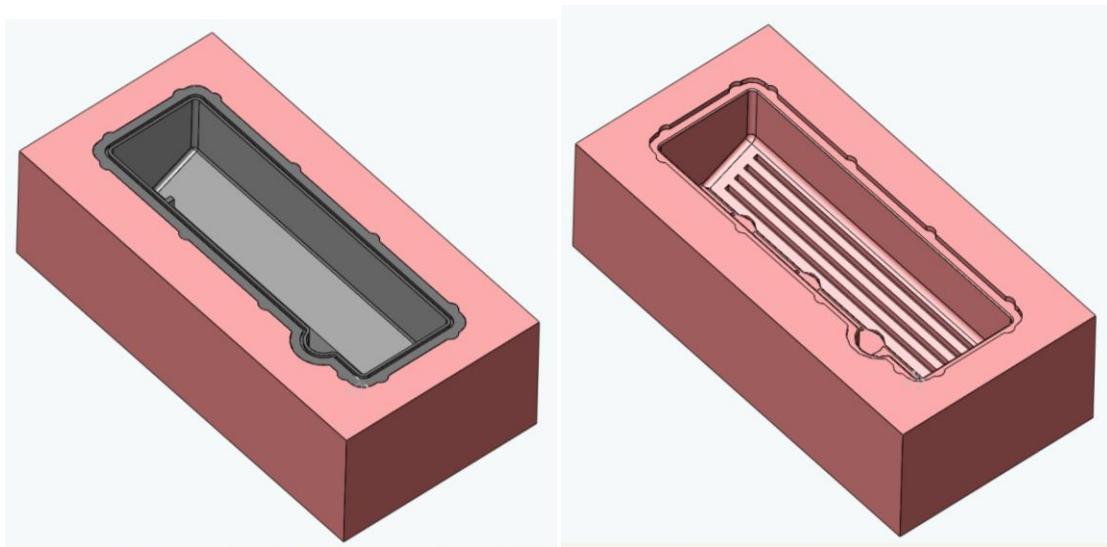
20. Select the Cavity Icon. The next screen should look like this... Select the E14 part file either from the screen or feature tree.



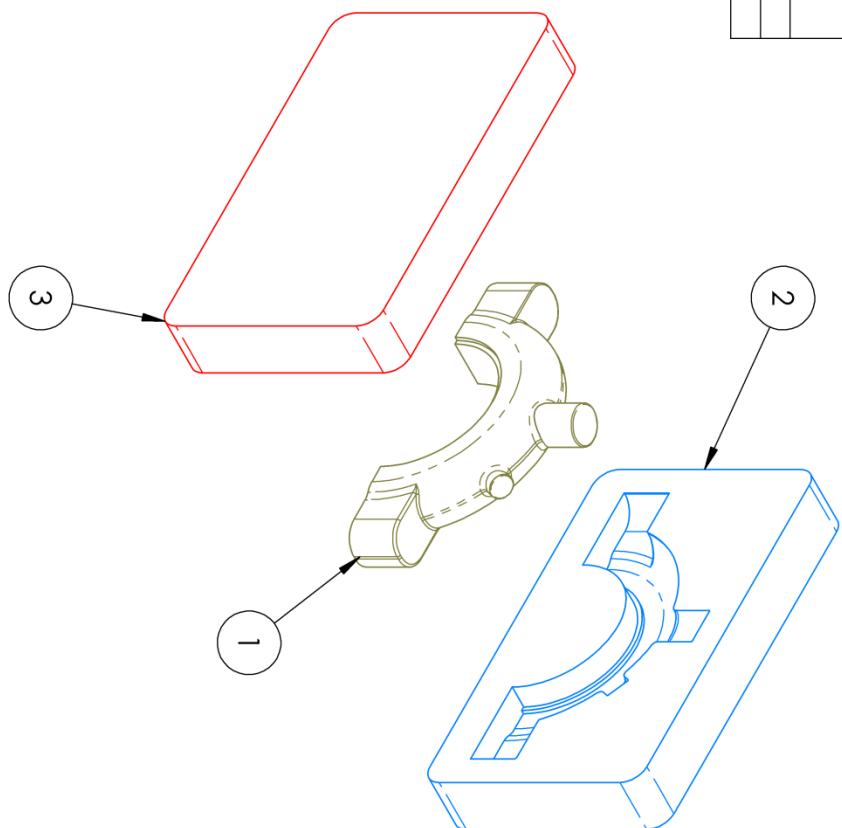
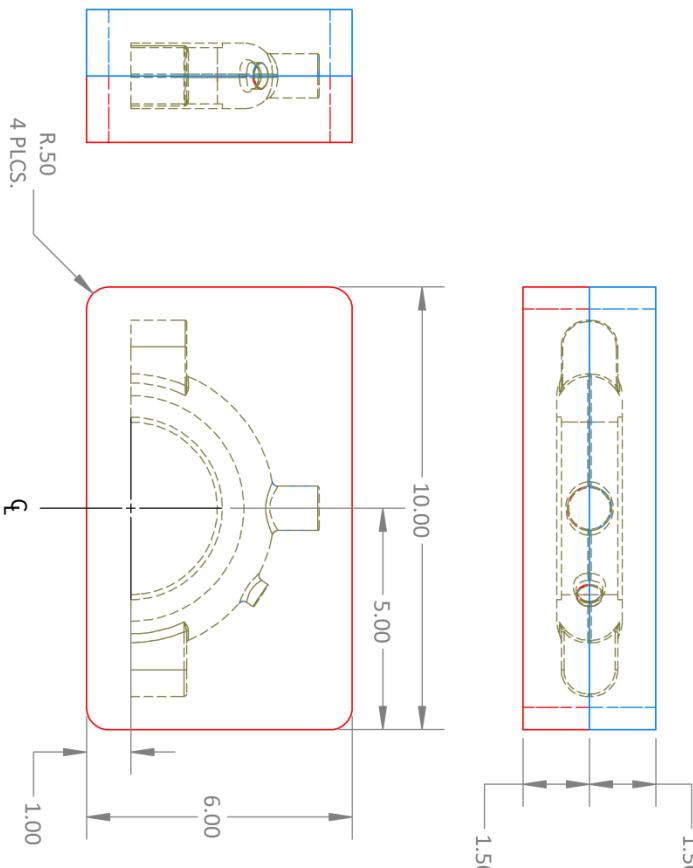
21. Once the cavity has been initiated you will get the “Resolve Result Ambiguity” box. Just hit okay. *This allows you to select the cavity or core separations available.*



22. Hide the E14 component and you should now see the cavity.



ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	SMALL STEAM ENGINE CLAMP 2		1
2	CAVITY		1
3	CAVITY SIDE 2		1

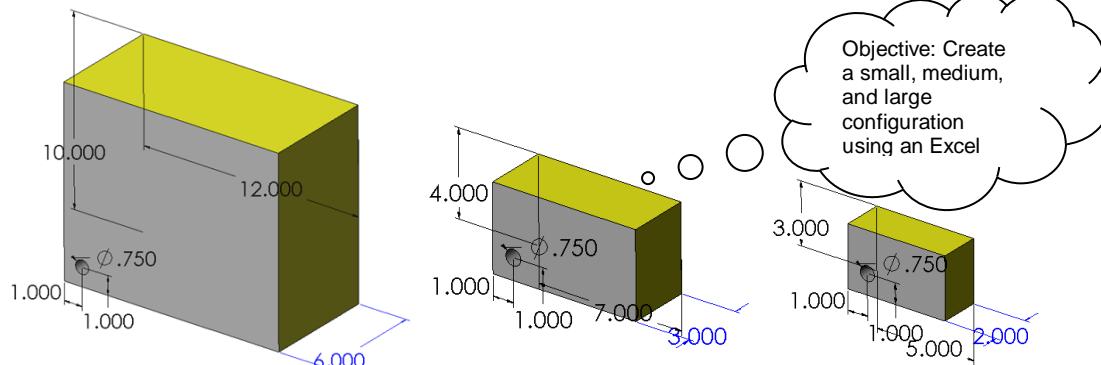


PROPRIETARY AND CONFIDENTIAL	
THE INFORMATION CONTAINED IN THIS DRAWING IS THE SOLE PROPERTY OF <INSERT COMPANY NAME HERE>. ANY REPRODUCTION IN PART OR AS A WHOLE WITHOUT THE WRITTEN PERMISSION OF <INSERT COMPANY NAME HERE> IS PROHIBITED.	
NEXT ASSY	USED ON
APPLICATION	FINISH
DO NOT SCALE DRAWING	
SIZE A DWG. NO. REV	
SCALE: 1:4 WEIGHT: SHEET 1 OF 1	

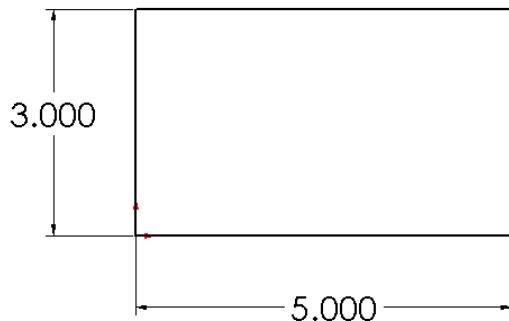
EXERCISE 16

Configurations with Design Tables

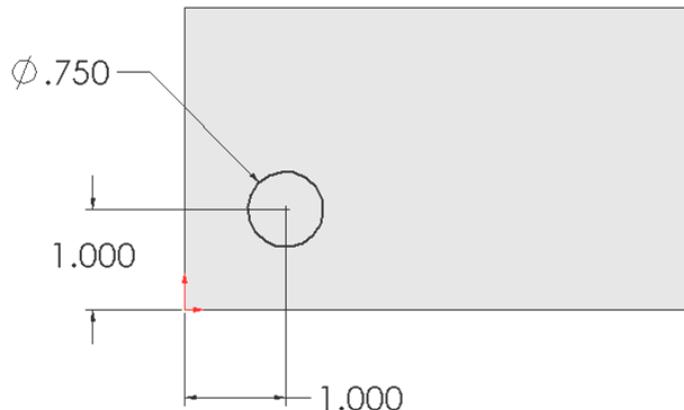
Design Tables can be very useful for designing multiple variations of the same part.



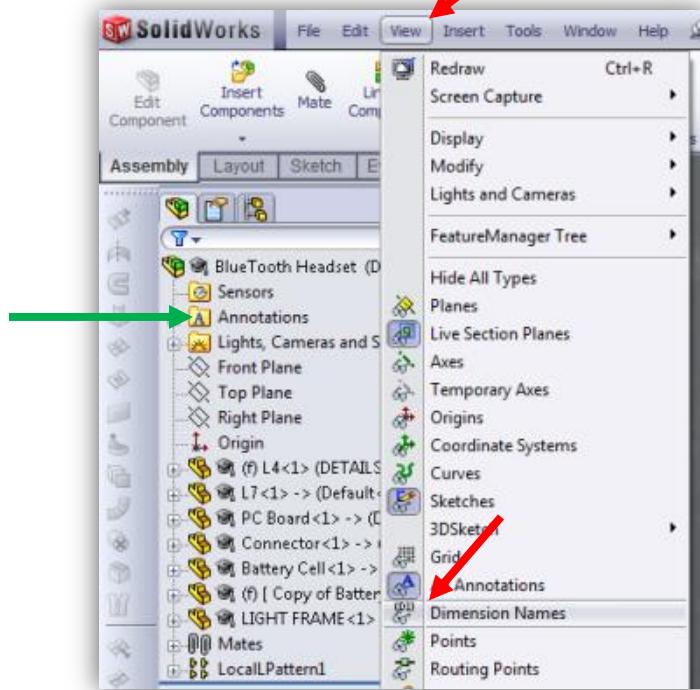
1. Sketch the following on the “Front” plane. Extrude 2”.



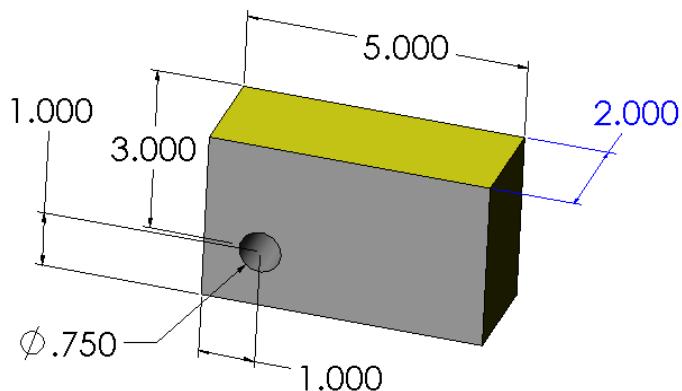
2. Sketch the circle and extrude cut through all.



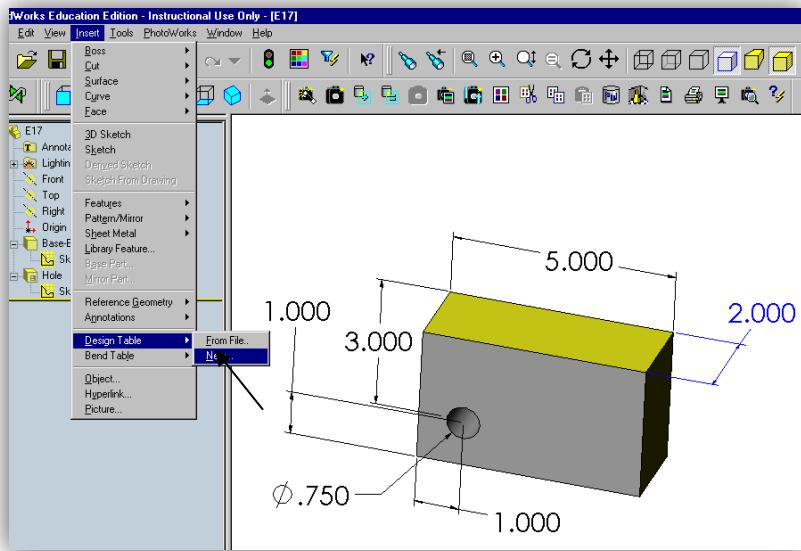
3. RMB click on the Annotations folder in the tree. Select “Show feature dimensions”.
4. Also, go to, “View/Dimension Names”



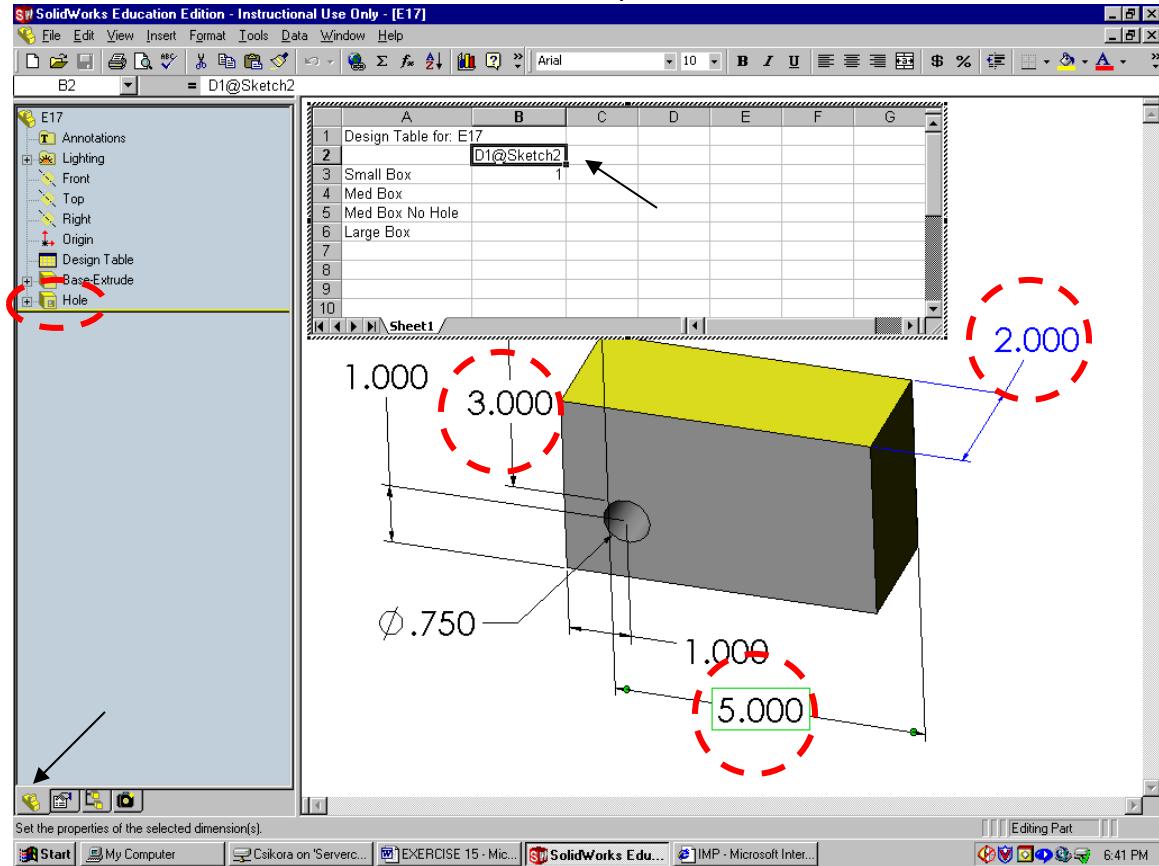
5. You should now zoom out and see all the dimensions. Arrange them so they are clear to see.



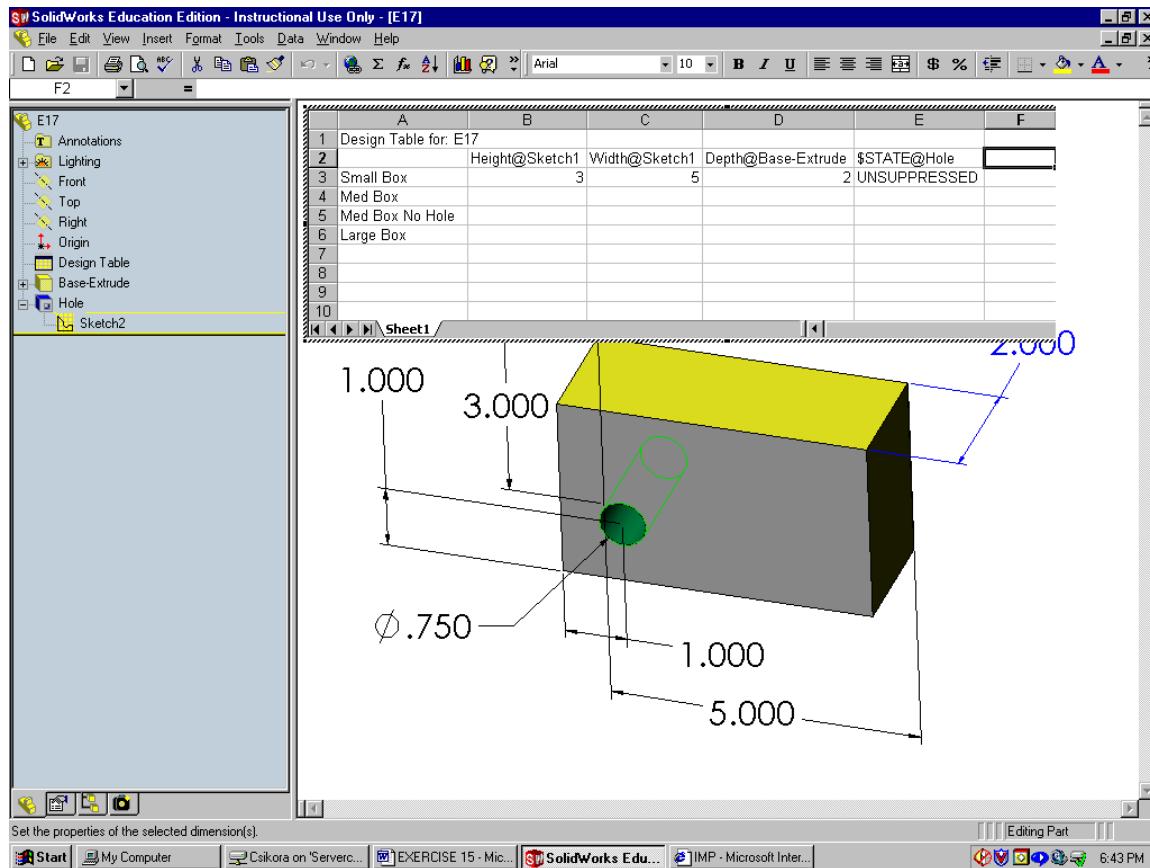
6. Go to “Insert/Design Table and select New”.



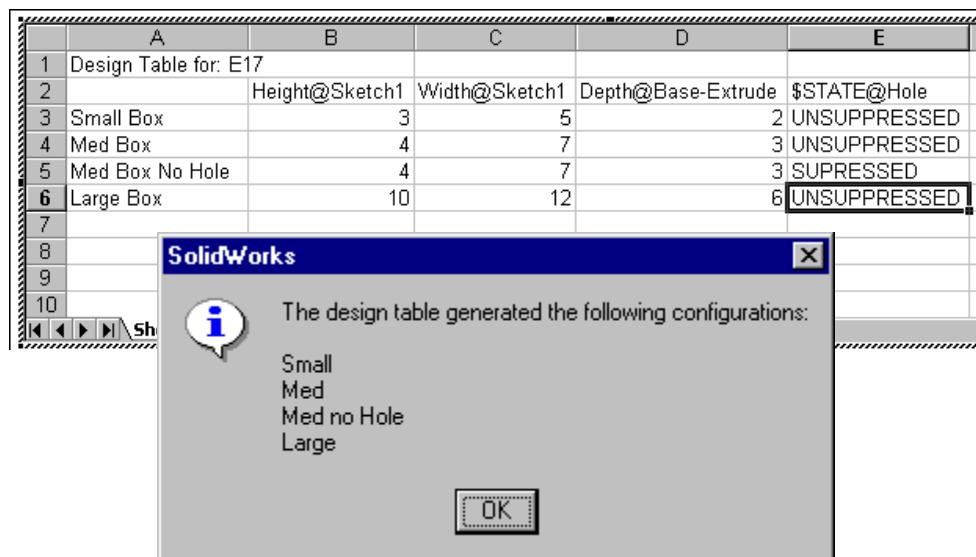
7. Type in the Small thru Large text in the left column. Then select cell 2-B and then double click on the dimension you wish to enter into the table.



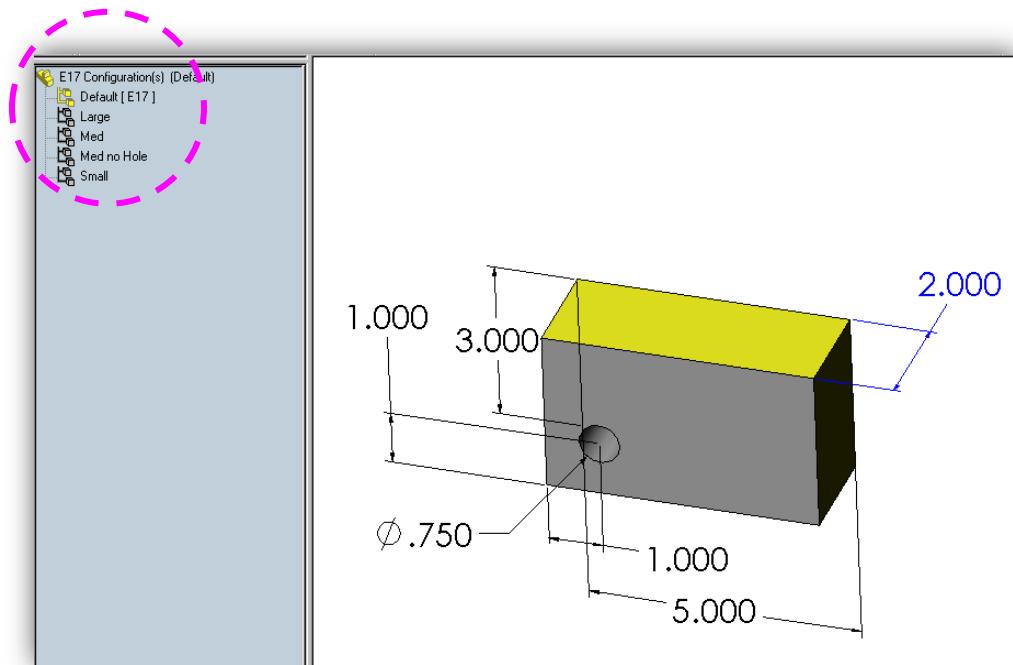
8. Once the four cells are filled in, type the following specification into the cells below.



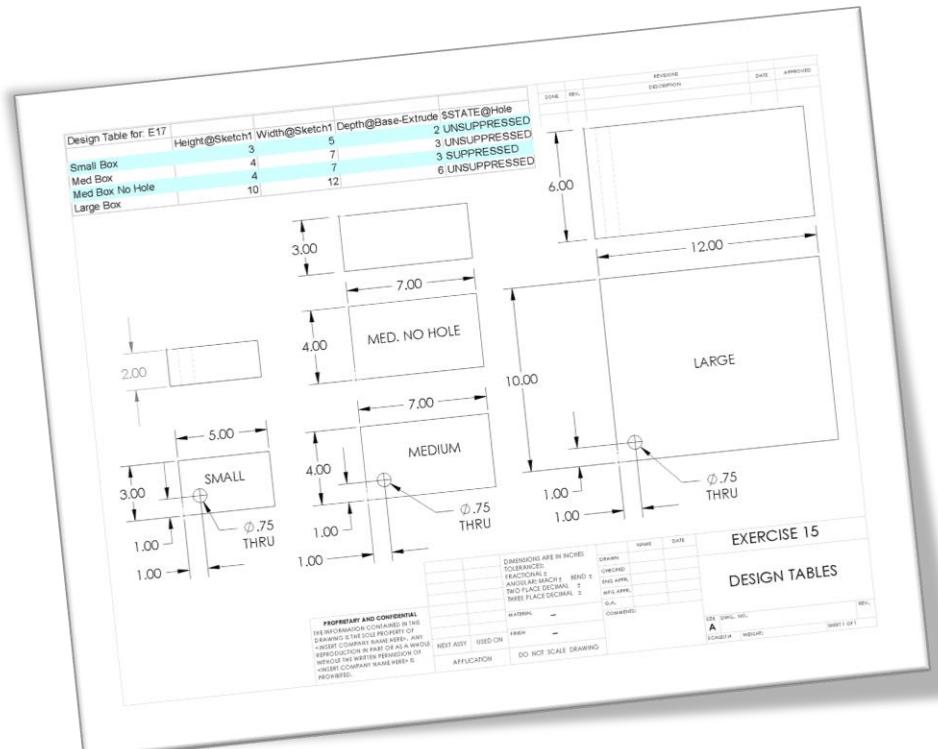
9. It should look like this when complete. Click in the view area and you will receive the following message if it was successful.



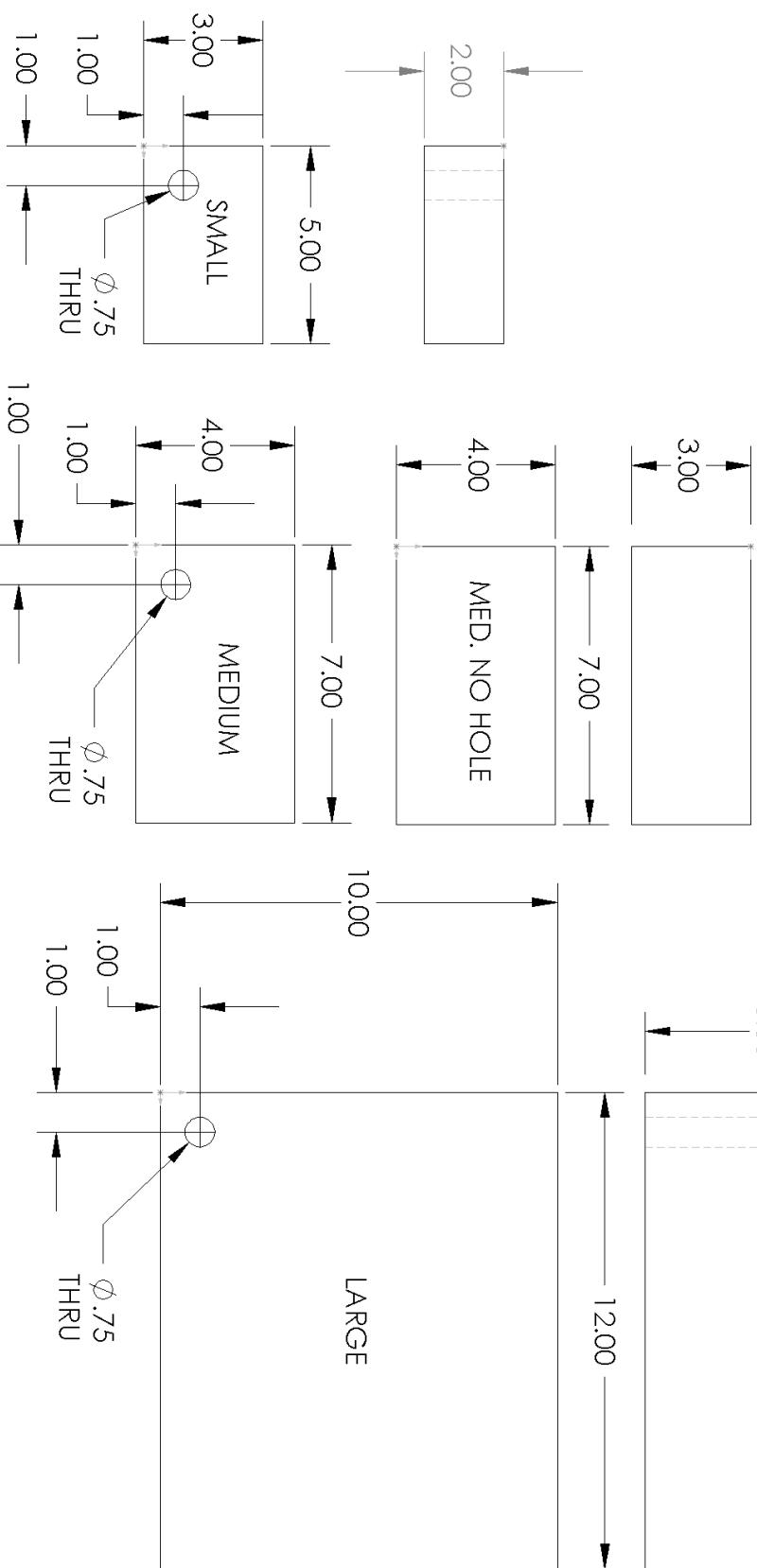
10. Click on the configurations manager tab and you will see the new configurations. Double click on any of the configurations to see it.



11. You are finished with the model. Make the Drawing.



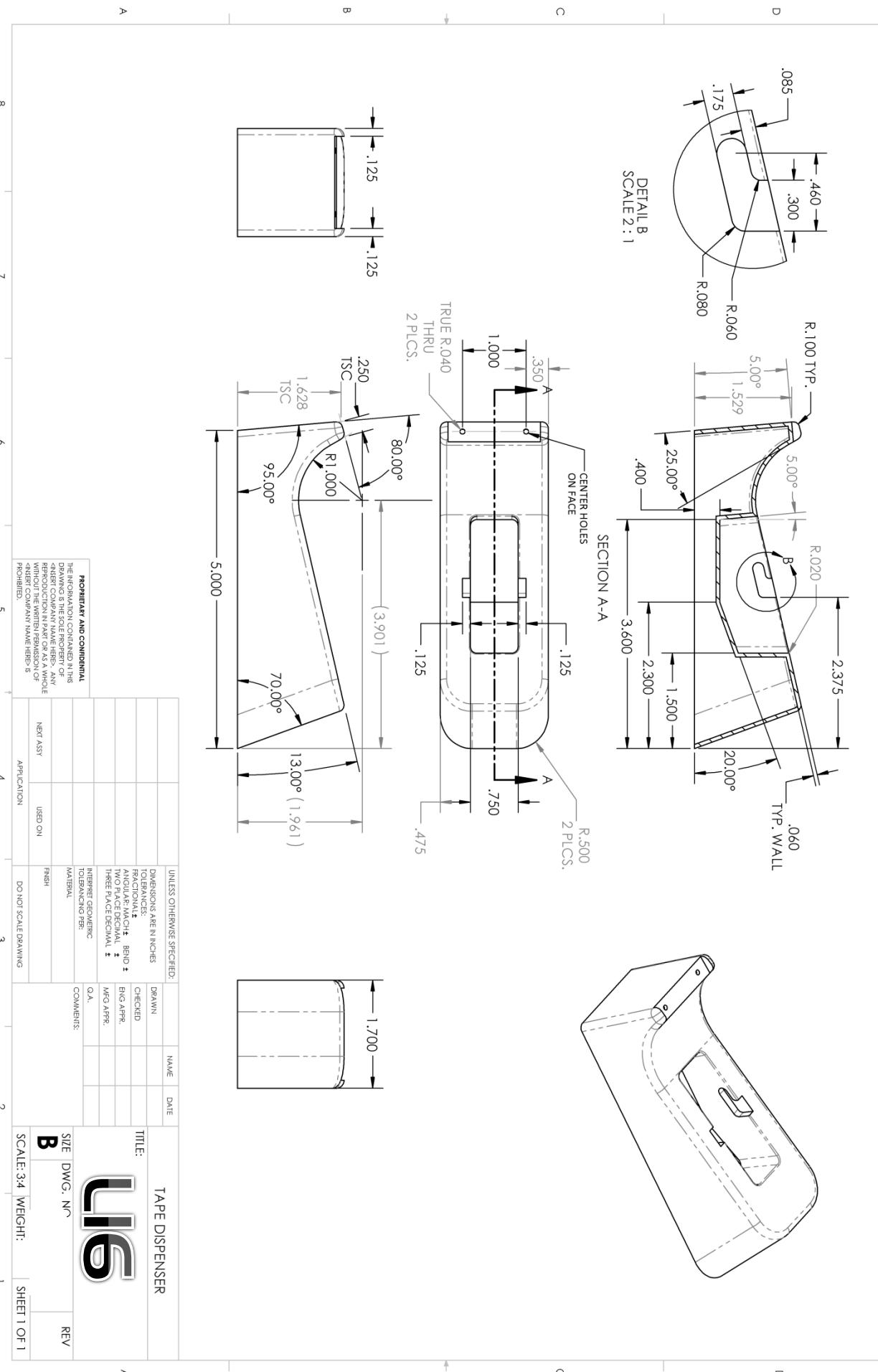
Design Table for: E17	Height@Sketch1	Width@Sketch1	Depth@Base-Extrude	\$STATE@Hole	REVISIONS	
					ZONE	REV.
Small Box	3	5	2	UNSUPPRESSED		
Med Box	4	7	3	UNSUPPRESSED		
Med Box No Hole	4	7	3	SUPPRESSED		
Large Box	10	12	6	UNSUPPRESSED		



		DIMENSIONS ARE IN INCHES		NAME	DATE
		TOLENCES:	DRAWN		
		FRACTIONAL [‡]		CHECKED	
		ANGULAR MACH [†]		ENG APPR.	
		TWO PLACE DECIMAL ±		MFG APPR.	
		THREE PLACE DECIMAL ±		Q.A.	
		MATERIAL —		COMMENTS:	
NEUT ASSY	USED ON	FINISH —		SIZE DWGS. NO.	
				REV.	
		APPLICATION	DO NOT SCALE DRAWING	SCALE 1:4	WEIGHT:
				SHEET 1 OF 1	

PROPRIETARY AND CONFIDENTIAL
THE INFORMATION CONTAINED IN THIS
DRAWING IS THE SOLE PROPERTY OF
<INSERT COMPANY NAME HERE>. ANY
REPRODUCTION IN PART OR AS A WHOLE
WITHOUT THE WRITTEN PERMISSION OF
<INSERT COMPANY NAME HERE> IS
PROHIBITED.

