CREO PARAMETRIC 8.0
ADVANCED

BY CHRISTOPHER F. SIKORA

COMPUTER AIDED DESIGN

2022

CAD 111 COURSE SYLLABUS

PRO/ENGINEER- CREO PARAMETRIC 8.0 ADVANCED

COURSE DESCRIPTION:

PRO/E- CREO ADVANCED
3 CREDIT HOURS
EXPLORATION OF THE ADVANCED THEORY AND APPLICATION OF SOLID MODELING TECHNIQUES FOR PRODUCT DESIGN AND MANUFACTURING.
PREREQUISITE: CAD 105 INTRO TO PRO/E-CREO OR CONSENT OF INSTRUCTOR.
(2 LECTURE HOURS, 2 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

CREO PARAMETRIC 7.0 ADVANCED (FREE/PDF. PROVIDED)
INSTRUCTIONAL VIDEOS OF LECTURE PROVIDED AT WWW.VERTANUX1.COM
CREO PARAMETRIC 7.0 LINK: SOFTWARE
NOTE: YOU MUST USE YOUR STUDENT .EDU EMAIL ADDRESS TO OBTAIN A STUDENT LICENSE.

EVALUATION SCALE:

A	90% TO 100%
B	80% TO 89%
C	70% TO 79%
D	60% TO 69%
F	BELOW 60%

POINTS:

EXERCISE	300 PTS
MID-TERM	300 PTS
FINAL	300 PTS
LABS	100 PTS
TOTAL	1000 PTS

GENERAL COURSE OUTLINE

DATE	WEEK	TOPIC
10/19	1.	E12 IMPORT AND EXPORT - 2D AND 3D TRANSLATION
	2.	E13 ADVANCED LOFTING BOTTLE
	3.	E14 BASIC BLOW MOLD CAVITY
	4.	E15 INTRO TO CAD/CAM - CREO TO MASTERCAM
	5.	E16 DESIGN/FAMILY TABLES
	5.	E17 SHEET METAL
	6.	E18 ADVANCED FAMILY TABLES
	7.	REVIEW FOR MID TERM
11/4	9.	MID TERM EXAM
	10.	E19 RELATIONS (AKA: EQUATIONS)
	11.	E20 CYLINDRICAL, CONICAL, LOFTED SHEET METAL
	12.	E21 SURFACING A SPOON
	13.	E22 3D IGES IMPORT
	14.	E23 DESIGN A COFFEE LID PROJECT (NO VIDEO)
	15.	E0 HOW TO MAKE A PORTFOLIO, FINAL EXAM REVIEW
12/15	16.	FINAL EXAM AND PORTFOLIO - ALL WORK IS DUE

REQUIRED HARDWARE

WINDOWS 10 64-BIT PC OR LAPTOP
MINIMUM 4 GB RAM (16 GB S BETTER)

SOFTWARE (REQUIRED)

CREO PARAMETRIC 8.0
MASTERCAM 2021 HLE

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.

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 TO BE COMPLETED DURRING LAB HOURS OR AT HOME. THEY ARE THERE TO HELP TEST AND SHARPEN YOUR SKILLS, AND ARE A GREAT RESOURCE FOR ADDITIONAL TRAINING.

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

CAD III TOTALS

EXERCISES (VIDEO LINKS AVAILABLE)

E12 - 20PTS. (*DWG FILE/WHEEL*)

E13 - 20PTS. (*BOTTLE*)

E14 - 20PTS. (*BOTTLE MOLD*)

E15 - 20PTS.-(*CAD/CAM*)

E16 - 20PTS. (*FAMILY TABLES*)

E17 - 20PTS. (*SHEET METAL II*)

E18 - 20PTS. (*FAMILY TABLES II*)

E19 - 20PTS. (*EQUATIONS/RELATIONS BOX*)

E20 --20PTS. (*SHEET METAL CYLINDER,CONE, LOFT*)

E21 - 20PTS. (*SPOON*)

E22 - 20PTS. (*IGES REPAIR*)

E23 - 60PTS. (*COFFEE LID DESIGN PROJECT*)

LABS (NO VIDEOS)

L18 - 25PTS. (*OFFICE PHONE*) P.67

L19 - 15PTS. (*CELL PHONE FRONT BEZEL*) P. 76

L20 - 15PTS. (*SMOKE DETECTOR FRONT BEZEL*) P. 79

L21 - 15PTS. (*SMOKE DETECTOR REAR BEZEL*) P. 80

L22 - 15PTS. (*CPU*) P. 99

L24 - 15PTS. (*NAGA PORT*) P. 116

EXAMS (VIDEO LINKS AVAILBLE)

300 MIDTERM

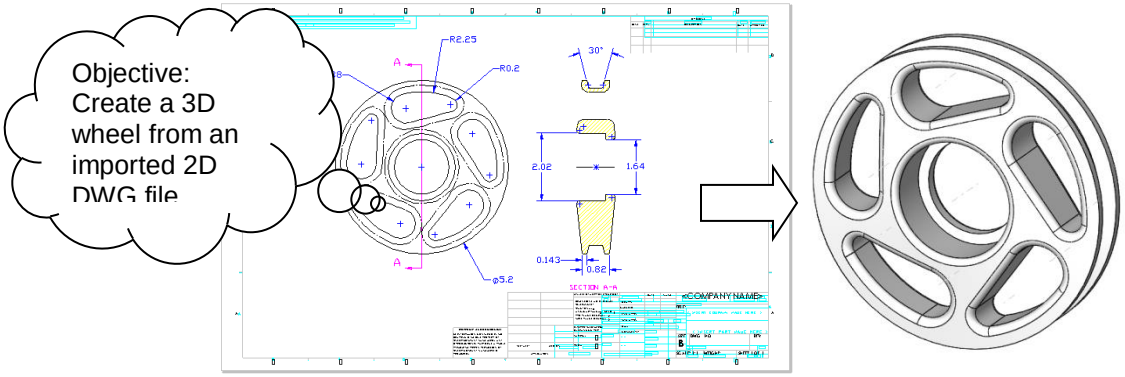
300 FINAL

1000 TOTAL POINTS POSSIBLE

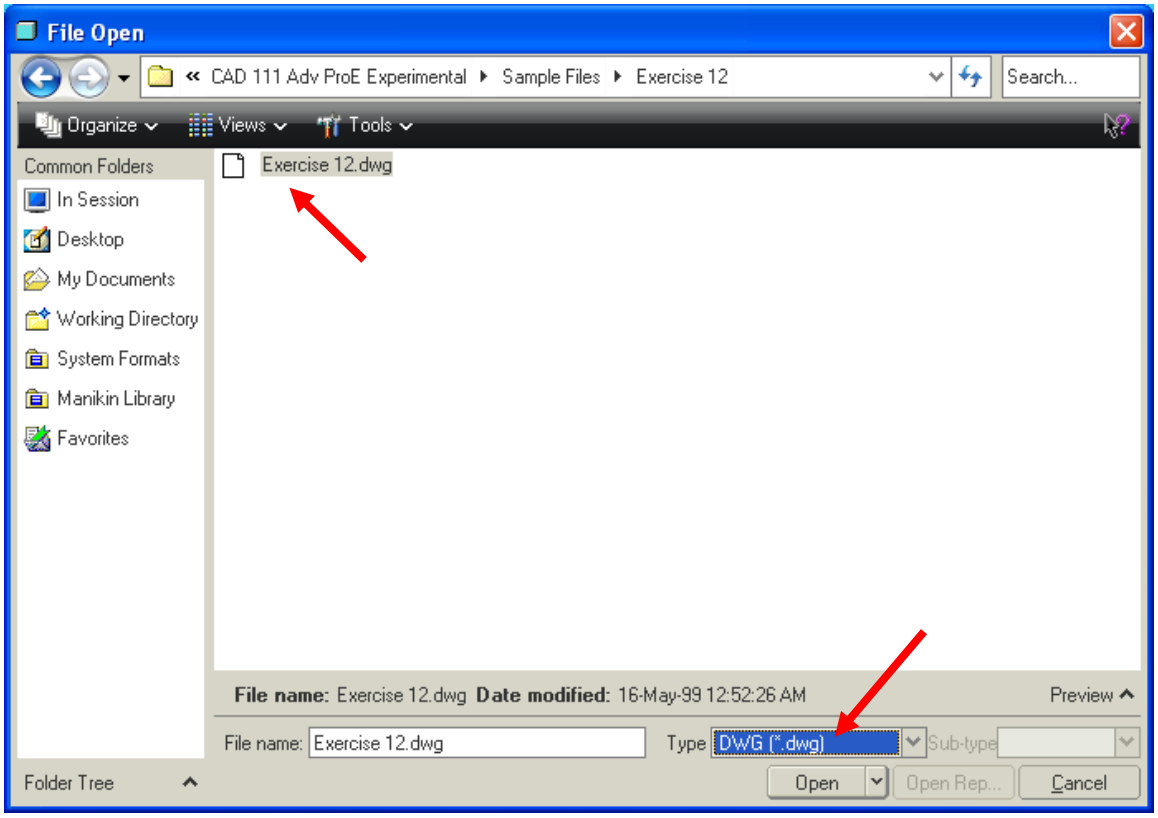
EXERCISE 12

Importing 2D DXF/DWG files

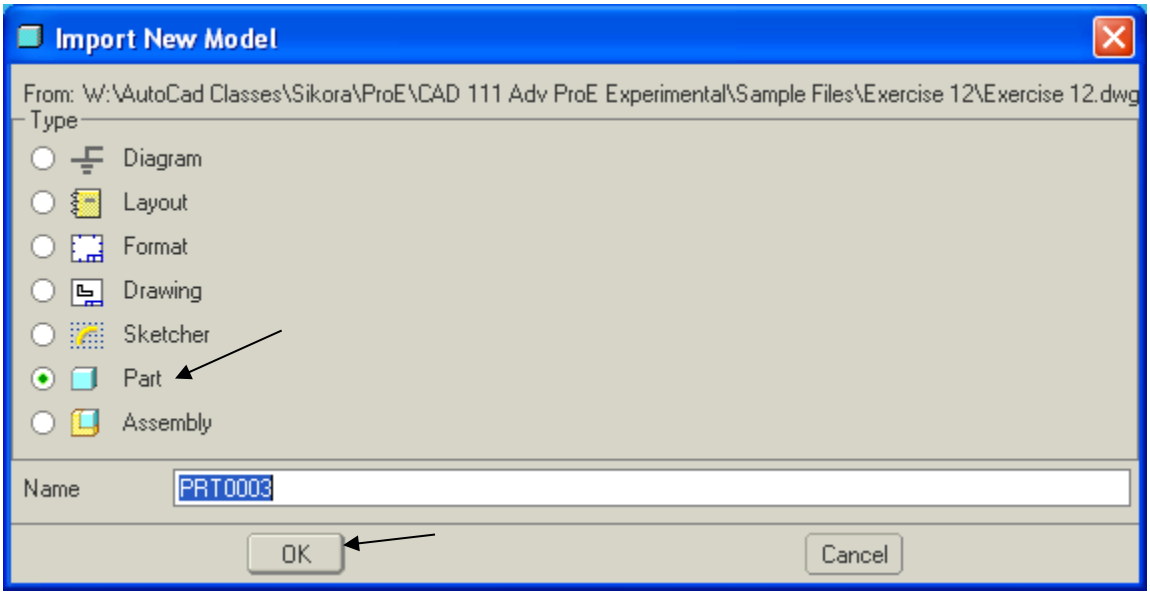
DWG and DXF files *can be very useful if imported into SolidWorks.*



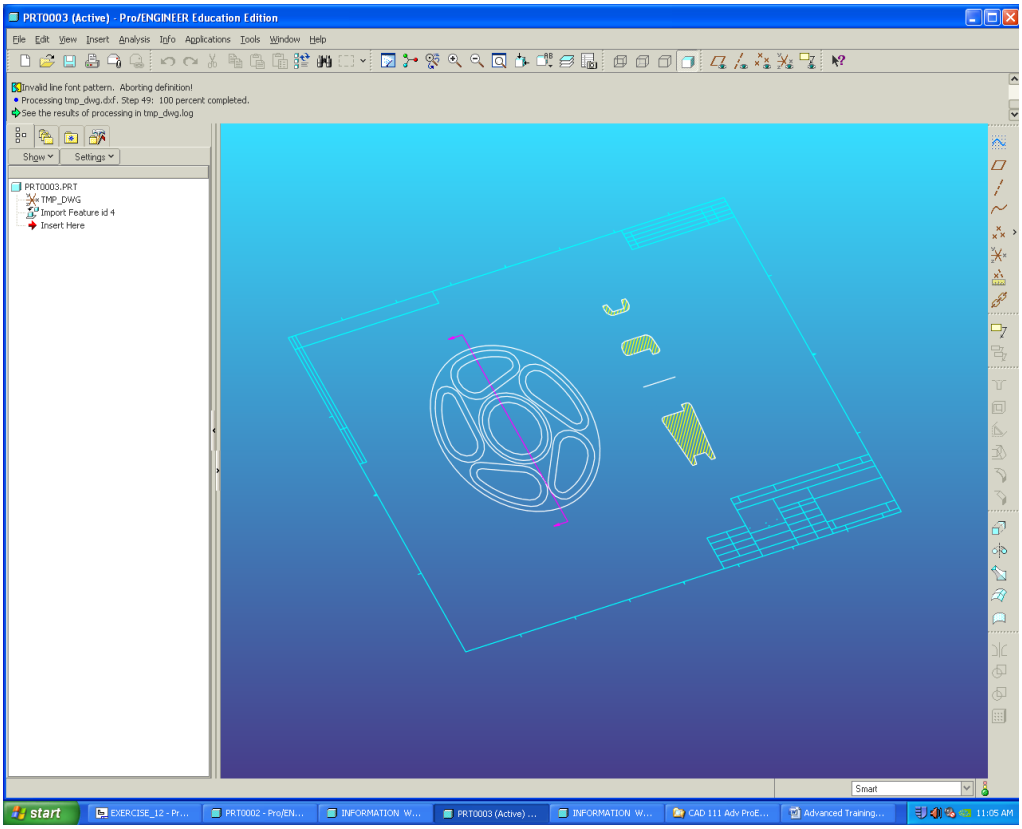
1. Go to file/open and select DWG from the options. Find the Exercise 12.dwg.



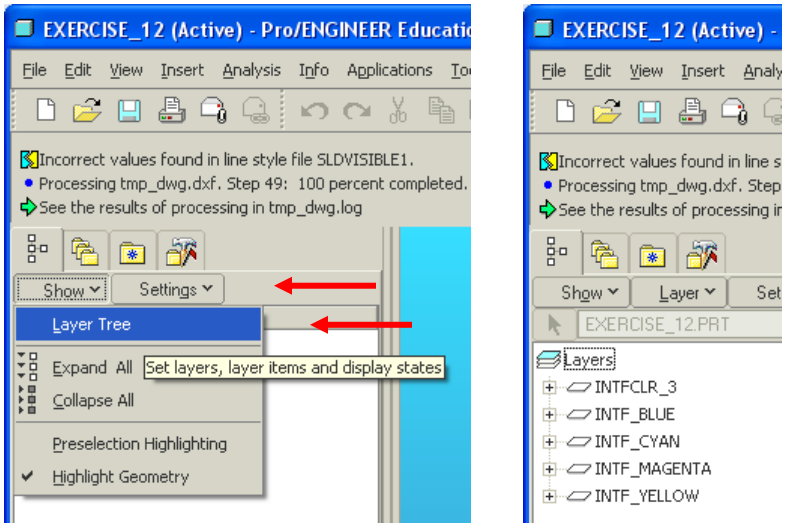
2. Import to a new Part.



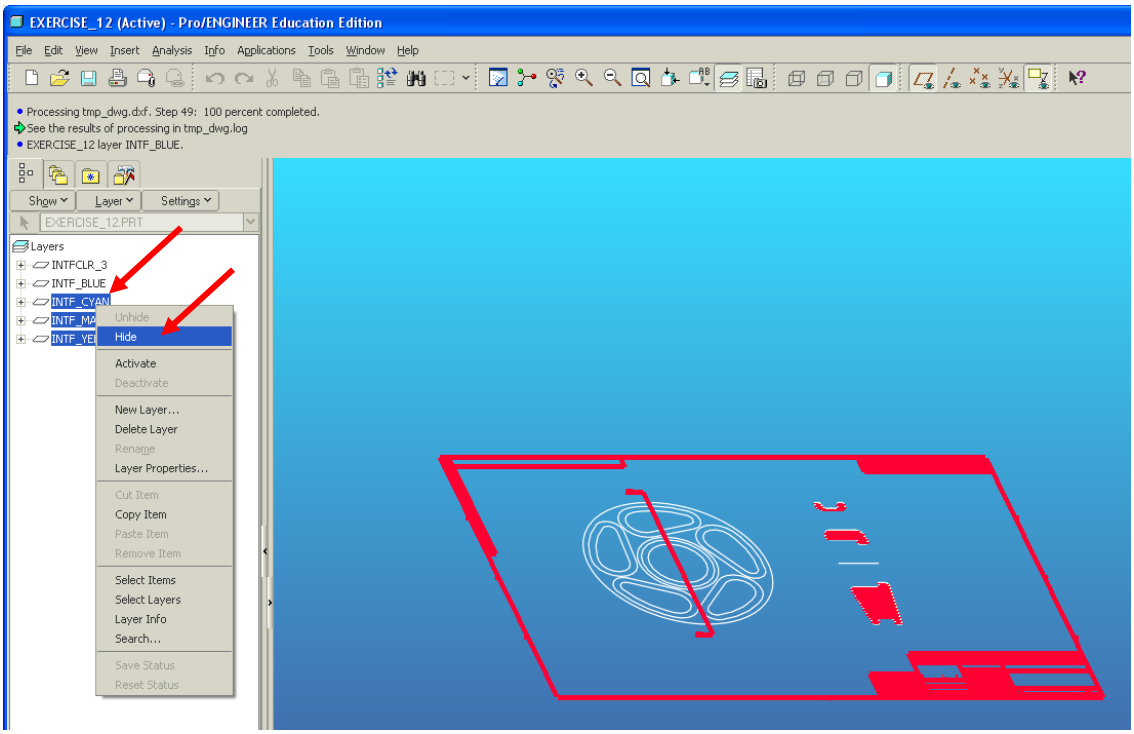
3. The next screen should look like this…



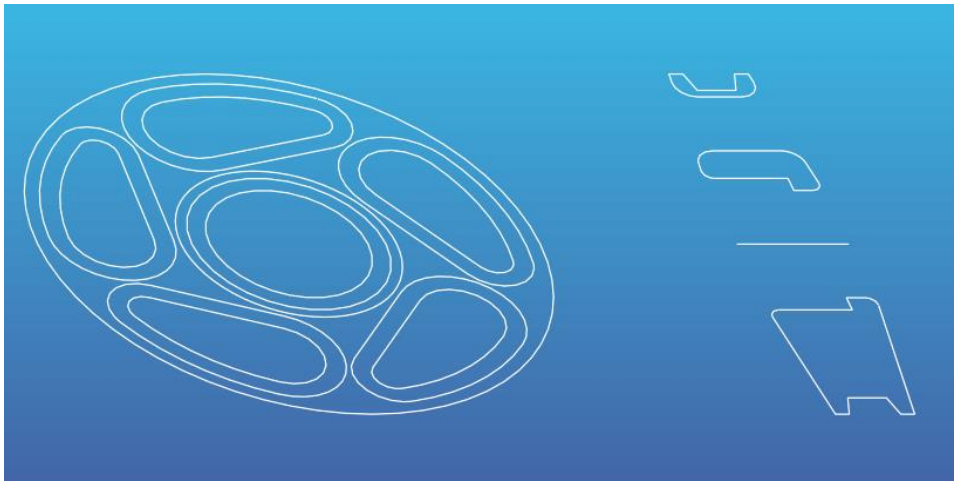
4. Go to “Show/Layer Tree”…



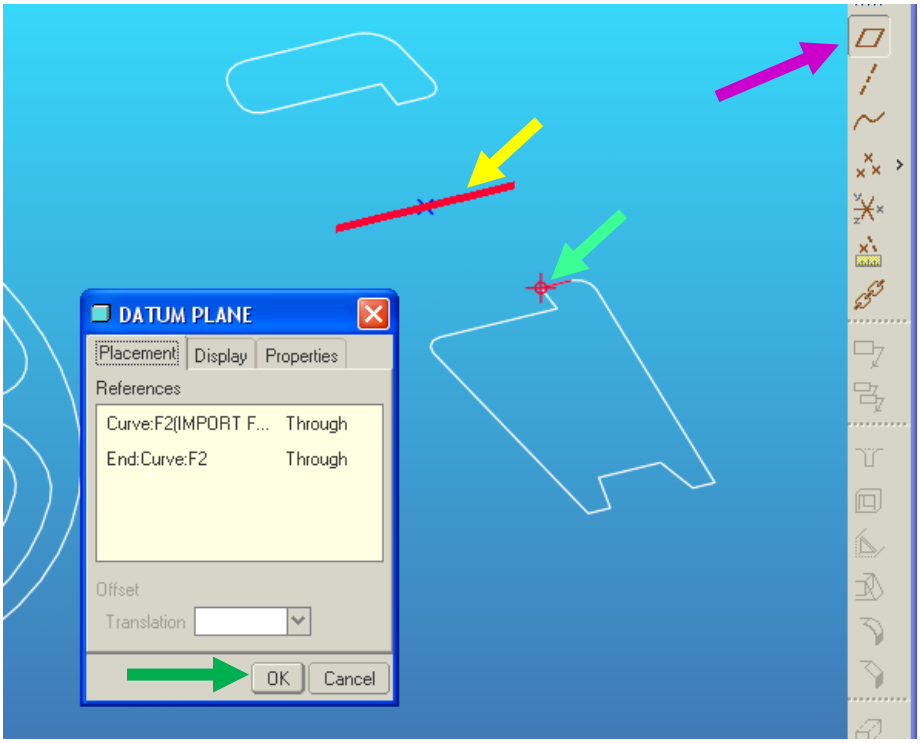
5. CTRL select the CYAN, MAGENTA, and YELOW layers, RMB click and select “Hide”.



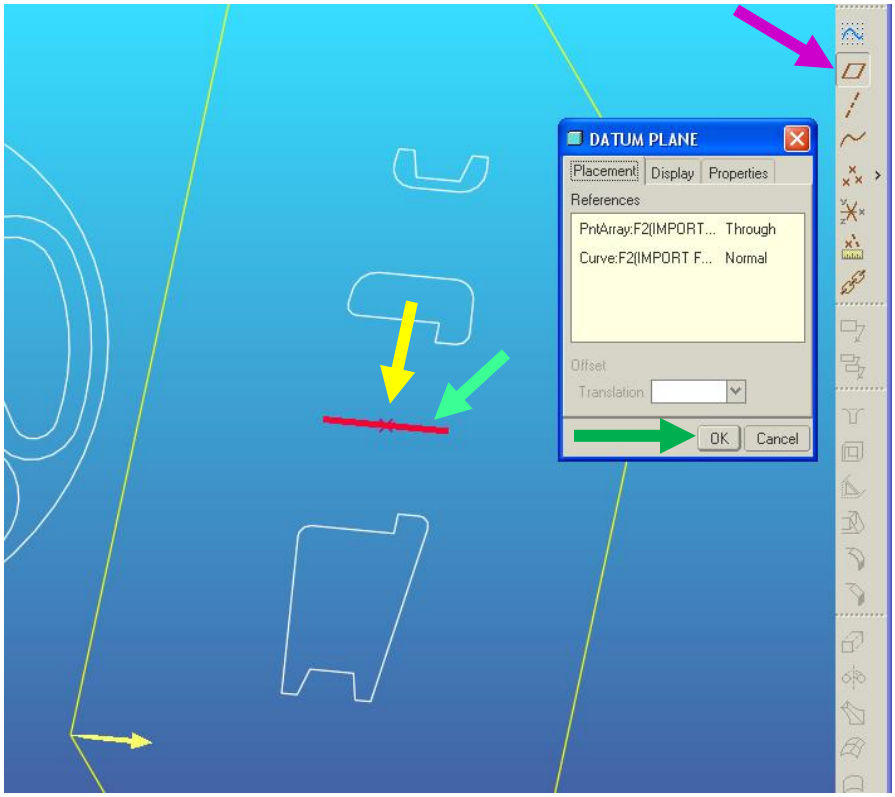
6. Once the other layers are disabled your drawing should look like this….



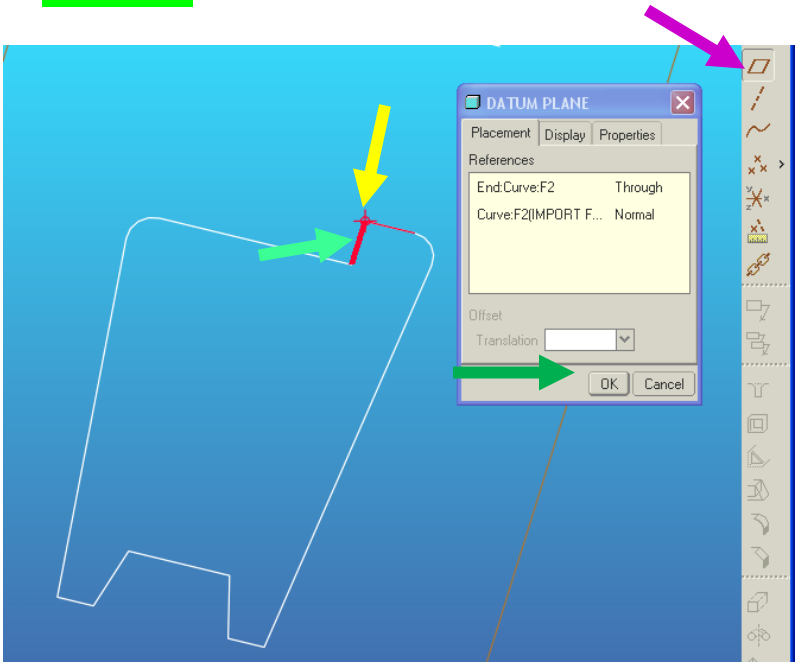
7. **Creating Reference and Sketch Planes:** CTRL Select the centerline and a vertex endpoint on the section view and then select the "DATUM PLANE creation icon. Hit "OK".



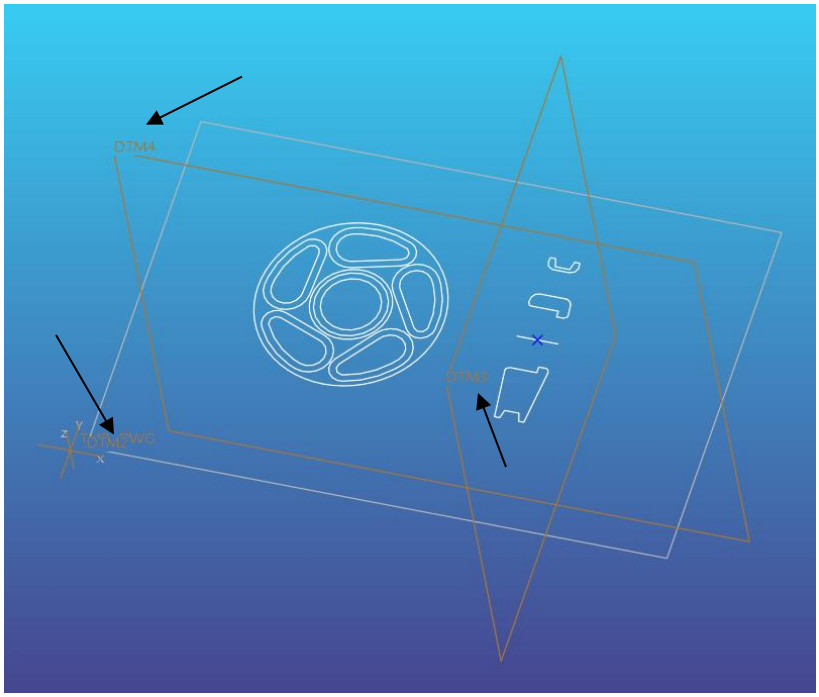
8. CTRL select the center point in the center of the centerline, and then CTRL select the centerline. Hit the "Plane" icon and then "OK".



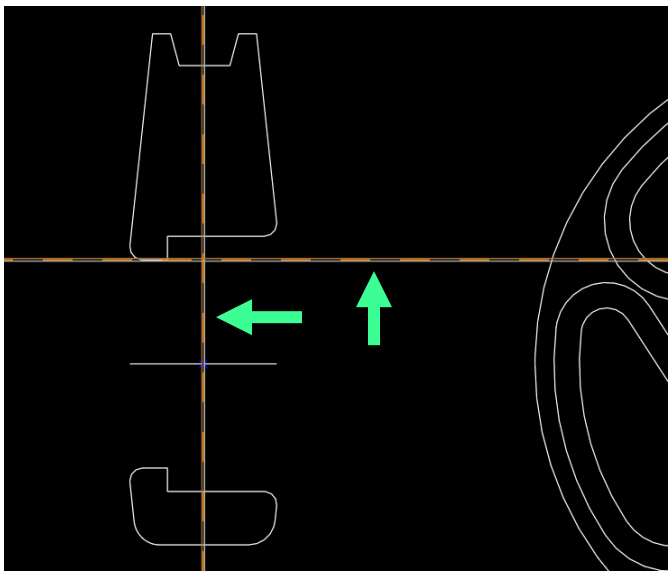
9. CTRL select the end point/vertex of the intersecting lines, and then select the vertical line and hit the "Plane" icon and then "OK".



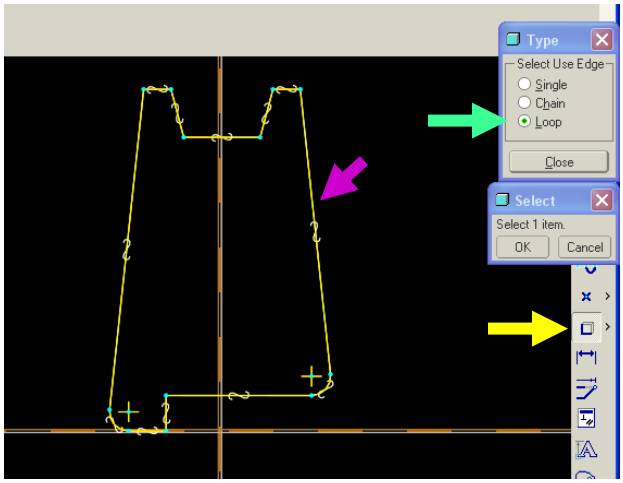
10. You should now have a three intersecting perpendicular planes to use for sketching and references.



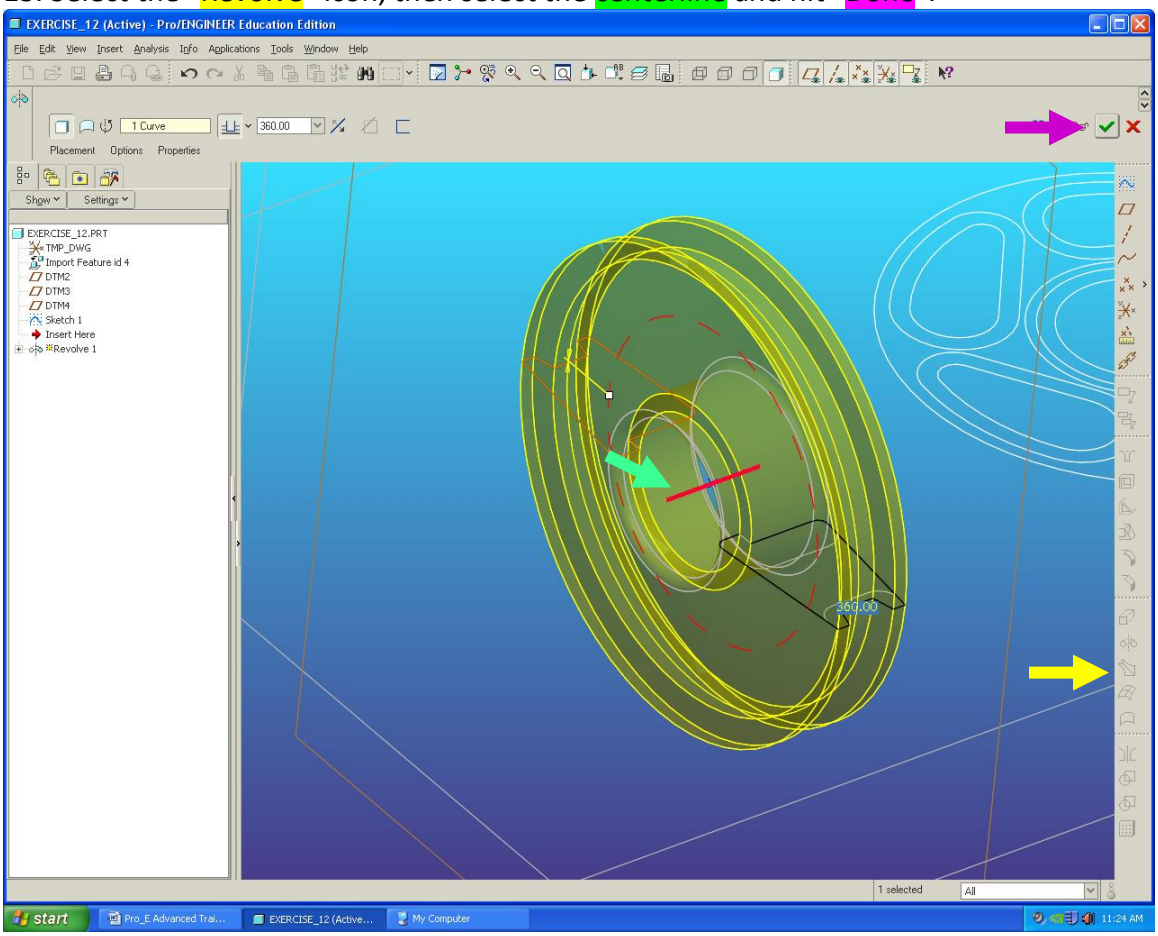
11. Start a sketch on the plane that is aligned parallel to your drawing. Select the vertical and horizontal datum planes as references.



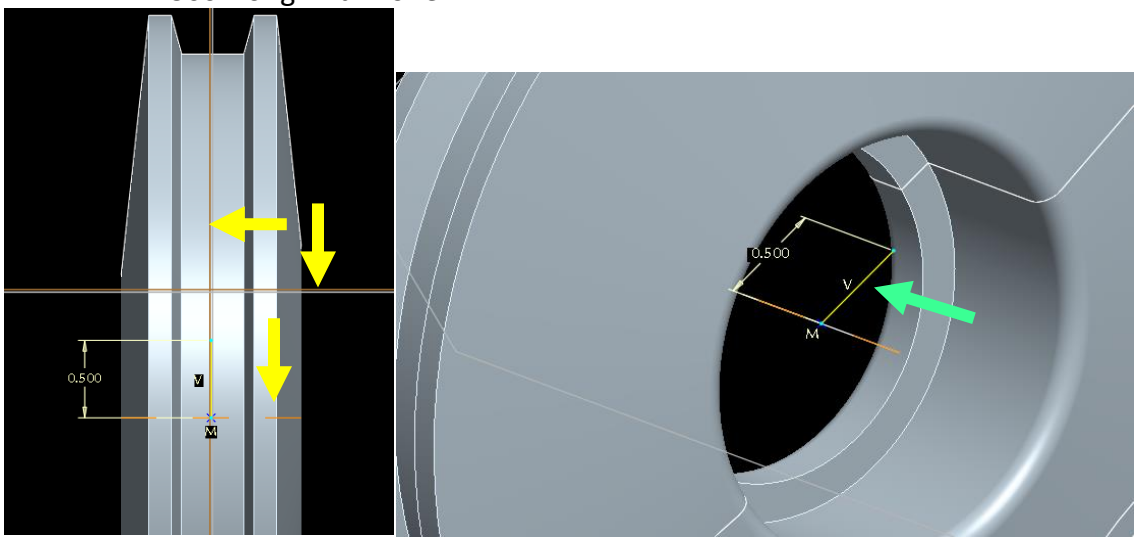
12. Select the "USE" icon, and chose "Loop", then select any edge of the profile.



13. Select the "Revolve" icon, then select the centerline and hit "Done".

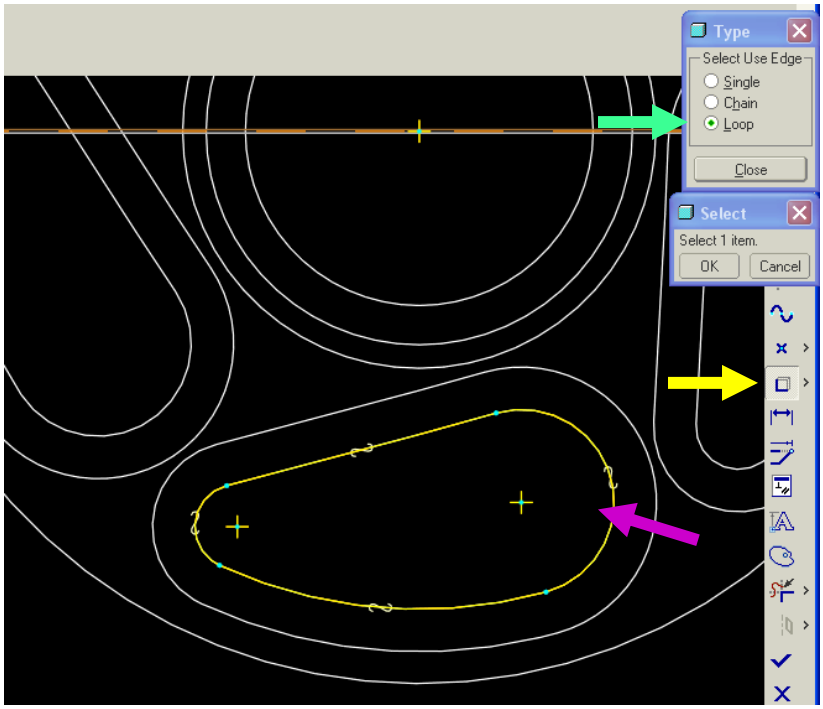


13. (SKIP THIS IF YOU ALREADY CREATED AN OFFSET DATUM) Start a sketch on the parallel plane to the drawing. Then select the centerline and vertical and horizontal planes as references. Draw a vertical line approximately .500" long. Hit "Done".

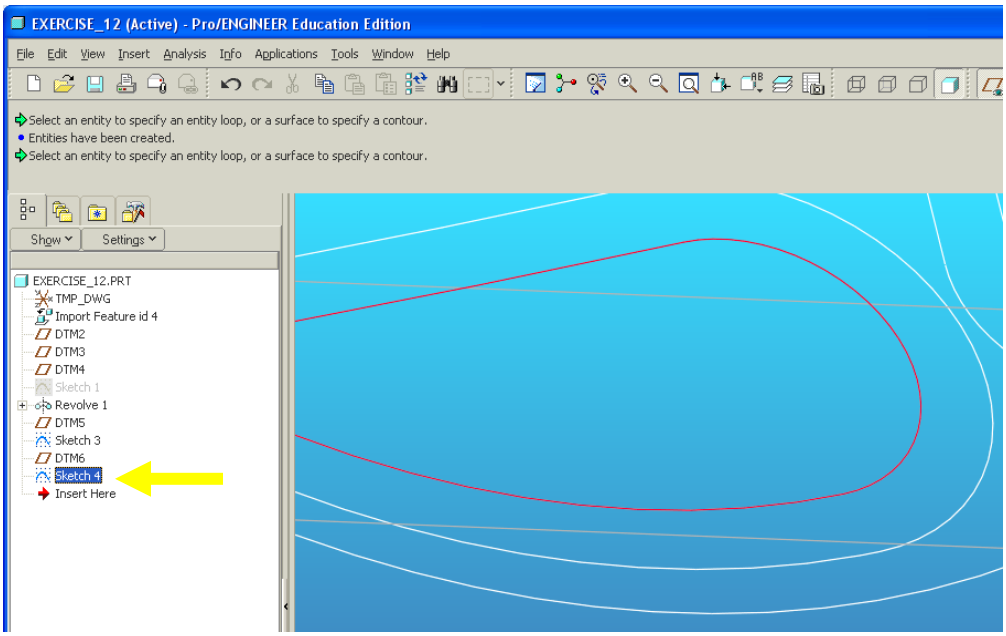


14. Select the endpoint and length of the ".500" line, and then go to the Plane icon. Hit "Done" *Note: turn off the view-points icon to help make selections easier*.

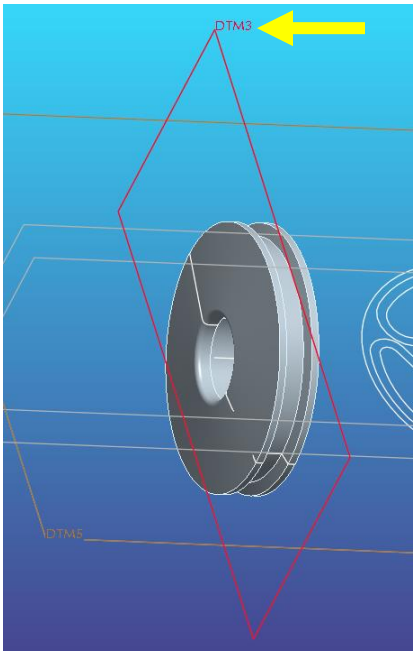
15. Start a new sketch on the same parallel plane to the drawing. Select the "USE" icon, and chose "Loop", then select any edge of the profile. Hit "Done".



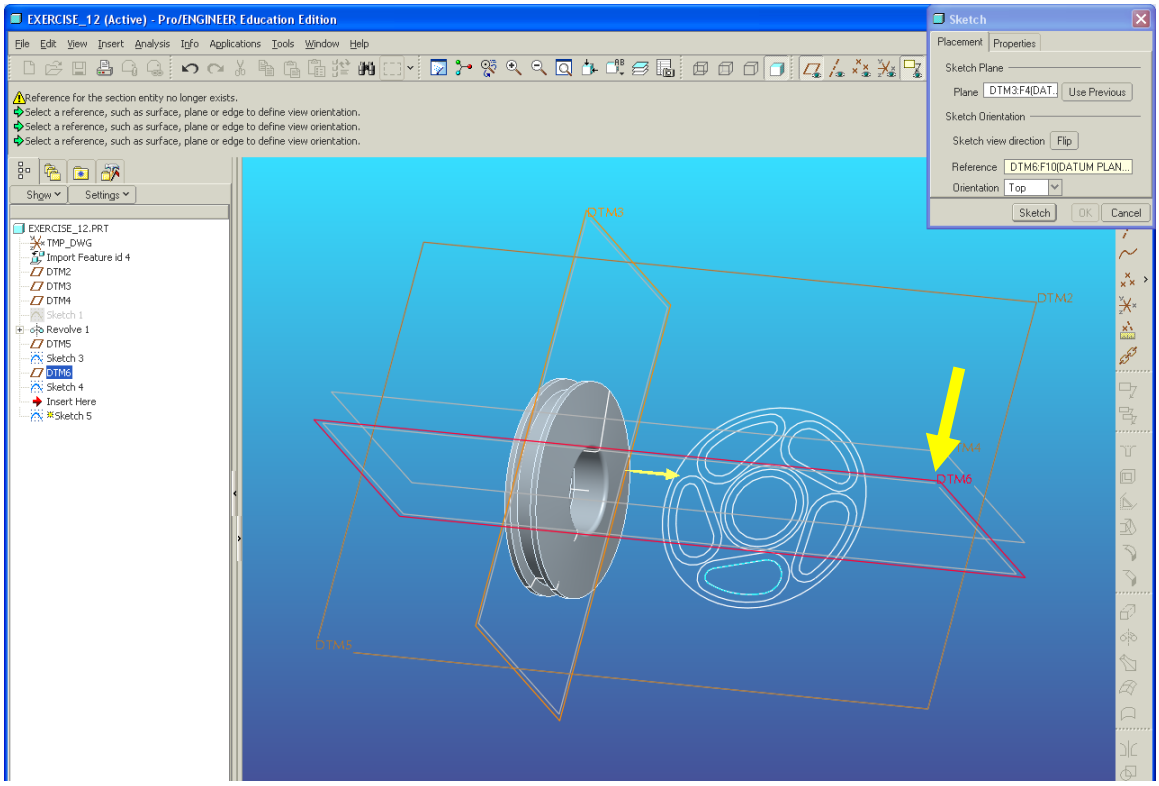
16. CTRL select the sketch from the feature tree. Then select “Edit/Copy” (CTRL-C).



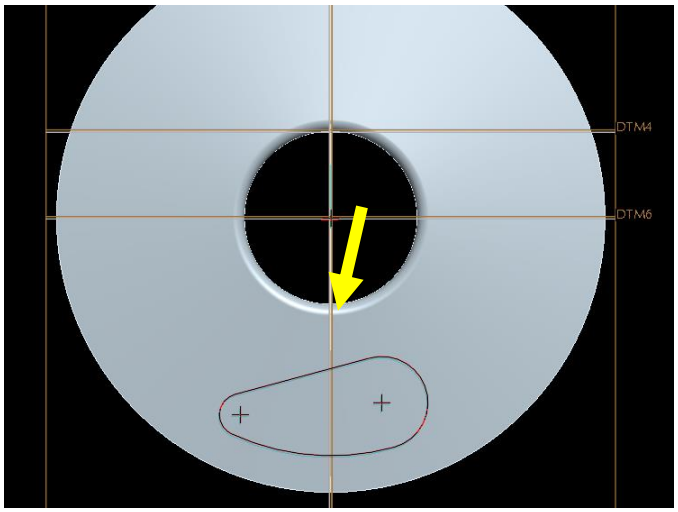
17. Select the Datum Plane that runs vertically through the model and go to “Edit/Paste”



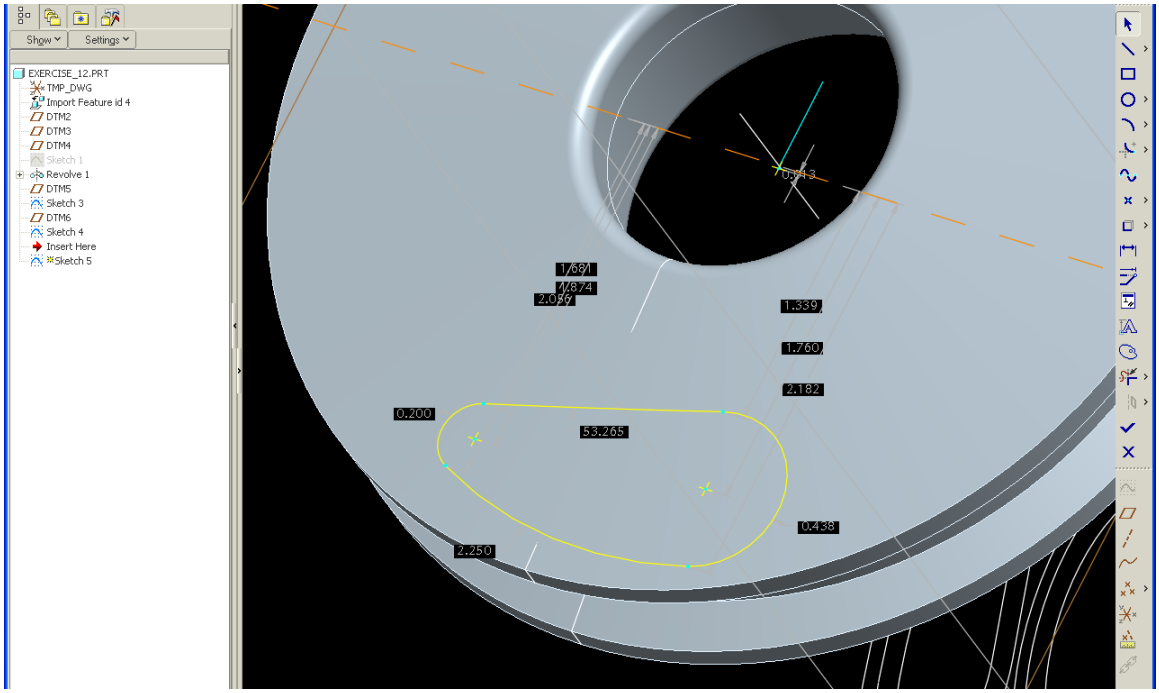
18. At the “Sketch” prompt select the datum plane you created that runs horizontally through the model that was created in steps 13 and 14 as a reference.



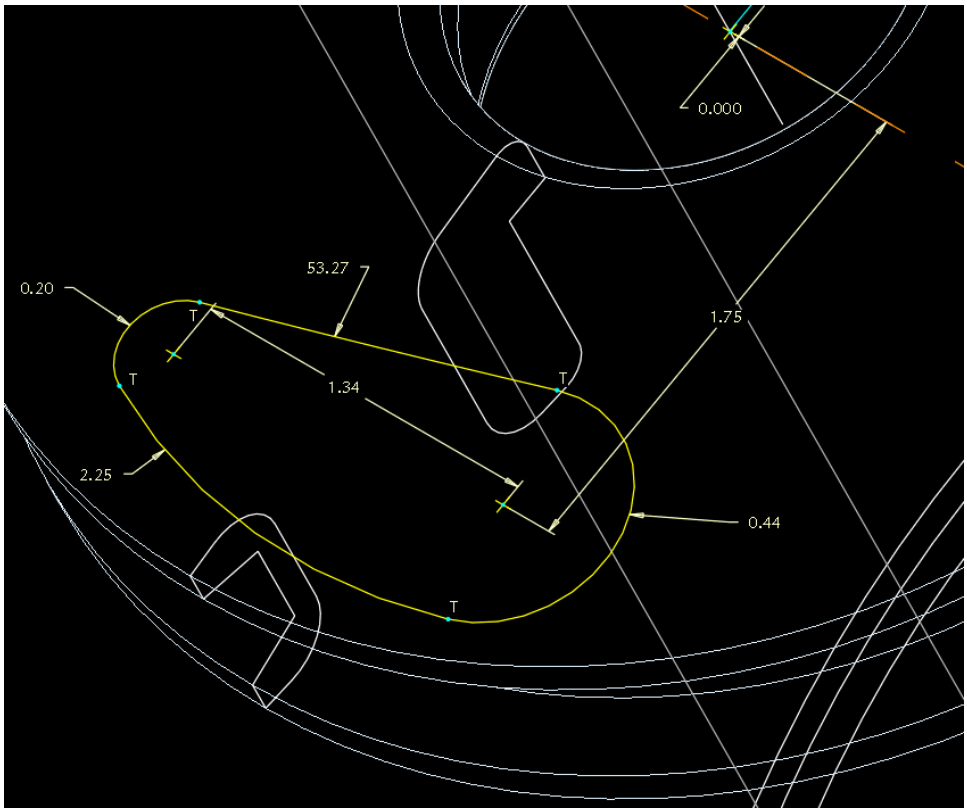
19. Now you can position with the cursor the approximate location for the cut out.



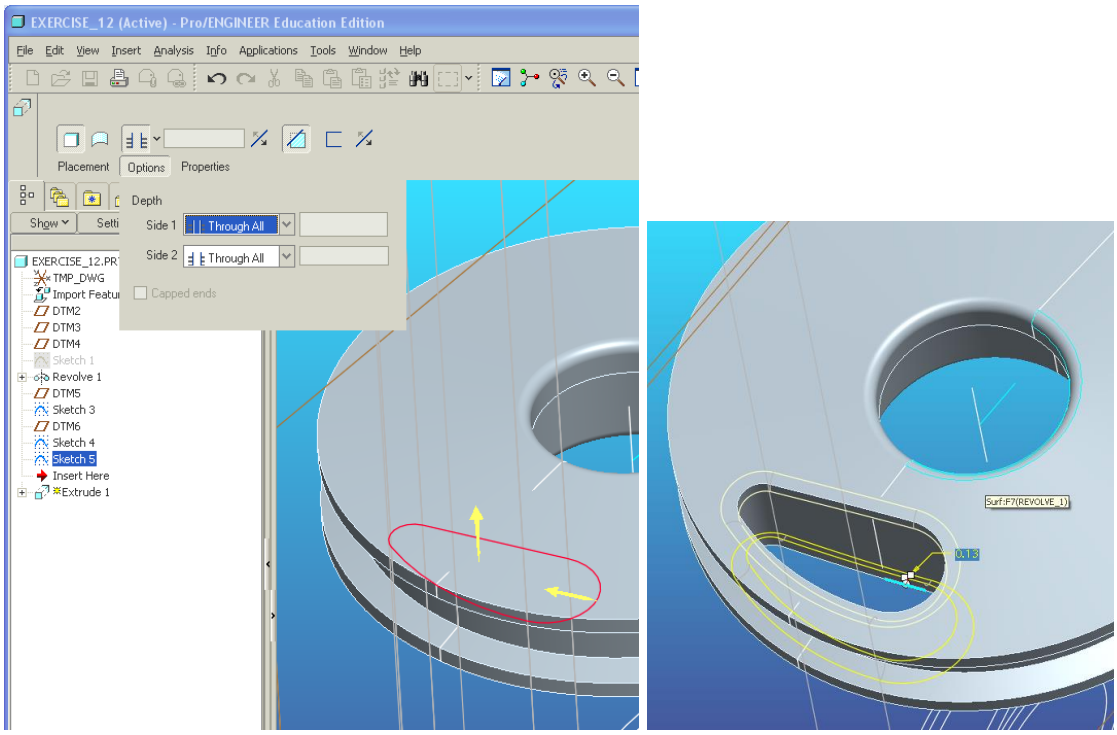
20. X



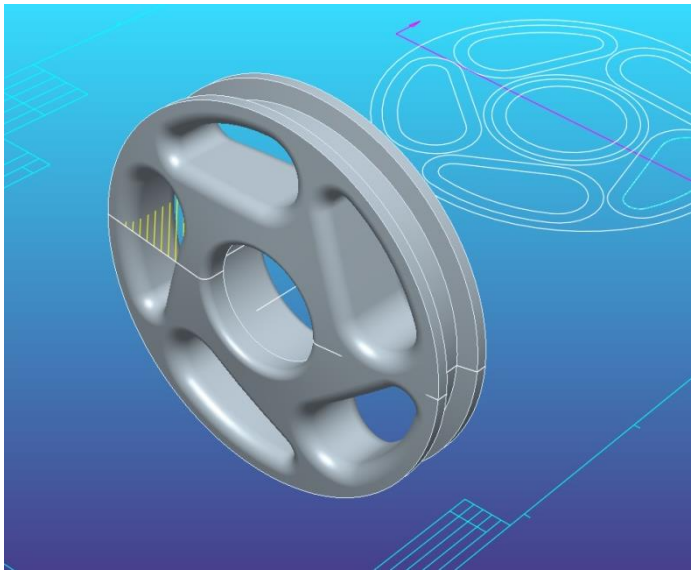
21. Insert tangent relations on all four connection points, then re-dimension as shown. Hit “Done”.



22. Go to extrude cut through all in both directions. Add .125” Rounds and circular patter 5 instances.



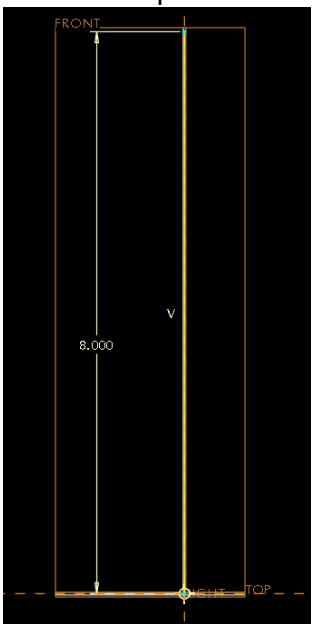
23. Completed model.



