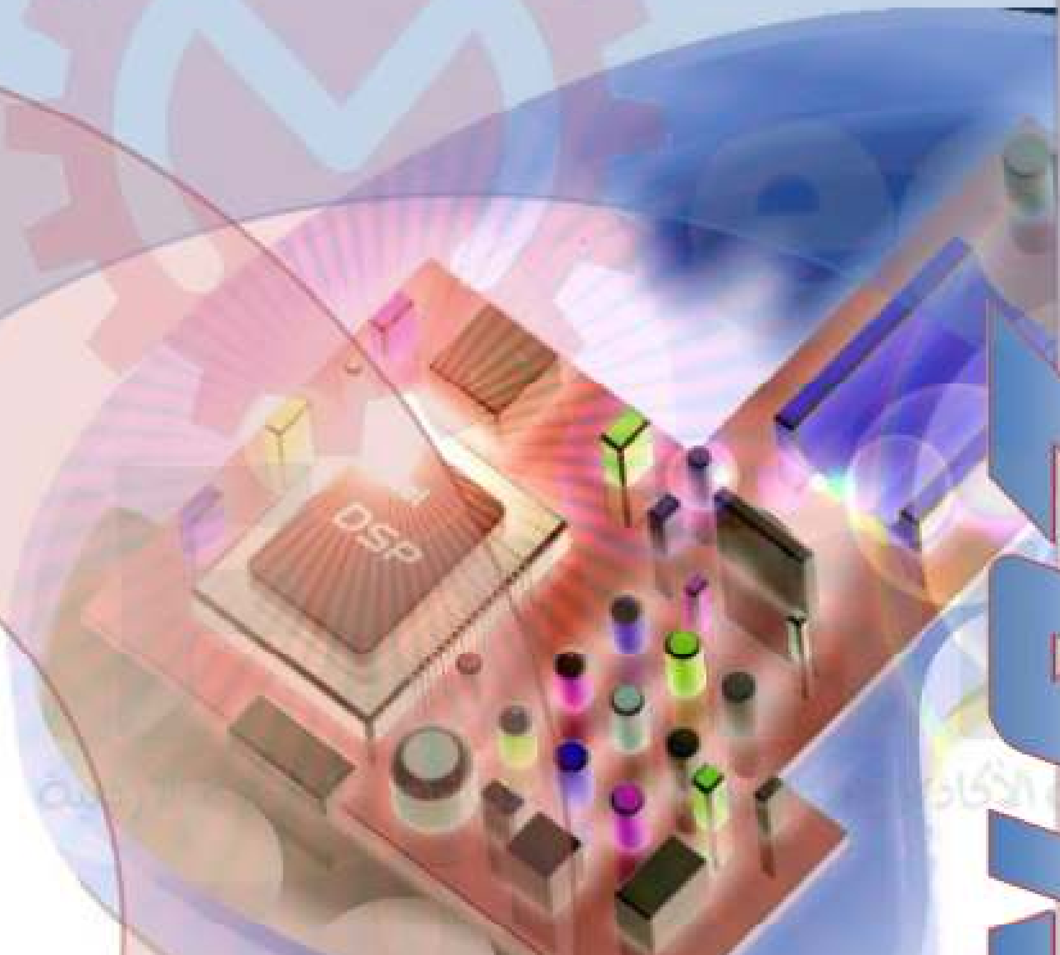
CREO Parametric 2.0
BASICS

BY CHRISTOPHER F. SIKORA

COMPUTER AIDED DESIGN

2013



Pro/ENGINEER (Creo 2.0) Basics 105

Course Description:
Pro/ENGINEER (Creo) Basics
3 credit hours
Exploration of the theory and application of solid modeling techniques for product design and manufacturing. Prerequisite: Intro to Engineering Drawings 101 or consent of instructor.

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 Basics pdf. and videos provided on
YouTube http://www.youtube.com/user/vertanux1
Or simply search the exercise number (example: E5 Creo)

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

General Course Outline

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

Chapters

1. Introduction to the Interface
   Modeling Theory - Sketching and Base Feature Geometry Creation.

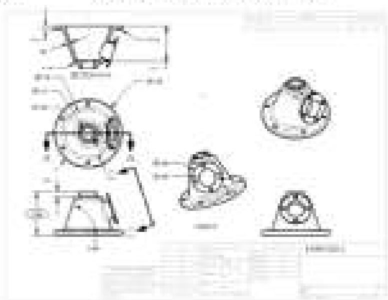
2. Part Modeling
   Revolved Method.

3. Secondary Feature Modeling (Draft, Offsetting Entities, Filleting)

4. Advanced part Modeling (Sweeps, and Circular Patterns)

5. Bottom-Up Assembly Modeling

6. Creating Drawings



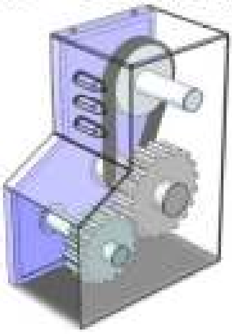
7. 3D Guide Curve using a surface

8. Swept Blend/Lofting

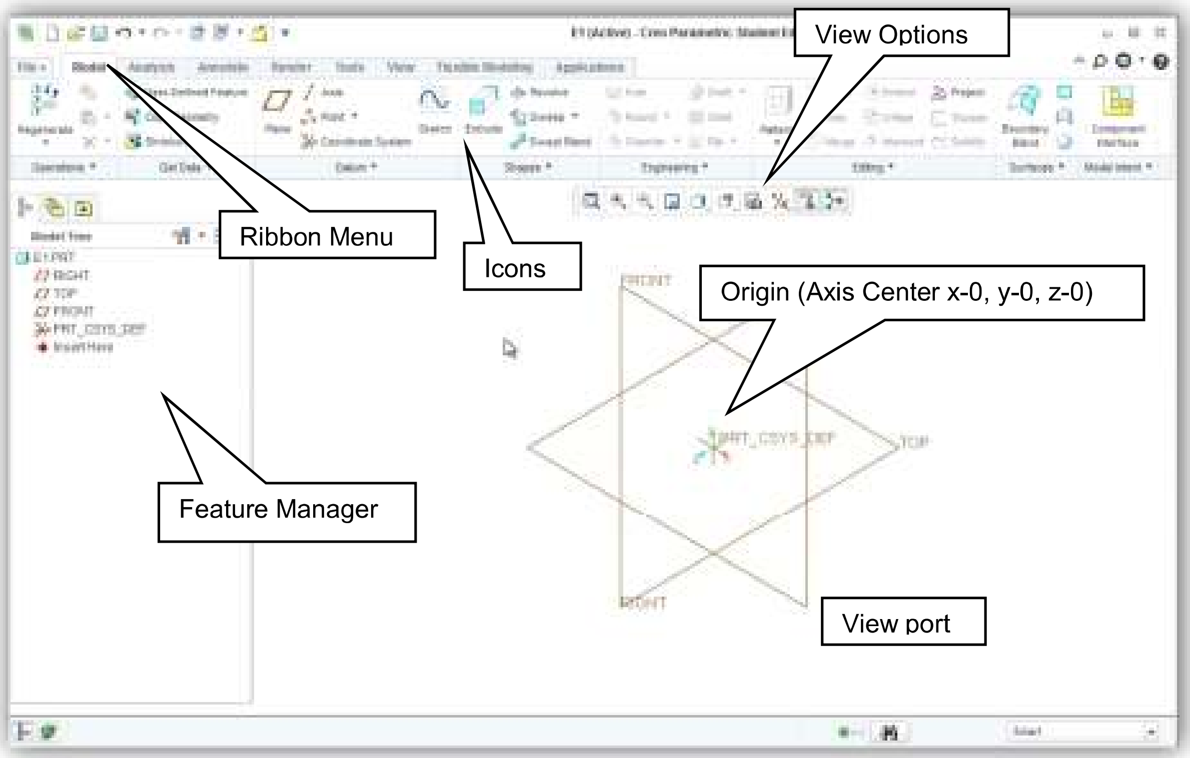
9. Introduction to Top-Down Assembly Modeling

10. Top-Down Assembly Editing

11. Bottom-Up Assembly Project



creo 2.0 Interface



Mouse Buttons

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

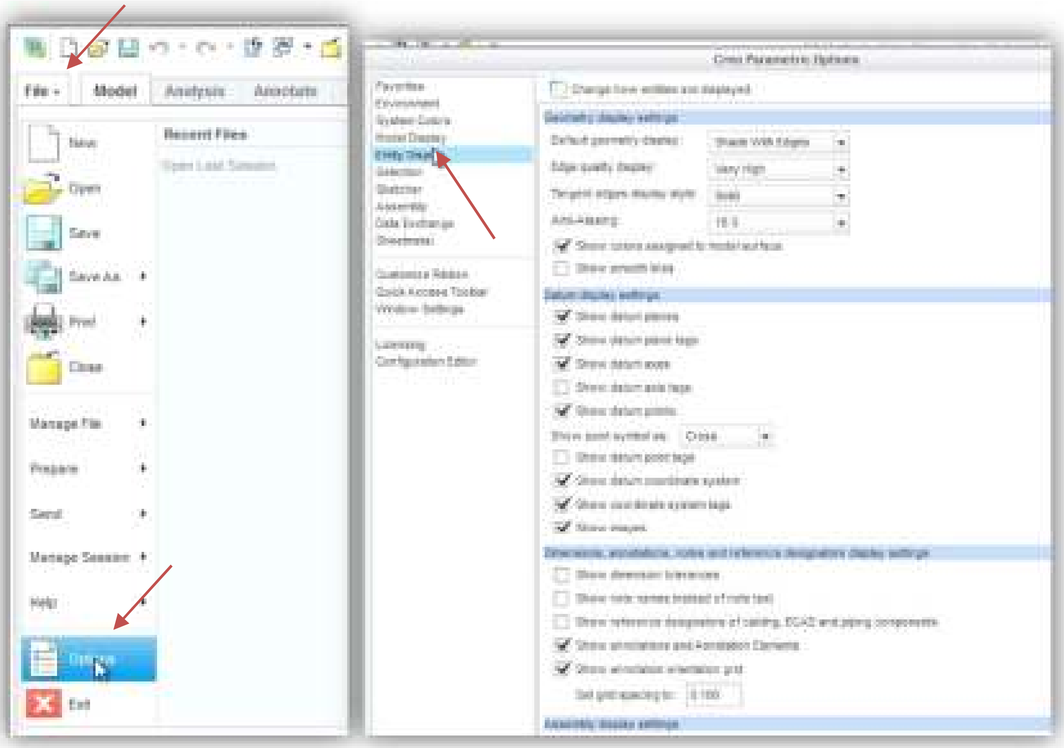
Right Button – Used for activating pop-up menu items, typically used when editing. (*Note: you must hold the down button for 2 seconds*)

Center Button – (option) Used for model rotation, dimensioning, **zoom** when holding Ctrl key, and **pan** when holding Shift key. It also cancels commands and line chains.

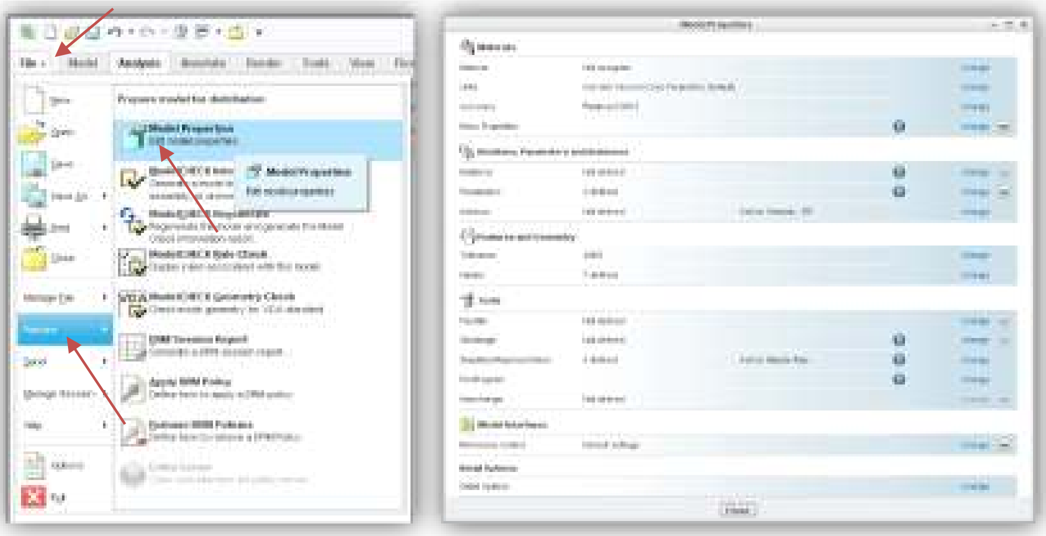
Center Scroll Wheel – (option) same as Center Button when depressed, only it activates **Zoom** feature when scrolling wheel.

“Options & Properties” menus *“The heart of* creo*”*

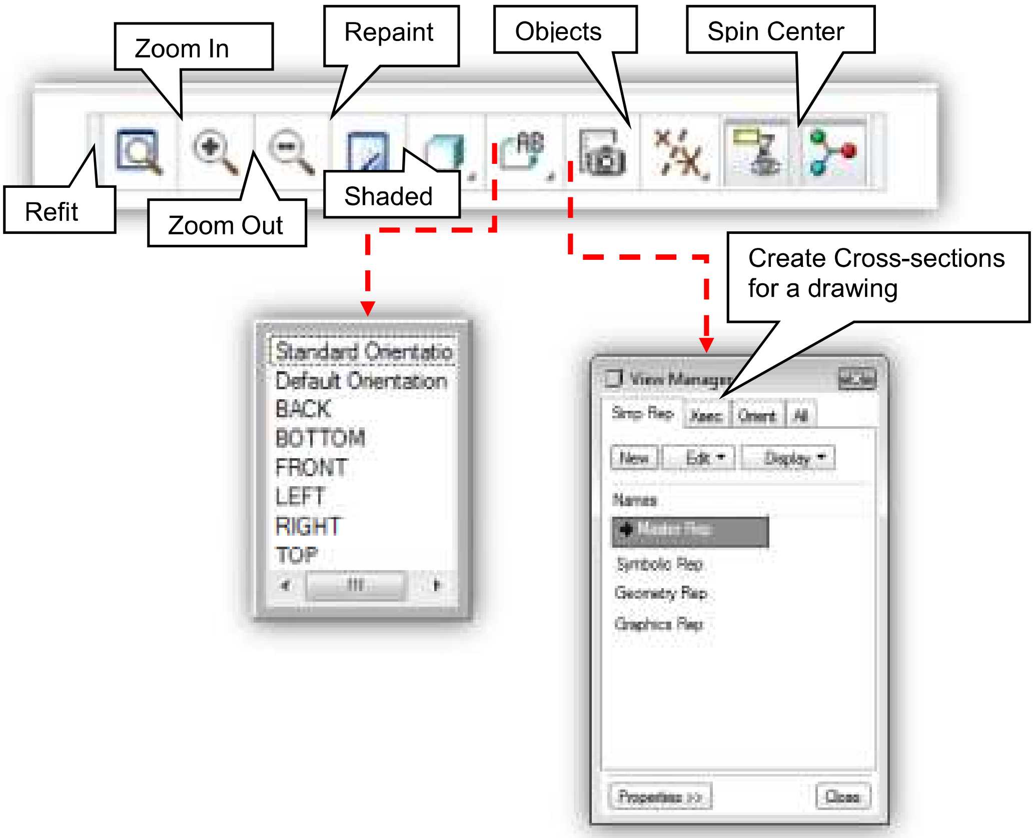
Selecting the “File” – “Options” pull down menu *(located at the top left side of the screen)* **opens the active documents Options.**



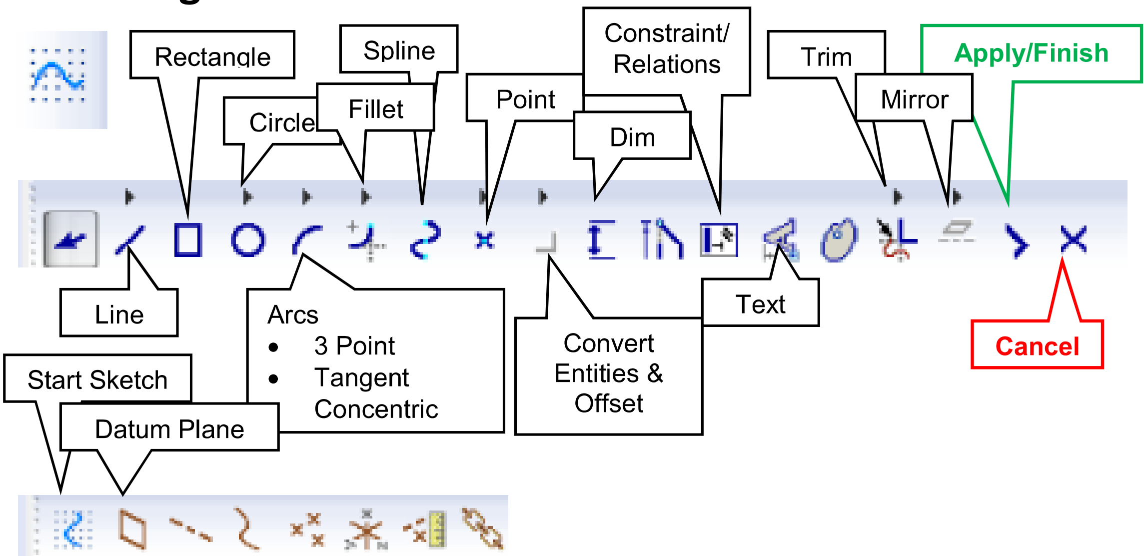
Model Properties



View options



Sketching

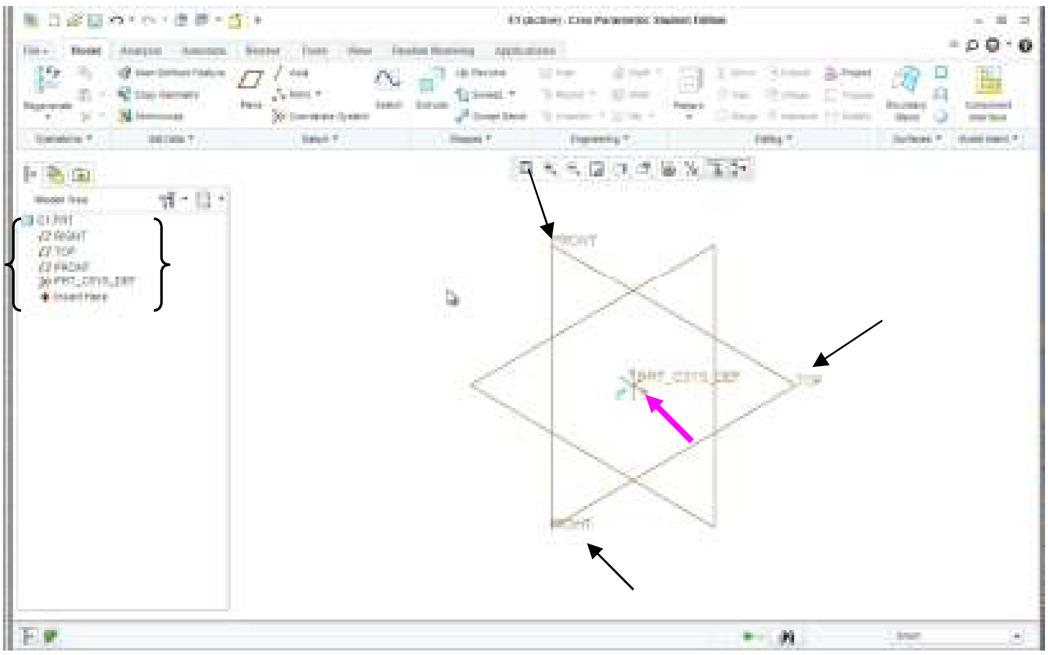


NOTE: If you do not see all of these icons on your interface you can customize the toolbars to bring them up. Right mouse button click on the top grey frame of the window and locate the "customize" option.

Where do you start a sketch?

Sketches can be created on any Datum Plane or Planar Face or Surface. Pro/E provides you with three datum planes centralized at the Origin (your zero mark in space)

NOTE: *Planes can also be created and will be discussed in more detail in the future. Also after completing a sketch always select the* ***Apply/Finish*** *check mark on the sketch toolbar, this will activate the extrude or revolve feature tools.*



To start a sketch Pre-select the plane or face you desire to sketch on and then select the Sketch Icon. ***NOTE****: You can select the planes from the "Feature Manager". You can also pre-select the feature you intend to use, and then you will be prompted to select a datum plane to begin your sketch.*

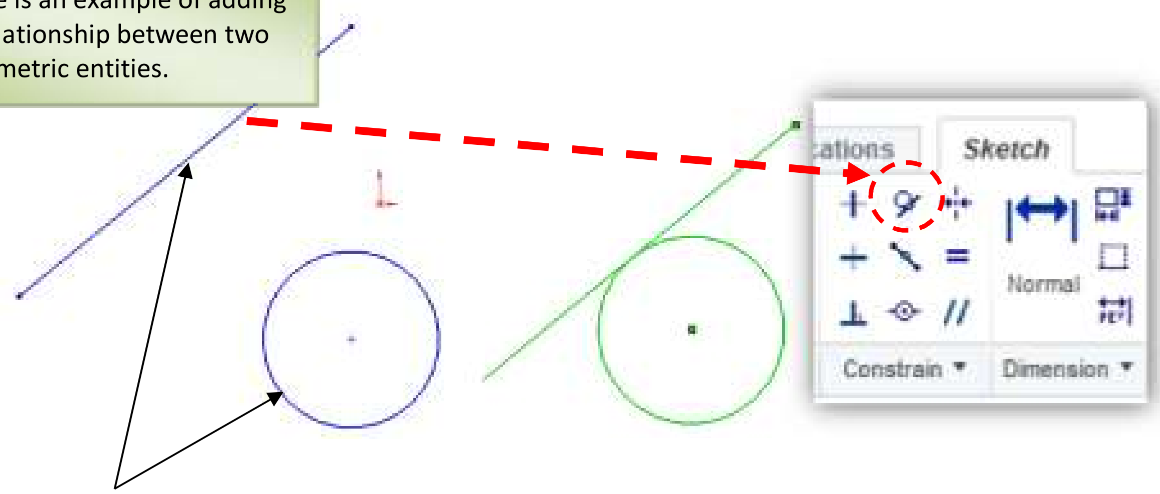
Sketch Options –

Controlling your geometry…

Creo - Pro/E uses two methods for constraining geometric entities.
Constraints and **Dimensions**

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

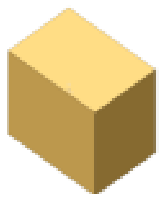
Here is an example of adding a relationship between two geometric entities.

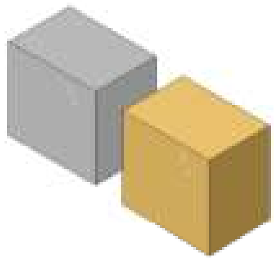


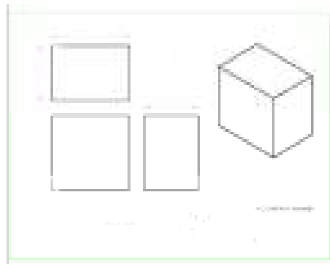
Cautious sketching can save time.

There are 3 primary file types in Creo, which include…

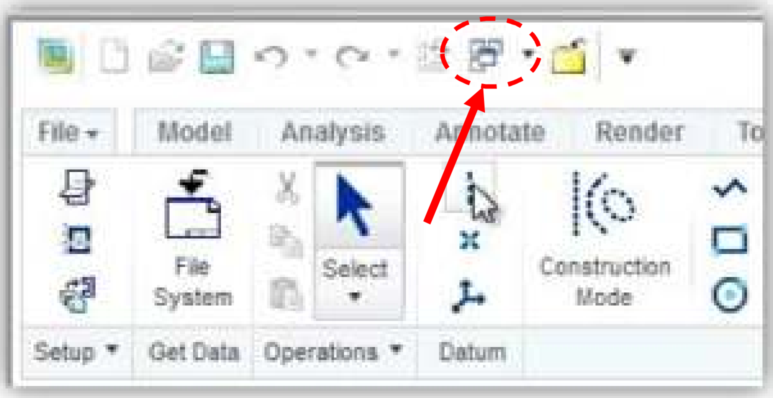
1. ***Part*** (.prt)
   Single part or volume.
2. ***Assembly*** (.asm)
   Multiple parts in one file assembled.
3. ***Drawing*** (.drw)
   The 2D layout containing views, dimensions, and annotations.





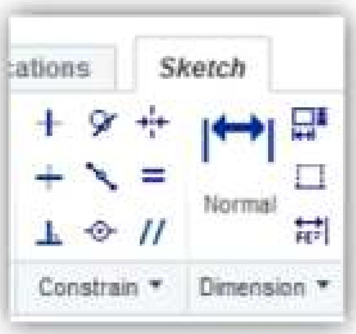


Switching between documents (Activating a document)



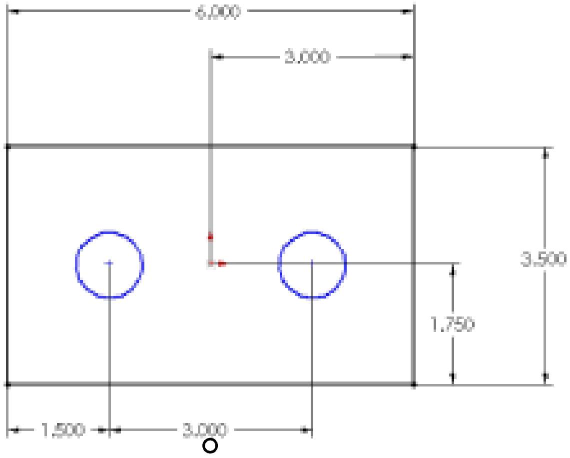
Select the Window pull-down menu and you will see the available documents. Click on the document you wish to work on from the list to "activate" it.

Sketch Constraints (Relations)



Constraint	Geometric entities to select	Resulting Constraint
Horizontal or Vertical	One or more lines or two or more points.	The lines become horizontal or vertical (as defined by the current sketch space). Points are aligned horizontally or vertically.
Collinear	Two or more lines.	The items lie on the same infinite line.
Perpendicular	Two lines.	The two items are perpendicular to each other.
Parallel	Two or more lines. A line and a plane (or a planar face) in a 3D sketch.	The items are parallel to each other. The line is parallel to the selected plane.
Tangent	An arc, ellipse, or spline, and a line or arc.	The two items remain tangent.
Concentric	Two or more arcs, or a point and an arc.	The arcs share the same centerpoint.
Midpoint	Two lines or a point and a line.	The point remains at the midpoint of the line.
Coincident	A point and a line, arc, or ellipse.	The point lies on the line, arc, or ellipse.
Equal	Two or more lines or two or more arcs.	The line lengths or radii remain equal.
Symmetric	A centerline and two points, lines, arcs, or ellipses.	The items remain equidistant from the centerline, on a line perpendicular to the centerline.

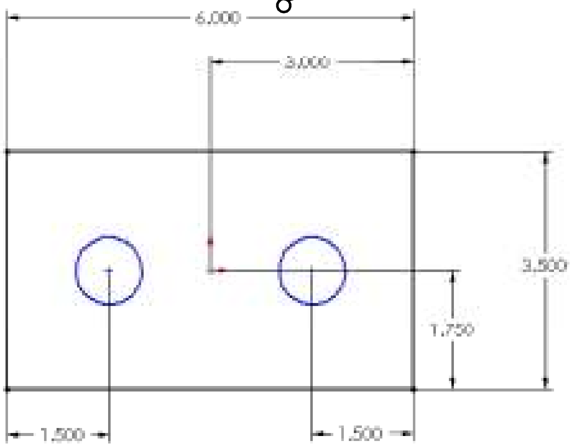
Controlling your geometry with dimensions…

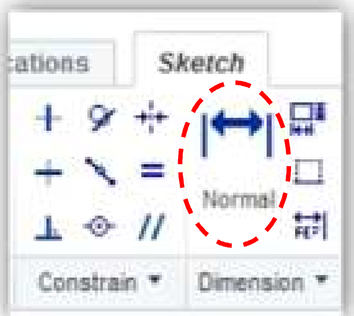


Dimensioning this way will enable the length of the bracket to change but the holes will always remain positioned to the left side.

Strong versus Weak Dimensions - Double click and change to make them Strong!

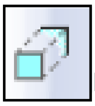
Dimensioning this way will enable the length of the bracket to change but the holes will always remain positioned to 1.5” off each side.





Solid Modeling Basics

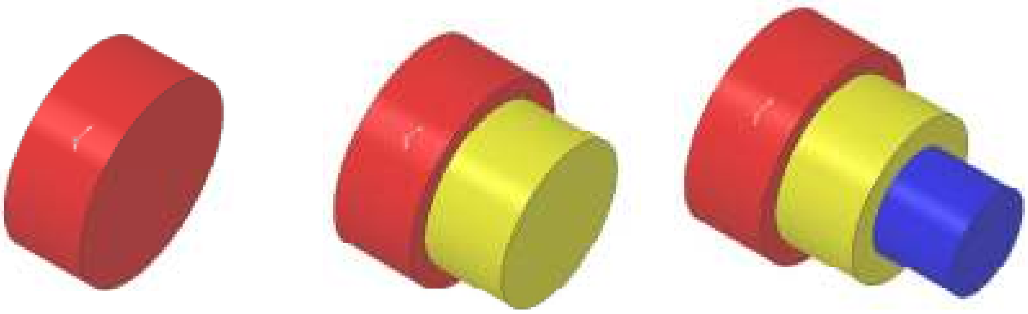
Layer Cake method

Extruded Boss/Base (Creates/Adds material)

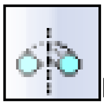
Extruded Cut (Removes material)

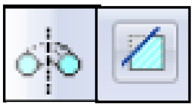
Ingredients:

- Profile



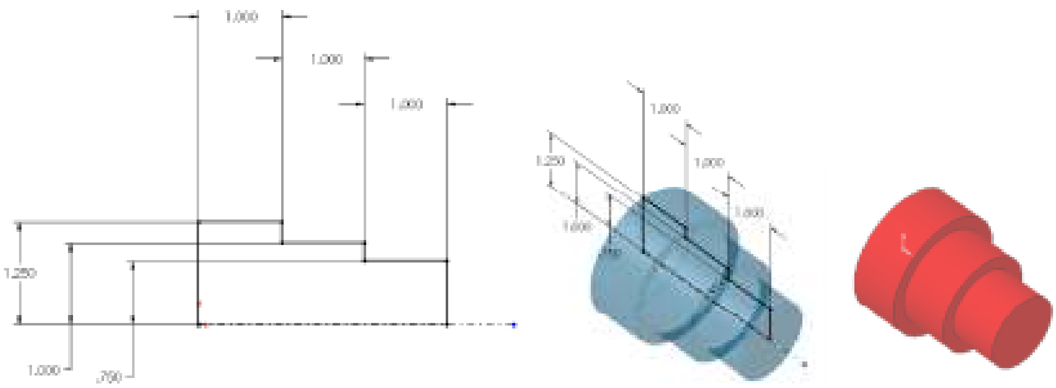
Revolve method

Revolve Boss/Base (Creates/Adds material)

Revolve Cut (Removes material)

Ingredients:

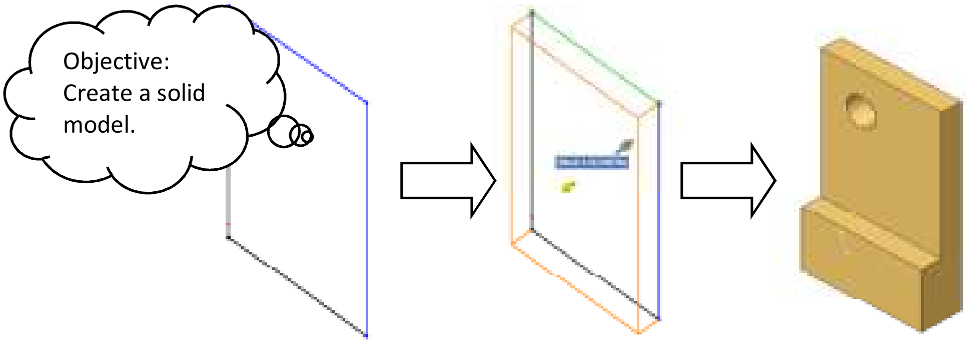
- Profile
- Center Line *(Note: The profile cannot cross over the center line!)*

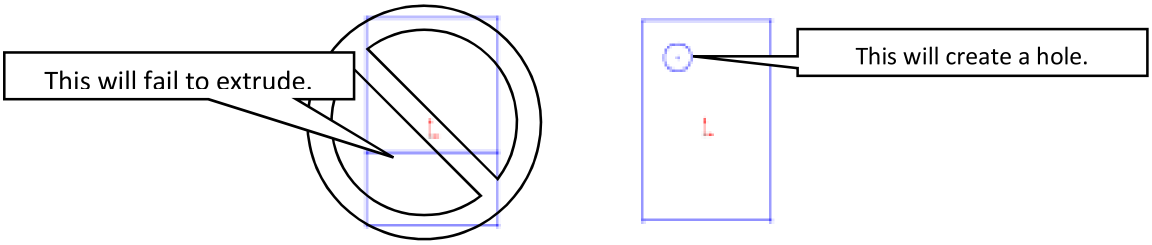


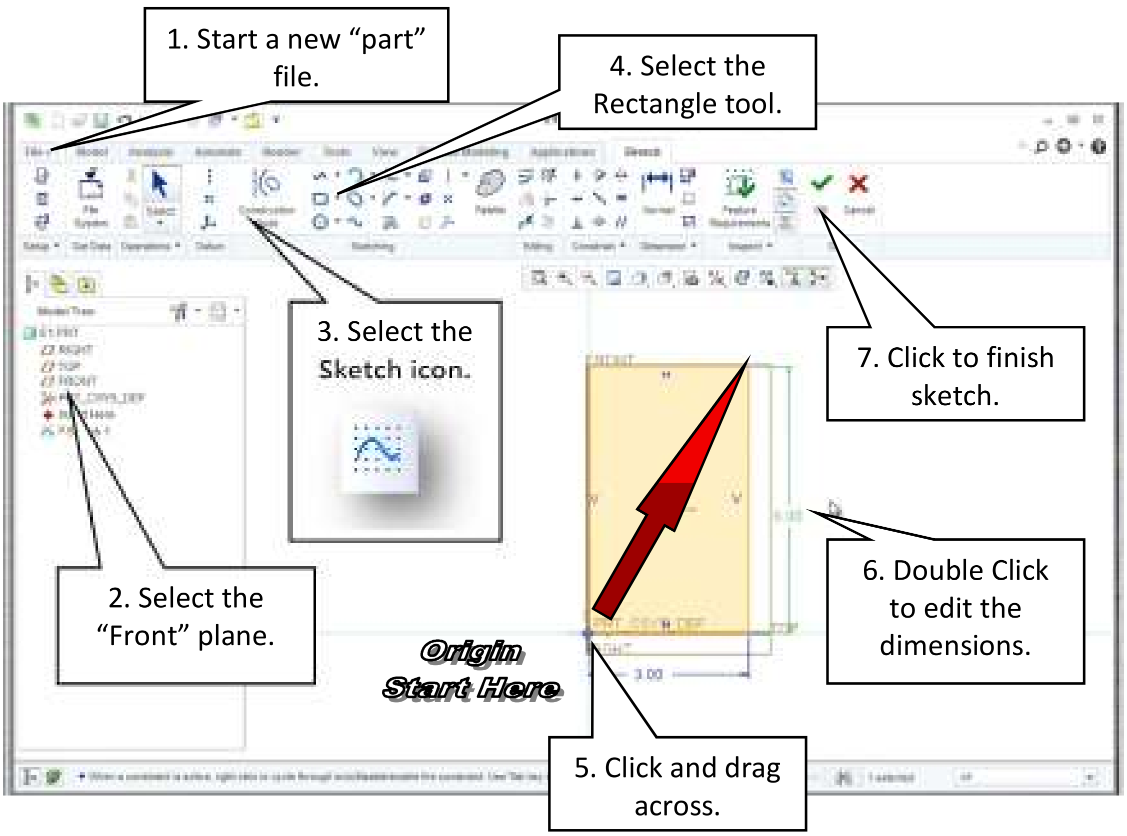
EXERCISE 1

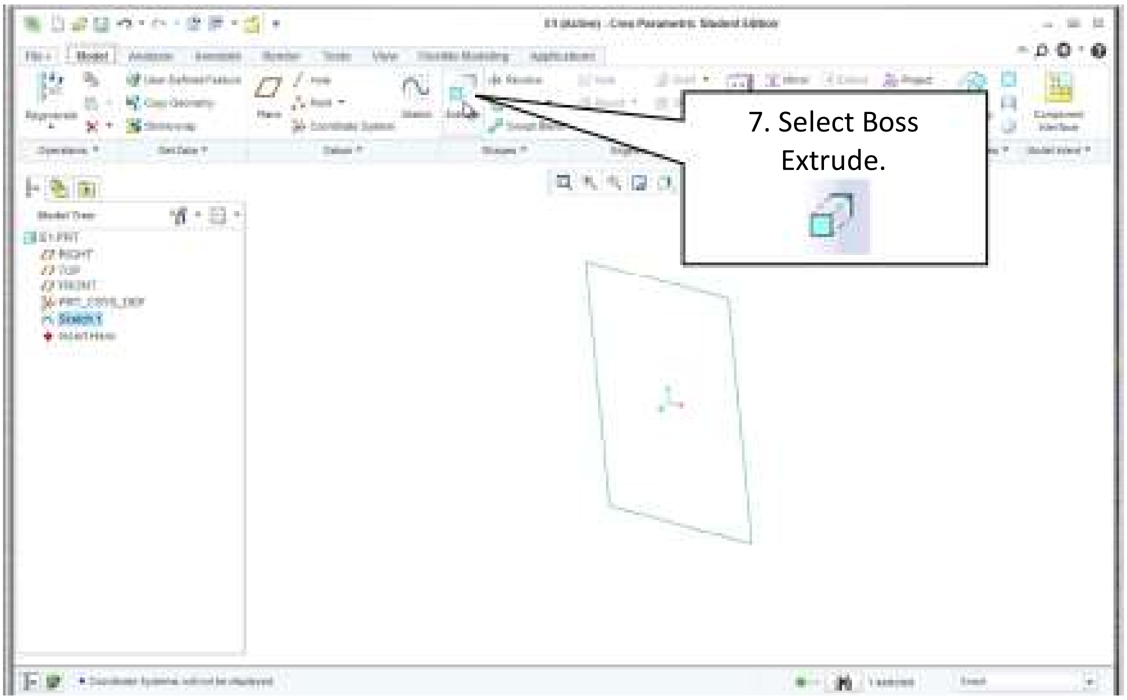
Introduction to basic part modeling

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

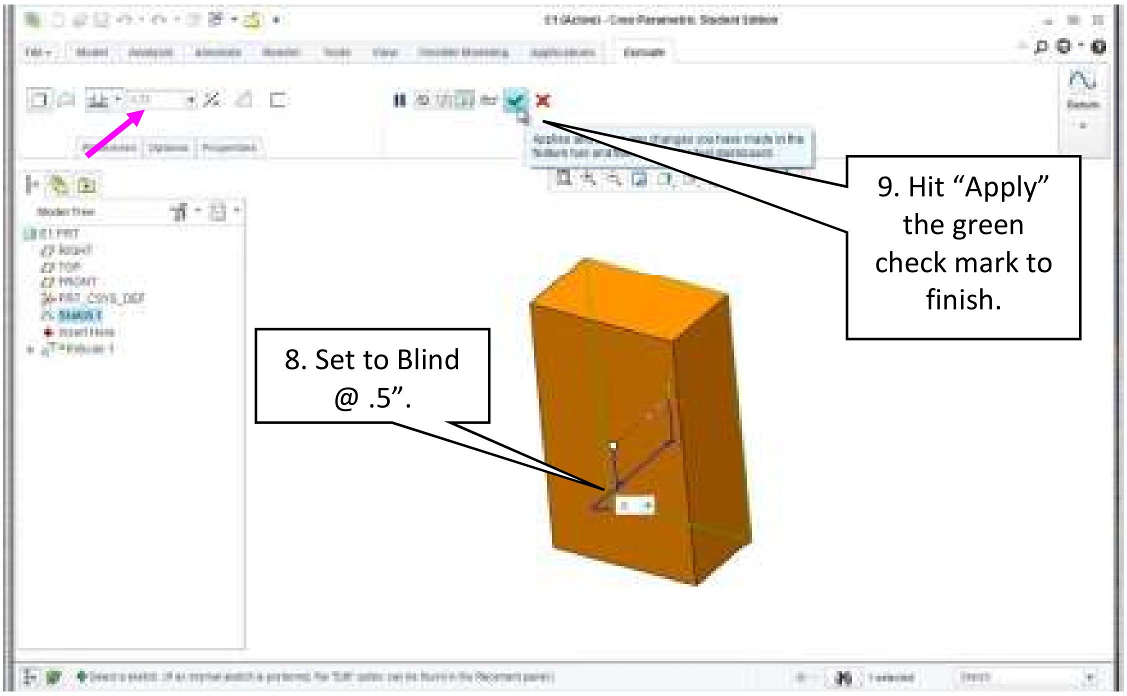


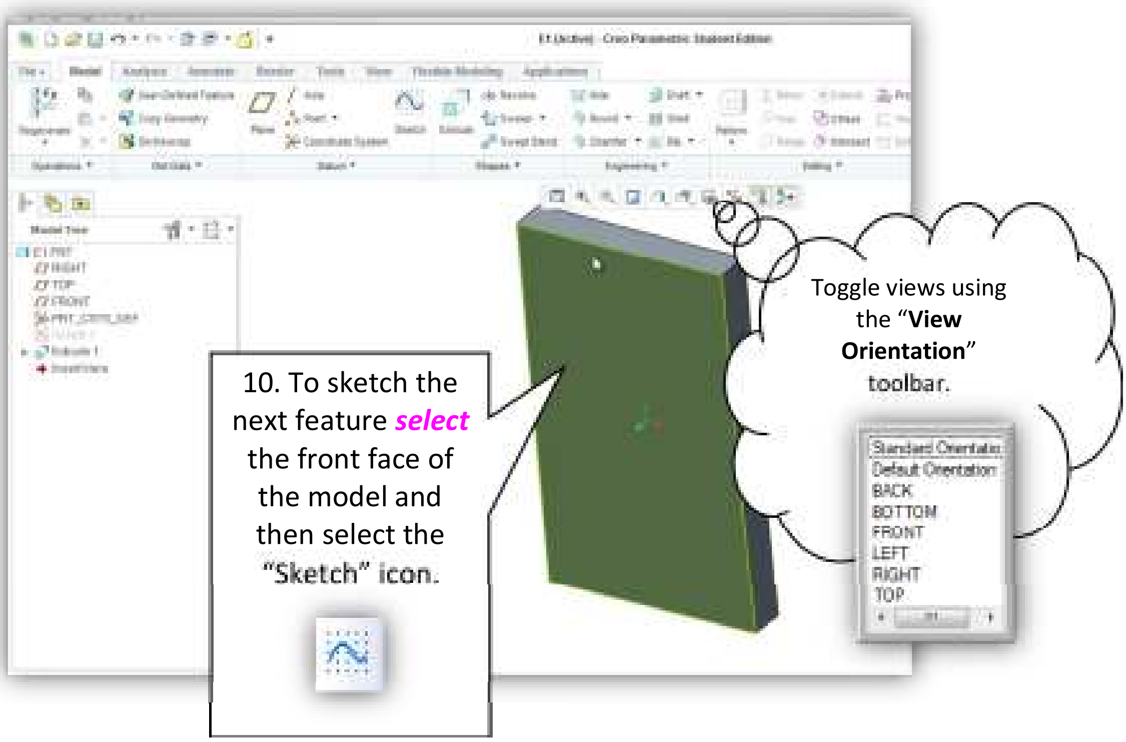




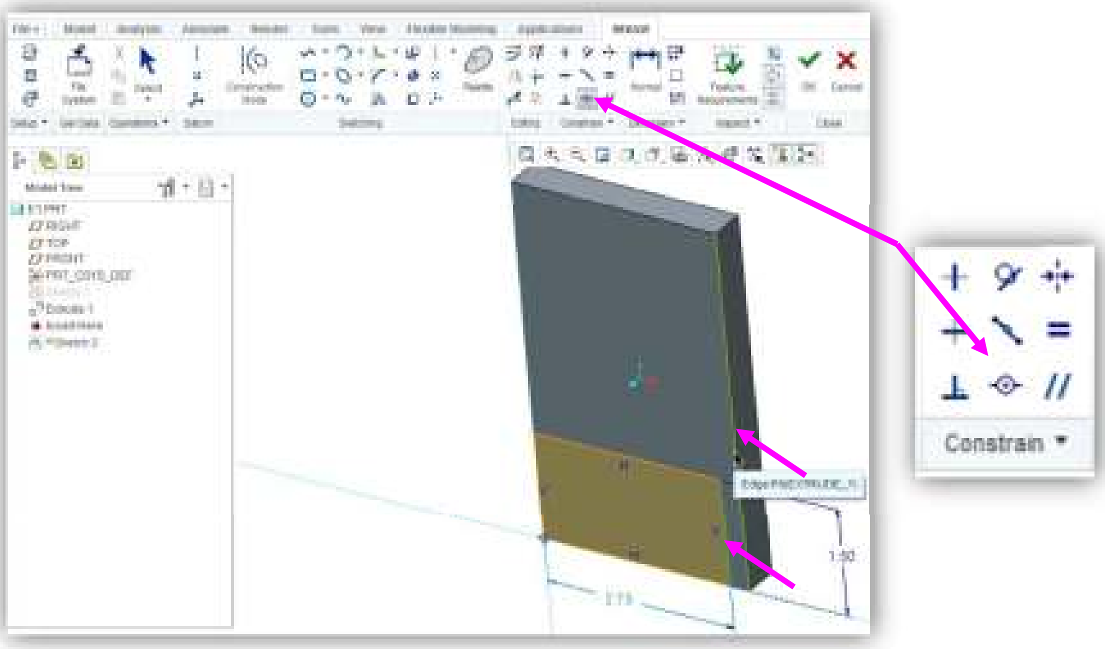


NOTE: When dimensioning use the dimension tool and make edge selections, mouse center button click to apply dimension.