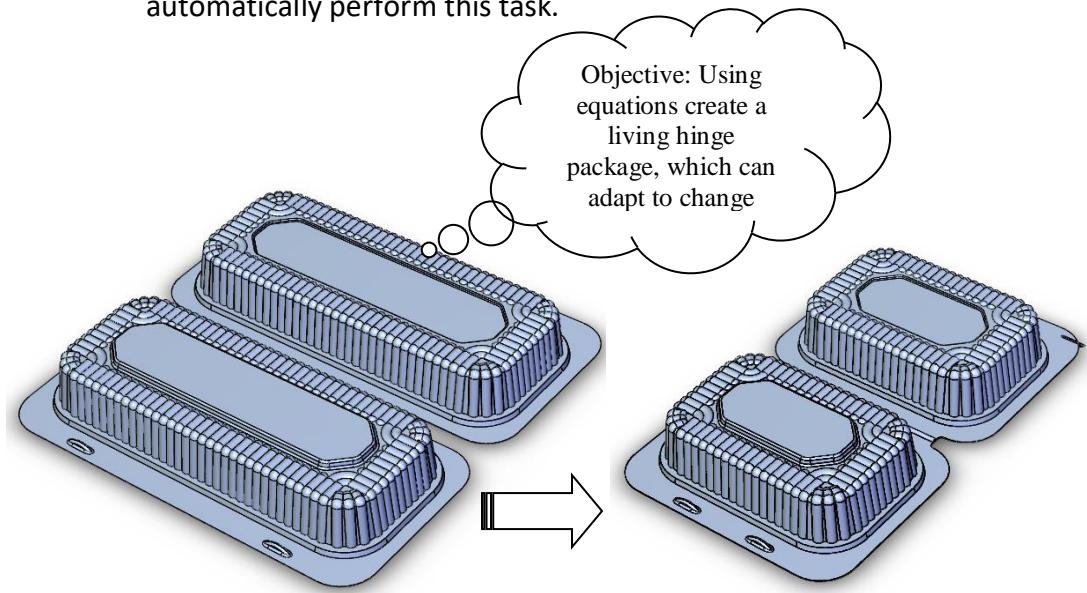
DESIGN TABLES

8
7
6
5
4
3
2
1

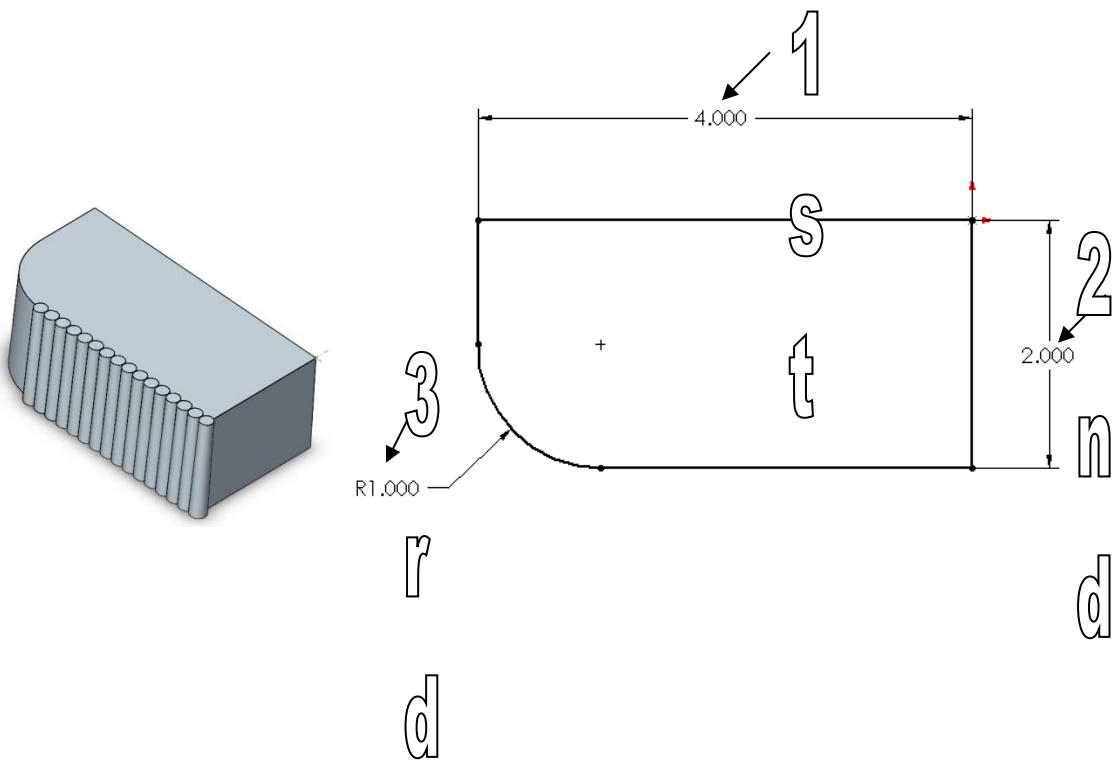
EXERCISE 17

Advanced Equations

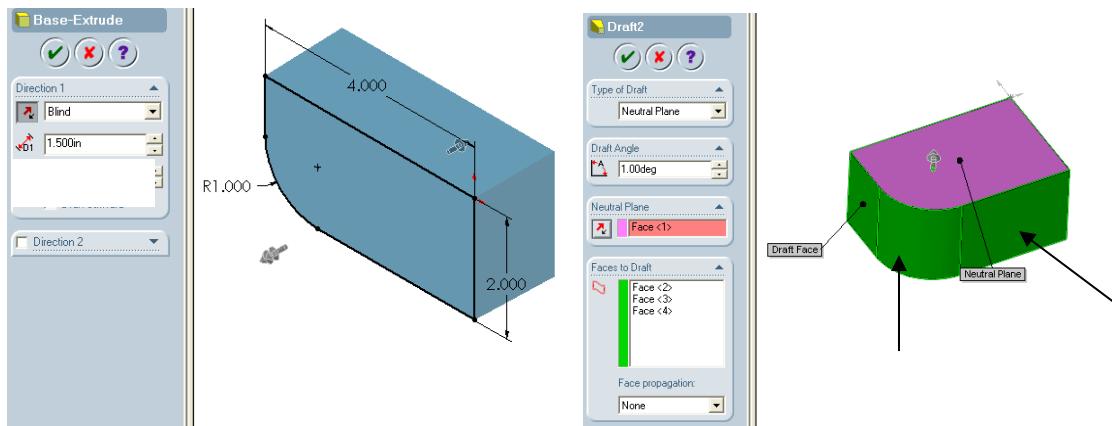
1. Here is an example of how to use equations. The images below represent the same model, but can easily be changed by double clicking on a dimension, and typing in a new value. This normally would create rebuild errors because the rib stack would need to be adjusted as well. Equations can be set up to automatically perform this task.



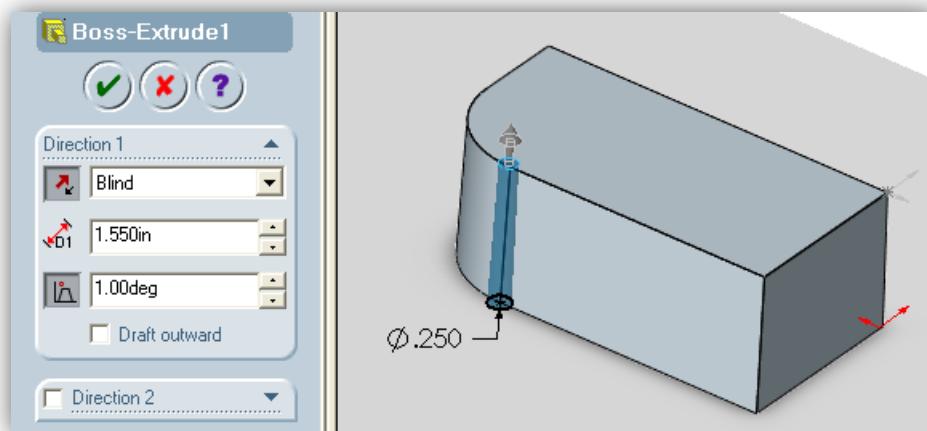
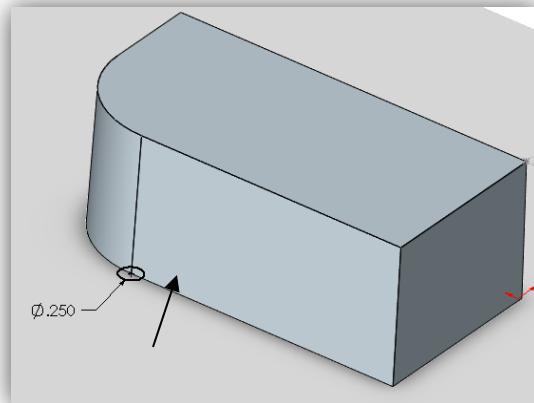
20. Sketch the following on the “Front” plane. Dimension in the order as show.



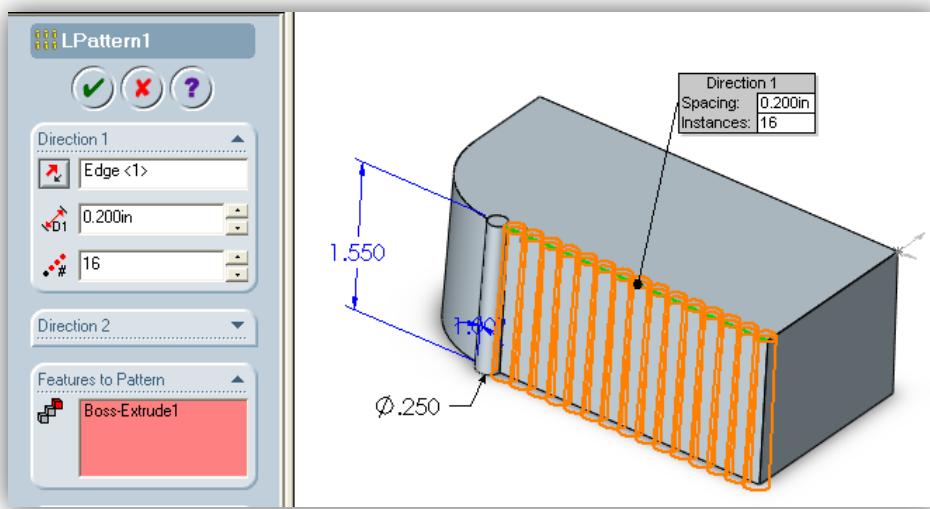
21. Extrude 1.5". Add 1° of draft when ready. (*Note: Do not extrude with draft*)



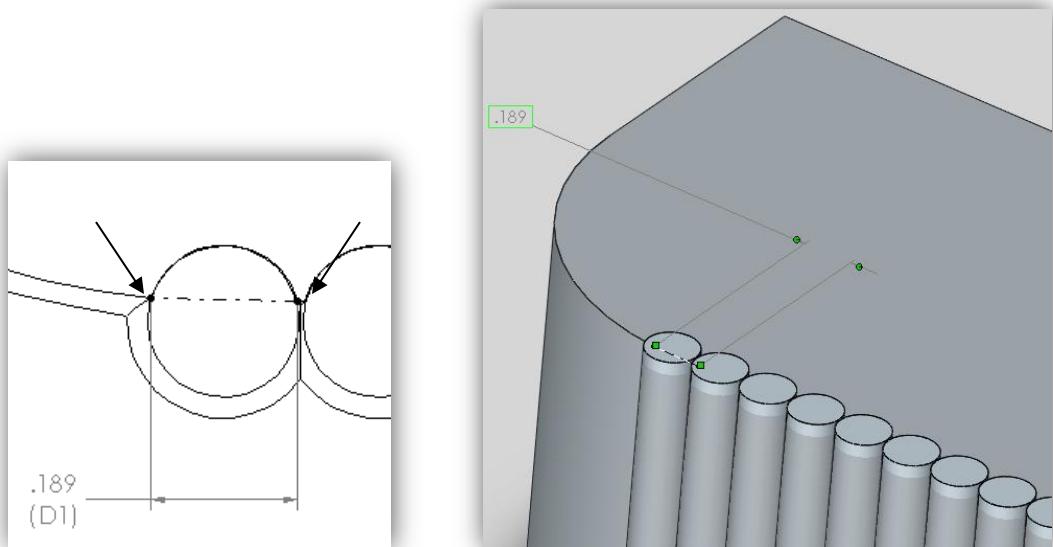
22. Start a sketch on the bottom surface, and draw the following. Extrude with 1° of draft when ready.



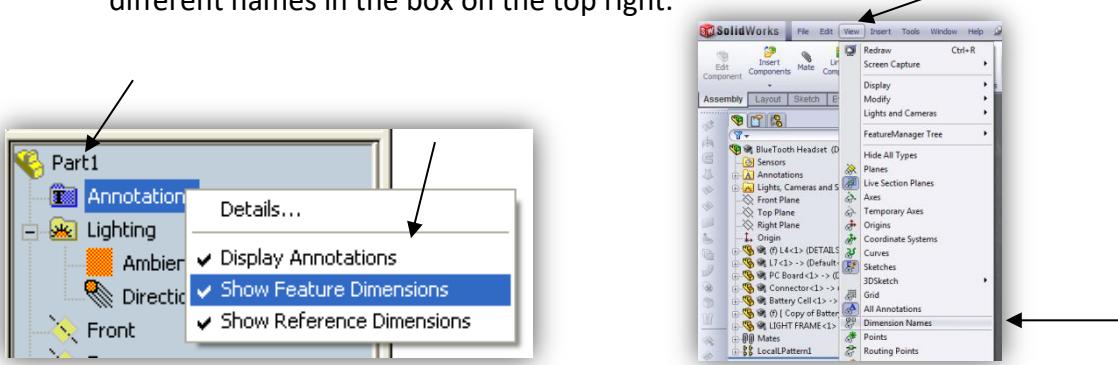
23. Create a linear pattern.



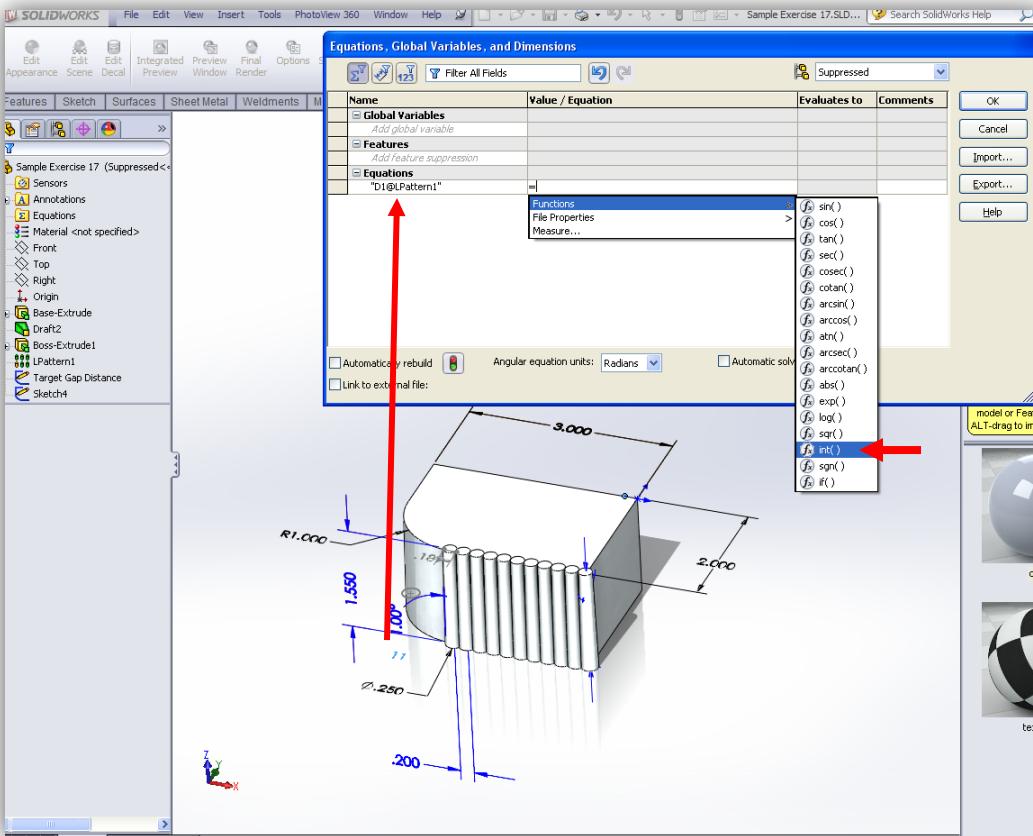
24. Create a new sketch; Draw the following with coincident relations on both sides of the cylinder. Let it become a driven dimension. Rebuild. Rename sketch to “Target gap distance”.

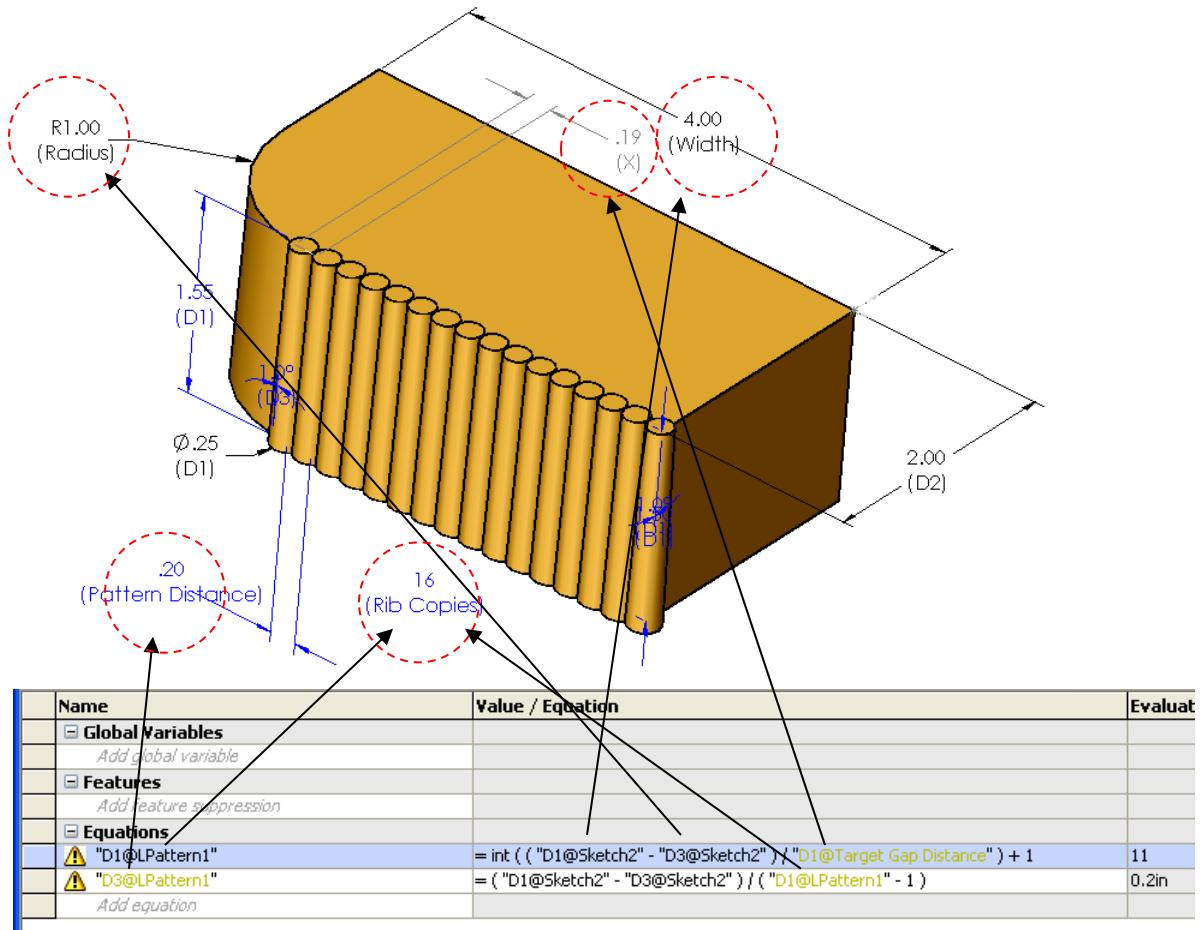


25. Activate the “View Feature Dimensions” and “Show Dimension Names”.
 26. Rename the dimensions as seen below. This can be done by simply RMB click on any dimension you would like to change. Go to Properties and entering different names in the box on the top right.

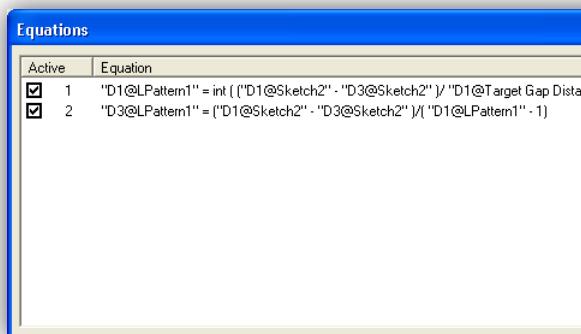


27. **Creating Equations.** Go to *Tools/Equations*. Then select double click in the cell below “Equations”. Click once on the desired dimensions to have the names automatically insert into the equations editor.





What are Equations inside SolidWorks?



They create mathematical relations between model dimensions, using dimension names as variables. When using equations in an assembly, one can set equations between parts, between a part and a sub-assembly, with mating dimensions, and so forth.

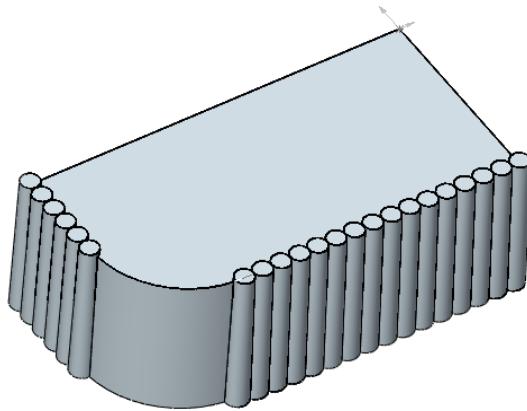
When deleting a feature or dimension that is used in an equation, you have the option of deleting the equation or not.

NOTE: Dimensions driven by equations cannot be changed by editing the dimension value in the model.

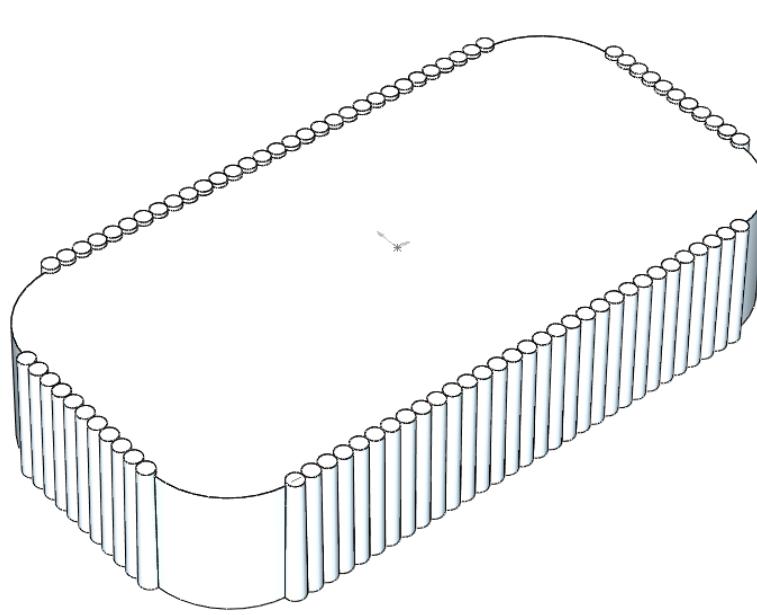
28. Here is an index of functions.

Function	Name	Notes
sin (a)	sine	a is angle expressed in radians
cos (a)	cosine	a is angle expressed in radians
tan (a)	tangent	a is angle expressed in radians
atn (a)	inverse tangent	a is angle expressed in radians
abs (a)	absolute value	returns the absolute value of a
exp (n)	exponential	returns e raised to the power of n
log (a)	logarithmic	returns the natural log of a to the base e
sqr (a)	square root	returns the square root of a
int (a)	integer	returns a as an integer
sgn (a)	sign	returns the sign of a
Constant		
pi	pi	3.14...

29. Add the same equations on the left side of the model.

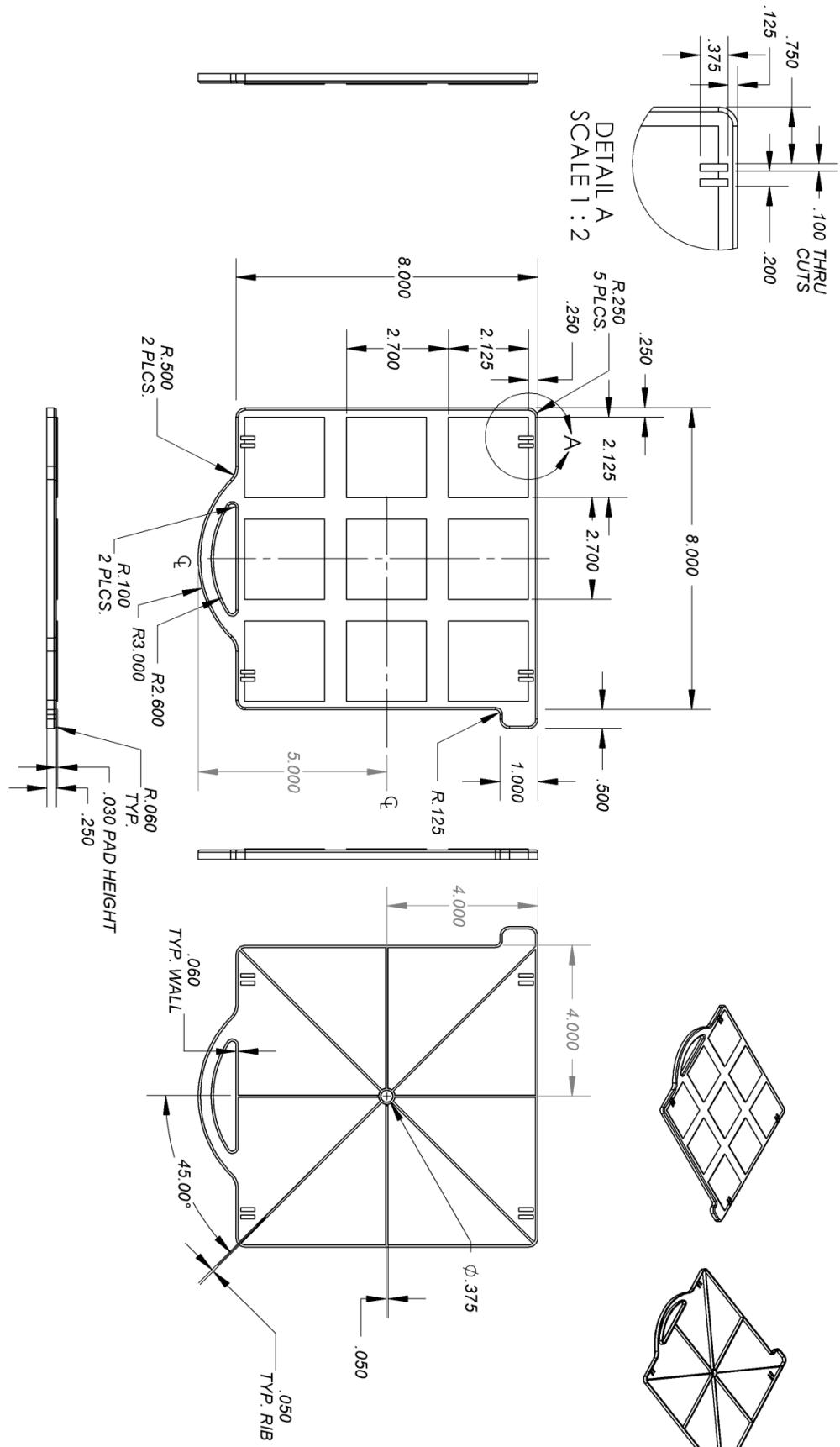


30. Mirror all, both sides.



Finished

PROPRIETARY AND CONFIDENTIAL
 THE INFORMATION CONTAINED IN THIS
 DRAWING IS THE SOLE PROPERTY OF
 <INSERT COMPANY NAME HERE>. ANY
 REPRODUCTION IN PART OR AS A WHOLE
 WITHOUT THE WRITTEN PERMISSION OF
 <INSERT COMPANY NAME HERE> IS
 PROHIBITED.



4	APPLICATION	DO NOT SCALE DRAWING
3	NEXT ASSY	USED ON
2	FINISH	
1		

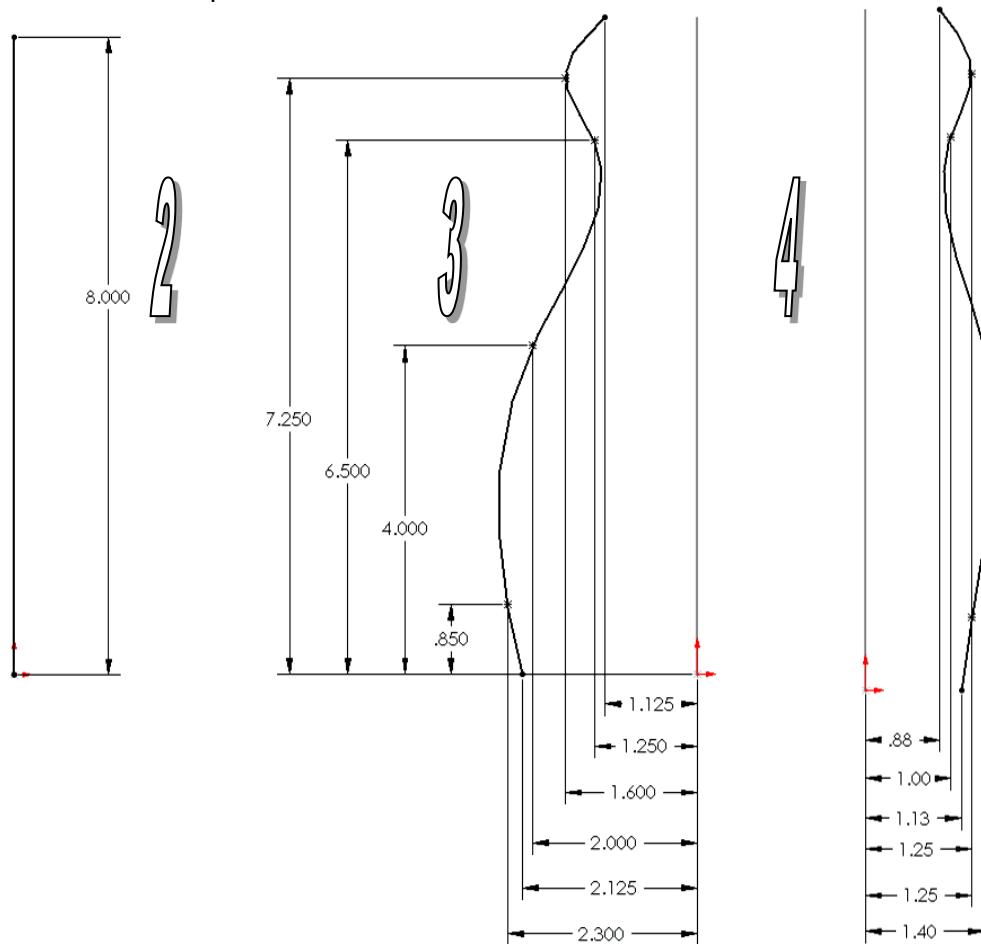
EXERCISE 18

Using Sweeps with Guide Curves

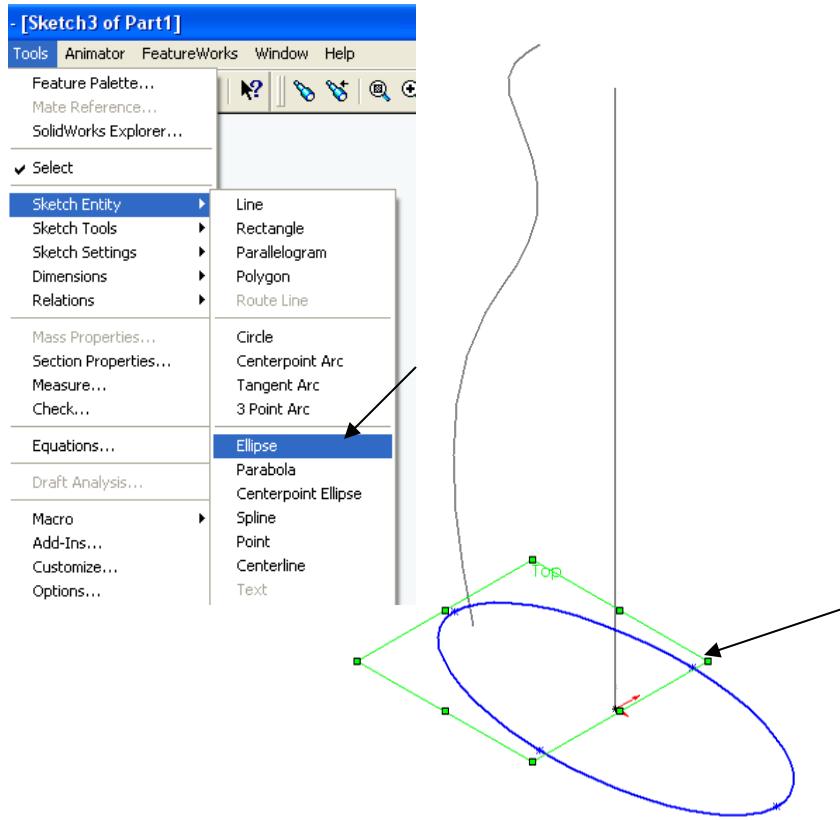
Sweeps can be beneficial when creating symmetric freeform geometry.



