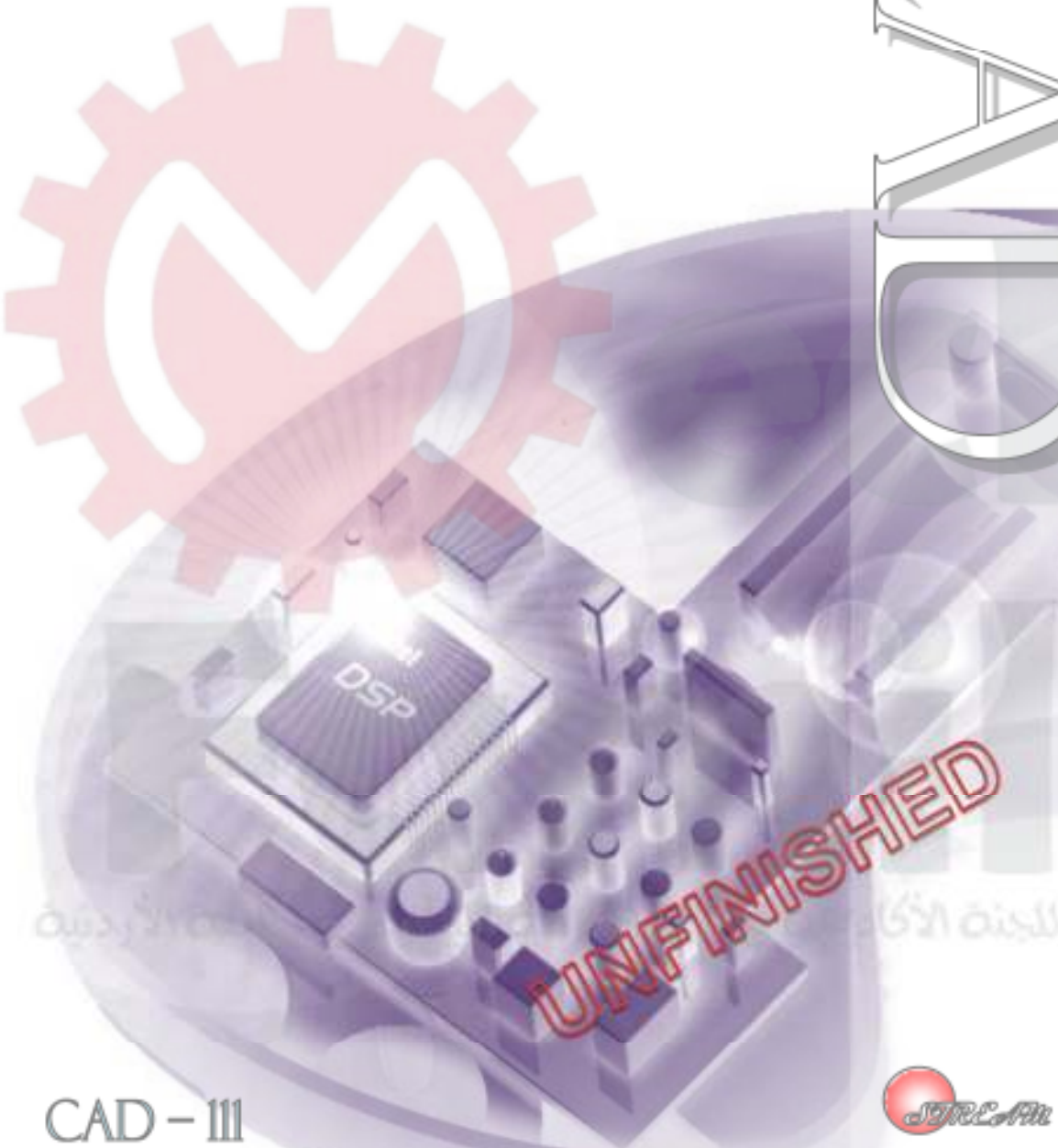
creo PARAMETRIC MODELING ADVANCED

BY CHRISTOPHER F. SIKORA 4-22-13

COMPUTER AIDED DESIGN

CAD



UNFINISHED

CAD – 111

PRO/ENGINEER ADVANCED DESIGN

LECTURE TRAINING STUDY GUIDE

CAD 111 COURSE SYLLABUS

Pro/ENGINEER (Creo 2.0) Advanced

Course Description:

Creo - Pro/E Advanced
3 credit hours
Exploration of the advanced theory and application of solid modeling techniques for product design and manufacturing. Prerequisite: Intro to Engineering Drawings 101 or consent of instructor.
(1 lecture hours, 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

Pro/E Wildfire Advanced (Free/pdf. provided)
Instructional videos of lecture are provided on YouTube vertanux1

Evaluation Scale:

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

Points:

Labs	300 pts
Mid Term	300 pts
Final	300 pts
Participation/Attendance	100 pts
Total	1000 pts

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General Course Outline

Date	Week	Topic
	1.	Import and Export – 2D and 3D Translation,
	2.	Advanced Swept Blends exercise
	3.	Mold Tools, Cast part, complex draft, setback fillets, and draft analysis.
	4.	Family/Design Tables
	5.	Pro/E Administration
	6.	Pro Engineer Relations aka: Equations
	7.	Phone Assembly
	8.	Advanced Sheet Metal Fabrication. Review for Mid Term
	9.	Mid Term Exam
	10.	(No Class) Break
	11.	Cylindrical and Conical Sheet Metal Parts
	12.	Modeling Quiz (game system plastic enclosure)
	13.	IGES Translation & Repair
	14.	Industrial Design Project (Coffee cup lid)
	15.	Desk Assembly. Review for Final
	16.	Final Exam

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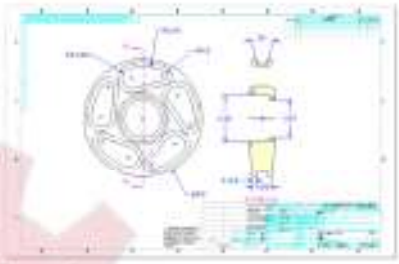
Exercises

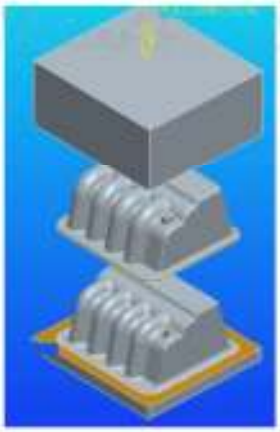
1. Converting 2D DXF/DWG files into 3D models.



2. Swept Blend and Threads

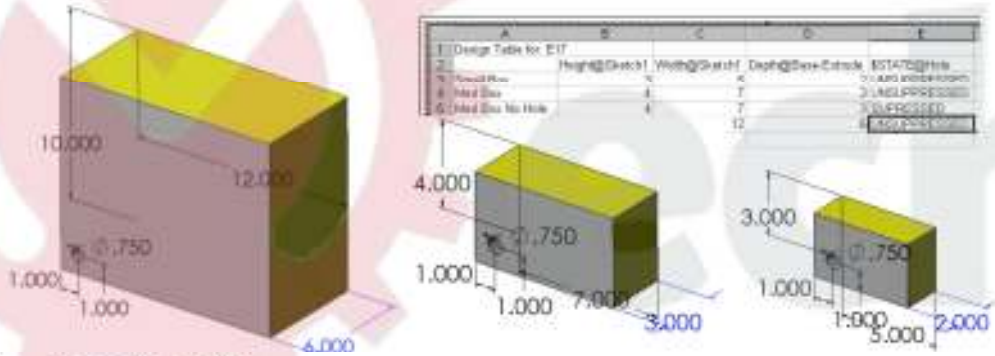
1. Blow Mold Construction

2. Introduction to Pro-Mold module (Cavity and Core creation)



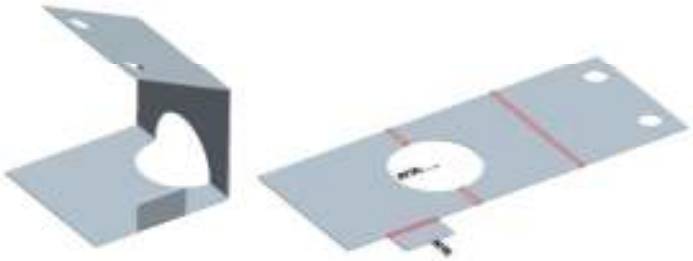
3. Relations: Applying Equations to 3D Models

4. Family Tables



5. Phone Assembly

6. Sheet Metal Methods



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7. Game System Quiz – DWG conversion

11. IGES Repair (Translation)

12. Cylindrical Sheet Metal parts

13. Conical Sheet Metal parts

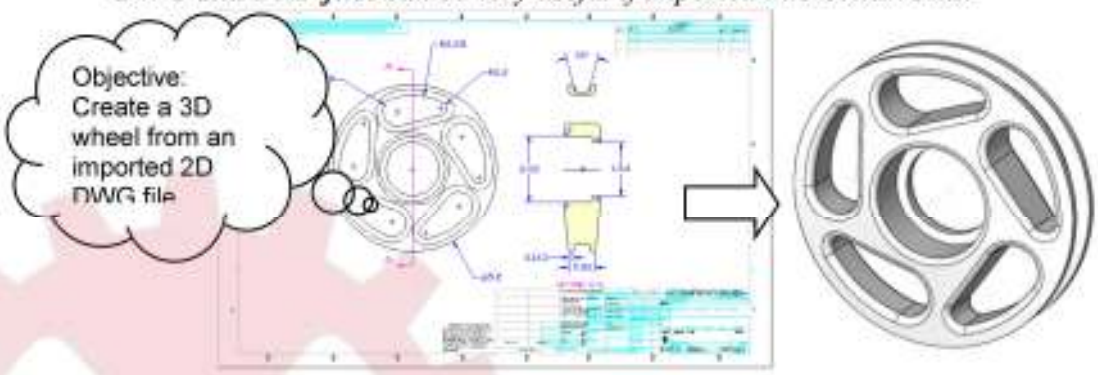
14. Industrial Design Project: Coffee Lid

15. Desk Assembly

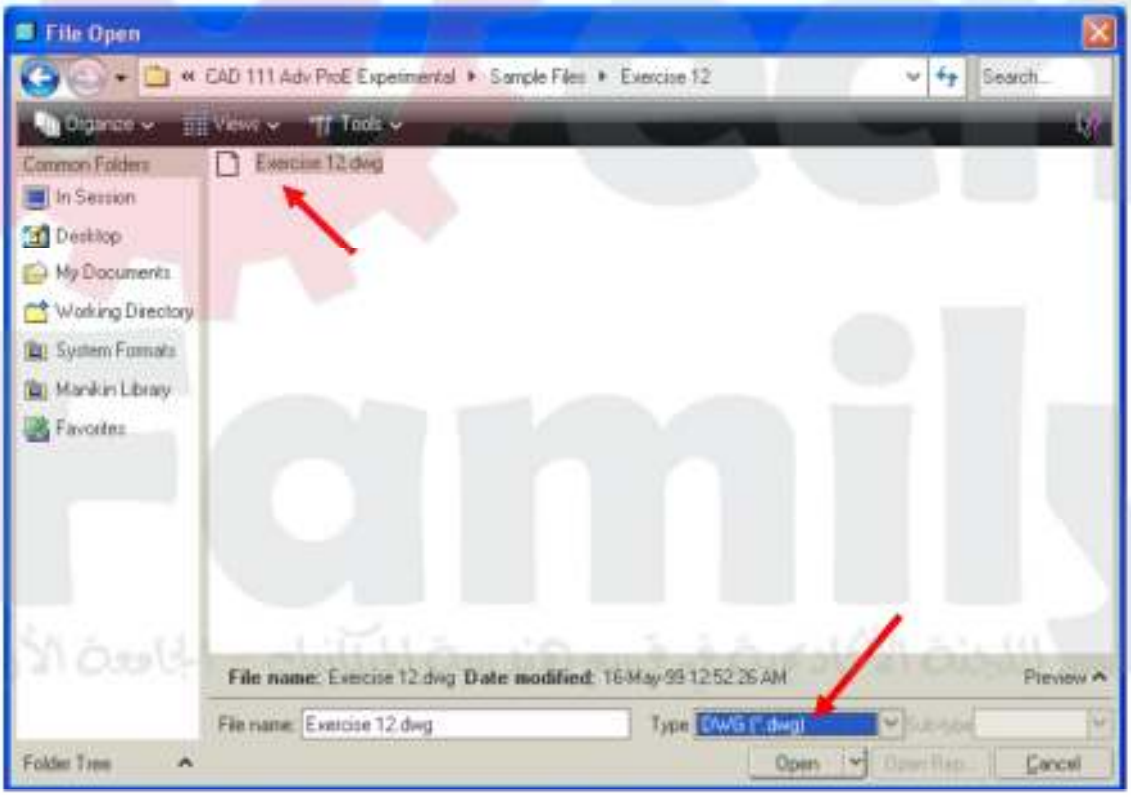
EXERCISE 12

Importing 2D DXF/DWG files

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

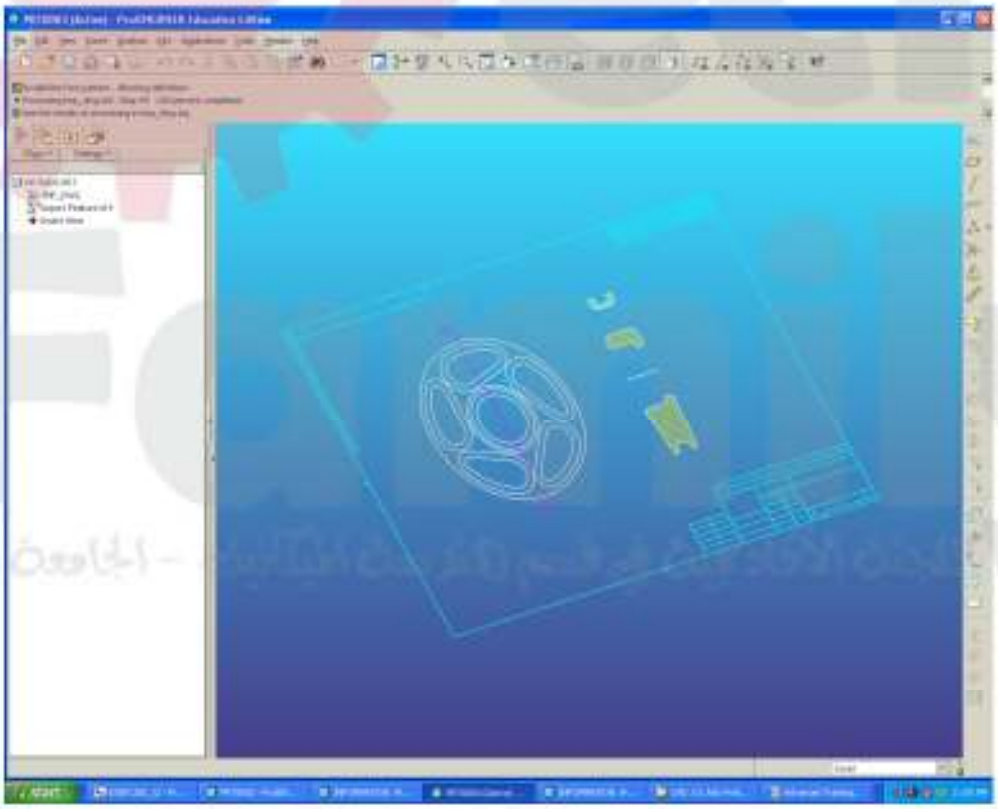


1. Go to file/open and select DWG from the options. Find the Exercise 12.dwg.

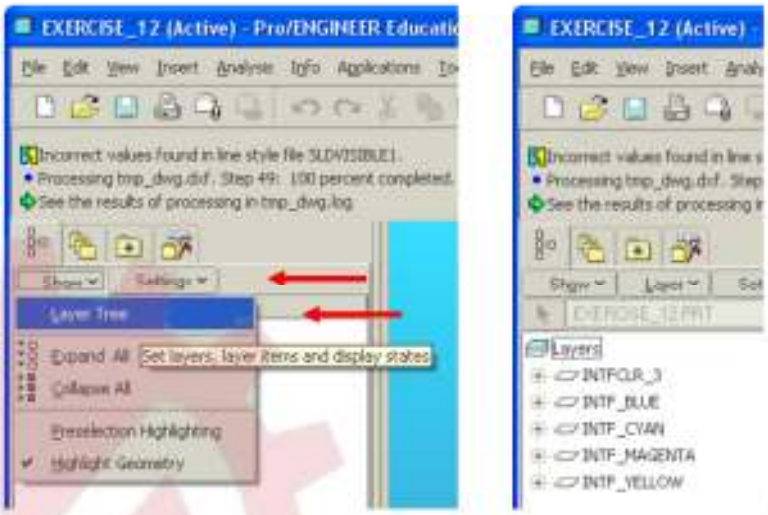


2. Import to a new Part.

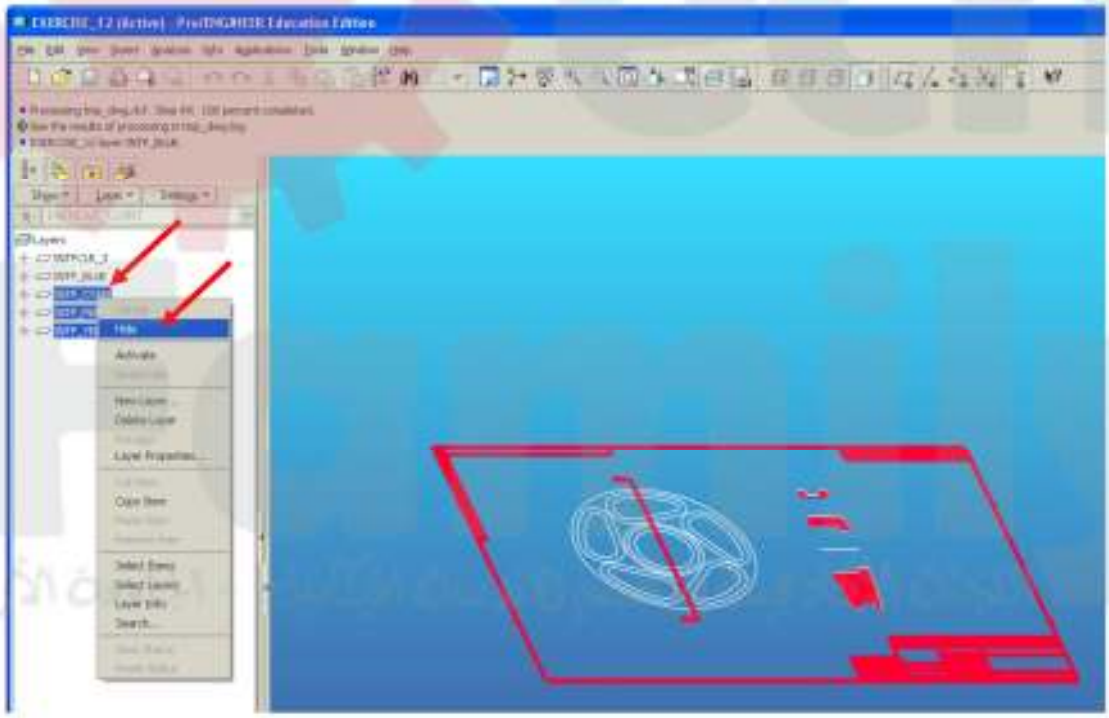
3. The next screen should look like this…



4. Go to “Show/Layer Tree”…



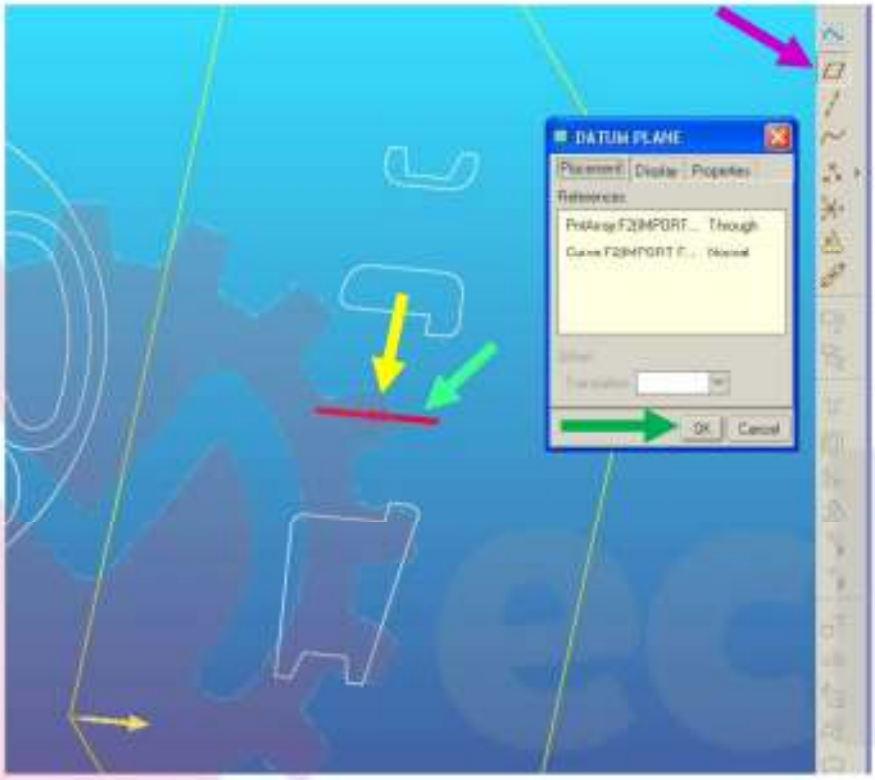
5. CTRL select the CYAN, MAGENTA, and YELOW layers, RMB click and select “Hide”.



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".

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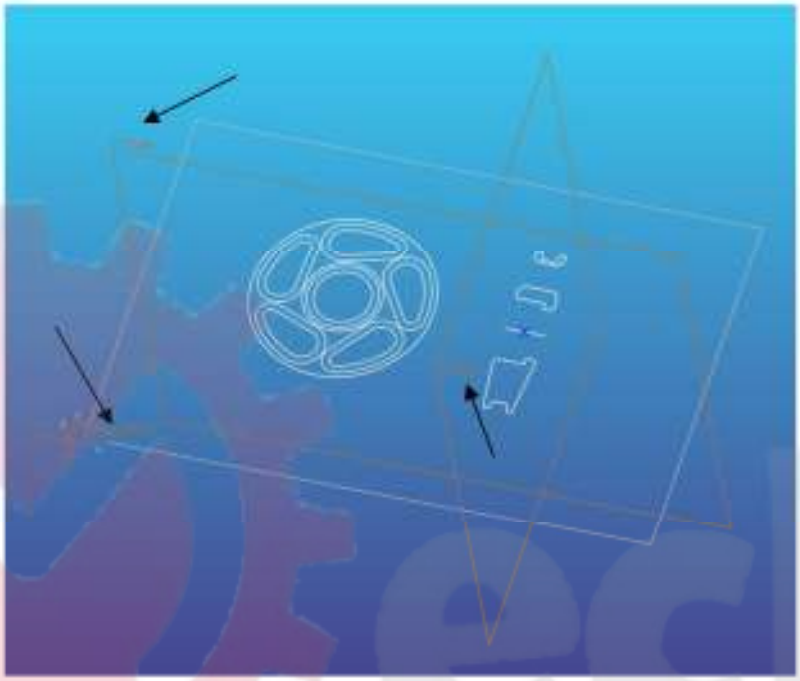
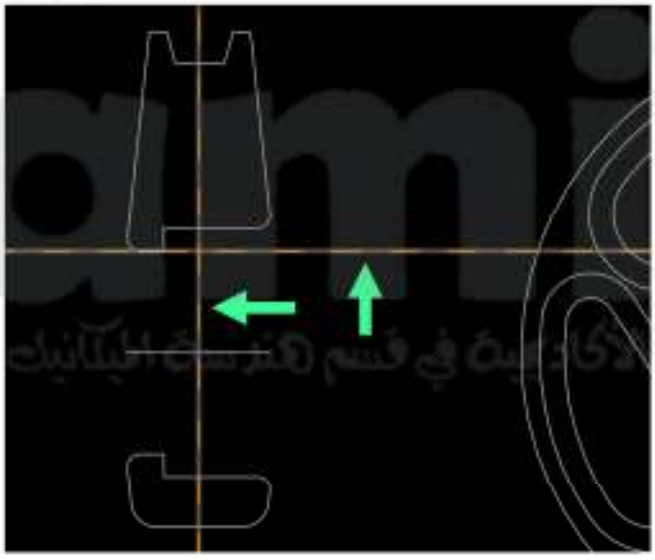
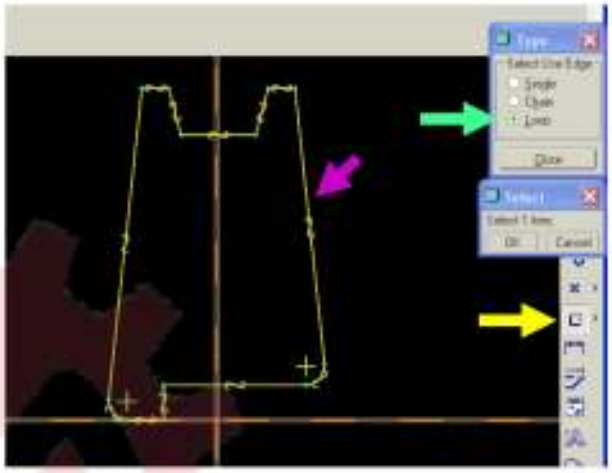
10. You should now have a three intersecting perpendicular planes to use for sketching and references.



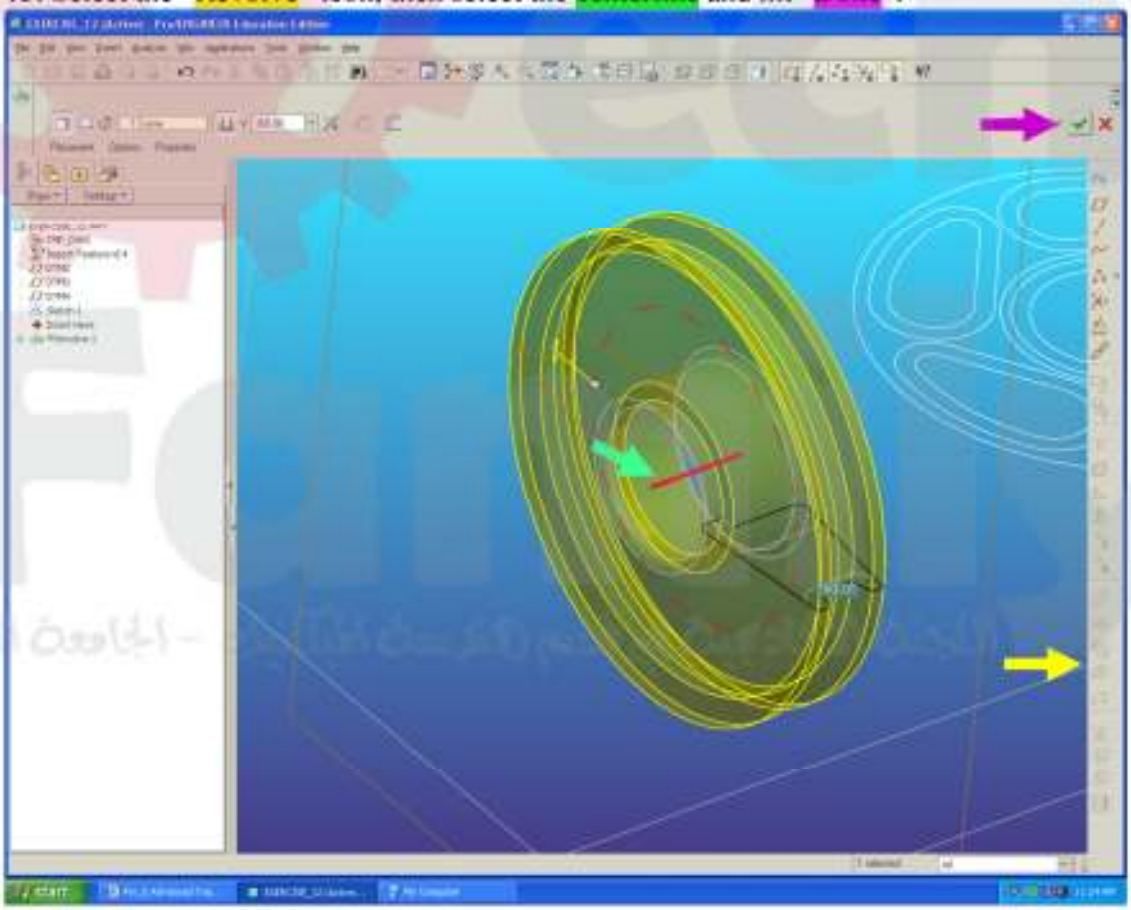
11. Start a sketch on the plane that is aligned parallel to your drawing. Select the vertical and horizontal datum planes as references.



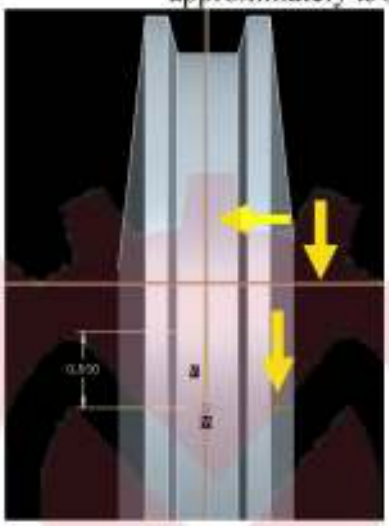
12. Select the "USE" icon, and chose "Loop", then select any edge of the profile.

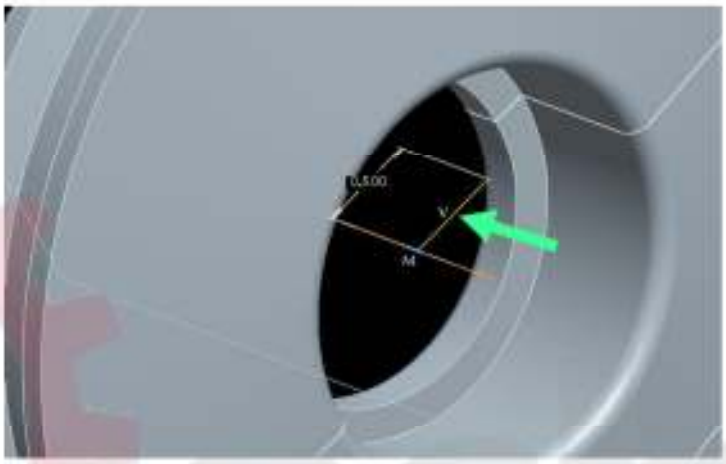


13. Select the "Revolve" icon, then select the centerline and hit "Done".

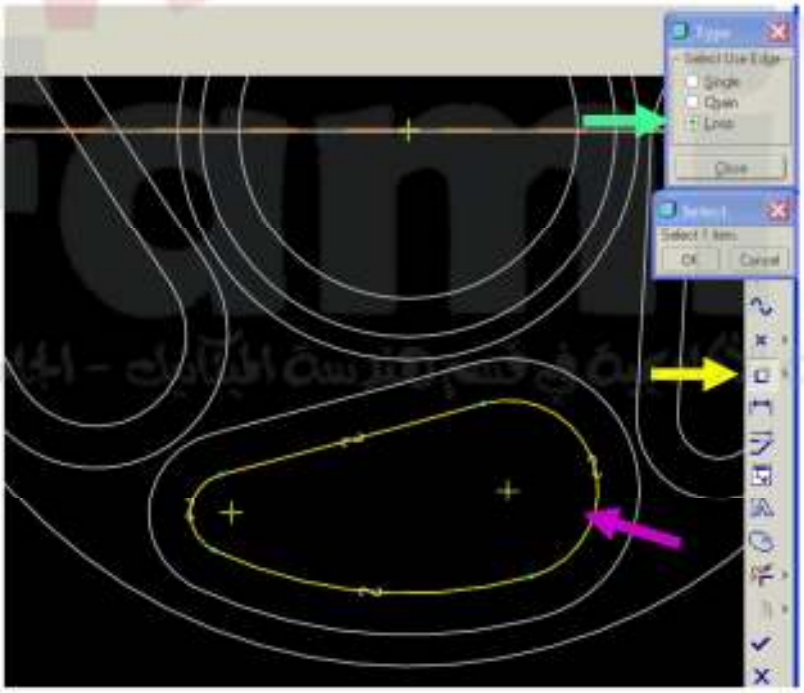


13. (SKIP THIS IF YOU ALREADY CREATED AN OFFSET DATUM) Start a sketch on the parallel plane to the drawing. Then select the centerline and vertical and horizontal planes as references. Draw a vertical line approximately .500" long. Hit "Done".

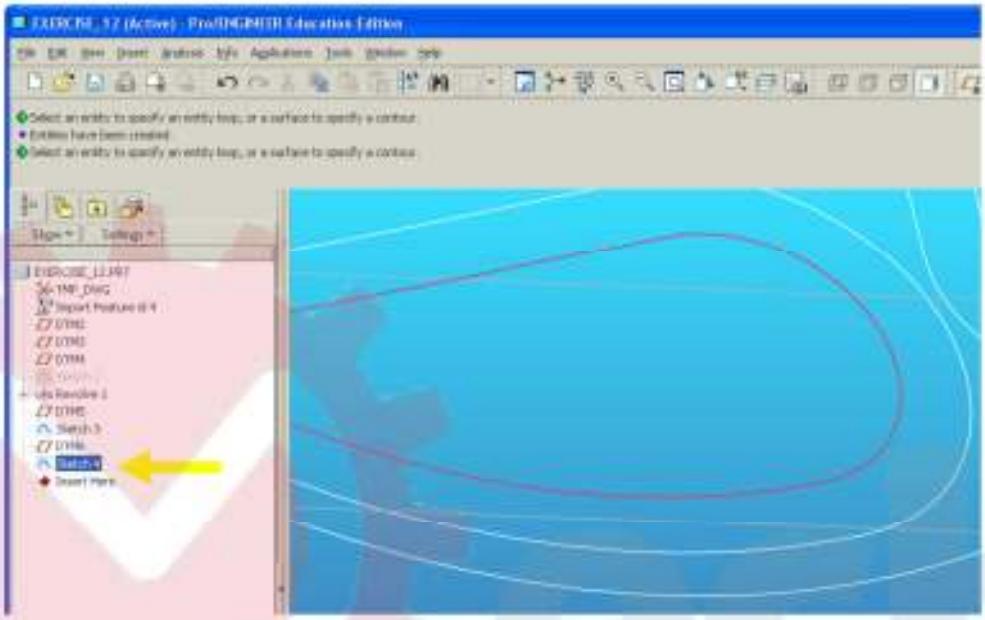




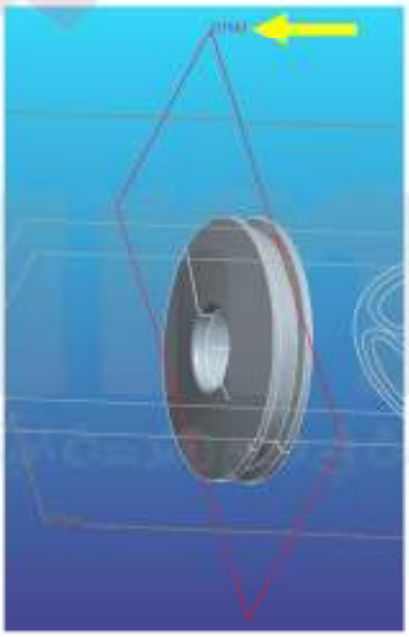
14. Select the endpoint and length of the ".500" line, and then go to the Plane icon. Hit "Done" *Note: turn off the view-points icon to help make selections easier.*
15. Start a new sketch on the same parallel plane to the drawing. Select the "USE" icon, and chose "Loop", then select any edge of the profile. Hit "Done".



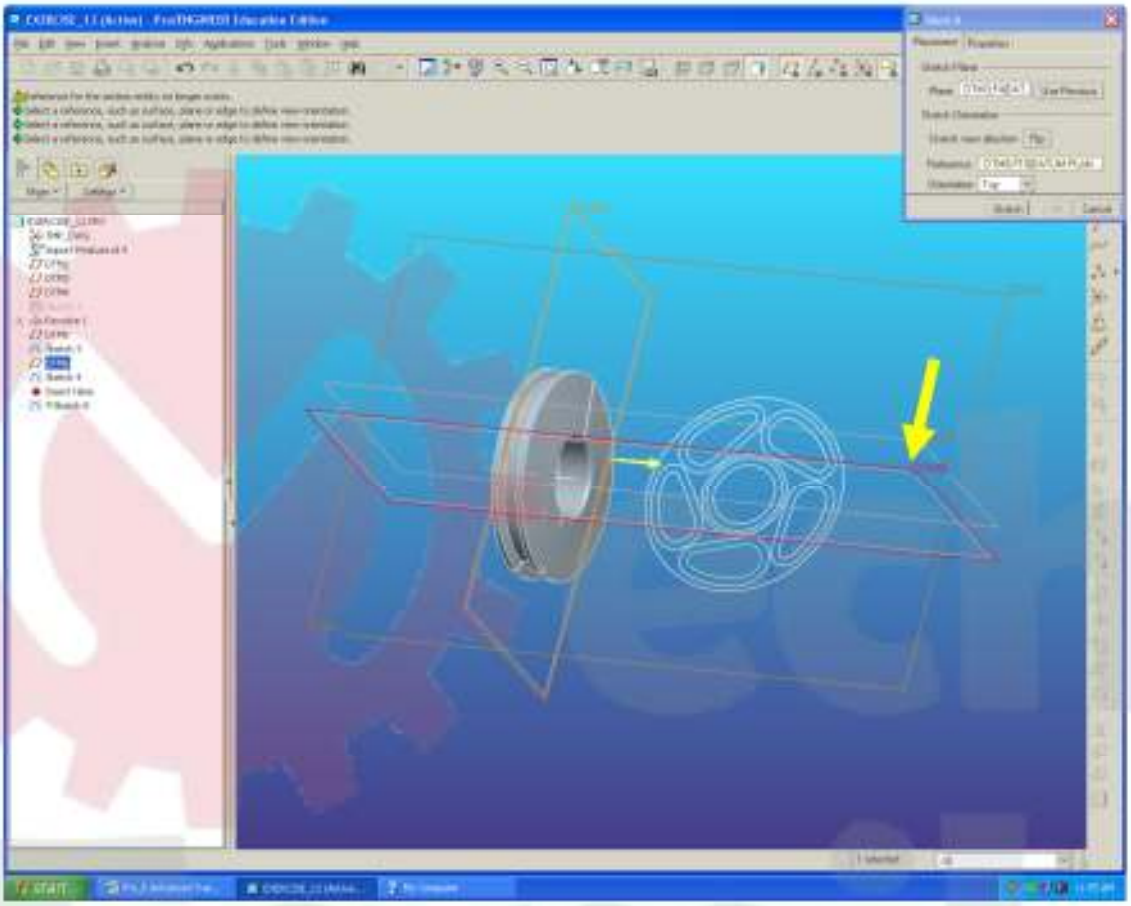
16. CTRL select the sketch from the feature tree. Then select “Edit/Copy” (CTRL-C).



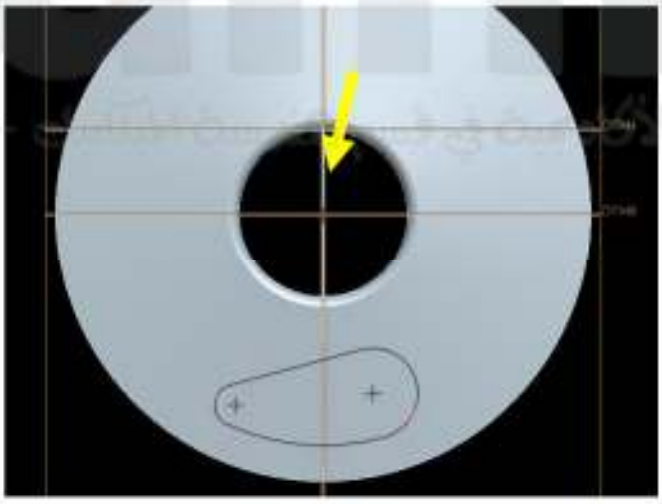
17. Select the Datum Plane that runs vertically through the model and go to “Edit/Paste”



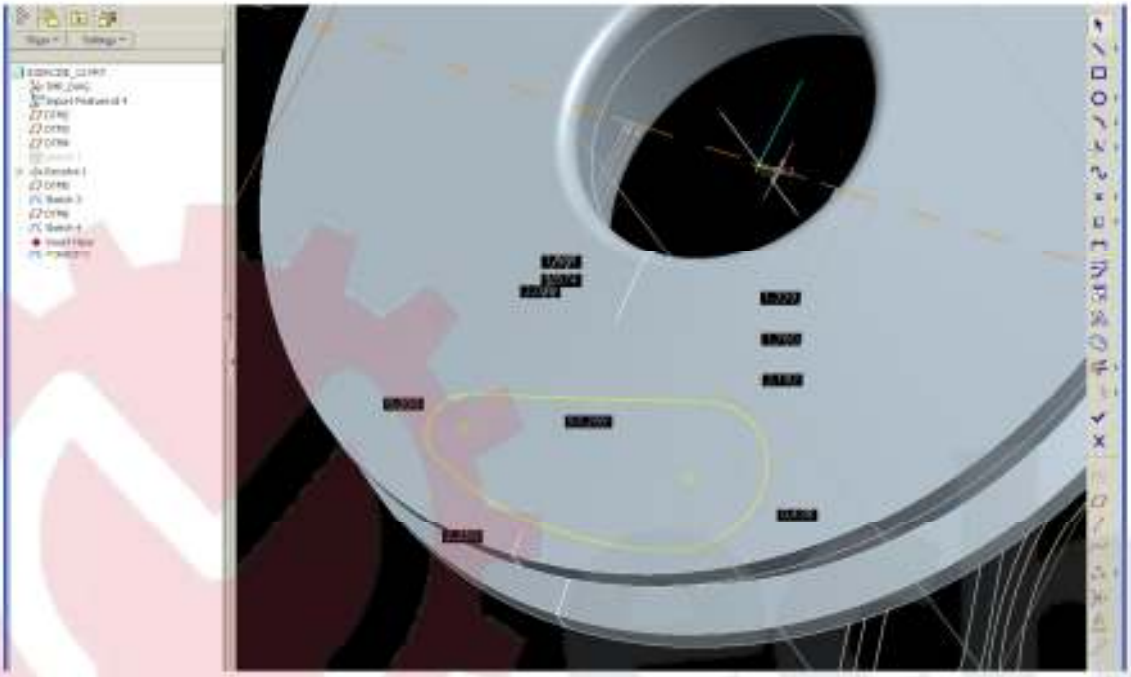
18. At the “Sketch” prompt select the datum plane you created that runs horizontally through the model that was created in steps 13 and 14 as a reference.



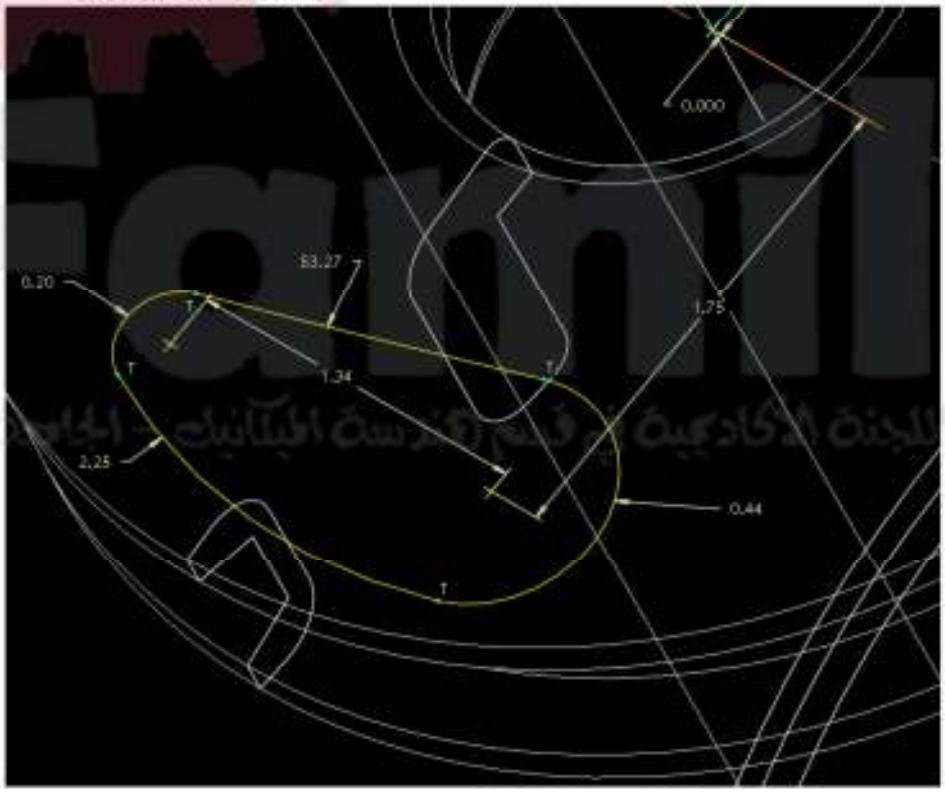
19. Now you can position with the cursor the approximate location for the cut out.



