

Computer Aided Drawing: An Overview

Dr. H. Hirani
Department of Mechanical Engineering
INDIAN INSTITUTE OF TECHNOLOGY BOMBAY
Powai, Mumbai-76
hirani@me.iitb.ac.in

Drawing: Machine/ Engineering/ Technical

- ◆ Drawing is a universal language. It communicates information faster and more clearly than the spoken words.
 - ◆ Industrial drawing allows viewers to imagine a building or machine before it is built.
 - ◆ During Industrial Revolutions many men were trained strictly as draftsmen. Some had engineering or production skills, but many more were copyists.
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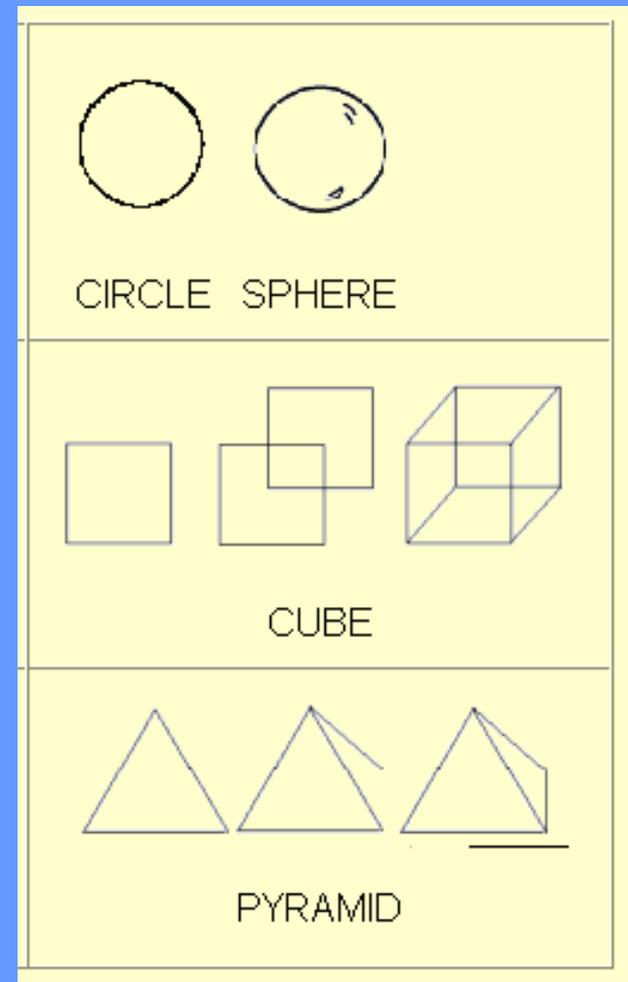
Advantages of CAD over manual drawing

- ◆ Difference between a business report written using Typewriter and Computer?
 - If Boss needs some changes?
 - If paper and printing is costly?
 - If there is a bit of change in two reports?
 - If a neat report is demanded?
 - If standards* are defined?

*ANSI Y14 American National Standard Drafting Manual

2-D versus 3-D drawings

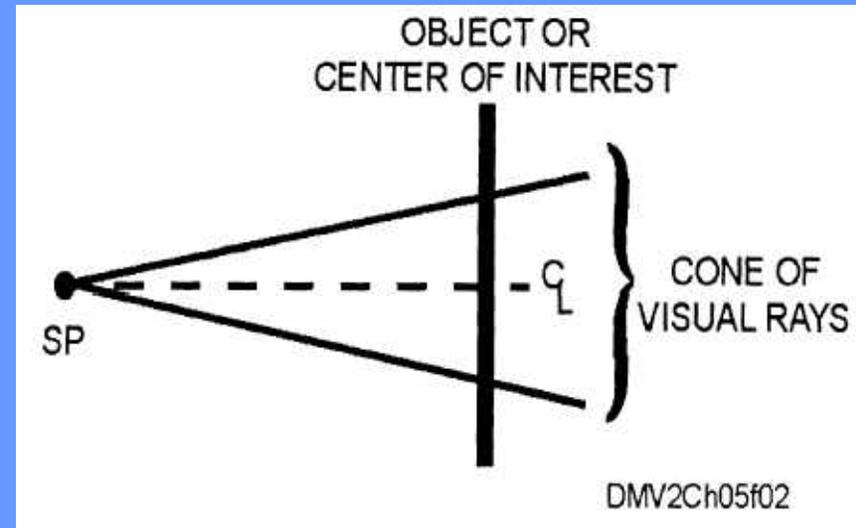
- ◆ 2-D: A concept of displaying real-world objects on a flat surface showing only height and width. This system uses only the X and Y axes.
- ◆ 3-D: A way of displaying real-world object in a more natural way by adding depth to the height and width. This system uses the X Y and Z axes.



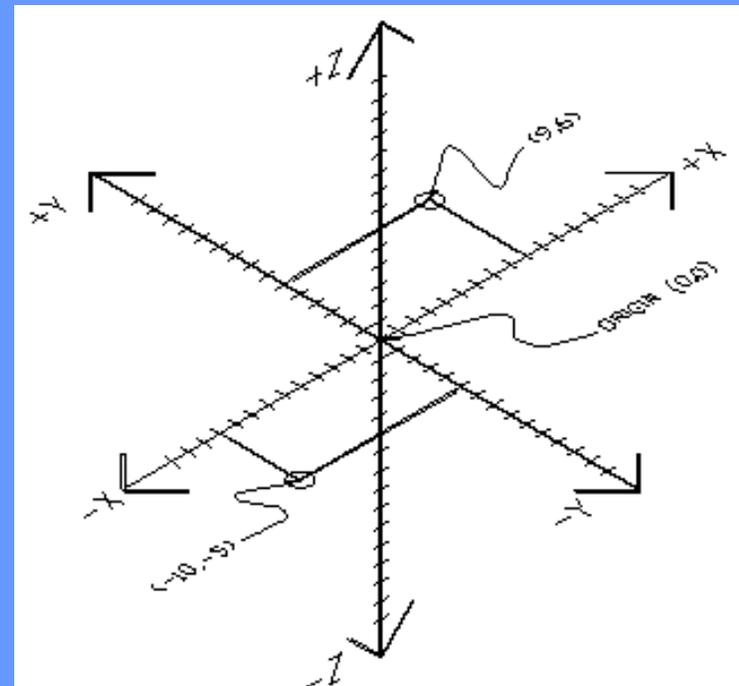
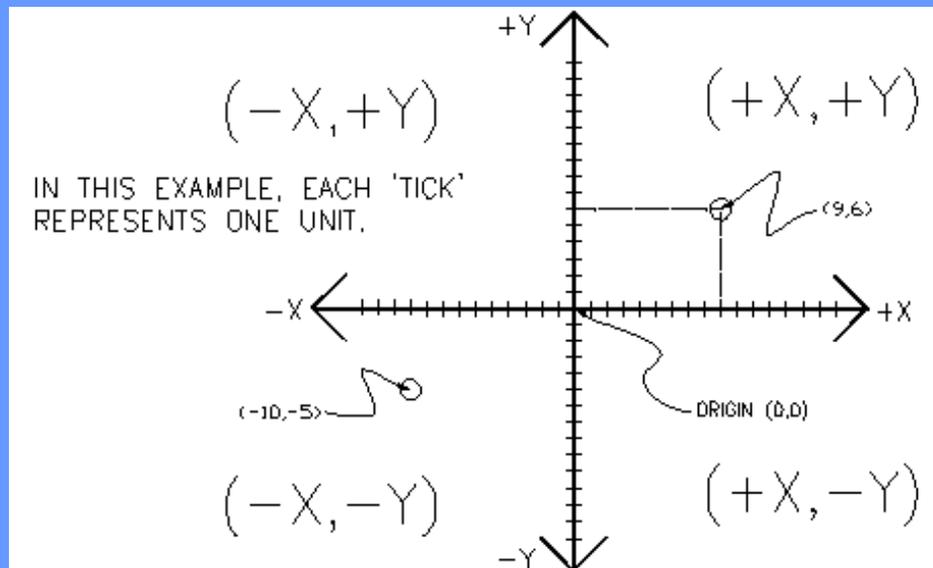
2-D versus 3-D drawings....cont



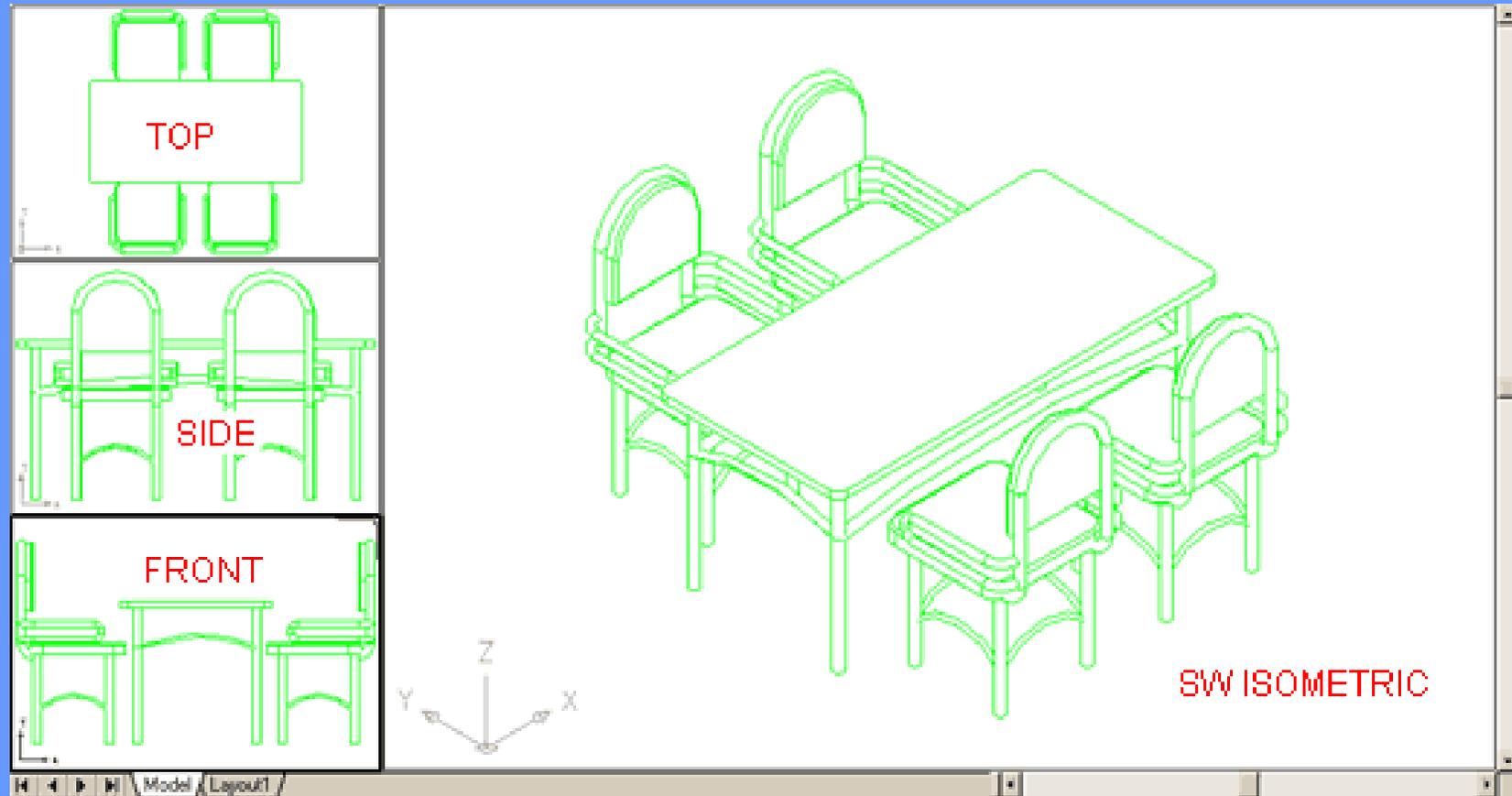
- ◆ A pictorial drawing or photograph shows an object as it appears to the observer, but not as it is. Such a picture cannot describe the object fully, no matter from which direction it is viewed, because it does not show the exact shapes and sizes of the several parts.



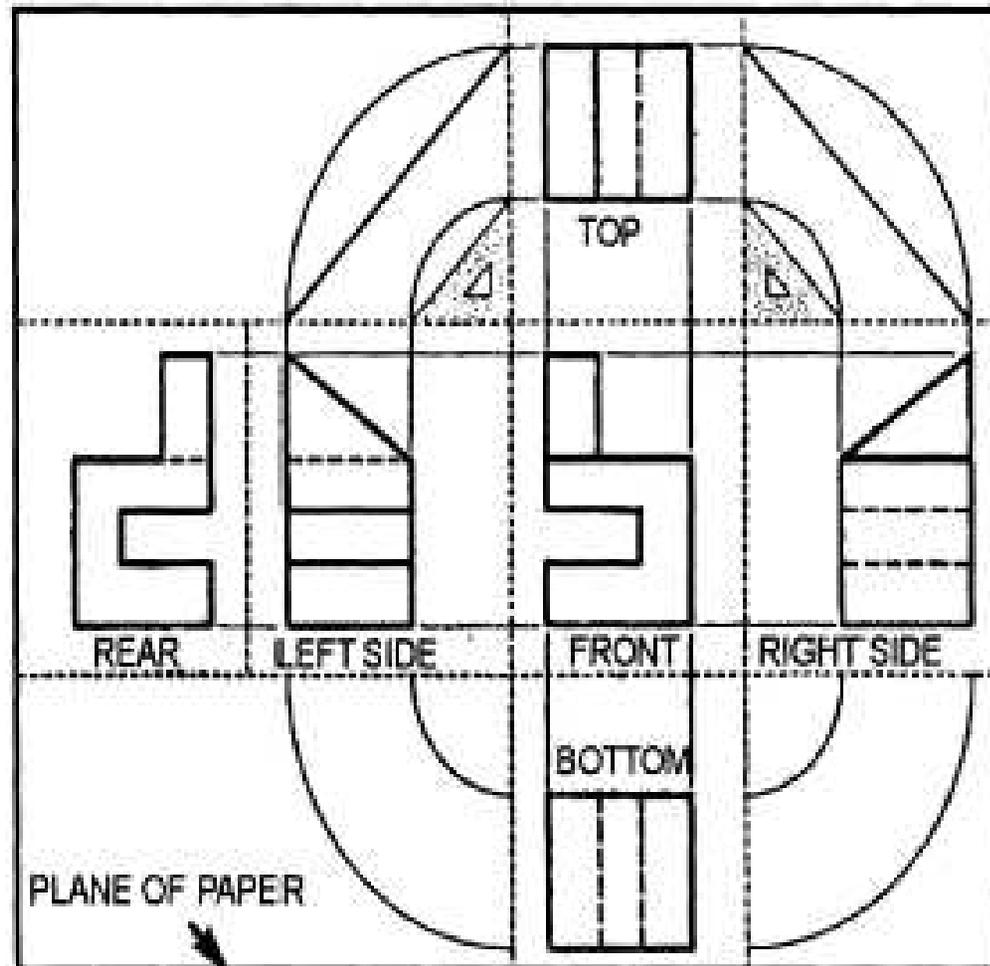
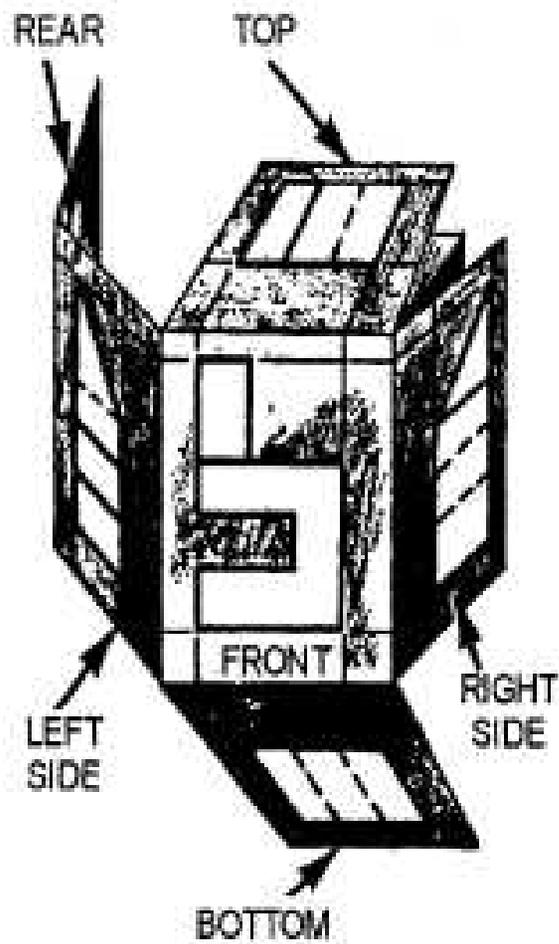
2-D versus 3-D drawings....cont



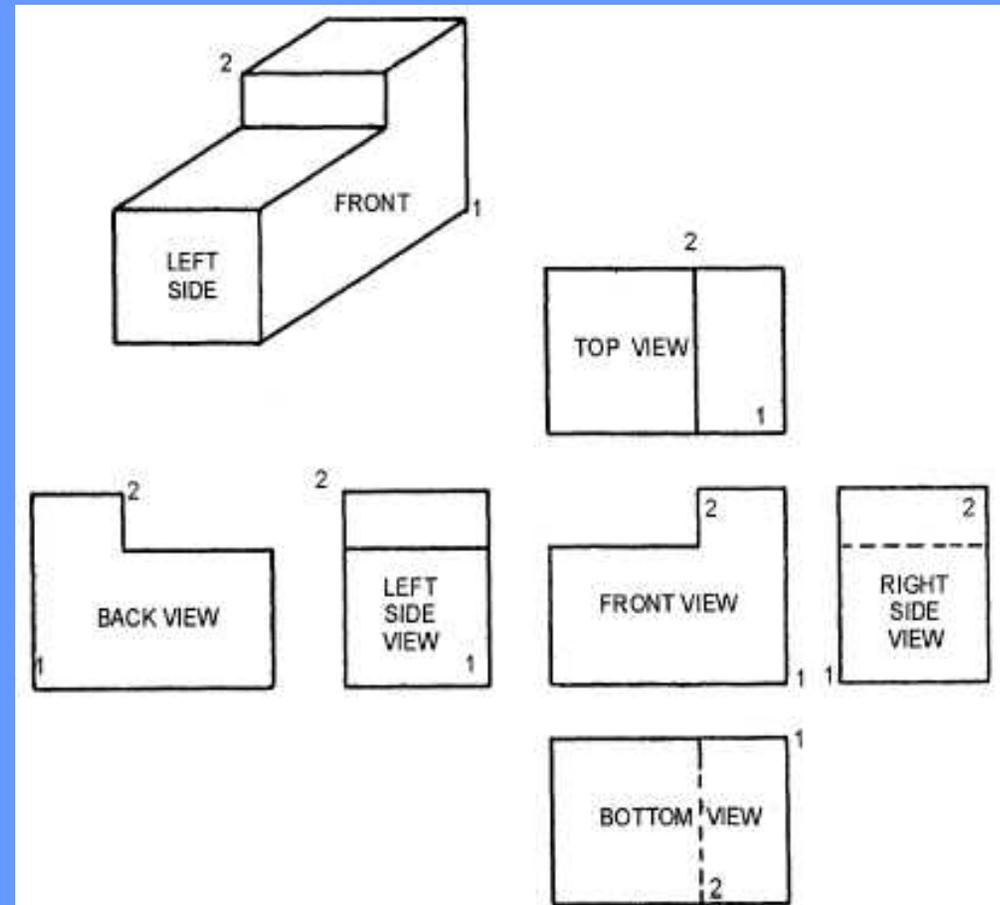
2-D versus 3-D drawings....cont



Parallel Projections



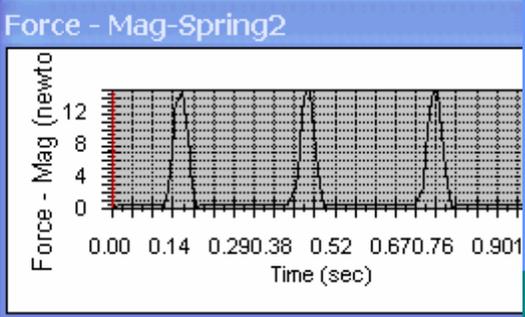
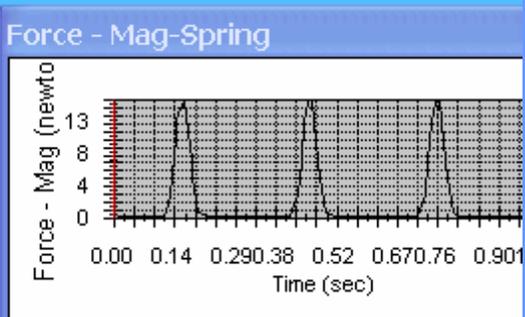
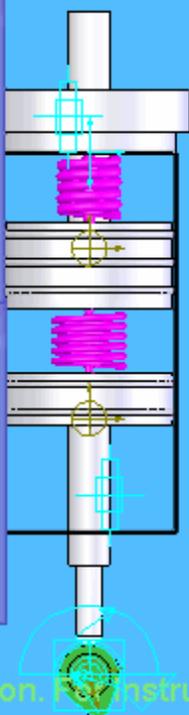
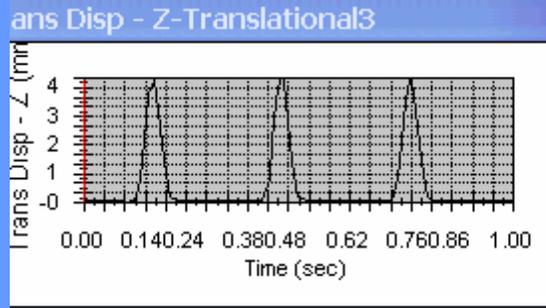
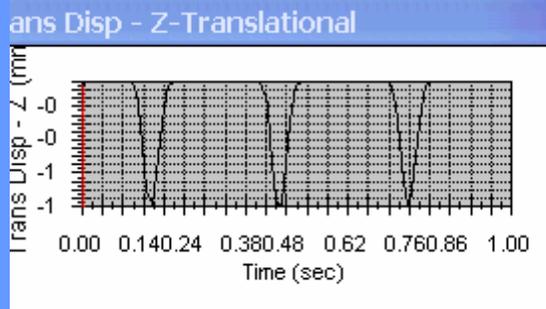
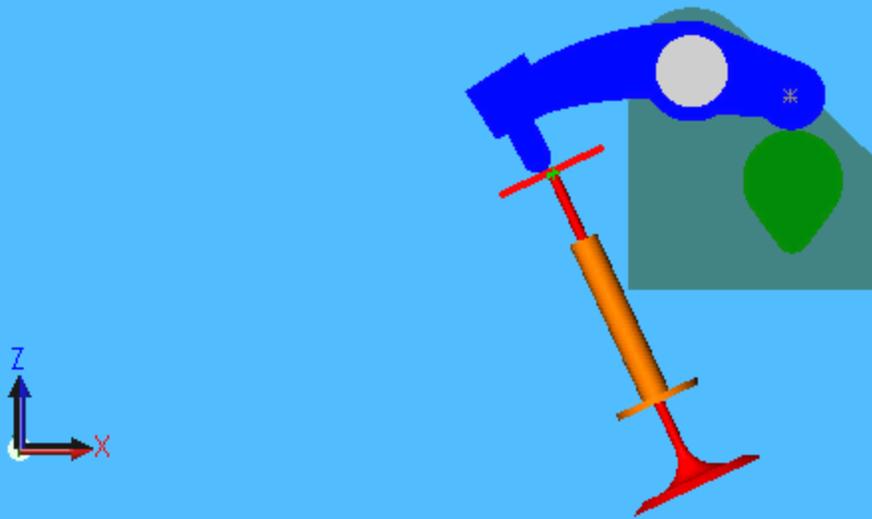
- ◆ Necessary views are those that show best the essential contours or shapes and have minimum number of hidden lines.



NOTE: Back, right side and bottom views can be deleted.

Why 3-D drawings ?

- ◆ Exploration of new design ideas becomes easier.
- ◆ Improvement in product design cycle:
 - Virtual assembly:
 - ◆ Very large assemblies containing thousands of parts.
 - ◆ Two way associativity
 - ◆ Detection of interference
 - Virtual Manufacturing:
 - Fastest way to create a 2-D drawing.
 - Better communication with non-engineers, sales and marketing personnel, production-line staff, vendors, partners, and clients.
 - Static and dynamic analyses of assemblies.



Educational Version. For Instructional Use Only

Overview of Pro/Engineer

- Pro/ENGINEER is a computer graphics system for modeling various mechanical designs & for performing related design and manufacturing operations.
- The system uses a 3D solid modeling system as the core, and applies the feature-based, parametric modeling method.

Pro/ENGINEER	Conventional CAD Systems
Parametric Model	Fixed-dimension Model
Feature-based Modeling	Primitive-based Modeling

Overview of Pro/E...Screen Layout

PRT0001 (Active) - Pro/ENGINEER Education Edition

File Edit View Insert Analysis Info Applications Tools Window Help

–Working window → Only one window as current or active window.

- Using the template default G:\PTC\templates\inlbs_part_solid.prt as the template.
- Datum planes will be displayed.

◆ Message window: Text window. Display one line information statement about highlighted menu-item.

Overview of Pro/E....Menus

- ◆ Each Pro/ENGINEER object opens in a “Graphics window”.
- ◆ The menu bar, which is part of the Pro/ENGINEER main window, contains menus with options for creating, saving, and modifying models.
 - **File menu**—Provides options for opening, creating, saving, renaming, printing, importing, and exporting files.
 - **Edit menu**—Provides options for editing features, including modifying, redefining, rerouting, suppressing, resuming, editing patterns, and deleting a feature.
 - **View menu**—Options for controlling display of the model and display performance.
 - **Insert menu**—Options for creating datum features (i.e. points, axes, and planes) and features, such as: holes, chamfers, cuts, etc.

Overview of Pro/E....Files

◆ Working Directory

– File menu > “Working Directory”

- ◆ Select directory where part /assembly/ drawing file will be saved.

◆ New File

– **Sketch:**

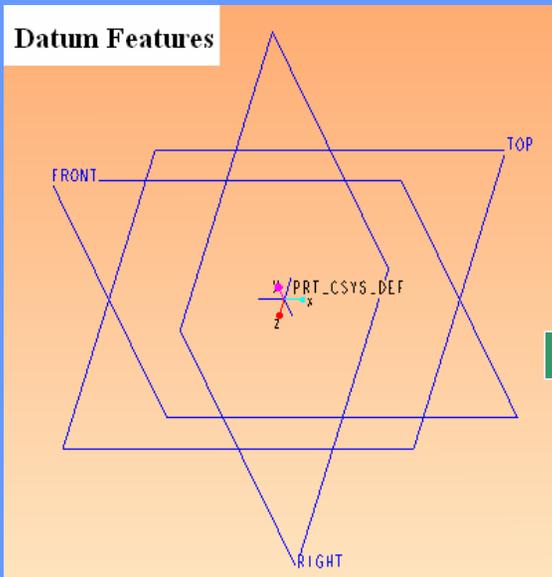
- ◆ This 2-D Section mode can be accessed directly from the MODE menu or from the Part and Assembly modes. S2dxxx.SEC

– **Part:** Prtxxx.prt

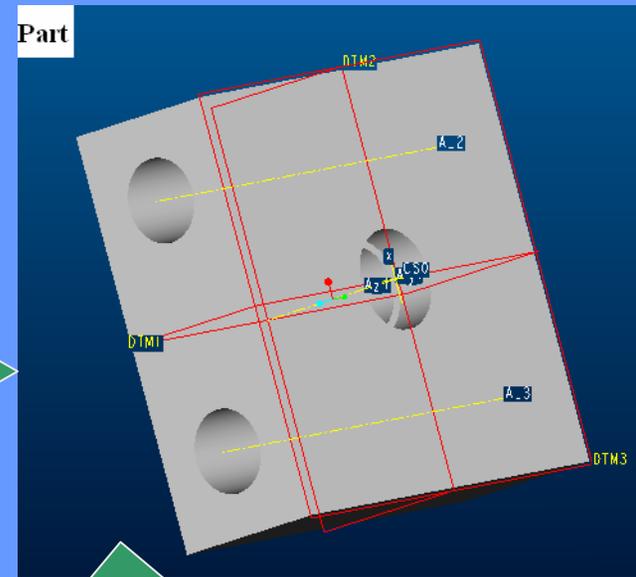
– **Assembly:** Asmxxx.asm

– **Drawing:** Drawxxx.drw

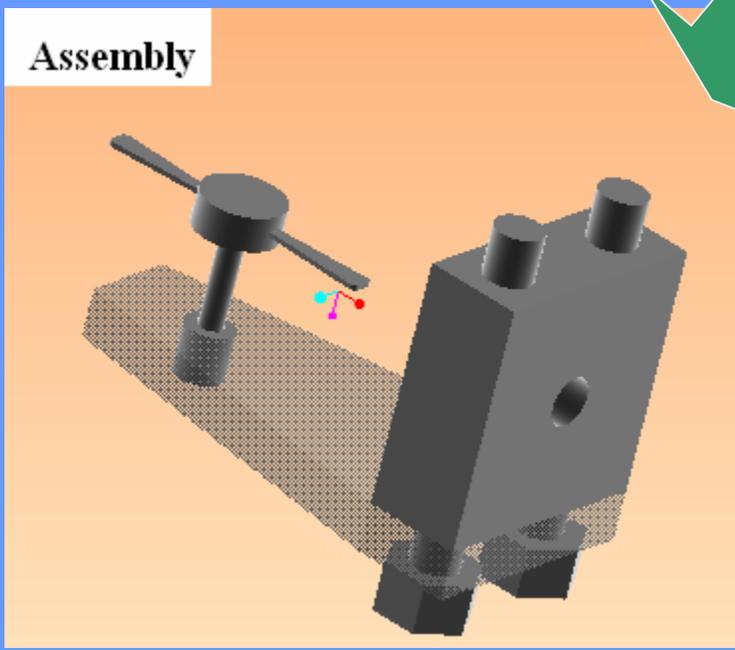
Datum Features



Part



Assembly



Overview of Pro/E....Model Information

- ◆ Obtaining a Feature List
 - Info > Feature List
- ◆ Getting Information about a Specific Feature
 - Info > Feature.
- ◆ Parent/Child Relations
 - Info > Parent/Child (Parents/*Children* > Highlight)
- ◆ Model Information
 - Info > Model

Datum plane

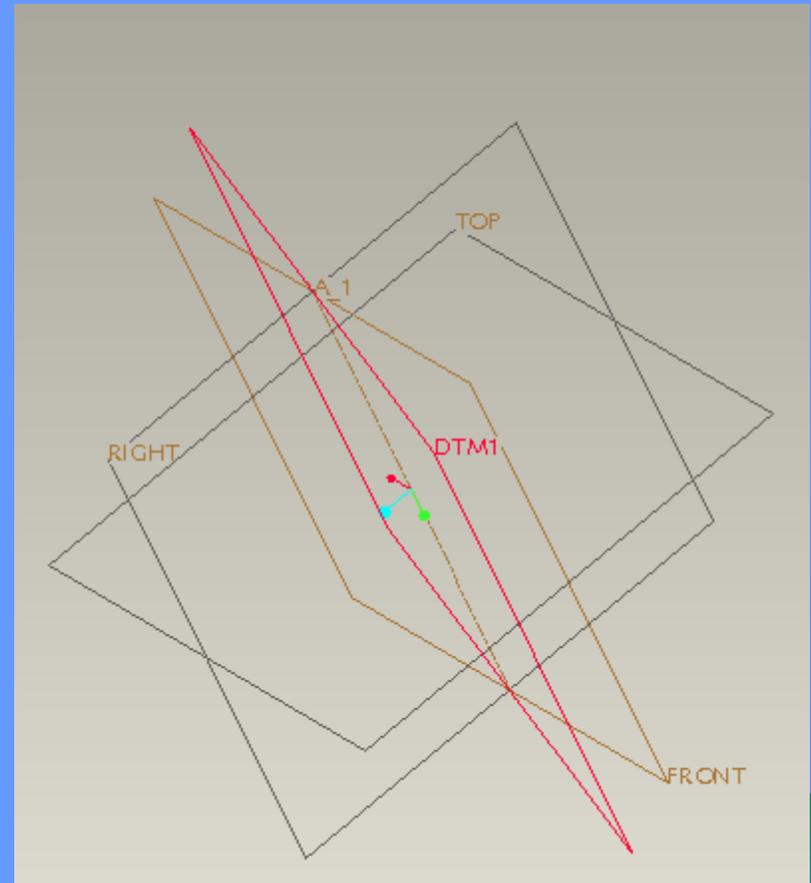
Datum coordinate system

Datum axis

Reference geometries

Reference Geometry

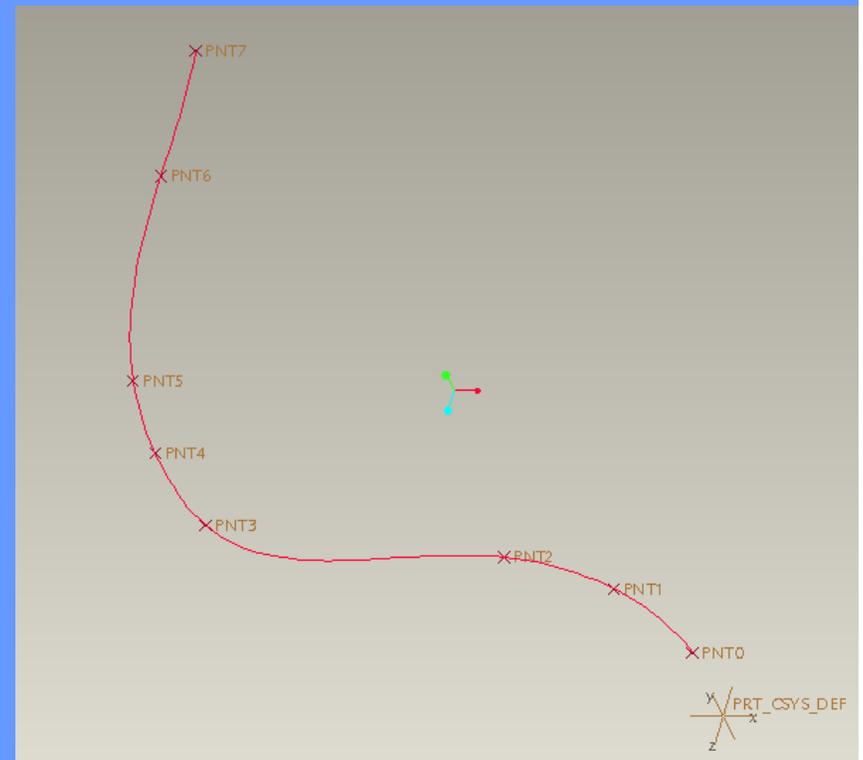
- ◆ Ex 1:
 - Create a datum plane at 15 degree angle with front plane and passing through intersection of all three planes.
 - Rename plane as "Sketch_Plane"



Reference Geometry

◆ Ex2:

- Create a series of datum points having dimensions: (0,10,0), (-5, 20,0), (-15,25,0), (-45,30, 0), (-50, 35, -5), (-52, 40, -10), (-52, 47, -30), (-52, 47.75, -45).
- Create a smooth datum curve through all points.