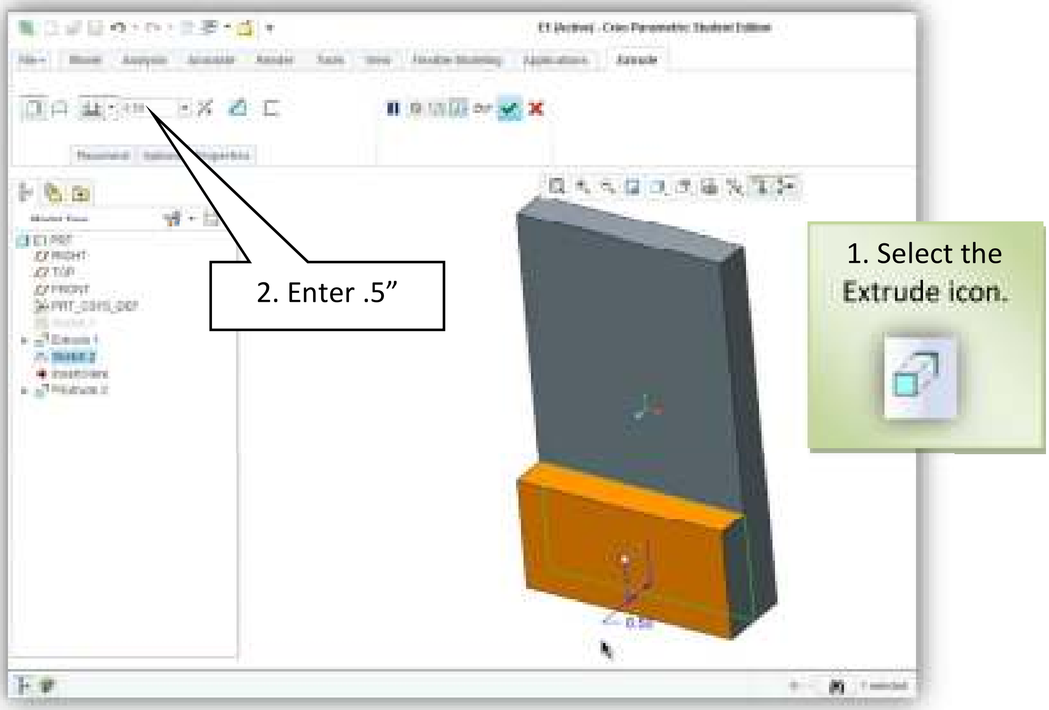




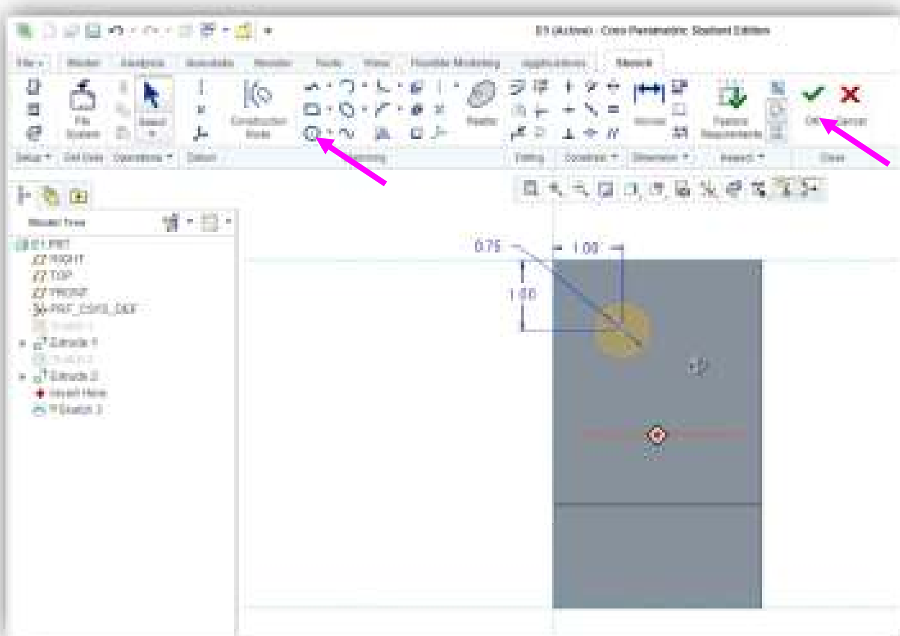
Adding a constraint – Ctrl Select both left edges of sketch and solid. Select Coincident



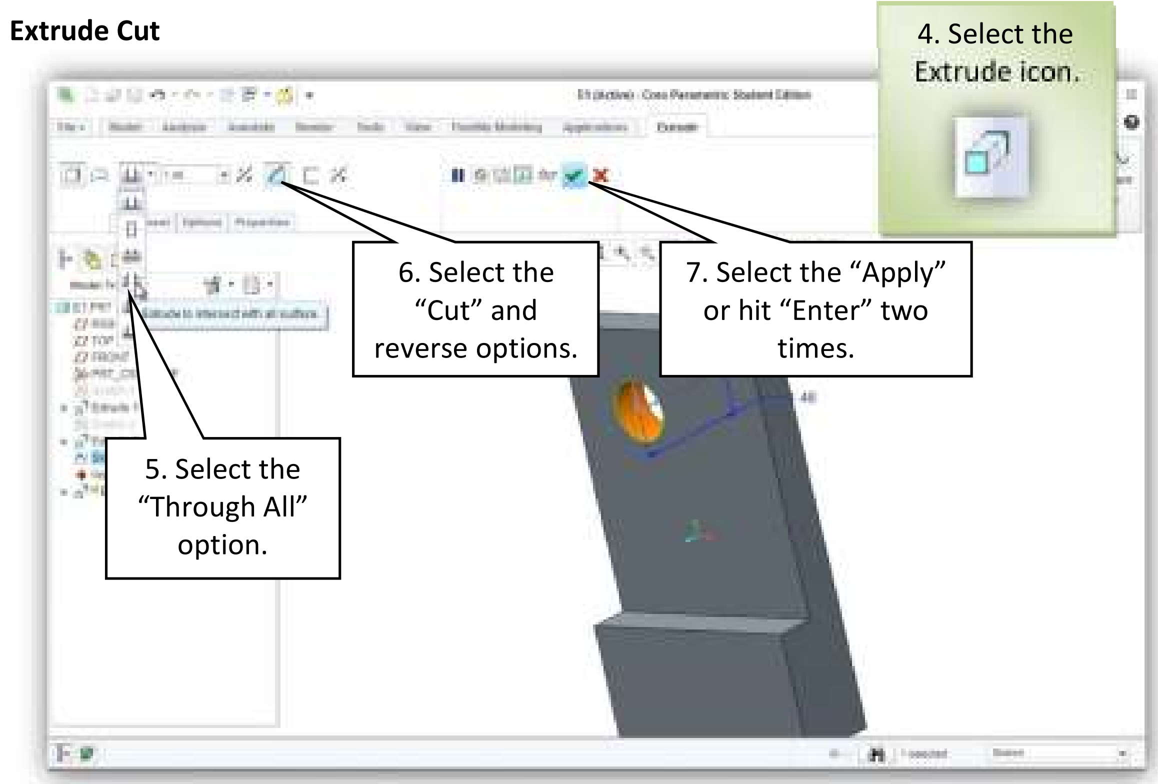
Extrude



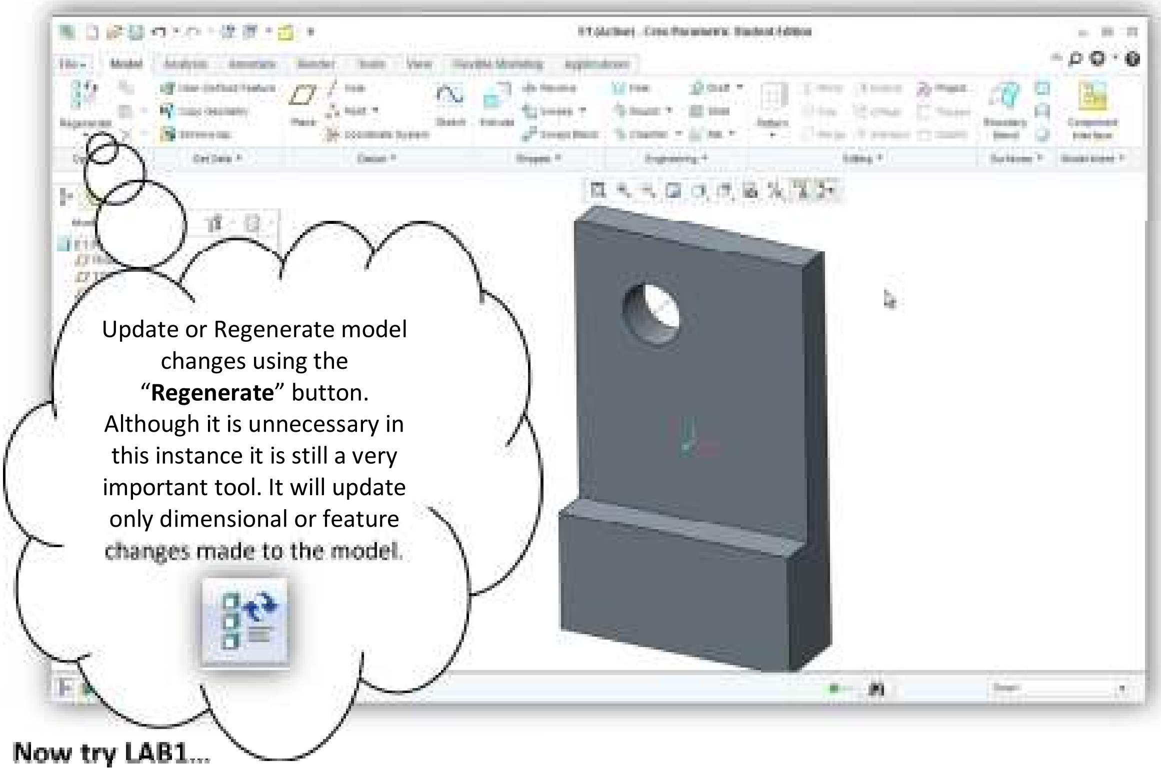
Select the face, select sketch icon and draw a circle on the face. Dimension, Hit “Ok”



Extrude Cut

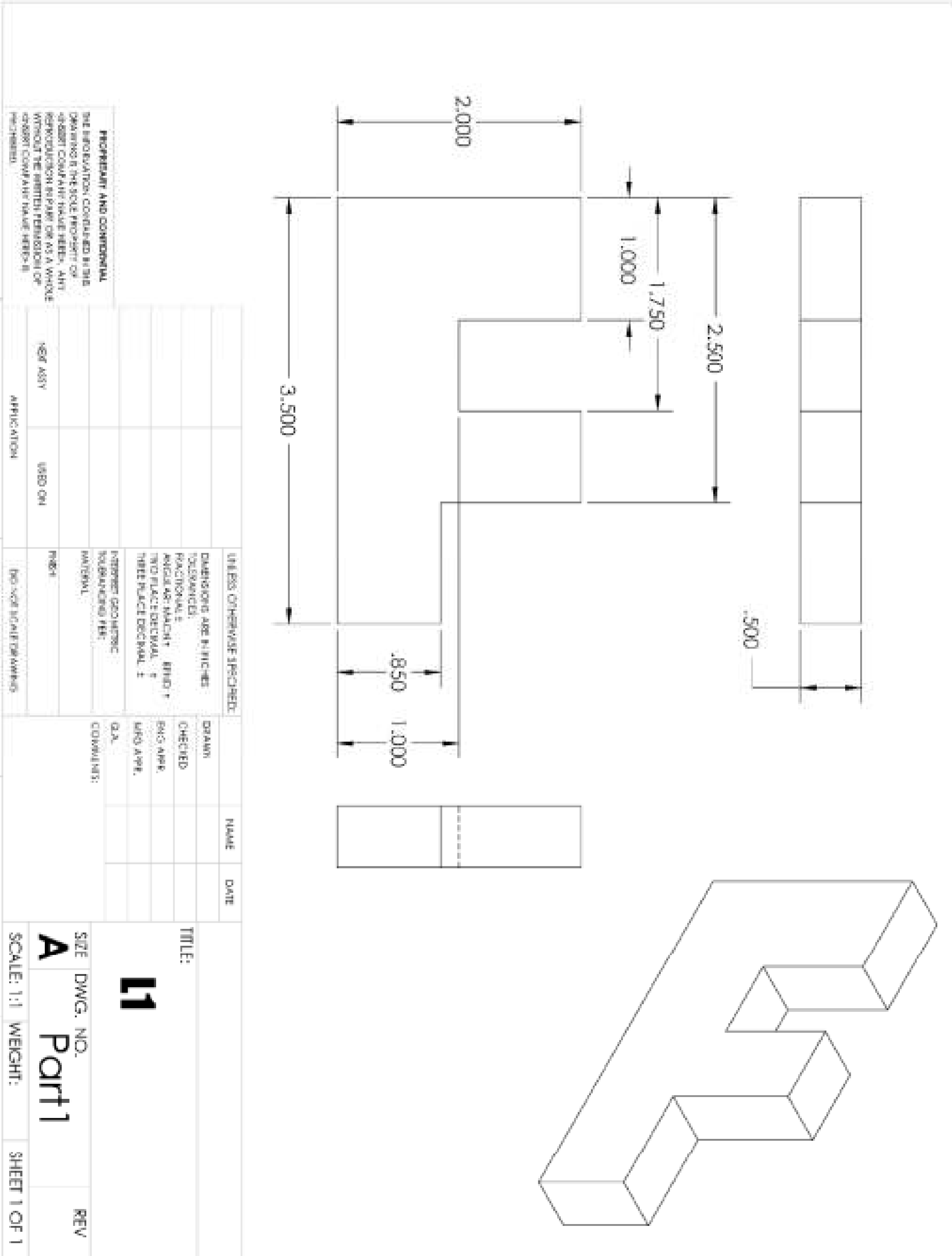


Go to file save and save-as "E1"

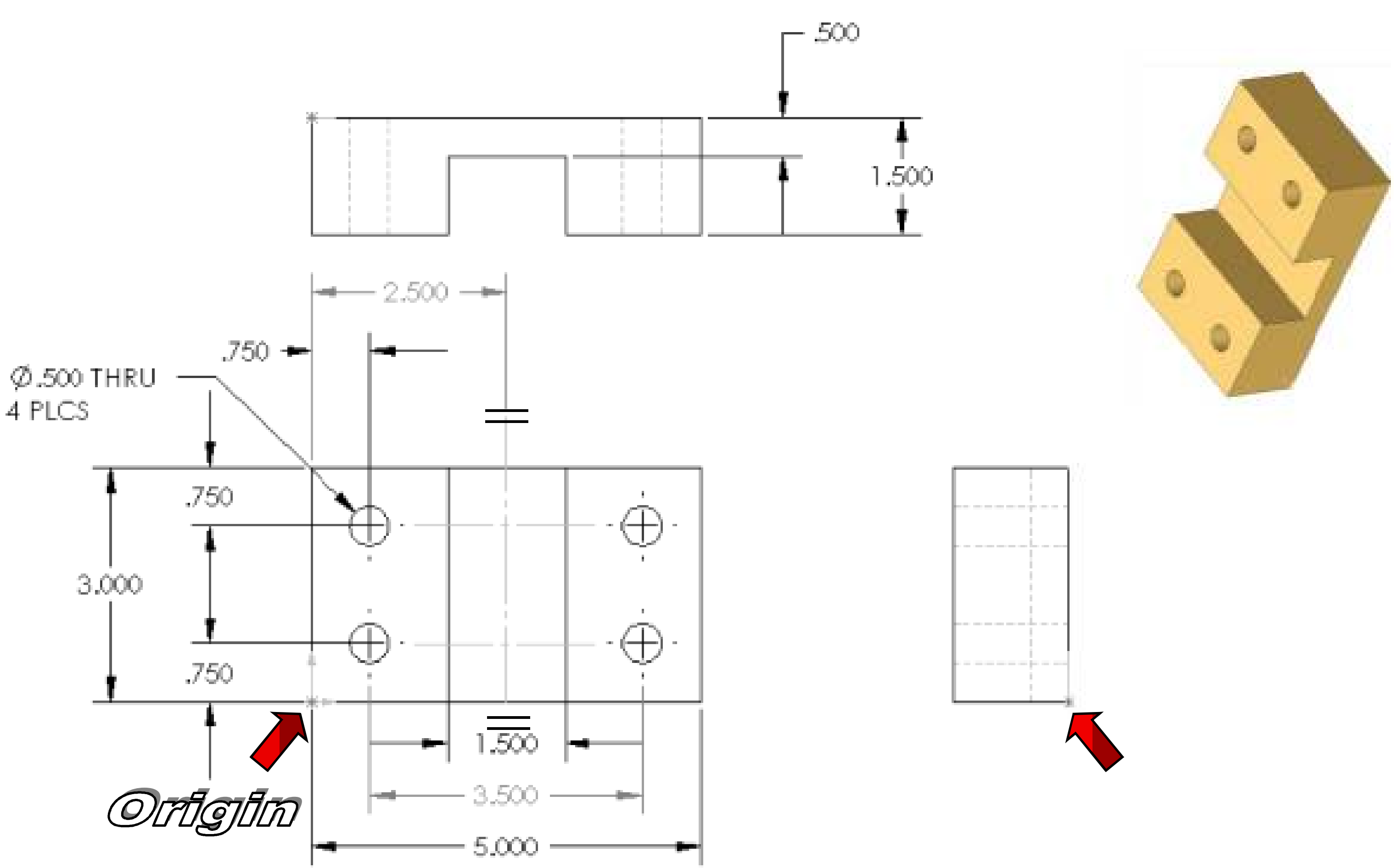


Now try LAB1…

NOTE: Patterns/Arrays and Mirroring will be covered in the next three chapters. Please try to model LAB 1 without using them. It's good practice to just dimension and sketch all geometry when first starting out learning this software.



2.000
1.000
1.750
2.500
3.500
.850
1.000
.500
TITLE:
L1
SIZE
A
DWG. NO.
Part1
REV
SCALE: 1:1
WEIGHT:
SHEET 1 OF 1



LAB 1B

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

PROPRIETARY AND CONFIDENTIAL

NEXT ASSY | USED ON

APPLICATION

DO NOT SCALE DRAWING

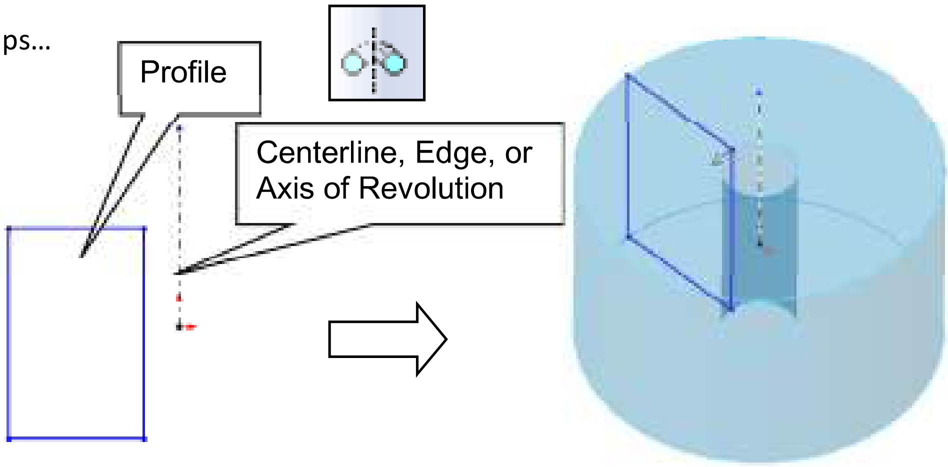
SIZE A

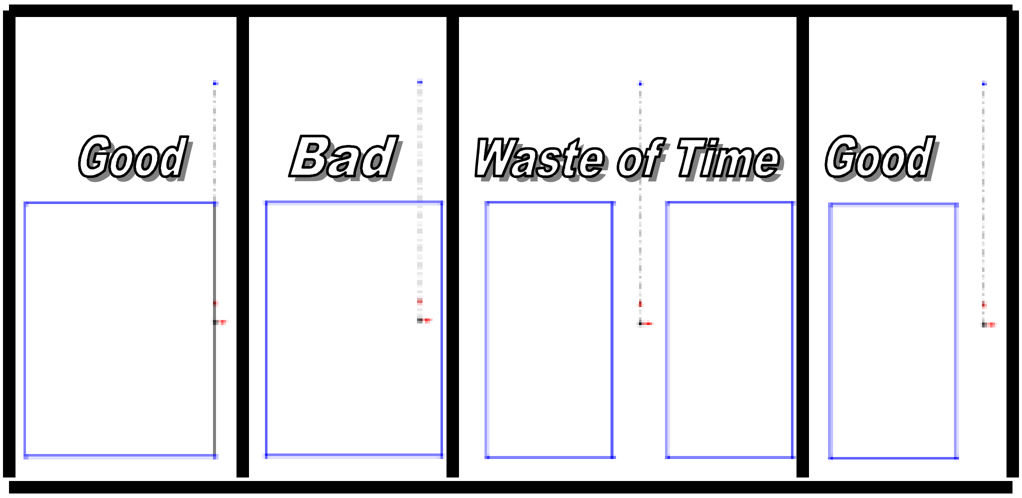
EXERCISE 2

Revolved Features

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

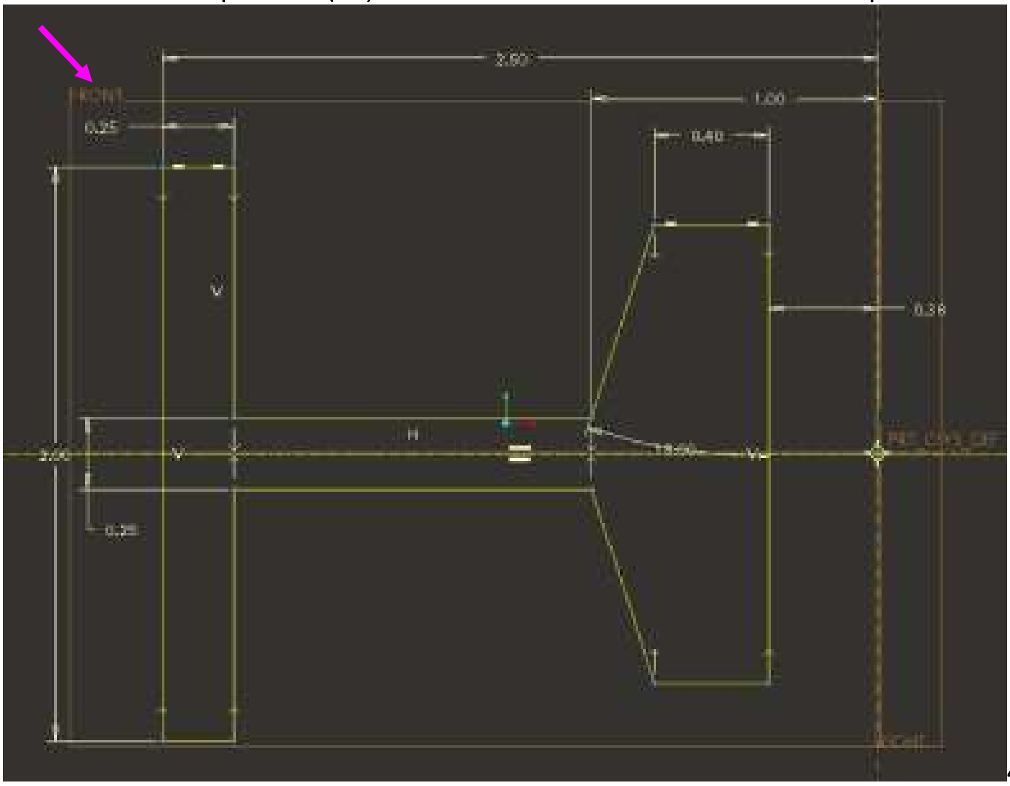
Tips…





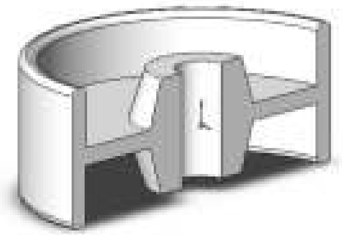
The profile should never cross over the centerline, nor should there be profiles on both sides of the centerline.

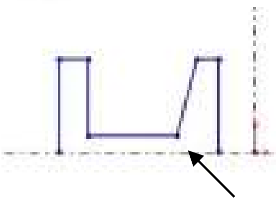
1. Create a new part file (E2) and then start a sketch on the “Front” plane.

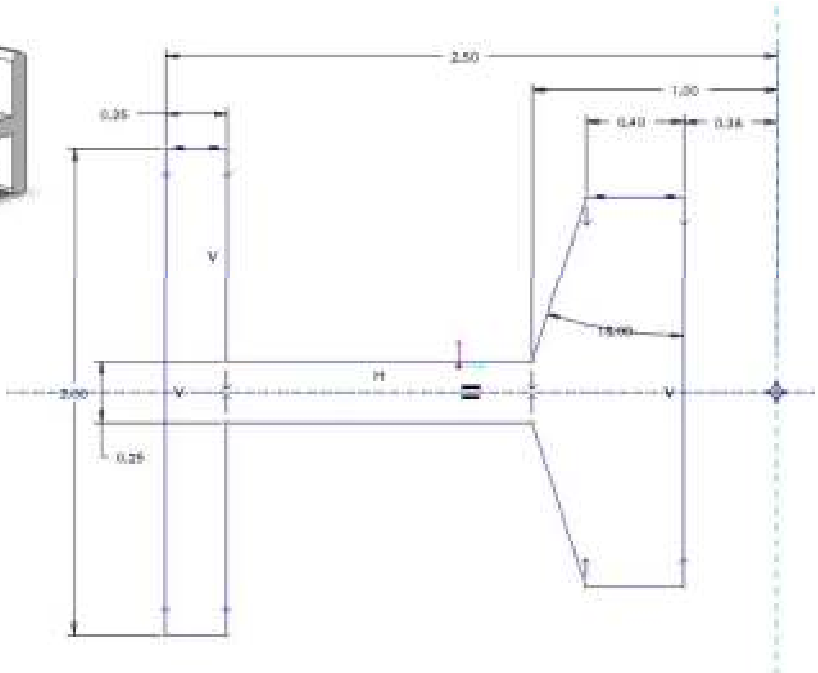


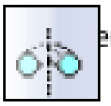
4

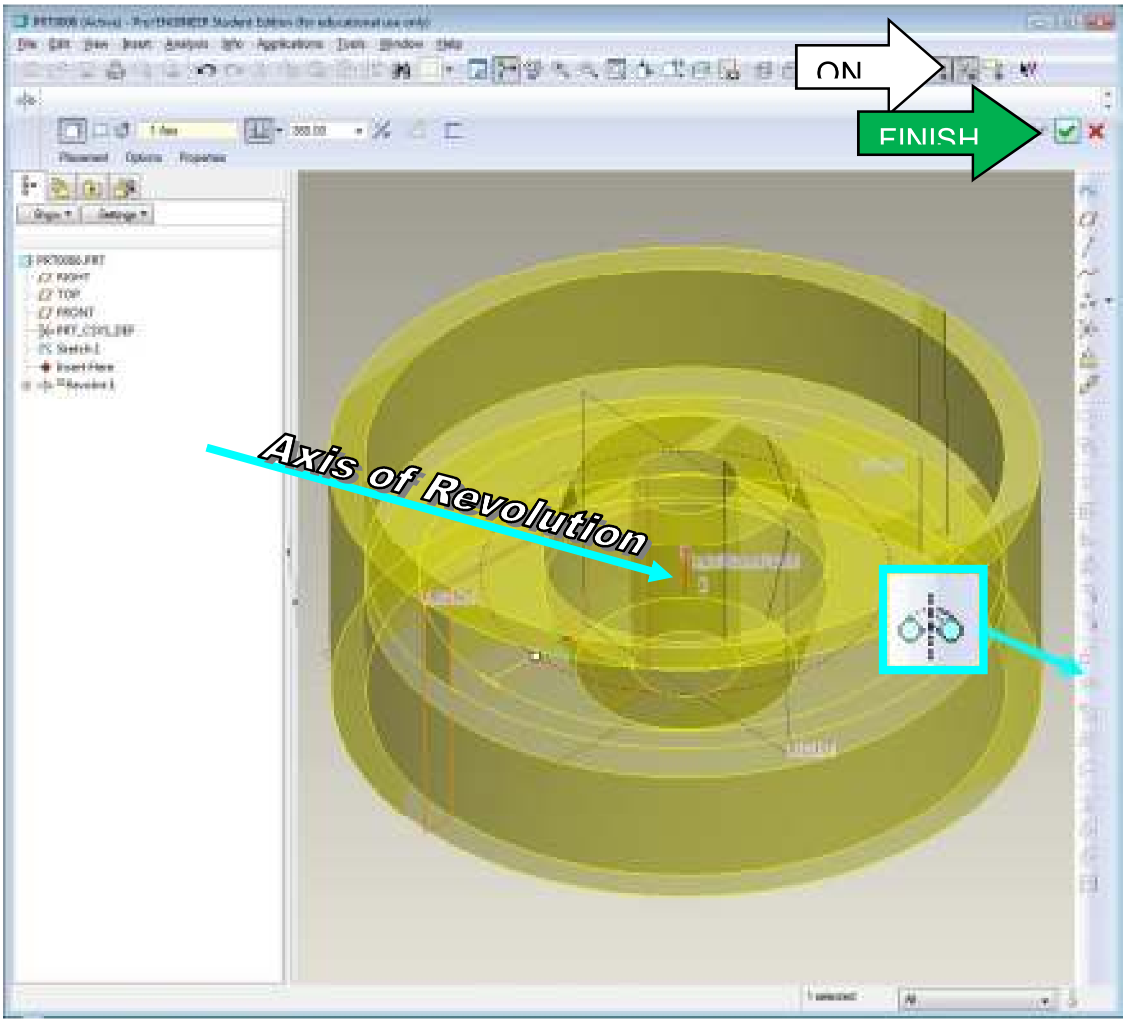
2. Sketch the following. Ctrl select the profile and the horizontal centerline, then using the “Mirror” tool to create a ¼ of the geometry and then mirror it to the other side. Make sure you finish adding the dimensions.



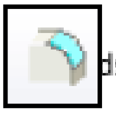


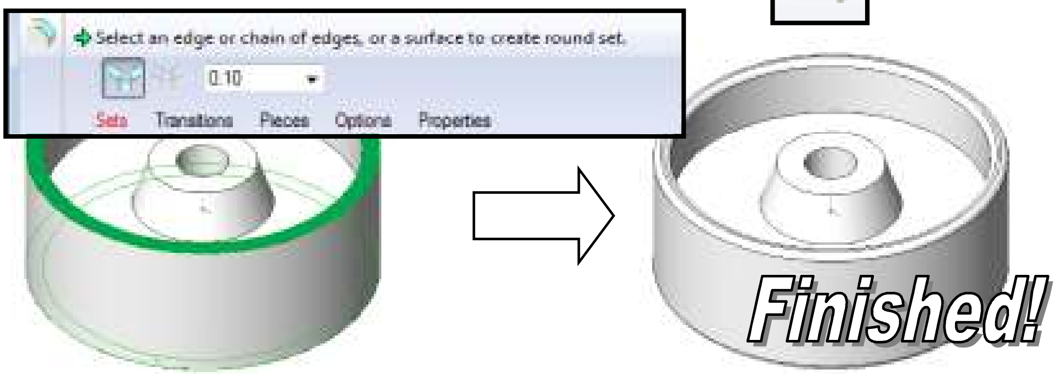


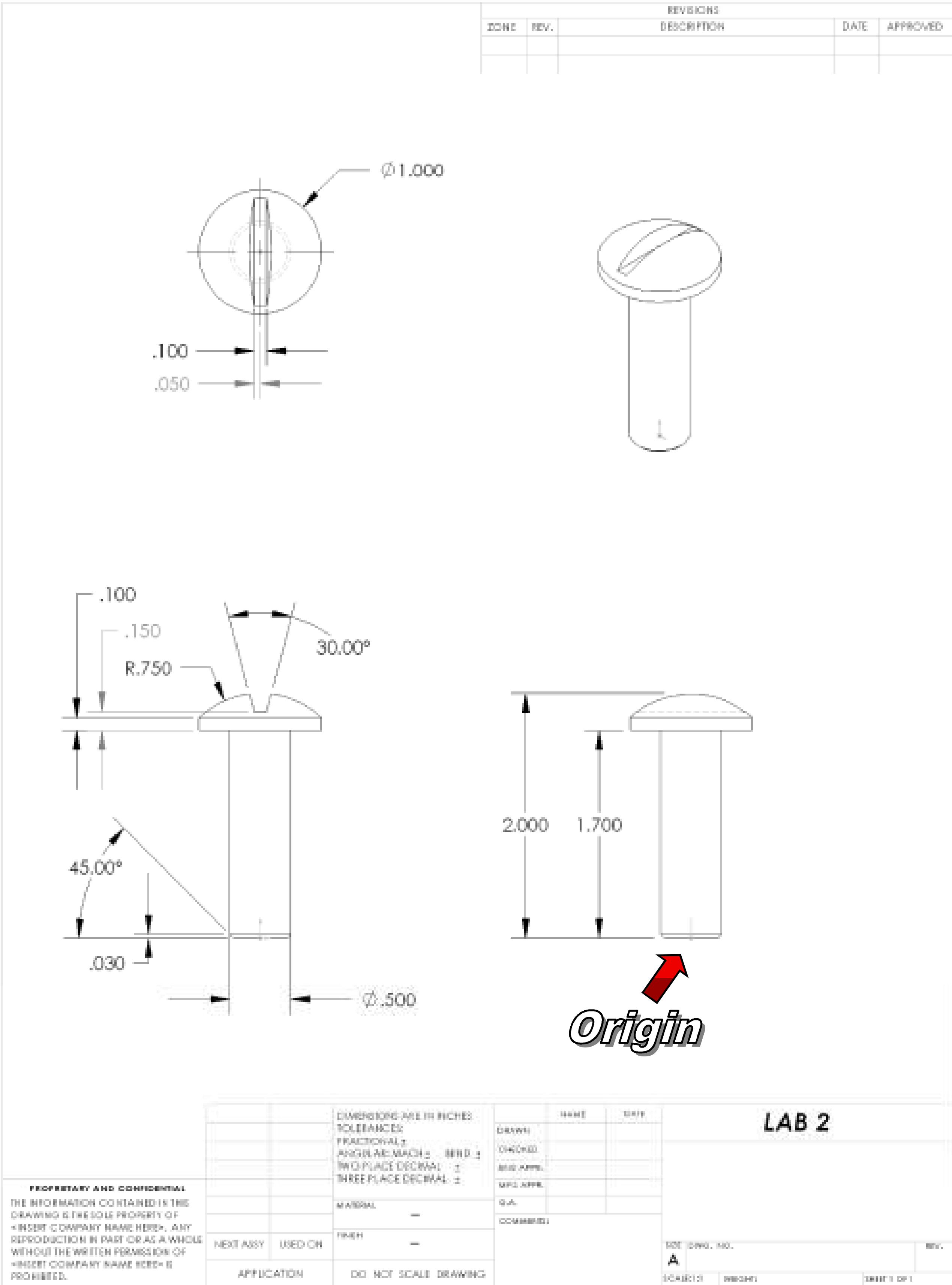
3. Select the **Revolve** feature icon.  en select the axis/centerline.

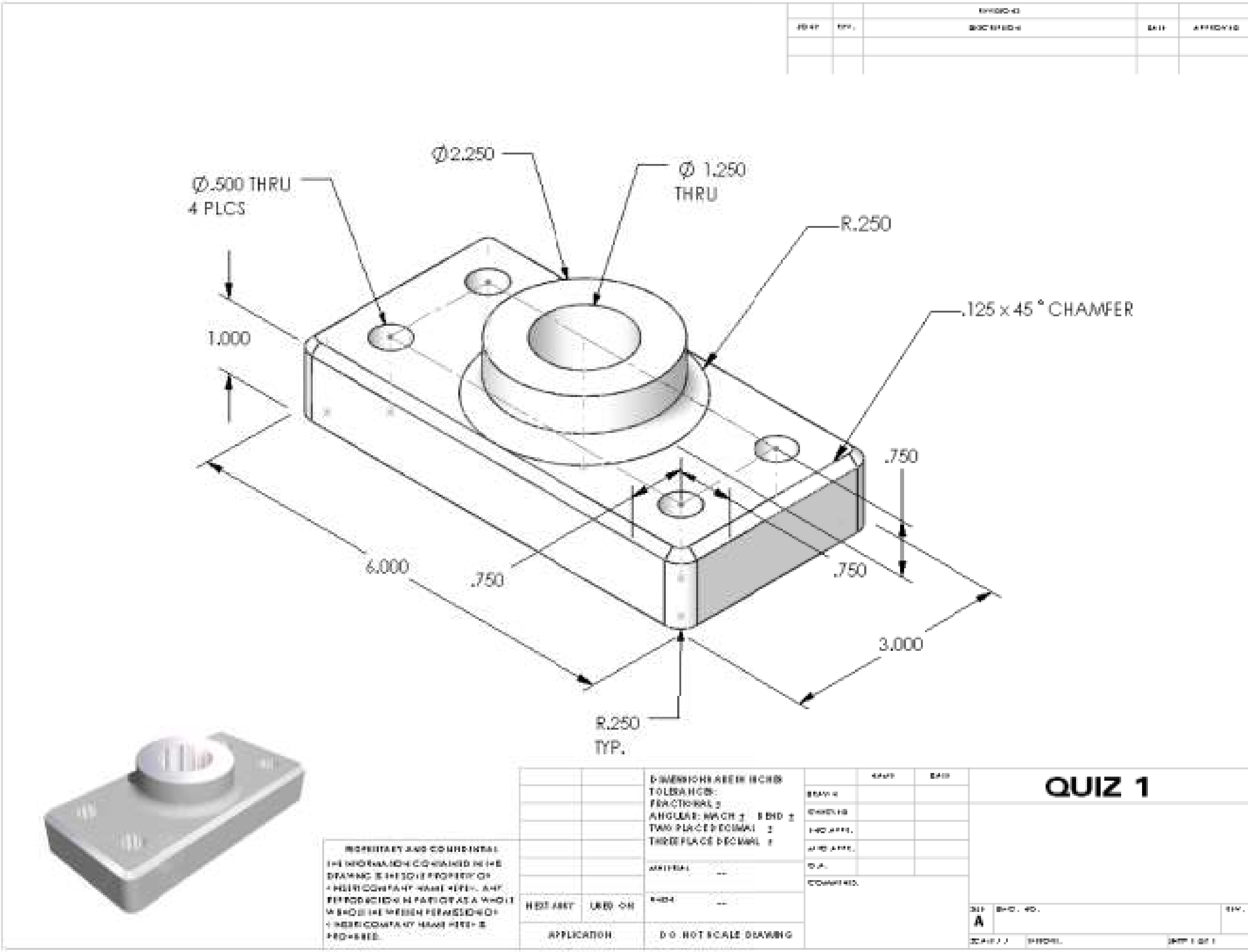


Rounds

4. Select the top and bottom edges and add a R.100”  ds/fillets.



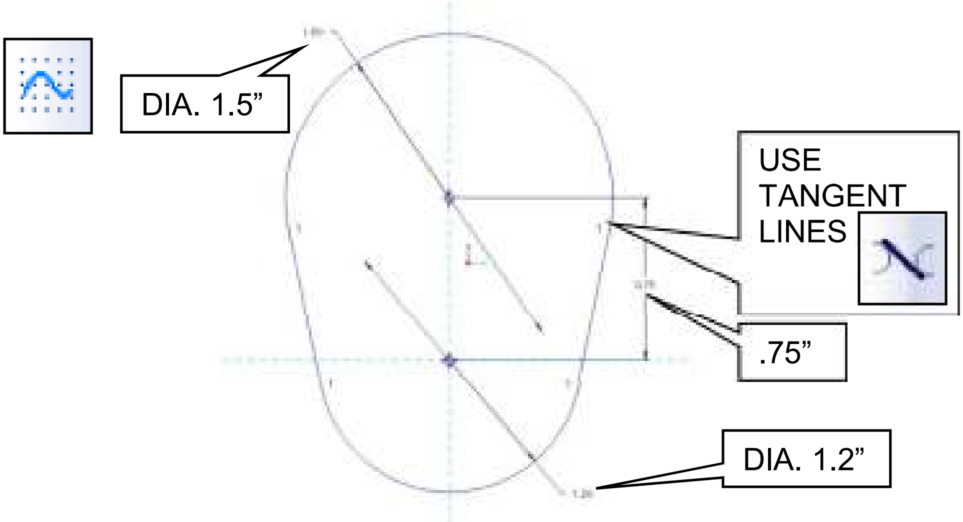

REVISIONS
ZONE
REV.
DESCRIPTION
DATE
APPROVED
Ø1.000
.100
.050
.100
.150
R.750
30.00°
45.00°
.030
Ø.500
2.000
1.700
Origin
DIMENSIONS ARE IN INCHES
TOLERANCES:
FRACTIONAL ±
ANGULAR: MACH ± BEND ±
TWO PLACE DECIMAL ±
THREE PLACE DECIMAL ±
MATERIAL
FINISH
NEXT ASSY
USED ON
APPLICATION
DO NOT SCALE DRAWING
NAME
DATE
DRAWN
CHECKED
ENG APPR.
MFG APPR.
Q.A.
COMMENTS:
LAB 2
SIZE
DWG. NO.
REV.
A
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.


Ø.500 THRU
4 PLCS
Ø2.250
Ø 1.250
THRU
R.250
.125 x 45 ° CHAMFER
1.000
.750
6.000
.750
.750
3.000
R.250
TYP.
QUIZ 1
NEXT ASSY
USED ON
APPLICATION
DO NOT SCALE DRAWING
A

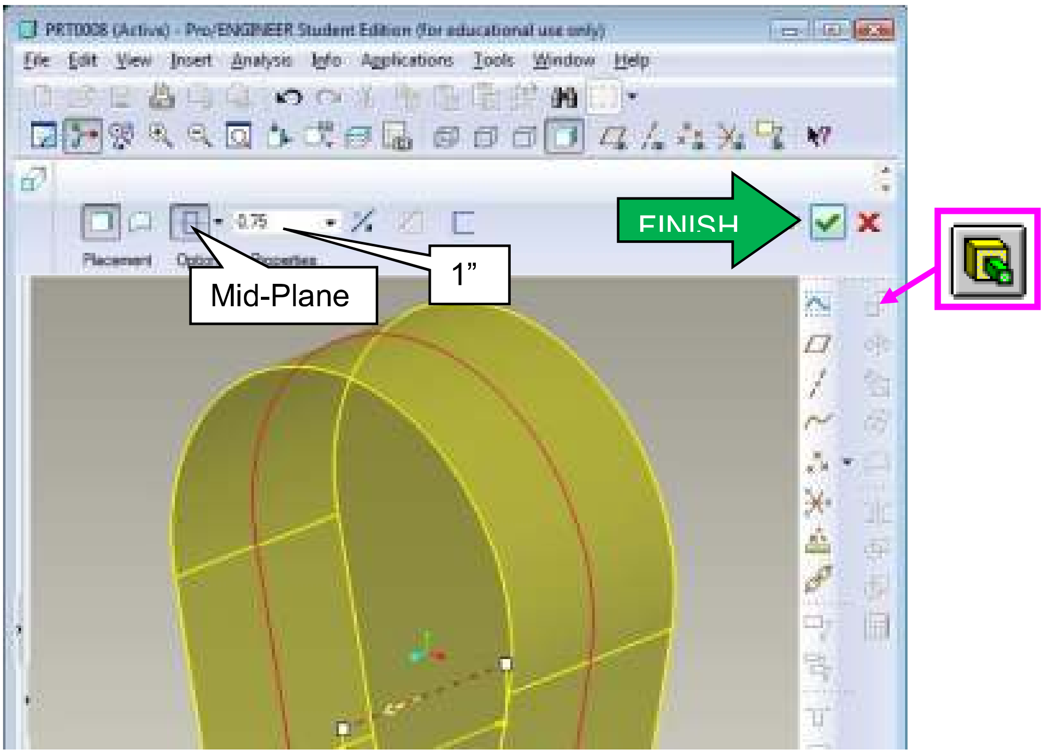
EXERCISE 3

Secondary Feature Modeling

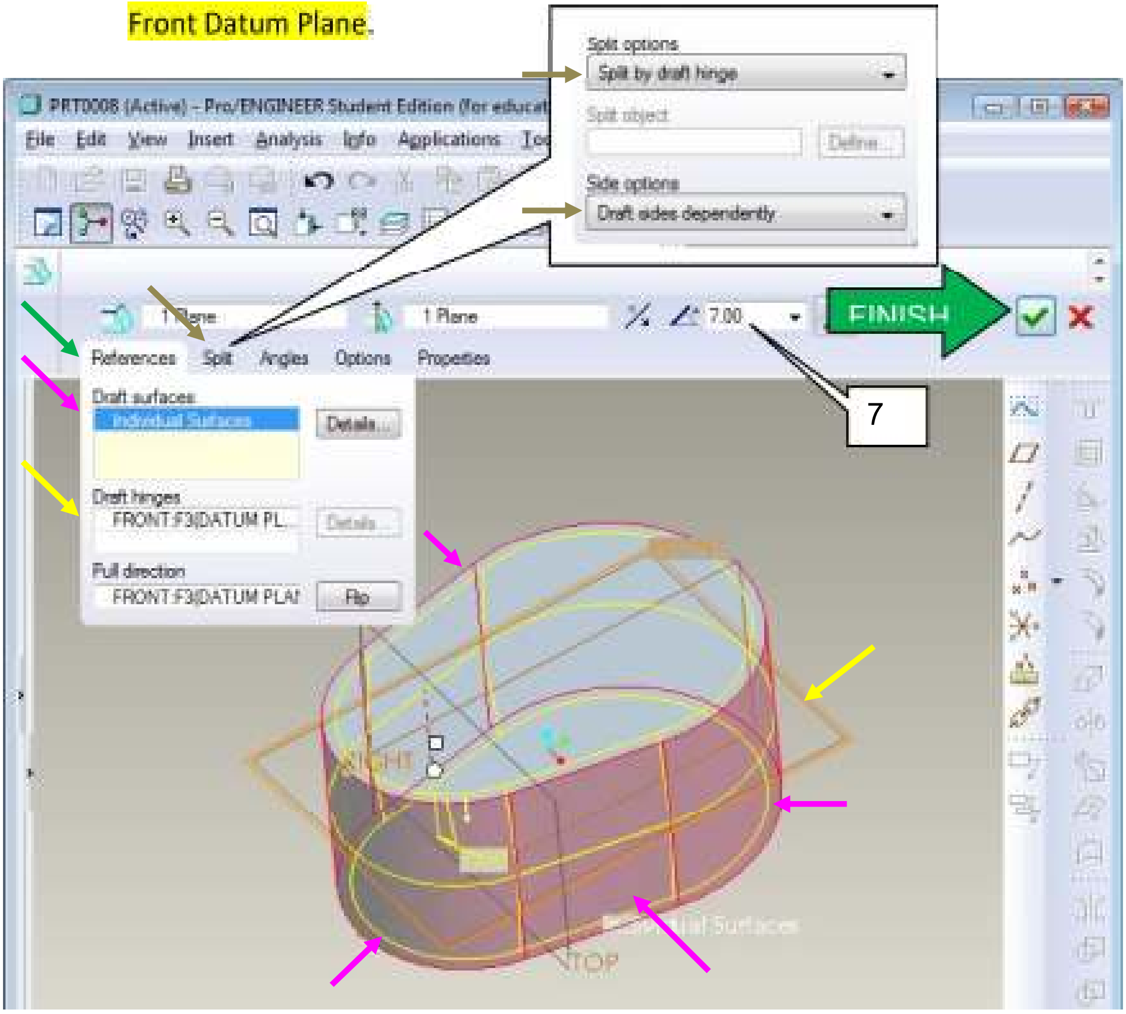
1. Sketch the geometry as show below on the "Front" plane.



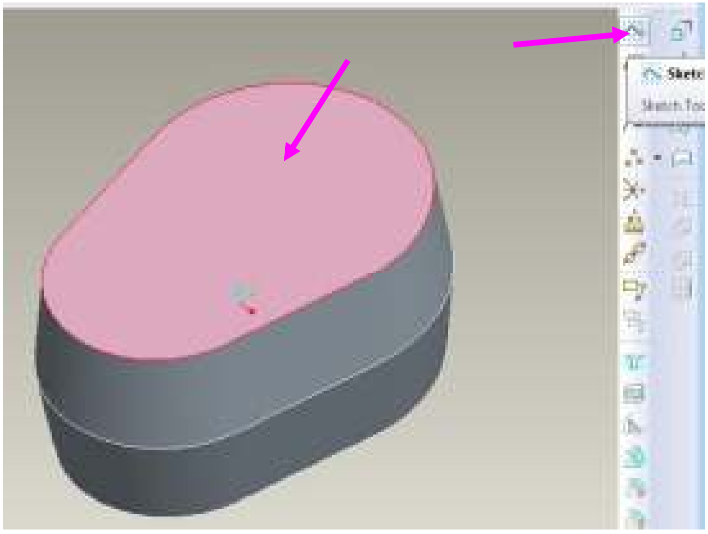
2. **Extrude**. Select Mid-Plane, 1".



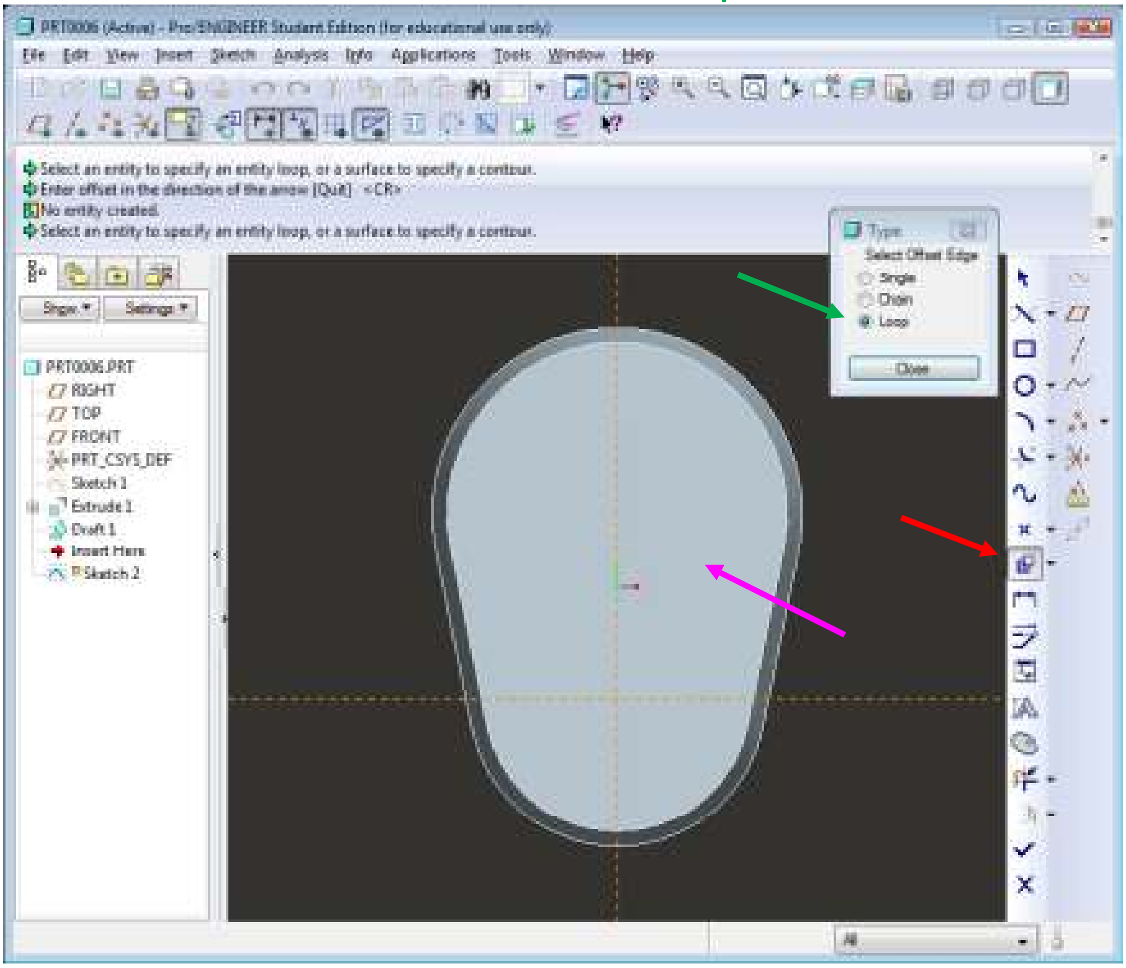
3. **DRAFT**: Select the Draft tool, and then References, Ctrl select all side faces of the model. Then Click on the draft hinges dialog box, and select the Front Datum Plane.