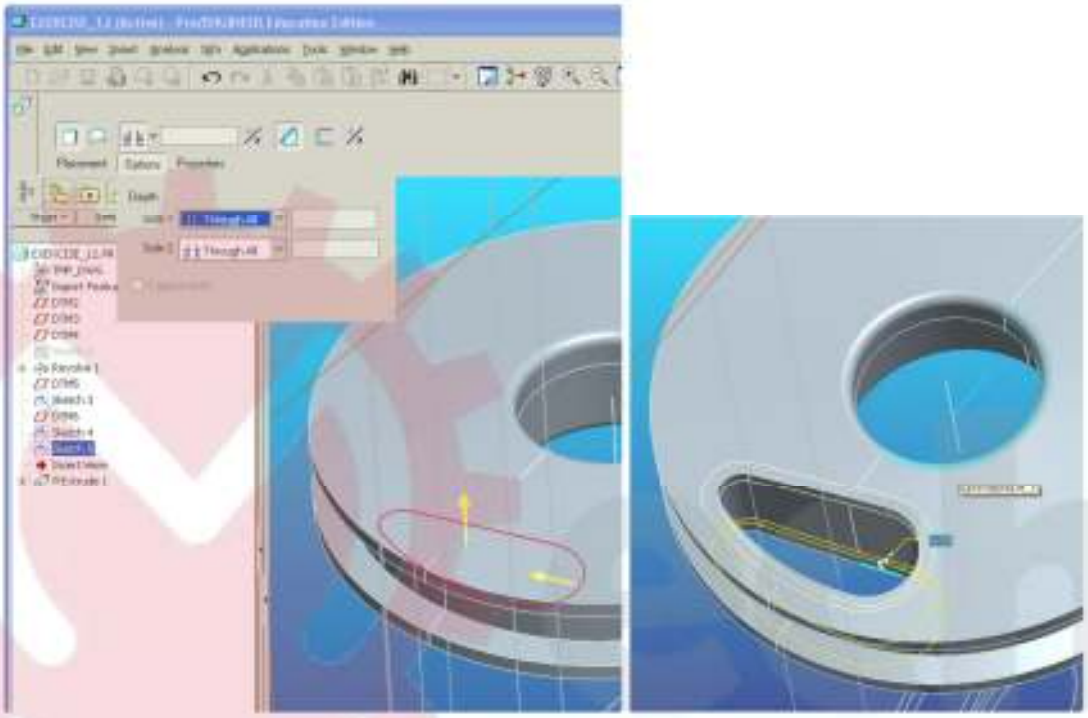
20. Xxx



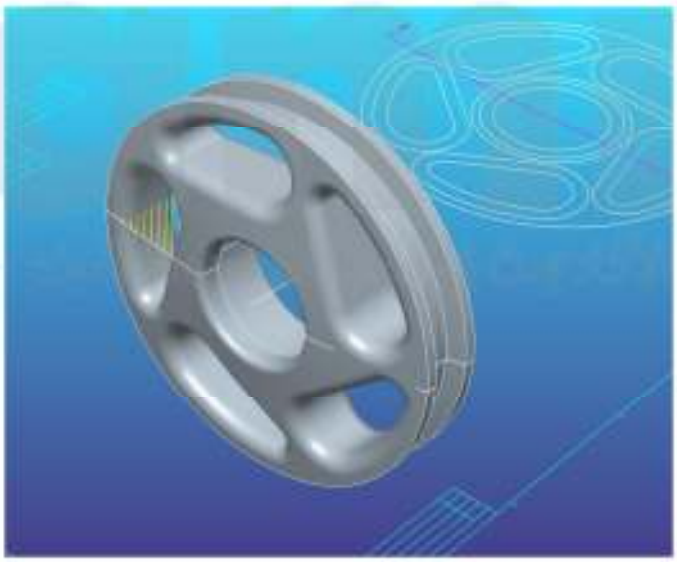
21. Insert tangent relations on all four connection points, then re-dimension as shown. Hit "Done".



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



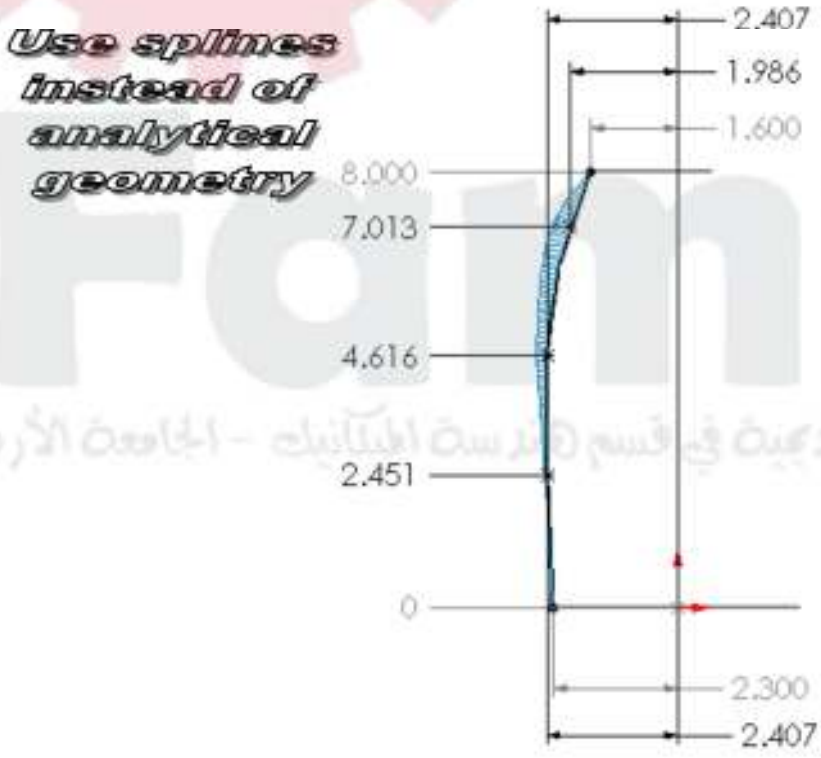
23. Completed model.



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".



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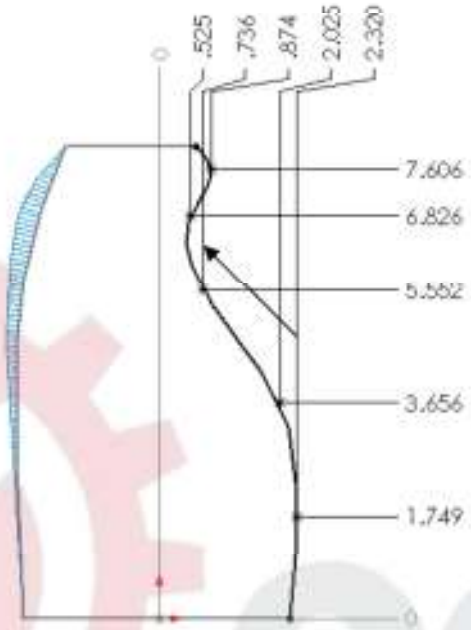
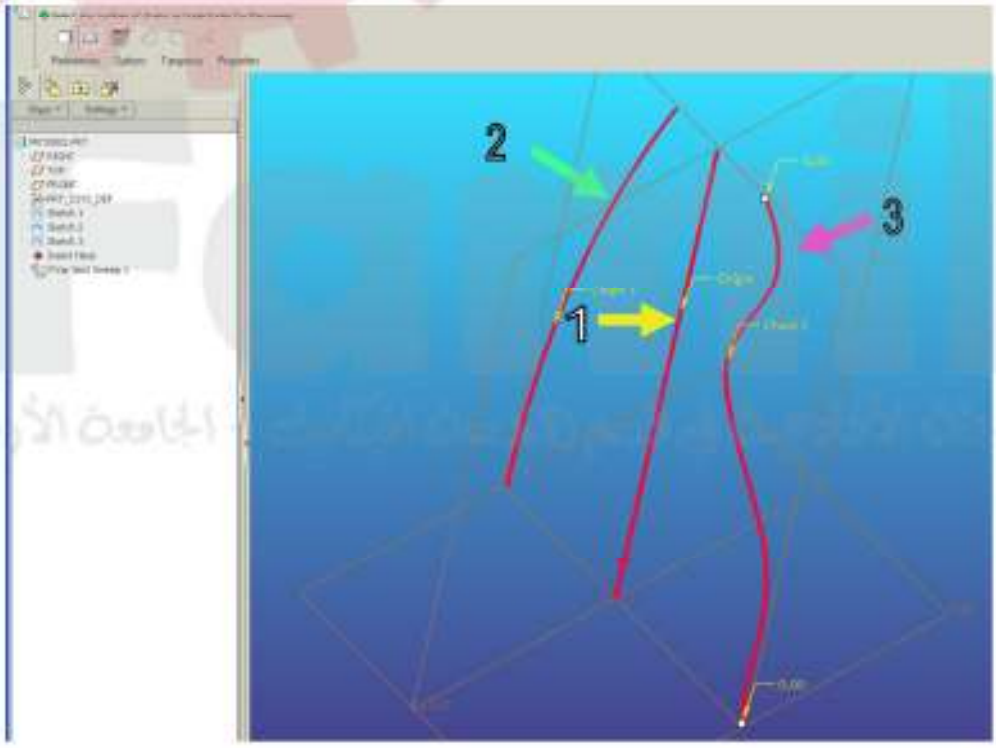
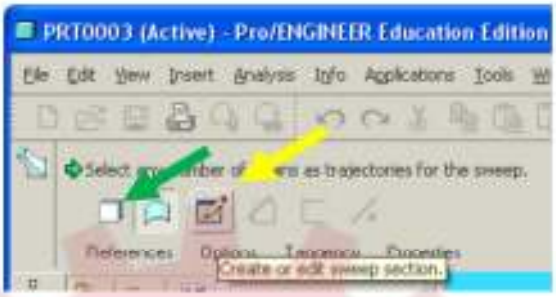
3. Sketch the following on the Front plane. (5 spline points)



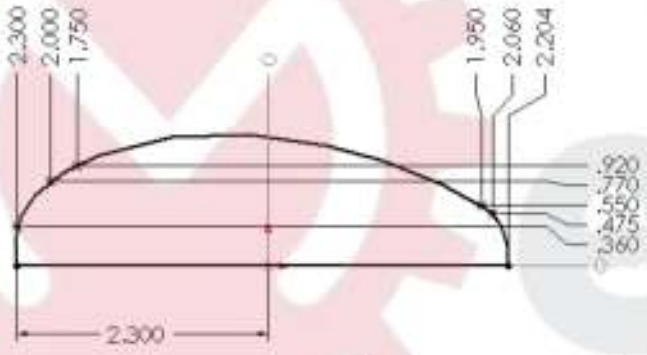
4. CTRL select **in order** the three sketches. Go to "Insert/Variable Section Sweep. 1 - Yellow, 2 - Green, 3 - Magenta.

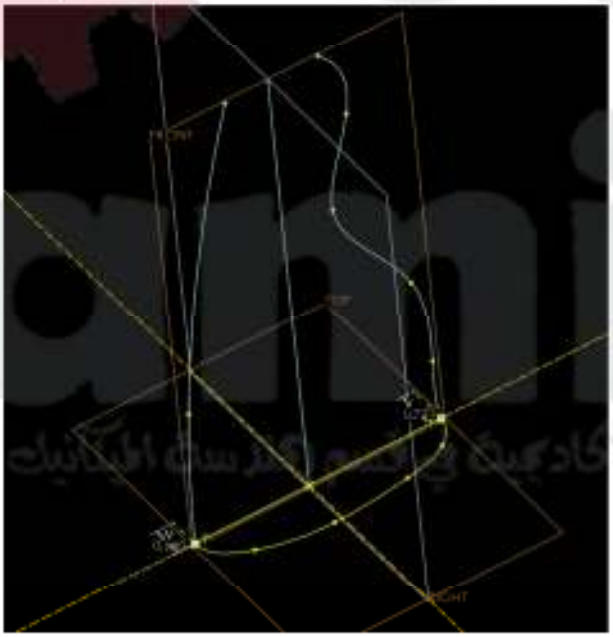


5. Select the "Edit sweep" icon from the command line, and don't forget to set it to a Solid.

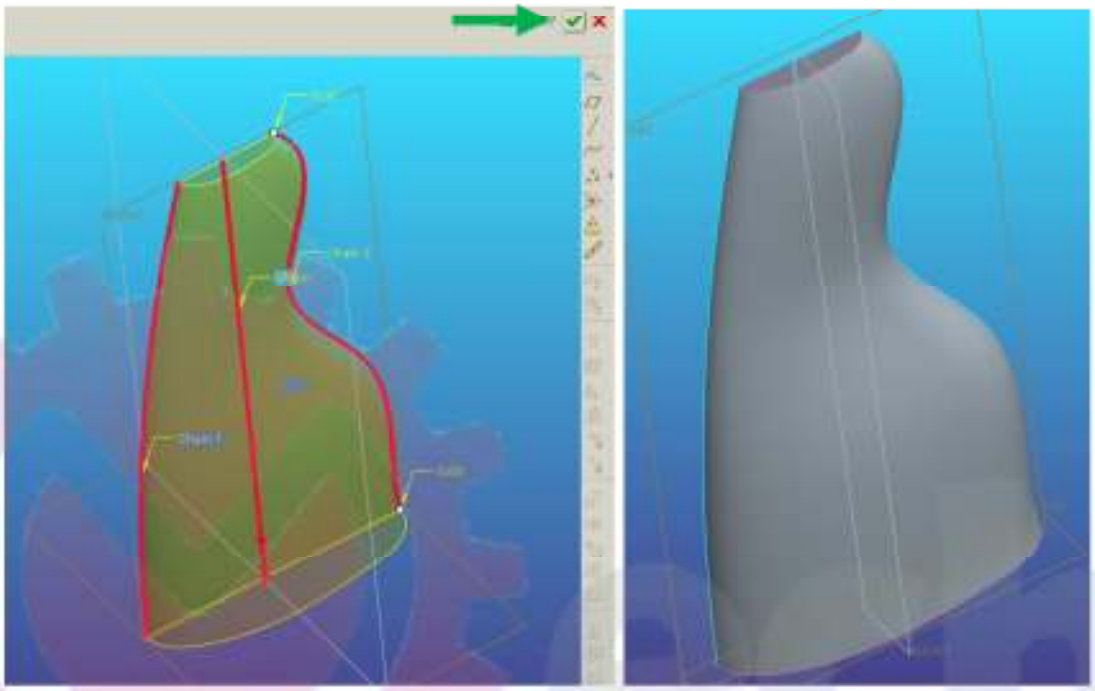


6. Draw the following profile.

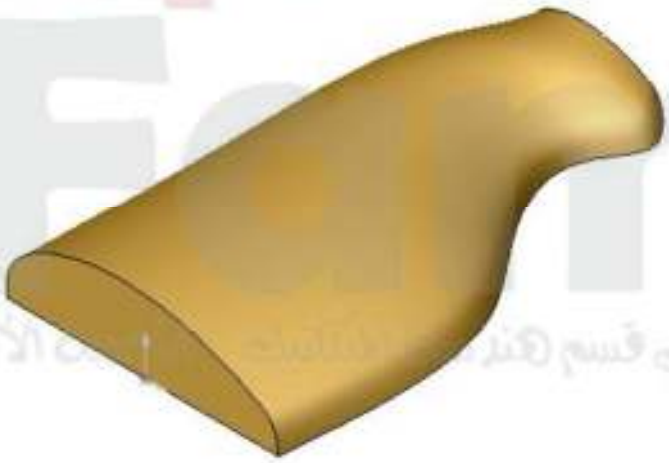




7. Select “Done”.



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”.

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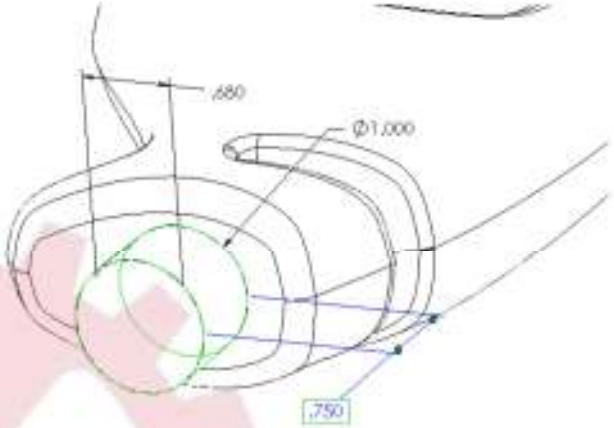
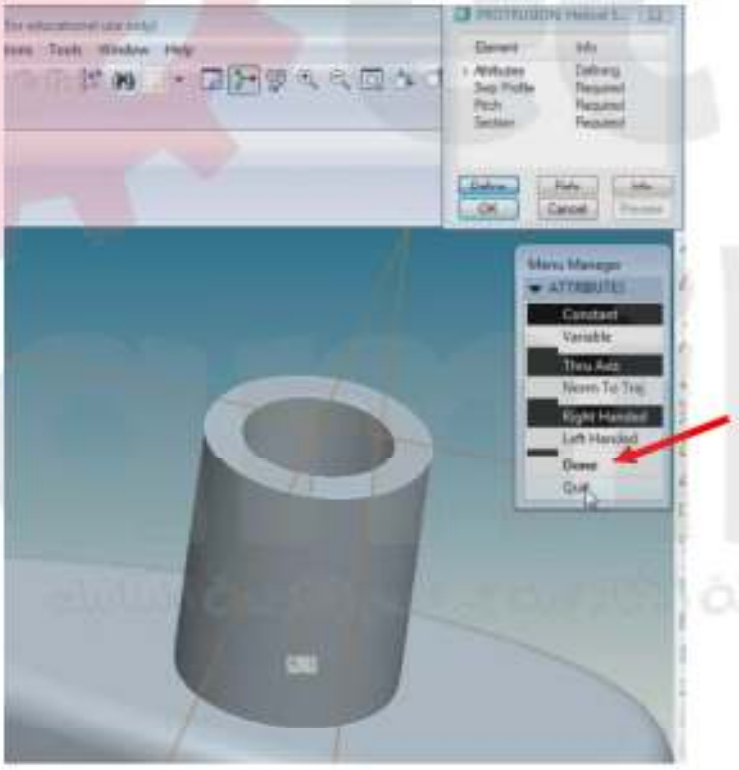
11. Insert the neck of the bottle as shown below.



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.

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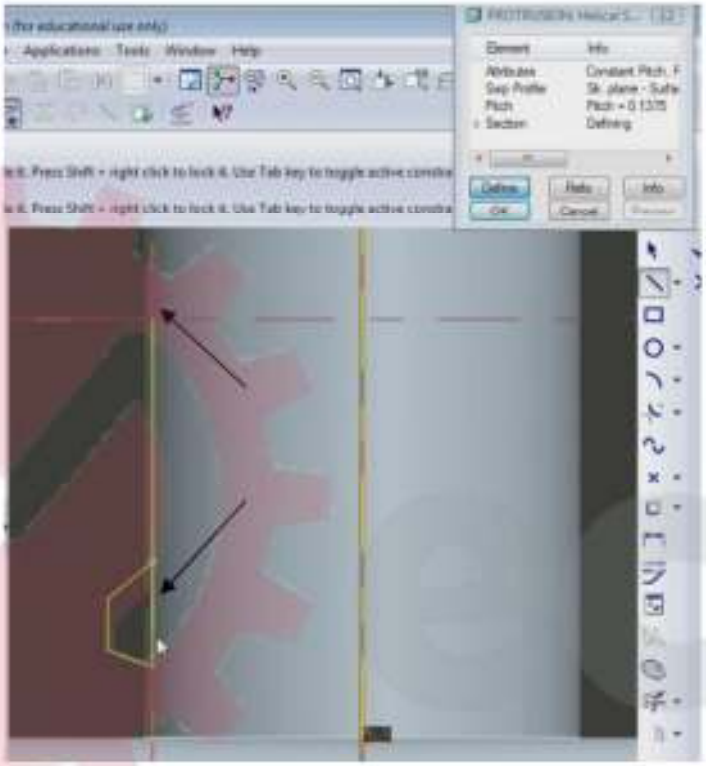
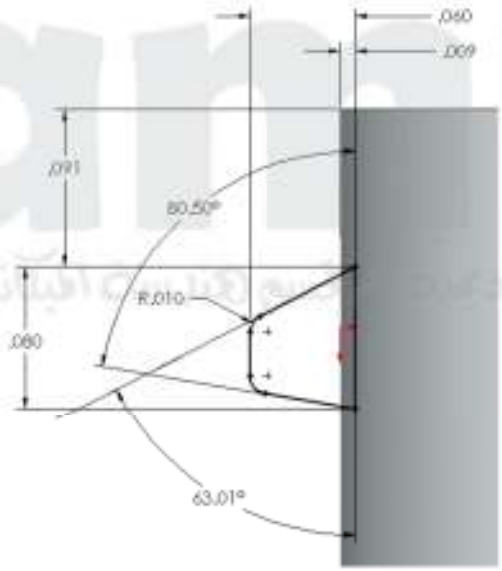
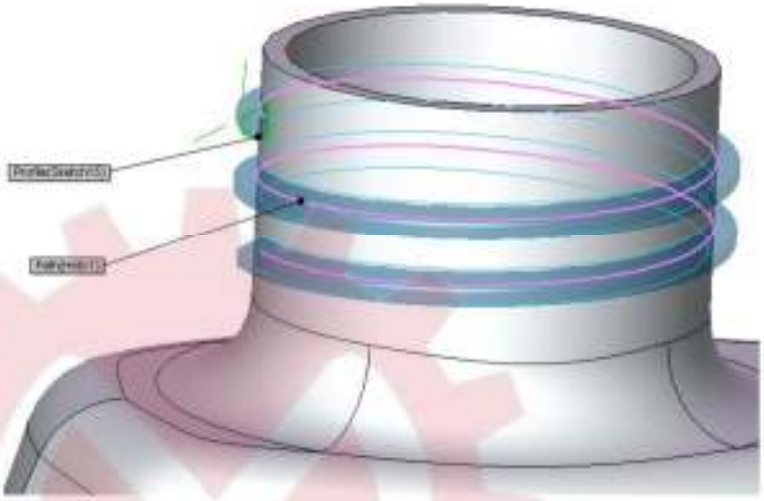
15. Now sketch the thread section profile. *Note: If you sketched the original line from the bottom up, draw as shown, however if you drew the original line from the top down begin the profile sketch at the top.*



16. Now just draw the geometry of the thread.

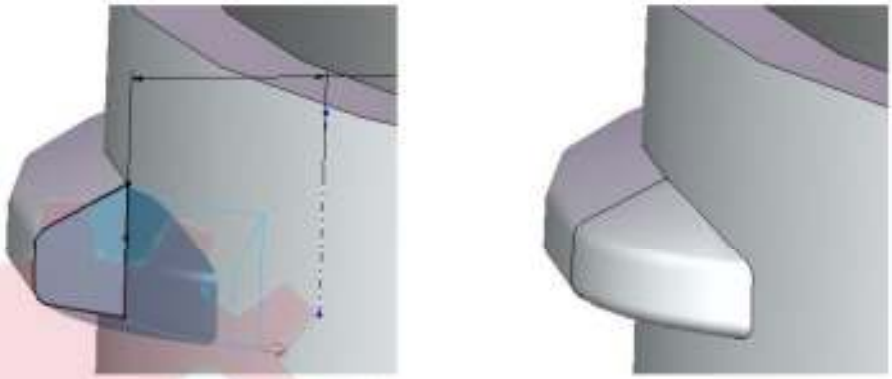


17. Hit “Preview, to view the Sweep feature. Select “OK” to finish.



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

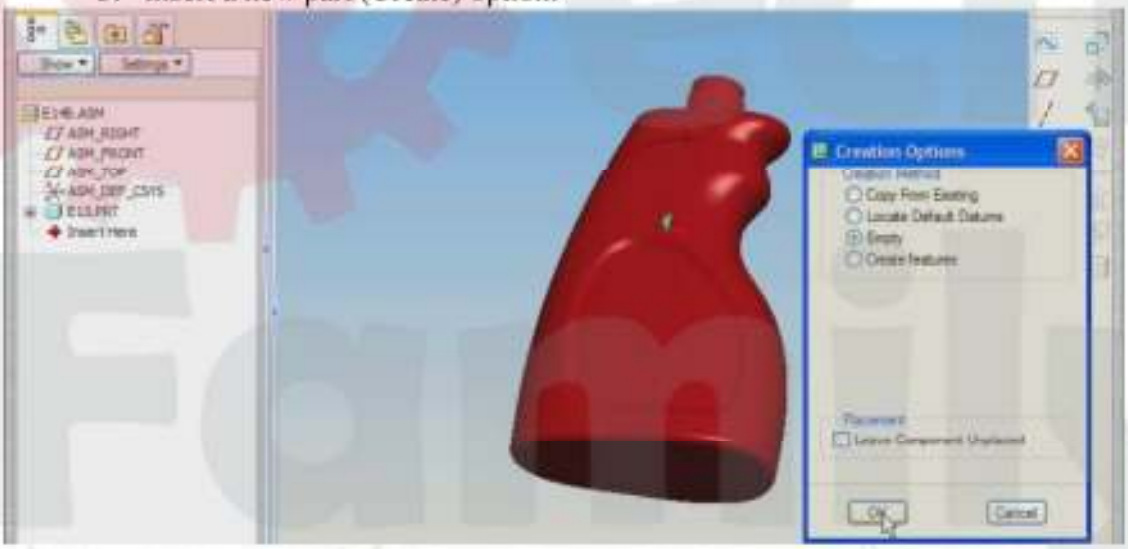
EXERCISE 14
Mold/Cavity Creation

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.

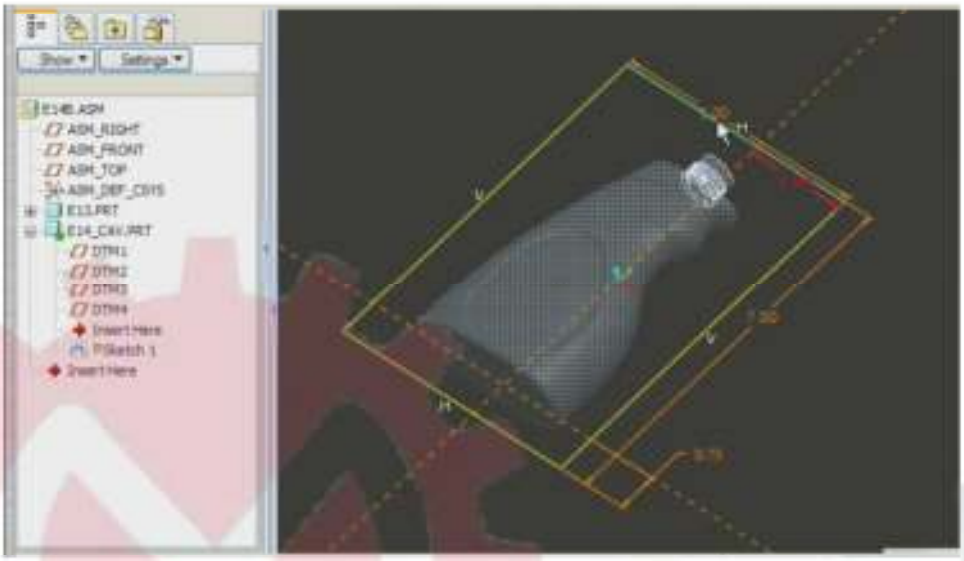
2. Insert the E_14_Bottle part file and use the default placement option.

3. Insert a new part (Create) option.

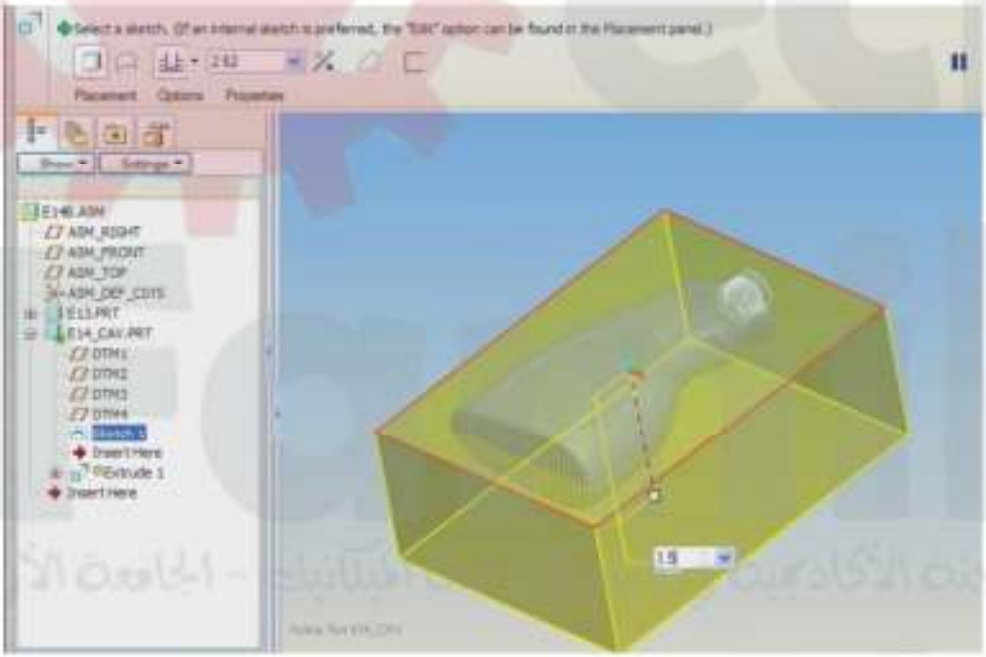


4. Name it "E14_Cav", then create Front, Top, and Right datum planes.

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

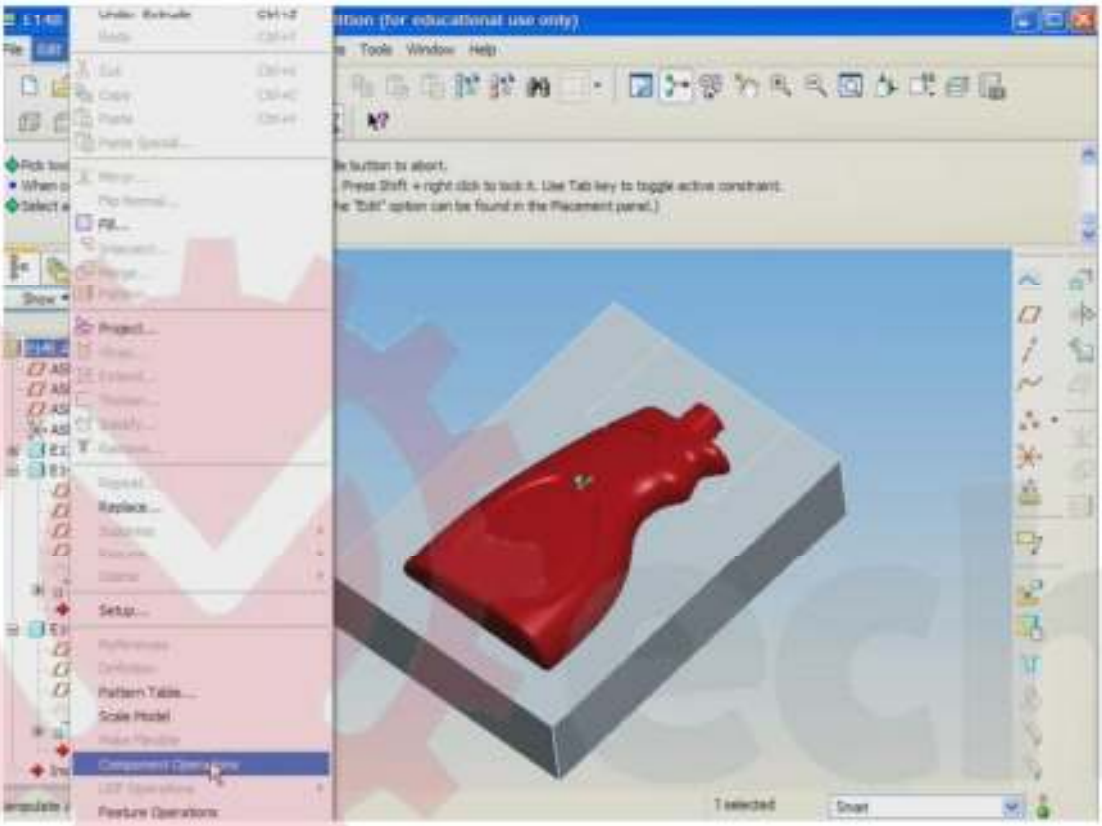


6. Extrude 1.5" down.

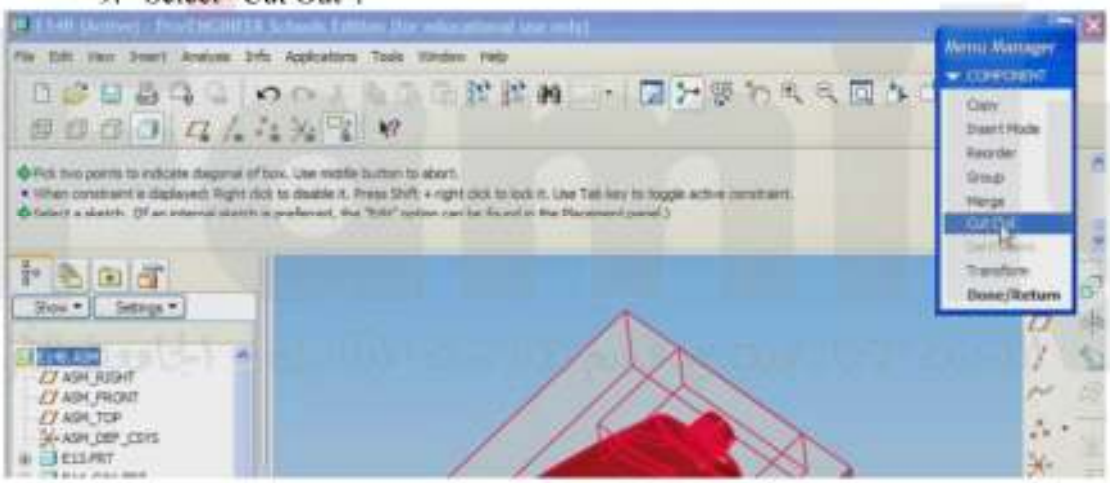


7. Right click on the Assembly marker (Top of the tree) Select Activate or Regenerate.

8. Go to Edit/Component Operations.

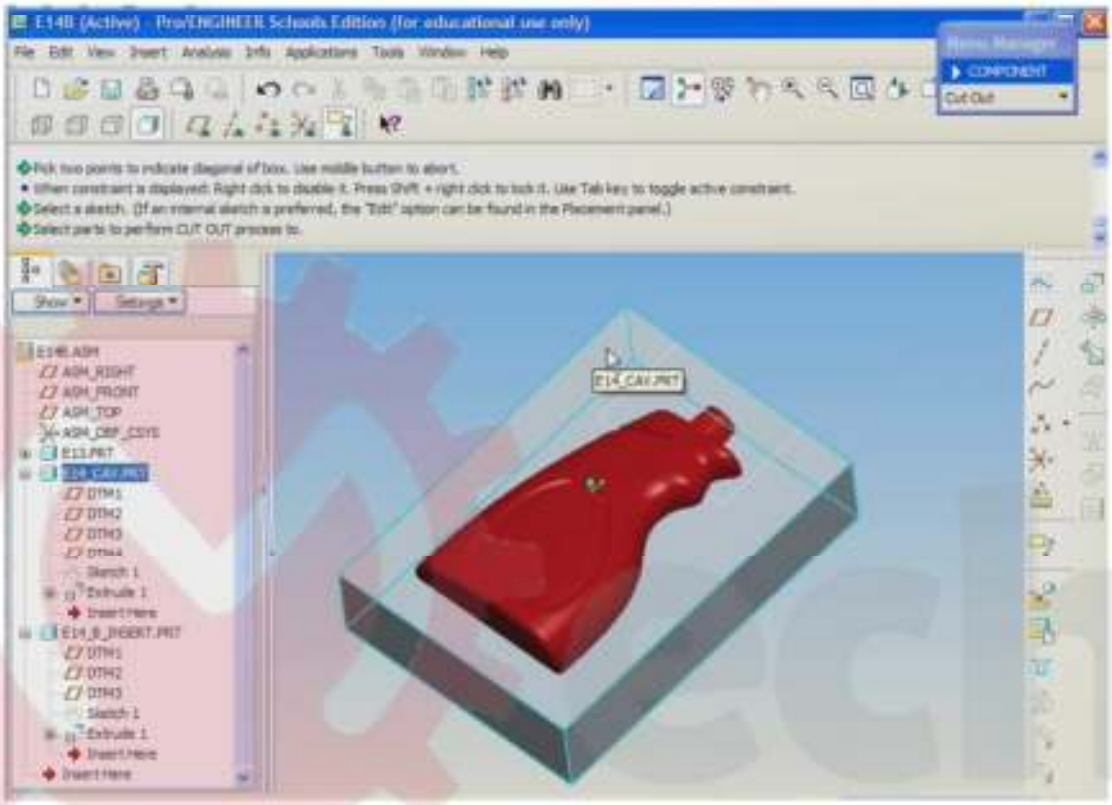


9. Select "Cut Out".



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.

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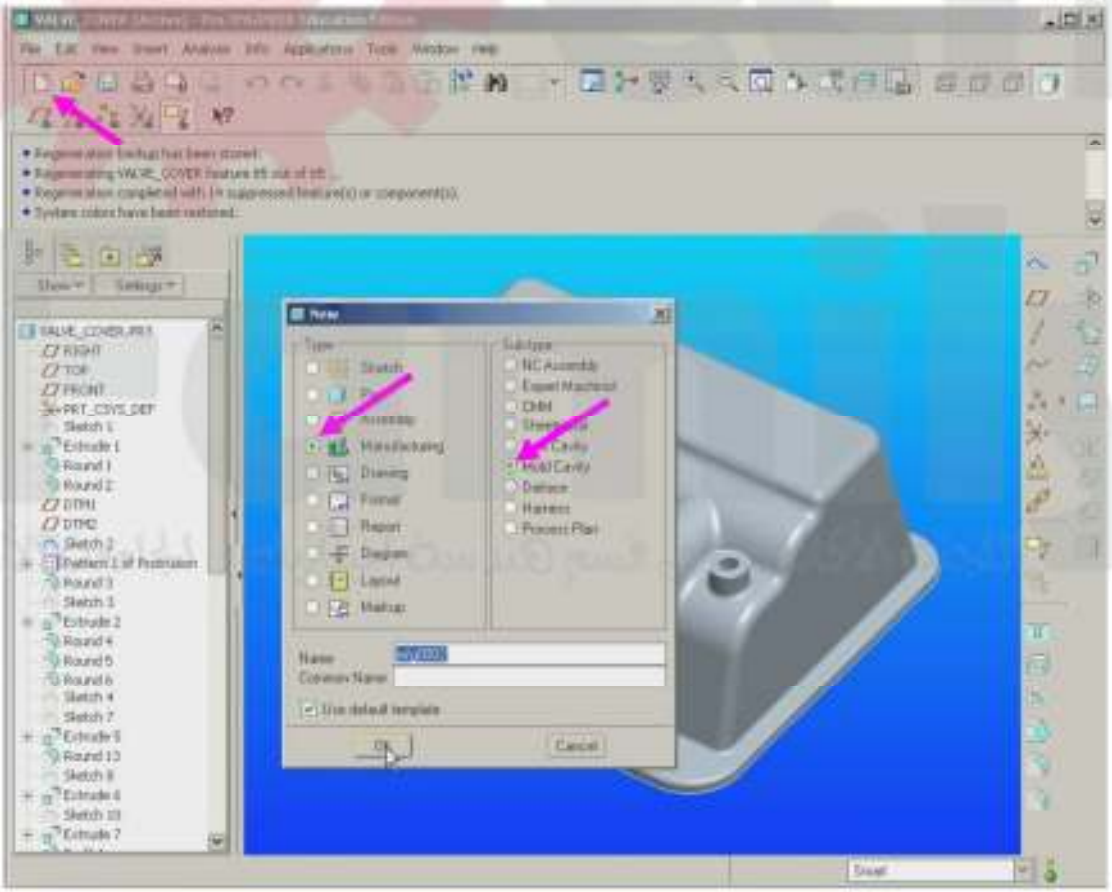
EXERCISE 15

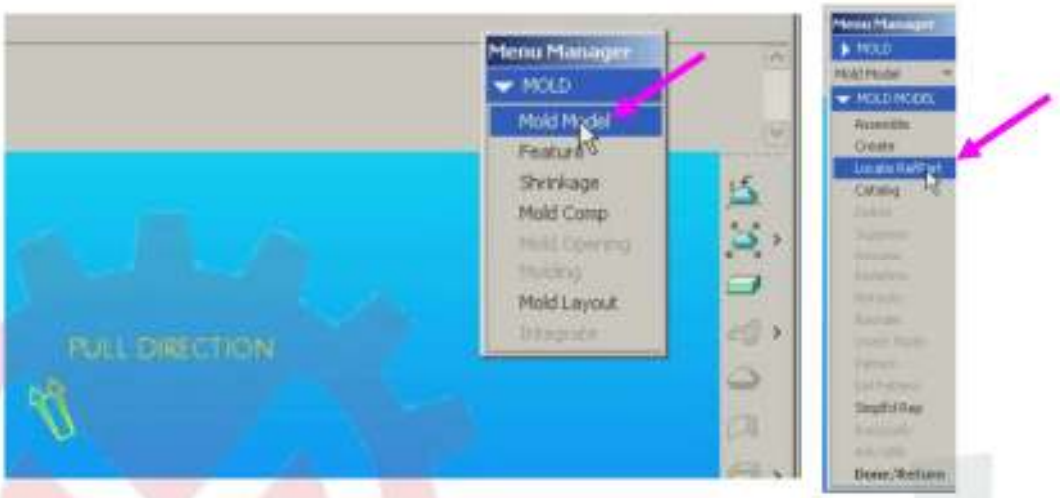
Cavity& Core Creation

***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.*

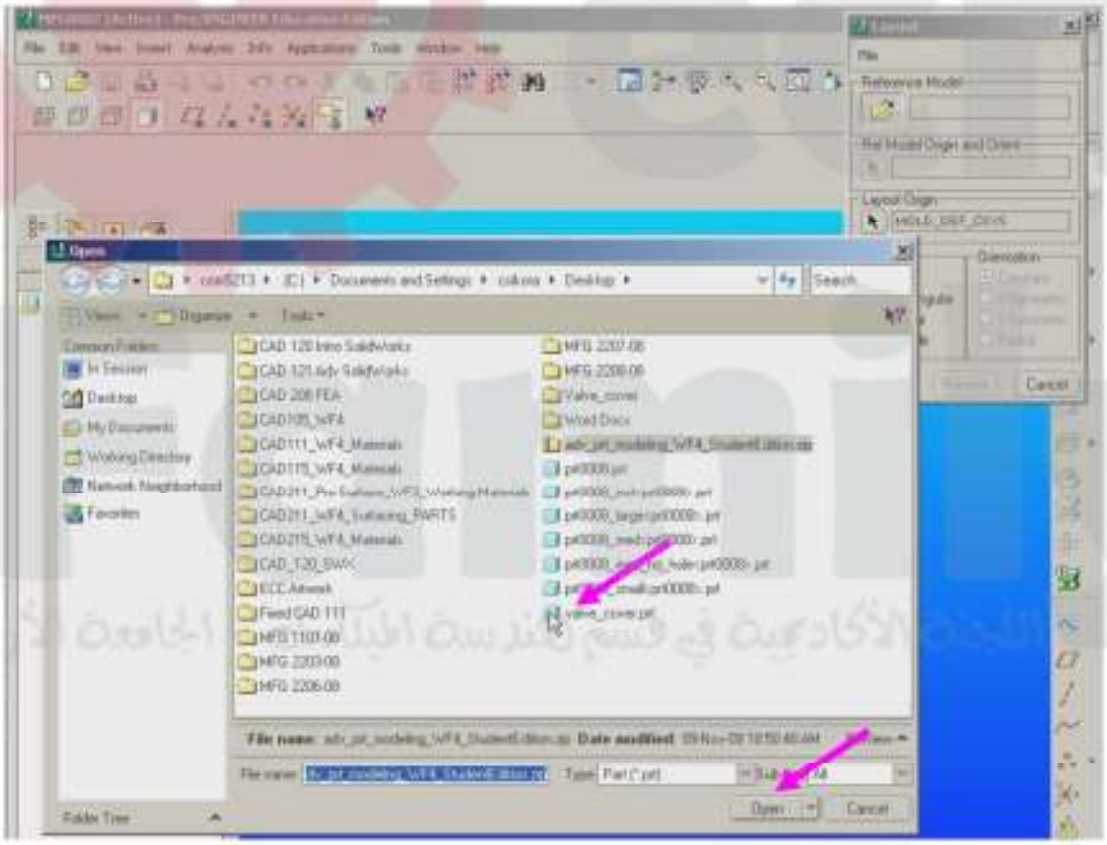
1. Start a new assembly using the Manufacturing and Mold Cavity options.