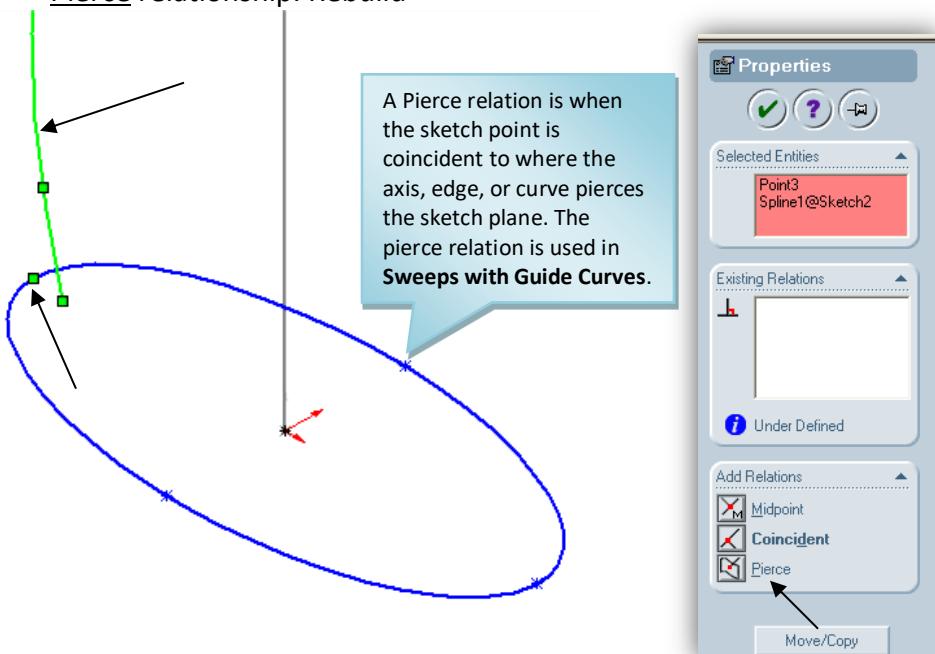
1. Start a sketch on the “Front” plane and draw a vertical line at 8” from the origin. Rebuild.
2. Using a spline draw the following on the “Front” plane. Rebuild.
3. Using a spline draw the following on the “Right” plane. Use the same vertical dims as in step 3. Rebuild.



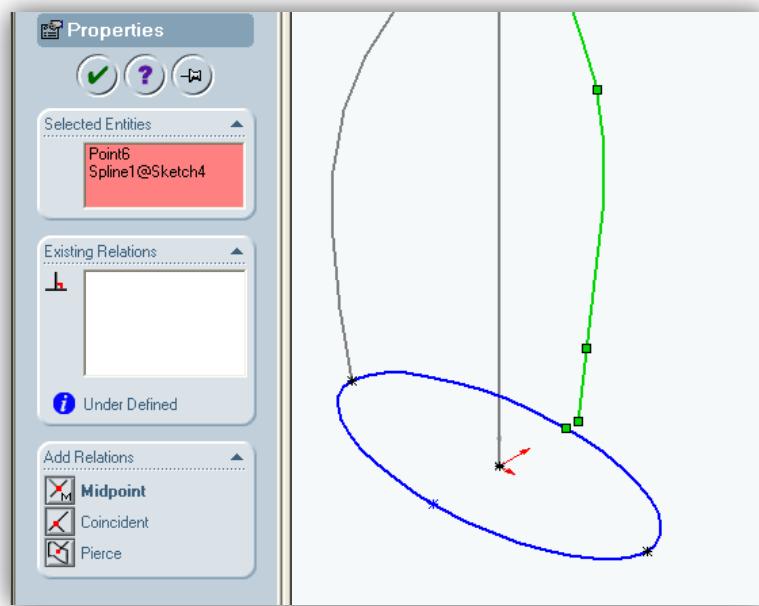
4. Sketch an Ellipse on the “Top” plane. This will be the profile.



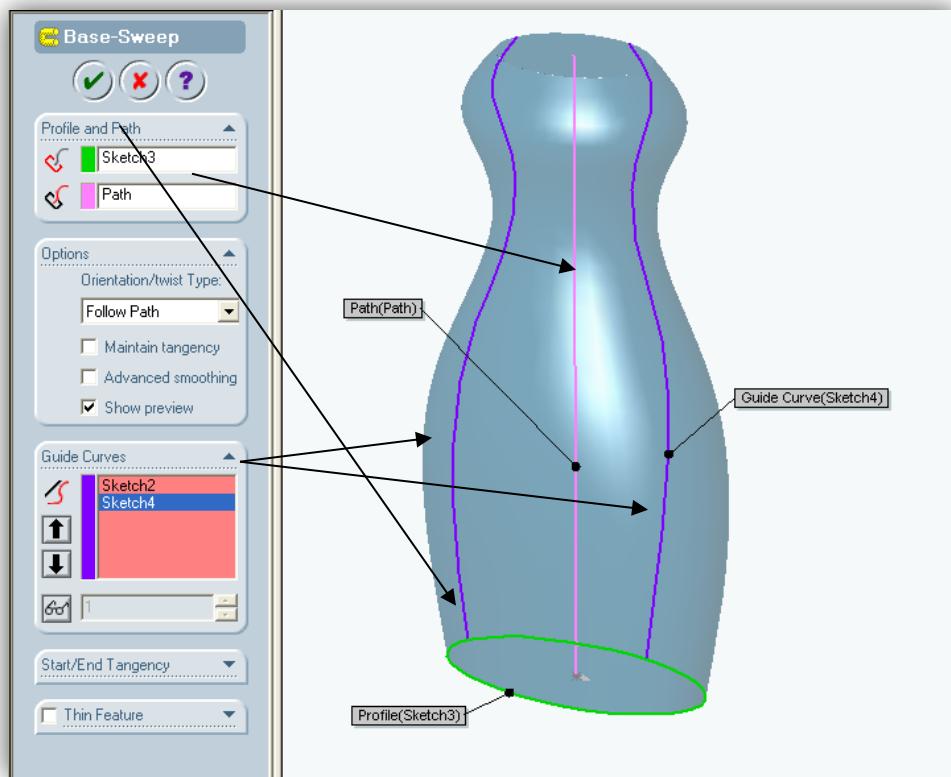
5. Hold CTRL and select the spline curve and the point on the Ellipse. Add a Pierce relationship. Rebuild



6. Repeat step 5. Using the right side spline.



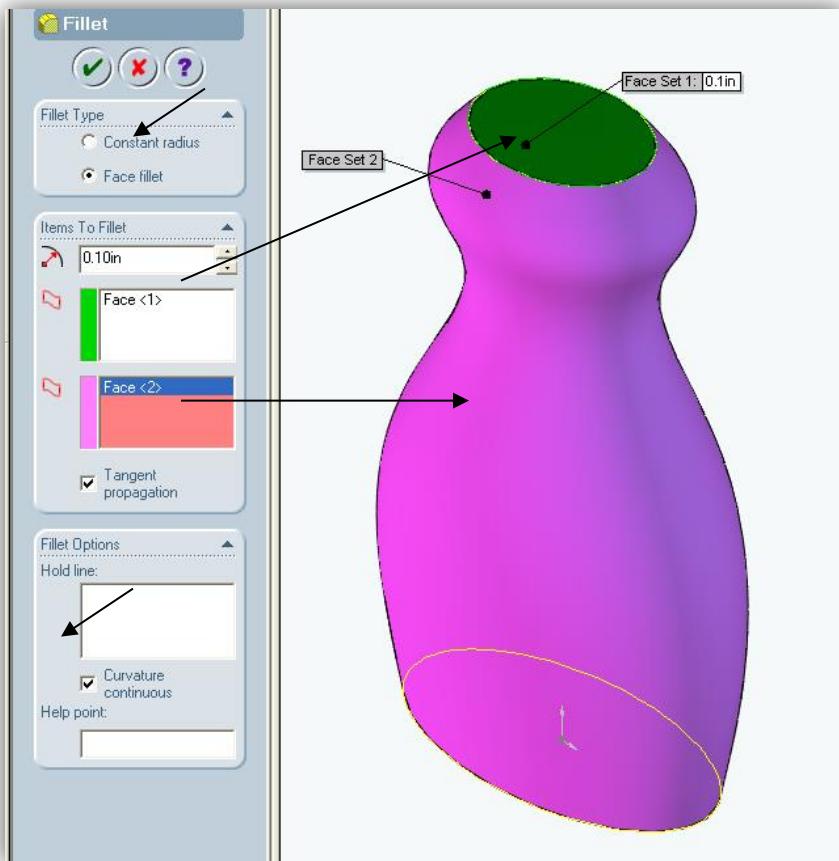
7. Sweep



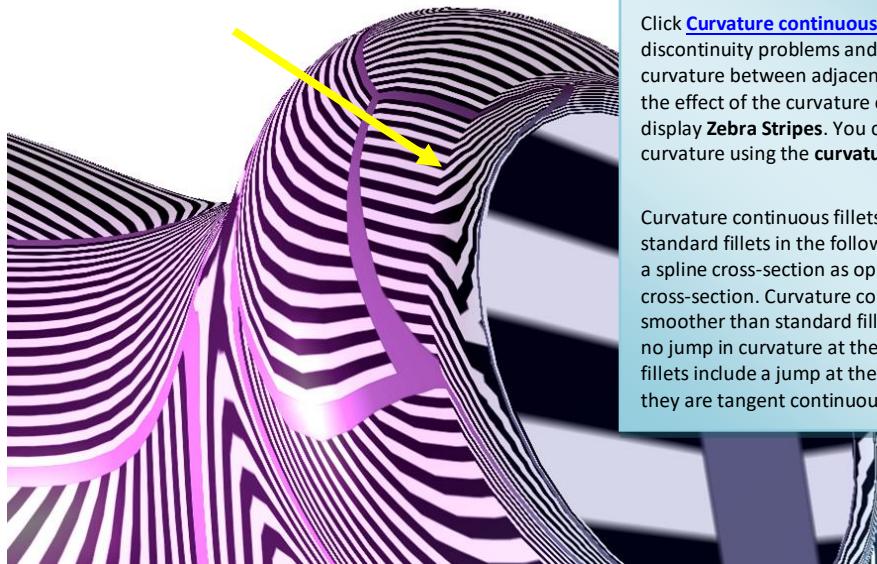
8. Sweep completed.



9. **Creating Curvature Continuous Face Blend Fillets.** Select the top face and side face of the bottle. Select the fillets icon. Input .250" radius and check curvature continuous.



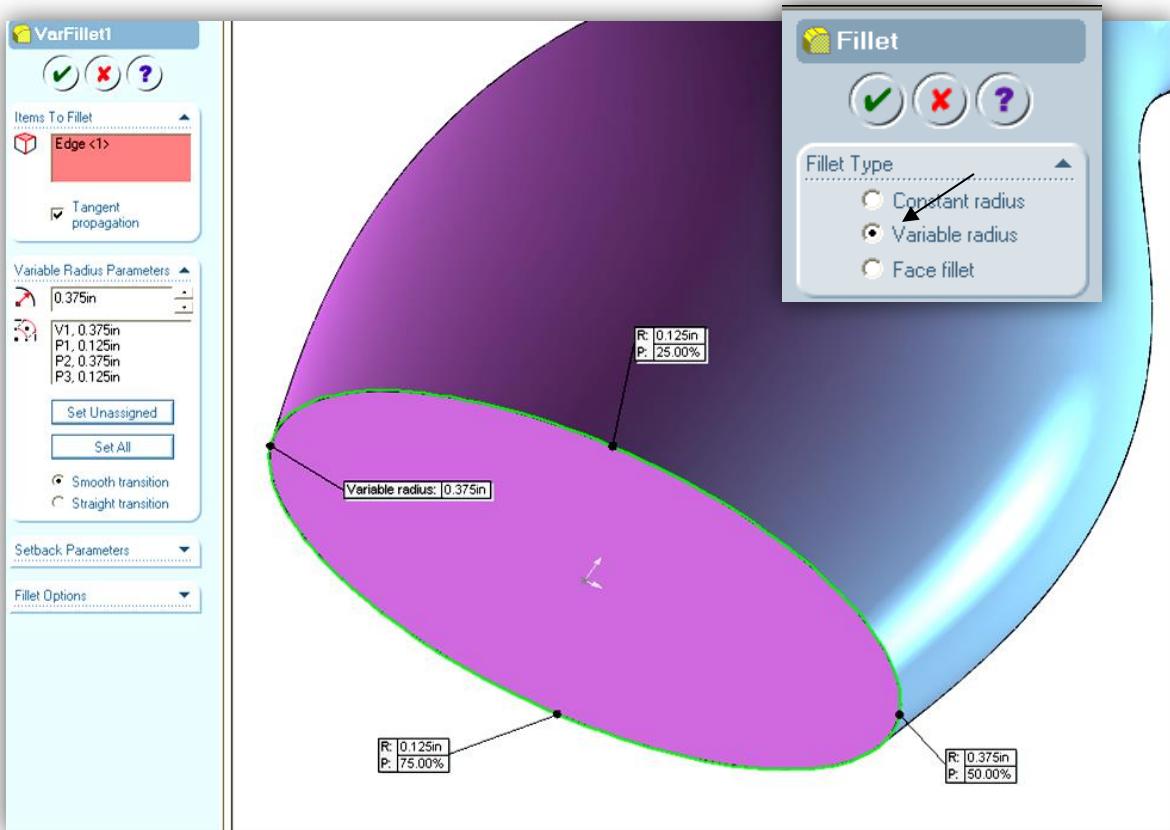
What are Curvature Continuous fillets?



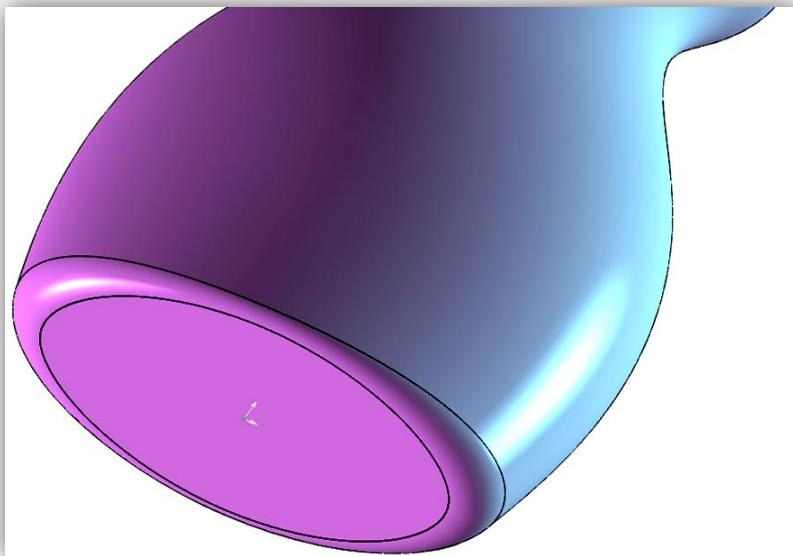
Click [Curvature continuous](#) to resolve discontinuity problems and create a smoother curvature between adjacent surfaces. To verify the effect of the curvature continuity, you can display **Zebra Stripes**. You can also analyze the curvature using the **curvature** tool.

Curvature continuous fillets differ from standard fillets in the following ways. They have a spline cross-section as opposed to a circular cross-section. Curvature continuous fillets are smoother than standard fillets because there is no jump in curvature at the boundary. Standard fillets include a jump at the boundary because they are tangent continuous at the boundary.

10. **Variable Radius Filleting.** Select Fillet tool, then select the Variable fillet. Select desired edge and it will break the edge into quadrants. Enter dims as shown.

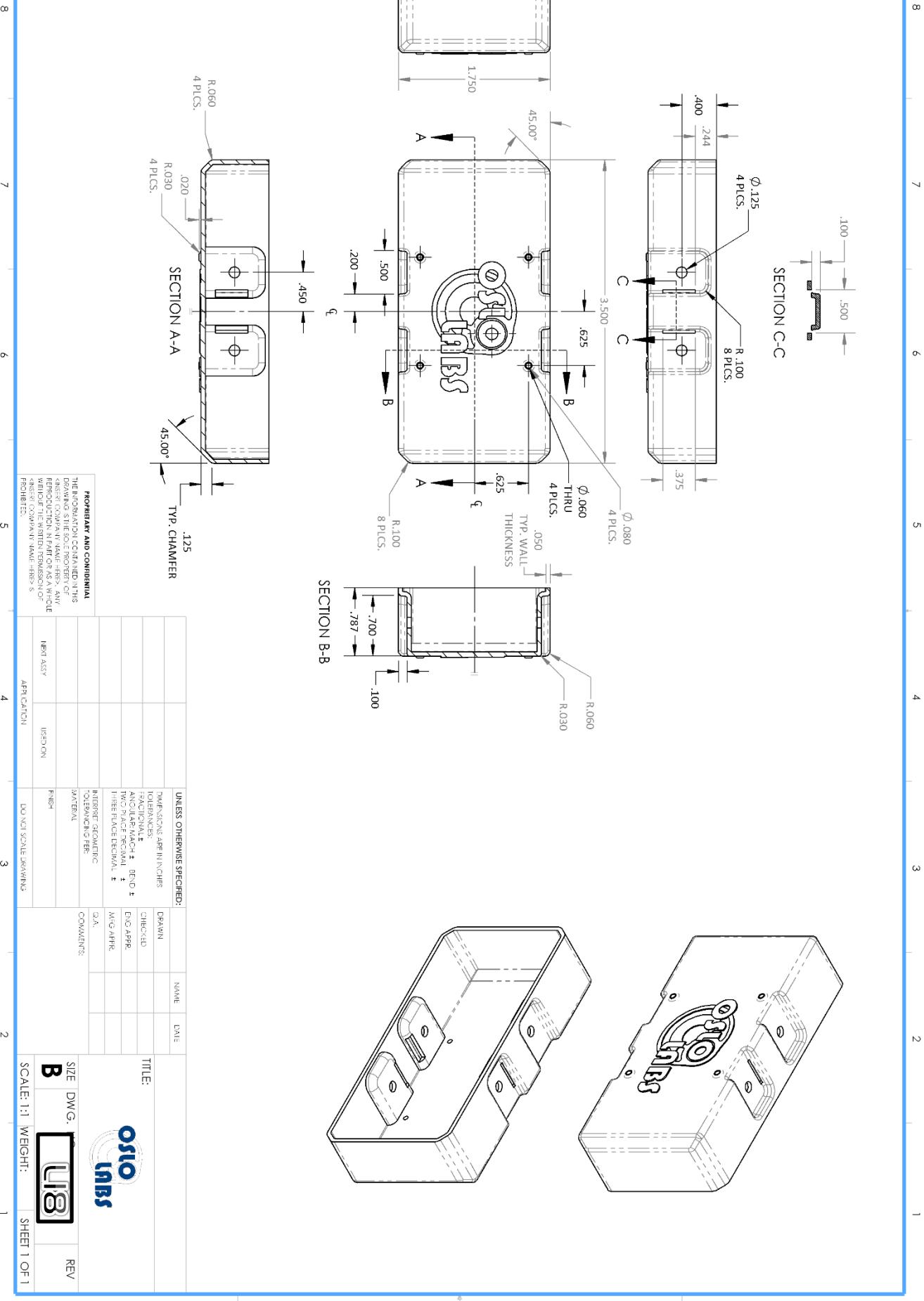


11. The fillet should appear as shown.



12. **Create a Thread** – Shell at .1", and add embossed text. You are finished.

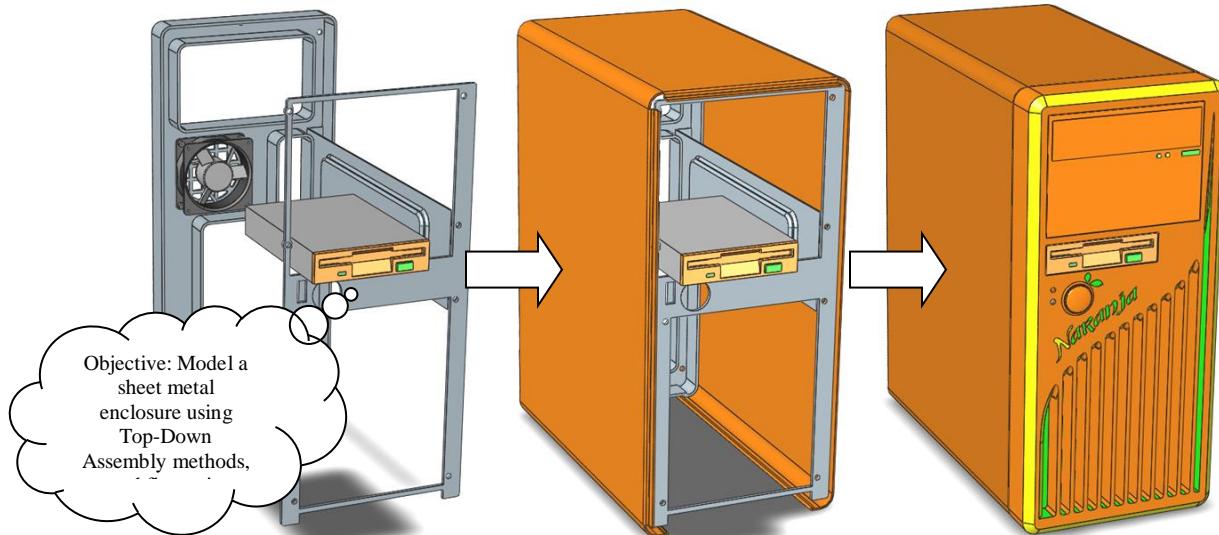




EXERCISE 19

Advanced Sheet Metal Design

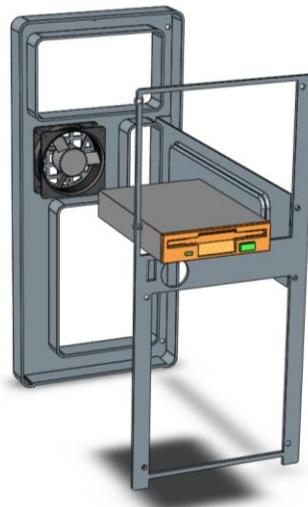
Sheet Metal part files can be very useful for extracting a flat pattern.



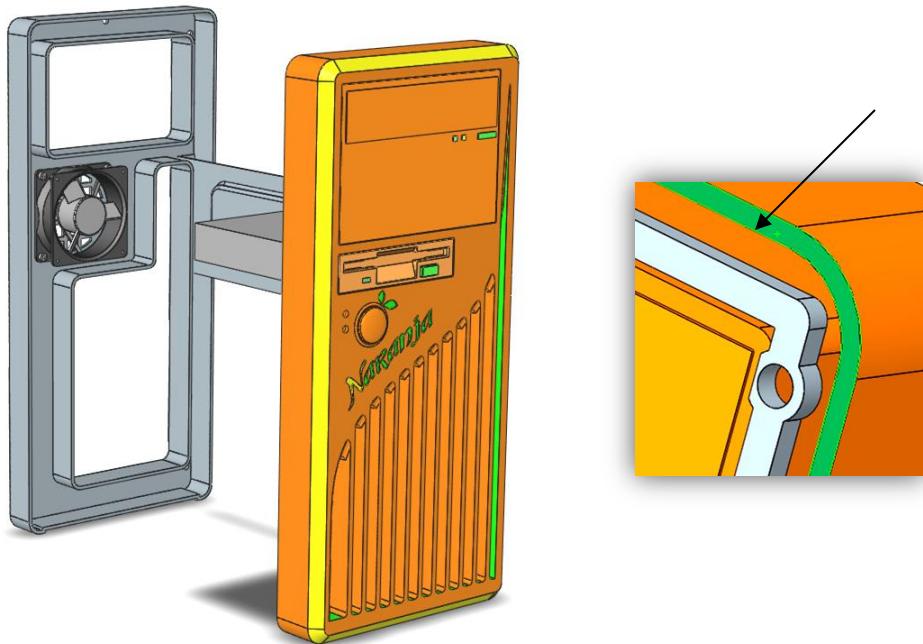
1. Go to file/open and select "E19". Hide the existing cover.

Sheet Metal Tool Bar:

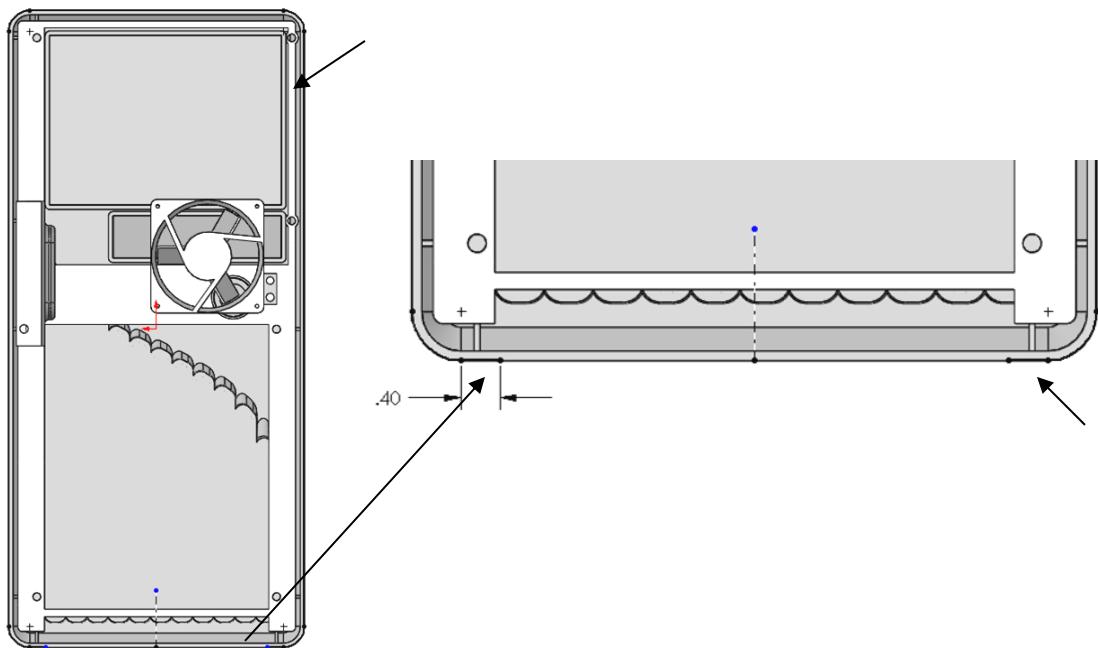
To activate the toolbar on the ribbon simply RMB click on any ribbon tab and check the Sheet Metal box.



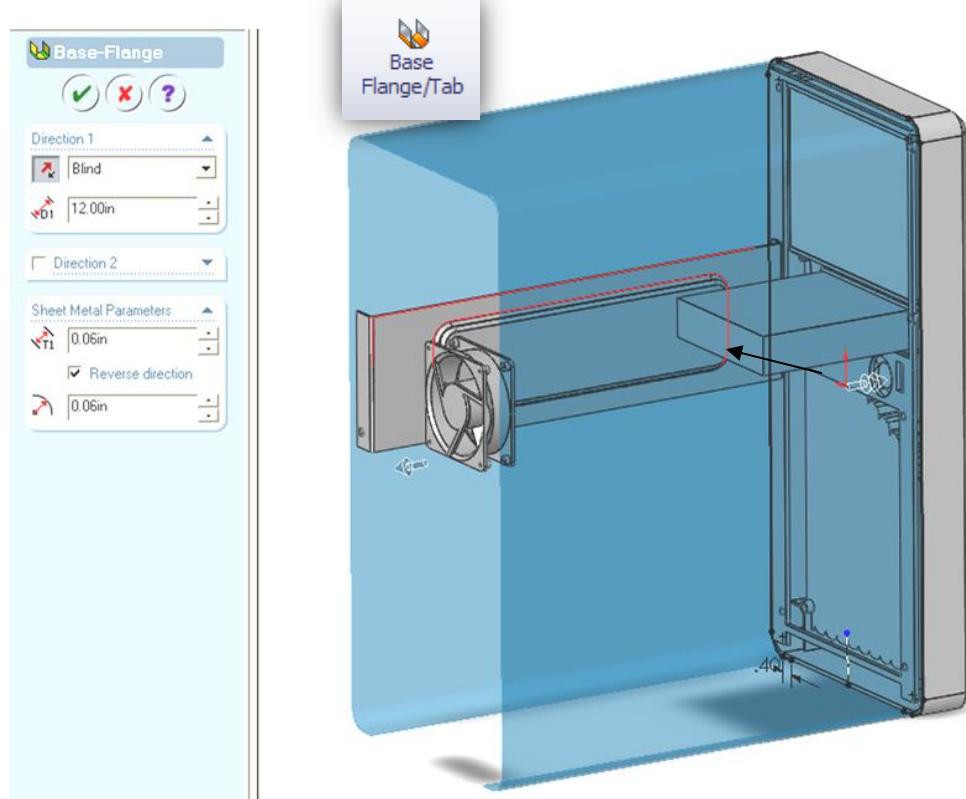
2. Insert a new part into the assembly; drop it on the “Front” plane of the assembly. Name it “Cover” (This will be the enclosure). Then select the inside shelled face. Convert Entities.



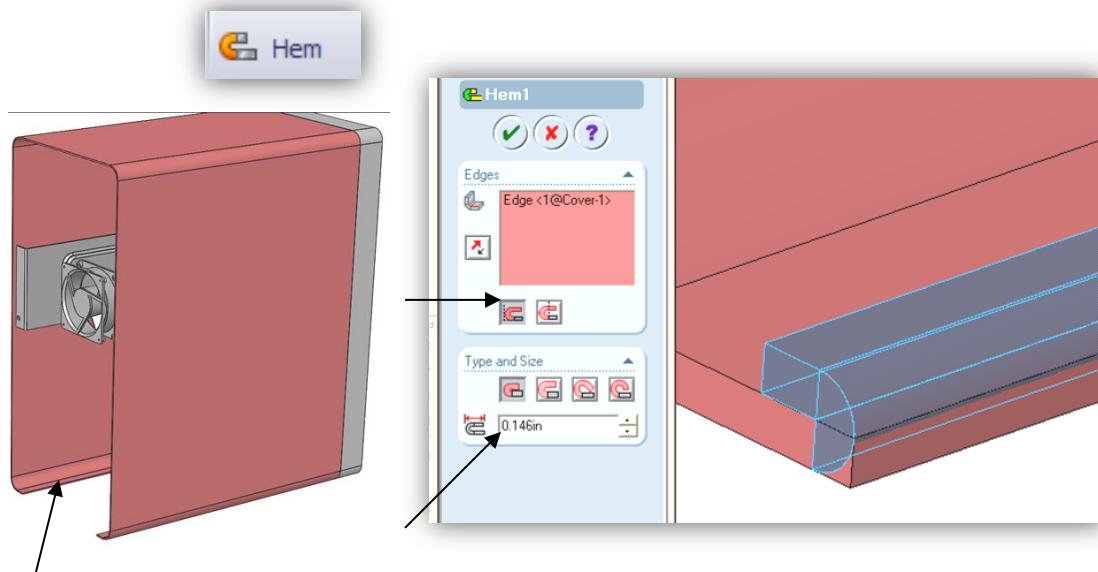
3. Sketch the bottom ends as shown.