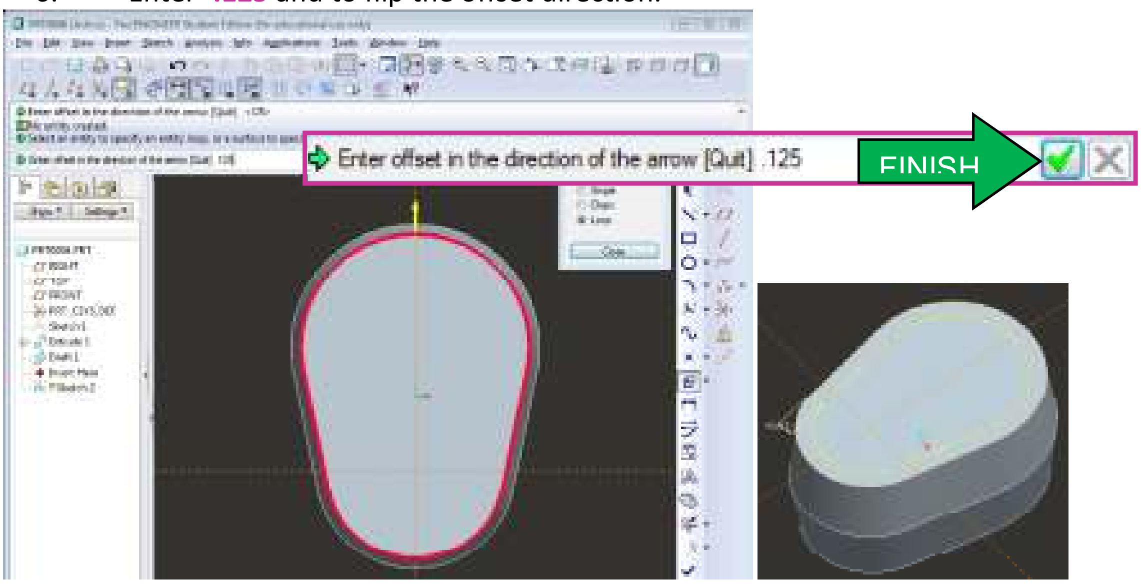
4. Select the top surface (LMB Click 2 x) on the model and start a sketch on it.



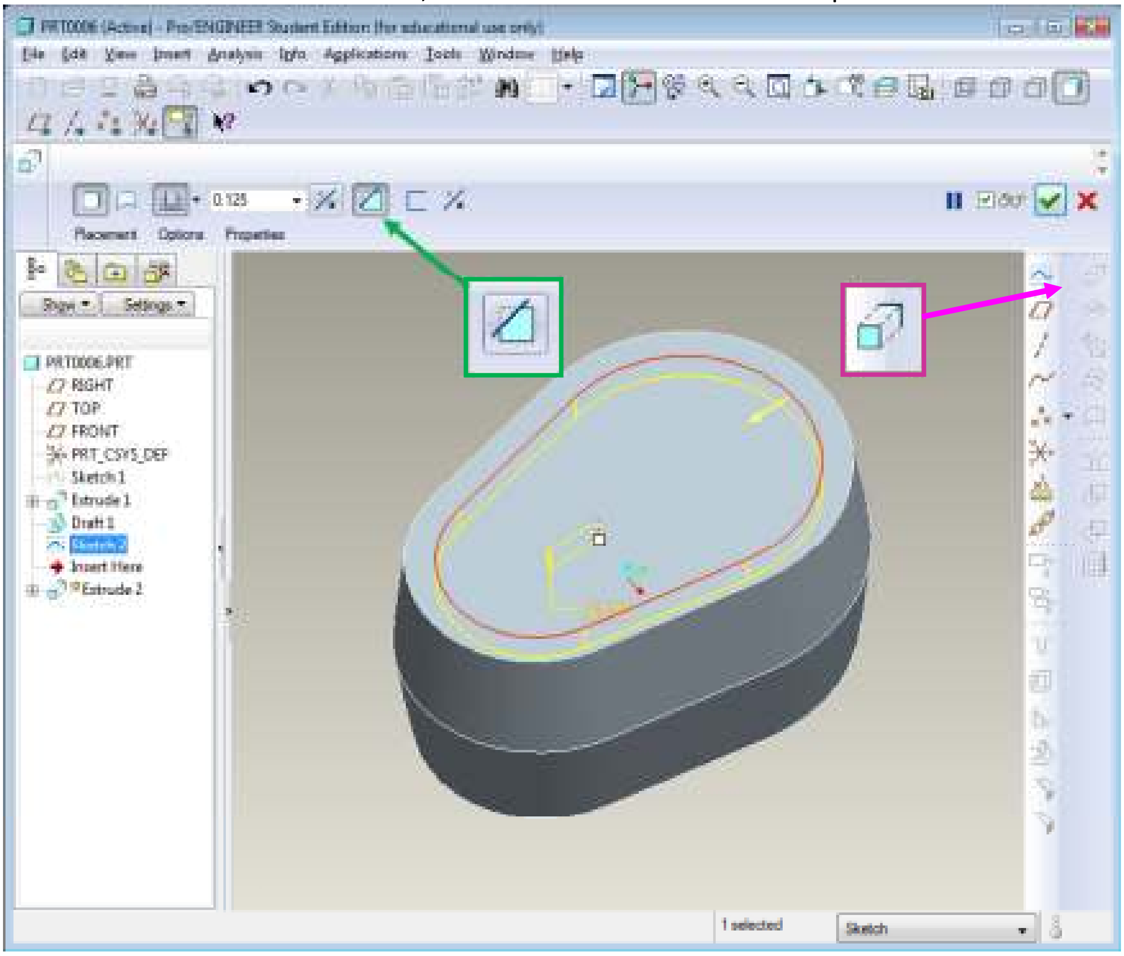
5. **OFFSET:** Select the **Offset** tool. Then select **Loop**. Then select the **face**.



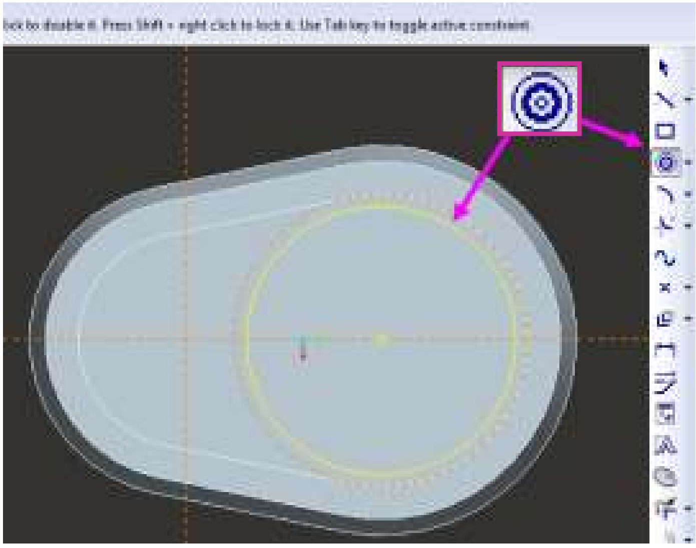
6. Enter **-.125** and to flip the offset direction.



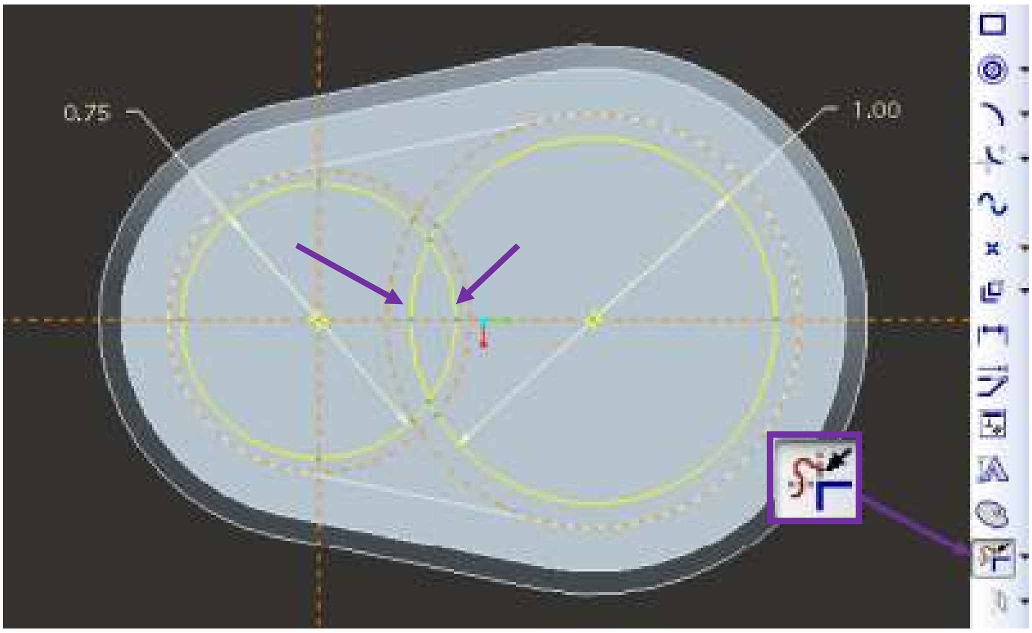
7. Select the **extrude** icon, and then set to **cut** and .125 depth.



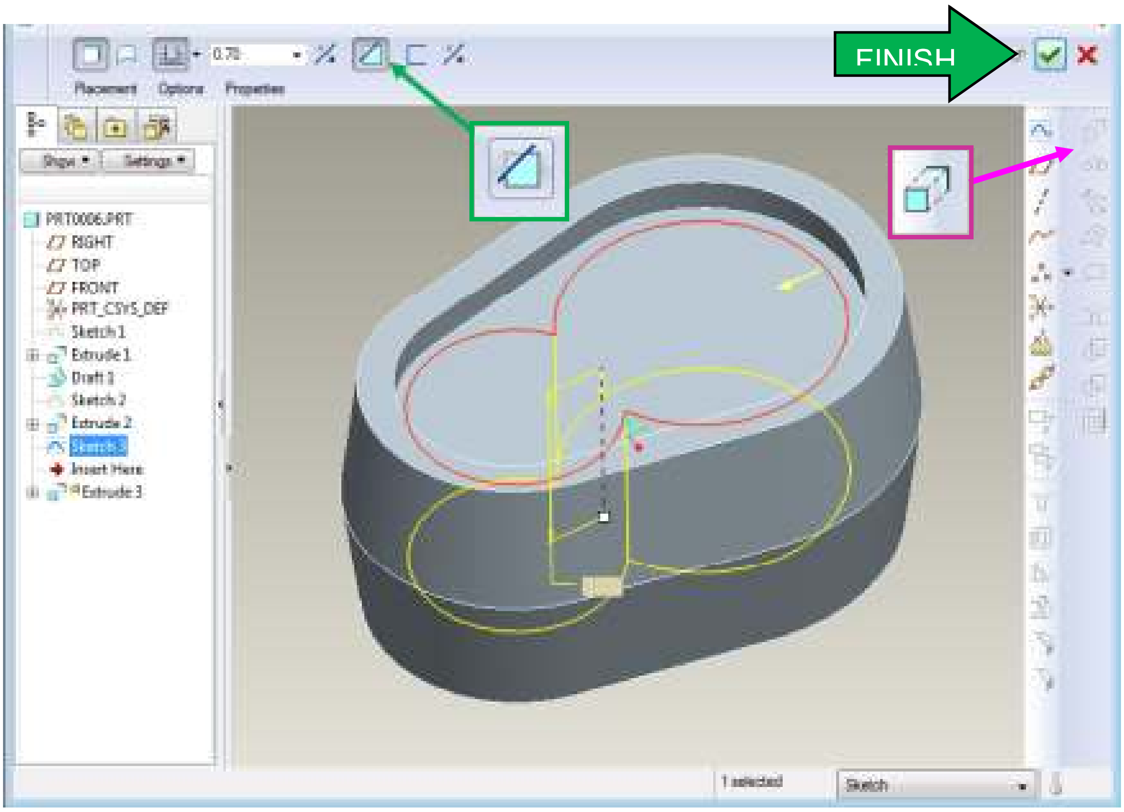
7. Select Concentric (Circle tool), then select the arc edge of the part.



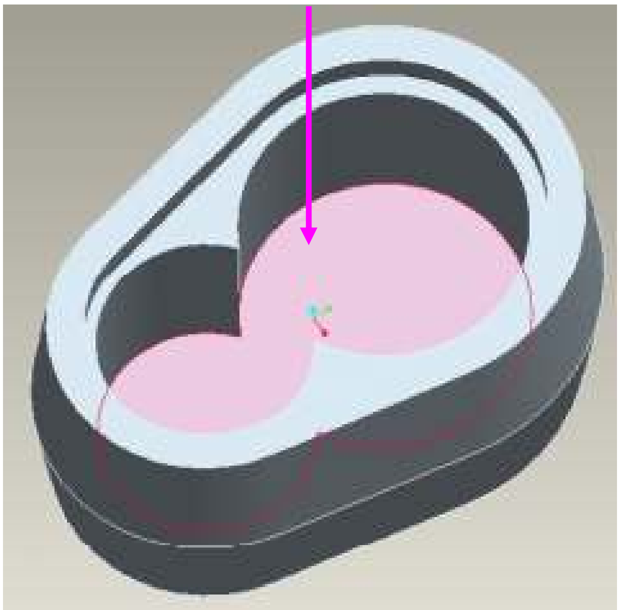
9. **Trim** the intersection.

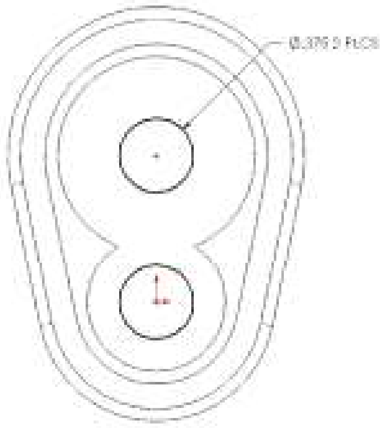


10. Select the extrude icon, and cut .700" depth.

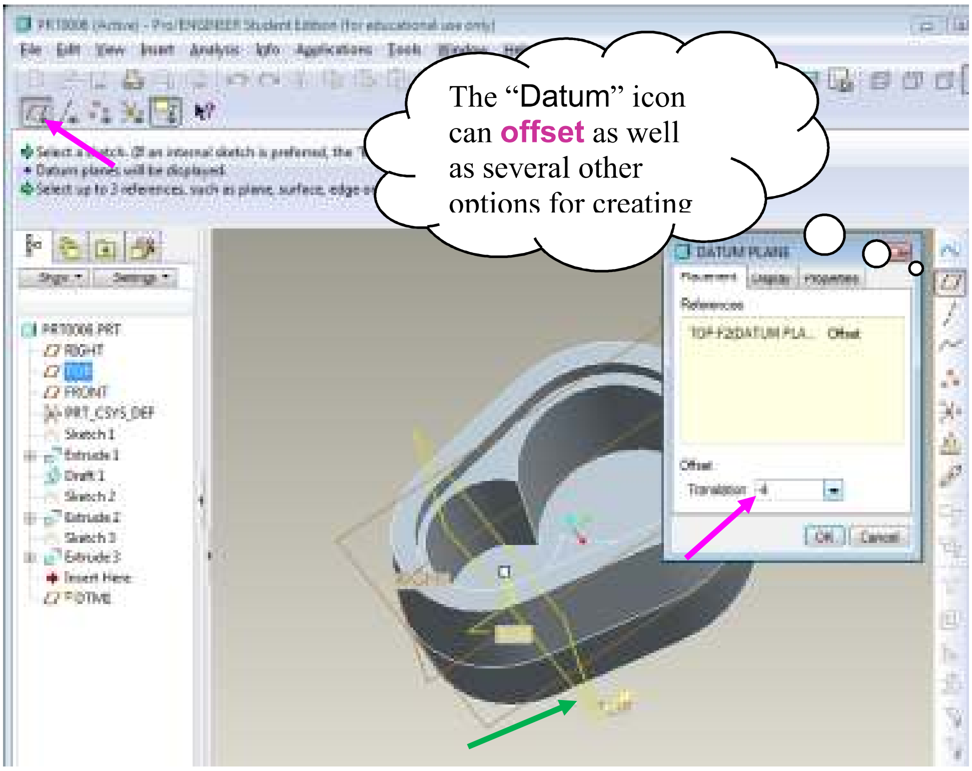


11. Select the base of the pocket and start a sketch. Draw the following two .375 DIA. circles, and extrude / cut "Through-all".

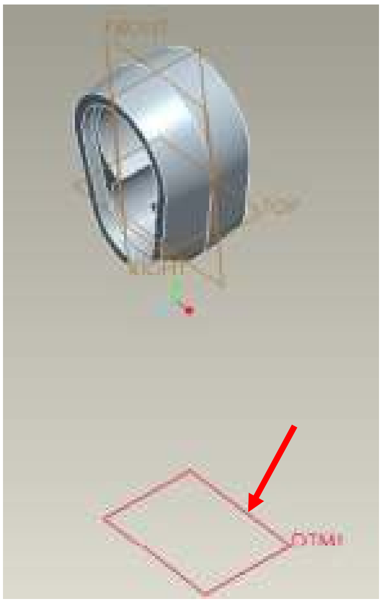




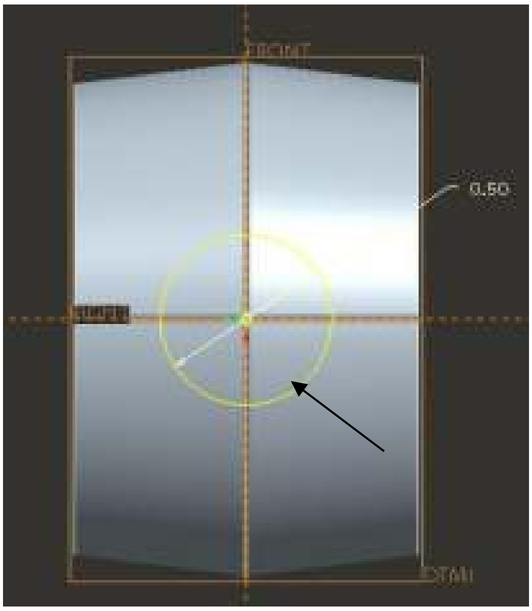
12. **DATUM PLANE OFFSET:** Select the Top datum plane, then select the Datum icon. Set to -4 offset.



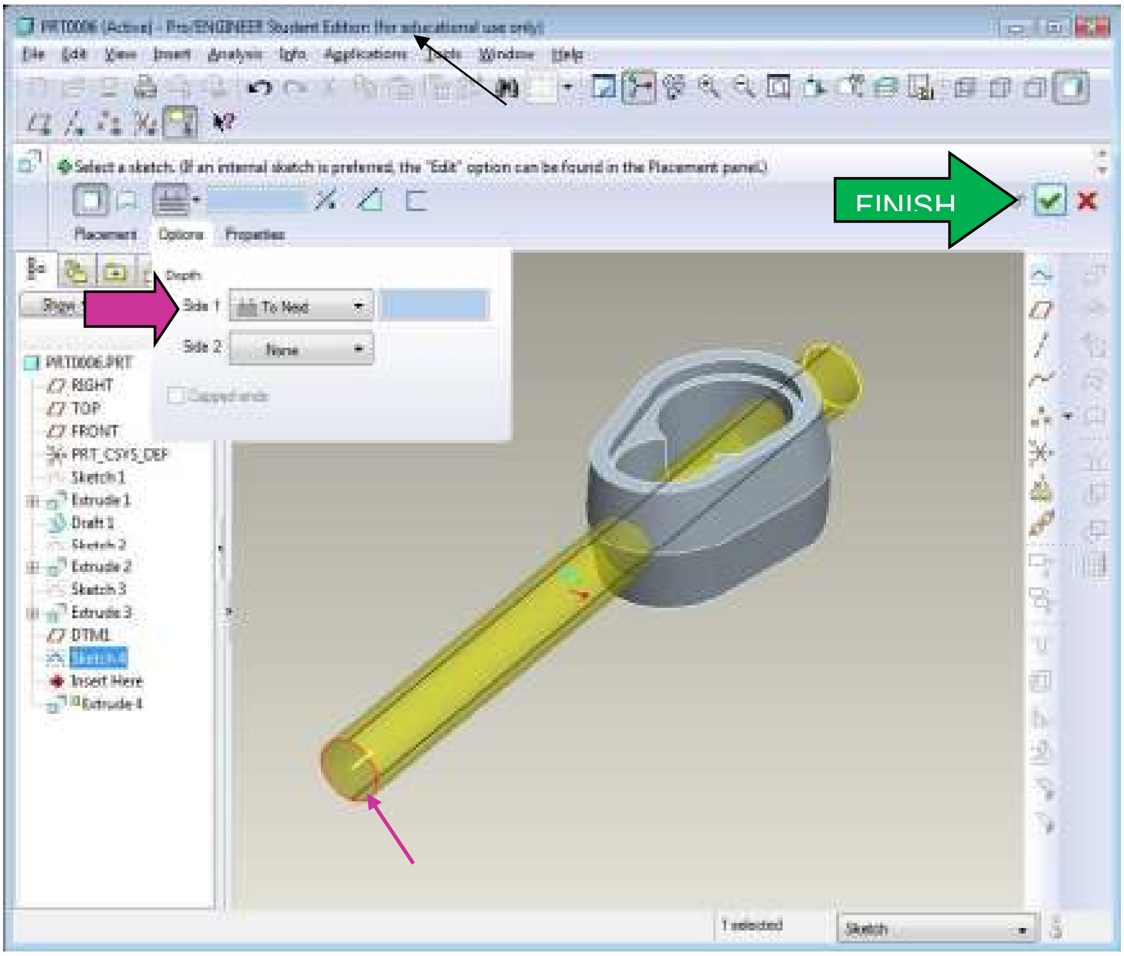
13. Start a sketch on "**DTM 1**" and draw a .5" dia. circle centered on the origin.

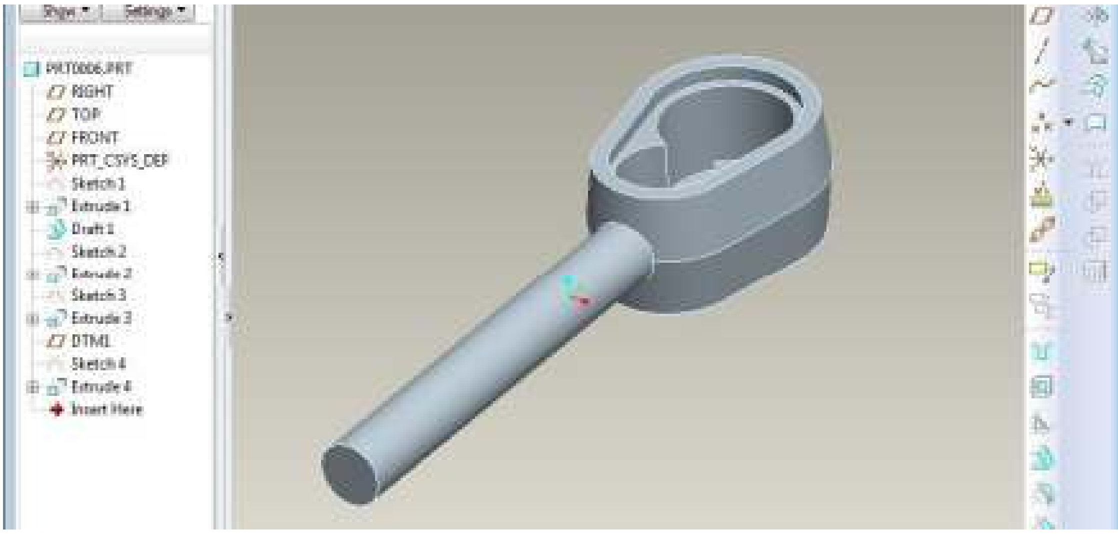


14. Extrude boss and use the "Up to next" option.

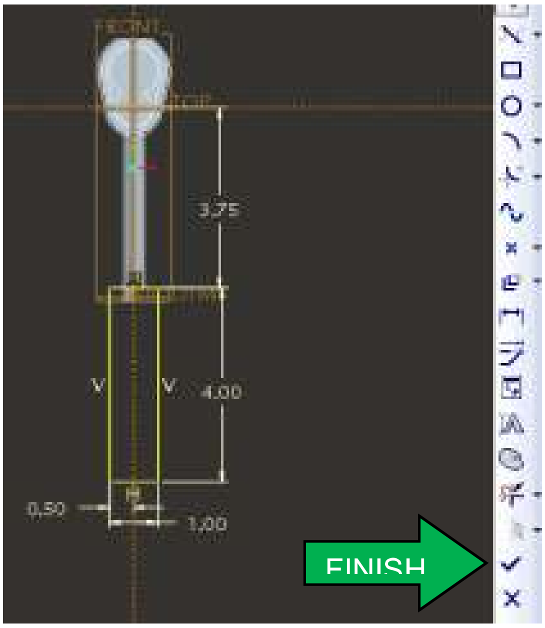


15. Select the circle and use the setting as shown in the illustration below.

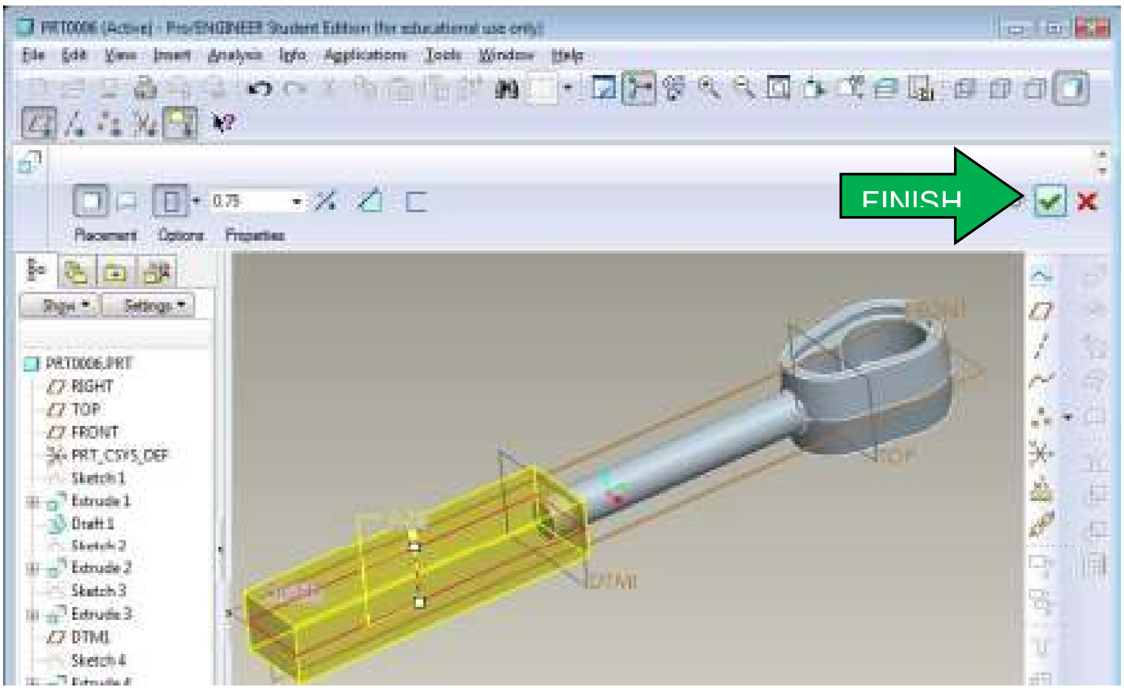




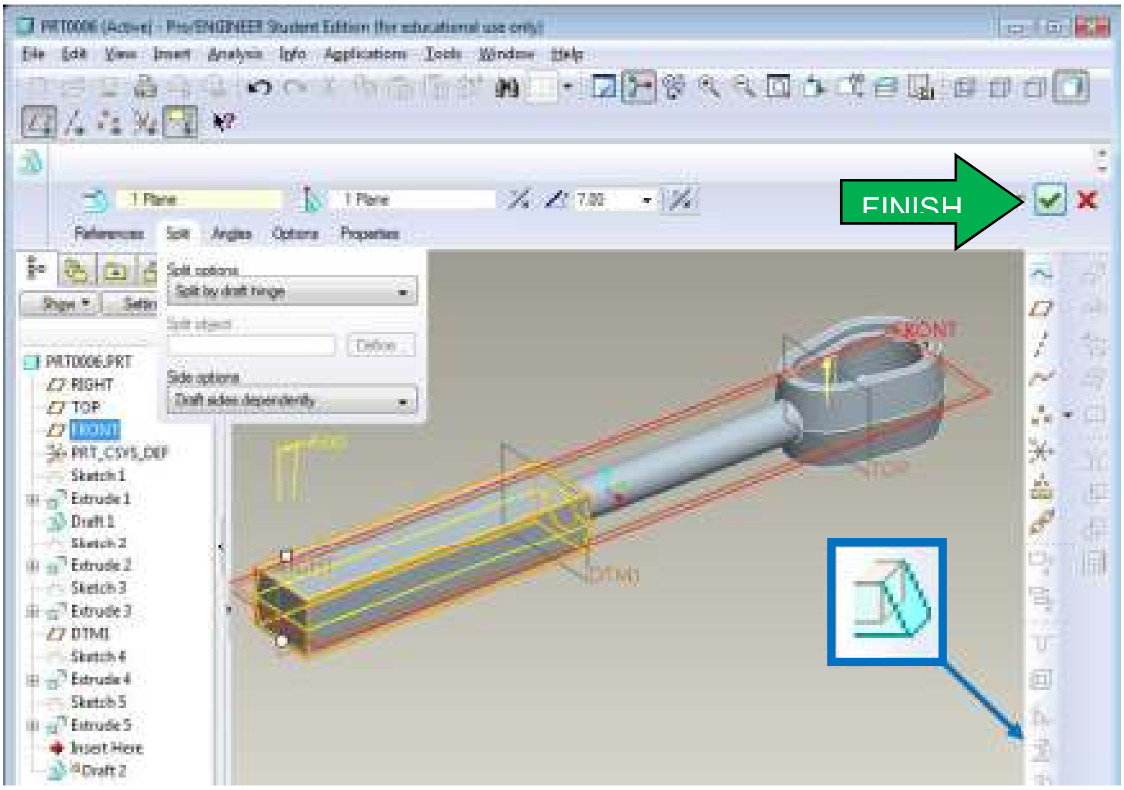
16. Start a sketch on the front datum plane and draw a rectangle with the following dimensions.



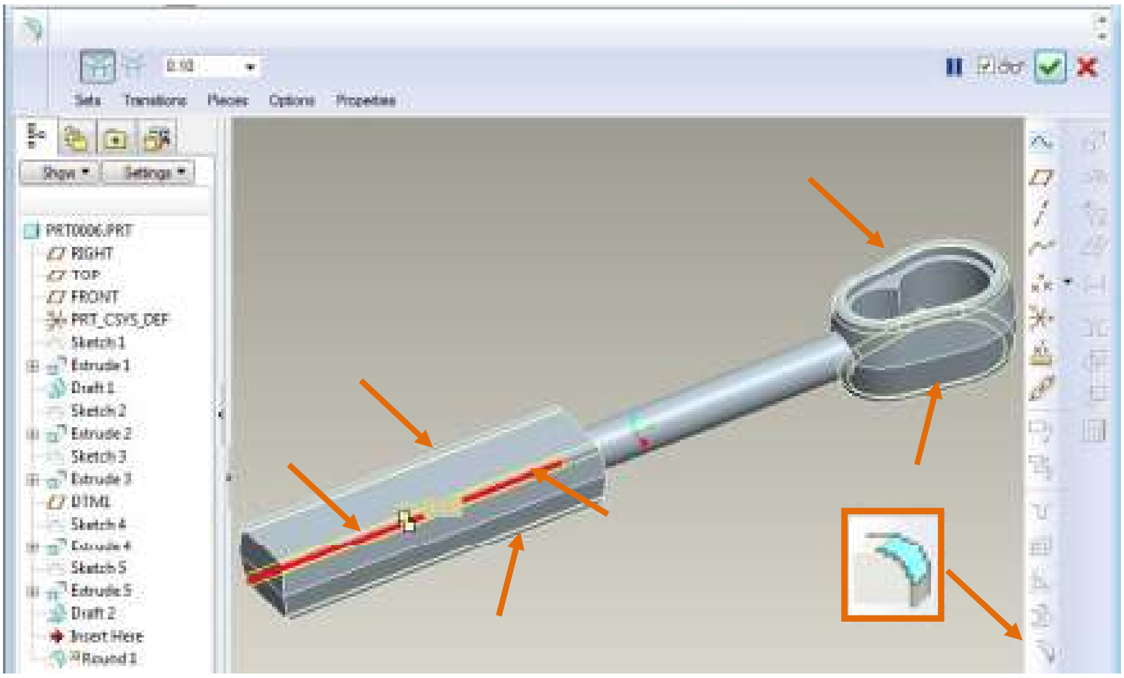
17. Extrude boss using the mid-plane option and .750 thick.



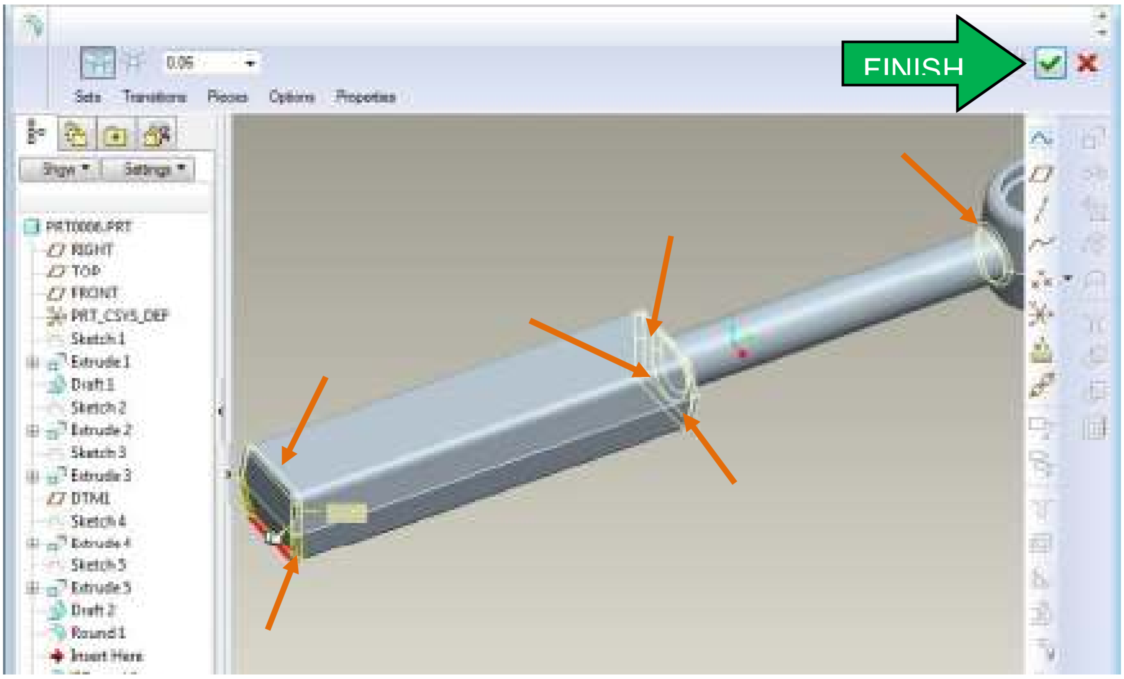
18. Using the **Draft** tool select the following faces and front plane and put 7° of draft on the side faces of the handle.



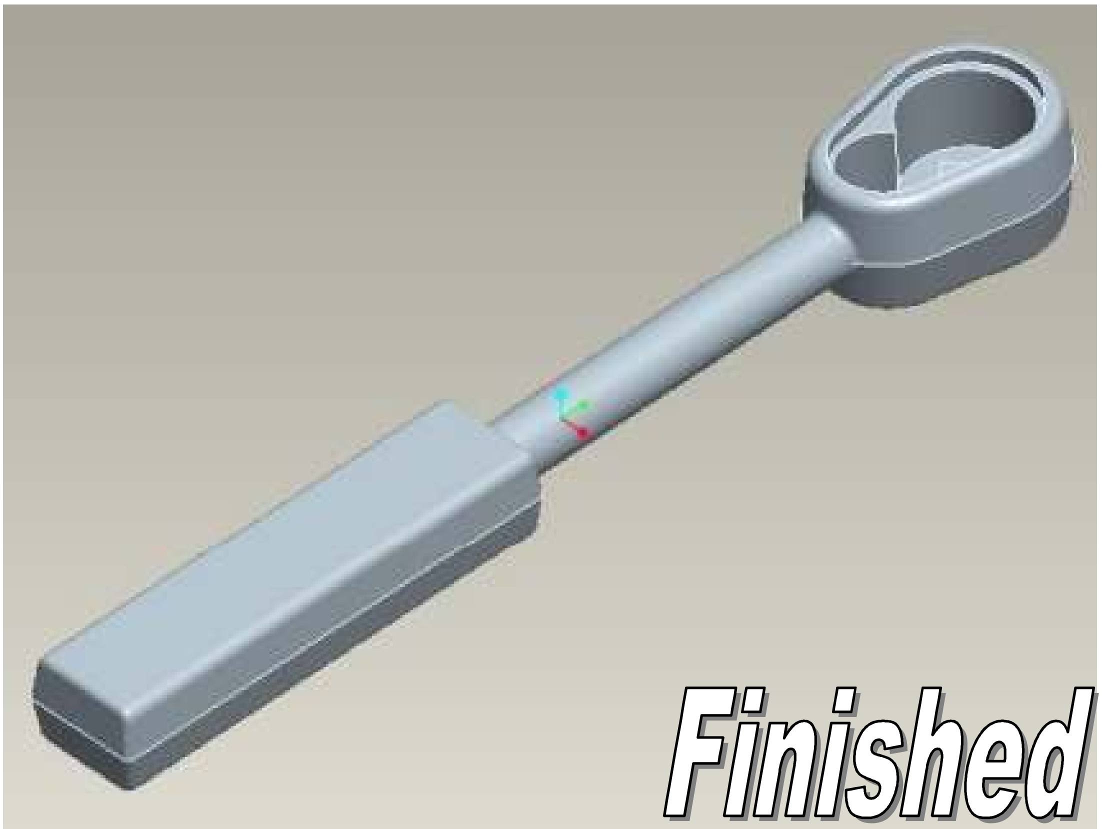
19. **Rounds:** Select the rounds/fillet icon, then select the edges as shown in the illustration below. Add .100".

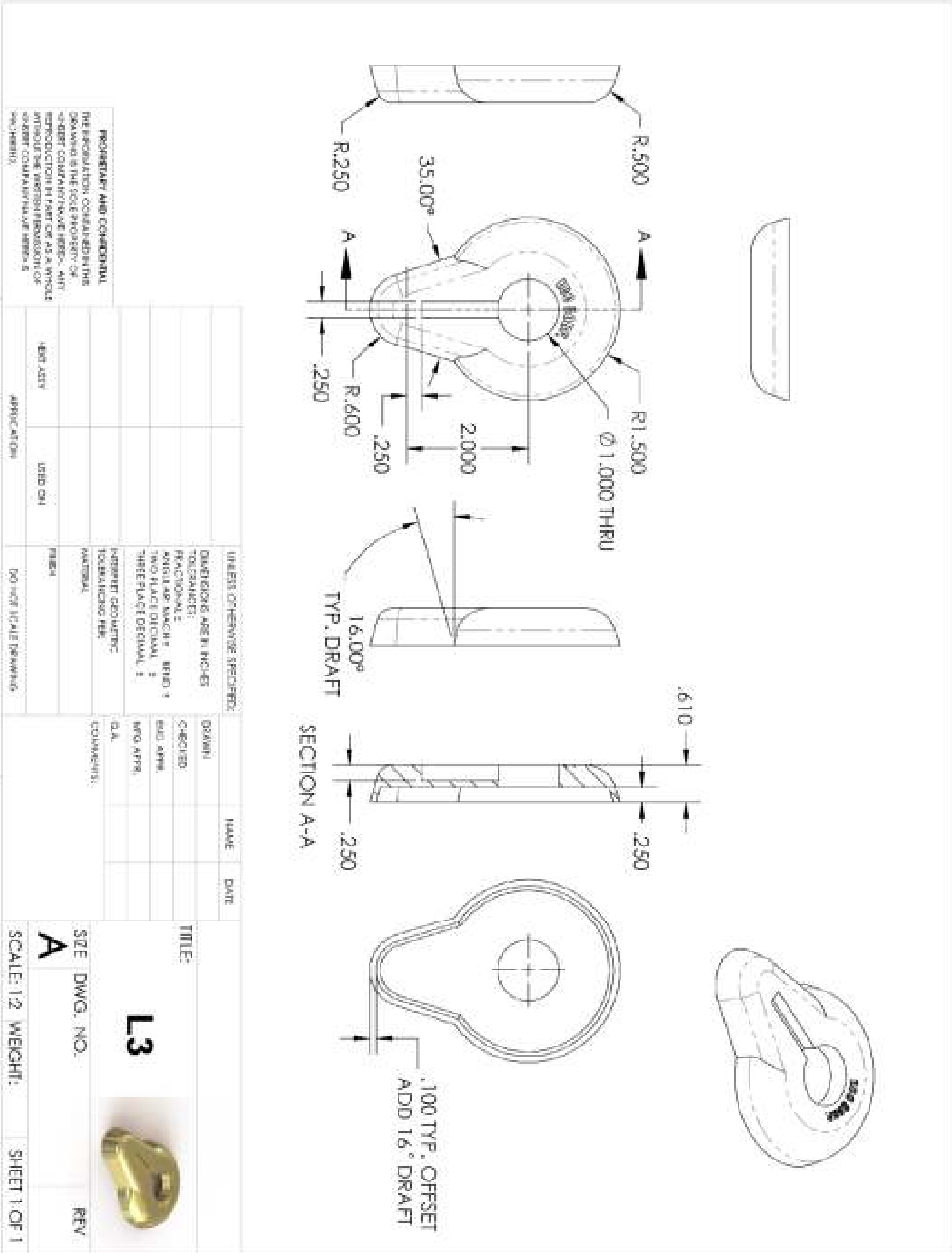


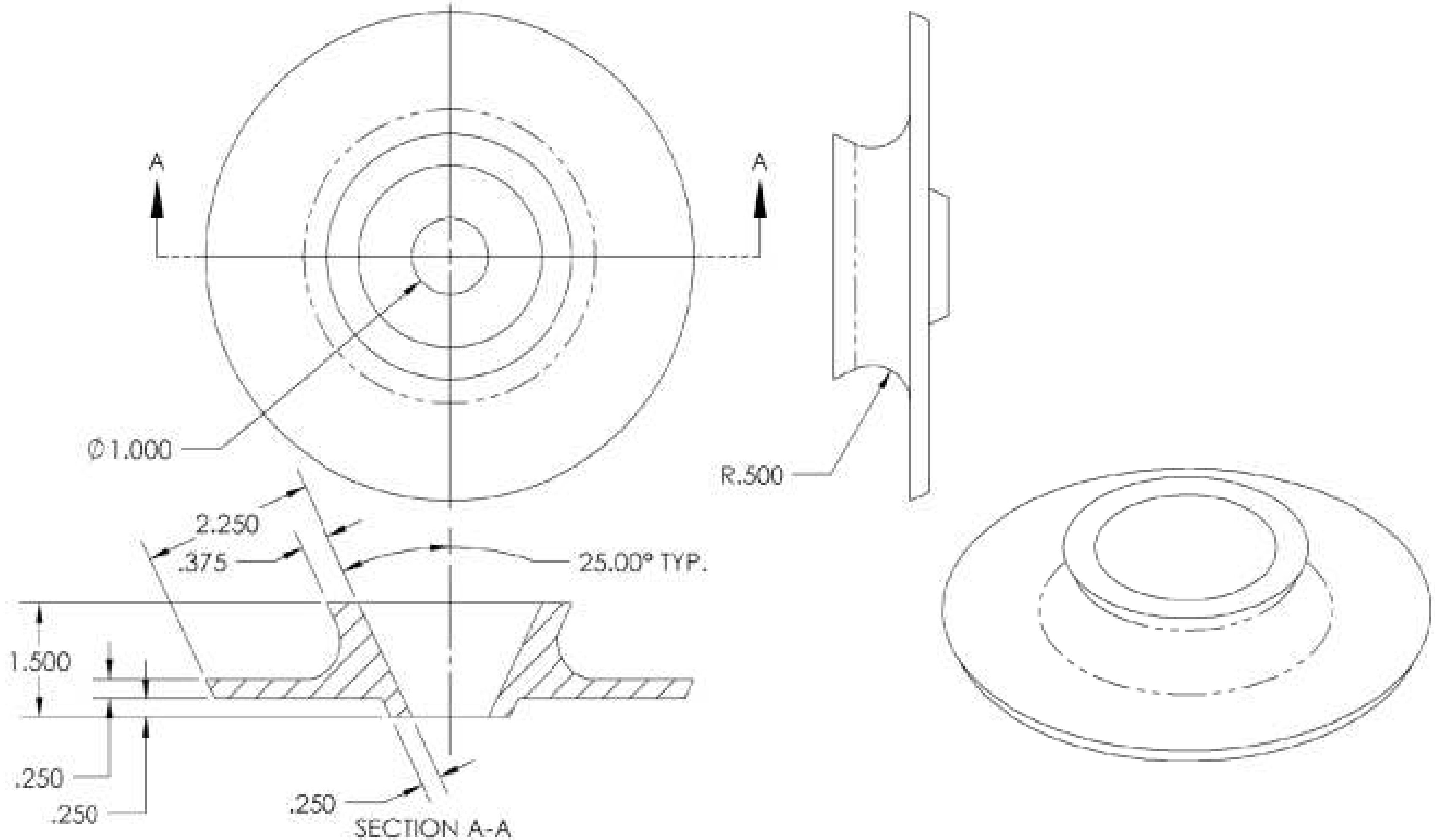
20. Add .060” Rounds to the following edges.