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

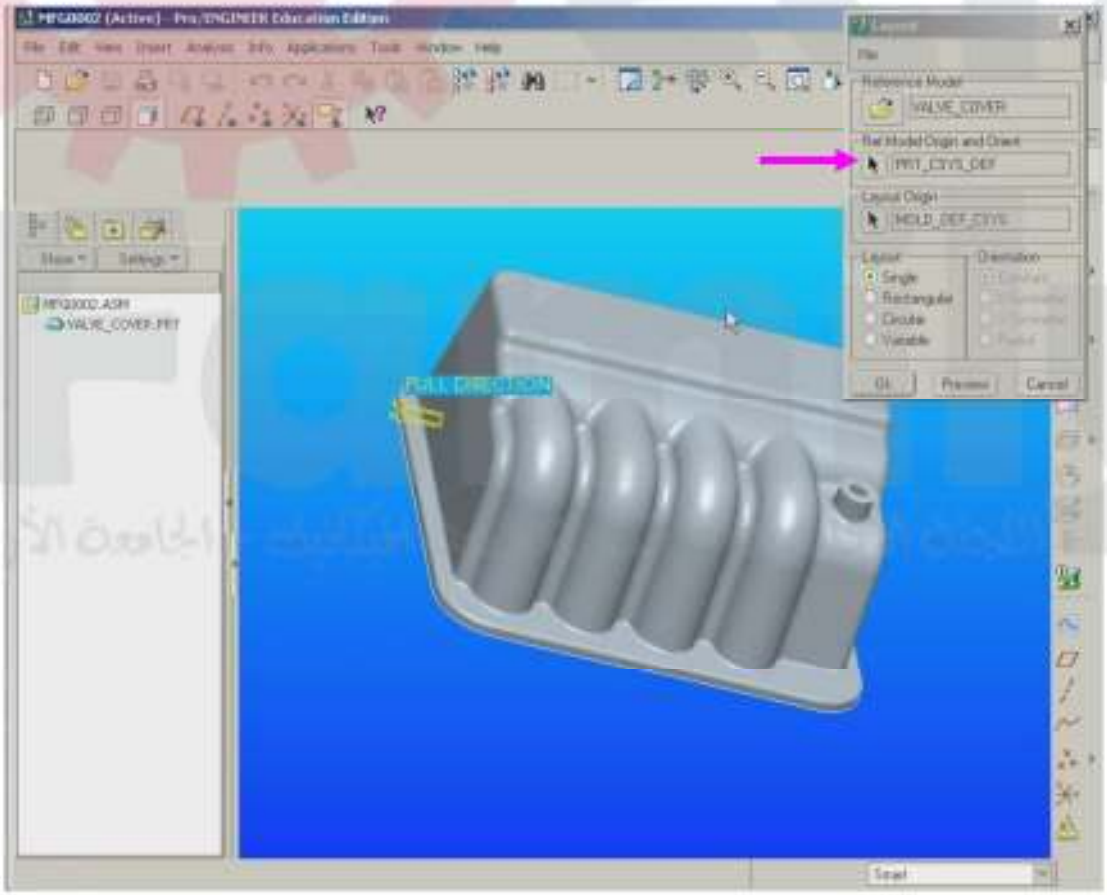


3. Locate the “valve_cover.prt” file. Hit open.

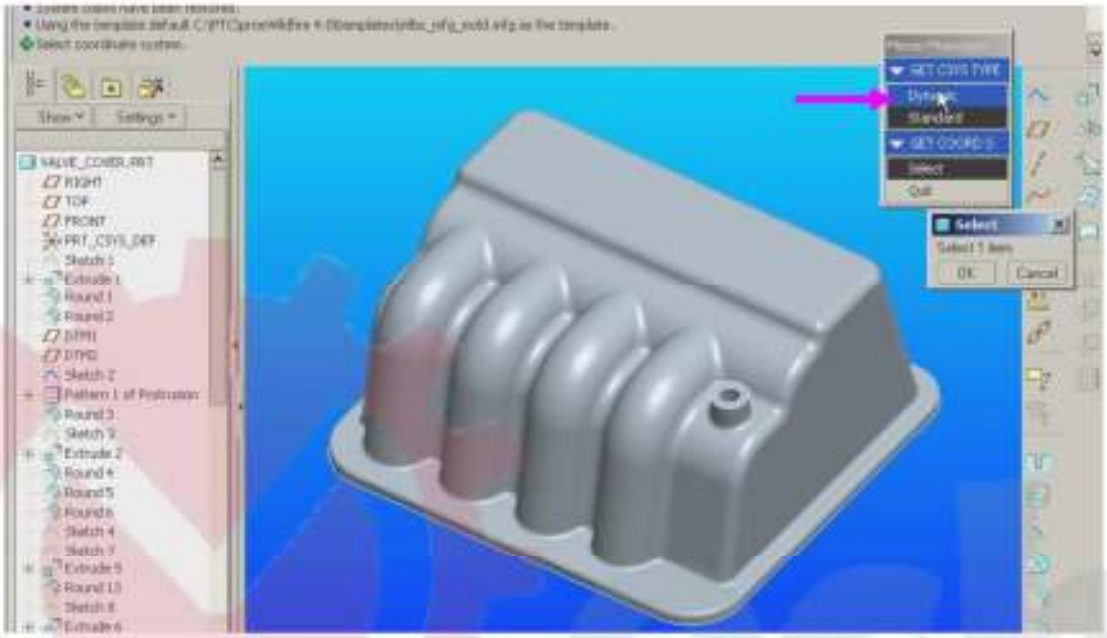


4. Select the "Same Model" option and hit ok.

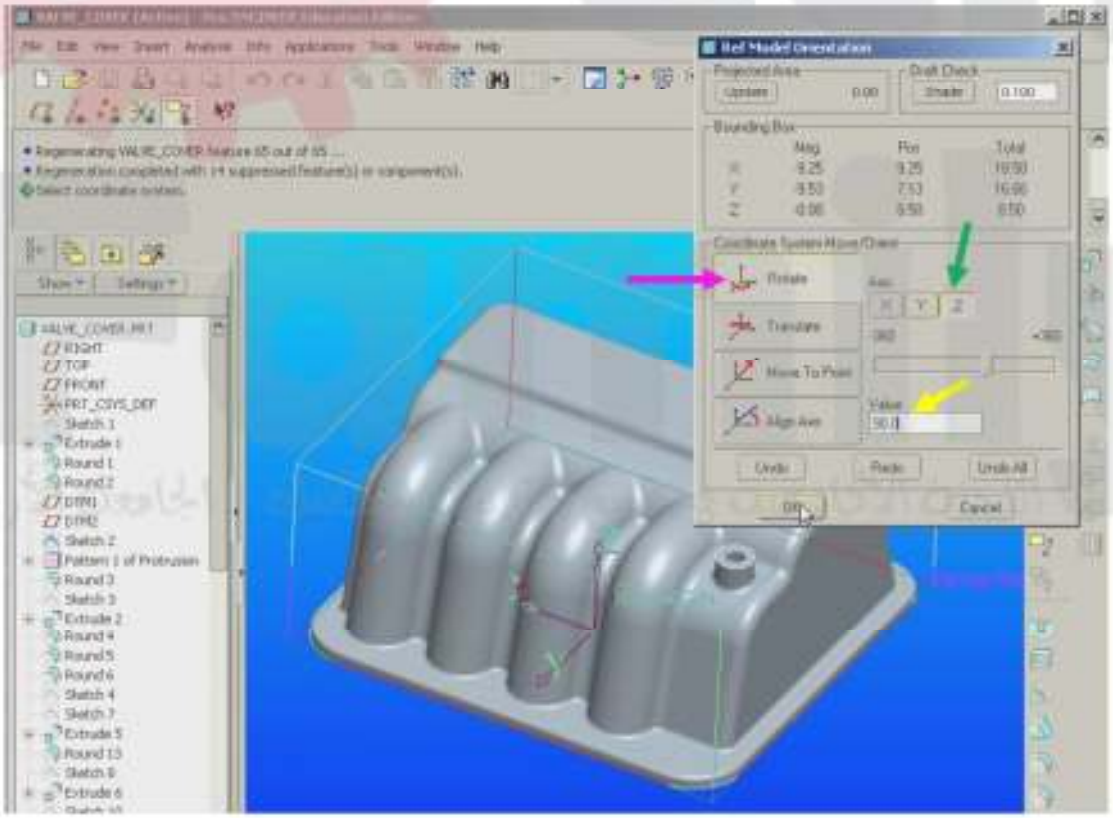
5. On the "layout" dialog box just hit the Ref Model arrow icon. Note: *This area gives the user the ability to align the pull direction with the parting line of the mold. Also the goal here is to have the mfg CSYS should have the "Z" direction pining up from the top of this model.*



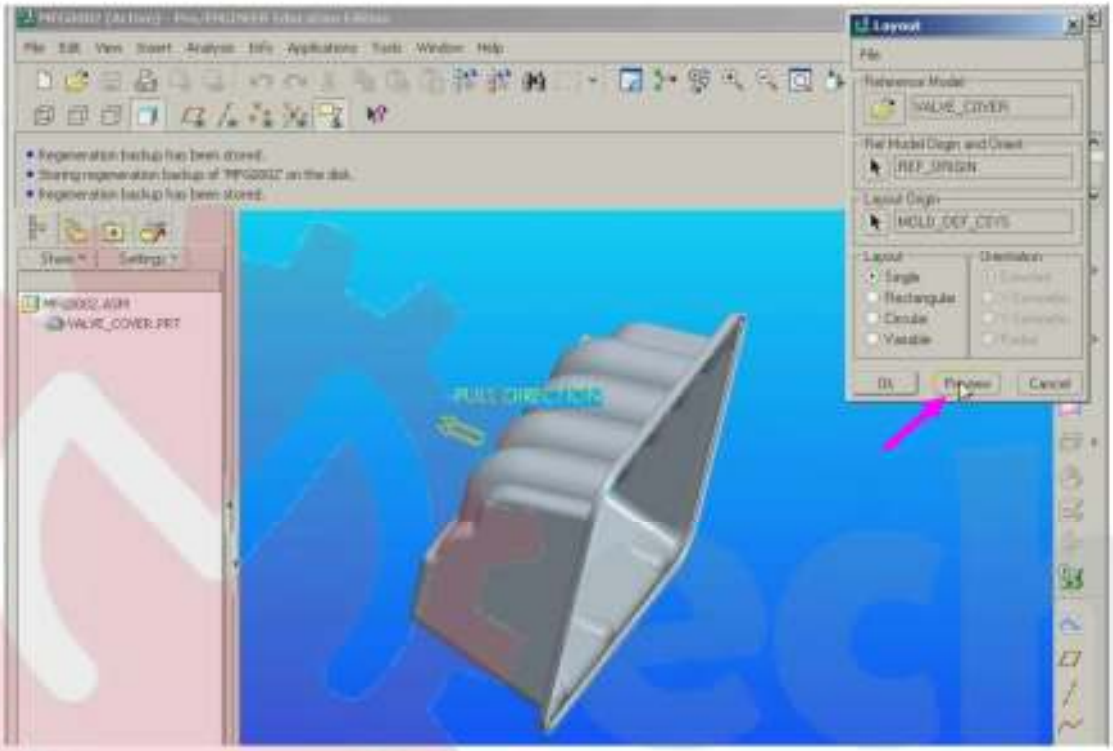
6. **Aligning the pull direction**. Select "Dynamic"



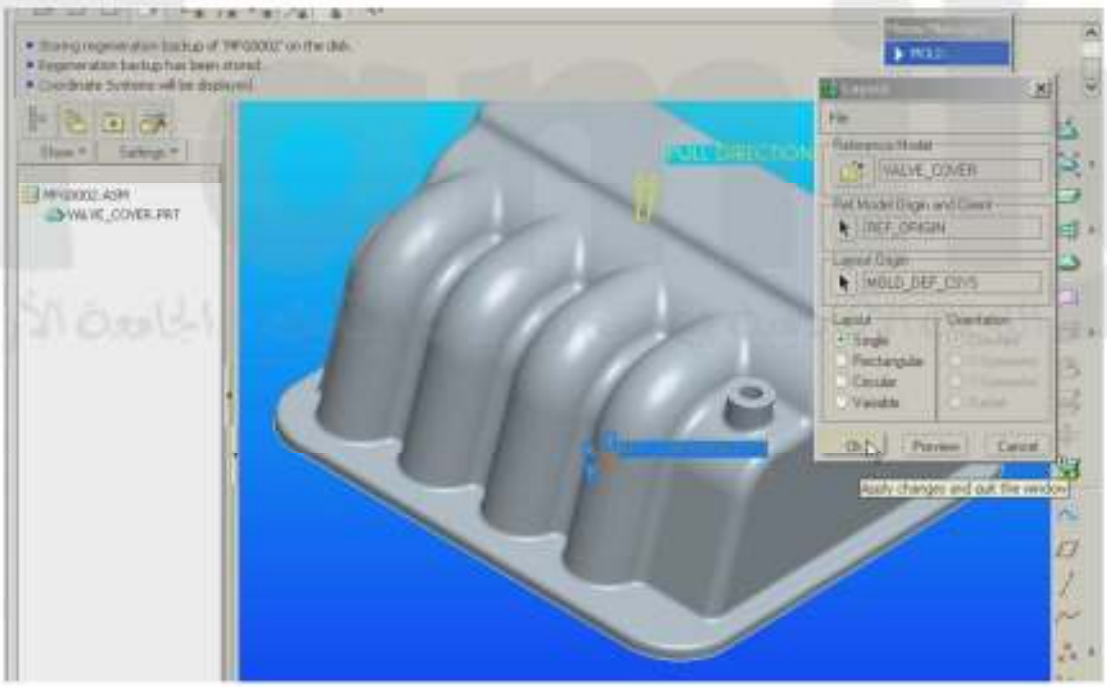
7. Select "Rotate", "Z", and enter "90°". Hit ok.



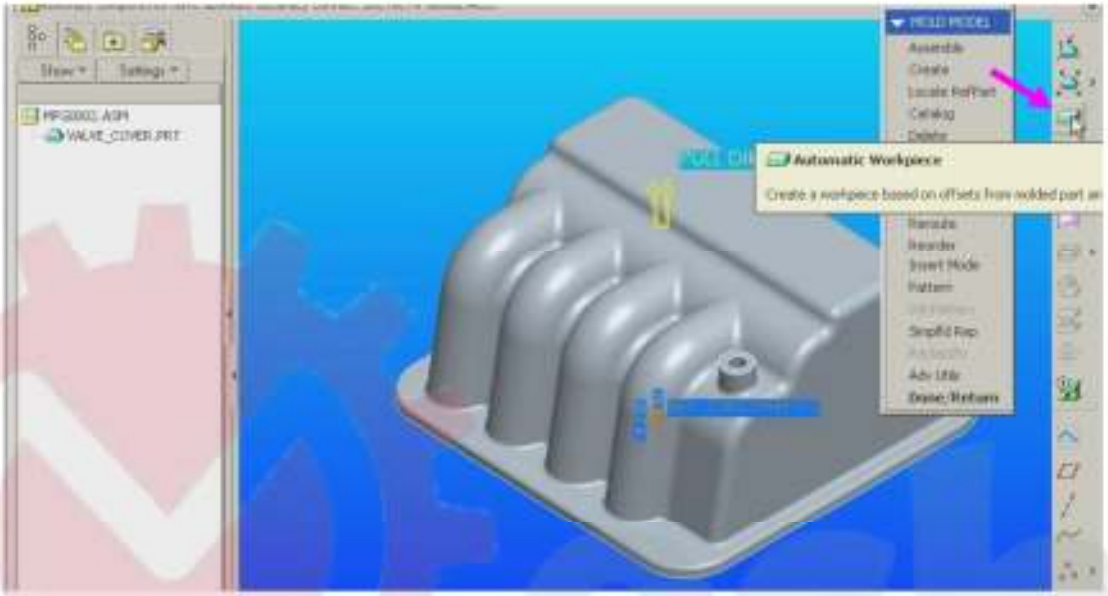
8. Now select “preview” to update the view. The “pull Direction” arrows should be pointing up and away from the top of your part.



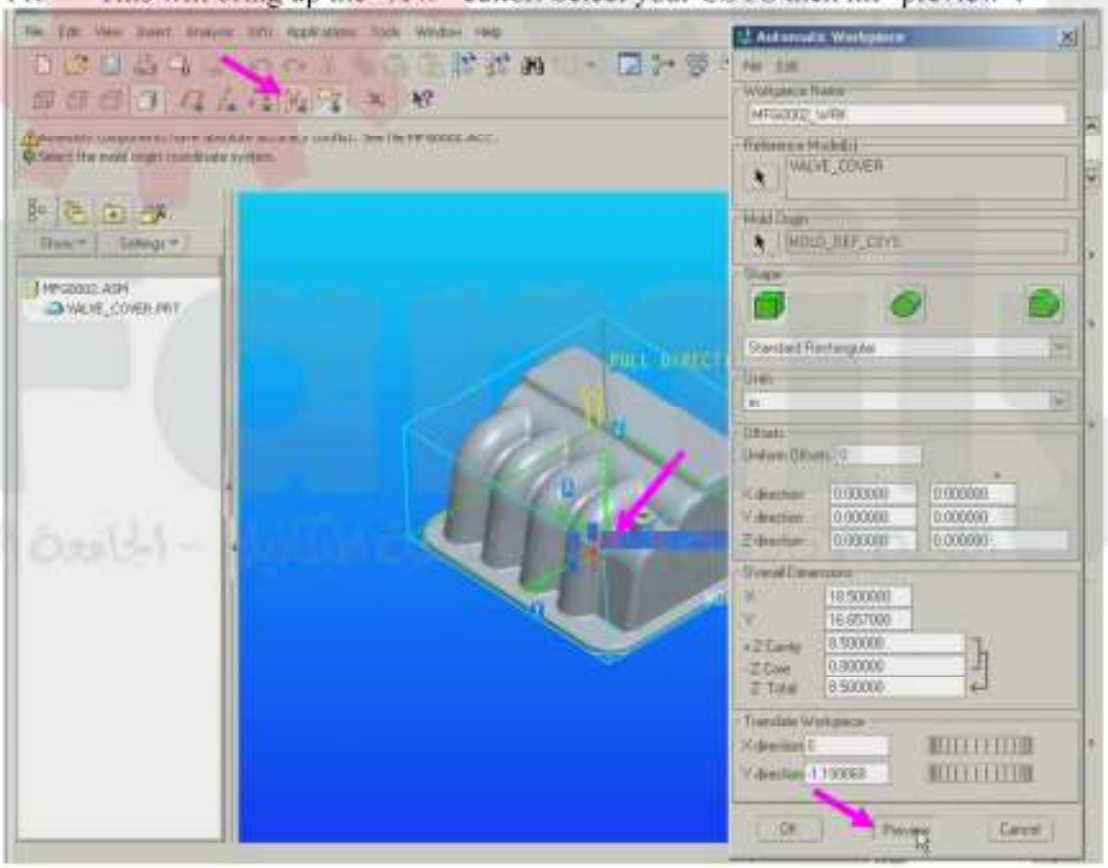
9. At this point we can now hit “OK” on the layout toolbar.



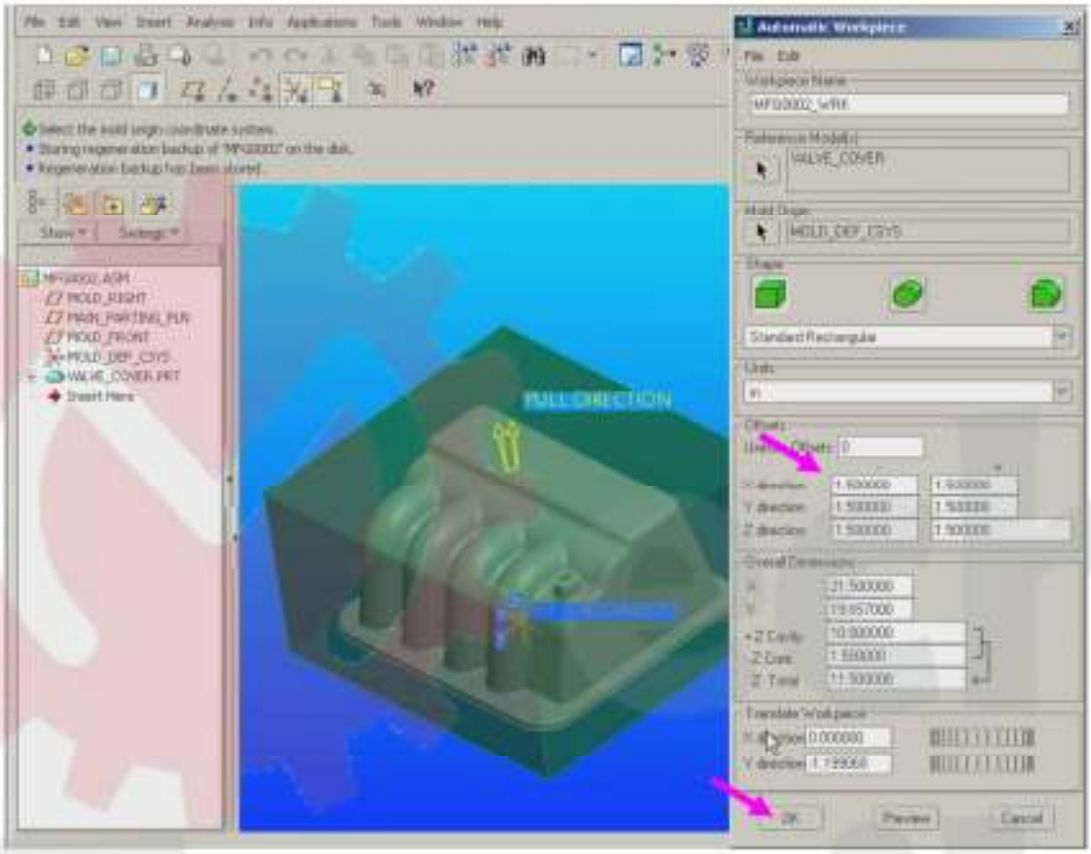
10. Once here select the “Automatic Workpiece” icon.



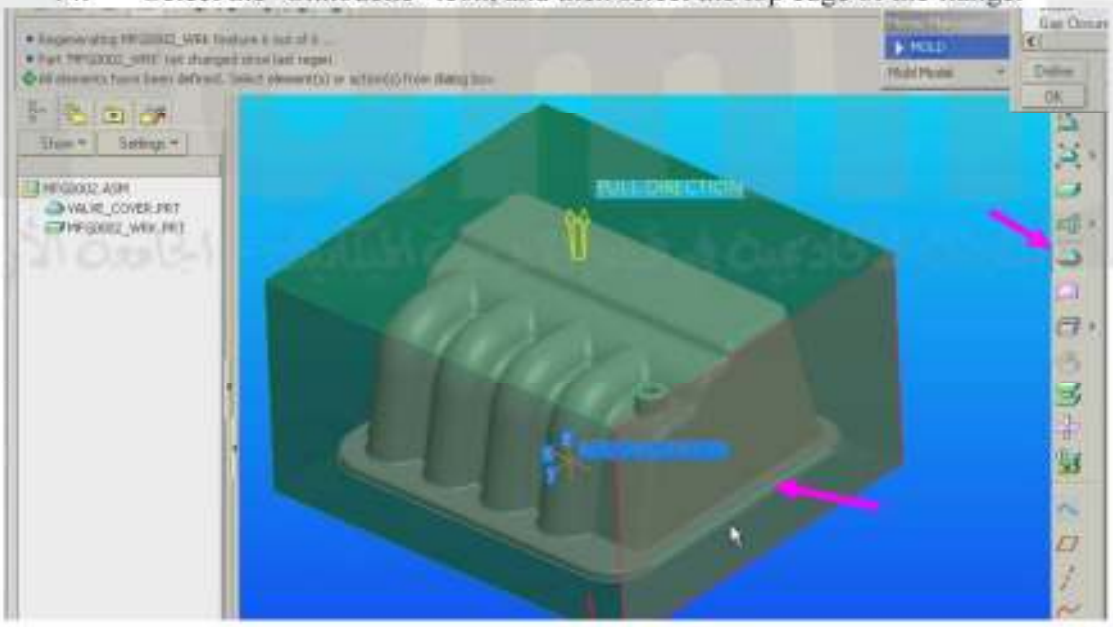
11. This will bring up the “AW” editor. Select your CSYS then hit “preview”.



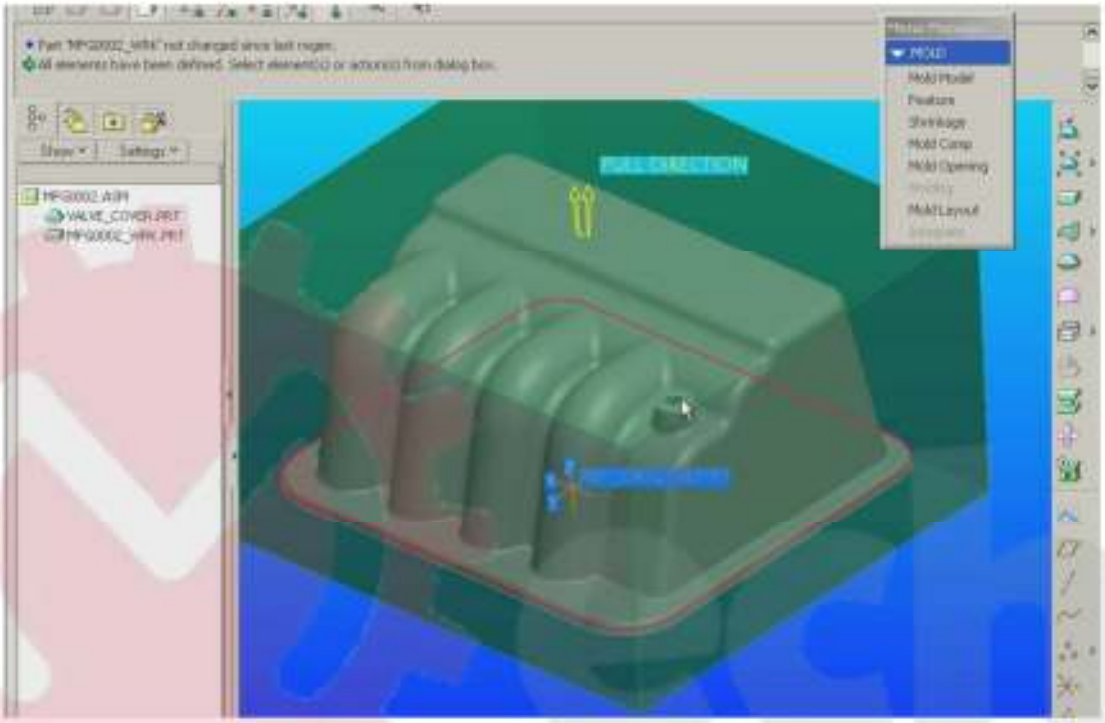
12. At this point you should now see a green transparent box around your work piece. We must not adjust the parameters as this is your stock to be machined.
13. Change the Offsets to 1.5 for all X, Y, and Z directions. Hit OK.



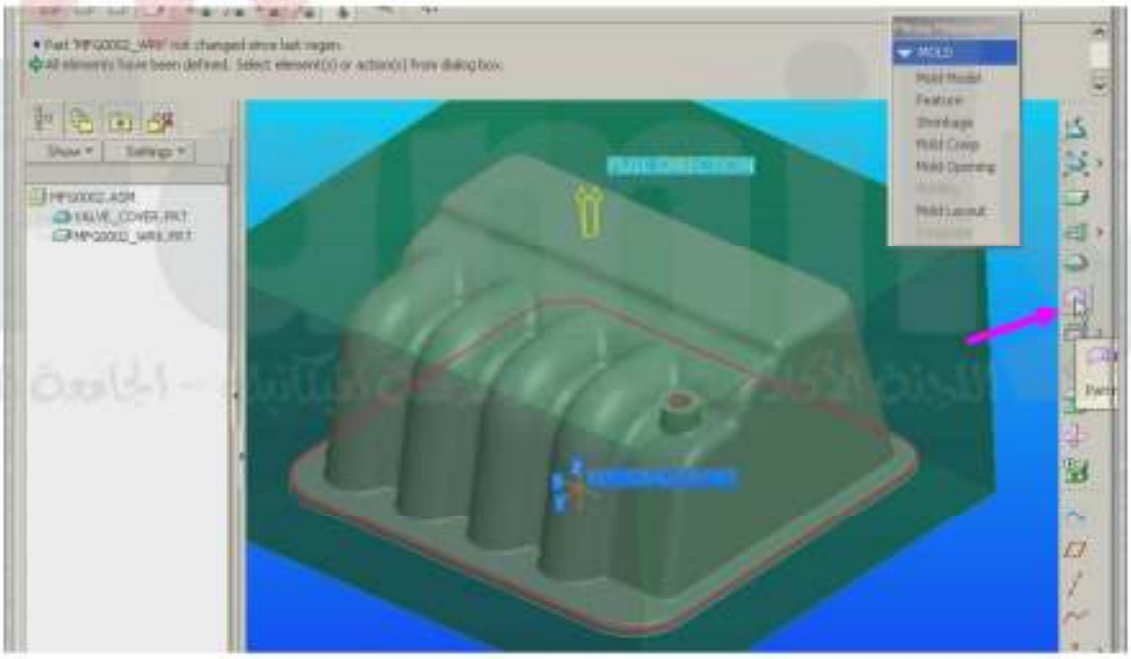
14. Select the "Silhouette" icon, and then select the top edge of the flange.



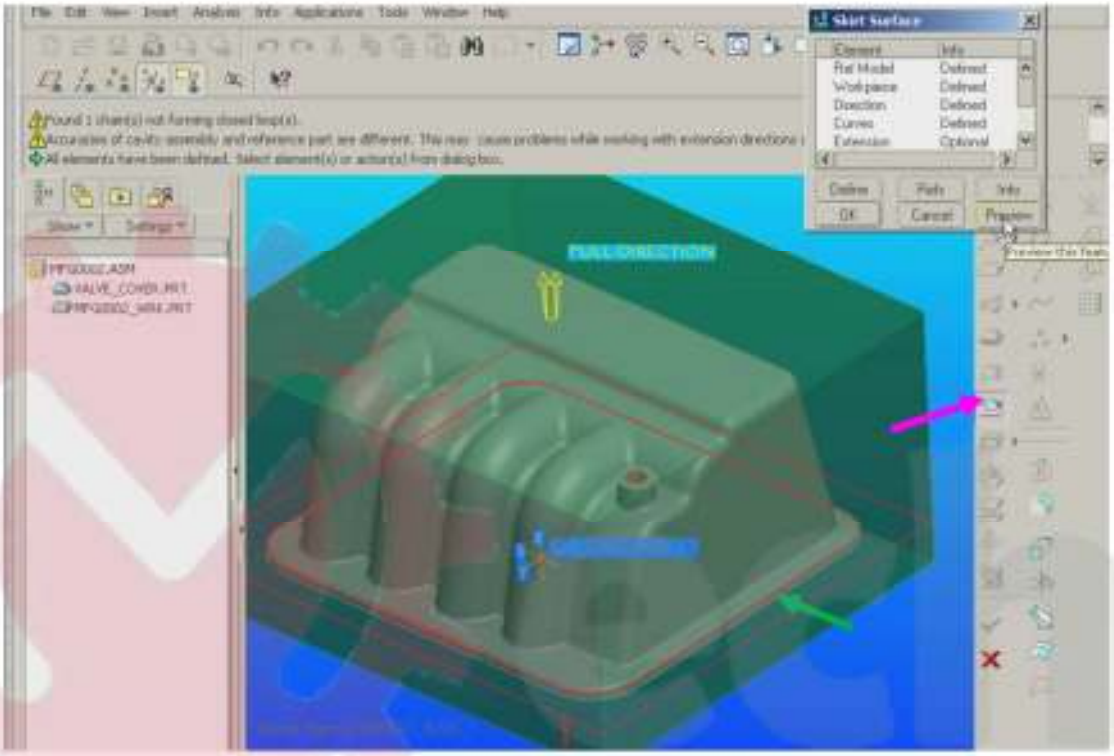
15. This generated parting line curves to be used to automatically generate surface shut-offs and boundary extensions to split the mold.



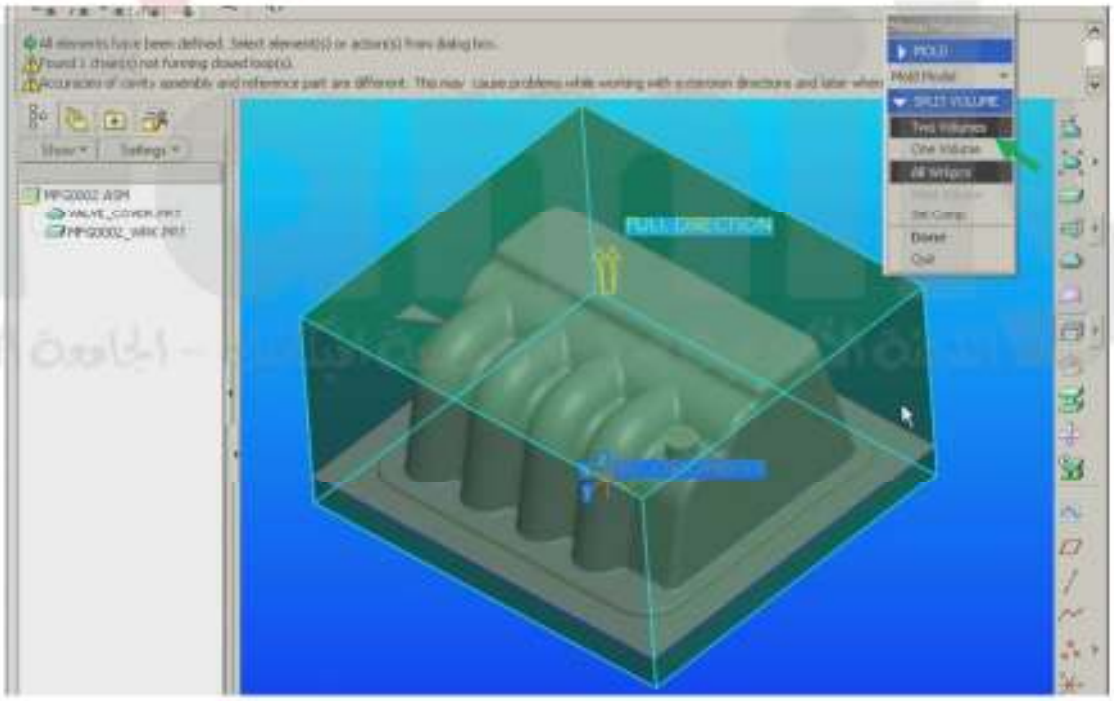
16. Select the parting surface icon.



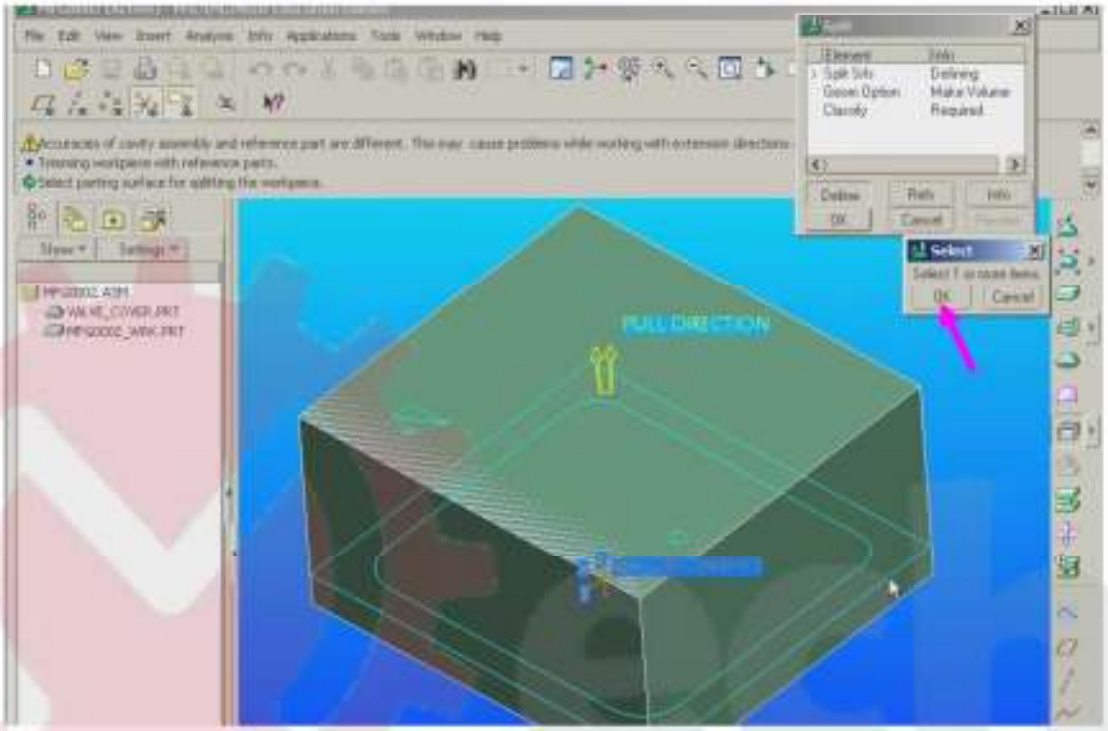
17. Select "**Skirt Surface**" and select the parting line **edge curve**, hit "preview" or "OK" to verify.



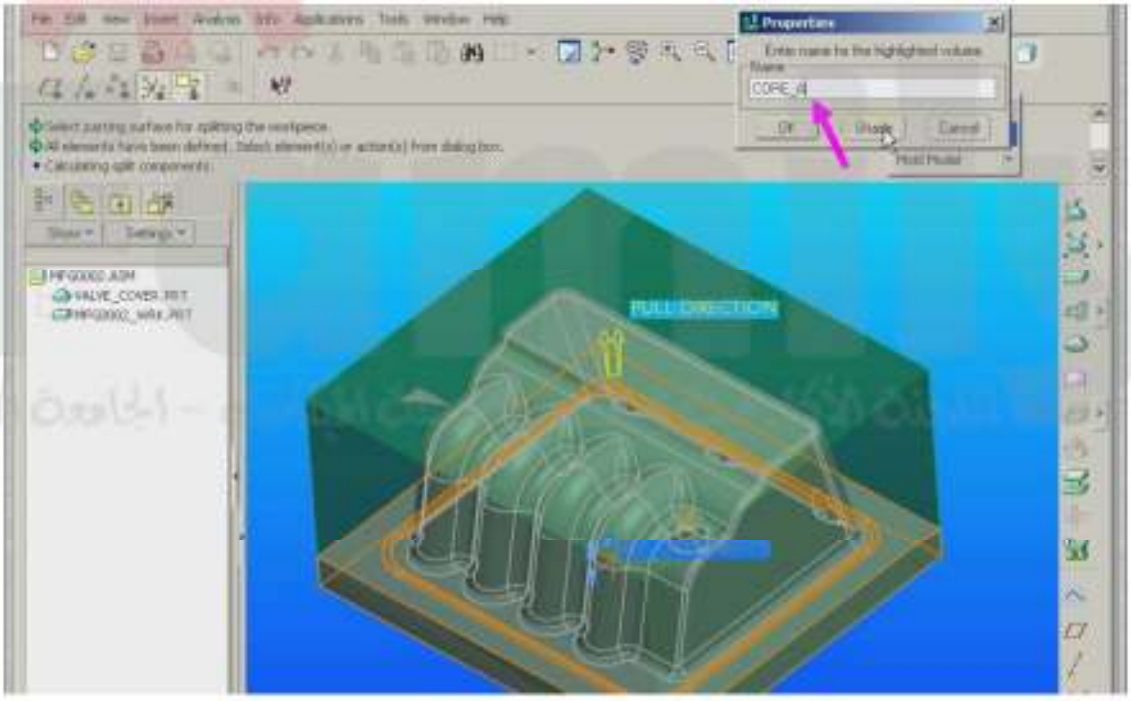
18. Select the "**Split Volume**" icon and make sure "**Two Volumes**" is selected.



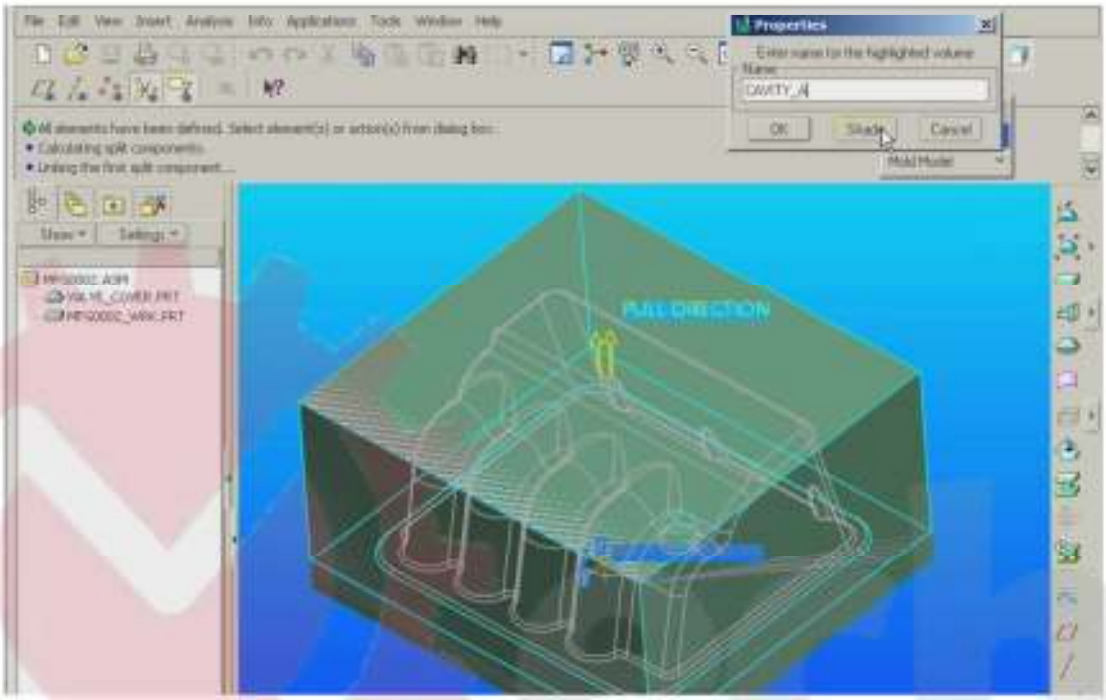
19. Select the skirt surface and hit “OK”.



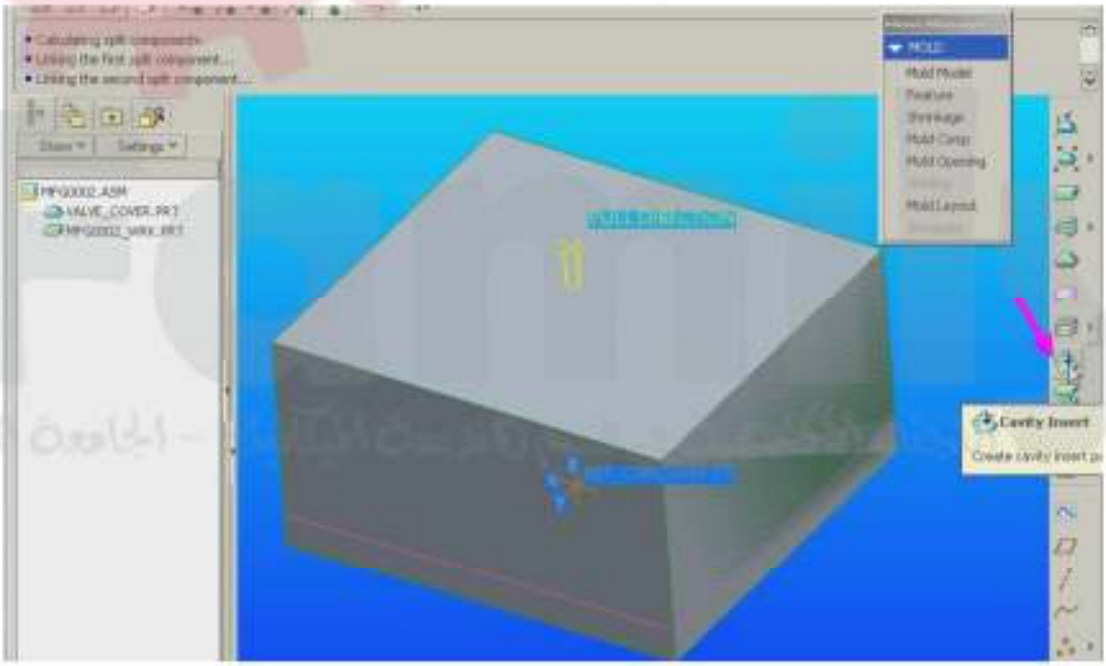
20. Enter a new name for the core, (CORE_A). Hit “OK”.