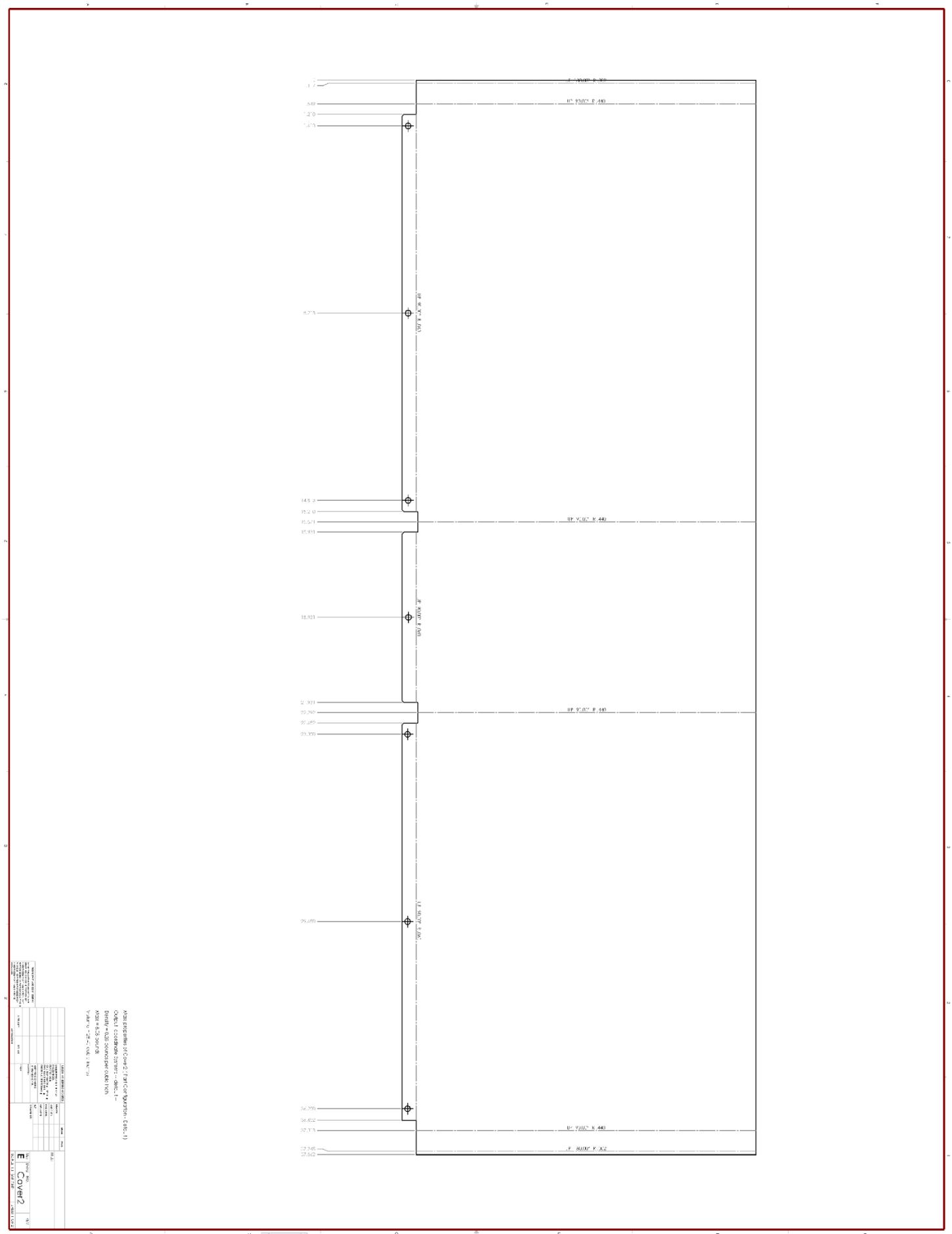


4. Use the Base Flange icon to extrude as follows.



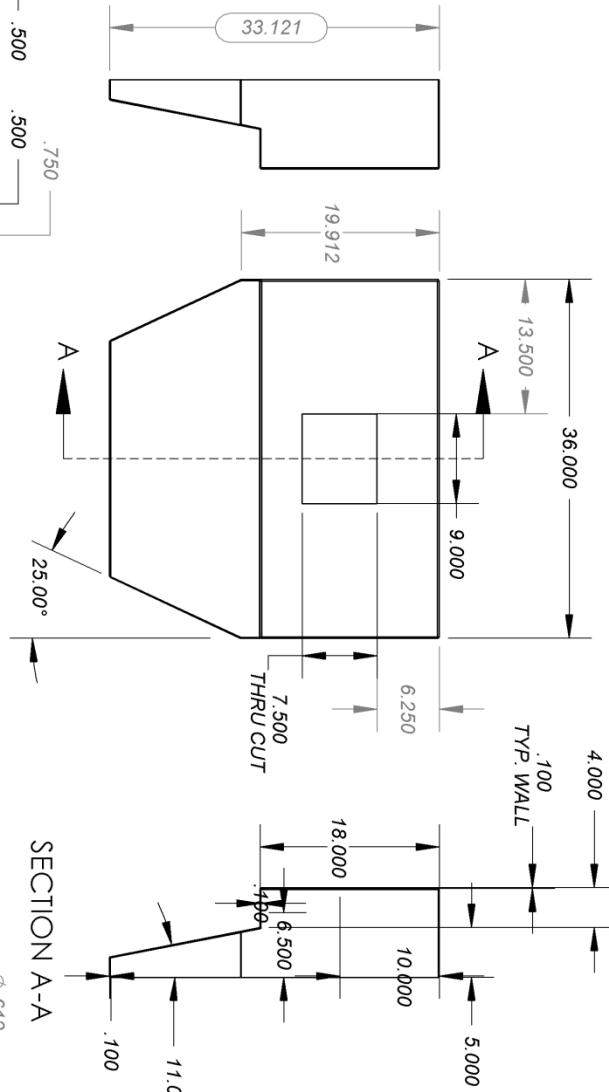
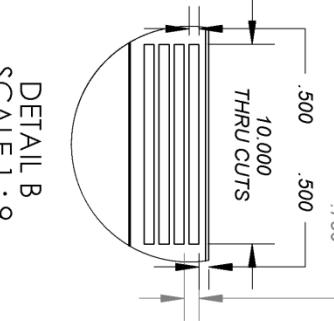
5. Use the "Hem" icon to remove sharp edges from the bottom of the cover.



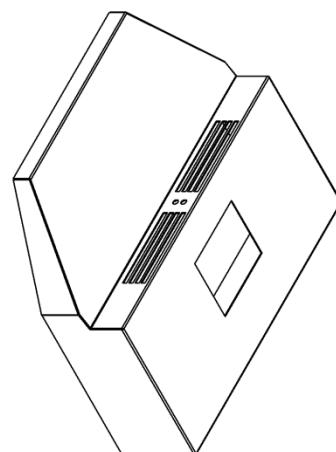
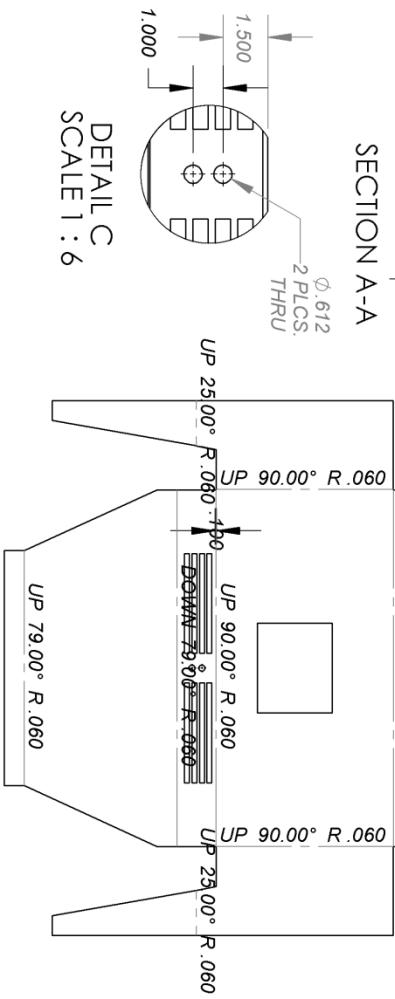


PROPRIETARY AND CONFIDENTIAL
THE INFORMATION CONTAINED IN THIS
DRAWING IS THE SOLE PROPERTY OF
<INSERT COMPANY NAME HERE>. ANY
REPRODUCTION IN PART OR AS A WHOLE
WITHOUT THE WRITTEN PERMISSION OF
<INSERT COMPANY NAME HERE> IS
PROHIBITED.

DETAIL B
SCALE 1:9



DETAIL C
SCALE 1:6



UNLESS OTHERWISE SPECIFIED:			
DIMENSIONS ARE IN INCHES	DRAWN	NAME	DATE
TOLERANCES:			
FRACTIONAL [±]			
ANGULAR: MACH [±]			
TWO PLACE DECIMAL [±]			
THREE PLACE DECIMAL [±]			
INTERPRET GEOMETRIC TOLERANCING PER: MATERIAL			
Q.A.			
COMMENTS:			

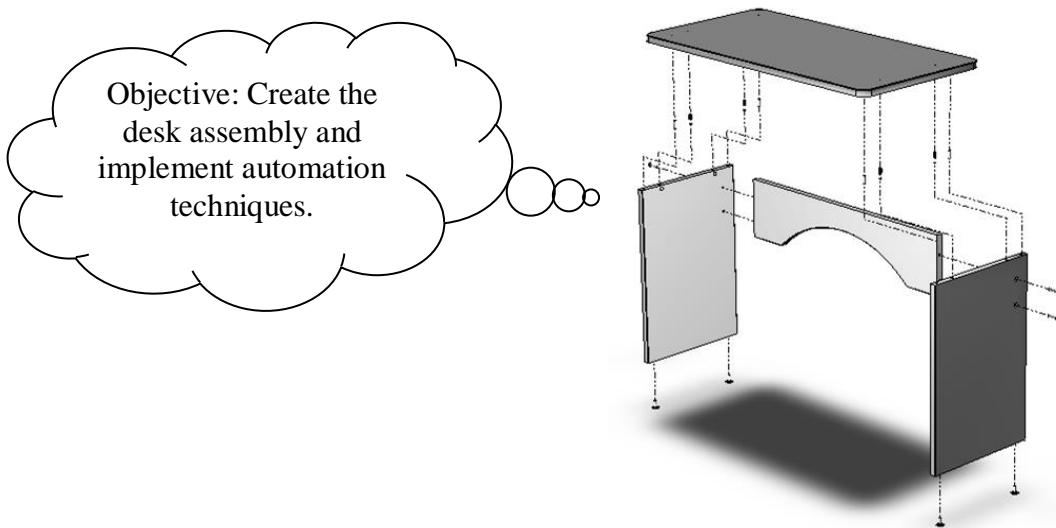
SIZE	DWG. NO.	REV
A		

SCALE: 1:18 WEIGHT: SHEET 1 OF 1

EXERCISE 20

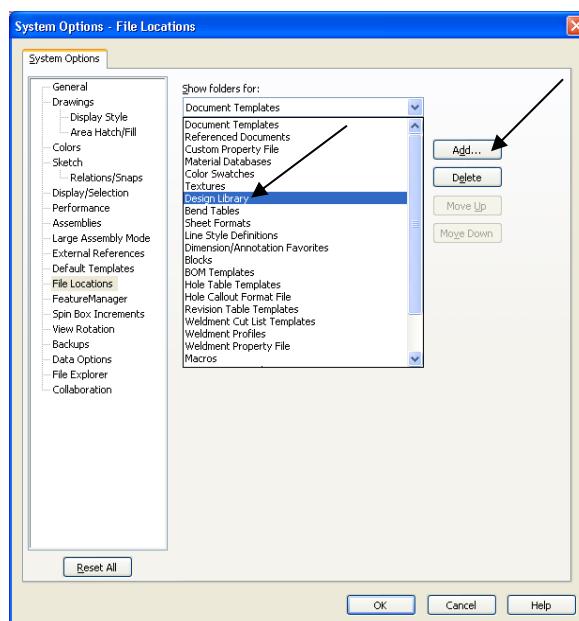
Assembly and Drawing Automation

Assembly and Drawing creation can be virtually automated through the use of many techniques capable in the SolidWorks software.

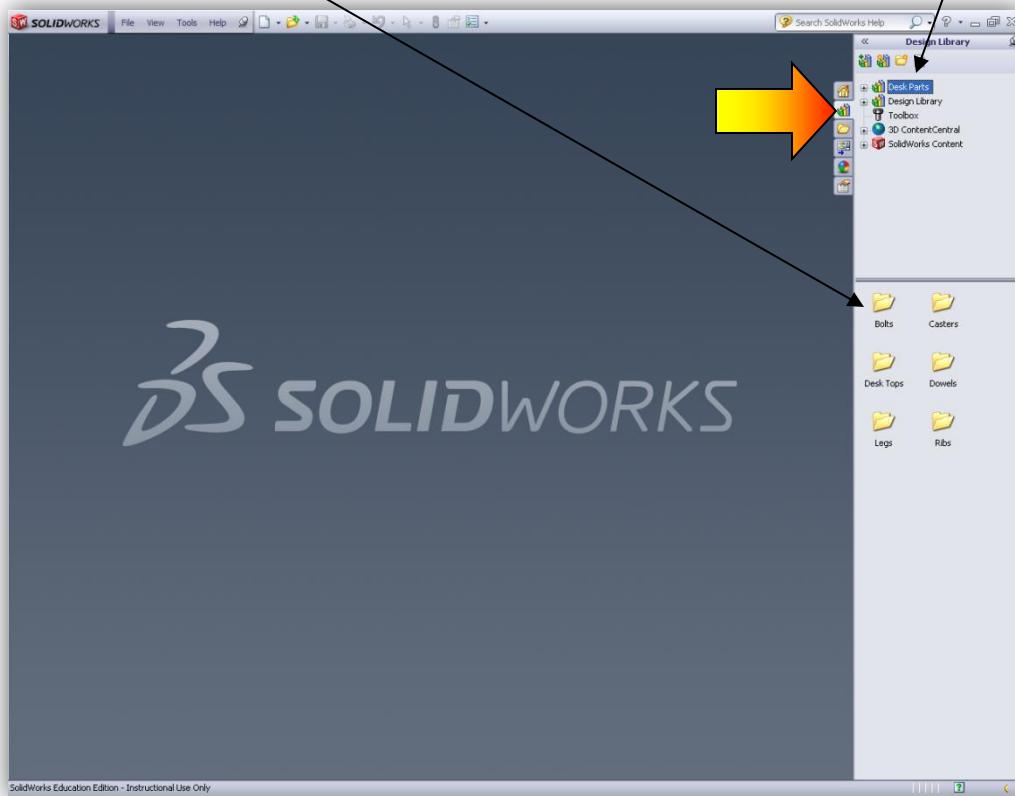


Creating a Hardware Library using the Design Library

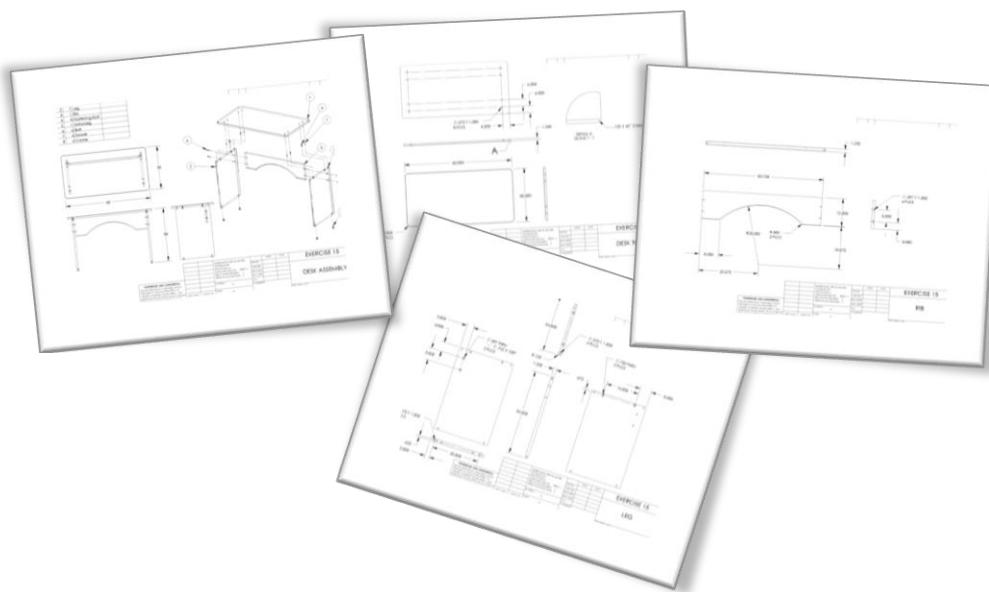
1. Install the Desk Parts sub folder (this contains all the hardware required for assembling the desk) into the E20 directory.
2. Then go to tools/options/system options/file locations and select Design Library from the pull down menu. Once selected click on the “add” button. This will enable you to browse to your Desk Parts folder.



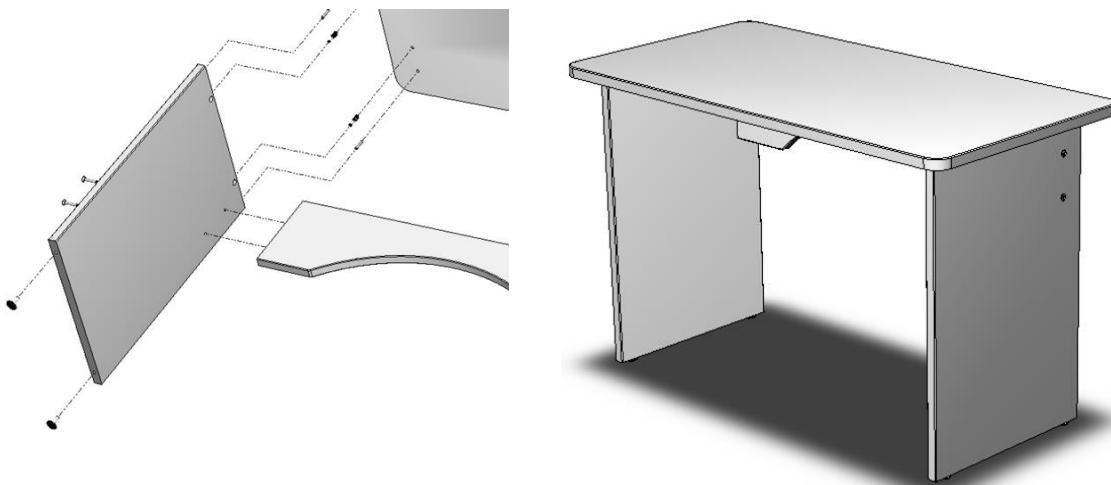
3. By doing this you have now enabled your parts to be accessed through the Design Library, Desk Parts folder. Enabling easy drag and drop access. Otherwise you can just use the Insert Component tool.



4. Now start a new part file and begin to create the attached parts.
NOTE: The drawings are missing dimensions. You can use the desk components in the Design Library to attain the proper measurements.



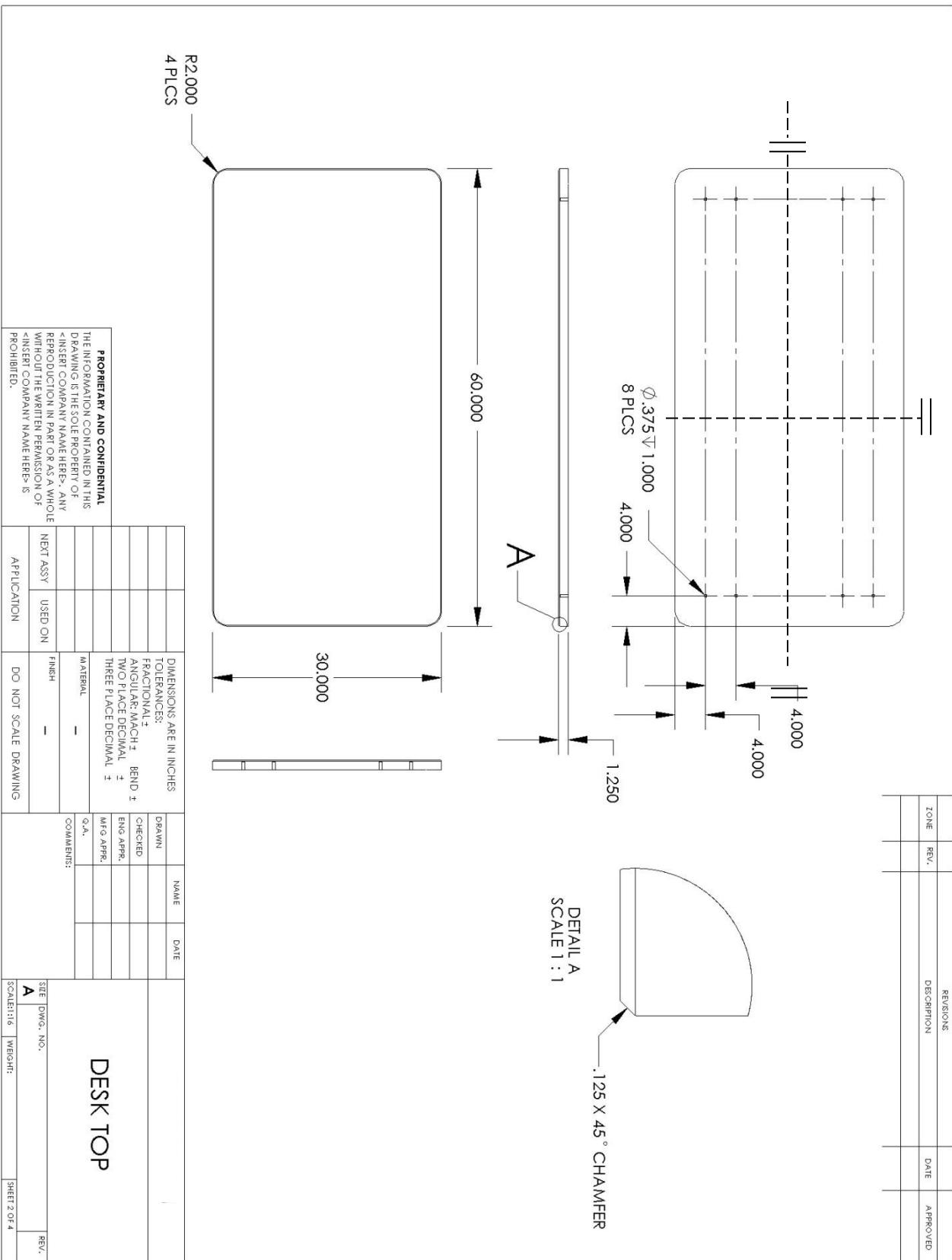
- When all the parts are finished, start an assembly and begin to assemble the components as shown in the assembly drawing provided.

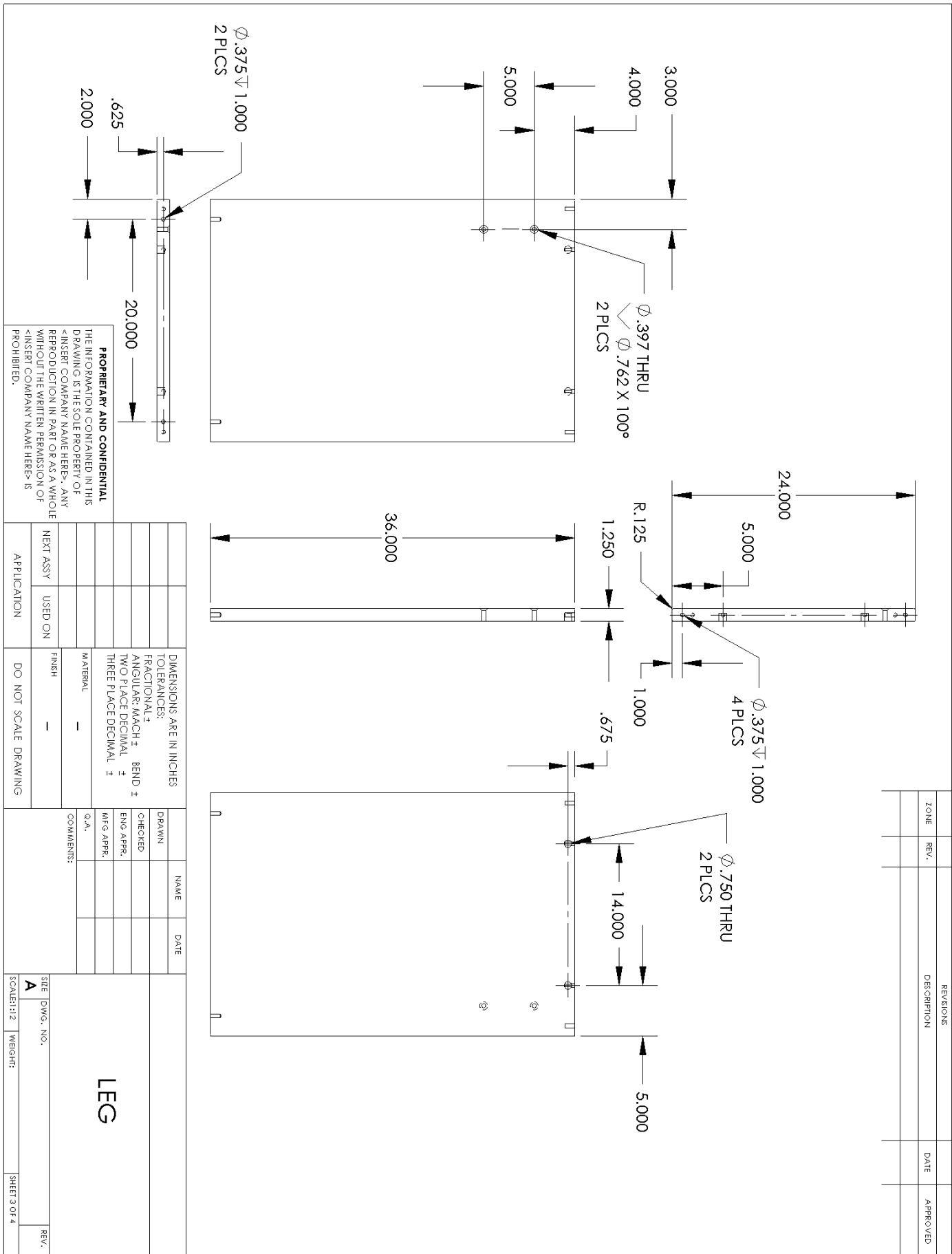


- Create a catalog image using Photoview 360. This is his how the finished model should appear after rendering.



REVISIONS		DATE	APPROVED
ZONE	REV.	DESCRIPTION	





ITEM NO.	QTY.	PART NO.	DESCRIPTION	ZONE	REV.	REVISIONS	DATE	APPROVED
1	1	Desk Top						
2	1	Leg						
3	1	Rib						
4	4	Fastening Bolt						
5	1	Mirror Leg						
6	4	Bolt						
7	4	Dowel						
8	4	Caster						

PROPRIETARY AND CONFIDENTIAL
THE INFORMATION CONTAINED IN THIS
DRAWING IS THE SOLE PROPERTY OF
<INSERT COMPANY NAME HERE>. ANY
REPRODUCTION IN PART OR AS A WHOLE
WITHOUT THE WRITTEN PERMISSION OF
<INSERT COMPANY NAME HERE> IS
PROHIBITED.

DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL ⁺ ANGULAR, MACH ⁺ TWO PLACE DECIMAL [±] THREE PLACE DECIMAL [±]			DRAWN	NAME	DATE			
			CHECKED					
			ENG APPR.					
			MFG APPR.					
			Q.A.					
COMMENTS:								
SIZE	DWG. NO.							
A								
SCALE: 1:20	WEIGHT:							
SHEET 1 OF 4								

DESK ASSEMBLY

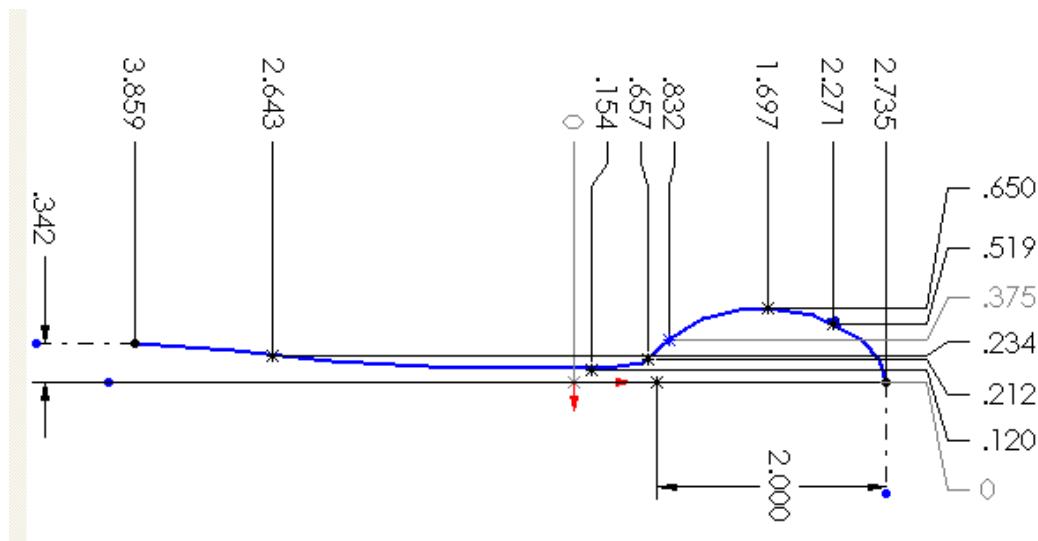
EXERCISE 21

Introduction to Surfacing

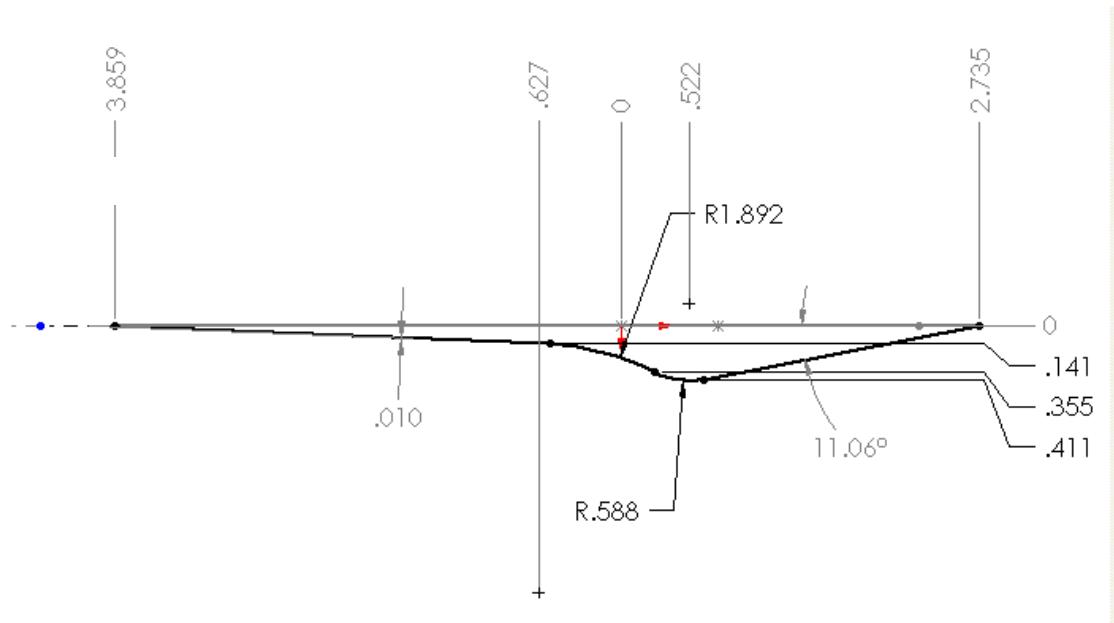
1. Here is an example of how to use surfaces. The spoon model will be used to introduce the user to the primary surfacing tools available in SolidWorks.



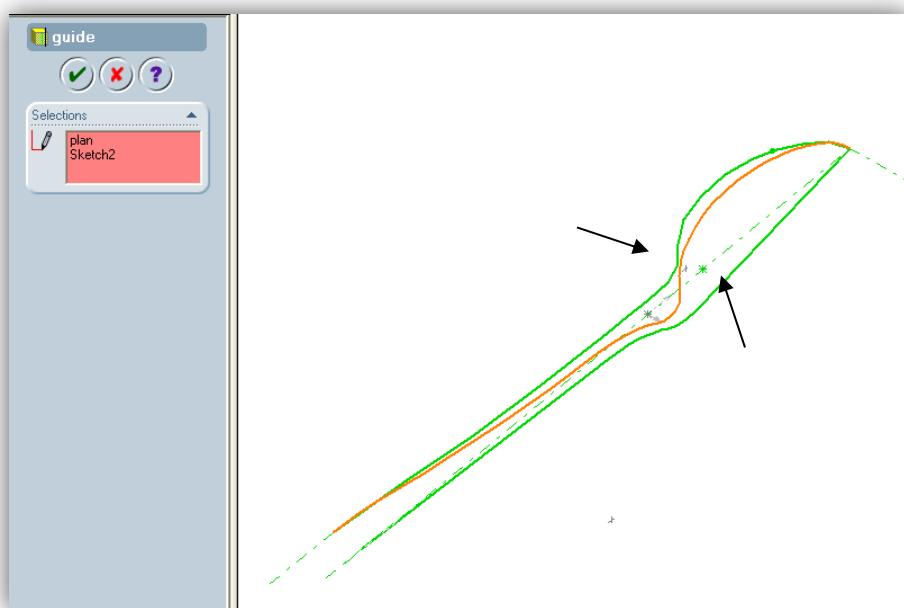
2. Sketch the following on the "Front" plane.



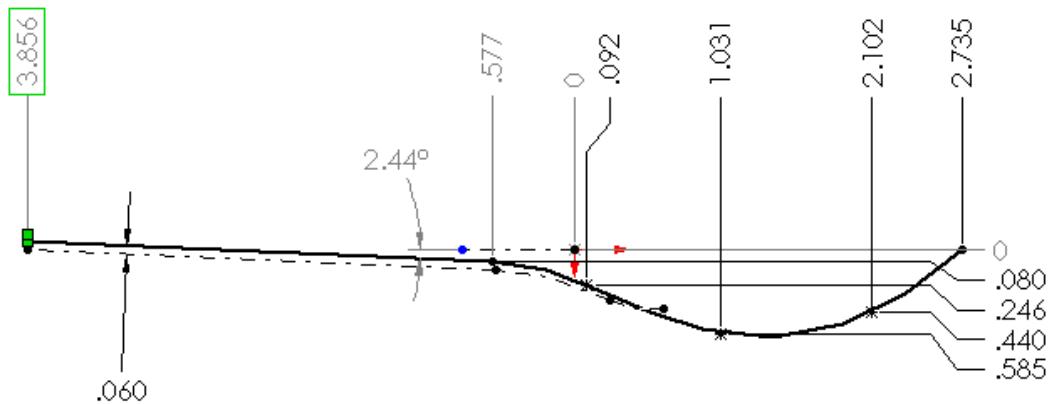
3. Sketch the following on the “Right” plane.