Finished




R.500
R.250
35.00°
A
A
R1.500
Ø1.000 THRU
R.600
.250
.250
2.000
16.00°
TYP. DRAFT
.610
.250
.250
SECTION A-A
.100 TYP. OFFSET
ADD 16° DRAFT
PROPRIETARY AND CONFIDENTIAL
UNLESS OTHERWISE SPECIFIED:
NAME
DATE
DRAWN
CHECKED
ENG APPR.
MFG APPR.
Q.A.
COMMENTS:
MATERIAL
FINISH
NEXT ASSY
USED ON
APPLICATION
DO NOT SCALE DRAWING
TITLE:
L3
SIZE
A
DWG. NO.
REV
SCALE: 1:2
WEIGHT:
SHEET 1 OF 1



PROPRIETARY AND CONFIDENTIAL

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

NEXT ASSY	USED ON
APPLICATION	

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:

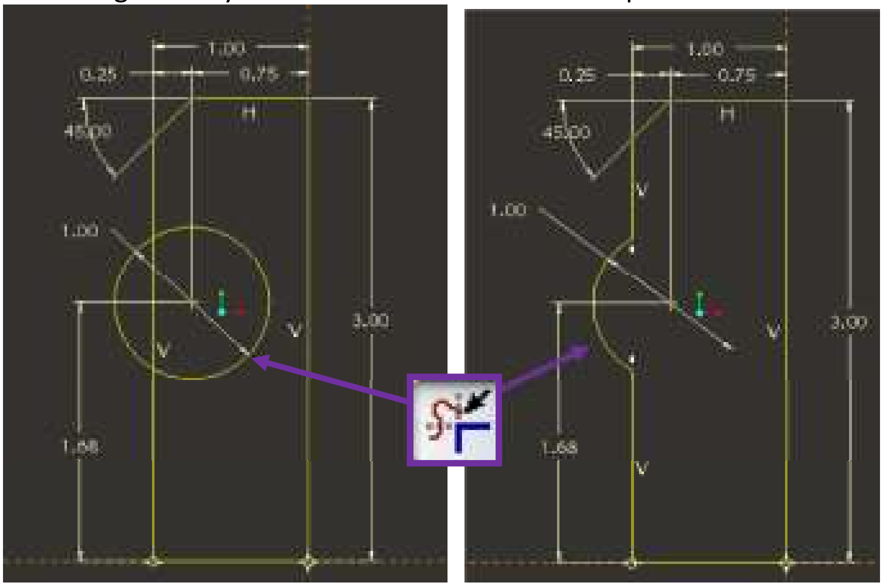
SIZE	DWG. NO.	REV
A	L3b	
SCALE: 1:2	WEIGHT:	SHEET 1 OF 1

5 4 3 2 1

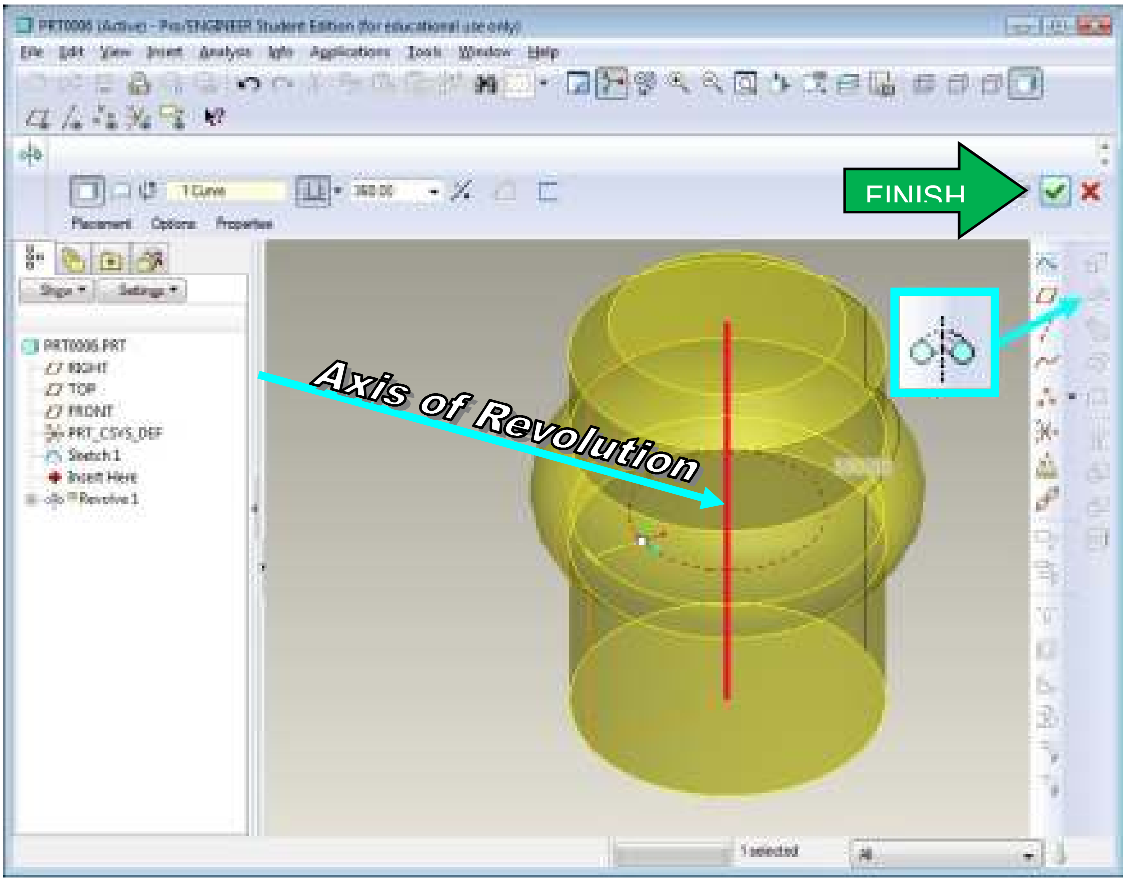
EXERCISE 4

Secondary Feature Modeling

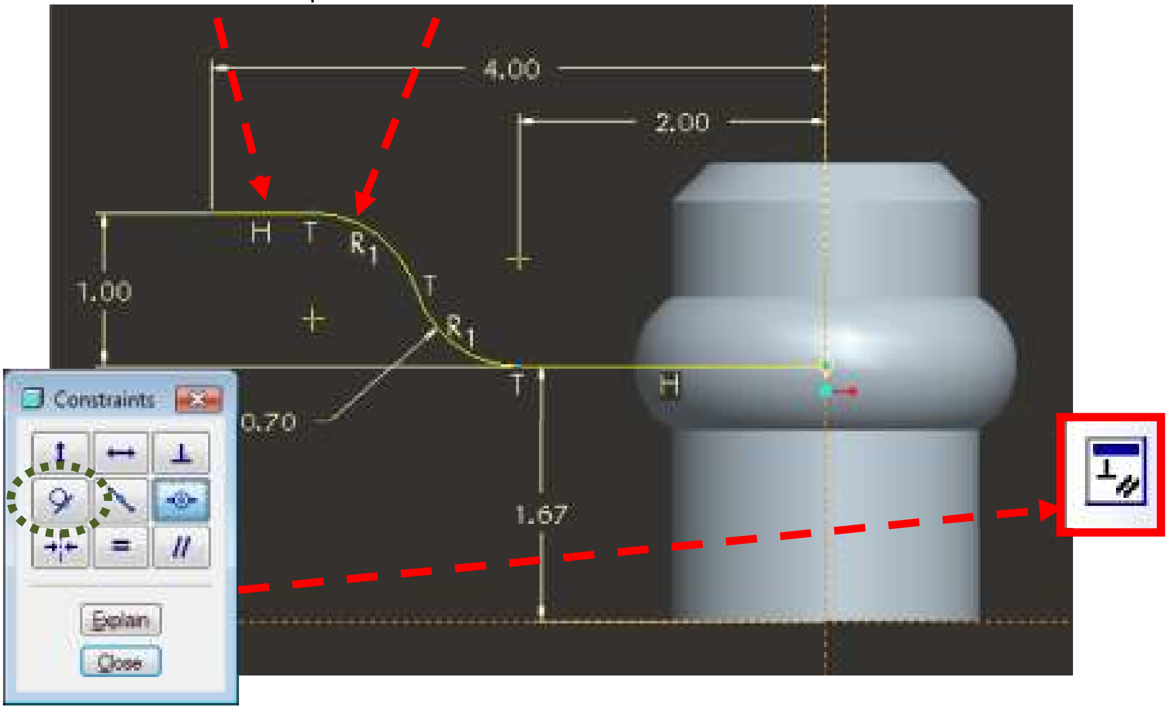
1. Sketch the geometry as show below on the “Front” plane. Then **Trim.**



8. **Revolve**.

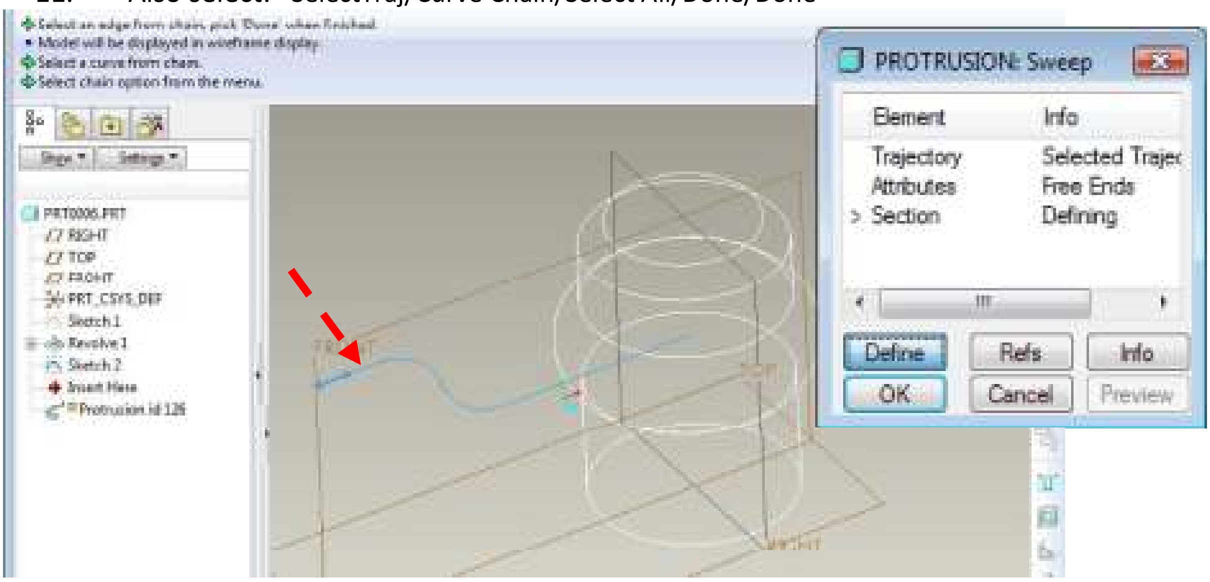


9. **Constraints:** Select the Front datum plane and sketch the following. Use the Constraint tool and select the **Tangent** option. Then select the left most horizontal line and the arc attached to it to establish a tangent relationship.

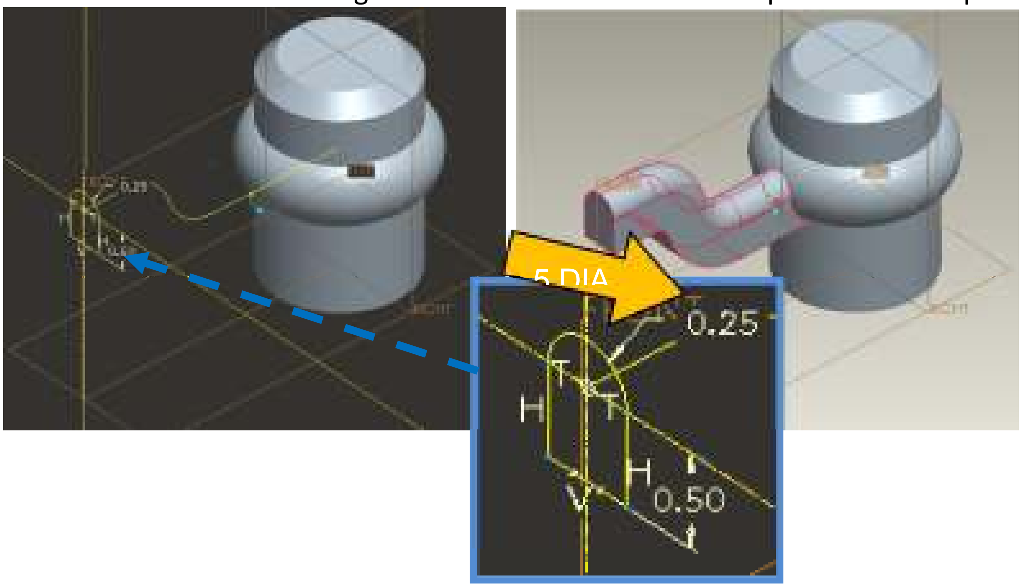


10. **Sweeps**: Use the pull-down menu "Insert/Sweep/Protrusion" Select the left side of the curve we just created to create a new sketch datum at the end.

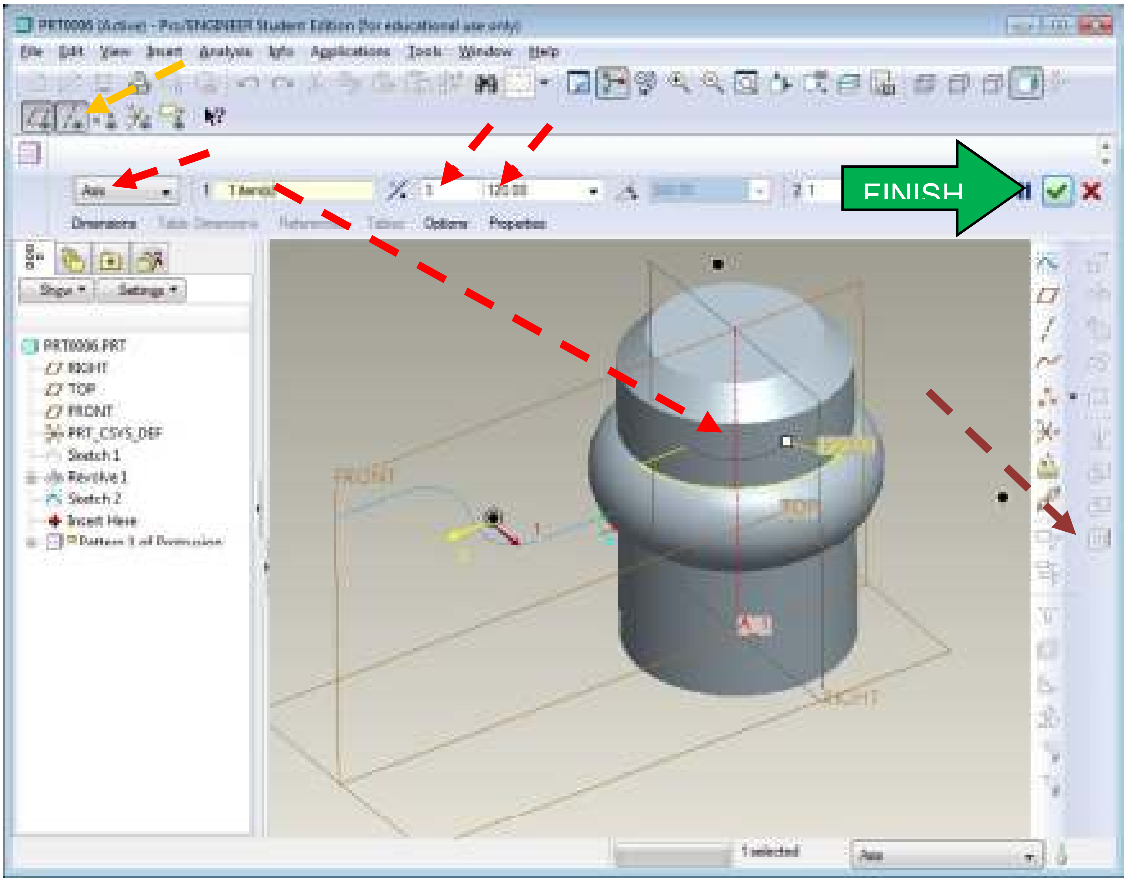
11. Also select: "SelectTraj/Curve Chain/Select All/Done/Done"



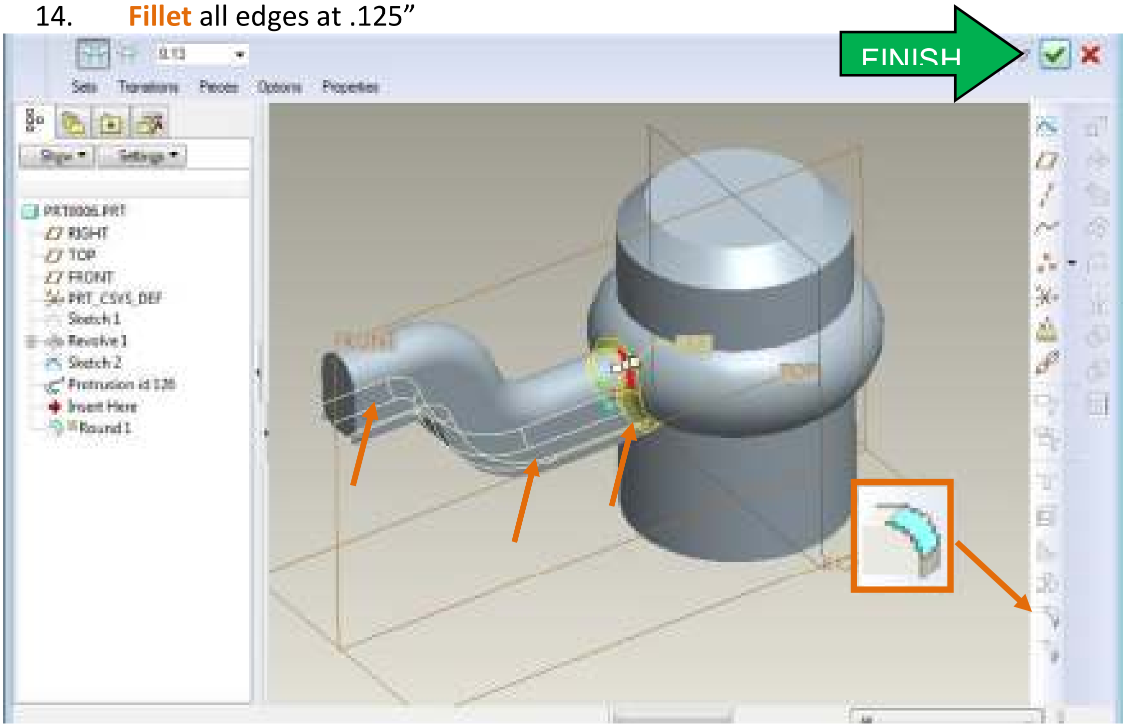
12. Draw the following sketch and select the "finish" option once complete.

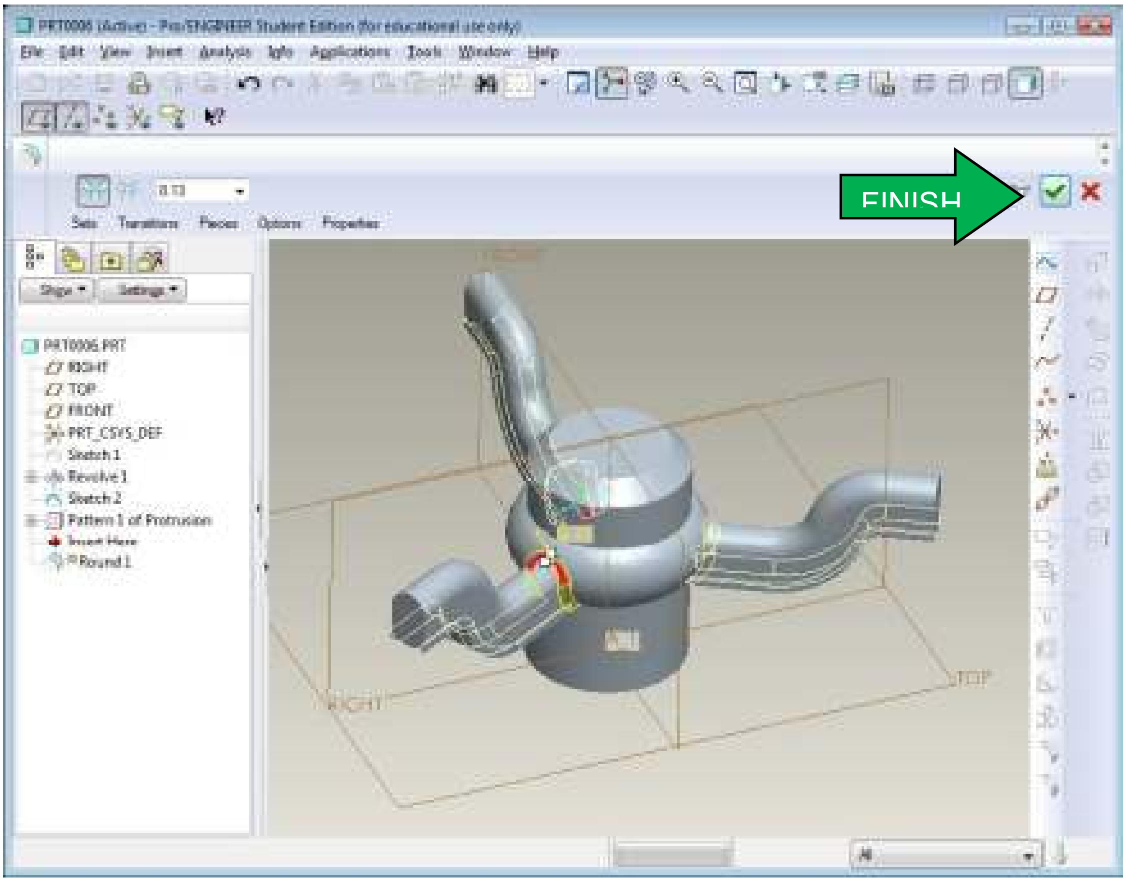


13. **Pattern** *Circular Pattern*: 360°/3 = 120° (*NOTE: First select the spoke to activate the icon.*) Select "Axis" also select the "view axis"

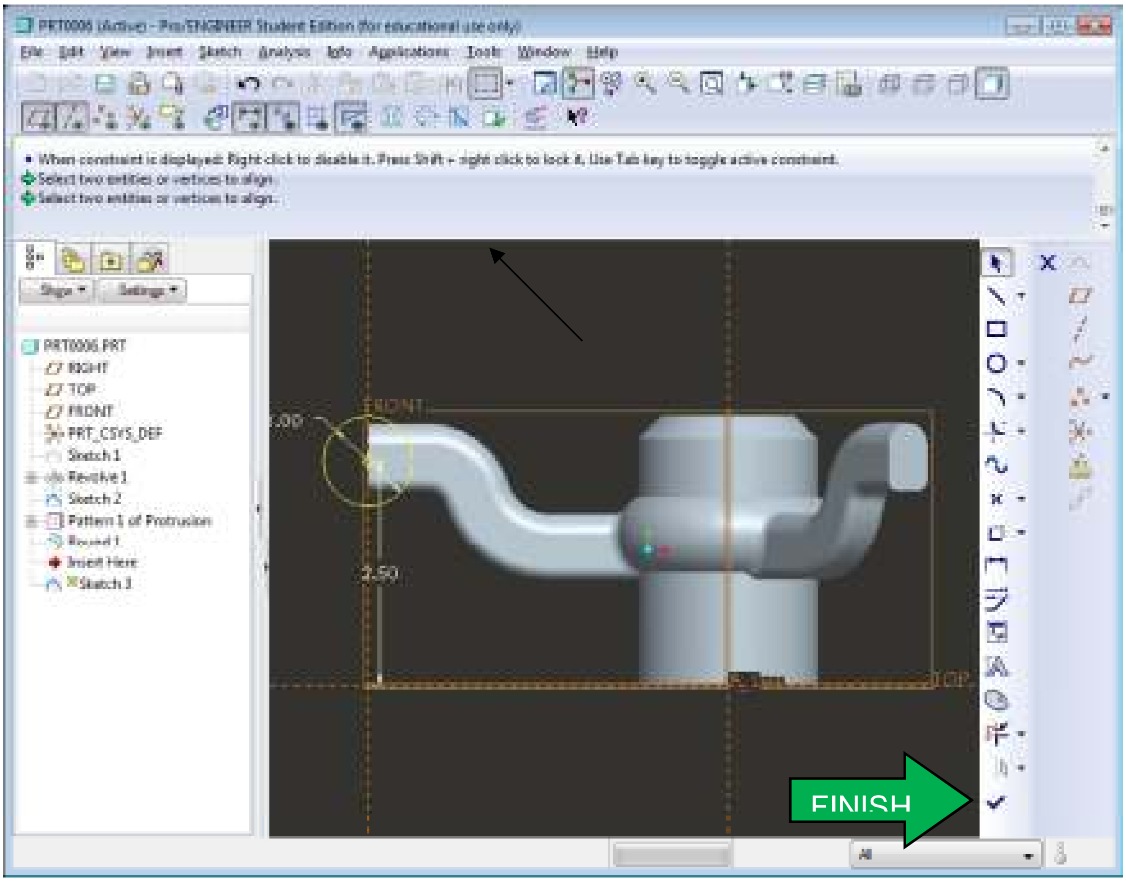


14. **Fillet** all edges at .125”

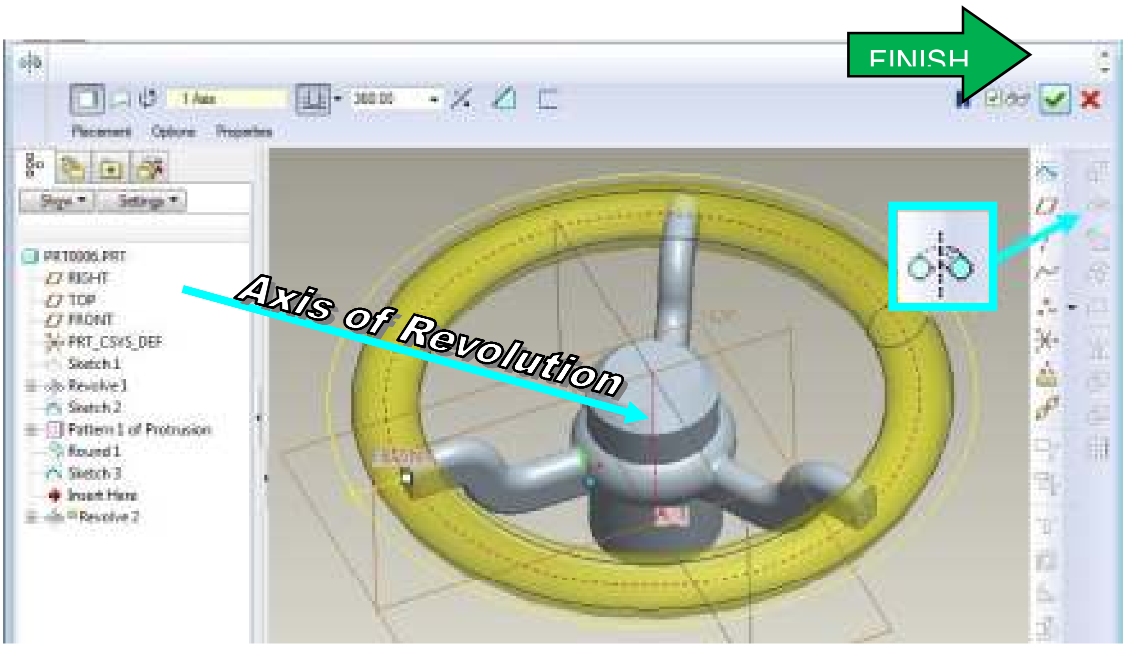




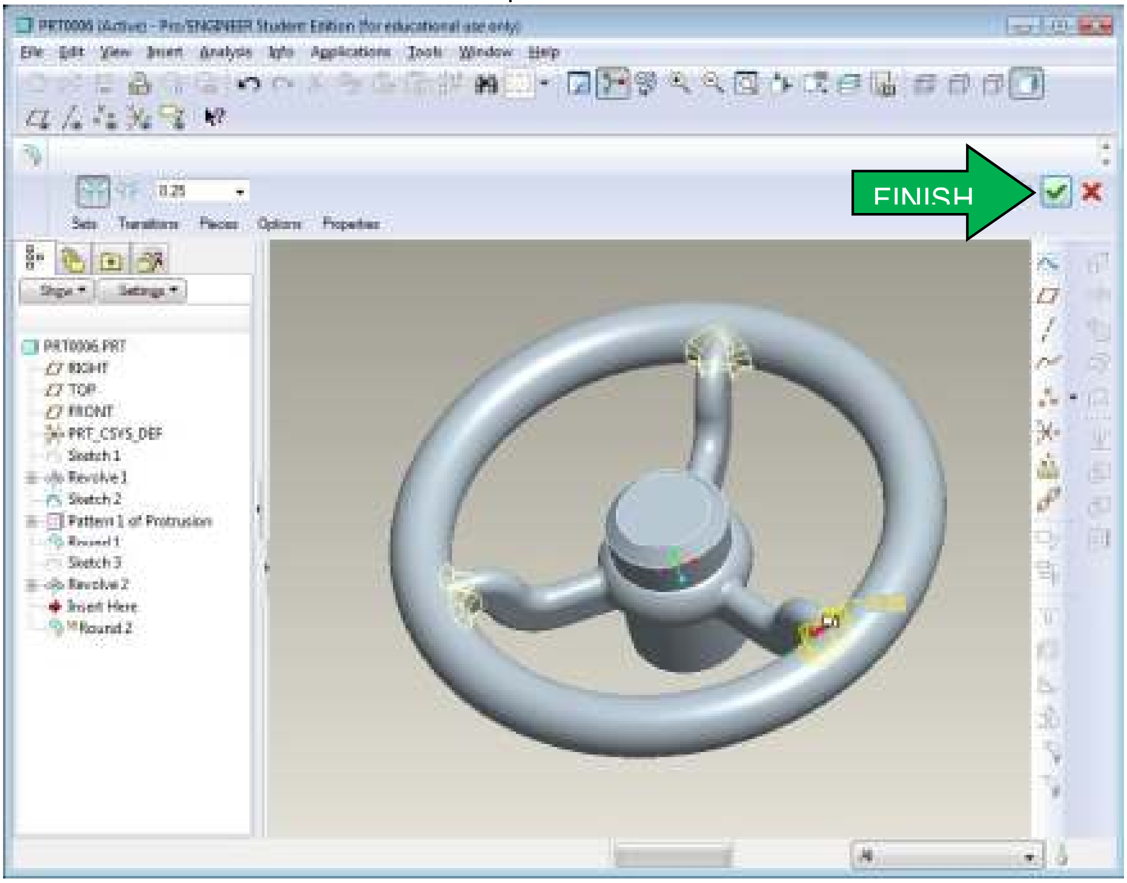
15. Select the “Front” plane and start a sketch on it. Rebuild after completion.



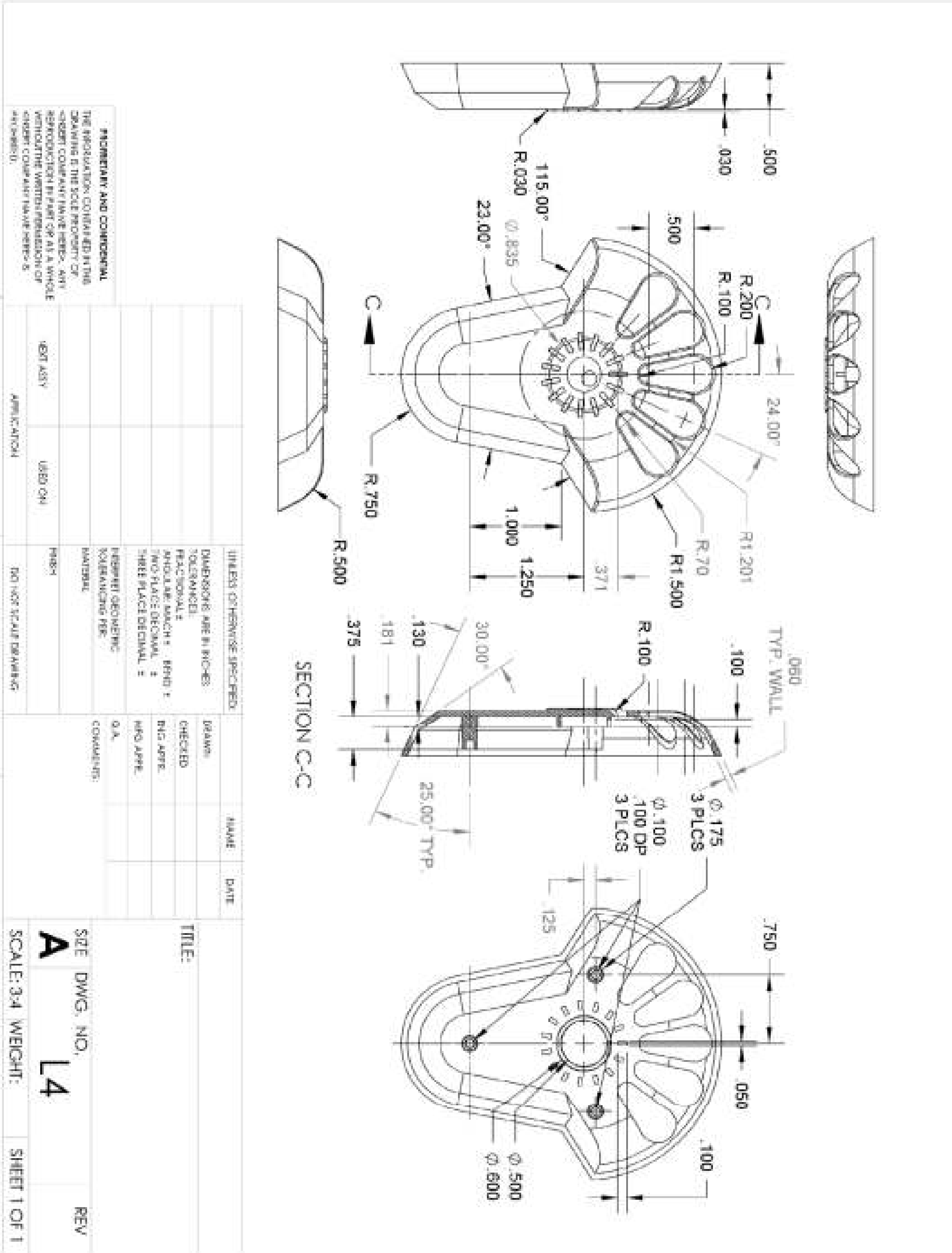
16. **REVOLVE**



17. Add .250" **Rounds** to the spoke – handle sections.



FINISHED

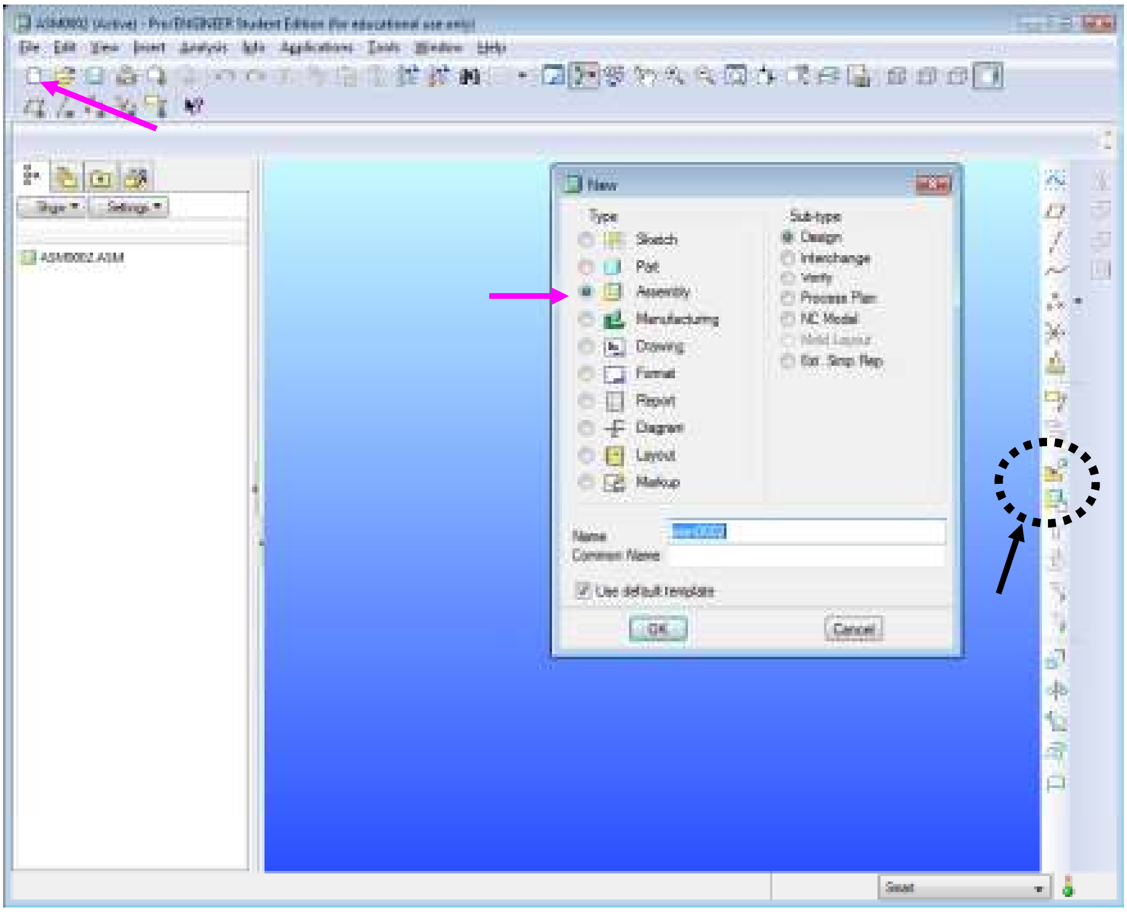


SECTION C-C
C
C
.500
.030
R.030
115.00°
23.00°
Ø.835
.500
R.200
R.100
24.00°
R1.201
R.70
R1.500
R.750
R.500
1.000
1.250
.371
R.100
.060 TYP. WALL
.100
30.00°
.130
.181
.375
25.00° TYP.
Ø.175 3 PLCS
Ø.100 .100 DP 3 PLCS
.125
.750
.050
.100
Ø.500
Ø.600
PROPRIETARY AND CONFIDENTIAL
UNLESS OTHERWISE SPECIFIED:
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
NEXT ASSY
USED ON
APPLICATION
NAME
DATE
DRAWN
CHECKED
ENG APPR.
MFG APPR.
Q.A.
COMMENTS:
TITLE:
SIZE
A
DWG. NO.
L4
REV
SCALE: 3:4
WEIGHT:
SHEET 1 OF 1

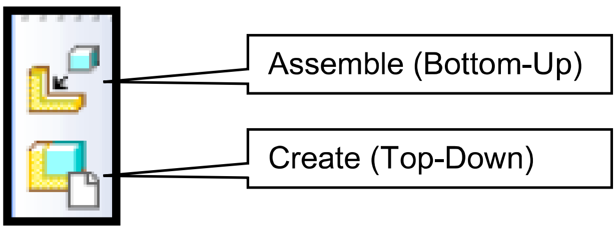
EXERCISE 5

Bottom-Up Assembly Creation

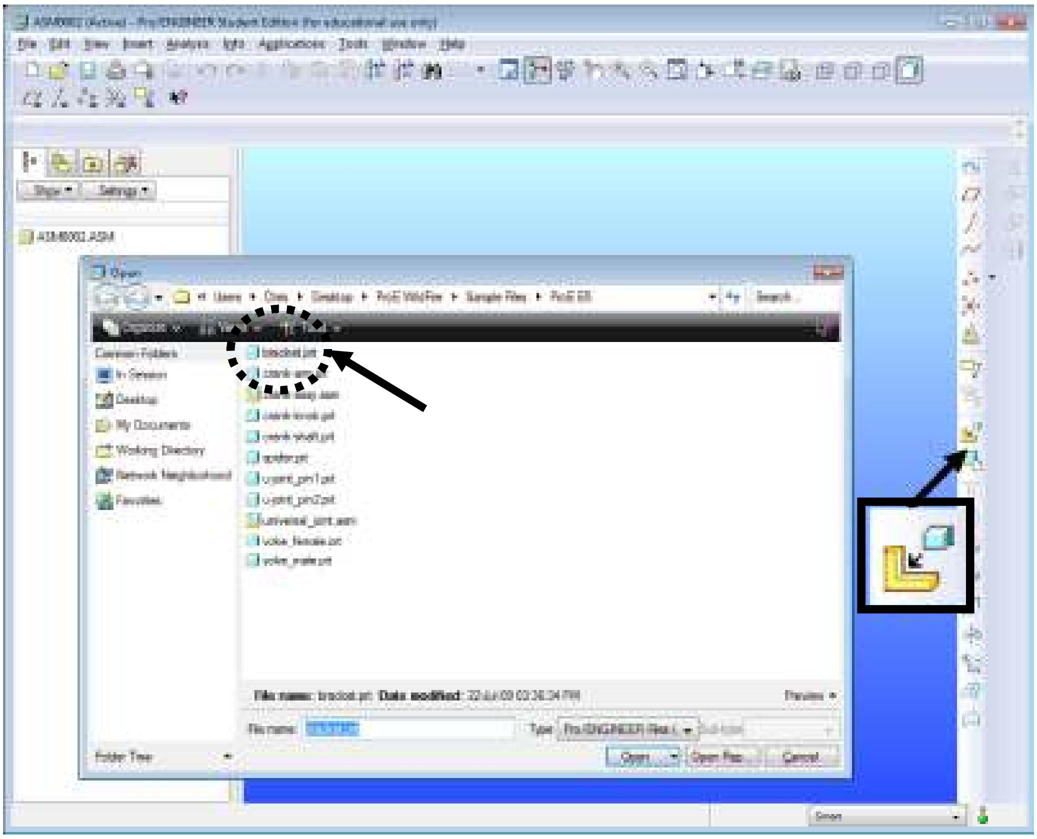
1. Go to “File/New and select the Assembly Template”.



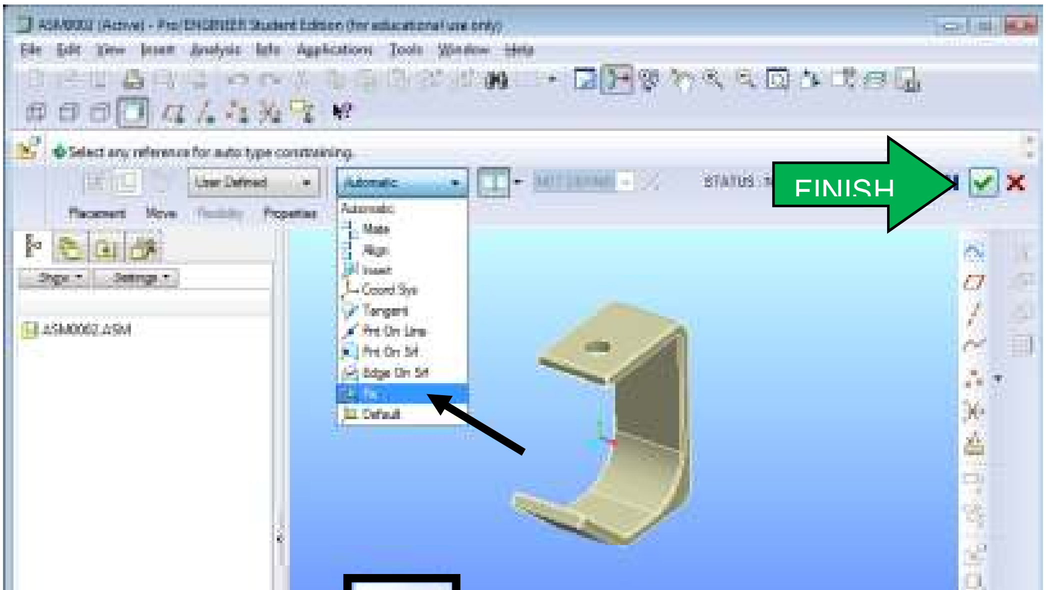
2. Assembly Tools.



3. To insert a part into the assembly select the **Assemble** icon. Select the *Bracket.prt,* and hit the “open” button at the bottom.

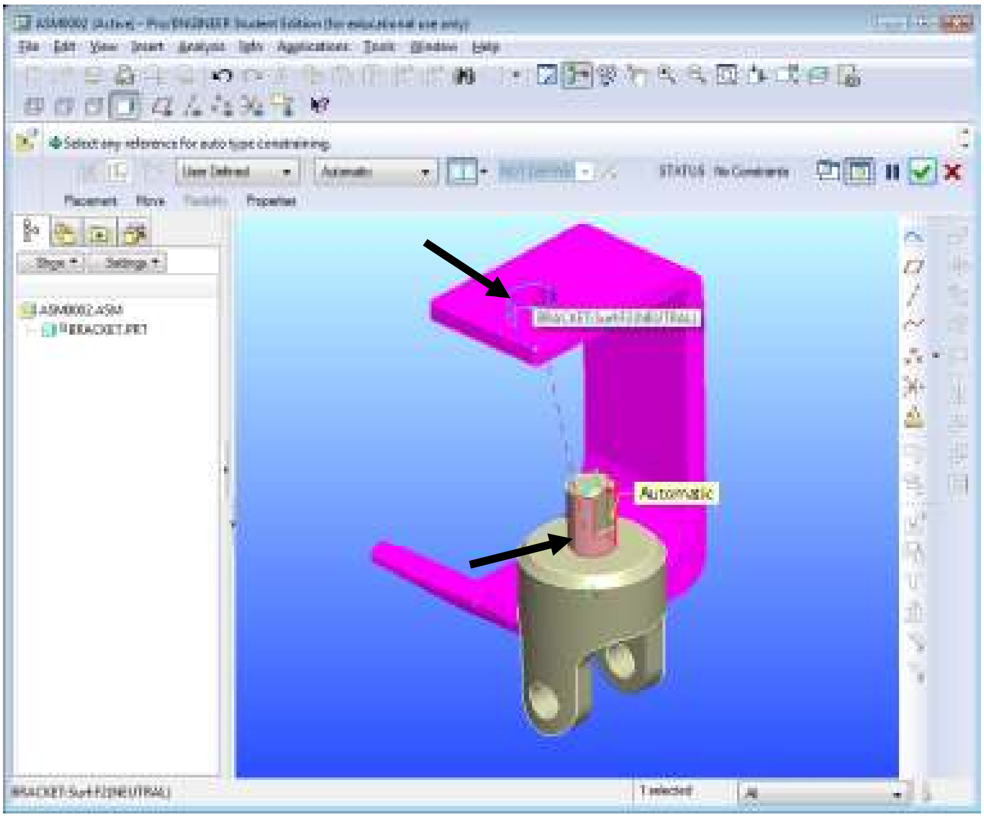


4. Select the ***Automatic*** pull down and select the ***Fix*** option.

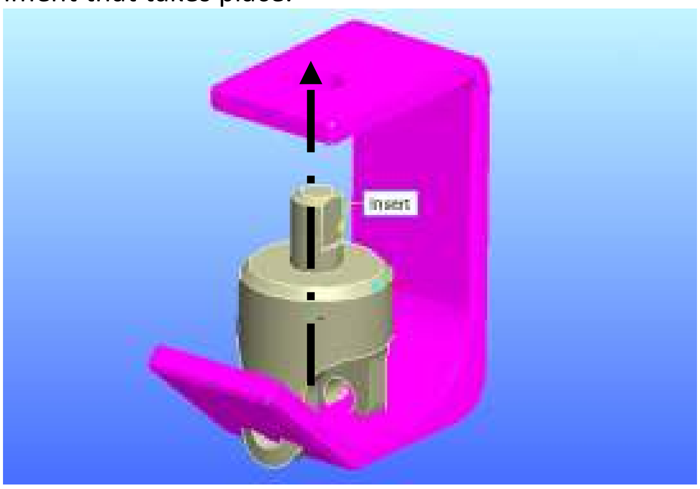


5. Select the Assemble  icon and then insert the ***yoke_male.prt***.

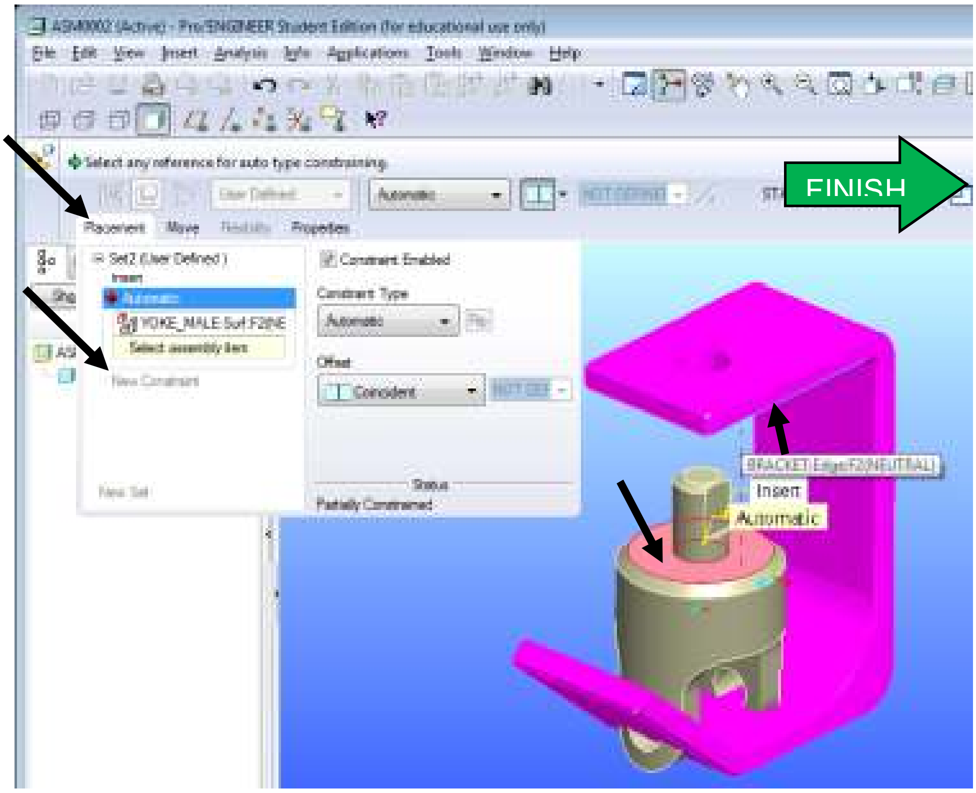
6. Select the radial surface of the ***yoke_male*** shaft and then select the surface of the hole on the ***bracket***.