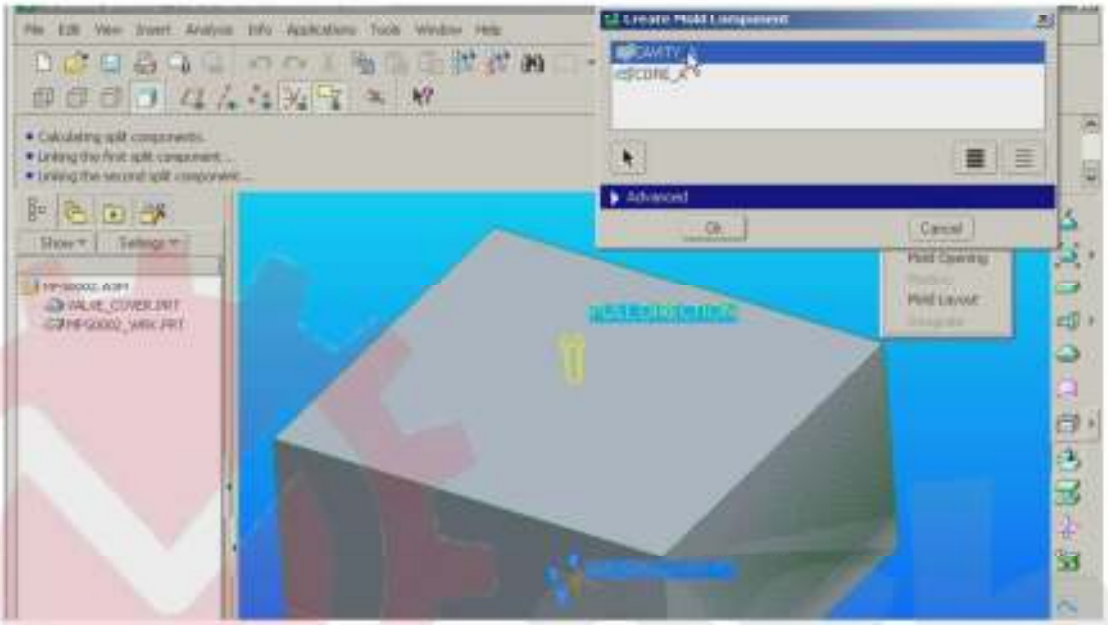
21. Enter (CAVITY_A) as the new name for the next section. “Hit



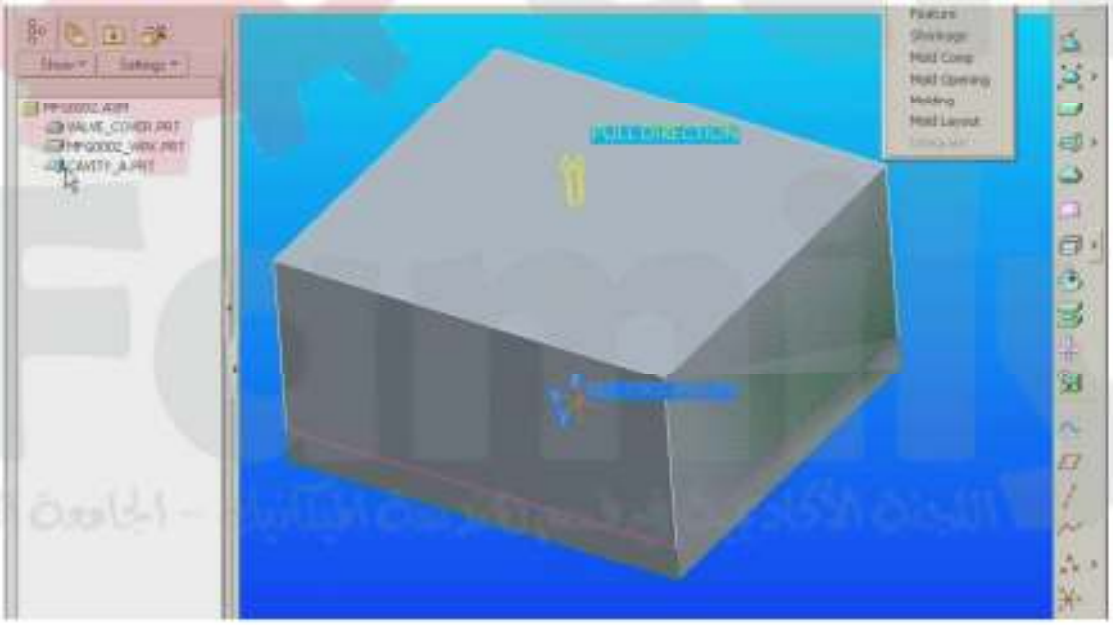
22. Select the “Cavity Insert” icon to create “official” cavity and core parts.



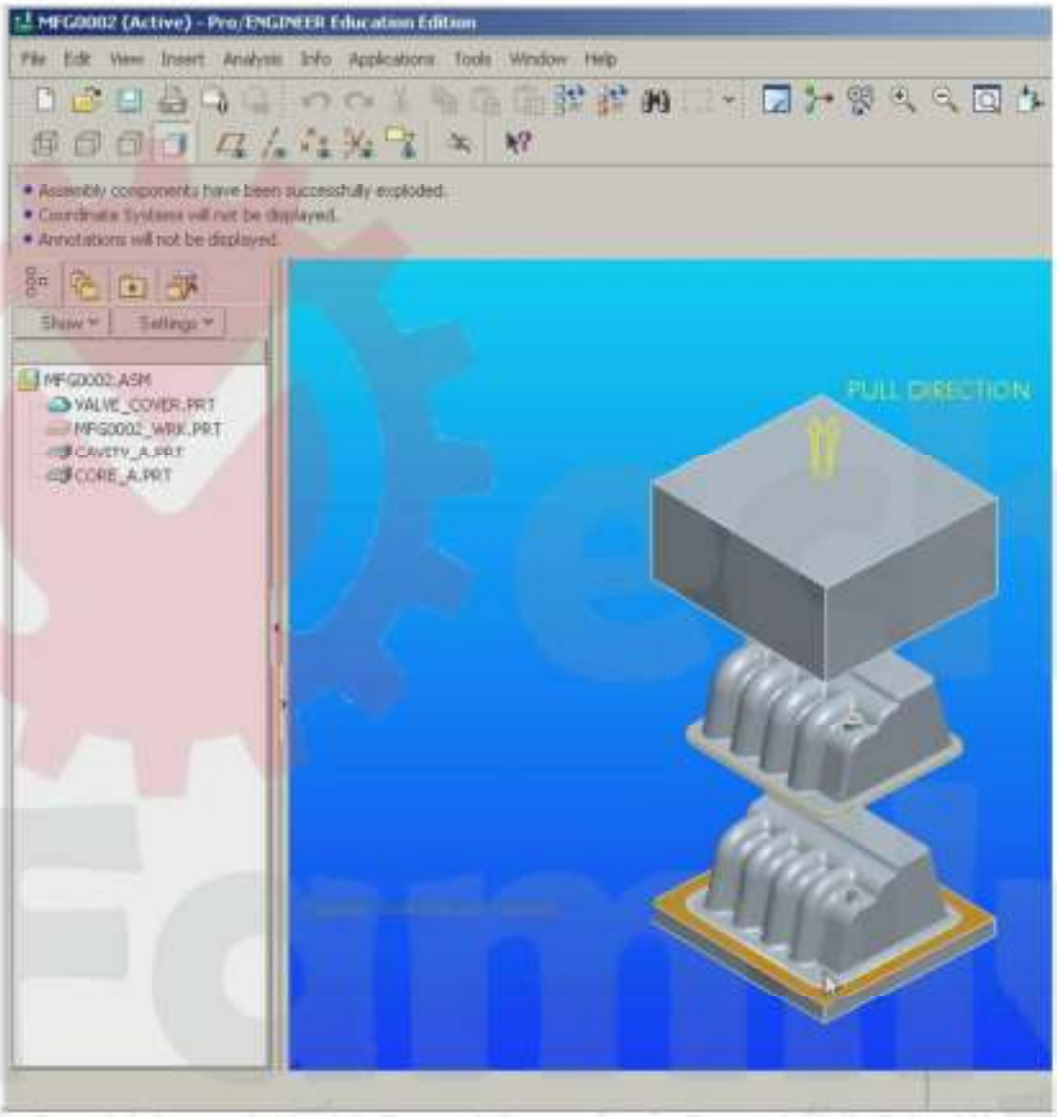
23. Select “CAVITY_A” from the list and hit “OK”.



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

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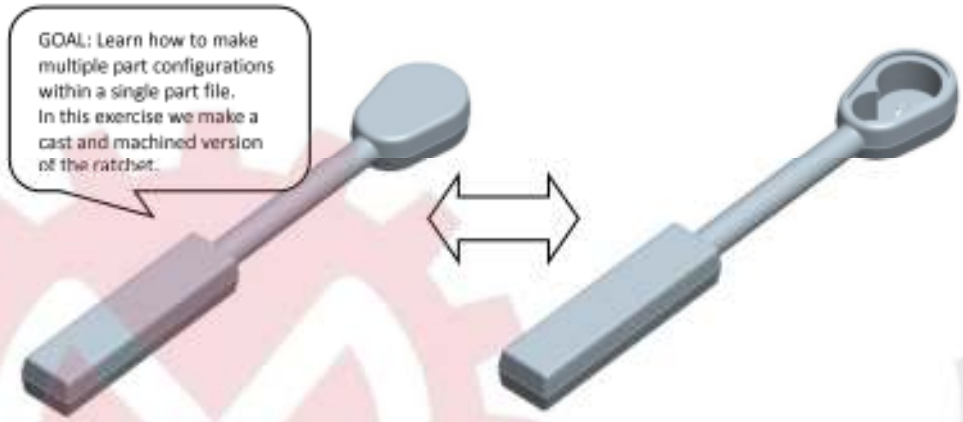
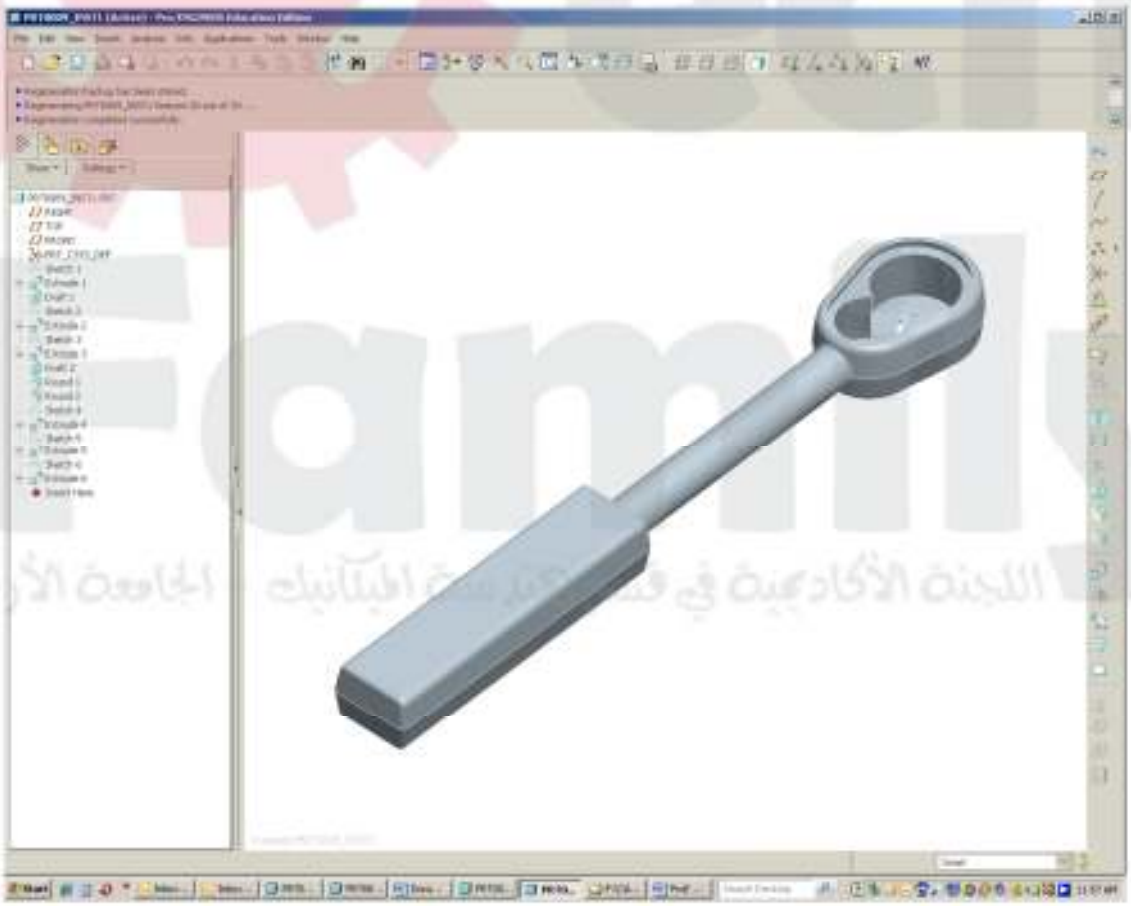
EXERCISE 16
Family Tables

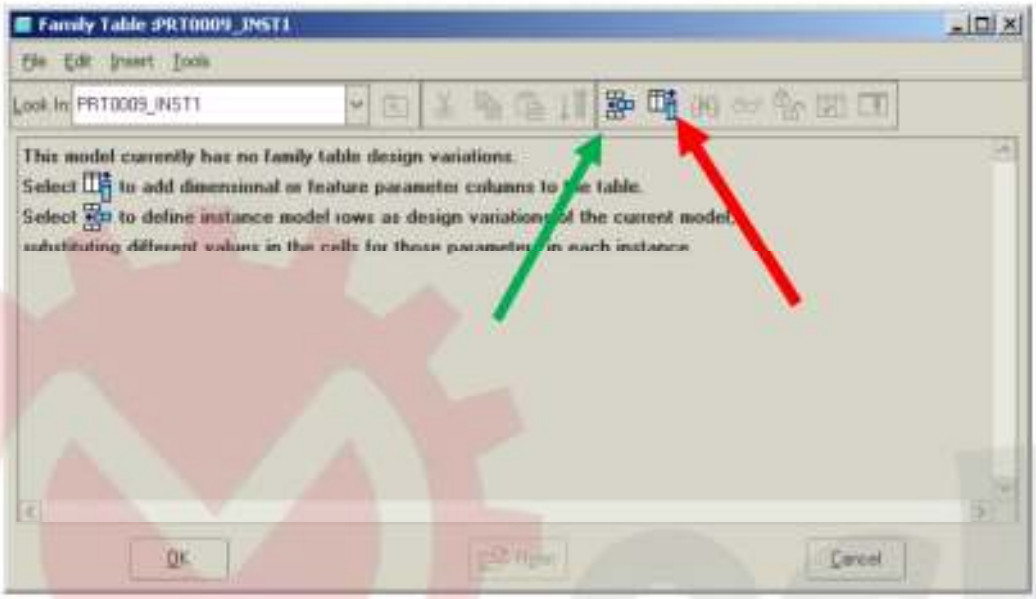
Family Tables enable you to create multiple part configurations derived from a single part file.



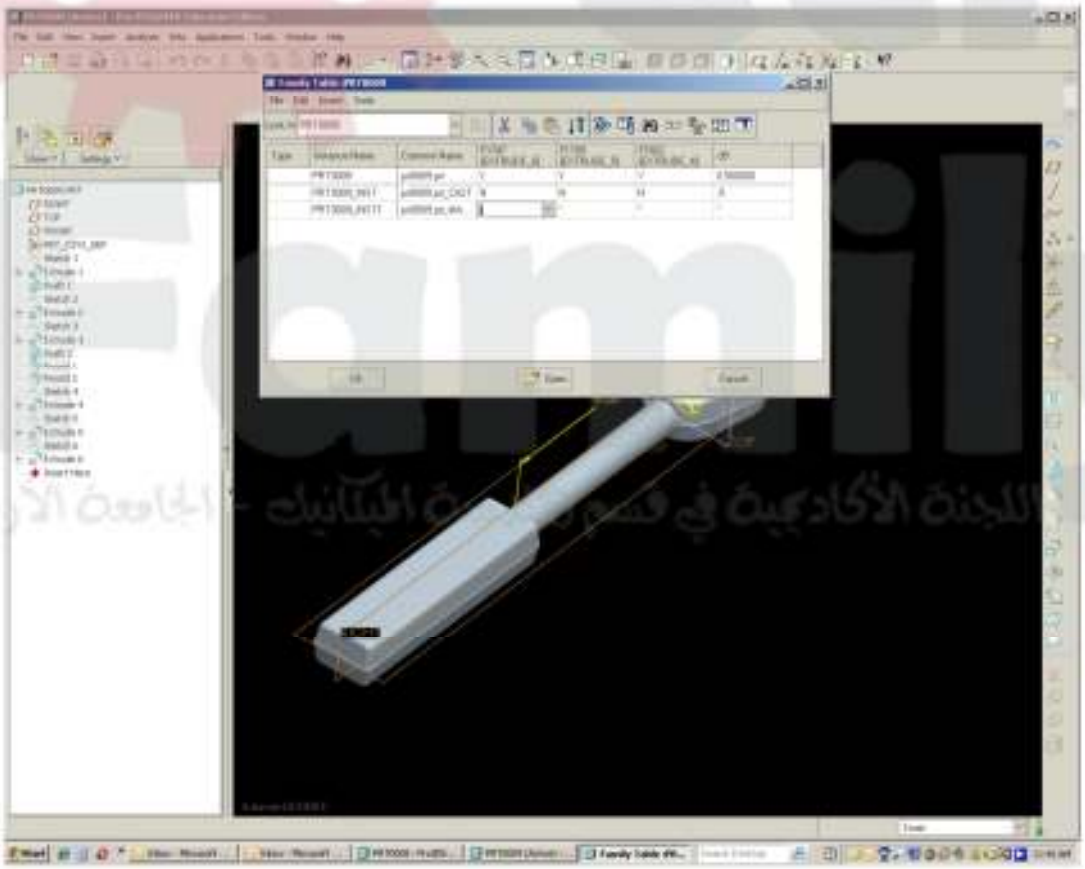
1. Open the Exercise_16 part file.



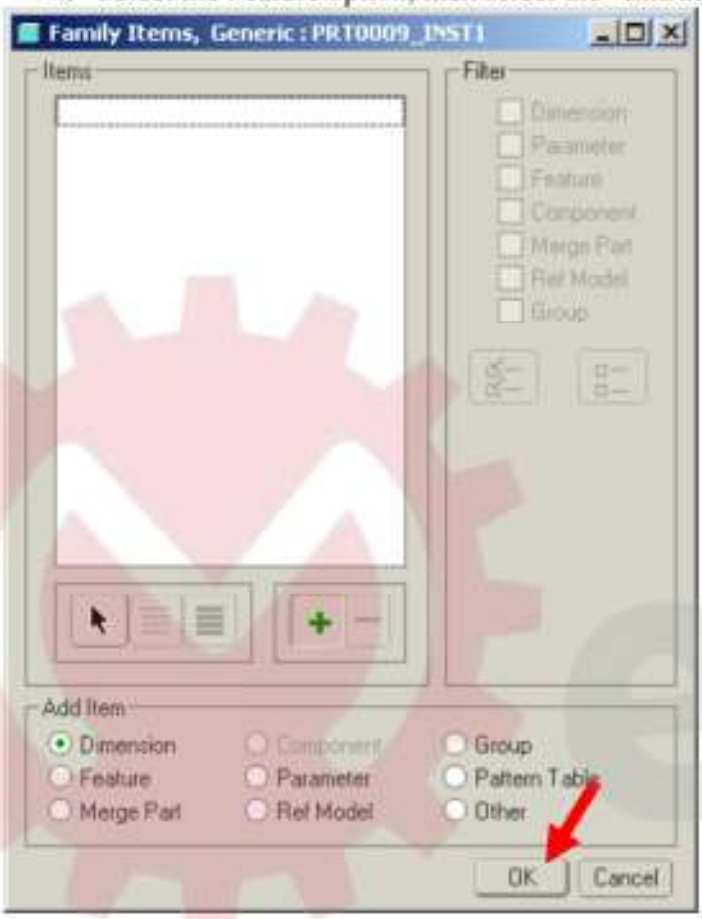
2. Go to the pull down menu- “Tools/Family Tables”



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



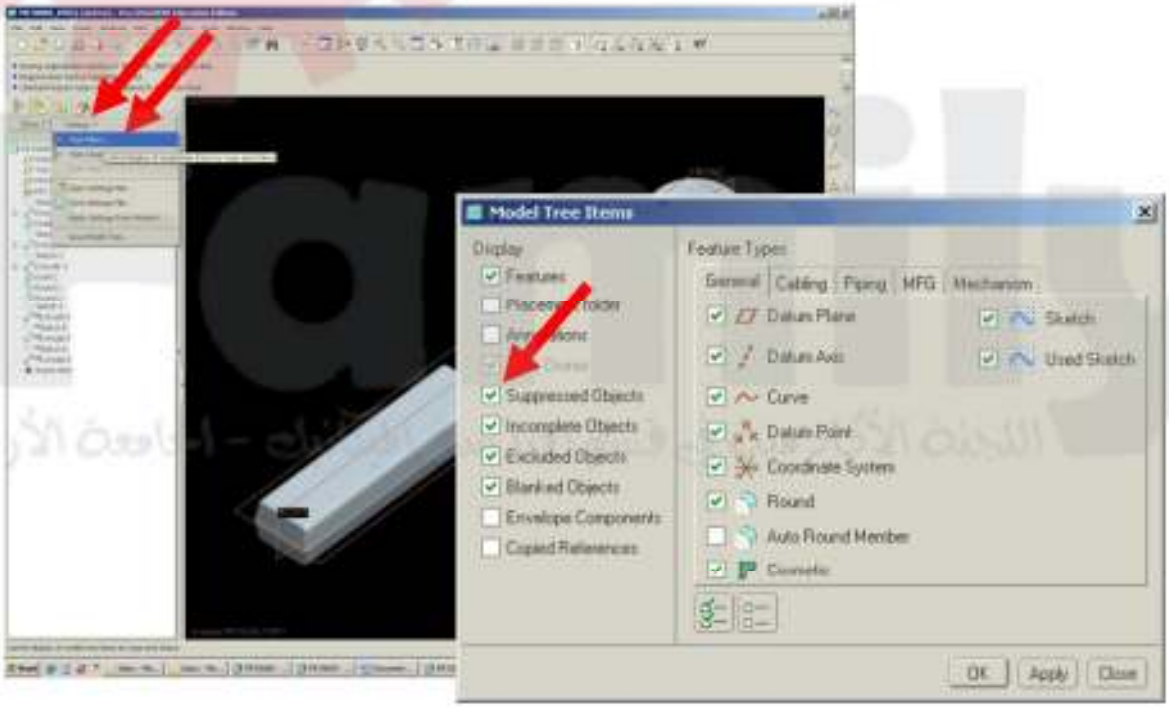
4. Select the Feature option, then select the "Extrude 4, 5, and 6"



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

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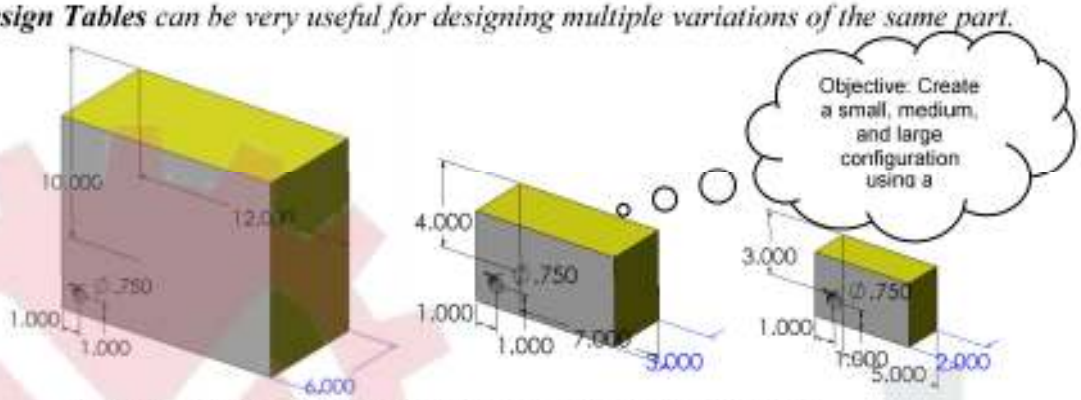
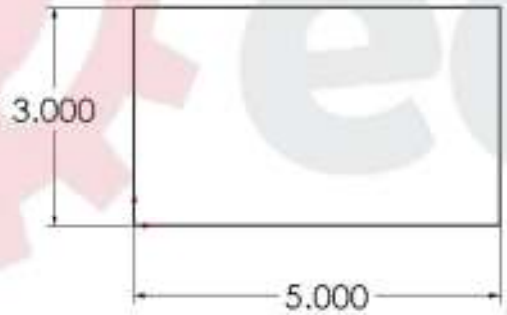
EXERCISE 16b
Configurations with Family Tables

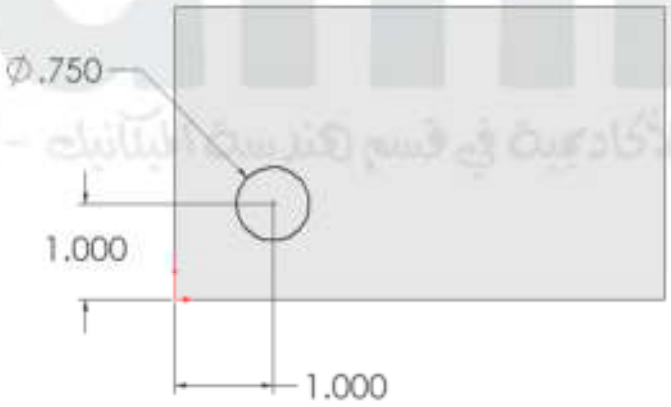
Design Tables *can be very useful for designing multiple variations of the same part.*



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*)



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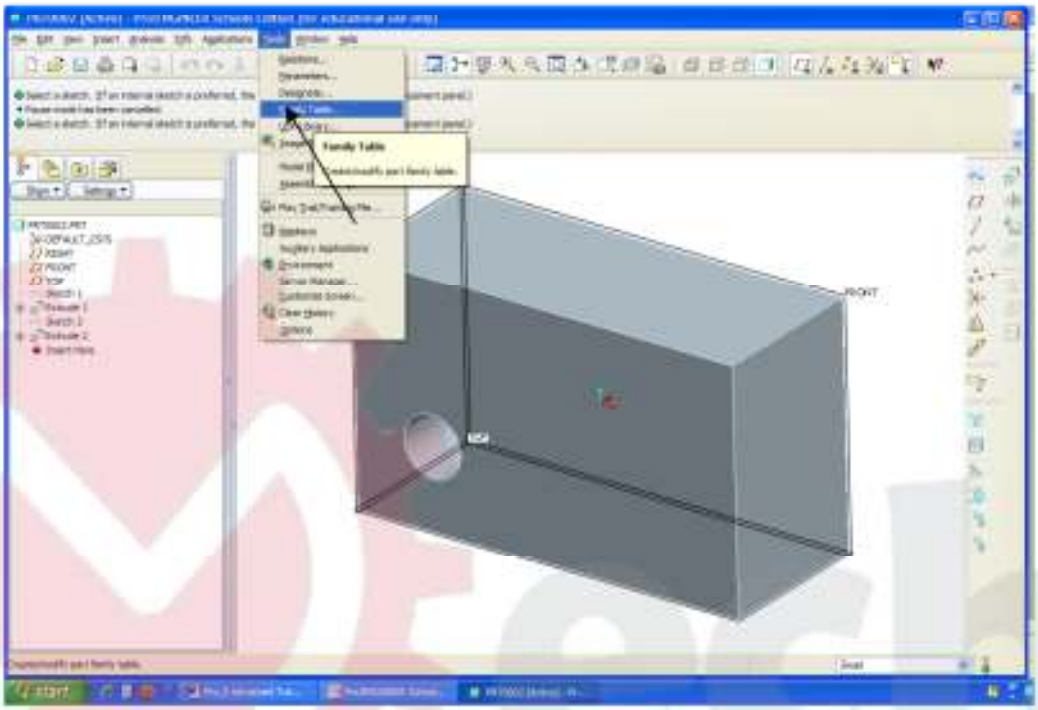
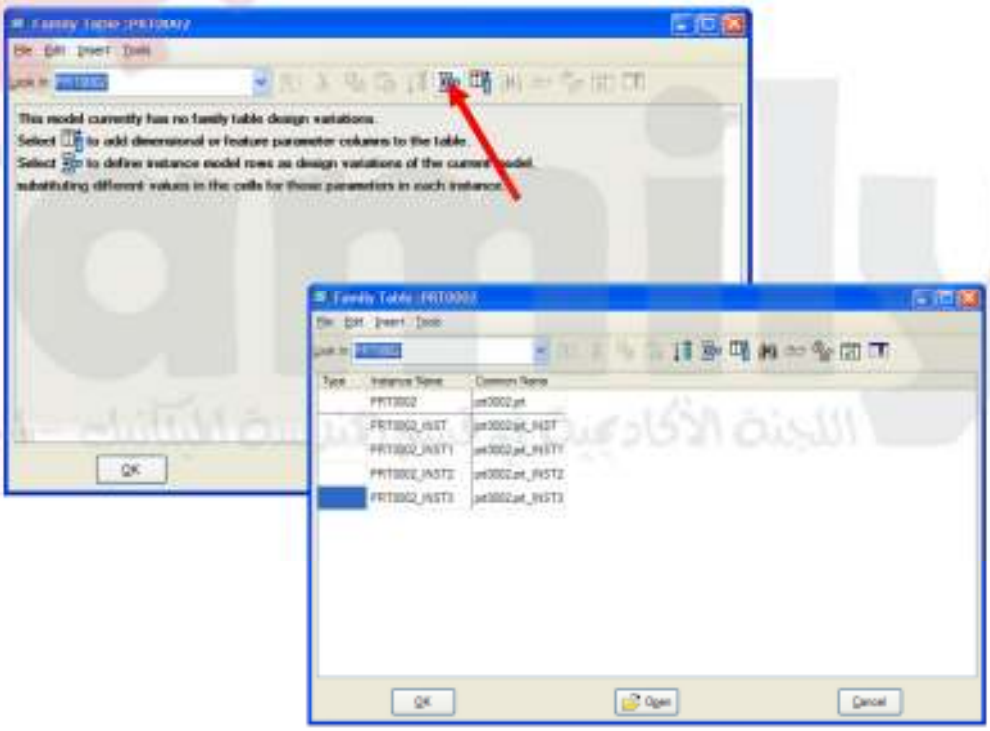
3. Click on the “Tools/Family Table” option.

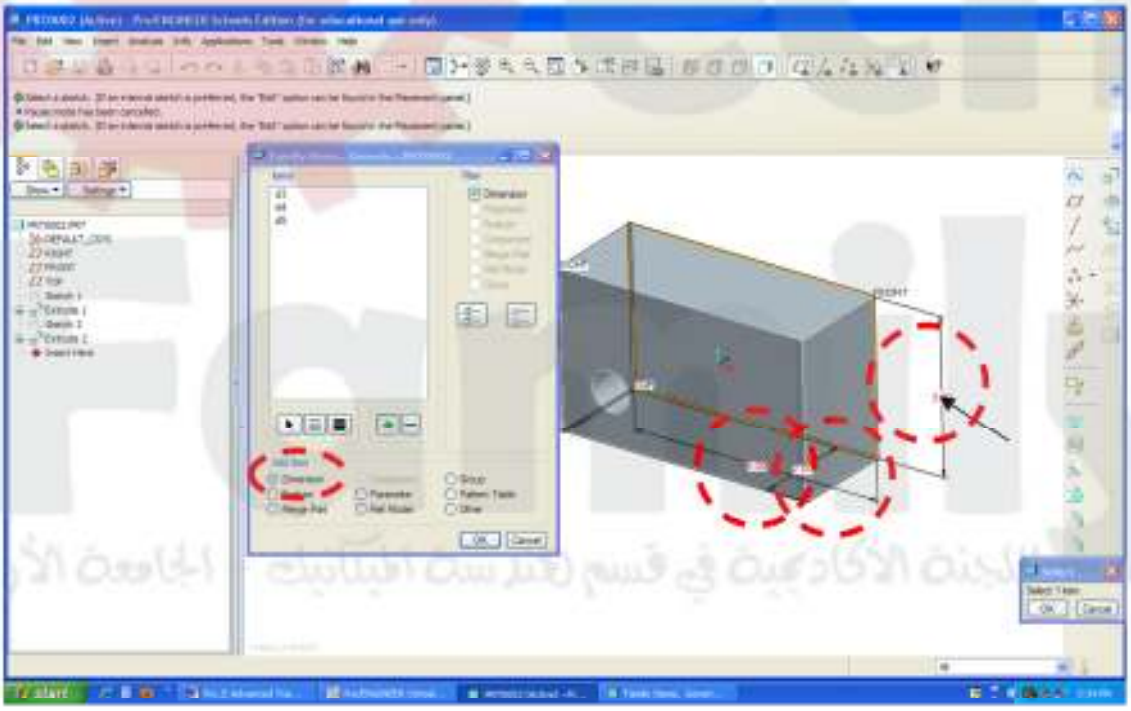


4. Select “Insert new instance” four times.

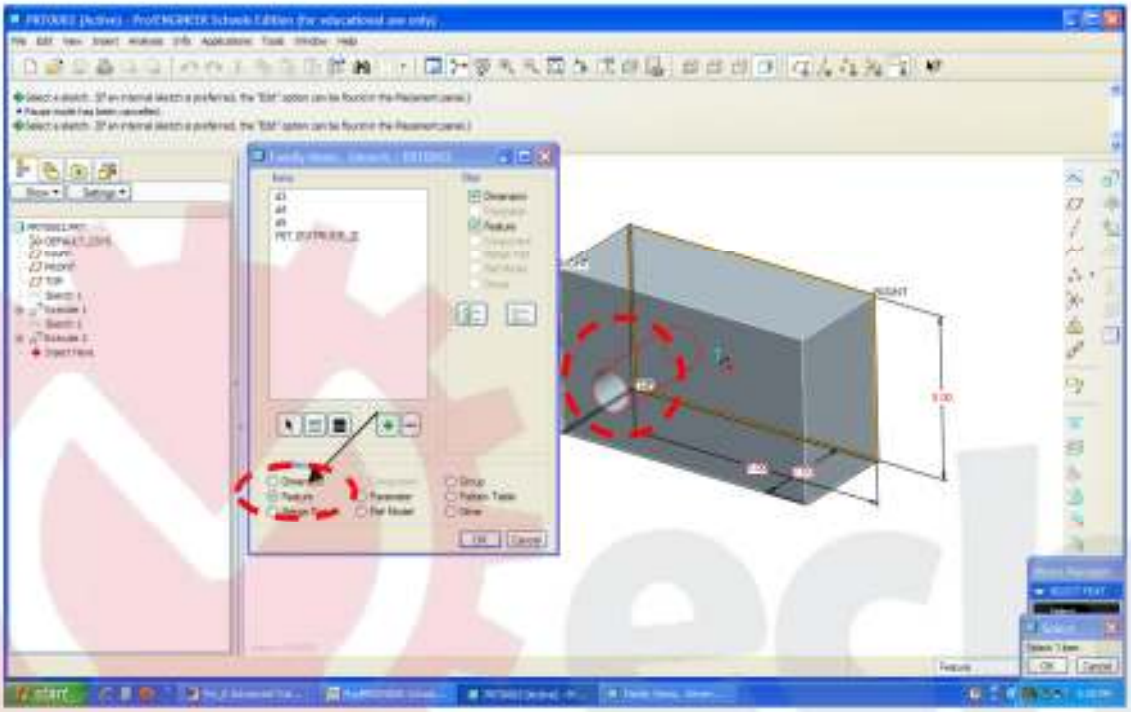


5. Select “Add/Delete” table columns”.

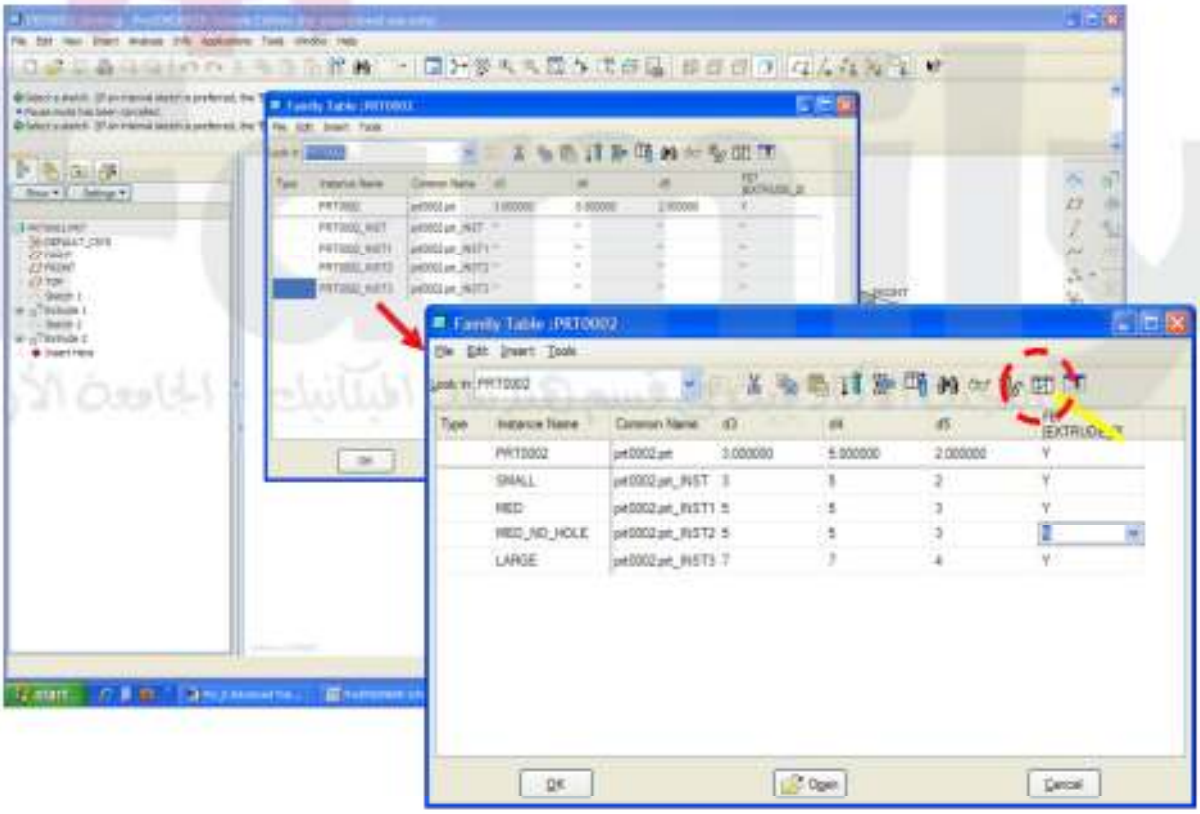
6. Select Dimensions then click select any model surface, the dimensions should appear, next, click on the 3”, 5”, and 2” dimensions on the model.



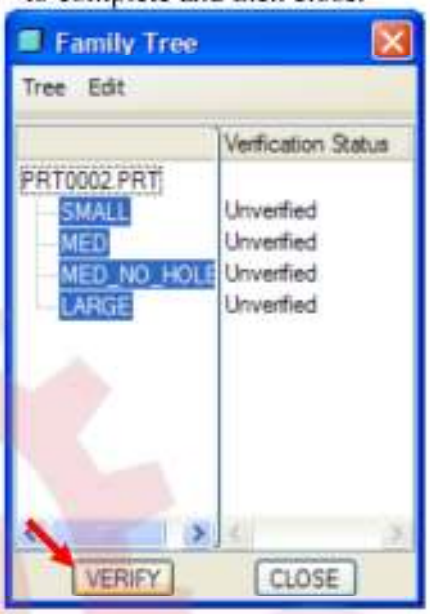
7. Select the "Feature" option, and continue to select the inside surface of the hole, not the edge of the hole. Hit "OK"



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



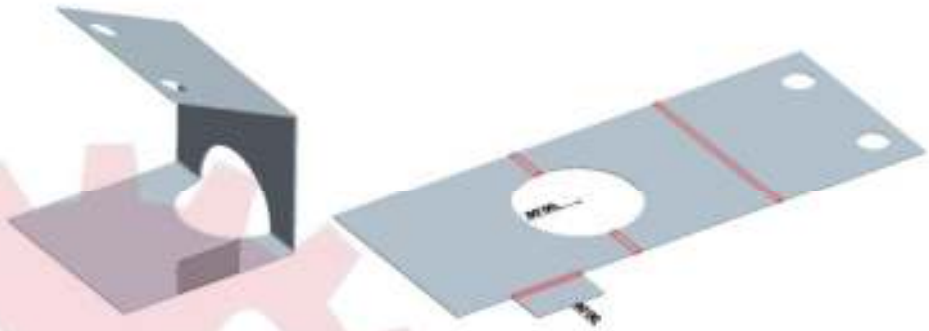
9. Click on "Verify" to complete and then close.



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.)

EXERCISE 17

Sheet Metal Fabrication

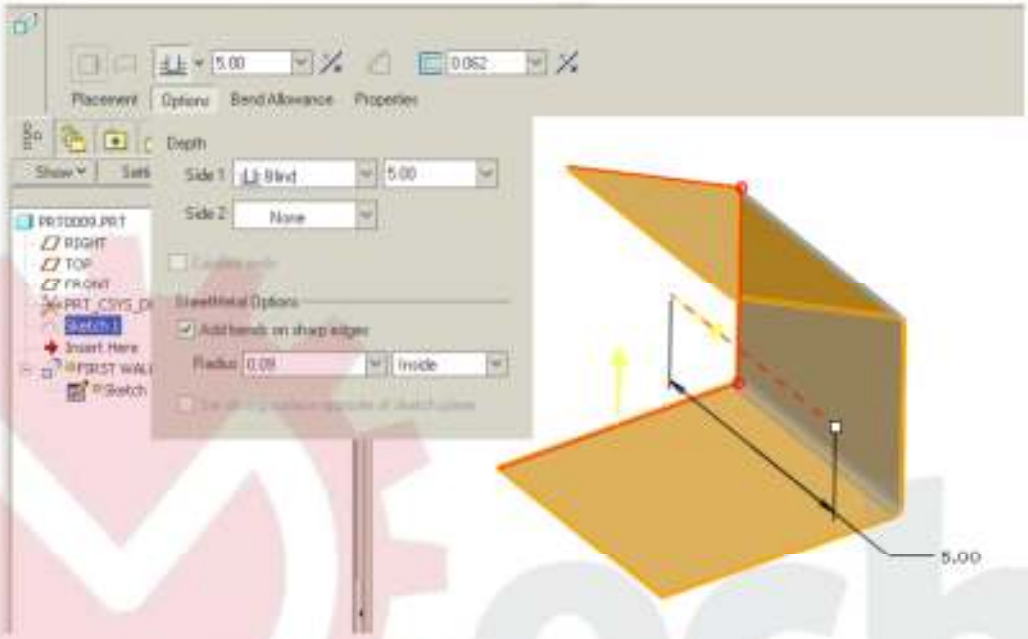


1. Start a new part file and select "Sheet Metal".

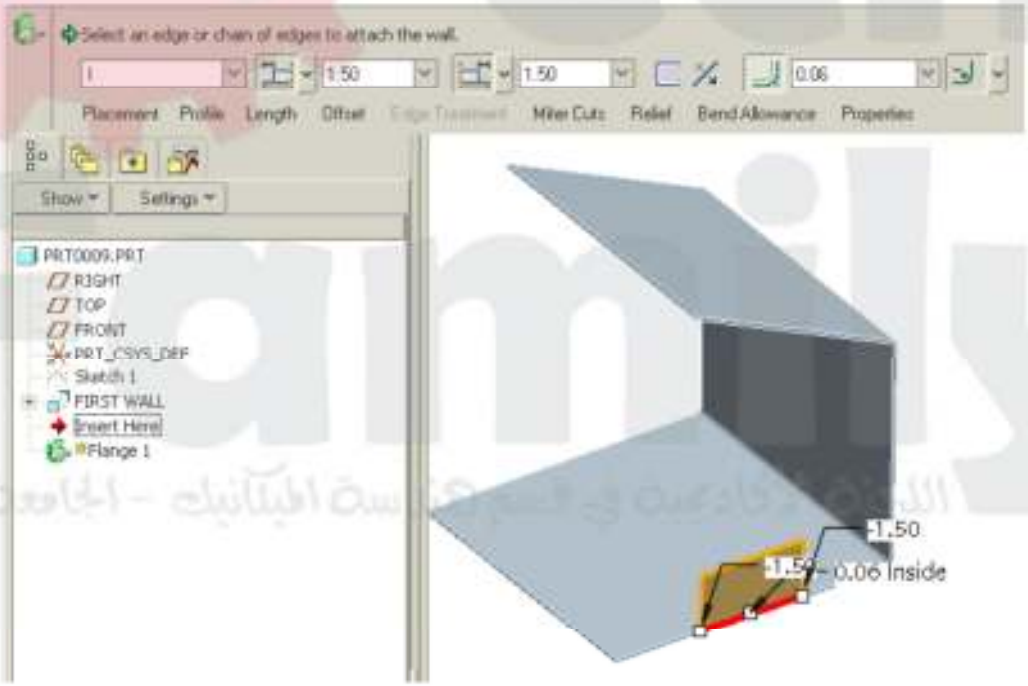
2. Sketch the following on the Front Plane

3.