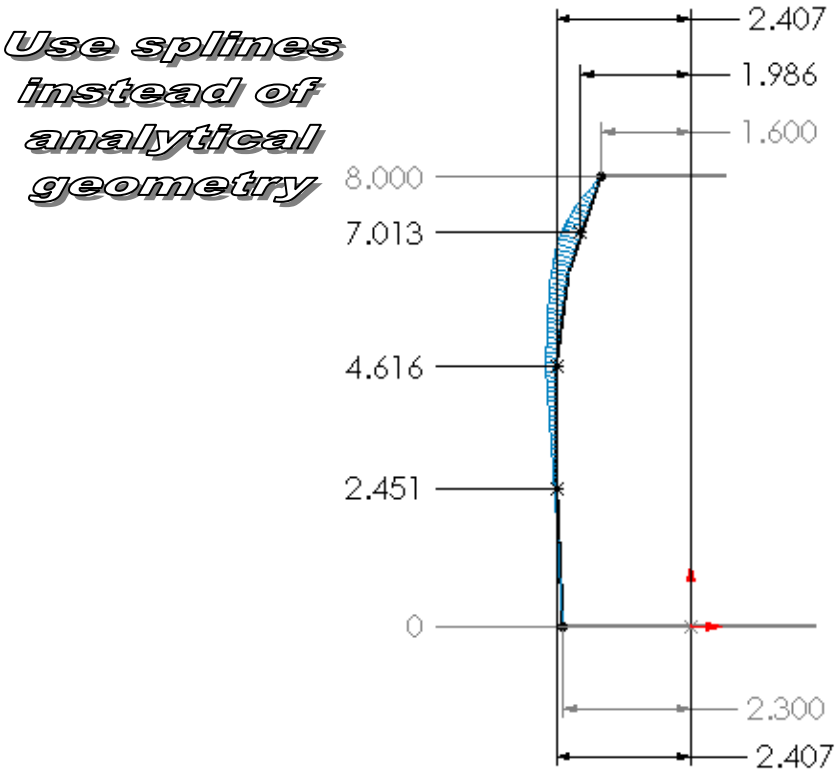
EXERCISE 13

Variable Section Sweeps

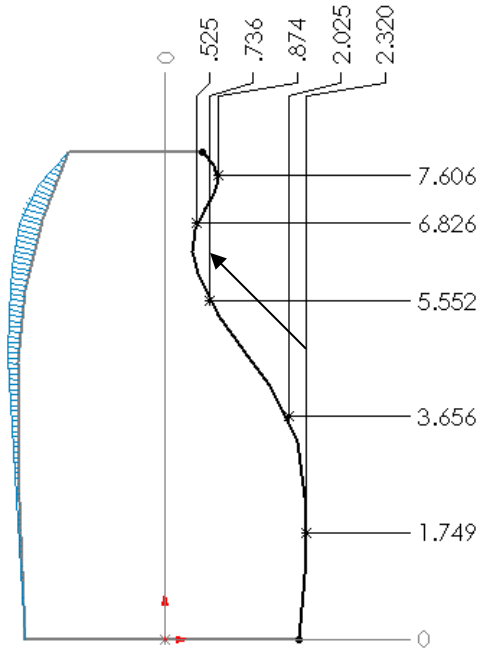
1. Sketch an 8” high vertical line on the front datum plane. Hit “Done”.



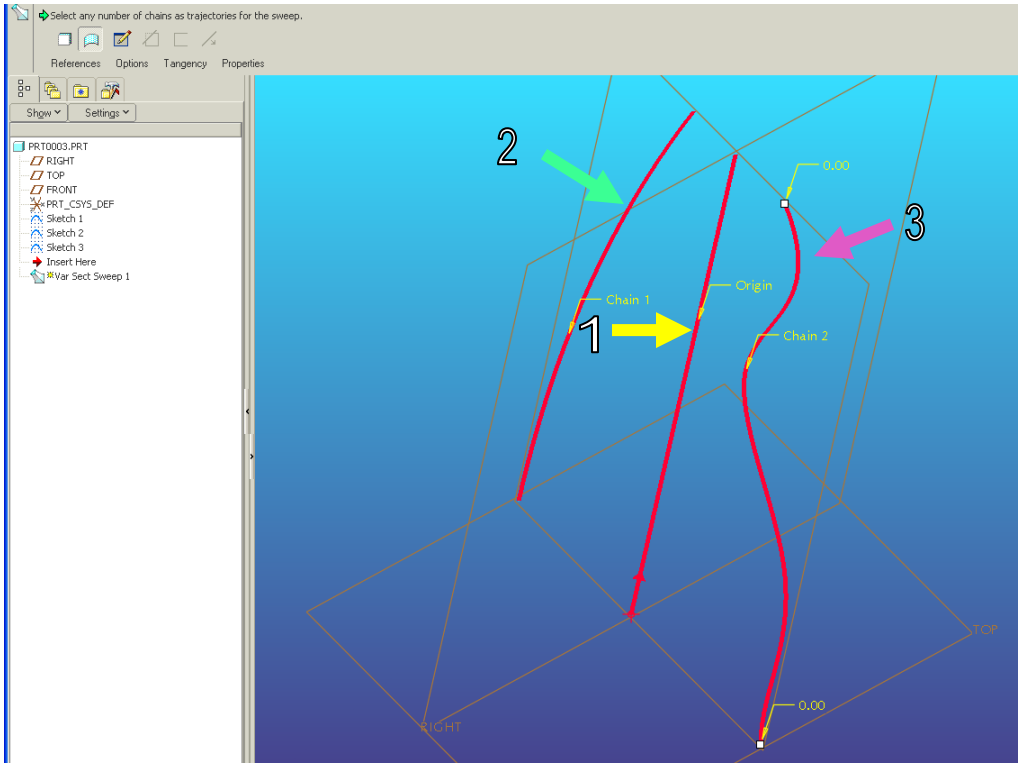
2. Select the “Front” plane and sketch a spline using 3 points. Hit “Done”.



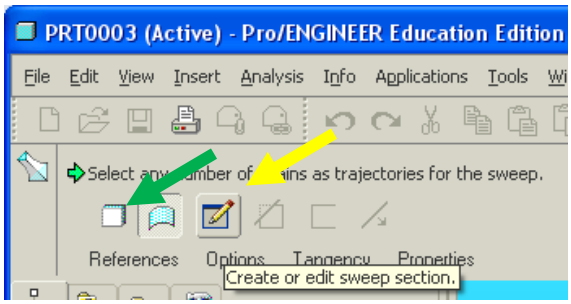
3. Sketch the following on the Front plane. (5 spline points)



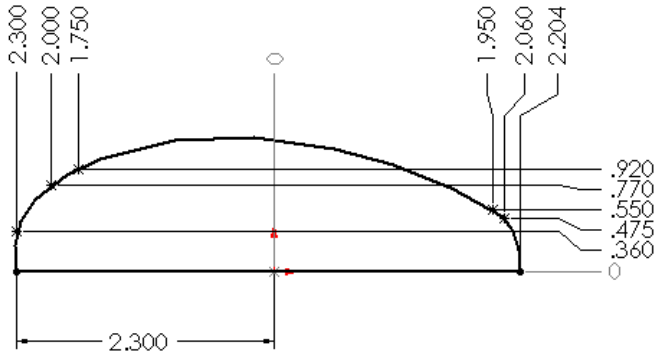
4. CTRL select **in order** the three sketches. Go to "Insert/Variable Section Sweep. 1 - Yellow, 2 - Green, 3 - Magenta.

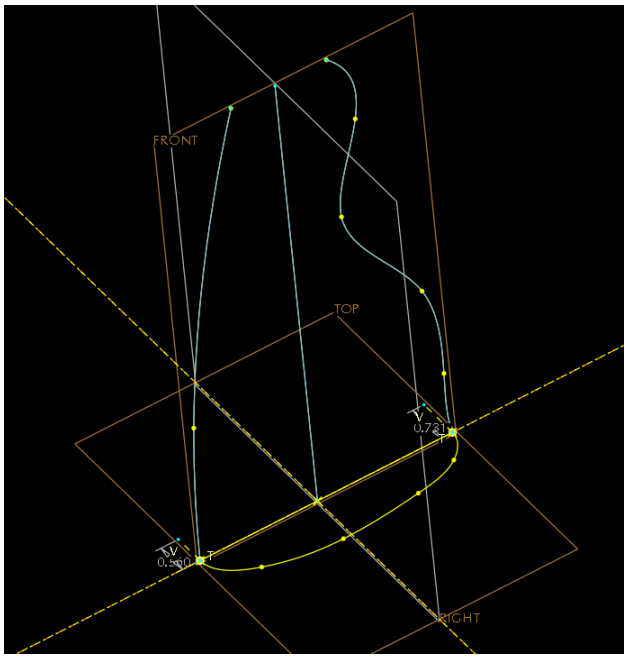


5. Select the “Edit sweep” icon from the command line, and don’t forget to set it to a Solid.

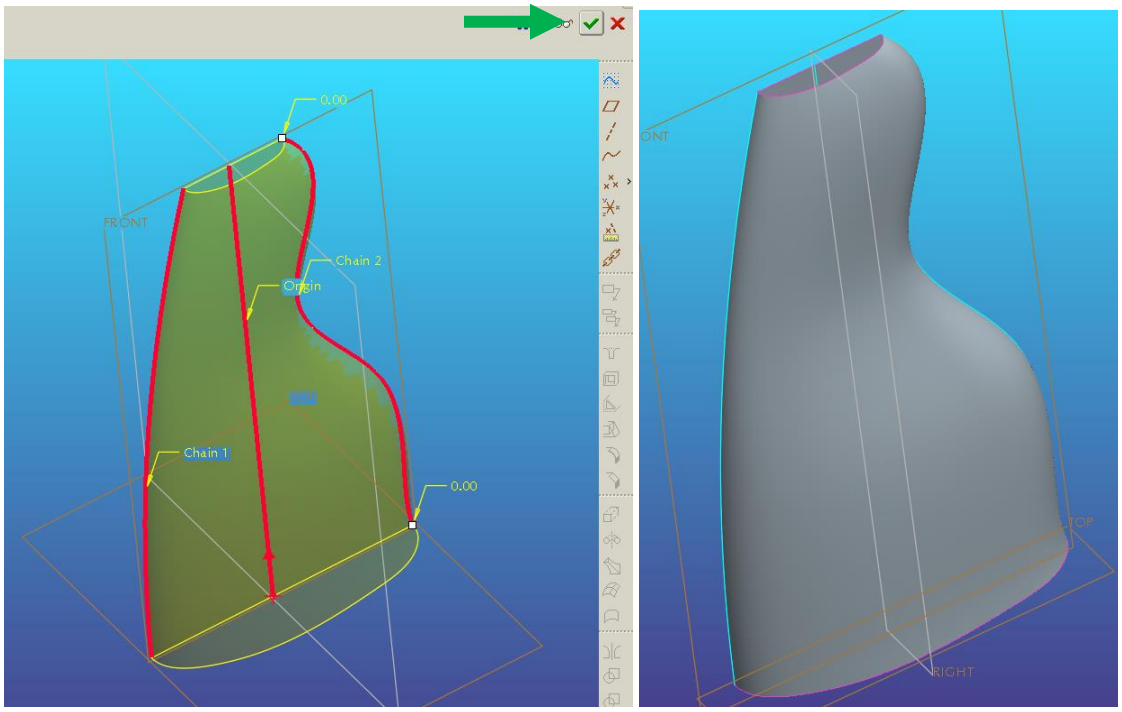


6. Draw the following profile.

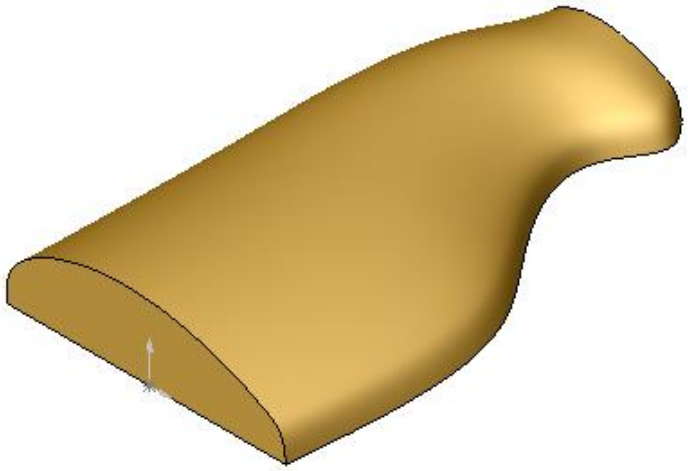




7. Select “Done”.

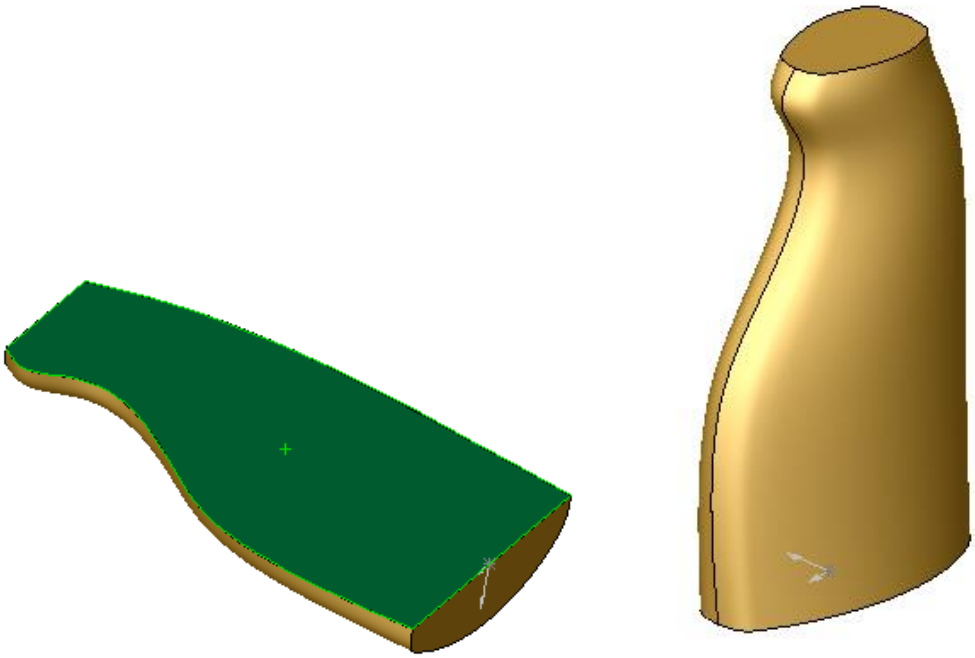


8. Sweep completed.

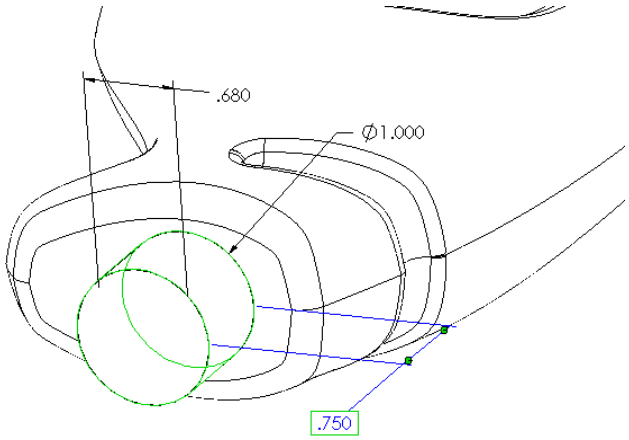


9. Creating Filets, Add .25” Rounds on the top and bottom edges before mirroring.

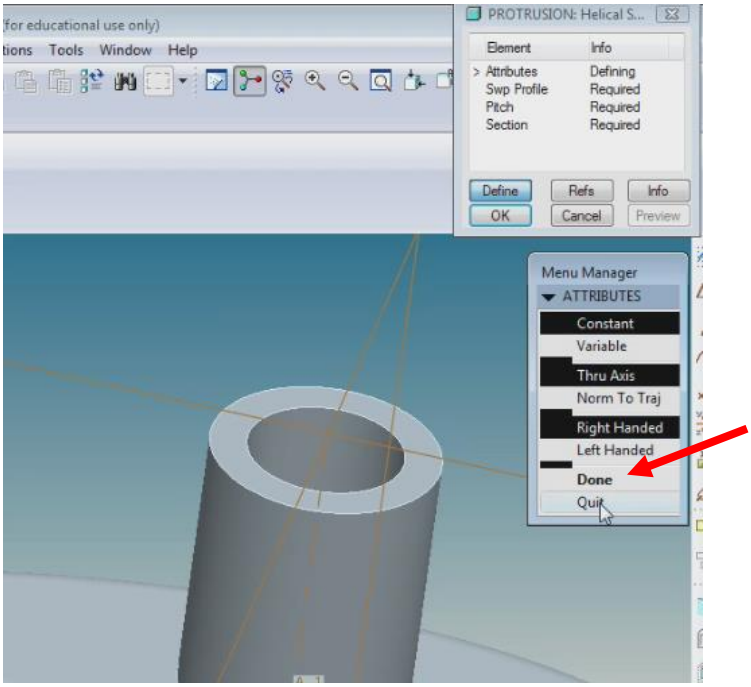
10. Select the back flat face and go to “Edit/Mirror”.



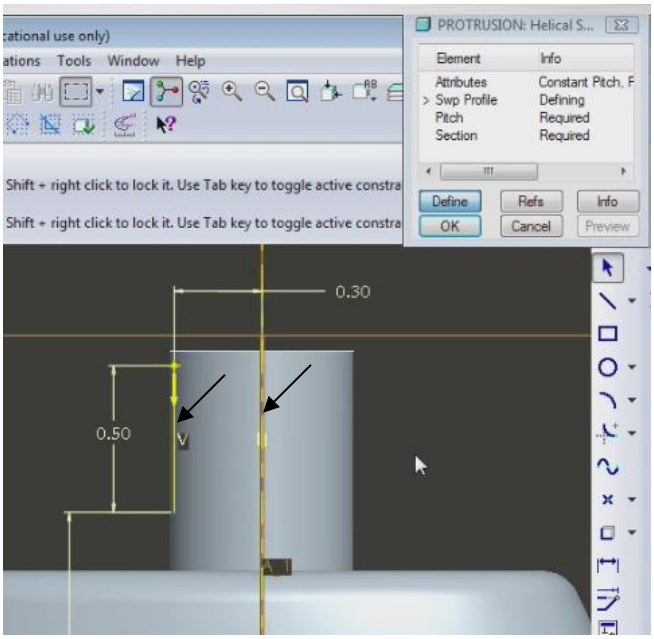
11. Insert the neck of the bottle as shown below.



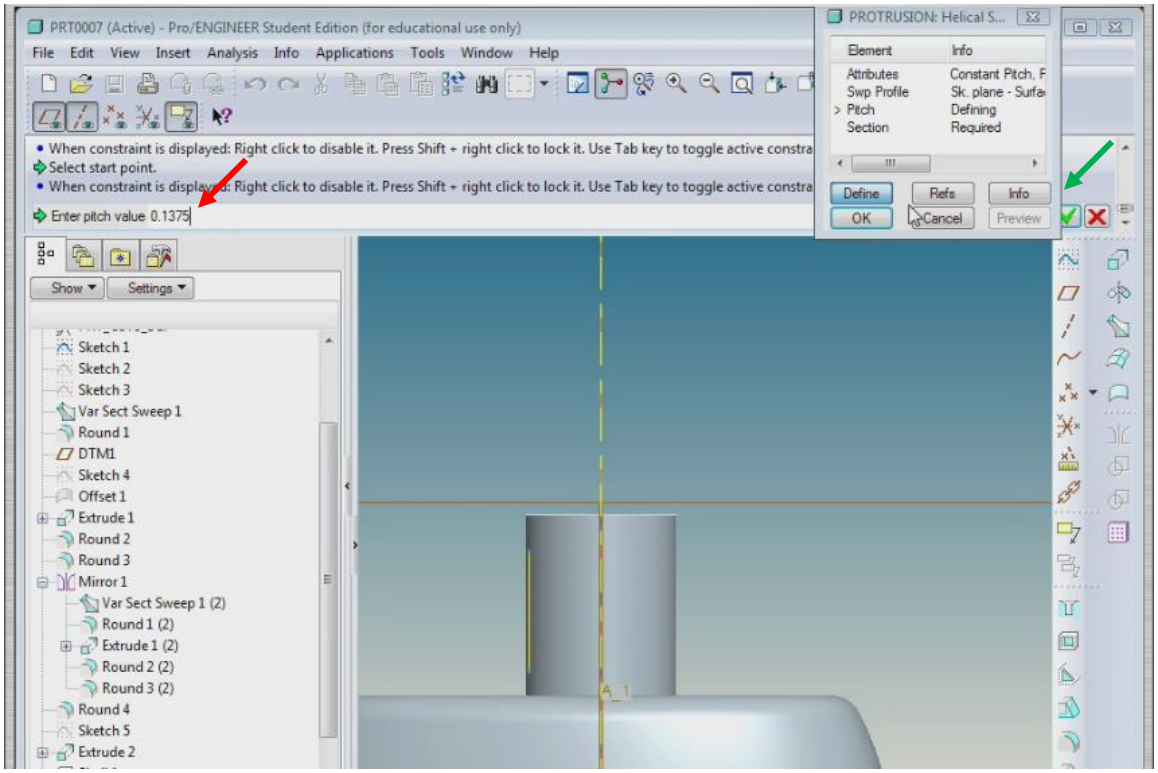
12. **Creating a Thread –** Go to “Insert/Helical Sweep/Protrusion”. Hit Done.



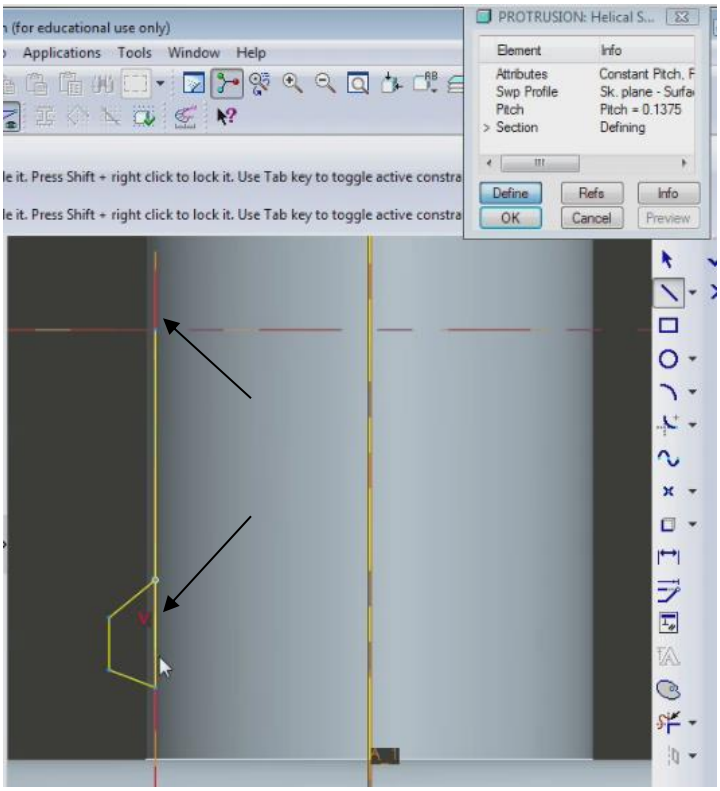
13. Select the right plane and sketch a vertical center line centered on the neck also sketch a solid line just slightly offset into the neck from the silhouette edge. Hit “Done” (Check mark on the sketch tools.)



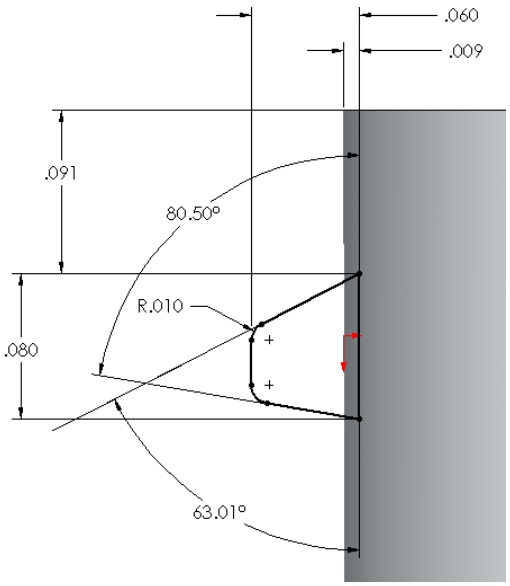
14. Set Pitch to .1375. Hit “Enter” or the Green Check mark.



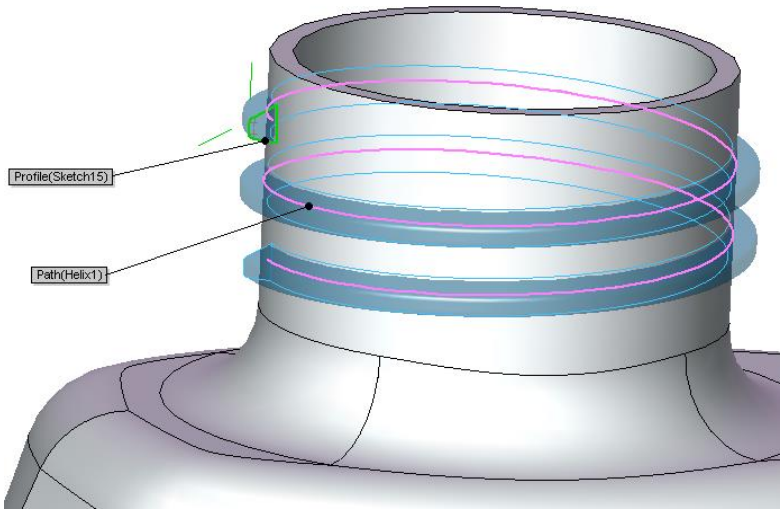
15. Now sketch the thread section profile. *Note: If you sketched the original line from the bottom up, draw as shown, however if you drew the original line from the top down begin the profile sketch at the top*.



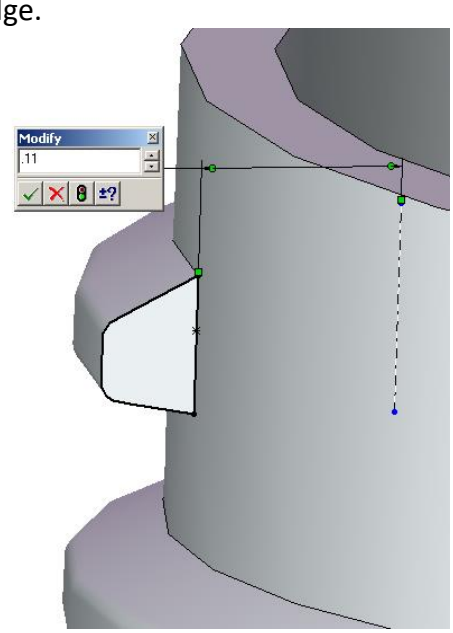
16. Now just draw the geometry of the thread.



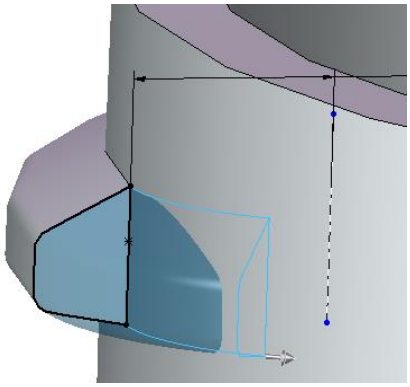
17. Hit “Preview, to view the Sweep feature. Select “OK” to finish.

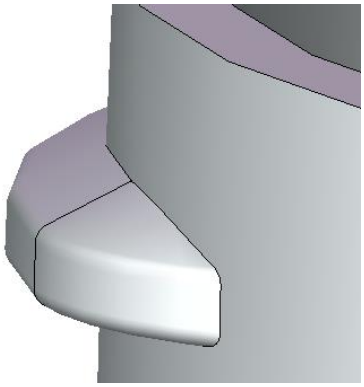


18. Select the end face of the thread, start a sketch and go to the “Use” and select “Loop”, then select the end face. Draw a vertical centerline .110” offset from the edge.



19. Revolve 56°.





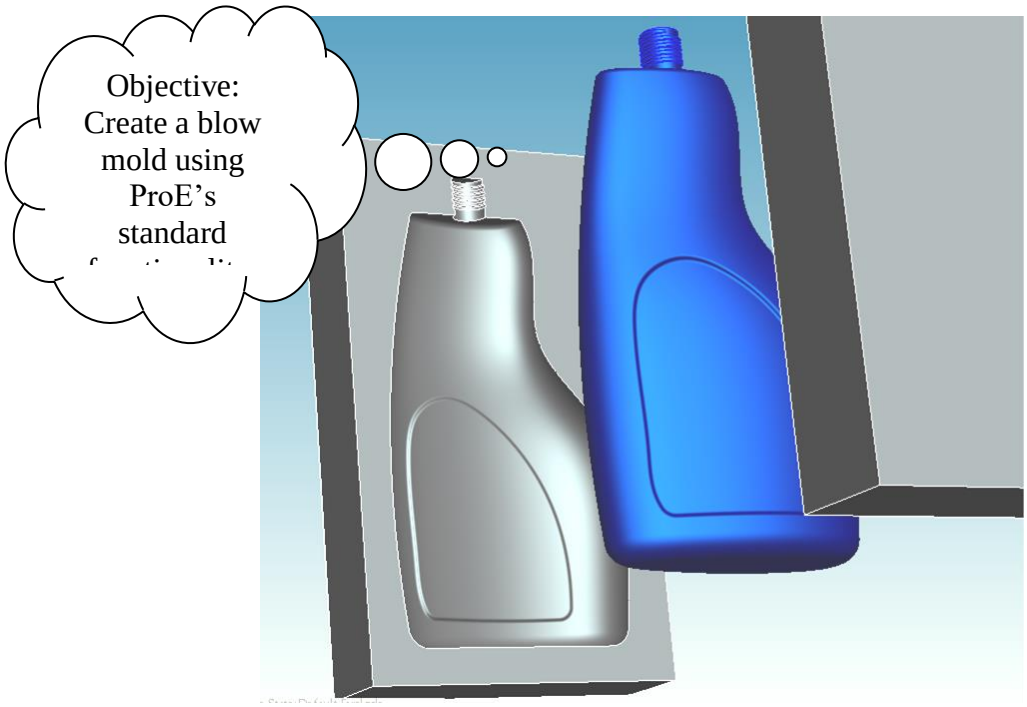
20. Complete the other side the same way. Add additional features to finish bottle. Shell at .050”.

Finished

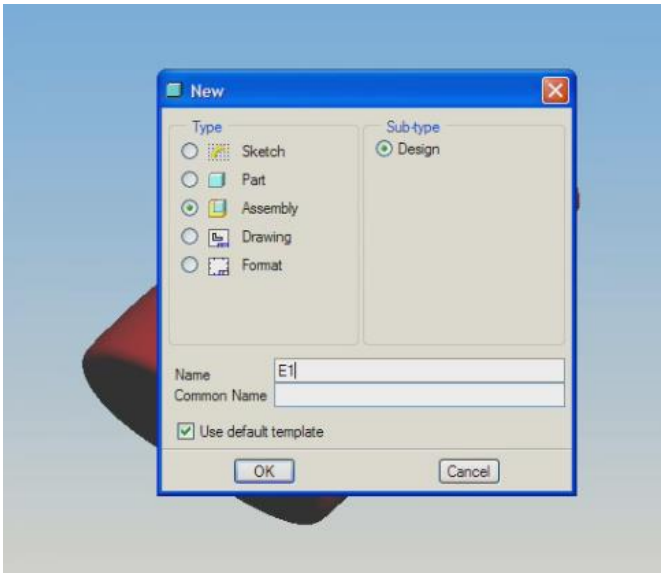
EXERCISE 14

Mold/Cavity Creation

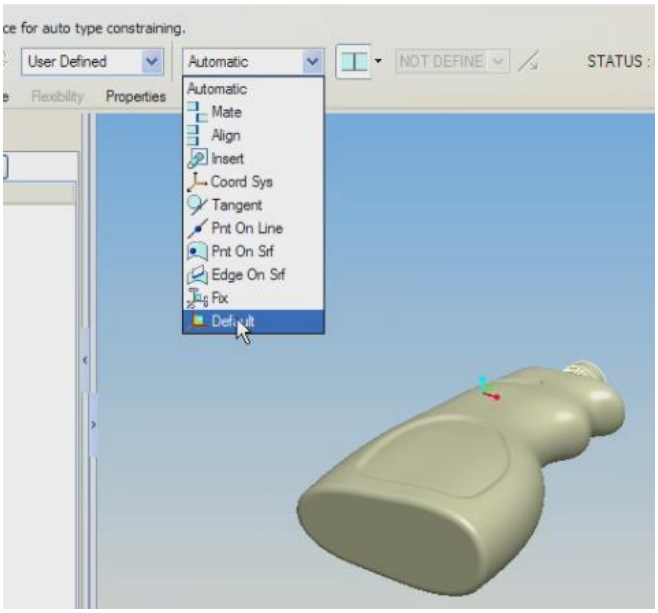
Pro Engineer *has a dedicated Mold package for automating the process of designing molds, however in this exercise we look at ProE's functionality by itself.*



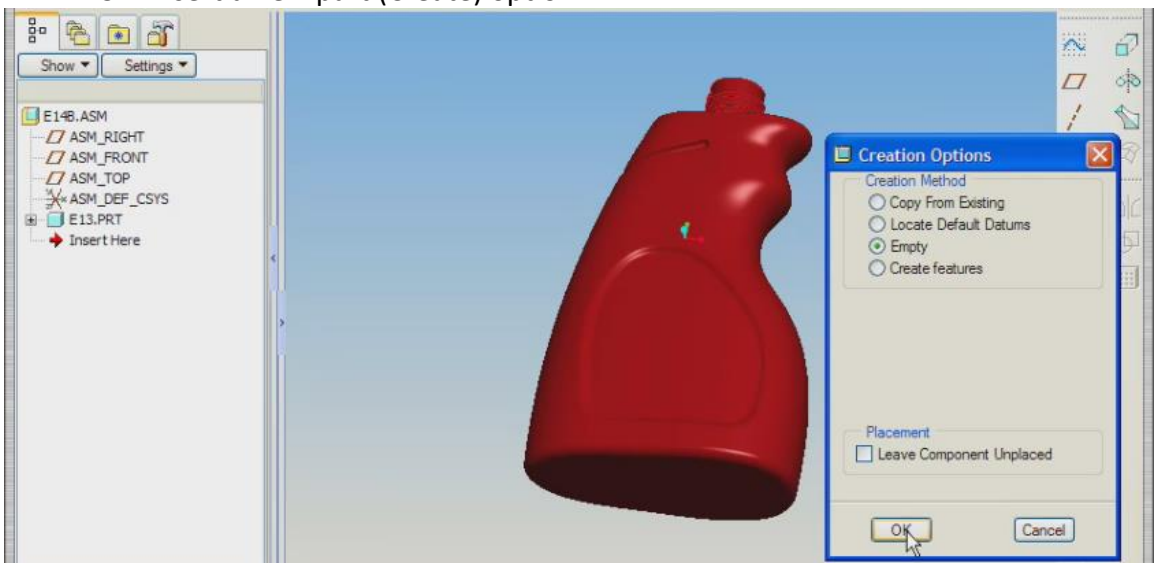
1. Using the Edit/Component Operations inside of an assembly helps us to create cavity sections of a mold. Begin by Starting a new assembly in ProE.



2. Insert the E_14_Bottle part file and use the default placement option.

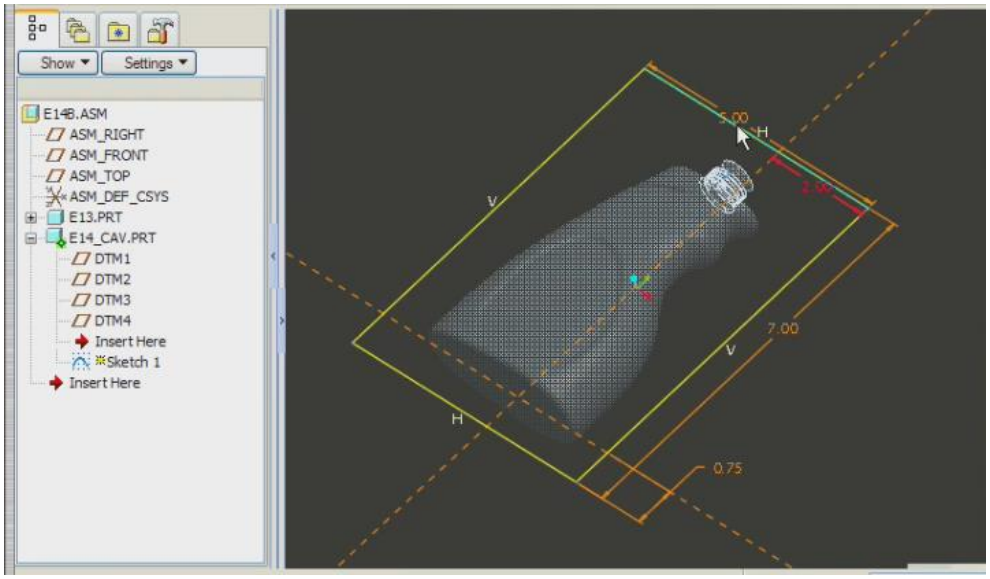


3. Insert a new part (Create) option.

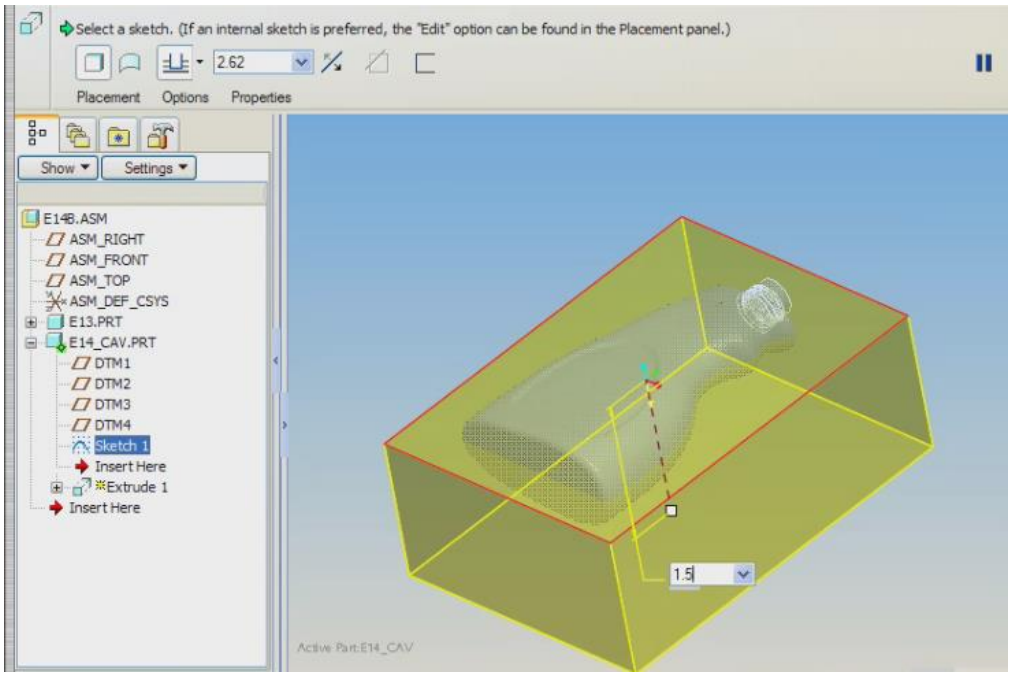


4. Name it “E14_Cav”, then create Front, Top, and Right datum planes.

5. Start a sketch on the front plane and draw a rectangle around the bottle.

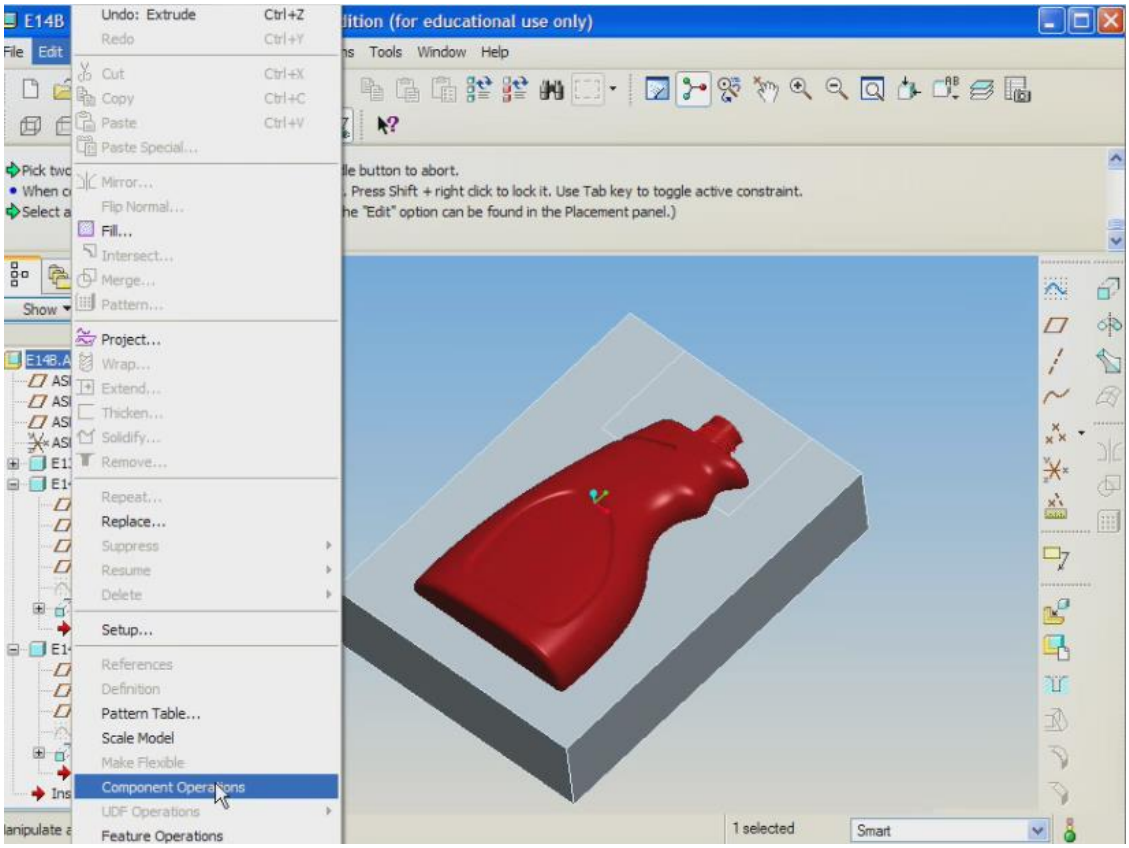


6. Extrude 1.5” down.

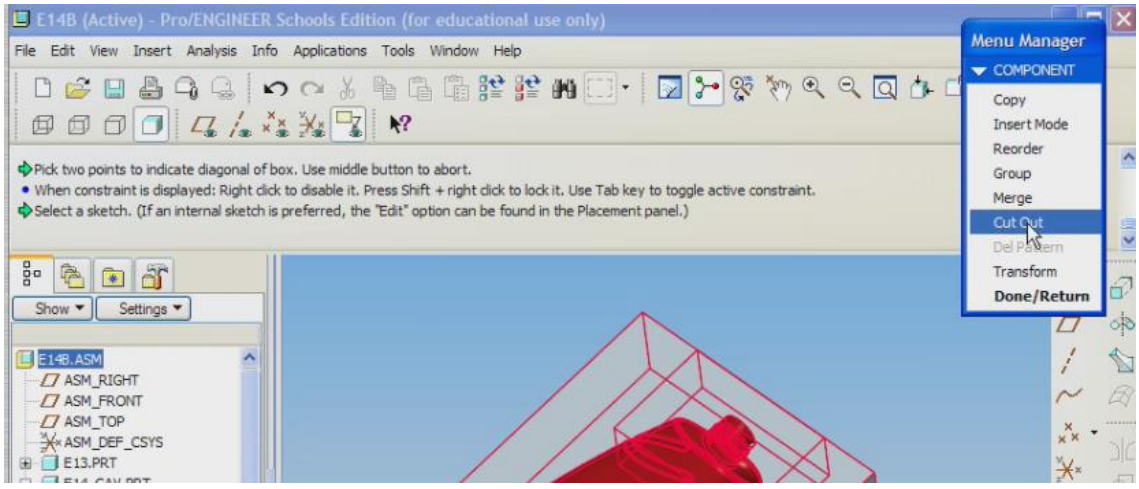


7. Right click on the Assembly marker (Top of the tree) Select Activate or Regenerate.

8. Go to Edit/Component Operations.

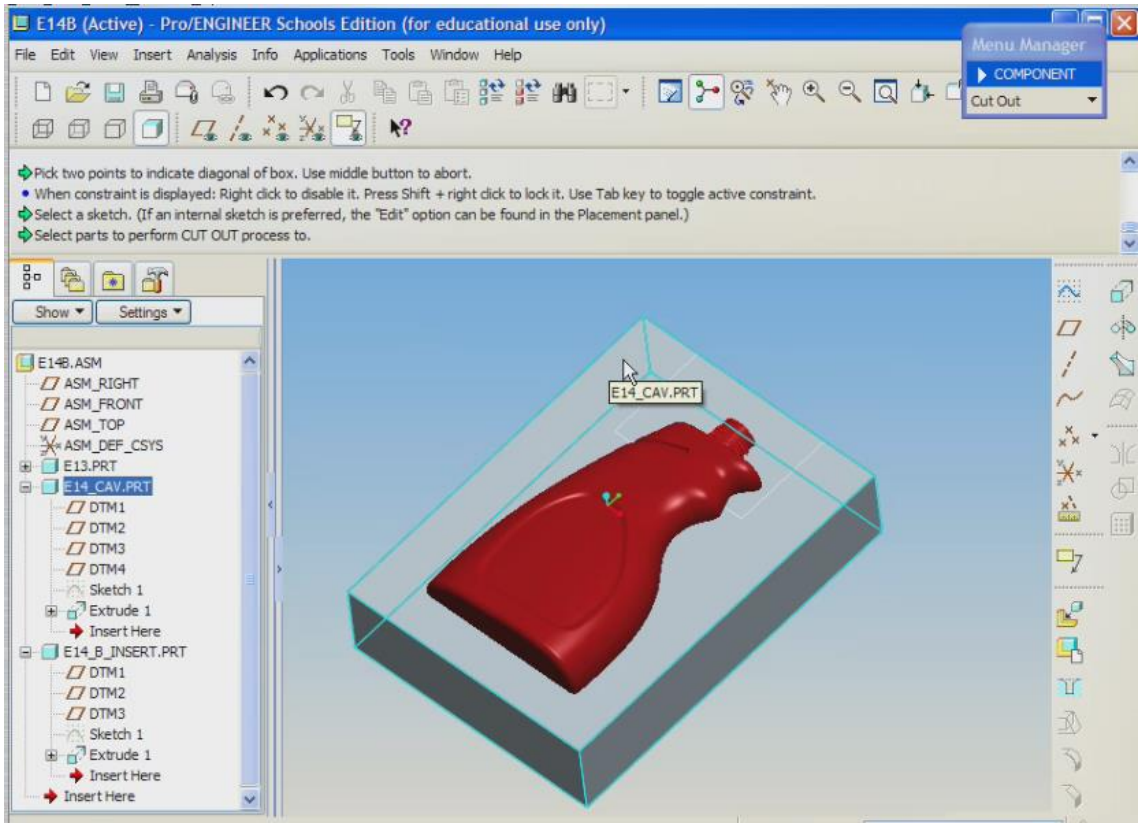


9. Select “Cut Out”.



10. Select the Cavity, then Center Mouse Button click once.

11. Select the Bottle, then Center Mouse Click two times.

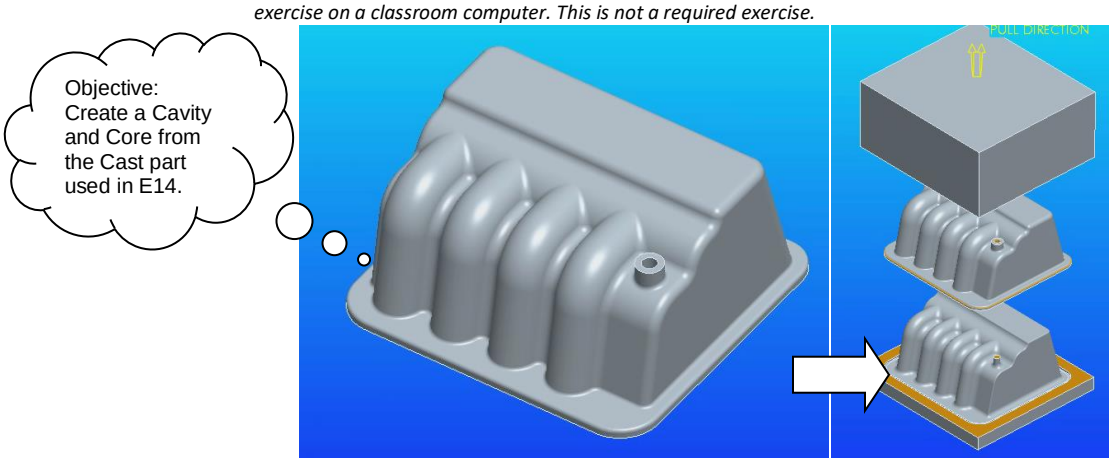


12. Go to View/Explode and separate the bottle from the cavity to view the Cavity.

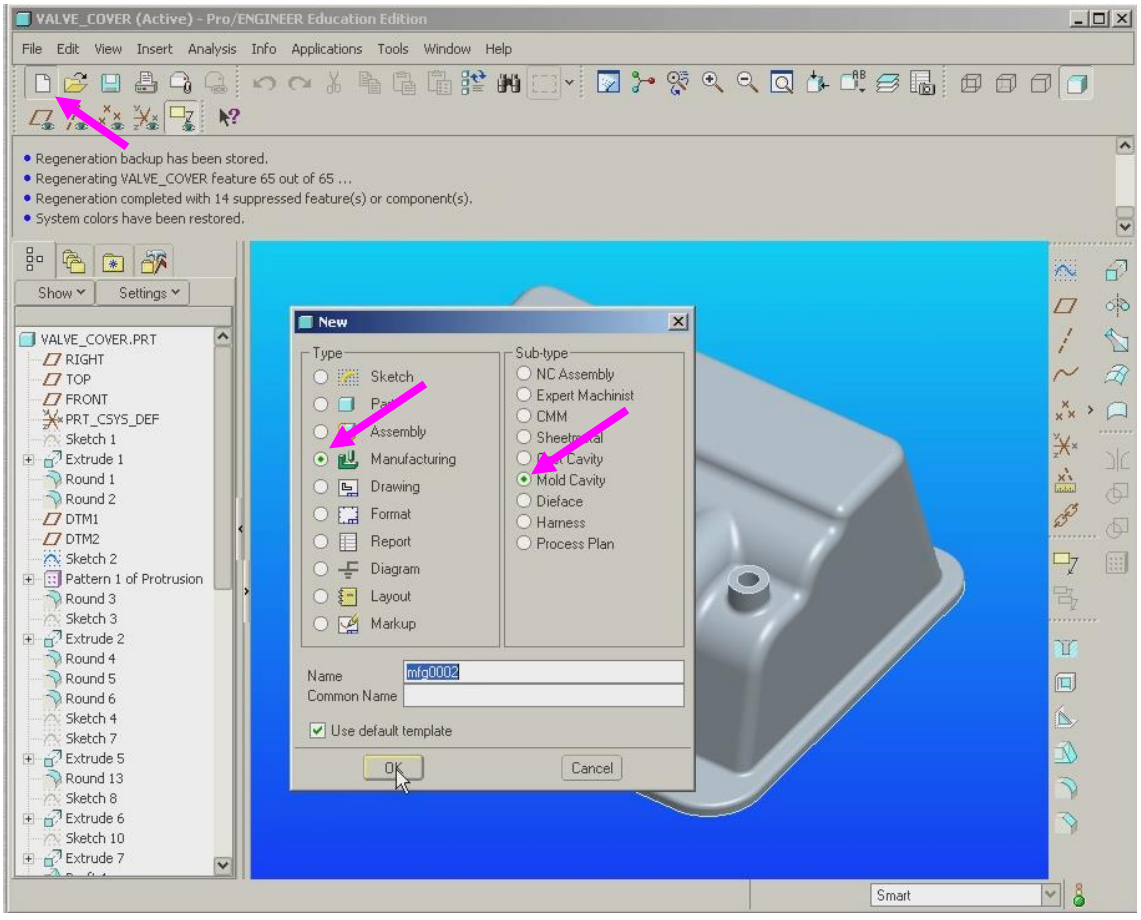
EXERCISE 15

Cavity& Core Creation *(Extra Credit)*

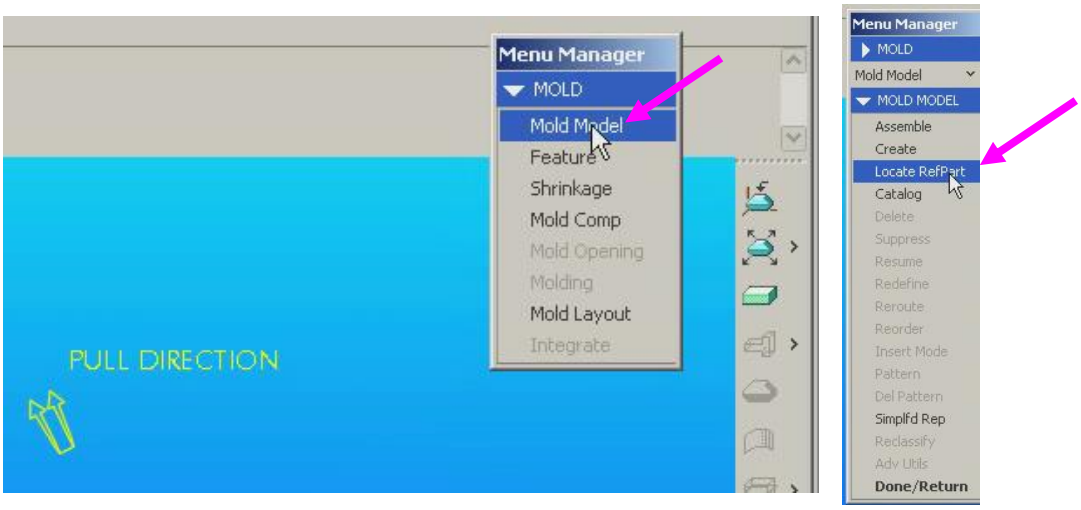
Cavities and Cores *can be created using ProMold.* *Note: ProMold is part of the Manufacturing Module and isn't available in the designer bundling that is used in the at-home student versions. In other words, you can only perform this exercise on a classroom computer. This is not a required exercise.*



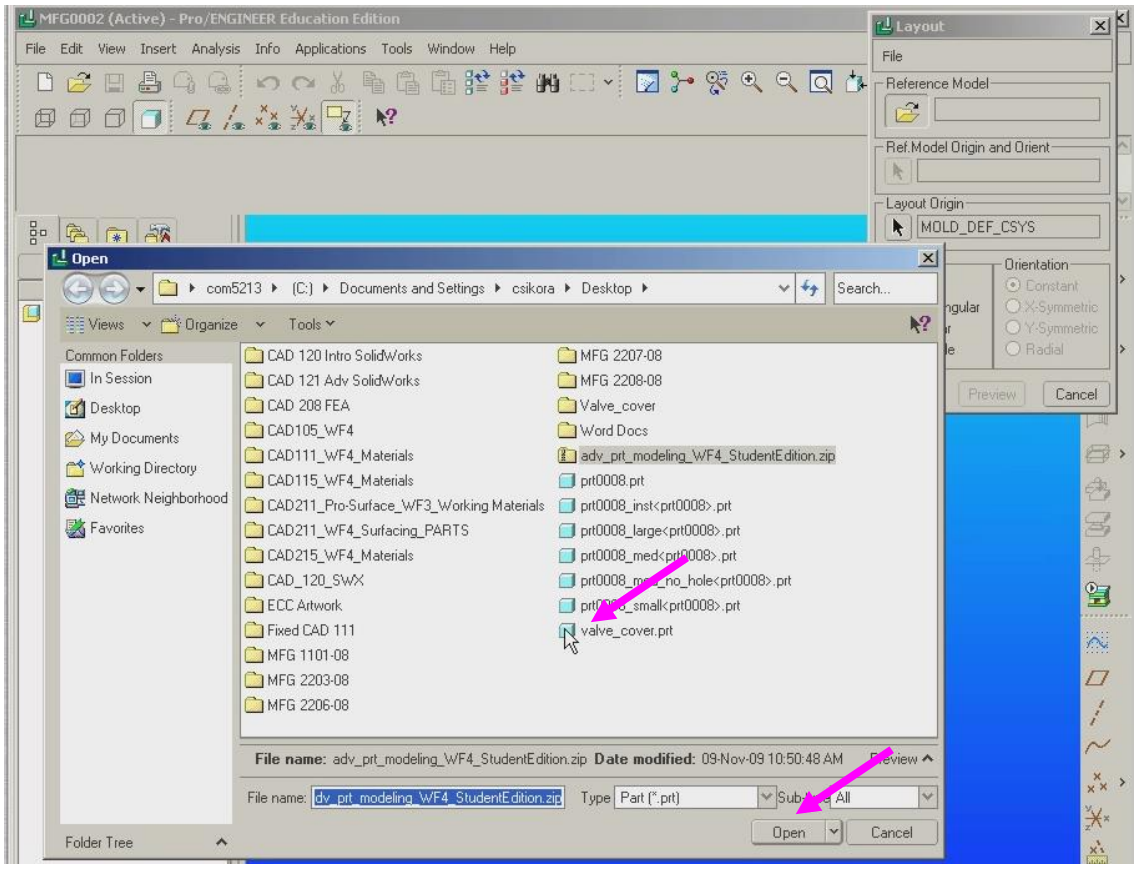
1. Start a new assembly using the Manufacturing and Mold Cavity options.