3-D Modeling of Simple Part

Datum features → Parent child relation

Geometric features

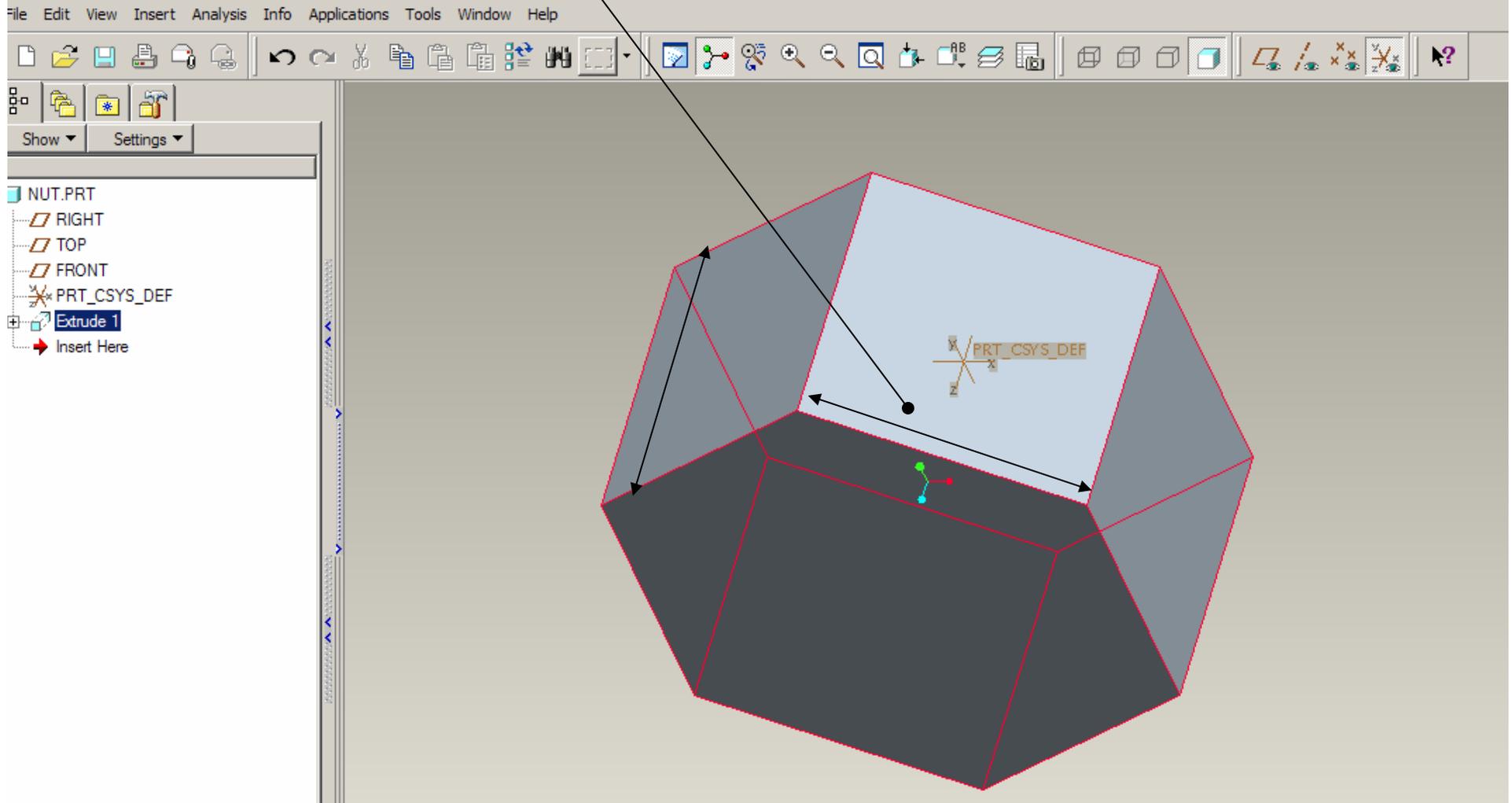
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Ex: Draw a hexagonal nut with each side 15 mm and depth 15 mm

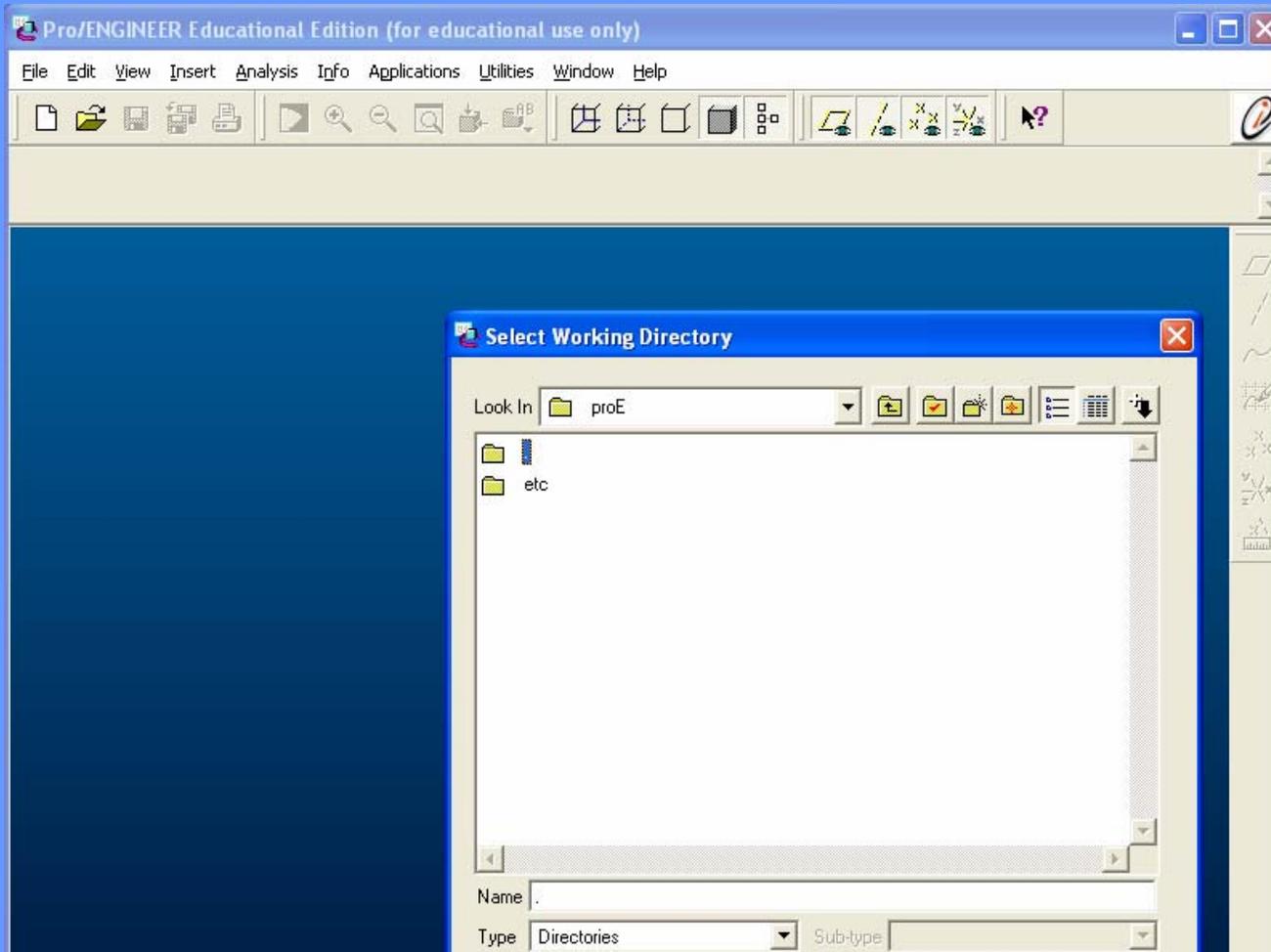


In hexagon each side is equal to the radius of circumscribed circle. Therefore in present exercise we will learn:

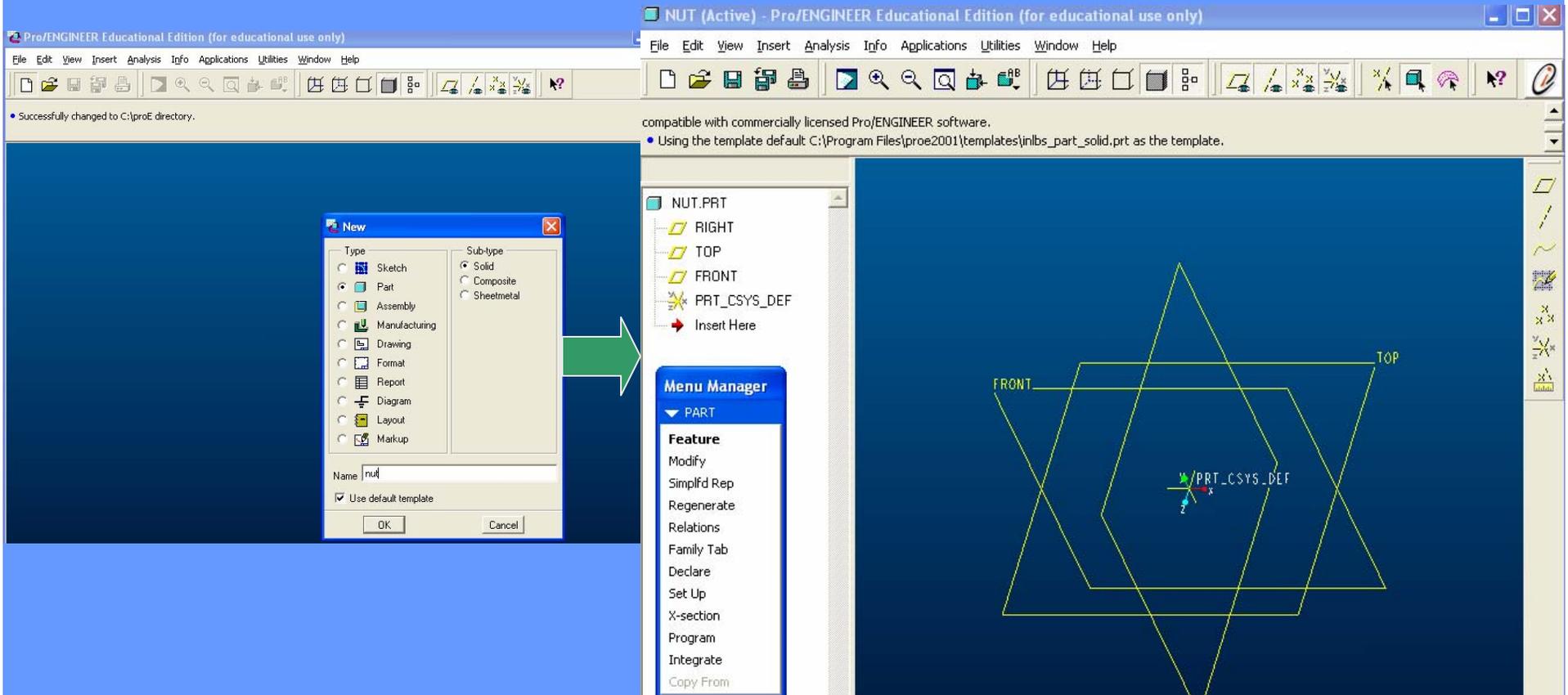
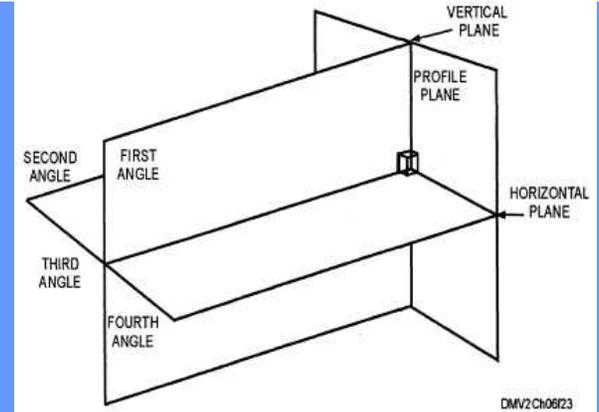
- ◆ How to draw a circle?
- ◆ How to draw centerlines?
- ◆ How to draw solid lines?
- ◆ How to trim arcs?
- ◆ How to make solid/hollow nut?
- ◆ How to create chamfer?

- ◆ Problem: Draw a hexagonal nut of 15mm side.

Step 1: Select working dir.



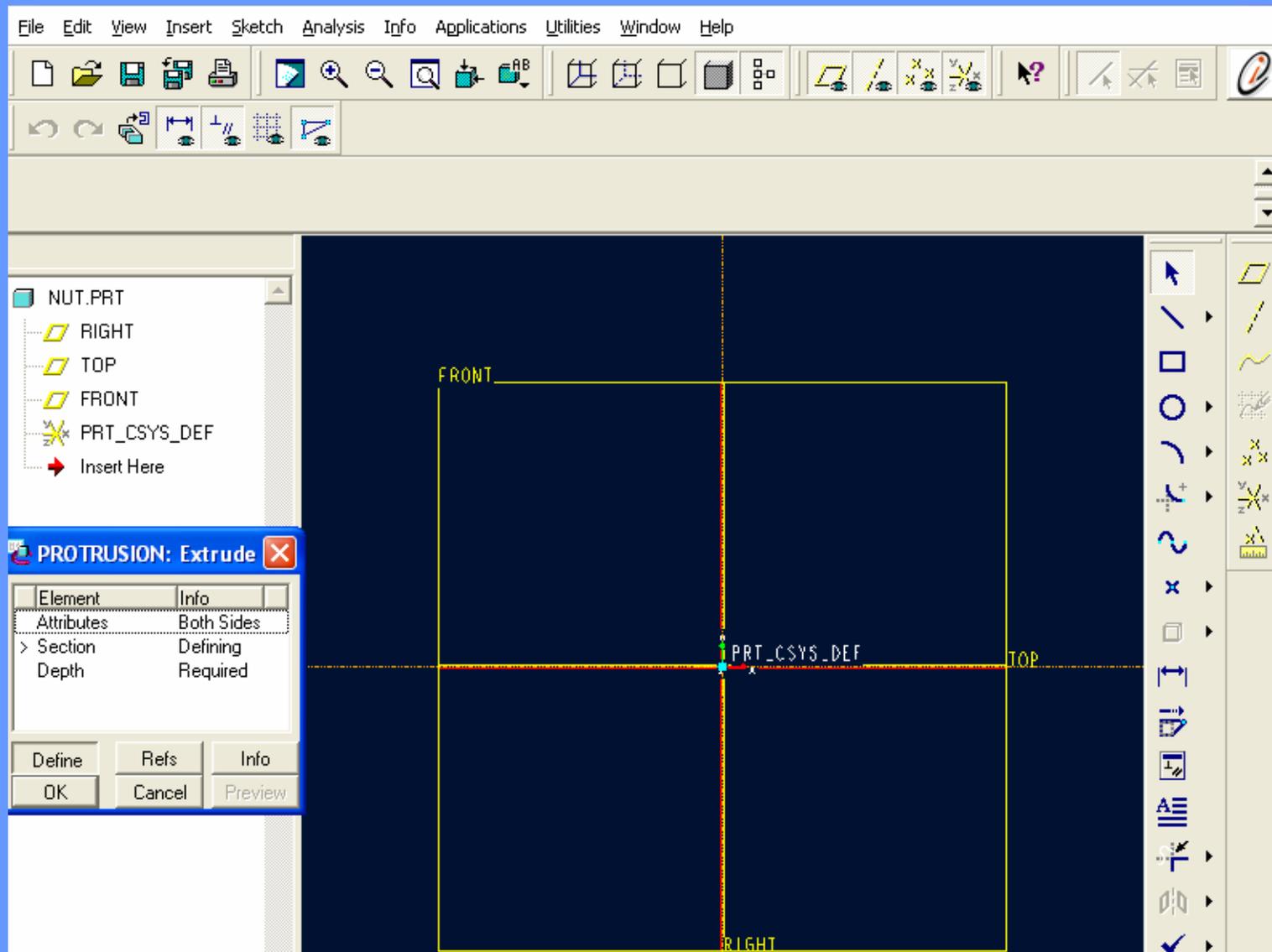
Step 2: File > New > part > Nut



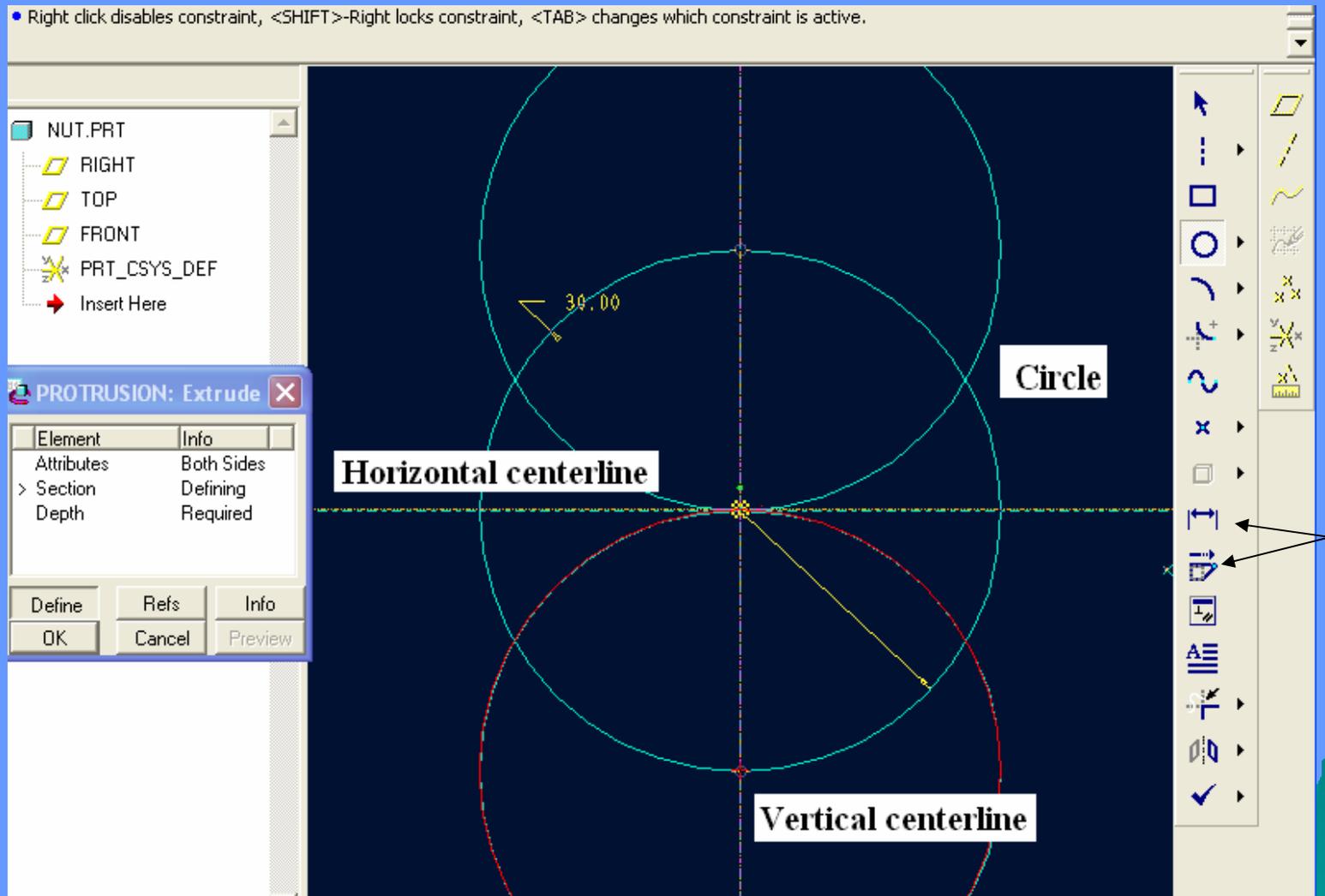
Step 3

EXTRUDE: Command raises the shape of a 2D outline into a 3D solid. For example, a circle would be extruded into a cylinder.

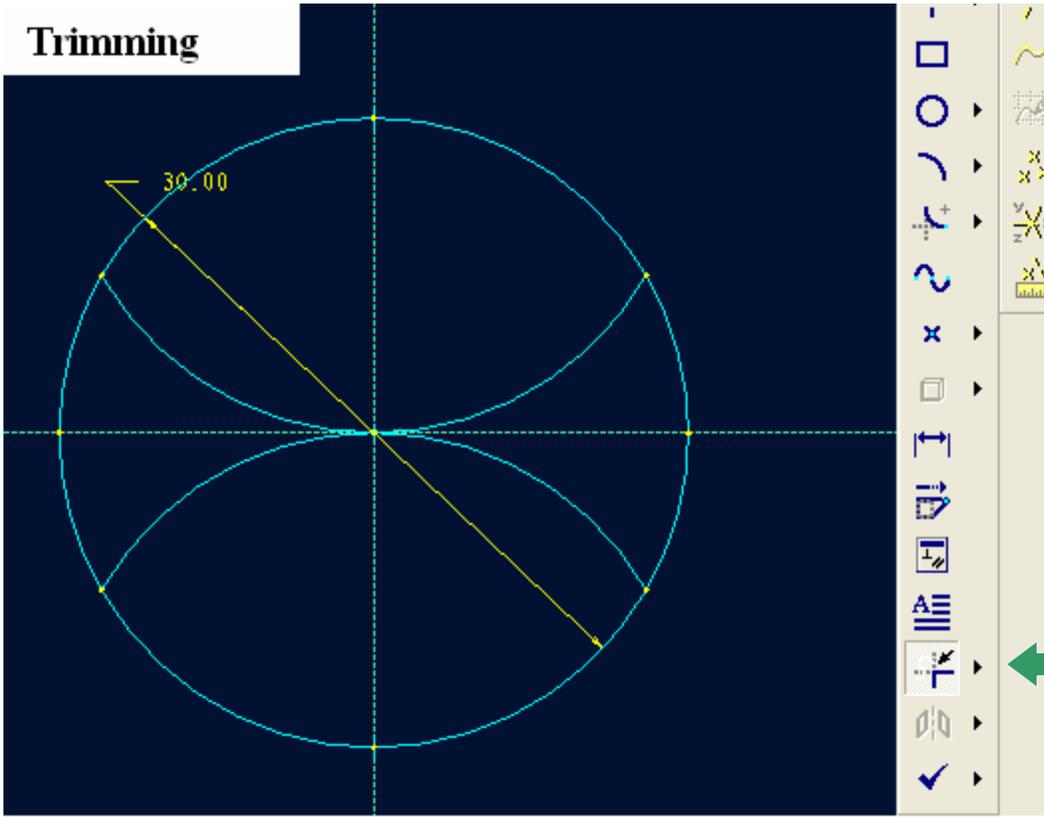
This feature requires a 2-D cross-section, so "Sketcher".



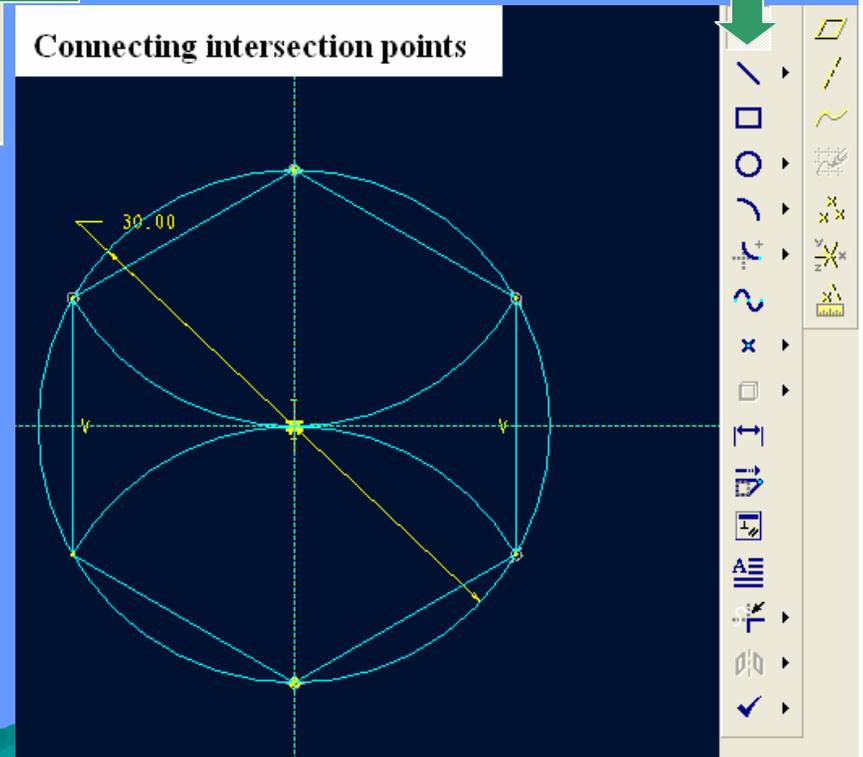
Step 4: Drawing circle and centerlines



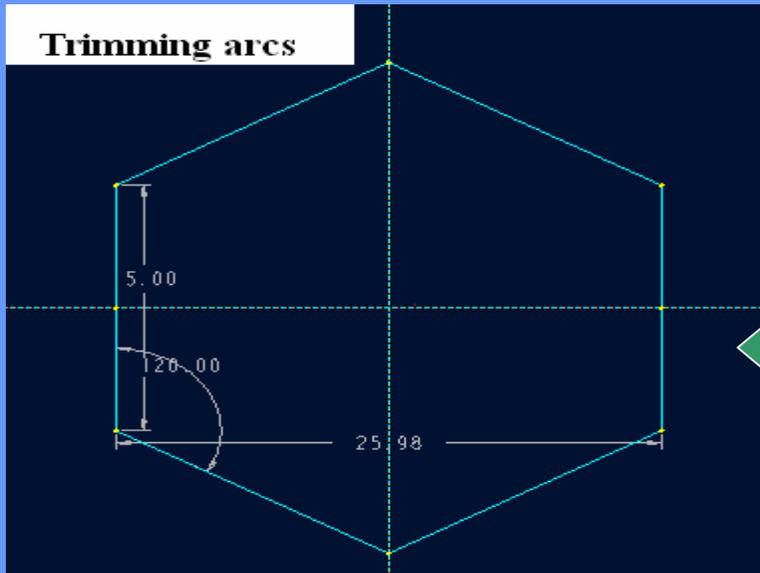
Trimming



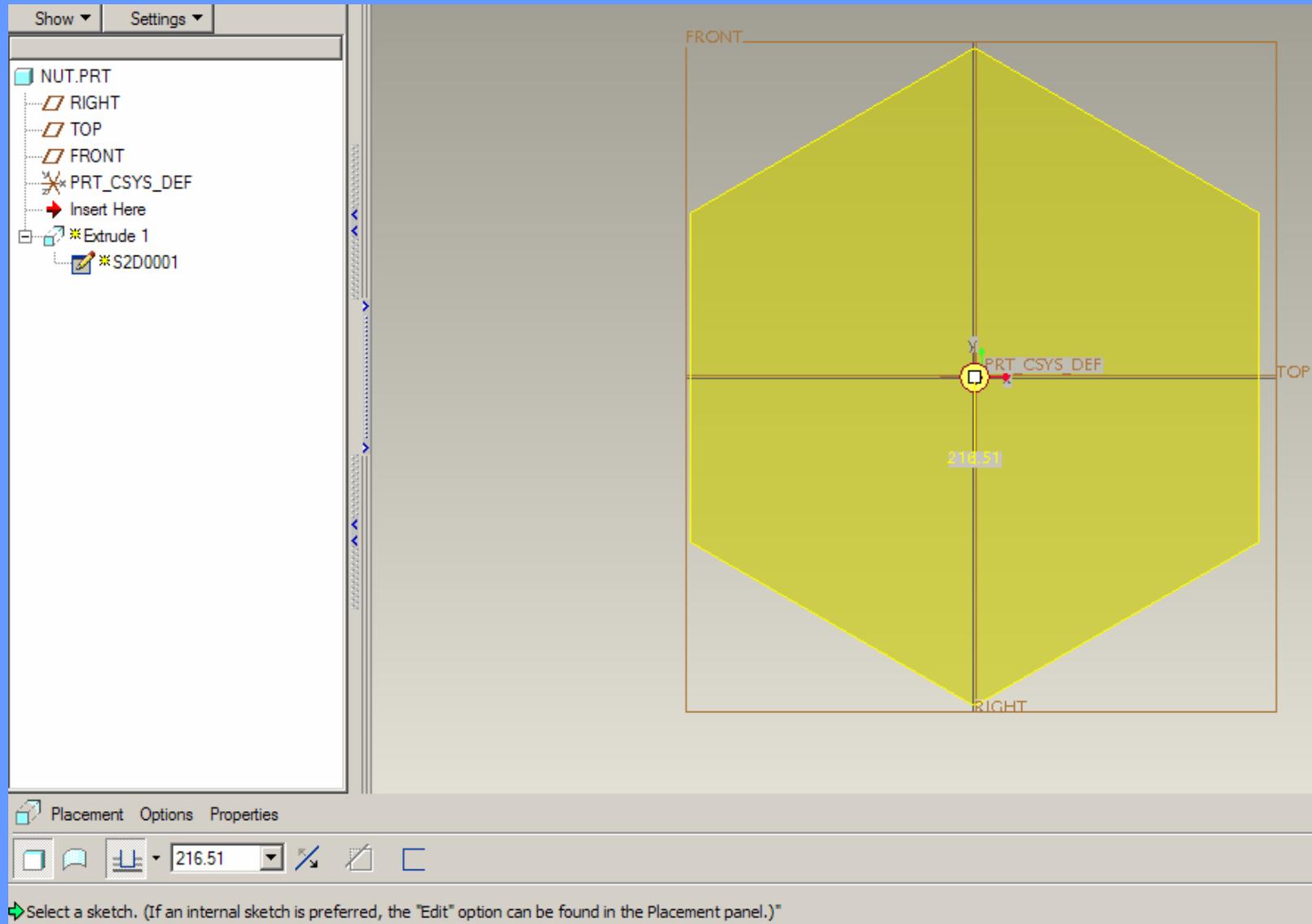
Connecting intersection points



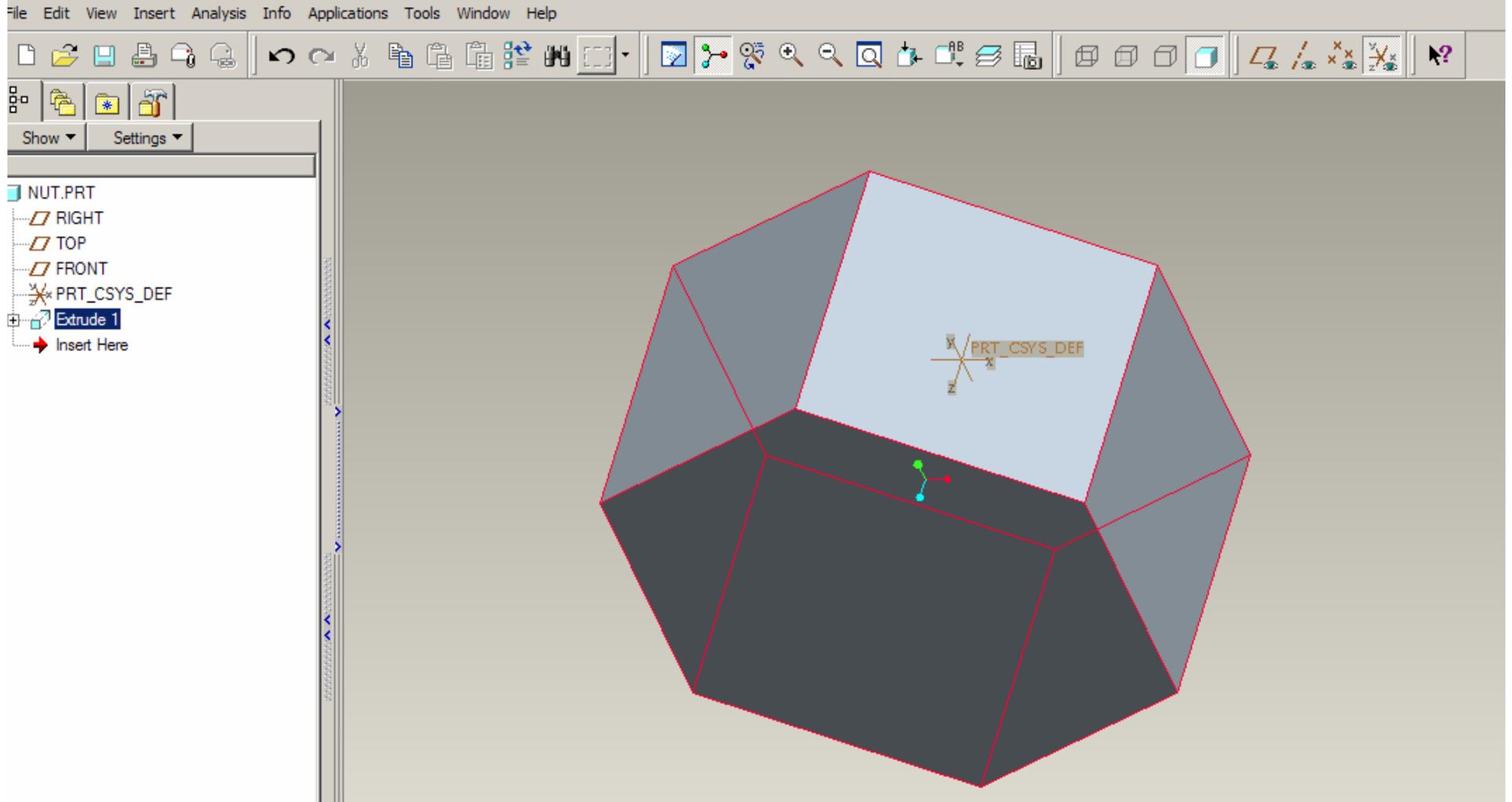
Trimming arcs



Completing sketch



Define depth = 15 mm



NUT.PRT

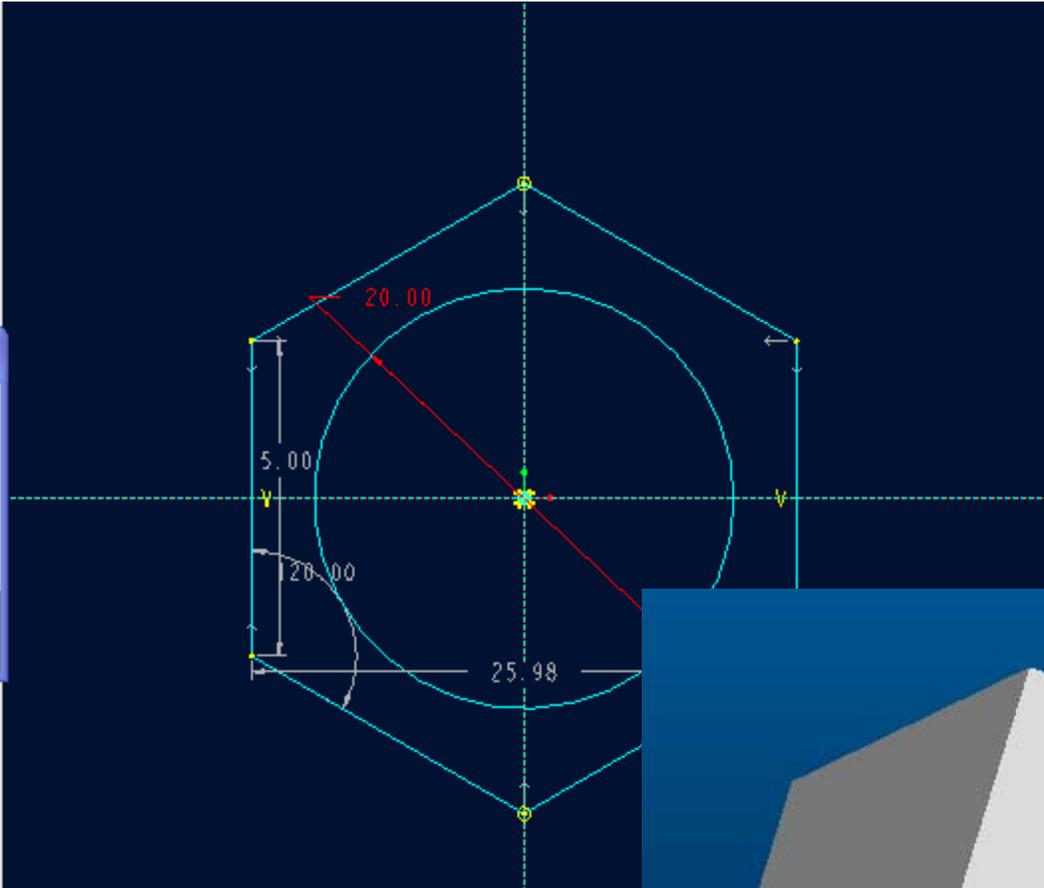
- RIGHT
- TOP
- FRONT
- PRT_CSYS_DEF
- Insert Here

PROTRUSION: Extrude

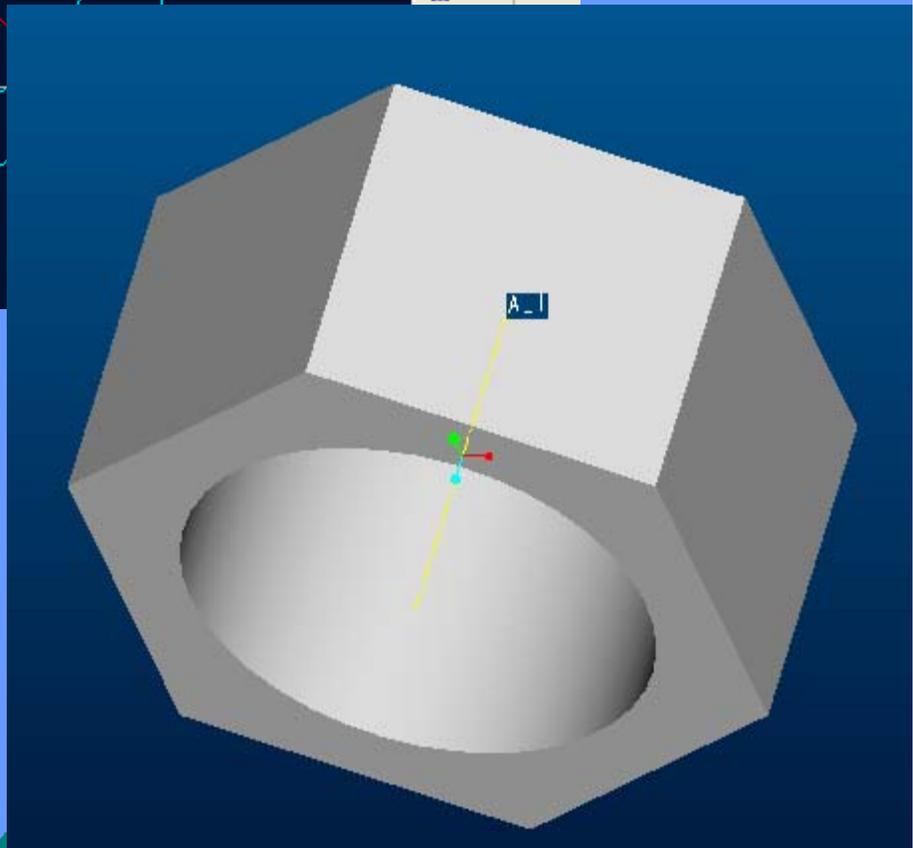
Element	Info
Attributes	Both Sides
> Section	Changing
Depth	Symmetrical BI

Define Refs Info

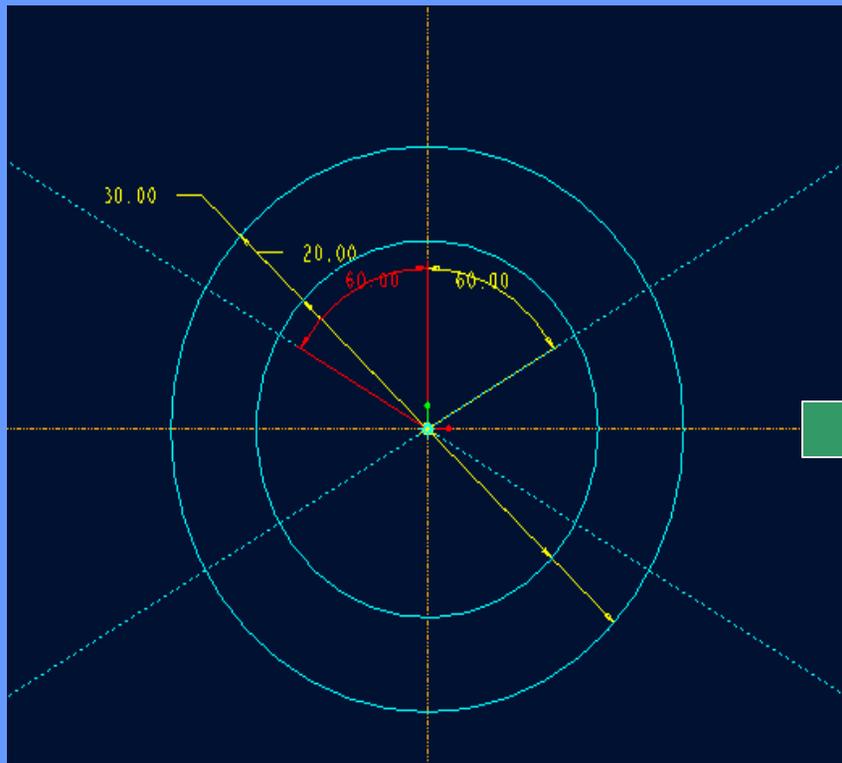
OK Cancel Preview



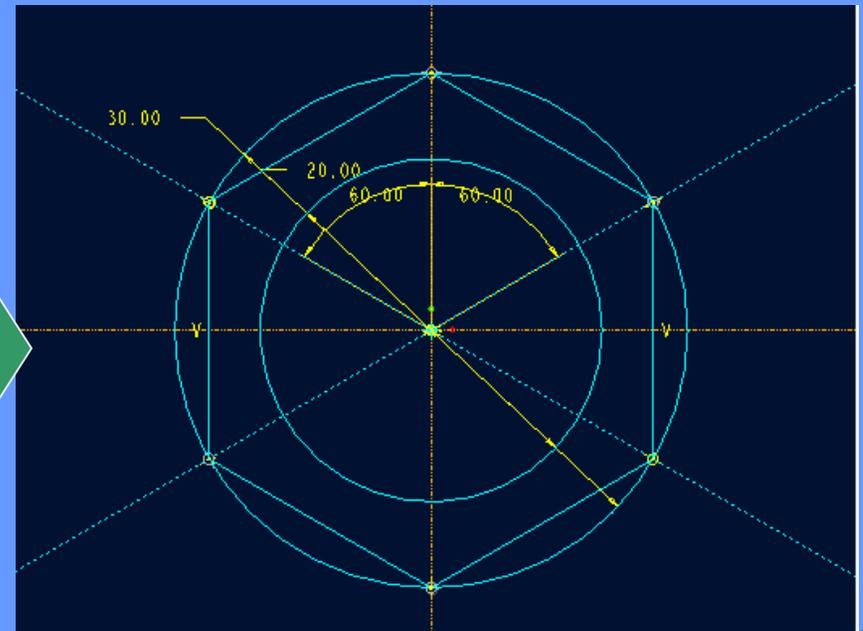
A vertical toolbar from a CAD software interface. It contains various icons for editing (like move, rotate, scale), viewing (like isometric, perspective), and other functions. The icons are arranged in a column on the right side of the interface.