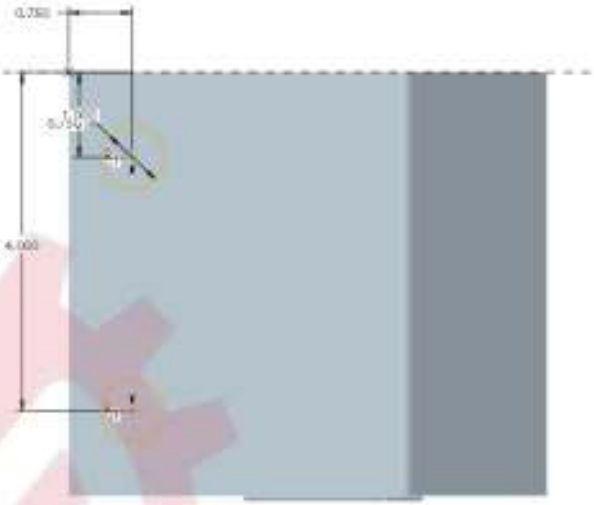
4. Select the Options Tab and fill in as shown.



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

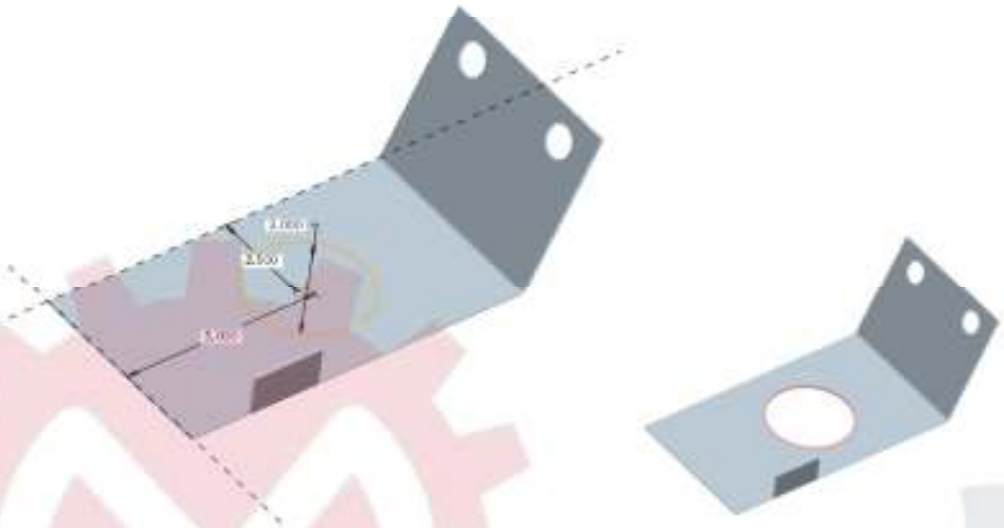


6. Start a sketch on the top flange and draw the following.

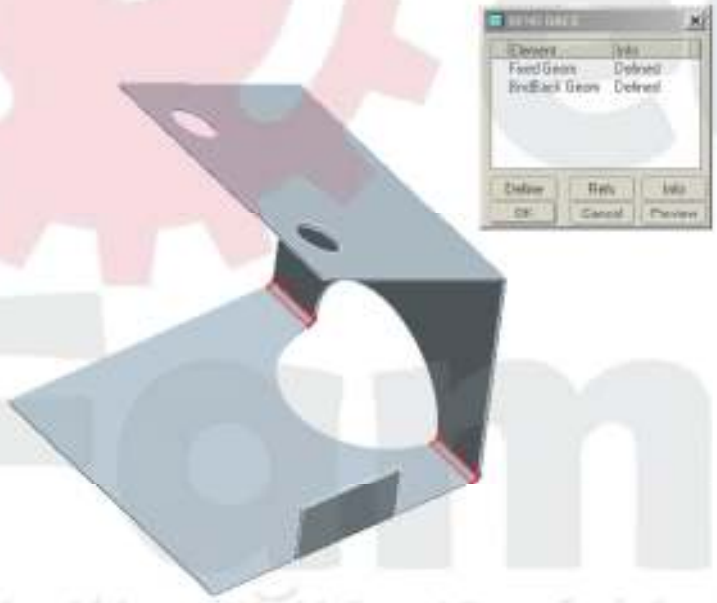


7. Unbend a bend using the "unbend tool" Select the Fixed face (bottom surface of flange), then select the actual bend surface.

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



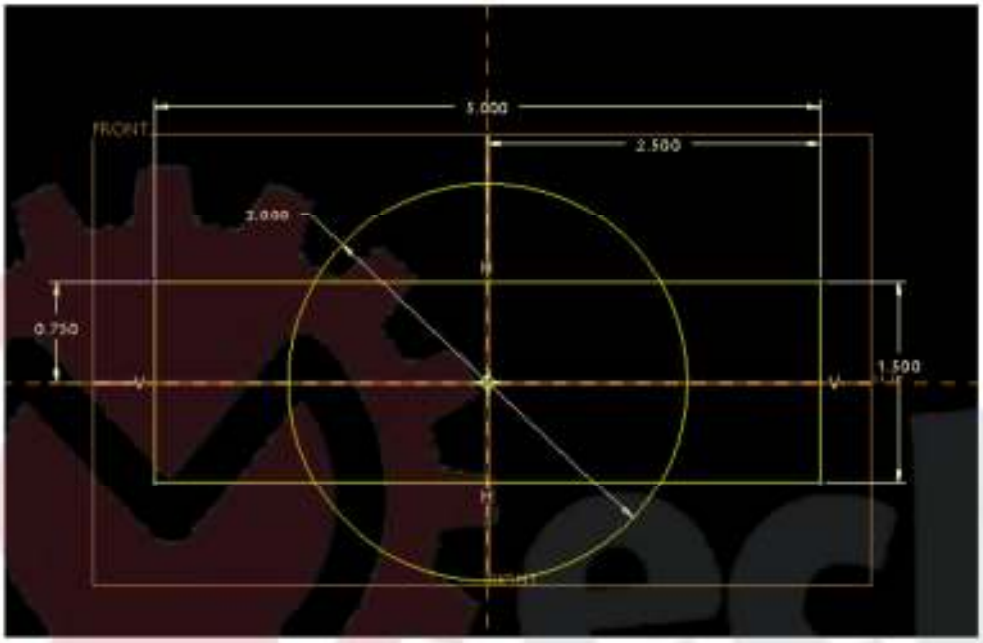
9. Use Bend Back.



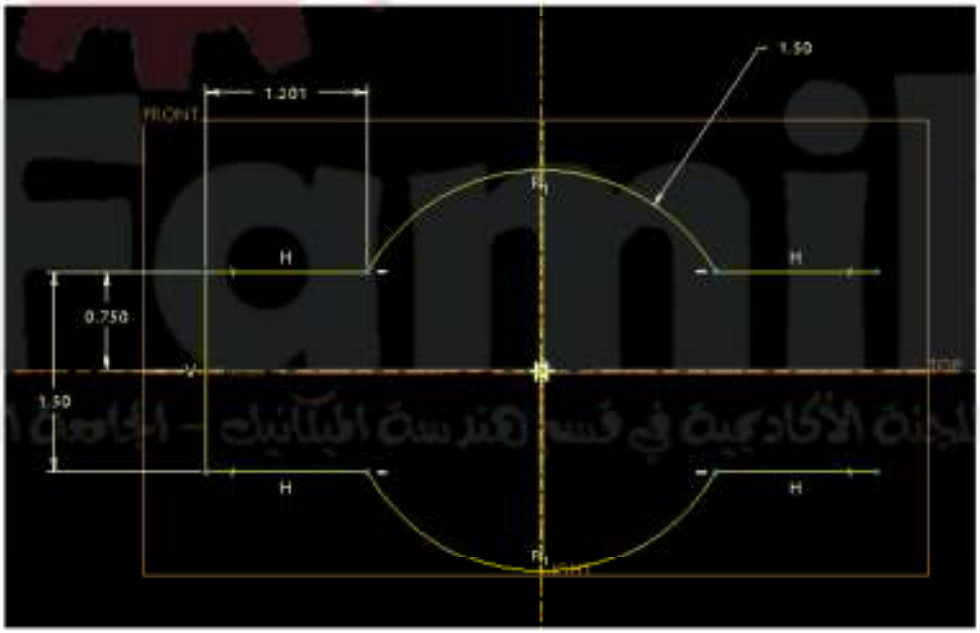
Tech Family
اللجنة الأكاديمية في قسم هندسة الميكاترونيك - الجامعة الأردنية

LAB 17

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

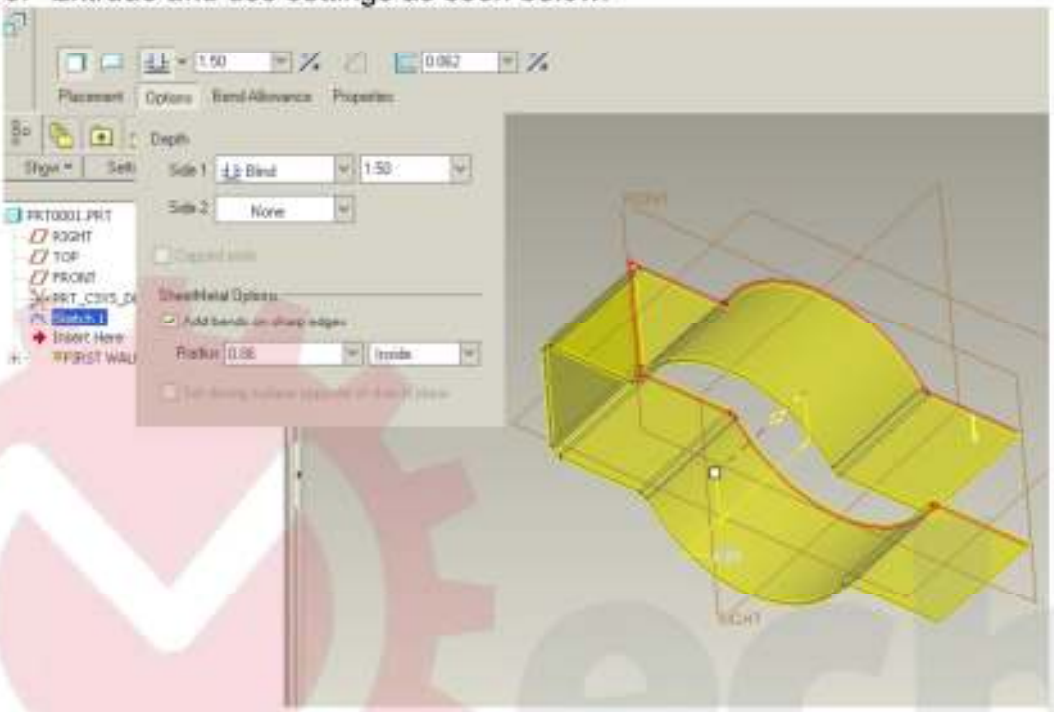


2. Trim

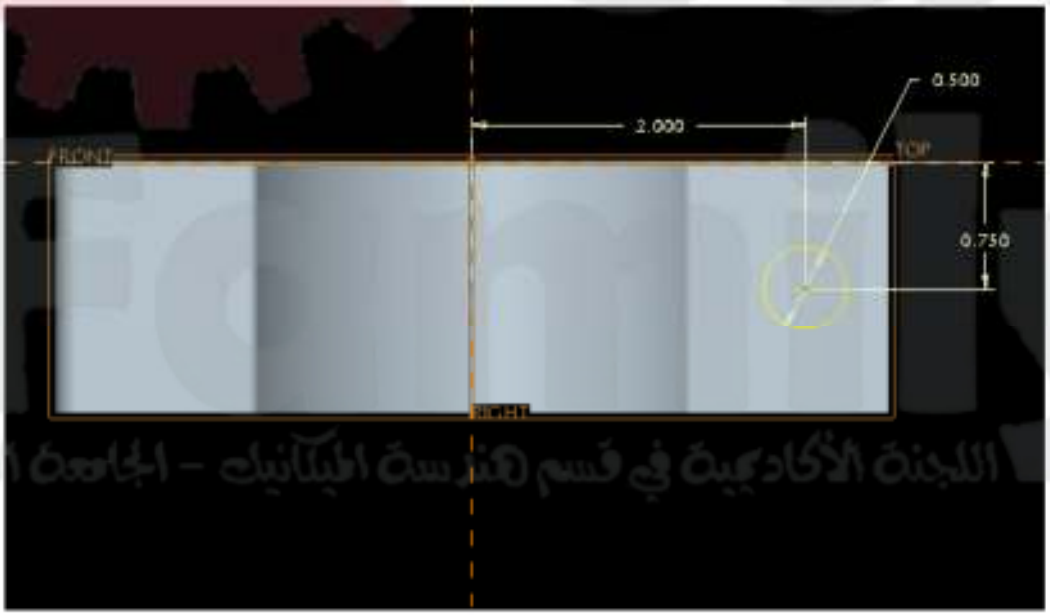


اللجنة الأكاديمية في قسم هندسة الميكاترونيكس – الجامعة الأردنية

3. Extrude and use settings as seen below.

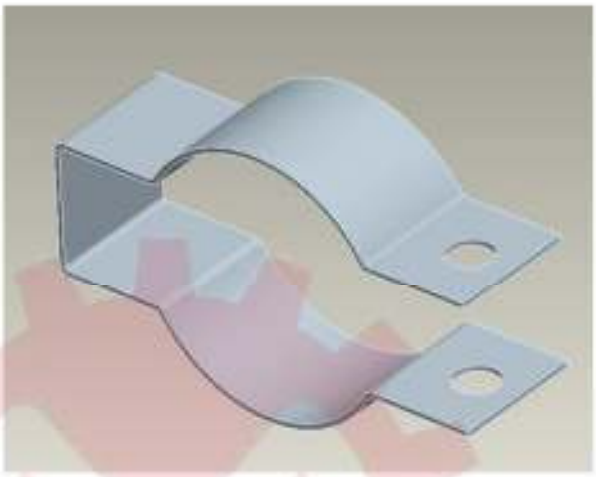


4. Start a sketch on the top surface of the flange, and draw the following.

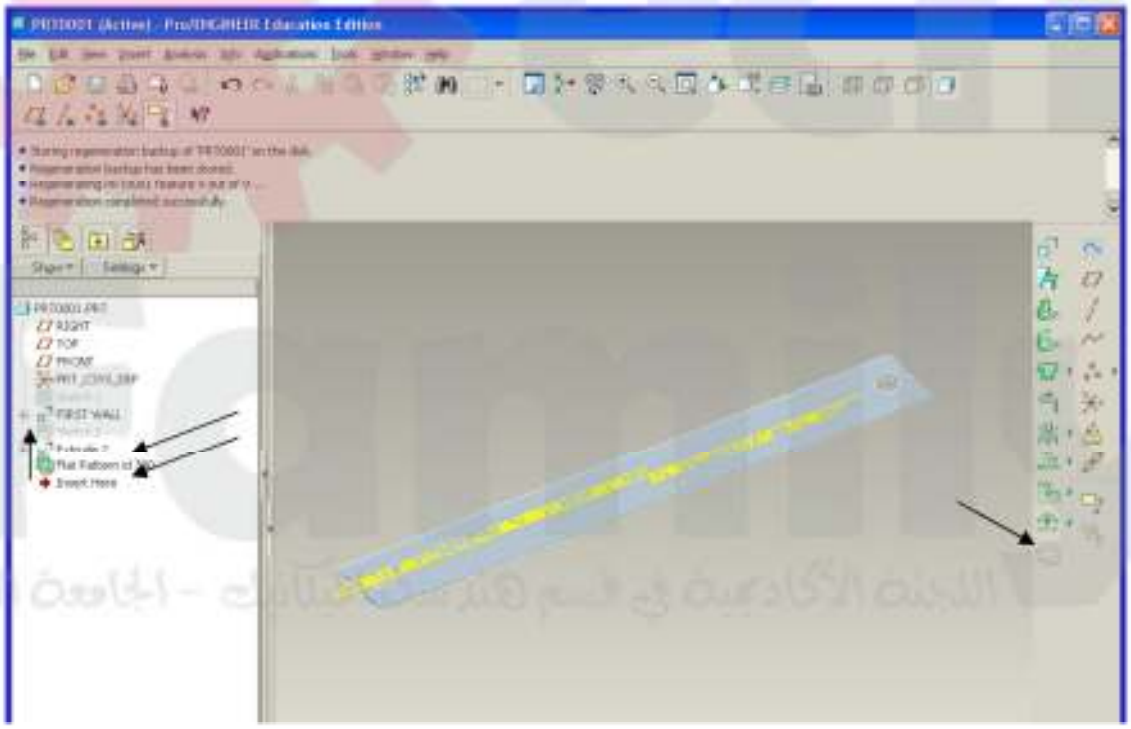


اللجنة الأكاديمية في قسم هندسة الميكانيك – الجامعة الأردنية

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.

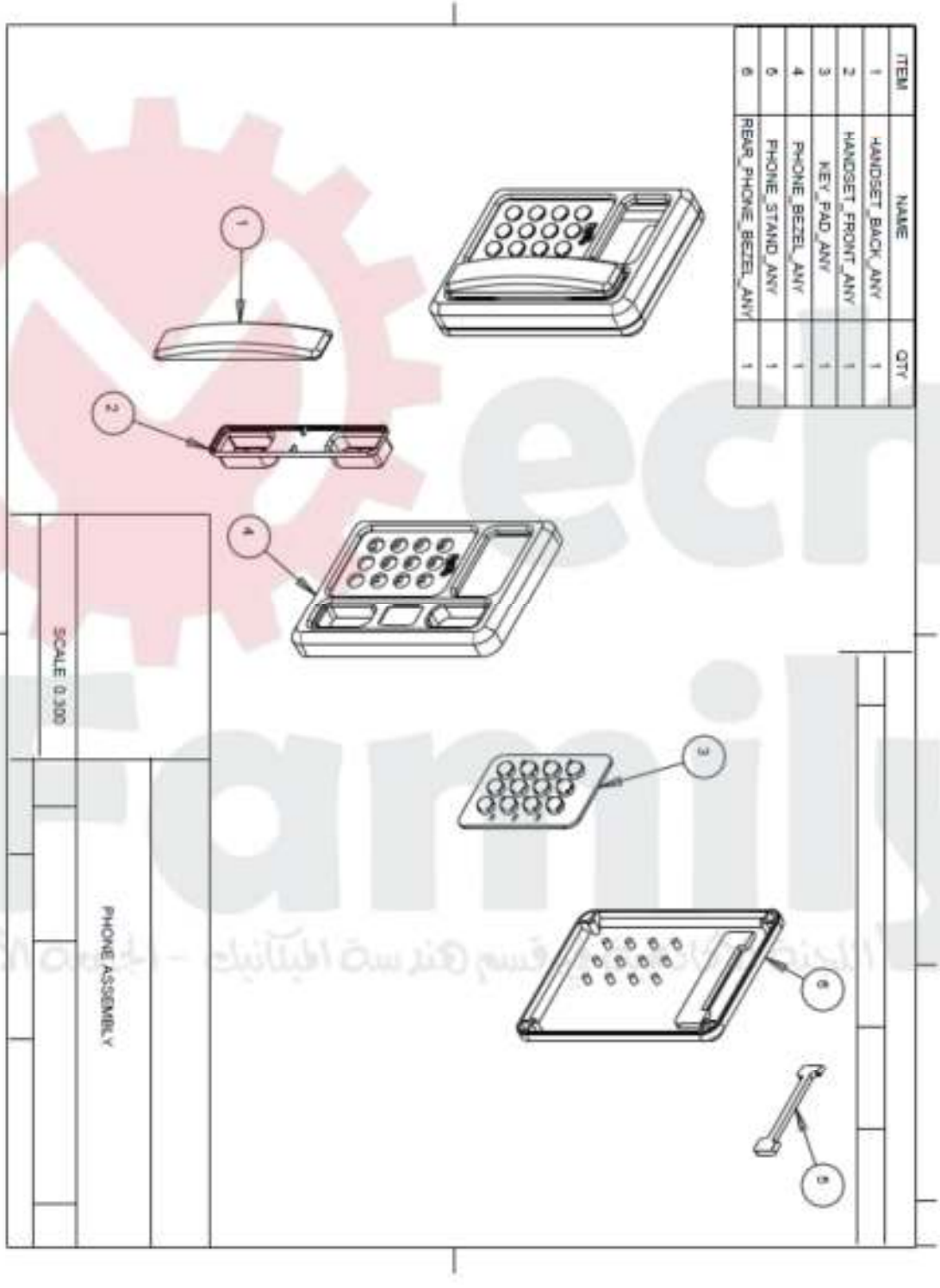


Done

EXERCISE 18

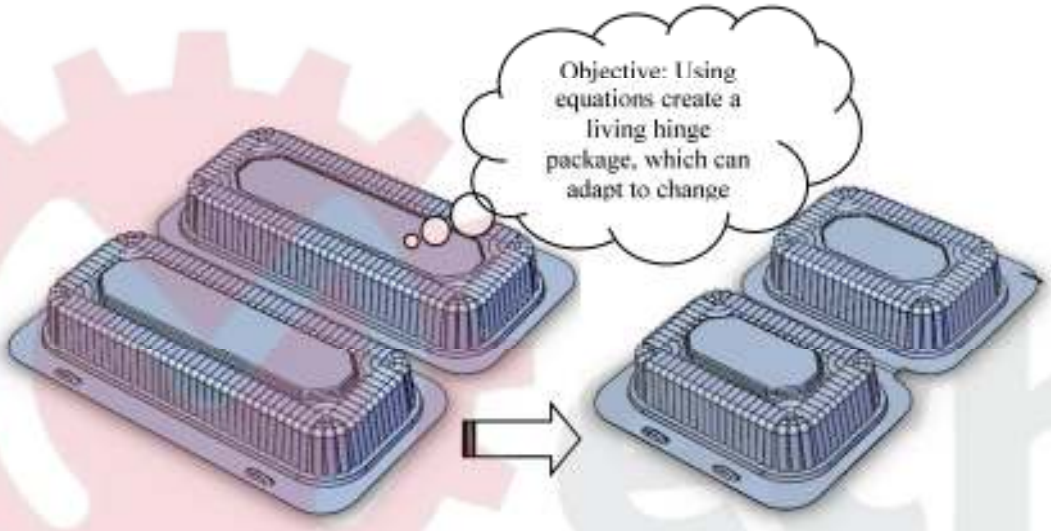
Phone Assembly and Drawing

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



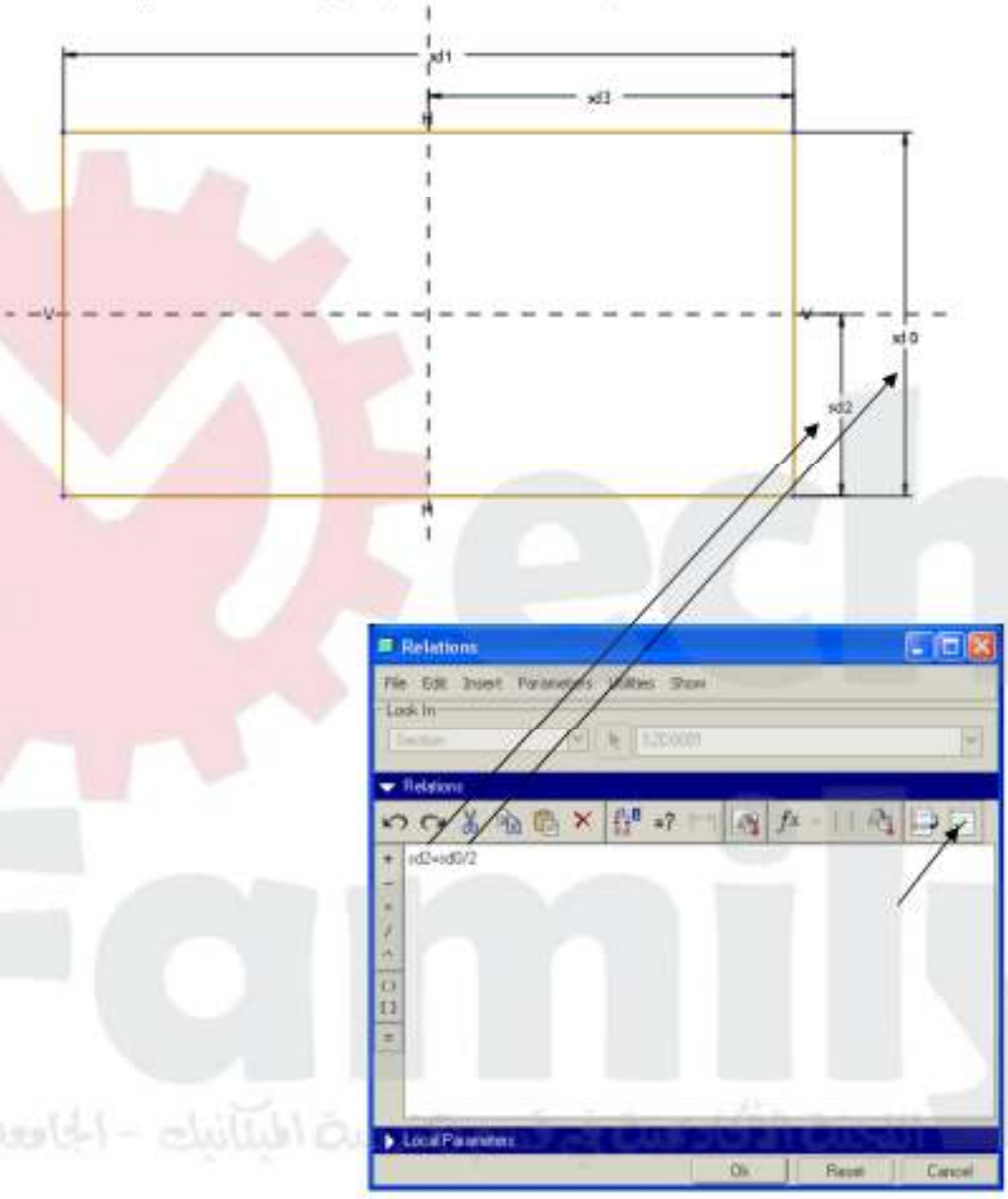
EXERCISE 19
Advanced Equations

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.

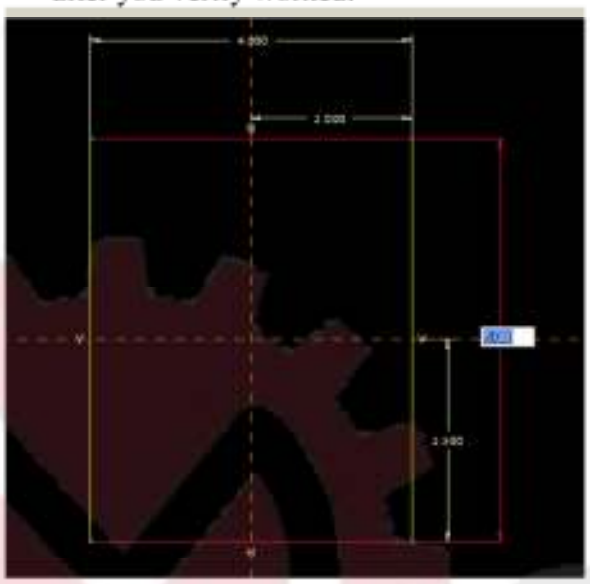


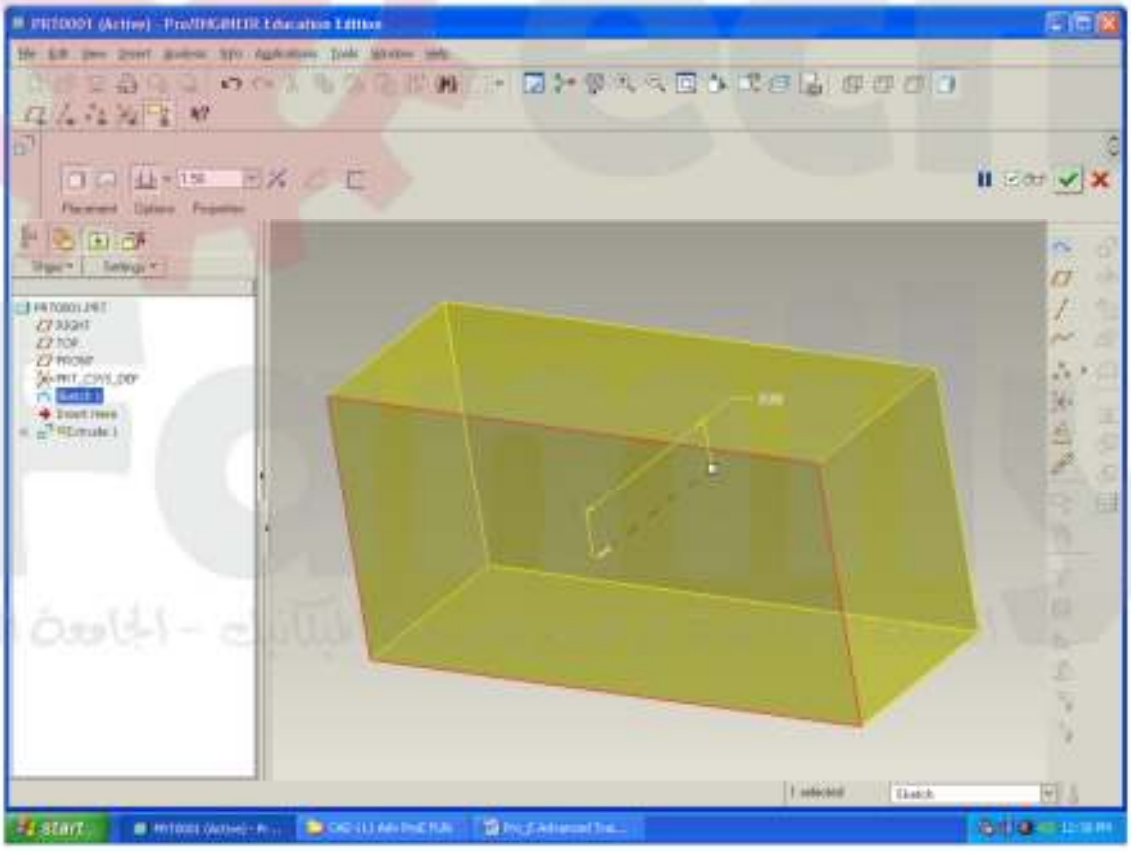
1. Sketch the following on the "Front" plane. Dimension in the order as show.

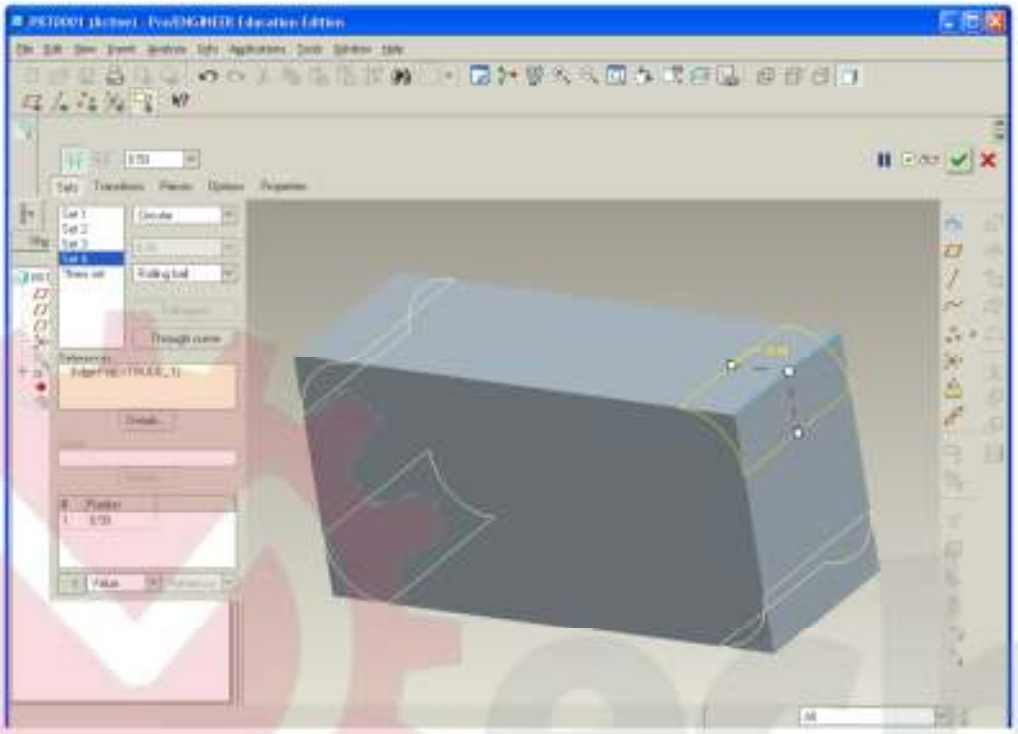
2. **Creating Equations**. Go to *Tools/Relations*. Click once on the desired dimensions to have the names automatically insert into the relations editor. Hit the verify icon on the right (Green check mark).



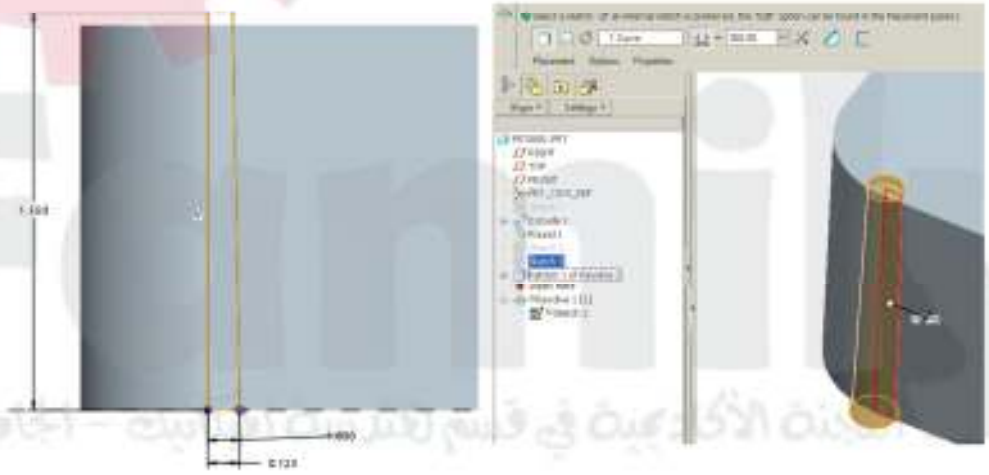
3. You're your equation by changing the 2" dimension to 5". Change it back after you verify worked.



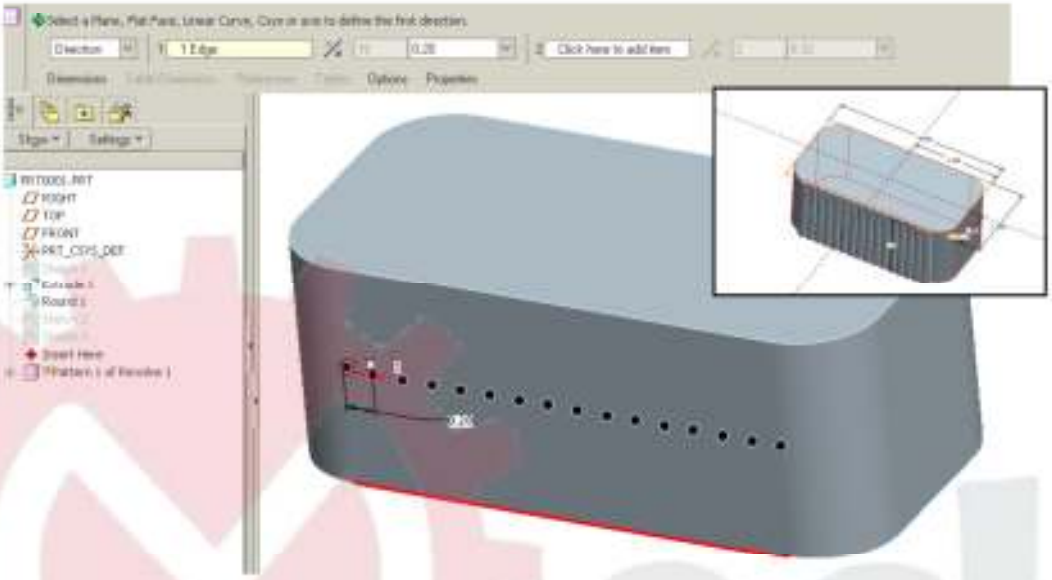




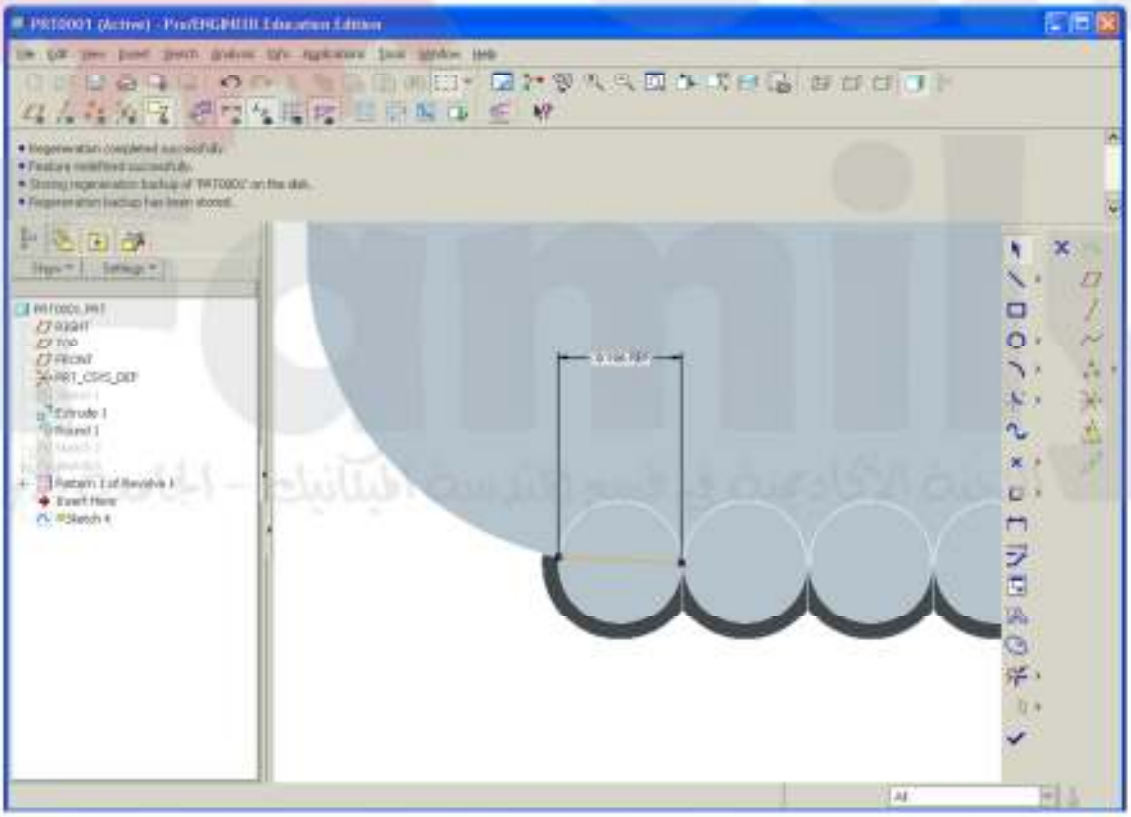
4. Start a sketch on the side face, and draw the following. Note, use Sketch/References to activate the tangent edge vertexes for reference of the center of the revolve feature. Revolve.



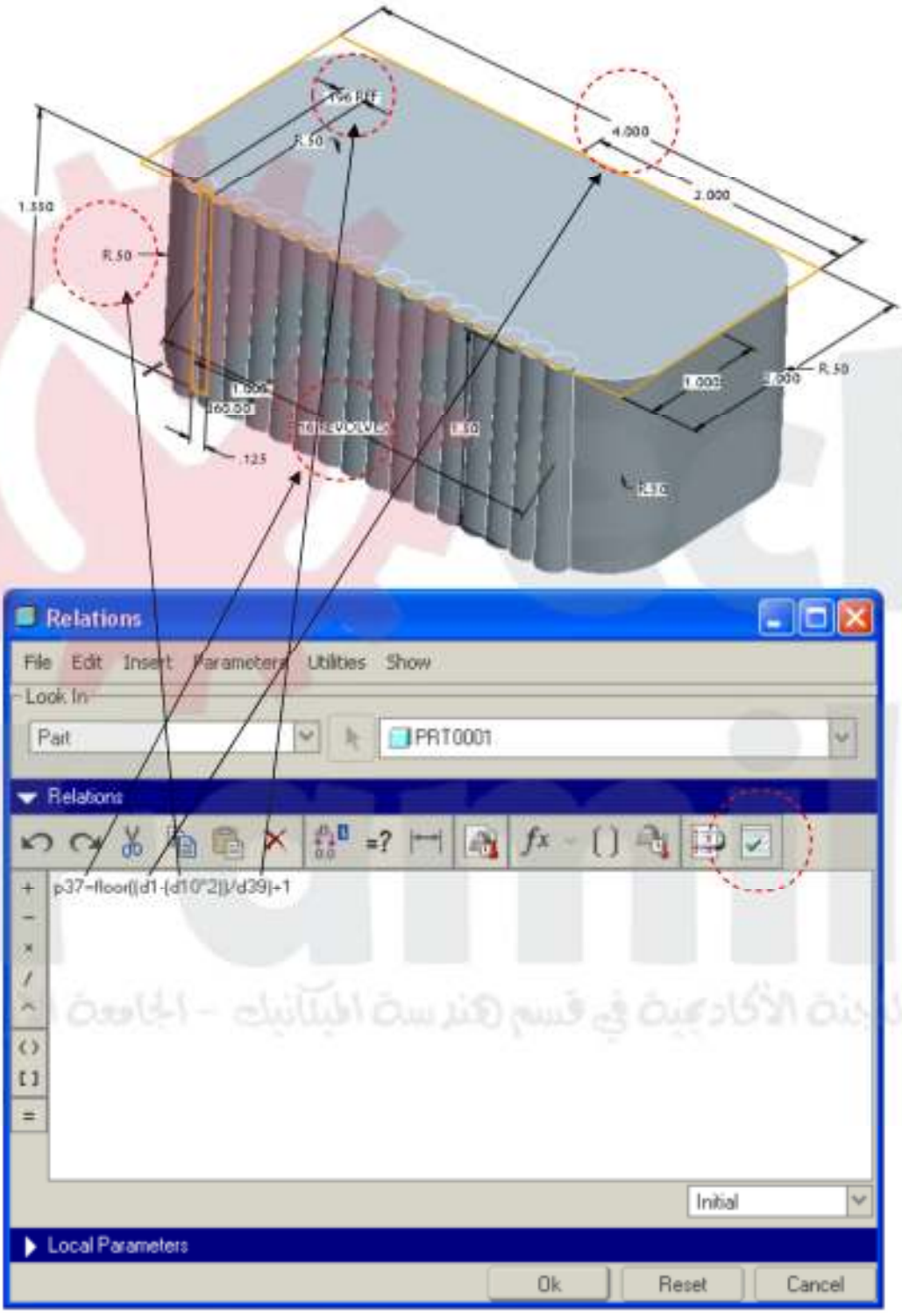
5. Create a linear pattern. 16 instances at .200” spacing.



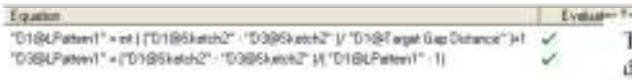
6. Create a new sketch; Draw the following with coincident relations on both sides of the cylinder (Use Sketch/References to select vertexes first). Let it become a driven dimension. Done.



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 Pro Engineer?



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.

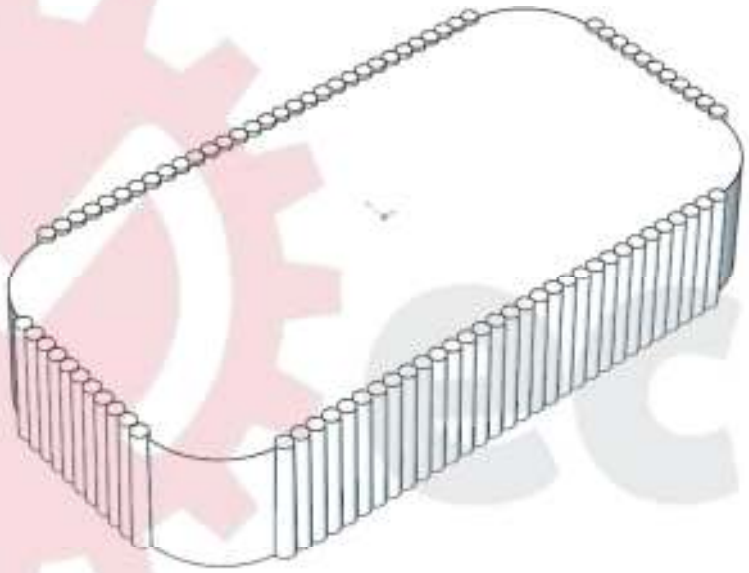
NOTE: Dimensions driven by equations cannot be changed by editing the dimension value in the model.