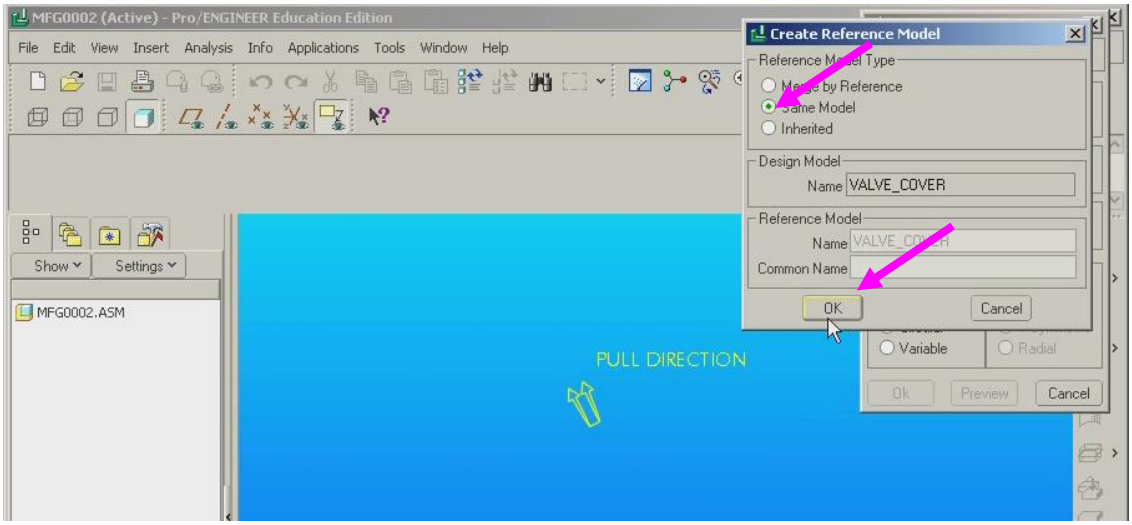
2. Once open select “Mold Model” and then “Locate RefPart”



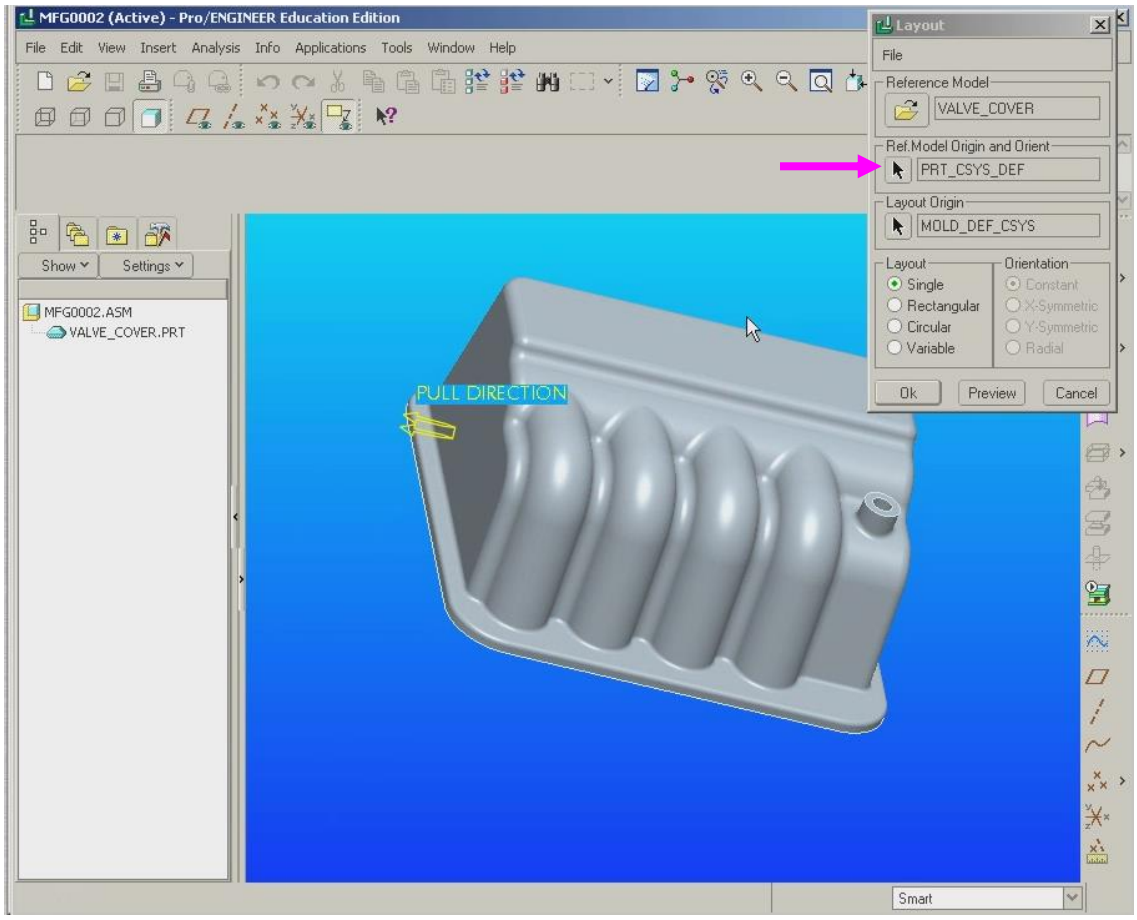
3. Locate the “valve_cover.prt” file. Hit open.



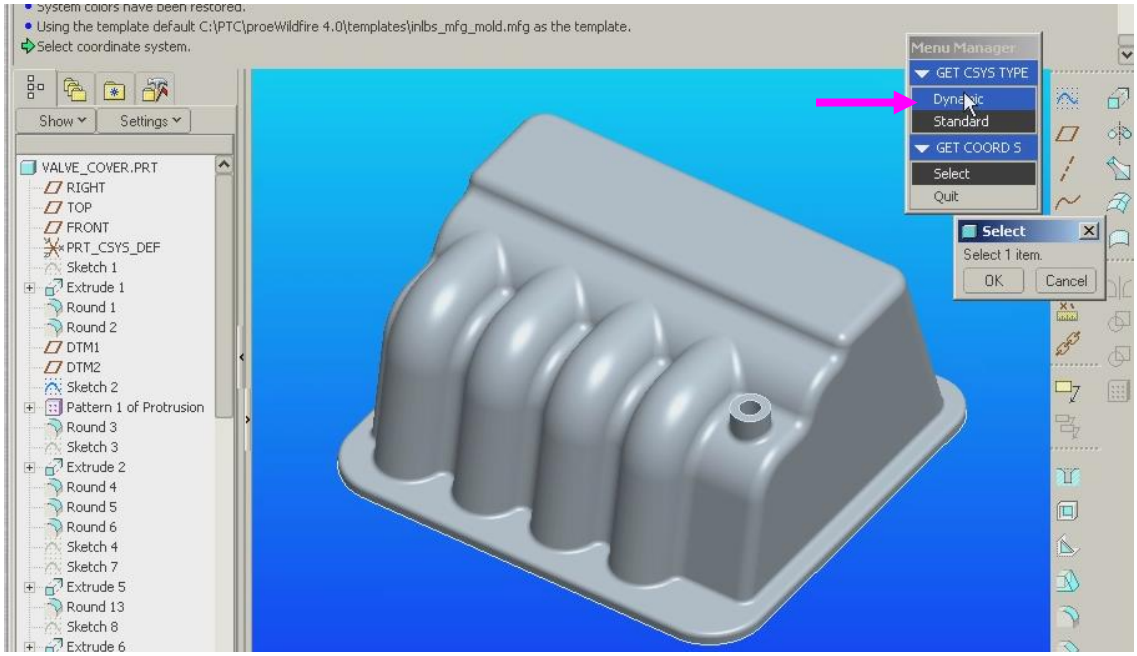
4. Select the “Same Model” option and hit ok.



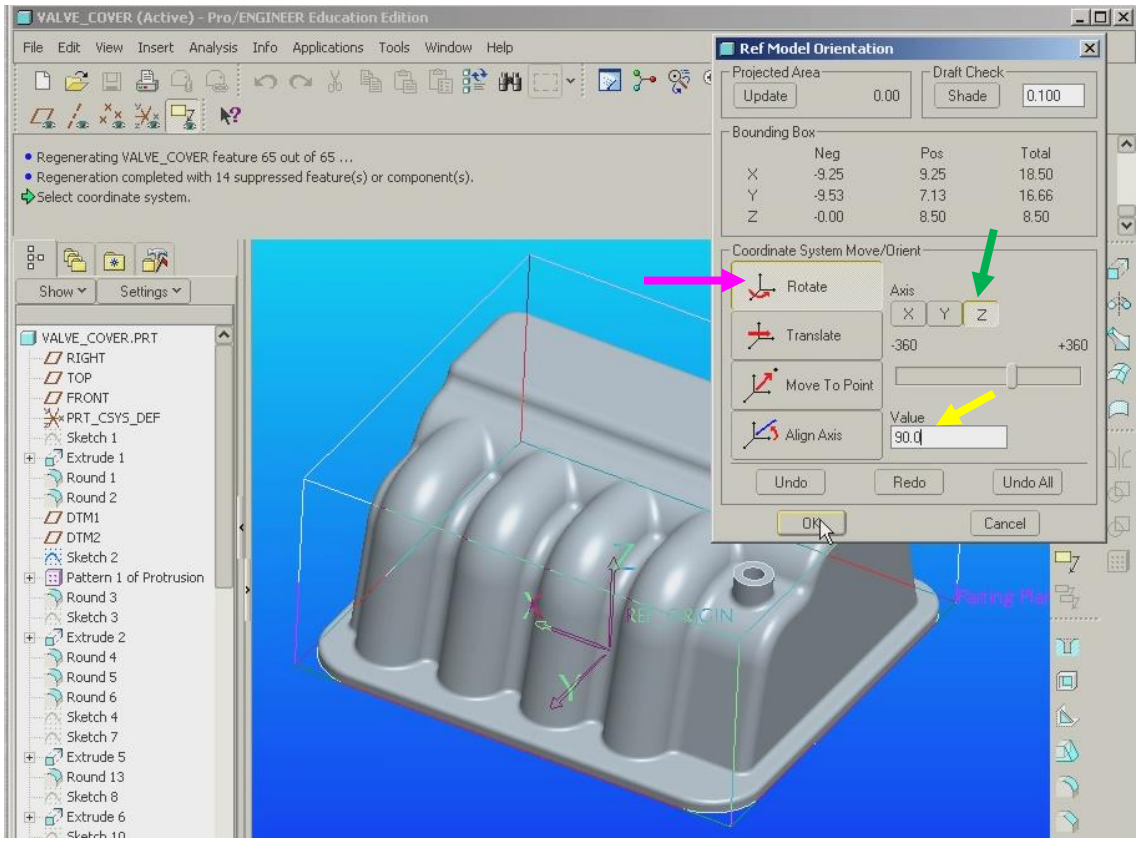
5. On the “layout” dialog box just hit the Ref Model arrow icon. Note: *This area gives the user the ability to align the pull direction with the parting line of the mold. Also the goal here is to have the mfg CSYS should have the “Z” direction pining up from the top of this model.*



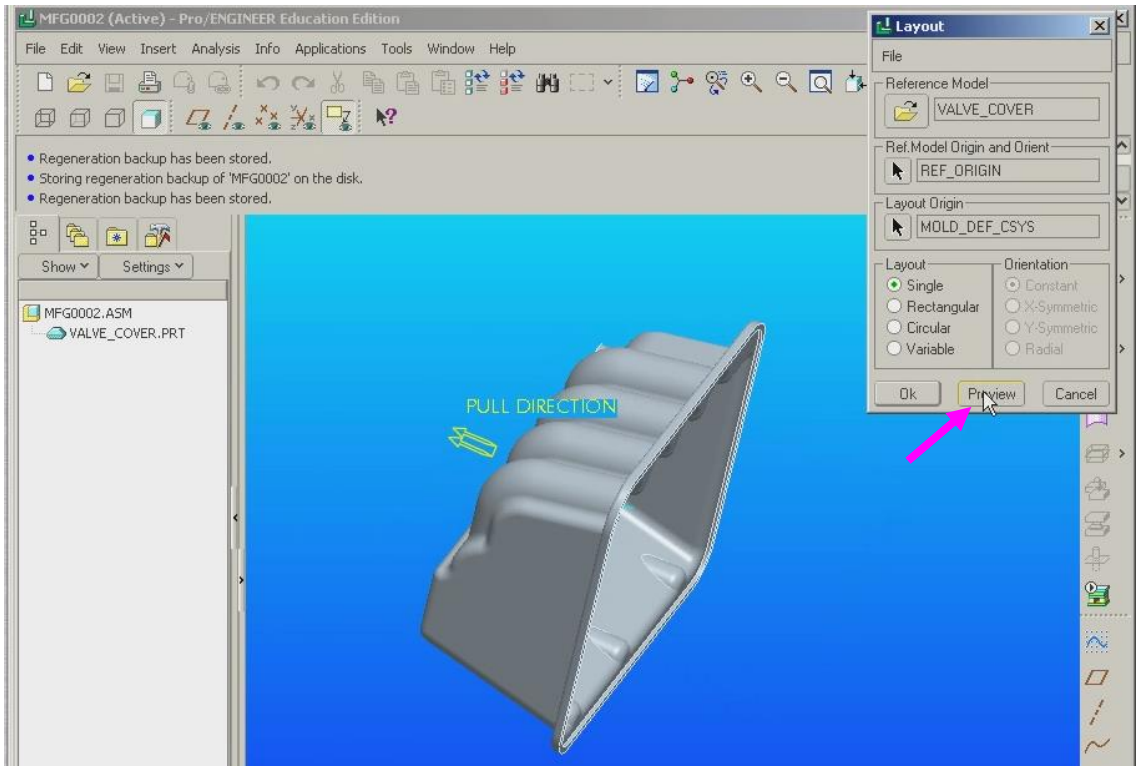
6. **Aligning the pull direction**. Select “Dynamic”



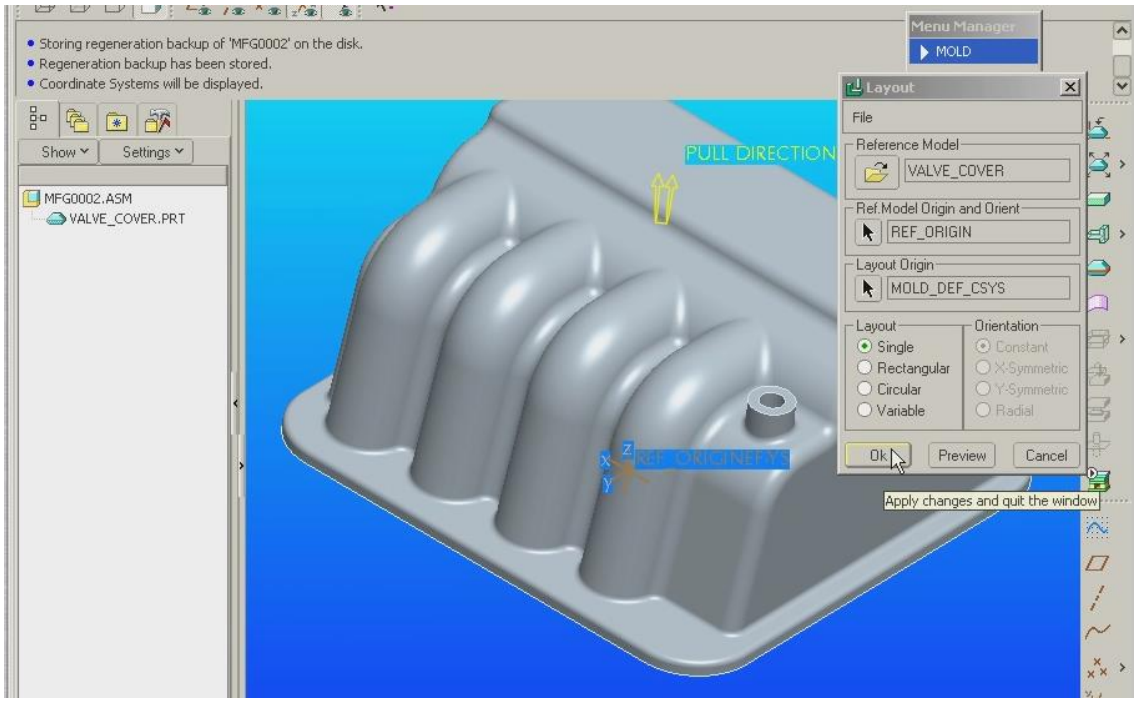
7. Select “**Rotate**”, “**Z**”, and enter “**90°**”. Hit ok.



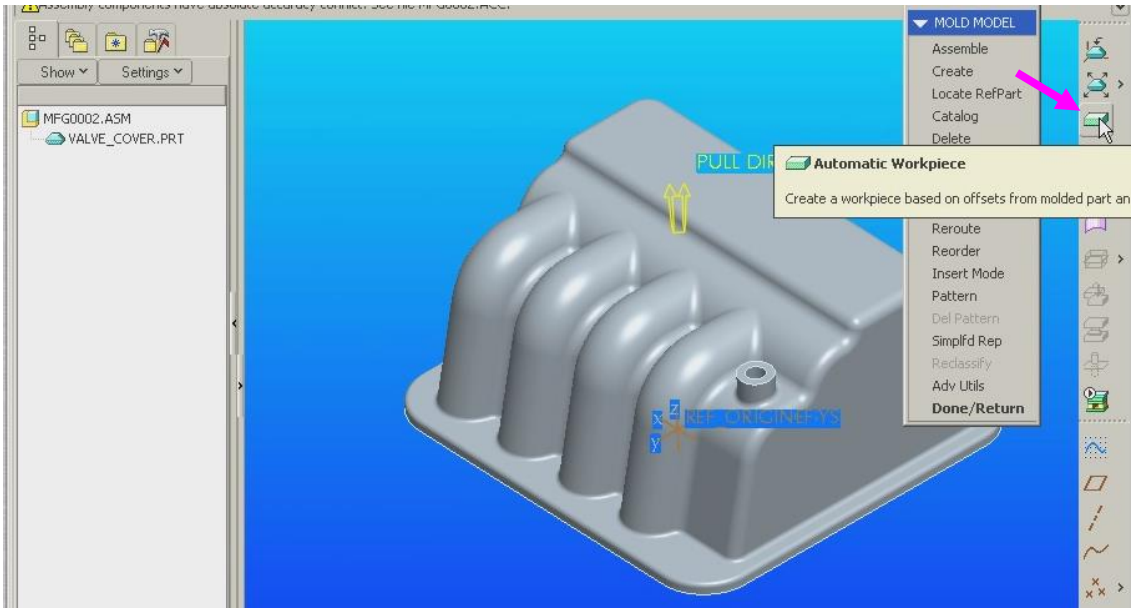
8. Now select “preview” to update the view. The “pull Direction” arrows should be pointing up and away from the top of your part.



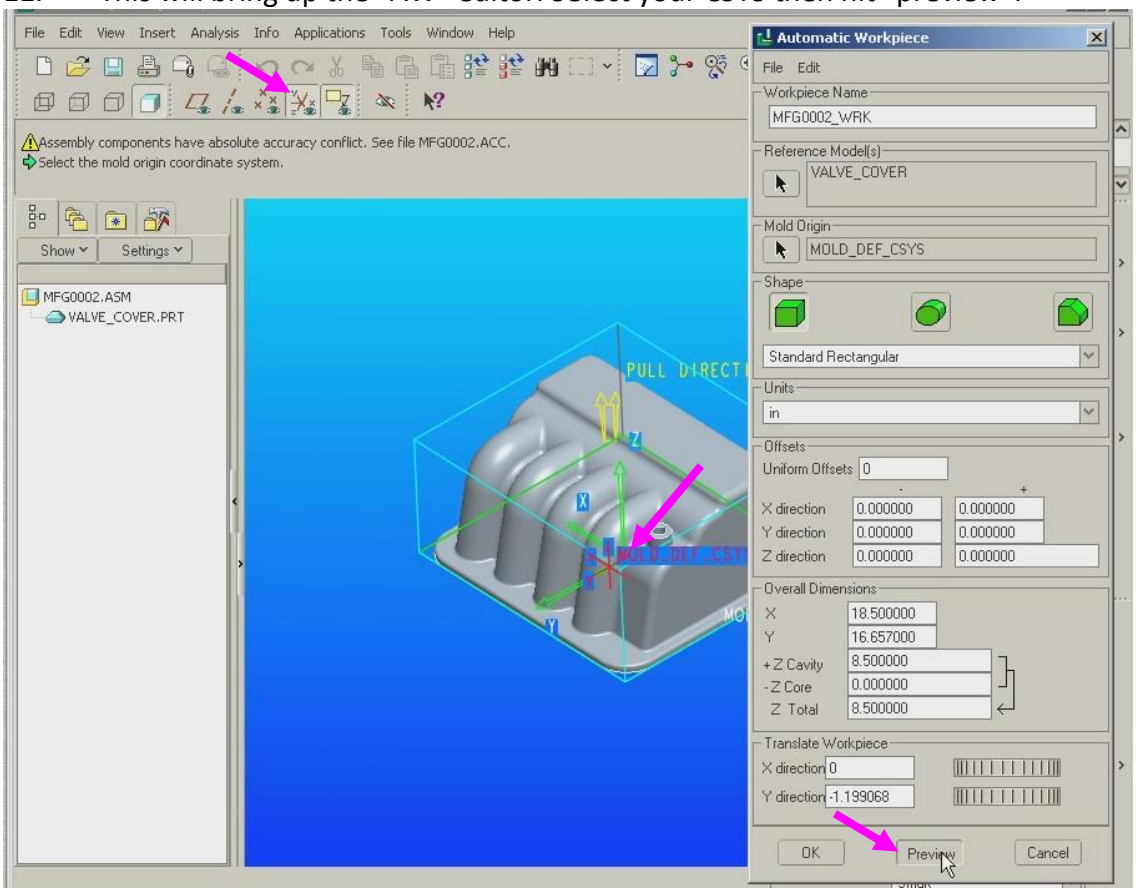
9. At this point we can now hit “OK” on the layout toolbar.



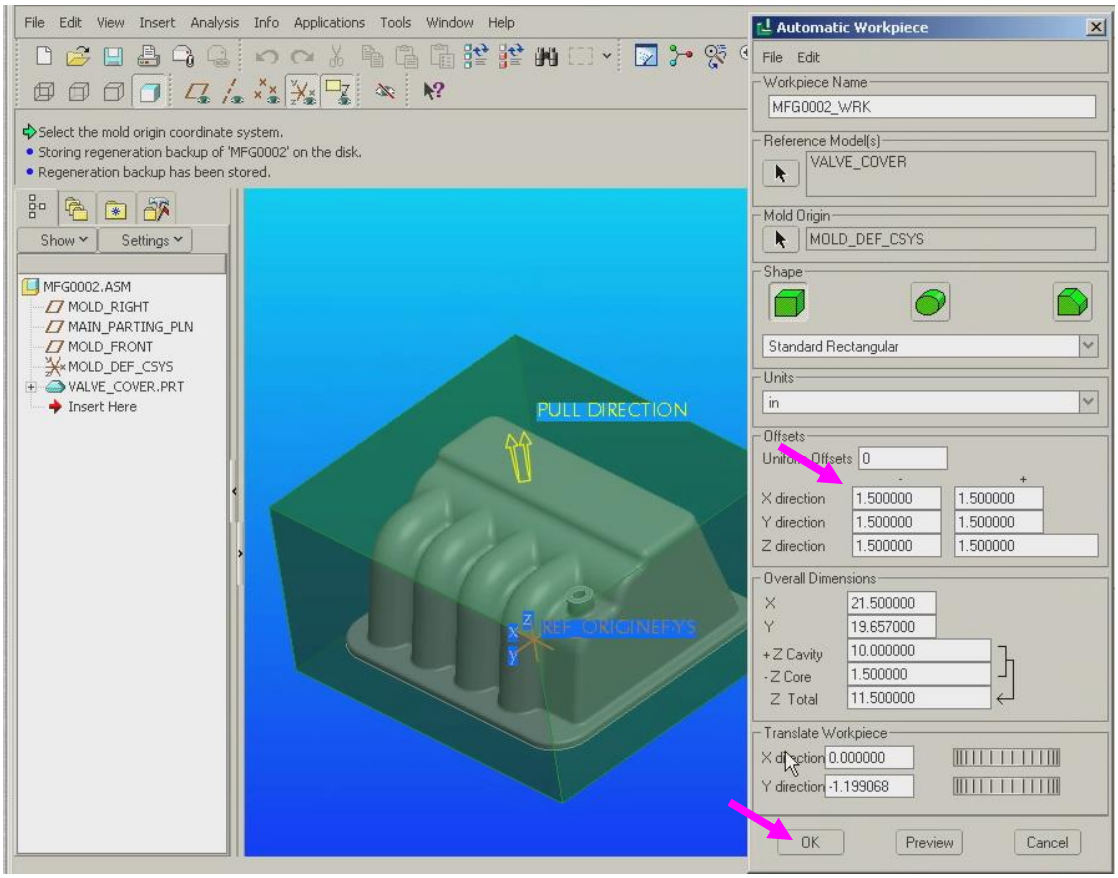
10. Once here select the “Automatic Workpiece” icon.



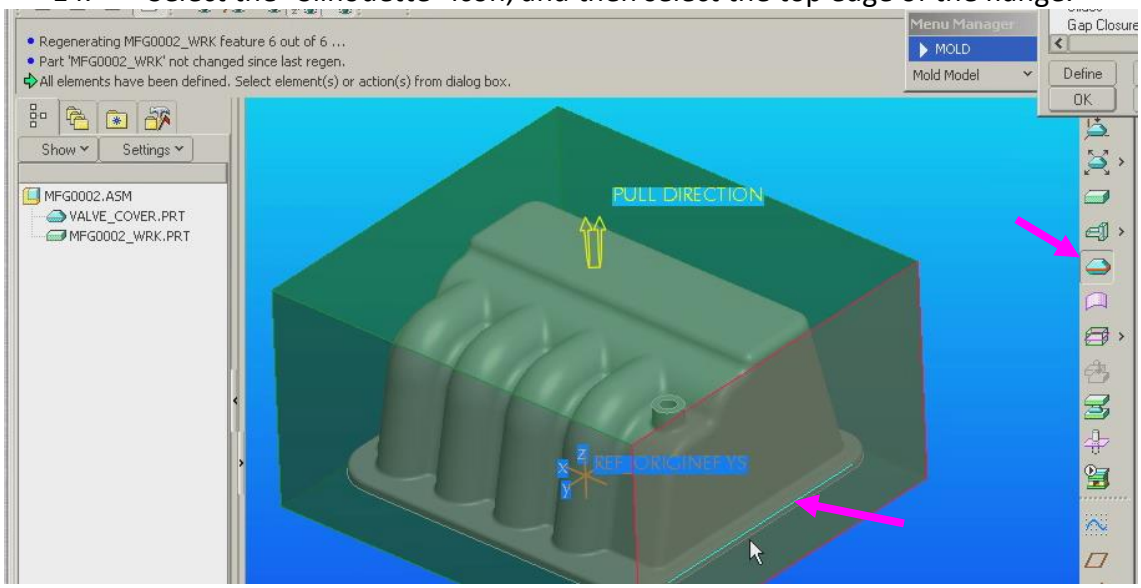
11. This will bring up the “AW” editor. Select your CSYS then hit “preview”.



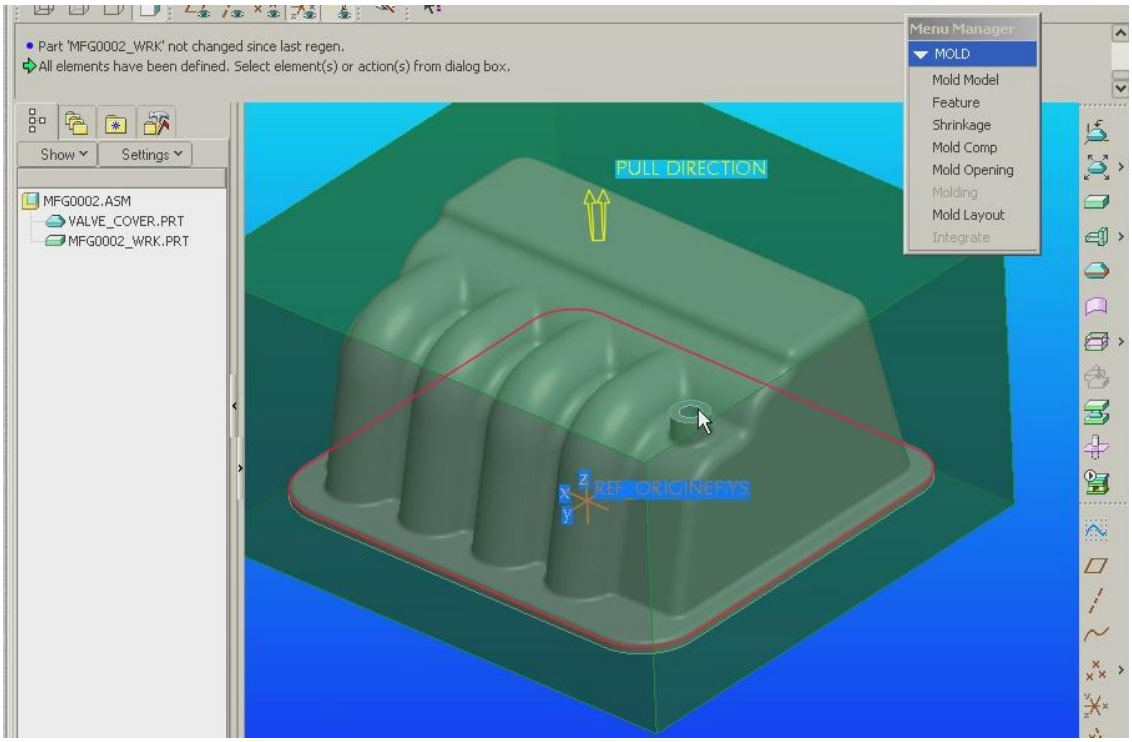
12. At this point you should now see a green transparent box around your work piece. We must not adjust the parameters as this is your stock to be machined.
13. Change the Offsets to 1.5 for all X, Y, and Z directions. Hit OK.



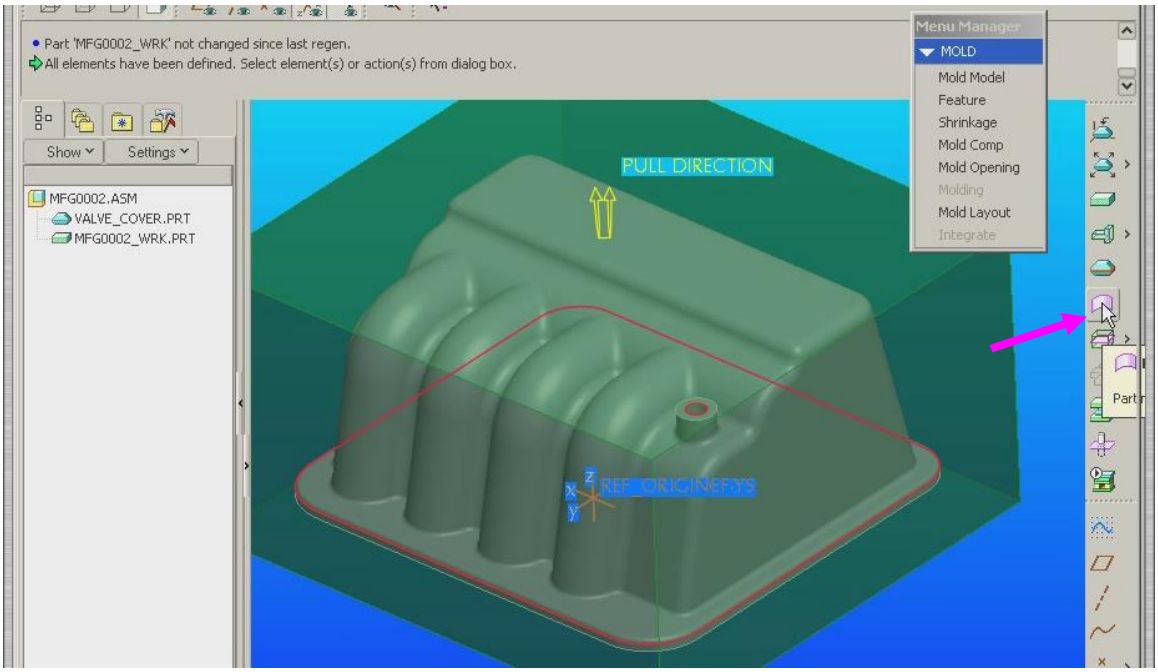
14. Select the “Silhouette” icon, and then select the top edge of the flange.



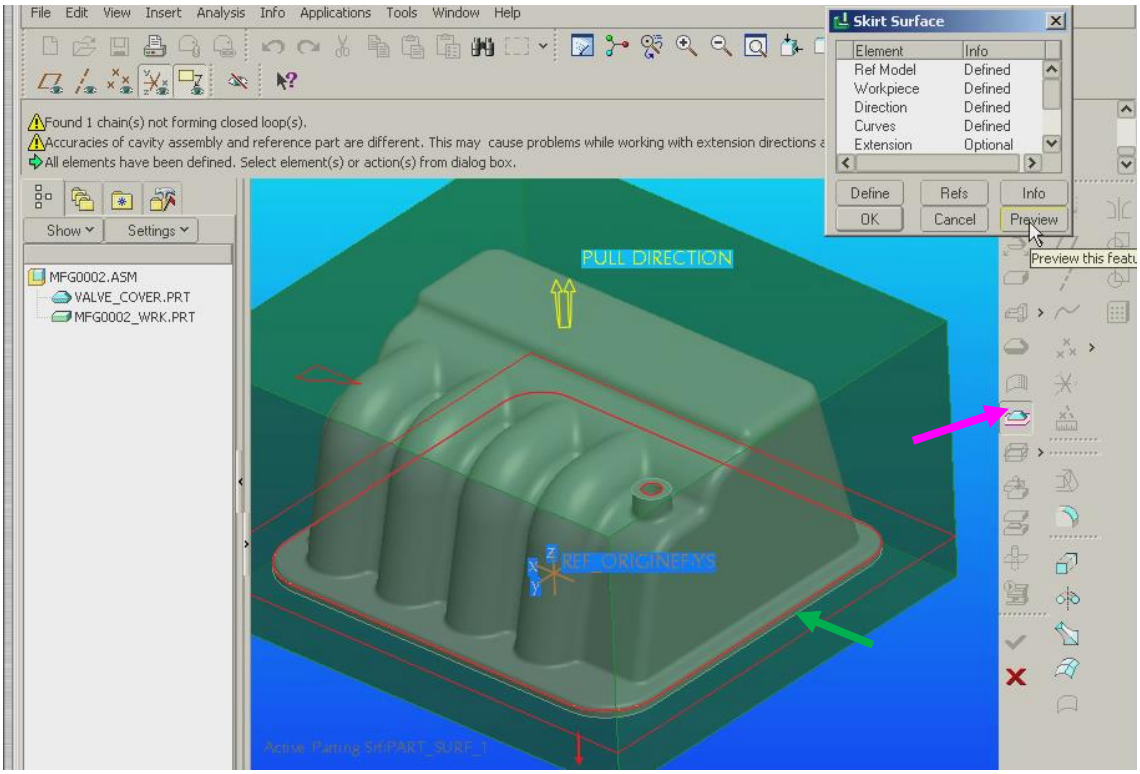
15. This generated parting line curves to be used to automatically generate surface shut-offs and boundary extensions to split the mold.



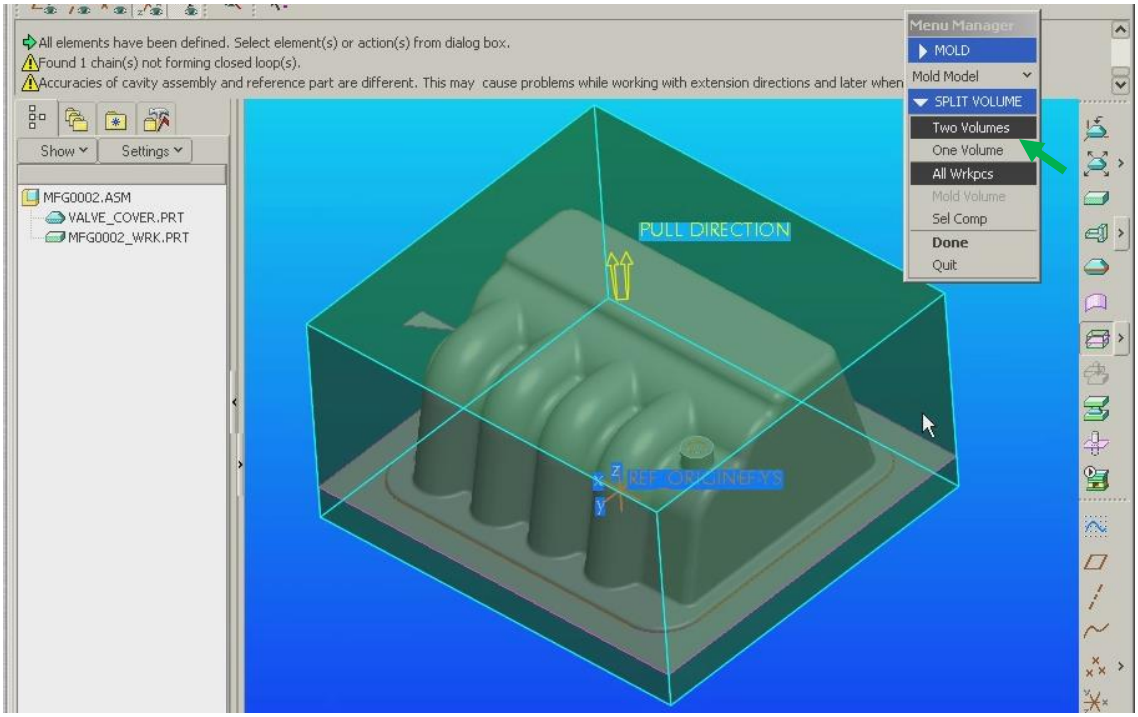
16. Select the parting surface icon.



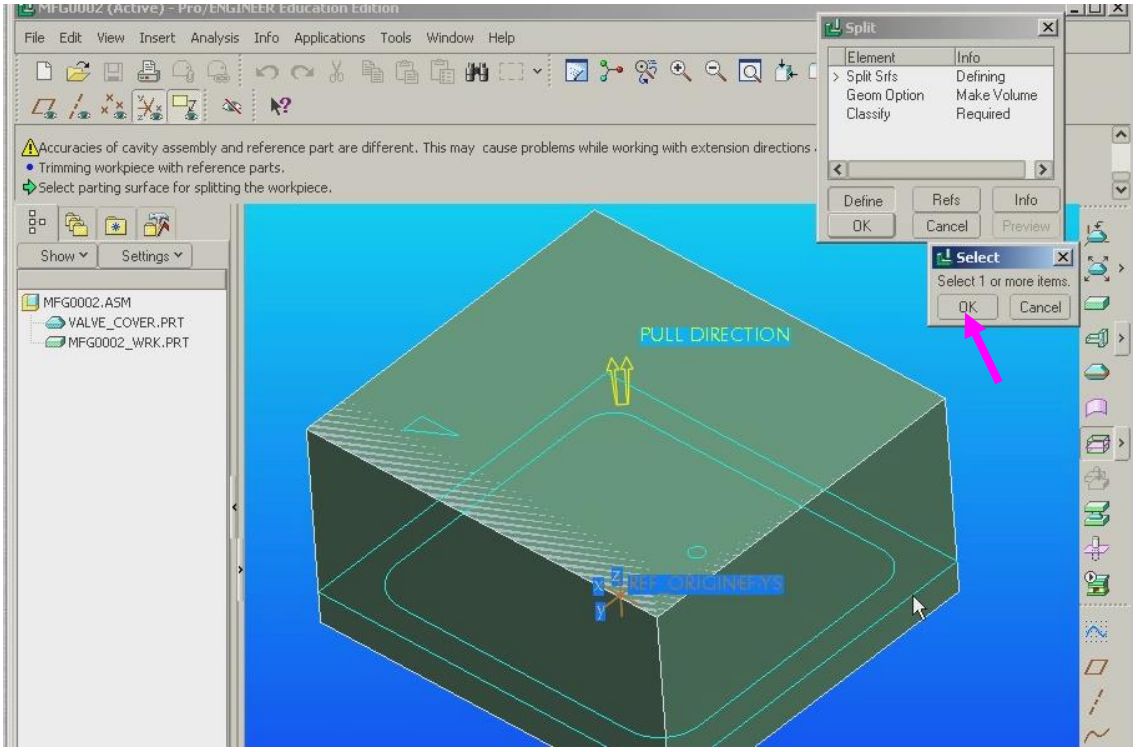
17. Select "**Skirt Surface**" and select the parting line **edge curve**, hit "preview" or "OK" to verify.



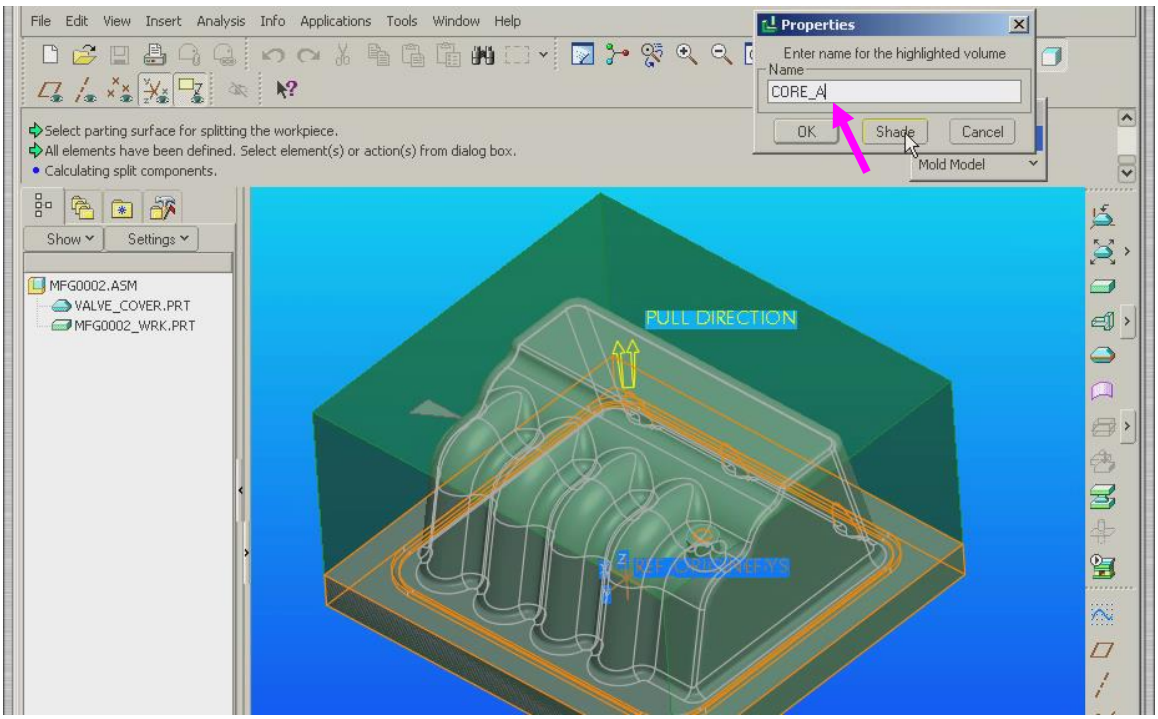
18. Select the "**Split Volume**" icon and make sure "**Two Volumes**" is selected.



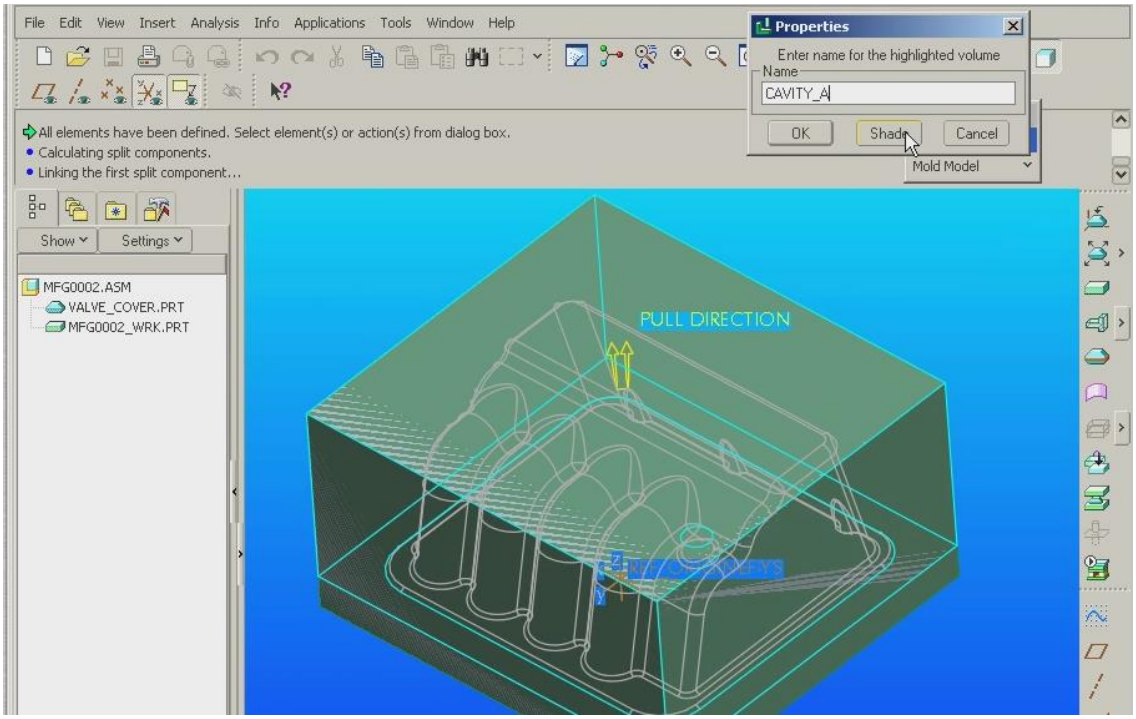
19. Select the skirt surface and hit “OK”.



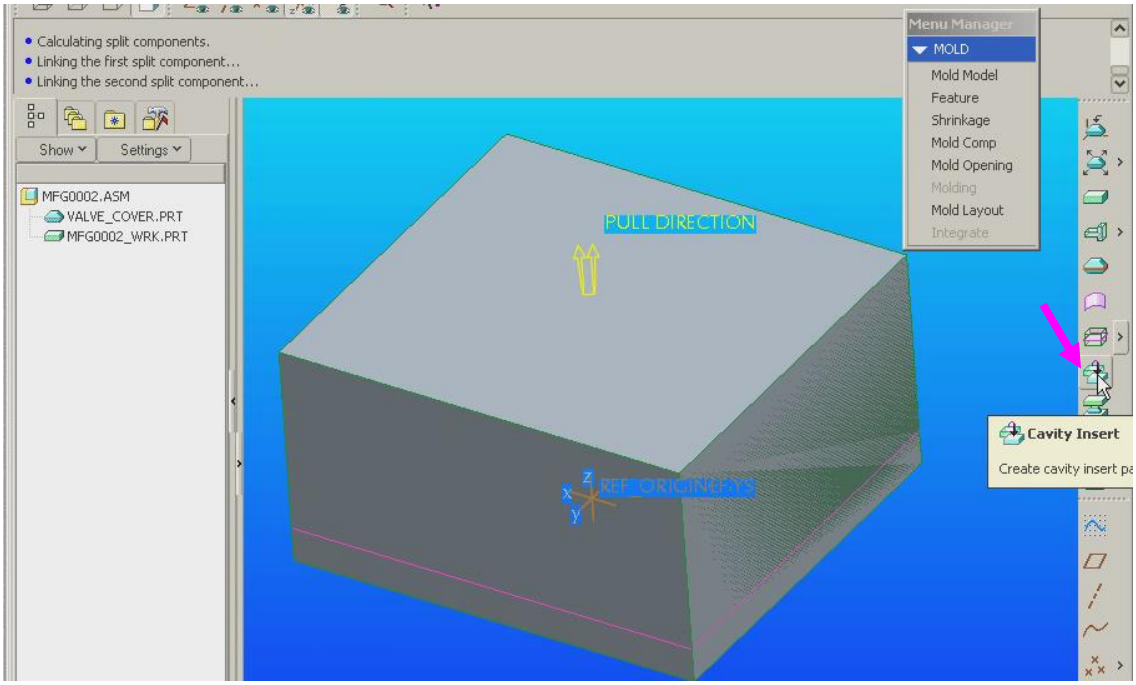
20. Enter a new name for the core, (CORE_A). Hit “OK”.



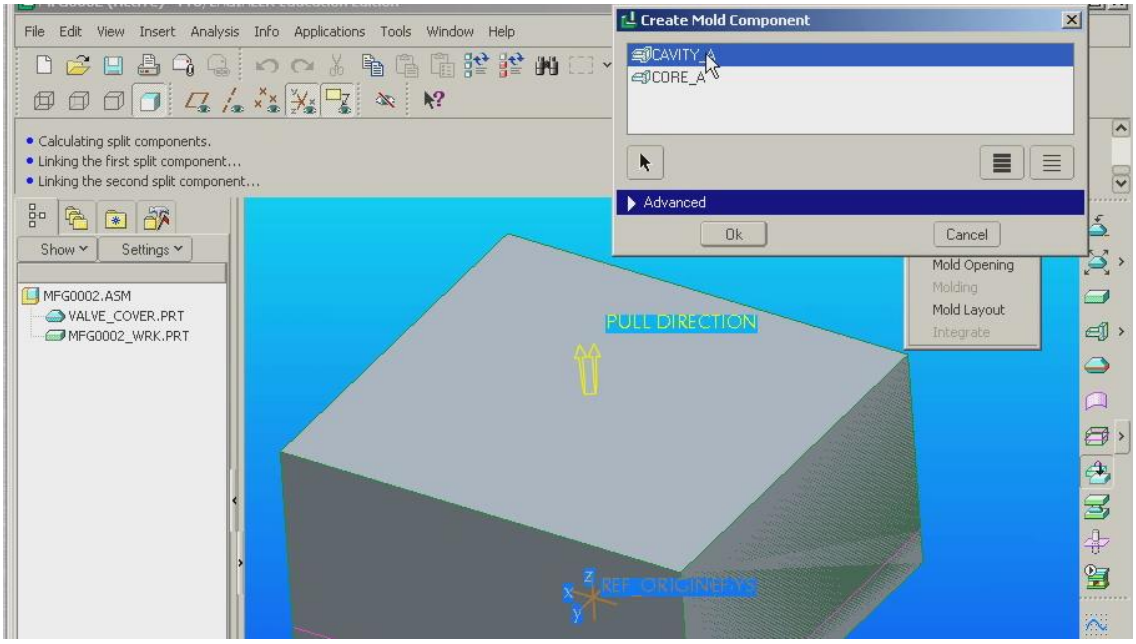
21. Enter (CAVITY_A) as the new name for the next section. “Hit



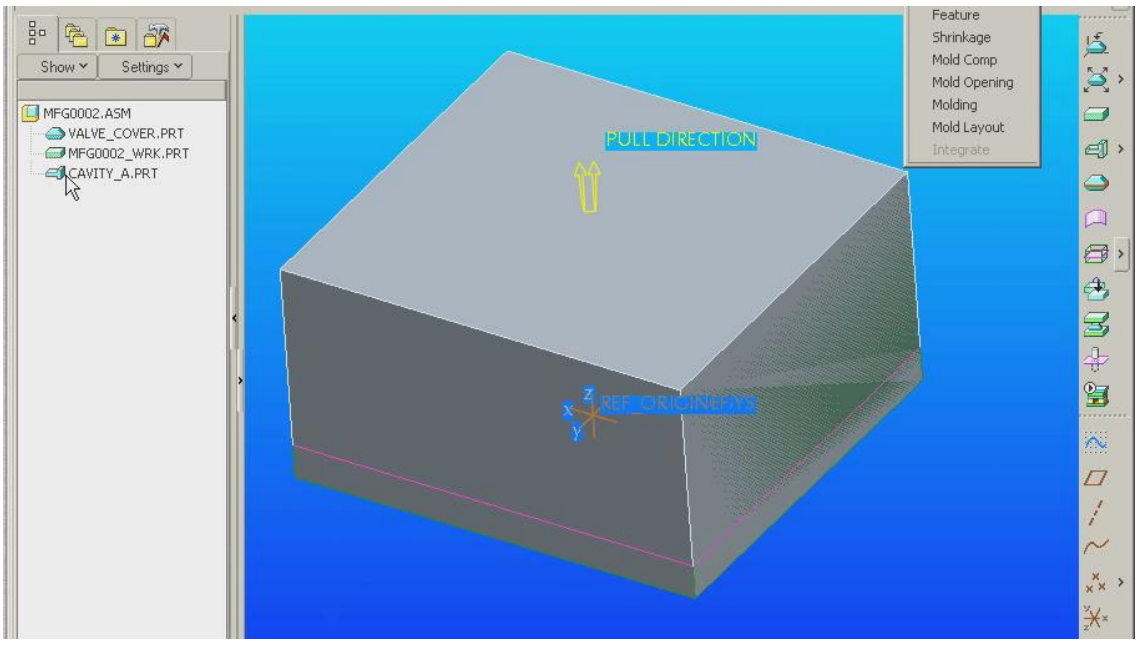
22. Select the “Cavity Insert” icon to create “official” cavity and core parts.