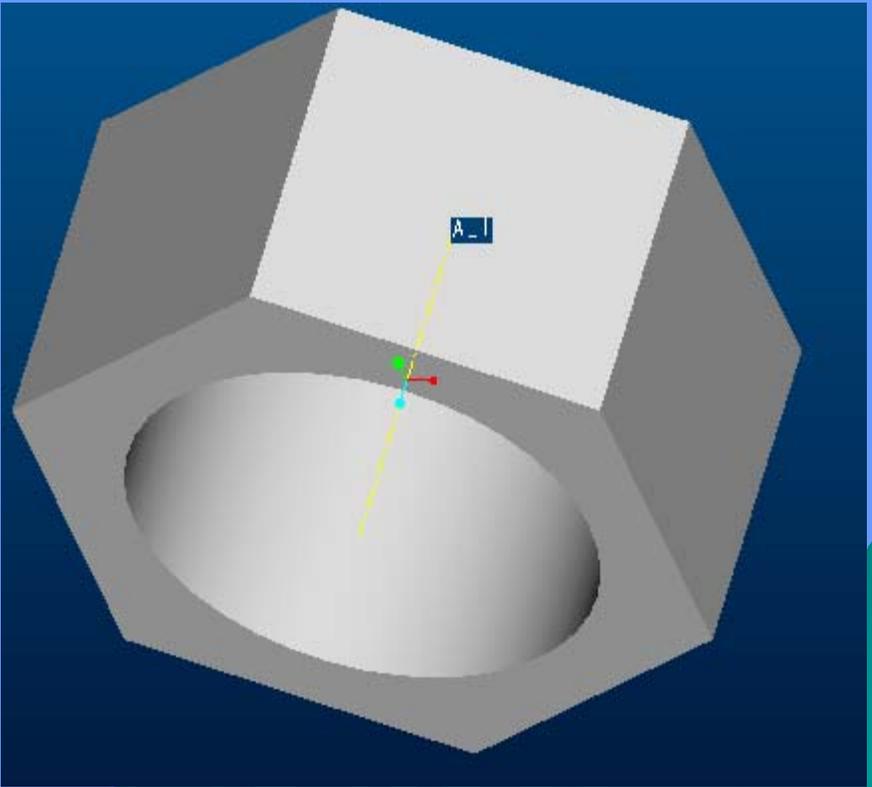
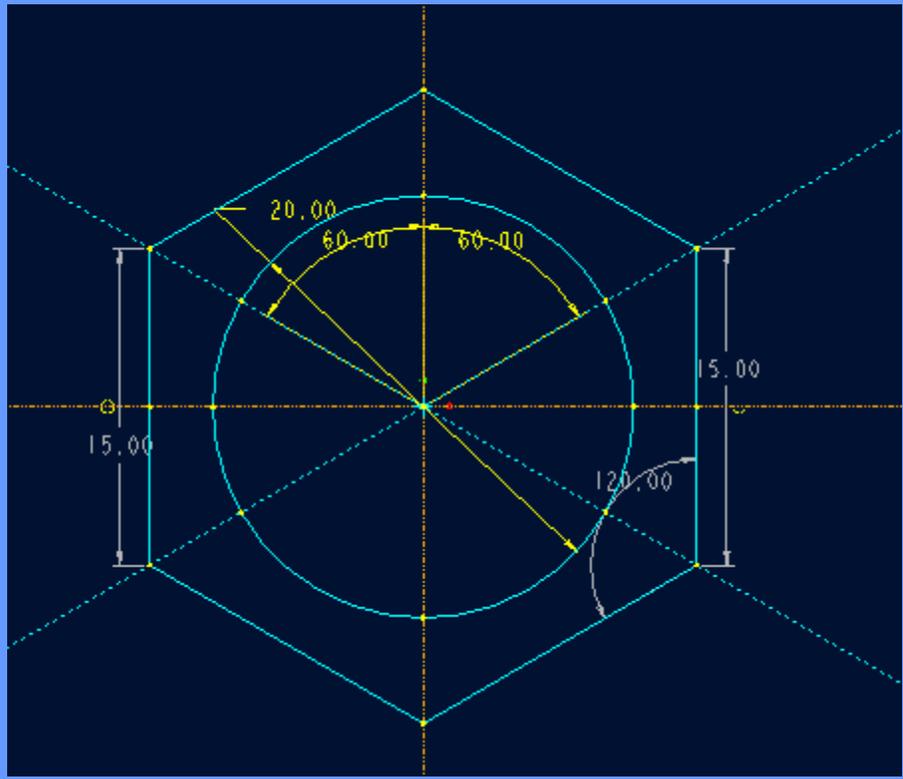


Alternative Method: using centerlines at 60 degree angles

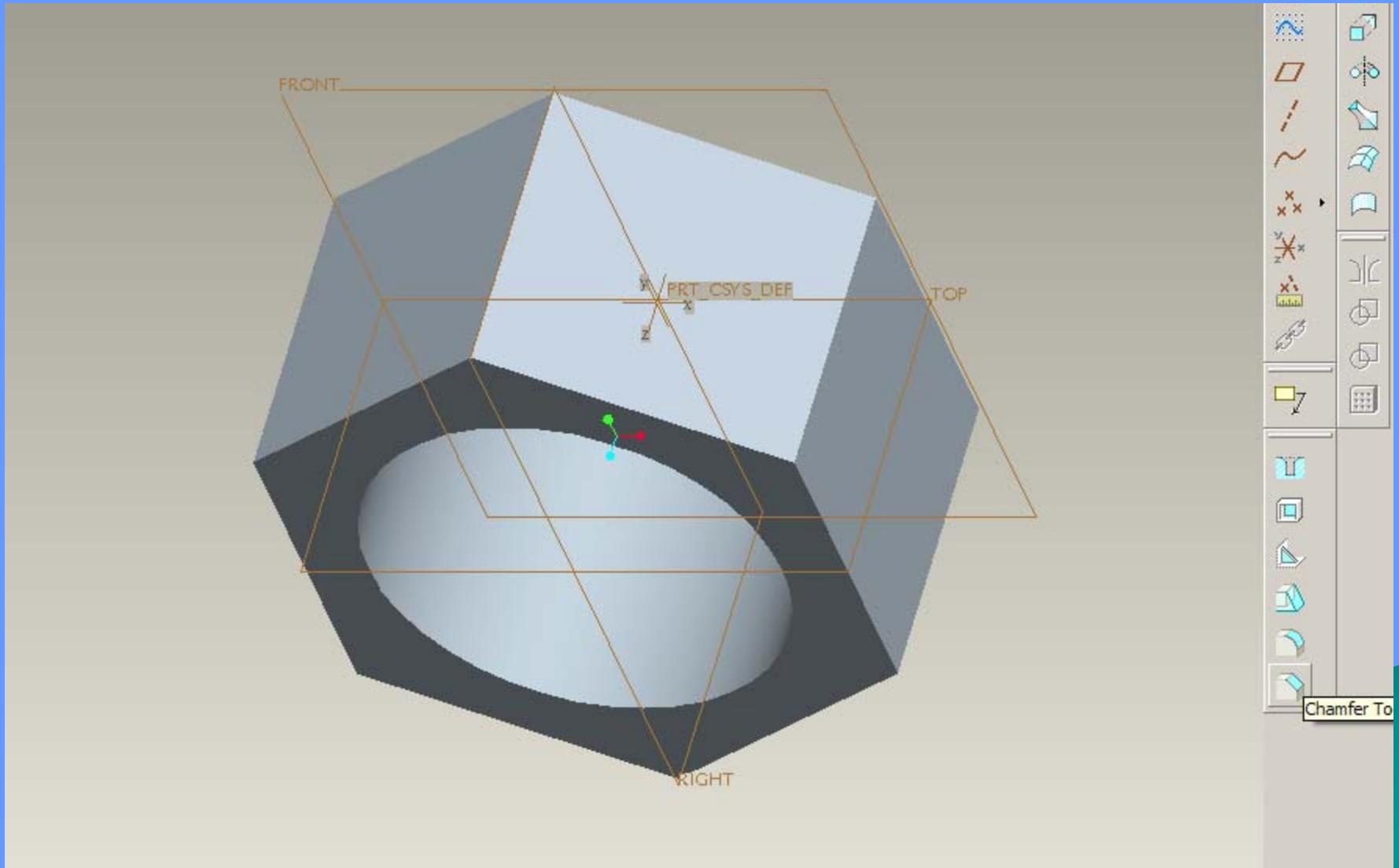


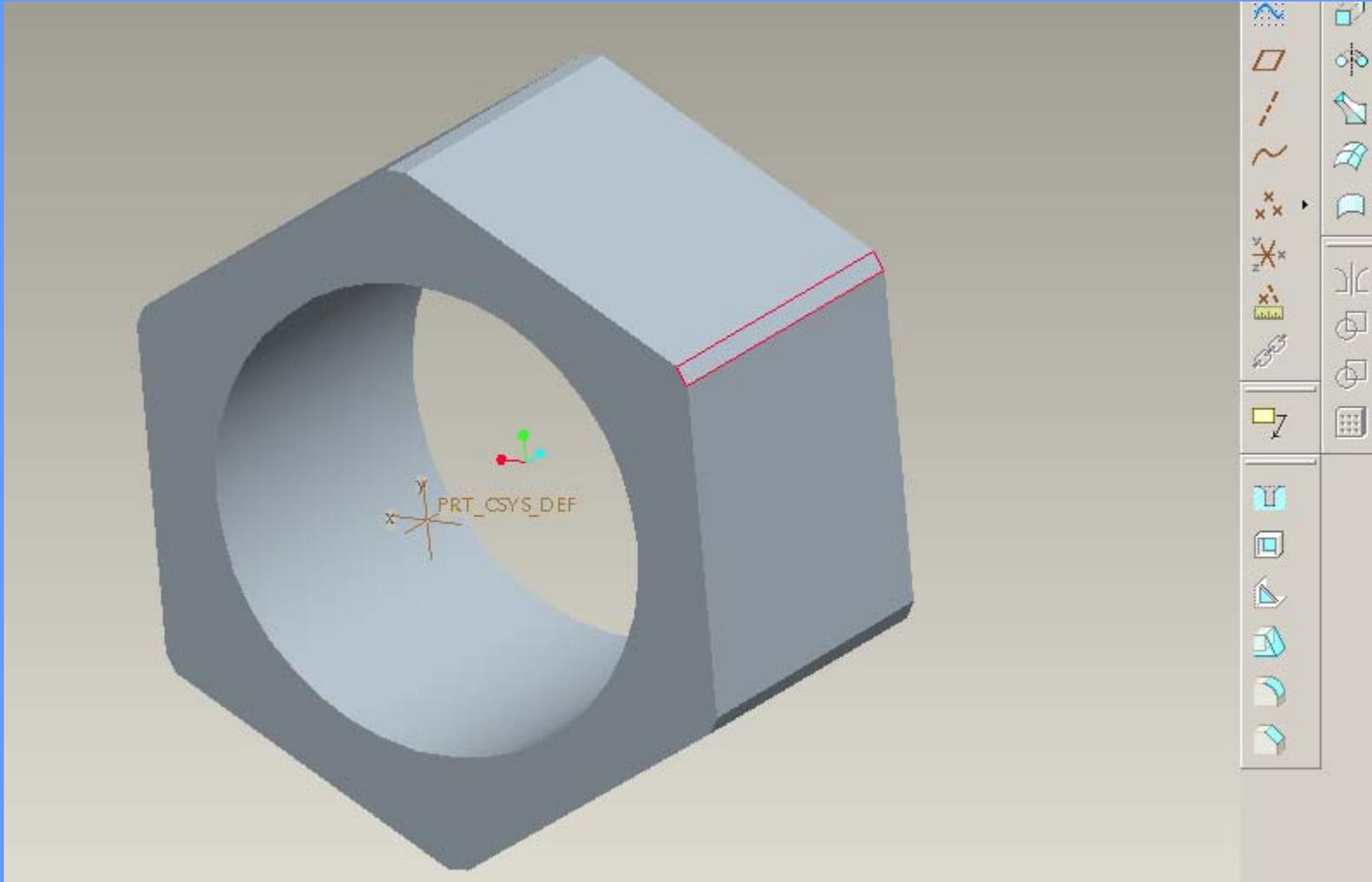
Dimensioning



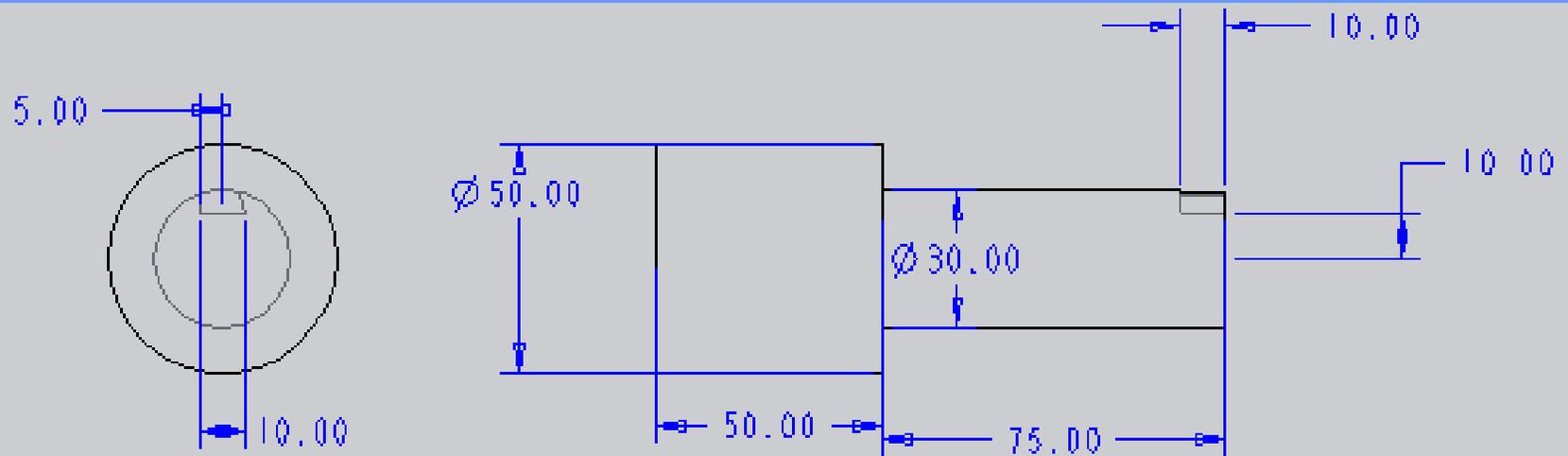
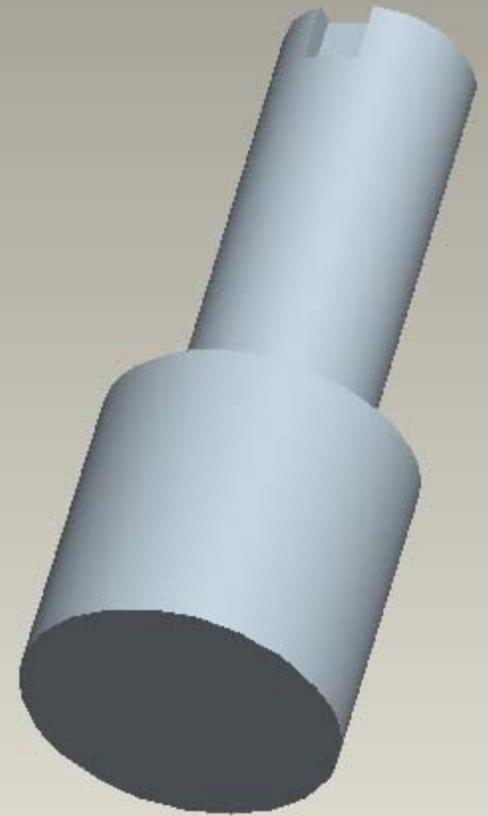


Create 6 chamfer features

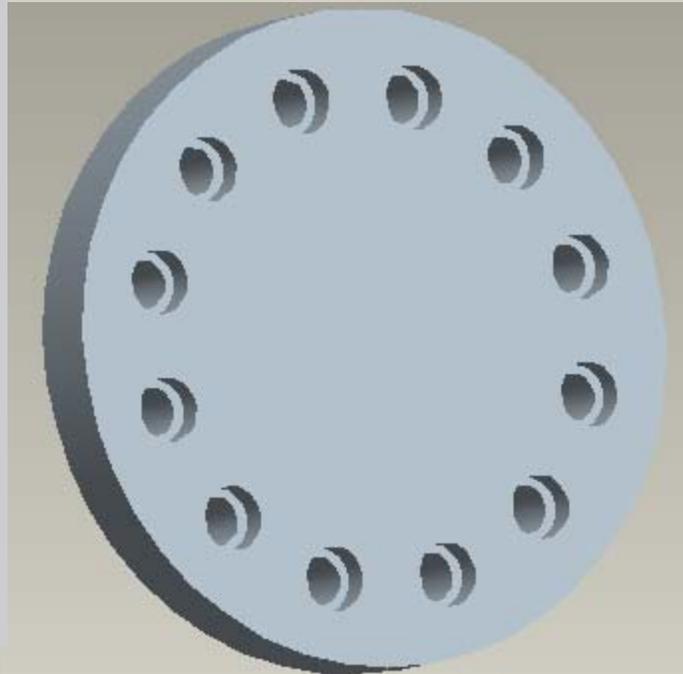
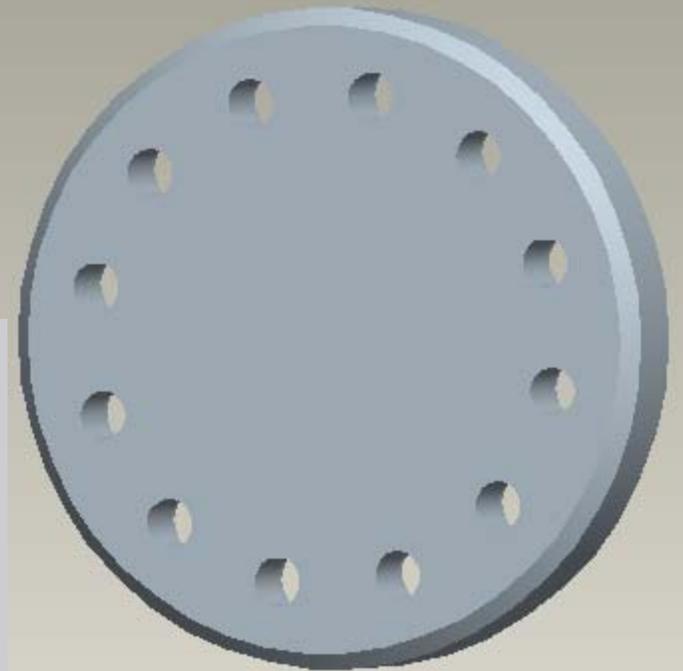
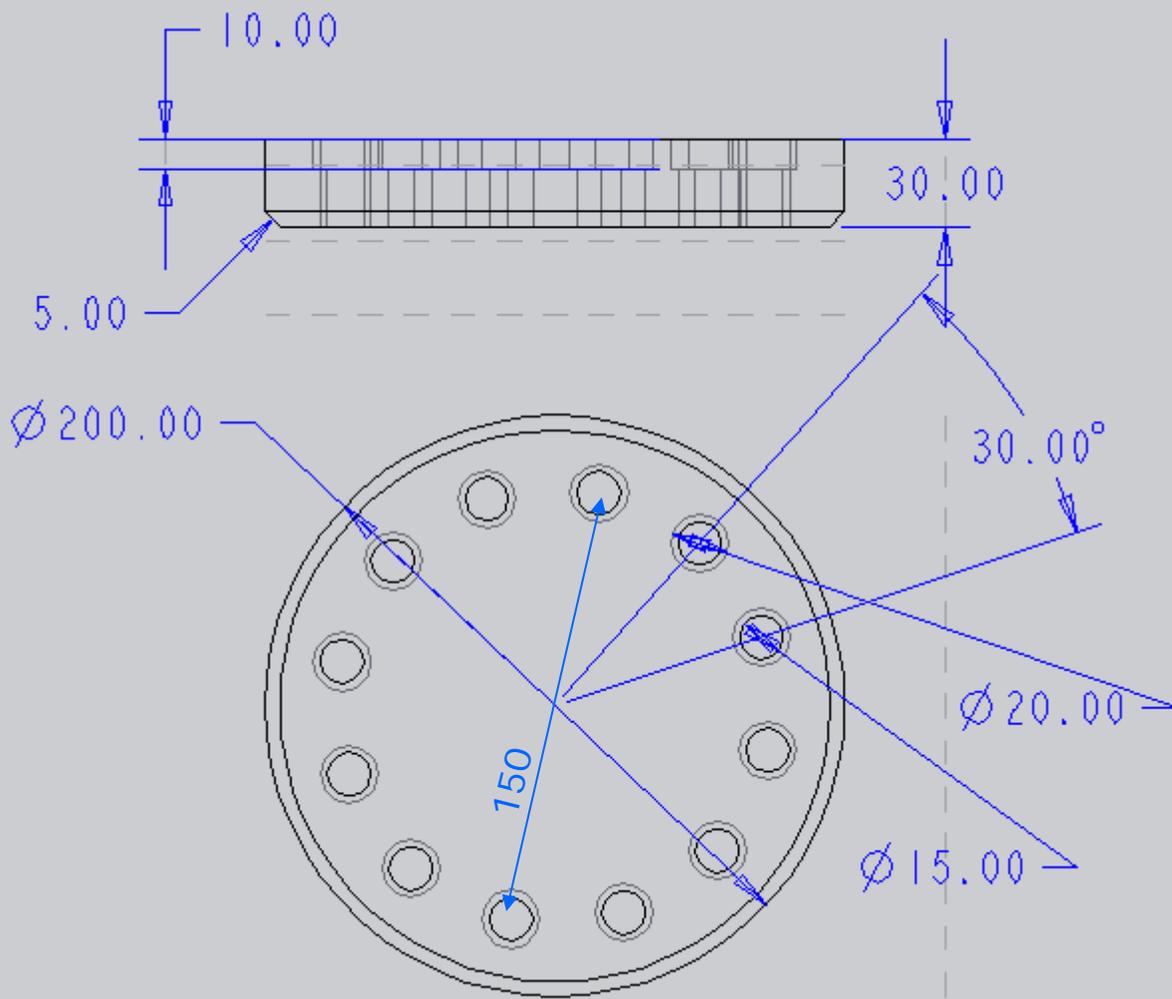




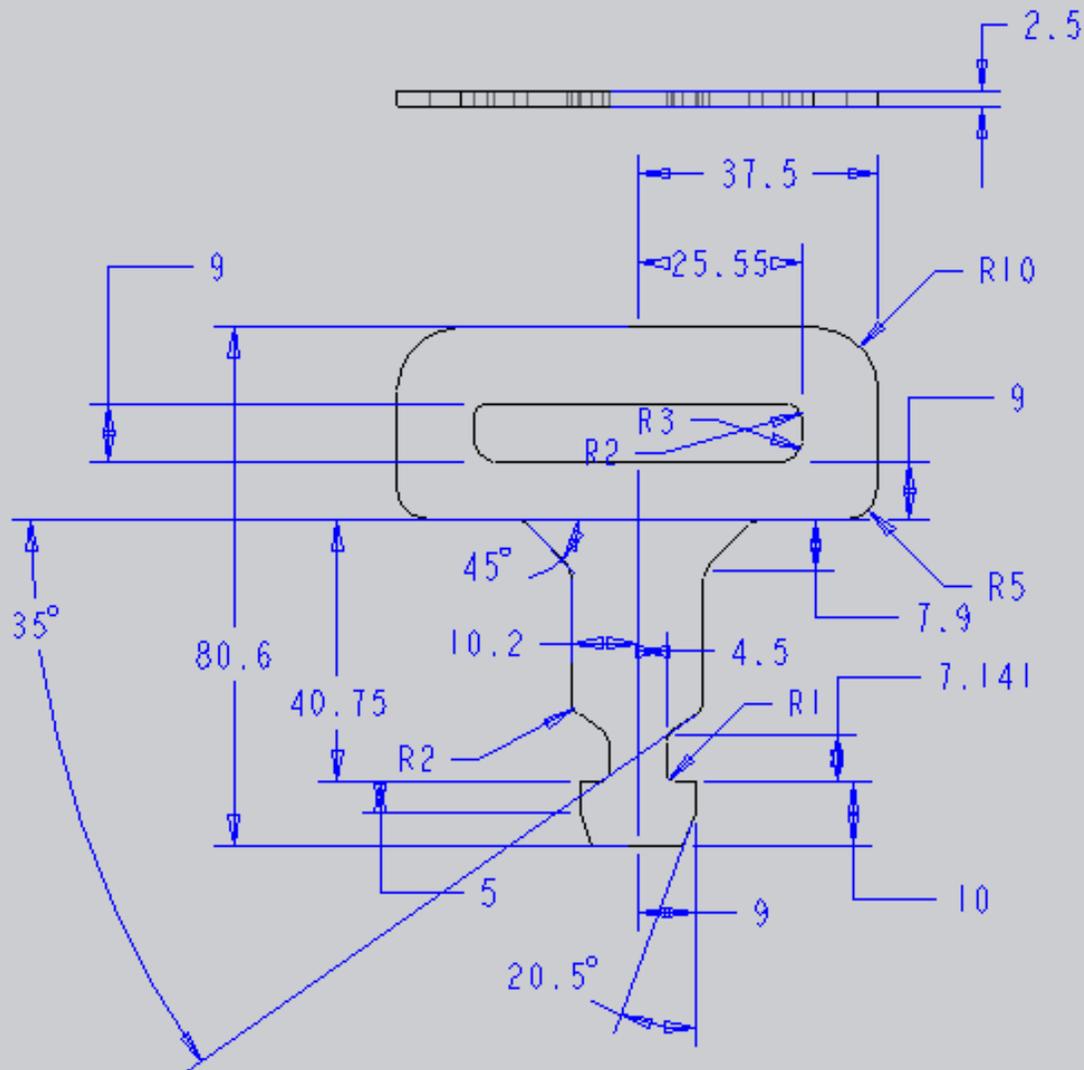
Exercise I



Exercise II



Exercise III



Part Modeling

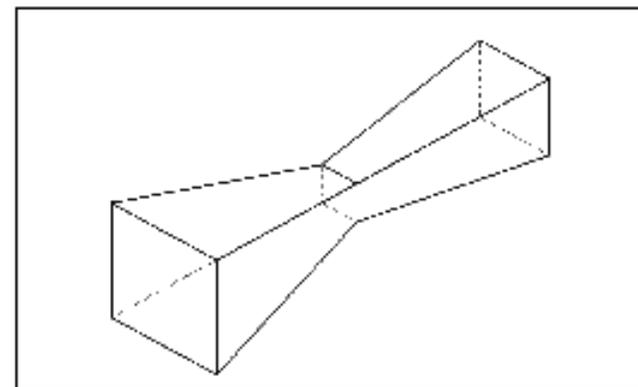
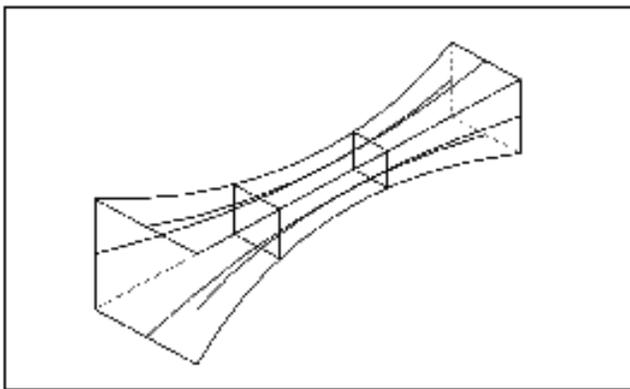
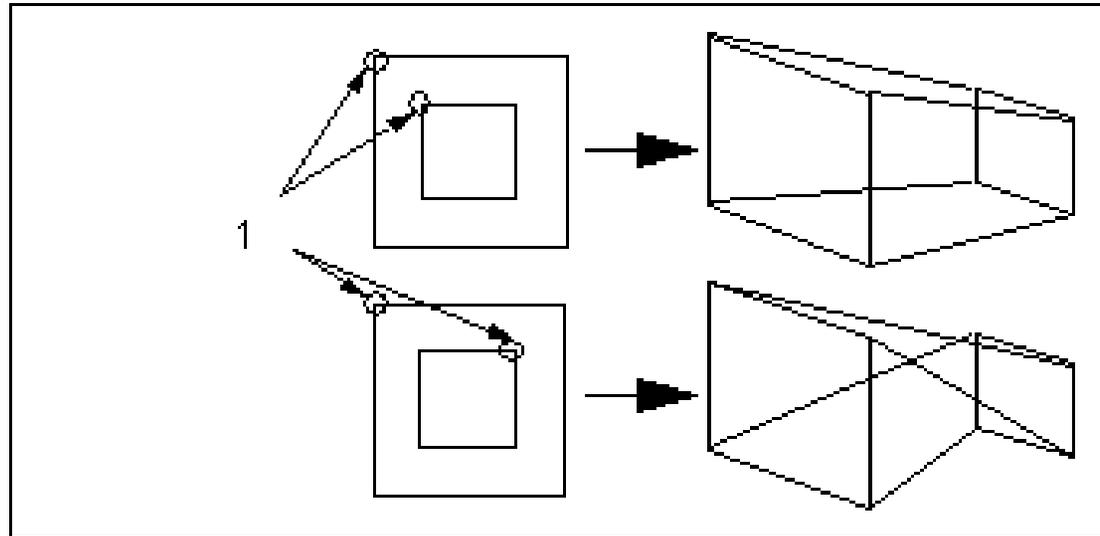
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Department of Mechanical Engineering
INDIAN INSTITUTE OF TECHNOLOGY BOMBAY

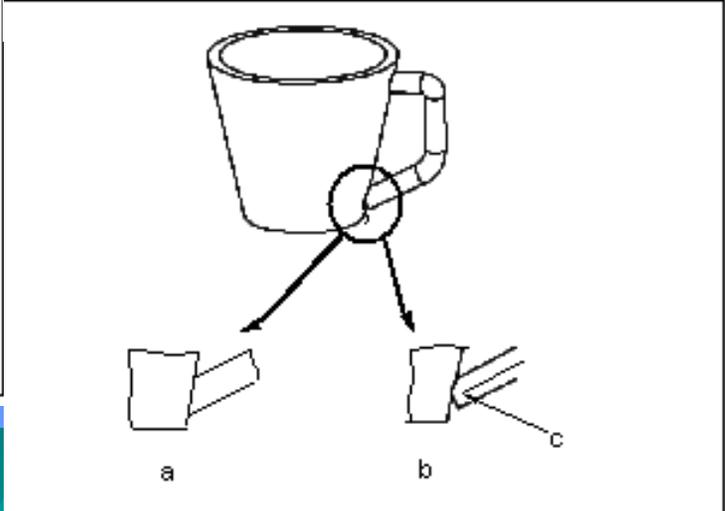
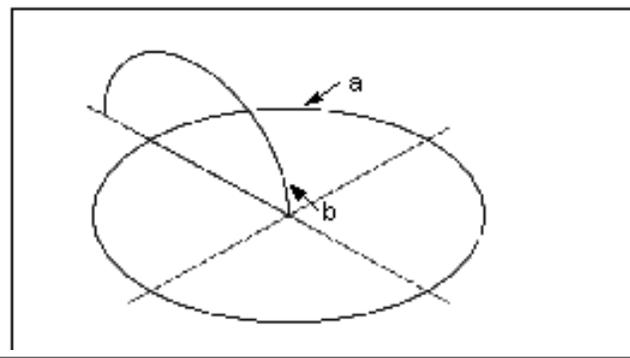
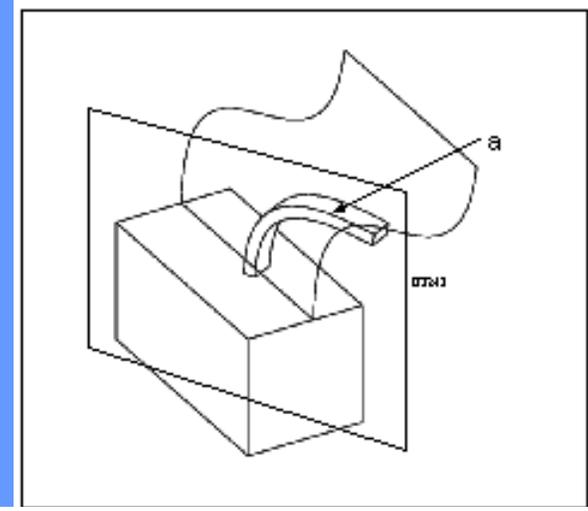
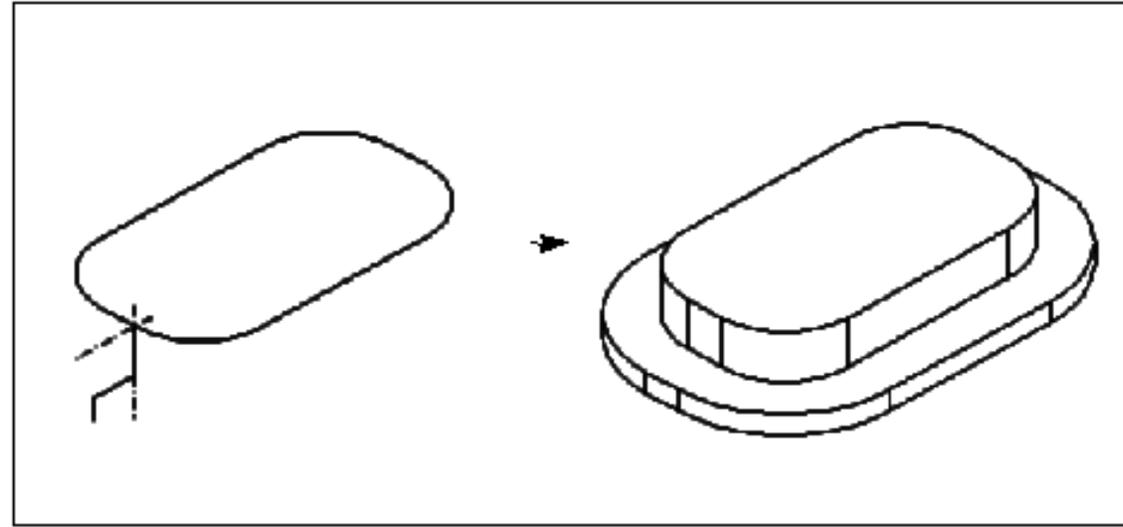
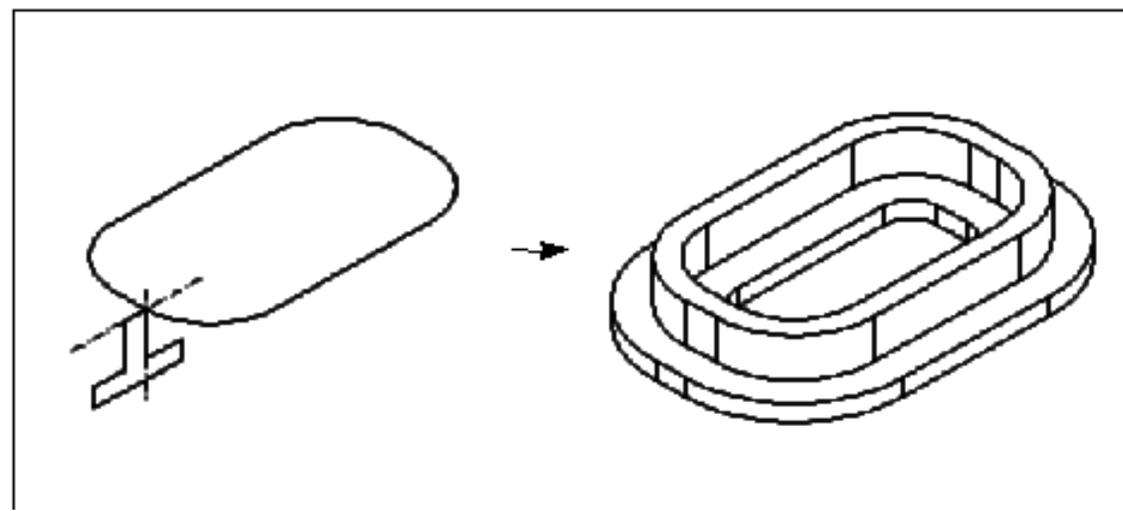
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hirani@me.iitb.ac.in