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...

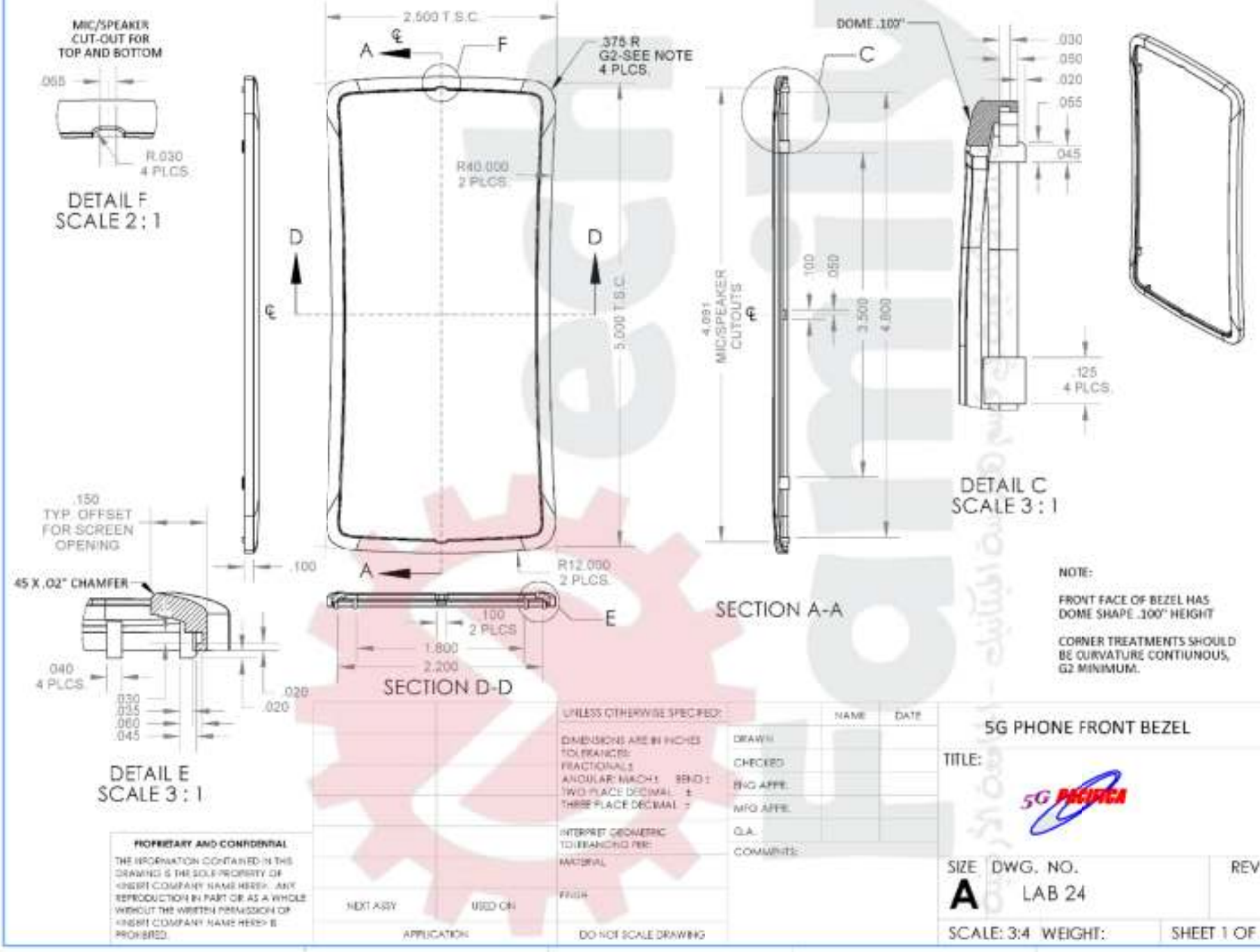
9. Add the same equations on the left side of the model.

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

Tech Family
اللجنة الأكاديمية في قسم هندسة الميكاترونيك - الجامعة الأردنية


MIC/SPEAKER CUT-OUT FOR TOP AND BOTTOM
.055
R.030 4 PLCS
DETAIL F SCALE 2 : 1
2.500 T.S.C.
A
F
.375 R G2-SEE NOTE 4 PLCS
R40.000 2 PLCS
D
D
5.000 T.S.C.
R12.000 2 PLCS
E
A
.100
.150 TYP. OFFSET FOR SCREEN OPENING
45 X .02" CHAMFER
.040 4 PLCS
.020
.030
.035
.060
.045
DETAIL E SCALE 3 : 1
.100 2 PLCS
1.800
2.200
SECTION D-D
DOME .100"
C
.030
.050
.020
.055
.045
.125 4 PLCS
4.091 MIC/SPEAKER CUTOUTS
.100
.050
3.500
4.800
SECTION A-A
DETAIL C SCALE 3 : 1
NOTE:
FRONT FACE OF BEZEL HAS DOME SHAPE .100" HEIGHT
CORNER TREATMENTS SHOULD BE CURVATURE CONTIUNOUS, G2 MINIMUM.
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN INCHES
TOLERANCES:
FRACTIONAL ±
ANGULAR: MACH ± BEND ±
TWO PLACE DECIMAL ±
THREE PLACE DECIMAL ±
INTERPRET GEOMETRIC TOLERANCING PER:
MATERIAL
FINISH
NEXT ASSY
USED ON
APPLICATION
DO NOT SCALE DRAWING
NAME
DATE
DRAWN
CHECKED
ENG APPR.
MFG APPR.
Q.A.
COMMENTS:
PROPRIETARY AND CONFIDENTIAL
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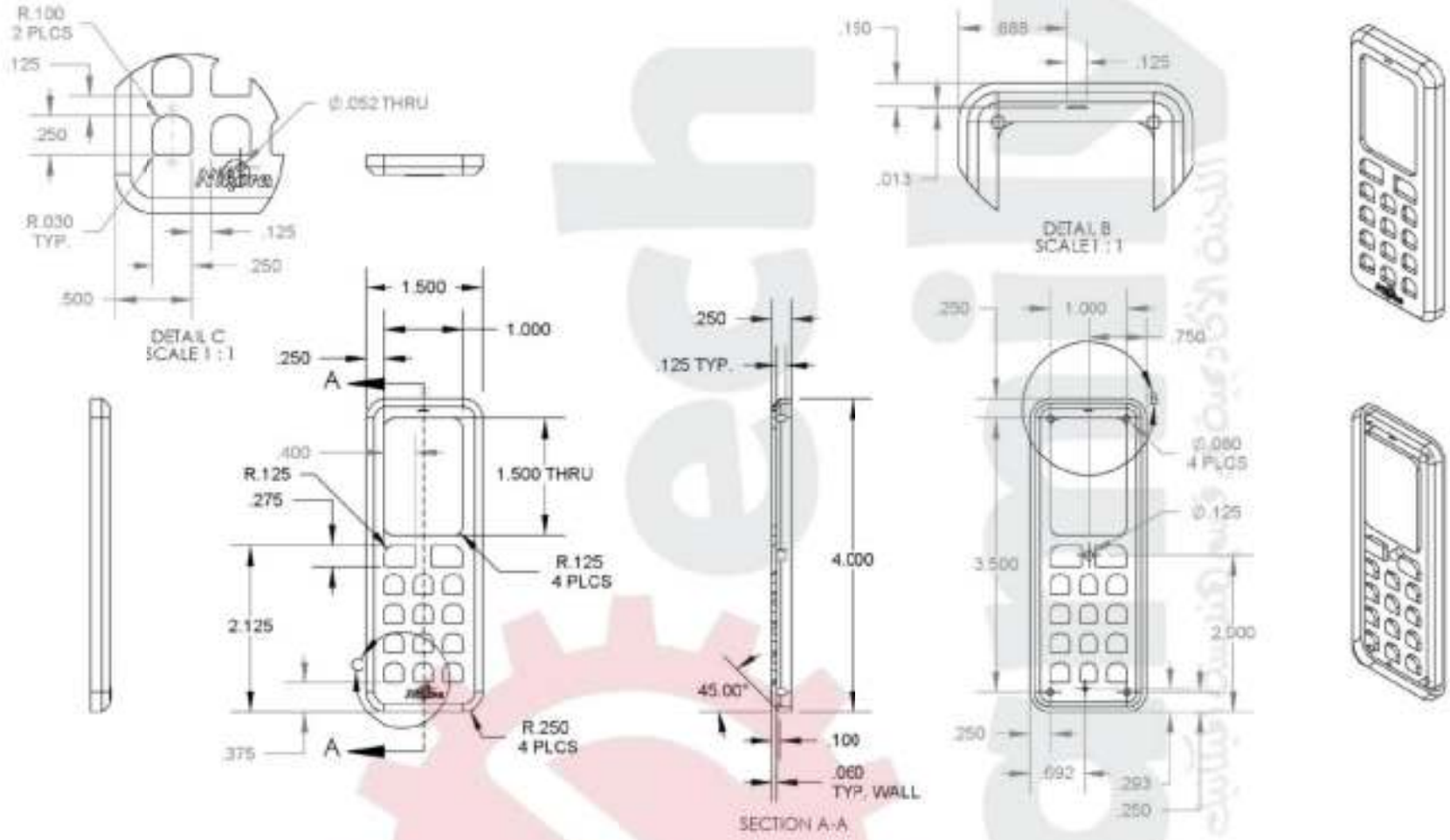
5G PHONE FRONT BEZEL
TITLE:
5G PROJECT
SIZE A
DWG. NO. LAB 24
REV
SCALE: 3:4 WEIGHT:
SHEET 1 OF 1
5
4
3
2
1



		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			

PROPRIETARY AND CONFIDENTIAL

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Nikora TECHNOLOGIES

TITLE:

LAB 19
CELLULAR PHONE
FRONT BEZEL
MODEL. L19

SIZE	DWG. NO.	REV
A		

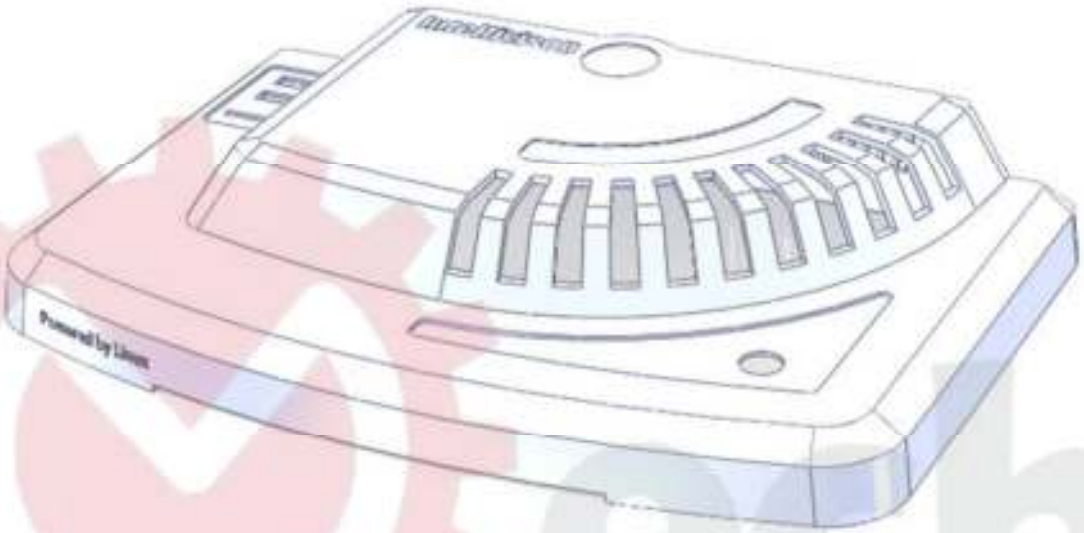
SCALE: 1:2 WEIGHT: SHEET 1 OF 1

5 4 3 2 1

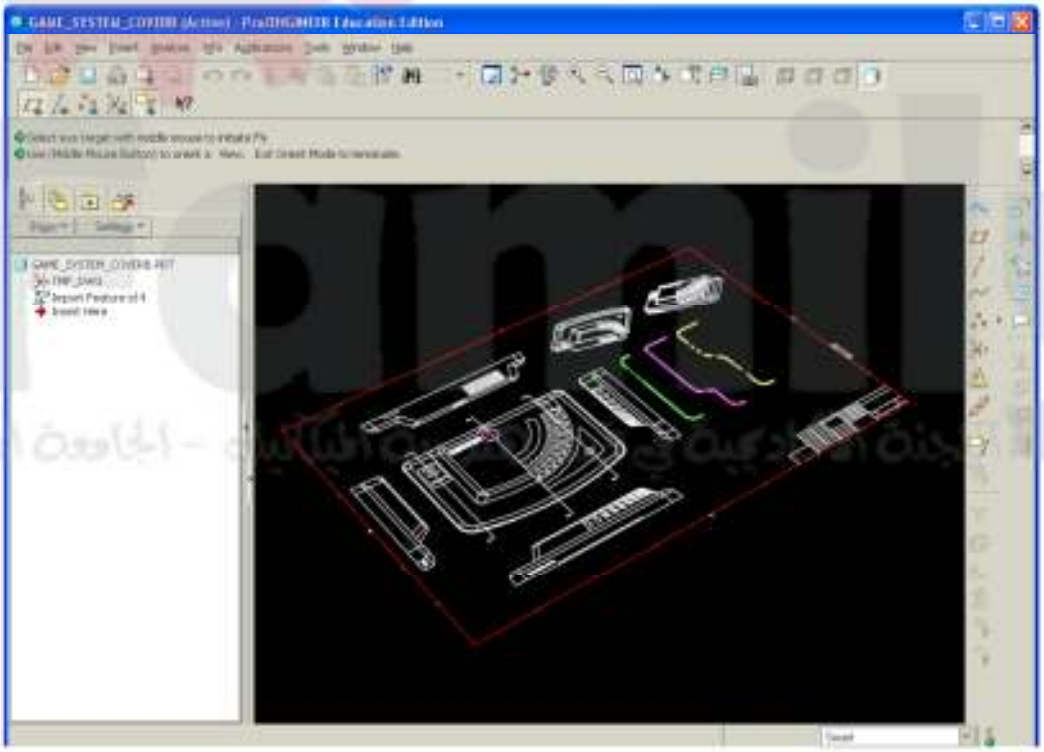
EXERCISE 20

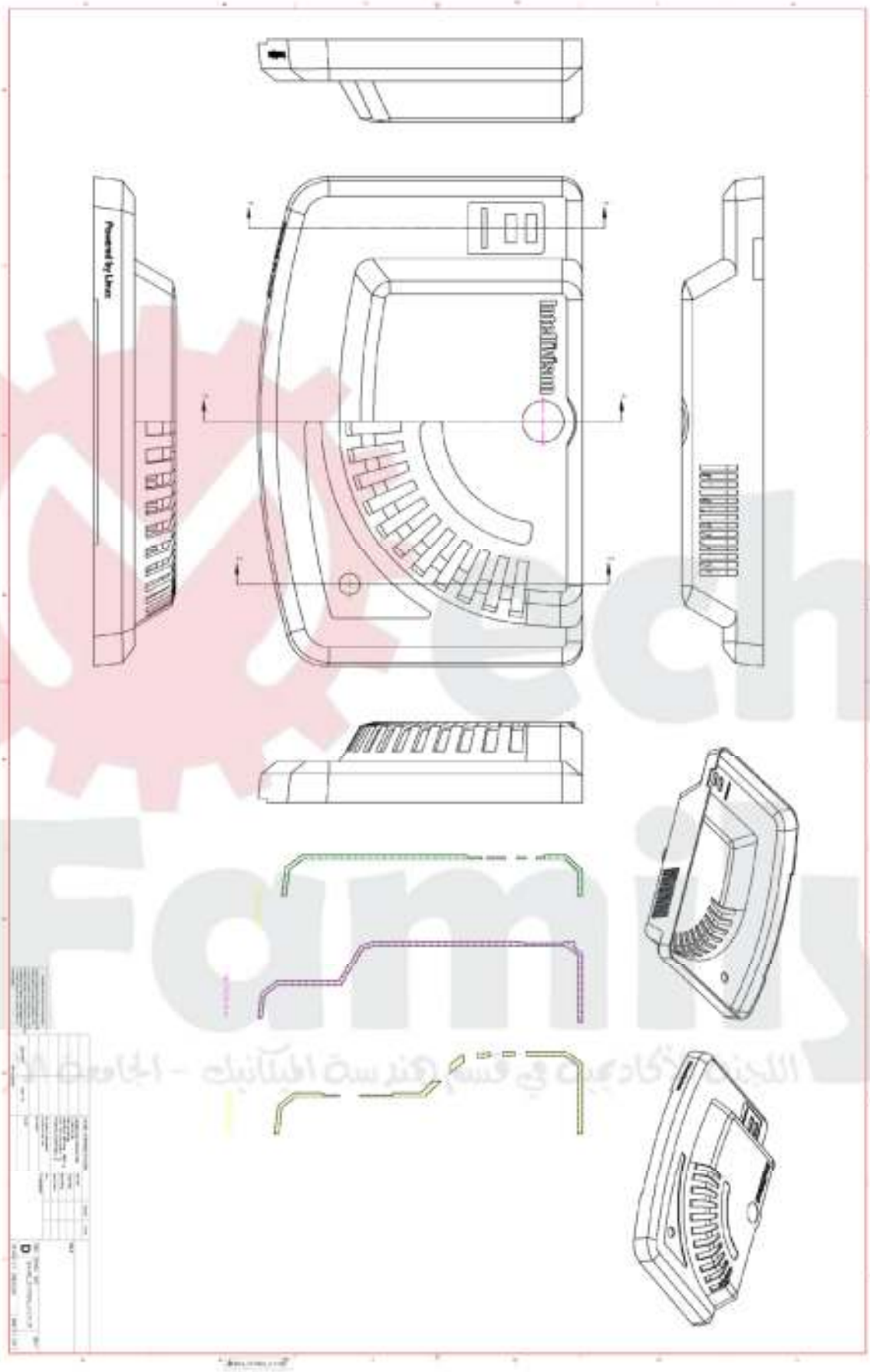
Quiz - Model a Video Game System Cover

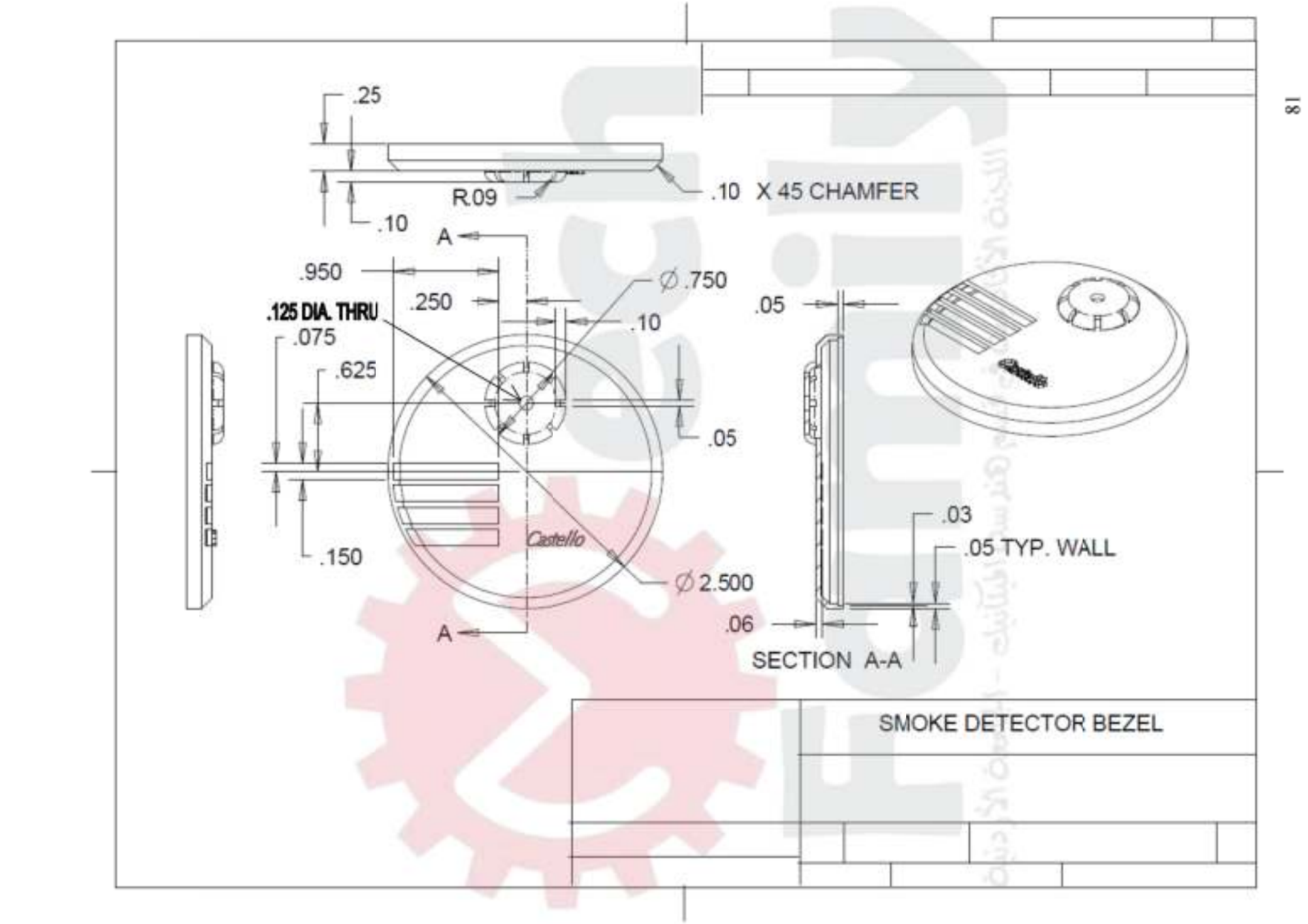
1. Open the GAME_SYSTEM_COVER.dwg

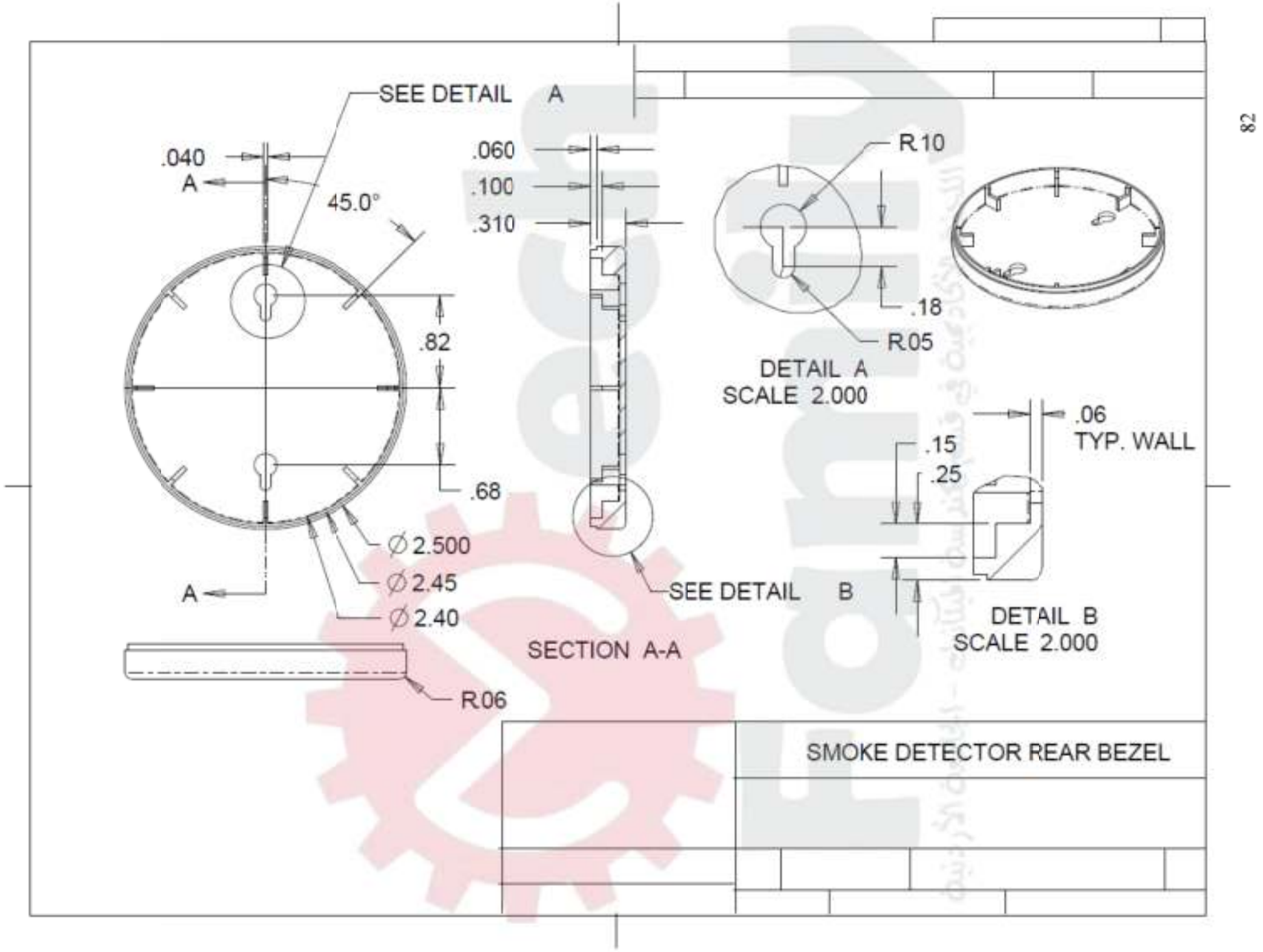


2. Import the DWG into a part file. Construct a model from the provided data.




Powered by Linux
Intel Inside


.25
.10
R.09
.10 X 45 CHAMFER
A
.950
.250
.125 DIA. THRU
.075
.625
.150
Ø.750
.10
.05
Castello
Ø2.500
A
.05
.03
.05 TYP. WALL
.06
SECTION A-A
SMOKE DETECTOR BEZEL

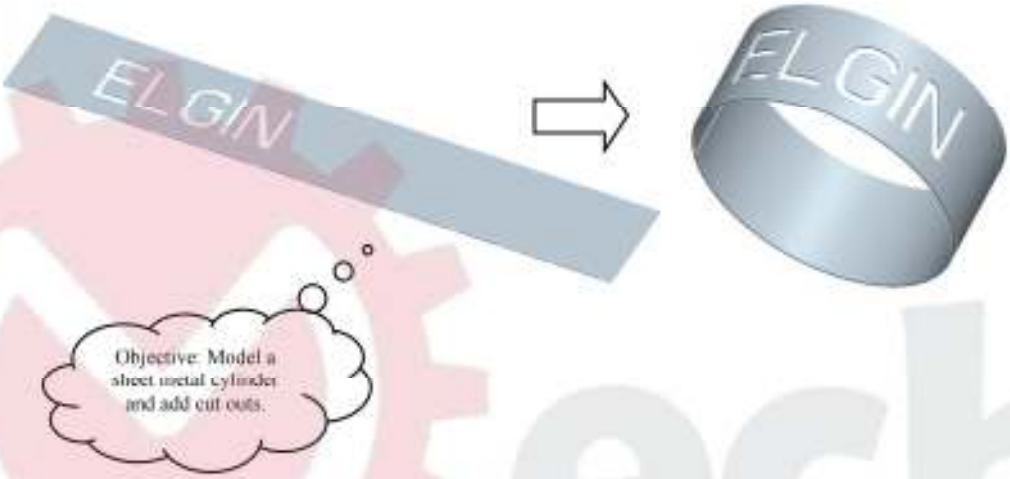

SEE DETAIL A
.040
A
45.0°
.82
.68
A
∅2.500
∅2.45
∅2.40
R.06
.060
.100
.310
SEE DETAIL B
SECTION A-A
R.10
.18
R.05
DETAIL A
SCALE 2.000
.06
TYP. WALL
.15
.25
DETAIL B
SCALE 2.000
SMOKE DETECTOR REAR BEZEL
82

EXERCISE 21

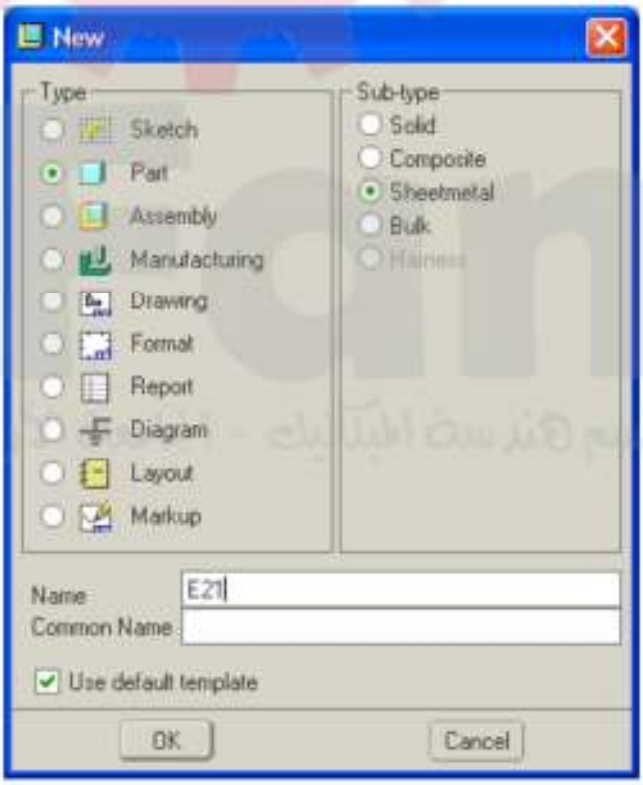
Sheet Metal II

Cylinders

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

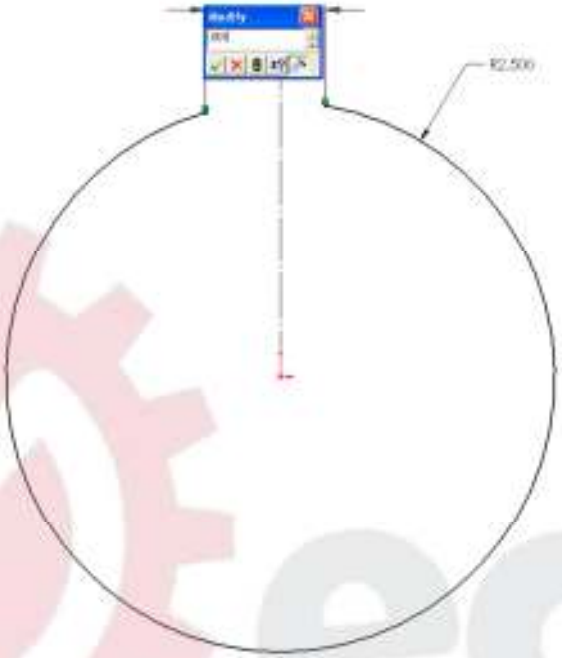


1. Go to file/new and select new part/sheet metal and save as "E21".

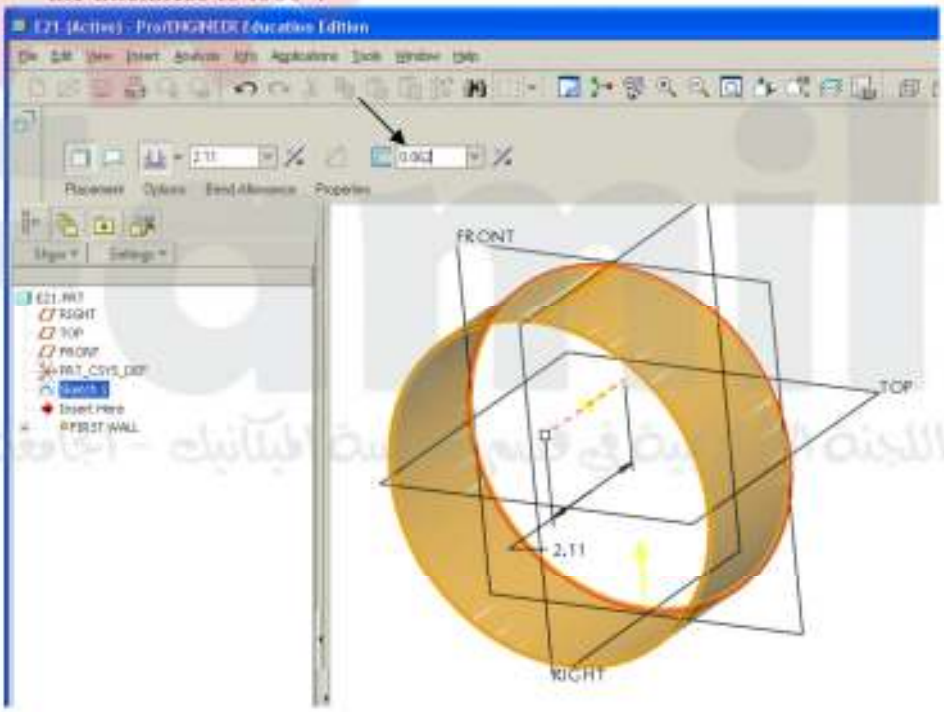


Mech Family
اللجنة الأكاديمية في قسم هندسة الميكانيك - الجامعة الأردنية

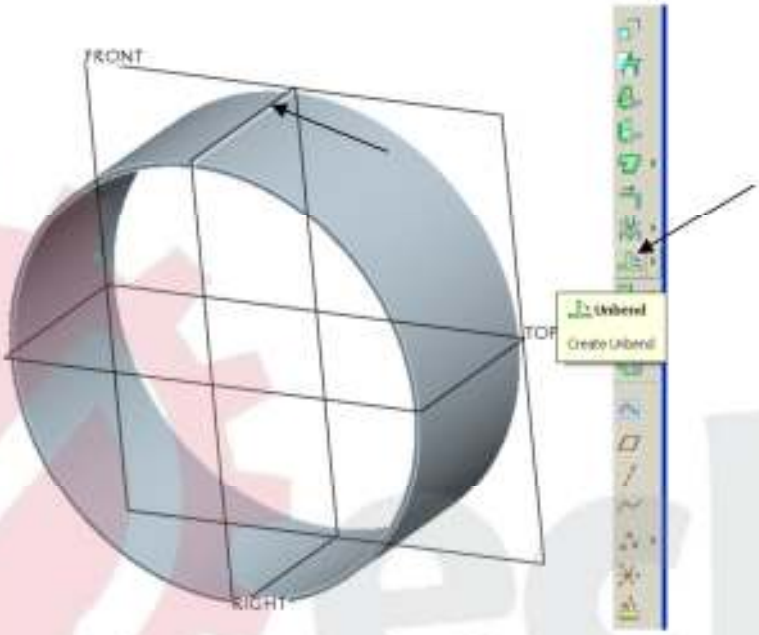
2. Draw the following sketch on the "Front" plane, use the "center point arc" tool. Make both ends of the arc symmetric to a vertical centerline. Space @ .01"



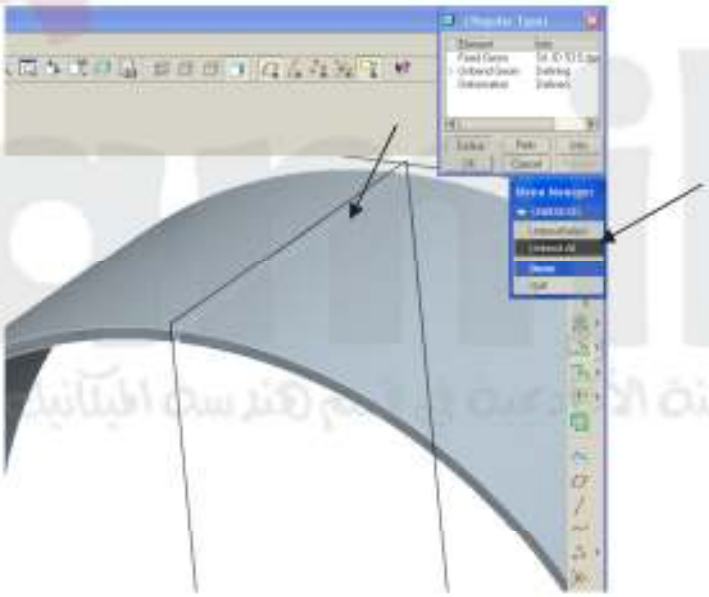
3. Boss Extrude blind 2.5". Notice that it should be creating a thin feature and set the thickness to .060".



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

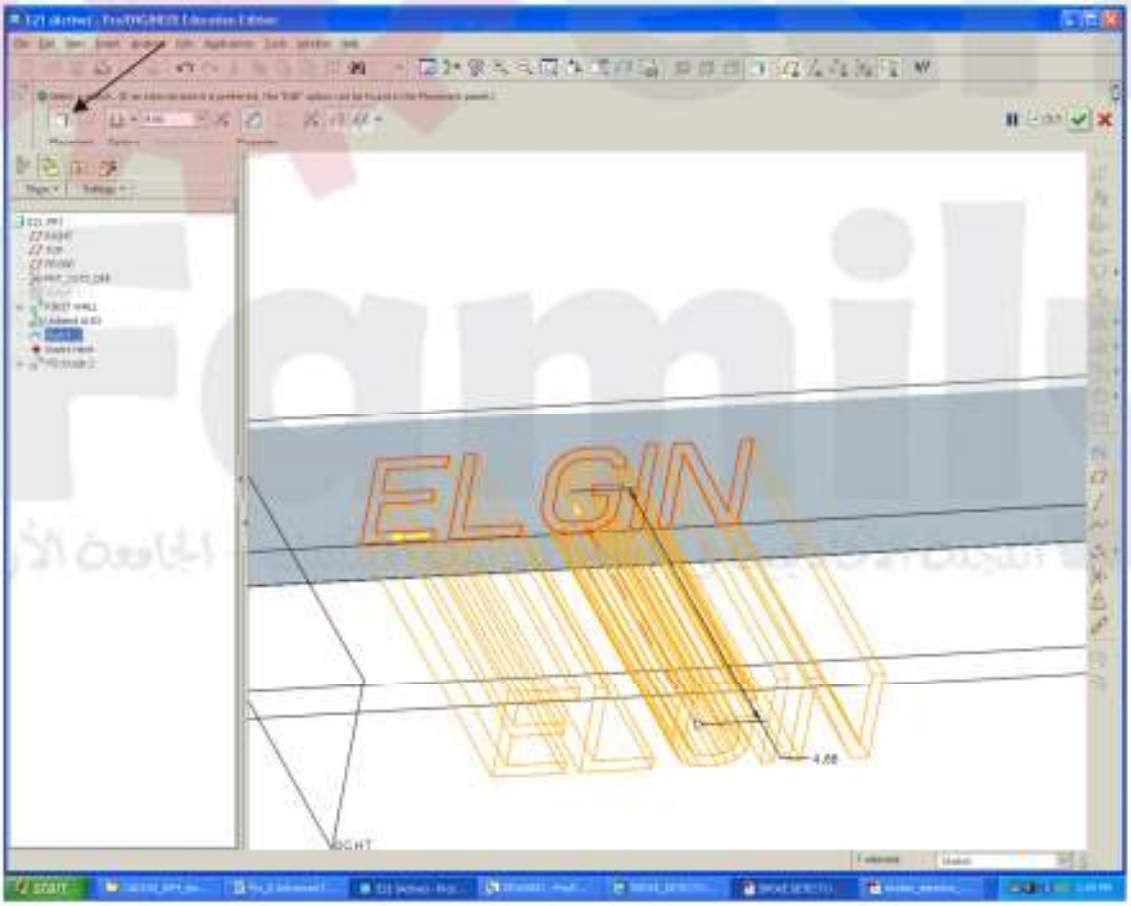


5. Select the “Flatten” icon to verify. Then select the face and start a sketch.

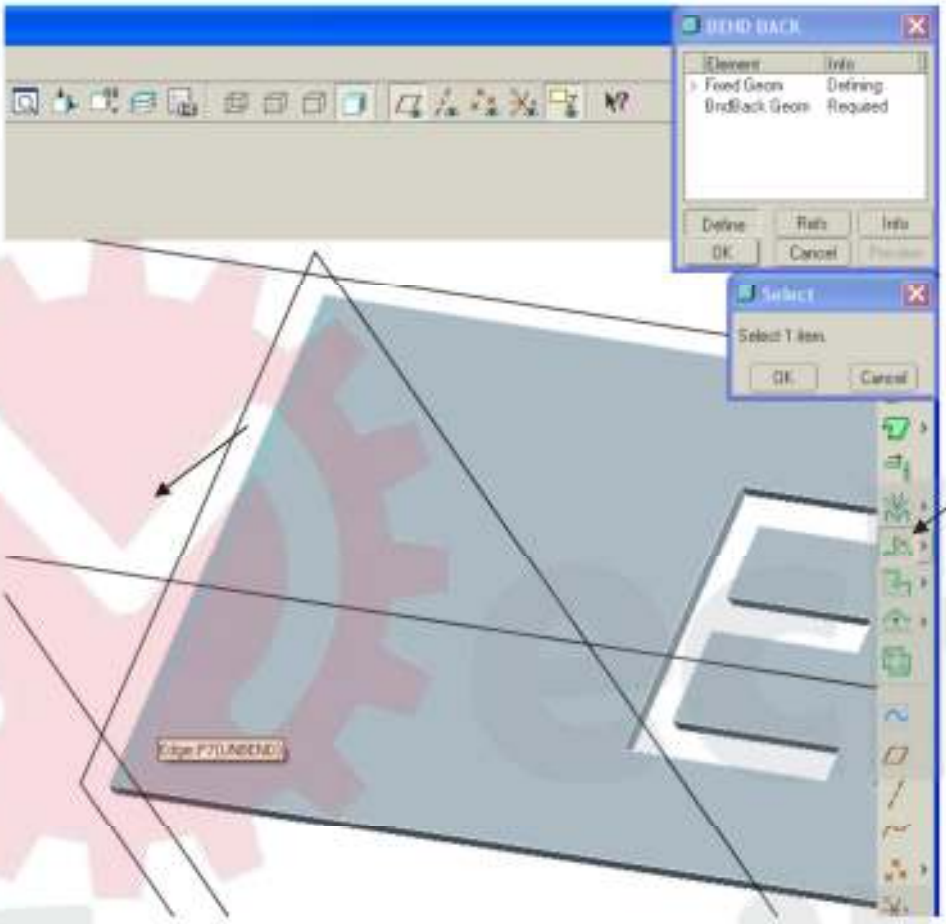


6. Use the Text tool. Write Elgin in the dialog box.

7. Select the "Flatten" icon, to refold it.



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



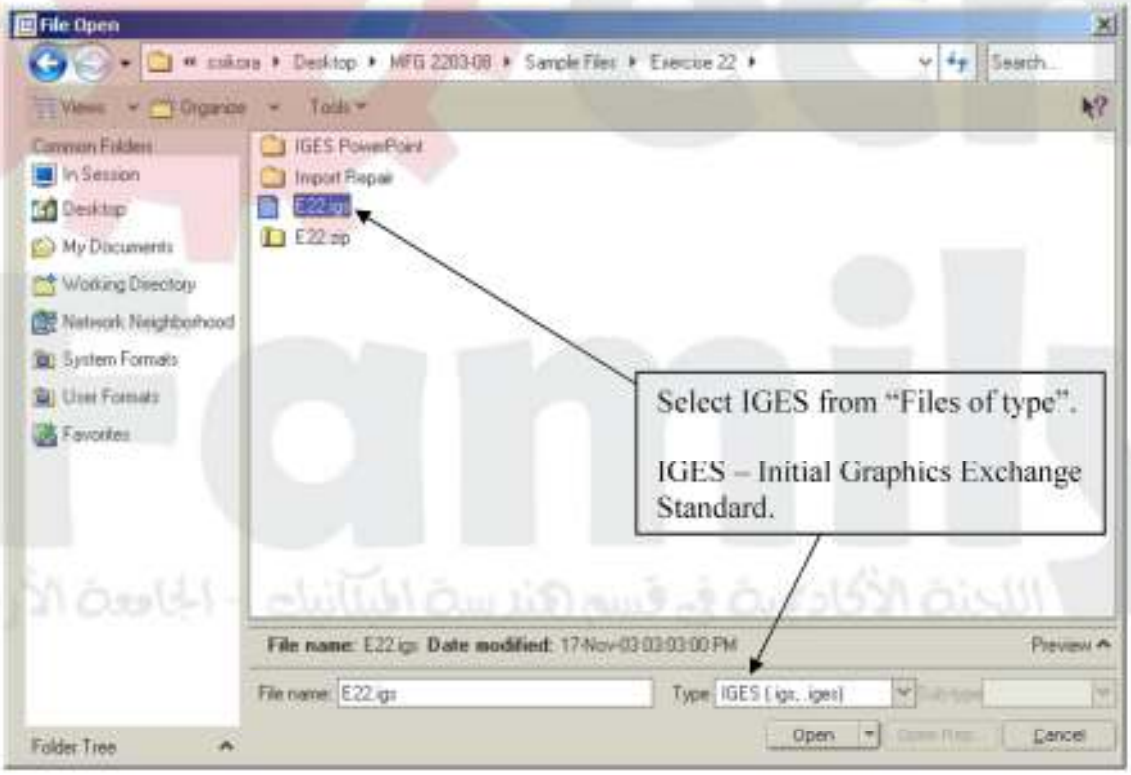
Finished

EXERCISE 22

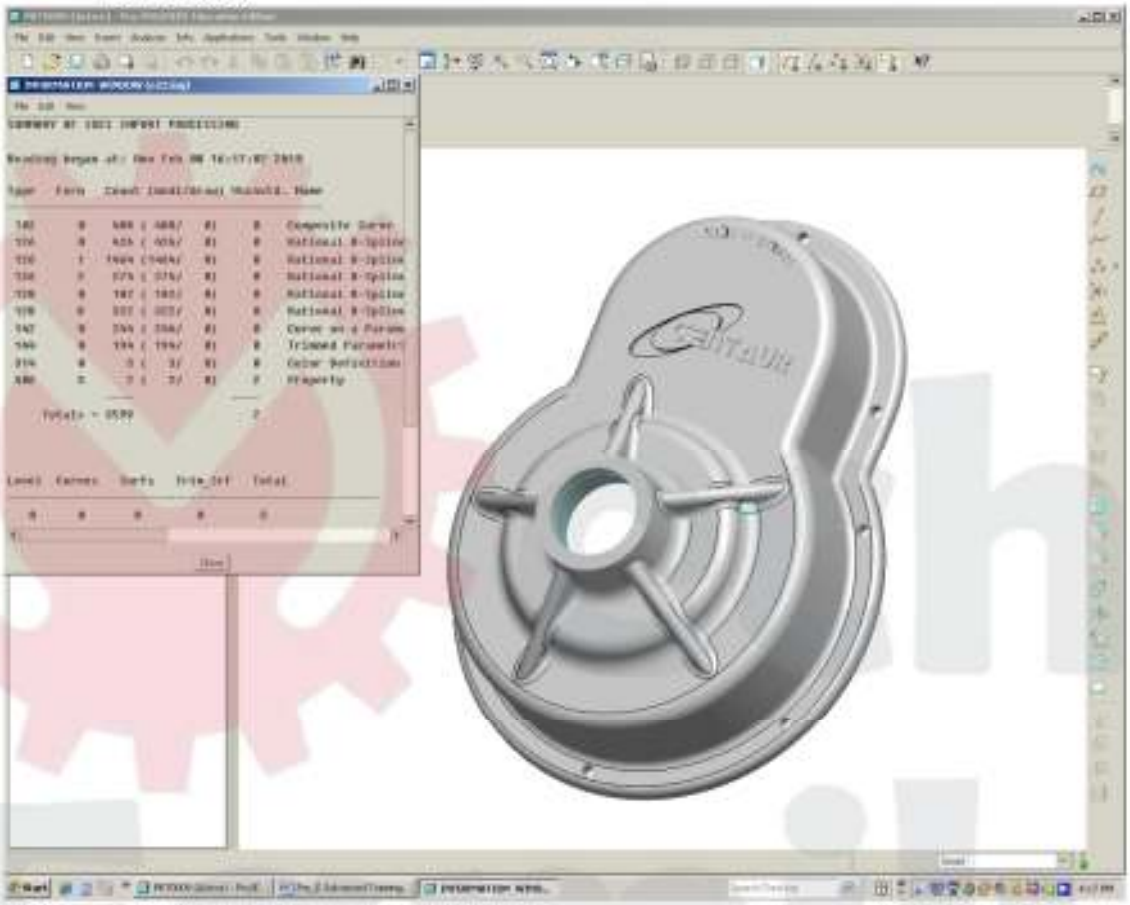
Imported 3D Model Repair

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.