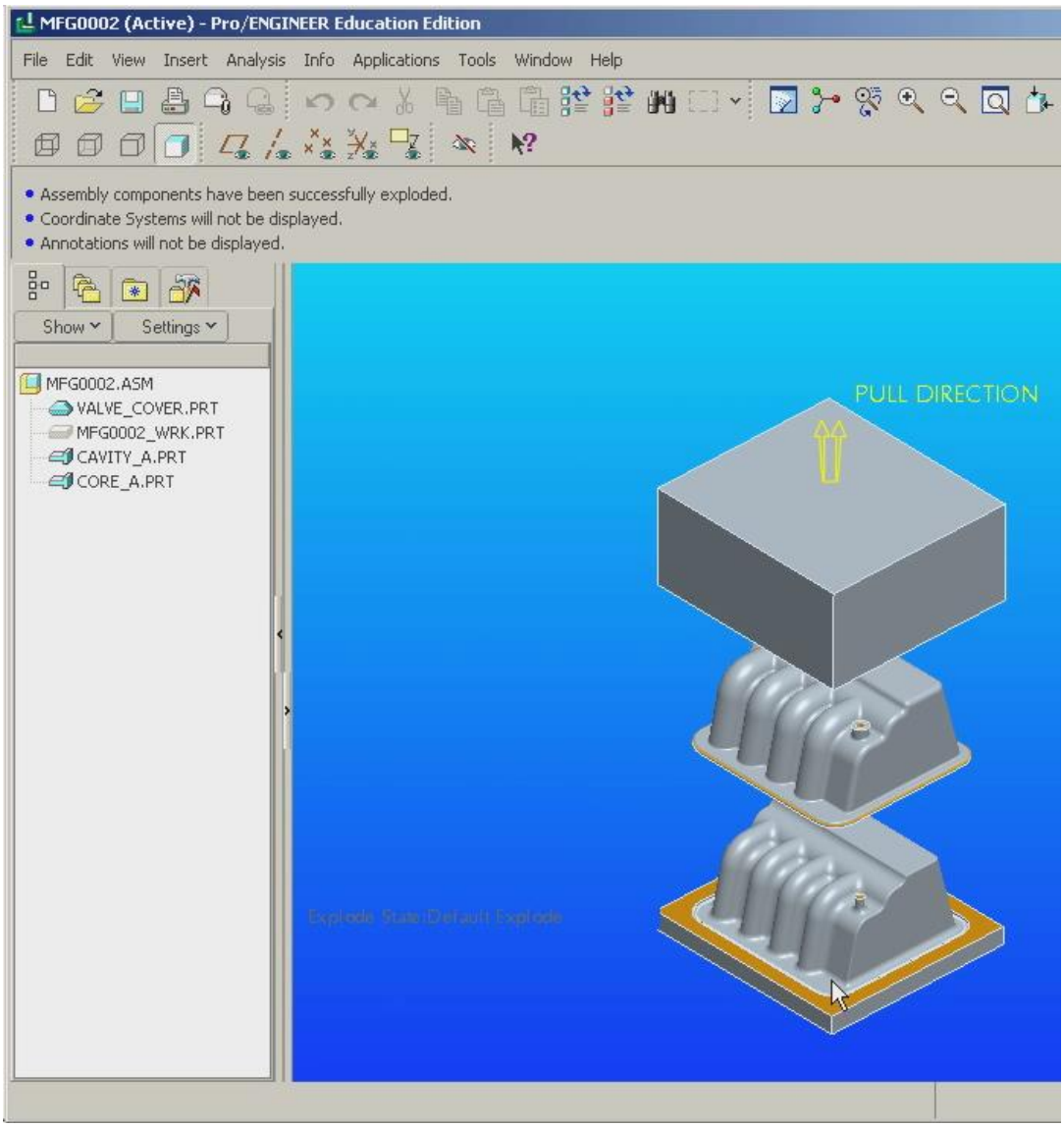
23. Select “CAVITY_A” from the list and hit “OK”.



24. Now “CAVITY_A” should appear in the feature tree as a part.



25. Repeat the same steps to create the “CORE_A” part.
26. Hide the MFG000x_WRK.PRT
27. Go to “View/Explode” to explode the assembly.
    Note: You can left click then right click on the skirt surface to find hide as an option.

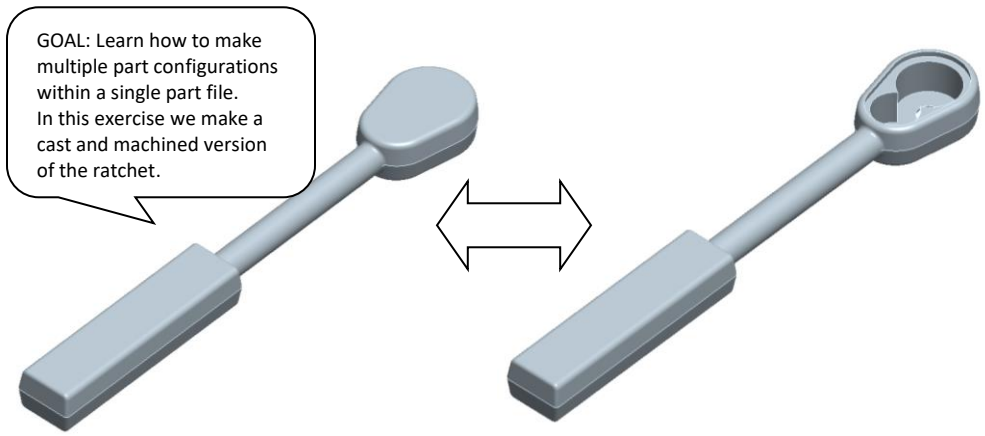


FINISHED

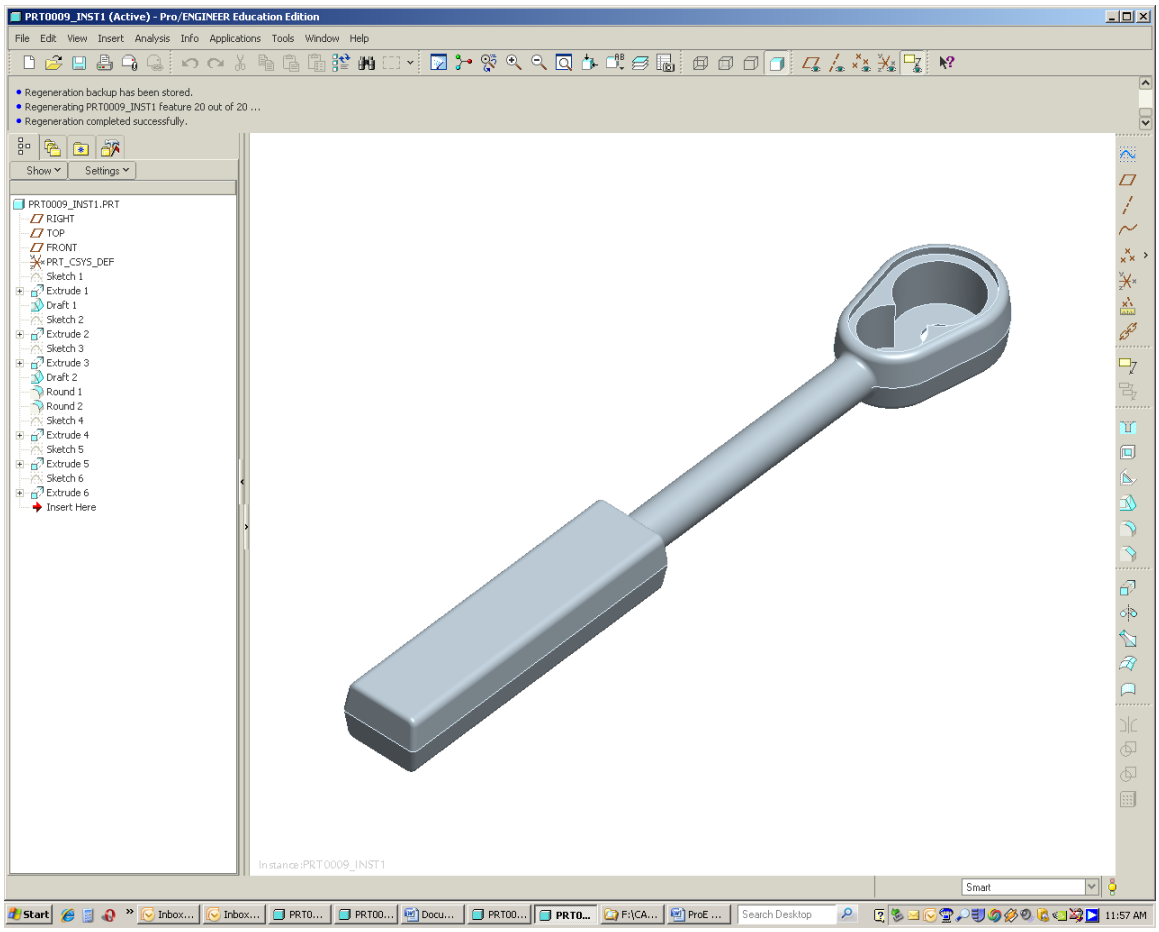
EXERCISE 16

Family Tables

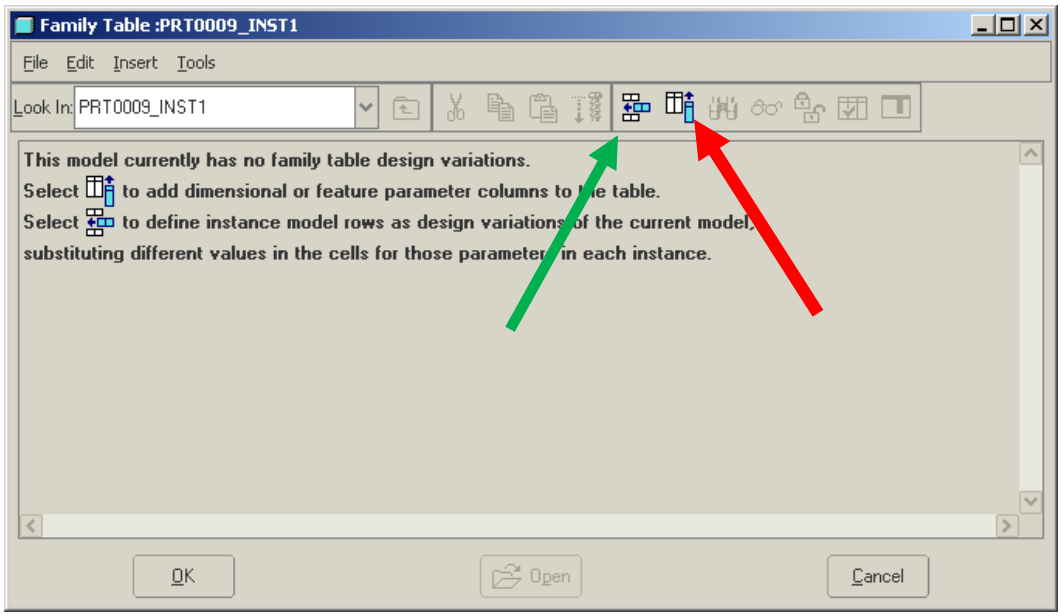
Family Tables enable you to create multiple part configurations derived from a single part file.



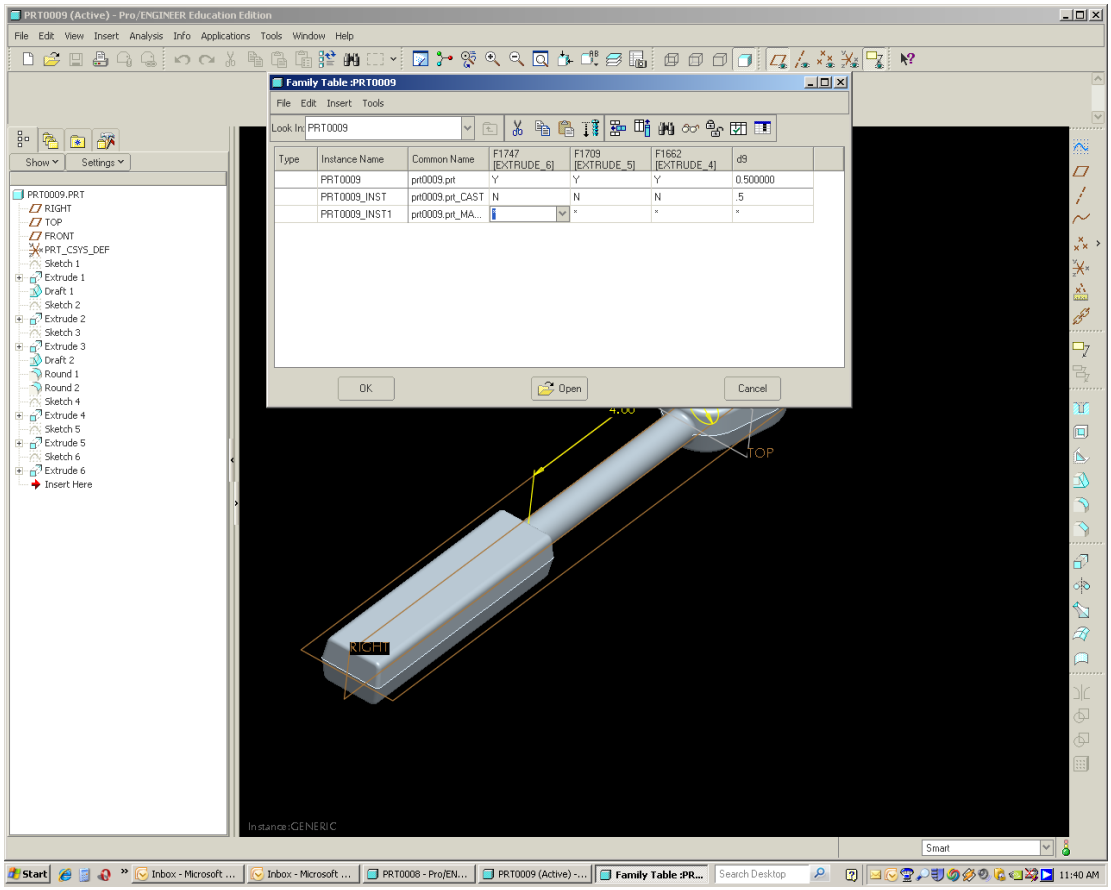
1. Open the Exercise_16 part file.



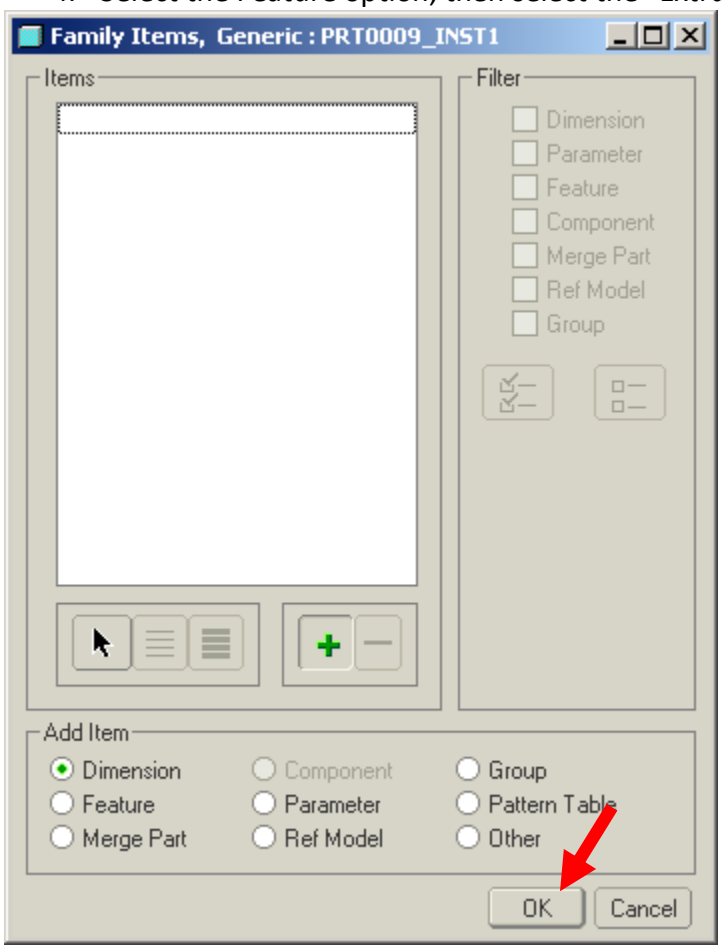
2. Go to the pull down menu- “Tools/Family Tables”



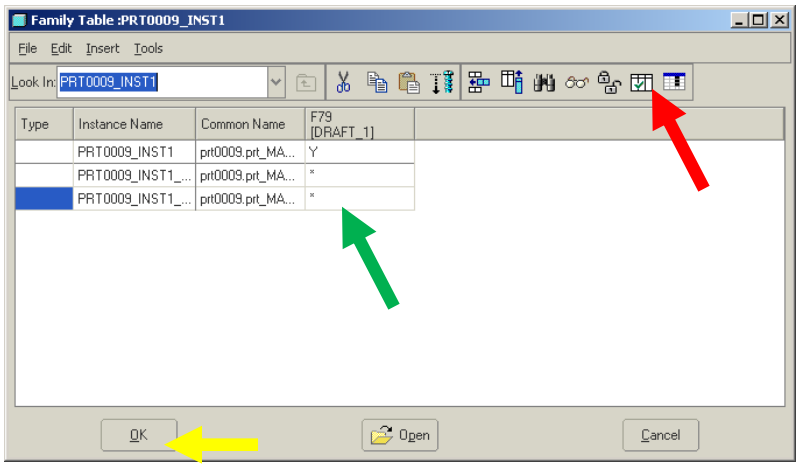
3. Select the **“Insert new insatnce”** two times. Then hit the “Add…” icon.



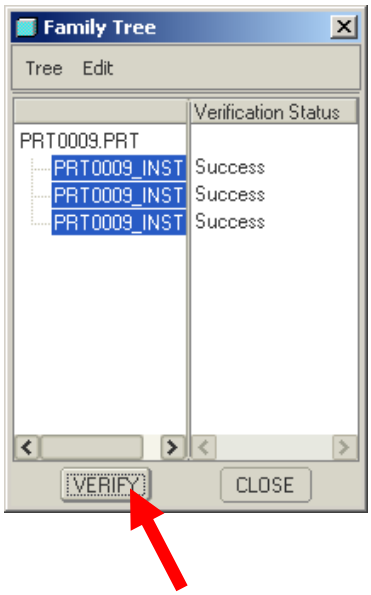
4. Select the Feature option, then select the “Extrude 4, 5, and 6”



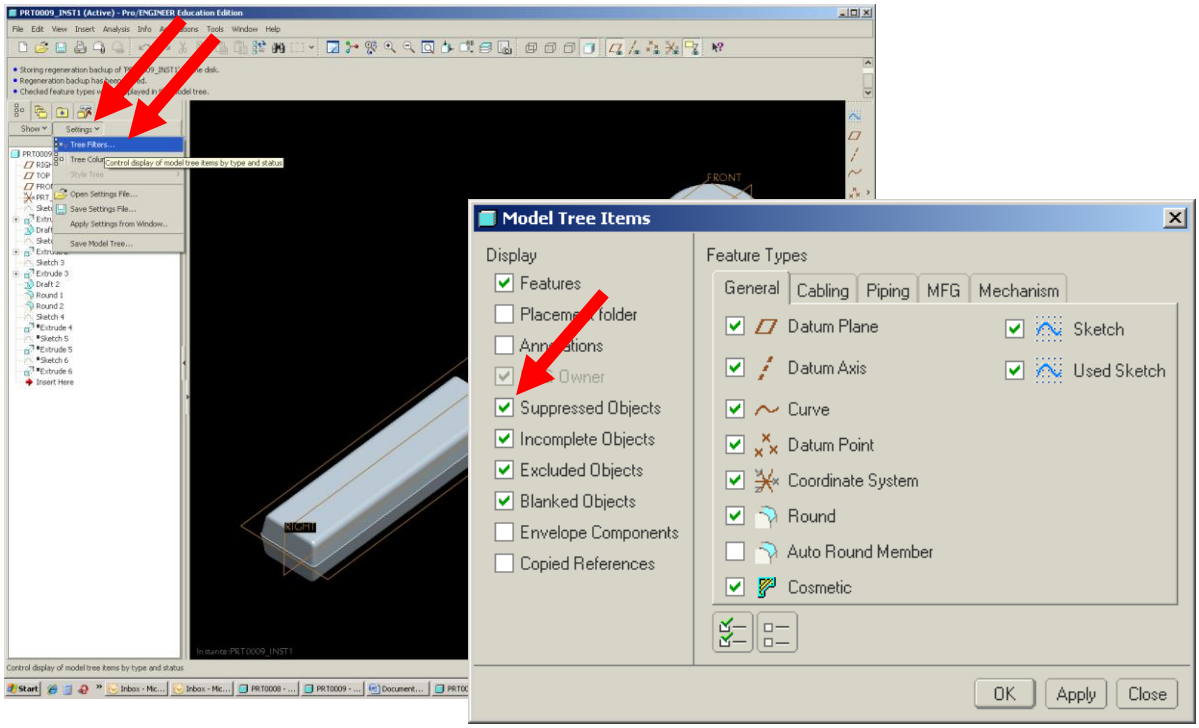
5. Select “OK”.
6. Select “**Verify**”
7. In the columns type “**N**” for no- to supress the feature, or “**Y**” for yes for the feature to be unsupressed. Hit “OK”.



8. Hit “Verify” once again on the smaller Family Tree box.

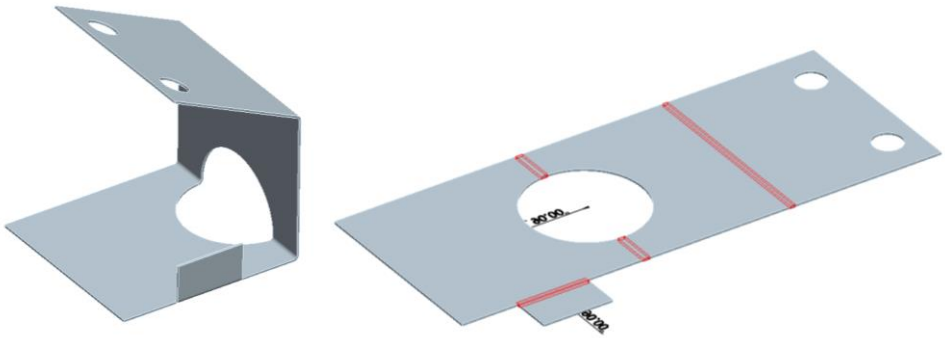


9. To view suppressed features on the tree select settings then Model Tree items.

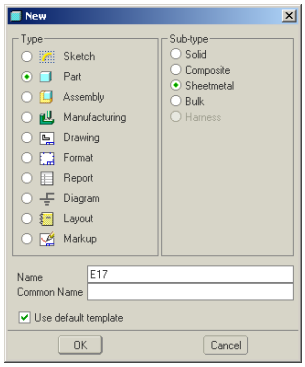


10. To open the additional instances go to File/Open, and select the original file, when it opens it will prompt you with a list of Family Parts available. FIN

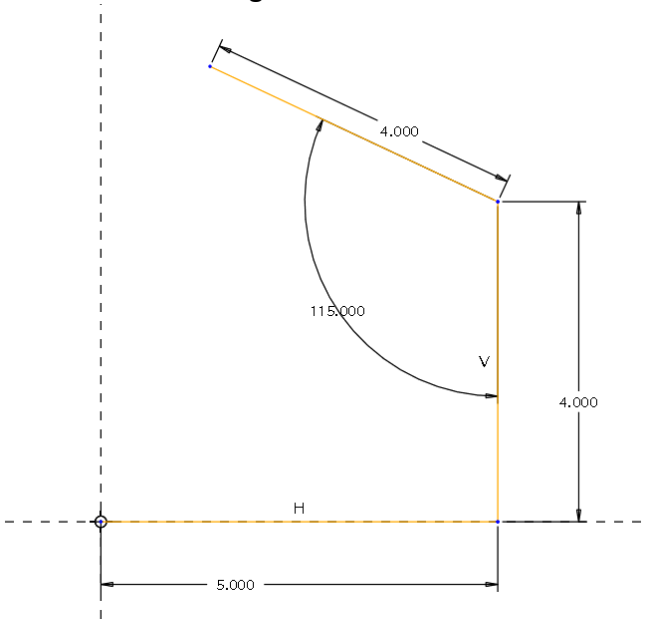
EXERCISE 17
Sheet Metal Fabrication



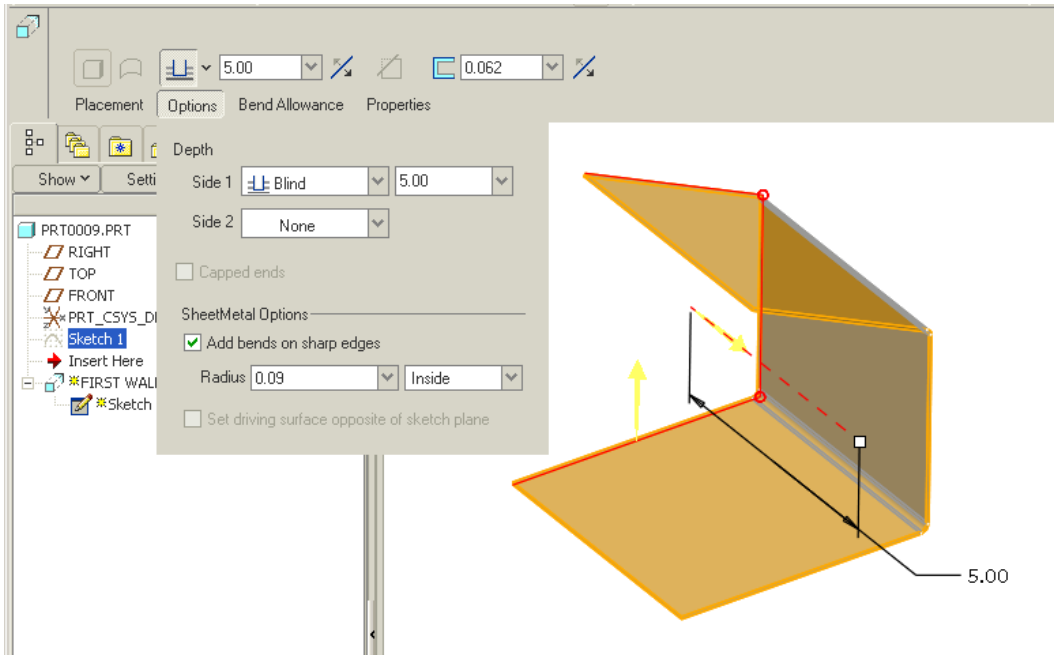
1. Start a new part file and select “Sheet Metal”.



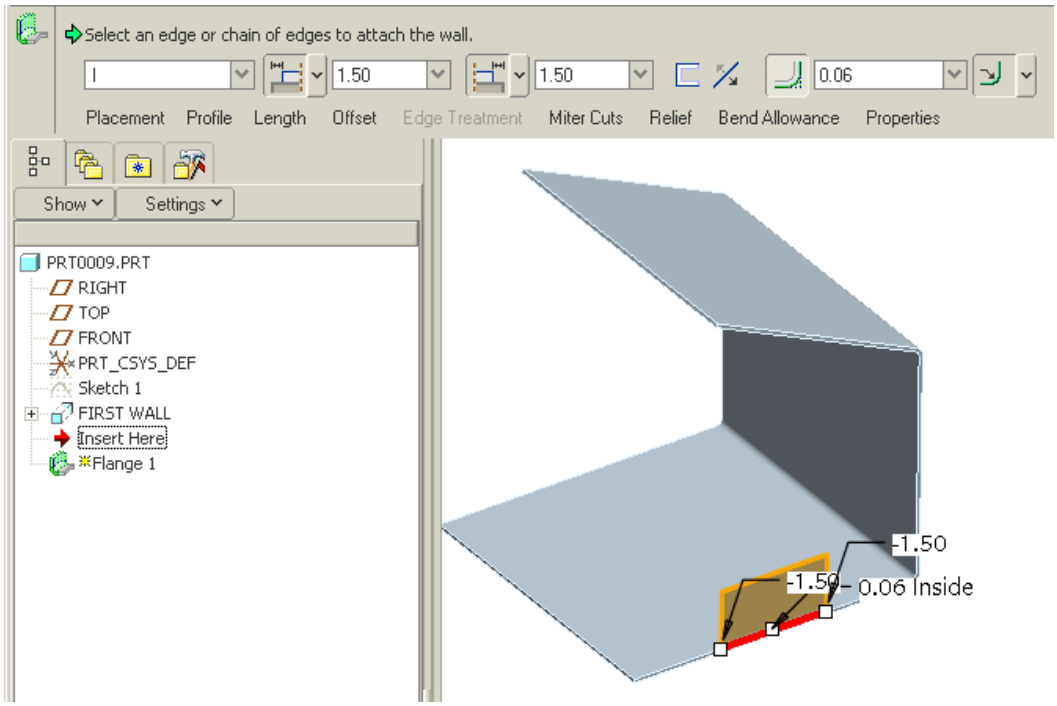
2. Sketch the following on the Front Plane

3. 

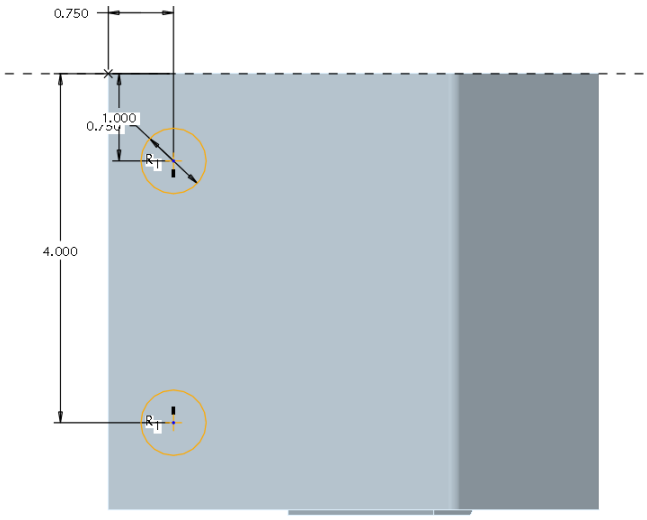
4. Select the Options Tab and fill in as shown.



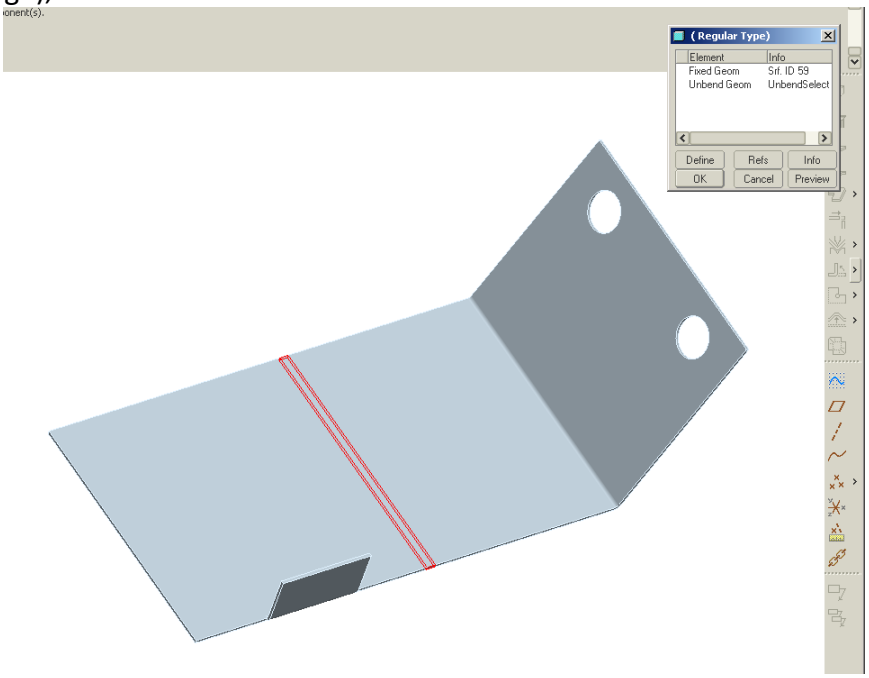
5. Add a tab, select profile and enter 1” length.



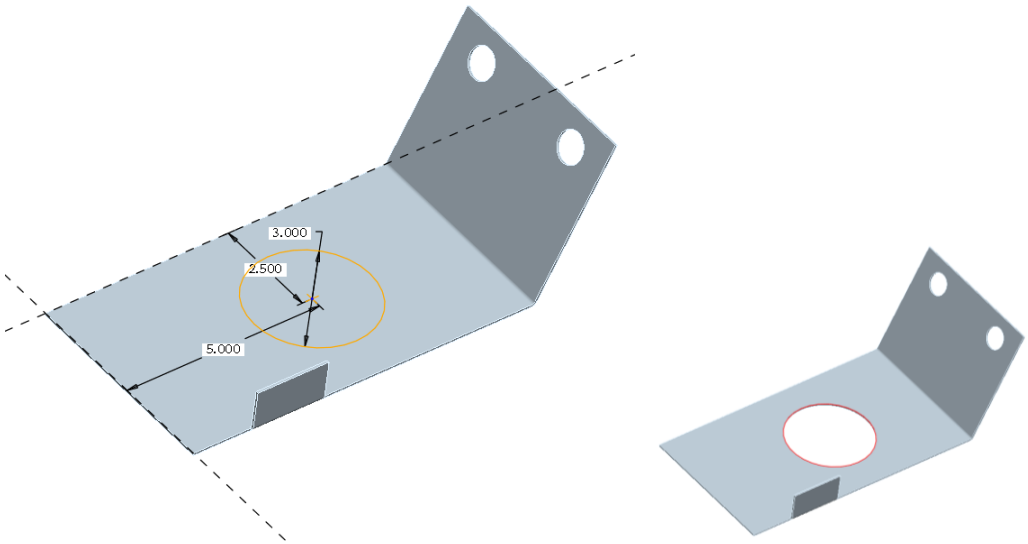
6. Start a sketch on the top flange and draw the following.



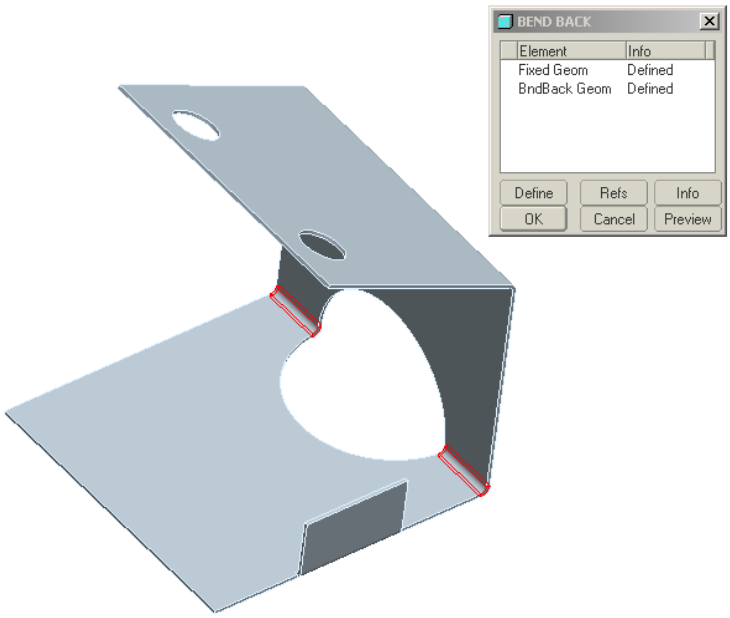
7. Unbend a bend using the “unbend tool” Select the Fixed face (bottom surface of flange), then select the actual bend surface.



8. Start a sketch on the bottom face and draw the following circle. Extrude Cut.

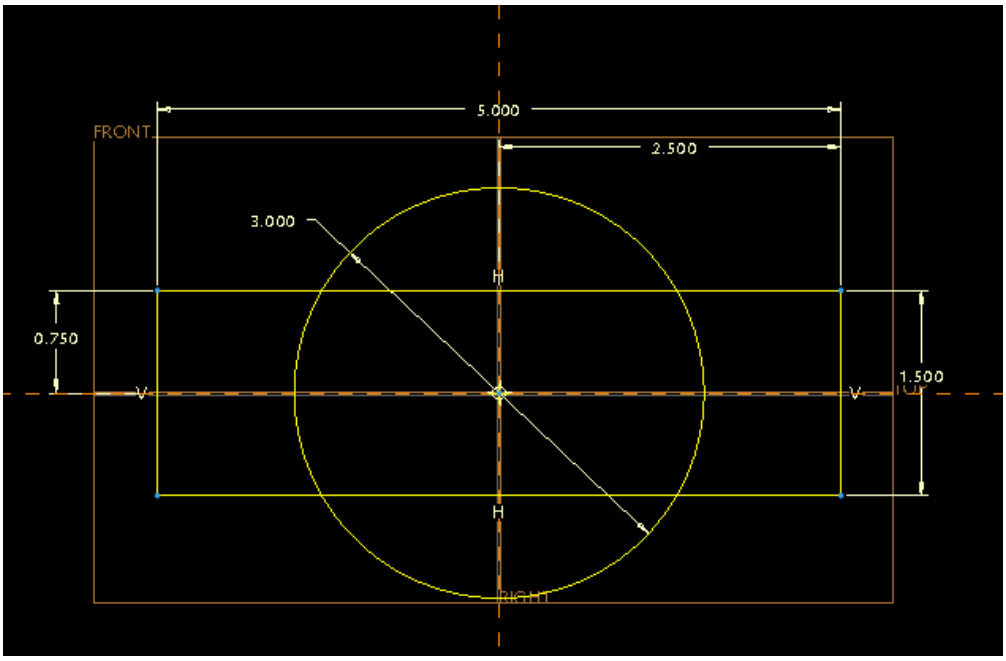


9. Use Bend Back.

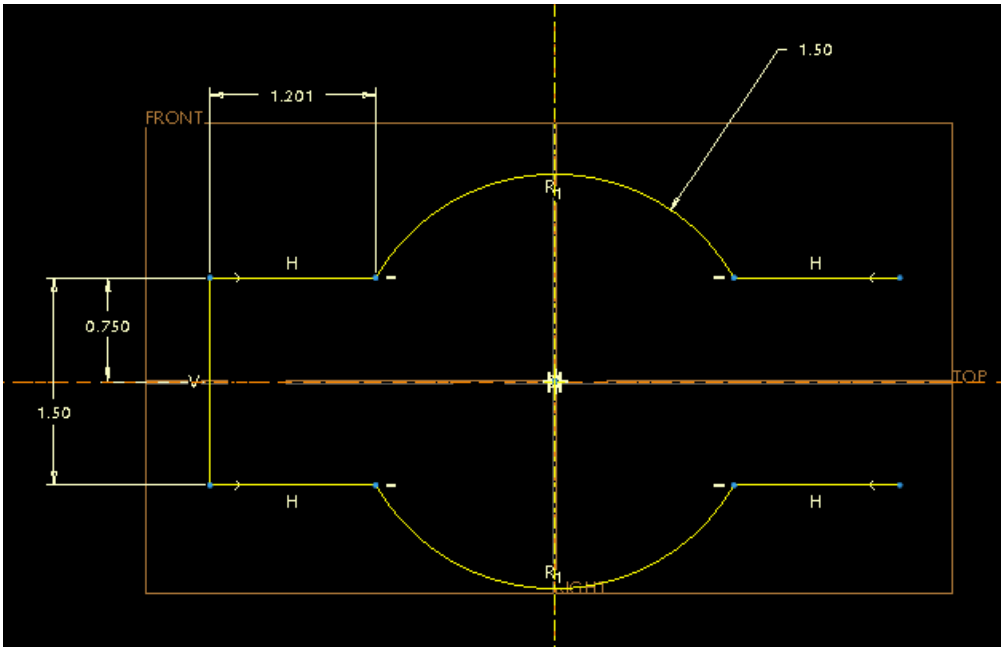


LAB 17

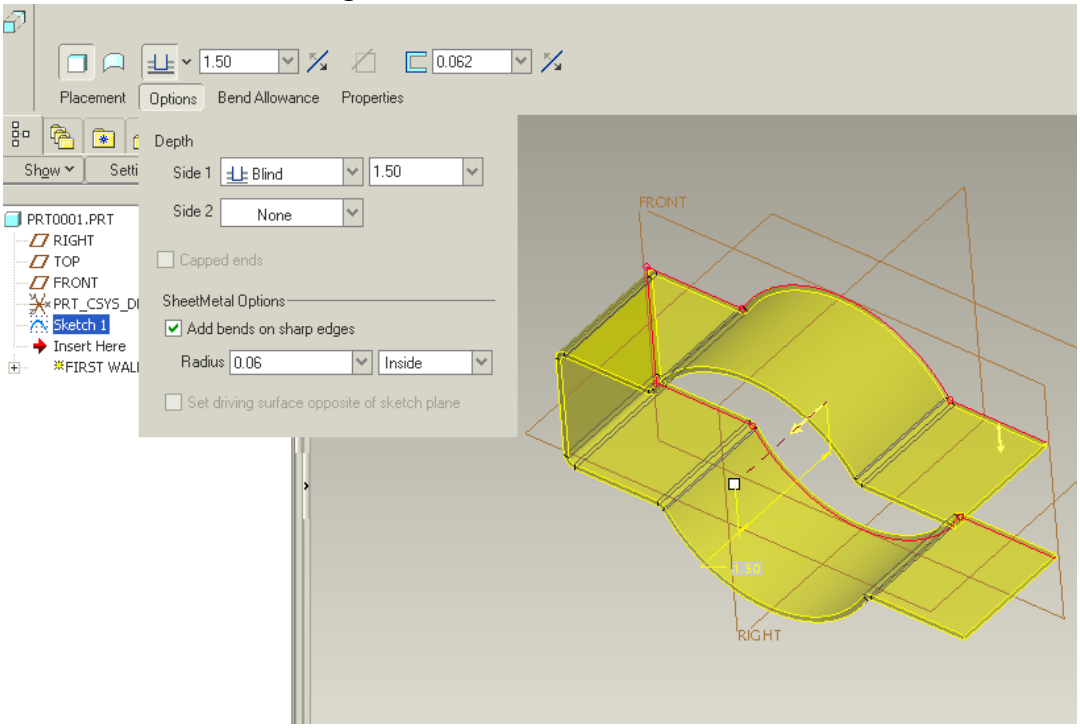
1. Start a new Sheet Metal part, and draw the following on the front plane.



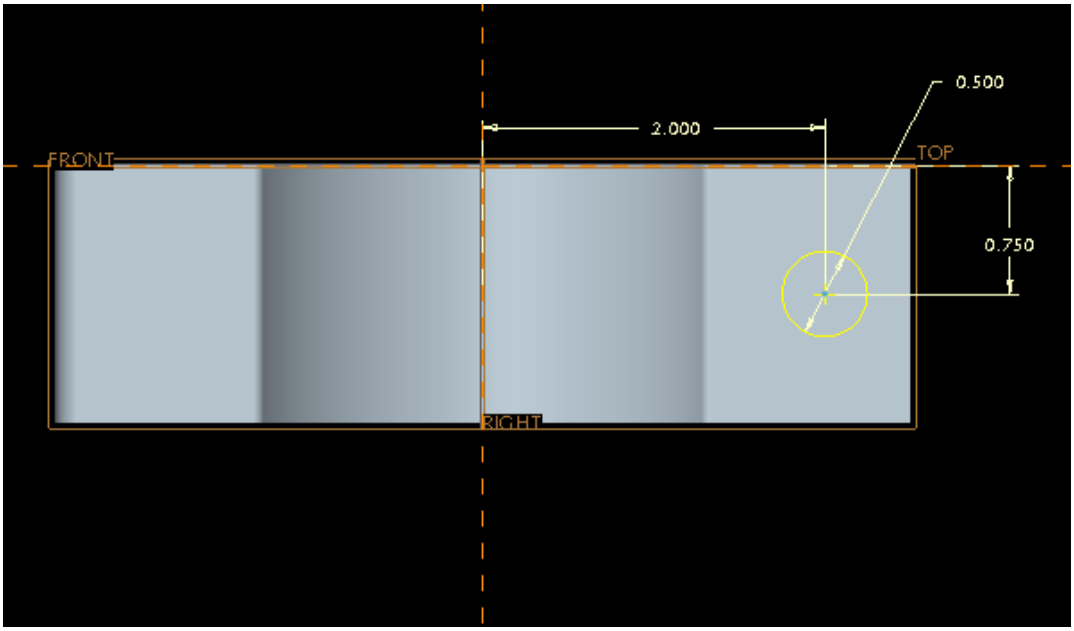
2. Trim



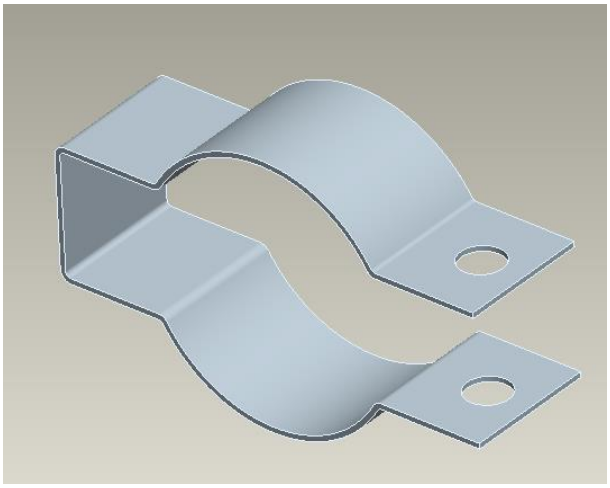
3. Extrude and use settings as seen below.



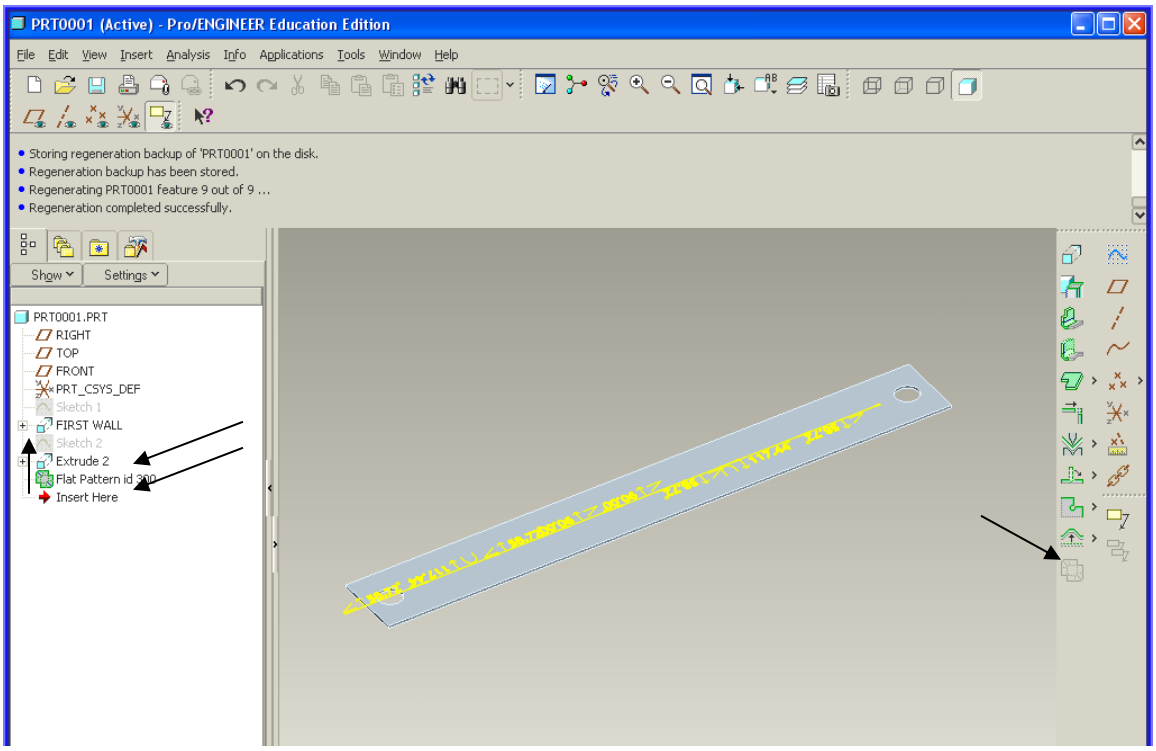
4. Start a sketch on the top surface of the flange, and draw the following.



5. Done



6. Select the Flatten tool and click on a fixed face of the model, then hit done. To Refold Drag the Red “Insert here” arrow above the Flat Feature in the tree.

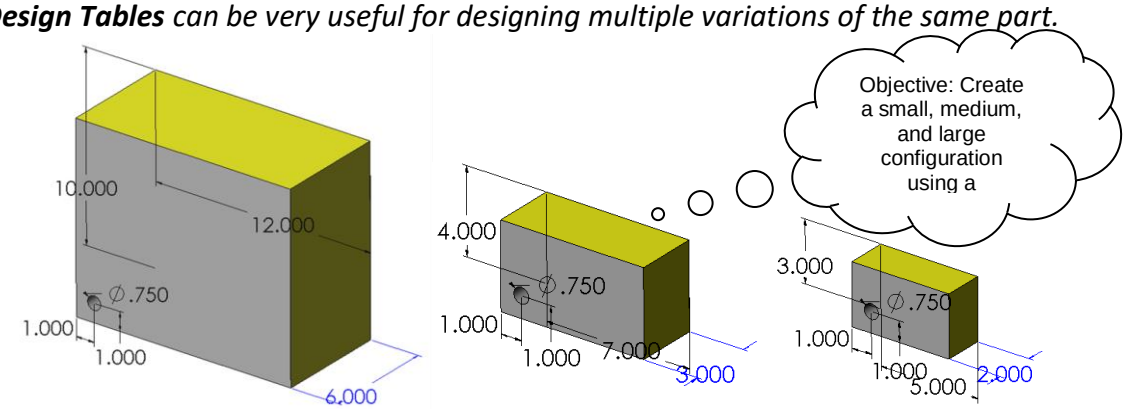


FINISHED

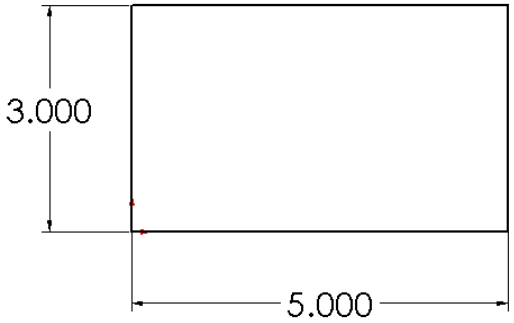
EXERCISE 18

Configurations with Family Tables

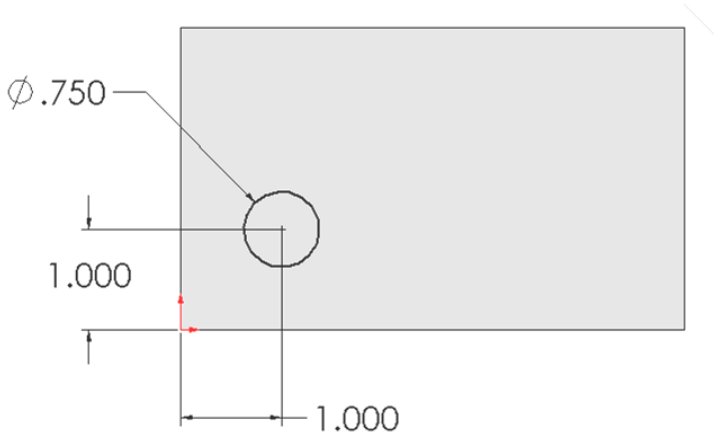
Design Tables *can be very useful for designing multiple variations of the same part.*



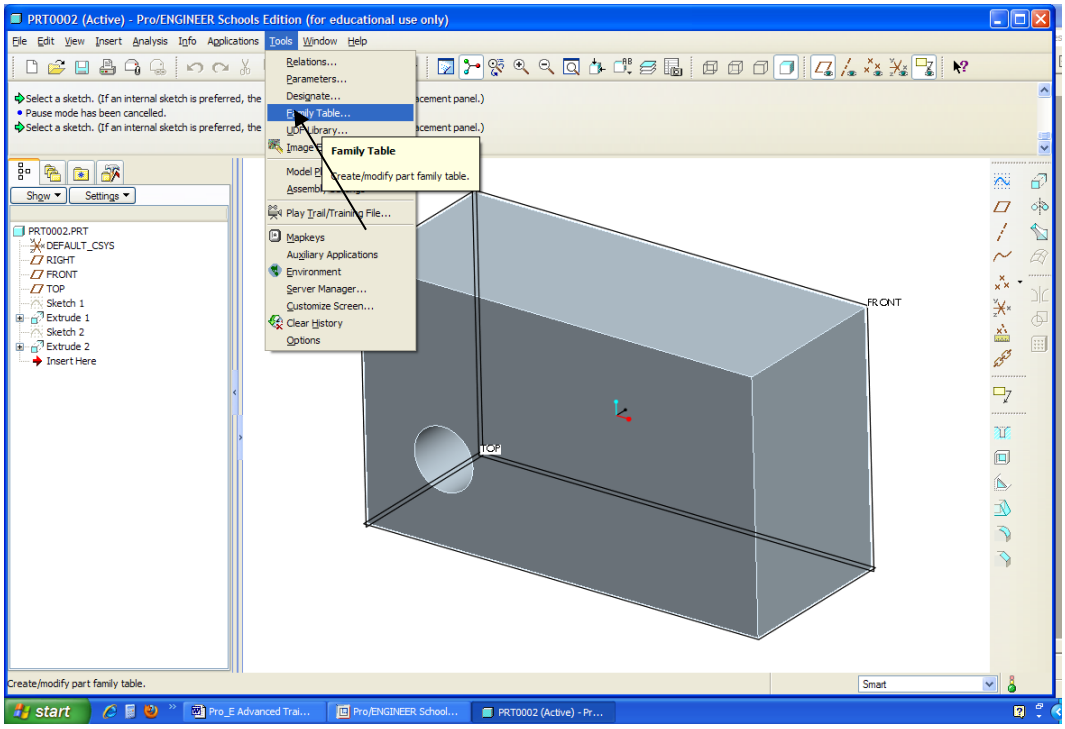
1. Sketch the following on the "Front" plane. Extrude 2 inches.