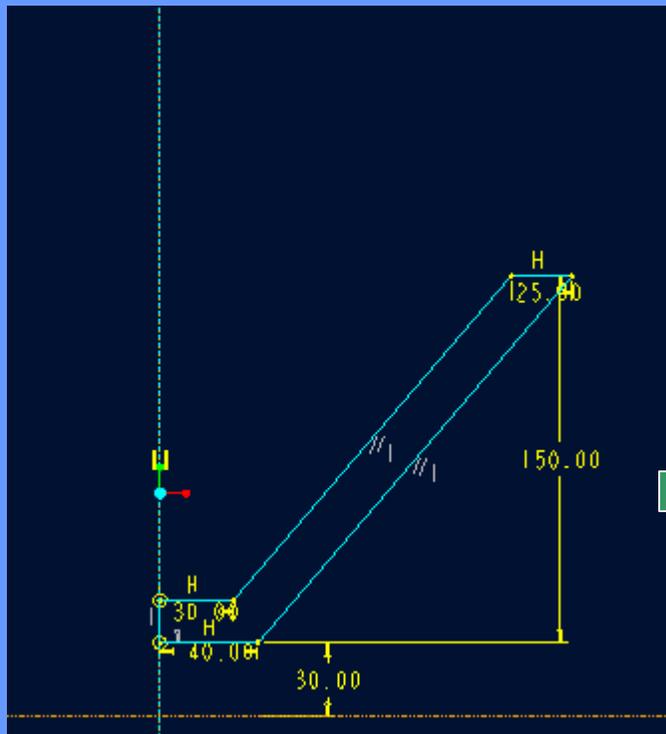
Different Blend Geometries



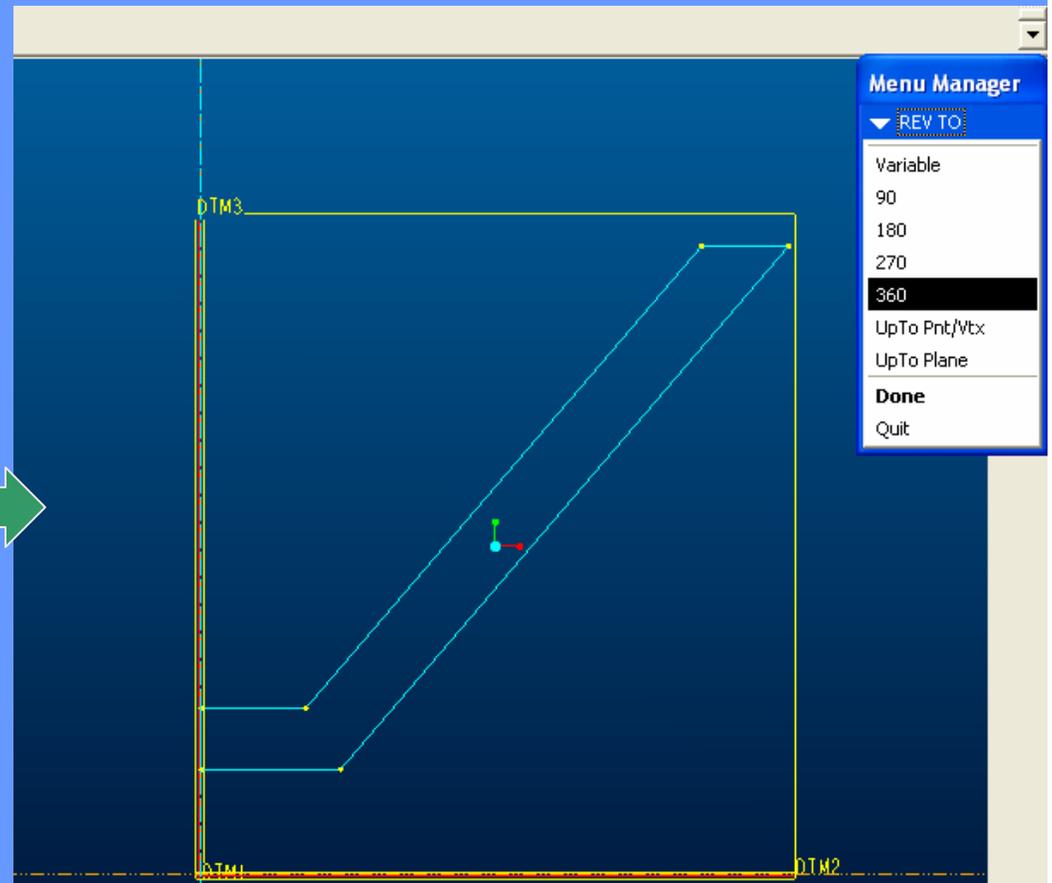
Sweeps



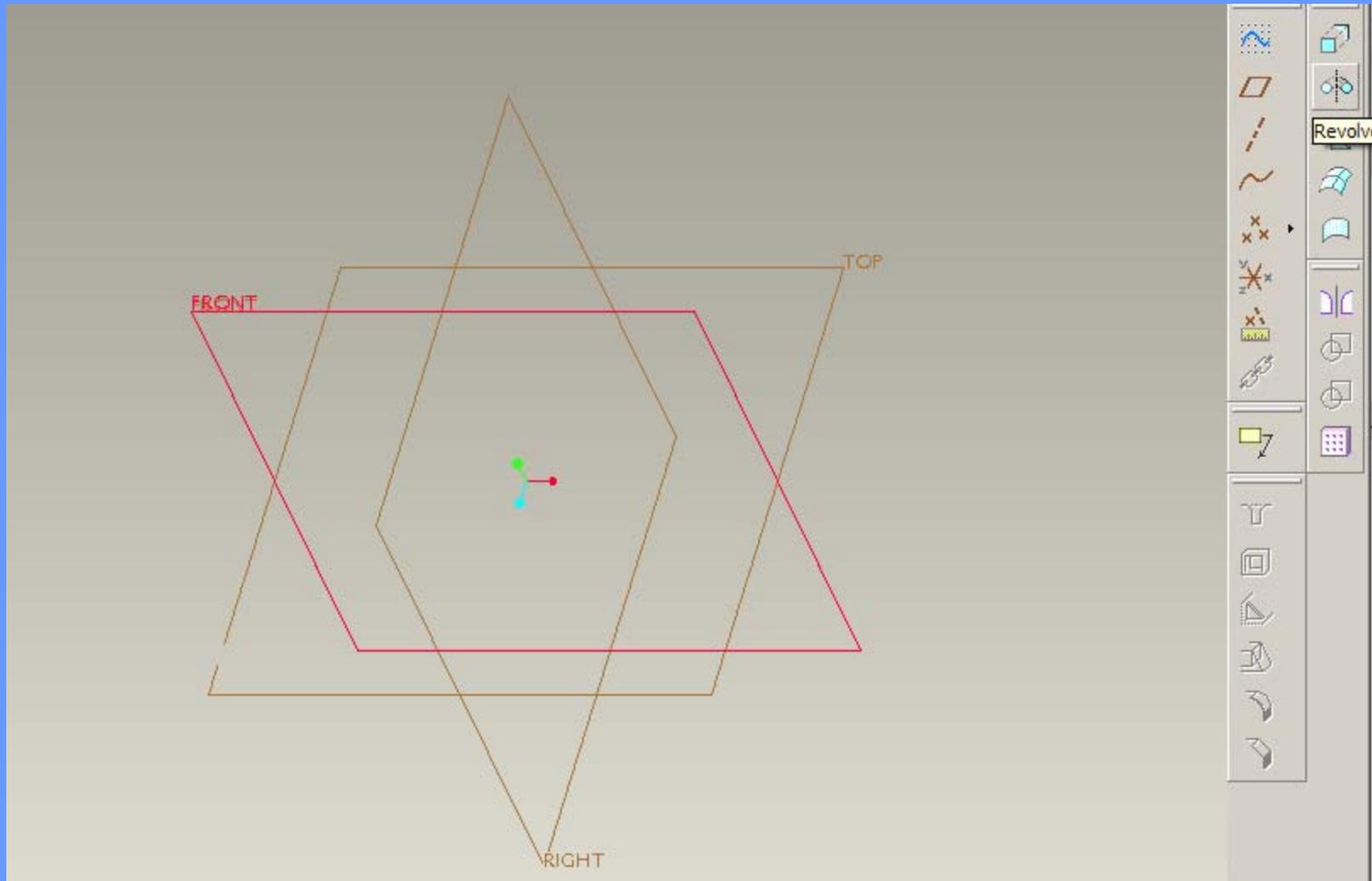
Part Modeling using Revolve



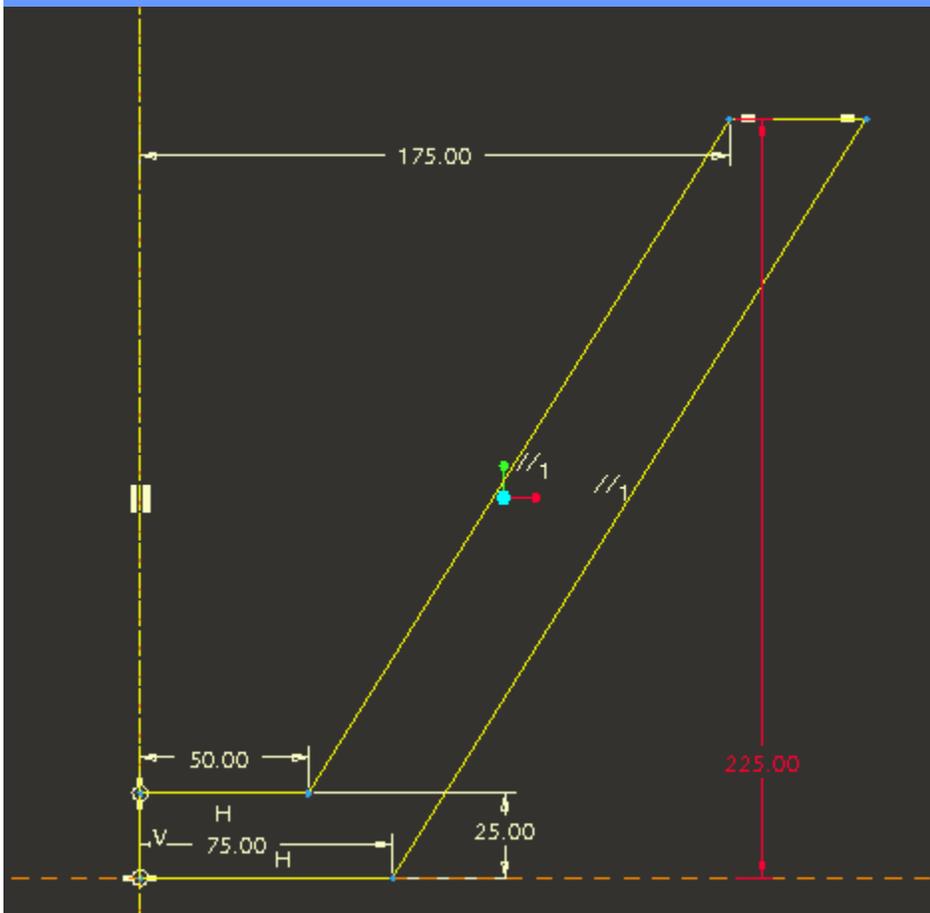
Sketch on one side of centerline



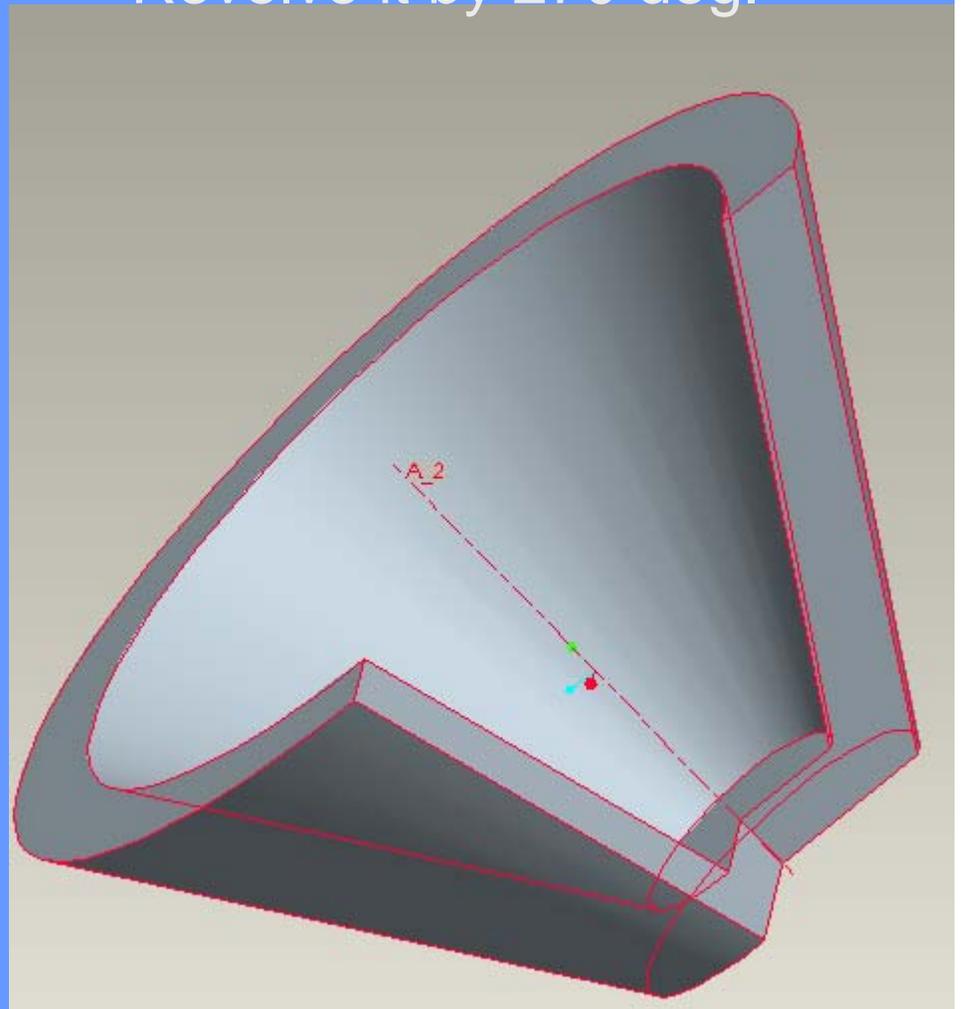
Step 1.. Choose Plane and Revolve tool



Step II: Sketch one side centerline and geometry given following figure



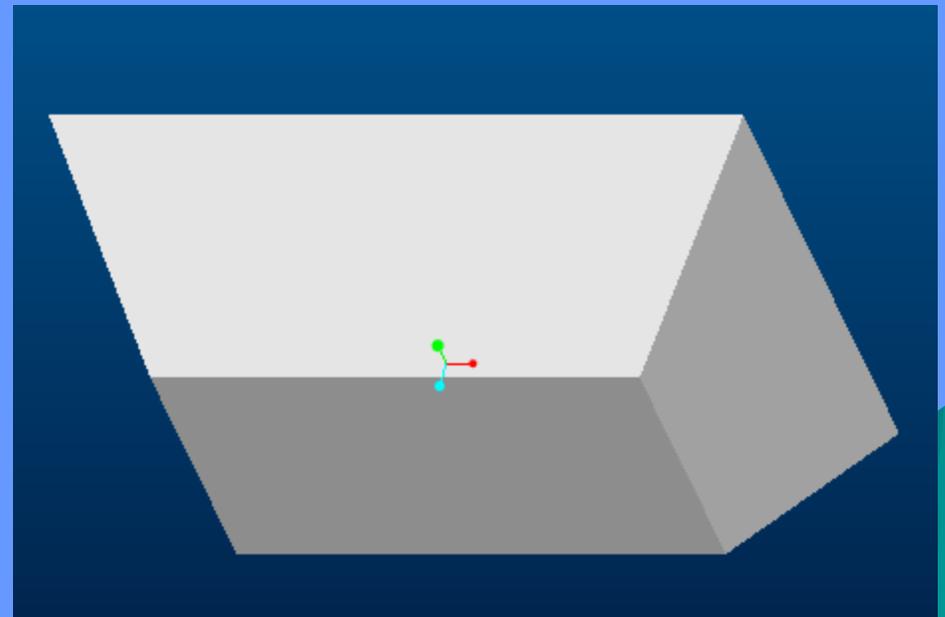
Revolve it by 270 deg.



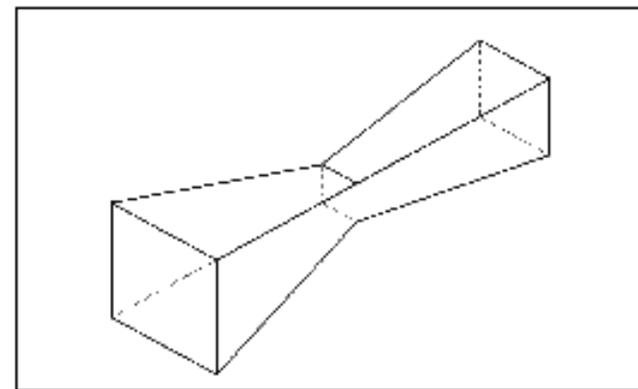
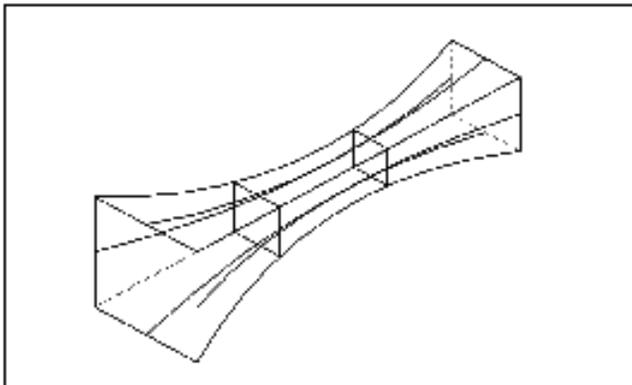
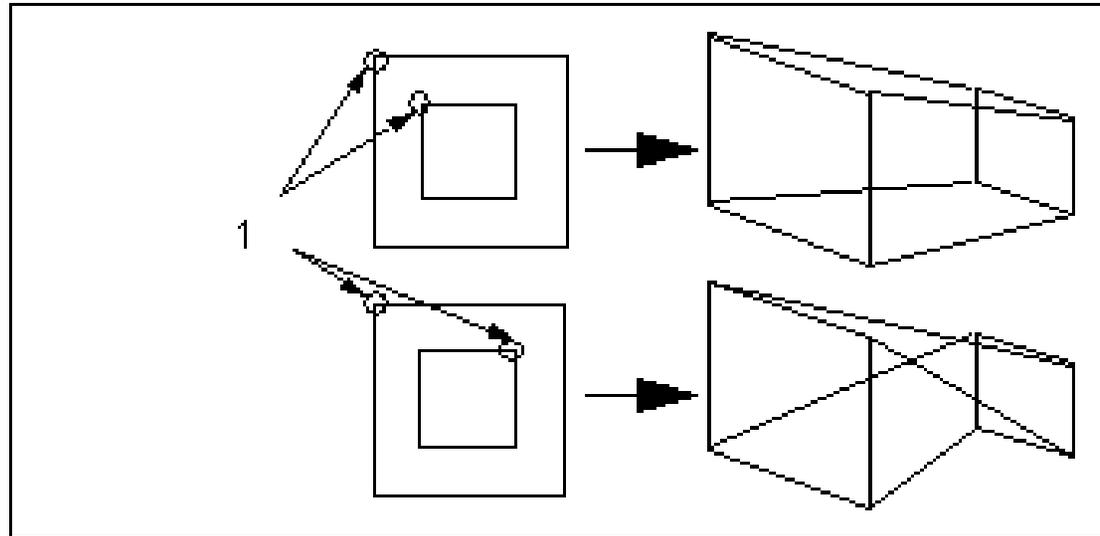
BLENDS

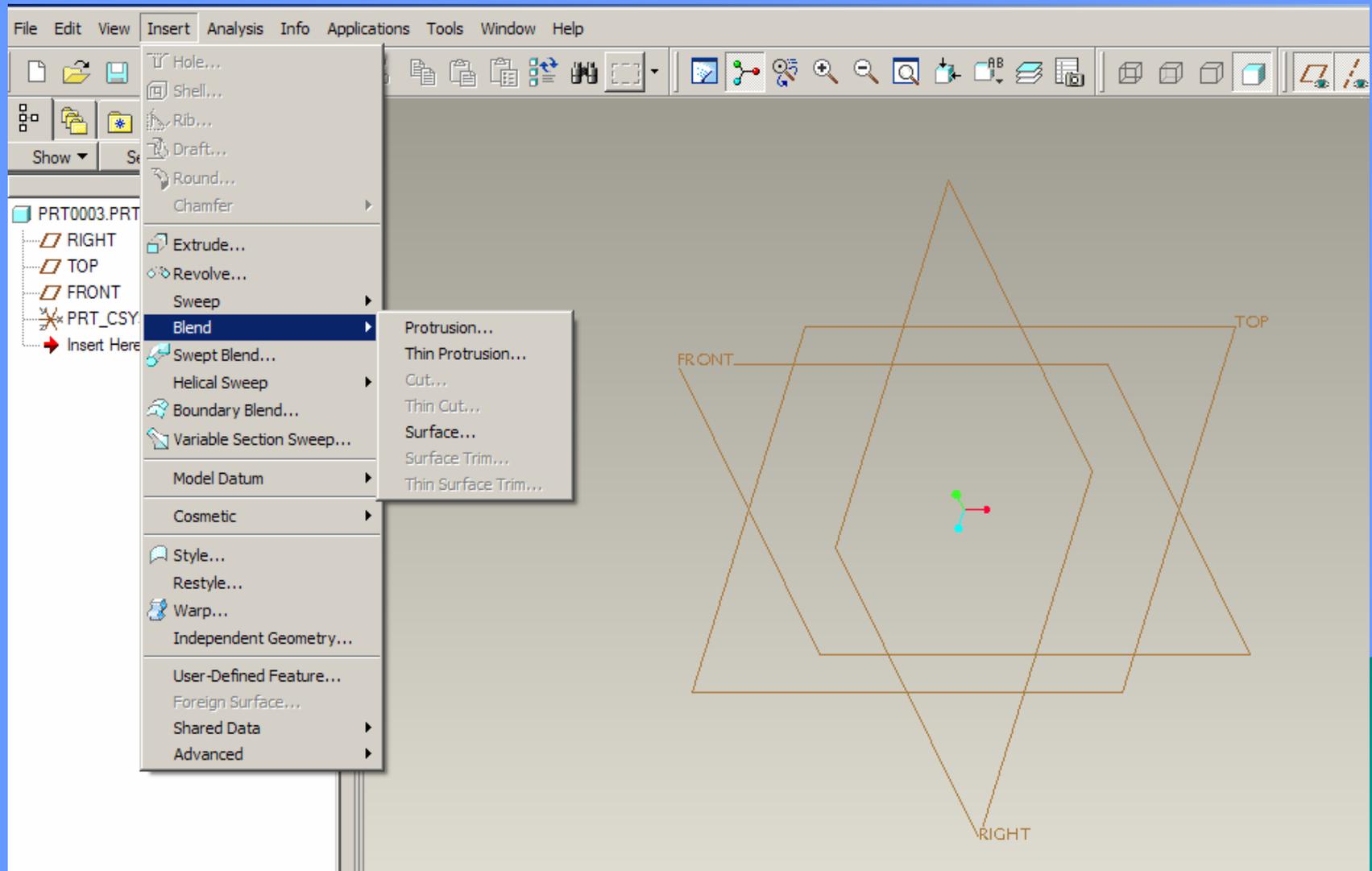
◆ Creation of Geometry between two or more sections of varying size, shape & location

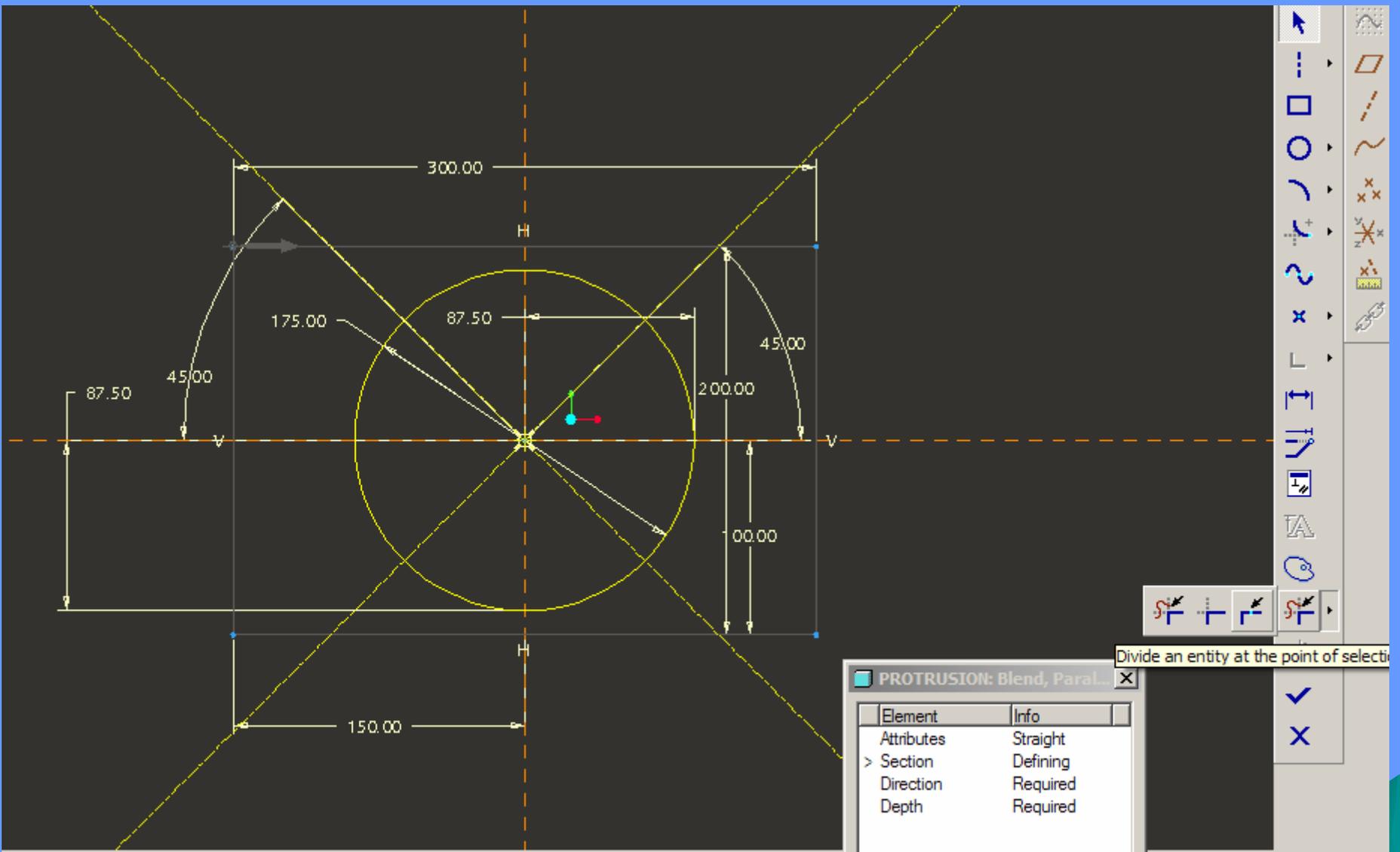
- Parallel Blends
- Rotational Blends
- General Blends



Blending Cross-sections







PROTRUSION: Blend, Paral...

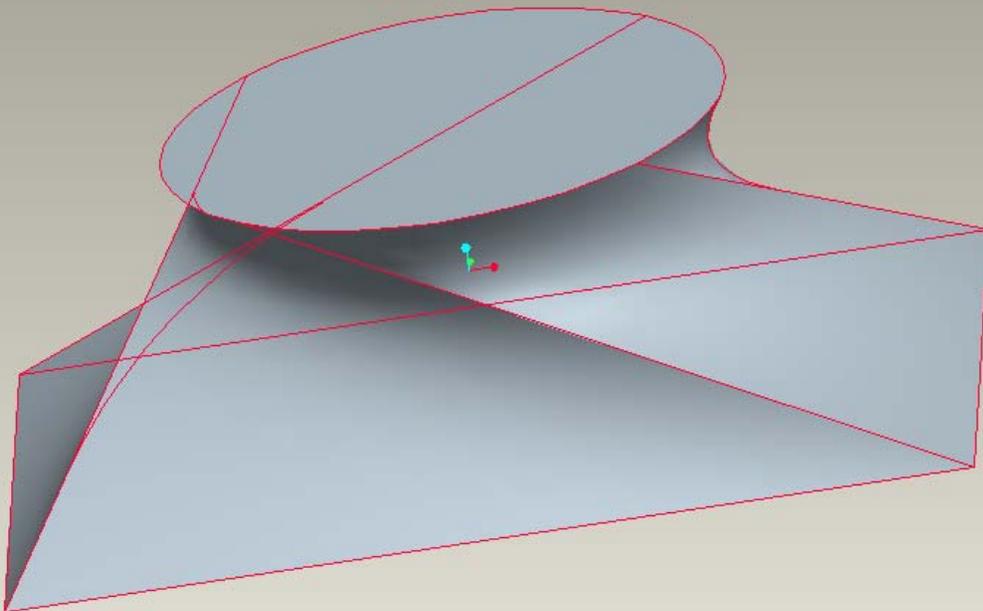
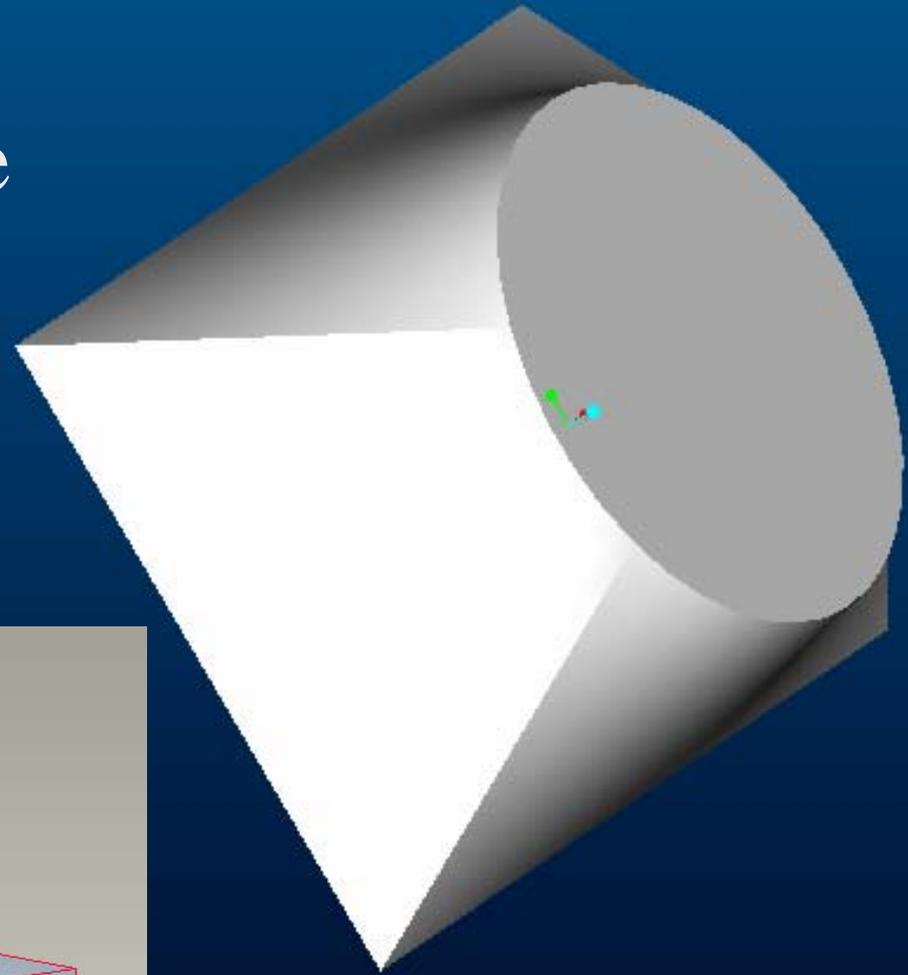
Element	Info
Attributes	Straight
> Section	Defining
Direction	Required
Depth	Required

Define Refs Info
OK Cancel Preview

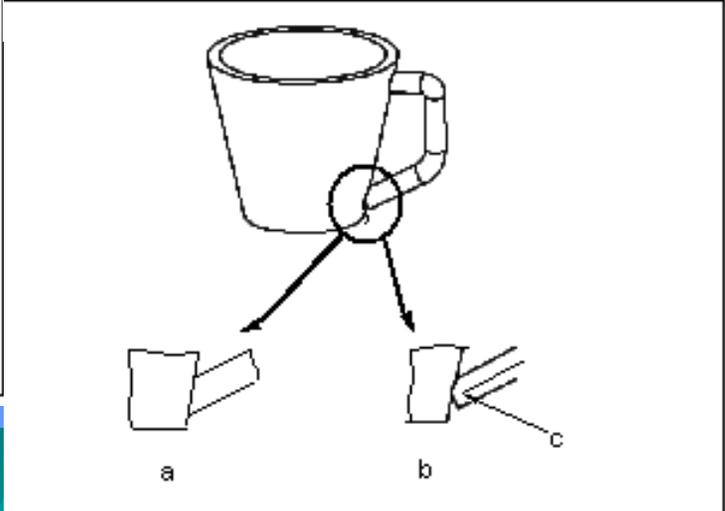
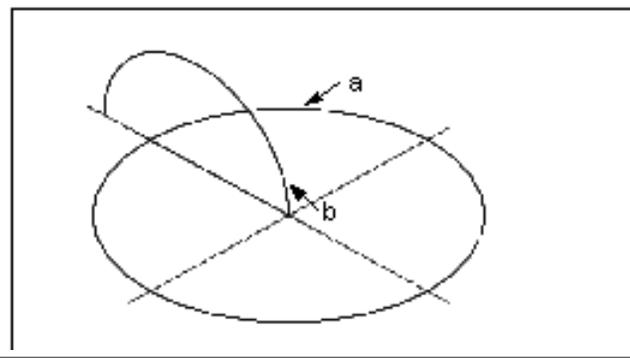
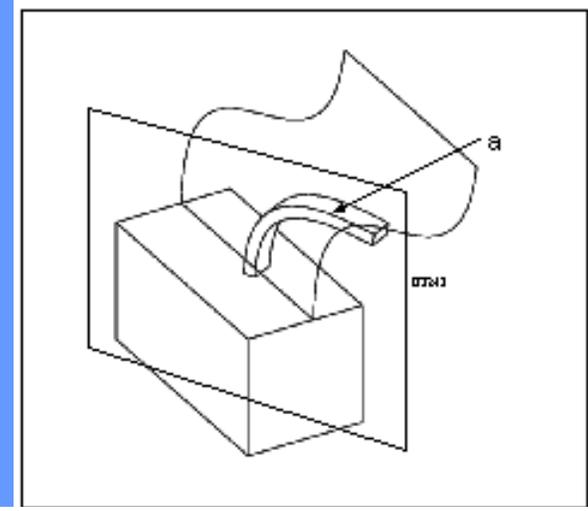
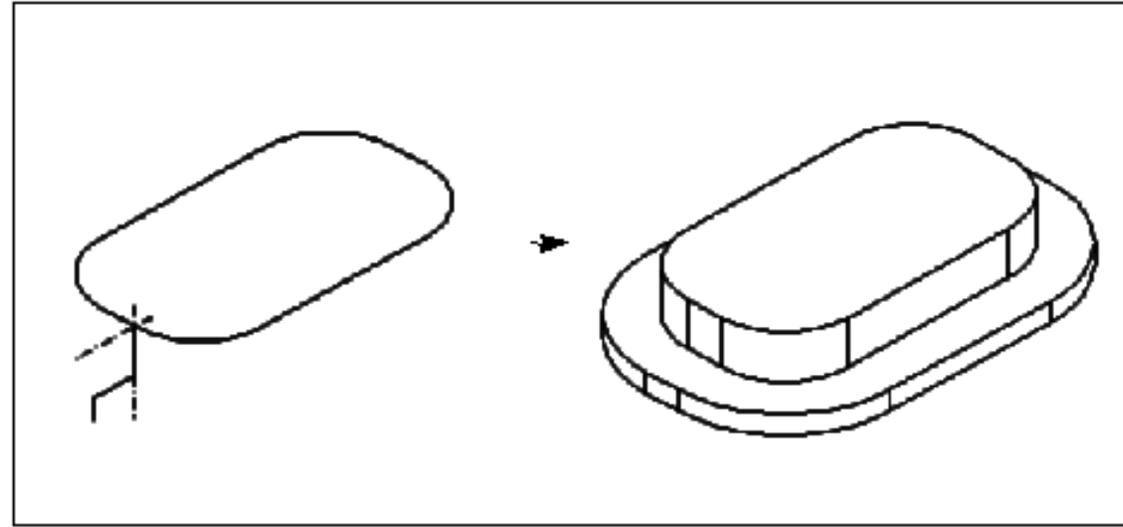
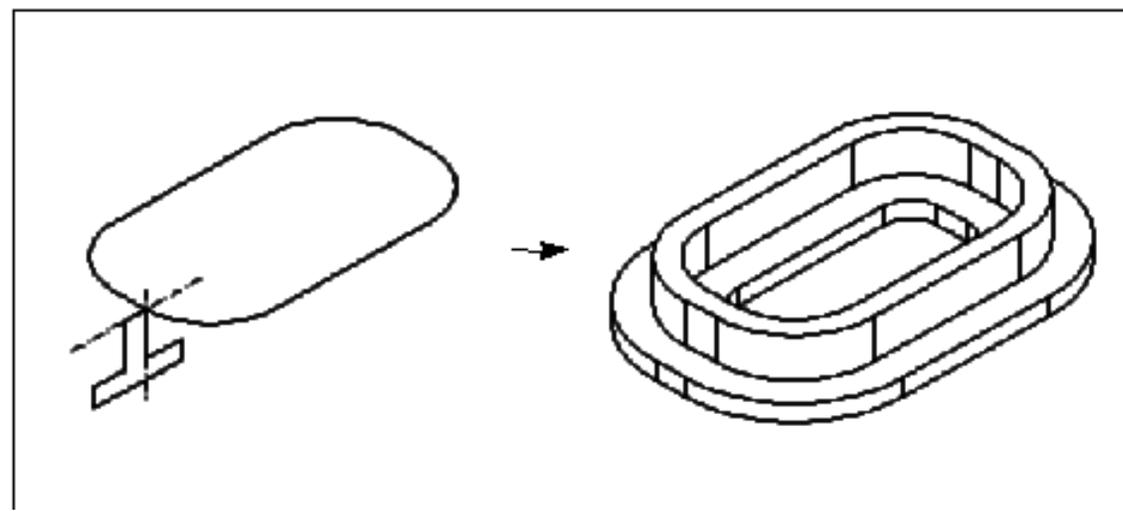
Divide an entity at the point of selection

right click to lock it. Use Tab key to toggle active constraint.

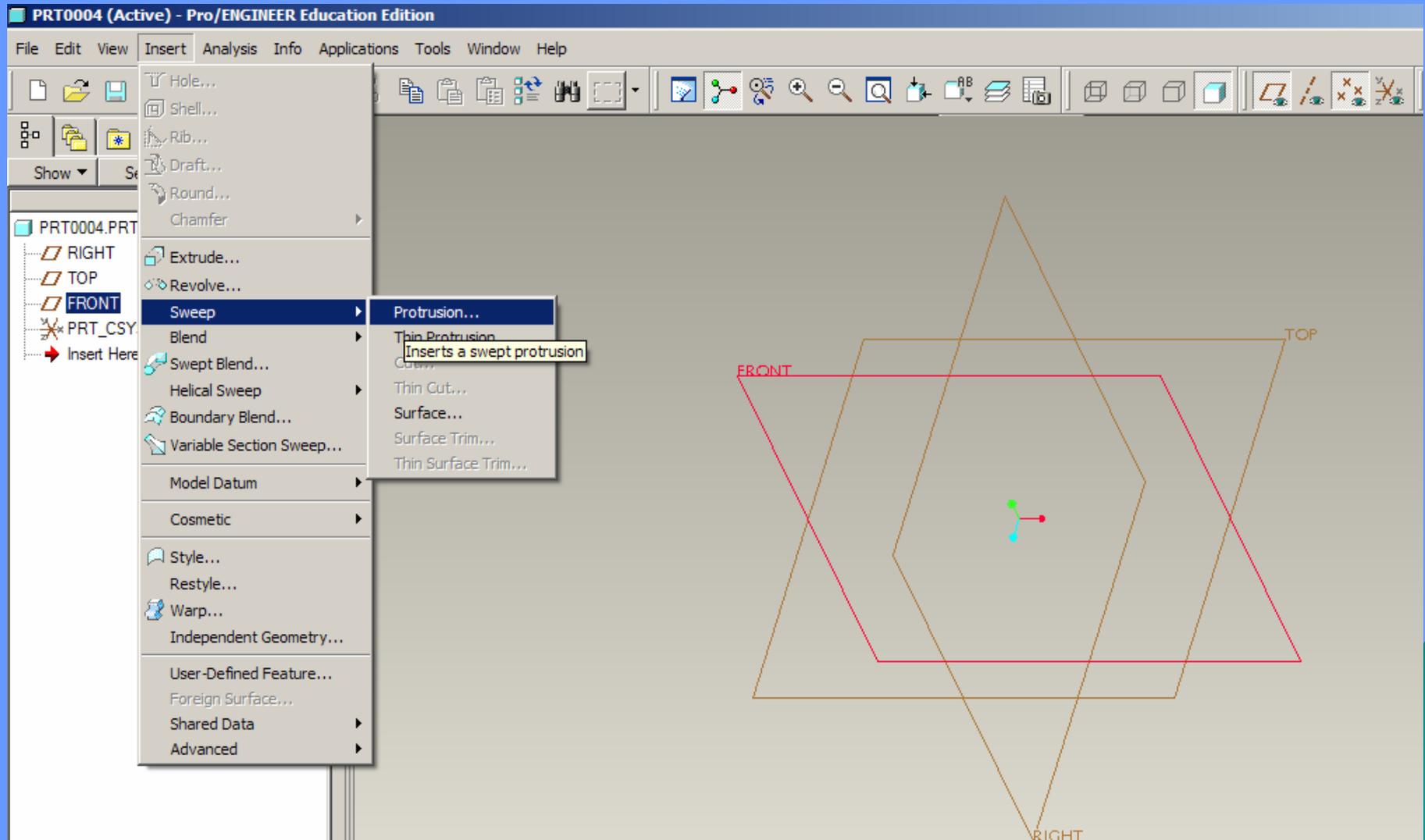
Each section
needs to have the
same number of
entities.