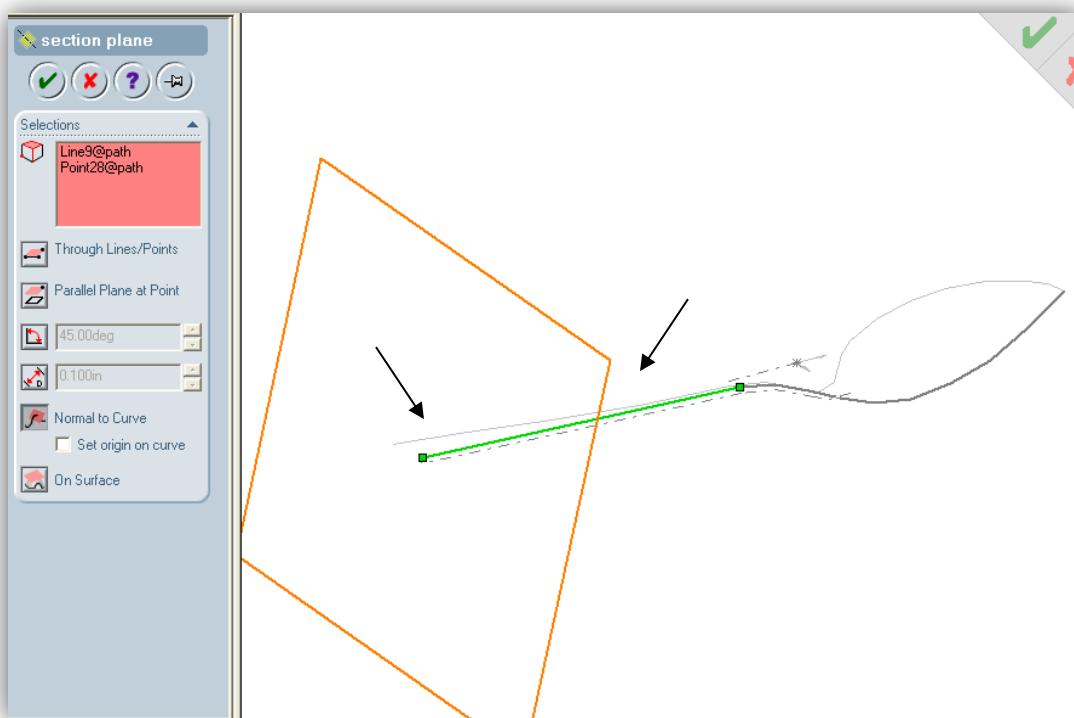
4. Go to Insert/Curve/Projected and select the two sketches to create a 3D projected curve. Change the feature name to “Guide”.



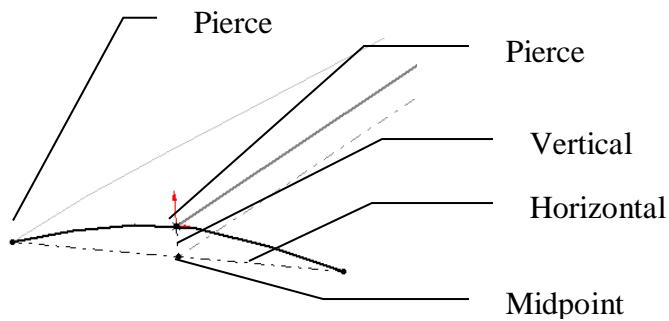
5. Draw the “Path” on the right plane.



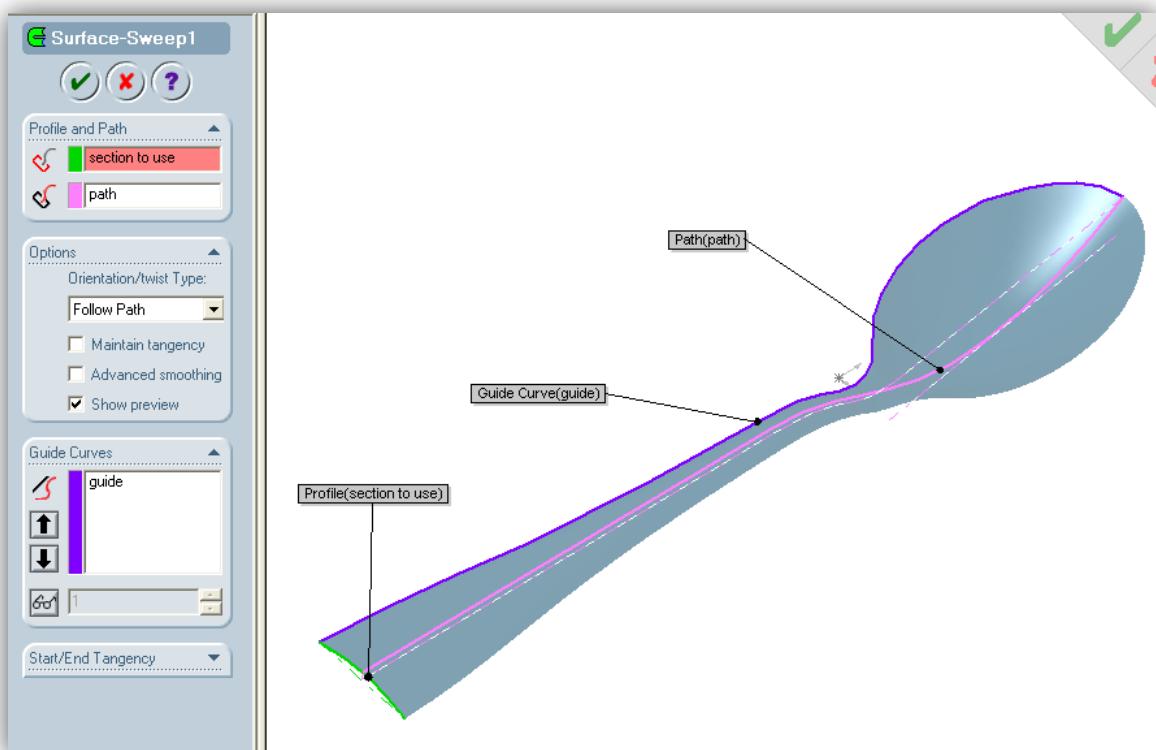
6. Create a new plane Normal to curve.



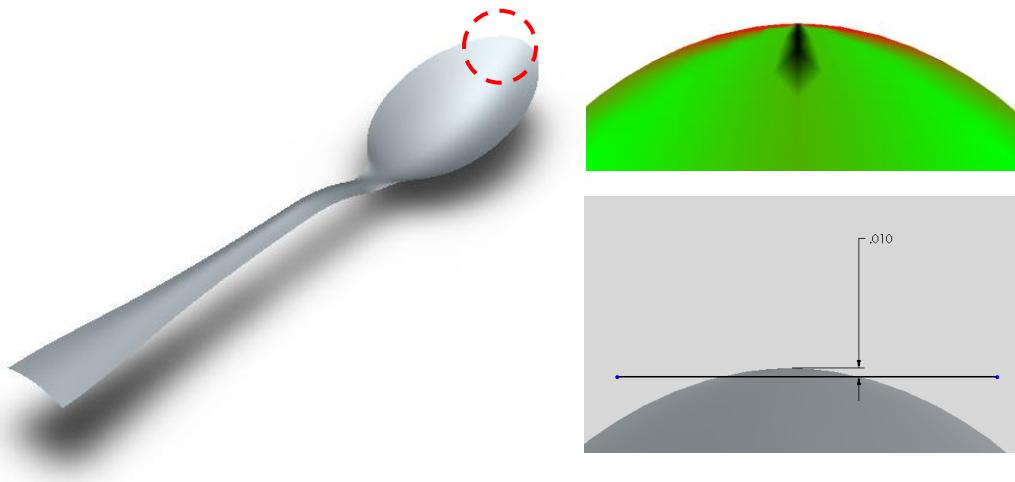
7. Start a sketch on the new plane and draw the following using a spline. Use relations to constrain the sketch.



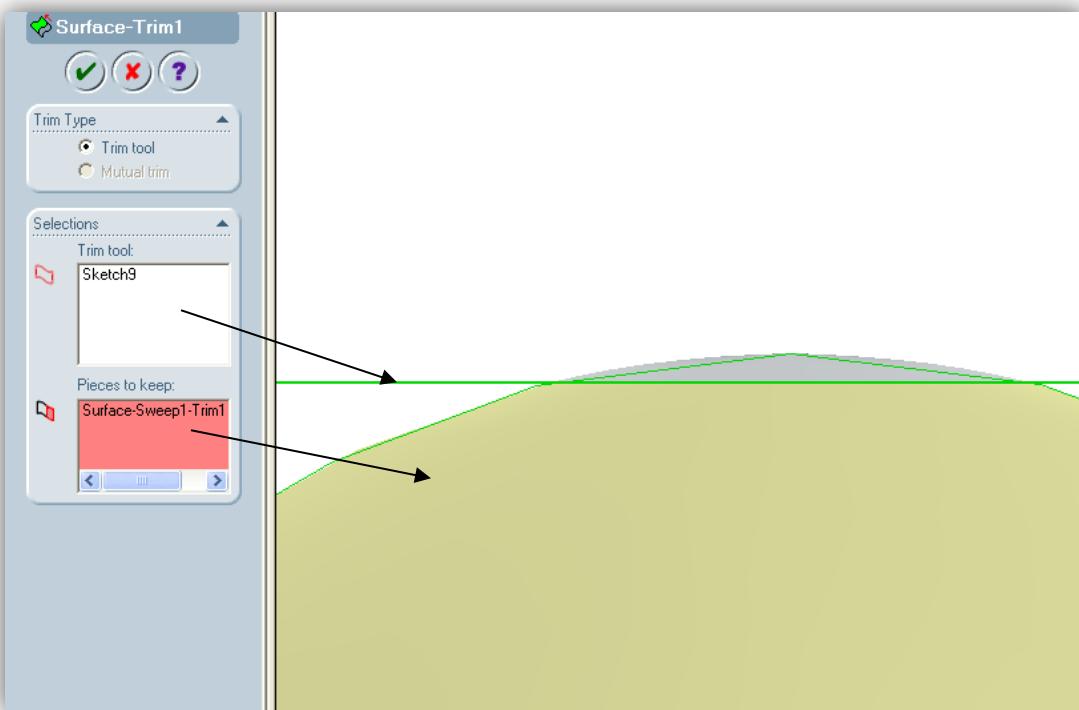
8. Add a surface sweep.



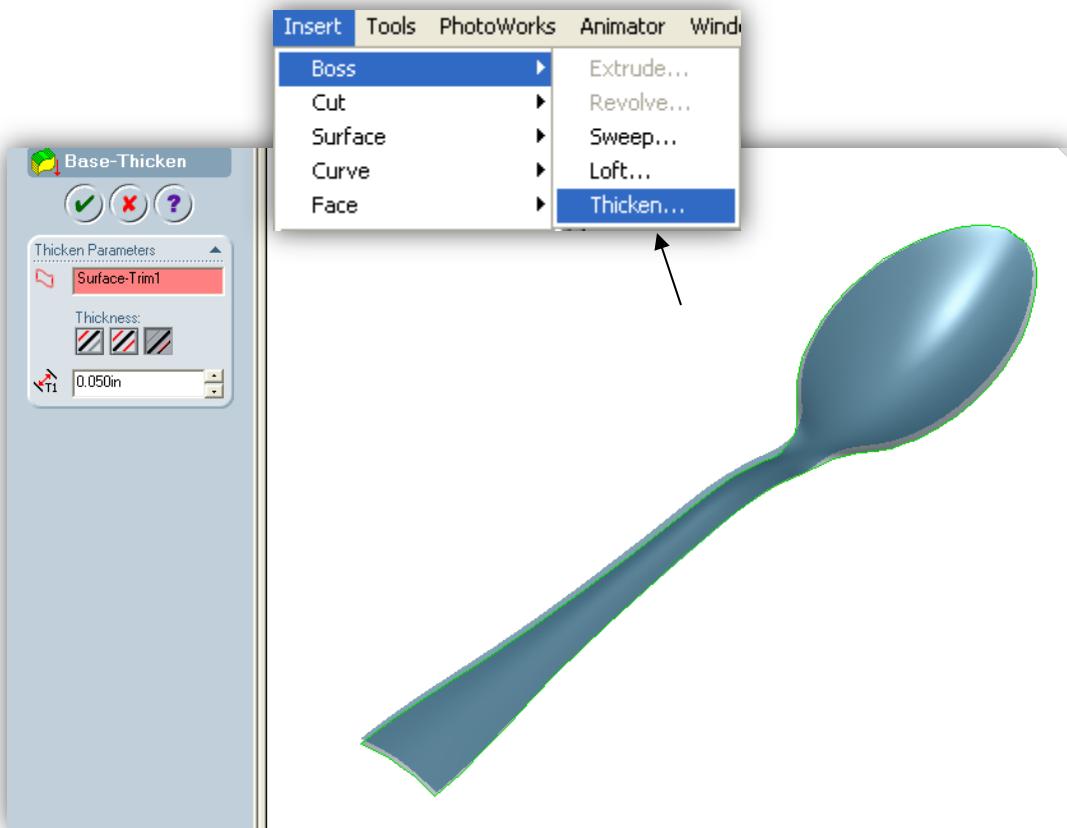
9. Check for surface flaws by right mouse clicking on the surface and select display curvature. Trim off the bad end by sketching a line on the front plane.



9. Select the trim surface icon.

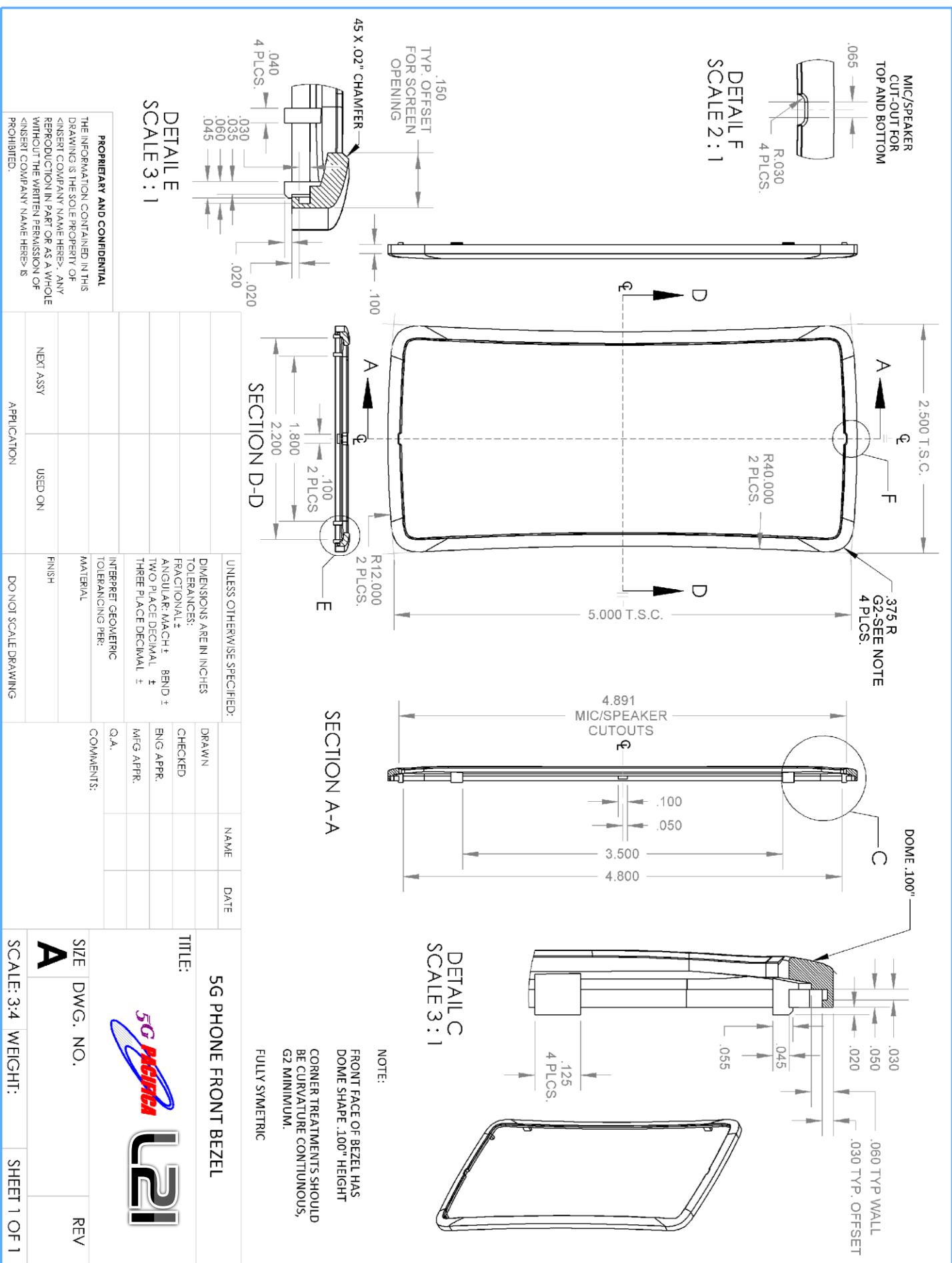


10. Select the Base-Thicken tool. Add .050".



11. You should now have a solid. Try and make the end on your own now.

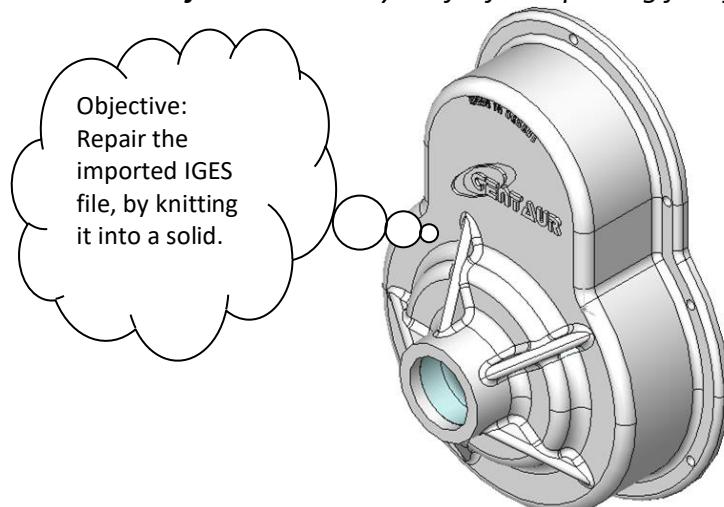




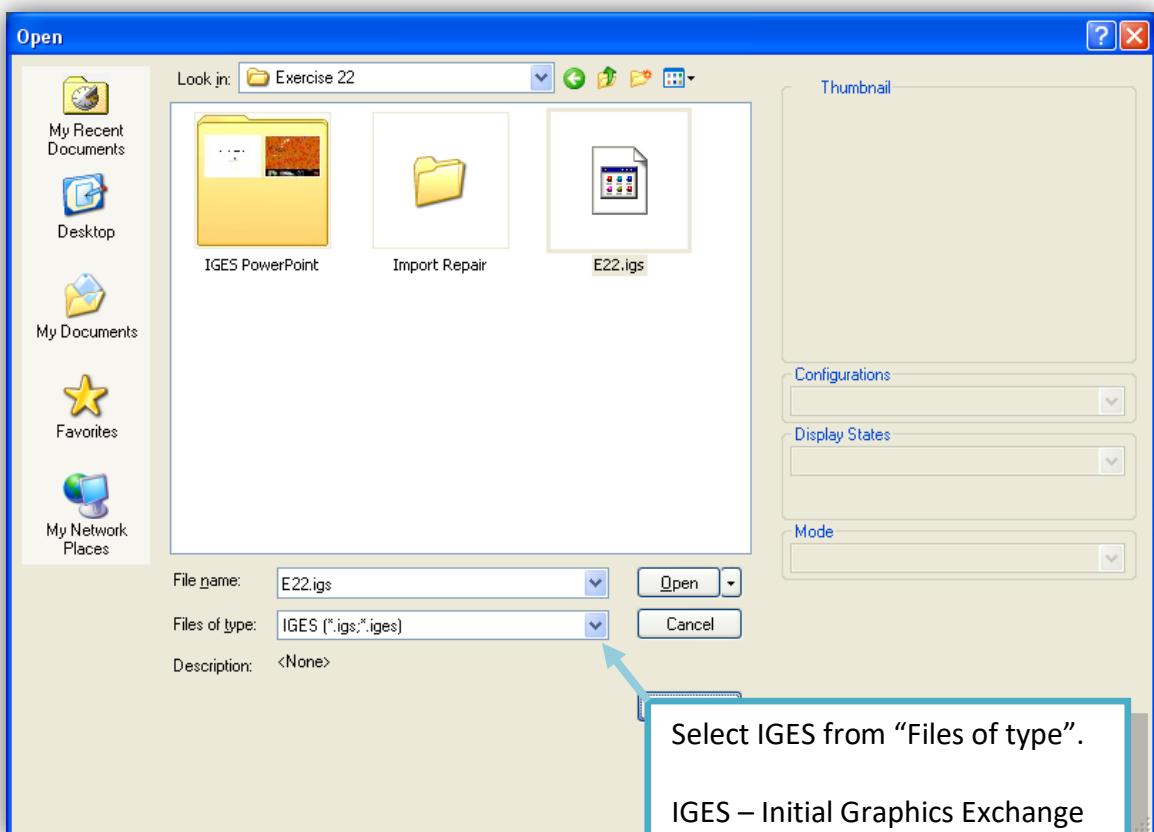
EXERCISE 22

Imported 3D Model Repair

IGES files can be very useful for importing files from other systems.

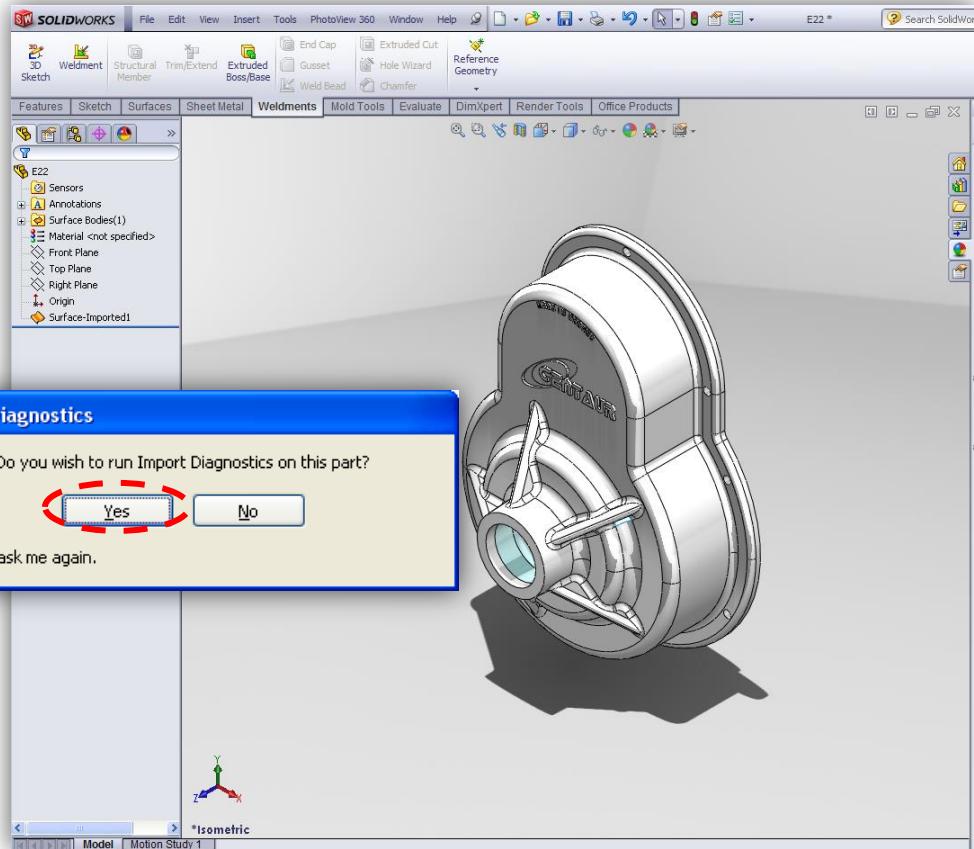


1. Go to File/Open, and select IGES.

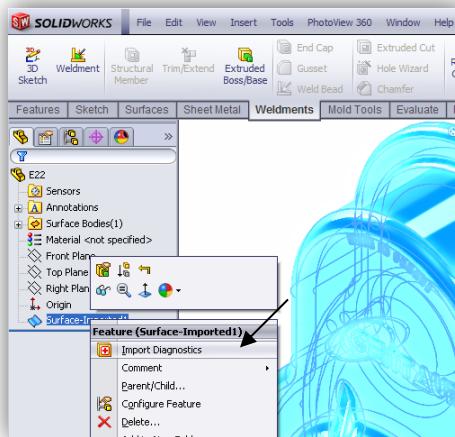


2. Once imported you will receive a message “Import Diagnostics”. Select “Yes”. Essentially in order to make use of this

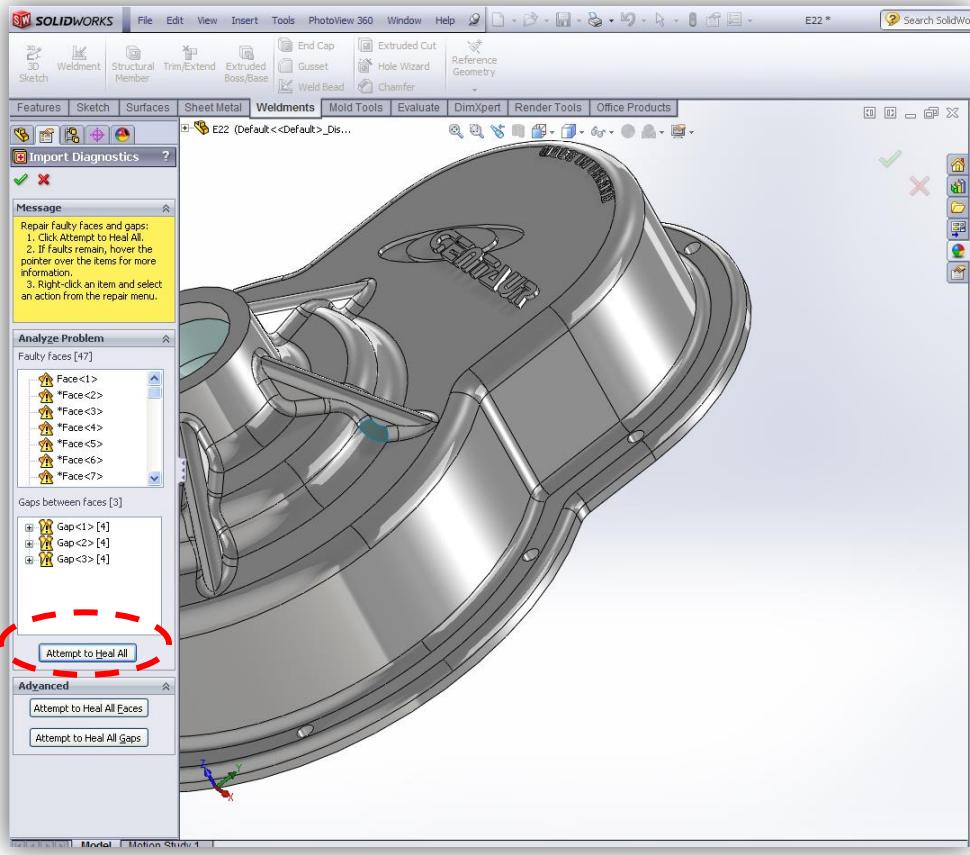
(make a cavity/mold) it is imperative that it be knit into a solid. This is an indication that there may be gaps in the surfaces.



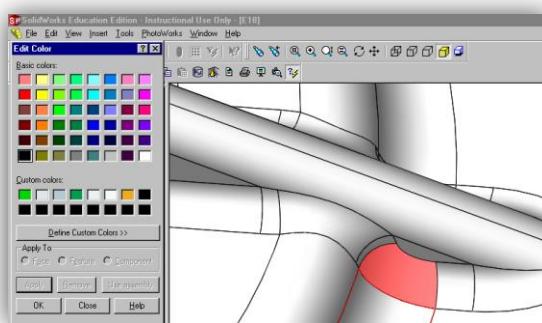
3. Note: If you fail to select Import Diagnostics you can simply RMB click on the surface in the feature tree, and select Import Diagnostics.



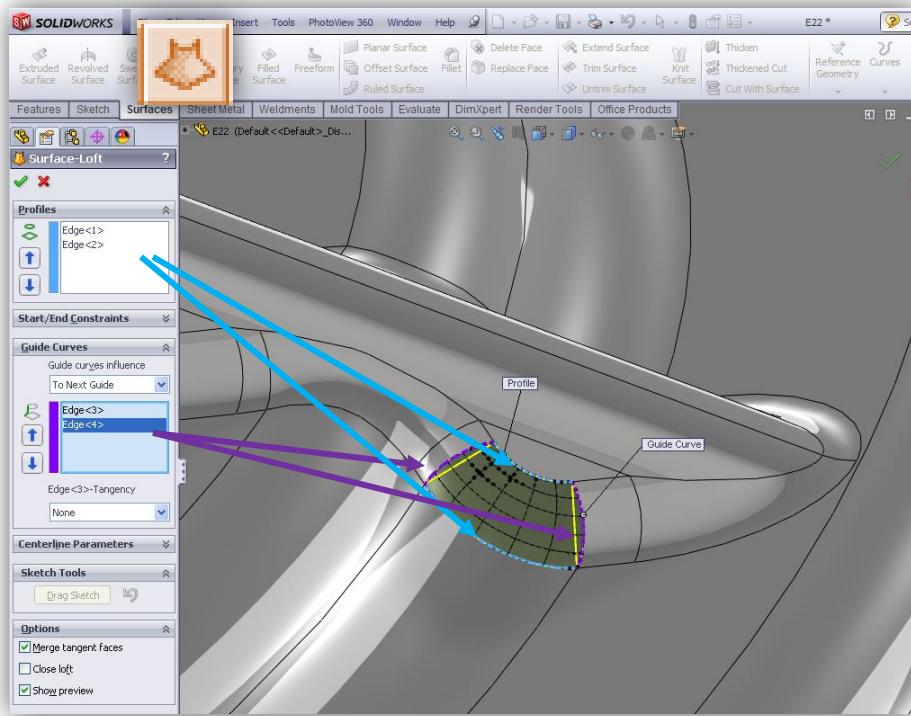
4. Select the Attempt to Heal in order to try and correct the gaps.



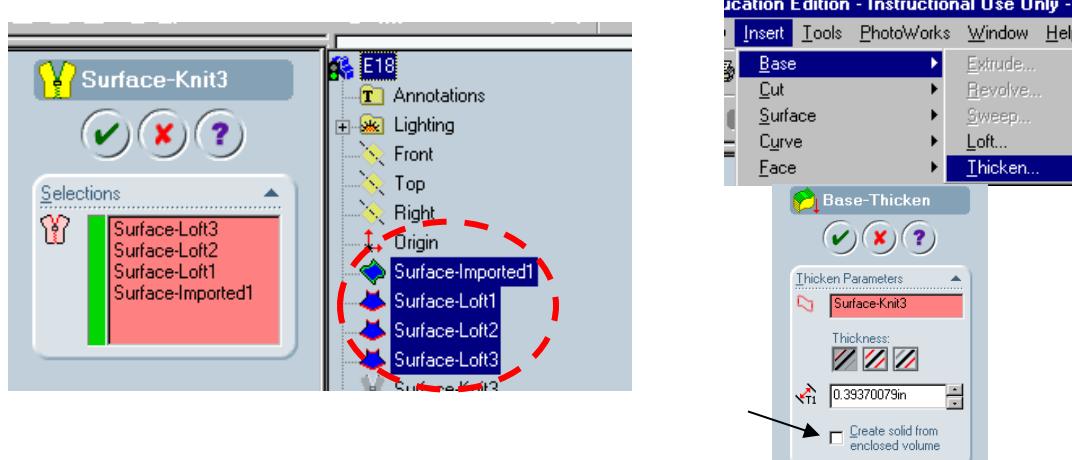
5. First try to have SWX automatically close the gaps by selecting the close All Gap Button. This works sometimes, but don't count on it. In this case it fails to close the gaps.
6. Now we must manually address the problem by creating surfaces to close the gaps. Use your surfacing toolbar.
7. Before canceling out of diagnosis be sure to locate and memorize the gaps. Changing the surface colors behind or around each gap easily does this.



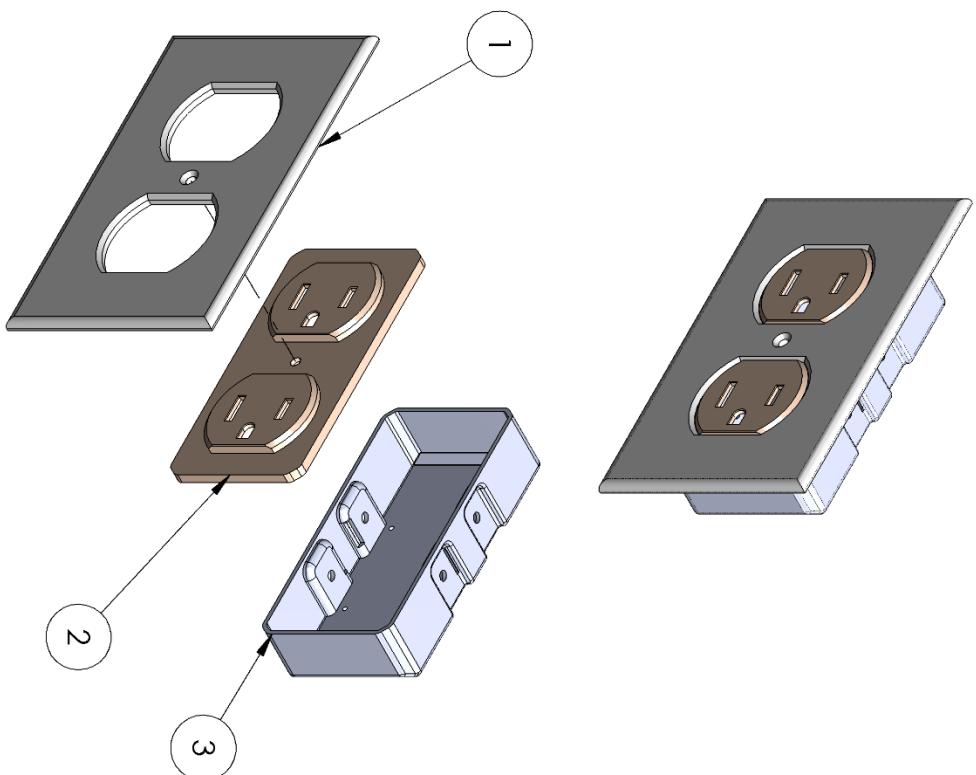
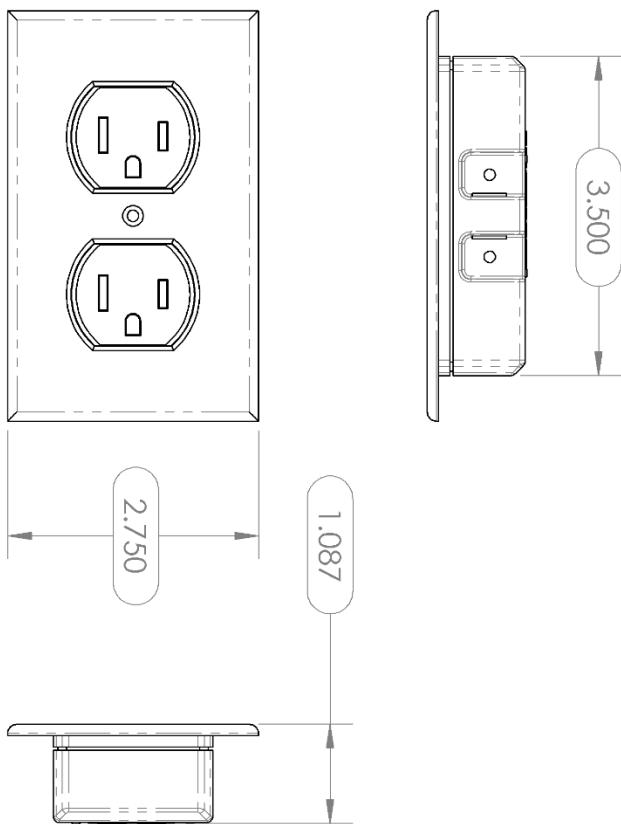
8. Activate the Surfaces tab and use the "Surface loft" tool. Select edges to use as Profiles and Guide Curves.