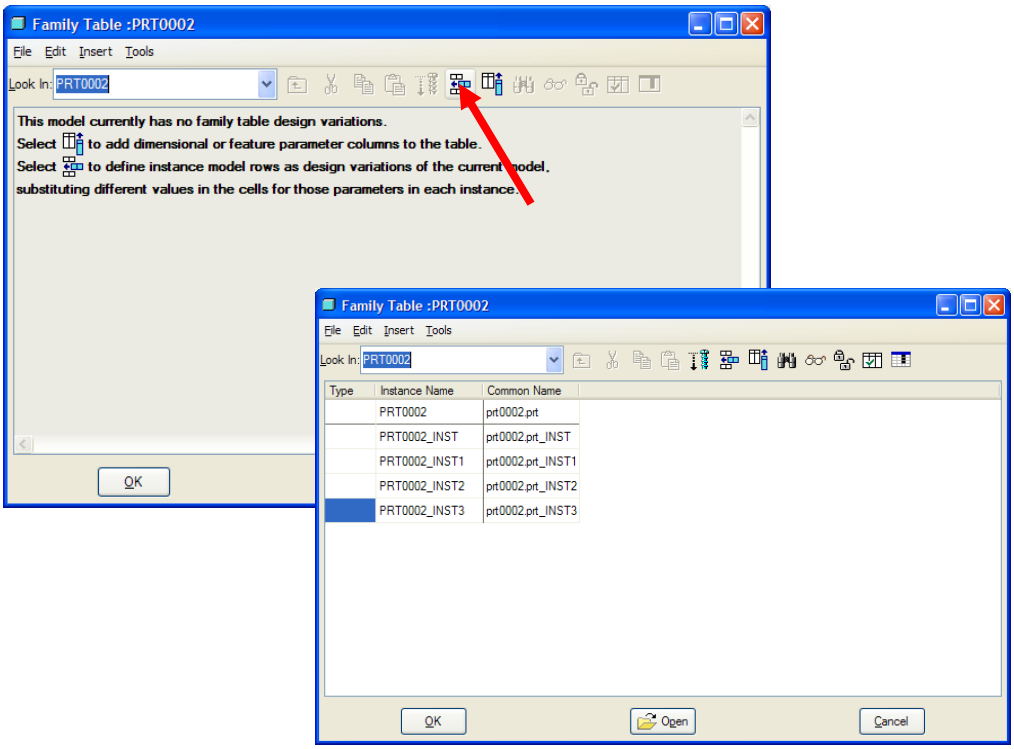
2. Start a new sketch the circle and extrude cut through all. (*Note: This hole needs to be a separate extrude feature or the exercise will fail to work correctly*)



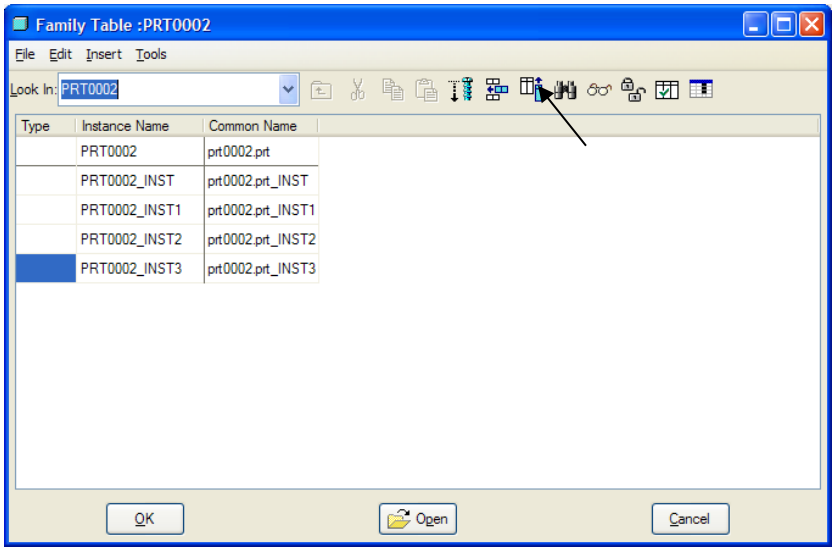
3. Click on the “Tools/Family Table” option.



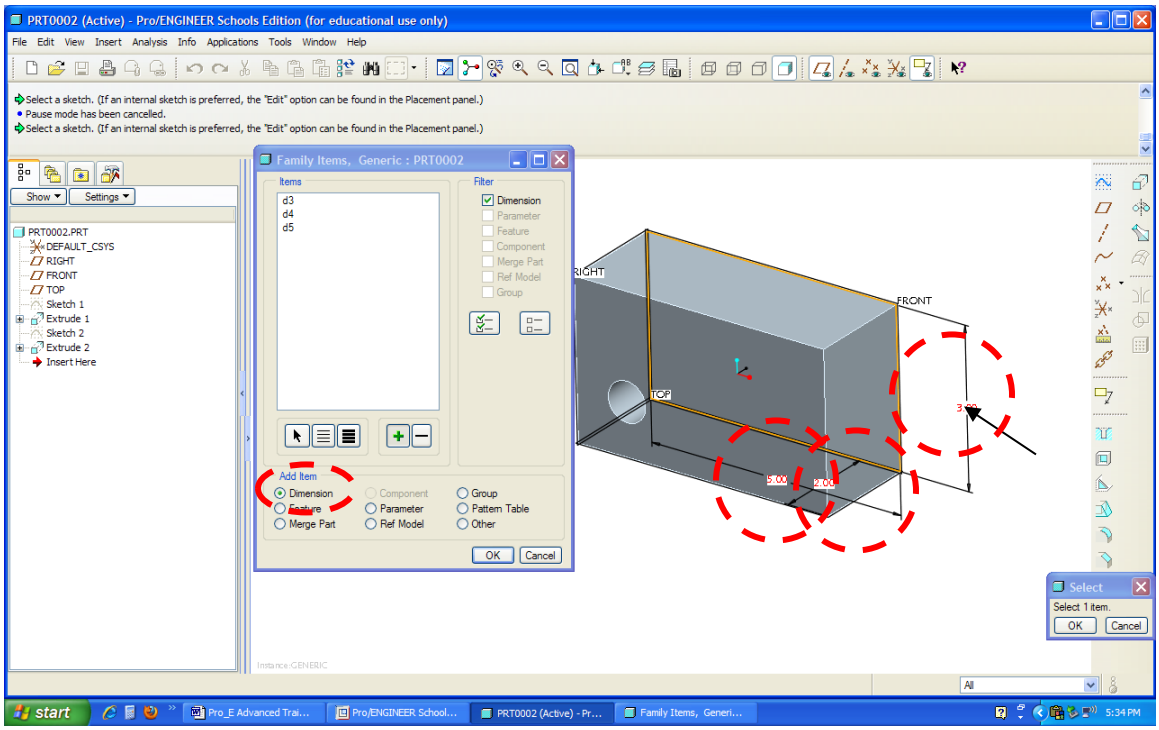
4. Select “Insert new instance” four times.



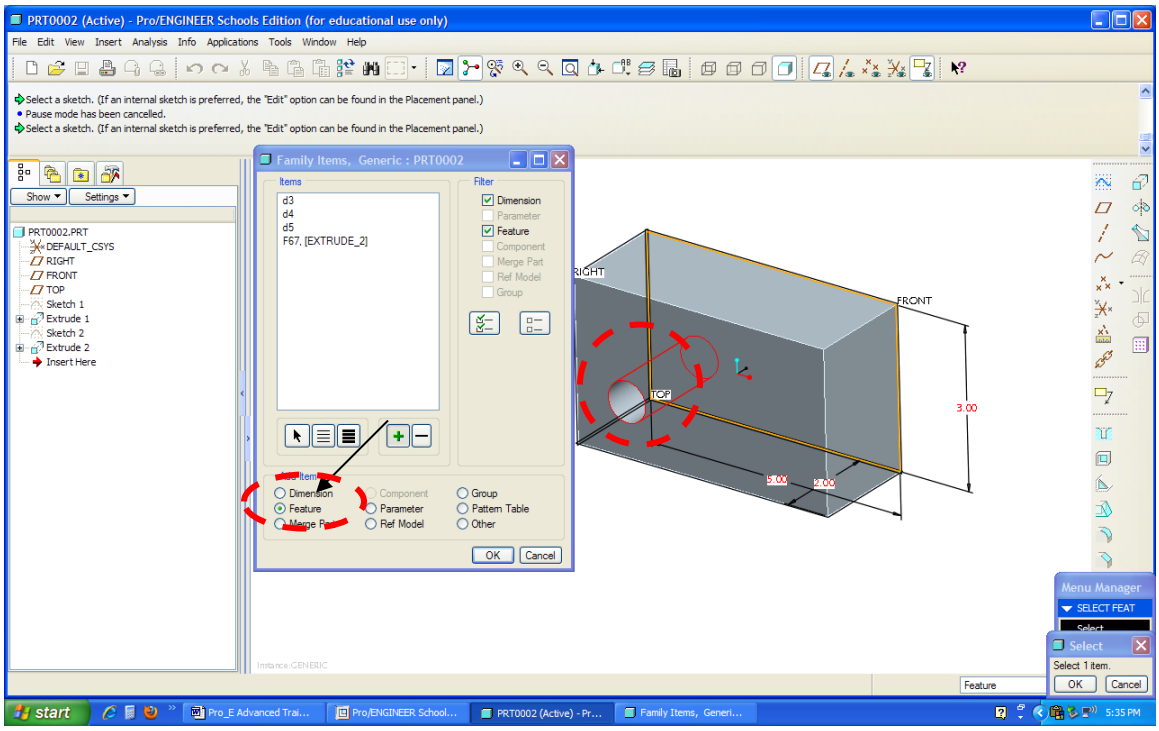
5. Select “Add/Delete” table columns”.



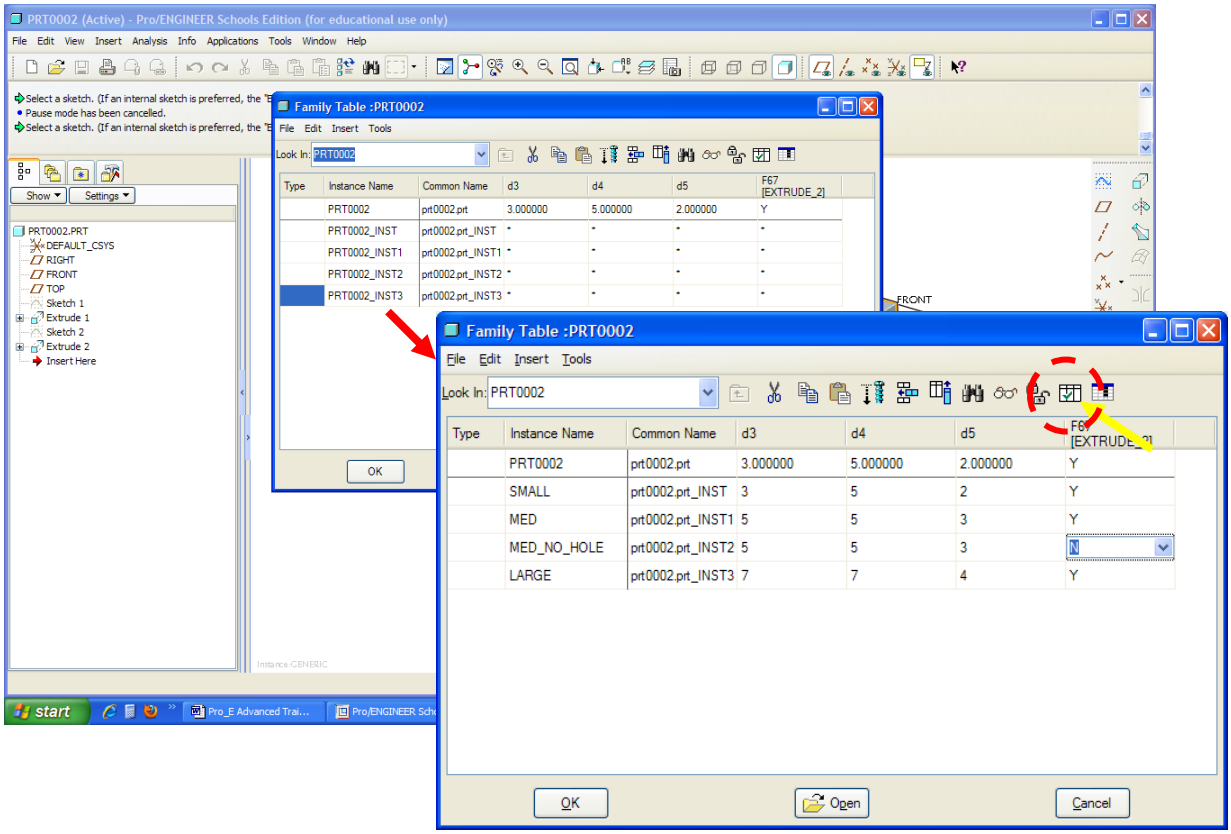
6. Select Dimensions then click select any model surface, the dimensions should appear, next, click on the 3”, 5”, and 2” dimensions on the model.



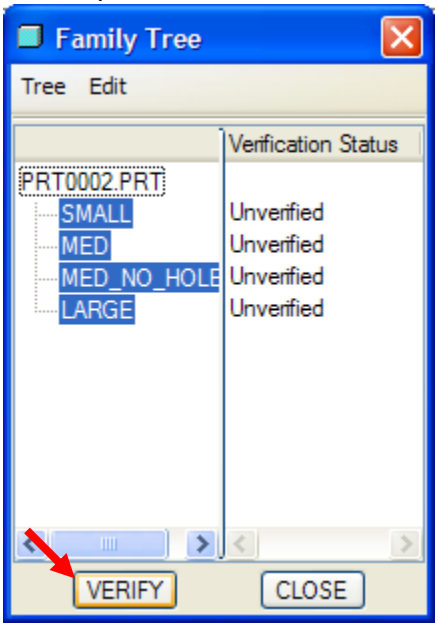
7. Select the “Feature” option, and continue to select the inside surface of the hole, not the edge of the hole. Hit “OK”



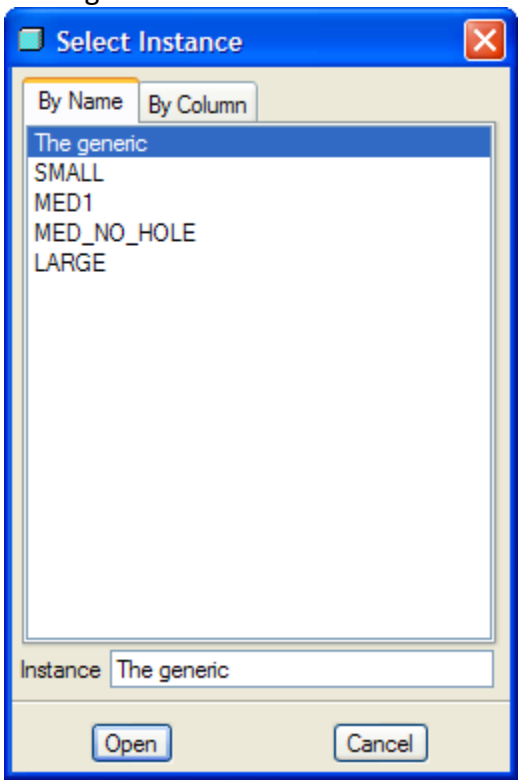
8. Now begin to insert the specifics into the table. Once complete hit “Verify”



9. Click on “Verify” to complete and then close.



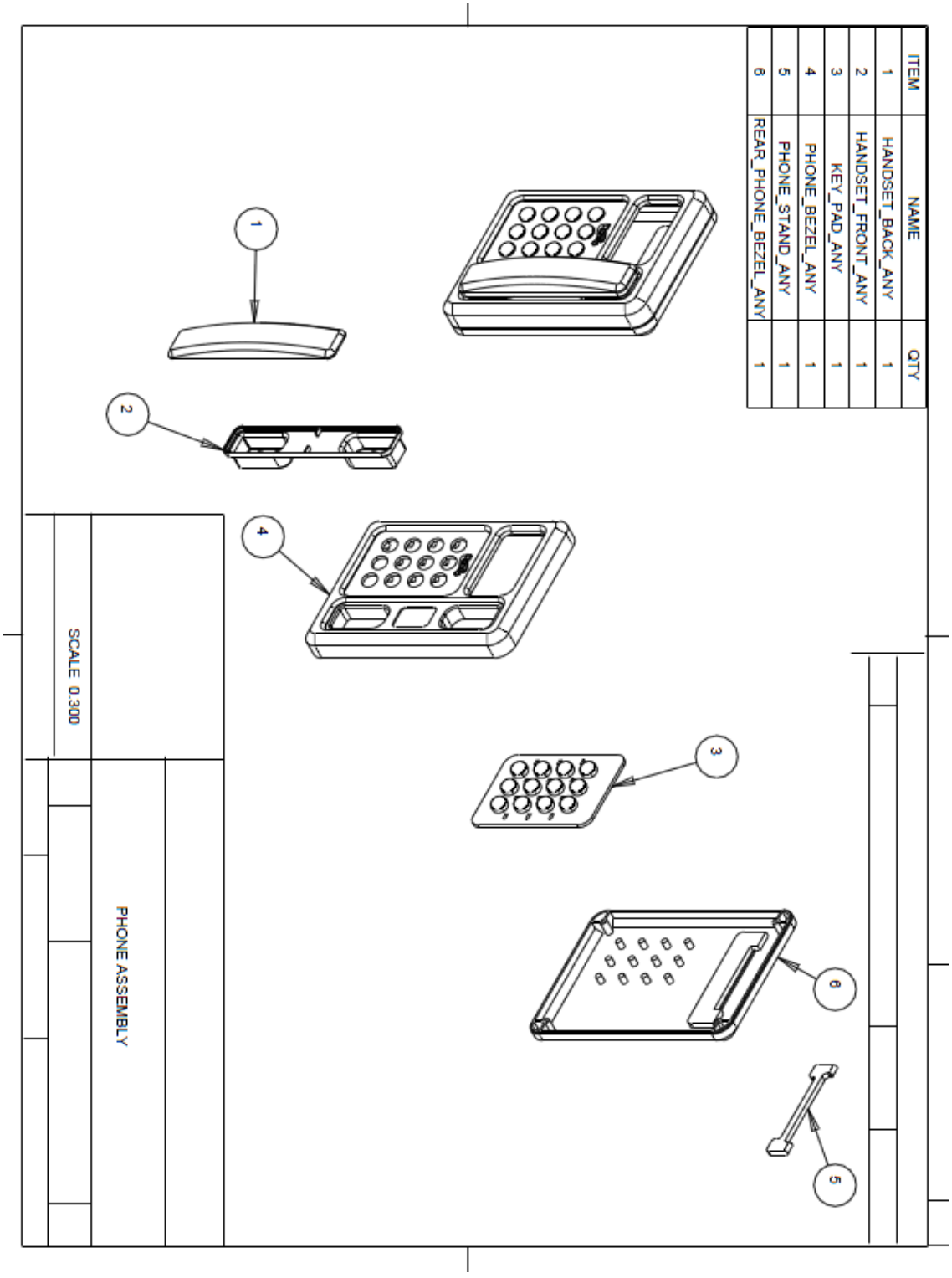
10. Now go to File/Open and select the E16 part, you should get this option. You are finished. (If you open any of the selected file options you will get the block with the changes added to the model in the table.)



LAB 18

Phone Assembly and Drawing

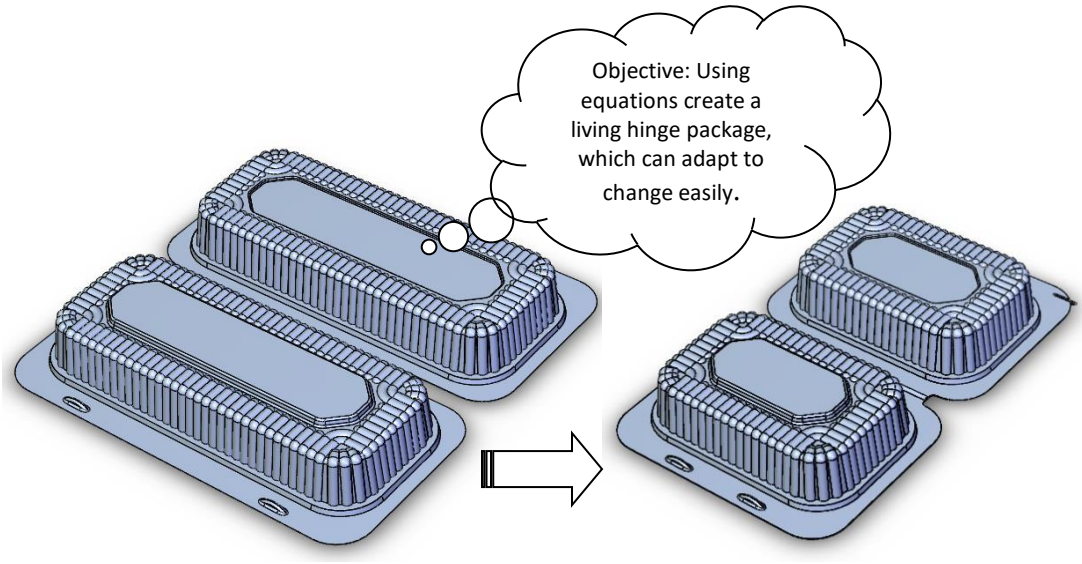
1. Take the Phone Lab parts built from the DWG files and assemble them and make the drawing below.



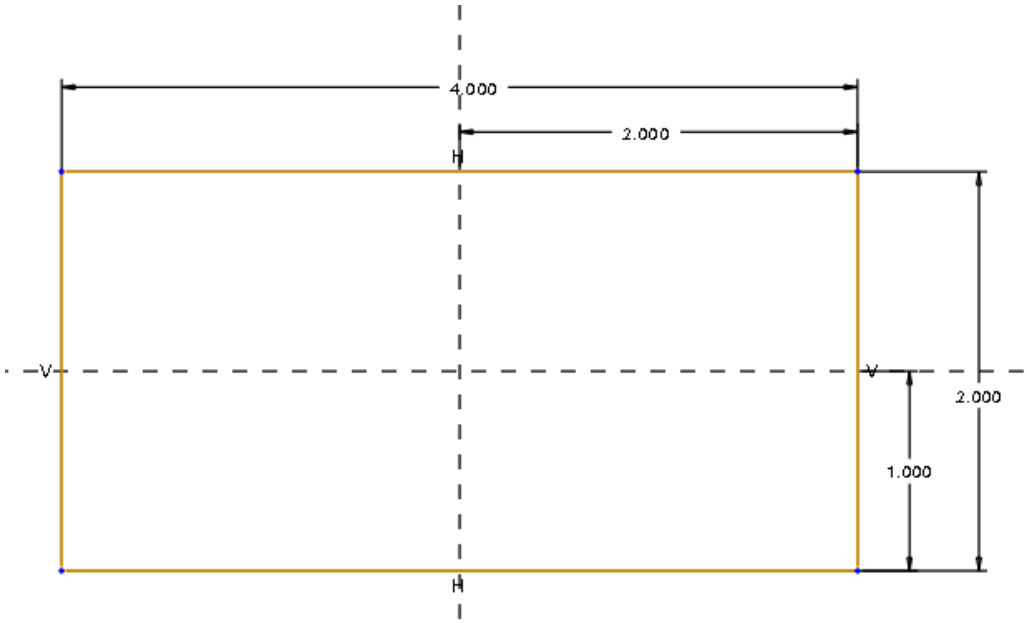
EXERCISE 19

Advanced Relations (Equations)

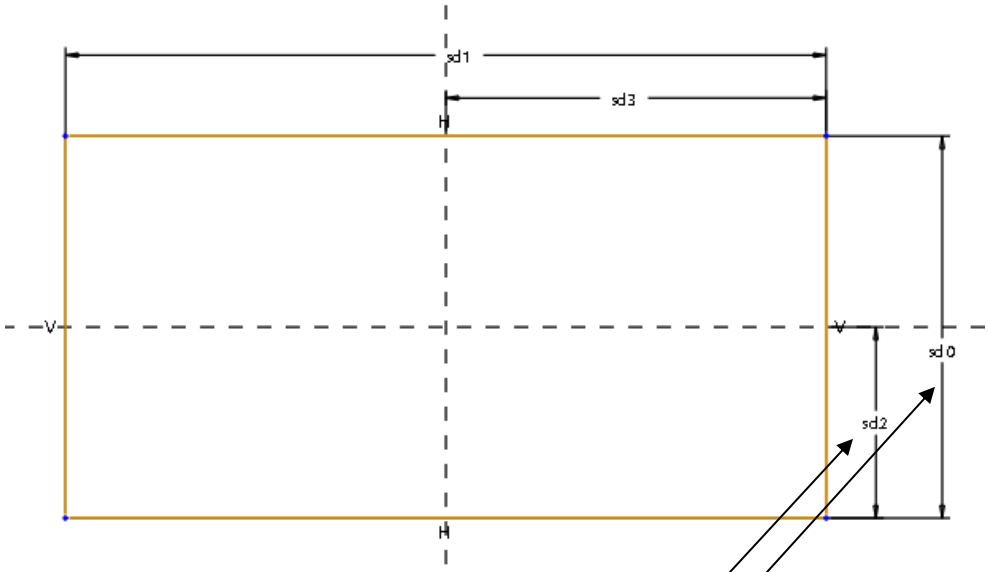
Here is an example of how to use equations. The images below represent the same model, but can easily be changed by double clicking on a dimension, and typing in a new value. This normally would create rebuild errors because the rib stack would need to be adjusted as well. Equations can be set up to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

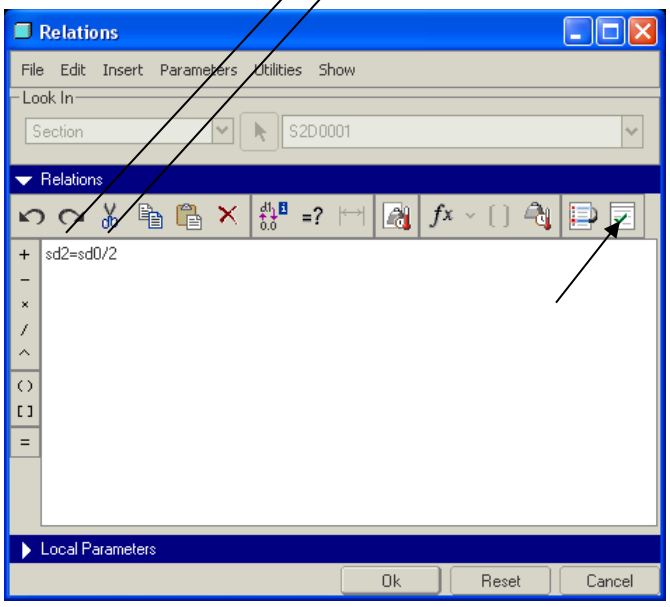


1. Sketch the following on the “Front” plane. Dimension in the order as show.

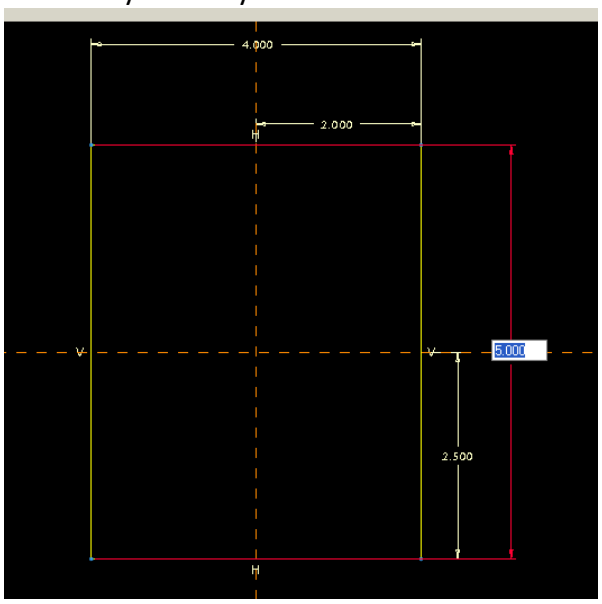


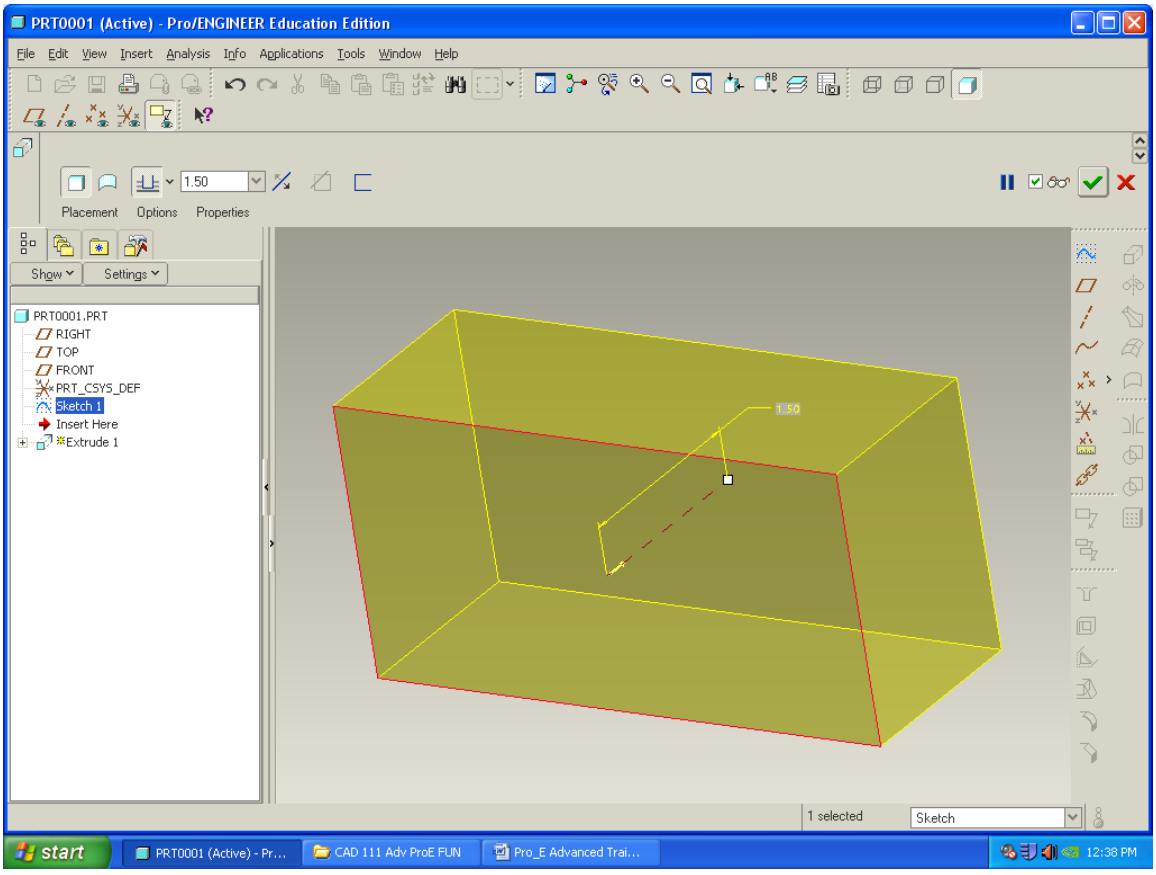
2. **Creating Equations**. Go to *Tools/Relations*. Click once on the desired dimensions to have the names automatically insert into the relations editor. Hit the verify icon on the right (Green check mark).

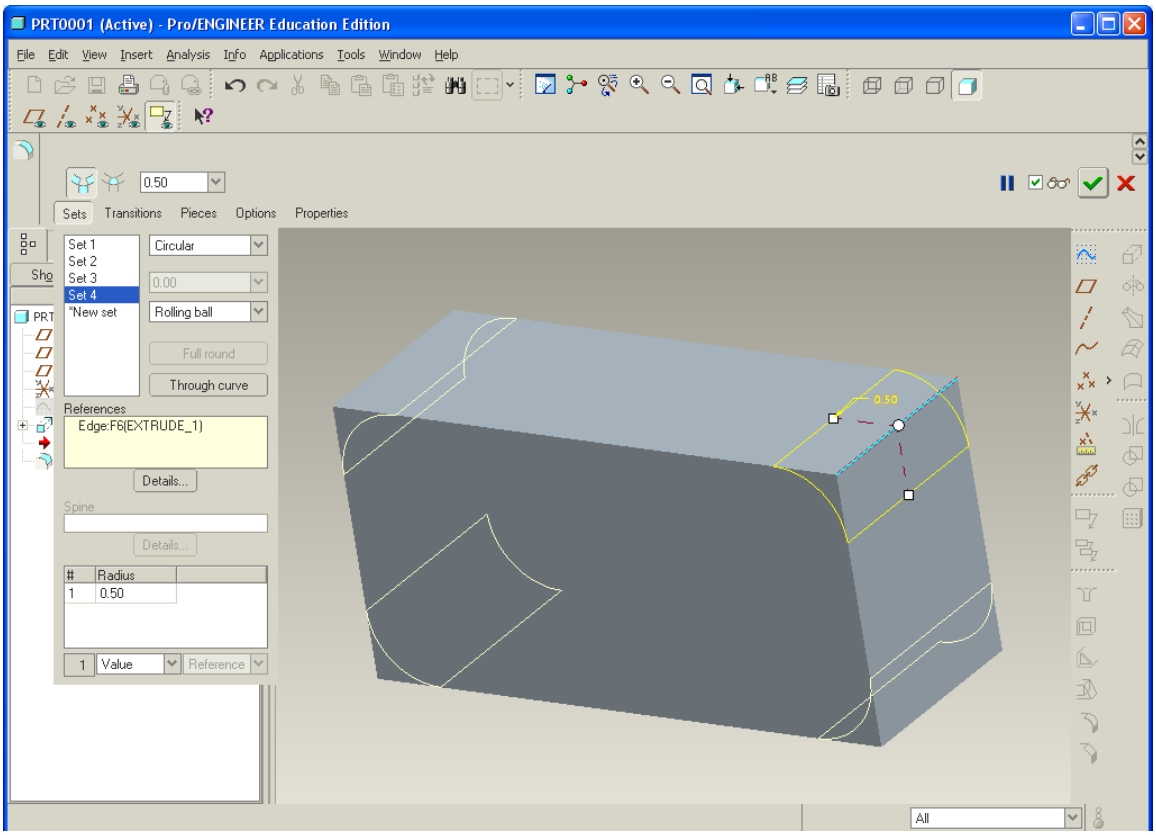




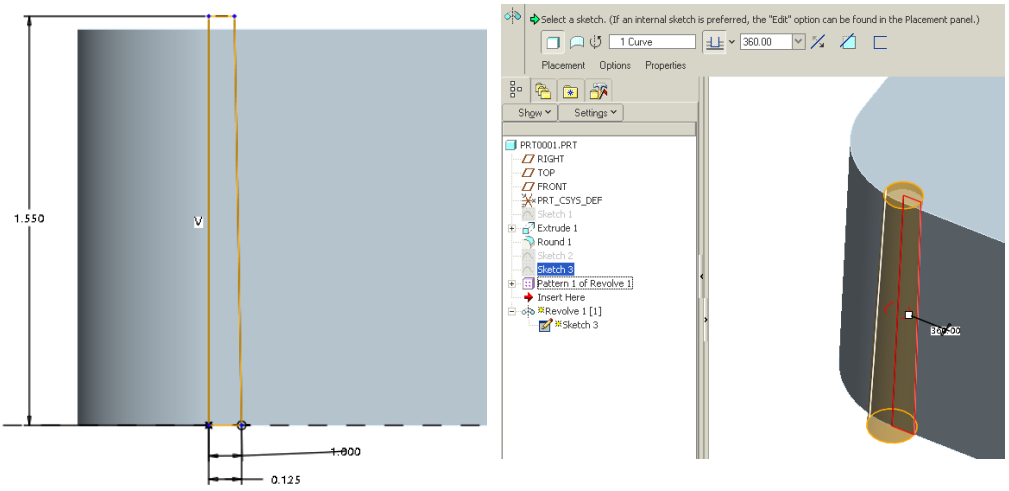
3. You're your equation by changing the 2" dimension to 5". Change it back after you verify worked.



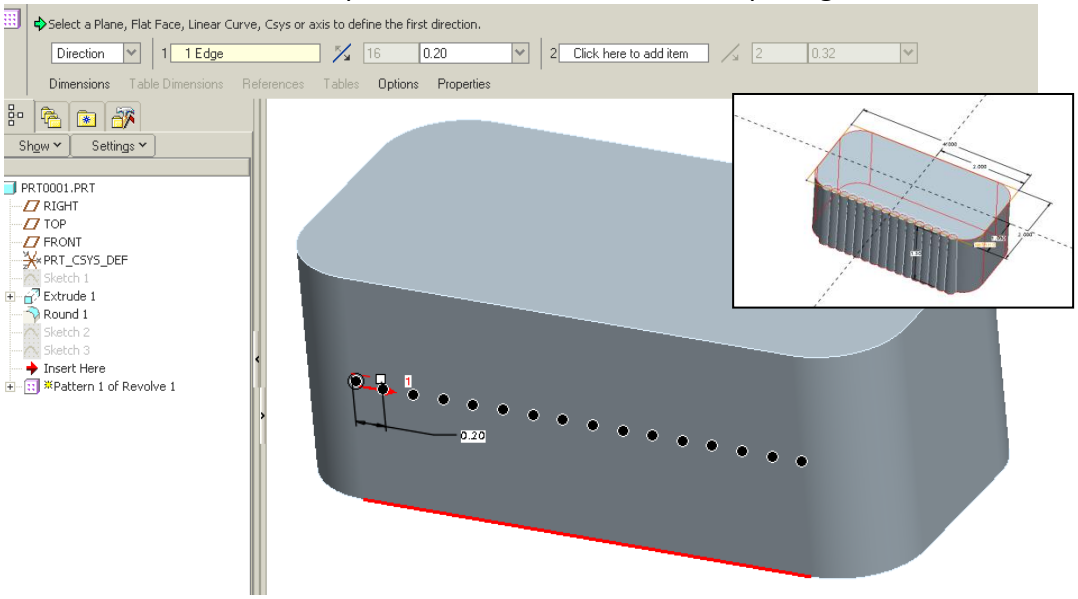




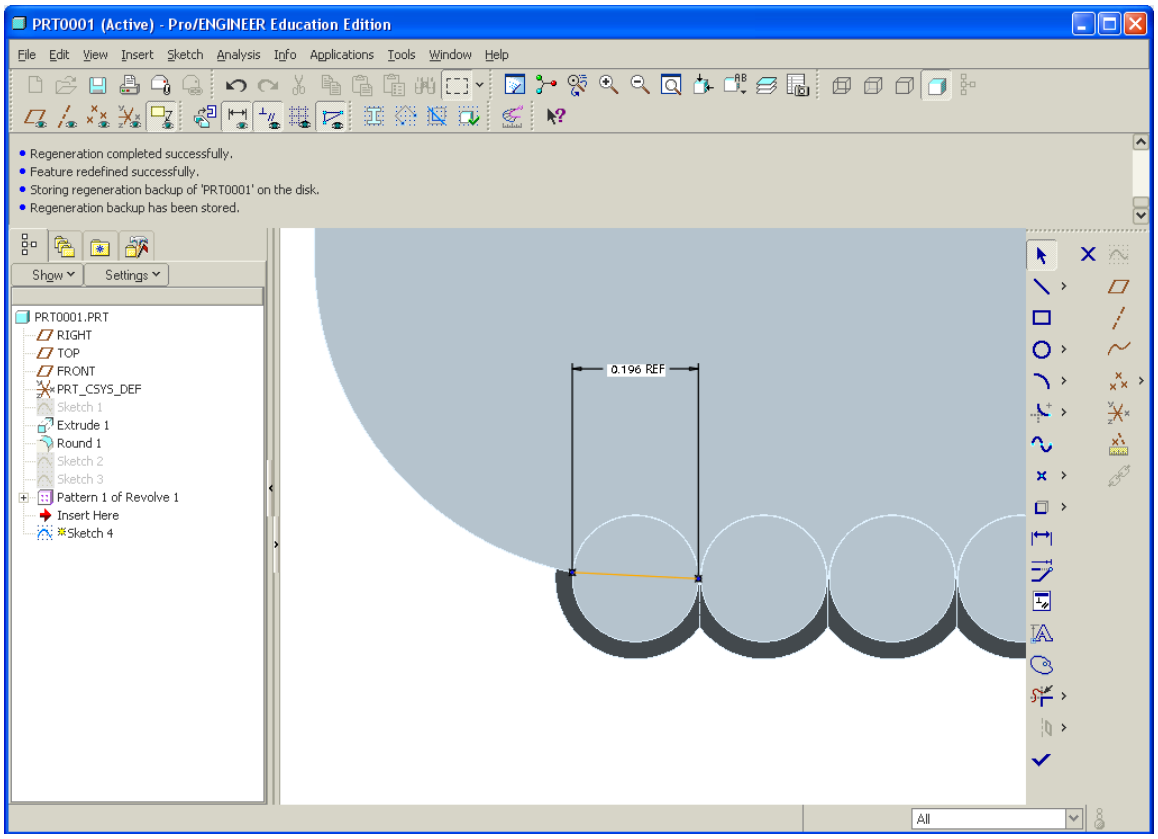
4. Start a sketch on the side face, and draw the following. Note, use Sketch/References to activate the tangent edge vertexes for reference of the center of the revolve feature. Revolve.



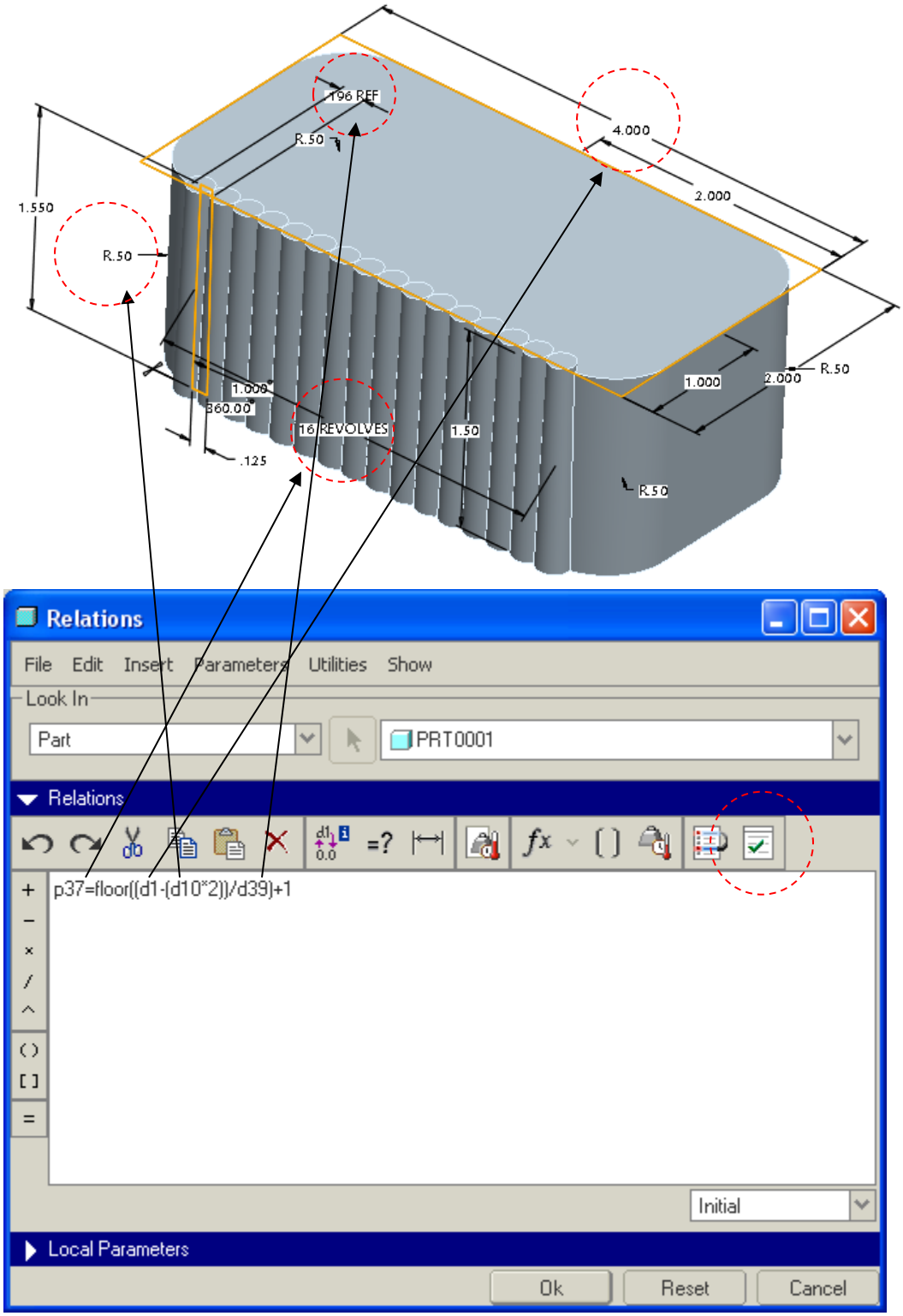
5. Create a linear pattern. 16 instances at .200” spacing.



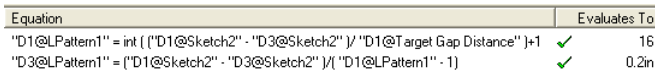
6. Create a new sketch; Draw the following with coincident relations on both sides of the cylinder (Use Sketch/References to select vertexes first). Let it become a driven dimension. Done.



7. **Creating Equations**. Go to *Tools/Relations*. Click once on the desired dimensions to have the names automatically insert into the relations editor.
(*Note: You will need to preselect the actual features on the model or from the feature tree to have the dimensions appear, The .196 REF dimension needs to be activated by selecting the "Sketch 3" from the feature tree.*)



What are Equations (Relations) inside Creo Parametric?



Equation		Evaluates To
"D1@LPattern1" = int (("D1@Sketch2" - "D3@Sketch2")/ "D1@Target Gap Distance")+1	✓	16
"D3@LPattern1" = ("D1@Sketch2" - "D3@Sketch2")/("D1@LPattern1" - 1)	✓	0.2in

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.

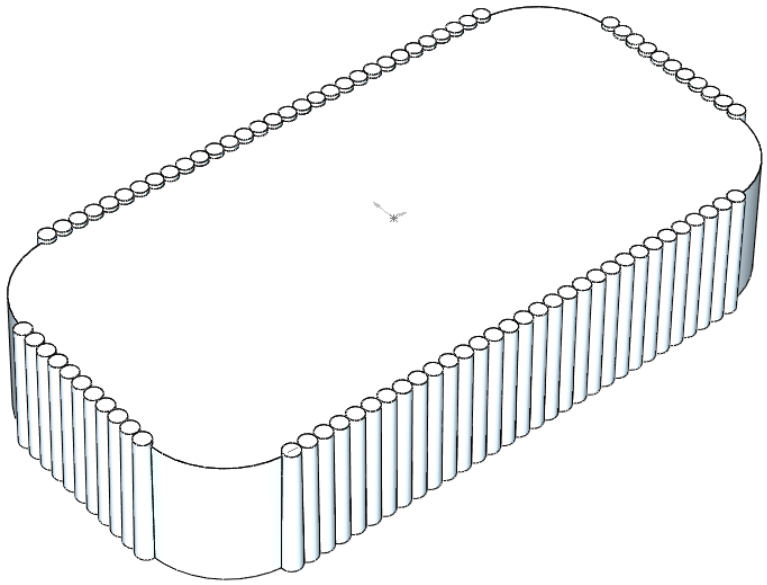
NOTE: Dimensions driven by equations cannot be changed by editing the dimension value in the model.

8. Here is an index of functions.

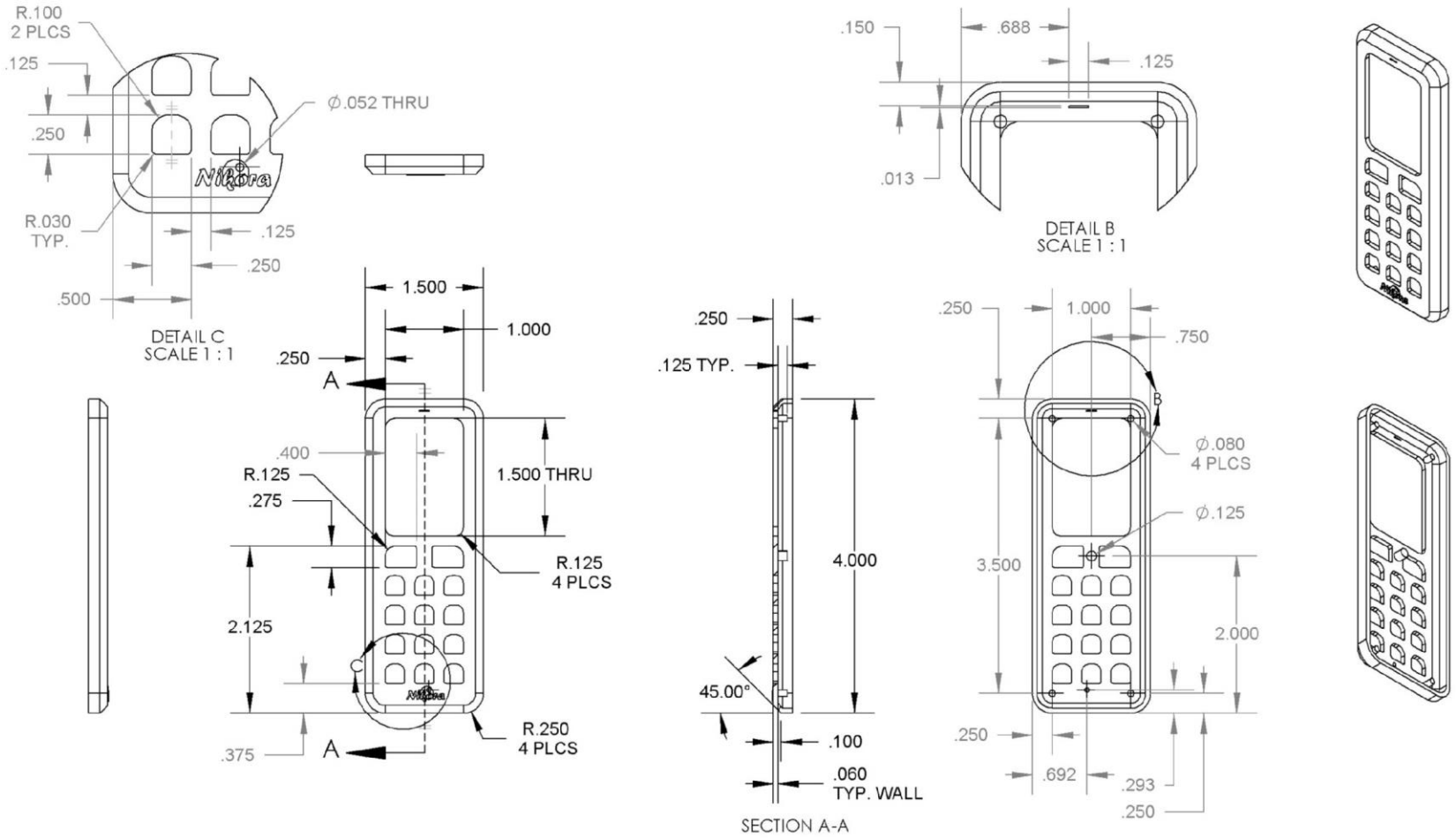
Function	**Name**	**Notes**
sin (a)	sine	a is angle expressed in radians
cos (a)	cosine	a is angle expressed in radians
tan (a)	tangent	a is angle expressed in radians
atn (a)	inverse tangent	a is angle expressed in radians
abs (a)	absolute value	returns the absolute value of a
exp (n)	exponential	returns 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
FLOOR	integer	returns a as an integer
sgn (a)	sign	returns the sign of a
Constant		
pi	pi	3.14...

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

10. Ctrl select the pattern and sketches associated with the equation, Right mouse click and select Group from the options. Mirror all, both sides.