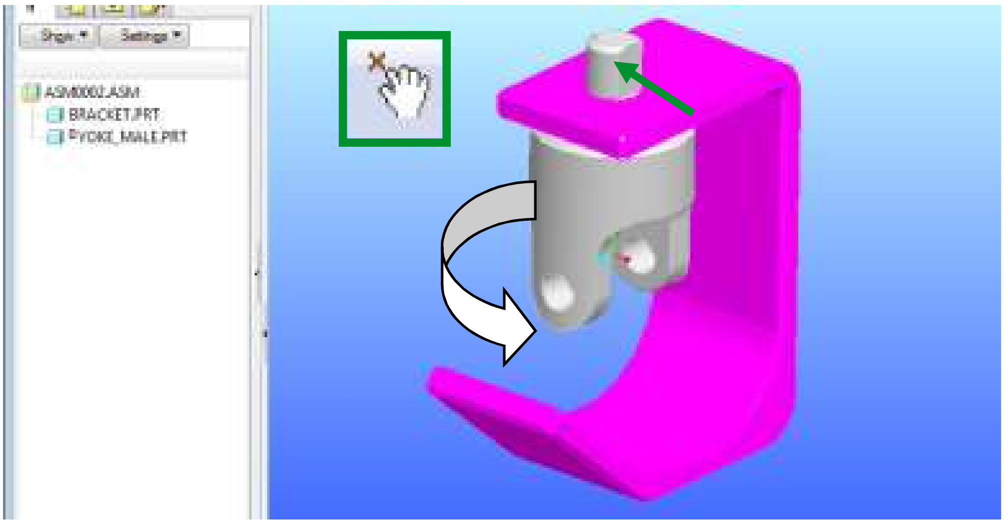
Notice the alignment that takes place.



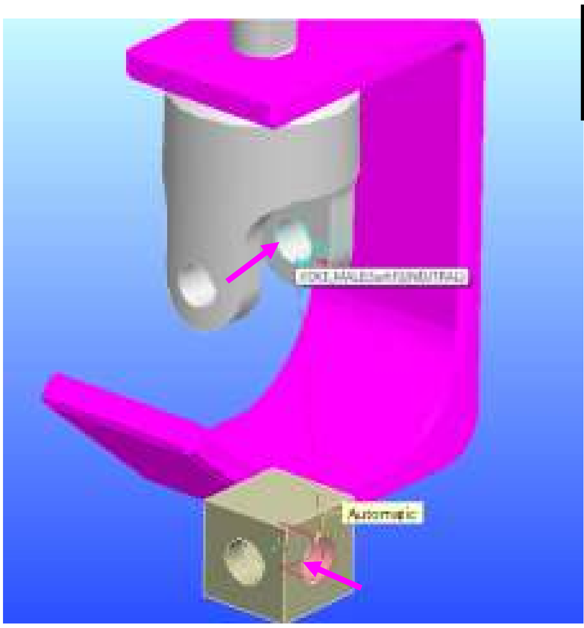
7. Select the **Placement** tab and then select **New Constraint** option. Then select the top surface of the ***yoke_male***, and the underside face of the top flange of the ***bracket***. *Note: make sure you deselect the **Allow Assumptions** icon to enable dynamic assembly motion (it's located at the bottom of the **Placement** tab).*

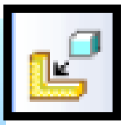


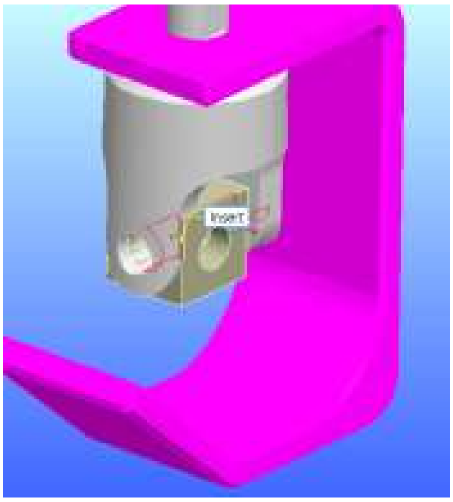
8. After applying the last constraint try moving the component using the **Drag Component** icon. Click on an edge of the yoke and drag with the left mouse button. It should spin in place only.



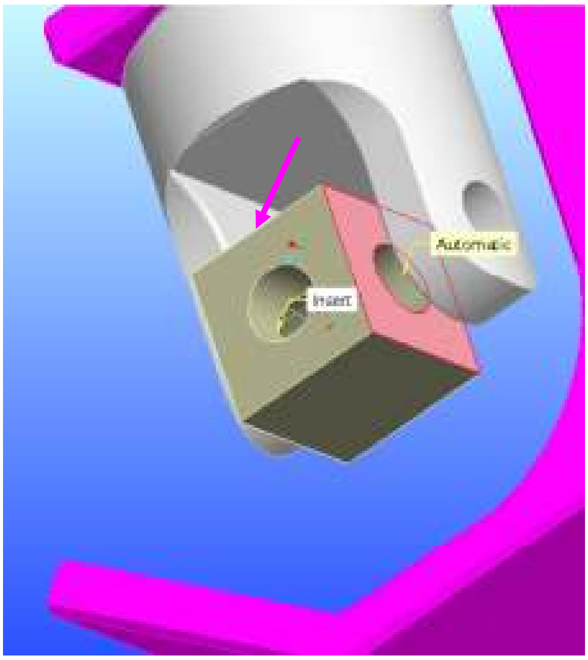
9. Insert the *spider.prt* and mate the cylindrical faces of the holes.

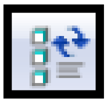


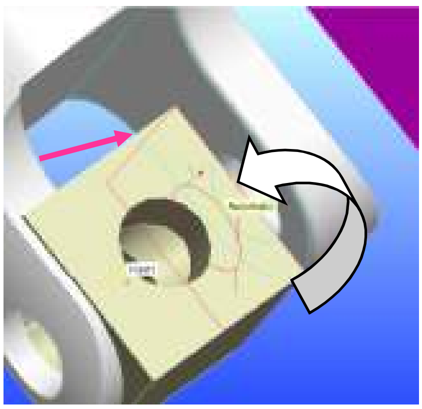




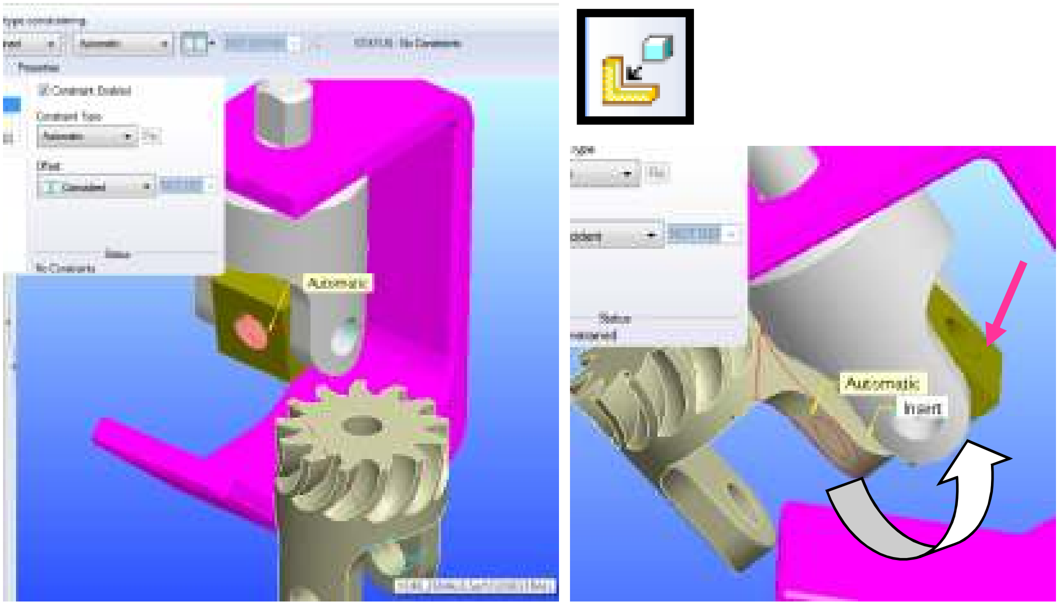
10. Select the side face of the *spider* and then the inside face of the *male_yoke* leg. You may need to rotate the assembly to see the correct faces. You may need to *Regenerate* after applying the last mate.



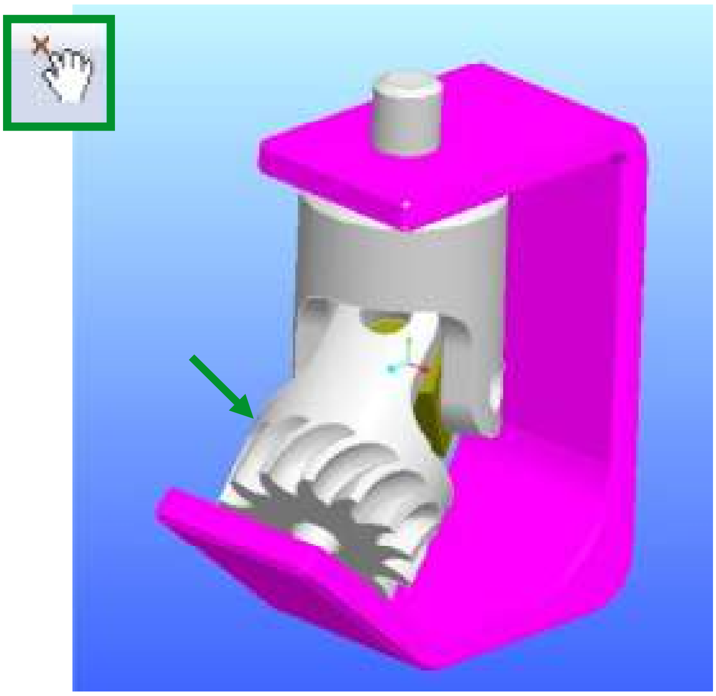




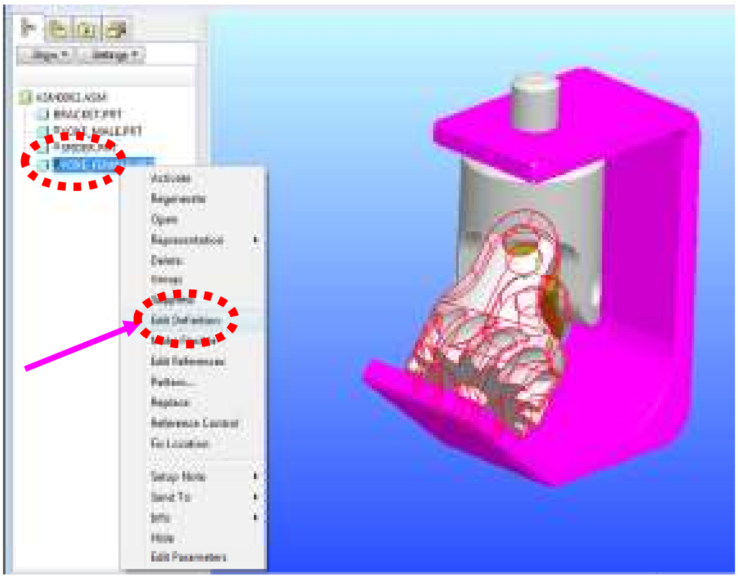
11. Select the concentric holes. Select the *yoke_female* leg and open face of the *spider*.



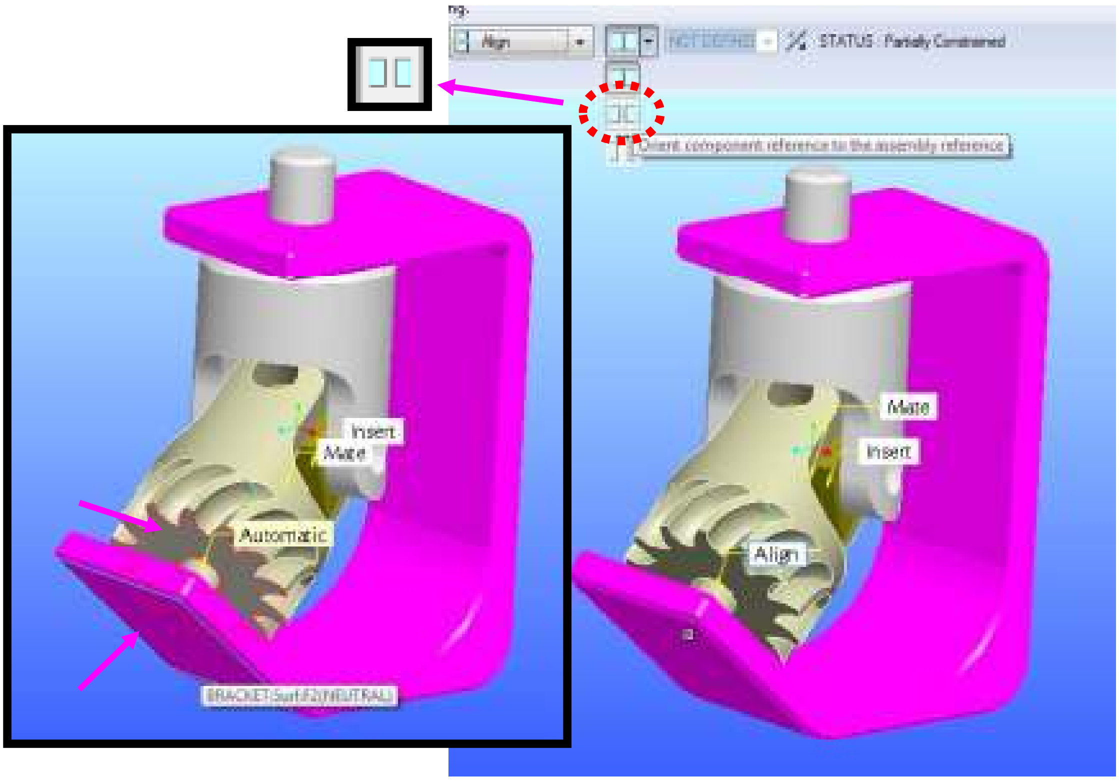
12. Use the **Drag Component** tool to locate the *yoke_female* near the bottom angled flange of the *bracket*.



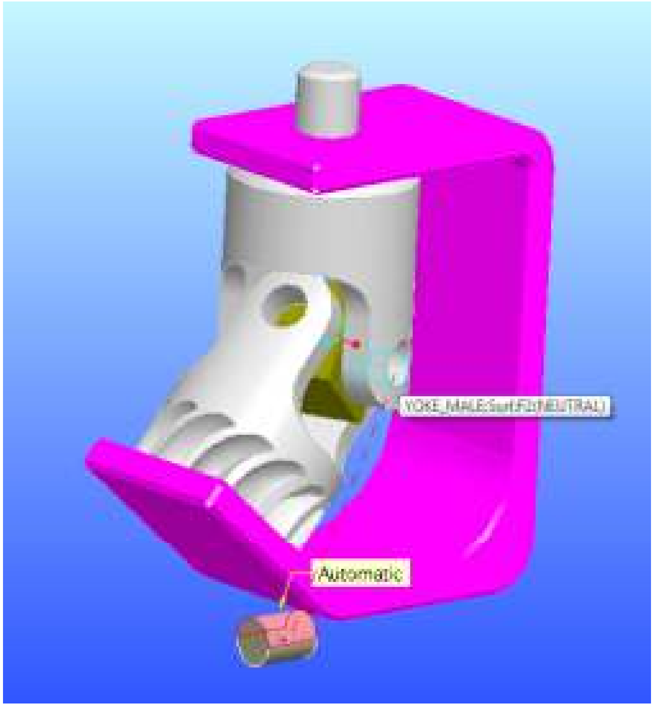
13. **Editing a Mate:** RMB select the Spider from the feature tree on the right of the screen. A pull-down menu will appear. Select Edit Definition.



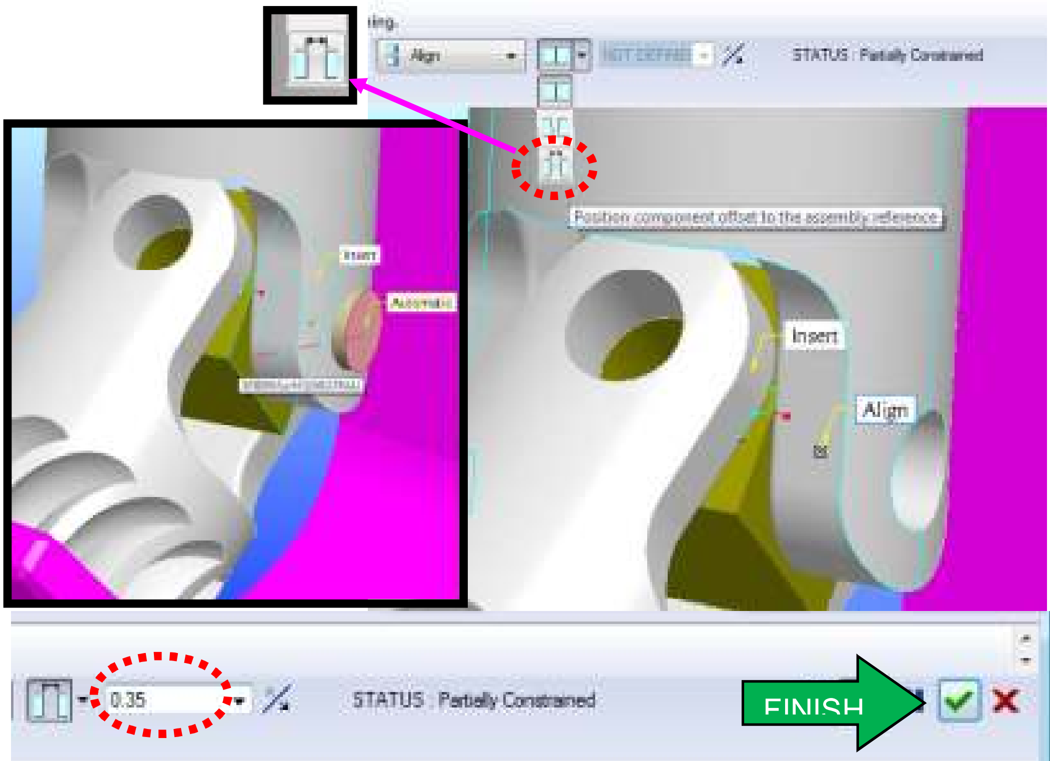
14. **Parallel Mate:** Select both bottom faces of the *yoke_female* and the angled flange of the *bracket*. Then select the *Orient to assembly reference* option to align parallel. (*Parallel is needed here because there is a small gap between the parts.)*



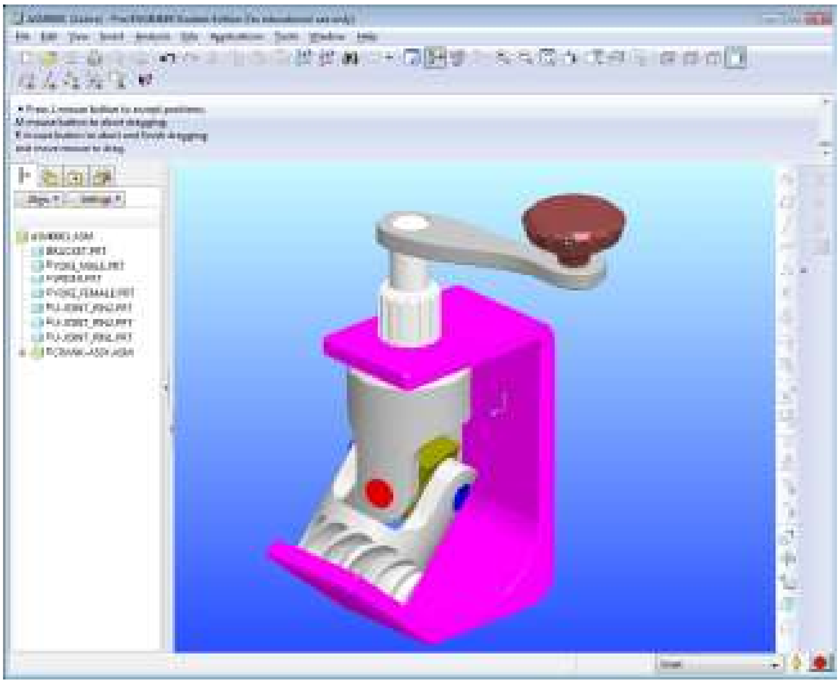
15. Insert the u-joint pin_2.prt, and select the cylindrical faces to mate.



16. **Distance Mate:** Select the end face of the *pin* and then select a parallel flat face the *spider*. Add a distance of .35"

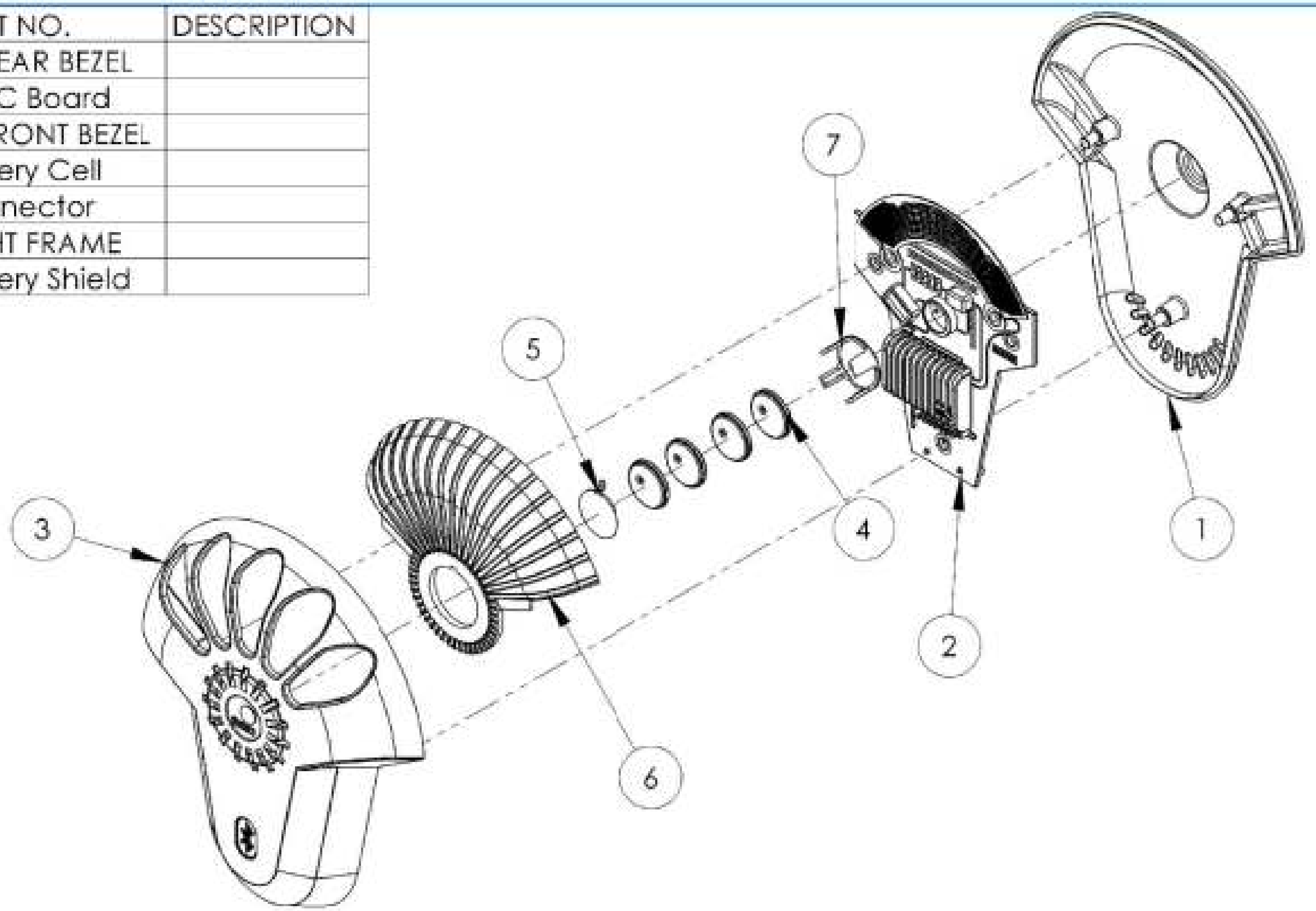


17. Attach the remainder of the components.



18. After completion you should be able to use the **Drag Component** icon to dynamically rotate the assembly.

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



PROPRIETARY AND CONFIDENTIAL

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

NEXT ASSY | USED ON
APPLICATION

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:

ORION

SIZE A | DWG. NO. L5b | REV

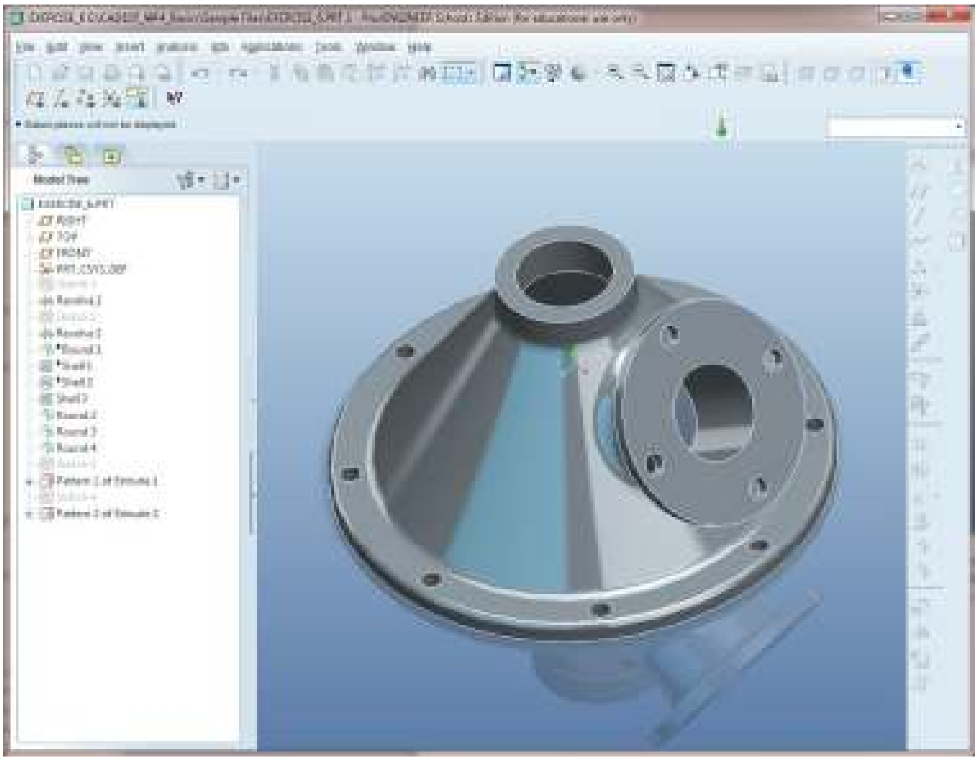
SCALE: 1:5 WEIGHT: | SHEET 1 OF 1

5 4 3 2 1

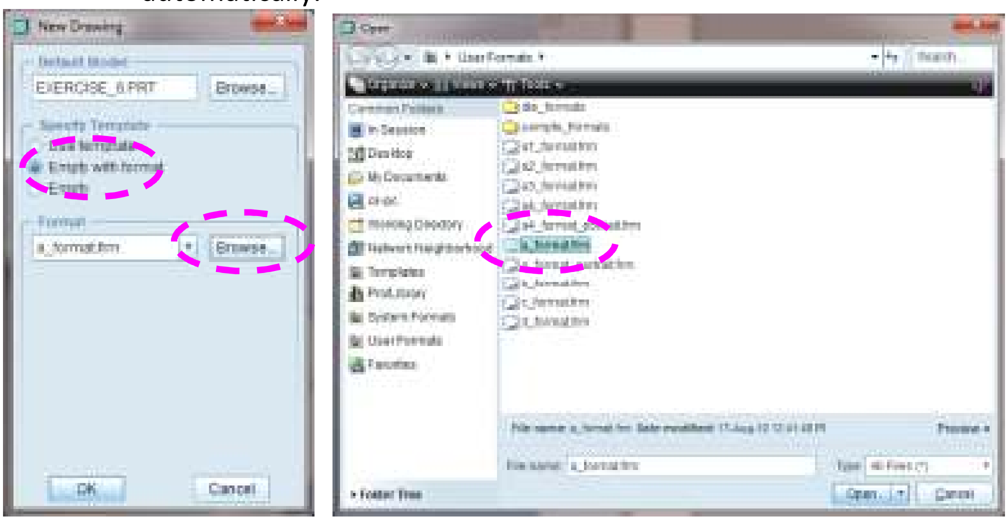
EXERCISE 6

Fundamental 2D Drawing Creation

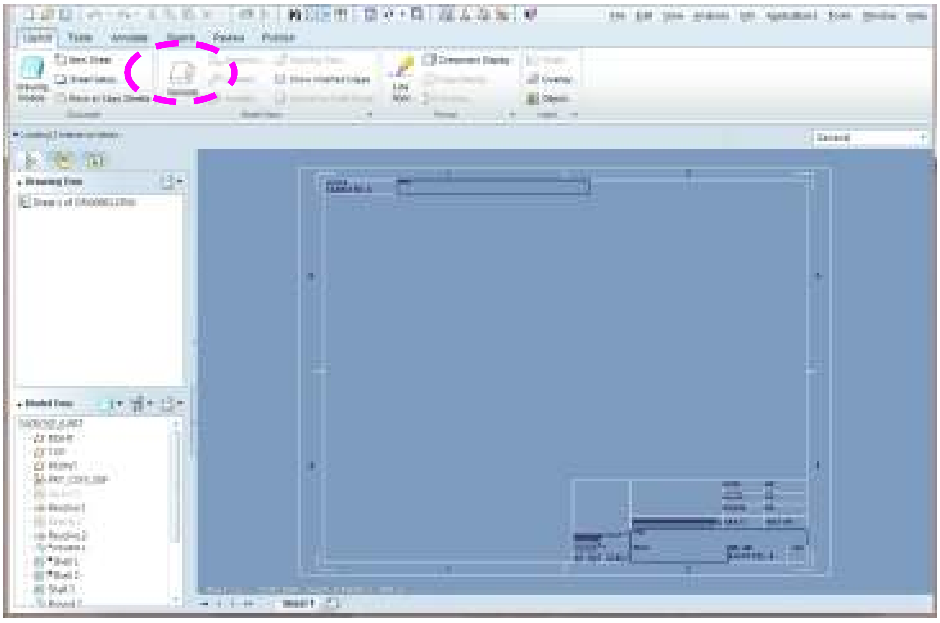
1. Open the “Exercise 6” part file.



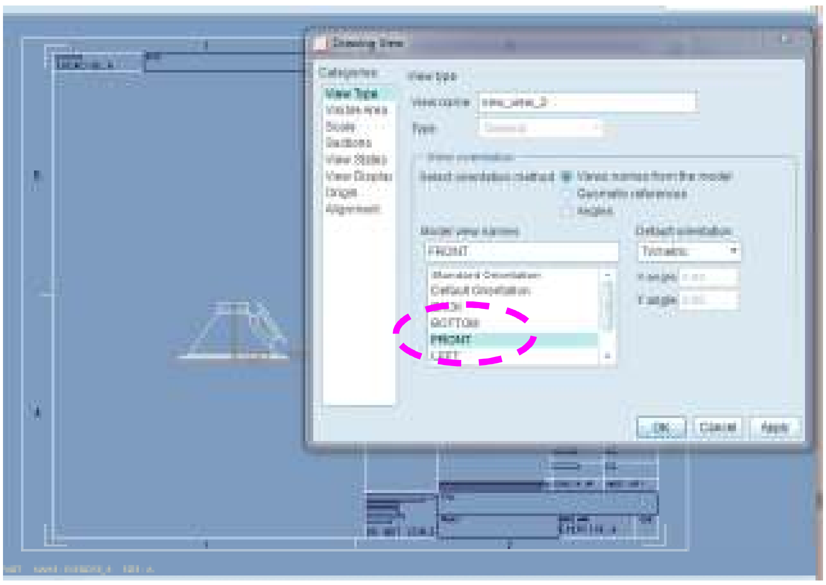
2. **View Layout**/Drawing Toolbar. Make sure Exercise 6 is shown in the “Default model” box, and select Empty with format, then select a_format. You may need to browse to find the part if it does not show automatically.



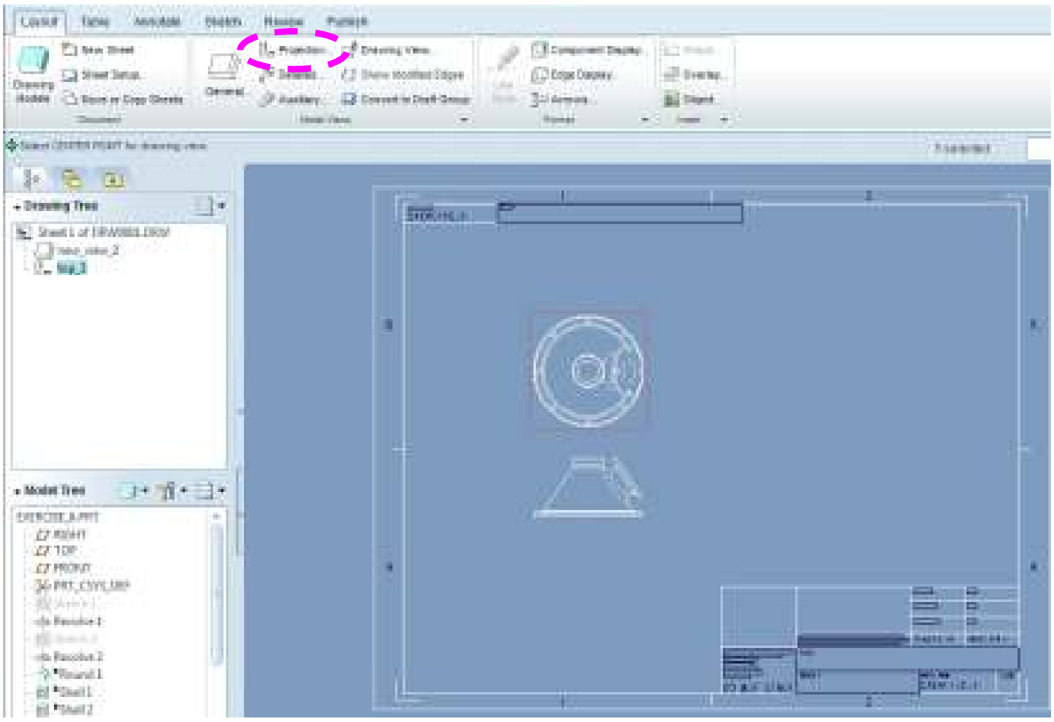
3. The standard a sheet should automatically show up.



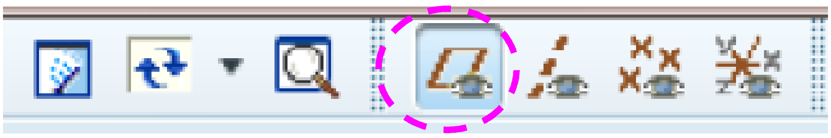
4. To insert views RMB (right mouse button) click/hold in the center of the drawing. Or select the “General” icon in the “Layout” tab tools ribbon.
5. Select “insert general view” from the list, and then left click to drop the new view in.
6. Select the “FRONT” option from the “Drawing View” dialog box and hit OK. *NOTE: If you lose the “Drawing View” dialog box simply double click on the drawing view itself to return it.*

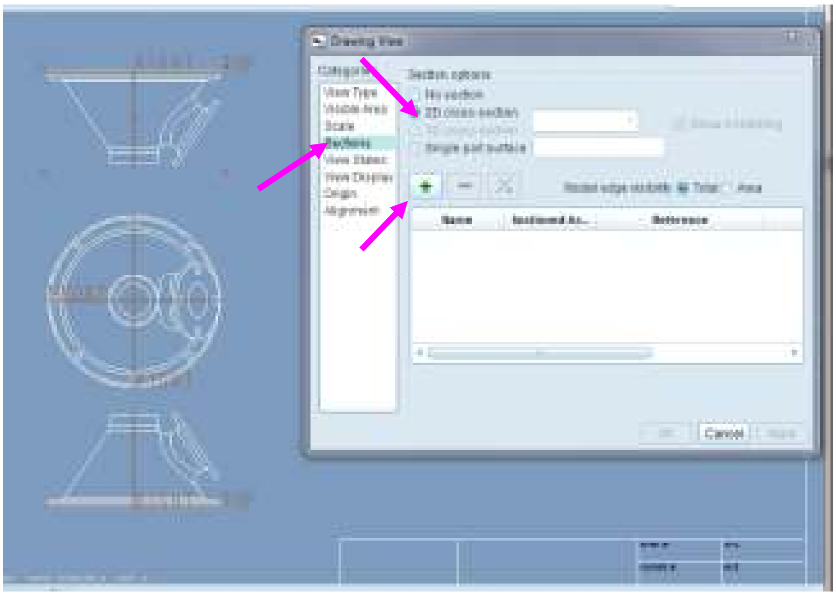


7. To move the views select the view then RMB click the “unlock view” option.
8. **Projection view**: Select the front view of the part then select the “Projection” option in the “Layout” tab ribbon.

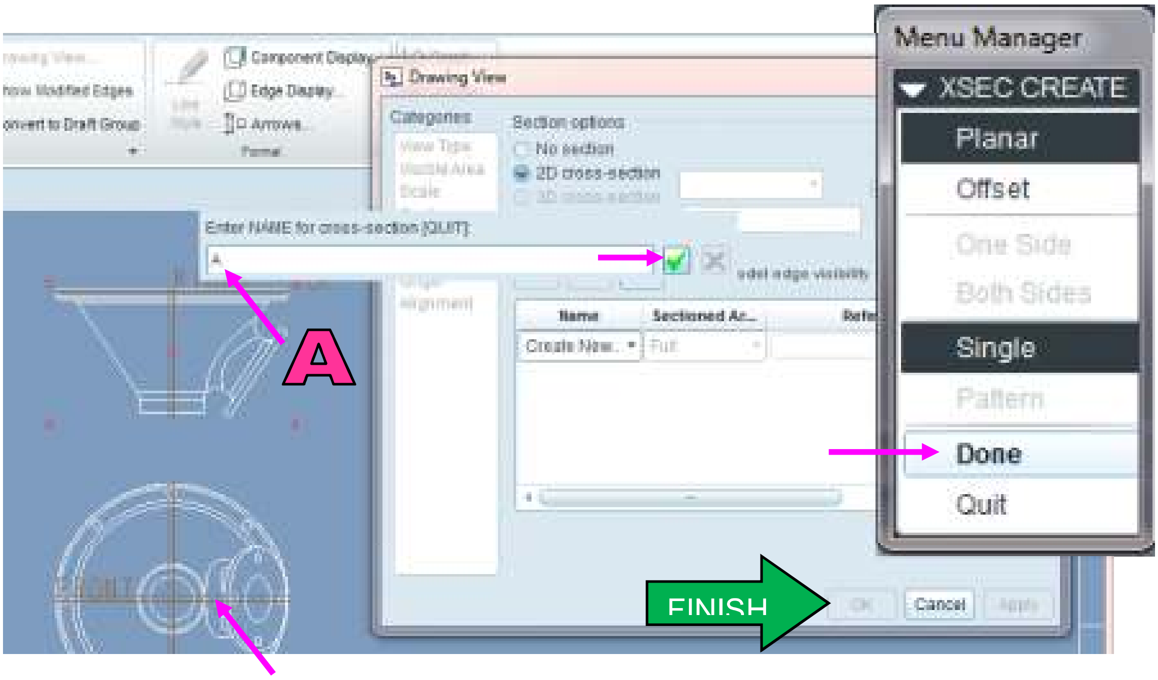


9. **Section Views**: Select the top view and repeat the projection view steps, and then move the pointer up, LMB click to drop the new view. Then double LMB click on the view to activate the options of that view.
10. Turn on/show the “Datum Planes”

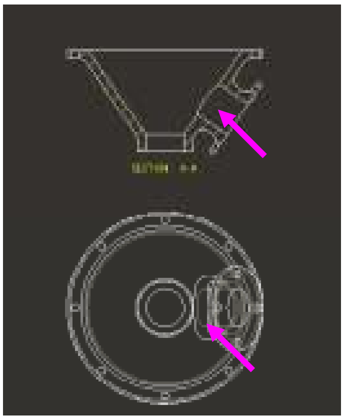


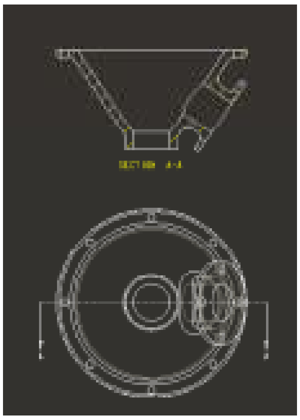


11. **Section Views**: Select the section option, in the menu manager select "Done", and then create new, then type "A" in the text box and hit the green check mark at the right of the screen. Then select the Plane option to the right, finally you can select the actual plane (horizontal) on the top view.

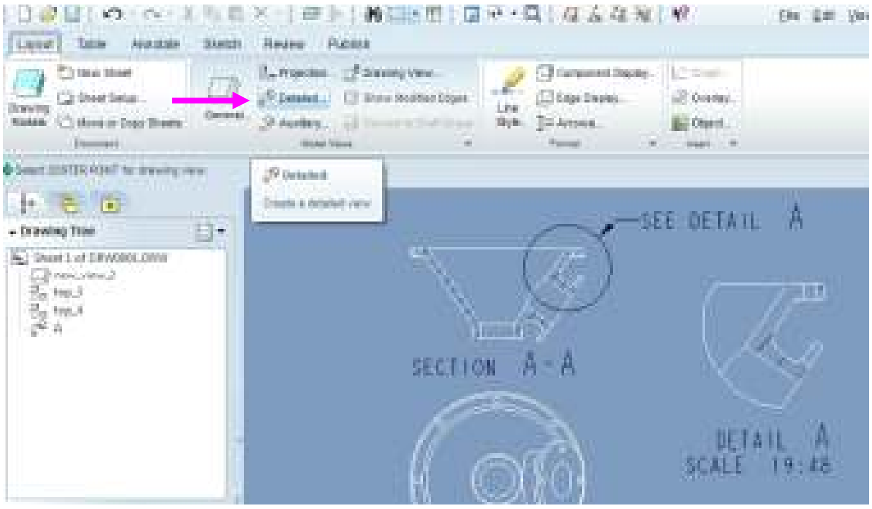


12. **Section view arrows**: You select and RMB click on the section view, then find “Add Arrows”, click on the Top view and they should appear.

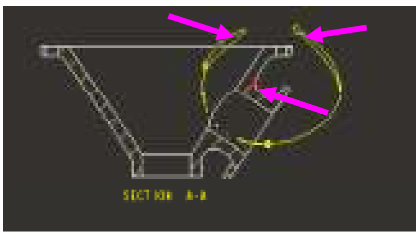


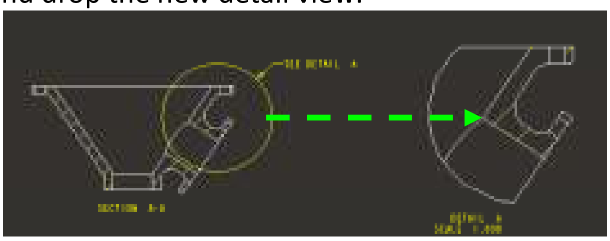


13. **Detail View**: Is added by selecting the “Detailed” tool in the “Layout” ribbon. (NOTE: Do not pre-select the view.)

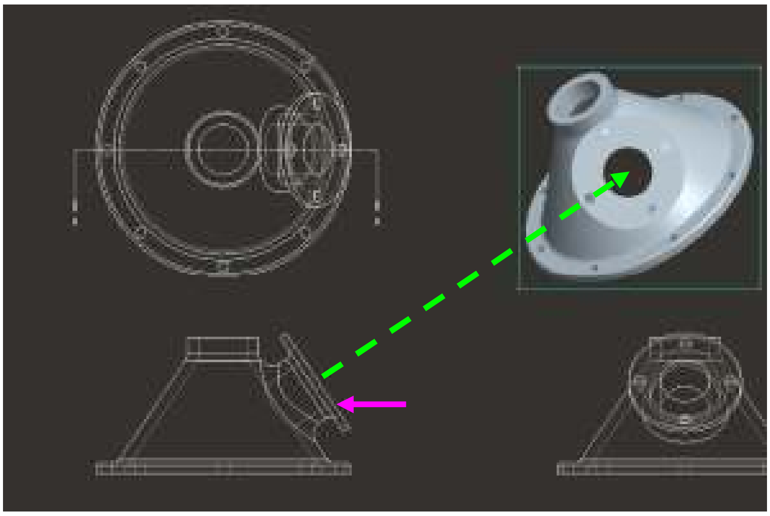


14. Select a center point for the view, and then sketch a spline around the area, and center mouse button click to close it. Then click to the right of the screen to locate and drop the new detail view.

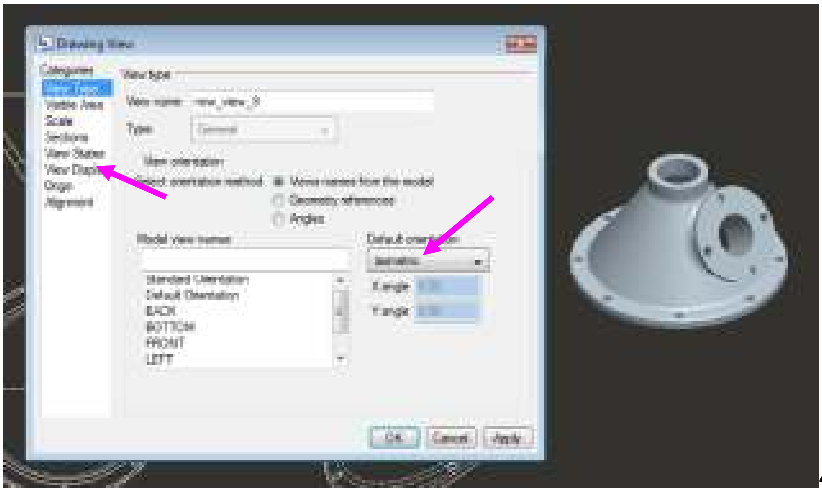




15. **Auxiliary views**: Are created by selecting the option then selecting the edge of the flange on the front view. Then select the drop point. Double click on the view to change its appearance.



16. **Isometric General Views**: Are created when you select the general view icon. Then select the location to drop the view. Double click on the view to change the appearance.



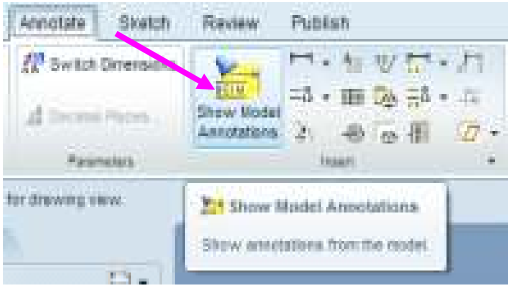
4

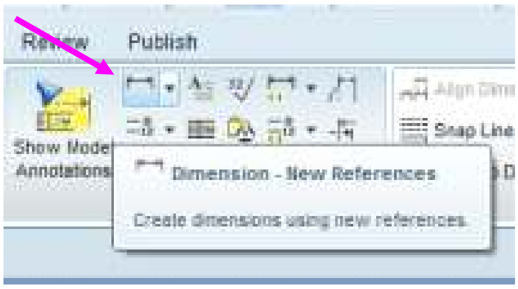
17. **View Display:** Can be used to change the views from solid to wireframe or hidden lines/HLR.

18. **Dimensions and Annotations (2 Methods)**: Select the ".