9. Once all gaps are patched use the “Knit” tool to stitch them together. Be sure to select all surfaces and the main body.
10. Then you can go to insert/base/thicken.
11. Be sure to select “Create solid from enclosed volume. You should now have a solid.



ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	LAB 3C		1
2	LAB 3d		1
3	LAB 3f		1



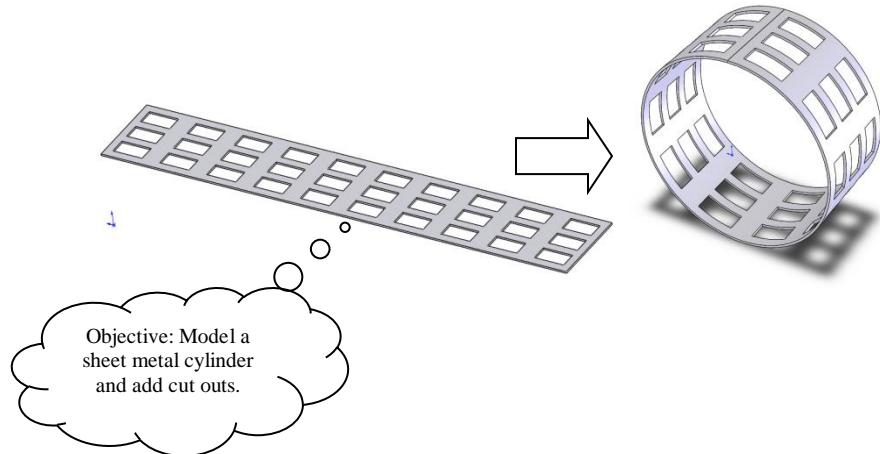
UNLESS OTHERWISE SPECIFIED:			
DIMENSIONS ARE IN INCHES	DRAWN	NAME	DATE
TOLERANCES: FRACTIONAL [±] ANGULAR MACH [±] TWO PLACE DECIMAL [±] THREE PLACE DECIMAL [±]	CHECKED		
ENG APPR.	MFG APPR.		
INTERPRET GEOMETRIC TOERANCING PER: MATERIAL	Q.A.	COMMENTS:	
NEXT ASSY	USED ON	SIZE DWG. NO.	
FINISH		A	REV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

PROPRIETARY AND CONFIDENTIAL
THE INFORMATION CONTAINED IN THIS
DRAWING IS THE SOLE PROPERTY OF
<INSERT COMPANY NAME HERE>. ANY
REPRODUCTION IN PART OR AS A WHOLE
WITHOUT THE WRITTEN PERMISSION OF
<INSERT COMPANY NAME HERE> IS
PROHIBITED.

EXERCISE 23

Sheet Metal Cylinders

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



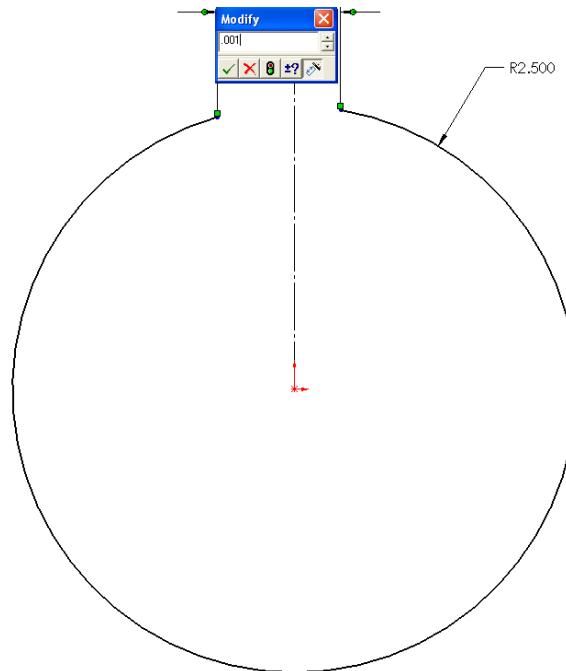
12. Go to file/new and select new part and save as “E23”.

Sheet Metal Tool Bar:

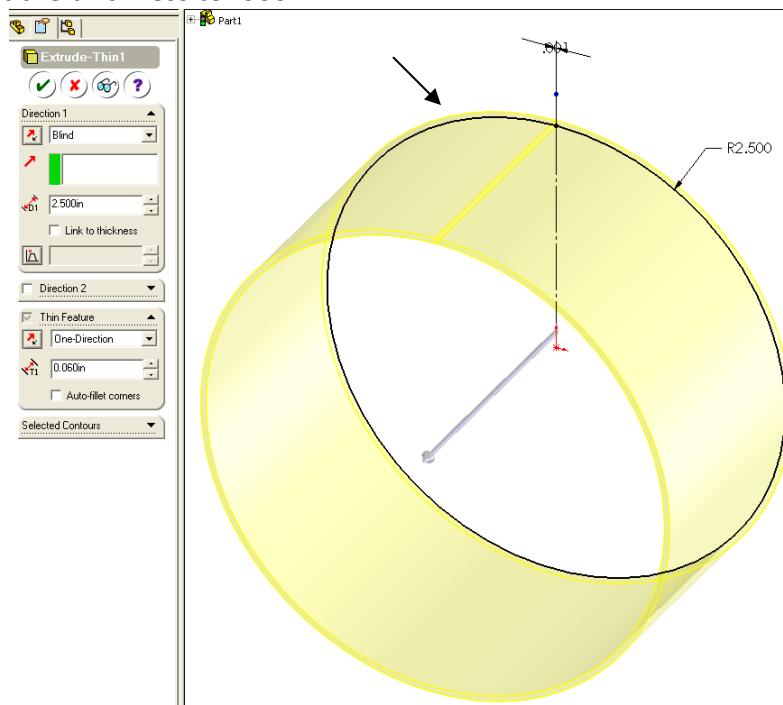
To activate the toolbar on the ribbon simply RMB click on any ribbon tab and check the Sheet Metal box.



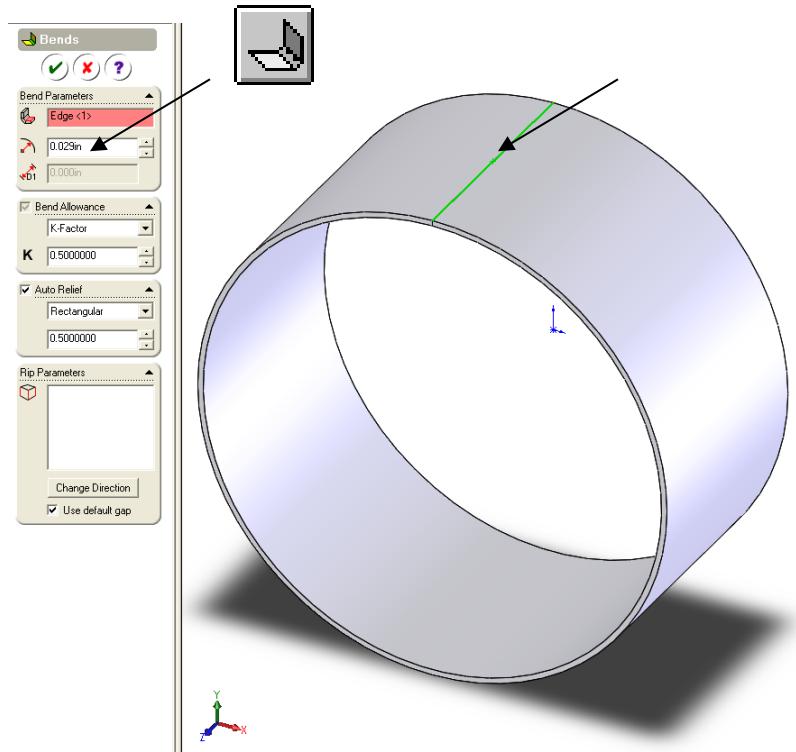
13. Draw the following sketch on the “Front” plane, use the “center point arc” tool. Make both ends of the arc symmetric to a vertical centerline. Space @ .001”



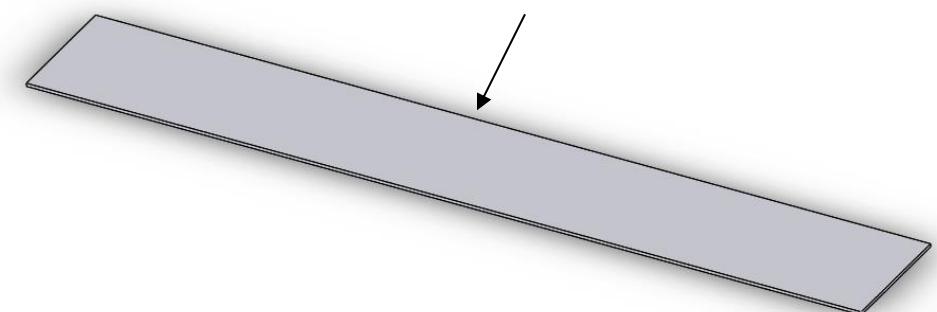
14. Boss Extrude blind 2.5”. Notice that it should be creating a thin feature and set the thickness to .060”.



15. Select the edge of the cylinder and select Insert Bends. Set radius to "0". Hit OK.



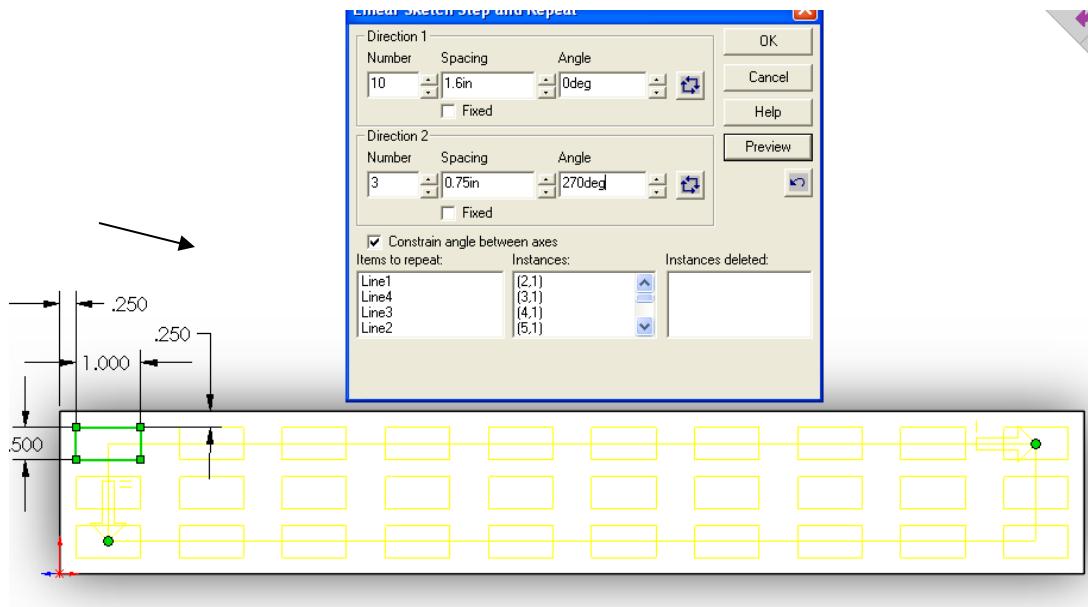
16. Select the "Flatten" icon to verify. Then select the flatten icon to fold back up.



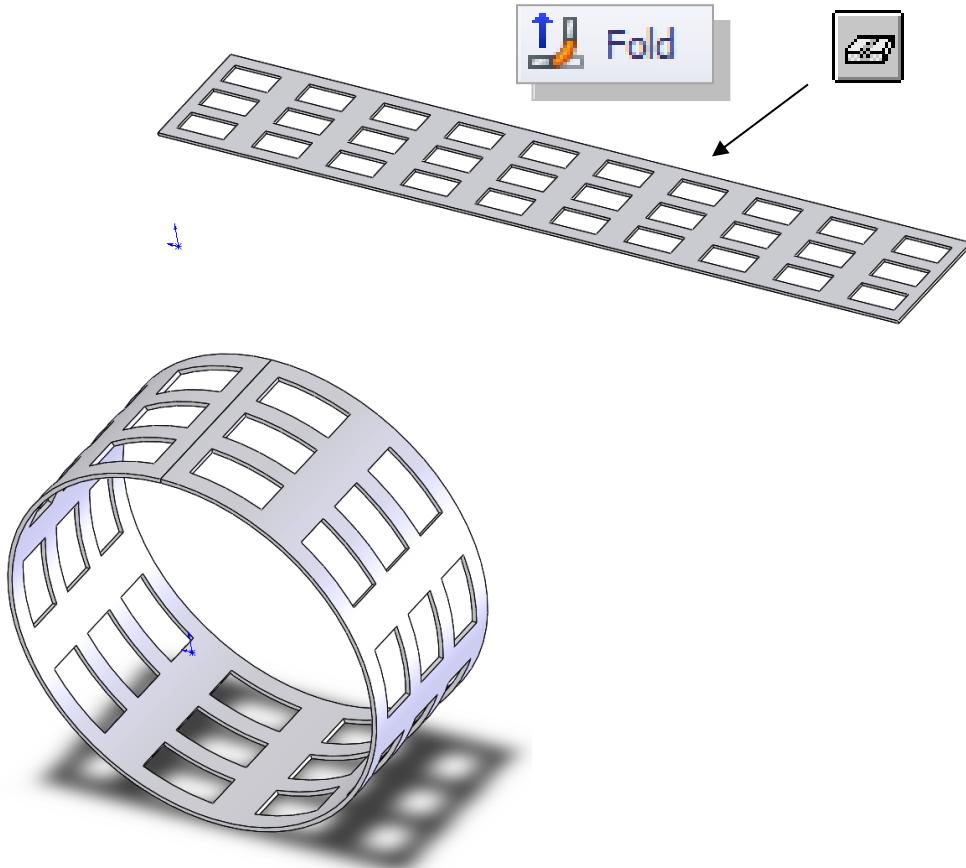
17. To create a cut pattern on the sheet metal in the flattened state you must first flatten it using the Unfold tool. And select the break edge to be fixed.



Draw a rectangle as seen below. Use the Sketch pattern tool to pattern rectangle.



18. Select the “Fold” icon, to refold it. Then you can use “Flatten” to verify.

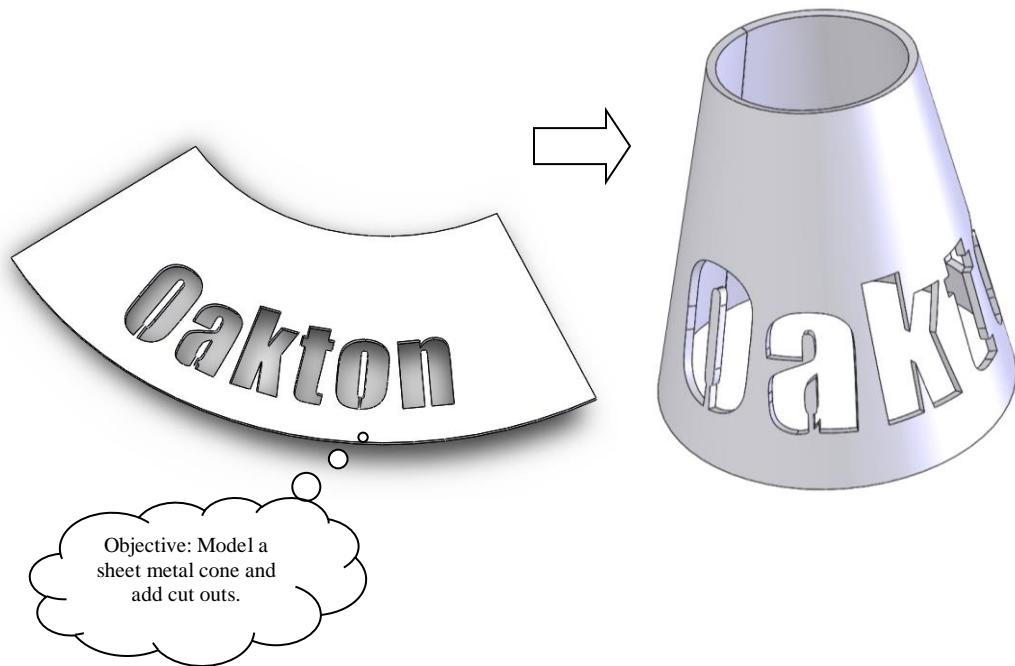


EXERCISE 24

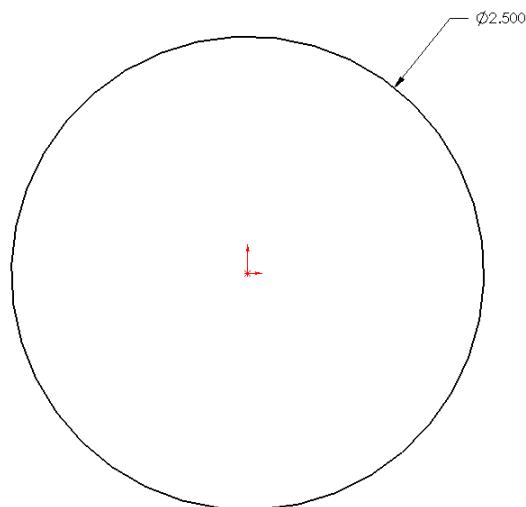
Sheet Metal

Modeling Conical Parts

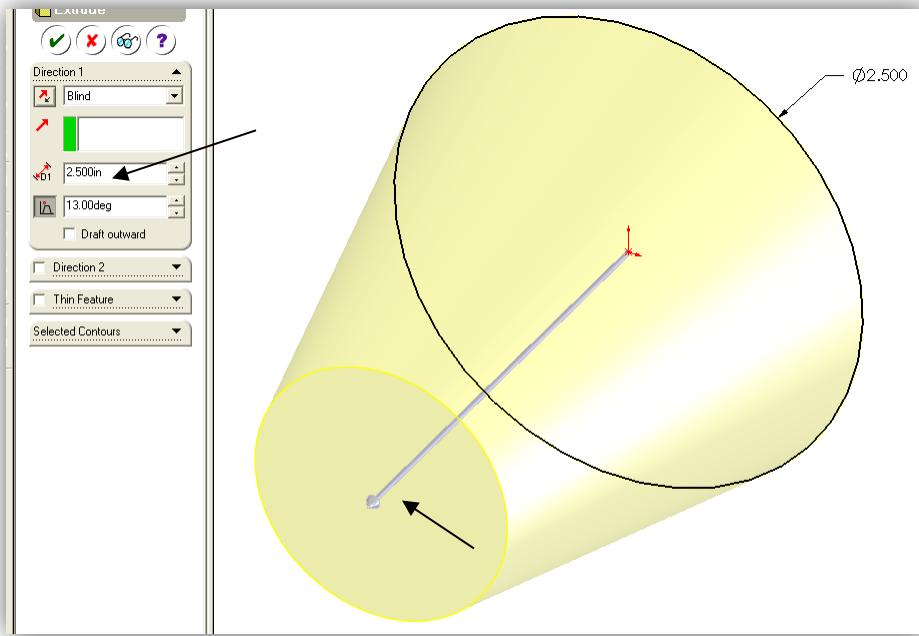
Sheet Metal part files can be very useful for extracting a flat pattern.



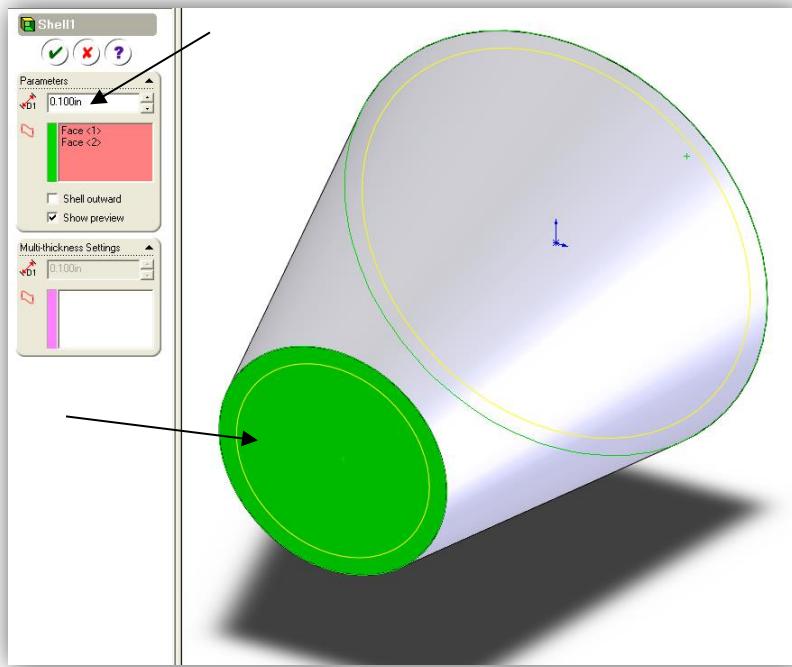
1. Go to file/new and select new part and save as “E24”.
2. Draw the following sketch on the “Front” plane, use the “circle” tool.



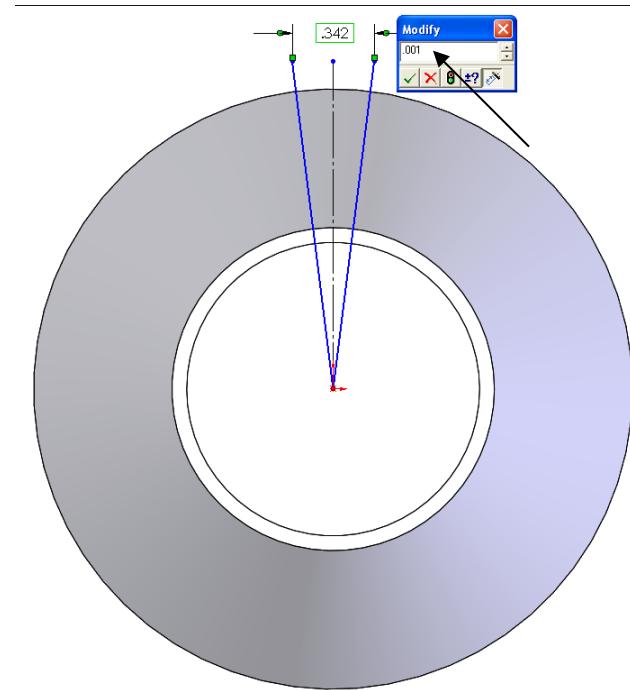
3. Boss Extrude blind 2.5" and add 13° Draft.



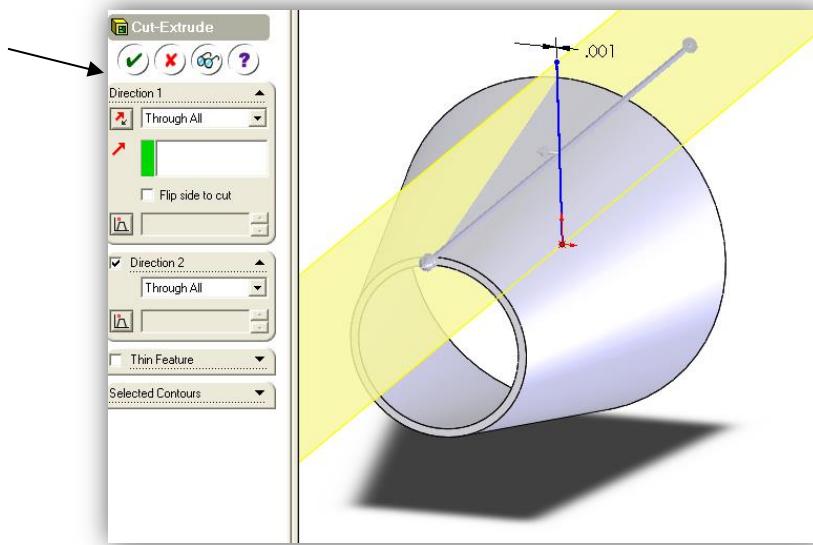
4. Select the front and back planar faces, then select the Shell feature. Set thickness to ".060". Hit OK.



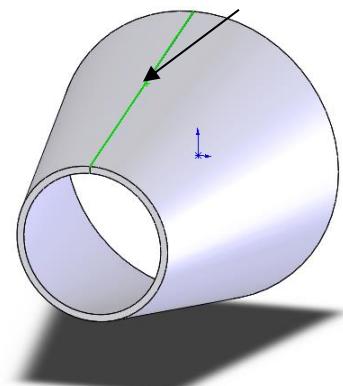
5. Select the “Front” plane and start a sketch on it. Draw the following angled cutout. Dimension the edges and set the thickness to .005”.



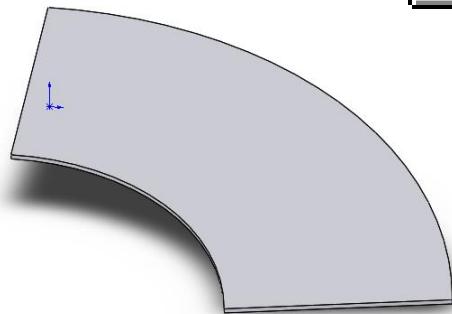
6. Cut Extrude Through All.



7. Select the cut edge of the part and then select the “Insert bends” icon.



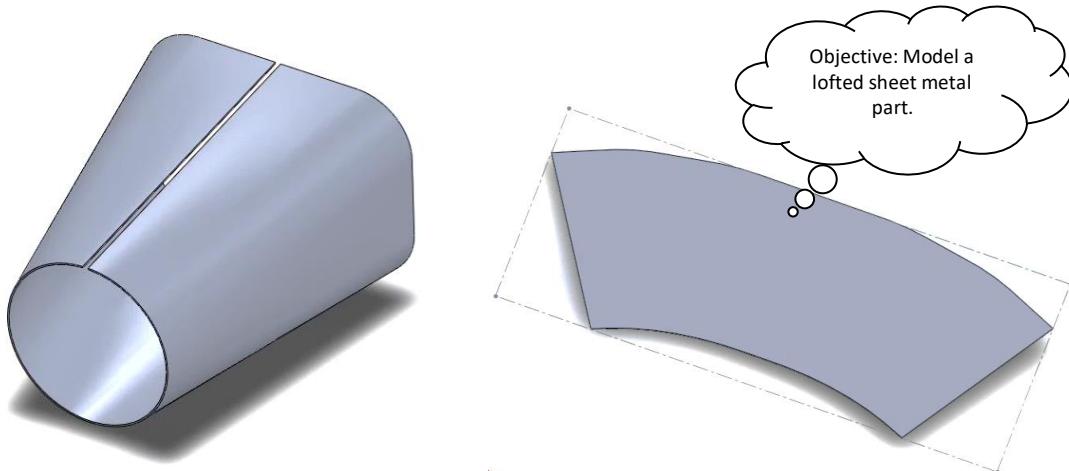
8. Select the Flatten icon to unfold.



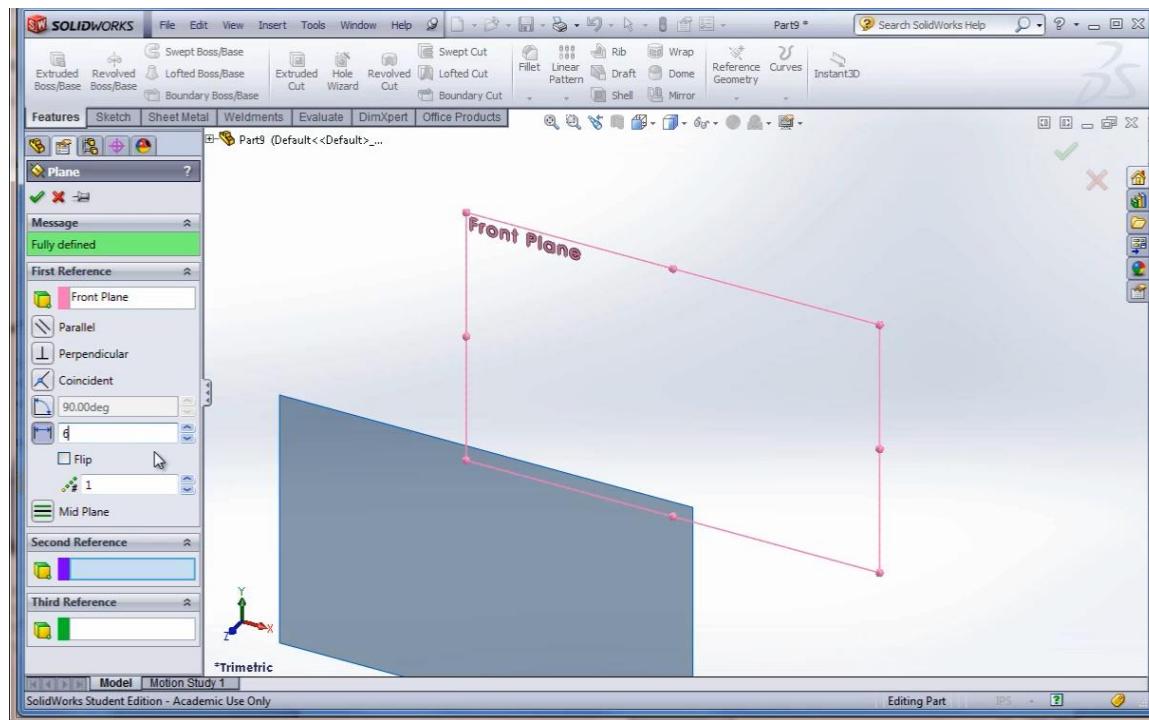
EXERCISE 25

Sheet Metal Lofts

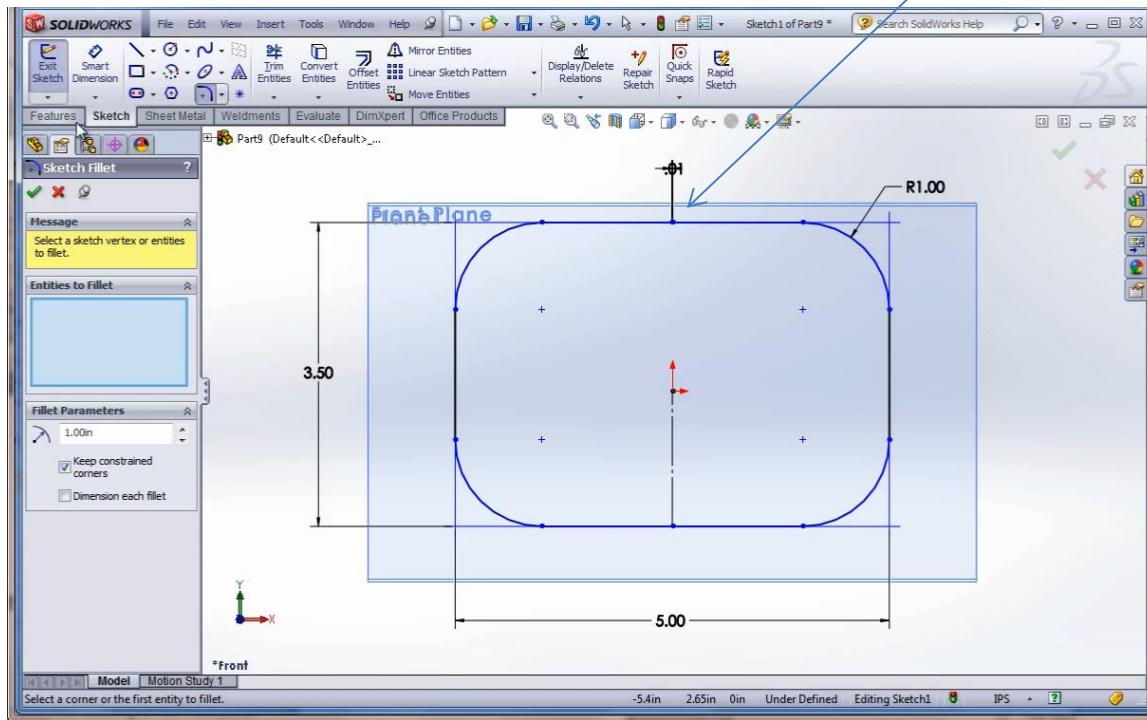
Lofted Sheet Metal creation can be very useful for complex sheet metal fabrication.