Finished



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

Nikora TECHNOLOGIES

TITLE:

LAB 19
CELLULAR PHONE
FRONT BEZEL
MODEL. L19

SIZE	DWG. NO.	REV
A		

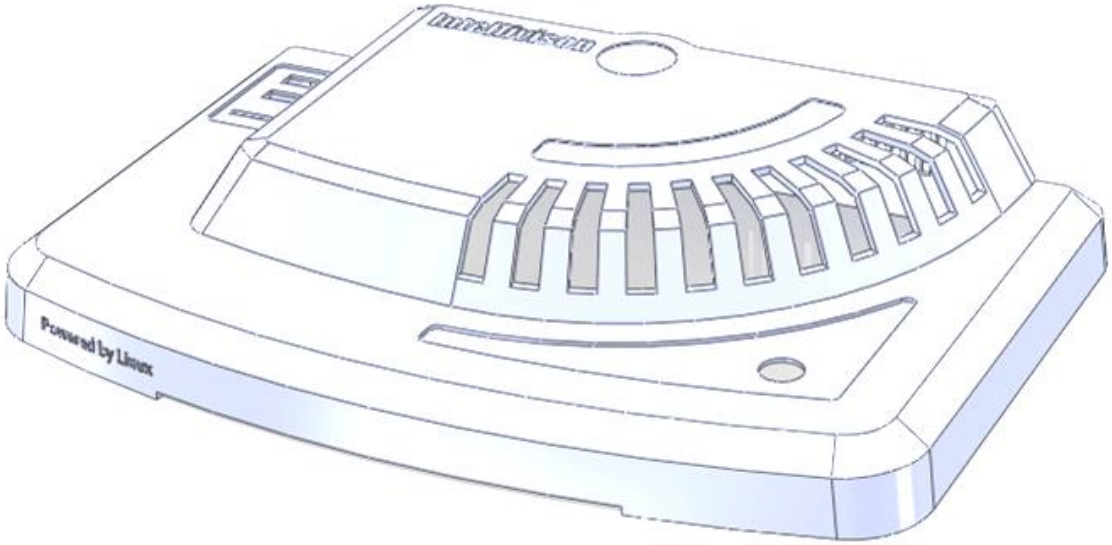
SCALE: 1:2	WEIGHT:	SHEET 1 OF 1

5 4 3 2 1

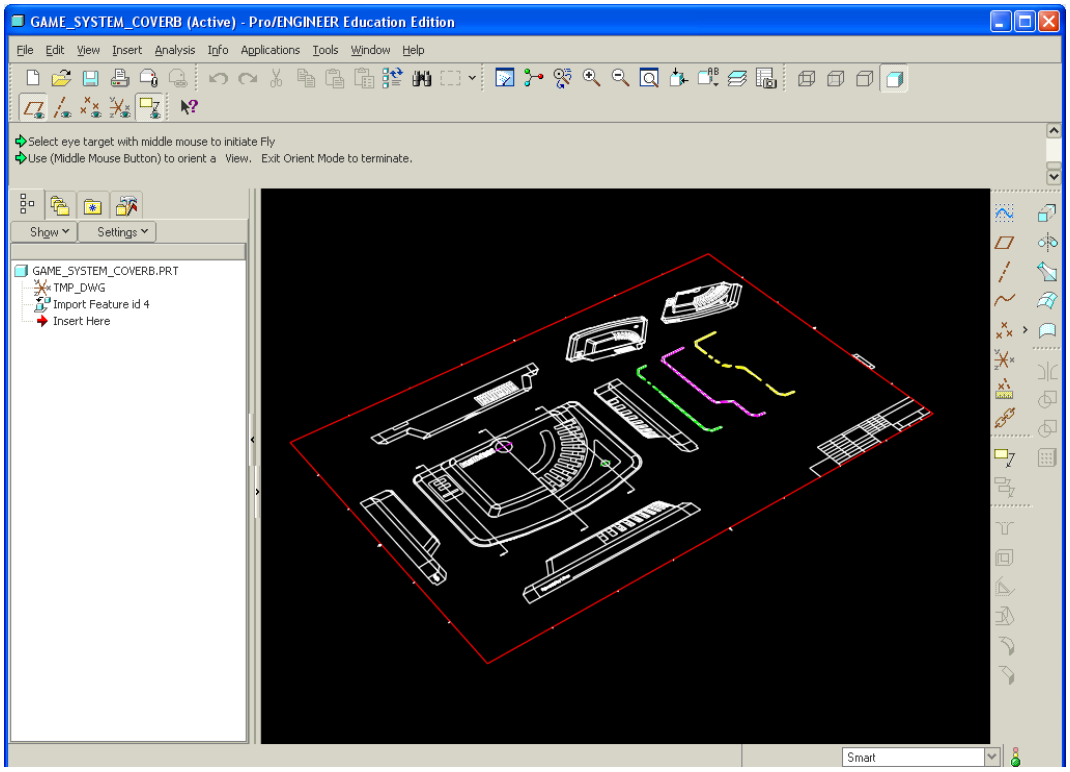
EXERCISE 20

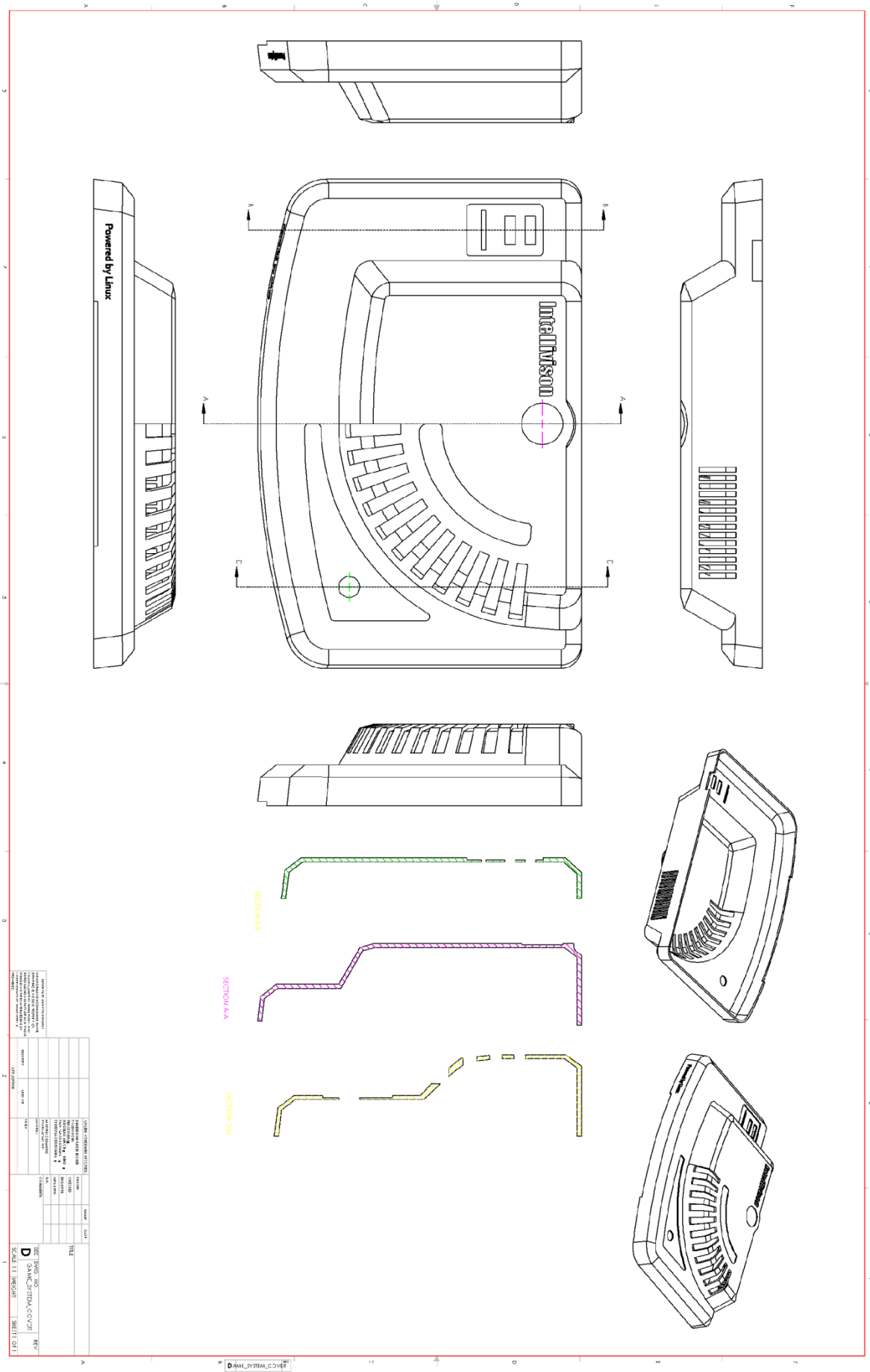
QUIZ - Model a Video Game System Cover (video with no sound available)

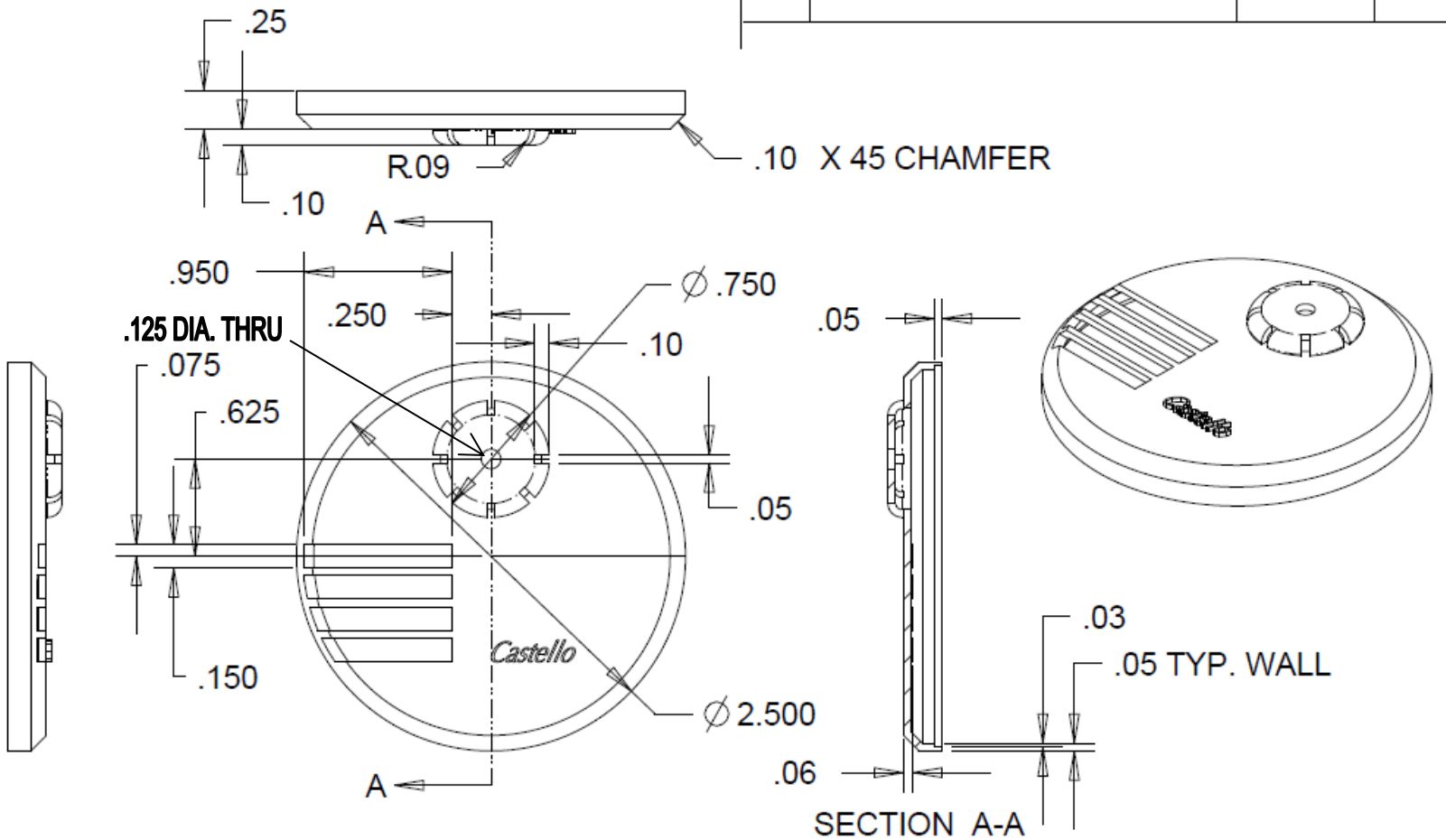
1. Open the GAME_SYSTEM_COVER.dwg



2. Import the DWG into a part file. Construct a model from the provided data.

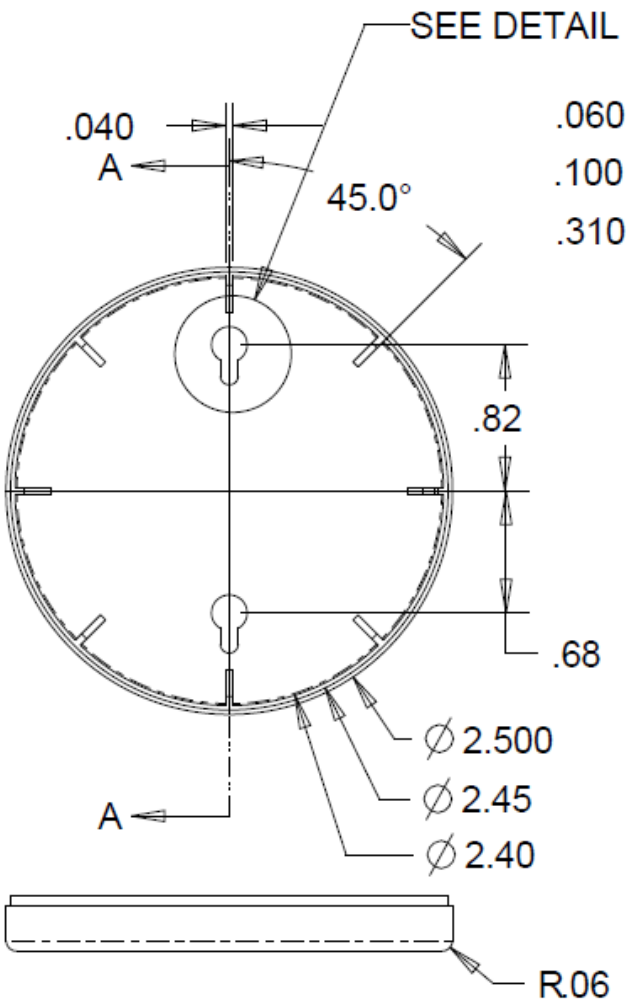


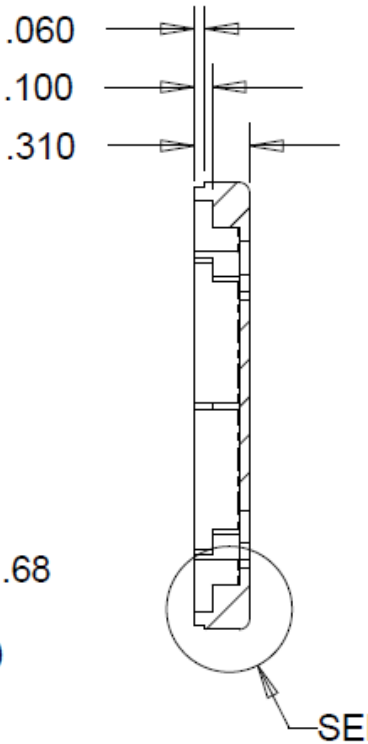

Powered by Linux
intellivision
SECTION B-B
SECTION A-A
SECTION D-D



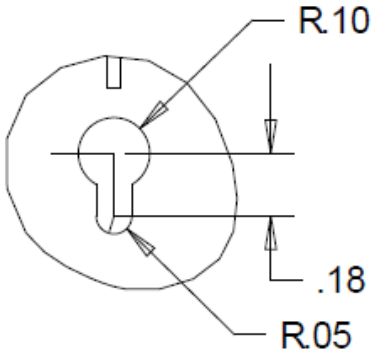
SMOKE DETECTOR BEZEL

LAB 20

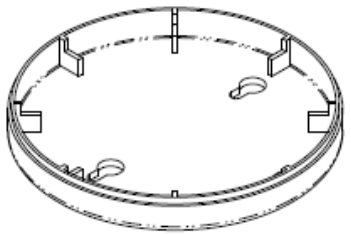


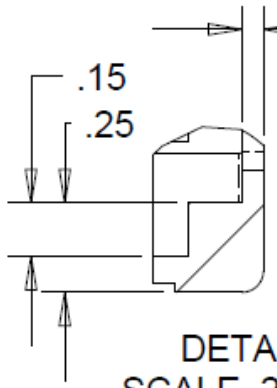


SECTION A-A



DETAIL A
SCALE 2.000





DETAIL B
SCALE 2.000

SMOKE DETECTOR REAR BEZEL

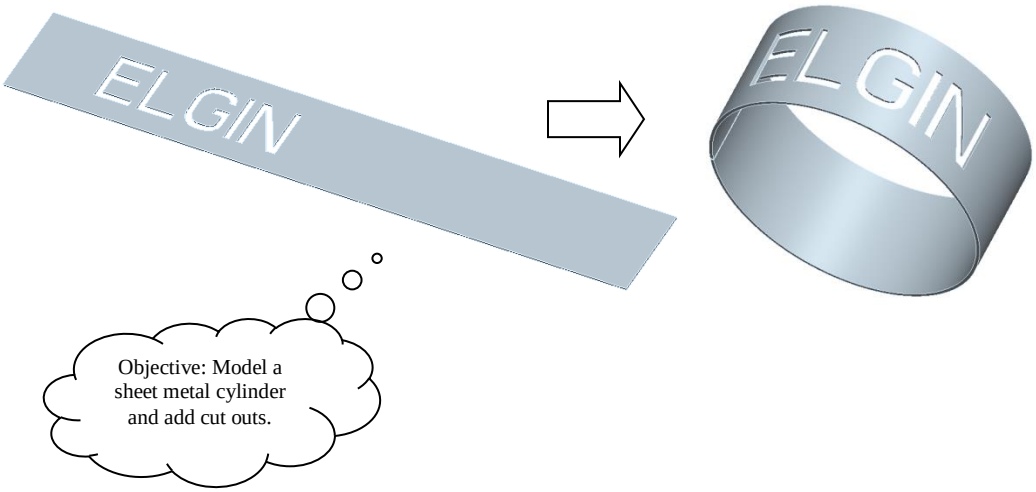
LAB 21

EXERCISE 21

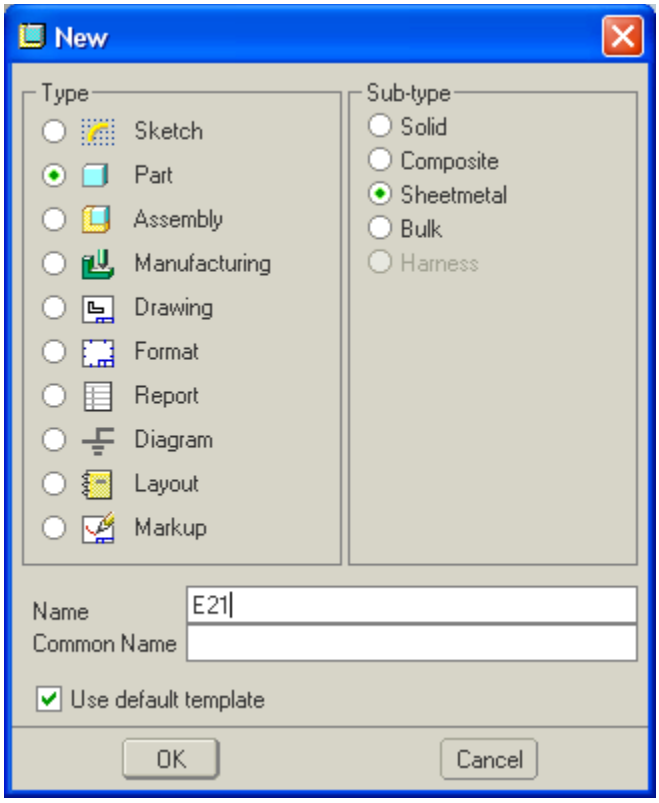
Sheet Metal II

Cylinders

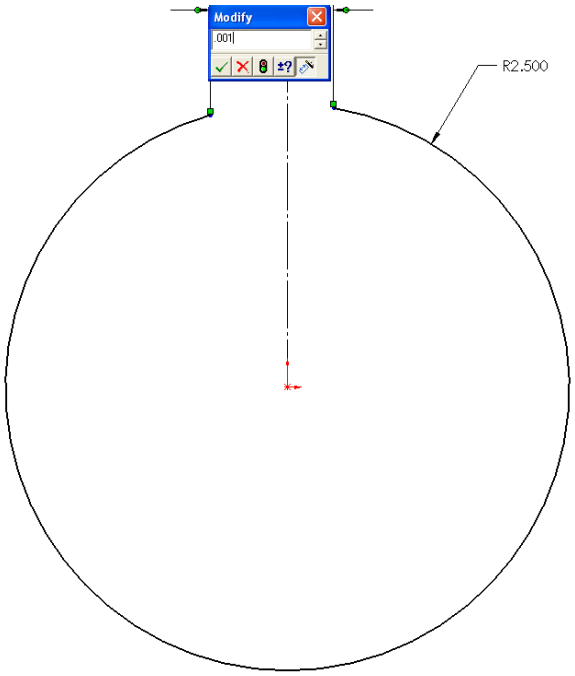
Sheet Metal part files *can be very useful for extracting a flat pattern.*



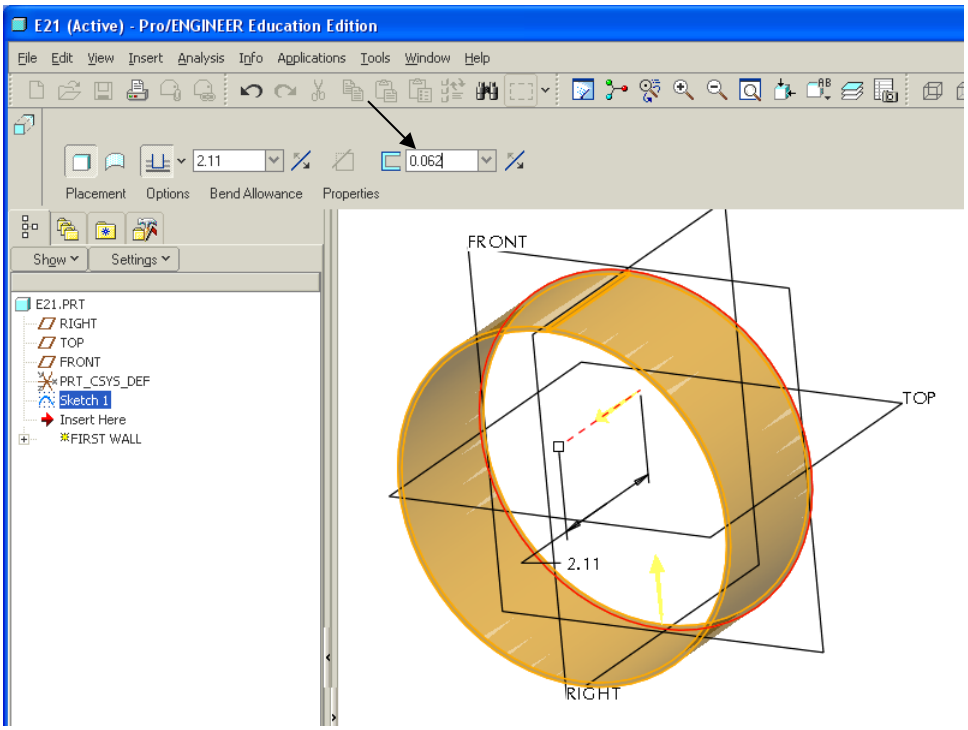
1. Go to file/new and select new part/sheet metal and save as “E21”.



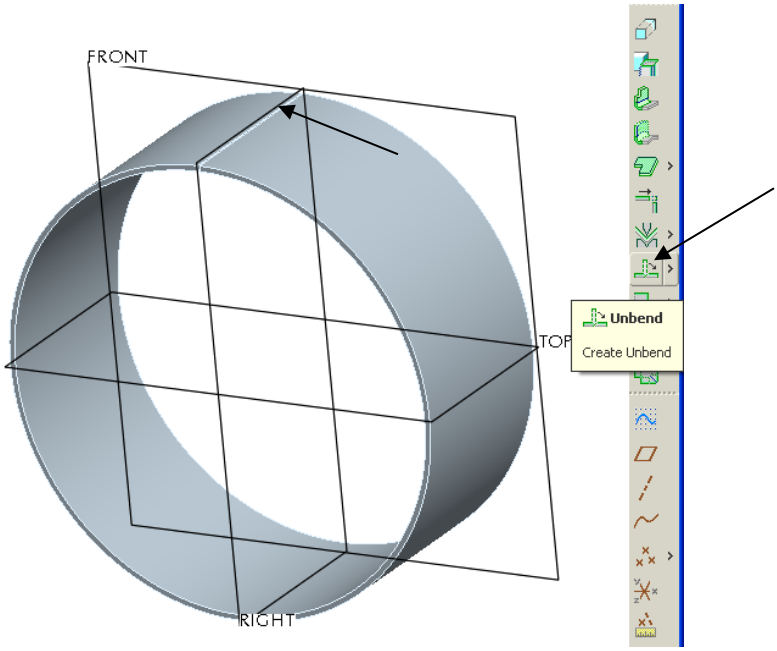
2. Draw the following sketch on the “Front” plane, use the “center point arc” tool. Make both ends of the arc symmetric to a vertical centerline. Space @ .01”



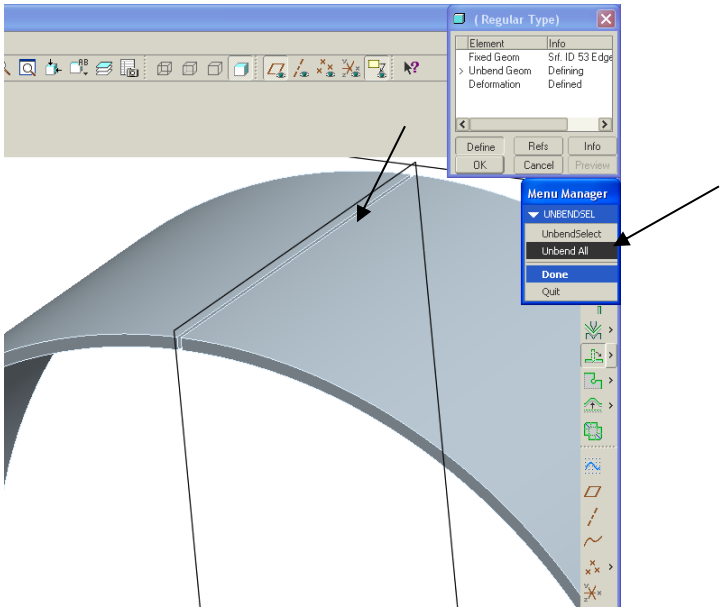
3. Boss Extrude blind 2.5”. Notice that it should be creating a thin feature and set the thickness to .060”.



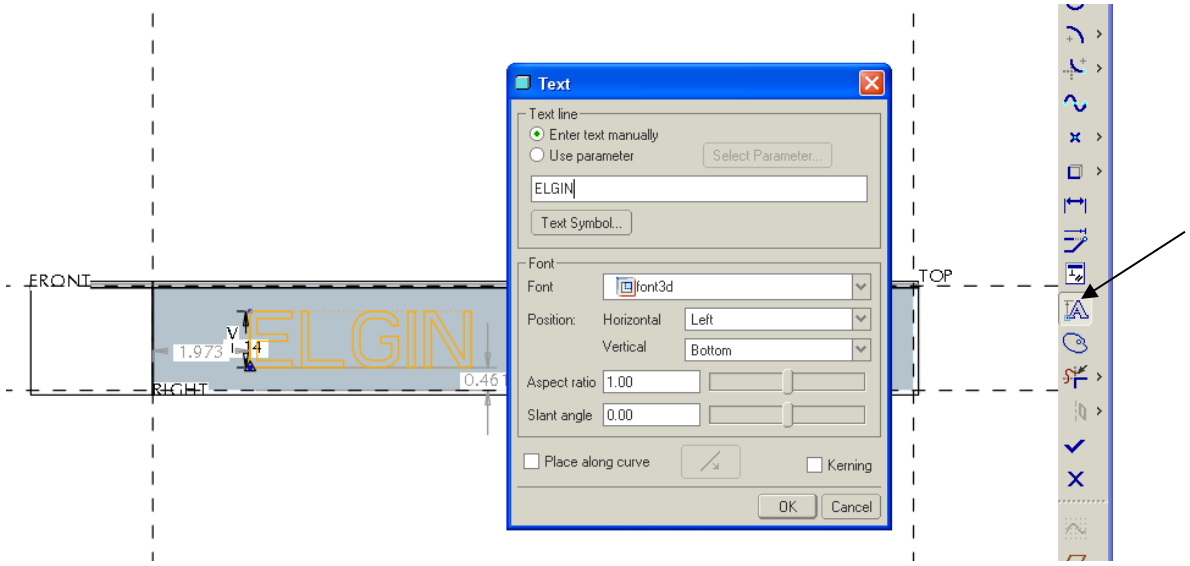
4. Select the edge of the cylinder and select Insert Bends. Set radius to “0”. Hit OK.



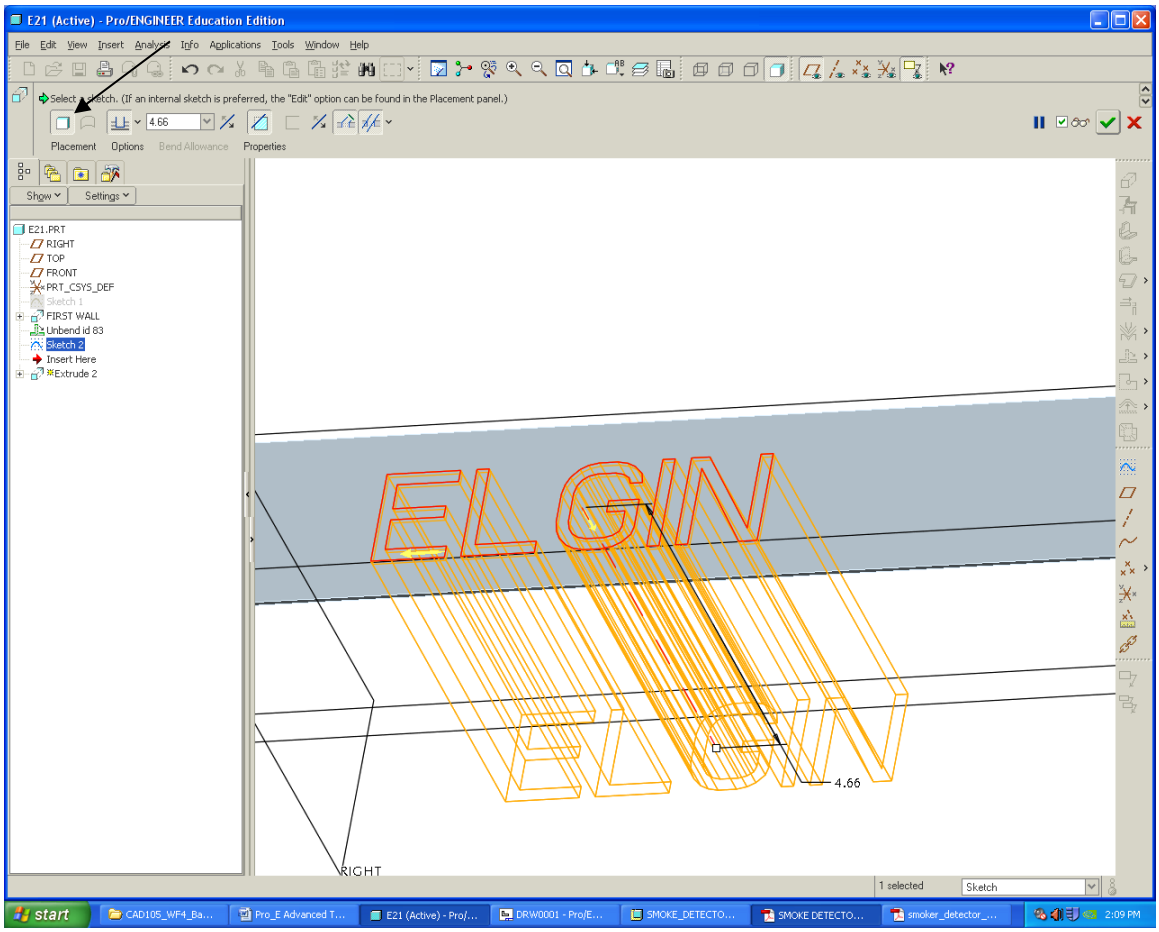
5. Select the “Flatten” icon to verify. Then select the face and start a sketch.



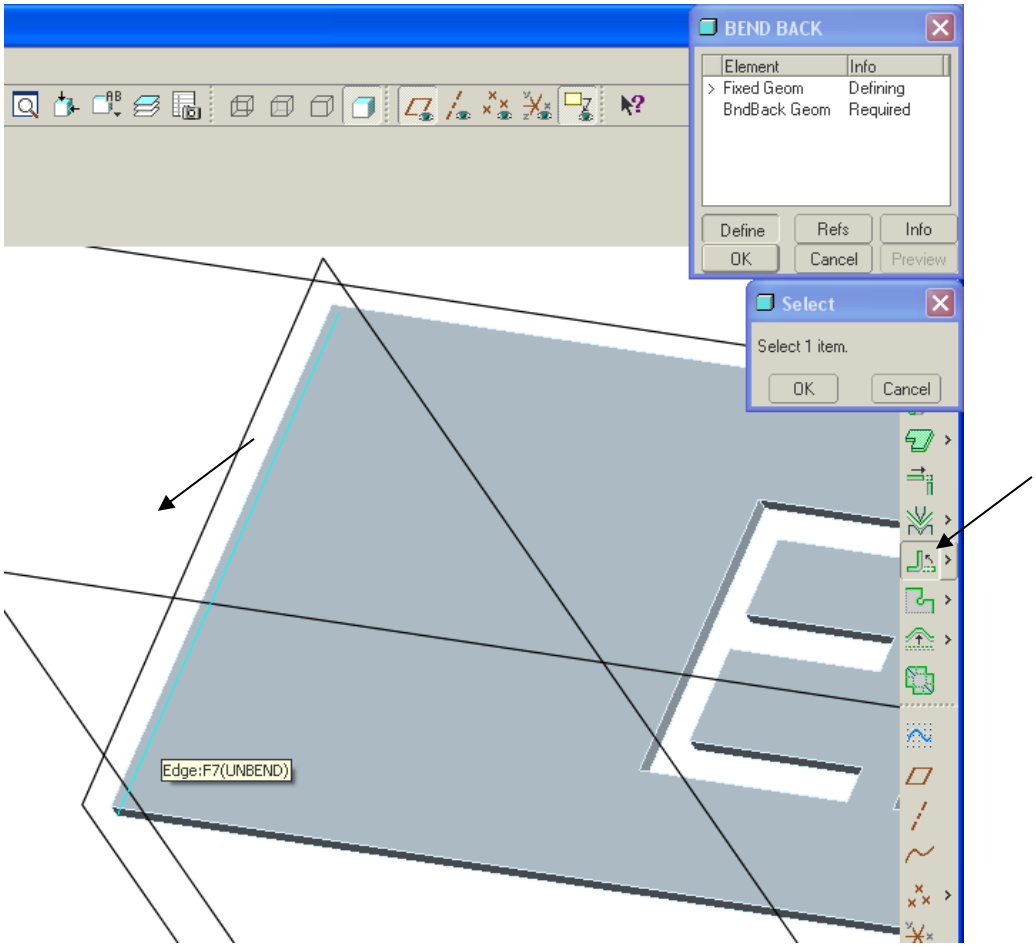
6. Use the Text tool. Write Elgin in the dialog box.



7. Select the “Flatten” icon, to refold it.



8. Use the Bend Back tool, select the fixed edge. Bend back All.

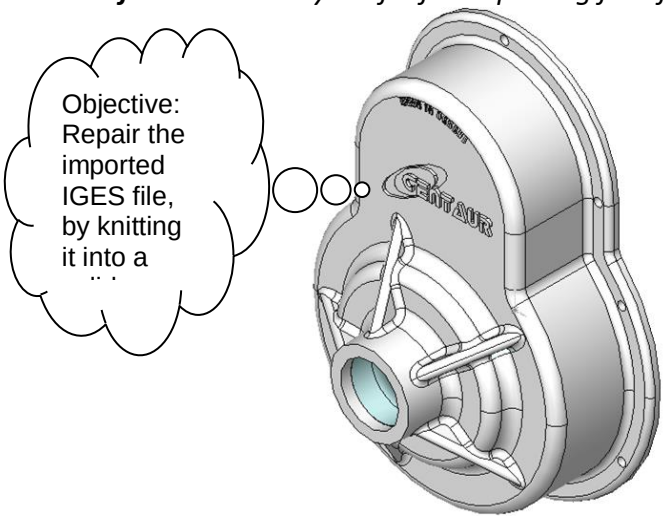


Finished

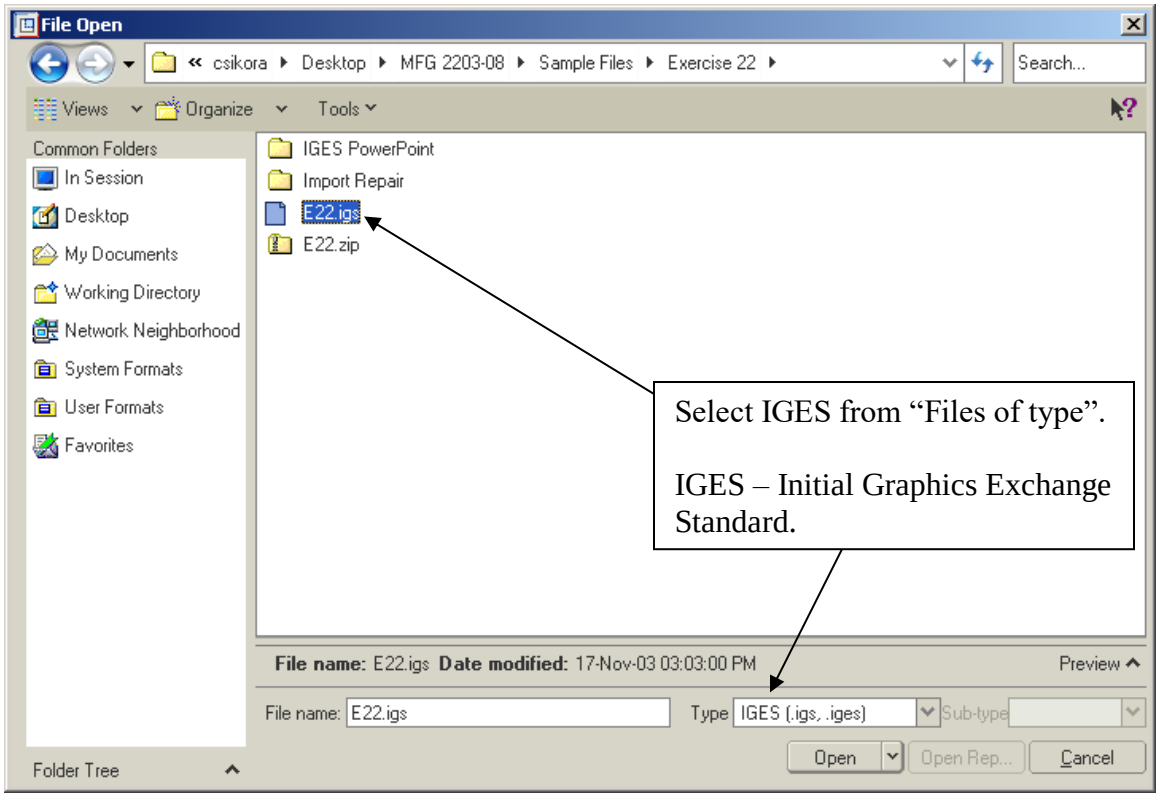
EXERCISE 22

Imported 3D Model Repair

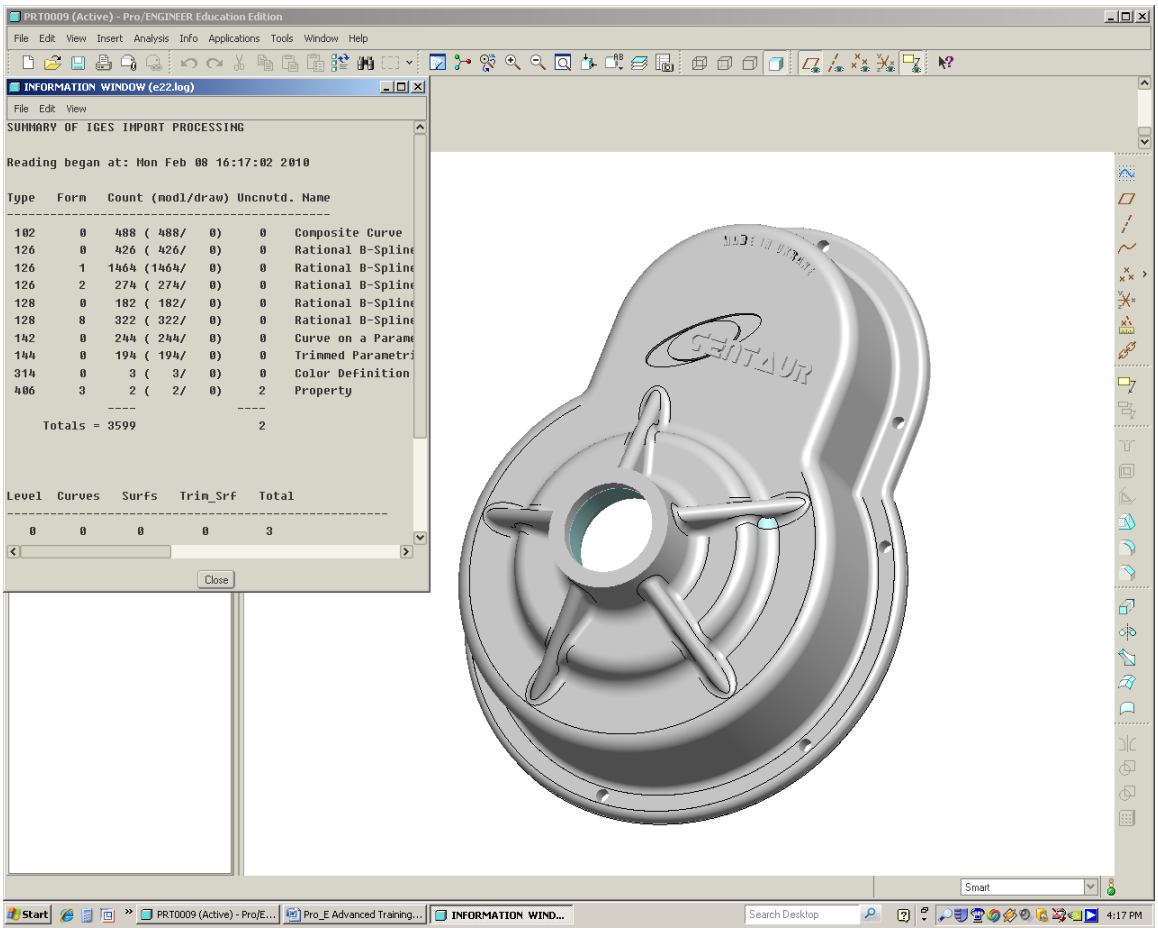
IGES files *can be very useful for importing files from other systems.*



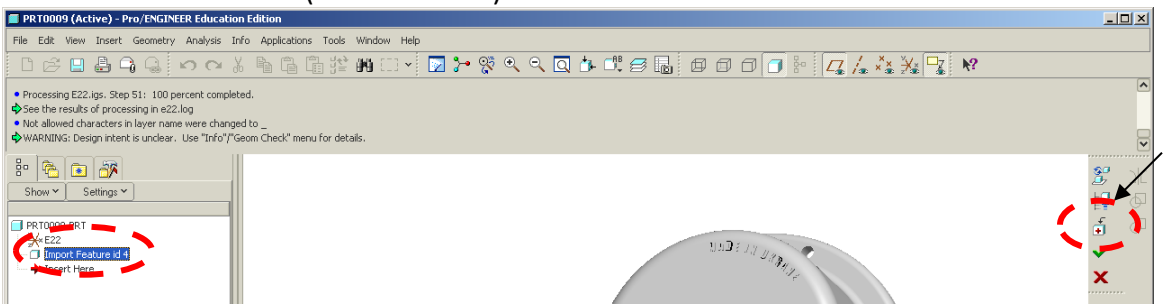
1. Go to File/Open, and select IGES. Open it into a "Part" file.



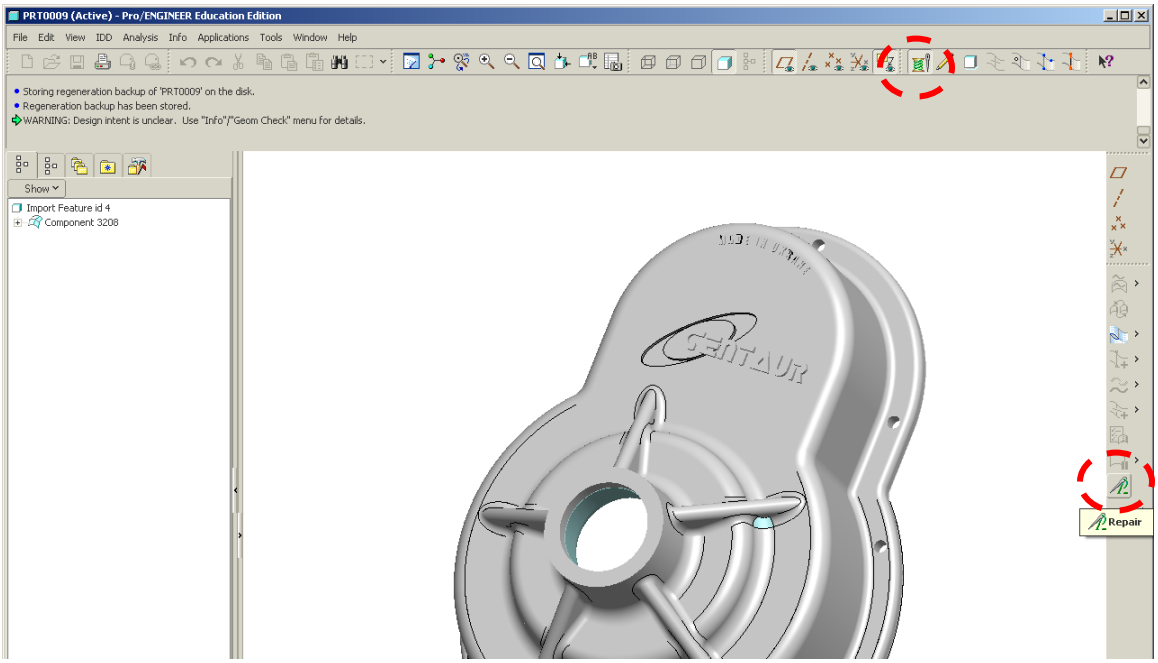
2. Once imported you will receive a message stating ProE was able to read in the file. It is very common that the file may have missing geometry or gaps and may only be a collection of surfaces versus a water tight solid. Essentially in order to make use of this (make a cavity/mold) it is imperative that it be knit into a solid.



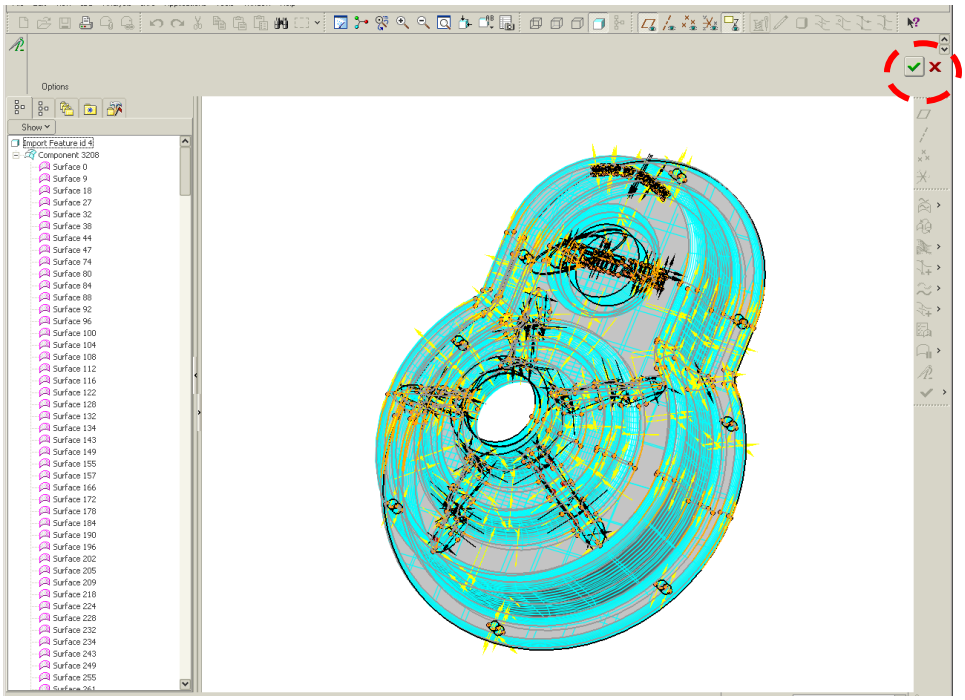
3. RMB click on the "Import Feature" in the feature tree, and Import Data Doctor icon (Red cross box).



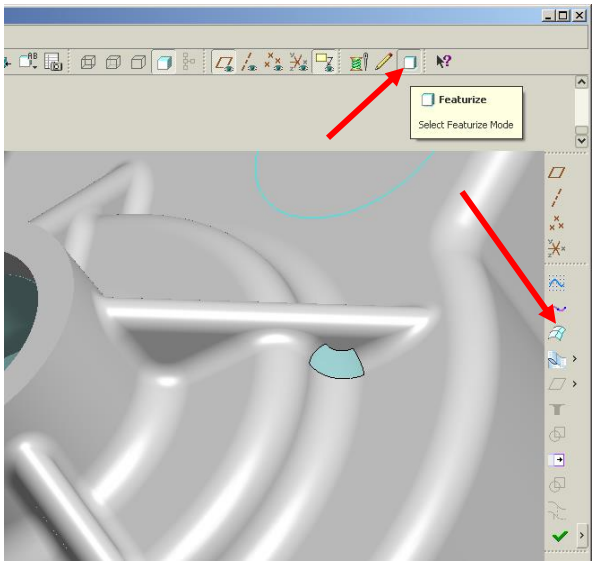
4. Select the Repair icon at the top right to open options, then select the Repair (Needle) icon to heal small gaps.



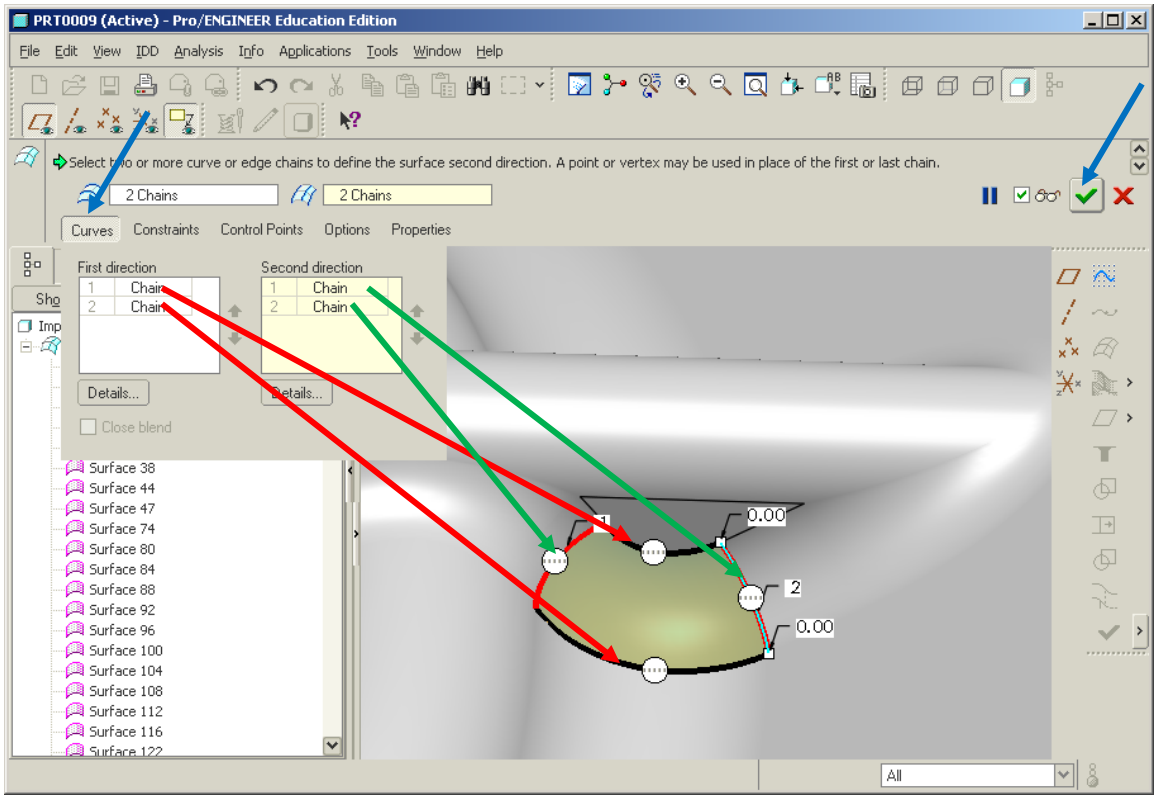
5. Hit the green check mark to apply.



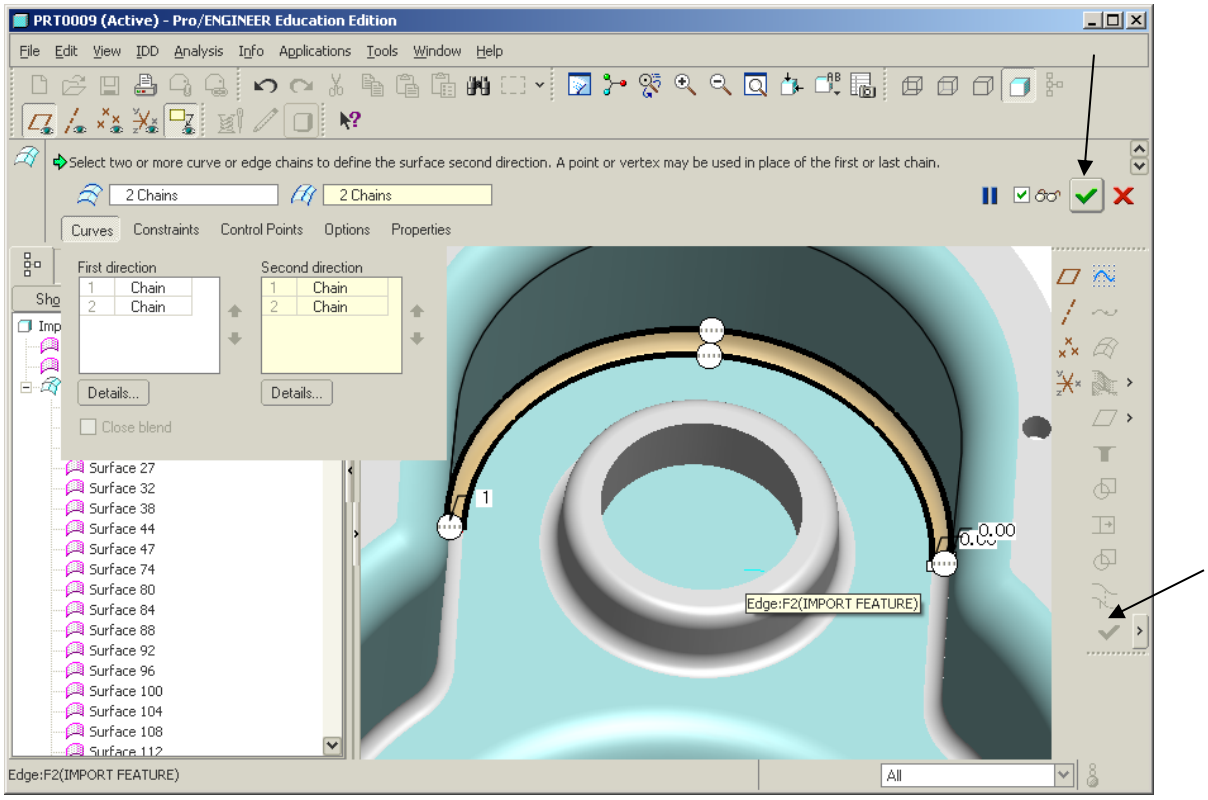
6. Now we must manually address the problem by creating surfaces to close the gaps. Select the Featurize icon, then use Boundary Blend Surface.



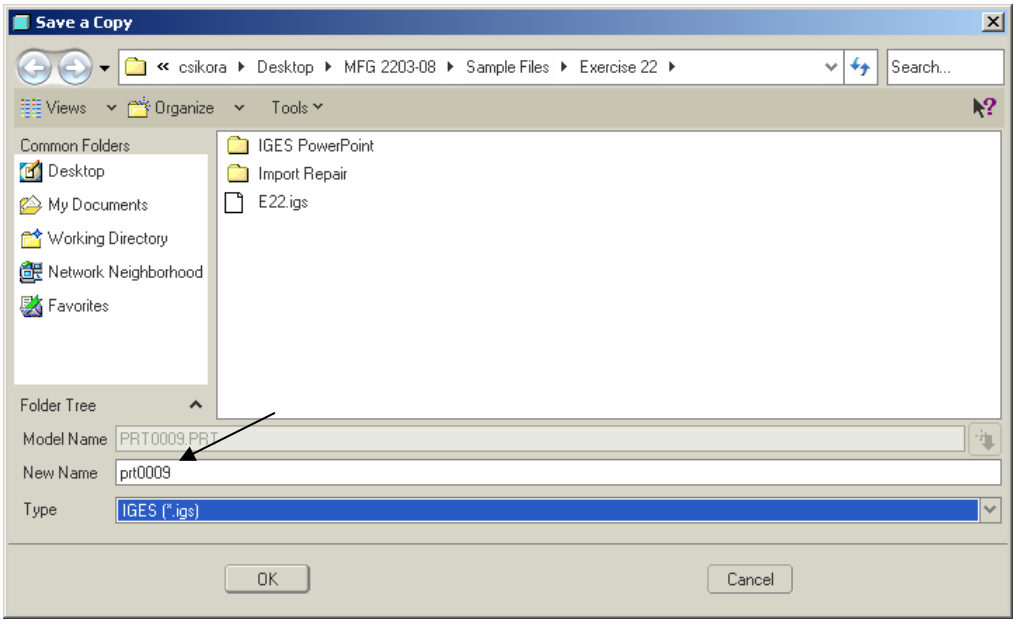
7. Select the "**Curves**" tab, then select the First Direction box, and then select the two red arrow **edges**. Then select the Second Direction box, and the two green arrow edges. Hit the Done icon.