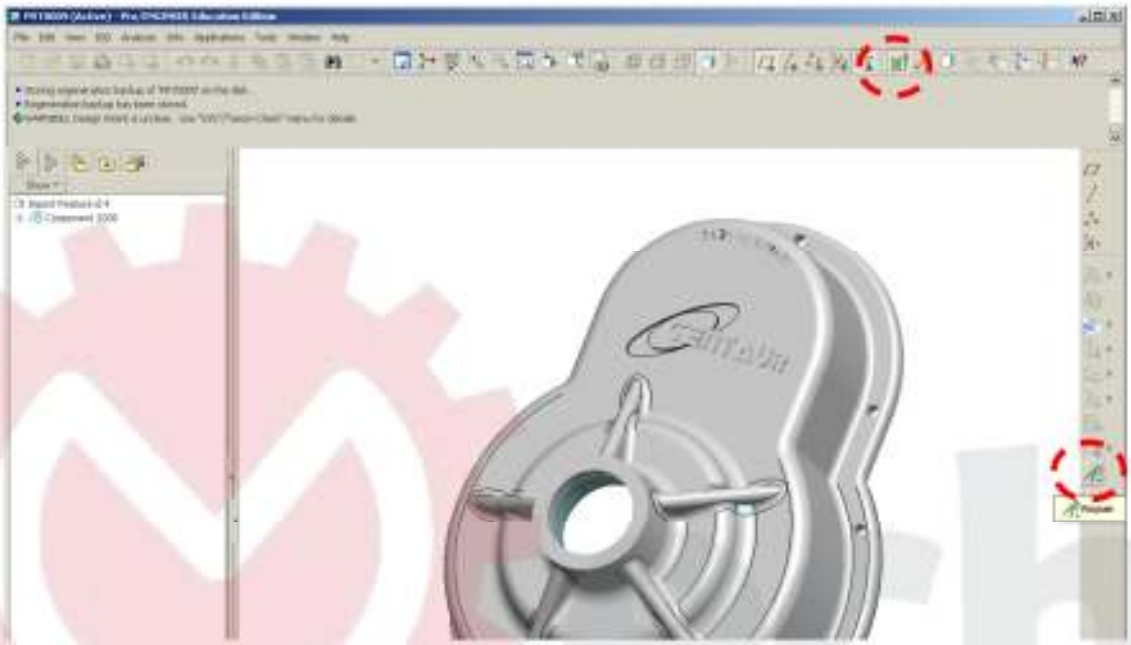


2. Once imported you will receive a message stating ProE was able to read in the file. It is very common that the file may have missing geometry or gaps and may only be a collection of surfaces versus a water tight solid. Essentially in order to make use of this (make a cavity/mold) it is imperative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



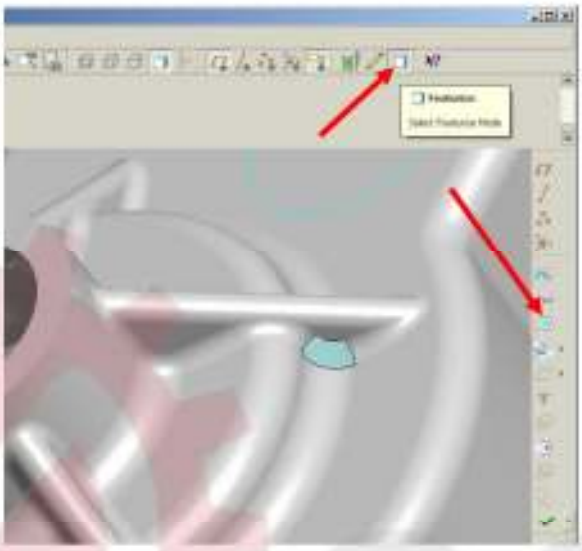
3. RMB click on the "Import Feature" in the feature tree, and Import Data Doctor icon (Red cross box).

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

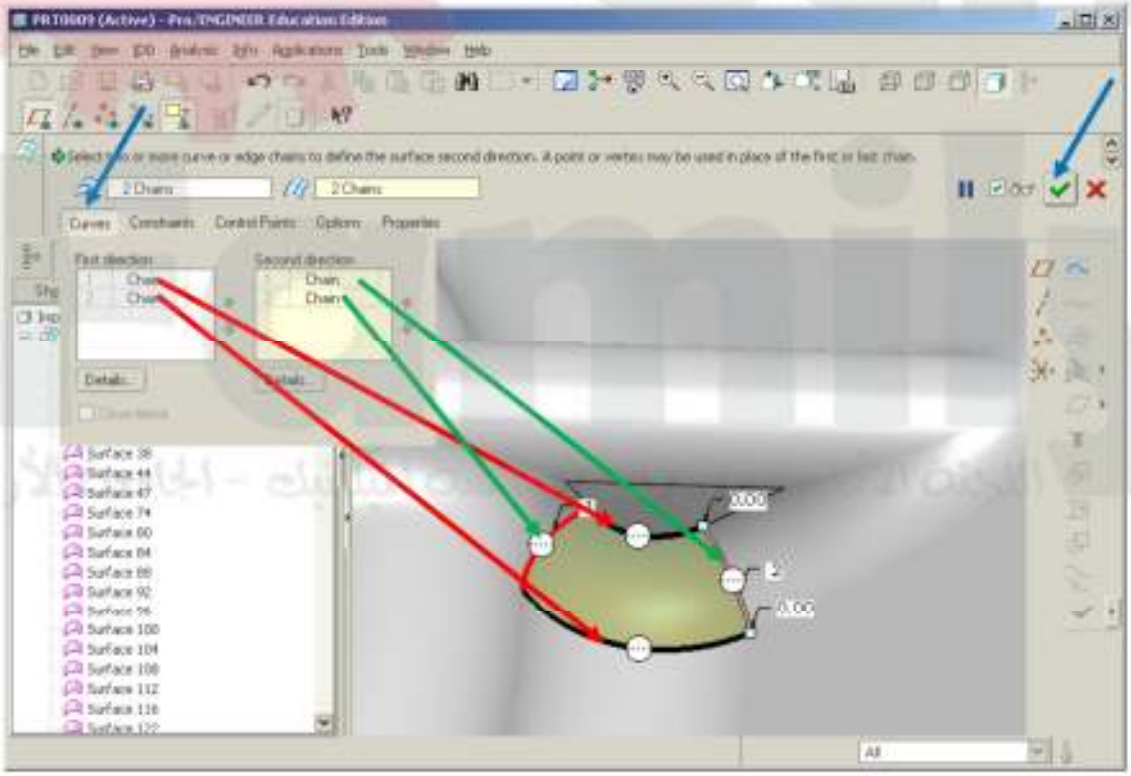


5. Hit the green check mark to apply.

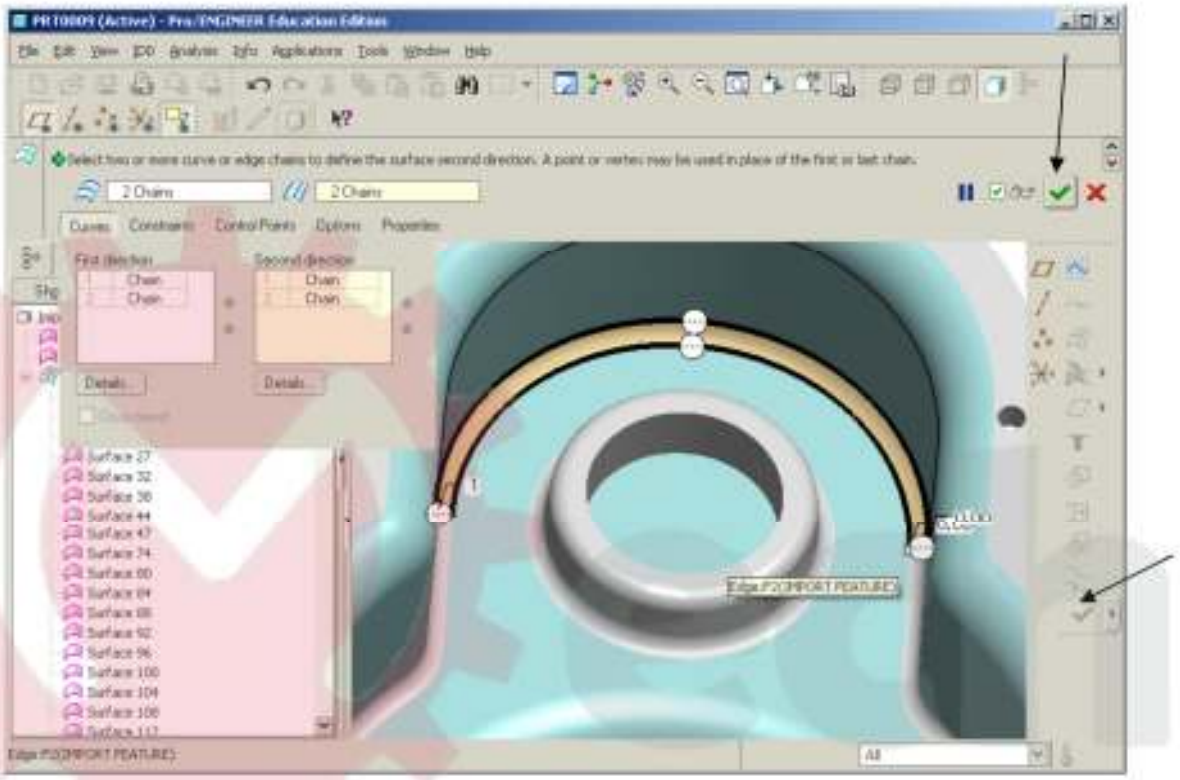
6. Now we must manually address the problem by creating surfaces to close the gaps. Select the Featurize icon, then use Boundary Blend Surface.



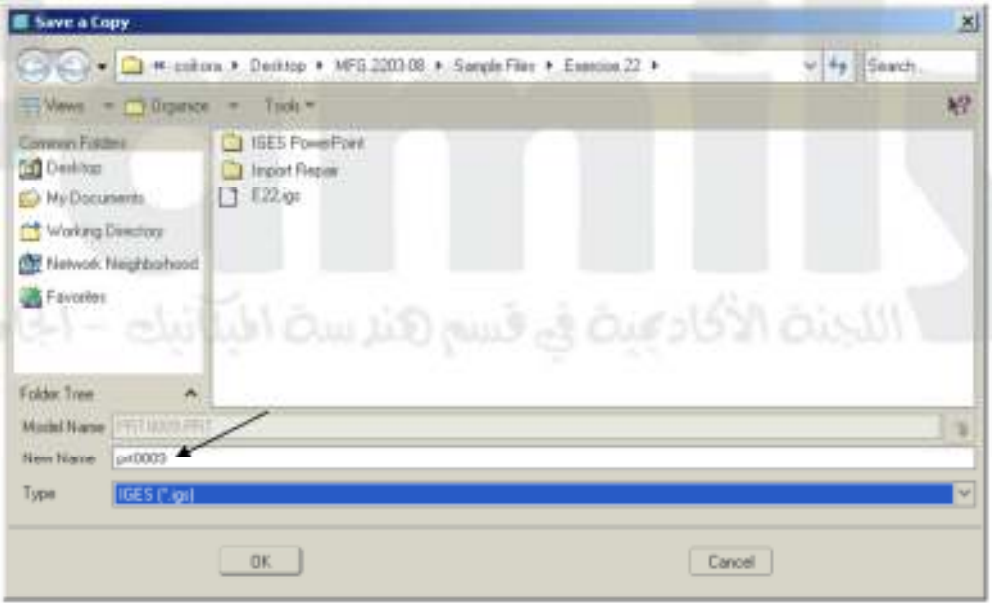
7. Select the "Curves" tab, then select the First Direction box, and then select the two red arrow edges. Then select the Second Direction box, and the two green arrow edges. Hit the Done icon.



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

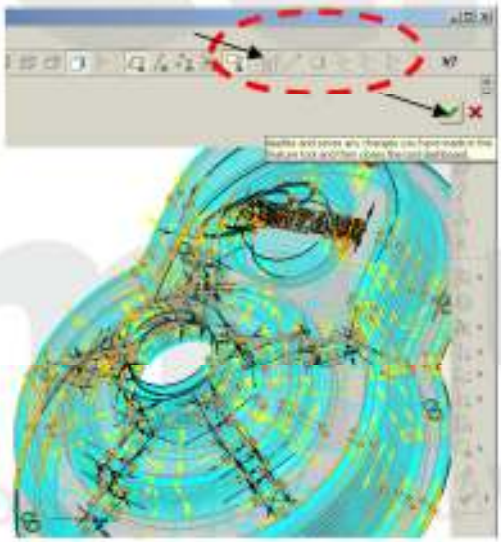


9. Save a copy and select IGES. Save as E22b.

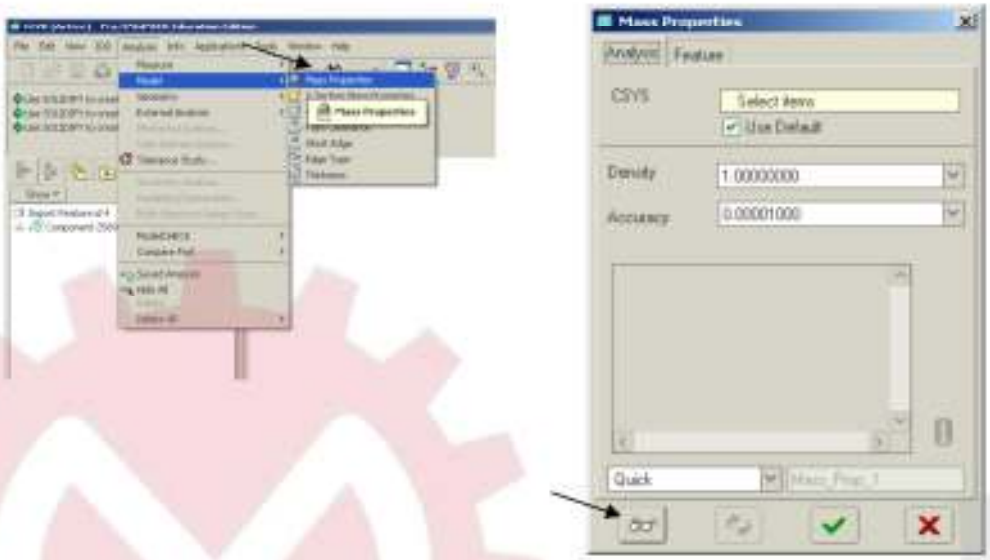


10. Select Solids and Shells.

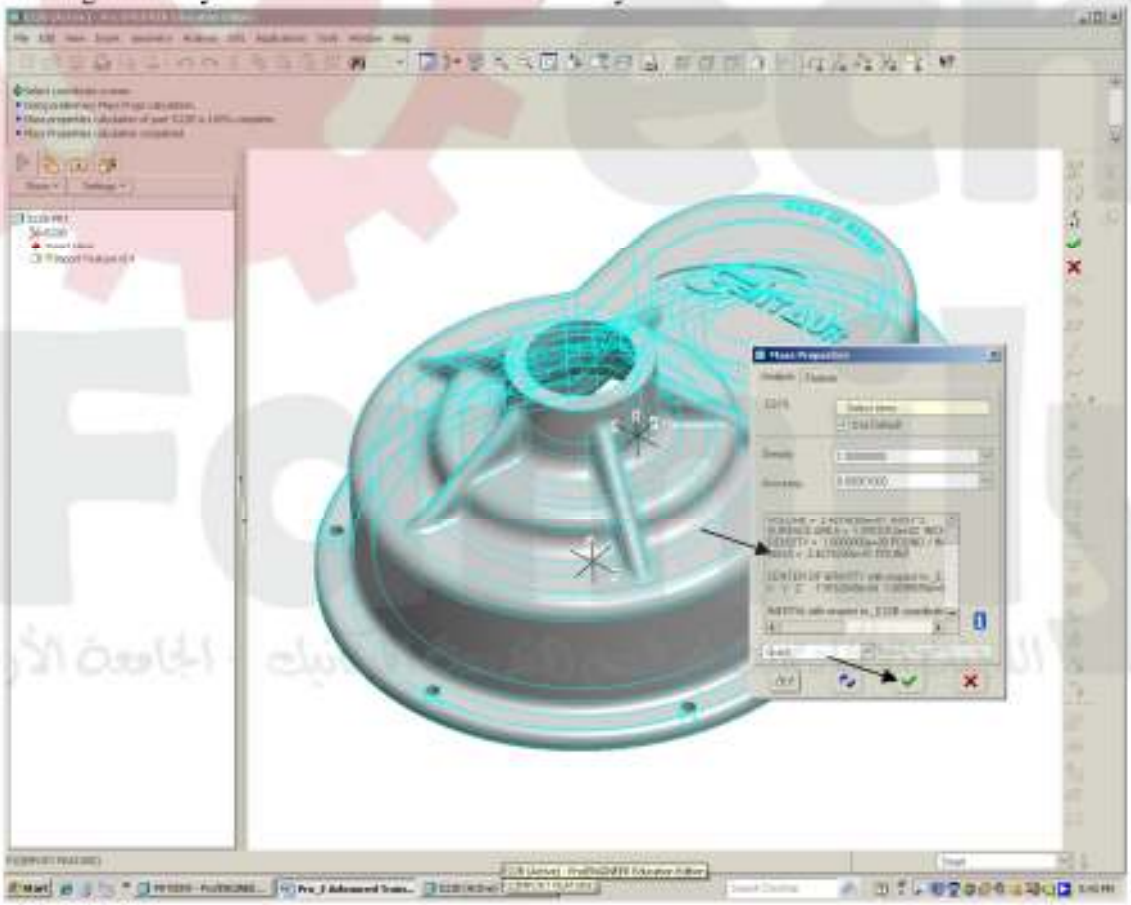
11. Re-import the new E22b IGES file. Step through the repair tools again.



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

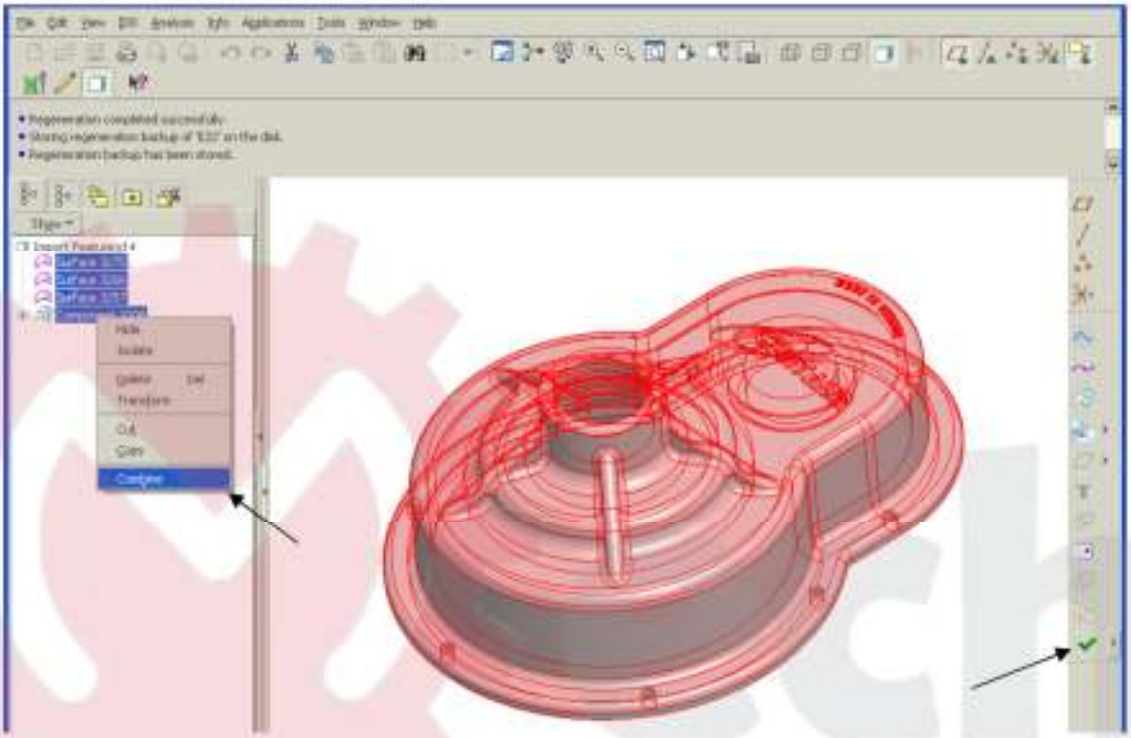


If it registers any other numbers than zeros, then you have a solid.

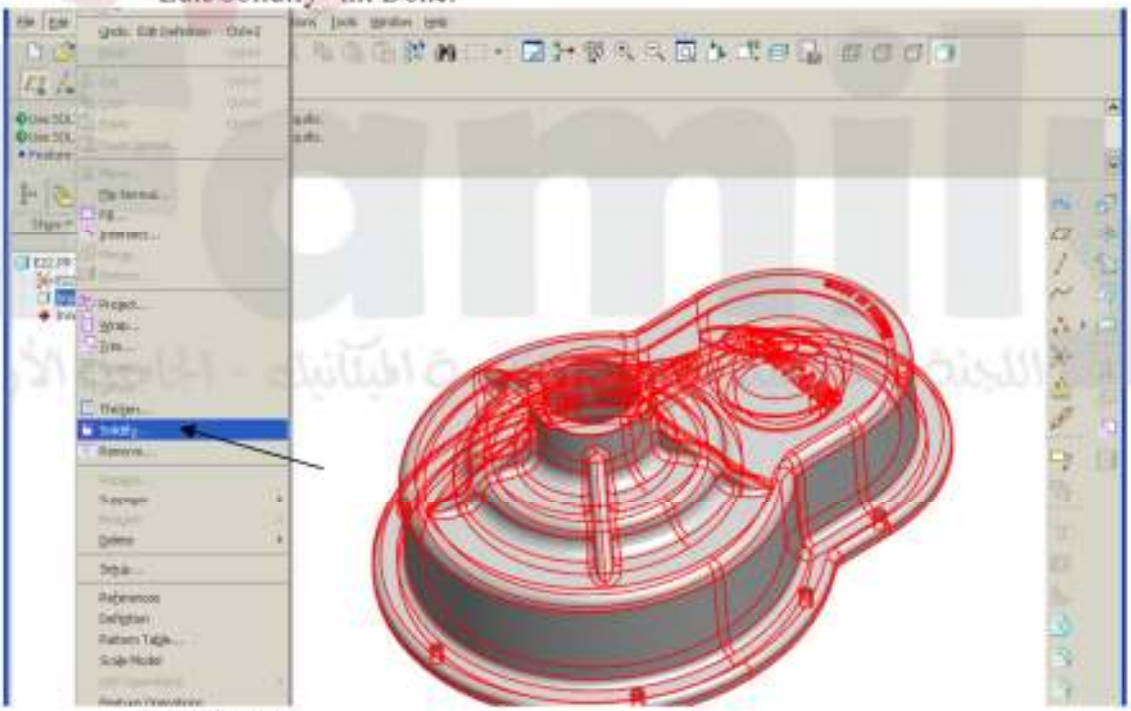


FINISHED

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.



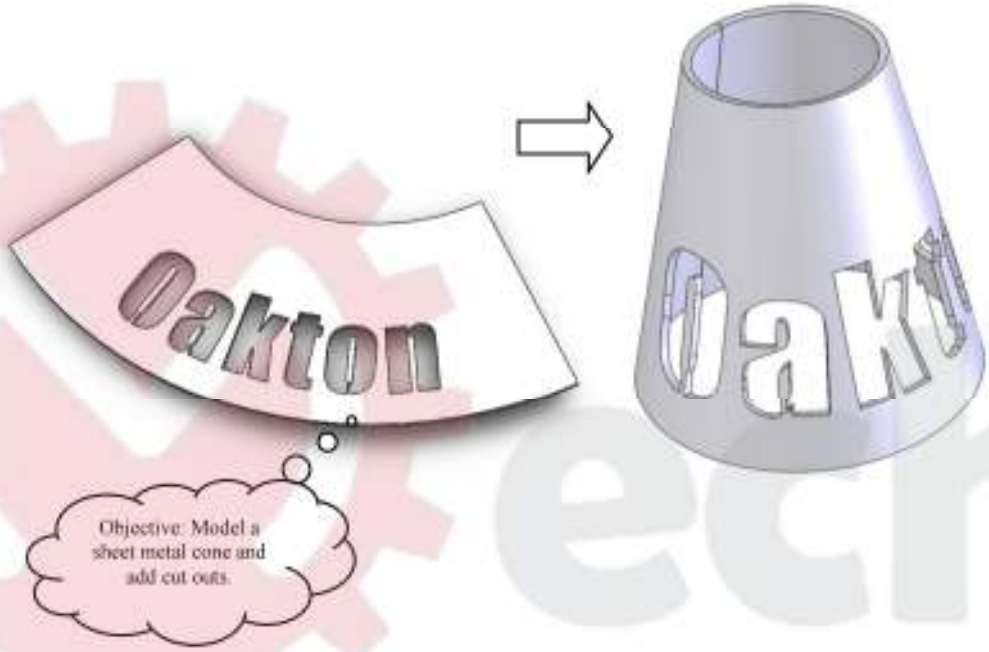
FINISHED

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.

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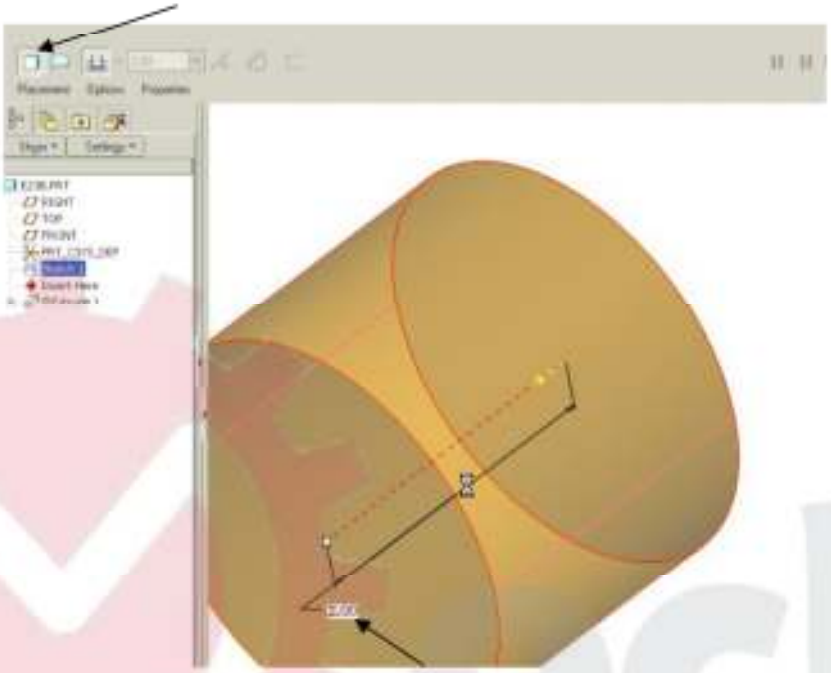
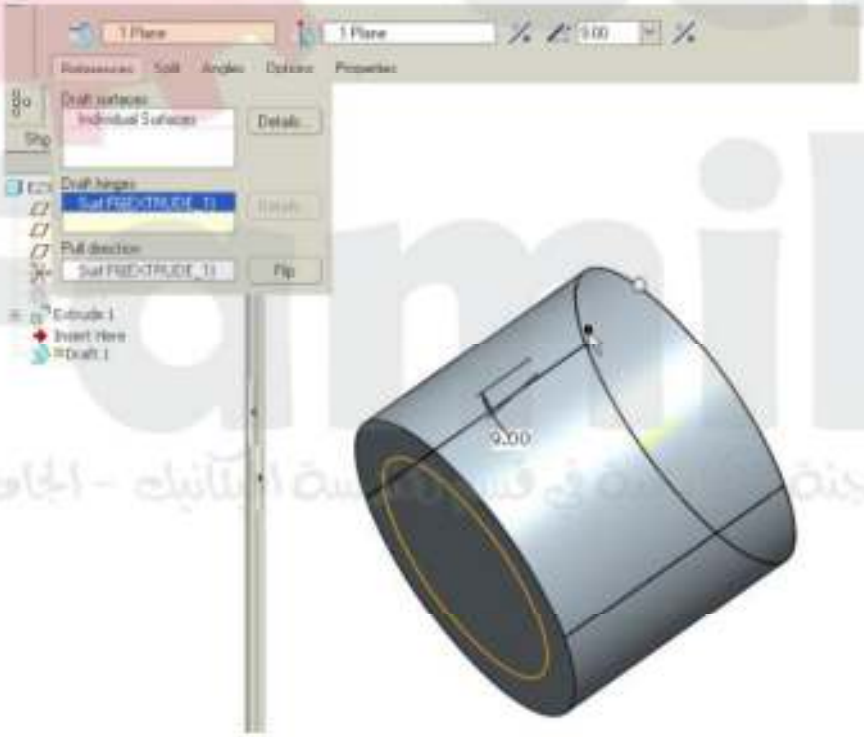
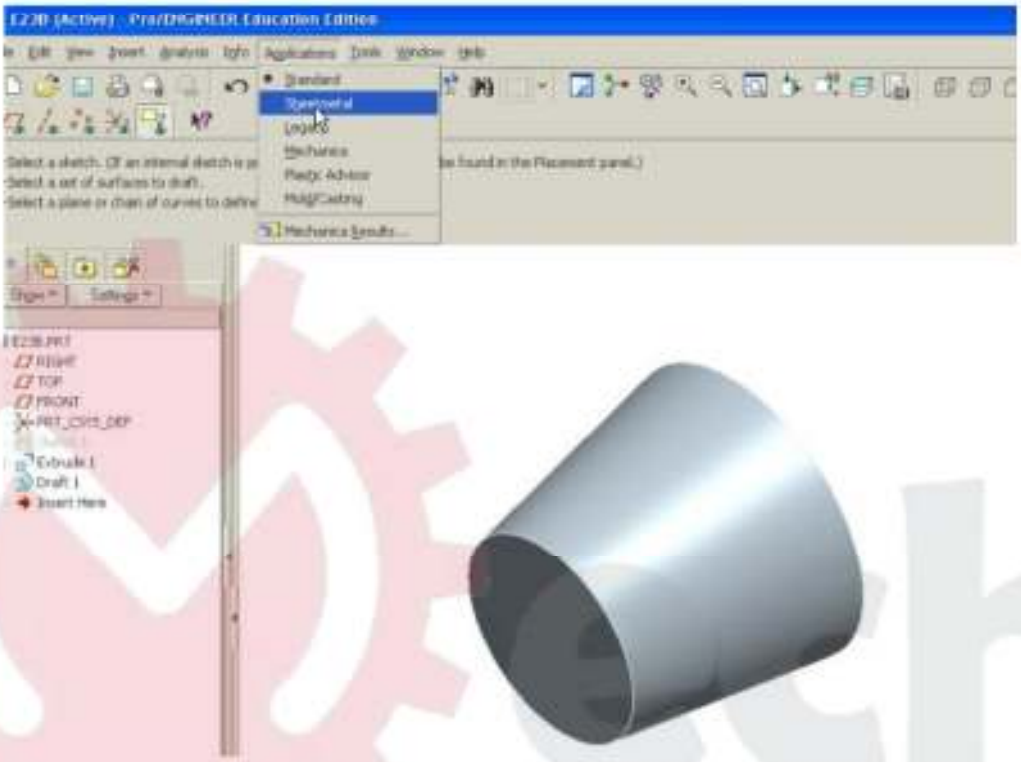
3. Boss Extrude blind 2.5”



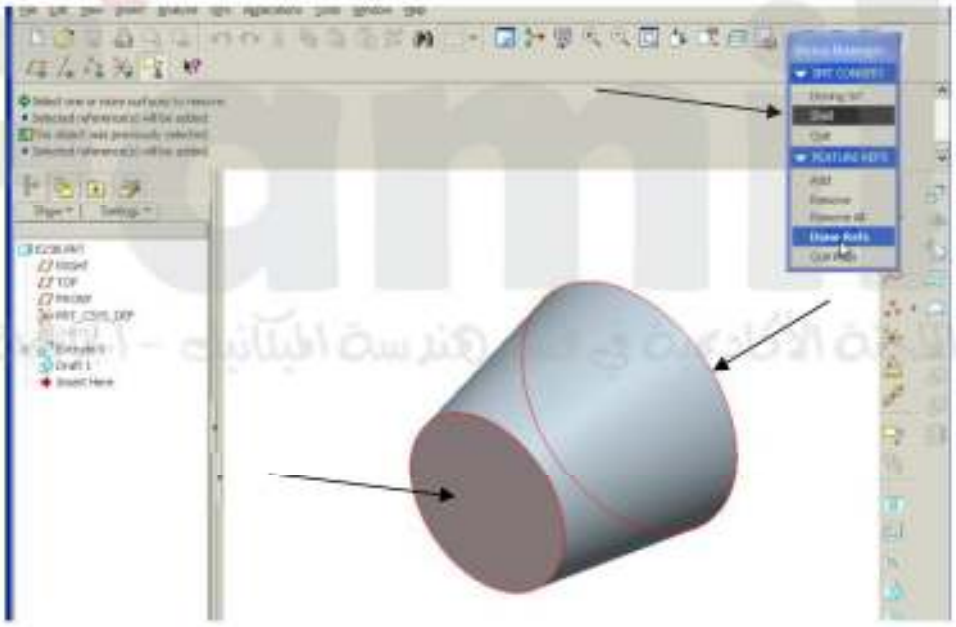
4. Add 13 degrees of draft.



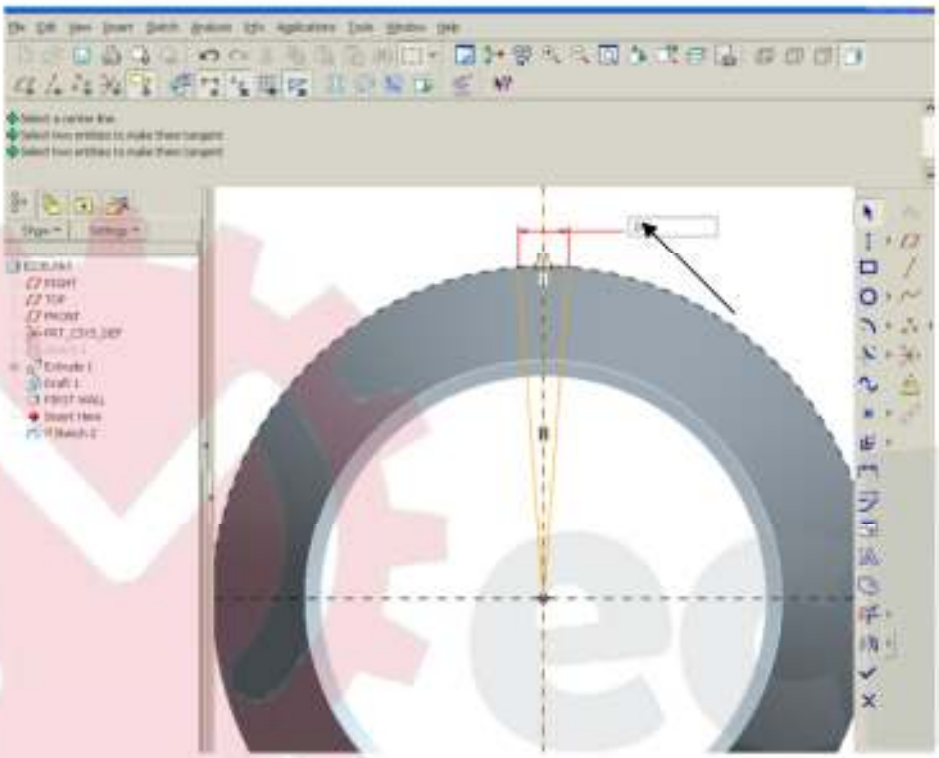
5. Switch to Sheetmetal.



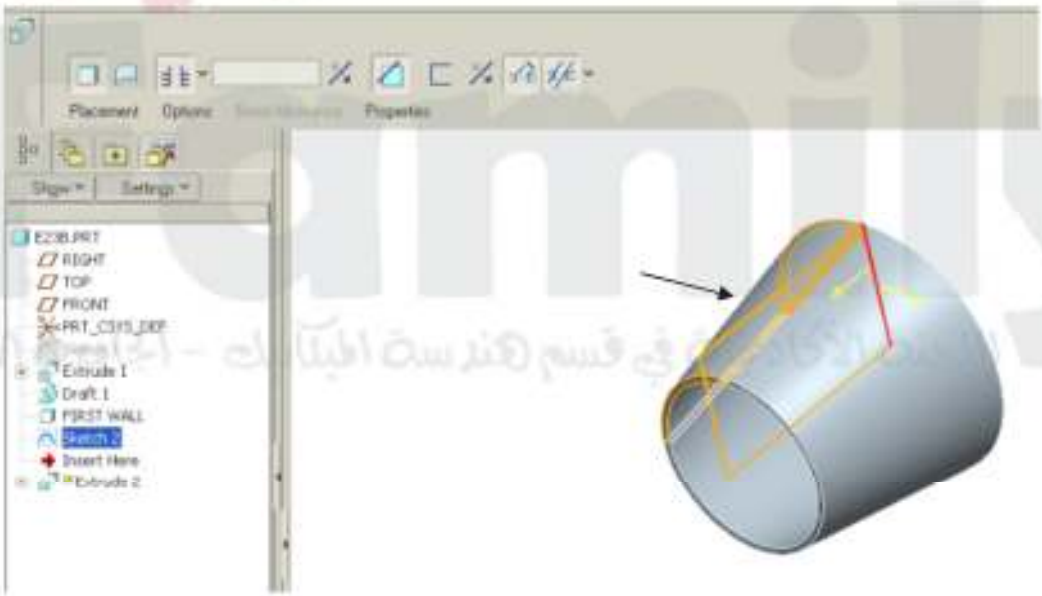
6. Select the front and back planar faces, then select the Shell feature. Set thickness to ".060". Hit Done Refs.



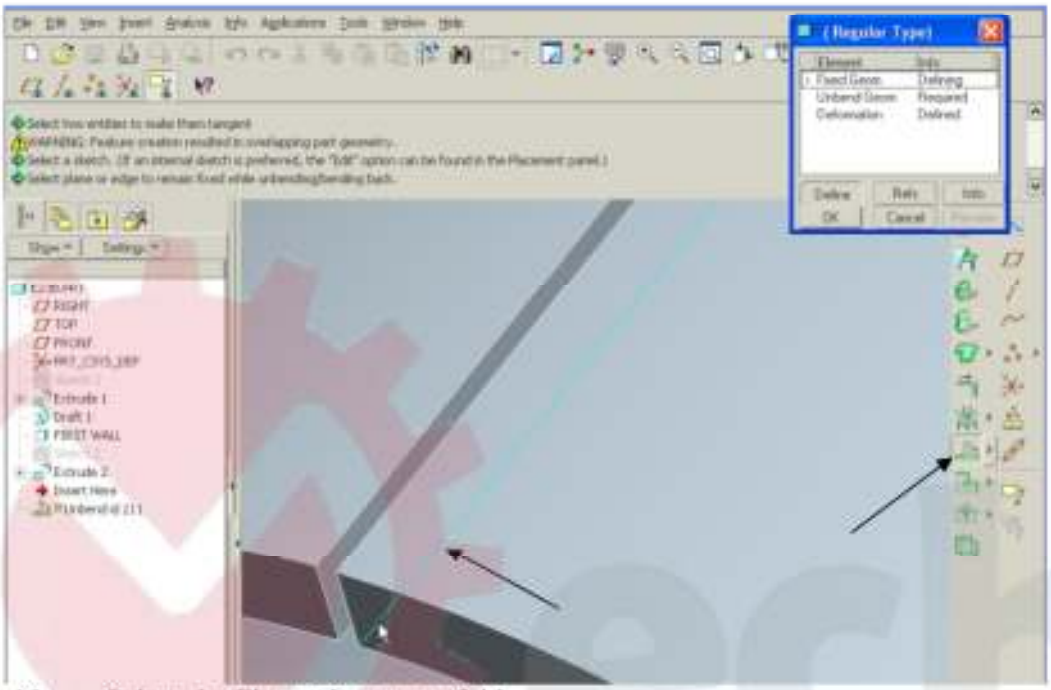
7. Select the “Front” plane and start a sketch on it. Draw the following angled cutout. Dimension the edges and set the thickness to .02”.



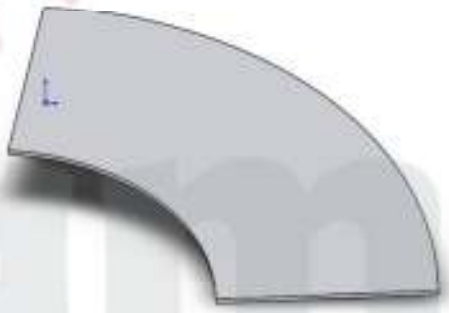
8. Cut Extrude Through All.