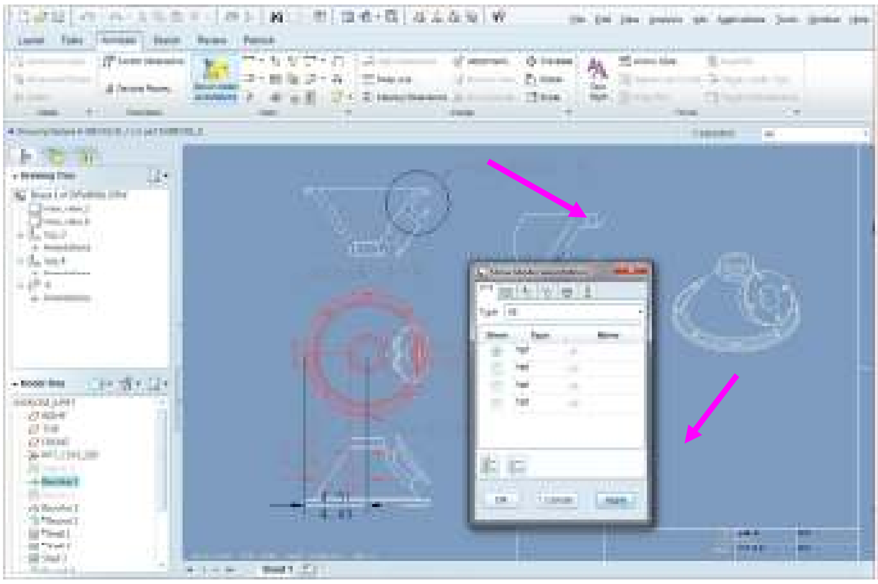
a. **Import (Show Model Annotations)** dimensions used to create the model

b. **Create (New References)** dimensions (Note: reference dimensions cannot be changed)



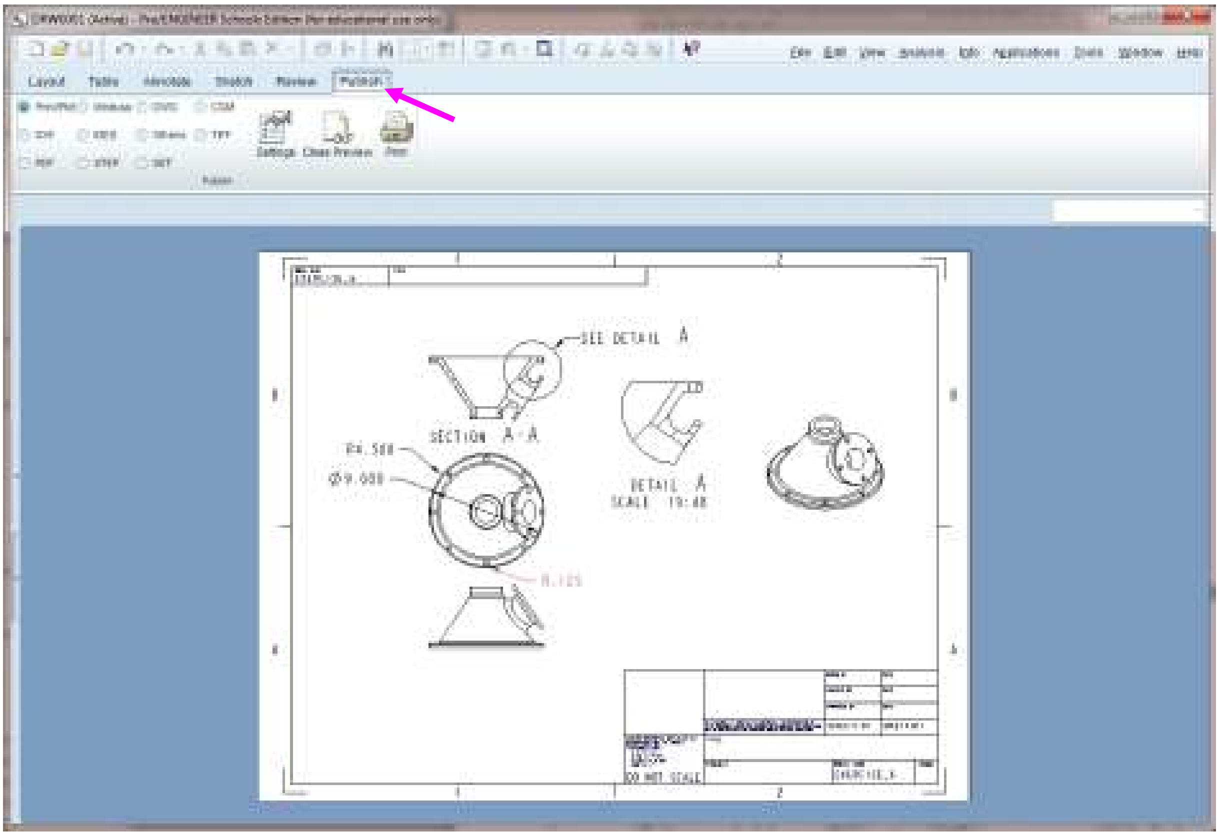


19. When importing dimensions try using the feature/view option versus inserting all the dimensions for the mode as it will cluster all them together. Feature helps reduce the cluster and yet the dimensions are editable, providing the benefit to edit the actual parts and assemblies in a bi-directional fashion from the drawing.

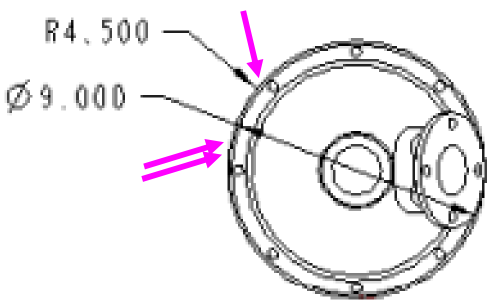


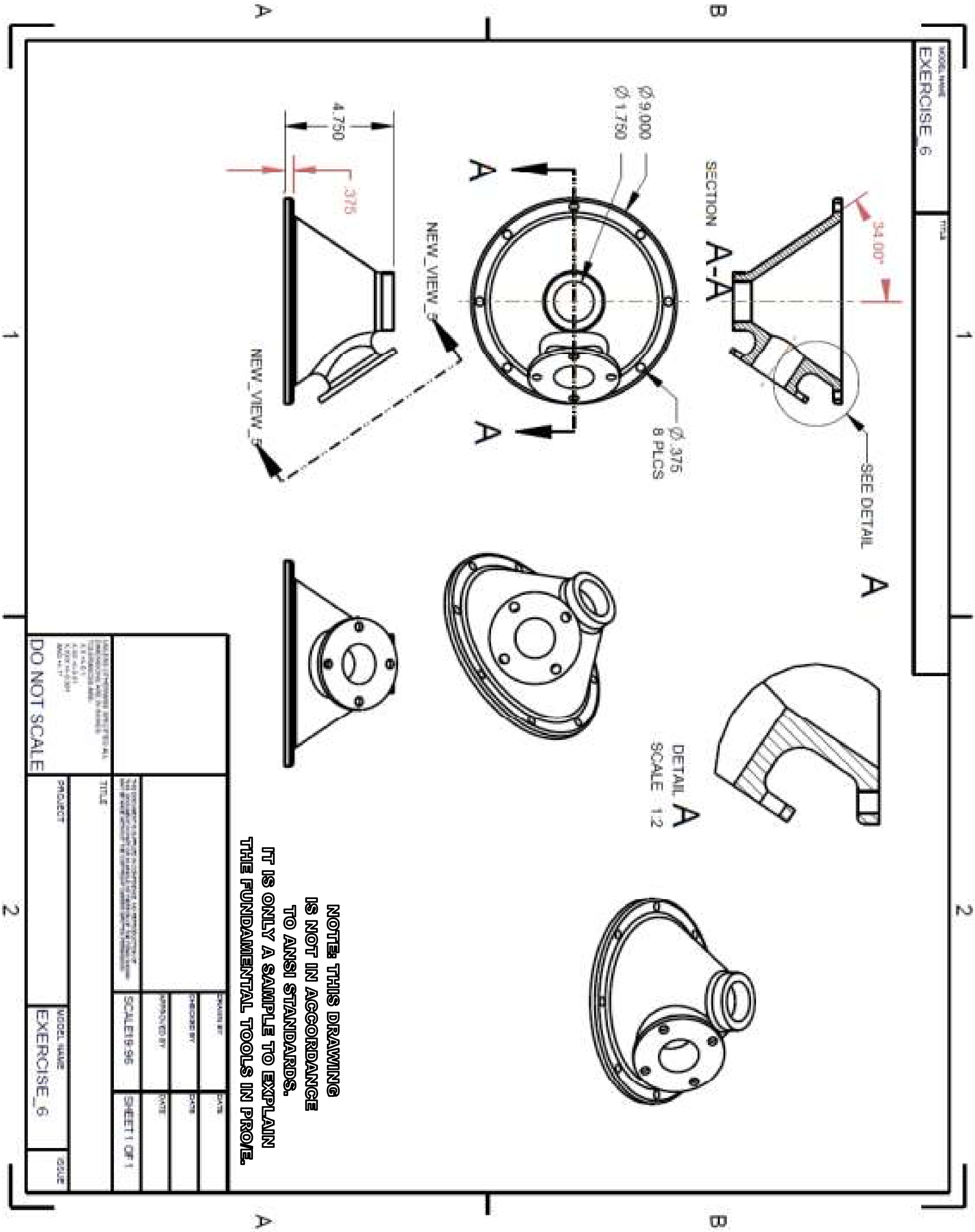
20. **Editing the Sheet:** use the "**Note**" tool to enter your name and part number.

21. **Printing:** Select the "Publish" tab for print and print preview options. Note if you find it difficult to print using the Pro Engineer printer tools select the "PDF" option and print from Adobe instead.

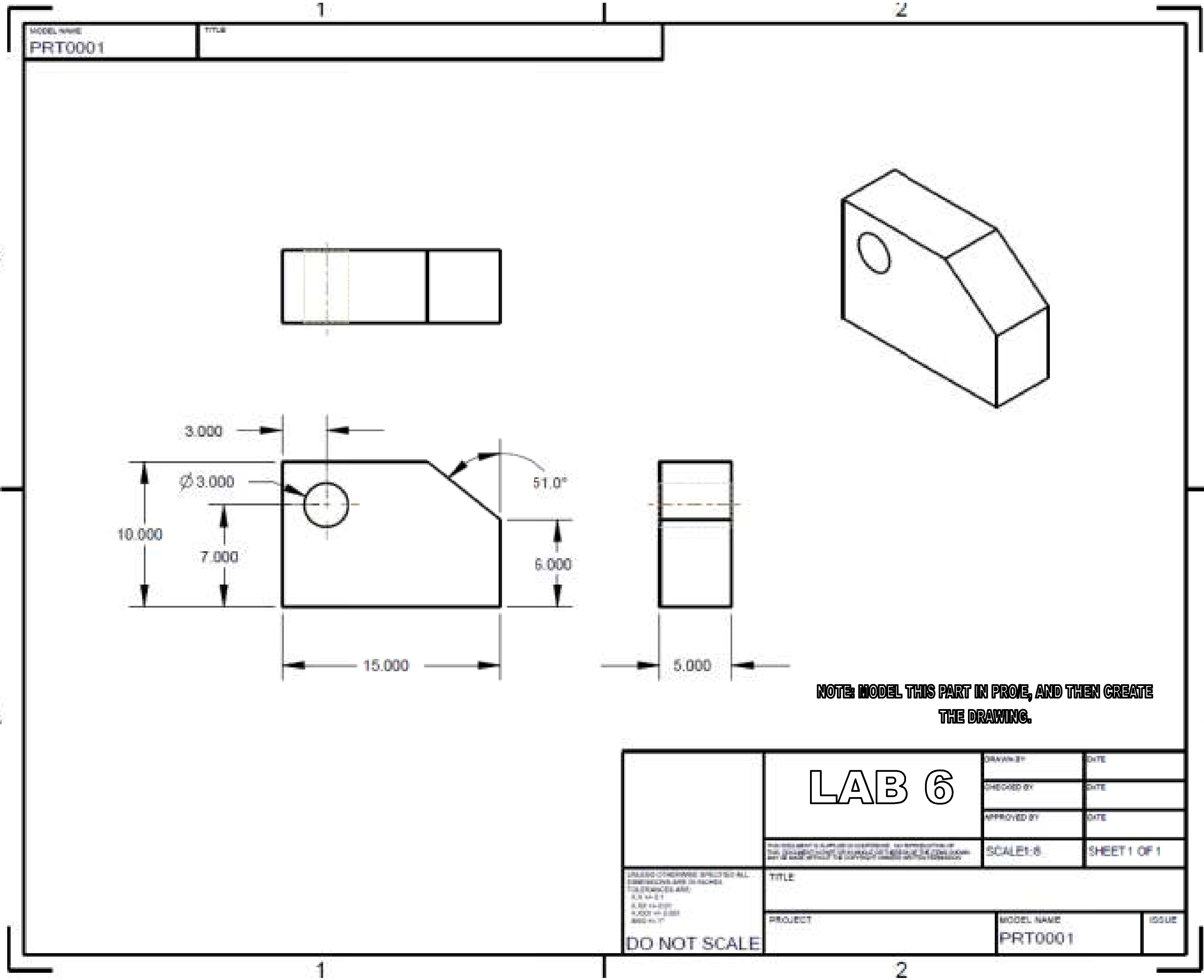


22. **Transitioning from Radius to Diameter** when dimensioning, is simply done by double clicking on the desired edge then middle click to drop a Diameter dimension. Versus a single click on an edge will result in a radius.



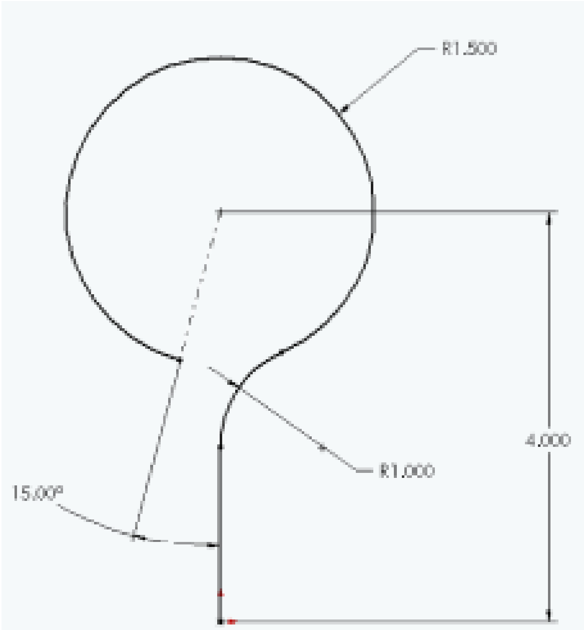


EXERCISE_6
SECTION A-A
34.00°
SEE DETAIL A
Ø 9.000
Ø 1.750
Ø .375
8 PLCS
4.750
.375
NEW_VIEW_5
DETAIL A
SCALE 1:2
NOTE: THIS DRAWING
IS NOT IN ACCORDANCE
TO ANSI STANDARDS.
IT IS ONLY A SAMPLE TO EXPLAIN
THE FUNDAMENTAL TOOLS IN PRO/E.
DO NOT SCALE
SHEET 1 OF 1

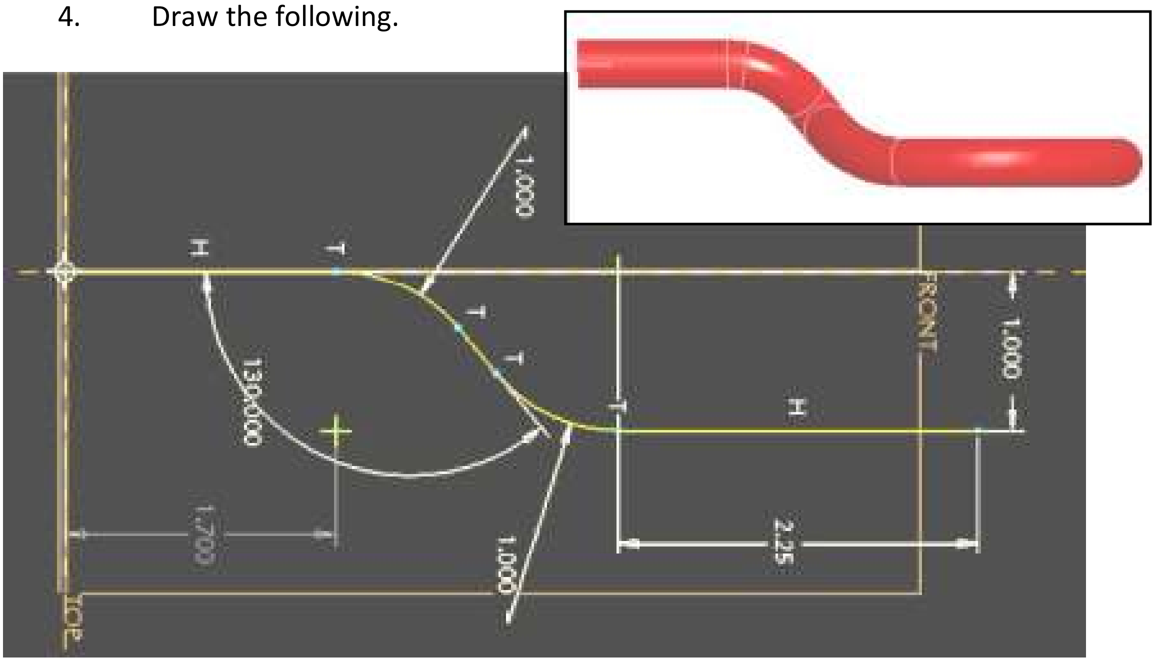

MODEL NAME
PRT0001
TITLE
3.000
∅3.000
10.000
7.000
51.0°
6.000
15.000
5.000
NOTE: MODEL THIS PART IN PRO/E, AND THEN CREATE THE DRAWING.
LAB 6
DRAWN BY
DATE
CHECKED BY
DATE
APPROVED BY
DATE
SCALE1:8
SHEET 1 OF 1
TITLE
PROJECT
MODEL NAME
PRT0001
ISSUE
DO NOT SCALE

EXERCISE 7
Projected Curves and Sweeping

1. Sketch this on the "Front" plane.

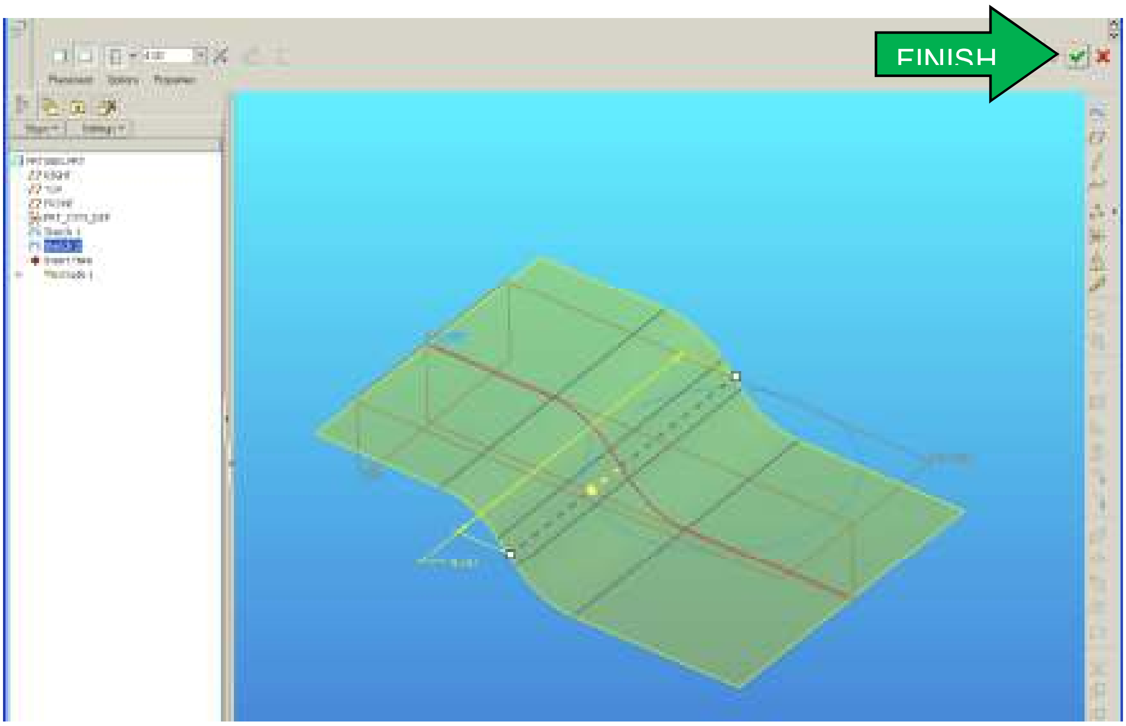


2. Hit "Done" to exit the sketch.
3. Select the "Right" plane and start a sketch on it.
4. Draw the following.

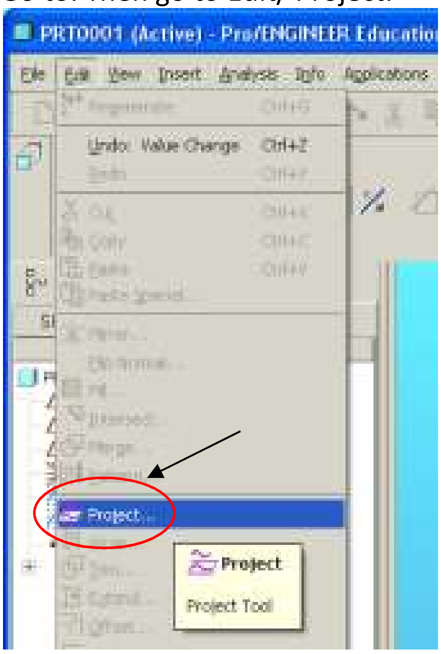


5. Hit "Done" to exit the sketch.

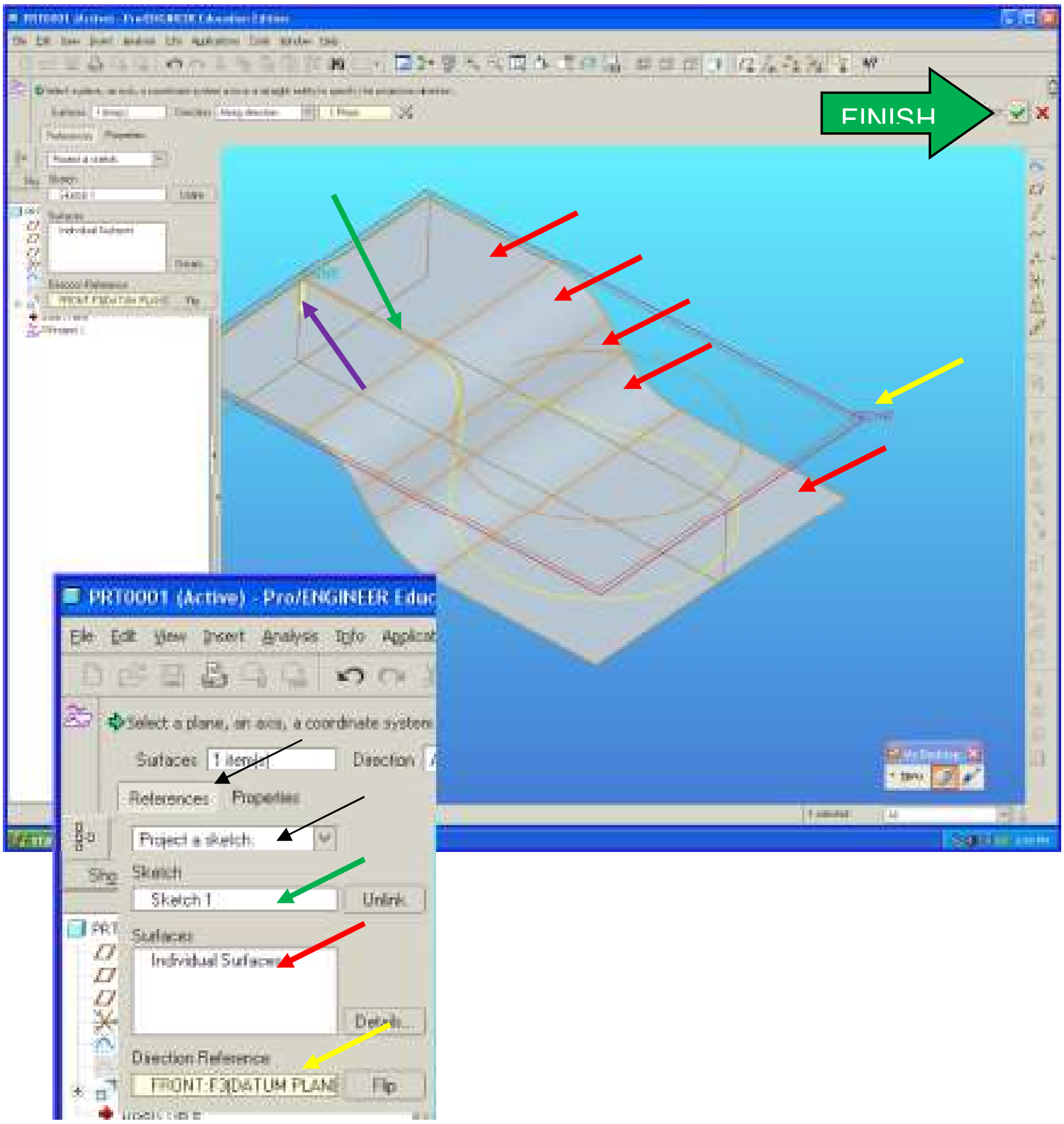
6. Extrude the curve Mid-Plane 4”. It should extrude as a surface. Hit the green check to apply.



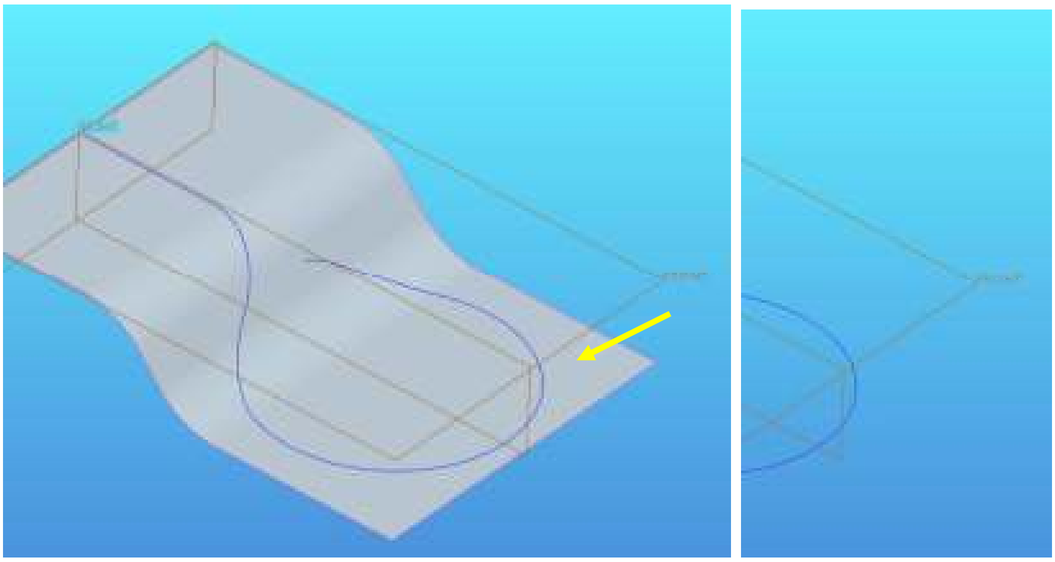
7. Go to. Then go to Edit/ Project.



8. Select References/Project a sketch/**Sketch1-Curve that you drew**/**CTRL select all surfaces**/**Select the Front Datum**/ **Flip the arrow**

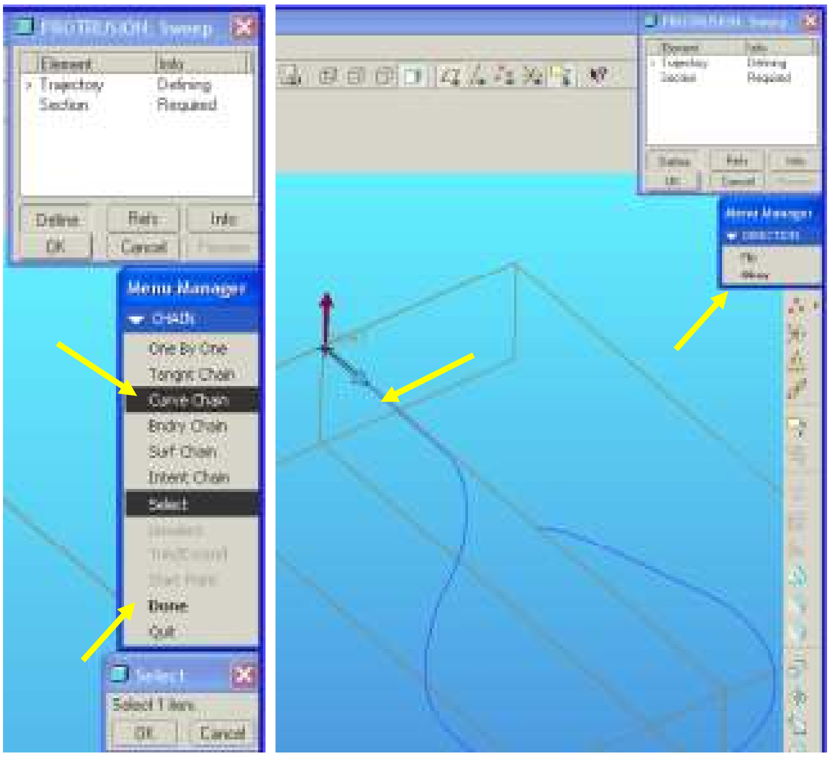


9. Select the surface and RMB click to find the “Hide” option.

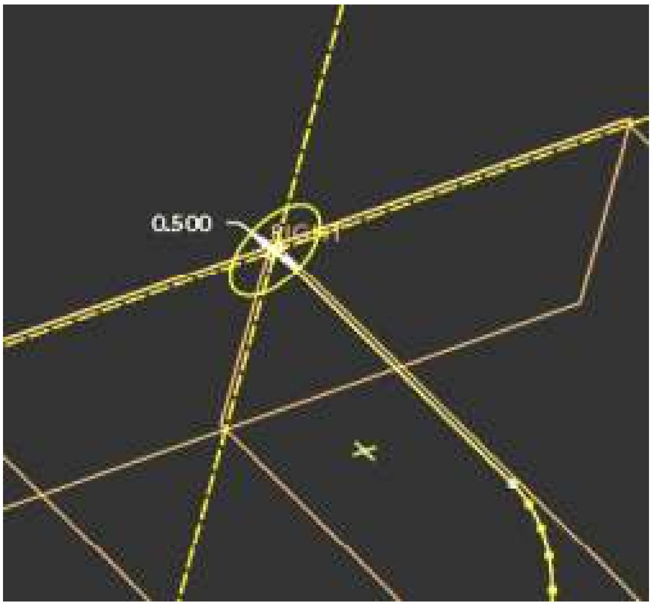


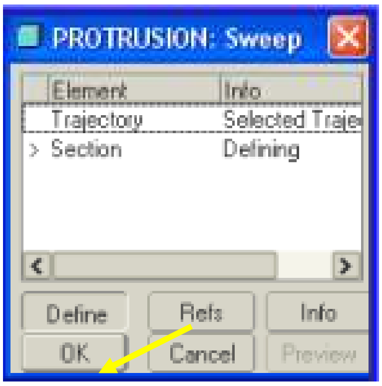
10. You should now have a single 3 Dimensional curve.

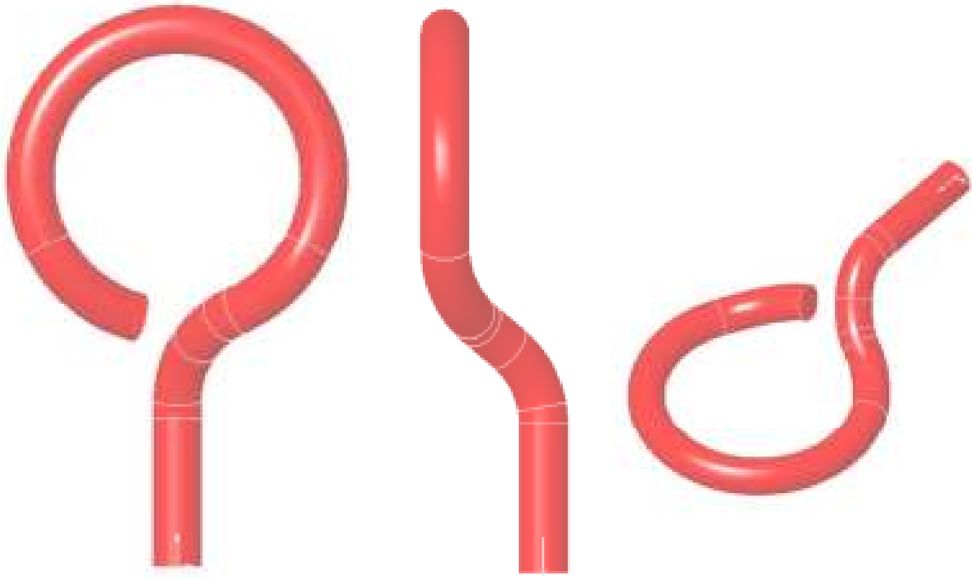
11. Hit the “Done” icon and Sweep/Protrusion using the curve as the Path and the circle as the Profile.
12. Also select: “SelectTraj/Curve Chain/Select All/Done/Done”



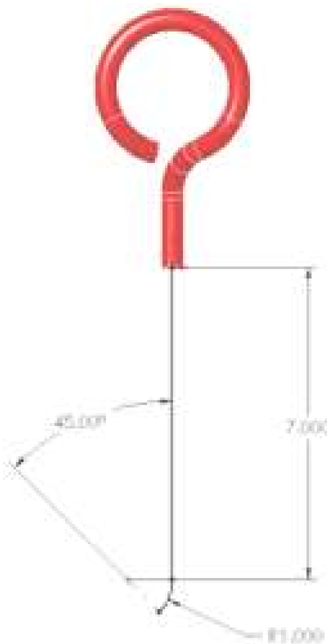
13. Draw a .500” circle at the intersection/end of the curve. Select “Done” and “OK”







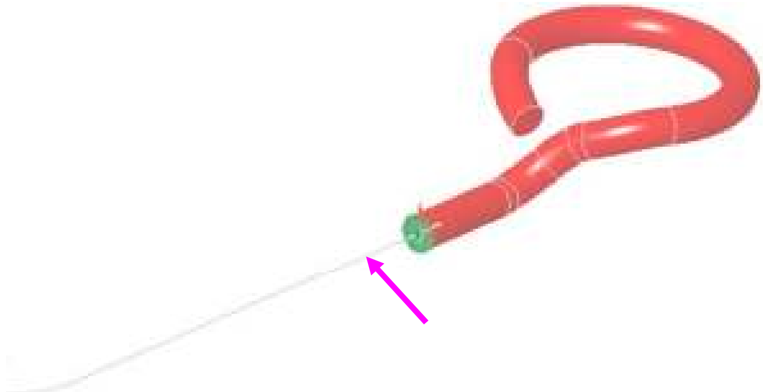
17. Start a sketch on the “Front” plane. Draw the following.



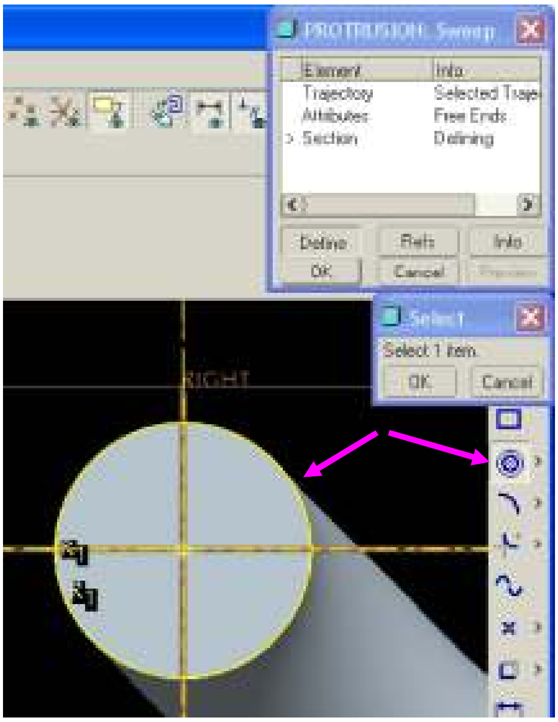
18. Select “Done” to exit the sketch.

19. Select Sweep/Protrusion using the curve as the Path and the circle as the Profile.

20. Also select: “SelectTraj/Curve Chain/Select All/Done/Done”

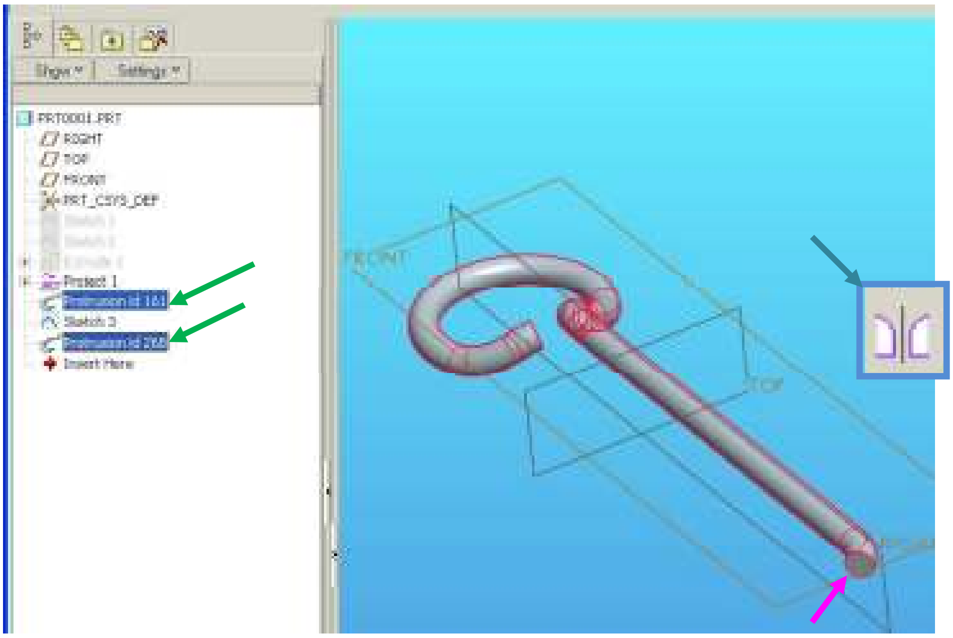


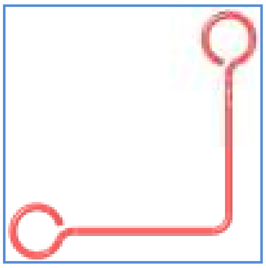
21. Select the concentric circle icon (buried under the circle tool). Select the edge of the face and click over the edge to assume an “Equal” diameter (R1/R1)



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

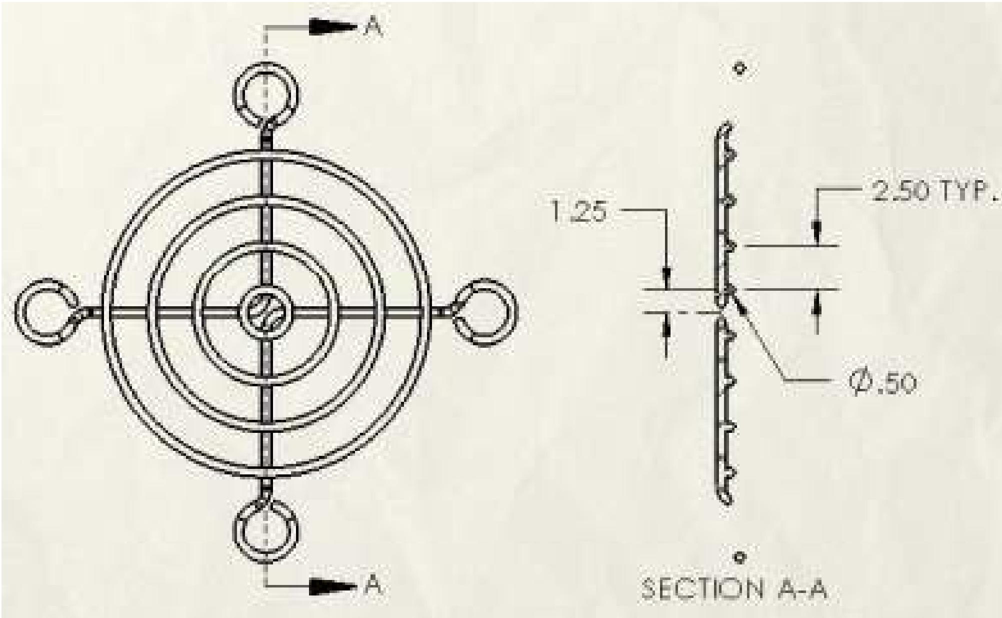
22. CTRL Select both **Protrusions** from the Feature Tree, and then select the **Mirror** icon. Then select the **end face** of the body.



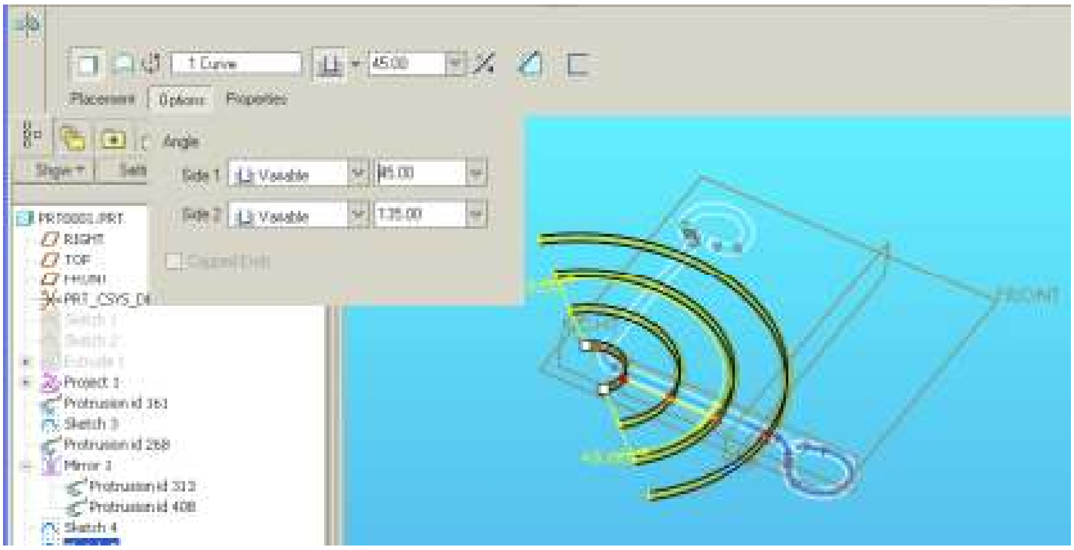


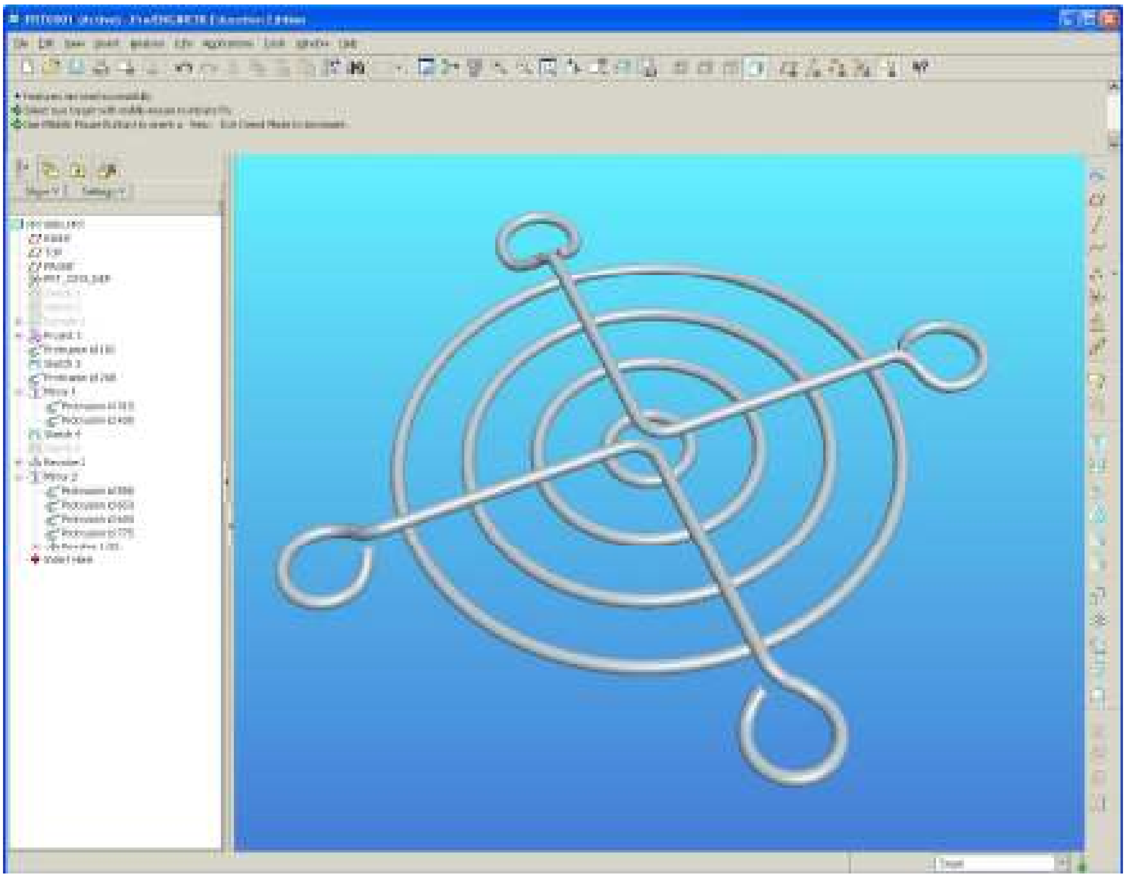
23. Now using the tools you have learned over the past 5 weeks finish the remainder of the model.