Sweeps



Step 1



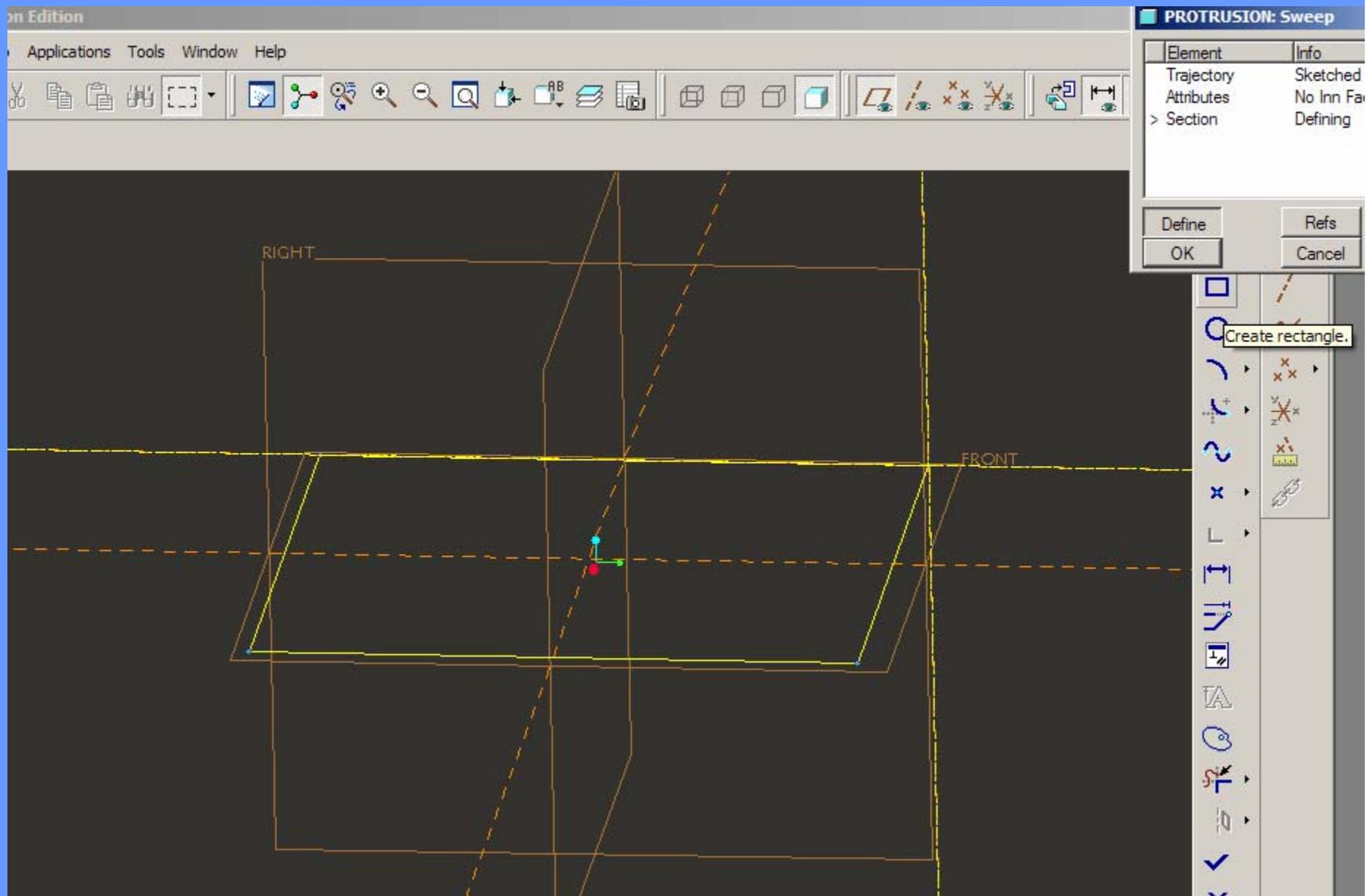
Step II.. Sketch trajectory

The image shows a CAD software interface with a sketch trajectory and a 'PROTRUSION: Sweep' dialog box. The sketch is a yellow closed curve on a dark background, centered around a coordinate system labeled 'PRT_CSYS_DEF'. The trajectory consists of a horizontal top edge, a horizontal bottom edge, and two vertical side edges, all with rounded corners. Dimensions are shown: 75.00 for the corner radii, 250.00 for the horizontal segments, and 500.00 for the vertical segments. A red vertical line is drawn on the right side of the sketch. The 'PROTRUSION: Sweep' dialog box is open in the top right corner, showing a table with the following content:

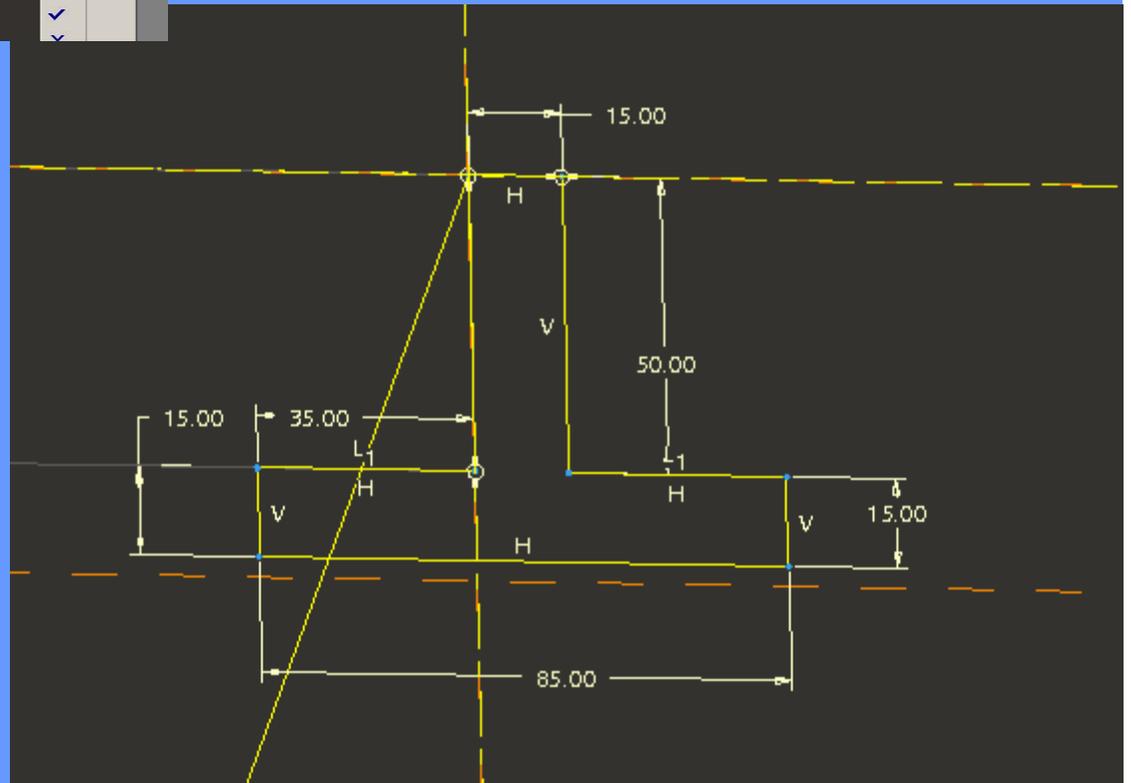
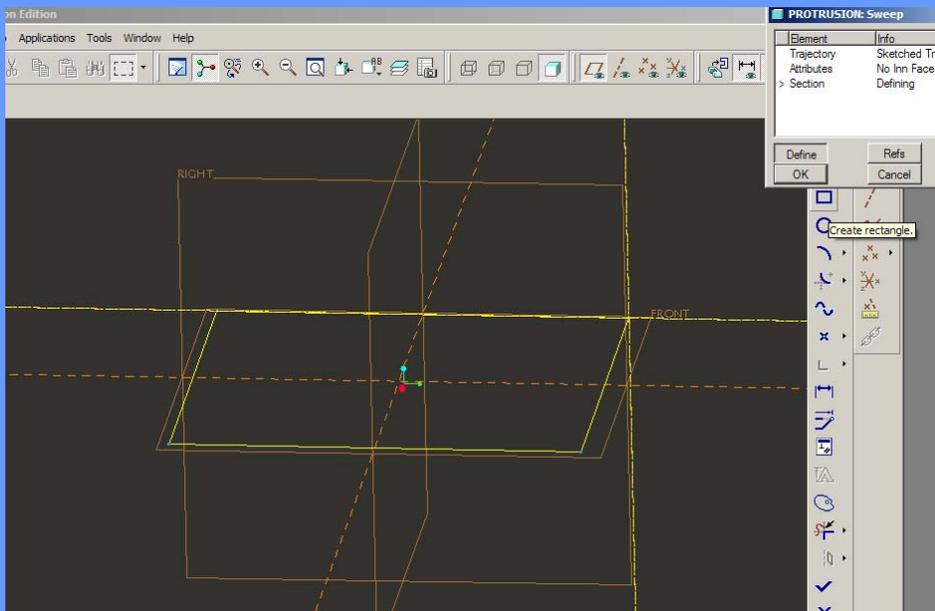
Element	Info
> Trajectory	Defining
Section	Required

Buttons for 'Define', 'OK', 'Refs', and 'Cancel' are visible. Below the dialog box is a vertical toolbar with various icons for sketching and editing.

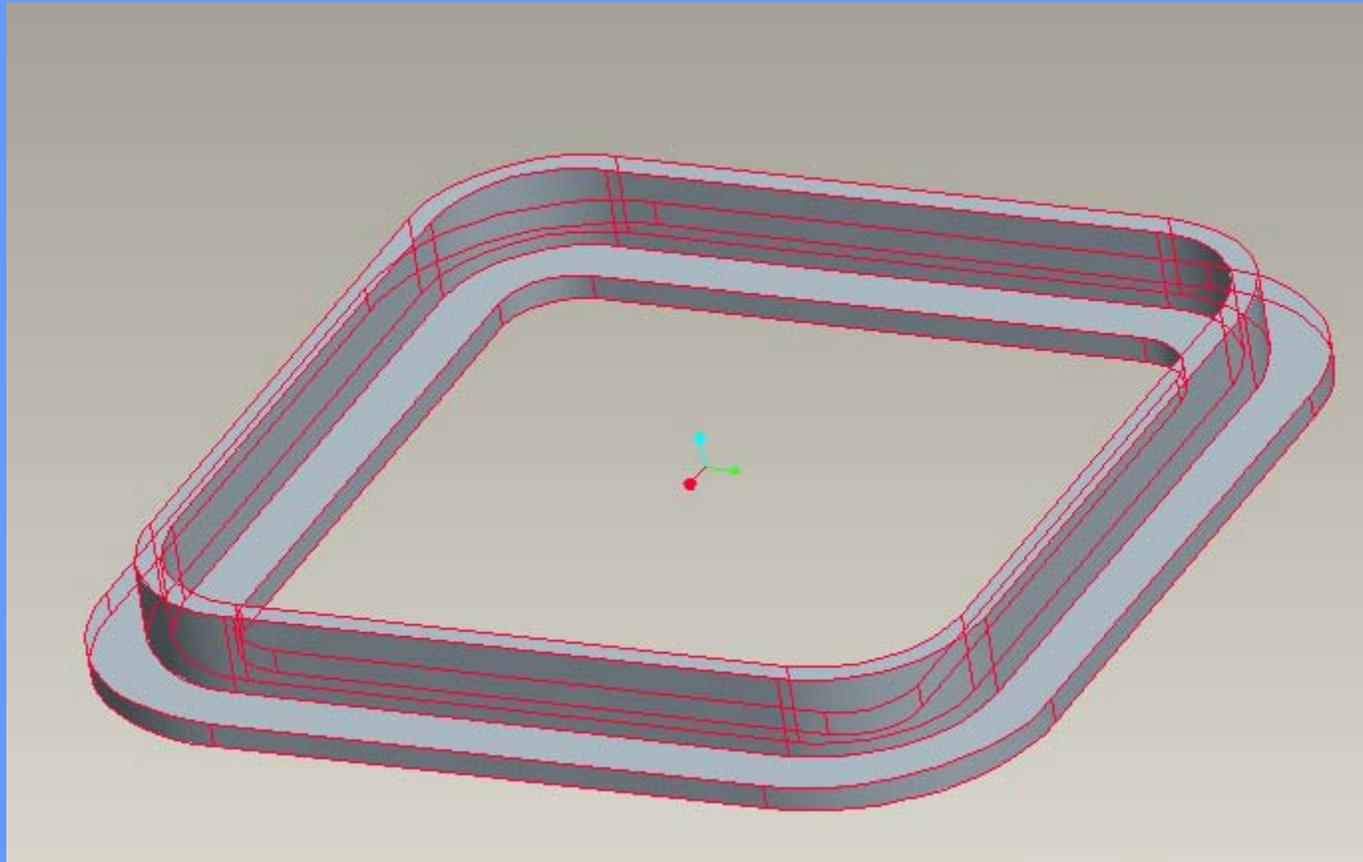
Step III



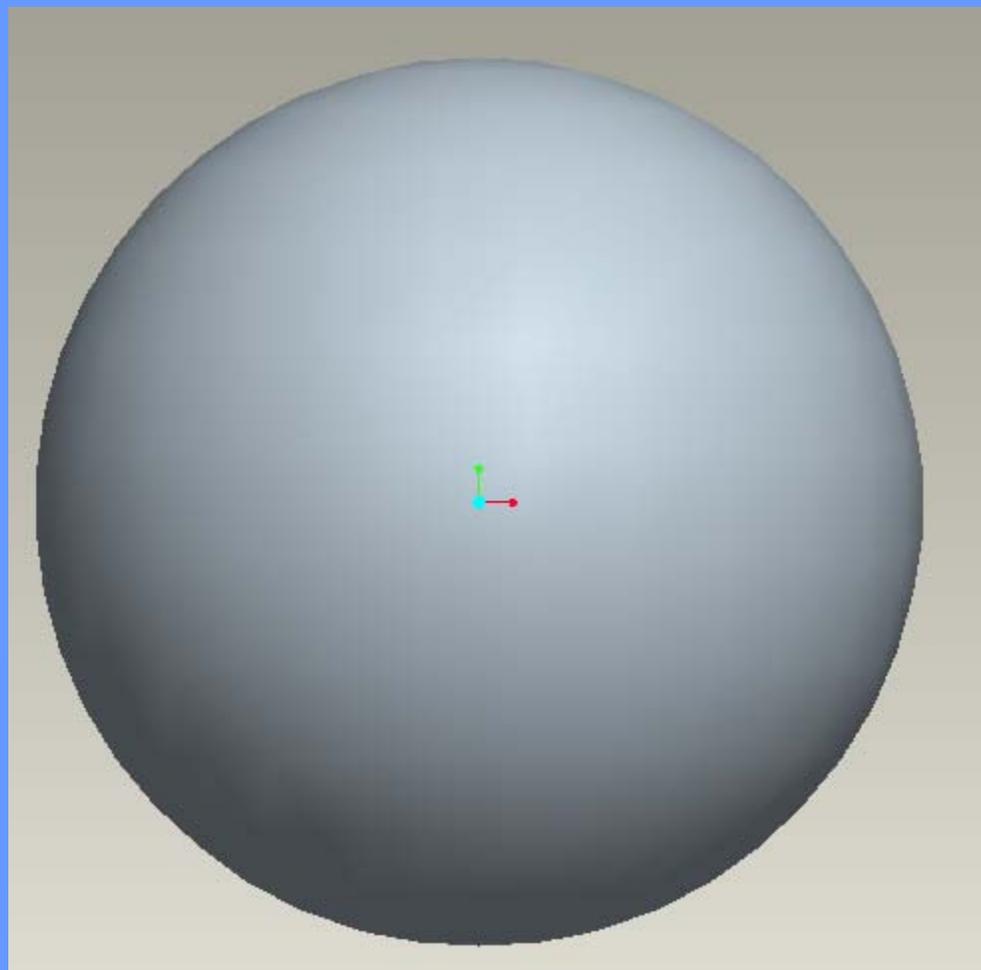
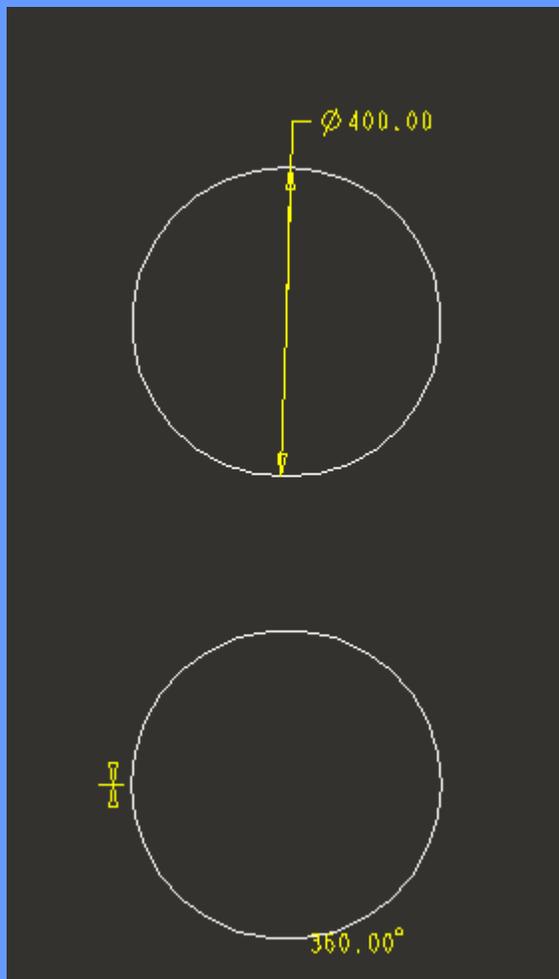
Step IV



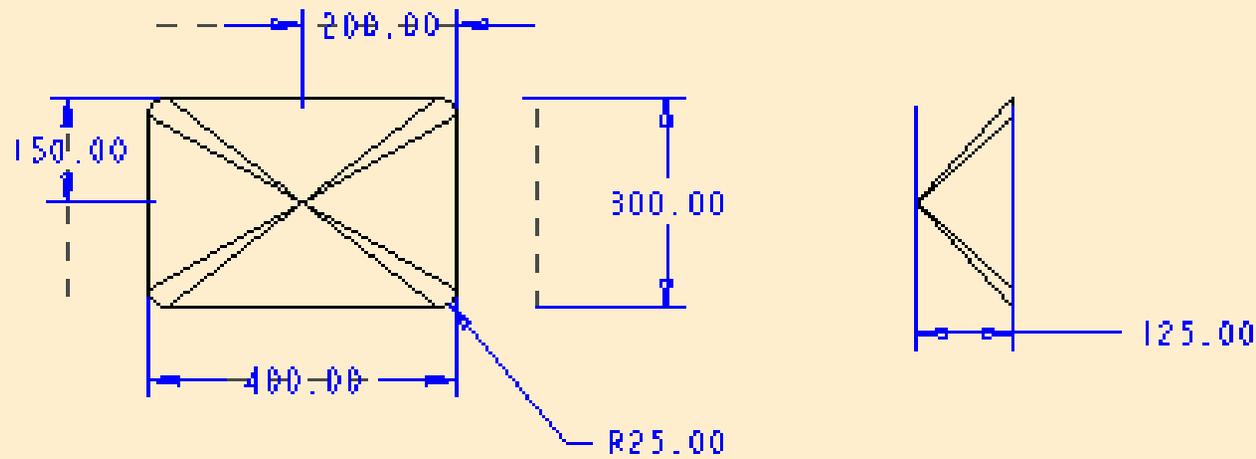
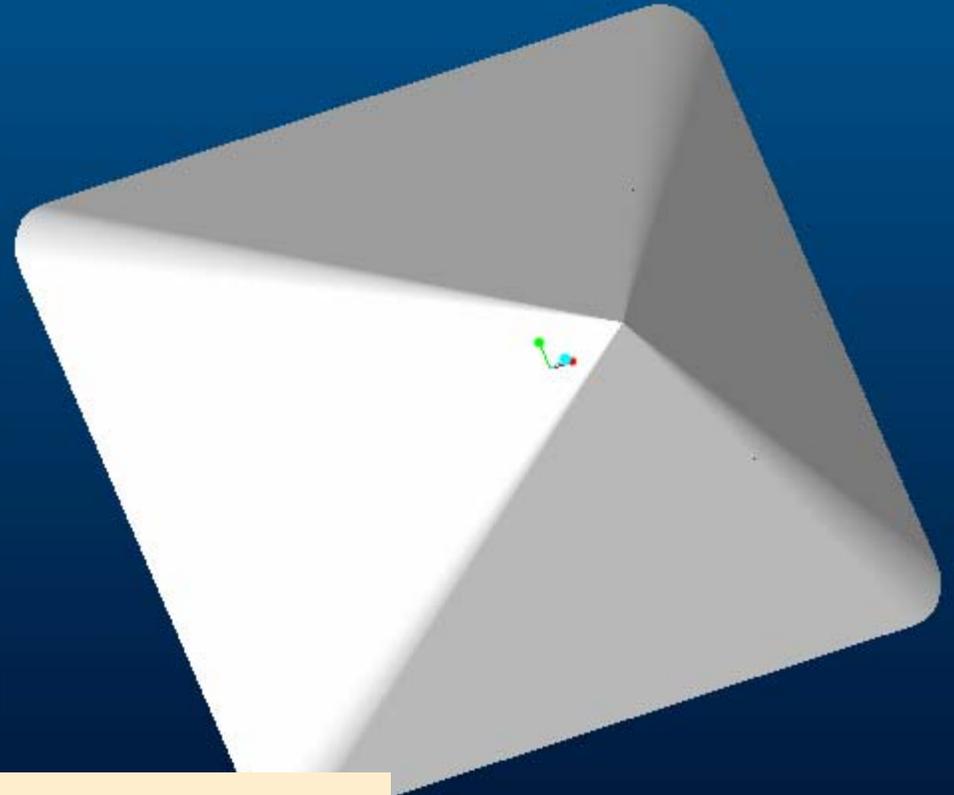
Step V



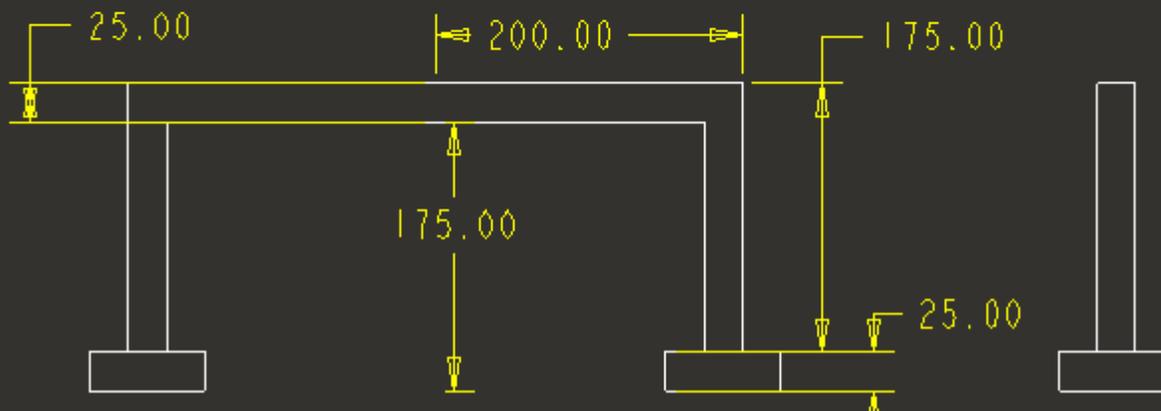
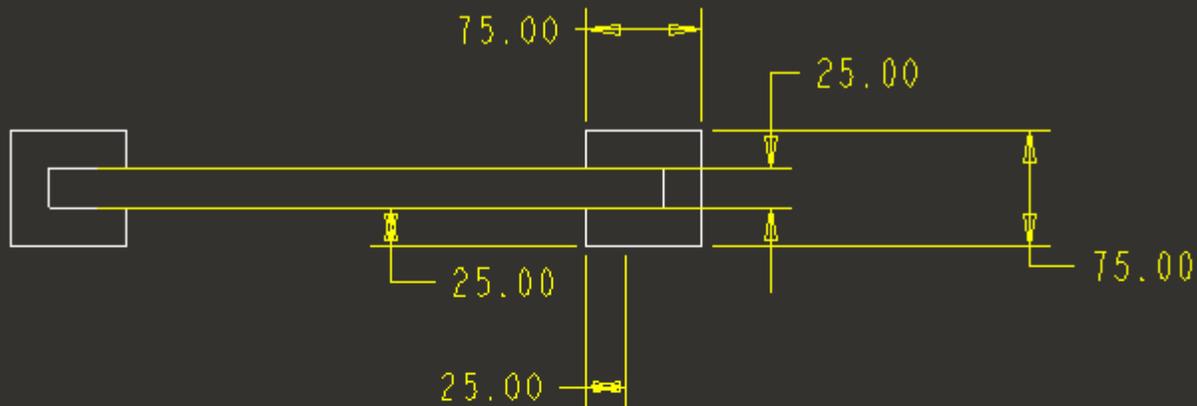
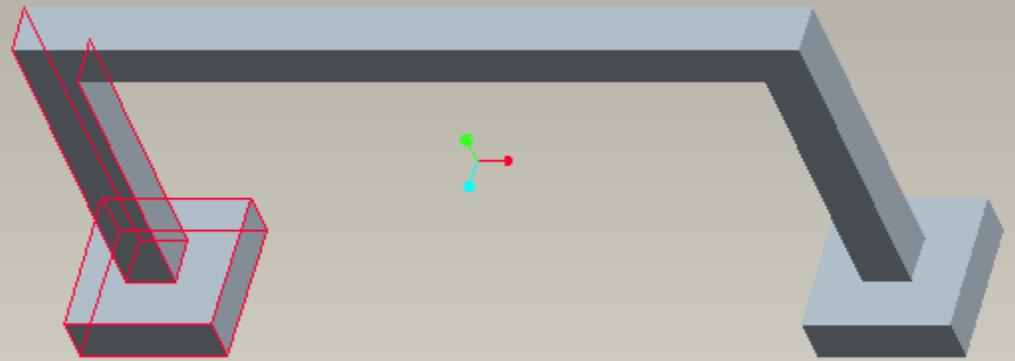
Exercise 1



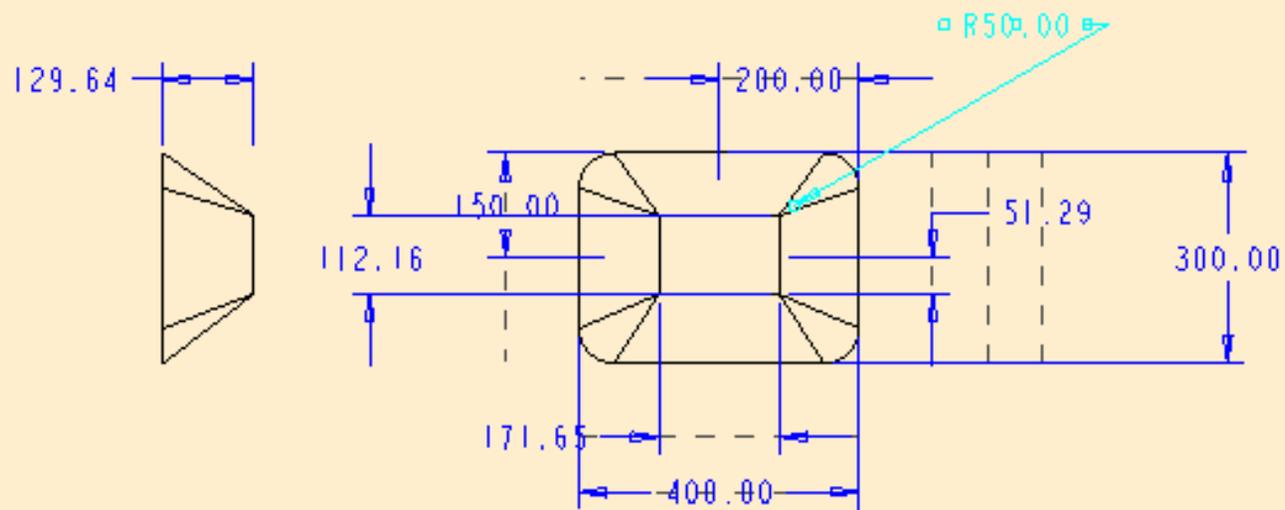
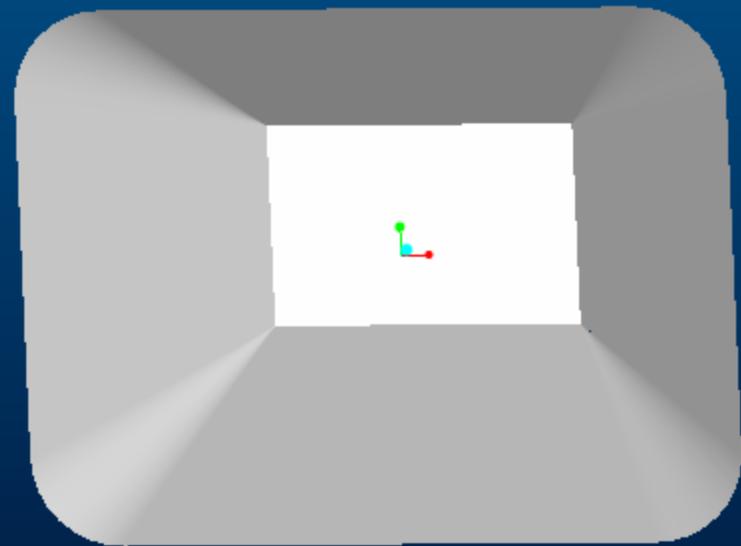
Exercise 2



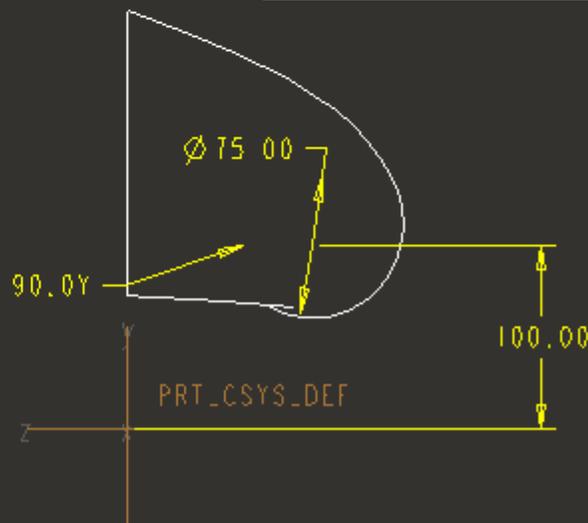
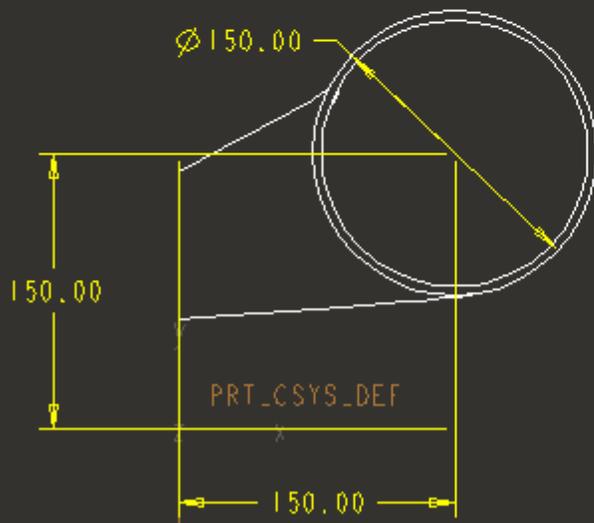
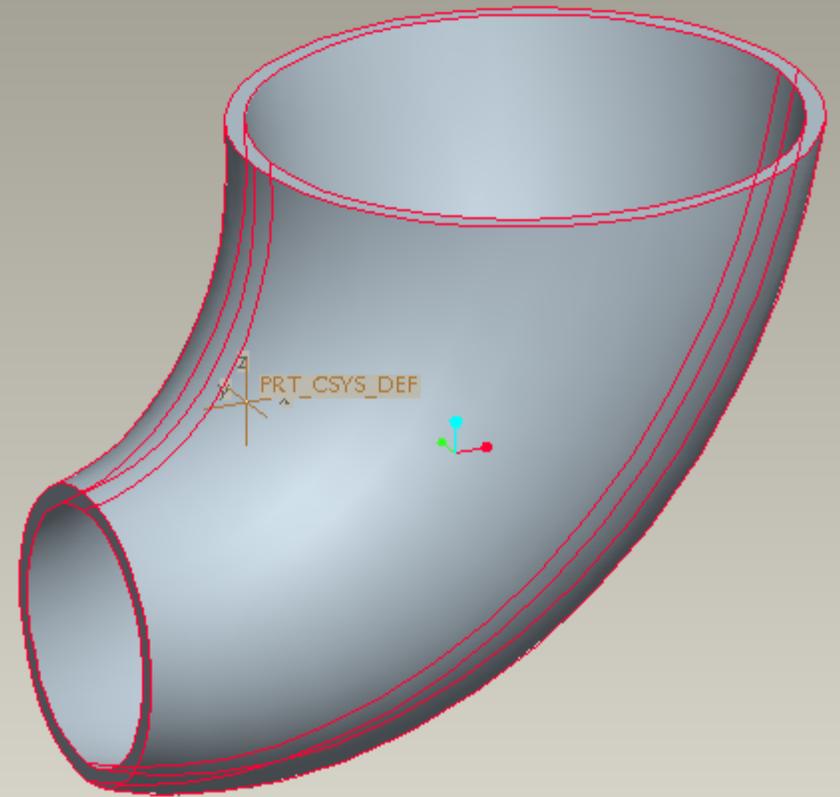
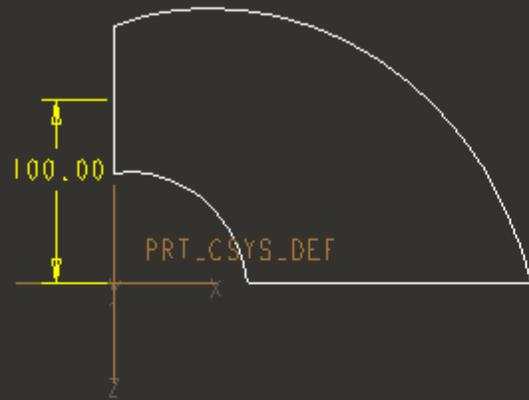
Exercise 3



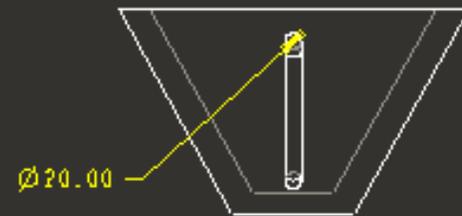
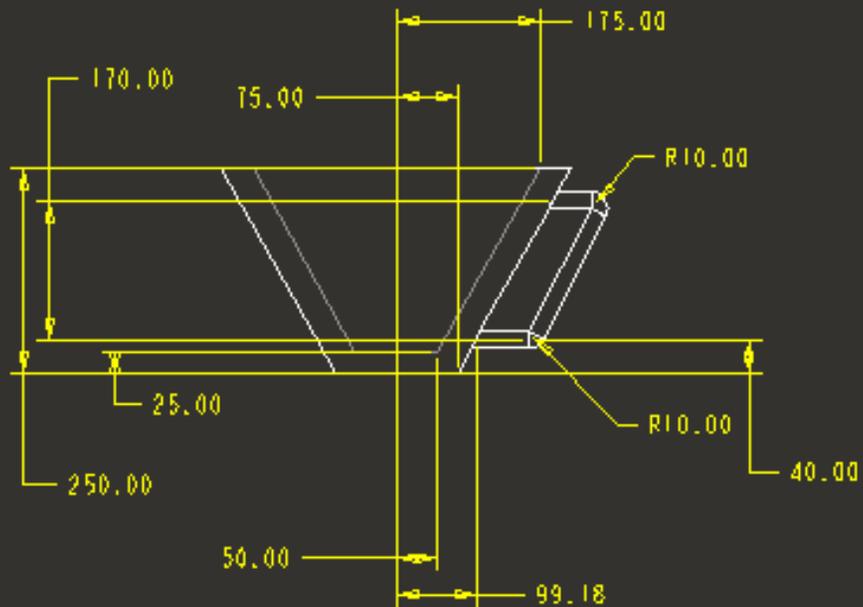
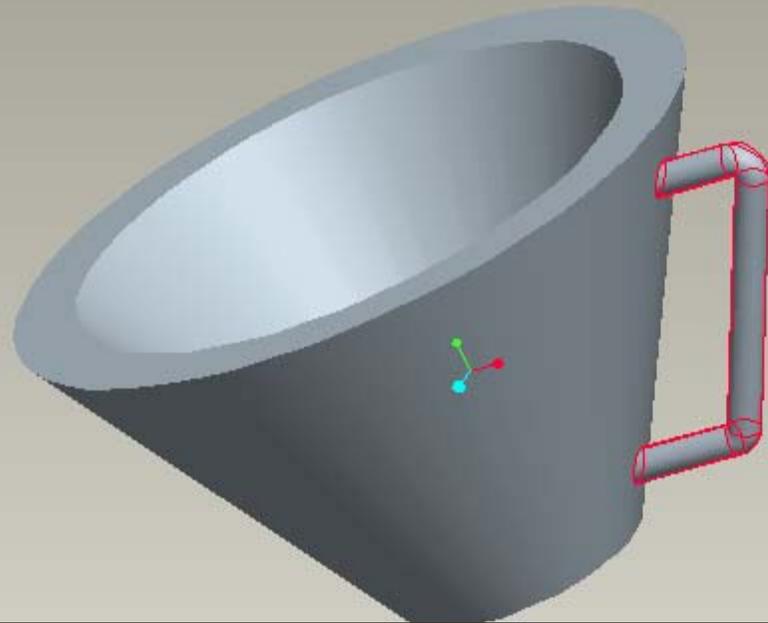
Exercise 4

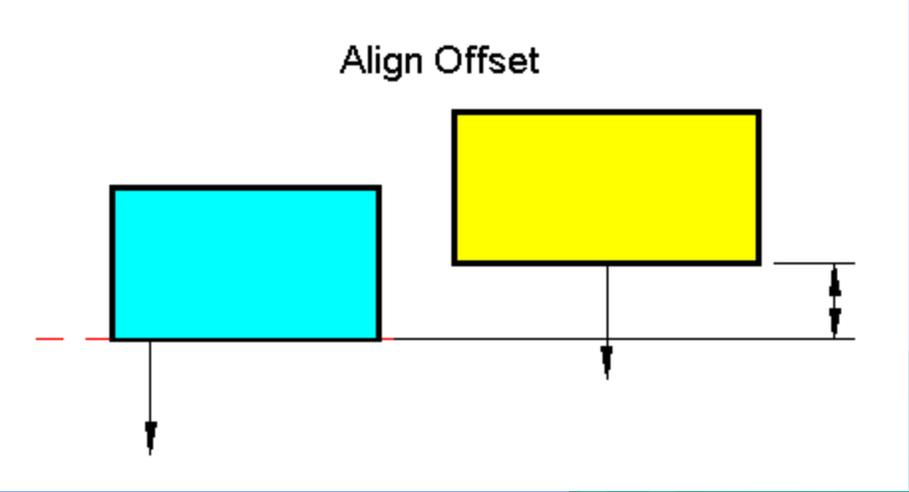
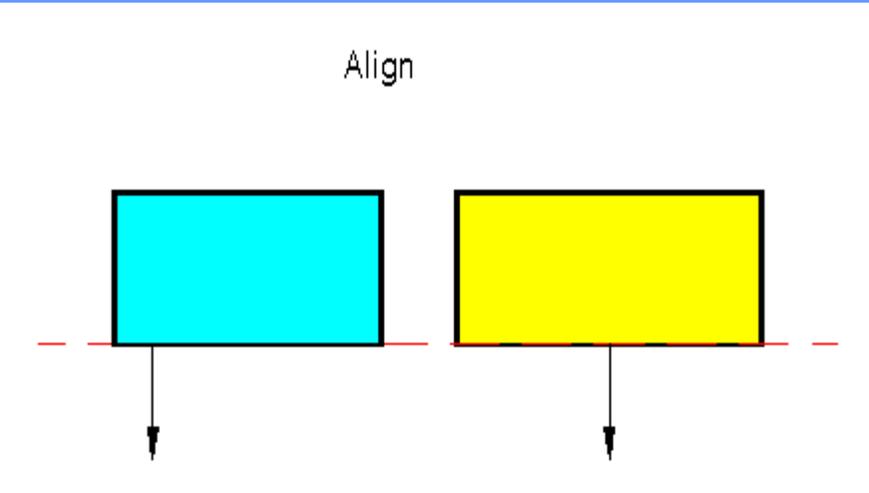
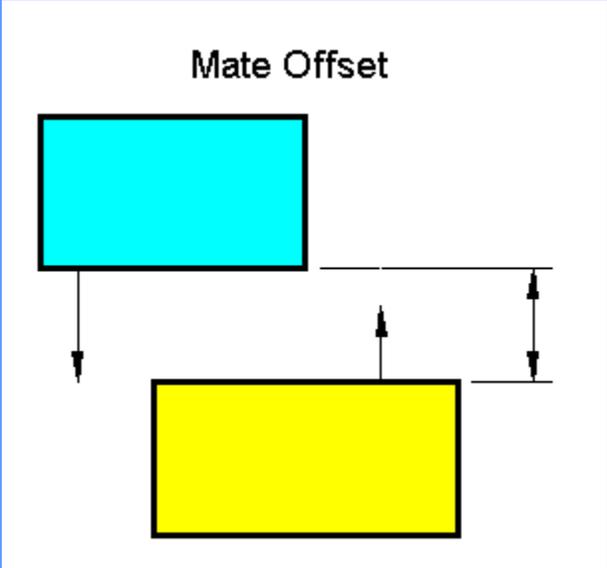
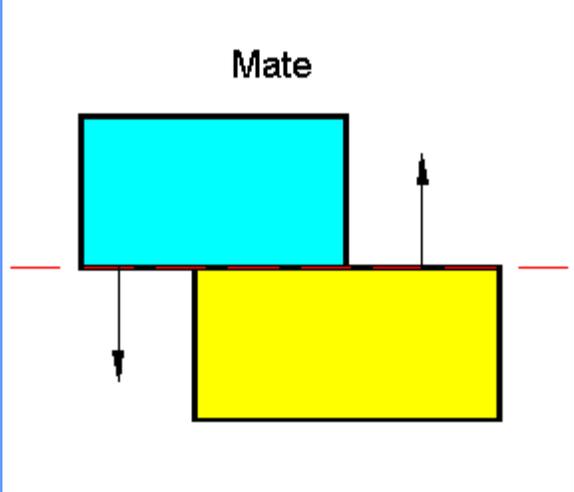