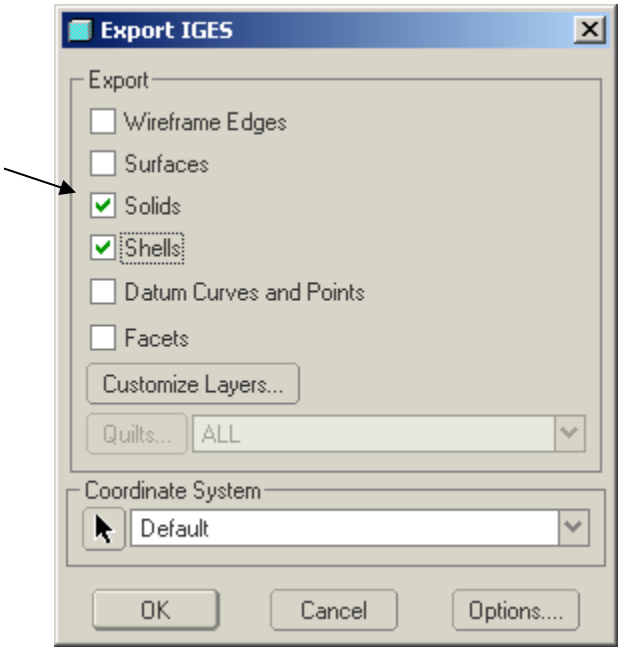
8. Finish repairing the other two gaps. Select the Done icon both times it appears.



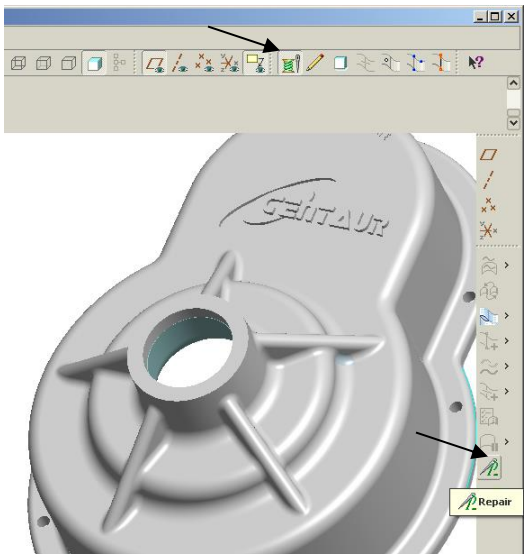
9. Save a copy and select IGES. Save as E22b.

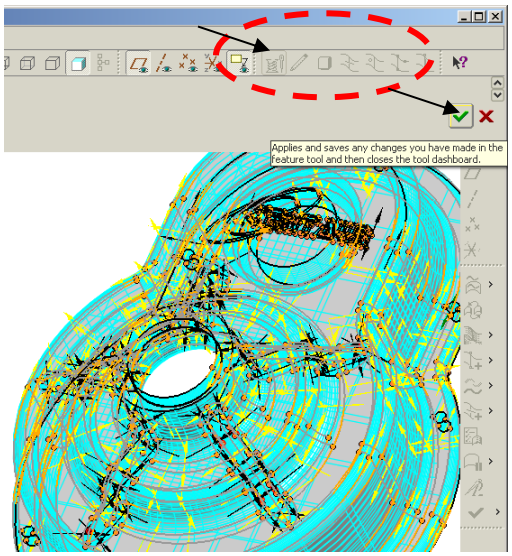


10. Select Solids and Shells.

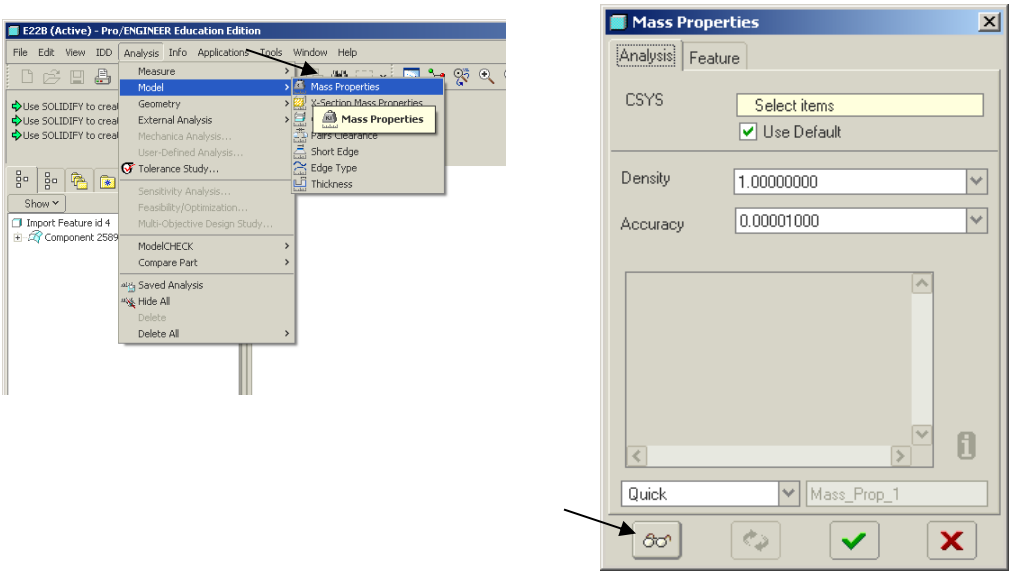


11. Re-import the new E22b IGES file. Step through the repair tools again.

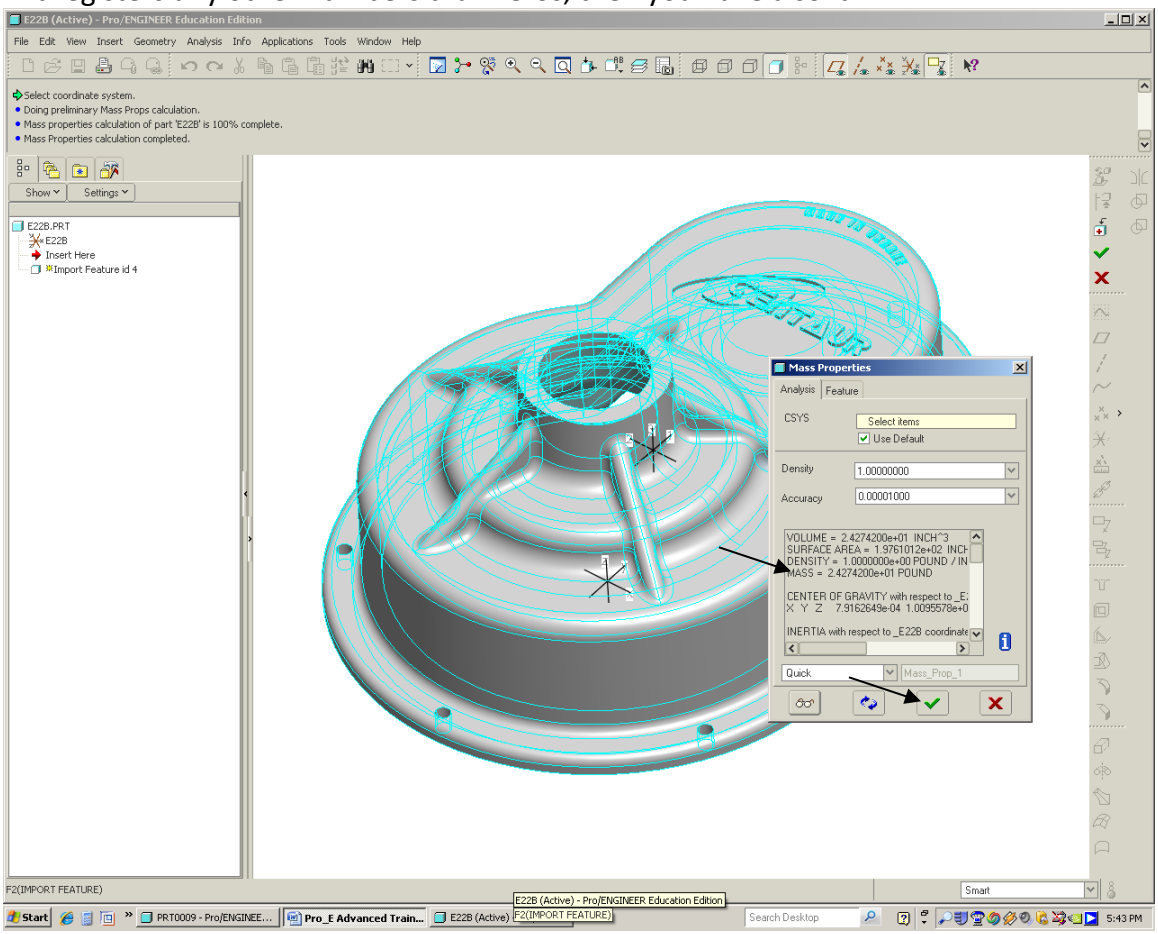




12. To verify if it is a solid got to Analysis/Model/Mass Properties

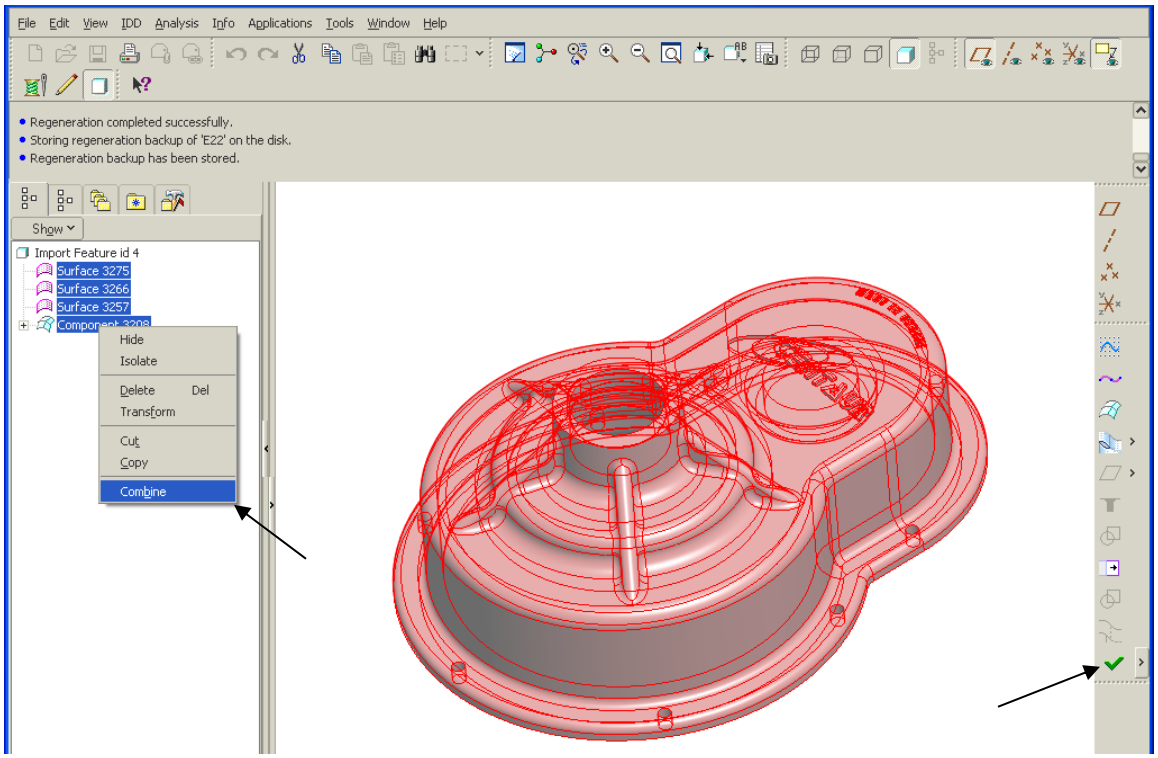


If it registers any other numbers than zeros, then you have a solid.

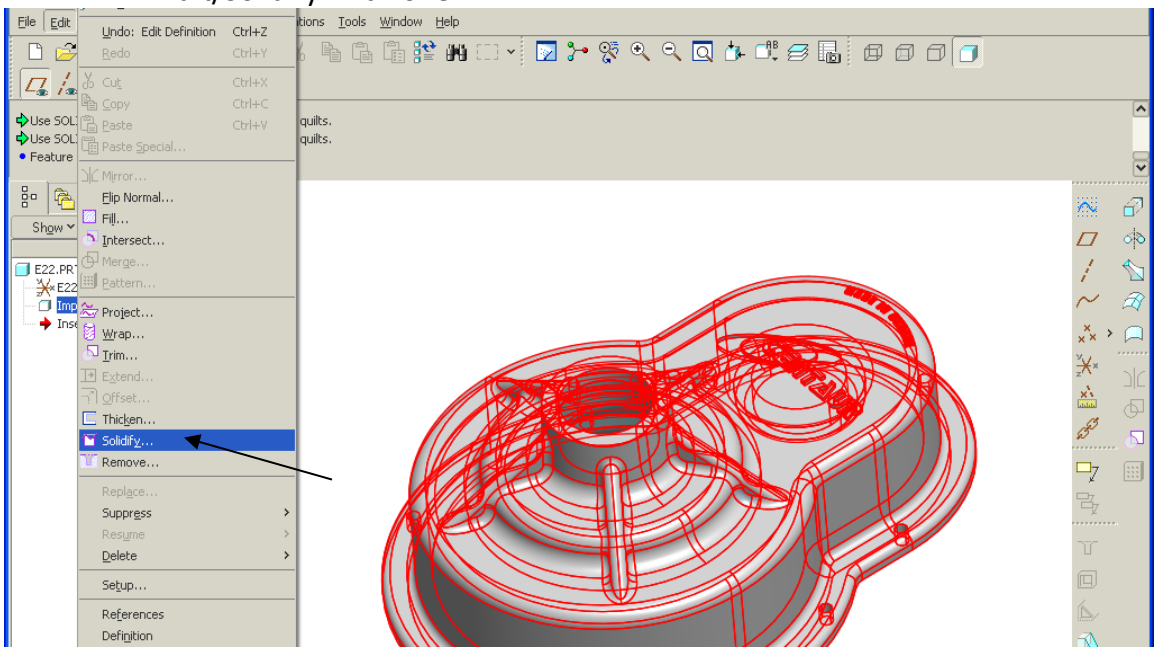


FINISHED

13. **Alternative** to exporting is to select all surfaces in the Doctor tool and RMB and select “combine”.



14. Hit Done, and Done. Then select the main feature from the Tree and go to “Edit/Solidify” hit Done.



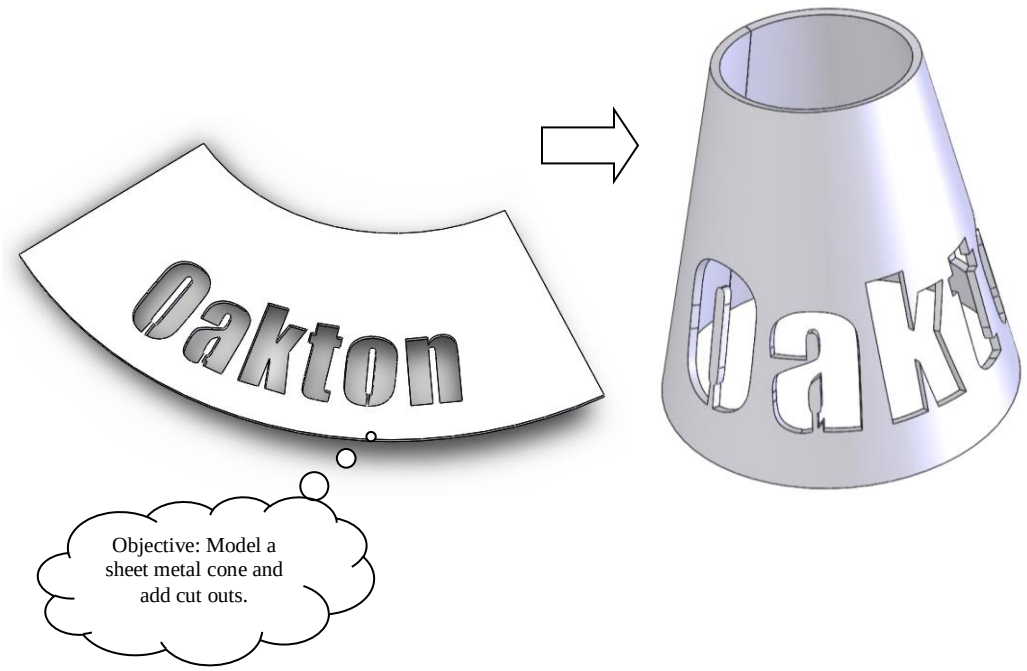
FINISHED

EXERCISE 23

Sheet Metal IV

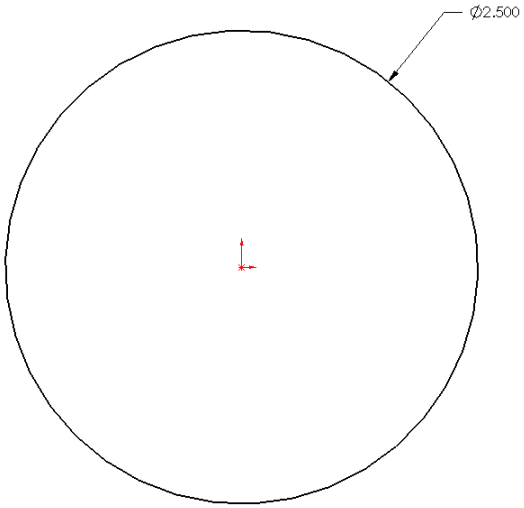
Modeling Conical Sheet Metal Forms

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

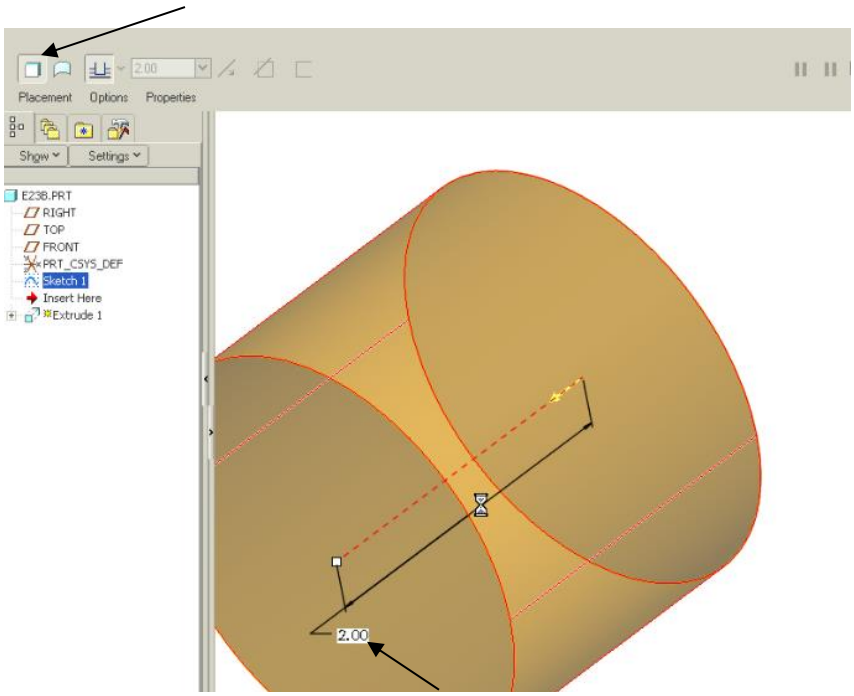


1. Go to file/new and select new part (NOT SHEETMETAL) and save as “E23”.

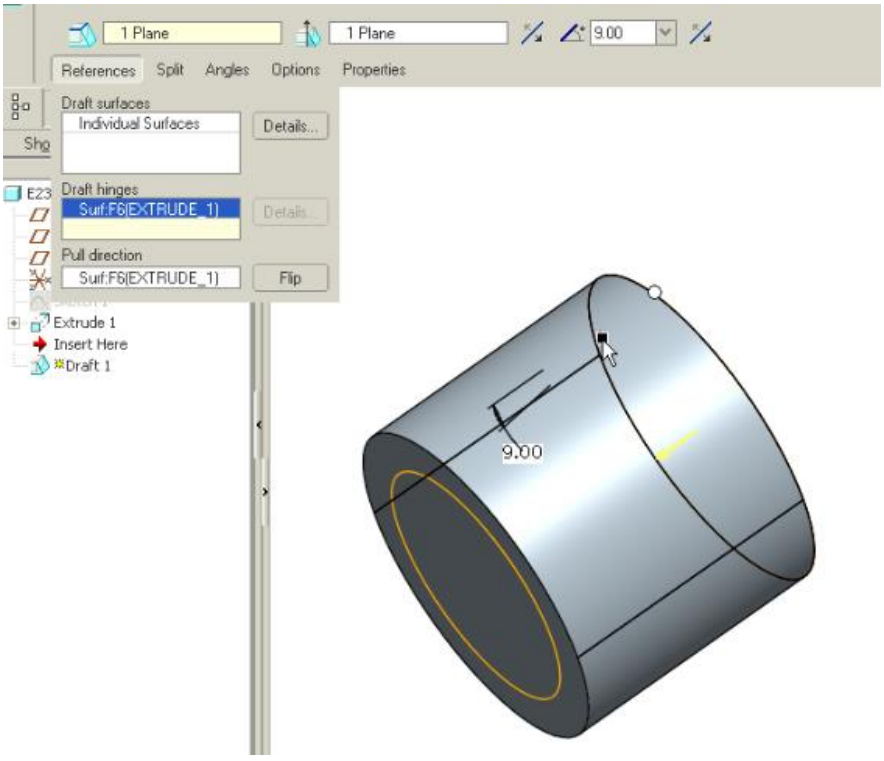
2. Draw the following sketch on the “Front” plane, use the “circle” tool.



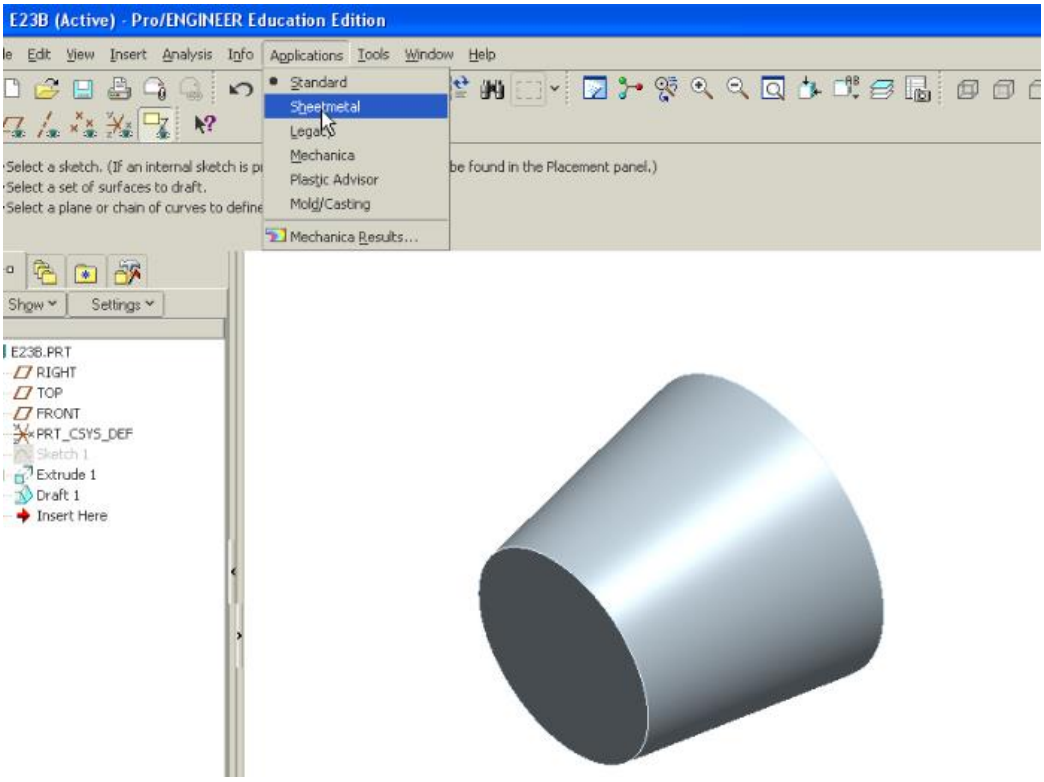
3. Boss Extrude blind 2.5”



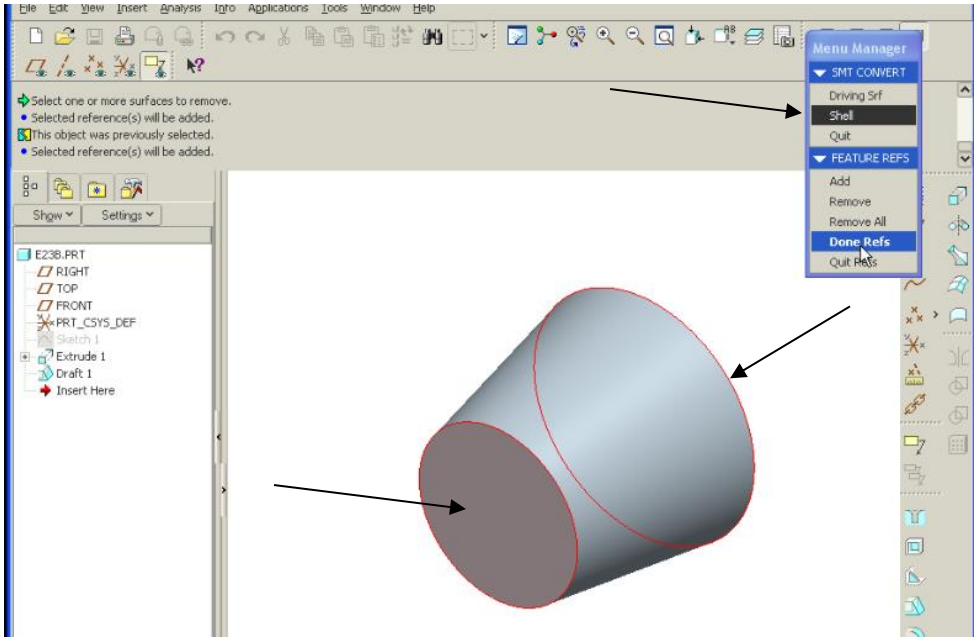
4. Add 13 degrees of draft.



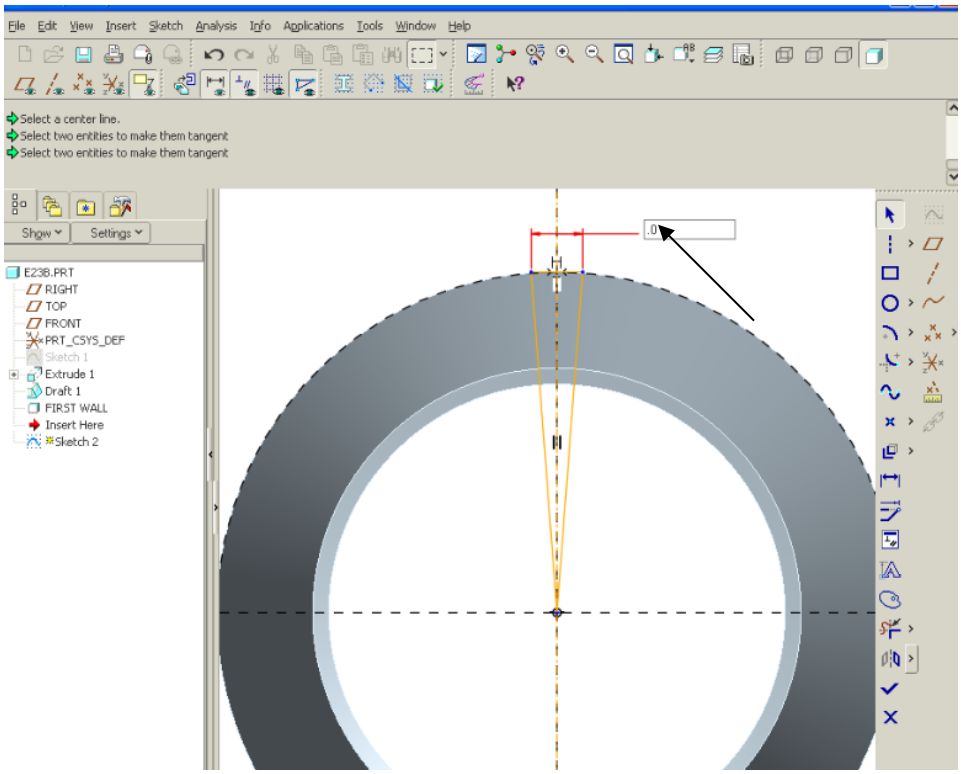
5. Switch to Sheetmetal.



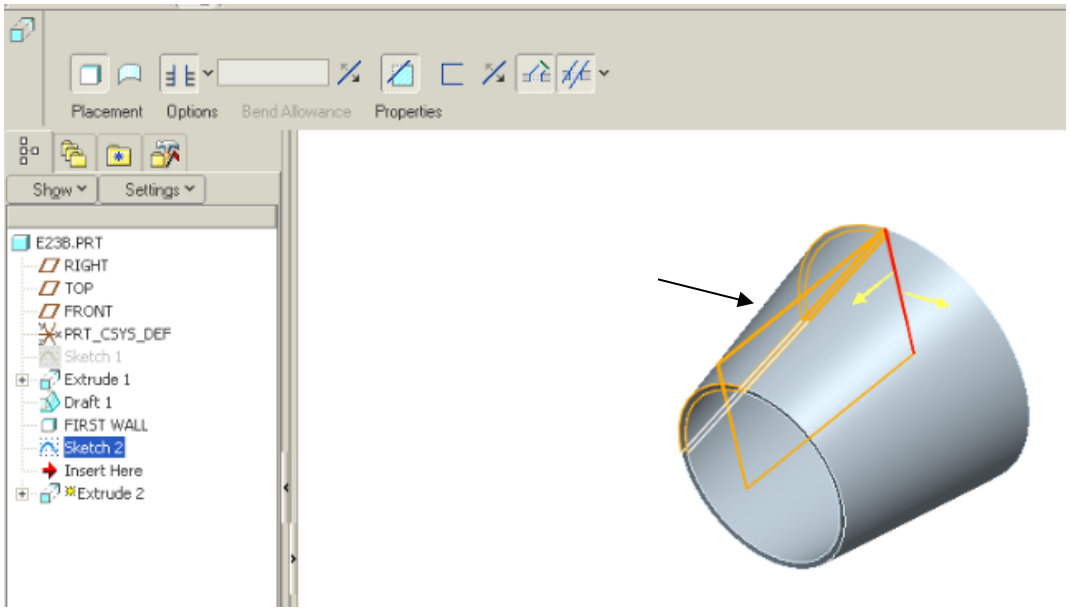
6. Select the front and back planar faces, then select the Shell feature. Set thickness to ".060". Hit Done Refs.



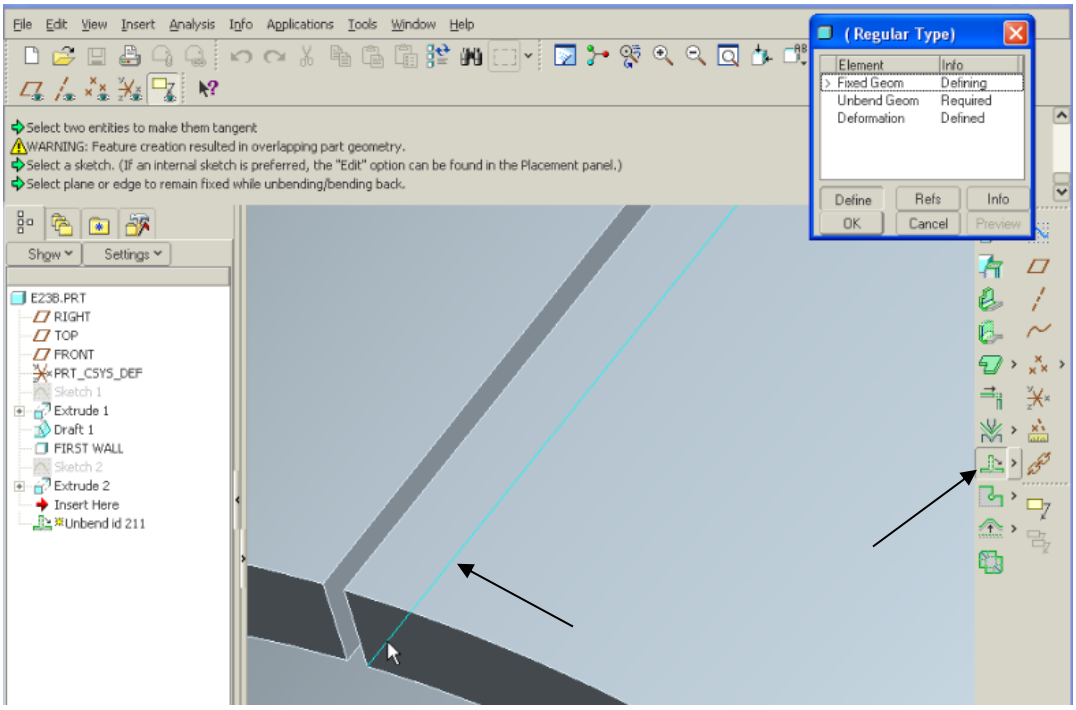
7. Select the “Front” plane and start a sketch on it. Draw the following angled cutout. Dimension the edges and set the thickness to .02”.



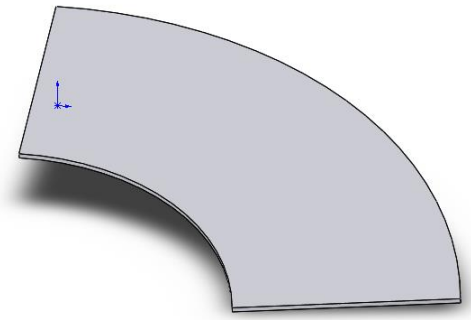
8. Cut Extrude Through All.

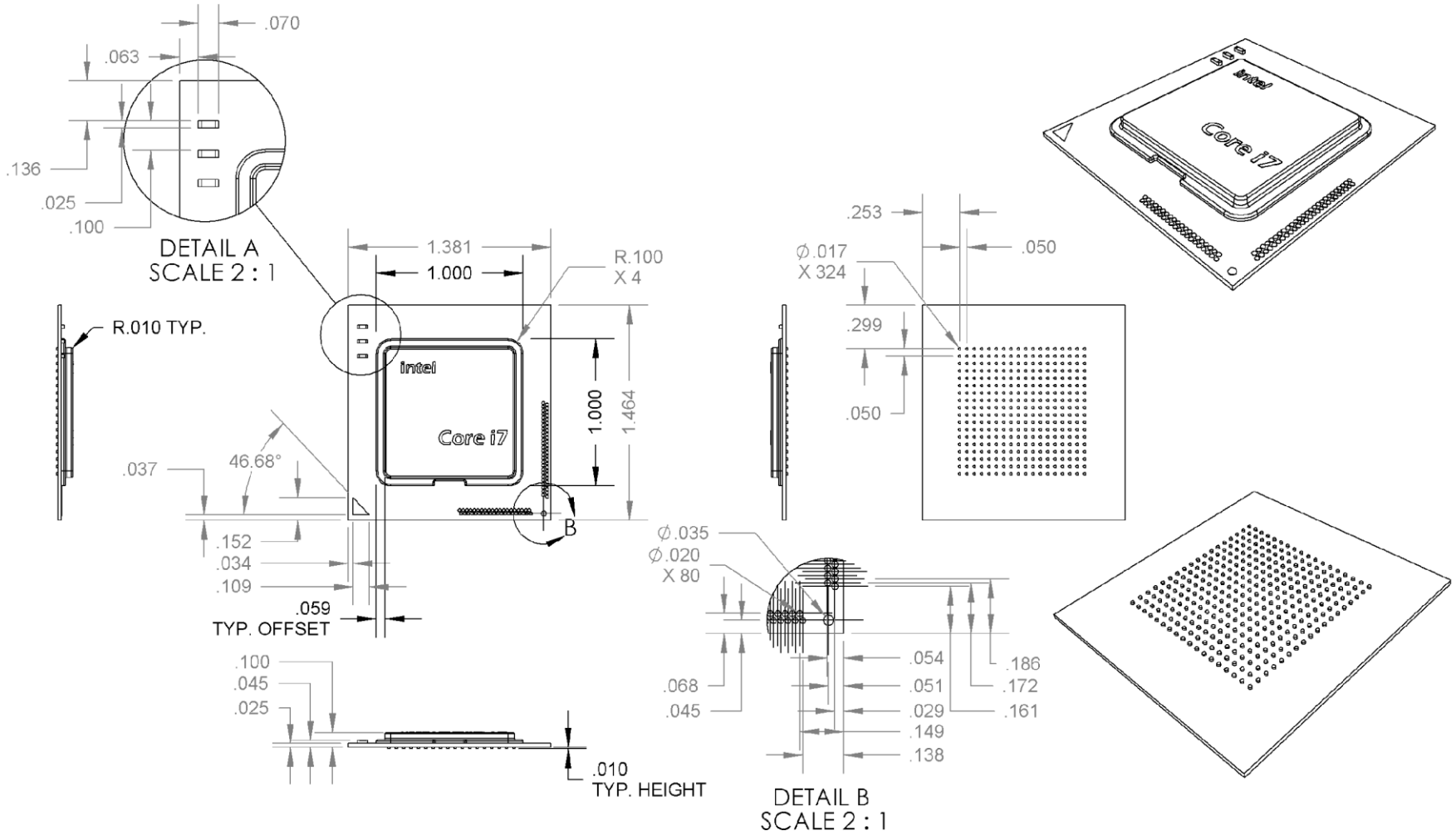


9. Select the “Unbend” icon then select cut edge of the part as the fixed edge.



10. Select the Flatten icon to unfold.





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

TITLE:

LAB 22

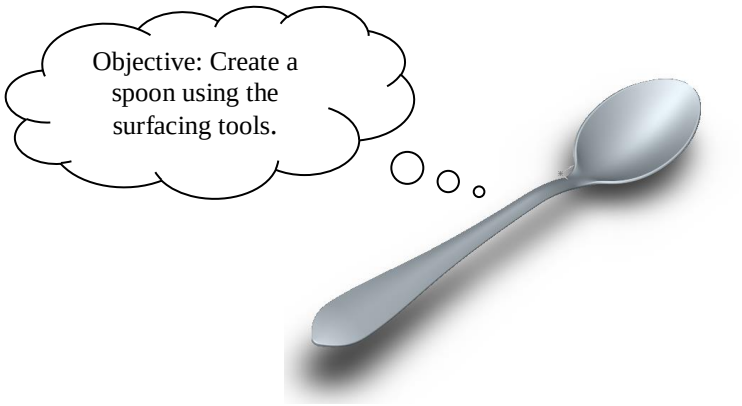
SIZE	DWG. NO.	REV
A	Core i7 920	
SCALE: 1:1	WEIGHT:	SHEET 1 OF 1

5 4 3 2 1

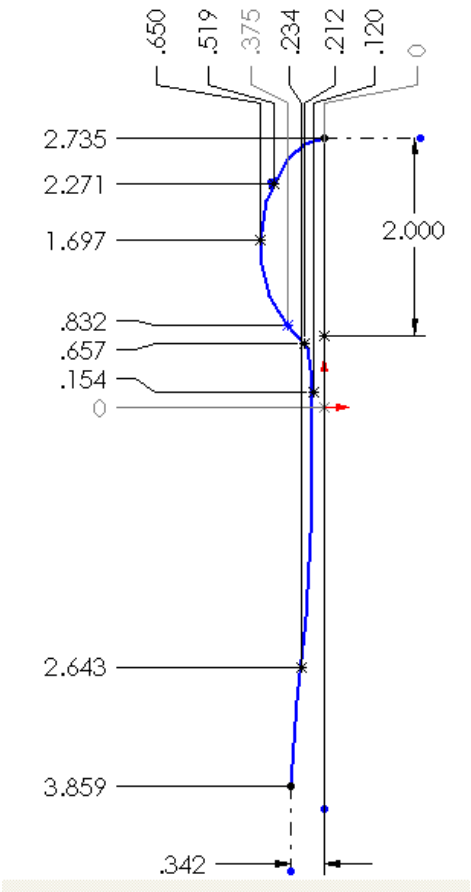
EXERCISE 21

Introduction to Surfacing

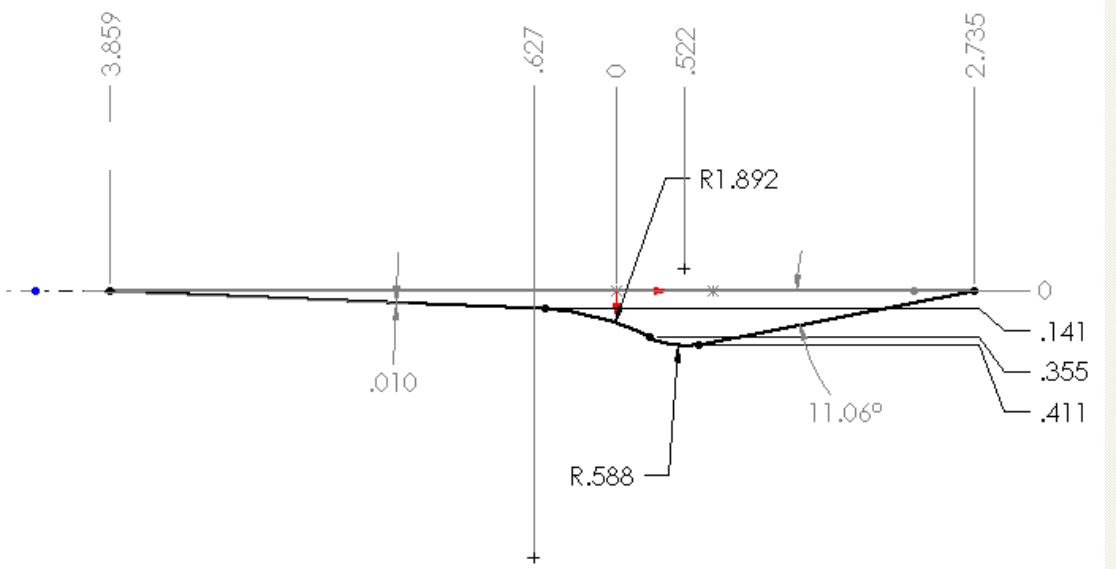
1. Here is an example of how to use surfaces. The spoon model will be used to introduce the user to the primary surfacing tools available in Creo.



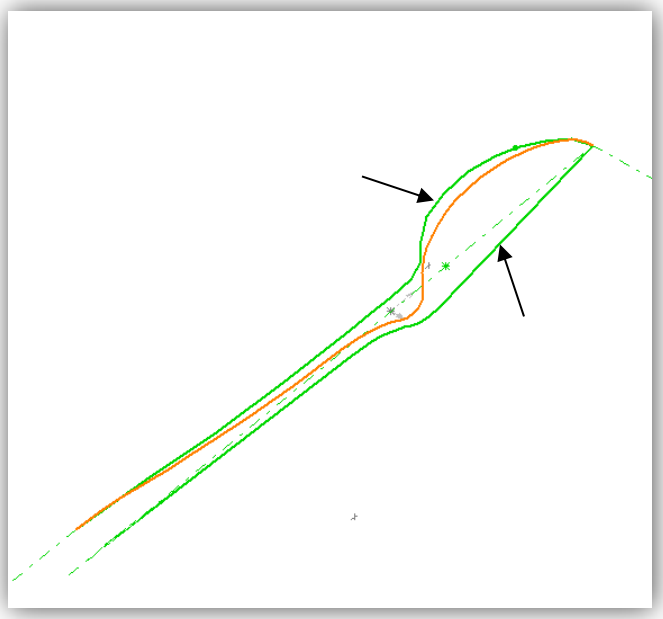
2. Sketch the following on the "Front" plane.



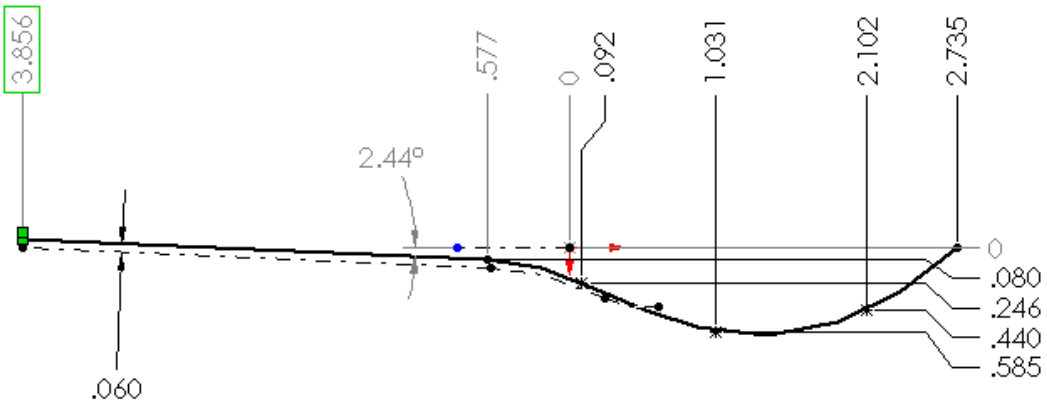
3. Sketch the following on the “Right” plane.



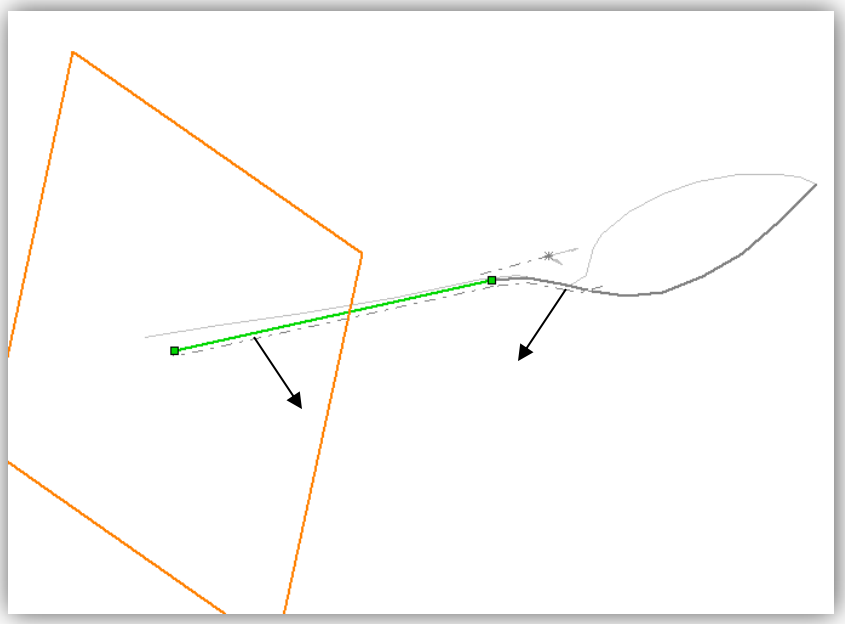
4. Go to Insert/Curve/Projected and select the two sketches to create a 3D projected curve. Change the feature name to “Guide”.



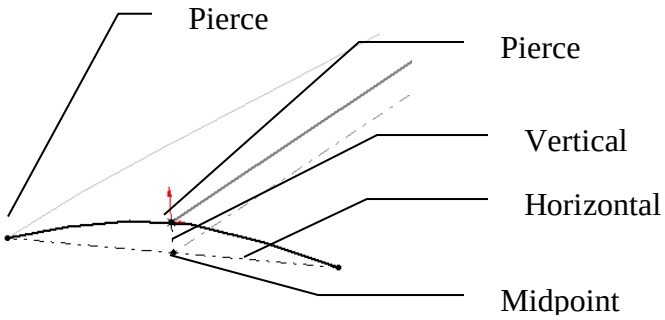
5. Draw the “Path” on the right plane.



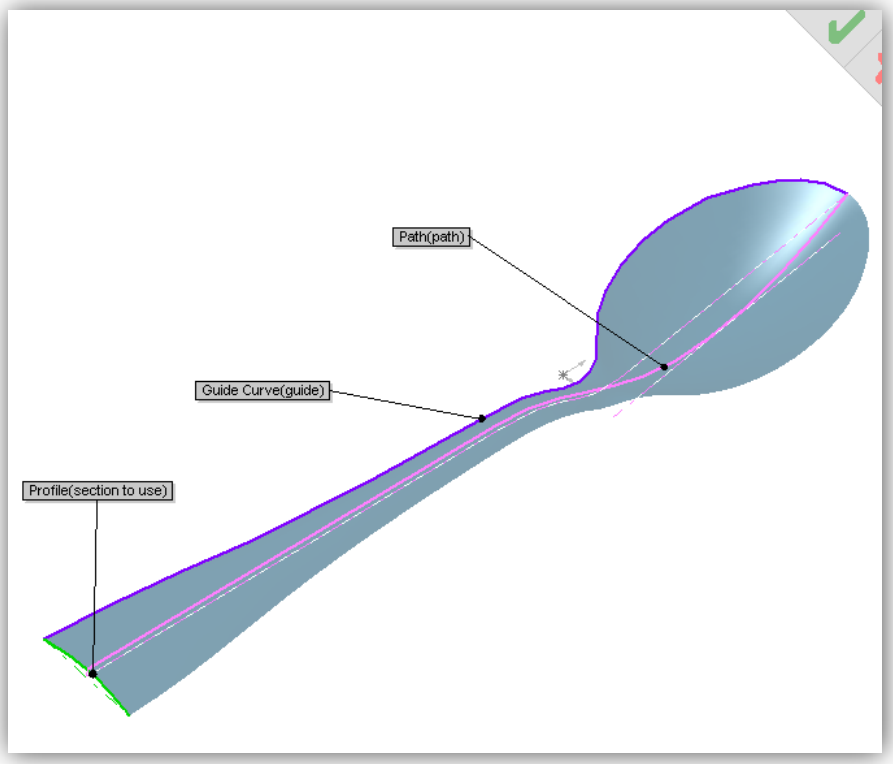
6. Create a new plane Normal to curve.



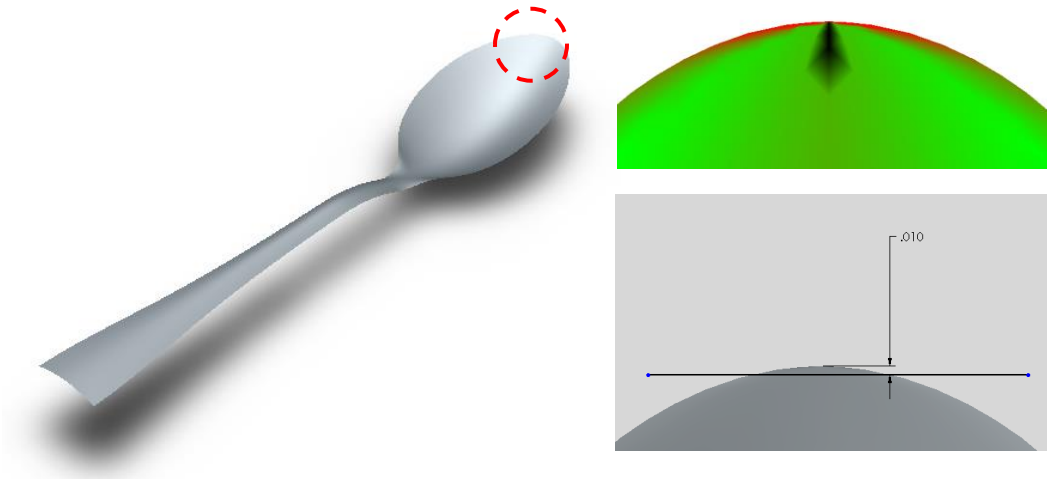
7. Start a sketch on the new plane and draw the following using a spline. Use relations to constrain the sketch.



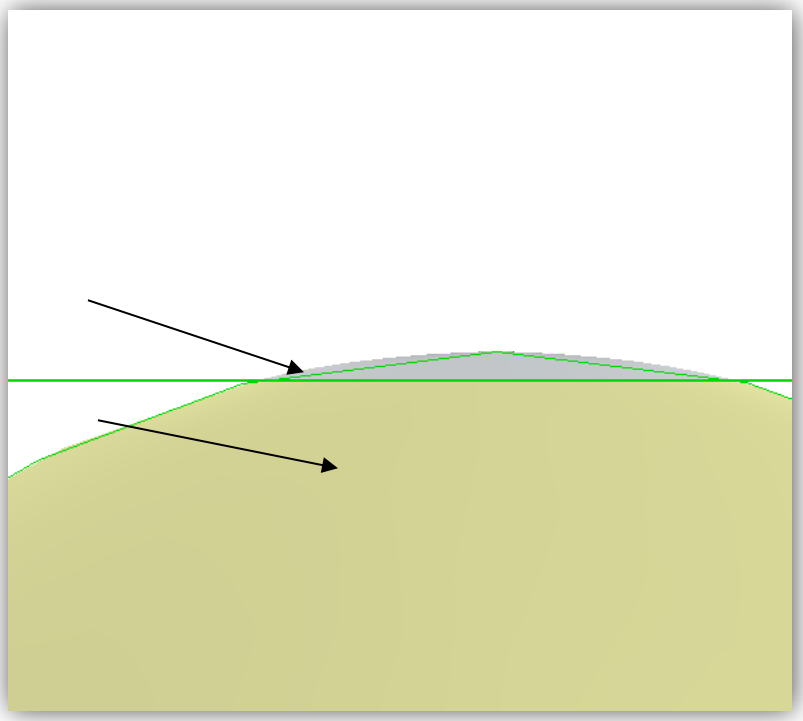
8. Add a surface sweep.



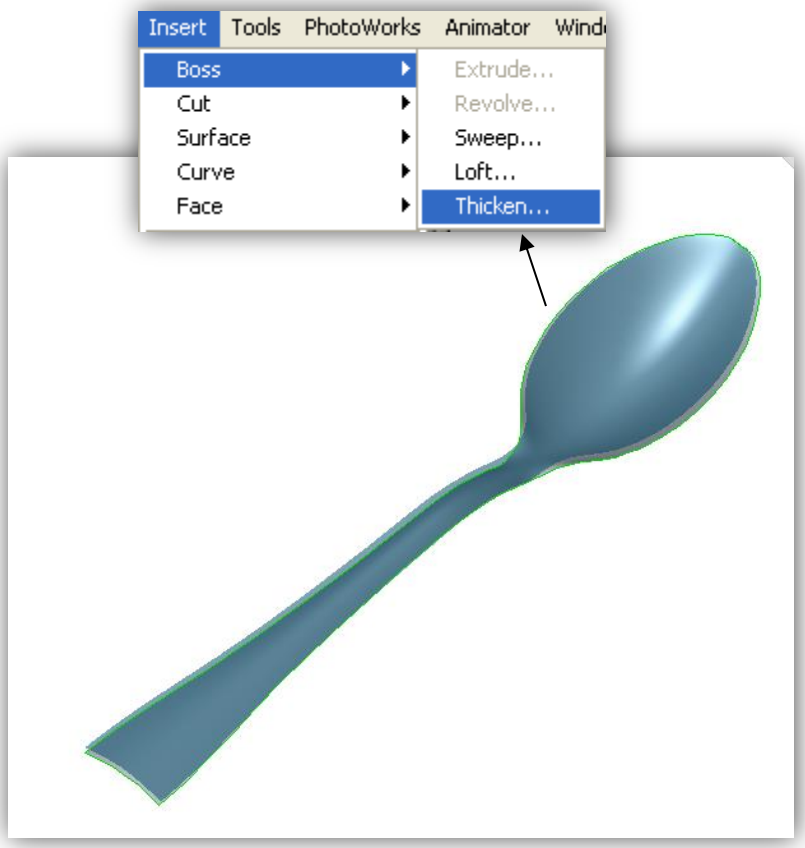
9. Check for surface flaws by right mouse clicking on the surface and select display curvature. Trim off the bad end by sketching a line on the front plane.