Hints to complete the model…

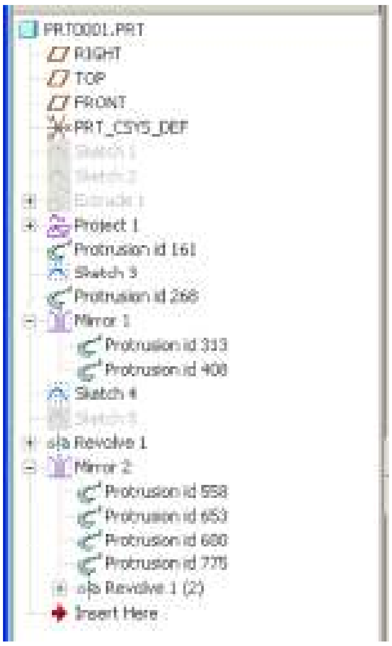


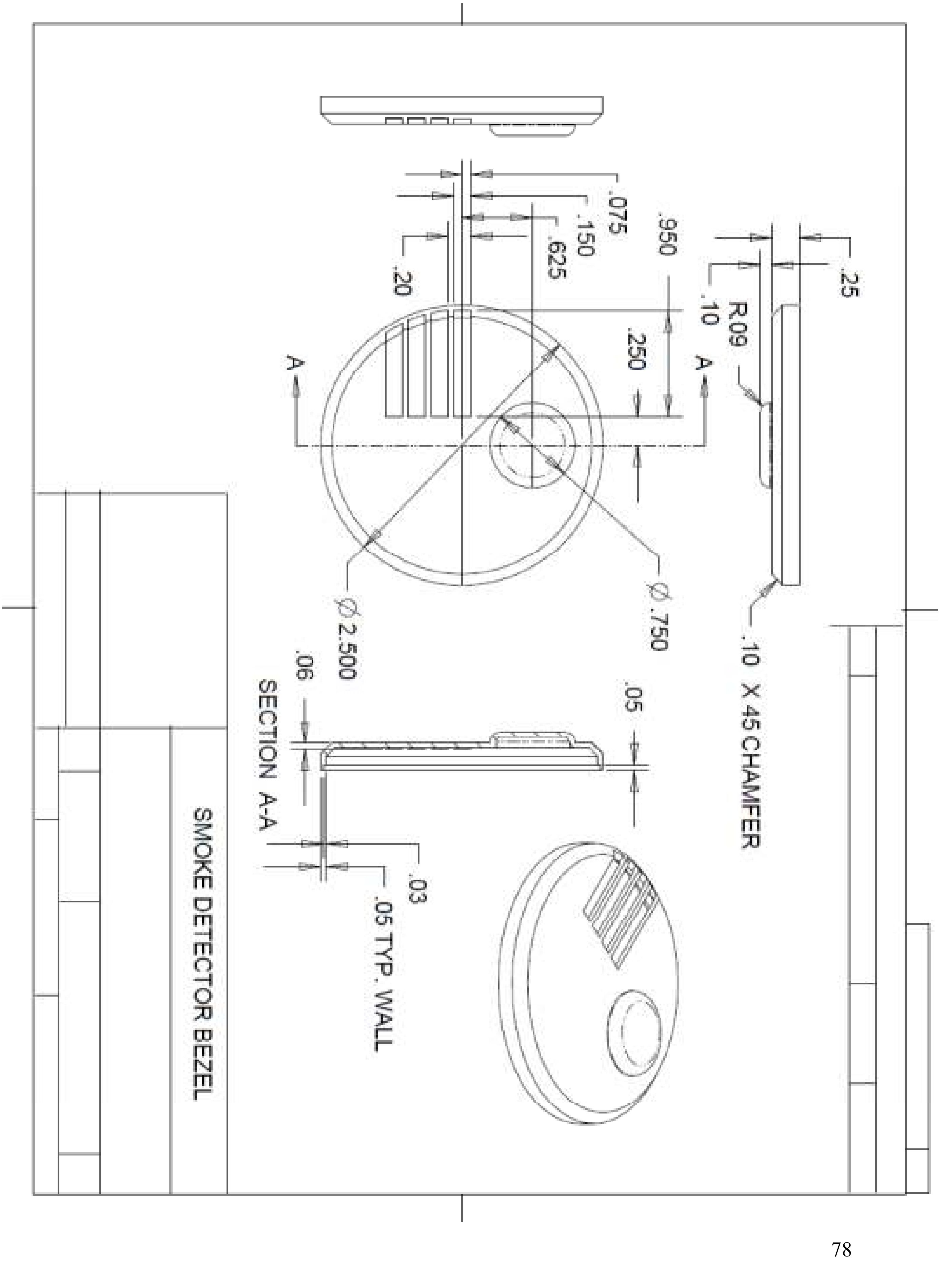
24. Revolve “Two Directions”





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

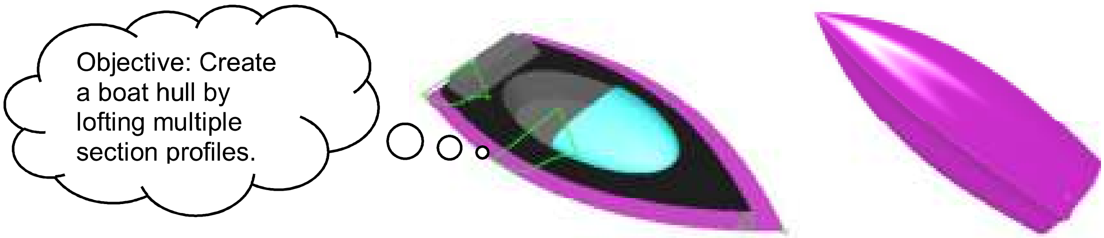



.075
.150
.625
.20
.950
.250
.25
.10
R.09
A
A
Ø .750
Ø 2.500
.10 X 45 CHAMFER
.05
.06
SECTION A-A
.03
.05 TYP. WALL
SMOKE DETECTOR BEZEL

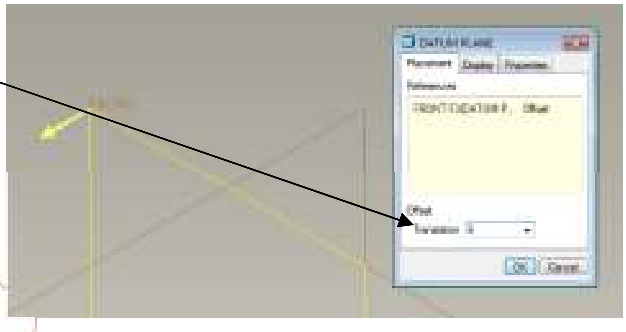
EXERCISE 8

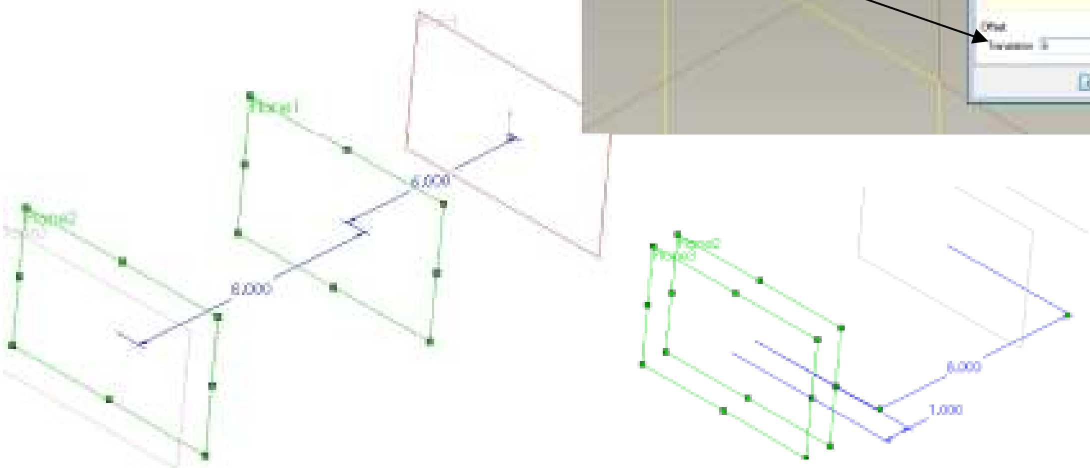
Swept Blend/Lofting

***Swept Blends** create a feature by making transitions between profiles. A Swept blend can be a base, boss, cut, or surface.*

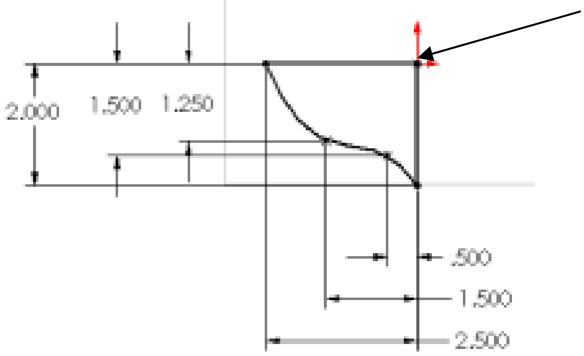


1. Create 4 datum planes beginning from the "Front" plane and offset from each other as shown.
   DTM Plane 1 – 6.00"
   DTM Plane 2 – 8.00"
   DTM Plane 3 – 1.00"

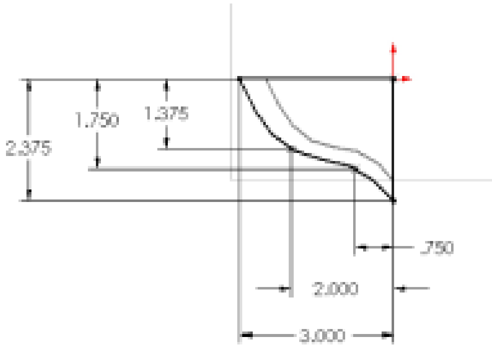




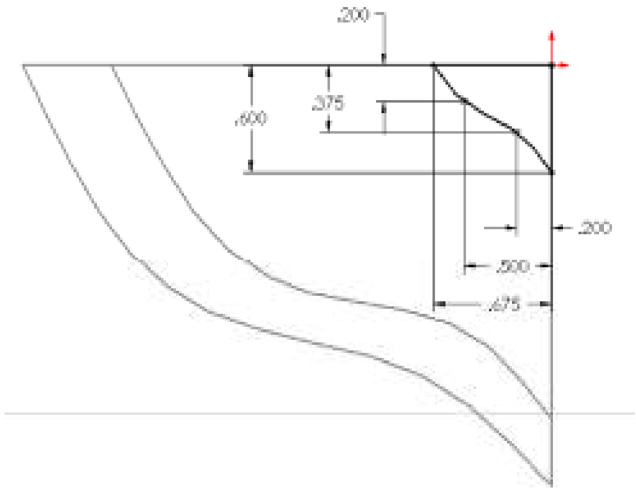
2. Sketch 1 on the "Front" plane should look like this… use the Spline tool.



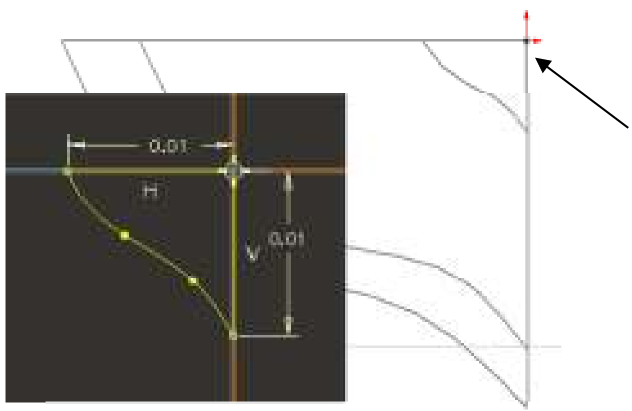
3. Sketch 2 on “DTM 1” should look like this…



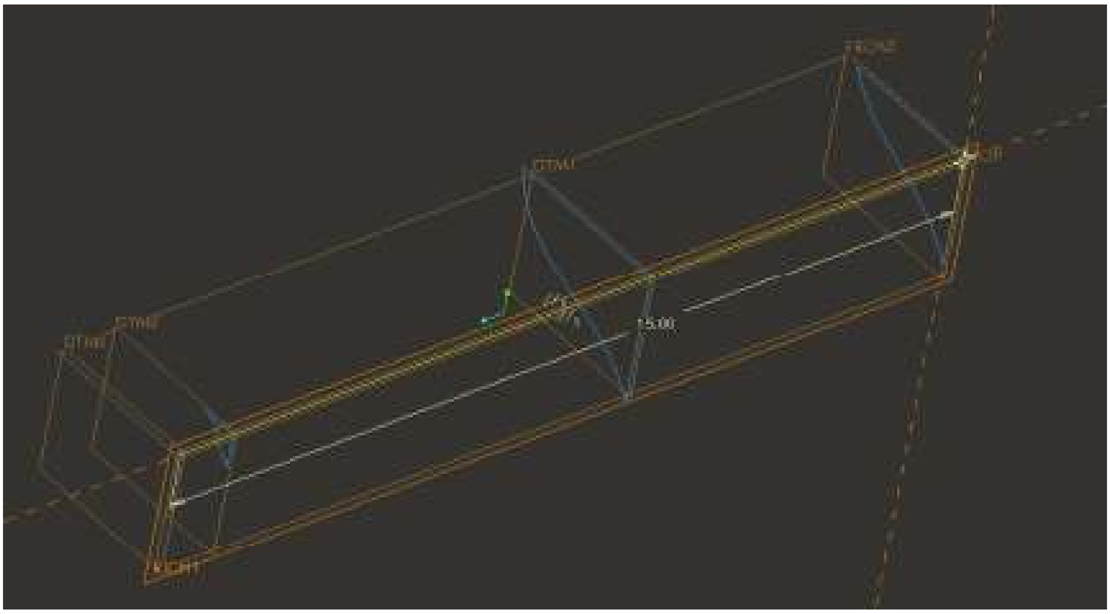
4. Sketch 3 on “DTM 2” should look like this…



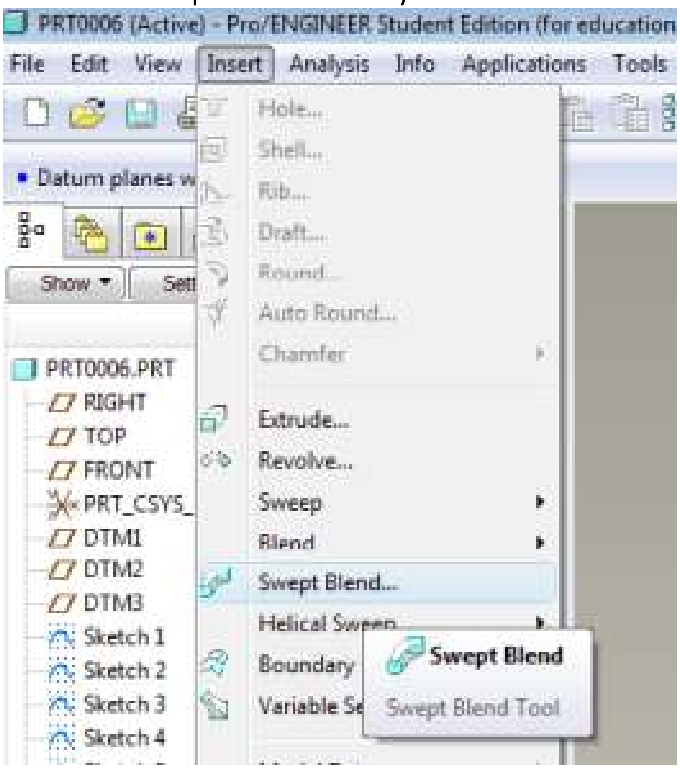
5. Sketch 4 on “DTM 3” should look like this… A (.010”) profile at the origin.



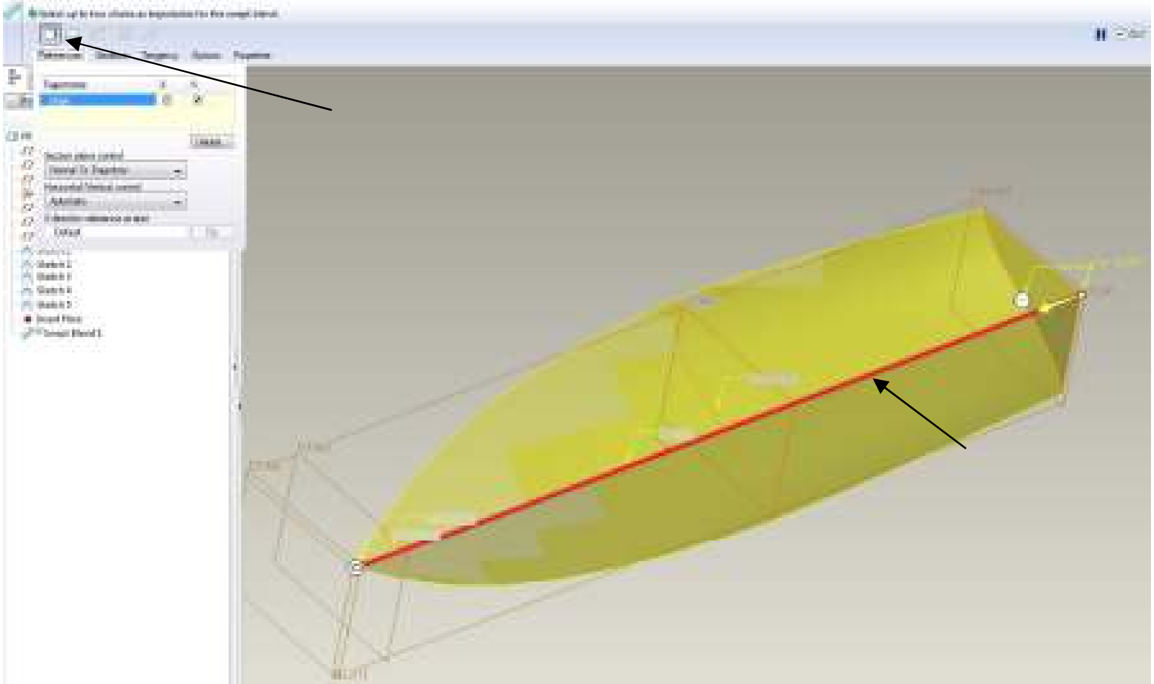
6. Select the Right datum plane ad draw a horizontal line at the origin and dimension it 15” long.



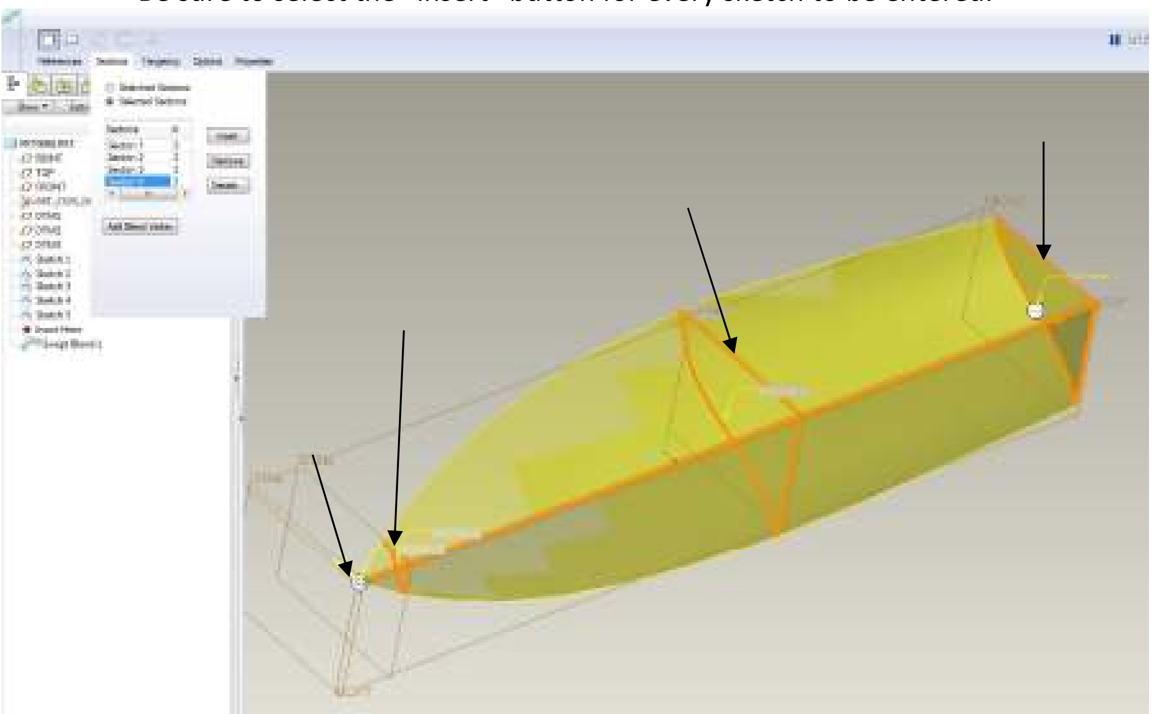
7. Swept Blend: Exit any sketches and select Insert/Swept Blend.



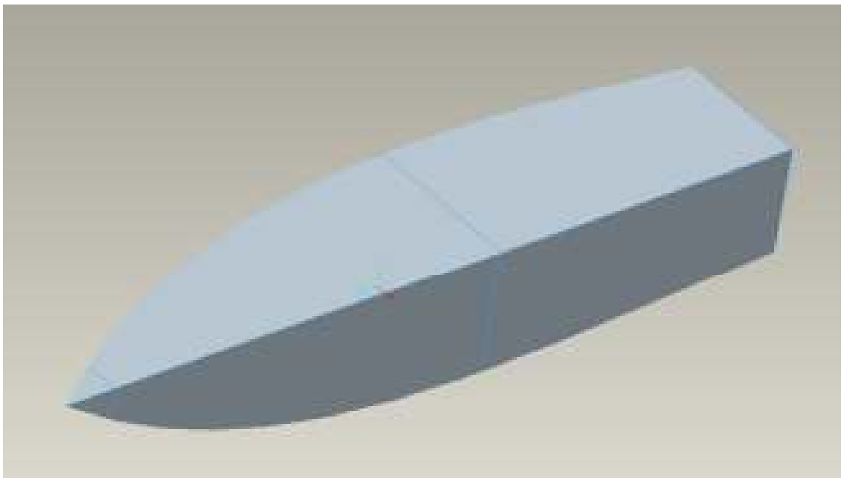
8. **References Trajectories:** Select the 15” line. Select the “Solid” option.



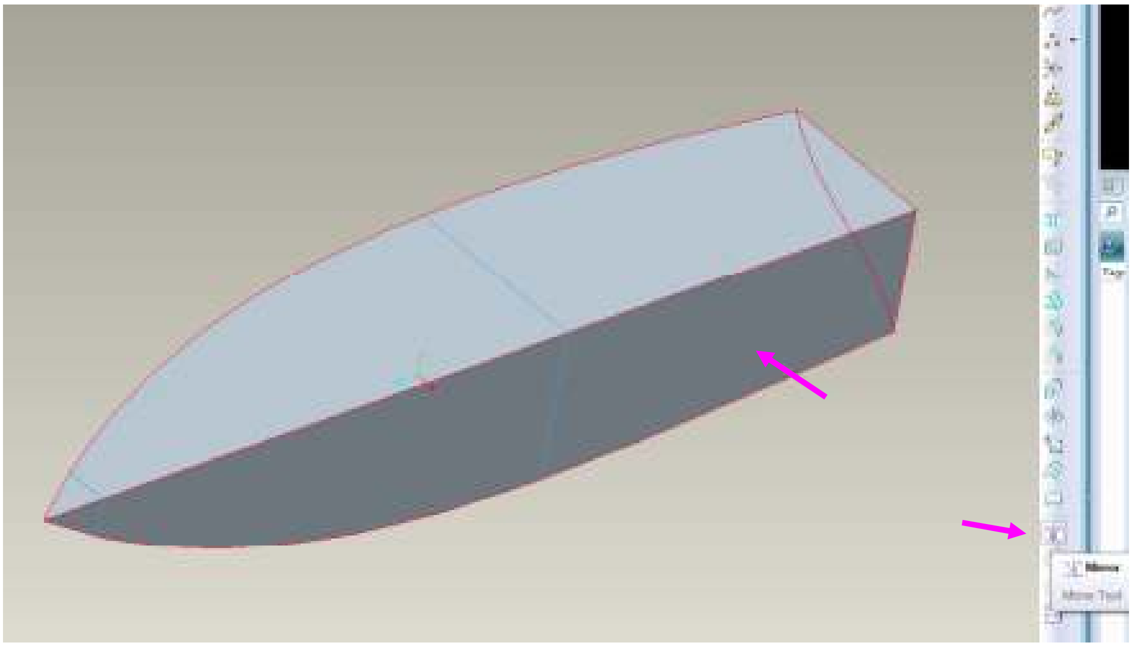
9. **Sections/Selected Sections**: Select the 4 sketches in order from back to front. Be sure to select the “Insert” button for every sketch to be entered.



10. You should have ½ a boat hull now…

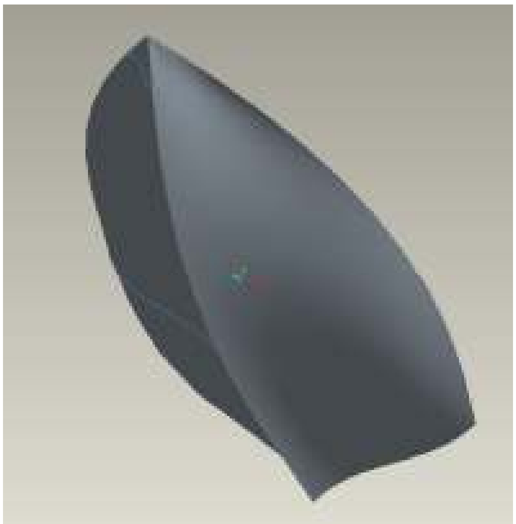


11. Use the Mirror feature and select the flat side face as the plane to mirror from.

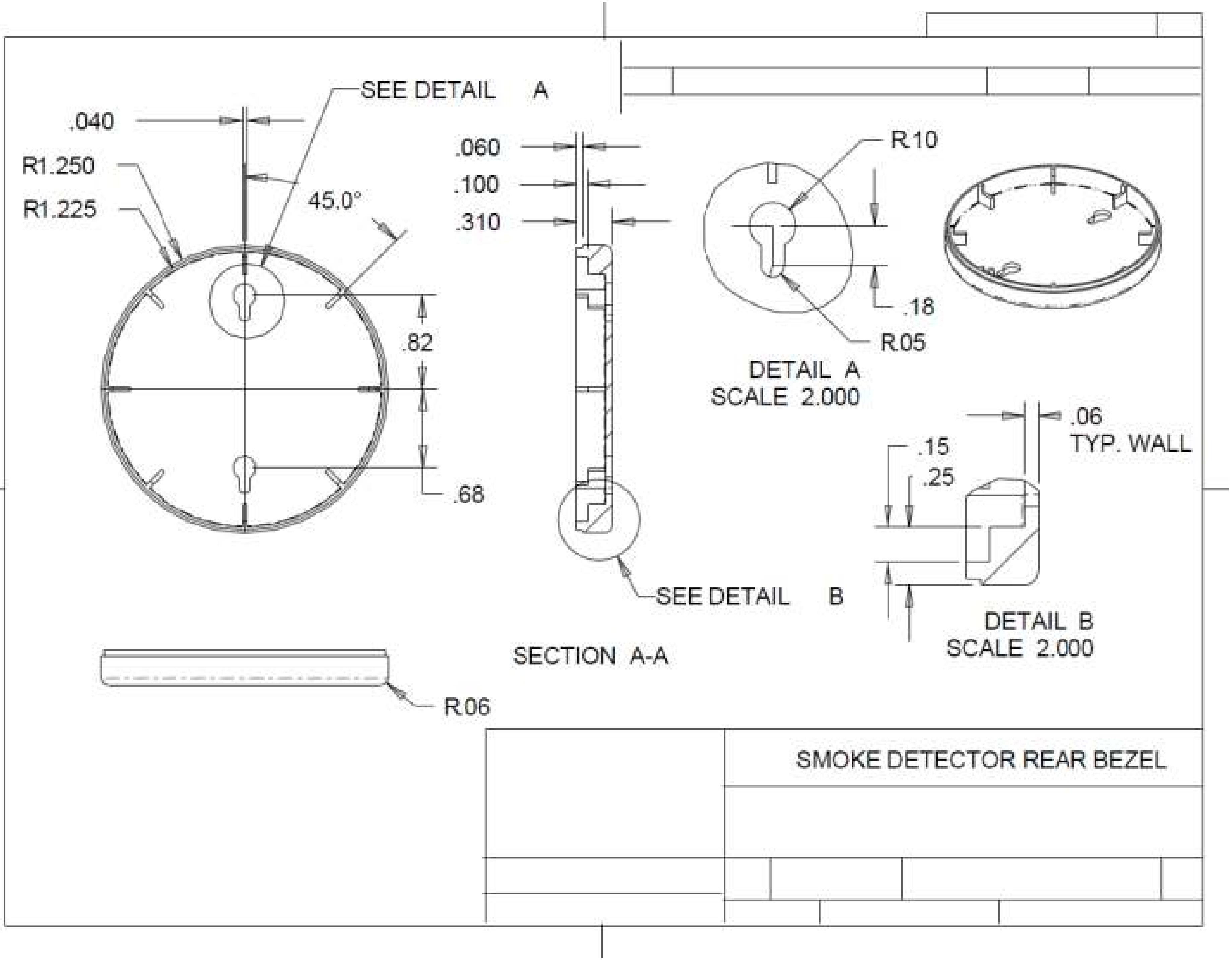


12. Select the hull one more time and hit the green check mark to apply.

13. You are finished with the boat Hull.



14. (Optional) Now dress it up for the contest…

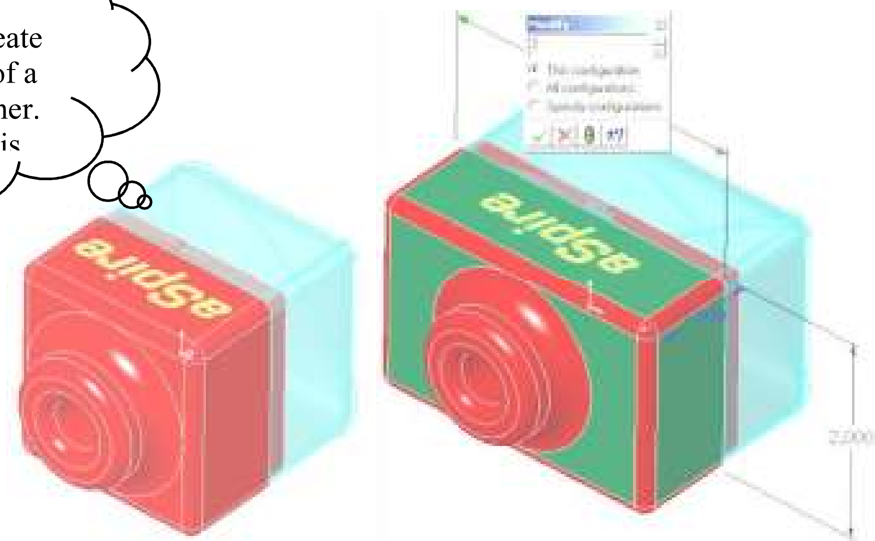

SEE DETAIL A
.040
R1.250
R1.225
45.0°
.82
.68
R06
.060
.100
.310
SEE DETAIL B
SECTION A-A
R10
.18
R05
DETAIL A
SCALE 2.000
.06
TYP. WALL
.15
.25
DETAIL B
SCALE 2.000
SMOKE DETECTOR REAR BEZEL

EXERCISE 9 V (WATCH VIDEO)

Top-Down Assembly Modeling

Top-Down Assembly Modeling *is creating parts inside an assembly.*

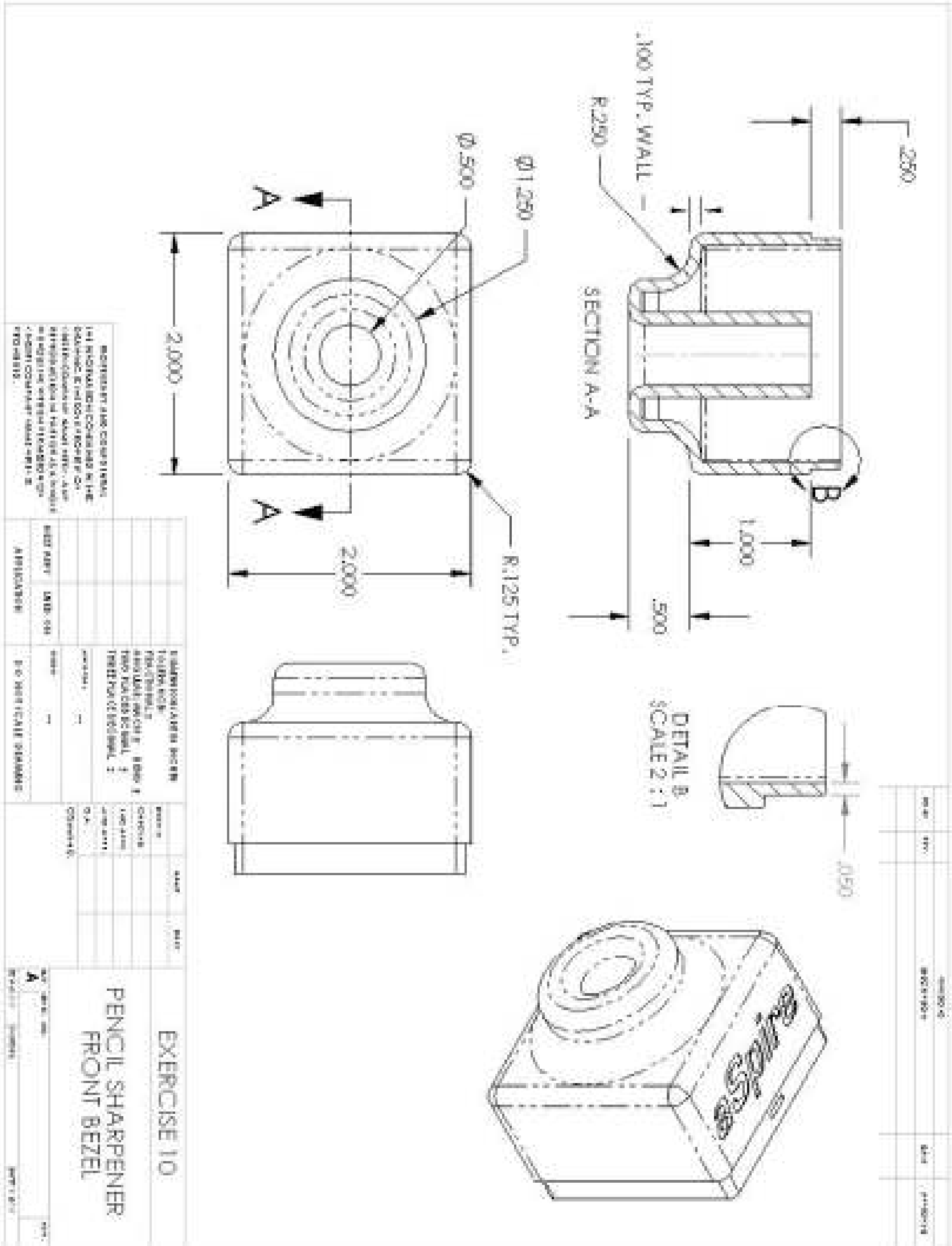
Objective: Create an assembly of a pencil sharpener. If the width is



1. Create a new assembly file.

2. Go to the Create icon.

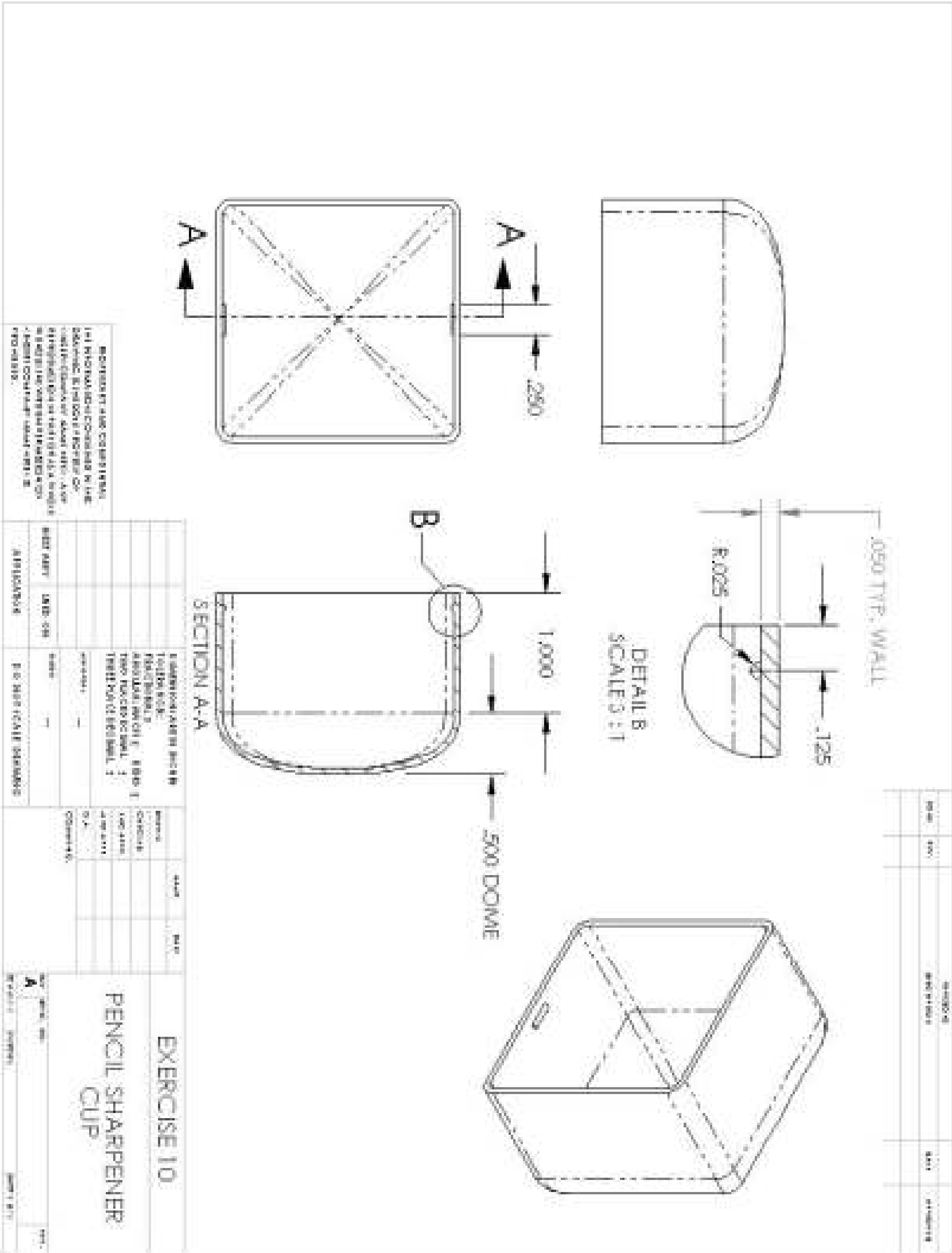
3. Save it as E9_Front and drop it on the “Front” plane. Create the following part from the drawing.

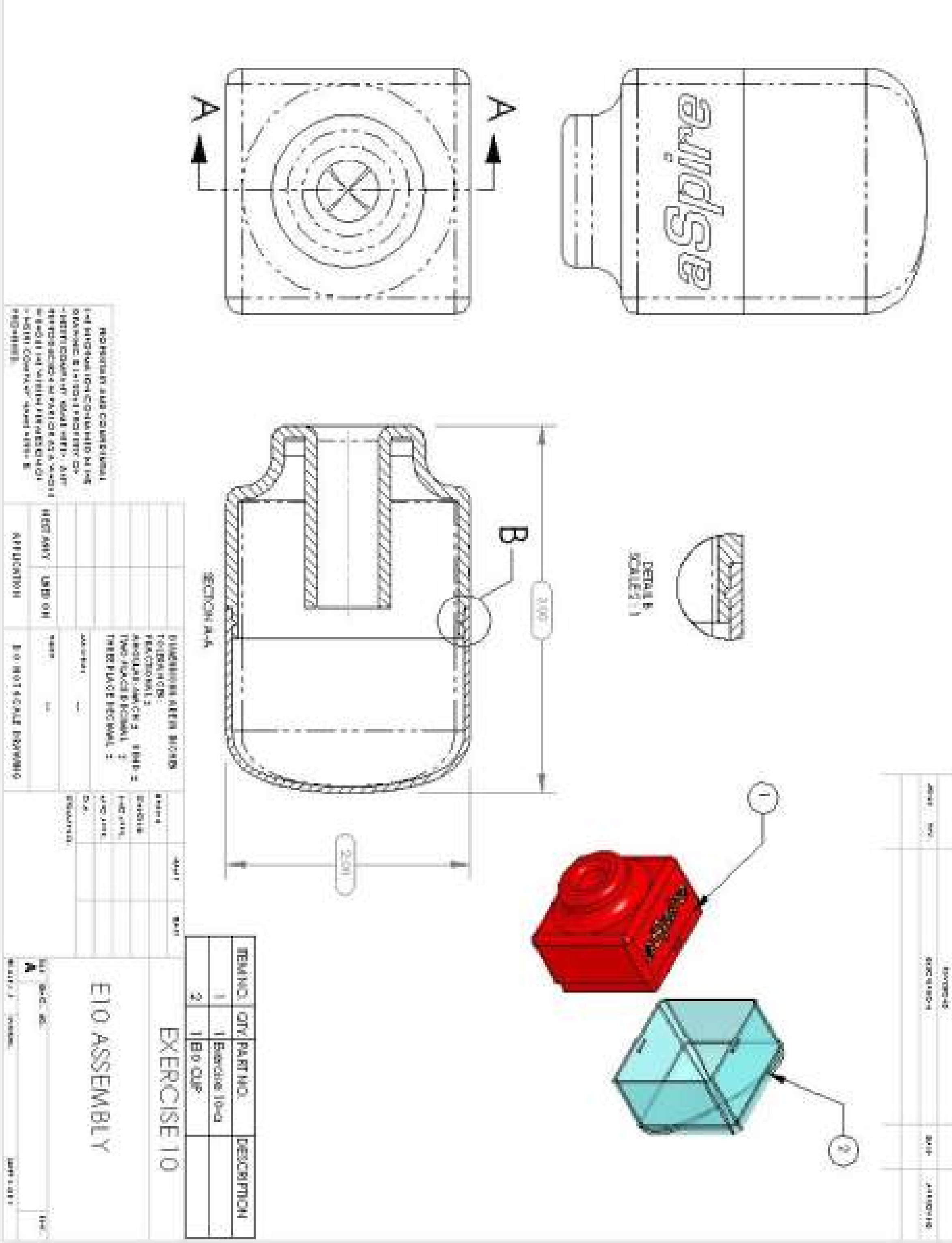


4.	When finished select the ***Activate*** option to exit part editing mode.

5.	Insert another new component and save it as E9_Reservoir.

6. Create the following model in the context of the assembly-using offset or convert entities from the E9_Front model.

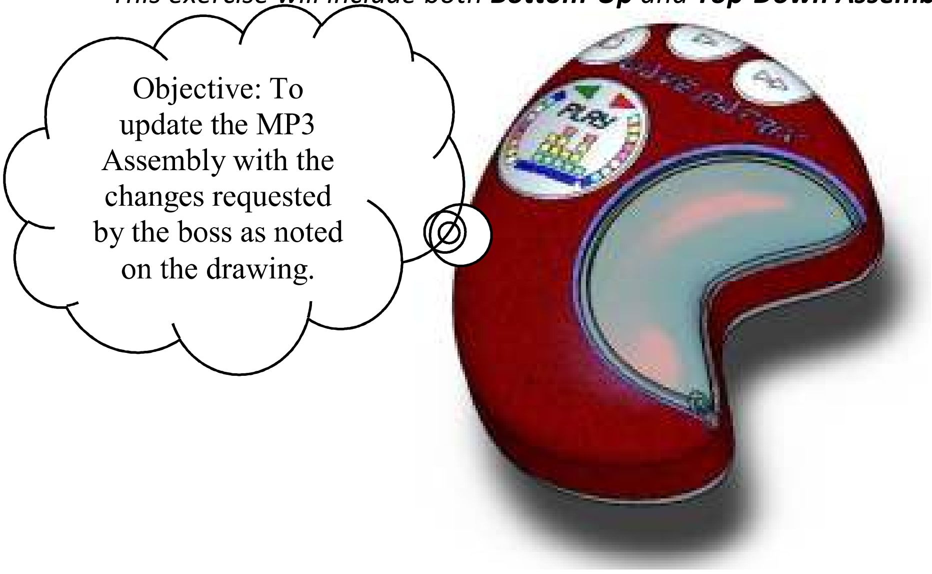



EXERCISE 10
E10 ASSEMBLY
SECTION A-A
ITEM NO.
QTY.
PART NO.
DESCRIPTION

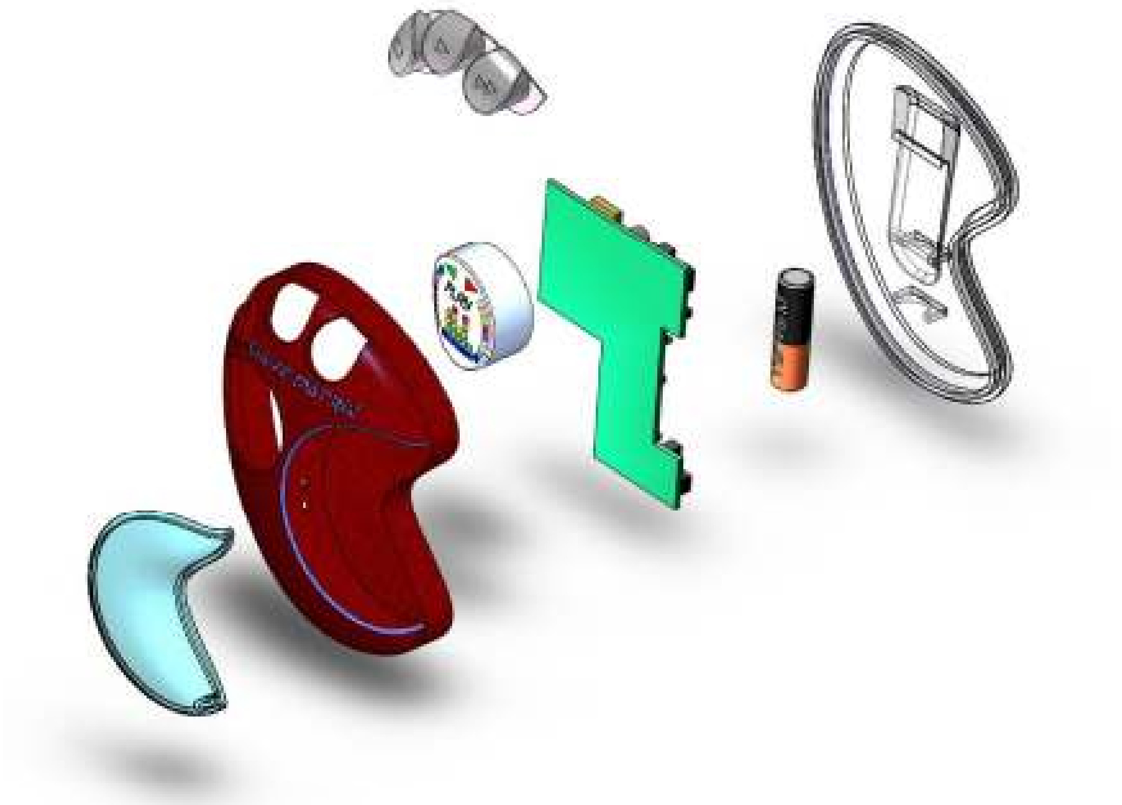
EXERCISE 10 V

Assembly Editing

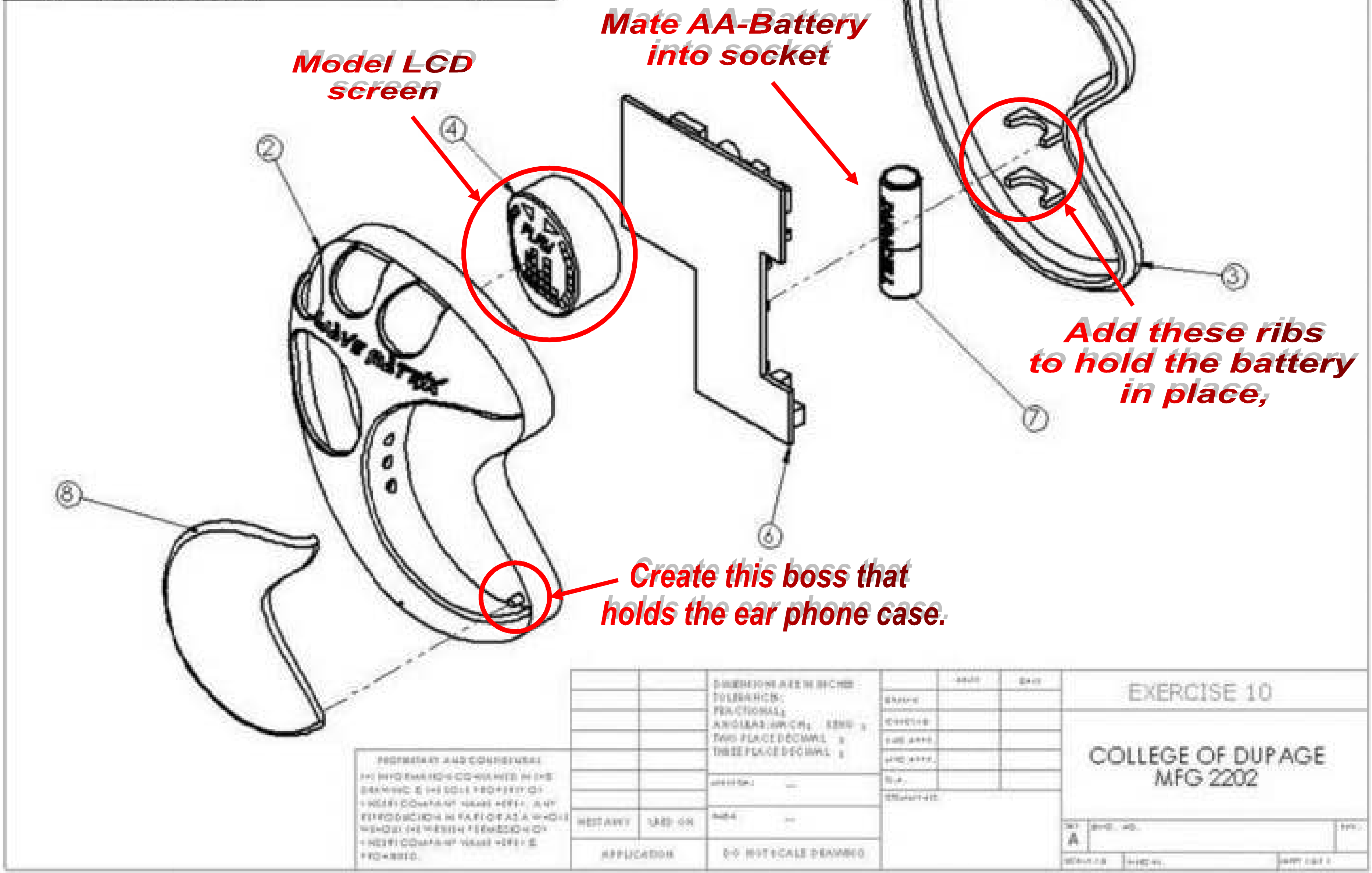
*This exercise will include both **Bottom-Up** and **Top-Down Assembly Modeling**.*



7. Open the file called E10_asm assembly and modify according to the instructions noted on the drawing provided. You will have to mate the Battery part file.