Exercise 5

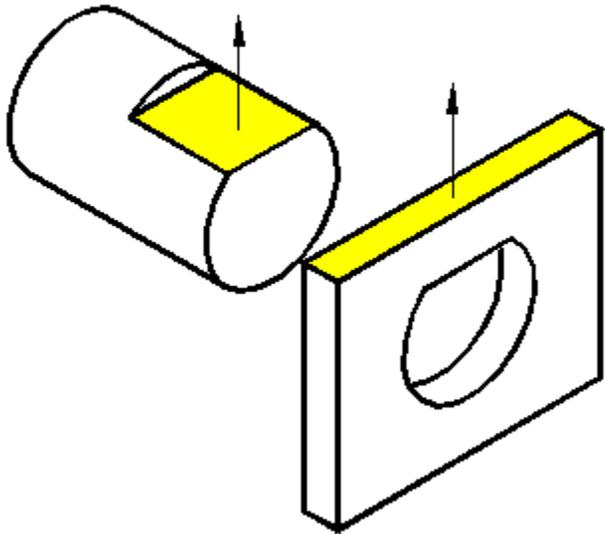


Exercise 6

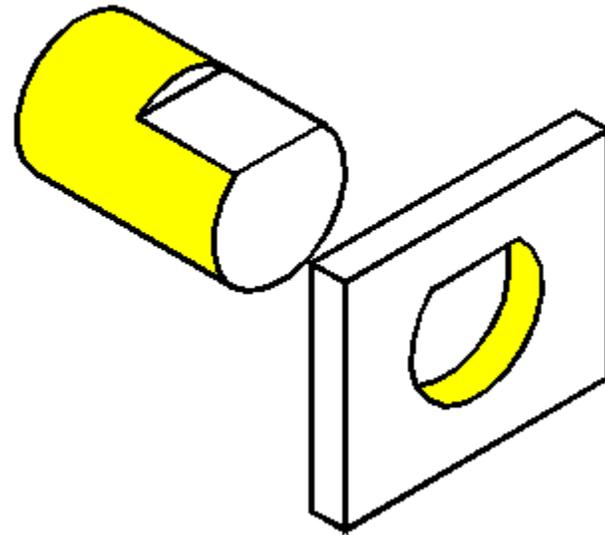




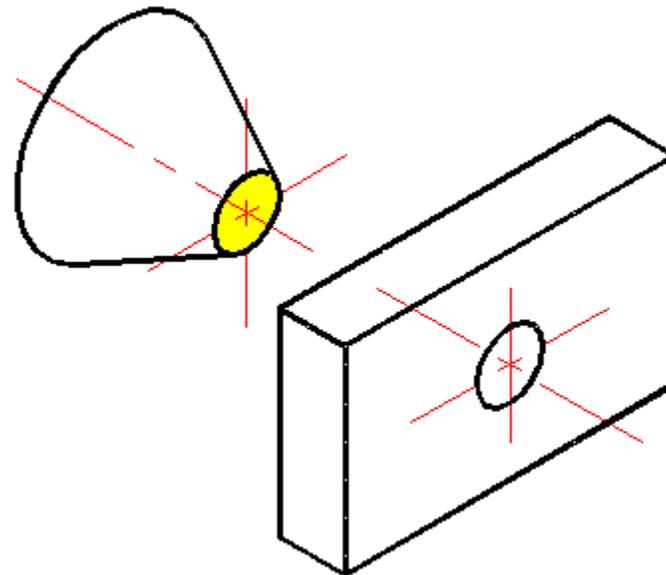
Orient

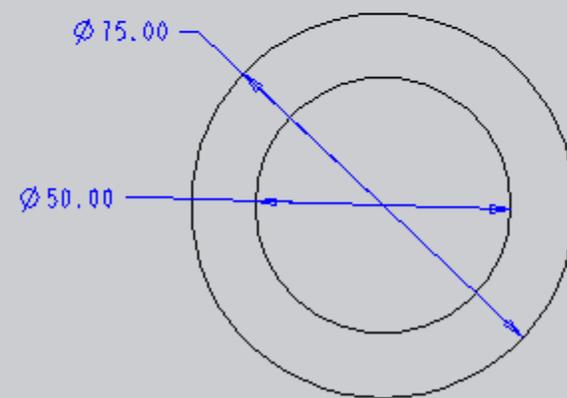
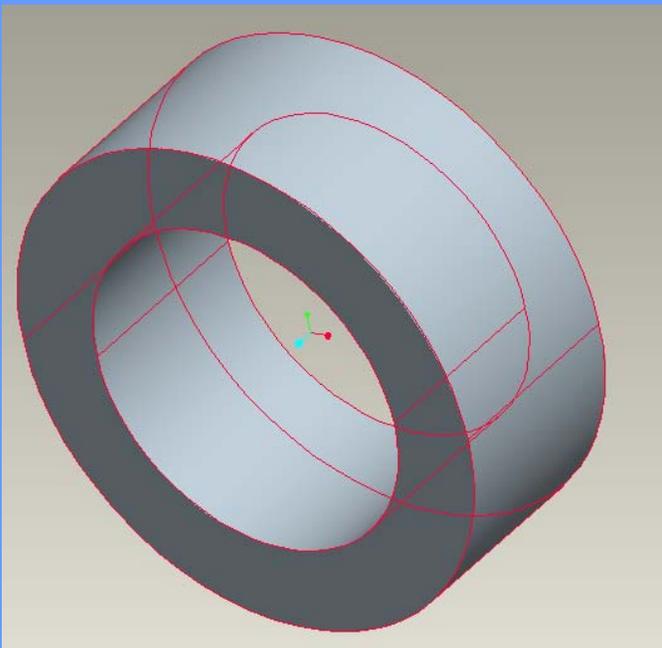
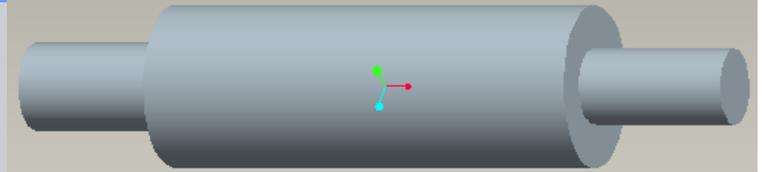
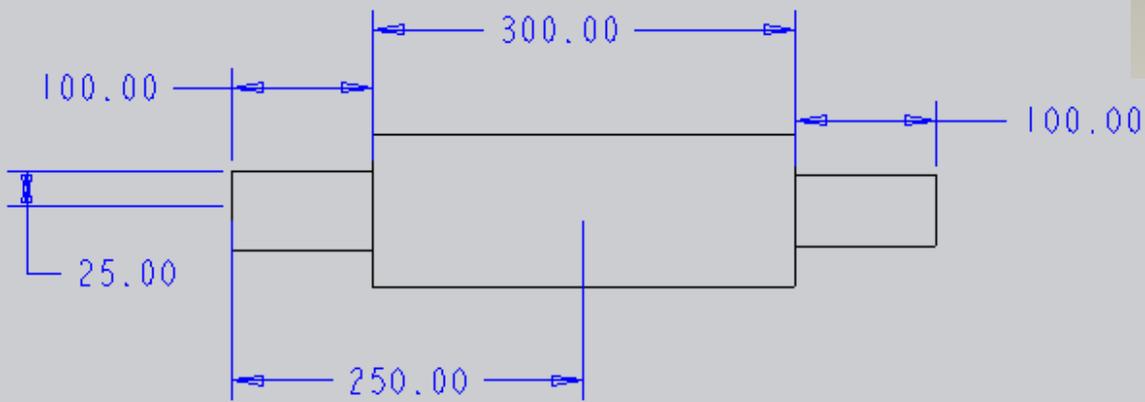


Insert



Align

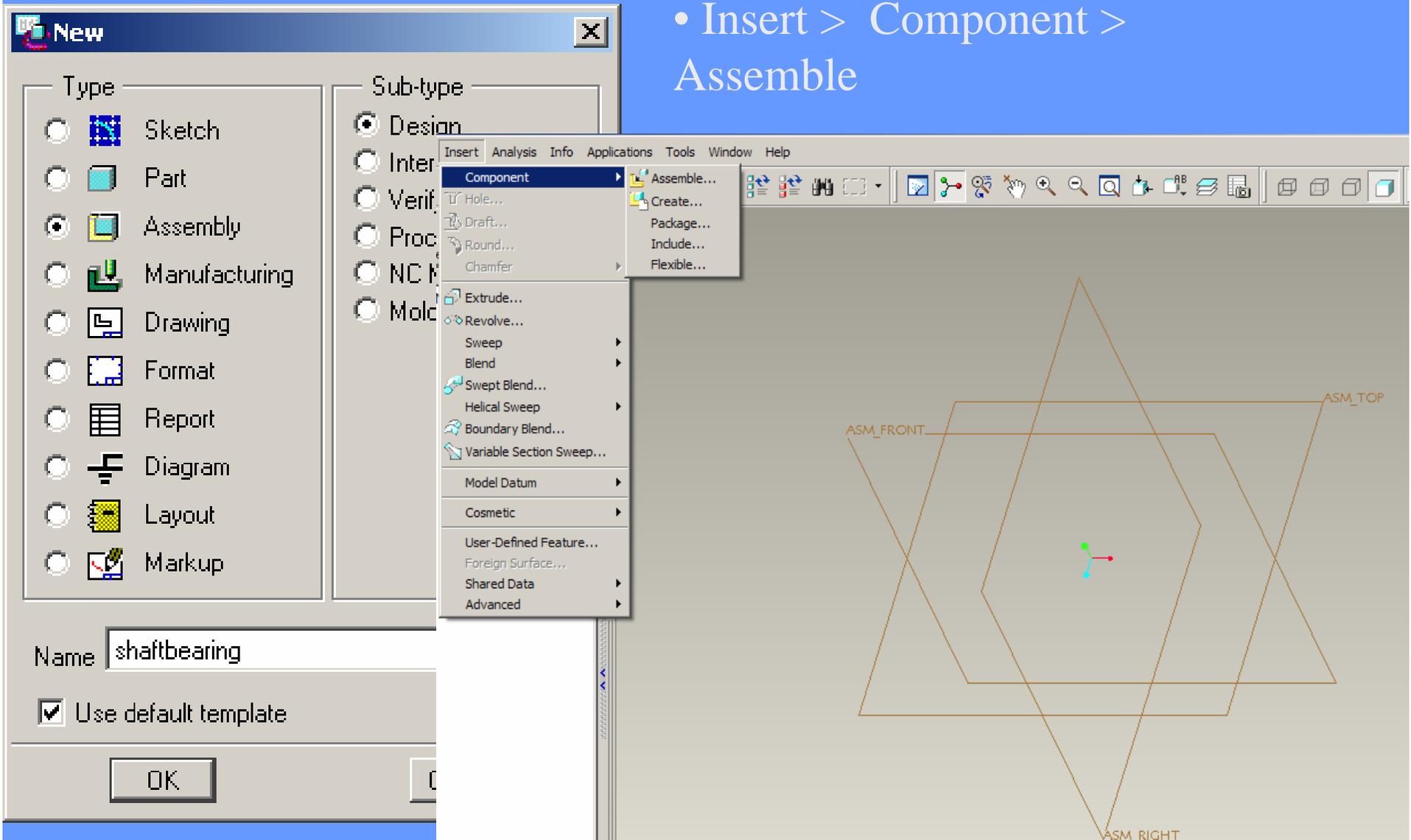


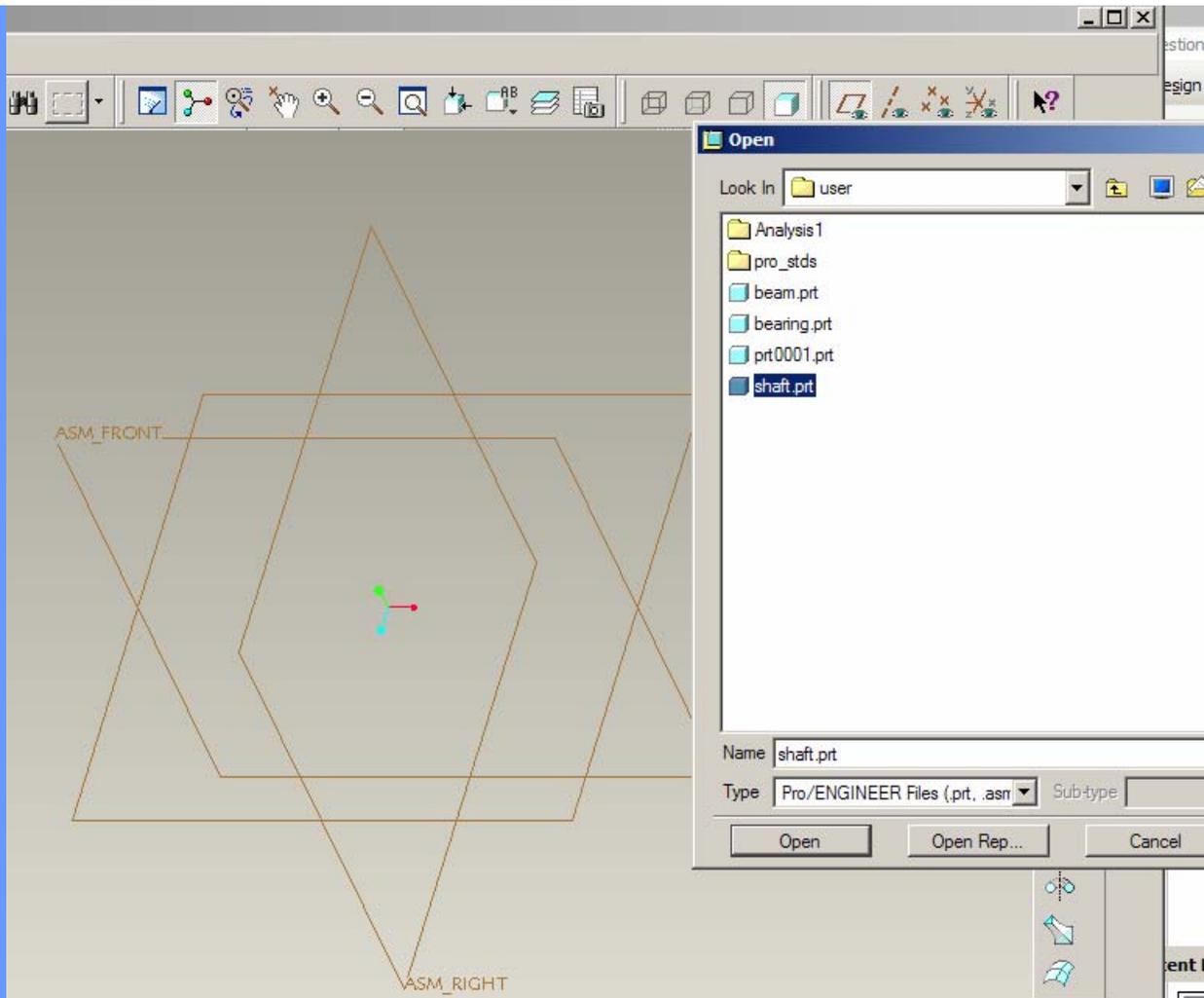


Placing Components

Datum plane labels are preceded by an “ASM_”.

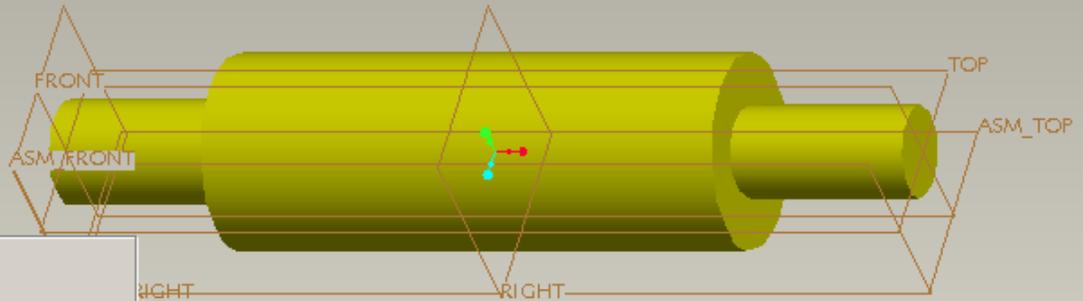
- Insert > Component > Assemble





- ◆ In general, three constraints are needed to position one part with respect to the assembly datum.
- ◆ Full associativity reflects any changes made on a part in all assemblies where that part is used.

SHAFTBEARING.ASM



Set1 (User Defined)

- Automatic

Select component item

Select assembly item

New Constraint

New Set

Constraint Enabled

Constraint Type: Automatic

Offset: Coincident 0.00

Status: No Constraints

Placement Move Flexibility Properties

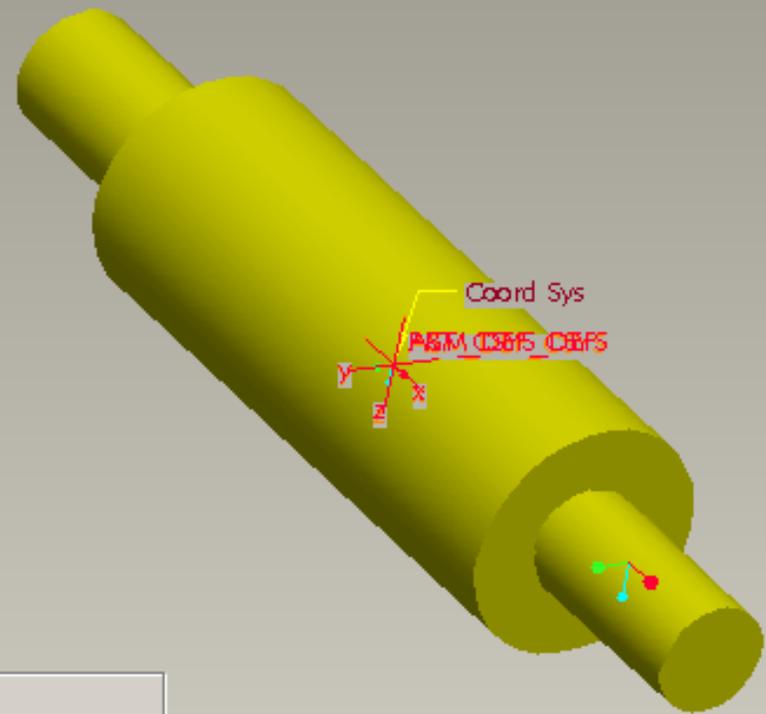
User Defined Automatic 0.00 STATUS : No Constraints

Select any reference for auto type constraining.



Show ▾ Settings ▾

SHAFTBEARING.ASM



Set1 (User Defined)

- Coord Sys
- PRT_CSYS_DEF:F4(CS)
- ASM_DEF_CSYS:F4(CS)

New Constraint

New Set

Constraint Enabled

Constraint Type

Coord Sys ▾

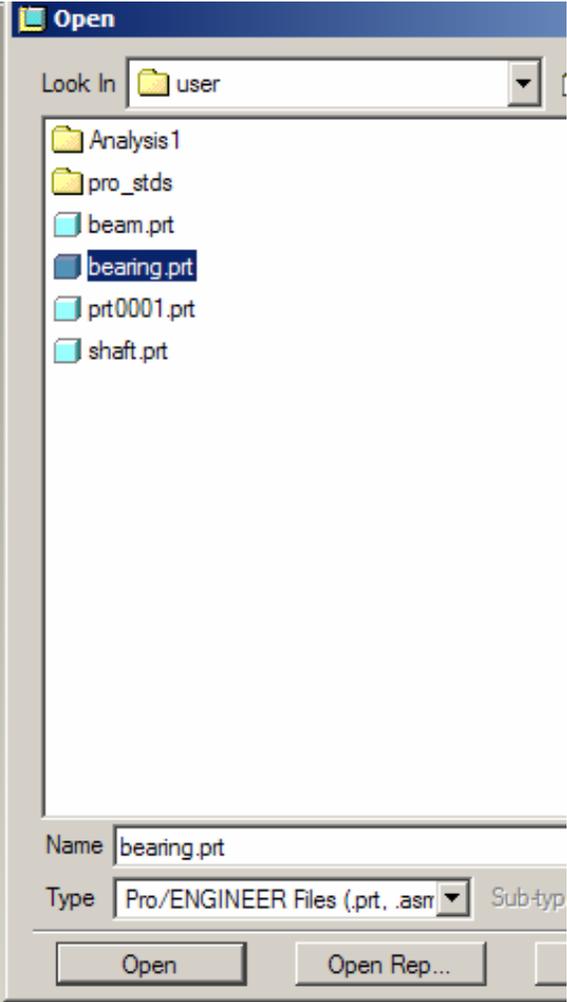
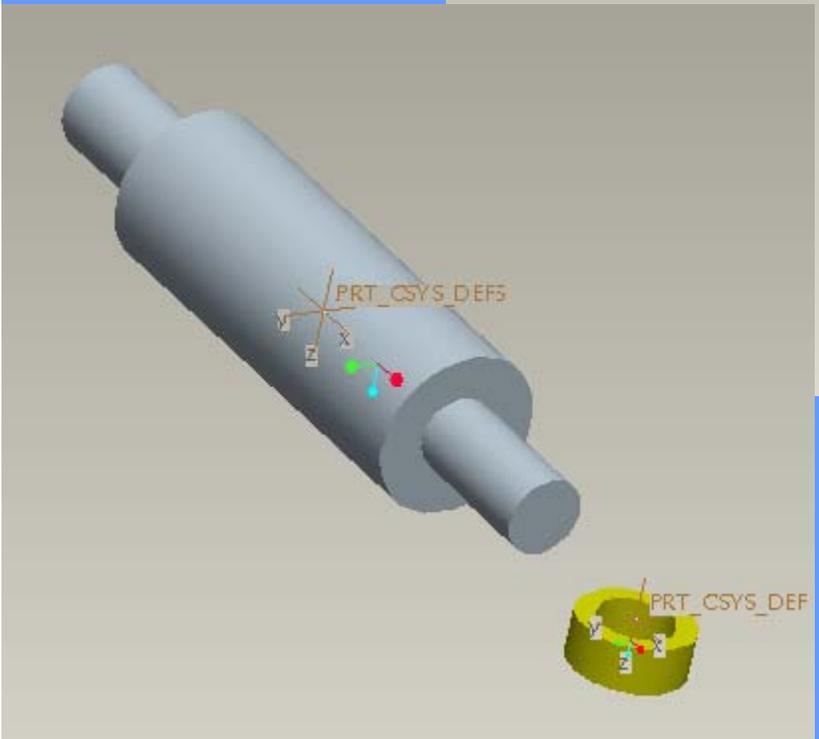
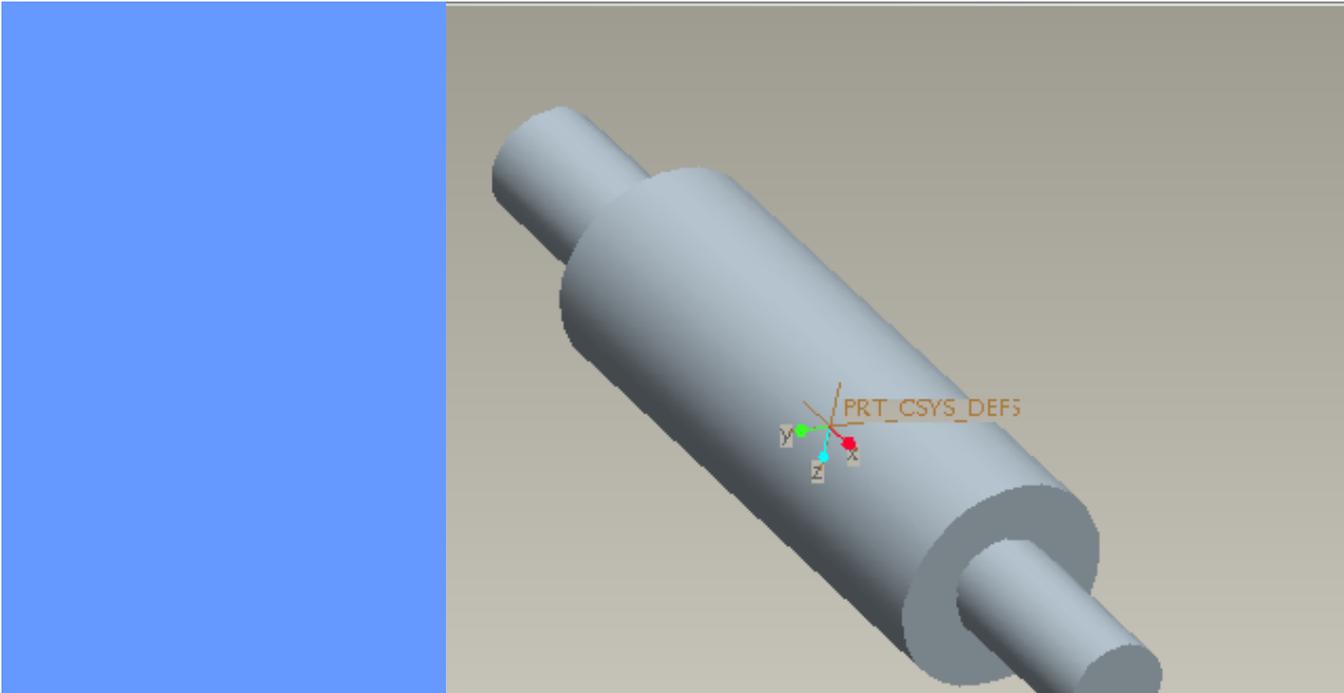
Offset

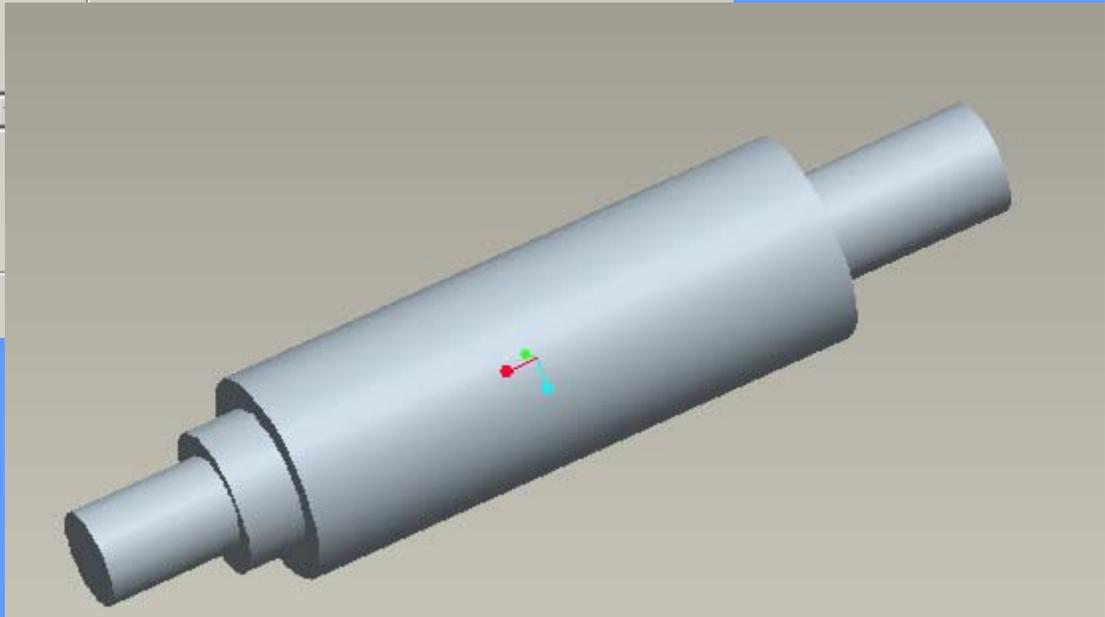
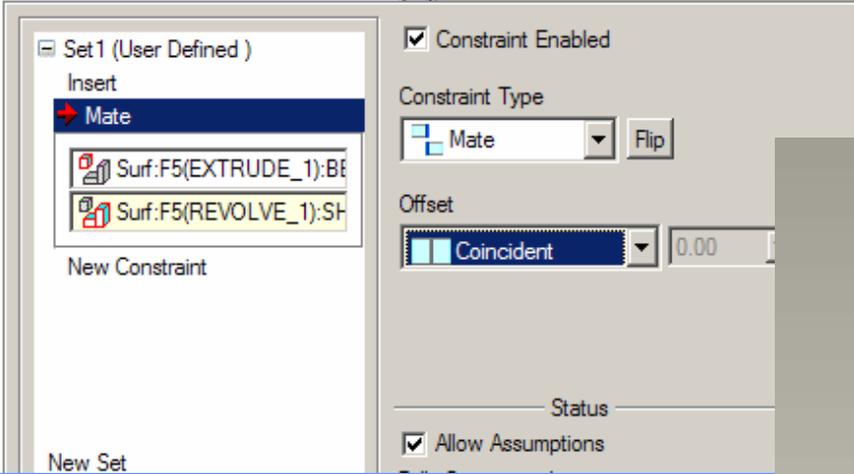
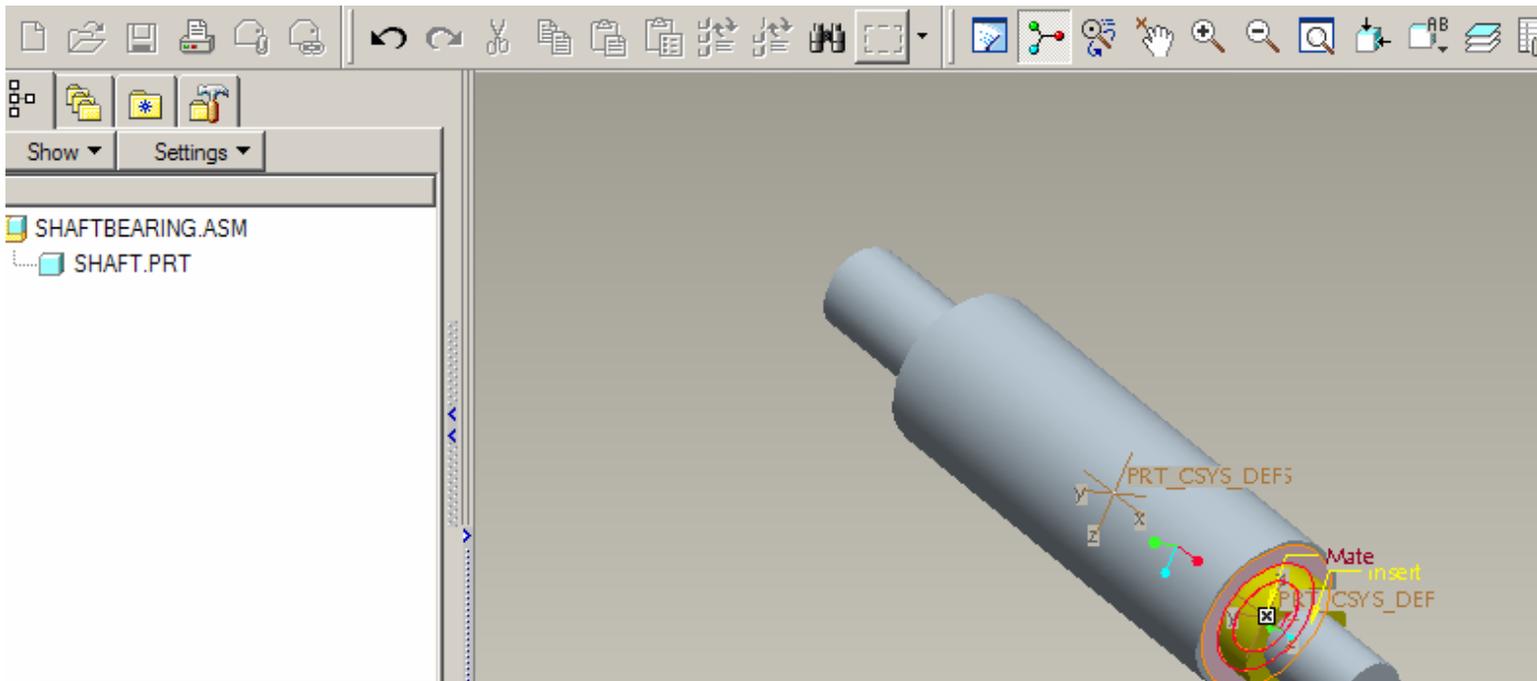
0.00

Status

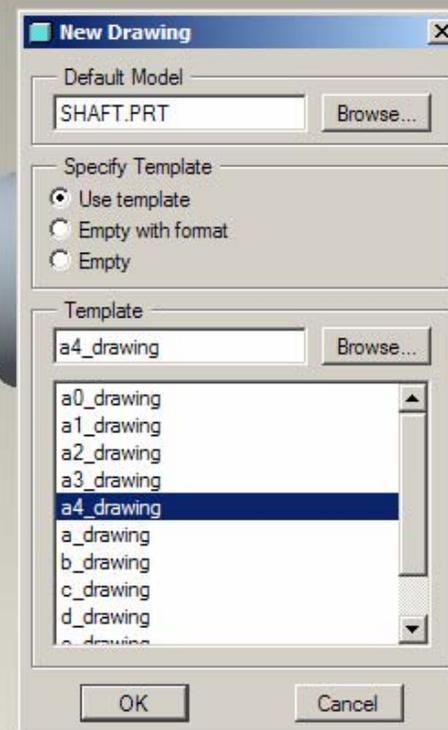
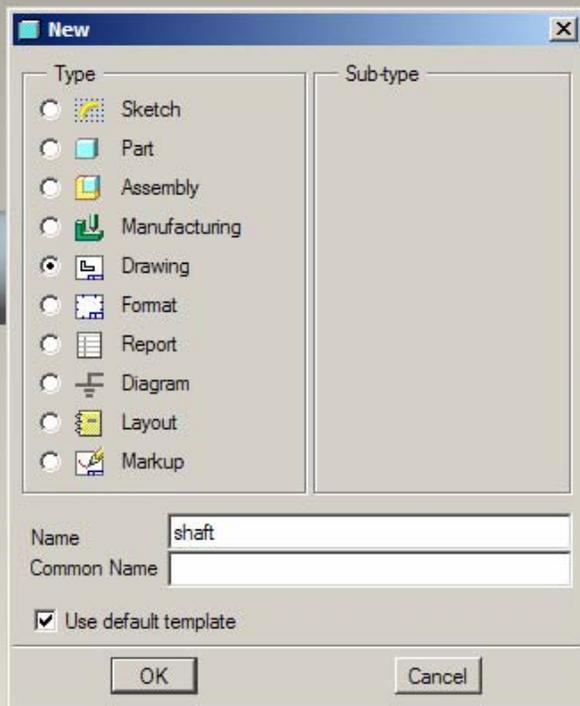
Fully Constrained

Placement Move Flexibility Properties

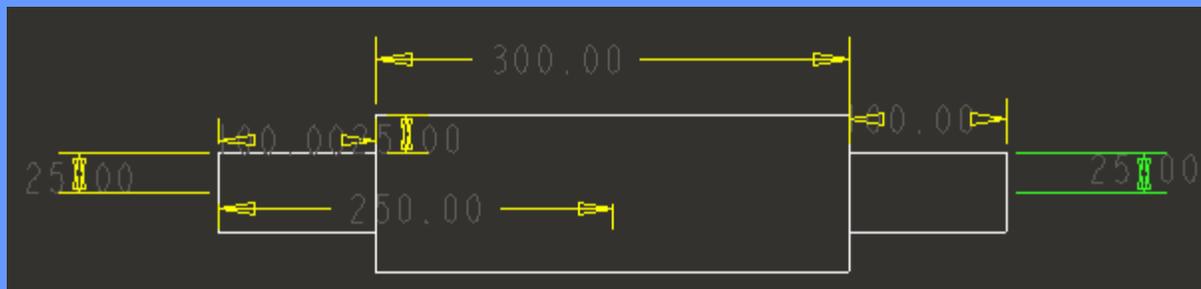
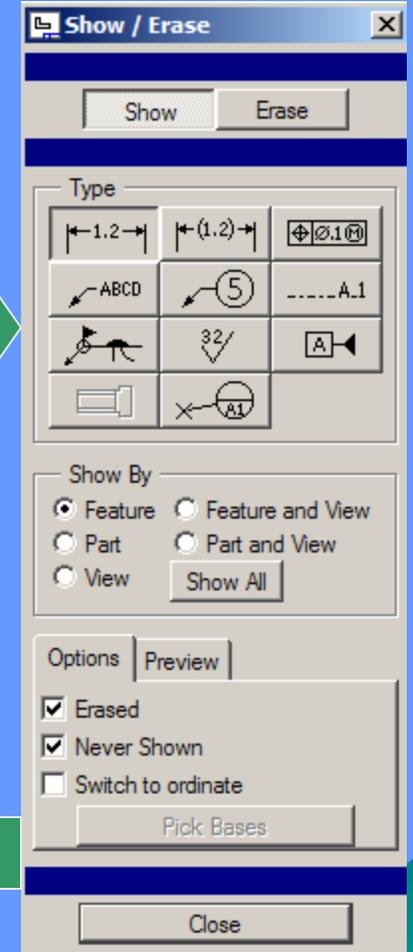
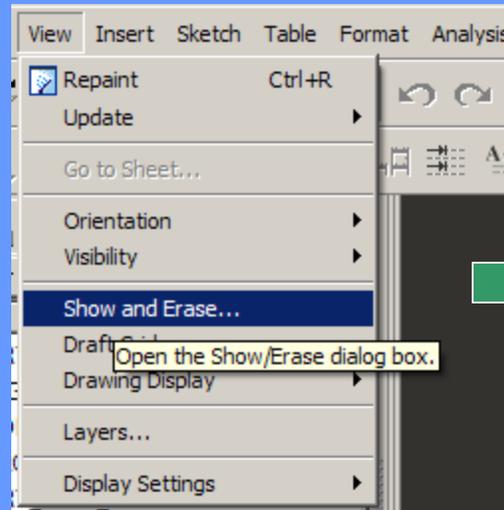
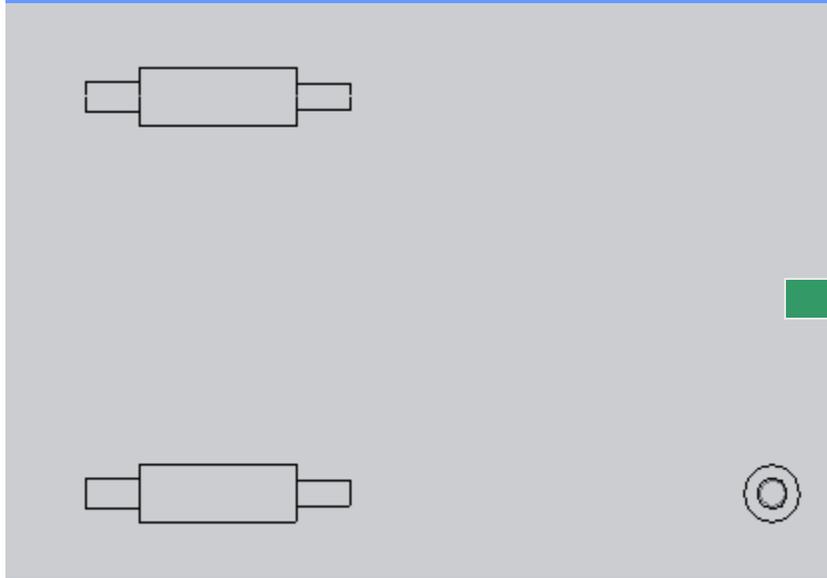




Drawing..