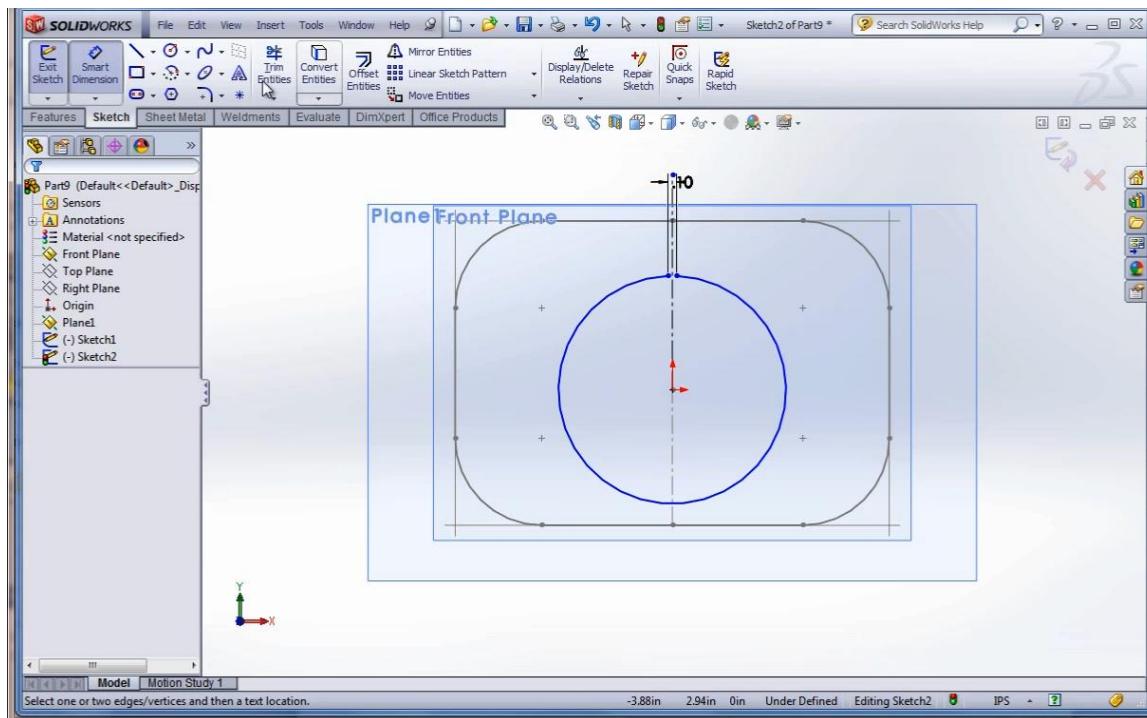
1. Begin by offsetting a plane 6" forward from the Front plane.



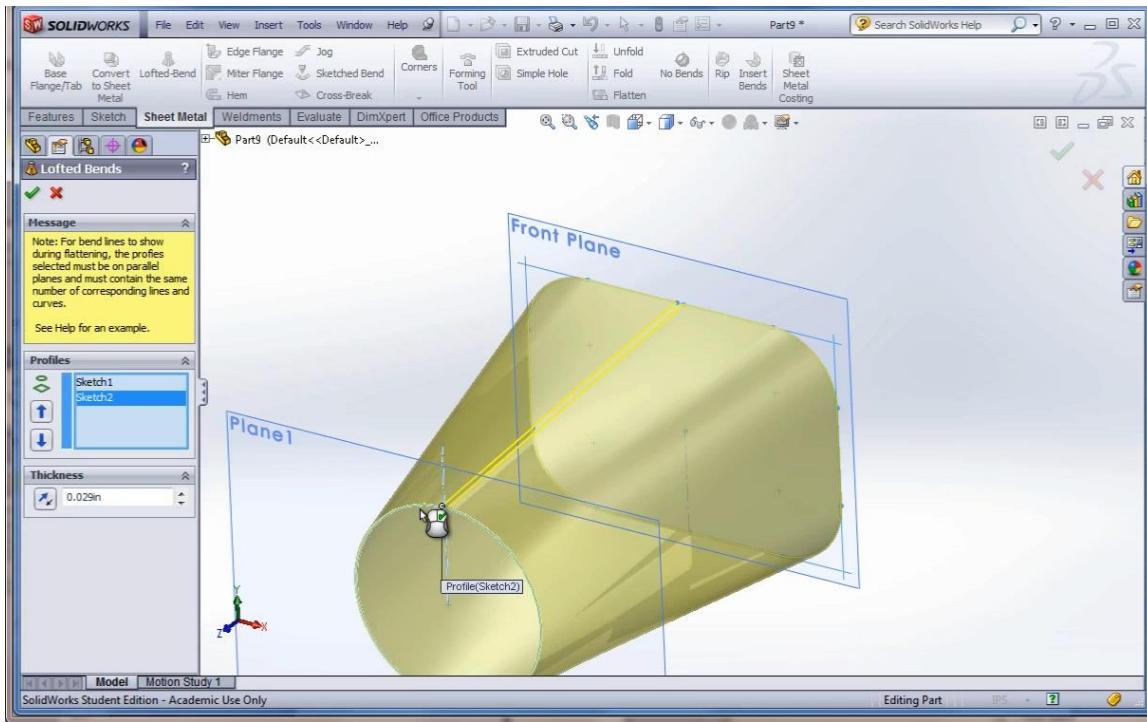
2. Sketch the following on the front plane. Leave a small open gap of .1 in the center top edge. Rebuild



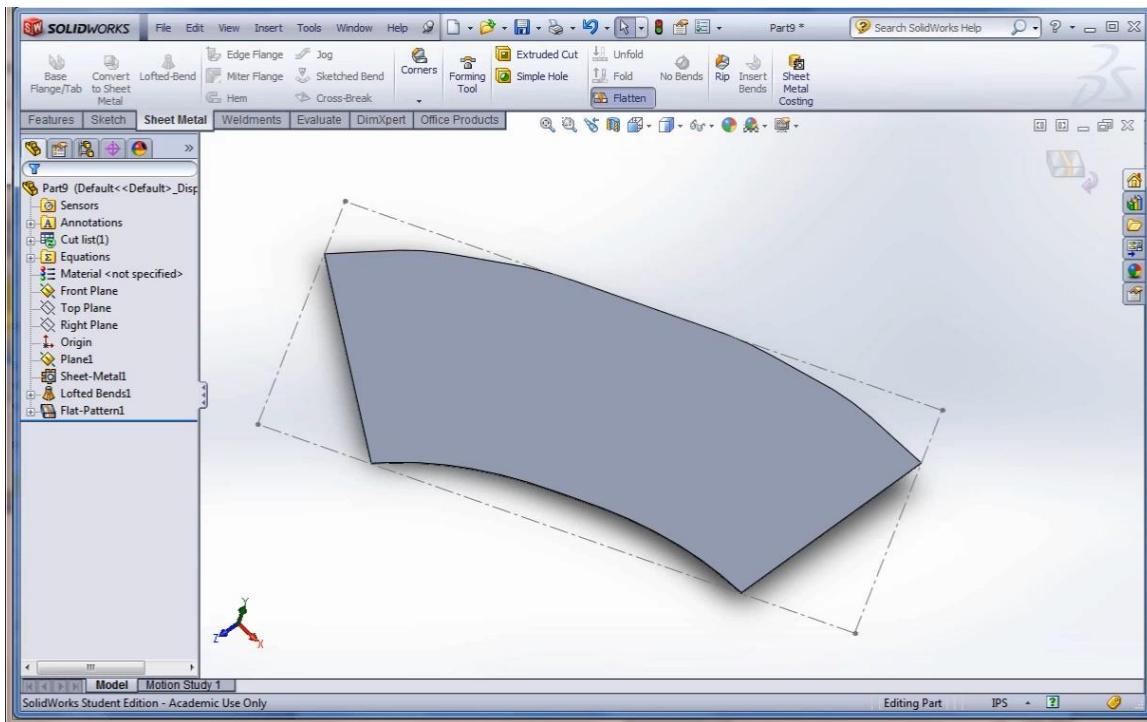
3. Sketch the following on the offset plane. 1" Radius. Rebuild



4. Select the Sheet Metal Loft tool, and select the sections.



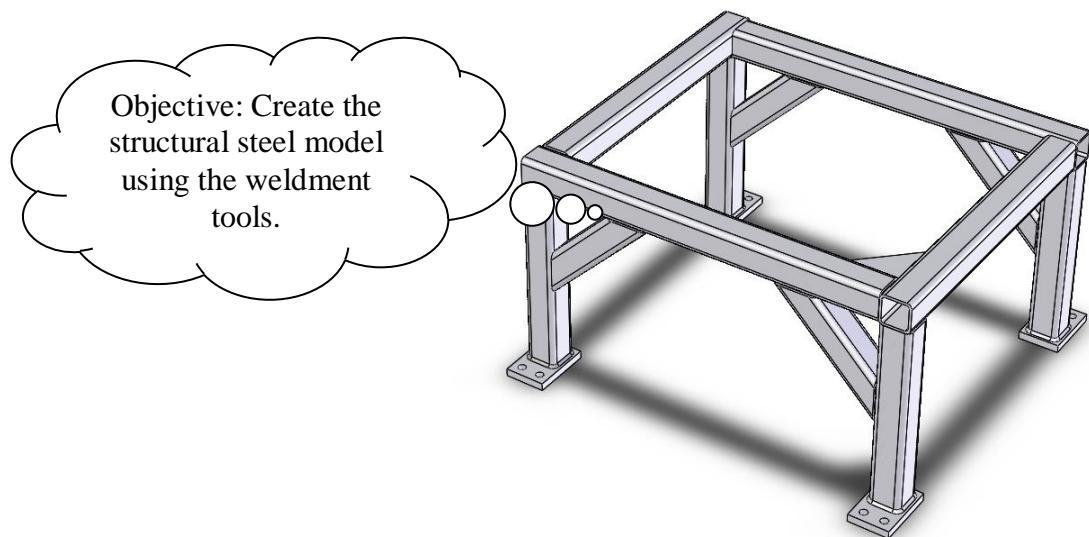
5. Flatten to verify.



EXERCISE 26

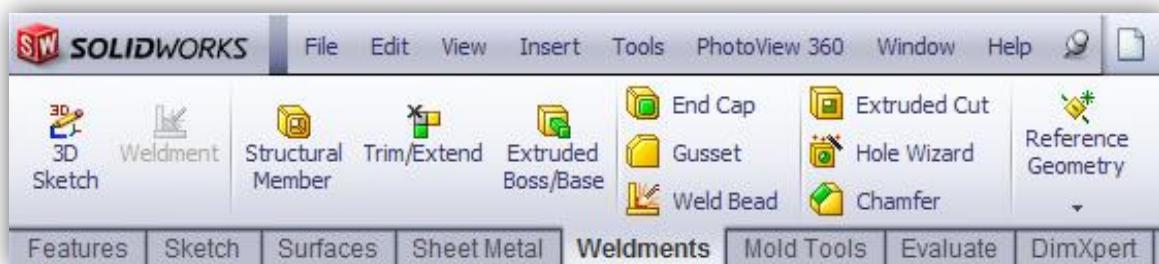
Weldments

Weldment creation can be very useful for automating layout designs and extracting cut lists.

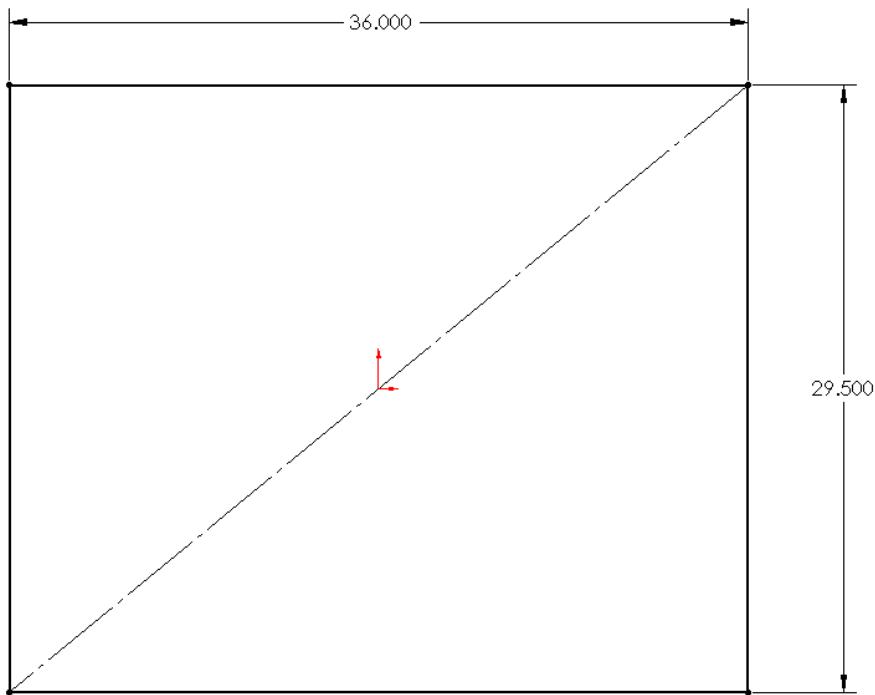


1. Start a new part file (ANSI inch) and save it as E25.

2. Activate the Weldments tool bar by RMB clicking on any ribbon tab, and select the “Weldments” checkbox.



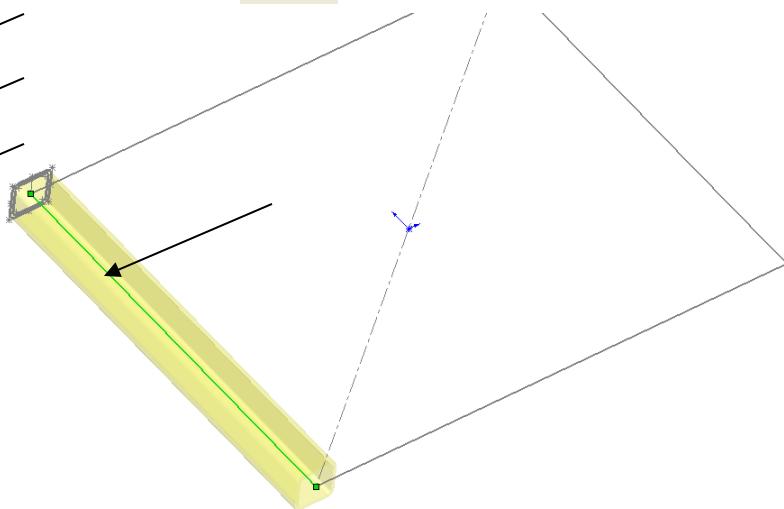
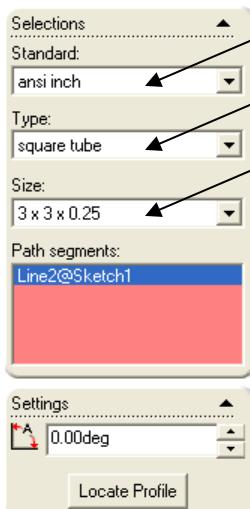
3. Start a sketch on the “Top” plane and draw the following.



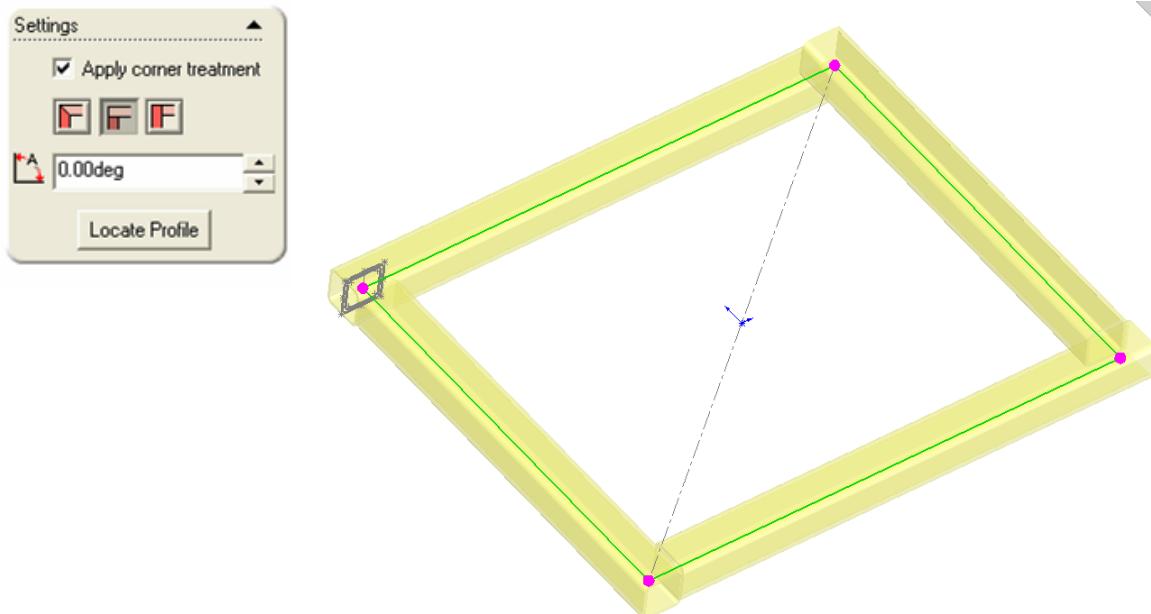
4. Select the Weldments Icon to convert the part to a weldment entity.



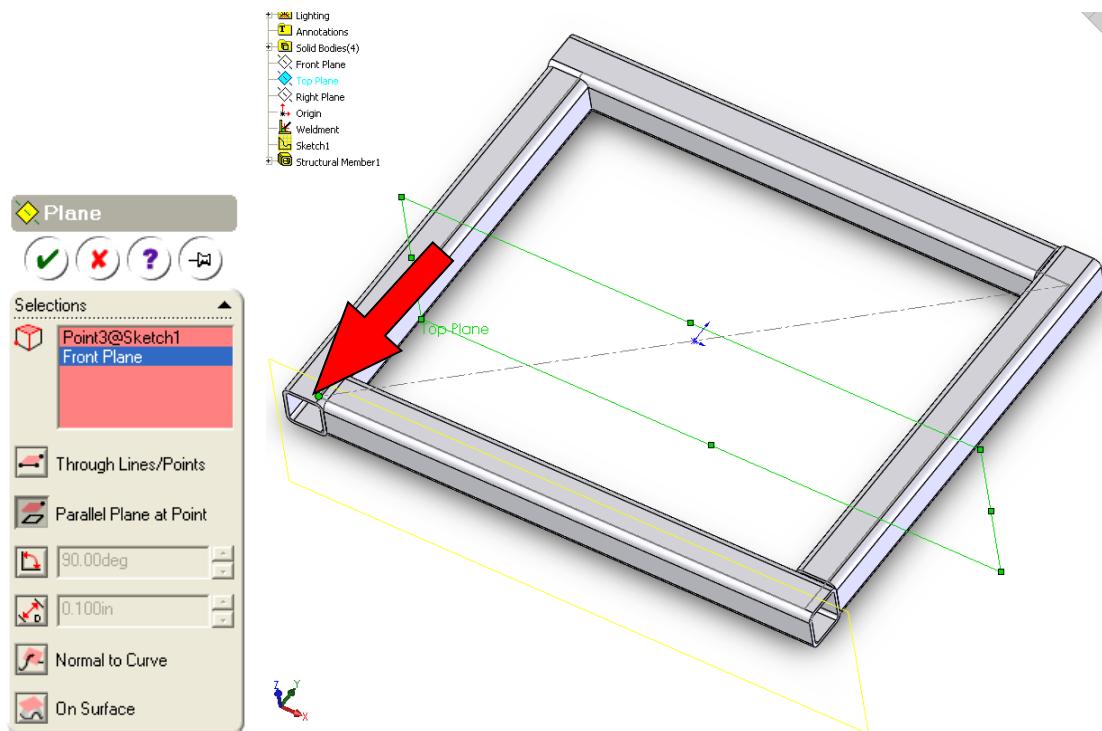
5. Select the structural member icon. Also select one sketch line. Input values.



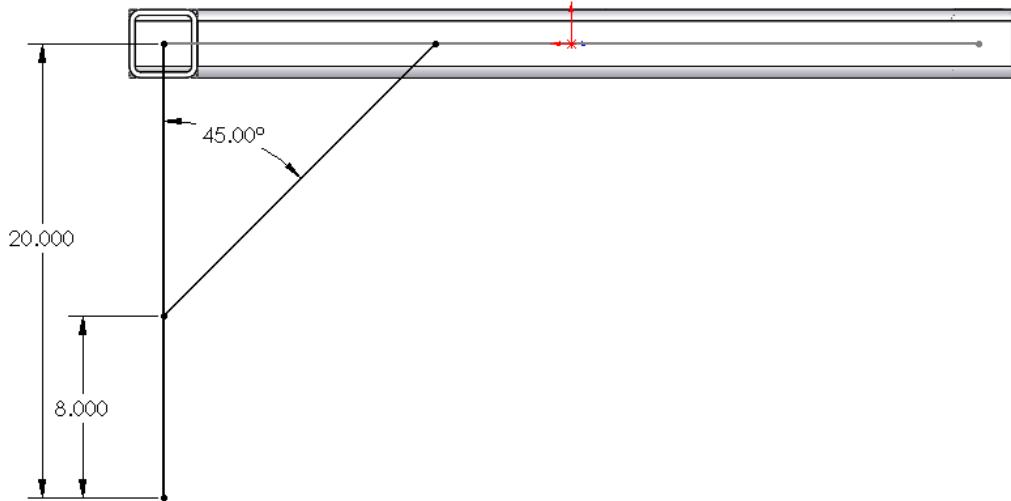
6. Select the other three sketch lines. Notice the end but settings.



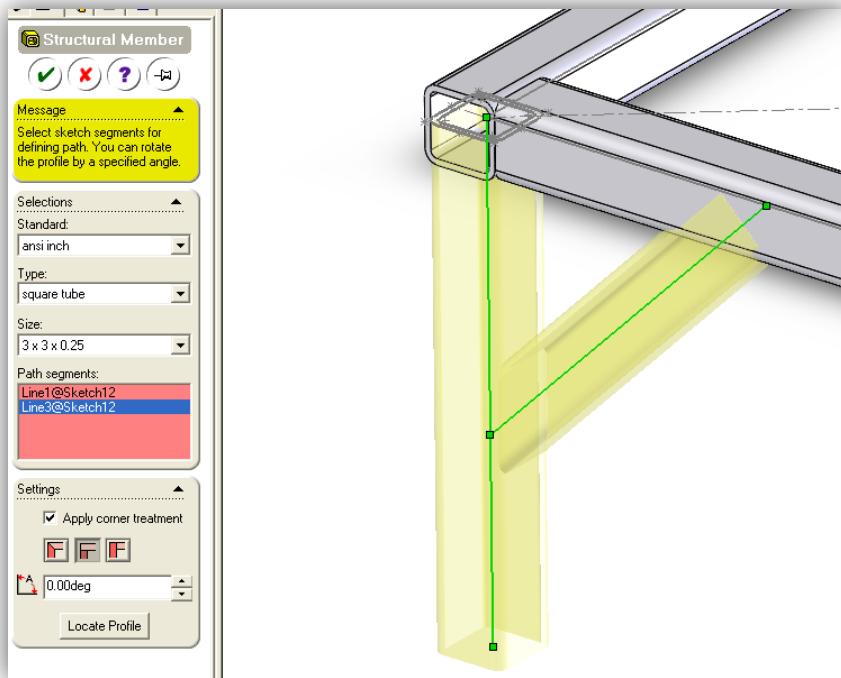
7. **Offsetting a plane parallel to a point.** Select the front plane and a corner of the layout sketch you created in step 3. Go to the Plane Wizard.



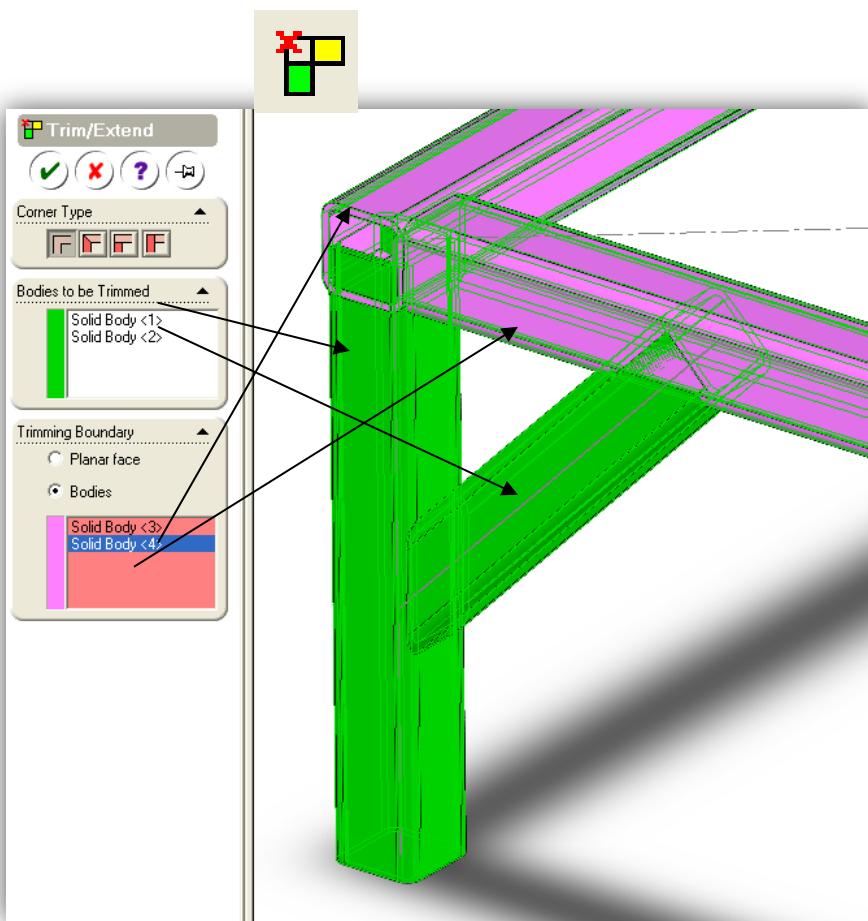
8. Sketch the layout of the leg. Start a sketch on the new plane and draw the following. Rebuild



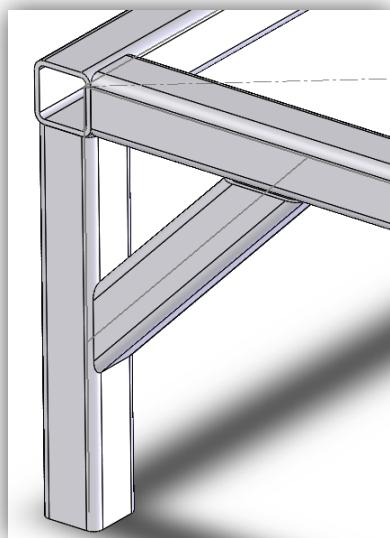
9. Select the Structural Members icon. Select the vertical and 45 degree angled line. Use the same settings as before. Hit the green check to apply.



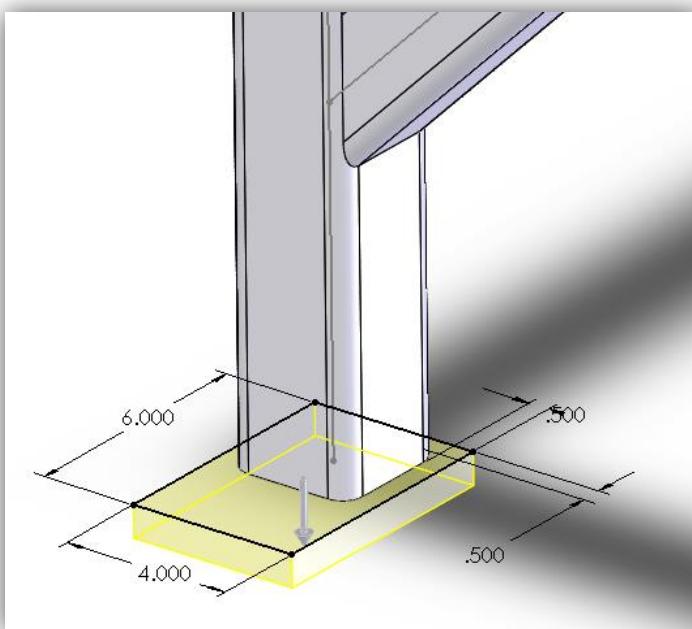
10. Select the Trim/Extend icon. Then select the Bodies and Boundaries.



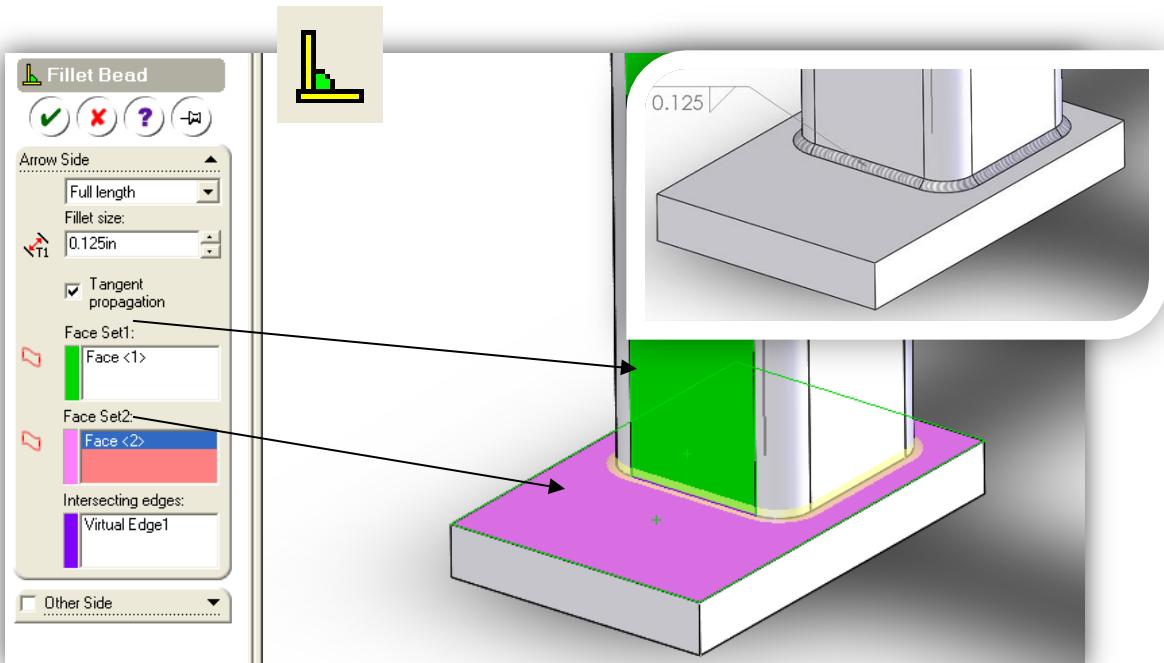
11. It should look like the image below.



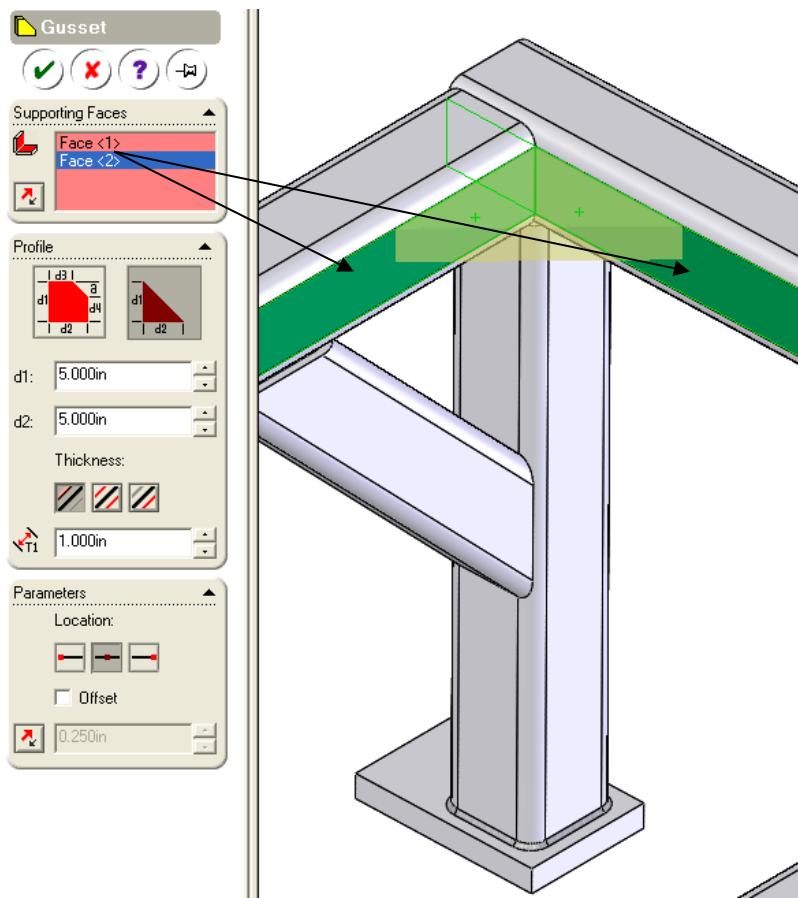
12. Select the bottom cut surface of the leg. Draw the following and extrude .75".



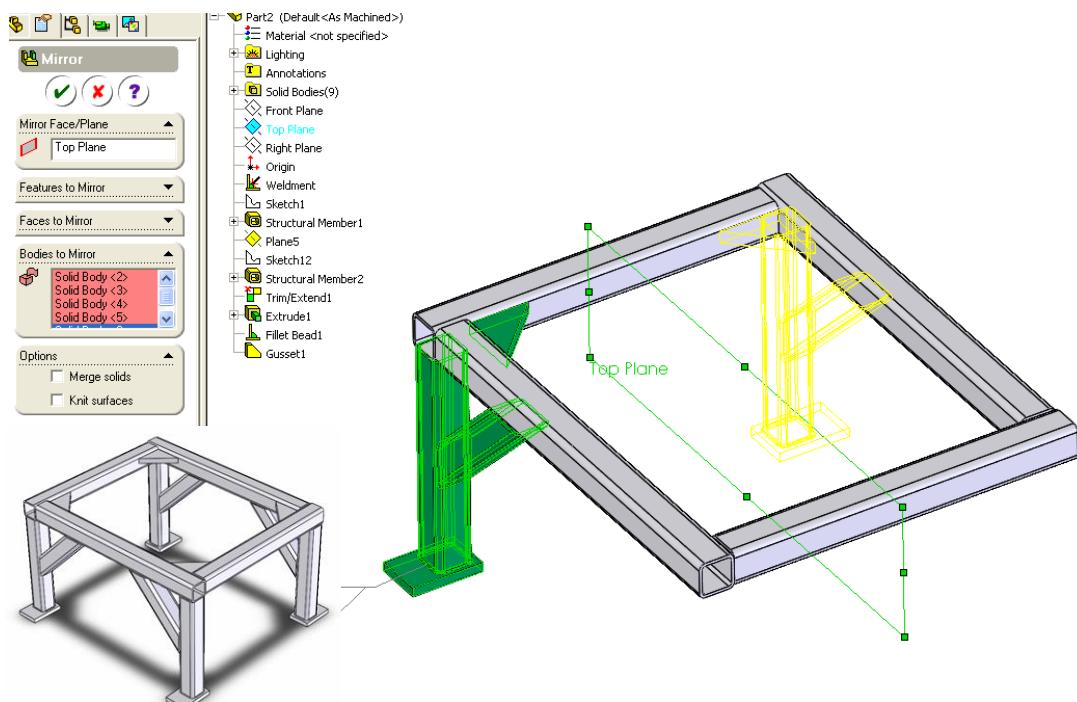
13. Select the Fillet Weld icon and input the following information.