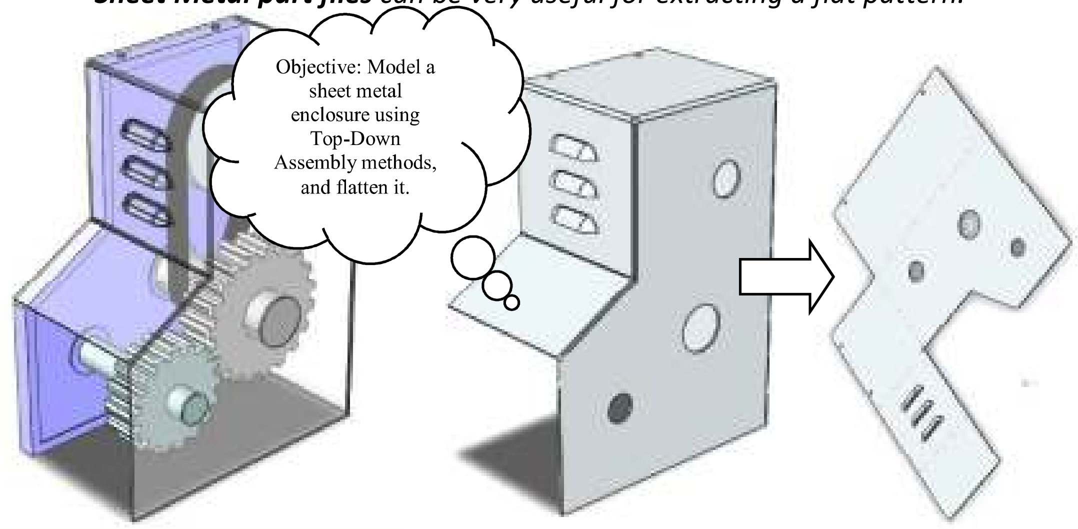


ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Original		1
2	Front Bezel		1
3	Rear Bezel		1
4	LCD		1
5	Button Array		1
6	PCB		1
7	AA Battery		1
8	Ear Phone Case		1

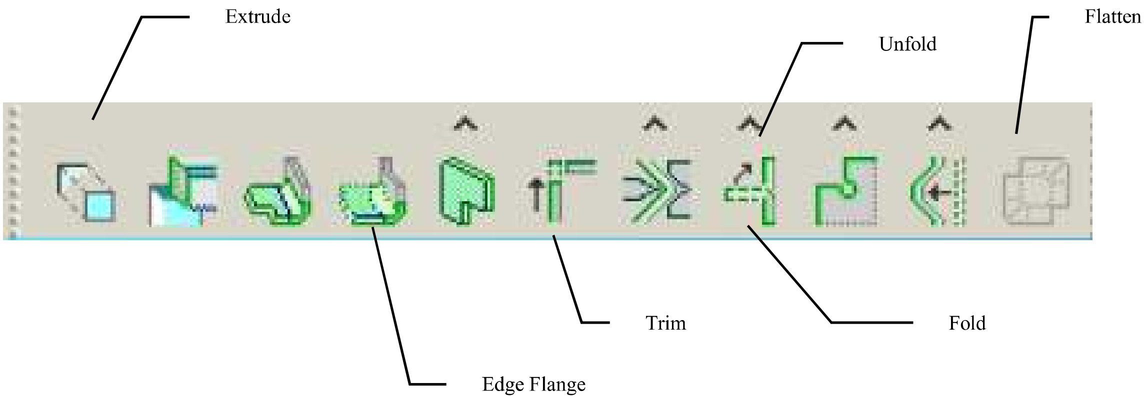


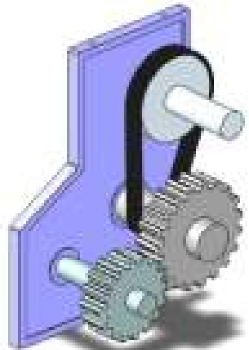
EXERCISE 11
Sheet Metal Design

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

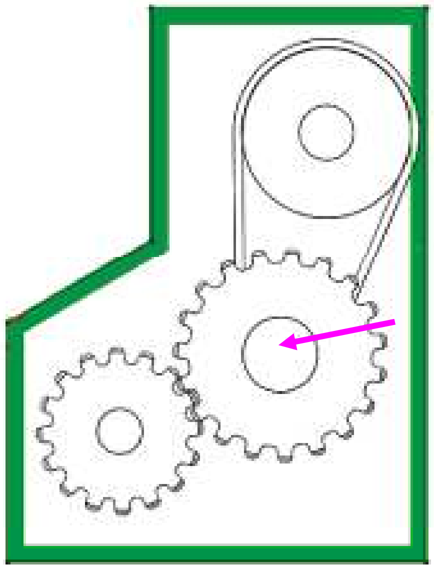


1. Go to file/open and select E11 for file type and locate "Gear Enclosure".

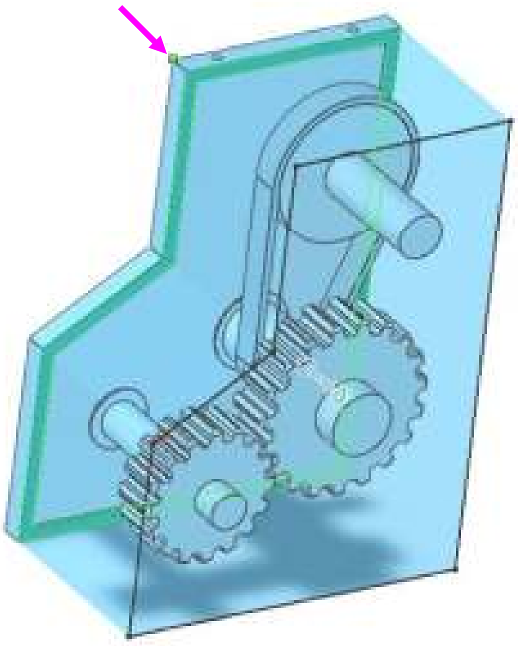




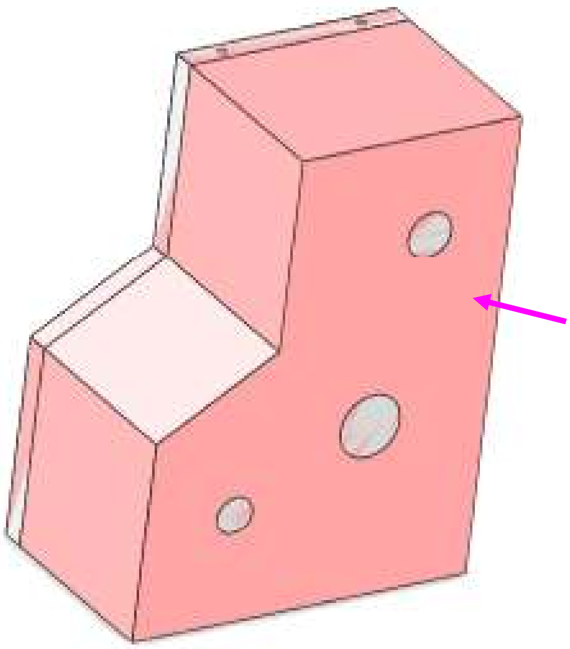
2. Insert a new part into the assembly; drop it on the end face of a gear shaft of the assembly. Name it “Cover 2” (This will be the enclosure) then select the front outside face. Convert Entities. “Offset”



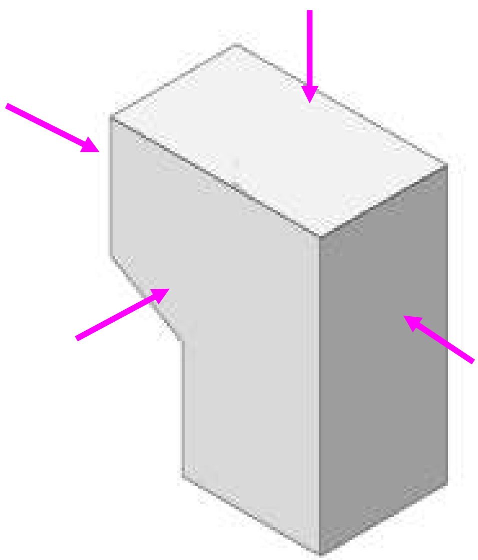
3. Extrude up to vertex.

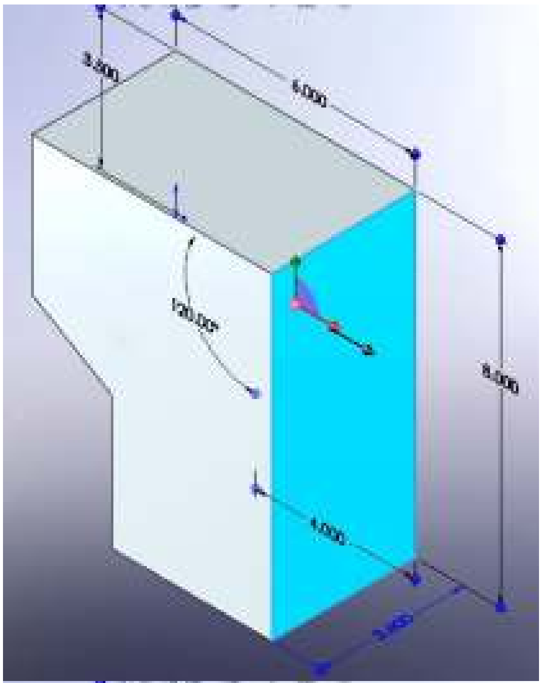


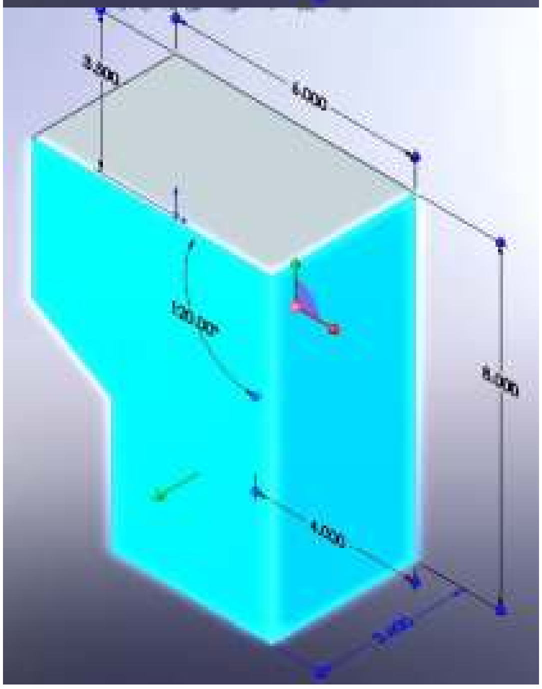
4. Once completed the assembly should look like this. Right Mouse click on the surface of the enclosure and select "open".

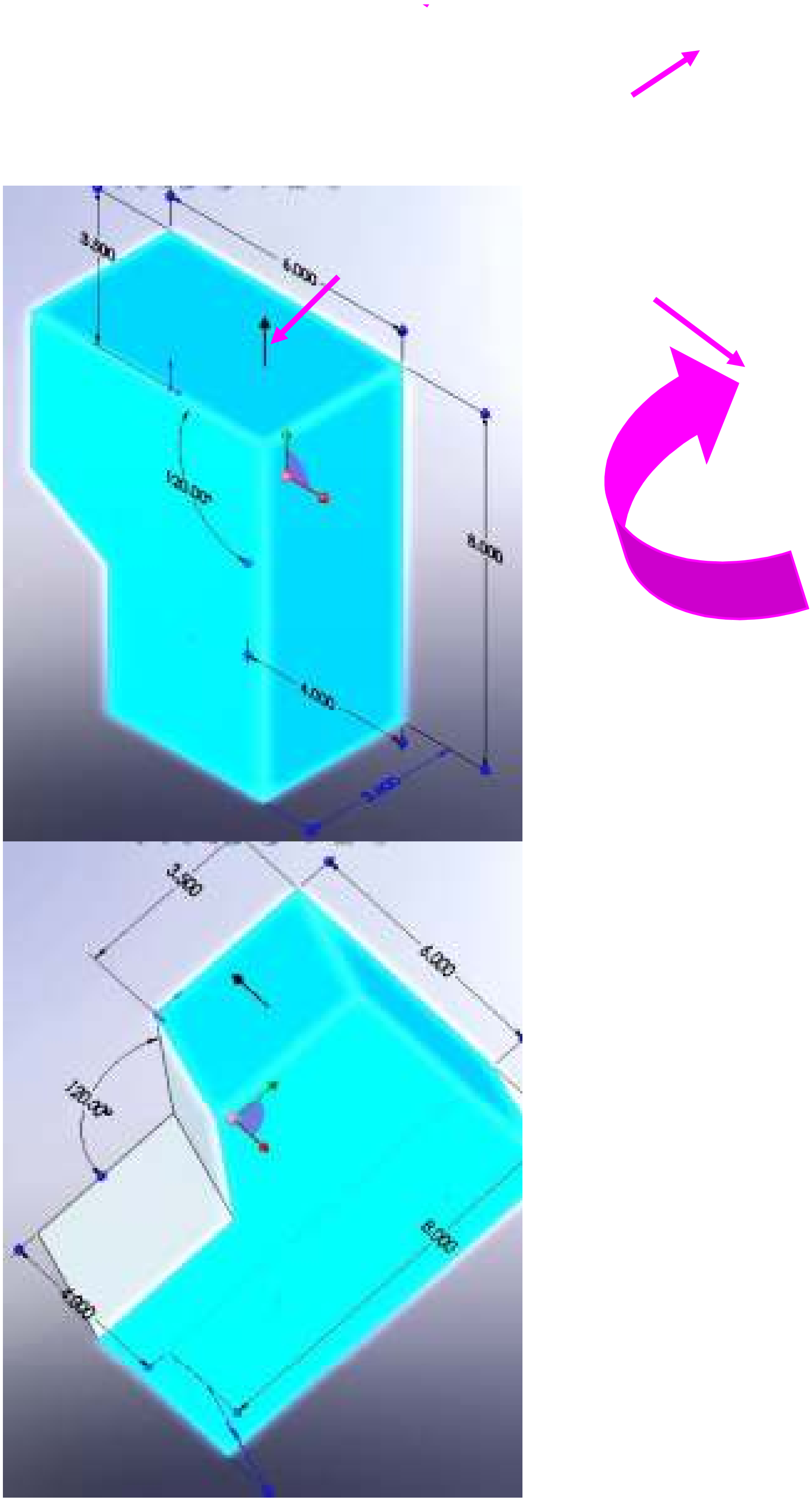


5. Convert to sheetmetal Go to an isometric view and "ctrl" select the four faces as shown.



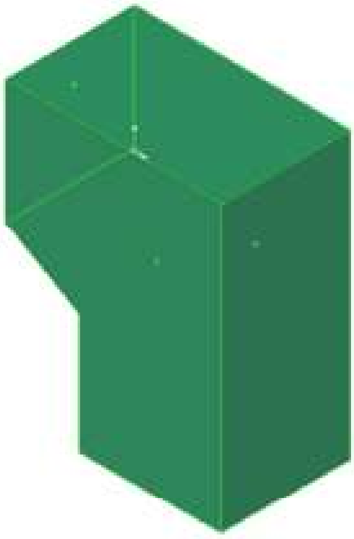

6.000
120.00°
8.000
4.000


6.000
120.00°
8.000
4.000

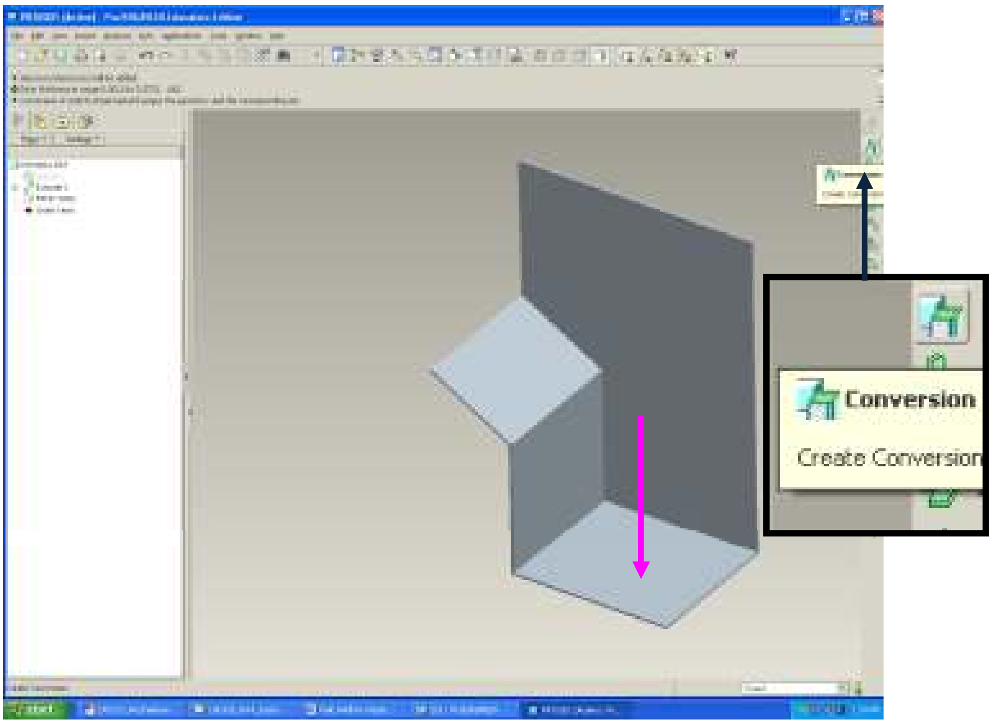


Rotate the view to select the fourth face.

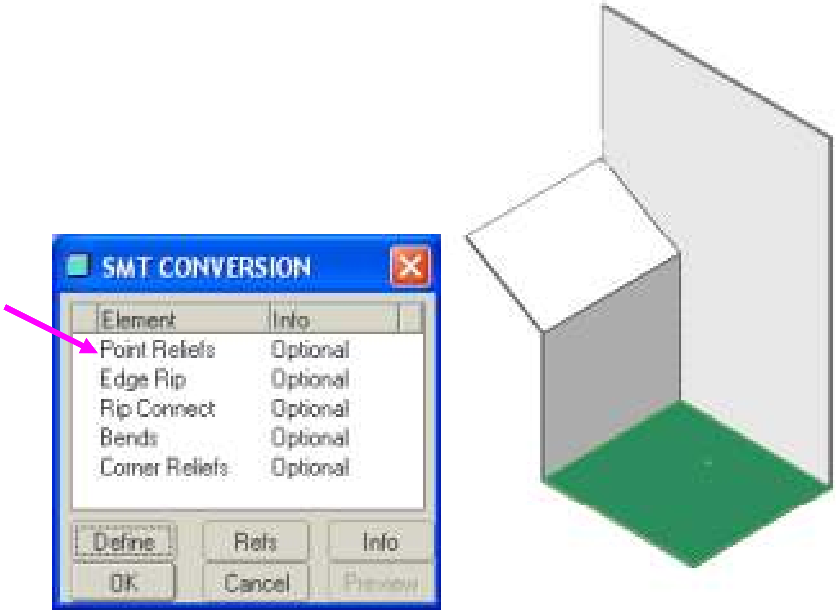
6. To convert to a sheet metal part, select the pull down menu “Application/Sheet metal” select the “shell” option.



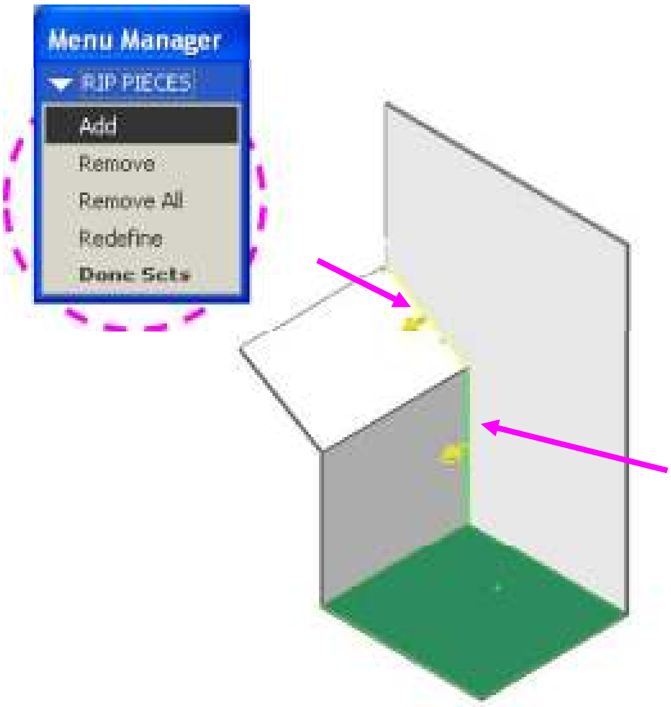
6. Select the bottom face and select the “Conversion” icon.



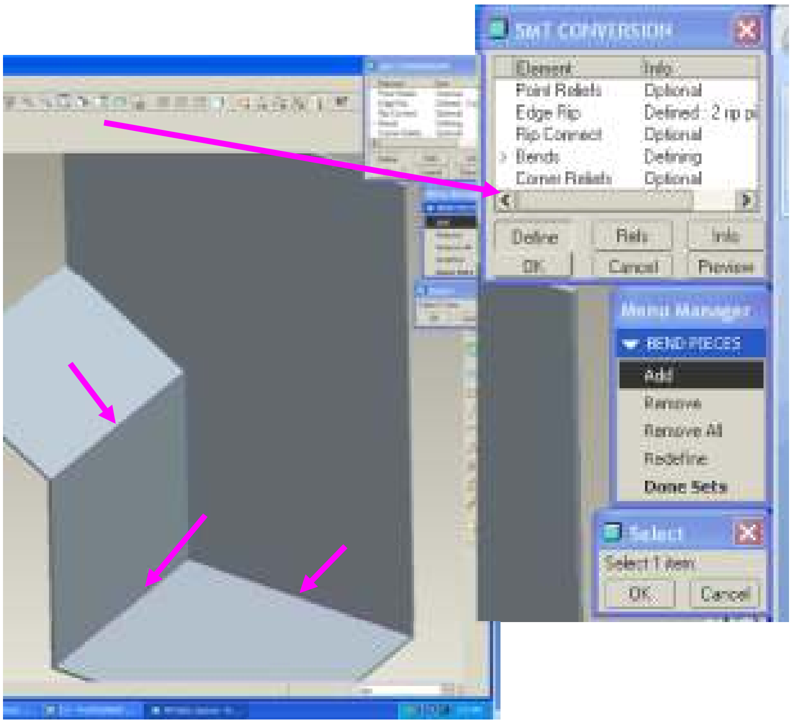
7. Go to the right view orientation and you should have this section view…



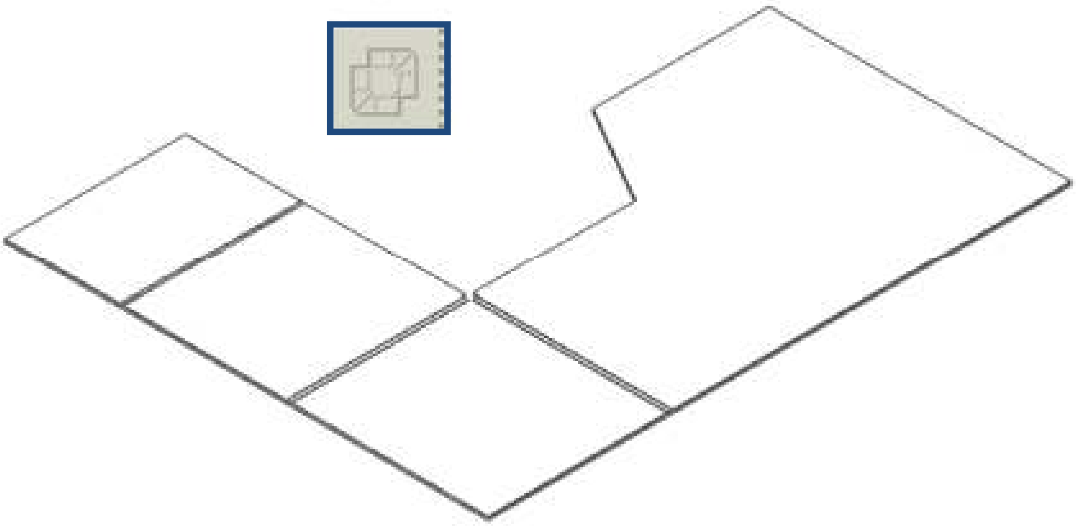
8. Click on the Rip parameters and select the two inside edges. Hit apply.



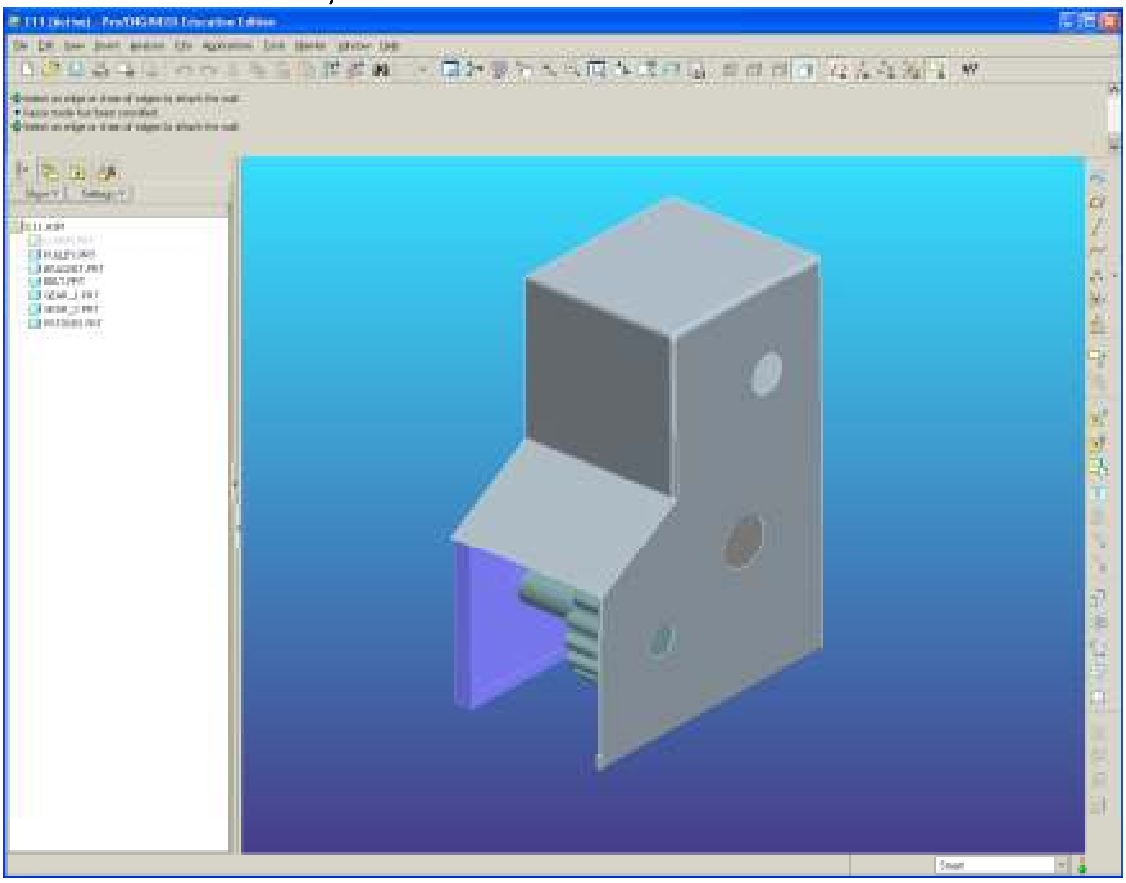
9. Double click on “Bends”. Hold the CTRL key while selecting. Hit done and OK.



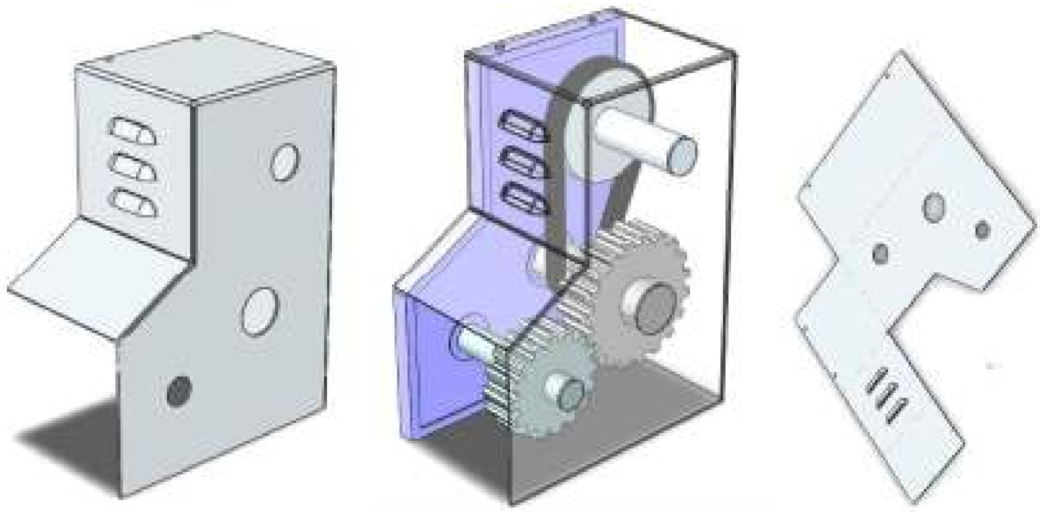
10. Select the flatten icon.



12. Return to the assembly.



13. Add holes and additional features.



14. The enclosure is now completed.

BONUS INFORMATION

ProE Creo Administration

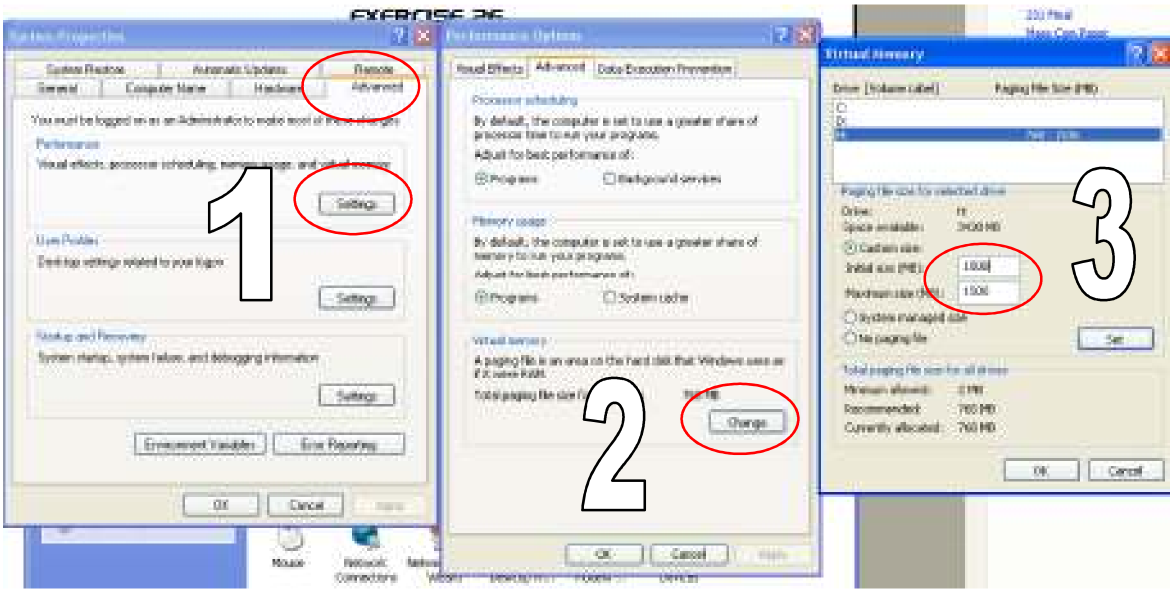
Finding adequate computer hardware to run Inventor can be challenging*, this lesson looks at the multiple aspects of selecting hardware as well as modifying settings inside Creo to allow it to run efficiently and trouble free.*

Selecting an Operating System (OS).

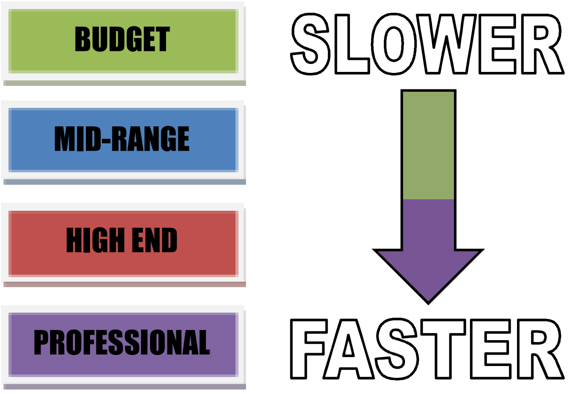
Windows XP Professional
Windows XP 64-Bit edition (Creo 3.0 may be the last release for Windows XP)
Windows Vista
Windows 7
Windows 8

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: 4GB of RAM should have 8GB Virtual RAM. And it is suggested to keep the initial size the same as the maximum size. It is rumored to help 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

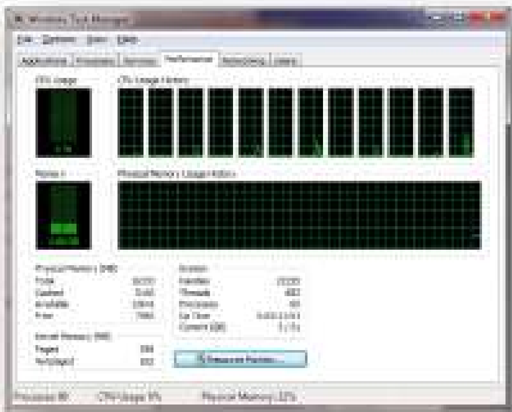
Most CPU manufacturers are beginning to deliver multiple core processors. This can be seen with the AMD Opteron which has up to 16 processing cores.

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

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 not every feature inside Creo is fully written to take advantage of multithreaded processes. However, using the Creo FEA Simulation, CFD, or Photolux rendering solutions one may discover 2x – 16x faster performance versus a single core processor. This is because these Creo applications do take 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 FX 600 erp.$159** (erp- estimated retail price)
    - **Quadro FX 2000 erp.$499**
    - **Quadro FX 4000 erp.$799**

- ATI ***FirePro*** series (not FireMV series)
    - **FirePro 3900 erp.$159**
    - **FirePro 5900 erp. $499**
    - **FirePro 7900**

- Intel ***Xeon***
    - **P**4000 *HD* integrated graphics *(must be P = Professional rated)*

These cards are considerably more expensive that mainstream cards but the benefits of experiencing less crashes or visual problems with Creo -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 may experience some graphical glitches from time-to-time.

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

MEMORY (RAM)

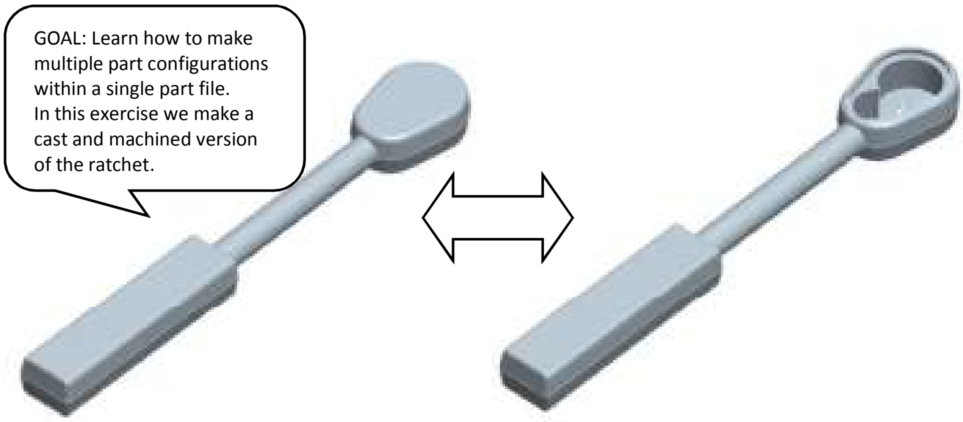
- 8– 16 GB For simple machined parts to average assemblies.
- 32+ GB for large assemblies.

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

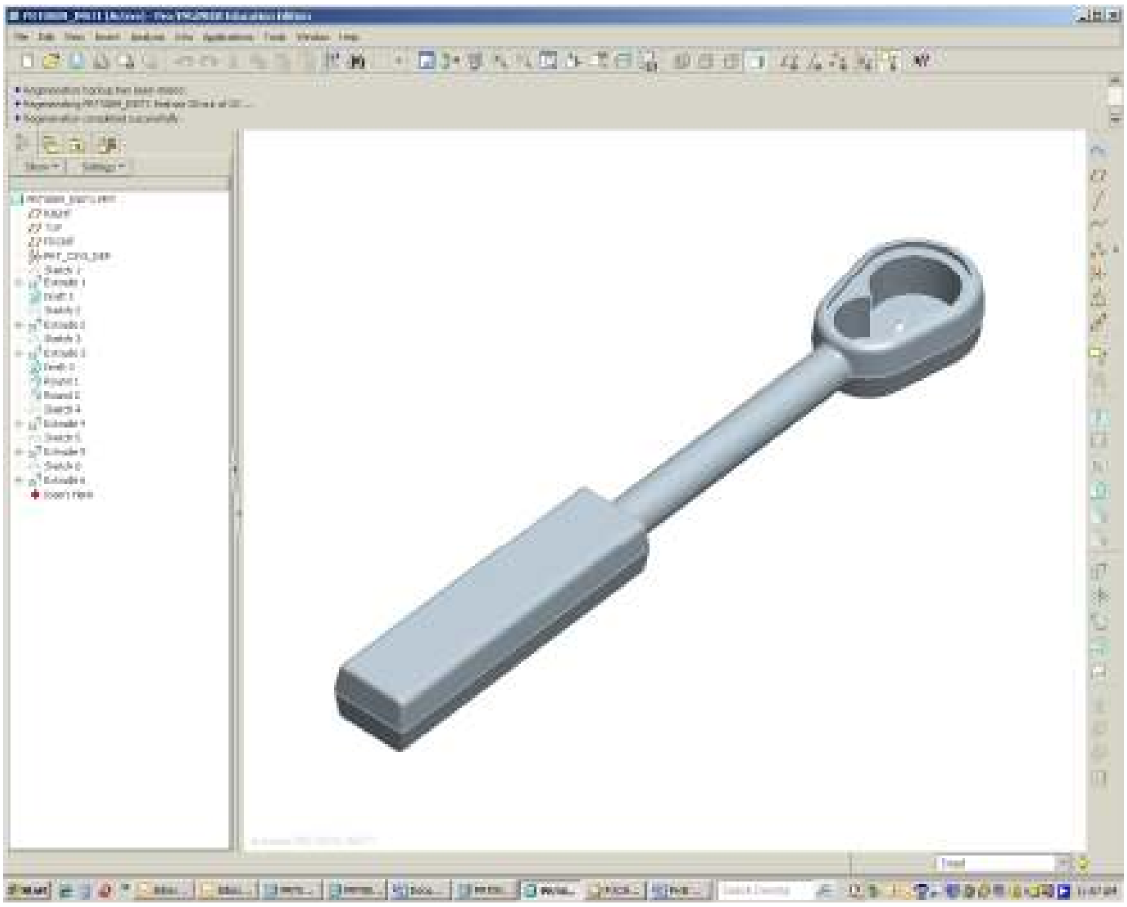
Bonus EXERCISE 3B

Family Tables

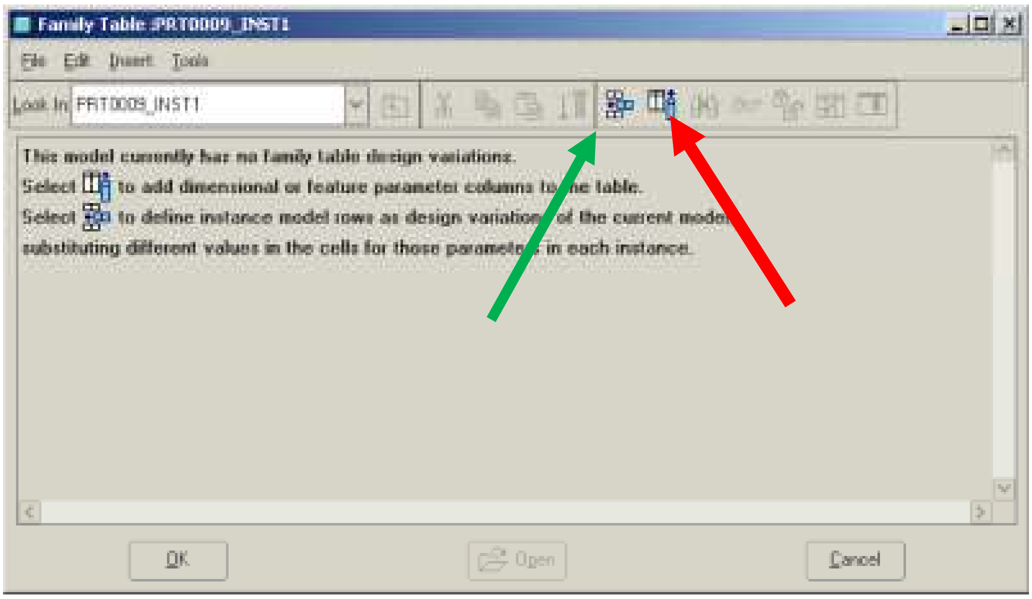
Family Tables enable you to create multiple part configurations derived from a single part file.



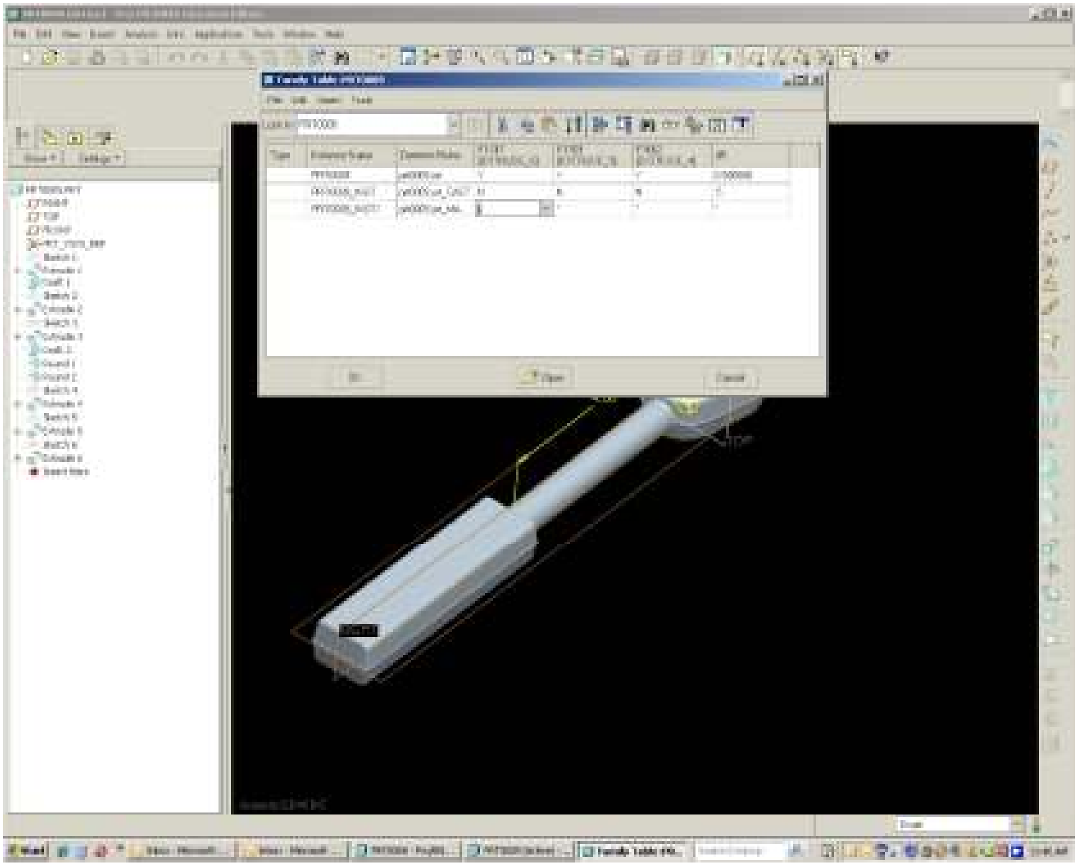
1. Open the Exercise_4_FAMILY part file.



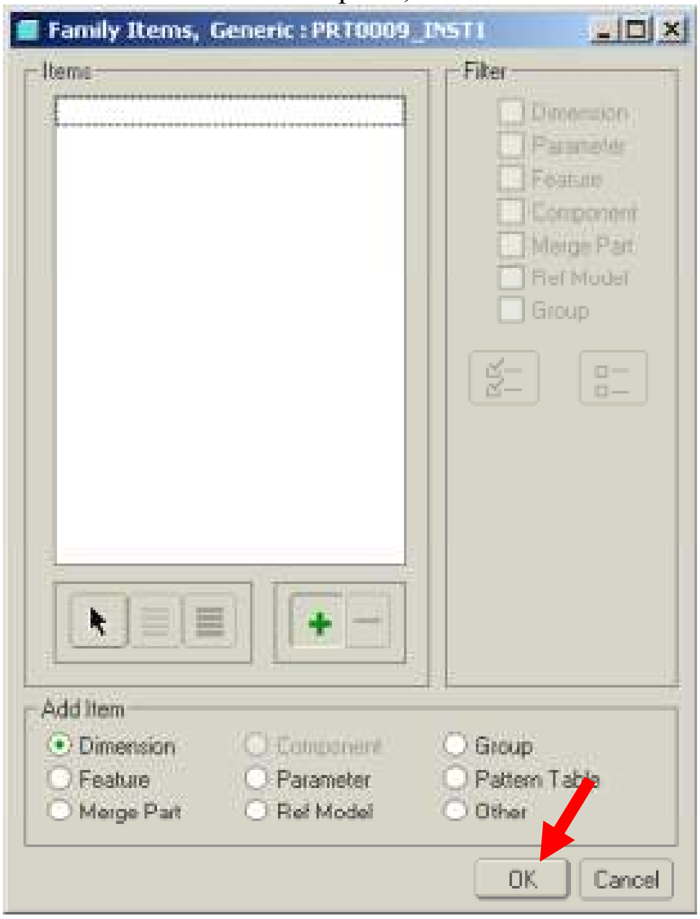
2. Go to the pull down menu- “Tools/Family Tables”



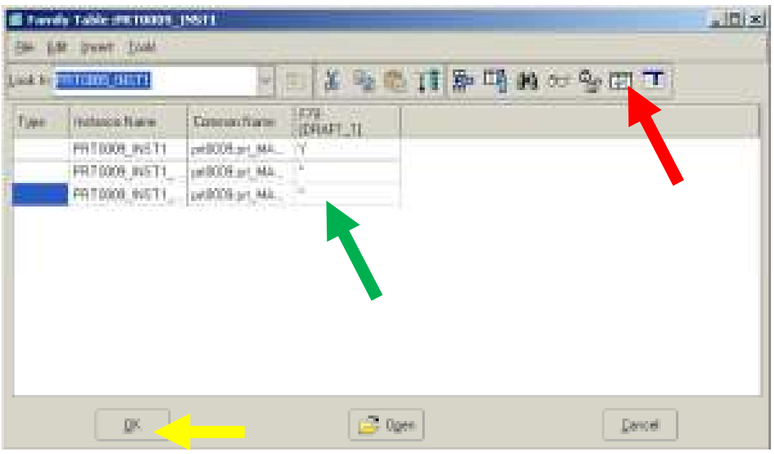
3. Select the **“Insert new insatnce”** two times. Then hit the “Add…” icon.



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

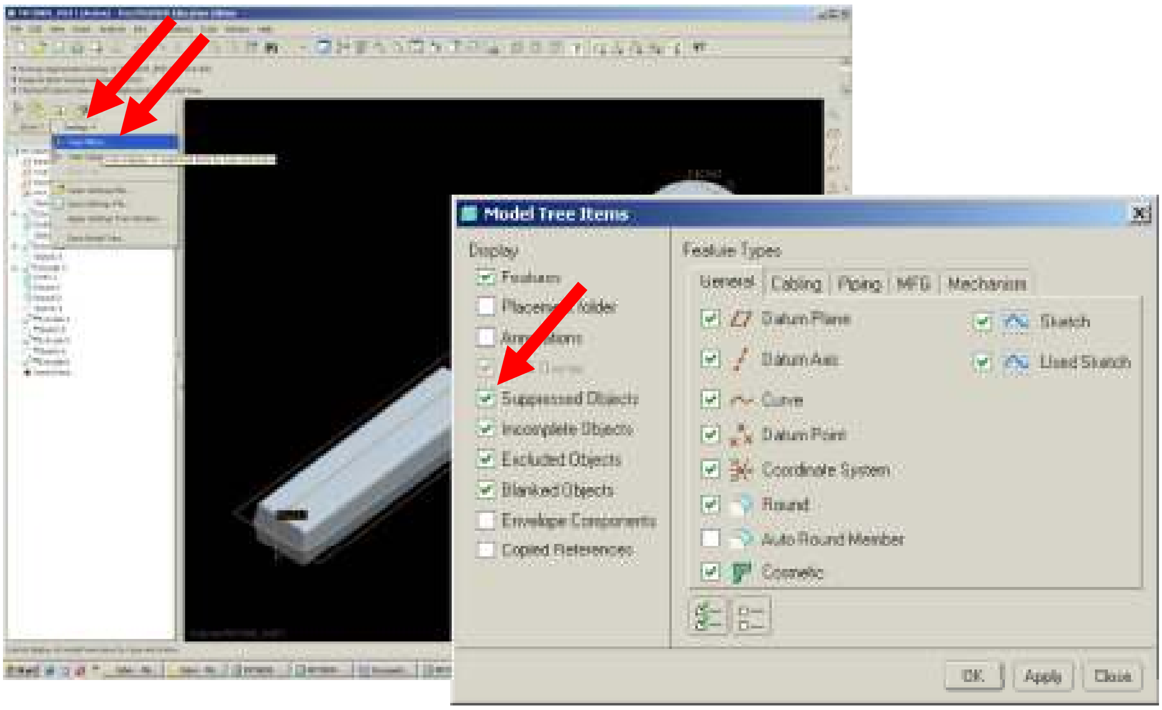


5. Select "OK".
6. Select "**Verify**"
7. In the columns type "**N**" for no- to supress the feature, or "**Y**" for yes for the feature to be unsupressed. Hit "OK".



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

CAD 105 TOTALS

E2 – 30pts

E3 – 30pts

E4– 30pts

E5– 30pts

E6– 30pts

E7– 30pts

E8– 30pts

E9– 30pts

E10– 30pts

E11– 30pts

Q1 -10pts

Smoke Detector Front – 10pts

Smoke Detector Rear – 10pts

MIDTERM – 300pts

FINAL – 300pts

ATTENDANCE & PARTICIPATION -70pts

<u>TOTAL - 1000pts</u>