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



10. Select the Flatten icon to unfold.



Green Family
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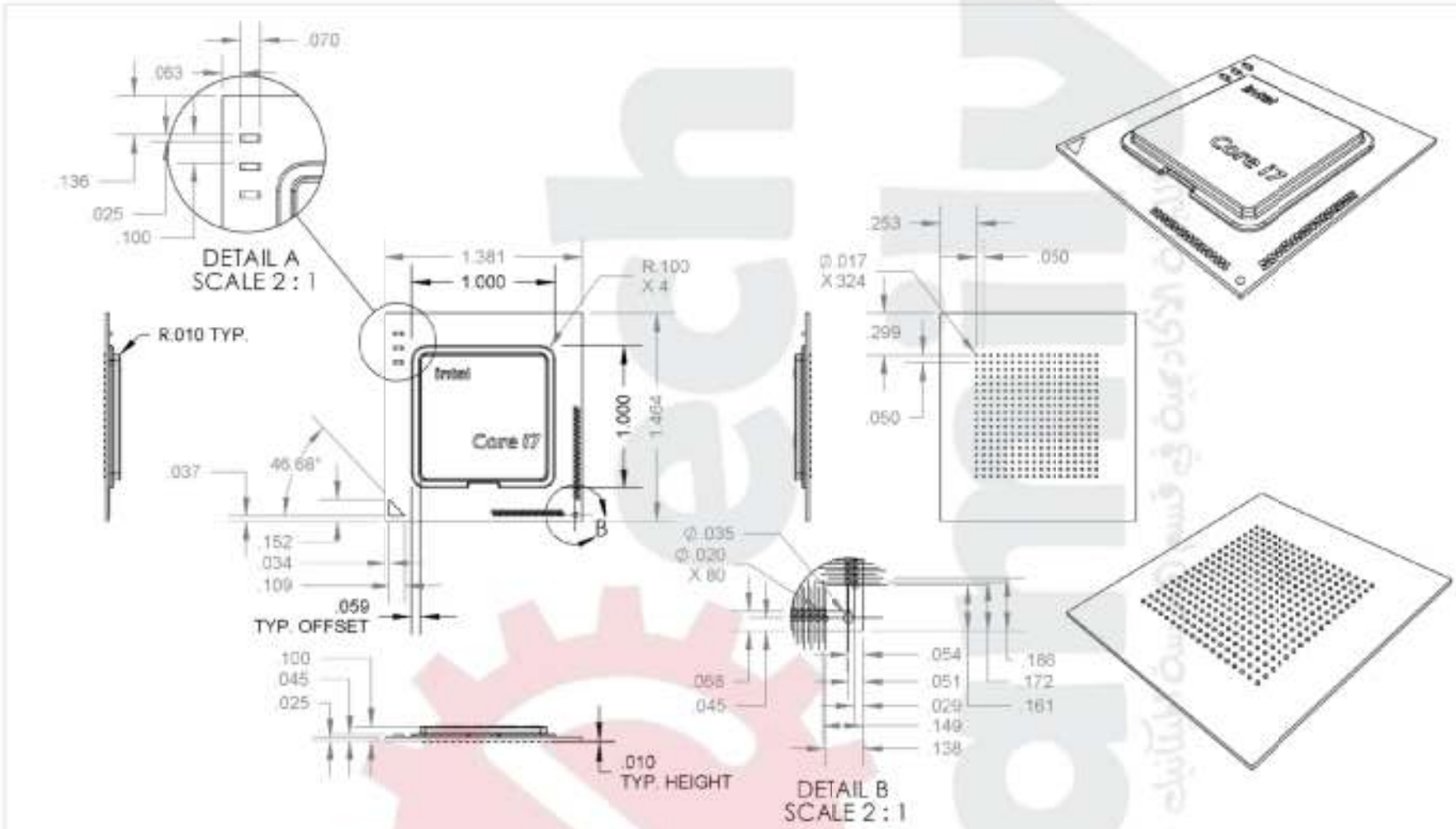

UNLESS OTHERWISE SPECIFIED:		NAME	DATE
DIMENSIONS ARE IN INCHES TOLERANCES: FRACTIONAL ± ANGULAR: MACH ± BEND ± TWO PLACE DECIMAL ± THREE PLACE DECIMAL ±	DRAWN		
	CHECKED		
	ENG APPR.		
	MFG APPR.		
INTERPRET GEOMETRIC TOLERANCING PER:	Q.A.		
MATERIAL	COMMENTS:		
FINISH			
NEXT ASSY	USED ON		
APPLICATION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

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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 24

Industrial Design Project

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

Part design for Thermoforming or Vacuum Forming

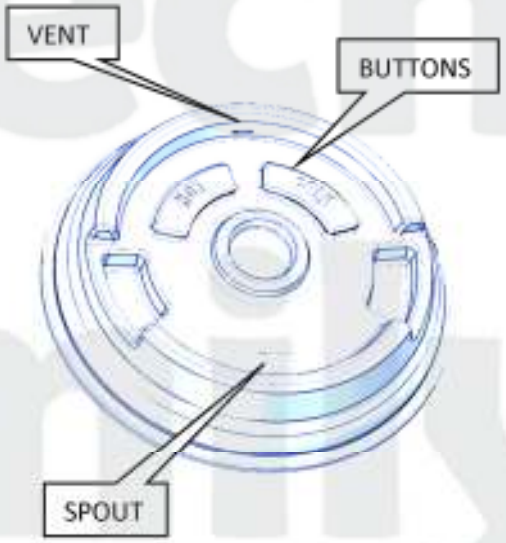
What is thermoforming?

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:wiki)

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

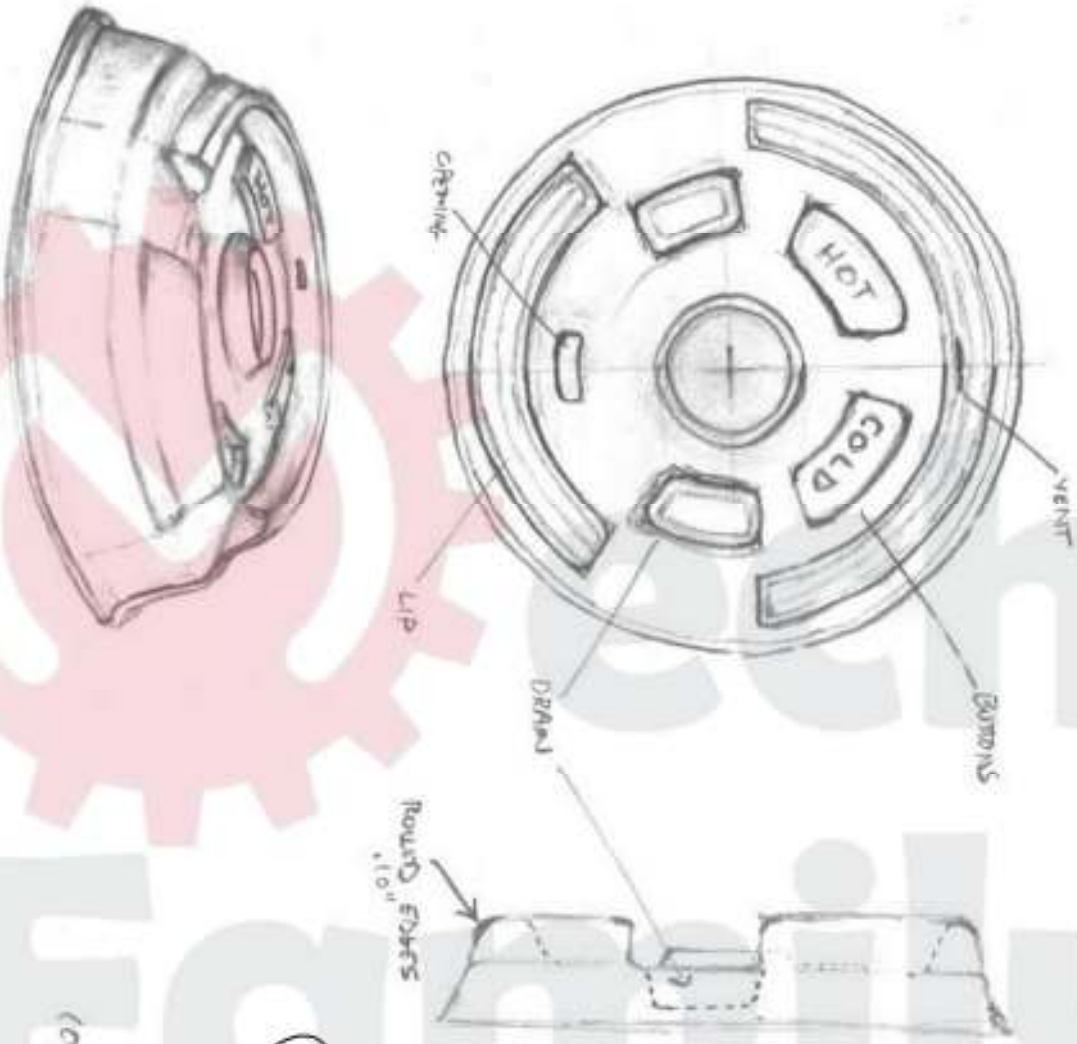


Once complete turn in all materials…

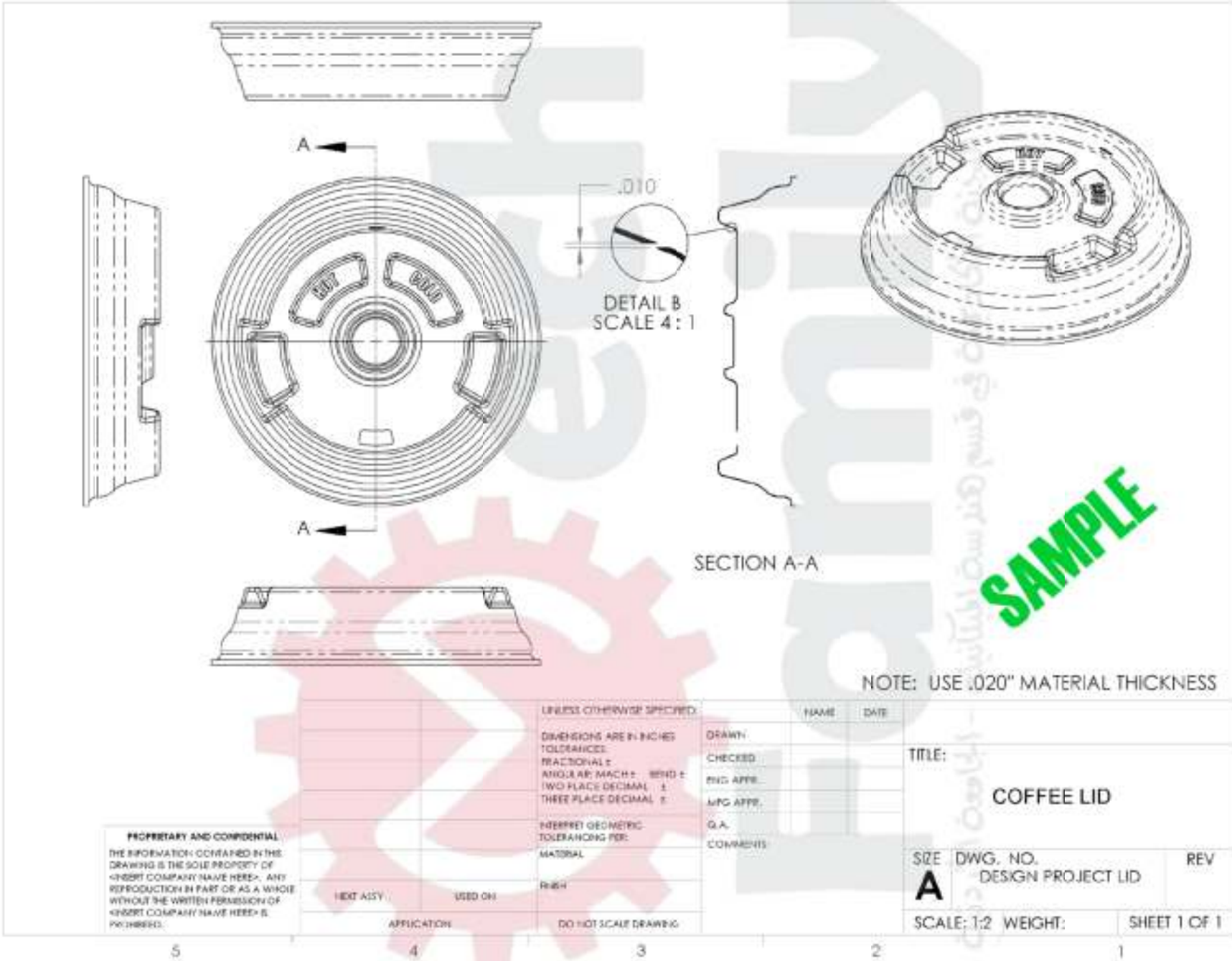
- 1 - Hand Sketch
- Pro Engineer model
- Pro Engineer drawing (print out copy)
- 3D Print Model (if available)
- Prepare brief summary of design and reasons for designing it the way you did.

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

HAND SKETCH



SAMPLE


A
A
.010
DETAIL B
SCALE 4 : 1
SECTION A-A
SAMPLE
NOTE: USE .020" MATERIAL THICKNESS
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN INCHES
TOLERANCES:
FRACTIONAL ±
ANGULAR: MACH ± BEND ±
TWO PLACE DECIMAL ±
THREE PLACE DECIMAL ±
INTERPRET GEOMETRIC TOLERANCING PER:
MATERIAL
FINISH
DO NOT SCALE DRAWING
NAME
DATE
DRAWN
CHECKED
ENG APPR.
MFG APPR.
Q.A.
COMMENTS
TITLE:
COFFEE LID
SIZE
A
DWG. NO.
DESIGN PROJECT LID
REV
SCALE: 1:2
WEIGHT:
SHEET 1 OF 1
NEXT ASSY
USED ON
APPLICATION
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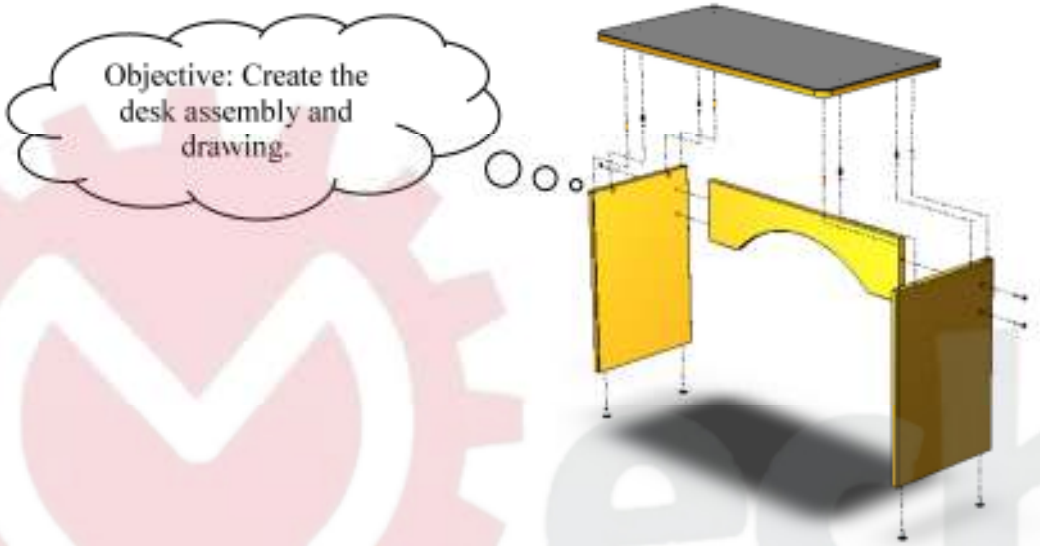
5
4
3
2
1

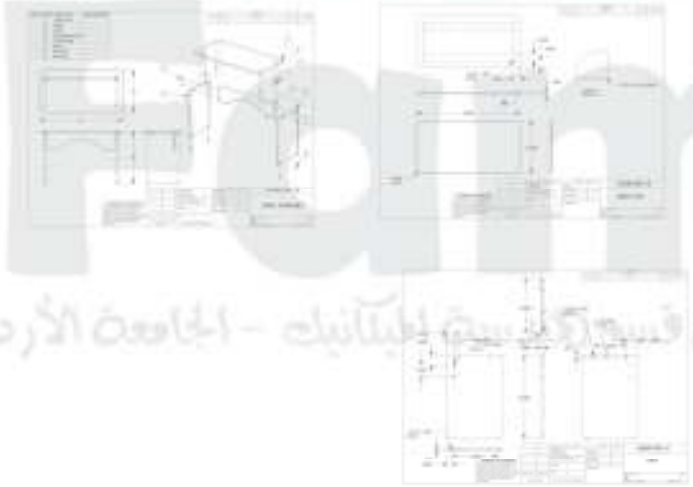
EXERCISE 25
Assembly and Drawing Automation

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.

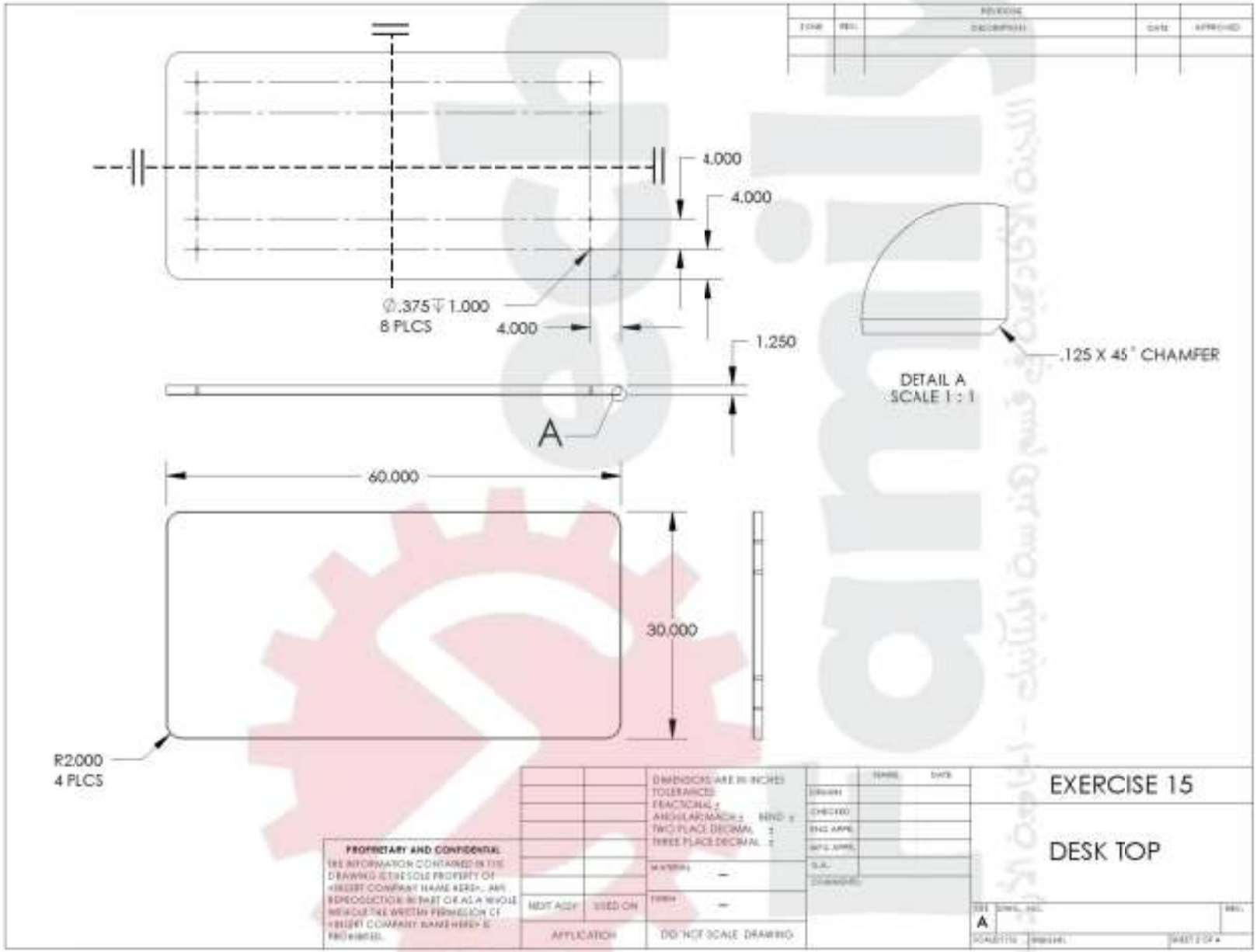
2. Now start a new part file and begin to create the attached parts.
NOTE: The drawings are missing some dimensions.

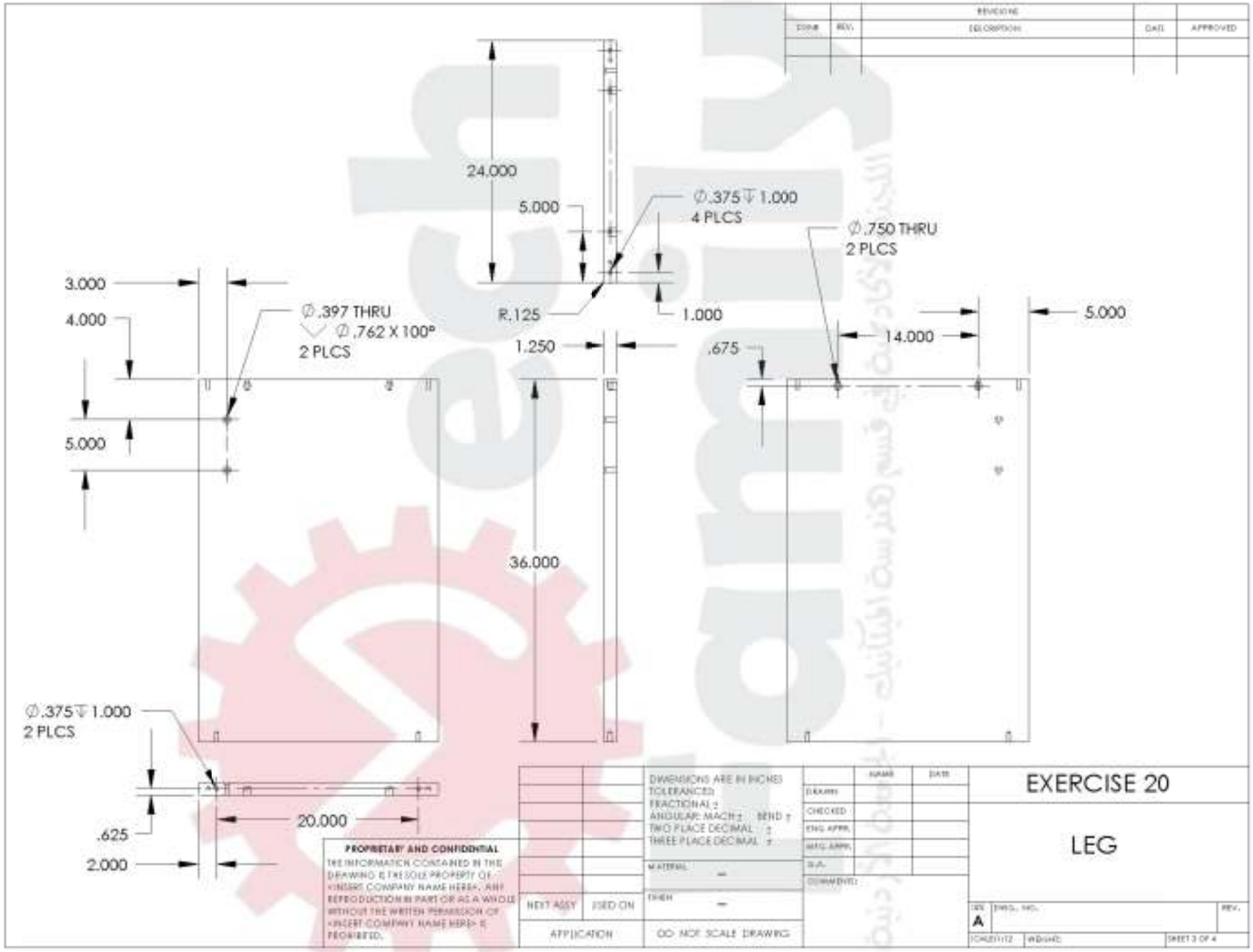


Family
اللجنة الأكاديمية في قسم هندسة الميكانيك - الجامعة الأردنية

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 Pro/E Photo Rendering utility. This is his how the finished model should appear after rendering.


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
EXERCISE 15
DESK TOP
PROPRIETARY AND CONFIDENTIAL
DO NOT SCALE DRAWING


24.000
5.000
Ø.375↧1.000
4 PLCS
Ø.750 THRU
2 PLCS
3.000
4.000
Ø.397 THRU
Ø.762 X 100°
2 PLCS
R.125
1.000
1.250
.675
14.000
5.000
5.000
36.000
Ø.375↧1.000
2 PLCS
.625
2.000
20.000
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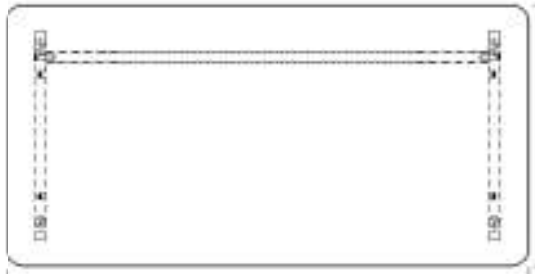
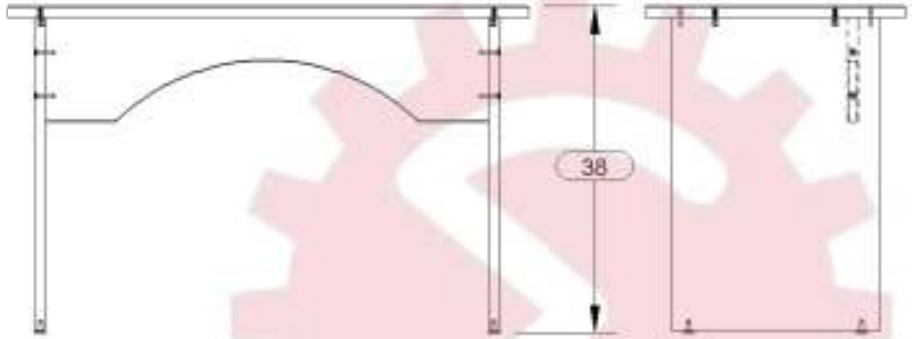
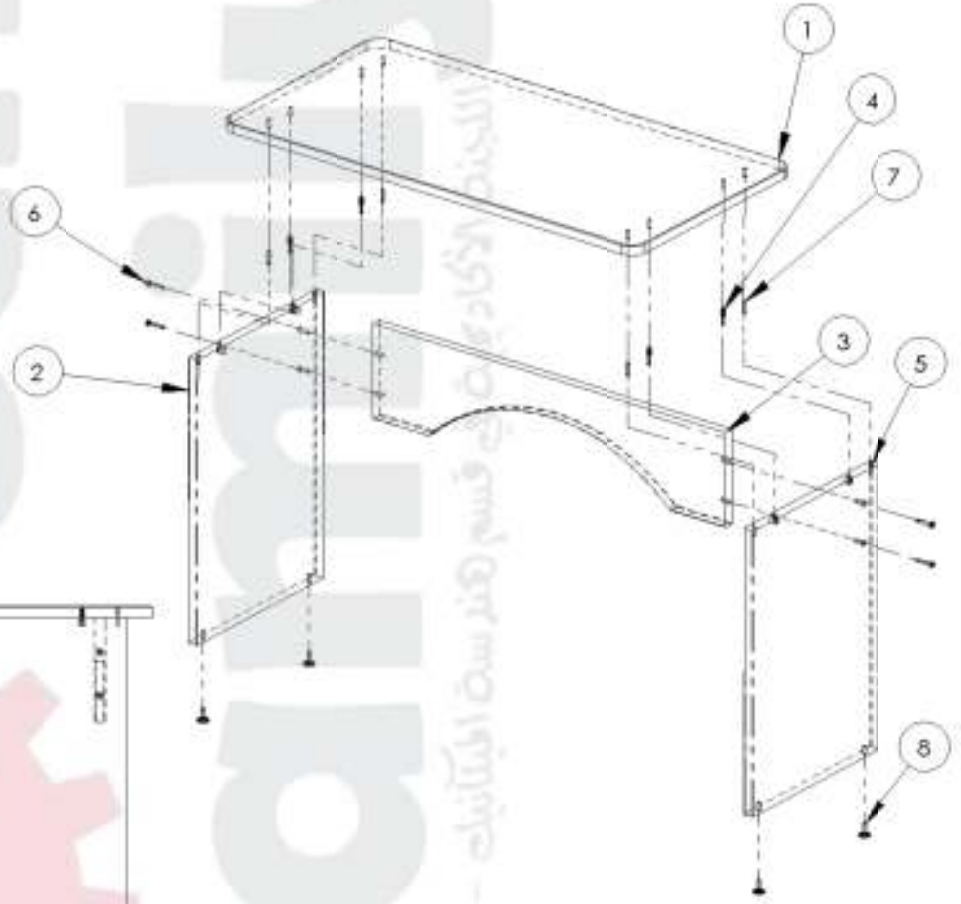
NEXT ASSY
USED ON
APPLICATION
DIMENSIONS ARE IN INCHES
TOLERANCES:
FRACTIONAL ±
ANGULAR: MACH ± BEND ±
TWO PLACE DECIMAL ±
THREE PLACE DECIMAL ±
MATERIAL
FINISH
DO NOT SCALE DRAWING
DRAWN
CHECKED
ENG APPR.
MFG APPR.
Q.A.
COMMENTS:
EXERCISE 20
LEG
SIZE A
SHEET 3 OF 4

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

ZONE	REV.	REVISIONS DESCRIPTION	DATE	APPROVED







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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 A | DWG. NO. | REV.

SCALE:1/2.5 | WEIGHT: | SHEET 1 OF 4

EXERCISE 21

Introduction to Surfacing

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

Mech Family
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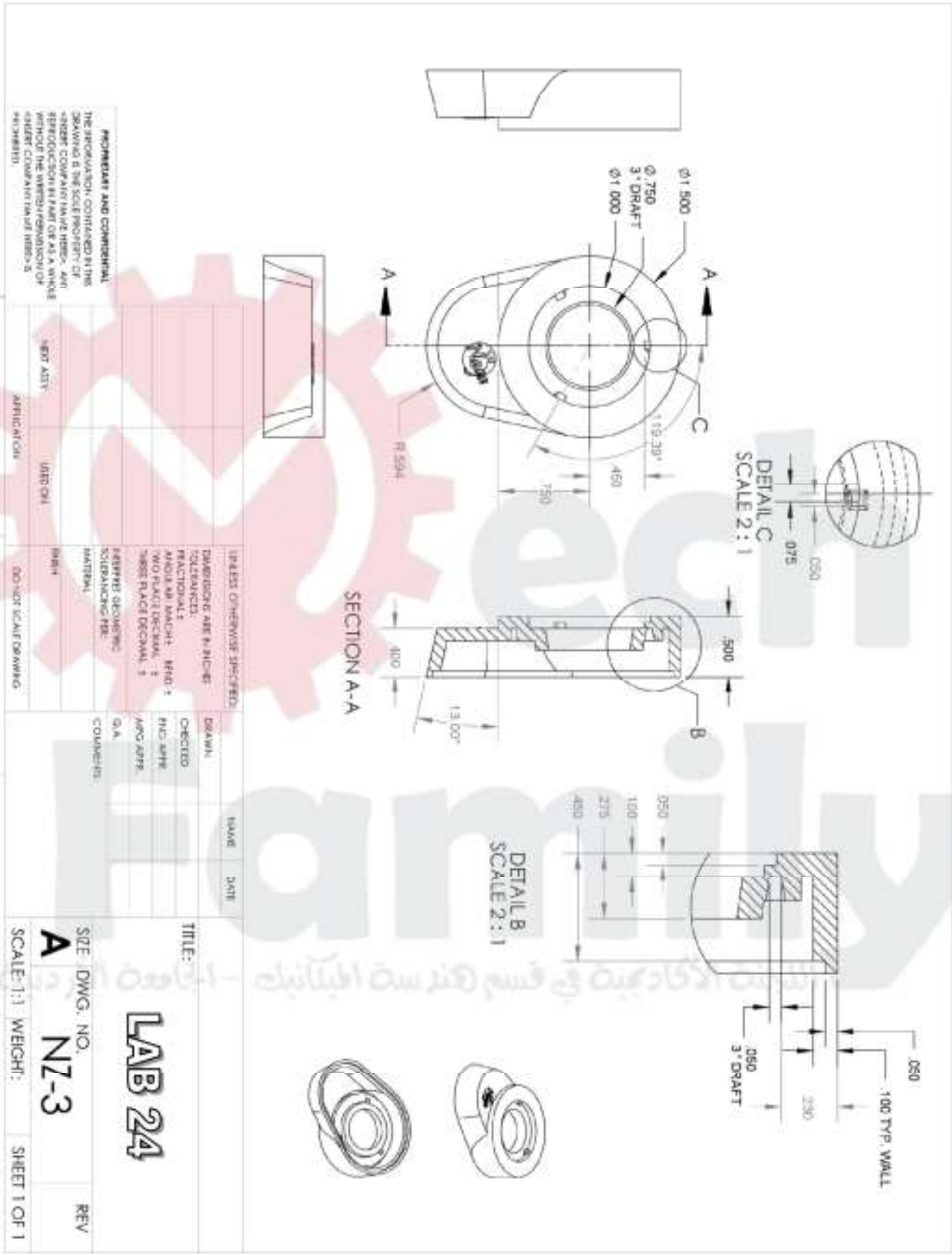


Mech Family
المؤسسة الأكاديمية في قسم هندسة الميكانيك - الجامعة الأردنية

EXERCISE 25

Pro/E Administration

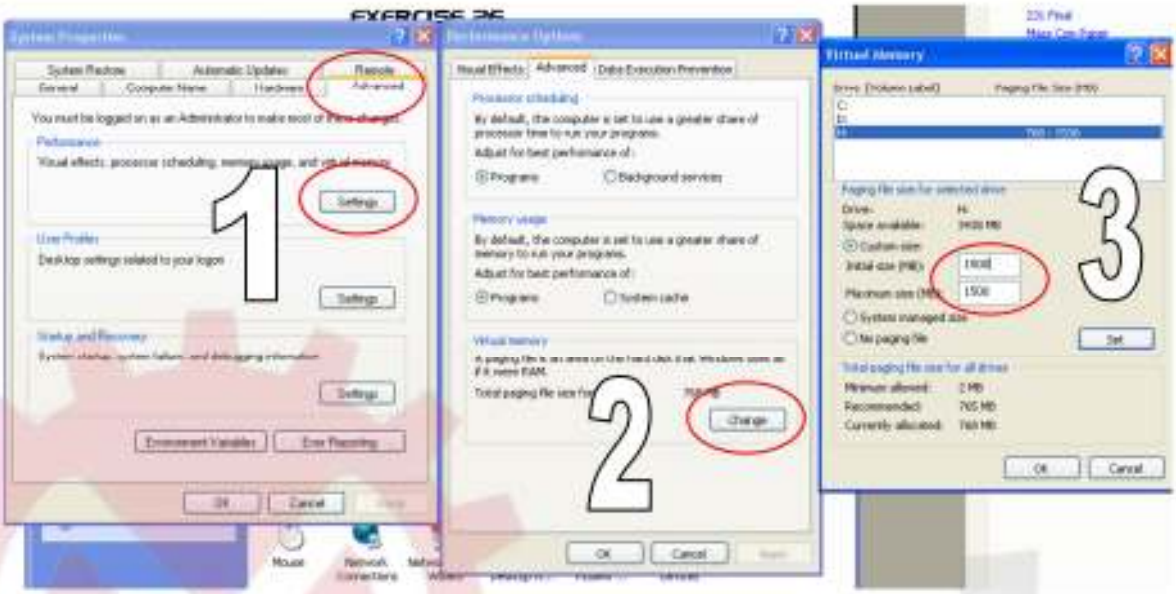
***Finding adequate computer Hardware to run Pro/E can be difficult**, this lesson looks at the multiple aspects of selecting hardware as well as modifying settings inside Pro/E to allow it to run efficiently and trouble free.*

Selecting an Operating System (OS).

Windows XP Professional
Windows XP 64-Bit edition
Widows Vista
Windows 7
Windows 8

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 2000 MB of Ram 4000 MB Virtual Ram. And keep the initial size the same as the maximum size. It is said that this prevents write errors.



Processors (CPU)

BUDGET

MAIN STREAM

HIGH END

EXOTIC

Intel

- Atom 1, 2 cores 2, 4 threads
- Celeron Dual Core
- Pentium Dual Core
- Centrino Mobile
- Core 2 Duo
- i5 - 4 cores, 4 threads
- i7 - 4 cores, 8 threads
- Xeon
- Core 2 Duo Extreme
- Quad Core Xeon
- Itanium IA64

AMD

- Sempron
- Athlon 64
- Athlon 64 X2
- Turion 64 Mobile
- Phenom 3 – 4 cores
- Phenom II 3 – 4 cores
- Opteron - up to 6 cores

(source: www.tomshardware.com)

اللجنة الأكاديمية في قسم هندسة الميكاترونيك - الأردنية

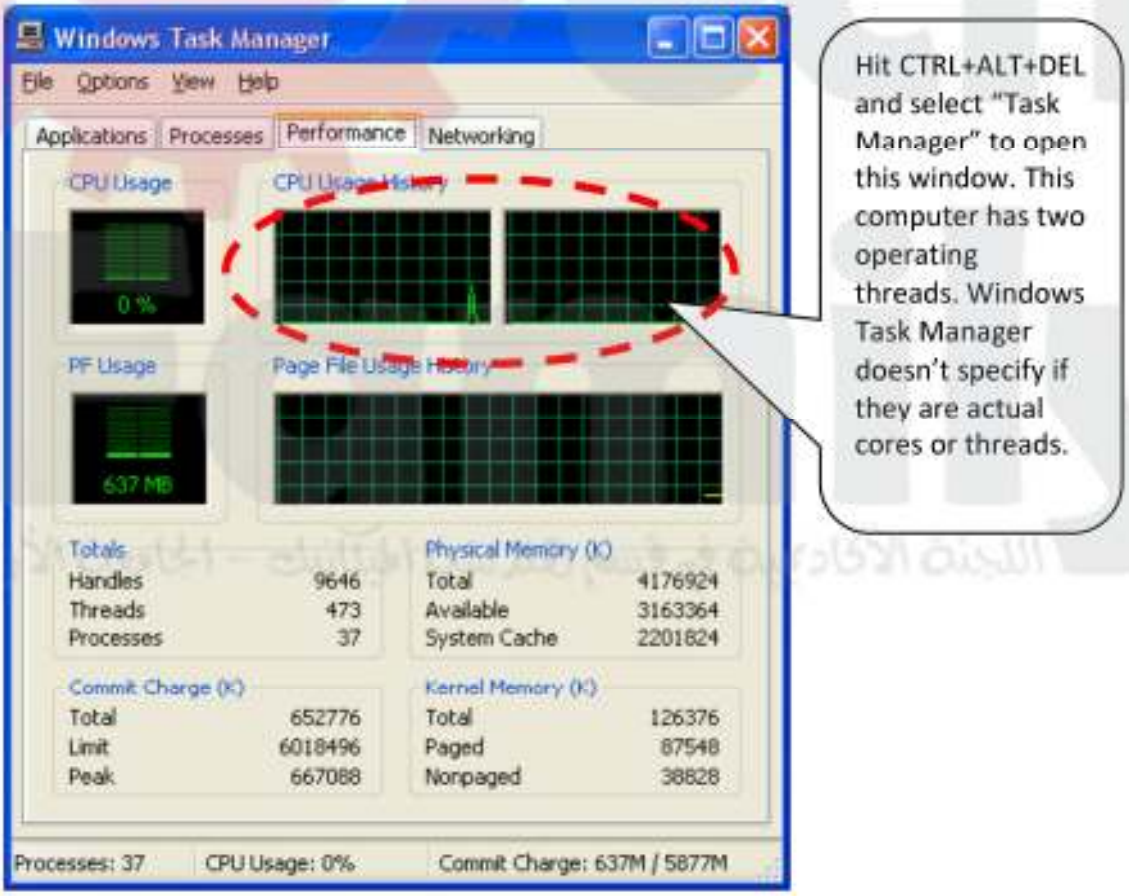
Multiprocessing

Most CPU manufacturers are beginning to deliver multiple core processors and even multiple threads. This can be seen with the AMD Opteron which has six processing cores, versus Intel's Core i7 processor that has four cores with eight threads due to their Hyper Threading technology.

What's the difference between a core and a thread?

Threads are process pipelines that are generally used exclusively to conduct a process. A core is an actual piece of hardware consisting of transistors and other processing elements required to make a Central Processing Unit. In Intel's case they developed a CPU core that can manage two processing pipelines or "threads" for multitasking. The secondary thread is not as fast as a sole core thread but can still enhance performance to a significant level.

How can I tell how many cores or threads I have on my PC?

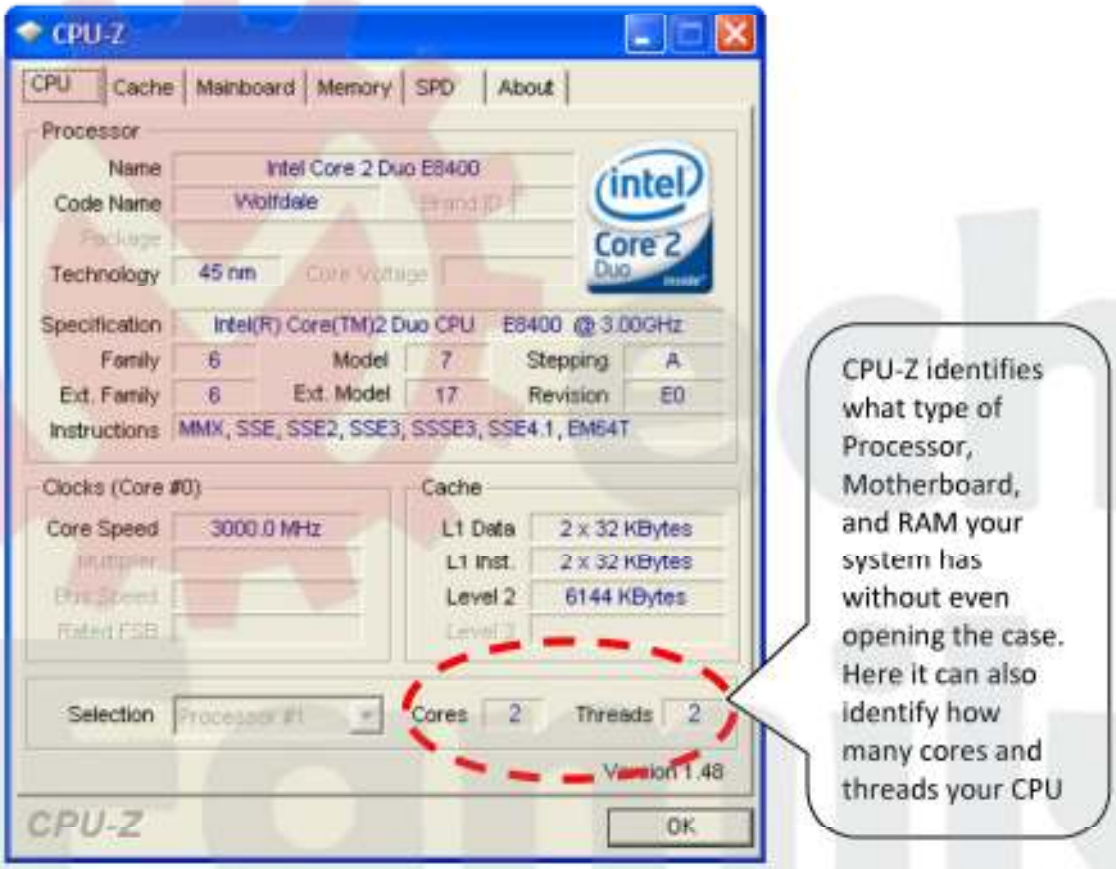


Which one will run Pro/E fastest? You can find benchmarks at www.spec.org specifically for Pro/E or you can look for the generic OpenGL benchmark results that usually use an OpenGL video game.

How to look inside your Computer without opening it!

You can use CPU-Z to identify your computer hardware. Download it for free at:

http://www.cpuid.com/cpuz.php



Question is: can Pro/E benefit from multiple cores? Currently one might find an average of 10 – 25% performance increase. This is because Pro/E is not fully written to take advantage of multithreaded processes.

The biggest benefit one might find is the ability to multitask while working with an FEA analysis or Photorealistic rendering. This is a long process and you could actually open up another window of Pro/E or Outlook and continue working while the analysis is running with little slow down in performance.

Graphics Cards

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

- NVIDIA ***Quadro*** series (not NVS series)
  - **Quadro FX 380 erp.$129** (erp- estimated retail price)
  - **Quadro FX 580 erp.$199**
  - **Quadro FX 1800 erp.$499**

- ATI ***FirePro*** series (not FireMV series)
  - **FirePro 3800 erp.$159**
  - **FirePro 5800 erp. $499**
  - **FirePro 8800**

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 Pro/E at work, DON'T SKIMP! Buy a professional grade video card. For home use the nVidia Geforce series is okay, but you will still experience some graphical glitches.

GRAPHICS CARD – Pro/E BENCHMARK (source: www.tomshardware.com)

TRANSLATOR INDEX

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						

Tech Family
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CAD 111 TOTALS

E13 – 30pts
E14– 30pts
E15– 30pts
E16– 30pts
E17– 15pts
E18– 30pts
E19– 30pts
E20– 15pts
E21– 15pts
E22-30pts
E23-15pts
E24-30pts

MIDTERM – 300pts
FINAL – 300pts
ATTENDANCE & PARTICIPATION -100pts
TOTAL - 1000pts

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