Default Views



File Edit

Regenerate

Undo Ctrl+Z
Redo Ctrl+Y

Cut Ctrl+X
Copy Ctrl+C
Paste Ctrl+V
Paste Special...
Repeat Last Format

Trim
Transform

Fill
Suppress
Delete
Remove
Move Sheet...

Attachment...
Move Item to Sheet...
Move Item to View...
Move Special...
Align Dimensions

Cleanup
Toggle Ordinate/Linear

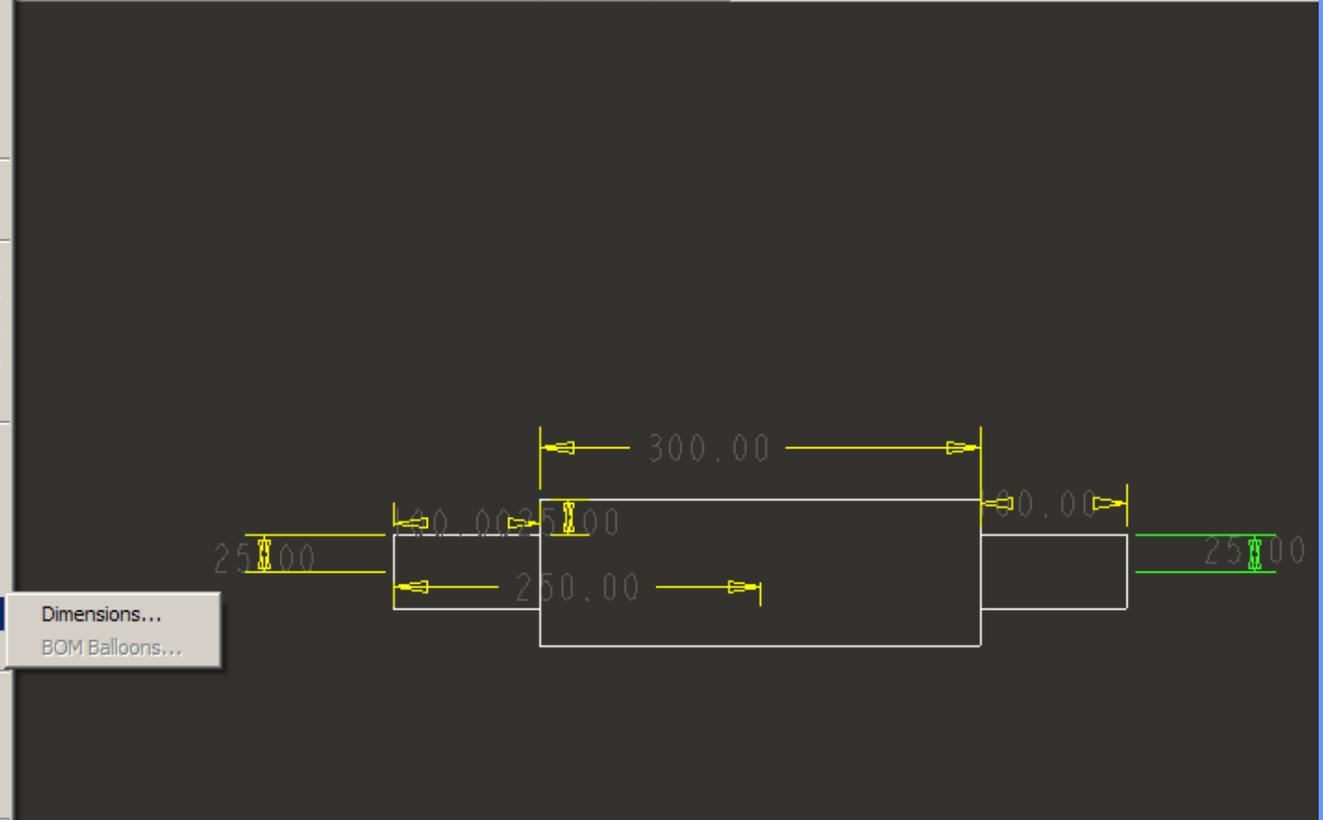
Convert to Draft Entities

Group
Value
Properties

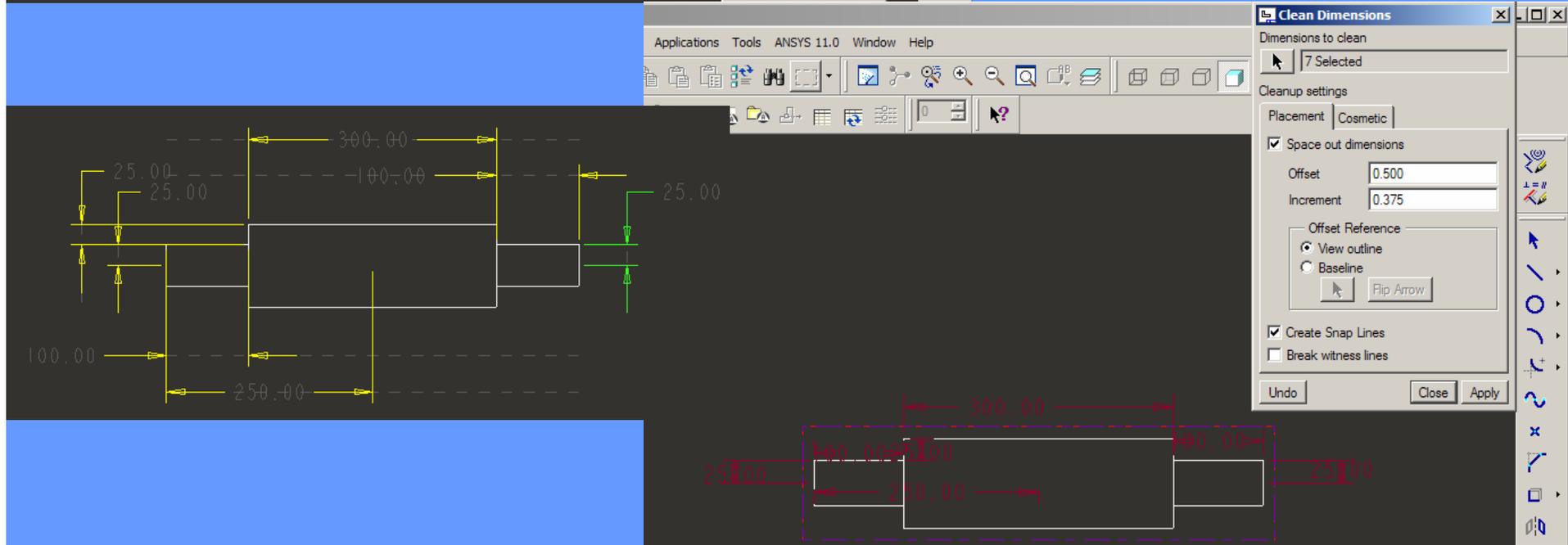
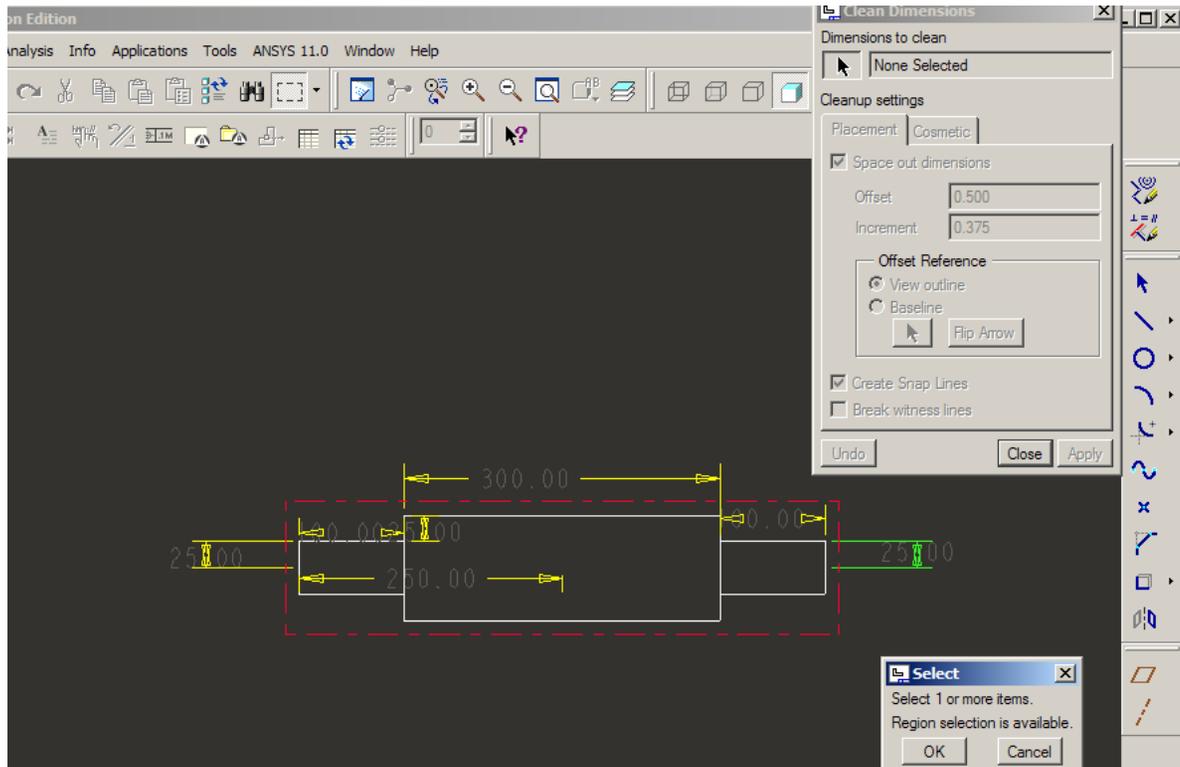
Analysis Info Applications Tools ANSYS 11.0 Window Help

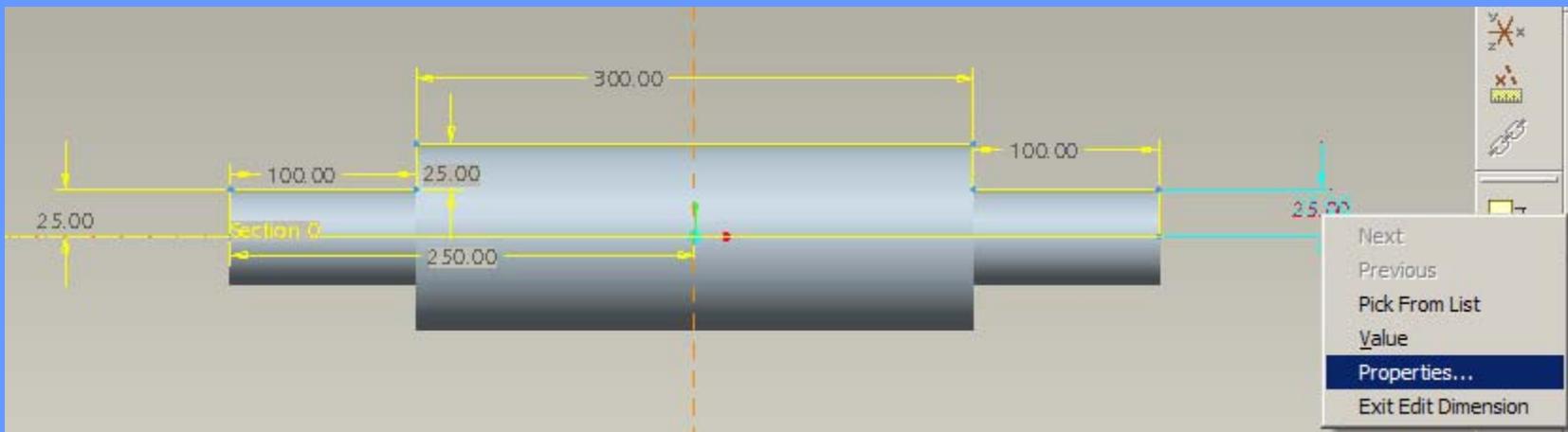
SHAFT-2008

- RIG
- TO
- FR
- PR
- Rev



Dimensions...
BOM Balloons...





Dimension Properties

Properties | Dimension Text | Text Style

Value and tolerance

Tolerance mode: (As Is)

Tolerance table: General

Nominal value: 25.00

Table name: [] []

Format

Decimal
 Fractional

Number of decimal places: 2

Angular dimension units: Degrees

Display

Basic
 Inspection
 Neither

Show as linear dimension

Text orientation: (As Is)

Chamfer style: (As Is)

Flip Arrows

Dual dimension

Position: Below
 To right

Dual decimal places: []

Witnessline display

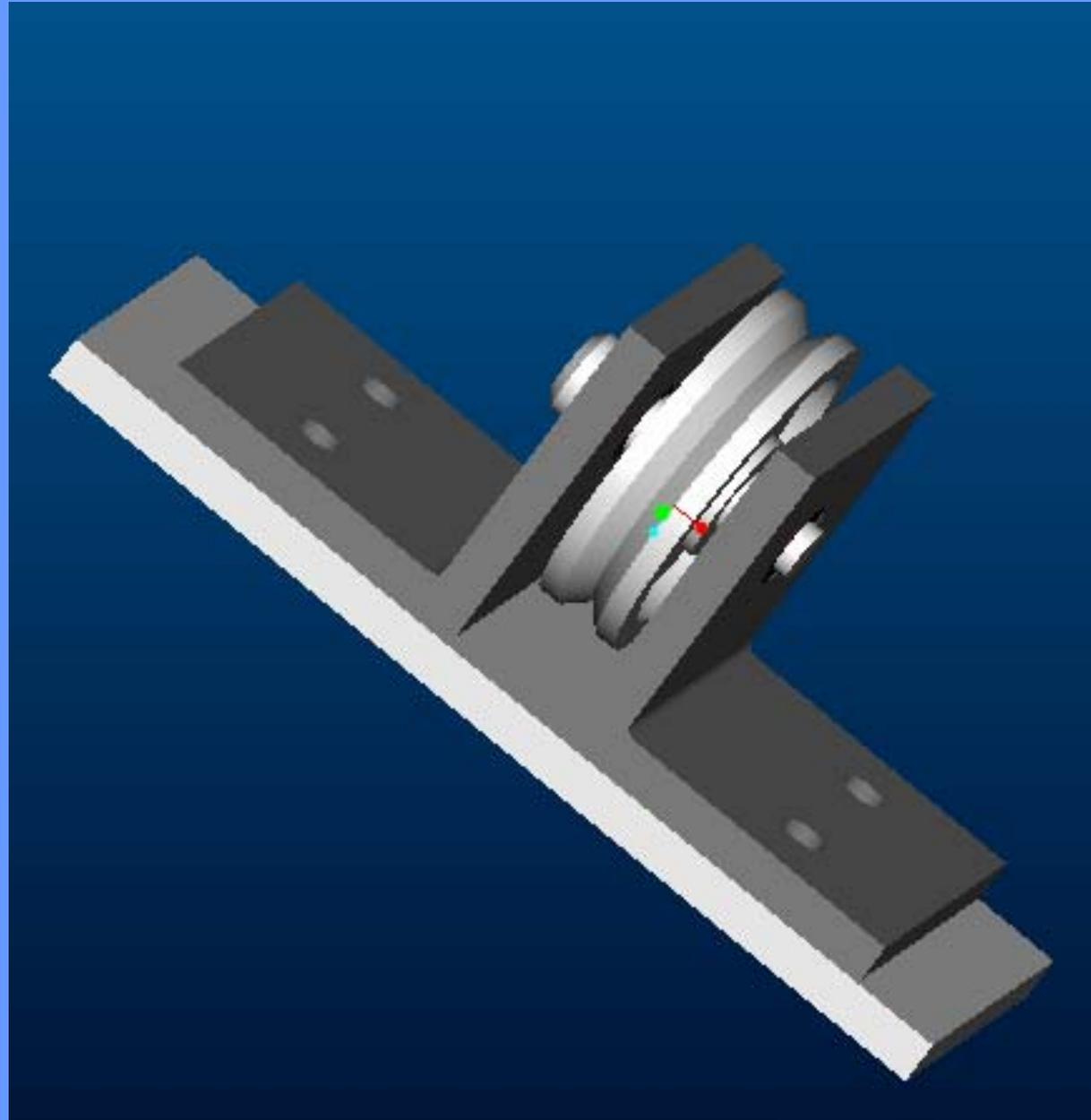
Show Erase... Default

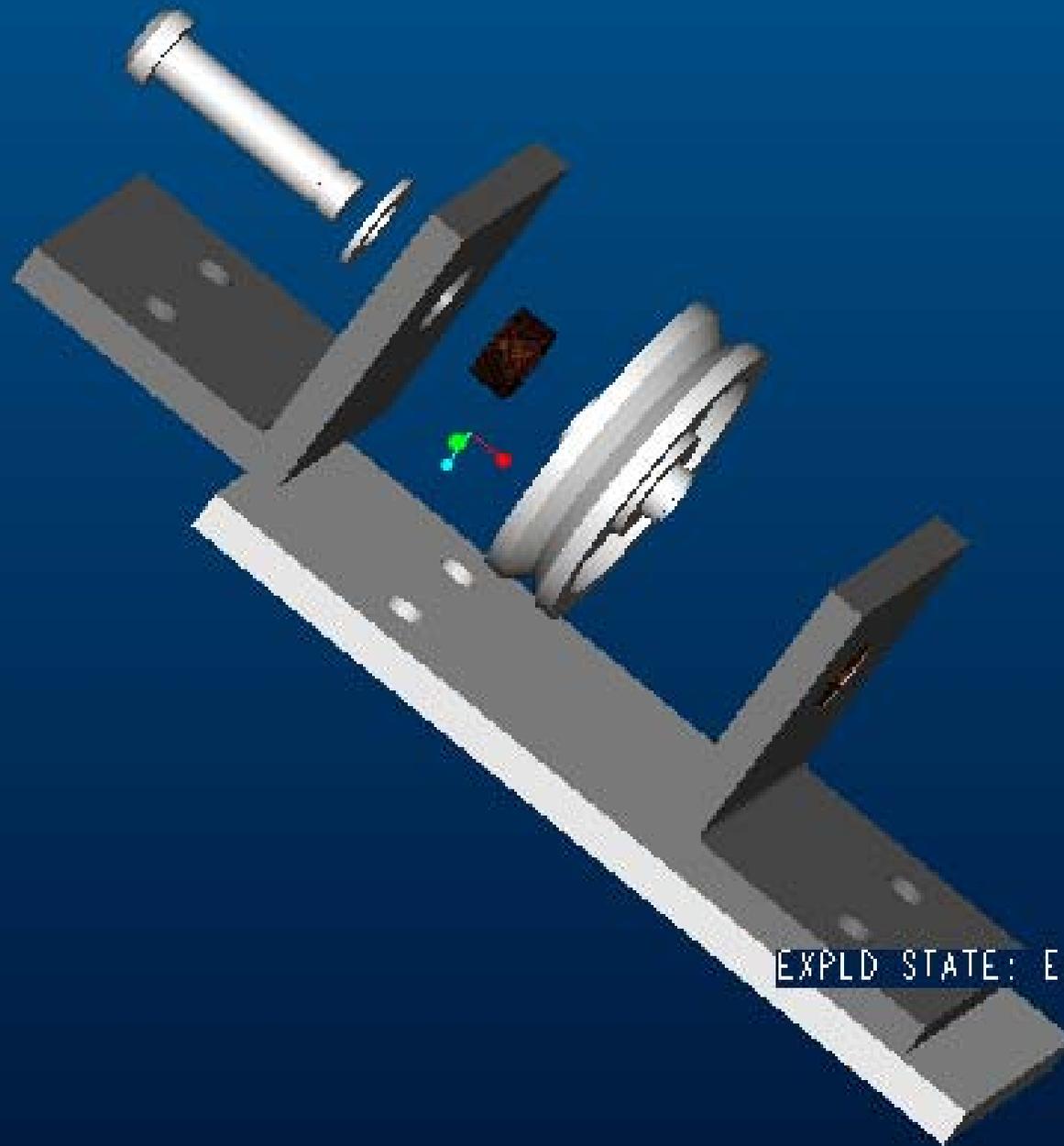
Move... Move Text... Edit Attach... Text Symbol...

Restore Values OK Cancel



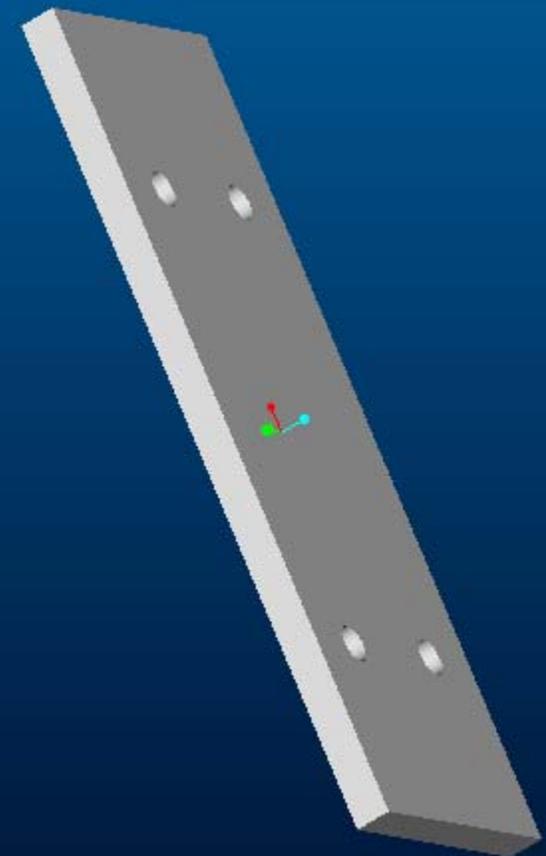
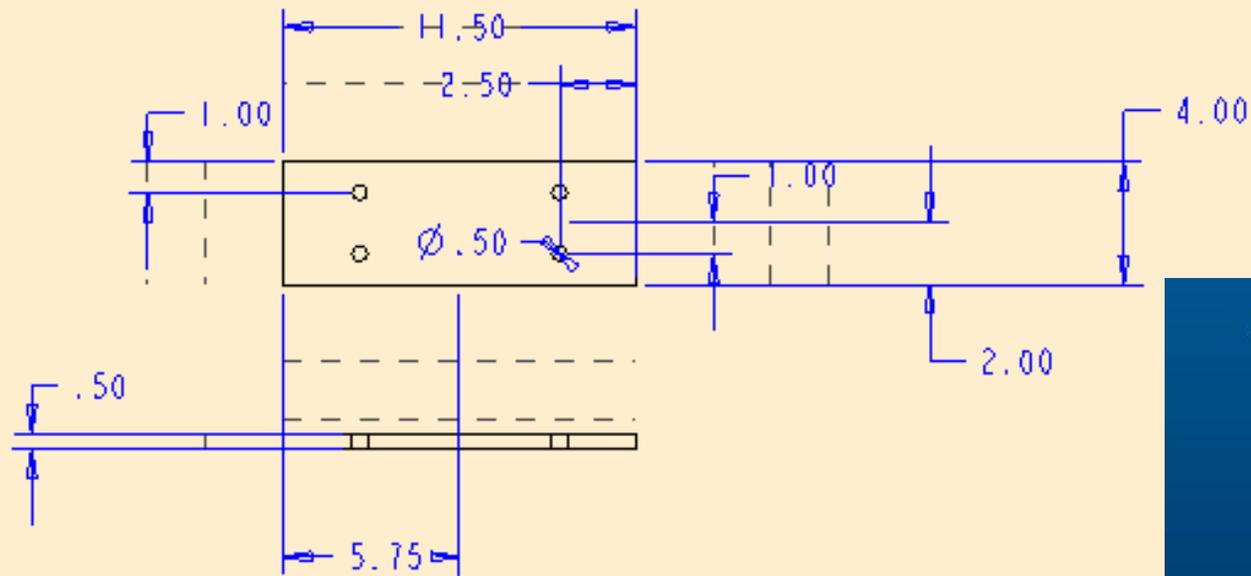
Exercise



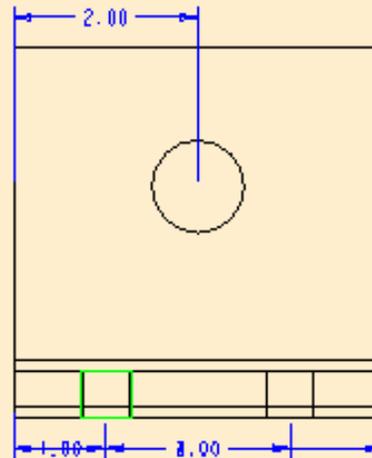
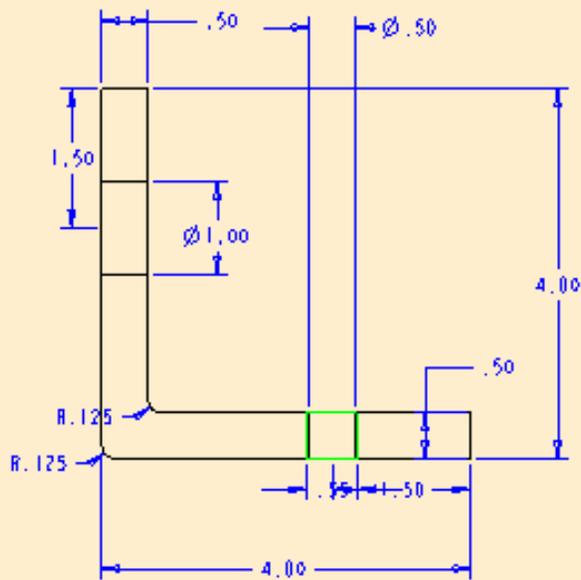
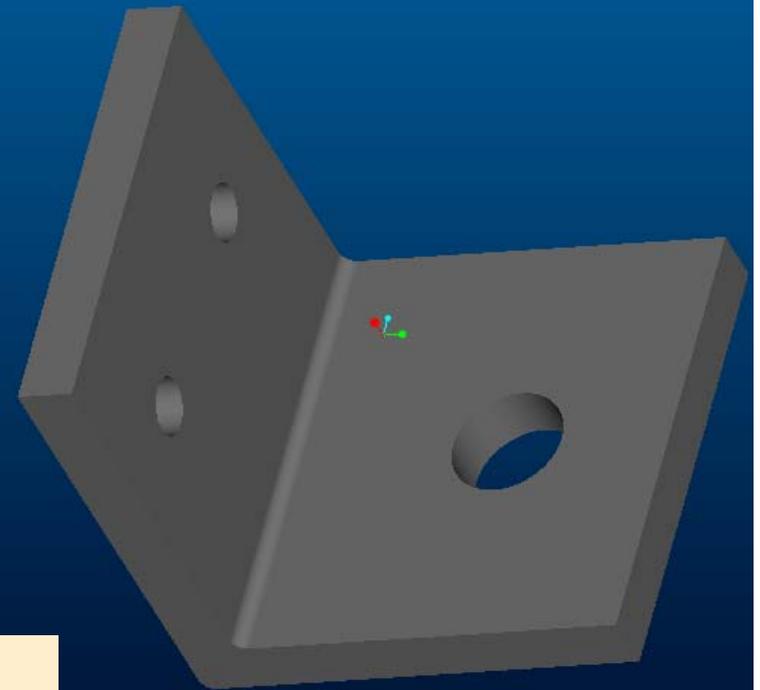


EXPLD STATE: EXP0001

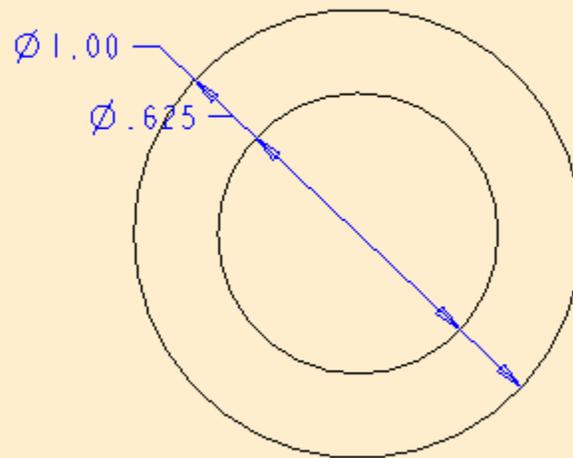
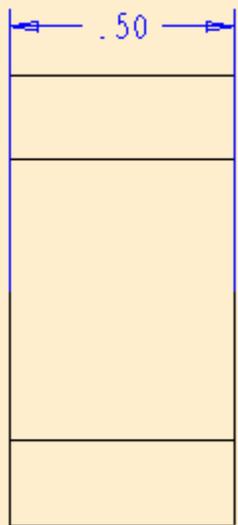
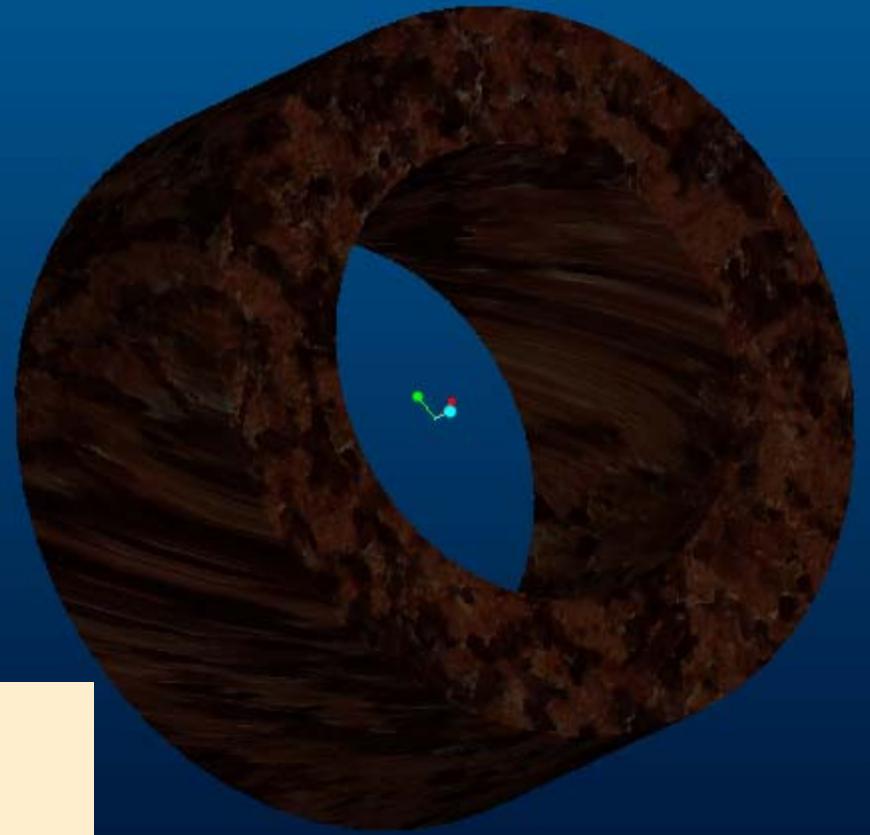
Exercise 8a: Base



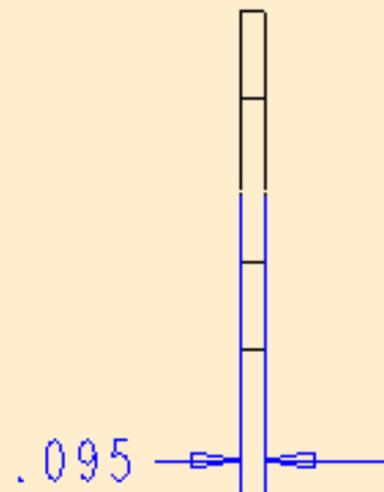
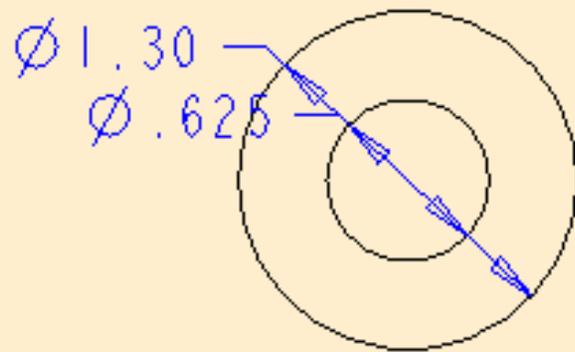
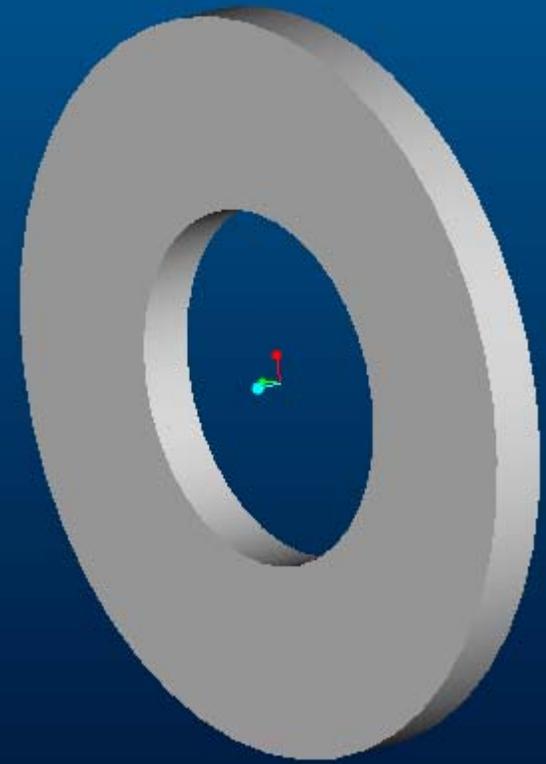
Exercise 8b: Pulley Support



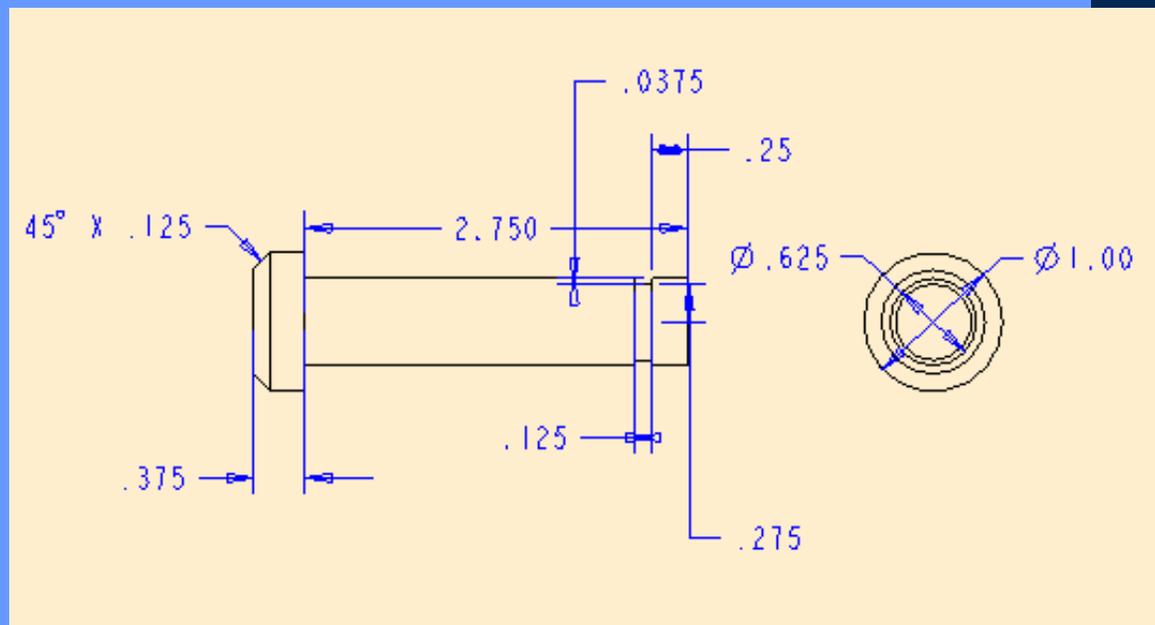
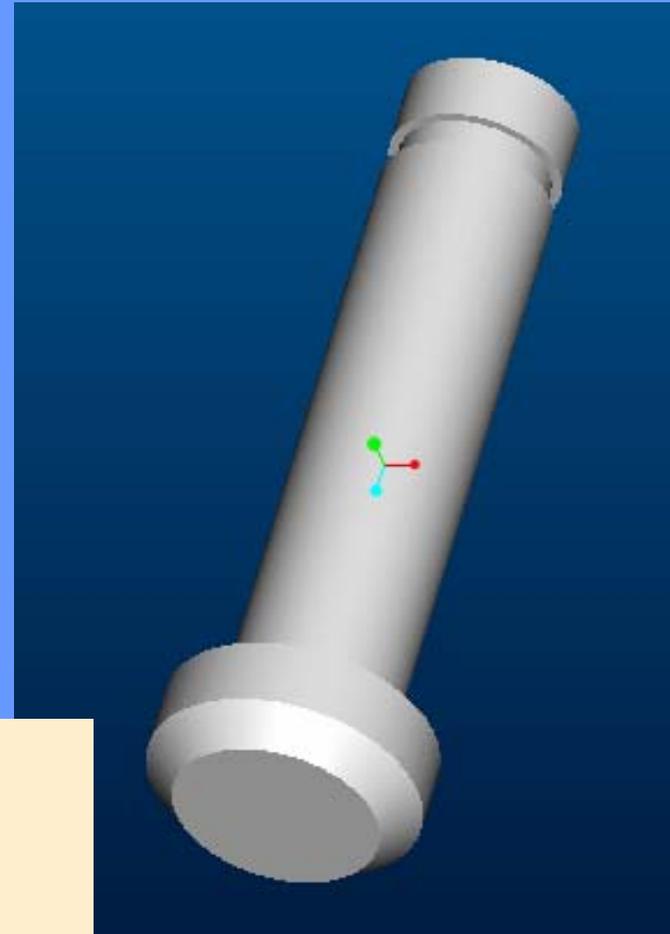
Exercise 9a: Bush