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 25

Industrial Design Project

1. **Design an original coffee lid.**

Part design for Thermoforming or Vacuum Forming

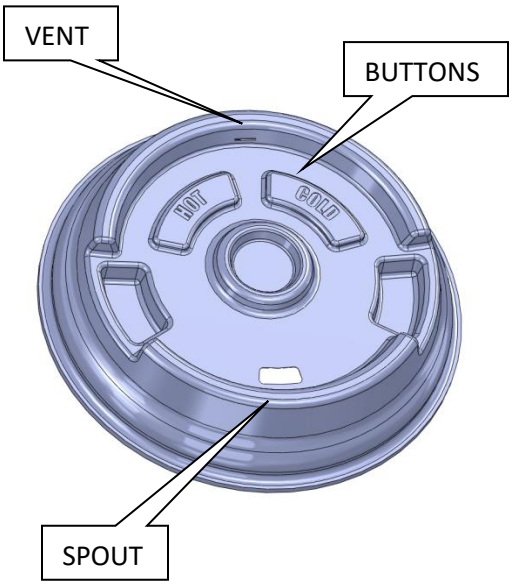
What is thermoforming?

Thermoforming is a manufacturing process for thermoplastic sheet or film. Specifically, it is more of a converting process, where plastic sheet or film is converted into a formed,

finished part. The sheet or film is heated in an oven to its forming temperature, then stretched into or onto a mold and cooled. Early generation thermoforming machines usually incorporated cal-rod type heaters, similar to heating elements found in conventional electric kitchen ovens. These are still used, but more modern equipment frequently uses quartz heaters or radiant-panel heaters for more efficient sheet heating and ease of zone control. Cast or machined aluminum is the most common mold material, although epoxy, wood and structural foam are sometimes used for prototypes, samples, and low volume production runs. Aluminum molds are normally water-cooled by a cooling tower or chiller system for faster production capabilities. Thermoforming differs from injection molding, blow molding, rotational molding, and other forms of processing plastics, and is primarily used in the manufacture of disposable cups, containers, lids, trays, blisters, clamshells, and other products. A thermoform machine can utilize vacuum only, or vacuum combined with air pressure, in the forming process. It can be as small and simple as a table-top sample former where small cut sheets of material are placed into a clamp and heated and formed, or as large and complex as a complete inline extrusion, thermoforming, trimming, granulating, and material handling system for continuous high-speed production. Many thermoforming companies do not extrude their own plastic sheet, but rather purchase it in roll-wound form for running on their forming equipment. Others purchase plastic resin in bulk pellet form and extrude the sheet for use on roll-fed or inline forming machines. (source:wikipedia)

STAGES IN PROCESS

1. Hand sketch a concept.
2. Choose the best concept for production
3. Measure and Draw the profiles
4. 3D model the Coffee Lid
5. Create production drawing

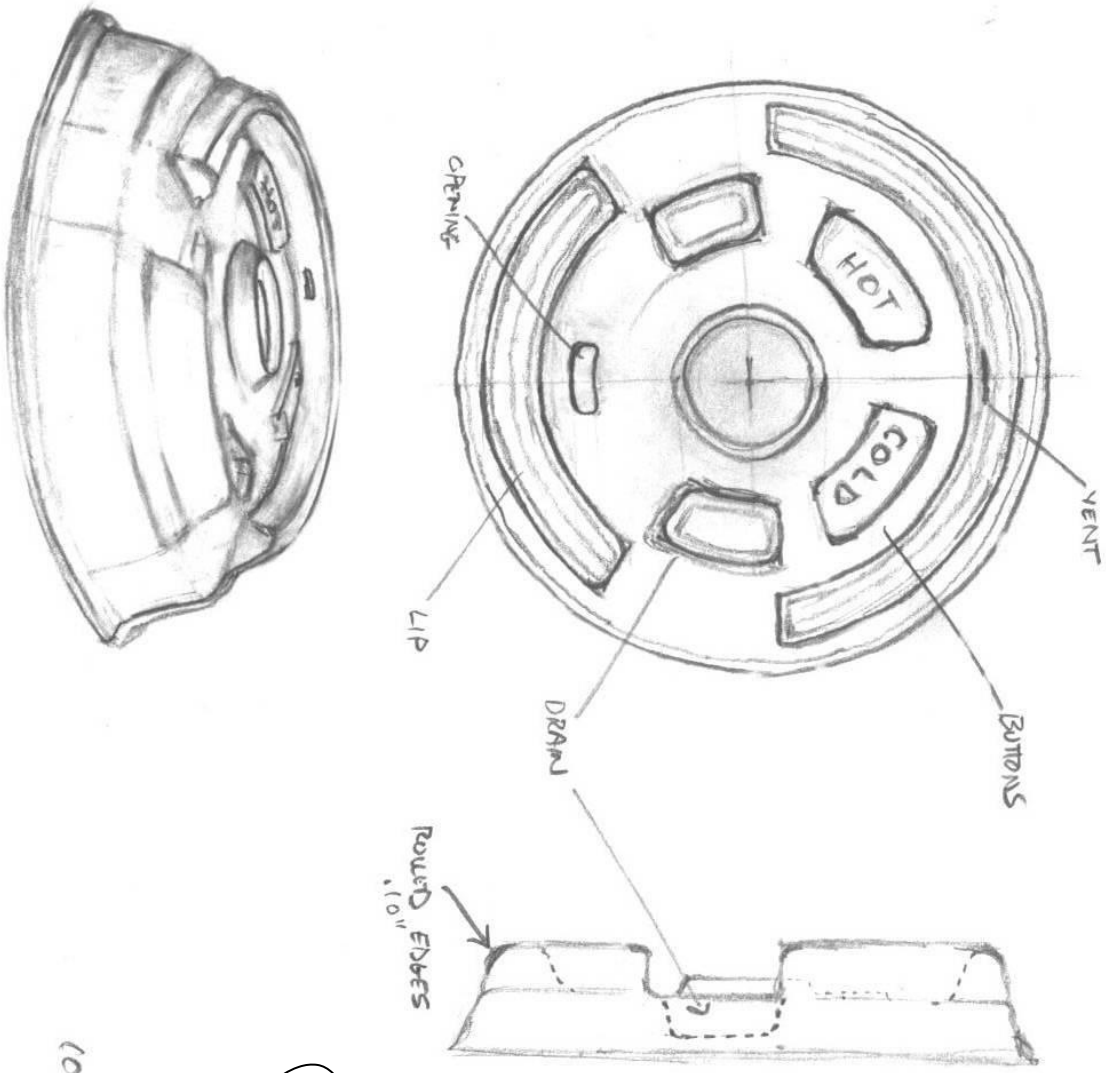


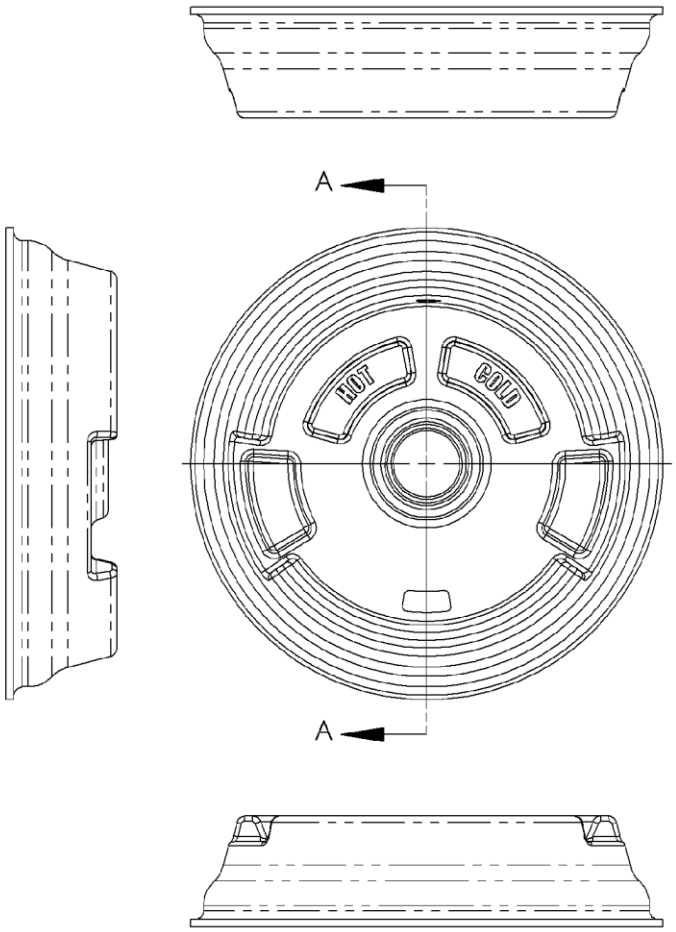
Once complete turn in all materials…

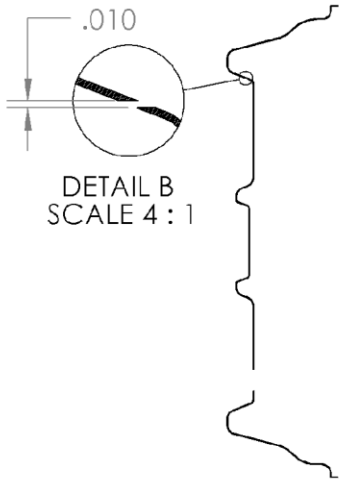
- 1 - Hand Sketch
- Creo model
- Creo drawing (print out copy)
- 3D Print Model (if available)
- Prepare brief summary of design and reasons for designing it the way you did.

You will be graded on the quality of your work and the level of detail used.

HAND SKETCH

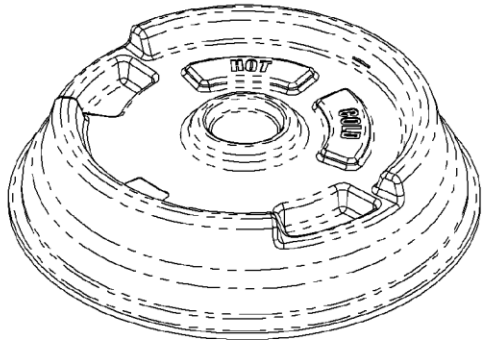






DETAIL B
SCALE 4 : 1

SECTION A-A



SAMPLE

NOTE: USE .020" MATERIAL THICKNESS

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

TITLE:

COFFEE LID

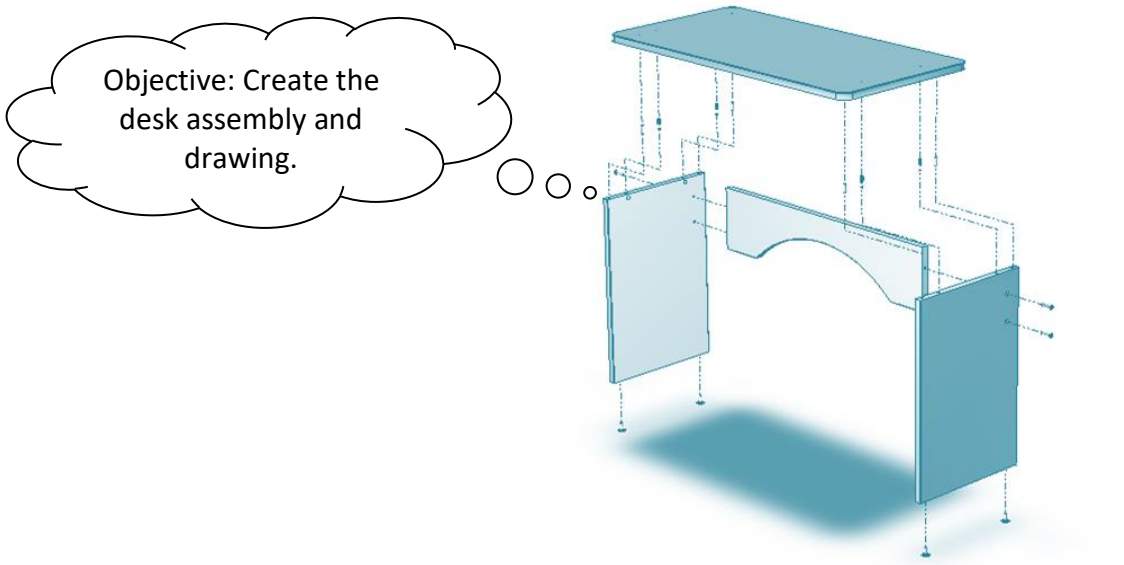
SIZE	DWG. NO.	REV
A	DESIGN PROJECT LID	
SCALE: 1:2	WEIGHT:	SHEET 1 OF 1

5 4 3 2 1

EXERCISE 26

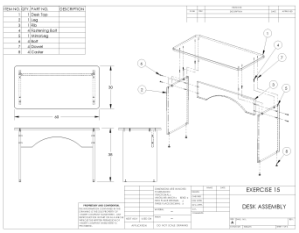
Assembly and Drawing Automation

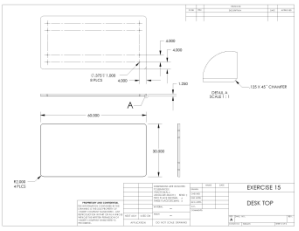
Assembly and Drawing *creation can be virtually automated through the use of many techniques capable in the Pro Engineer software.*

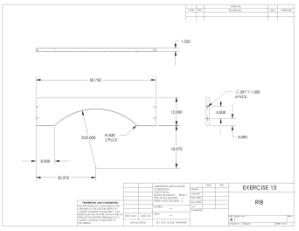


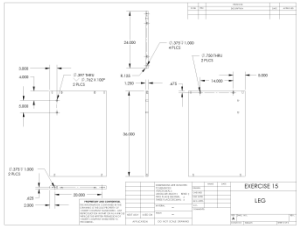
1. Create an E24 directory; use this folder to store all your parts for this exercise.

2. Now start a new part file and begin to create the attached parts.
   NOTE: The drawings are missing some dimensions.

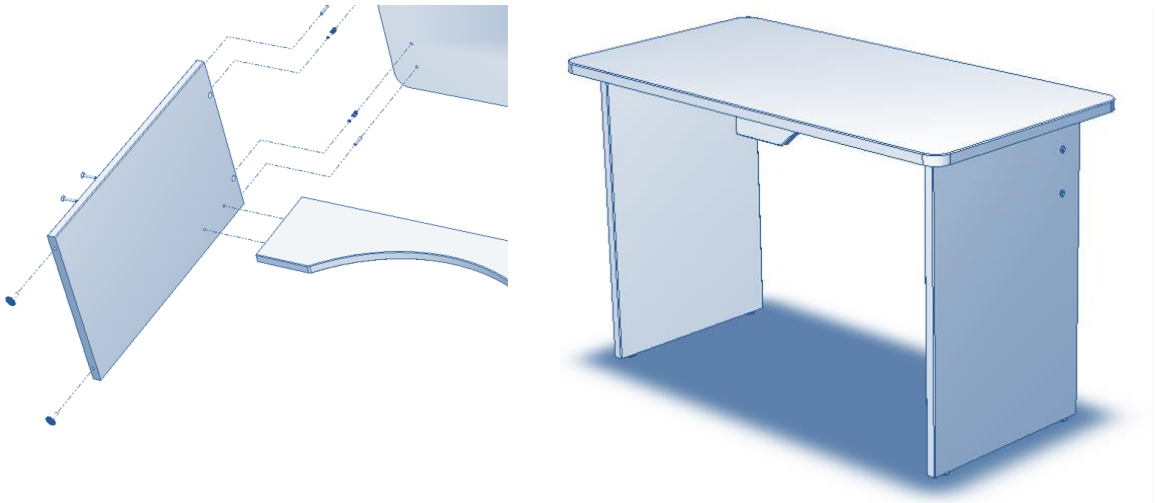




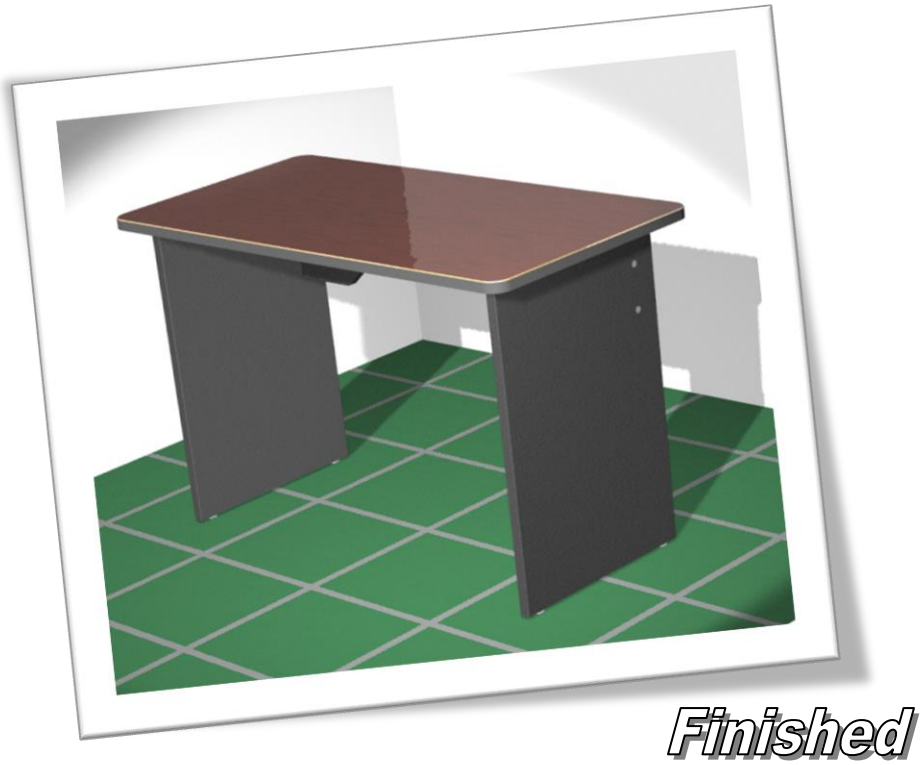


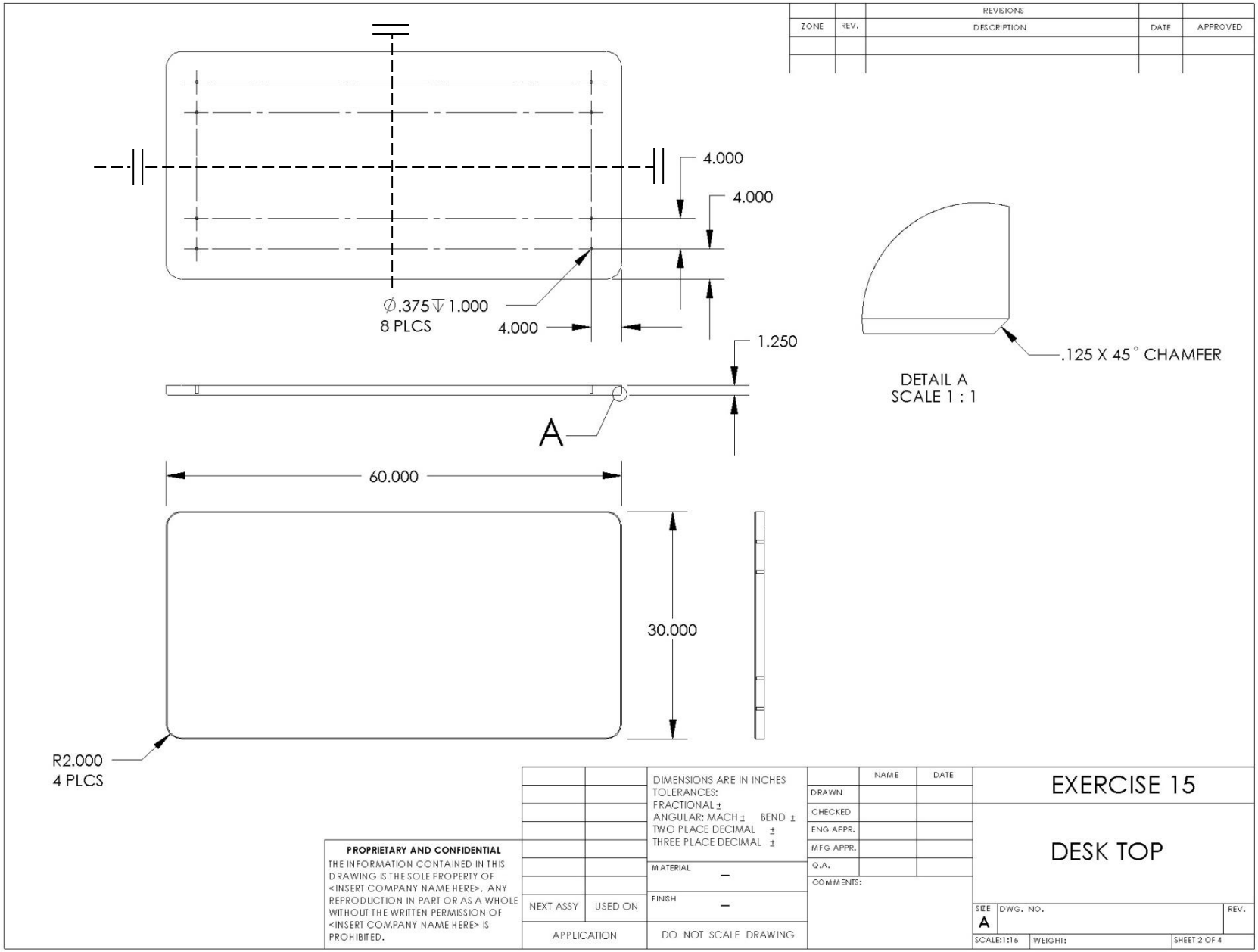


3. When all the parts are finished, start an assembly and begin to assemble the components as shown in the assembly drawing provided.

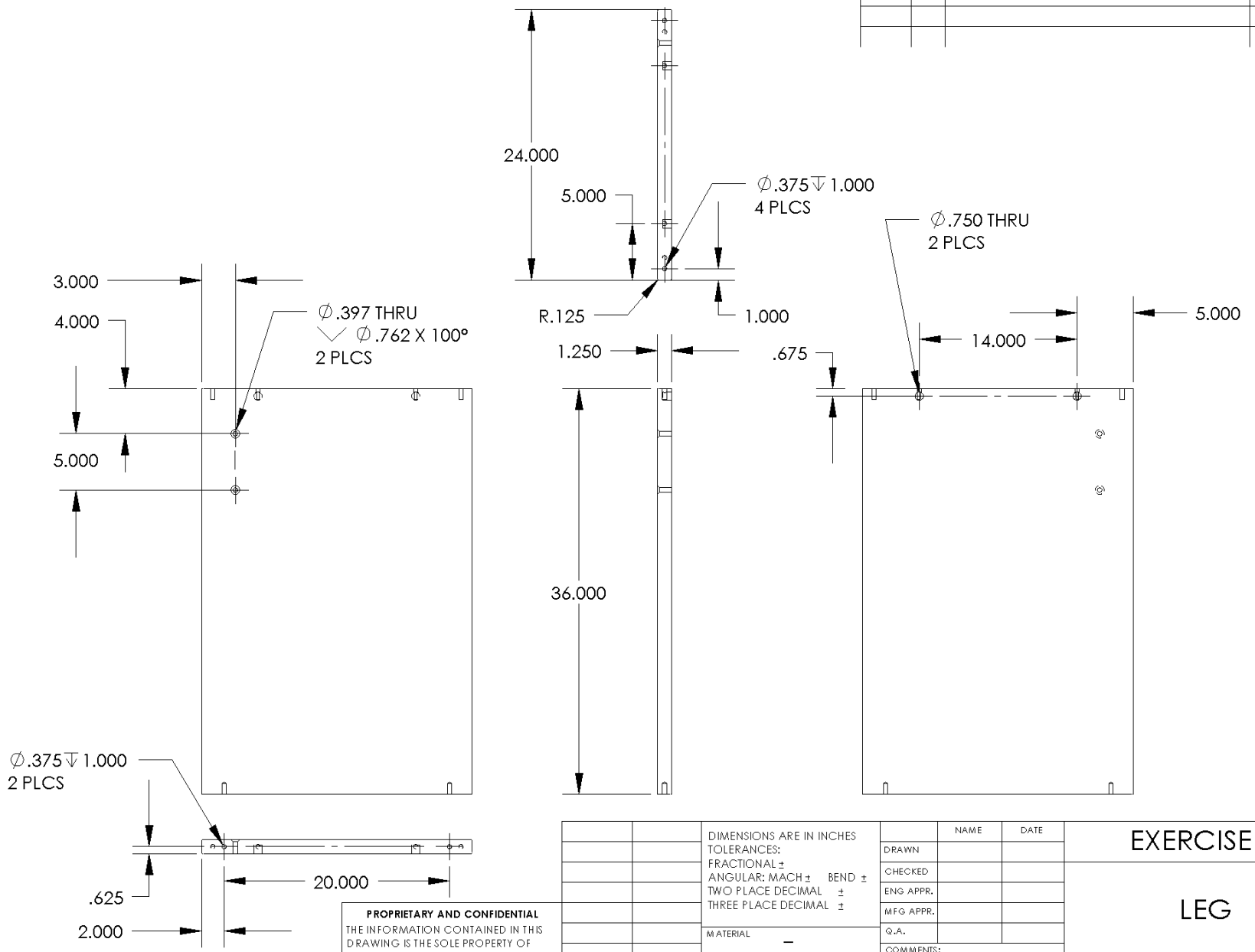


4. Create a catalog image using Creo Photo Rendering utility. This is his how the finished model should appear after rendering.




REVISIONS
ZONE
REV.
DESCRIPTION
DATE
APPROVED
4.000
4.000
Ø.375 ↧ 1.000
8 PLCS
4.000
1.250
A
60.000
30.000
R2.000
4 PLCS
DETAIL A
SCALE 1 : 1
.125 X 45° CHAMFER
DIMENSIONS ARE IN INCHES
TOLERANCES:
FRACTIONAL ±
ANGULAR: MACH ± BEND ±
TWO PLACE DECIMAL ±
THREE PLACE DECIMAL ±
MATERIAL —
FINISH —
DO NOT SCALE DRAWING
NEXT ASSY
USED ON
APPLICATION
NAME
DATE
DRAWN
CHECKED
ENG APPR.
MFG APPR.
Q.A.
COMMENTS:
EXERCISE 15
DESK TOP
SIZE A
DWG. NO.
REV.
SCALE:1:16
WEIGHT:
SHEET 2 OF 4
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.

REVISIONS				
ZONE	REV.	DESCRIPTION	DATE	APPROVED



NEXT ASSY	USED ON
APPLICATION	

DIMENSIONS ARE IN INCHES
TOLERANCES:
FRACTIONAL ±
ANGULAR: MACH ± BEND ±
TWO PLACE DECIMAL ±
THREE PLACE DECIMAL ±

MATERIAL —

FINISH —

DO NOT SCALE DRAWING

	NAME	DATE
DRAWN		
CHECKED		
ENG APPR.		
MFG APPR.		
Q.A.		

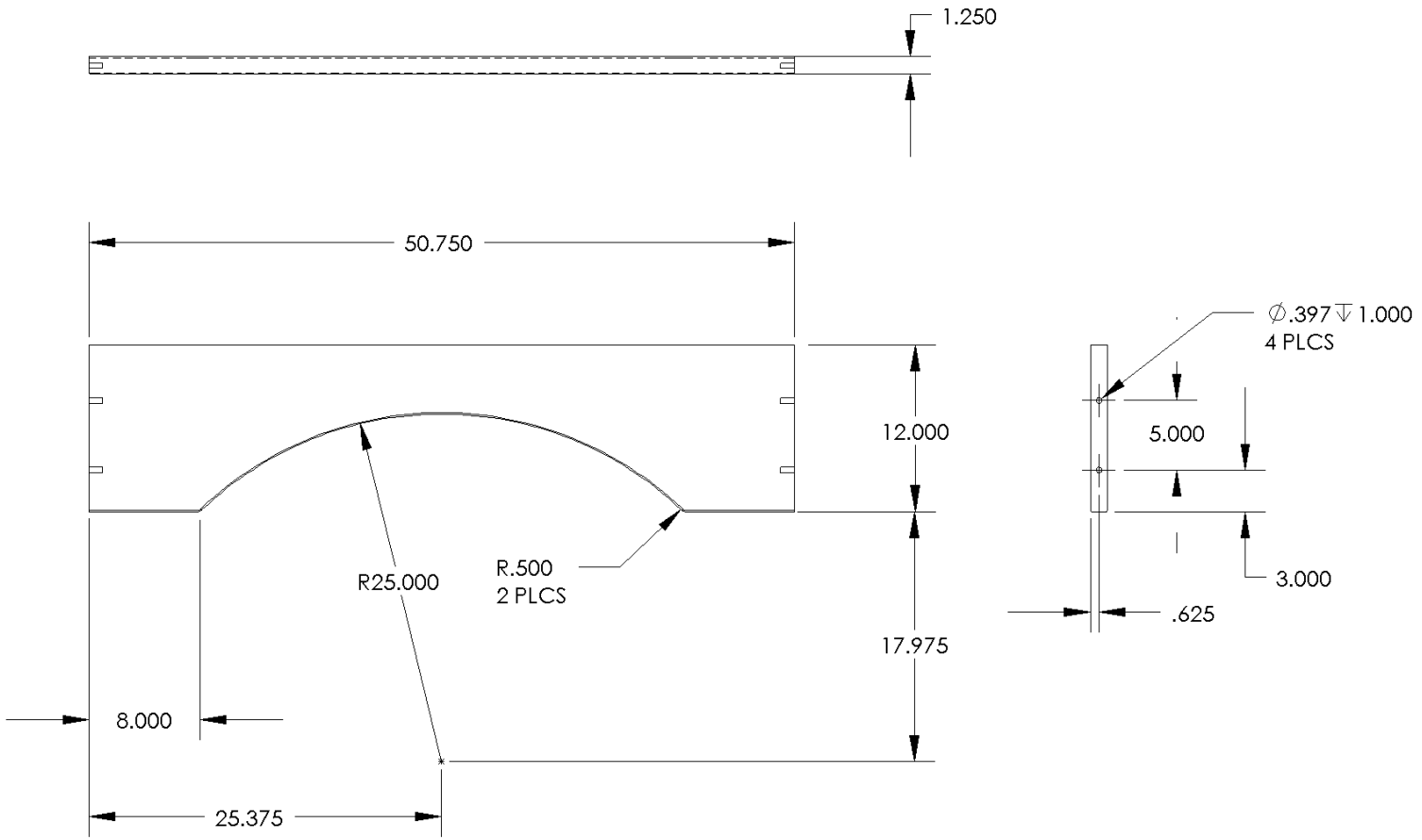
COMMENTS:

EXERCISE 20

LEG

SIZE	DWG. NO.	REV.
A		
SCALE:1:12	WEIGHT:	SHEET 3 OF 4

ZONE	REV.	REVISIONS DESCRIPTION	DATE	APPROVED



NEXT ASSY	USED ON
APPLICATION	

DIMENSIONS ARE IN INCHES
TOLERANCES:
FRACTIONAL ±
ANGULAR: MACH ± BEND ±
TWO PLACE DECIMAL ±
THREE PLACE DECIMAL ±

MATERIAL —

FINISH —

DO NOT SCALE DRAWING

	NAME	DATE
DRAWN		
CHECKED		
ENG APPR.		
MFG APPR.		
Q.A.		

COMMENTS:

EXERCISE 20

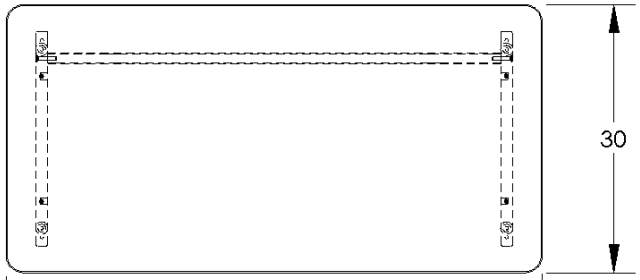
RIB

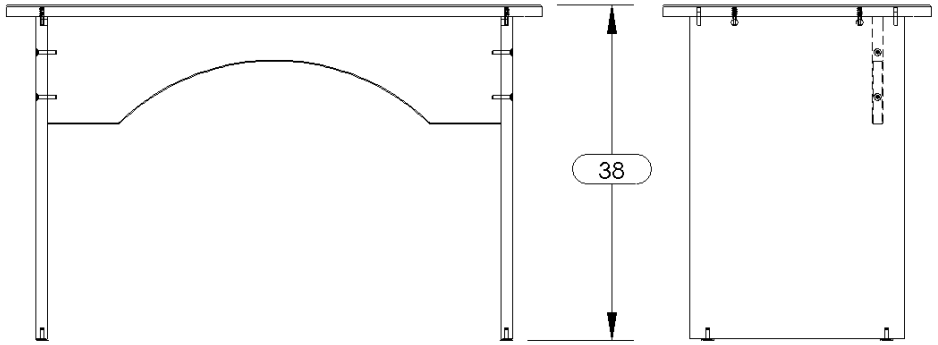
SIZE	DWG. NO.	REV.
A		

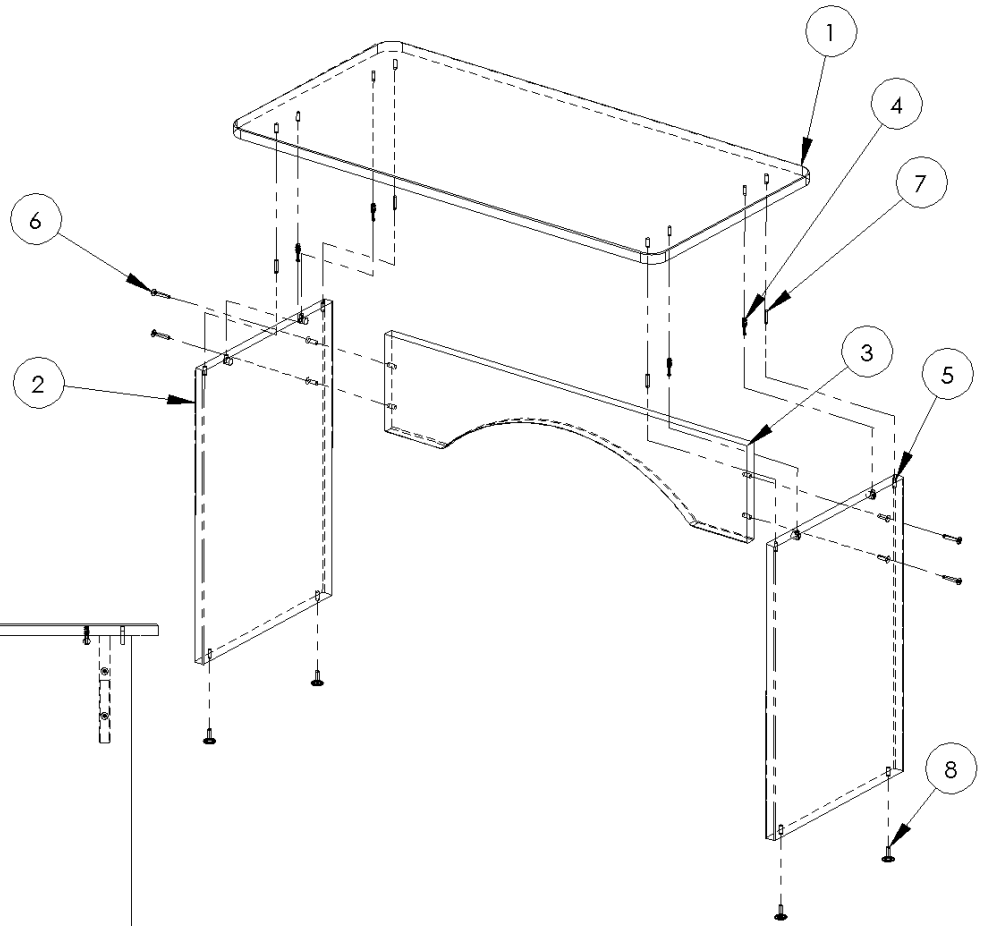
SCALE:1:12 | WEIGHT: | SHEET 4 OF 4

ITEM NO.	QTY.	PART NO.	DESCRIPTION
1	1	Desk Top	
2	1	Leg	
3	1	Rib	
4	4	Fastening Bolt	
5	1	MirrorLeg	
6	4	Bolt	
7	4	Dowel	
8	4	Caster	

		REVISIONS		
ZONE	REV.	DESCRIPTION	DATE	APPROVED







NEXT ASSY	USED ON
APPLICATION	

DIMENSIONS ARE IN INCHES
TOLERANCES:
FRACTIONAL ±
ANGULAR: MACH ± BEND ±
TWO PLACE DECIMAL ±
THREE PLACE DECIMAL ±

MATERIAL —

FINISH —

DO NOT SCALE DRAWING

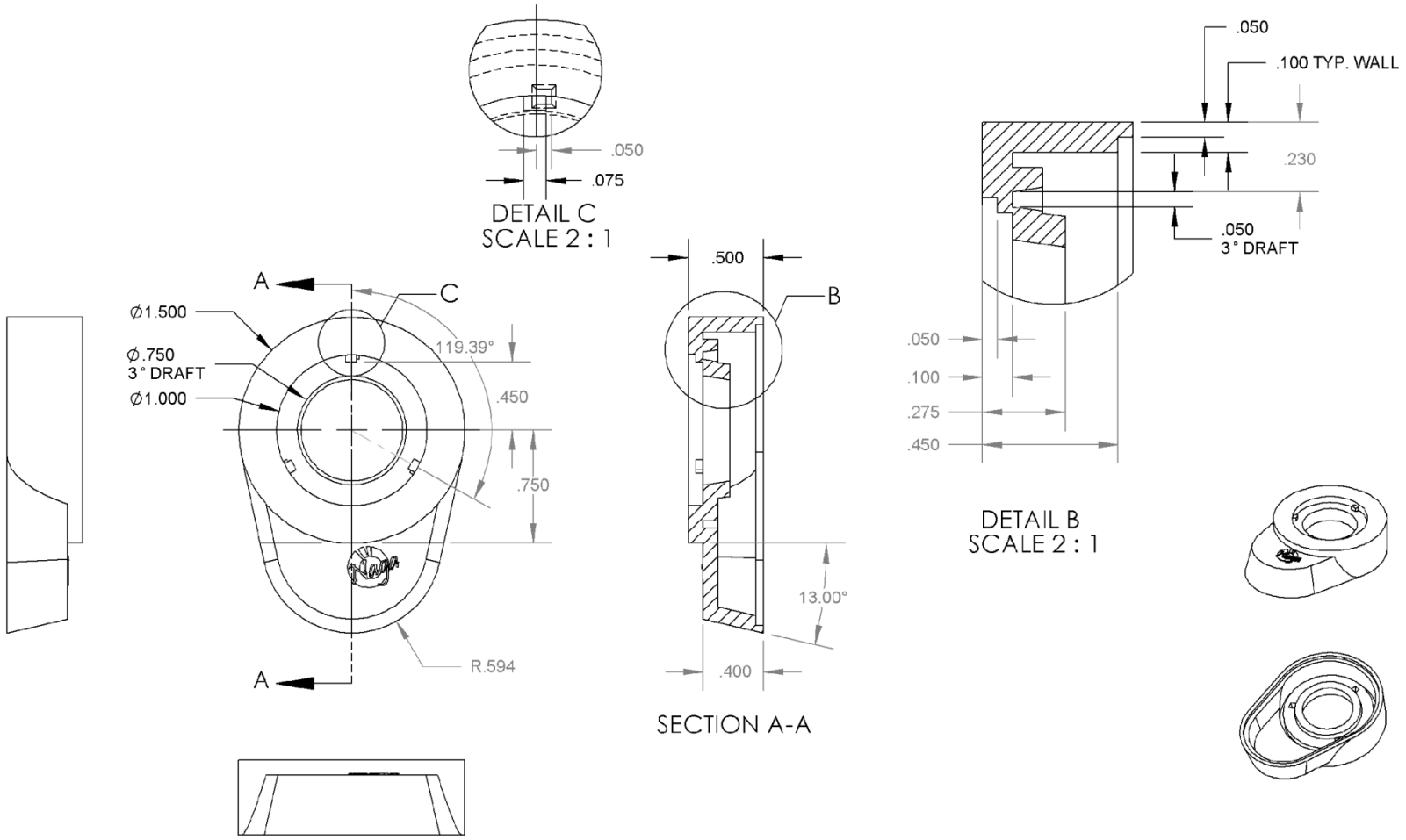
	NAME	DATE
DRAWN		
CHECKED		
ENG APPR.		
MFG APPR.		
Q.A.		

COMMENTS:

EXERCISE 20

DESK ASSEMBLY

SIZE	DWG. NO.	REV.
A		
SCALE:1:20	WEIGHT:	SHEET 1 OF 4



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

TITLE: LAB 23

SIZE	DWG. NO.	REV
A	NZ-3	

SCALE: 1:1	WEIGHT:	SHEET 1 OF 1

5 4 3 2 1

SUPPLIMENTAL

Creo - CAD Administration

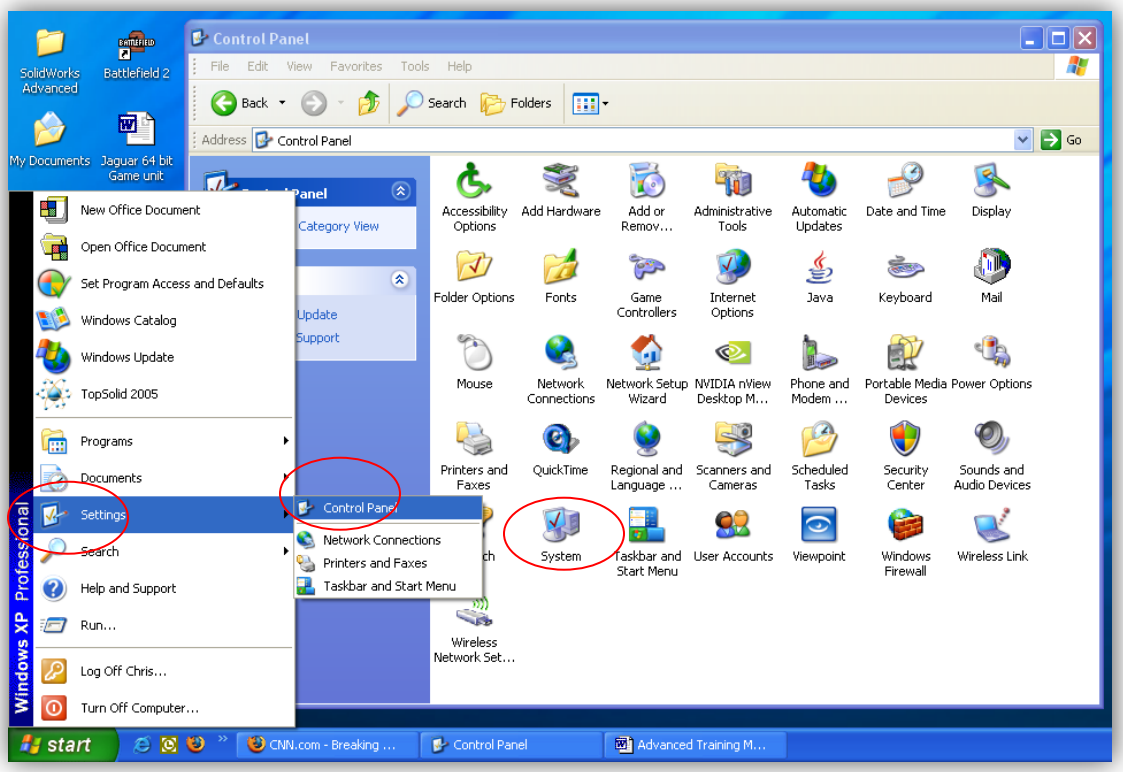
***Finding adequate computer hardware to run Creo can be challenging**, this lesson looks at the multiple aspects of selecting hardware as well as modifying settings inside Creo to allow it to run efficiently and trouble free.*

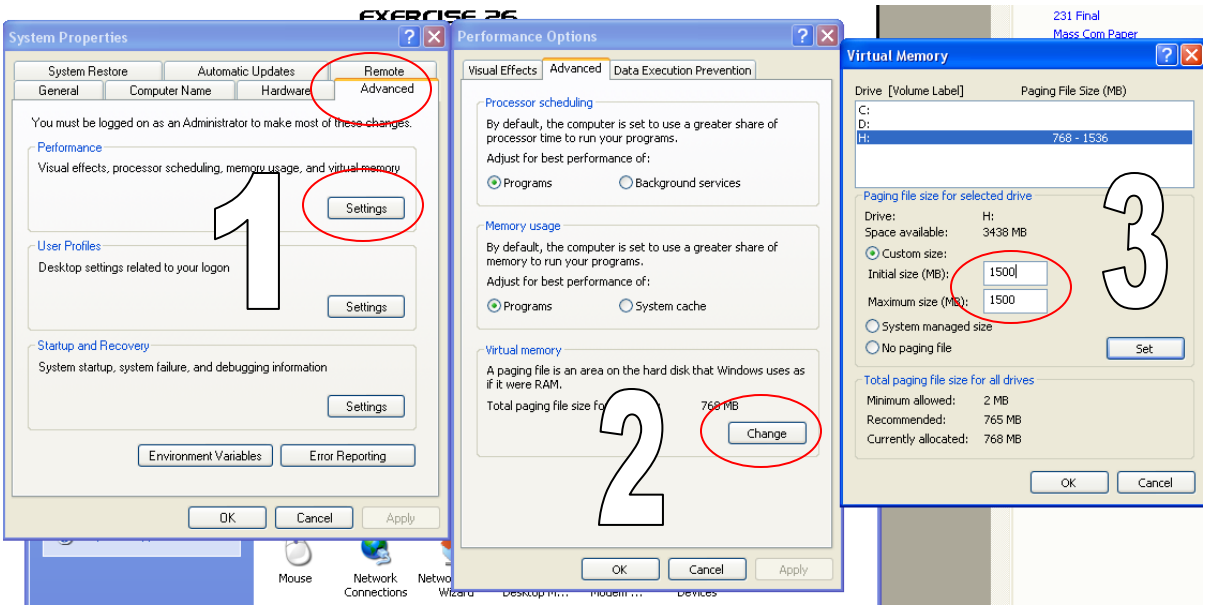
Selecting an Operating System (OS).

Windows 7 64-bit
Windows 8 64-bit
Windwos 10 64-bit

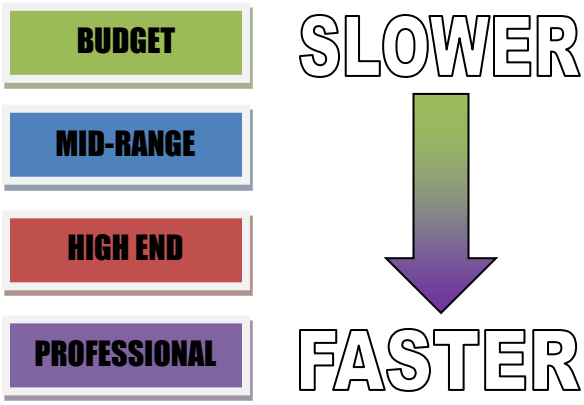
Virtual Memory Settings inside the OS. It may be a good idea to increase or adjust your virtual memory setting. The norm would be x2 – x3 your current amount of ram.

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

- Atom
- Celeron
- Pentium
- Core i3
- Core i5
- Core i7
- Xeon

AMD

- Sempron
- Athlon II
- Phenom X2,3,4,6
- VISION A4,6,8,10
- FX Series
- Opteron

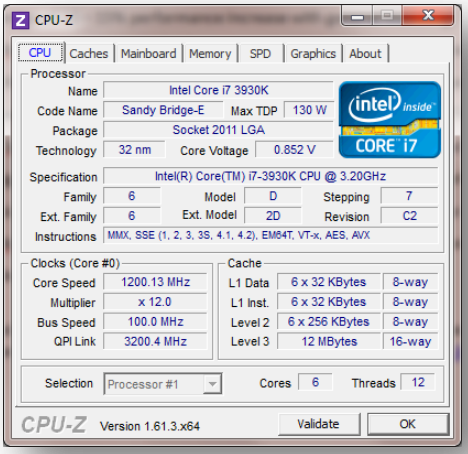
Multiprocessing

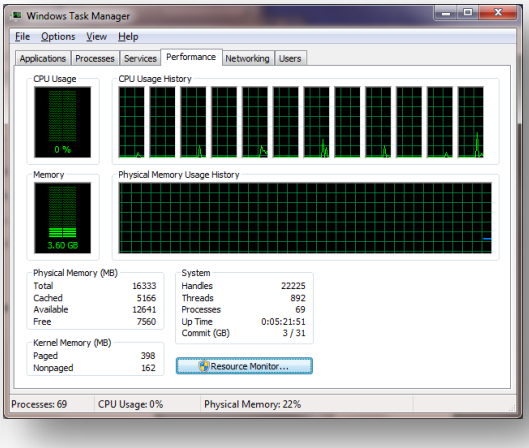
Most CPU manufacturers are beginning to deliver multiple core processors. This can be seen with the AMD FX which has up to eight processing cores.

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

The question is: “Can Creo benefit from multiple cores?” Currently one might find an average of 10 – 15% performance increase with general modeling. This is because Creo is not fully written to take advantage of multithreaded processes. However, using the Creo Simulation, CFD, or Photoreal rendering solutions one may discover 2x – 12x faster performance versus a single core processor. This is because these Creo 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 Creo 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.





Graphics Cards

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

- **NVIDIA *Quadro*** series (not NVS series)
    - **Quadro P600**
    - **Quadro P2000**
    - **Quadro P4000**

- **ATI *FirePro*** series (not FireMV series)
    - **FirePro w2100**
    - **FirePro w5100**
    - **FirePro w7100**

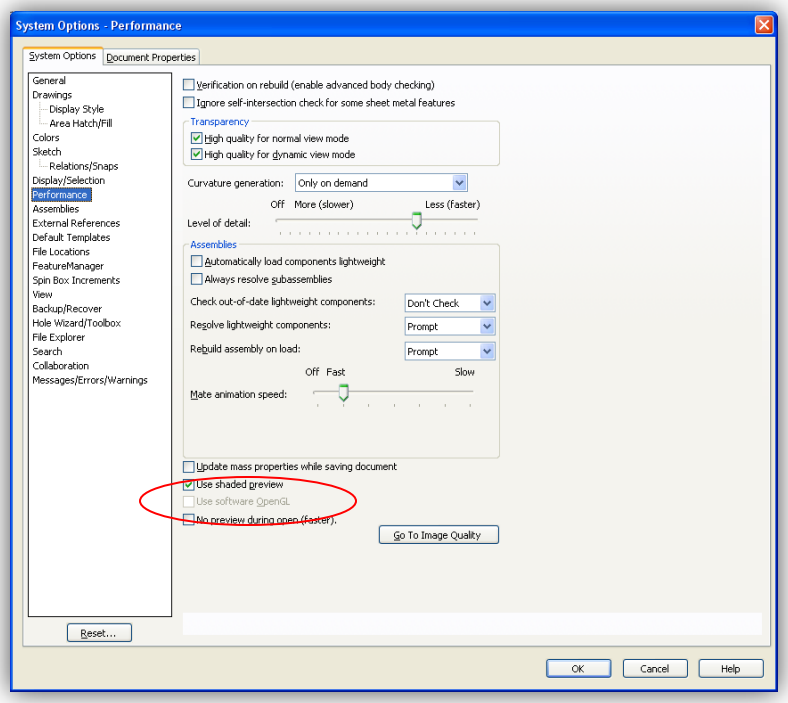
- **Intel *Xeon***
    - **P7**000 *HD* integrated graphics *("P" = Professional rated)*

These cards are considerably more expensive that mainstream cards but the benefits of experiencing less crashes or visual problems with Pro/E outweigh the cost.

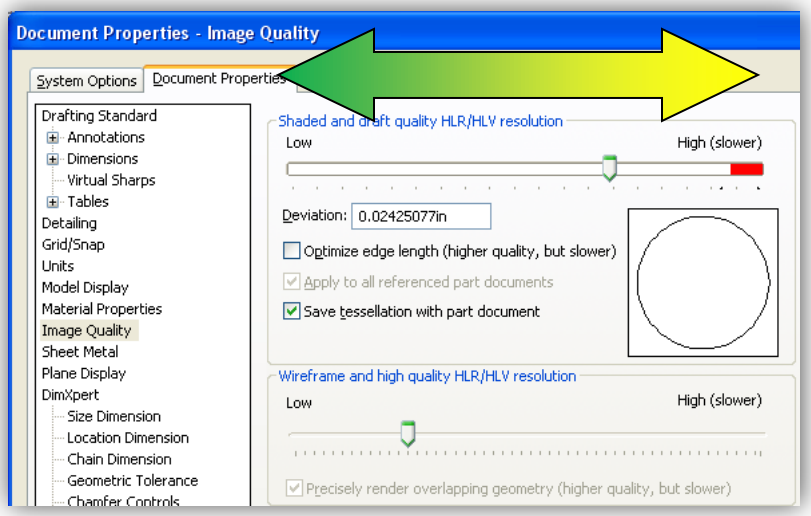
If you are using Creo at work, DON'T SKIMP! Buy a professional grade video card. For home use the nVidia Geforce or AMD Radeon series are fair, but you will still experience some graphical glitches.

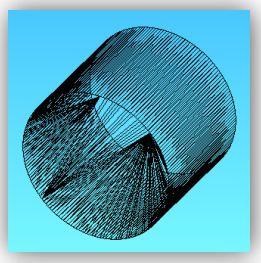
GRAPHICS CARD – Creo BENCHMARK (source: www.tomshardware.com)

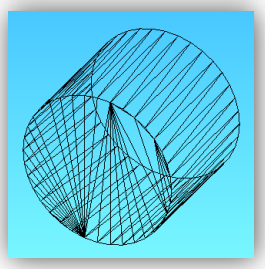
With all Creo documents closed go to Tools/Options/System and turn on software OpenGL.



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







MEMORY (RAM)

4.0 – 16.0 GB From simple machined parts to complex assemblies. The more RAM the better.

3.0 GB+ Requires Windows XP/Vista/7 64 Bit Editions

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.

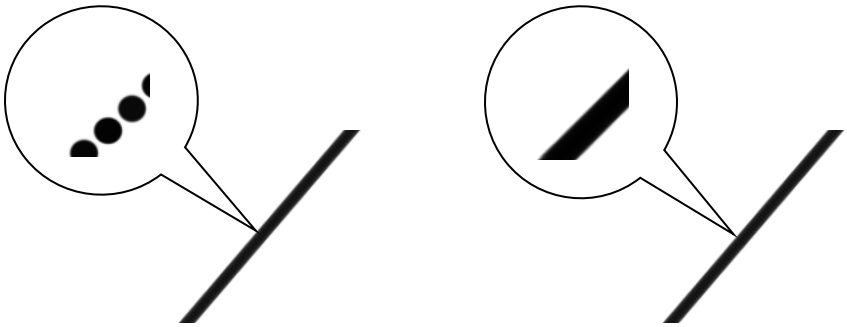
With the growing number of software applications available in the Computer Aided Design, Analysis, and Machining community (CAD/CAE/CAM), one might discover communication between applications to be difficult, because of incompatibilities between each application. Translators have been developed for this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

In the beginning CAD/CAE/CAM applications where 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 translator. 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 NX), 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.



Raster Vector

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 Creo. 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			X	
ADD-INS	DLL	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 Spatials 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 – **D**rawing E**x**change **F**ile. Support for versions AutoCAD R12 – R2011.

IGES – **I**nitial **G**raphics **E**xchange **S**pecification. 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 – **St**andard for the **E**xchange of **P**roduct 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 – **V**erband **d**er **A**utomobilindustrie (German Automotive Specification)

CGR – **C**atia **Gr**aphics format

HCG – Highly **C**ompressed **G**raphics format

CADKEY – Imports native Cadkey part file solids data only.

SolidEdge – Imports native SolidEdge part file solids data only.

UGII – Imports **U**ni**g**raphics (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.