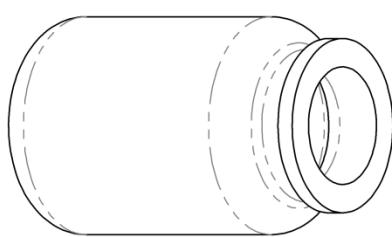
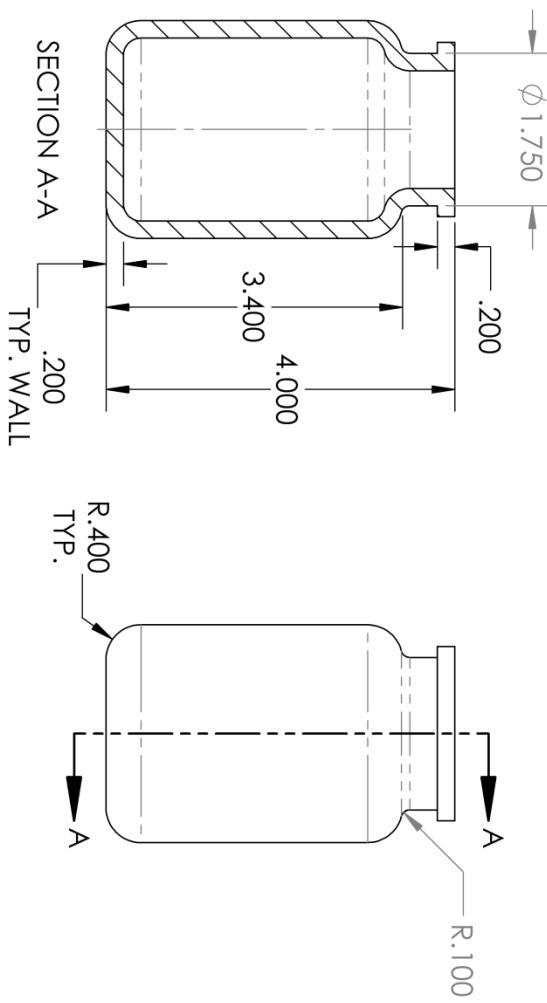
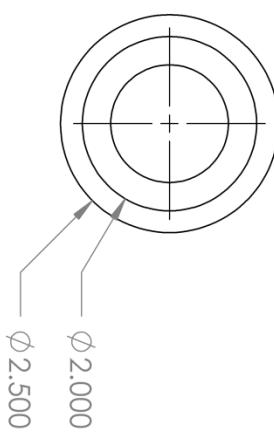
14. Select the gusset icon and input the following information.



15. Use the mirror bodies' tool to mirror the other legs into place. Finished.



GOAL:
FIND THE INTERNAL
VOLUME OF THE BOTTLE.



PROPRIETARY AND CONFIDENTIAL
THE INFORMATION CONTAINED IN THIS
DRAWING IS THE SOLE PROPERTY OF
<INSERT COMPANY NAME HERE>. ANY
REPRODUCTION IN PART OR AS A WHOLE
WITHOUT THE WRITTEN PERMISSION OF
<INSERT COMPANY NAME HERE> IS
PROHIBITED.

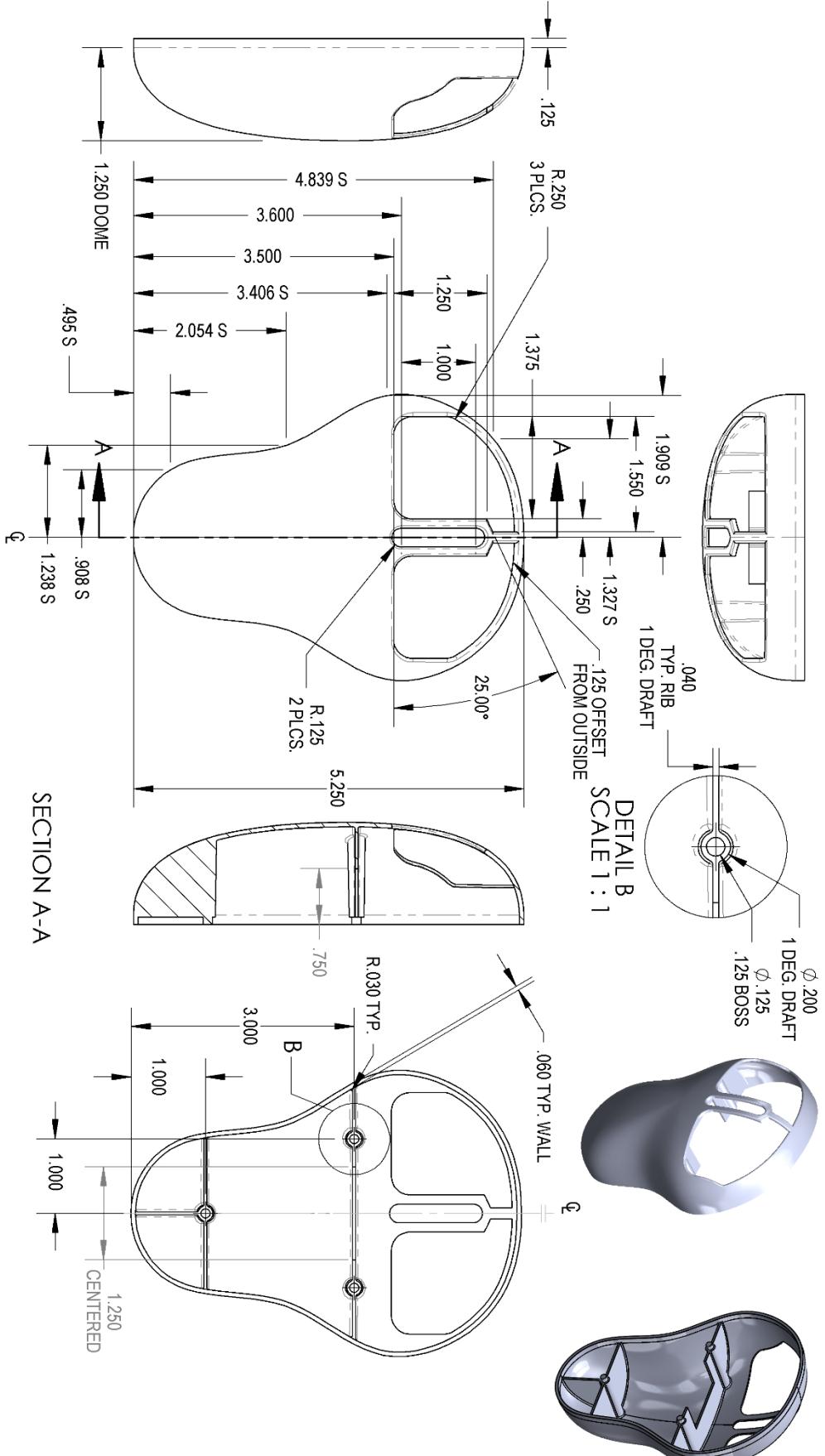
UNLESS OTHERWISE SPECIFIED:			NAME	DATE	BOTTLE VOLUME LAB	
DIMENSIONS ARE IN INCHES	DRAWN	TOLERANCES:				
FRACTIONAL ¹	CHECKED	ANGULAR: MACH ² ±	BEND ±			
TWO PLACE DECIMAL ±	ENG APPR.	THREE PLACE DECIMAL ±				
MFG APPR.						
INTERPRET GEOMETRIC TOLERANCING PER:	Q.A.	COMMENTS:				
MATERIAL						
NEXT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

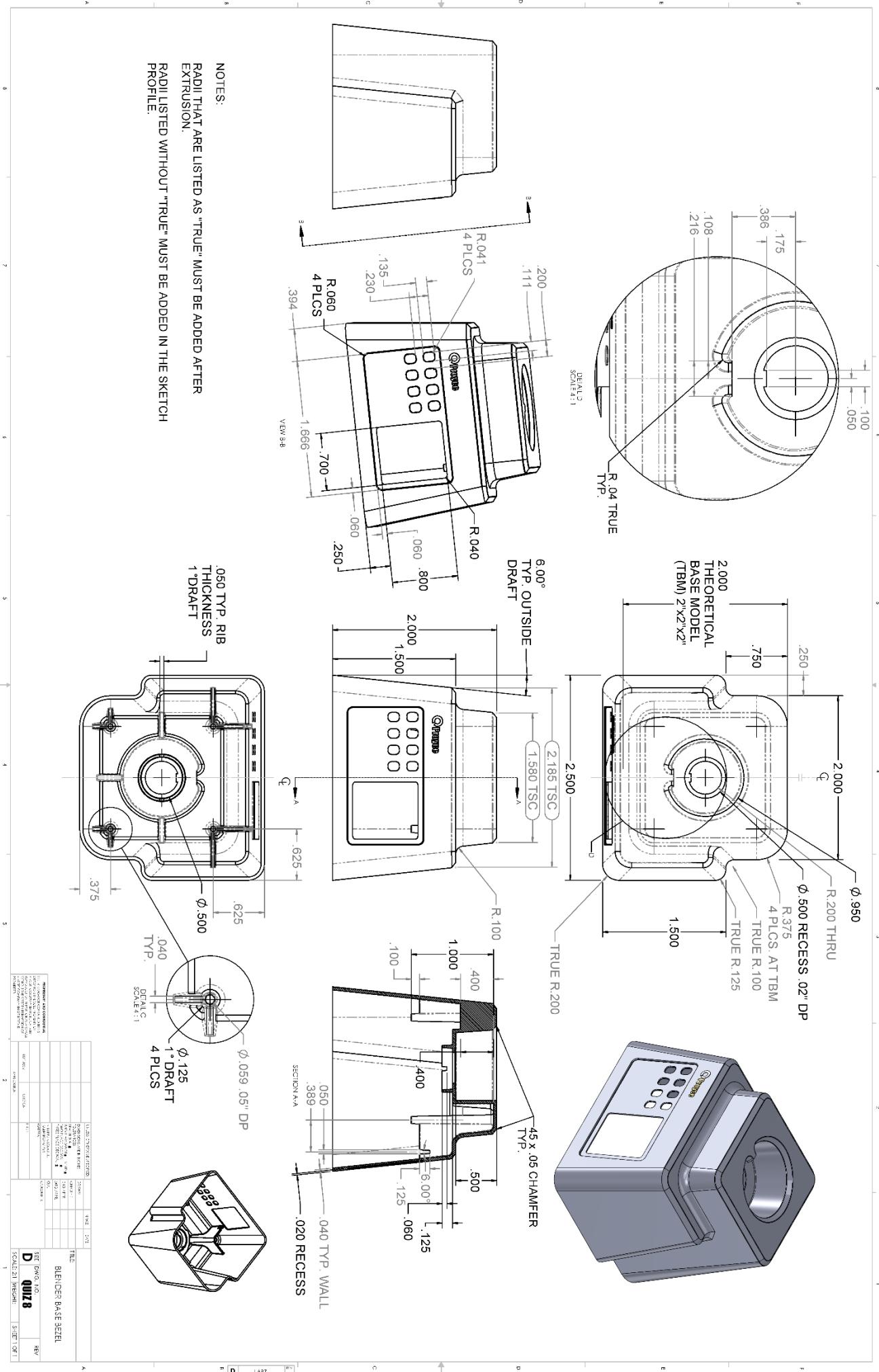
A

SIZE DWG. NO. REV

L26

PROPRIETARY AND CONFIDENTIAL THE INFORMATION CONTAINED IN THIS DRAWING IS THE SOLE PROPERTY OF <INSERT COMPANY NAME HERE>. ANY REPRODUCTION IN PART OR AS A WHOLE WITHOUT THE WRITTEN PERMISSION OF <INSERT COMPANY NAME HERE> IS PROHIBITED.	
NEXT ASSY	USED ON
FINISH	APPLICATION
	DO NOT SCALE DRAWING





SUPPLEMENT

SolidWorks - CAD Administration

Finding adequate computer hardware to run Solidworks can be challenging, this lesson looks at the multiple aspects of selecting hardware as well as modifying settings inside SolidWorks to allow it to run efficiently and trouble free.

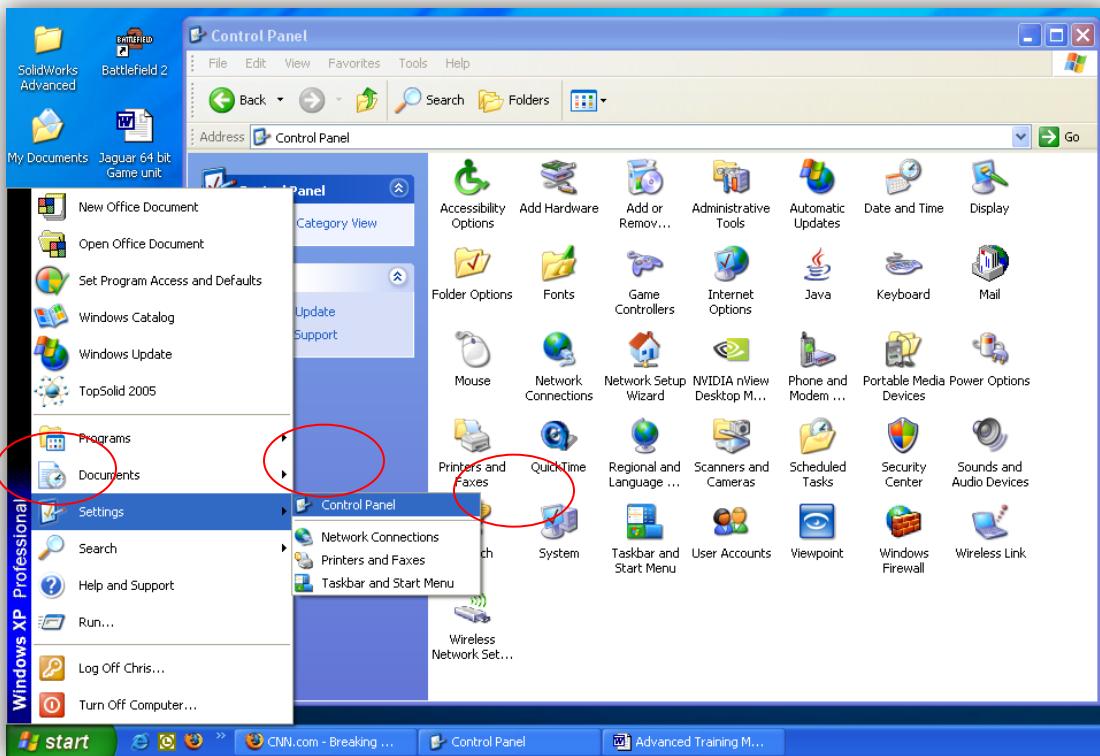
Selecting an Operating System (OS).

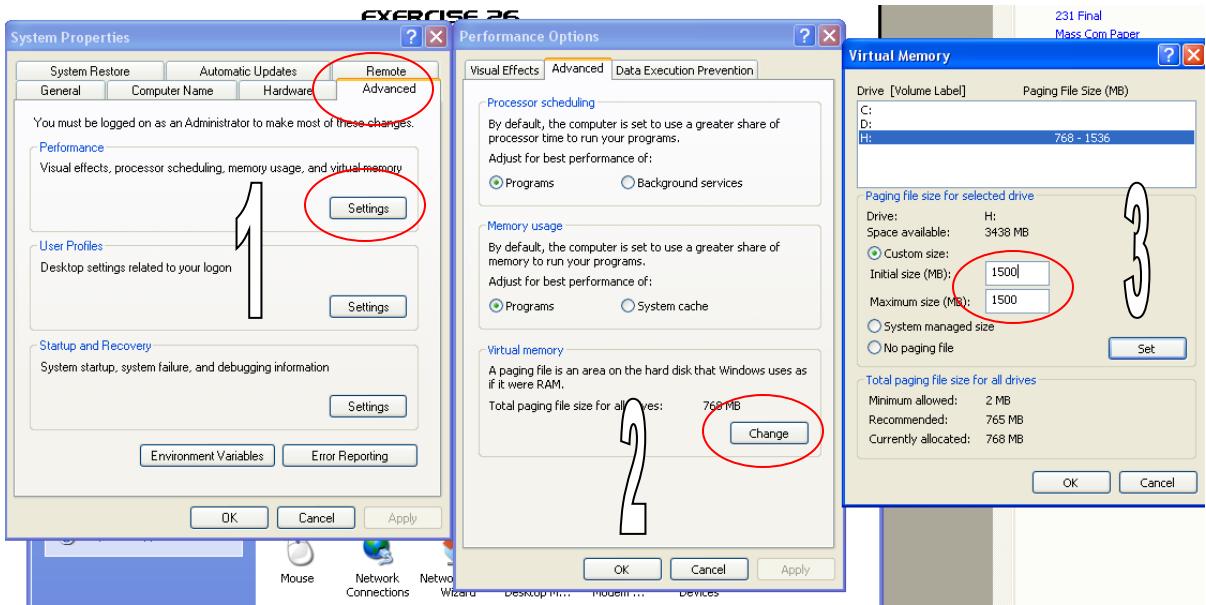
Windows 10

Windows 11

Virtual Memory Settings inside the OS. It may be a good idea to increase or adjust your virtual memory setting. The norm would be x2 – x3 your current amount of ram.

Example 512MB of Ram 1000 – 1500 MB Virtual Ram. And keep the initial size the same as the maximum size. It is said that this prevents write errors.





Processors (CPU)

Intel

Celeron
Pentium
Core i3
Core i5
Core i7
Core i9
Xeon



SLOWER

FASTER

AMD

Athlon
Ryzen 3
Ryzen 5
Ryzen 7
Ryzen 9
Threadripper
EPYC

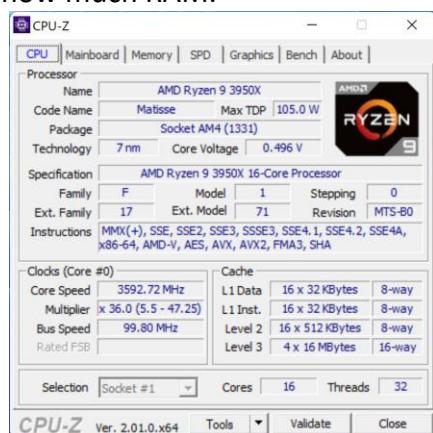
Multiprocessing

CPU manufacturers deliver multiple core processors. This can be seen with the AMD 5995WX Threadripper which has 64 processing cores and 128 Threads.

Which one will run SolidWorks fastest? You can find benchmarks at www.spec.org specifically for SolidWorks or you can look for the generic OpenGL benchmark results that usually use an [OpenGL](#) video game.

The question is: “Can SolidWorks benefit from multiple cores?” Currently one might find an average of 10 – 15% performance increase with general modeling. This is because SolidWorks is not fully written to take advantage of multithreaded processes. However, using the SolidWorks Simulation, CFD, or Photoview rendering solutions one may discover 2x – 64x faster performance versus a single core processor. This is because these SolidWorks applications do take full advantage of multithreaded processing. The biggest benefit one might find is the ability to multitask while working with an FEA analysis. This is a long process and you could actually open up another window of SolidWorks or Outlook and continue working while the analysis is running with little slow down in performance.

To check out what your computer has inside without opening the case download the free version of CPUID – CUP-Z <http://www.cpuid.com/softwares/cpu-z.html> Or ctrl-alt-del and start task manager to see how many threads your CPU has, as well as how much RAM.



GPU | Graphics Cards

Here are a few brands that are in the Professional Category and actually have specific drivers that are written to run SolidWorks at its best.

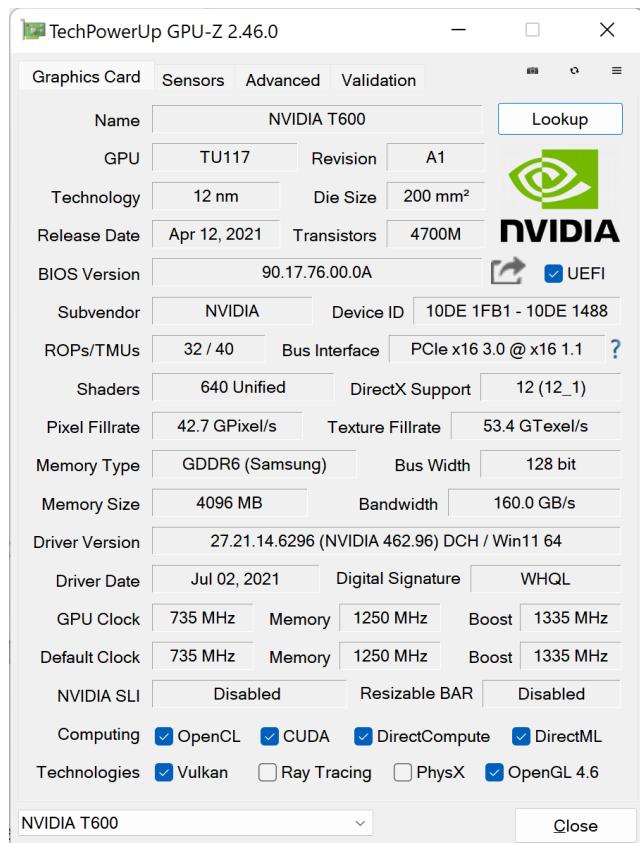
- **NVIDIA Quadro** series (not NVS series)

- **Quadro T600 (Entry Level)**
- **Quadro A2000**
- **Quadro A4000**

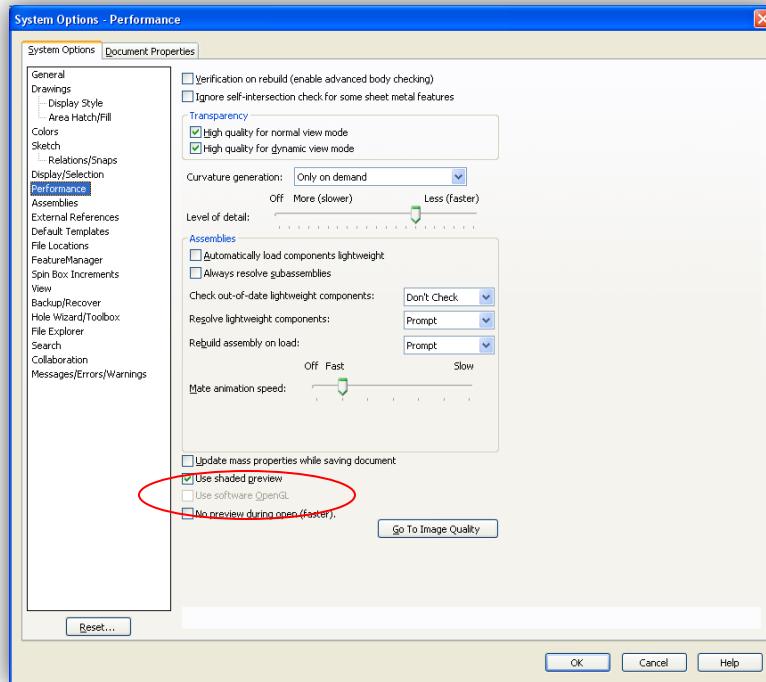
- **AMD Radeon Pro** series

- **W6400 (Entry Level)**
- **W6600**
- **W6800**

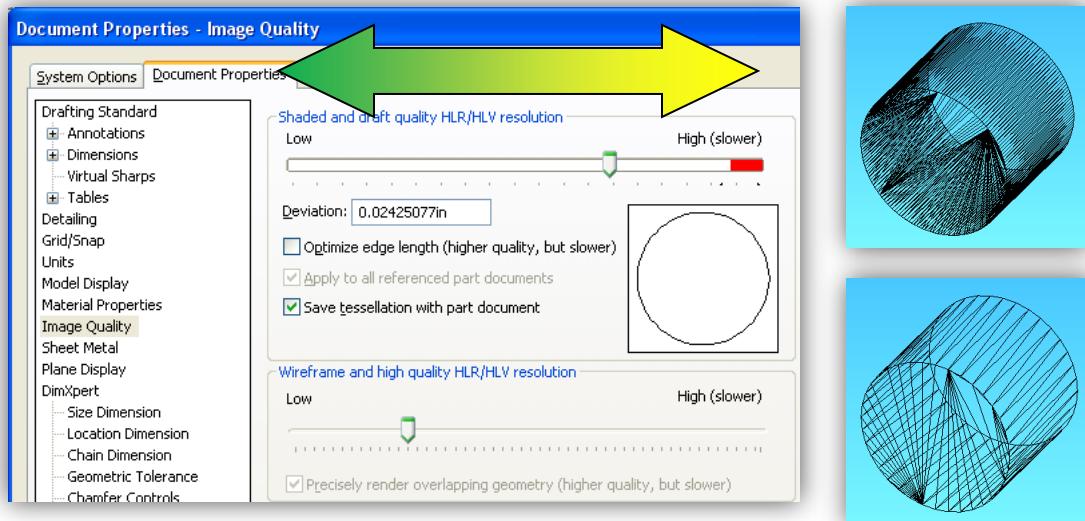
GRAPHICS CARD – GPU BENCHMARK (source: www.passmark.com)



With all SolidWorks documents closed go to Tools/Options/System and turn on software OpenGL.



If your graphics rotation of models is slow try adjusting the image quality. This is located in the document properties.



MEMORY (RAM)

8.0 – 64.0 GB From simple machined parts to complex assemblies. The more RAM the better.

File Translation for the CAD/CAE/CAM Industries

Outline

1. History of Translation in the industry (What is it, why was it needed?)
 - a. 1979 Society of Manufacturing Engineers (SME) meeting.
 - b. General Electric challenges the CAD industry.
2. Primary/Neutral translators
 - a. STEP (Standard for the Exchange of Product Model Data)
 - i. AP (Application Protocols)
 - ii. Configurations
 - iii. Parametric Data
 - b. IGES (Initial Graphics Exchange Specification)
3. Types of data
 - a. Raster
 - b. Vector
 - i. 2D Geometry
 - ii. 3D Geometry
 1. Curves
 2. Surfaces
 3. Solids (B-Rep)
 - c. Configurations
 - d. PD (Parametric Data)
4. Extracting information from translators.
 - a. Variation
 - b. Application
5. What the future holds.

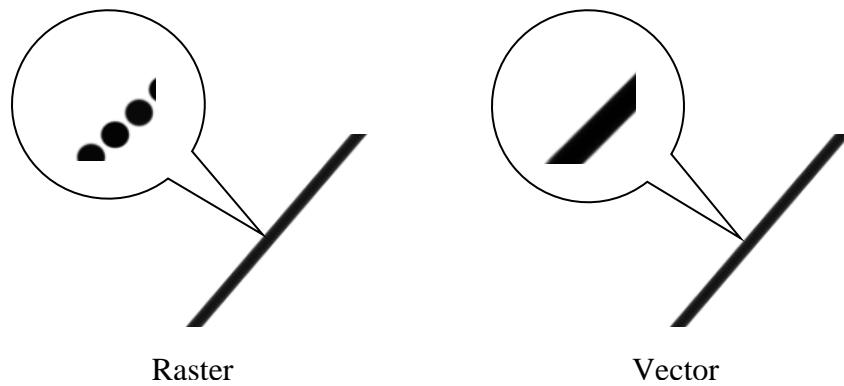
With the growing number of software applications available in the Computer Aided Design, Analysis, and Machining community (CAD/CAE/CAM), one might discover communication between applications to be difficult, because of incompatibilities between each application. Translators have been developed for this reason.

In the beginning CAD/CAE/CAM applications were primarily used in the Scientific, and Aerospace communities. This CAD technology was developed in 1969 and within a decade had reached a point where various companies had developed multiple CAD applications. Users of these applications quickly discovered transferring data between systems was very difficult. Specialized programs had been written to address the task but no standards had been set. Frustration finally peaked at a Society of Manufacturing Engineers (SME) meeting in the fall of 1979. An attendee from General Electric (GE) challenged a panel of CAD vendors to create a universal translator to be used between the various CAD applications. Soon after the meeting, the panel had determined that such a task was possible, and set in motion the collaborative foundation to what would later be recognized as the Initial Graphics Exchange Specification (IGES). The IGES format is the most common translator used today, and it continues to evolve and improve with help of user feedback. For more information on IGES visit the web page. <http://www.nist.gov/iges/>

Since IGES there have been several other translators. One in particular named STEP (Standard for the Exchange of Product Model Data) was developed specifically by the Aerospace industry. Boeing and McDonald Douglas had been attempting to share data between their CAD systems. They discovered IGES was not adequate, and not well

regulated. Variation was becoming an issue between CAD vendors due to poor regulation of the translator. They decided quality had to be better enforced to eliminate and reduce variation. They employed the International Organization for Standardization (ISO) to develop, regulate, and manage the initiative. A team consisting of Boeing, McDonald Douglas (Unigraphics), Catia, and Computer Vision was formed to assist with laying the foundation. They assembled scalable algorithms that could be transplanted into the base code known as Application Protocols (AP). These AP's are used to identify the specific tasks available in the translator. For example AP 203 is the most common in that it has the ability to translate Solids data between systems. Another AP is the 214, which was later added to enhance the ability to transfer configuration information for manufacturing and inspection purposes. General Motors (GM) frequently uses the AP 214.

Visual CAD data comes in many forms. A basic break down of this reveals two types. Raster and Vector. Raster data consists of multiple microscopic dots, which make up an image. Much like a photograph this method proves inadequate for CAD because of inaccuracy and it is limited to only two dimensional (2D) representation and manipulation capabilities.



Vector geometry on the other hand proves robust in its ability to provide both 2D and three dimensional (3D) attributes, making it the criterion form for translating data. Vector geometry consists of continuous lines, arcs, splines, surfaces, or solid geometry representations. Dimensional attributes are applied to identify position and proportions. Enabling them to be easily modified, translated or scaled by simply changing values.

The ultimate goal of the translator is to eventually enable all data created on one CAD system to be flawlessly translated into another. This capability is currently being worked on by ISO to be introduced into the STEP translation algorithm sometime in the future. No date or deadline has been set until the CAD industry can come up with a reasonable standard to do so. Several CAD vendors already have methods of either reading in native CAD files from other systems and rebuilding from scratch the Parametric Data (PD) or even directly translating the information.

The following is a list of common translators and native CAD formats capable of being translated between SolidWorks. The list details if the file can be imported or exported, and what specific data can be obtained by it.

TRANSLATOR	EXTENSION	IMPORT	EXPORT	VECTOR	RASTER	2D	3D	PD*
PARASOLID	X_T, X_B	X	X	X			X	
ACIS	SAT	X	X	X			X	
DWG	DWG	X	X	X		X		
DXF	DWG	X	X	X		X	X	
IGES	IGES, IGS	X	X	X		X	X	
STEP	STEP, STP	X	X	X			X	
VDAFS	VDA	X	X	X			X	
CGR	WRL	X	X	X		X		
HCG	HCG		X	X		X		
CADKEY	PRT	X		X			X	
SOLIDEDGE	PAR	X		X			X	
UGII	PRT	X		X			X	
MDT	DWG	X		X		X	X	
INVENTOR	IPT	X		X			X	
PRO/ENGINEER	PRT,XPR,ASM,XAS	X	X	X		X	X	
HOOPS	HSF		X	X			X	
VRML	WRL	X	X	X			X	
VIEWPOINT	MTS		X	X			X	
REALITY WAVE	ZGL		X	X			X	
EDRAWING	EPRT,EASM,EDRW	X		X		X	X	
JPEG	JPEG,JPG		X			X	X	
TIFF	TIFF		X			X	X	
STL	STL		X	X				
ADD-INS	DLL	X					X	

Parasolid – Is the core-modeling kernel utilized inside SolidWorks. Two types of parasolid translation are supported – standard (.x_t) and binary (.x_b); both translate data flawlessly between native parasolid based systems. However, binary files are typically smaller in size.

ACIS – Developed by Spatial Technologies – a DASSAULT SYSTEMES company. Portions of this kernel are seamlessly integrated within SolidWorks. For example Spatial's deformable surface husk technology is recognized as SolidWorks “Shape” feature. ACIS is the core-modeling kernel for many 3D CAD applications and is a good choice for translating clean and efficient models between systems using this kernel. A complete list of current ACIS kernel partners can be found at the internet address listed below.

DWG/DXF – Drawing Exchange File. Support for versions AutoCAD R12 – R2011.

IGES – Initial Graphics Exchange Specification. A common translator type, works with most systems. However, incomplete translations are frequent due to lack of strict regulations on software developer's data creation.

STEP 203/214 – Standard for the Exchange of Product model data. A well-regulated format – software developers must follow strict regulations to offer STEP as a certified integrator. AP203 (Application Protocol) supports 3D geometry only. AP214 has additional support for configurations, commonly used by General Motors.

VDAFS – Verband der Automobilindustrie (German Automotive Specification)

CGR – Catia Graphics format

HCG – Highly Compressed Graphics format

CADKEY – Imports native Cadkey part file solids data only.

SolidEdge – Imports native SolidEdge part file solids data only.

UGII – Imports Unigraphics (Siemens NX) native part file solids data only.

MDT – Mechanical Desk Top, (MDT 6.0 installed required to operate) has support for parametric entities. Will not import 2D drawing data.

Inventor - Imports native Inventor part file solids data only.

Pro/Engineer-Creo – Imports Part and Assembly files. Capable of exporting v.20 part files. Has support for parametric entities from versions 16 – Creo 1.0.

In the past three years, data translation has made massive leaps in capabilities, which help end users communicate more efficiently with one another. Unfortunately there are still some CAD vendors holding back the progress made to have flawless data communication between all users. These vendors believe empowering any and all individuals to access data generated on proprietary CAD applications can be detrimental to the bottom line of their company. They actually have been known to encrypt their native files to prevent others who have not purchased the native CAD application from accessing the data, forcing them (usually tier 2 and 3 vendors) to purchase the application in order to better serve the customer.

In summary, there have been vast improvements in translation between systems over the past decade. One can assume the final goal will eventually be attained, which will break down the communications barriers, enabling virtually all CAD systems to communicate flawlessly between one another.