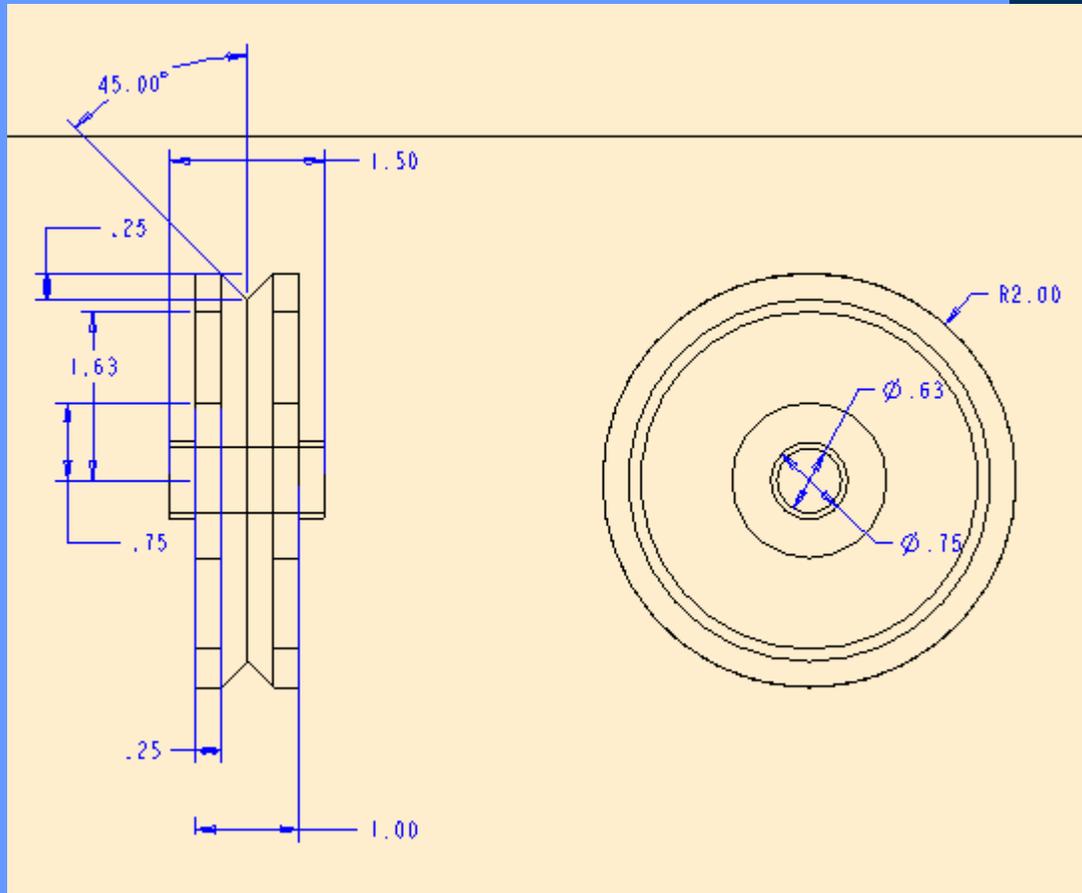
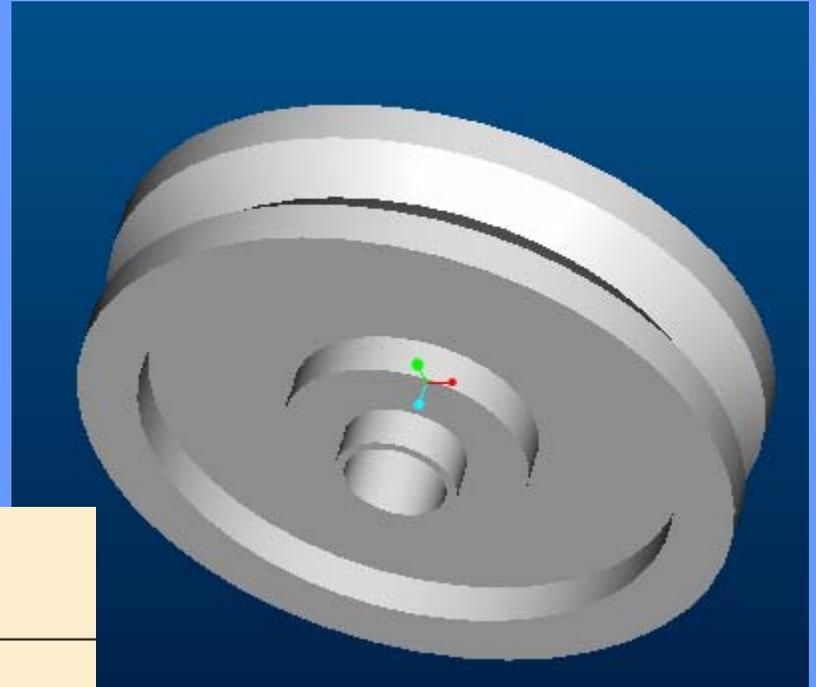
Exercise 9b: Washer



Exercise 9c: Axle Rod

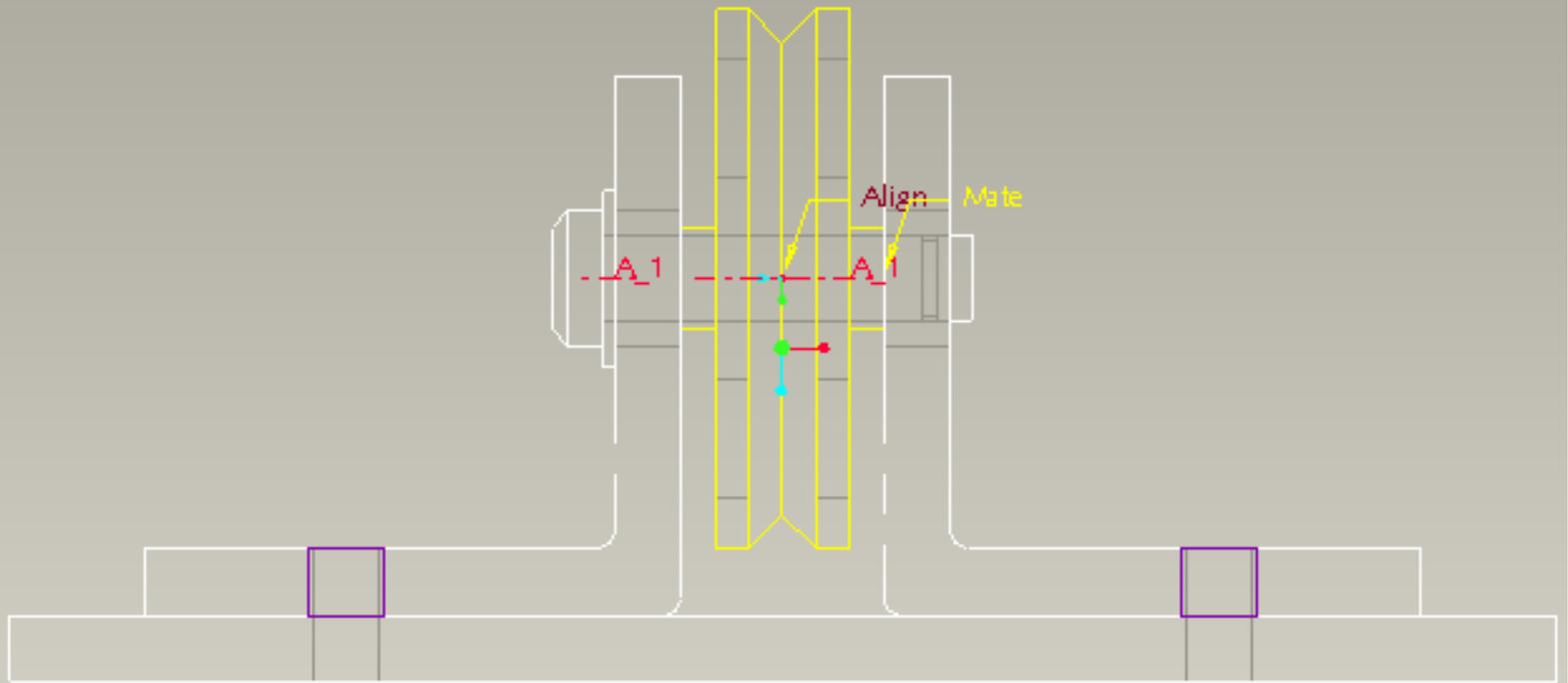


Exercise 10: Pulley

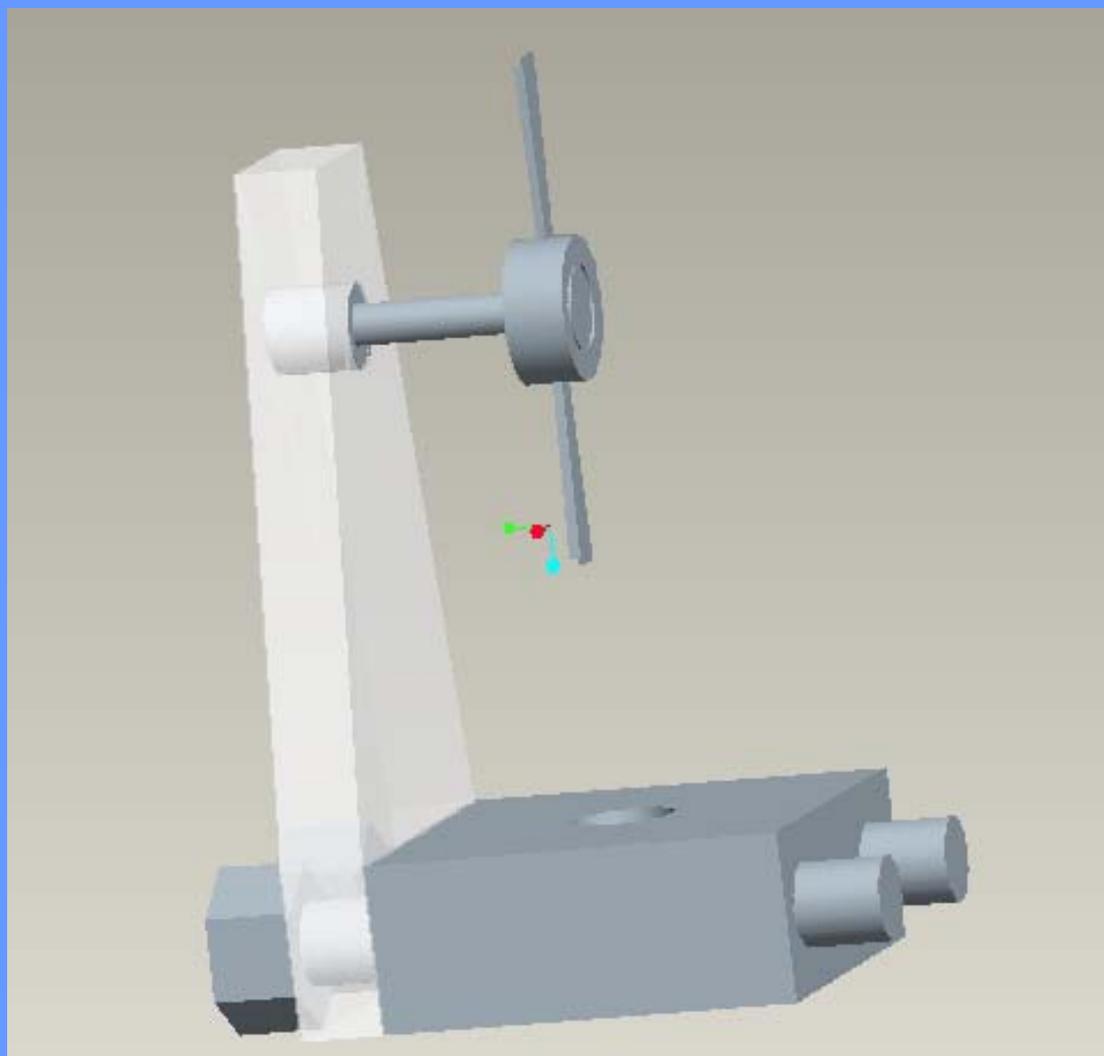


Exercise 11

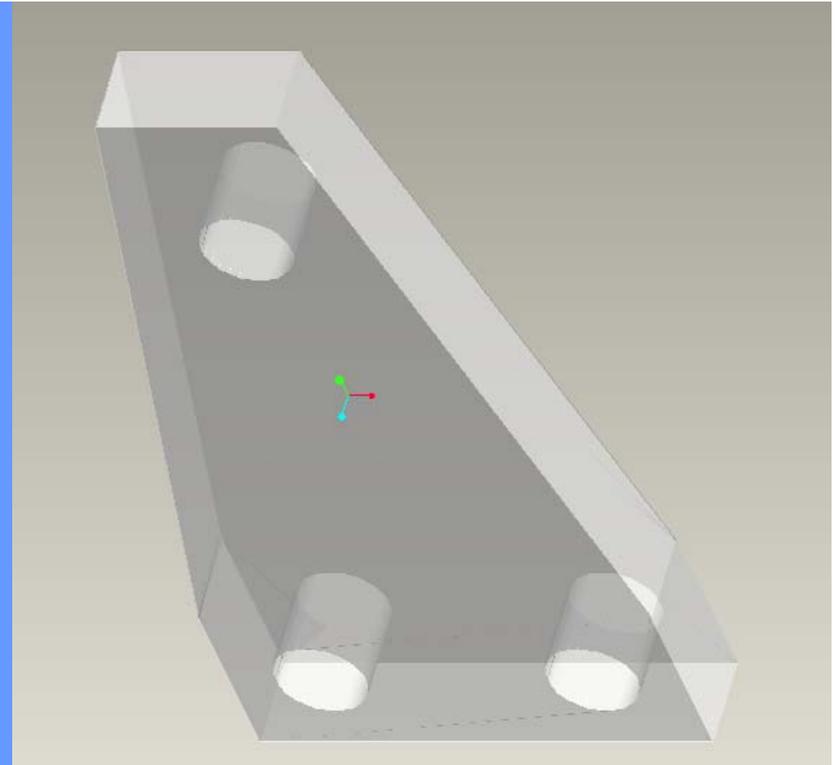
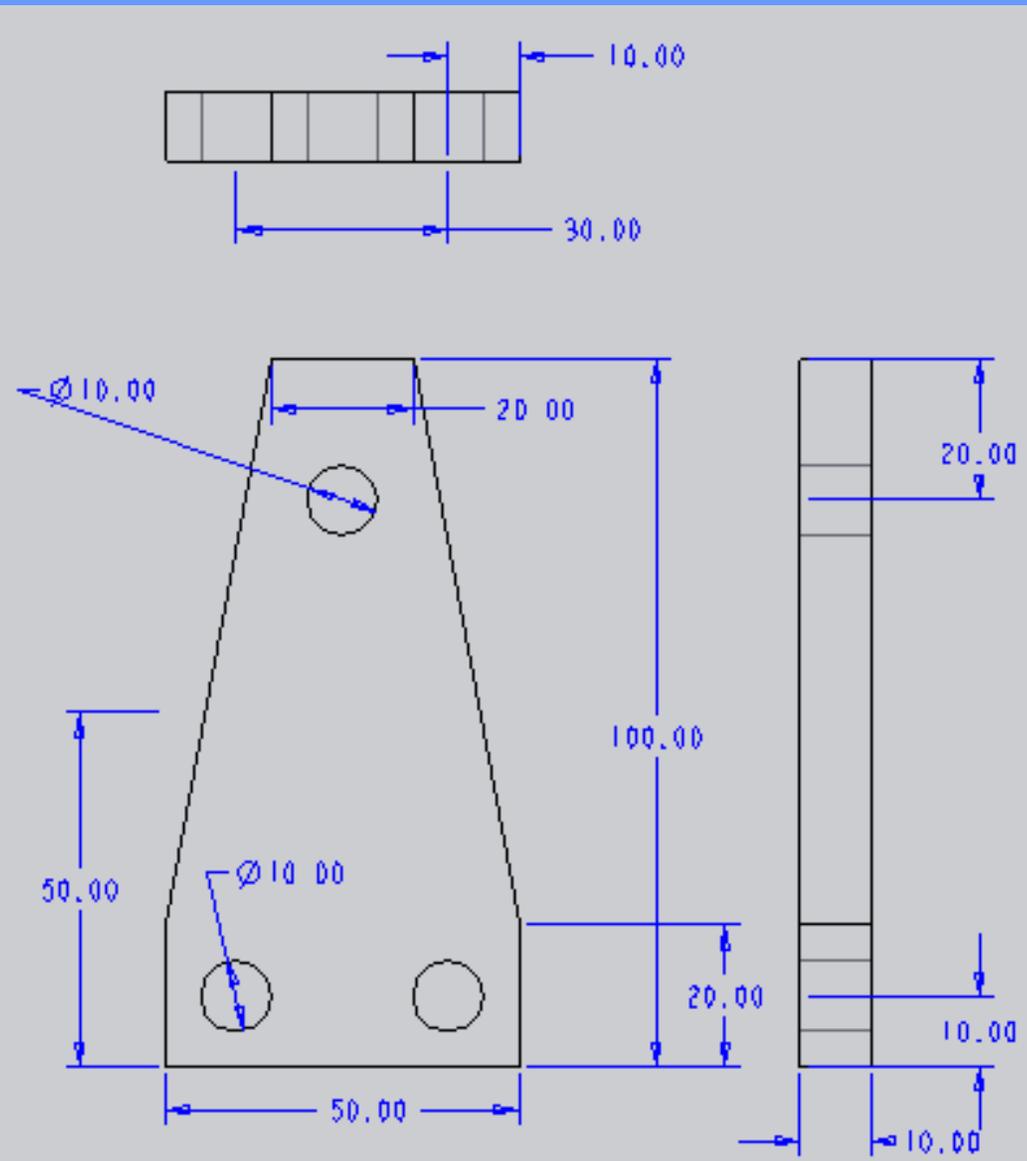
- PULLEY-SUPPORT.ASM
 - PULLEY-BASE.PRT
 - SUPPORT.PRT
 - SUPPORT.PRT
 - BUSH.PRT
 - BUSH.PRT
 - WASHER.PRT
 - AXLE-ROD.PRT



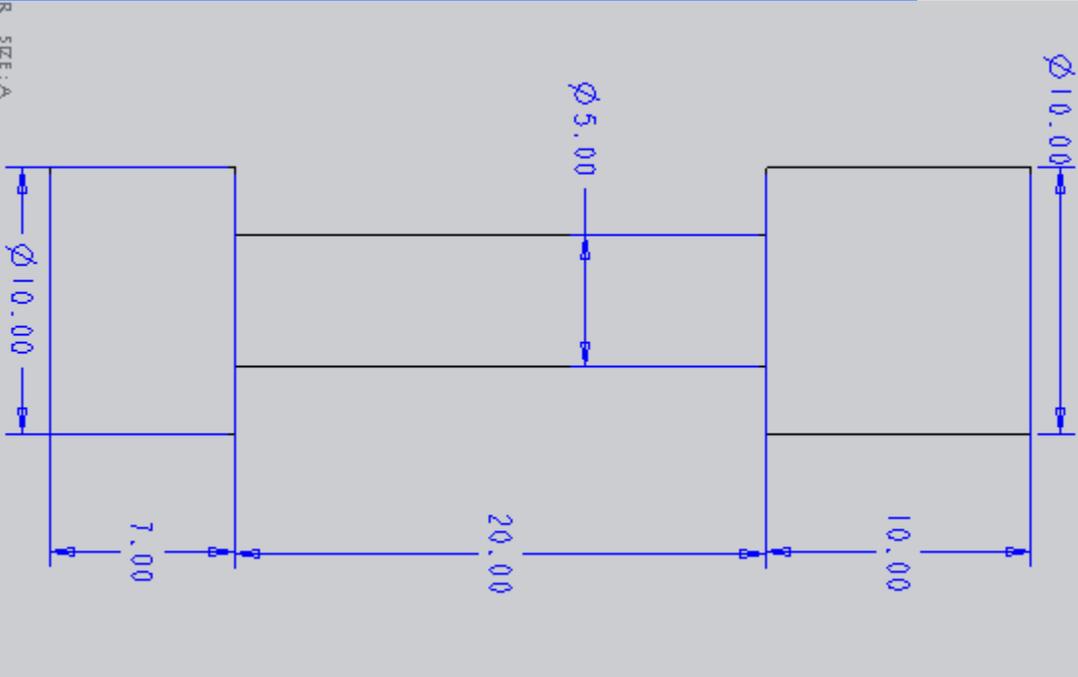
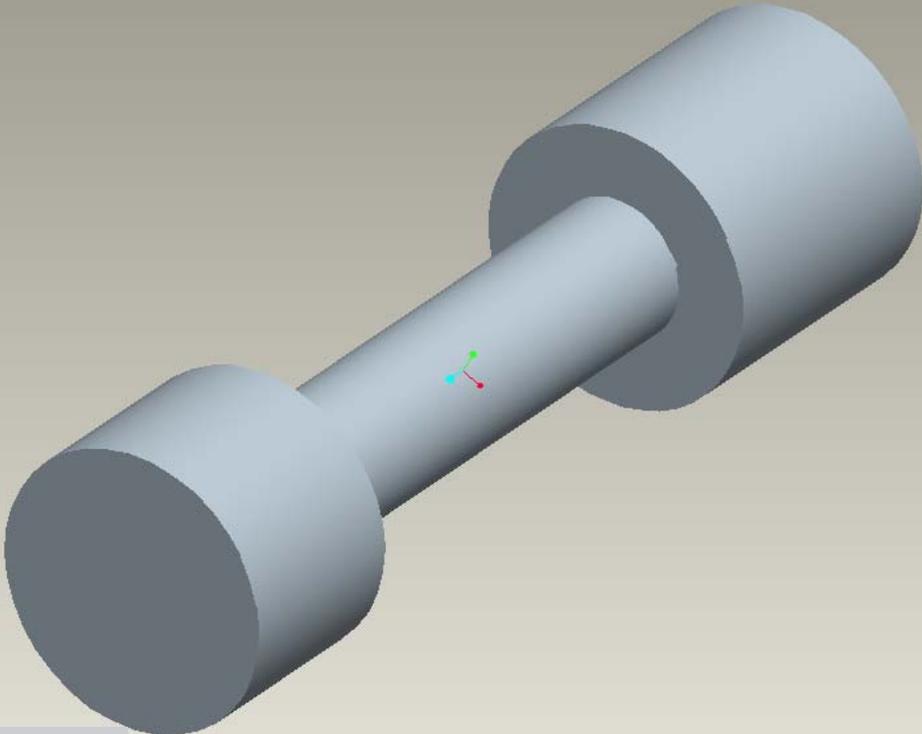
Exercise



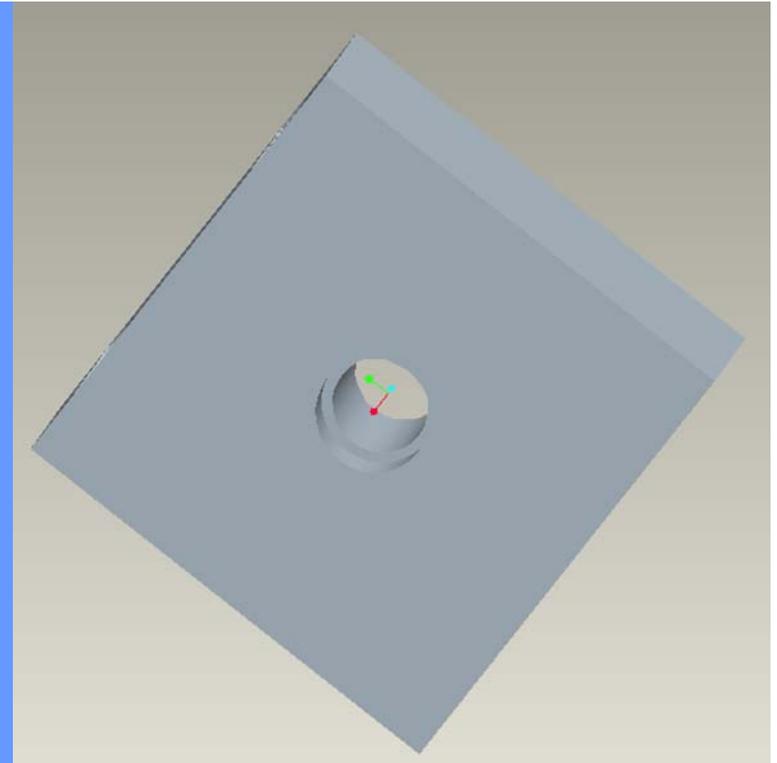
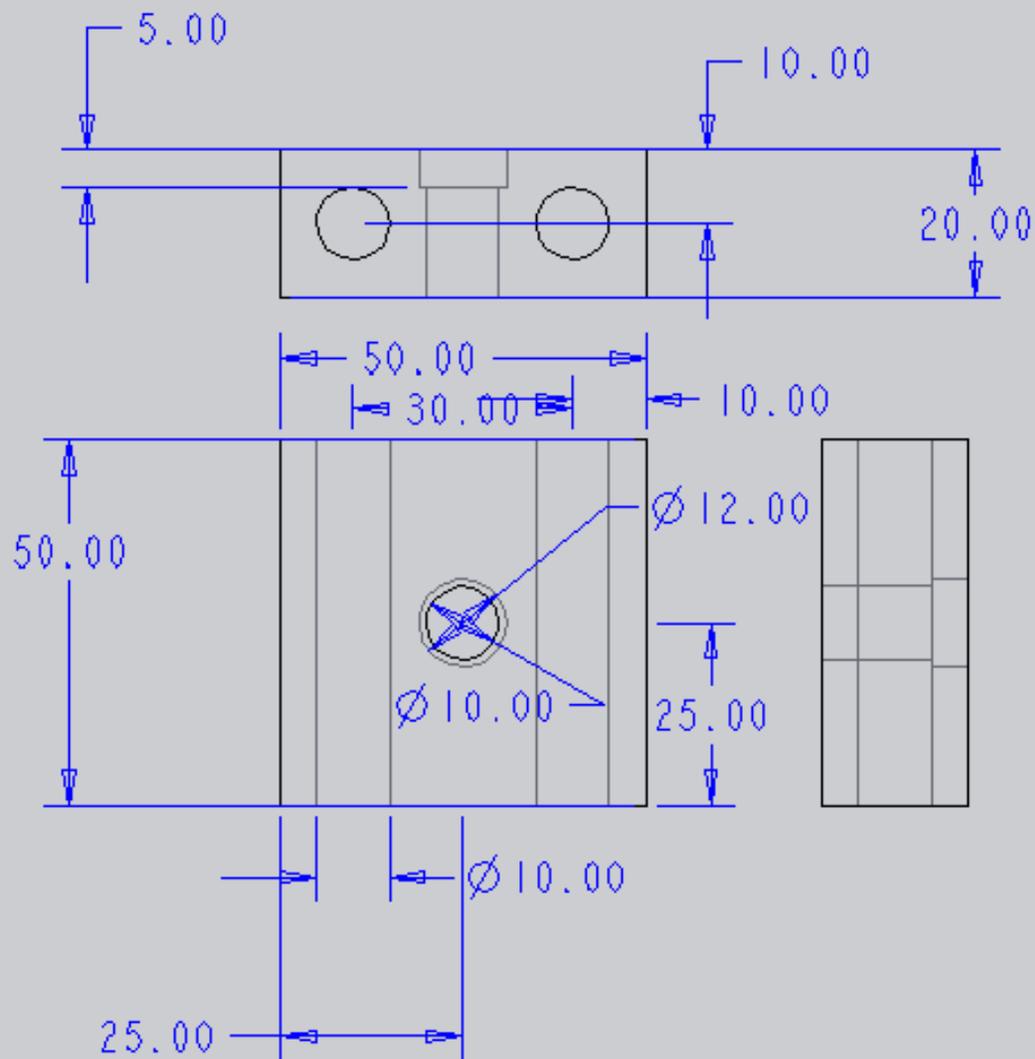
Bracket



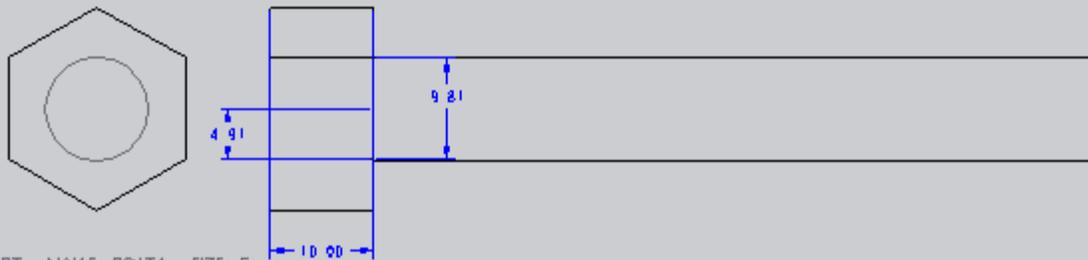
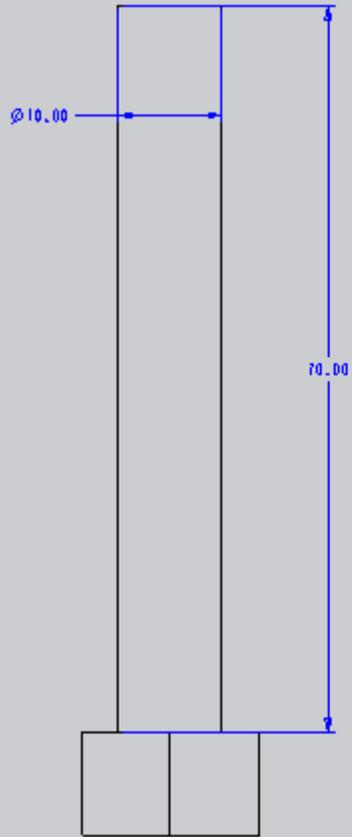
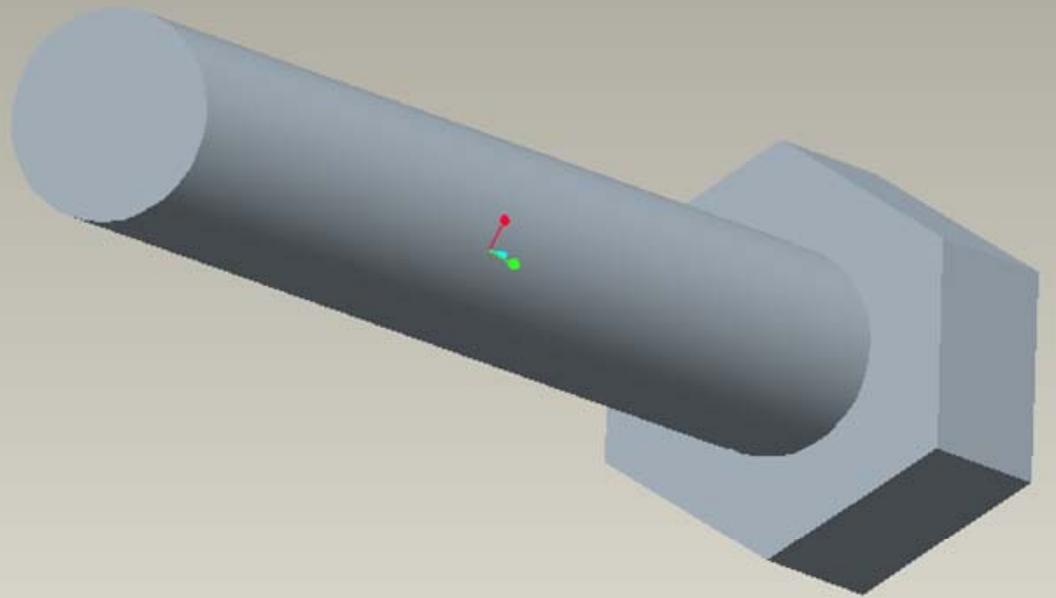
Connector



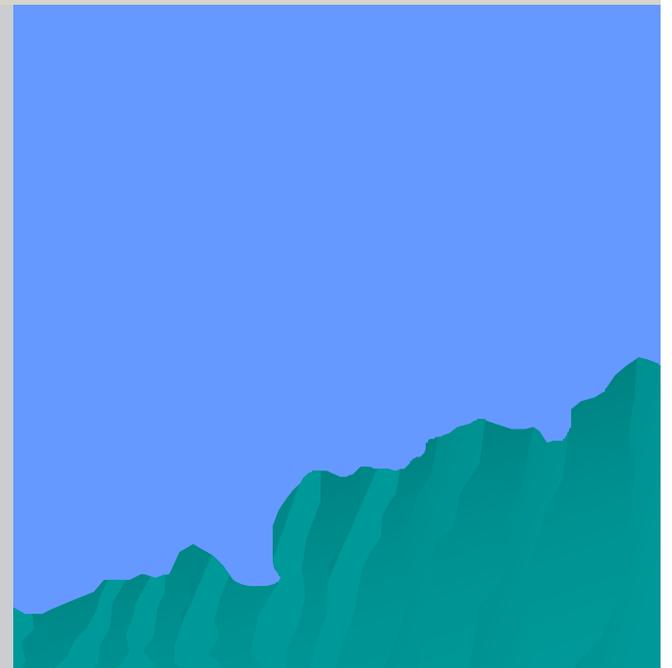
Base Block



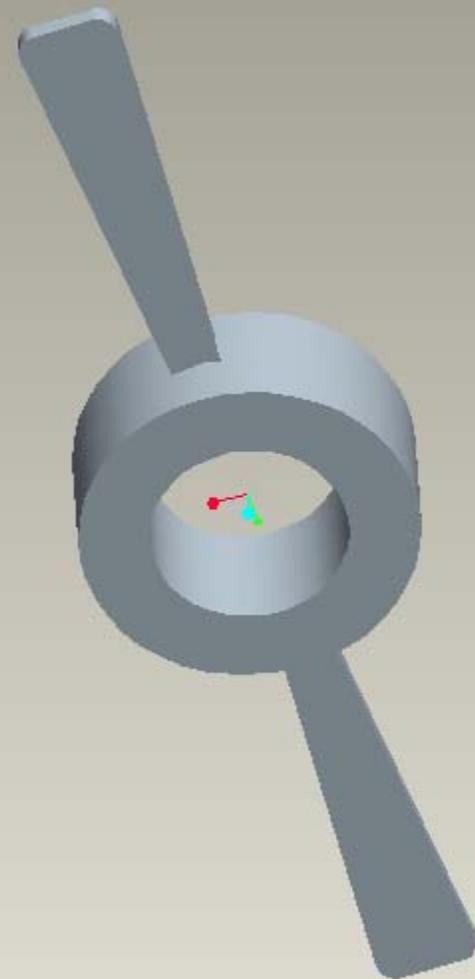
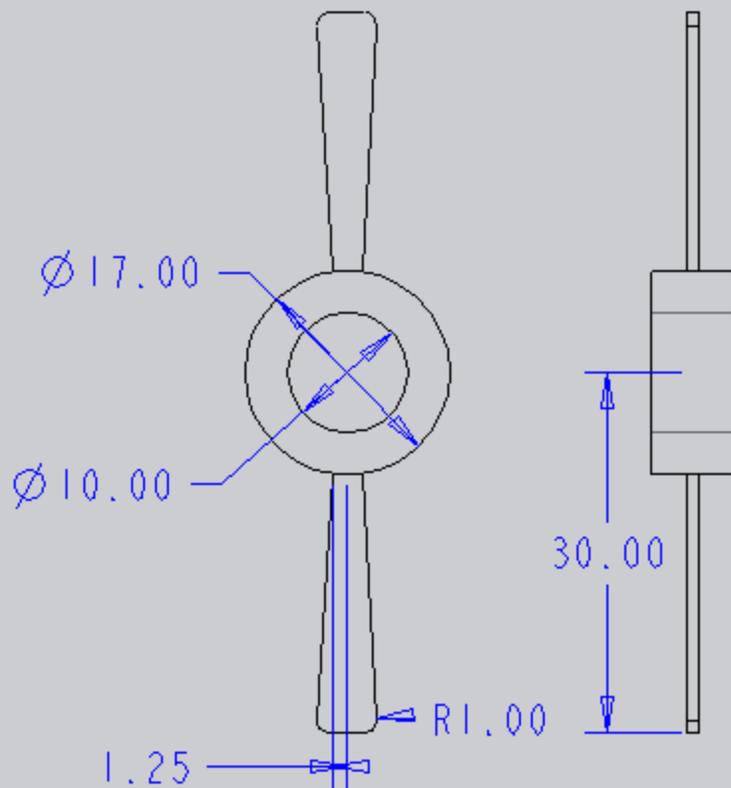
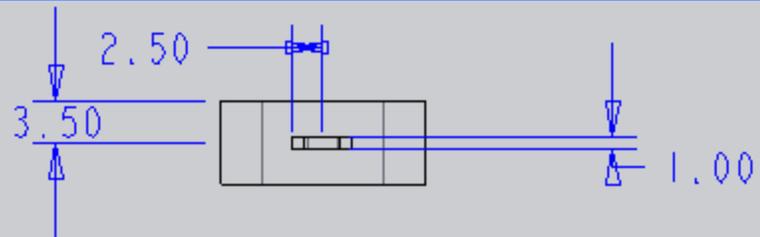
Bolt



TYPE - PART NAME - BOLT1 SIZE - C



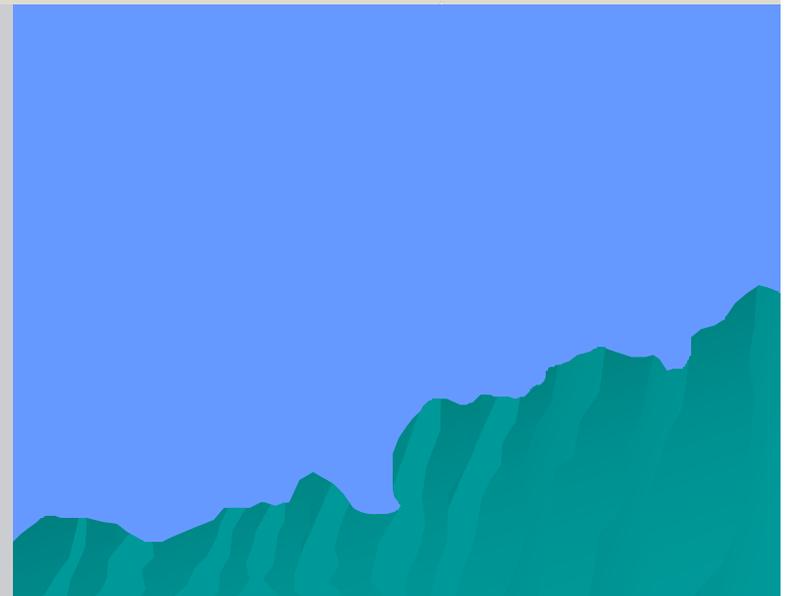
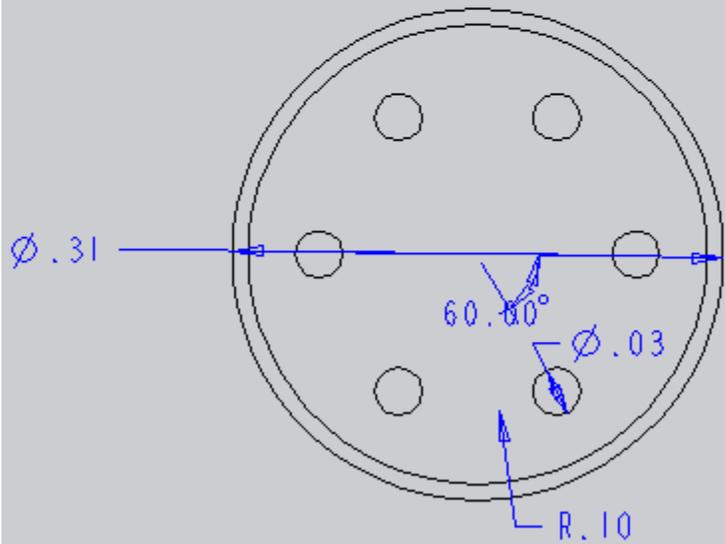
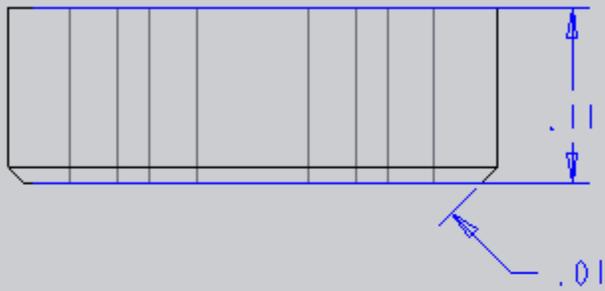
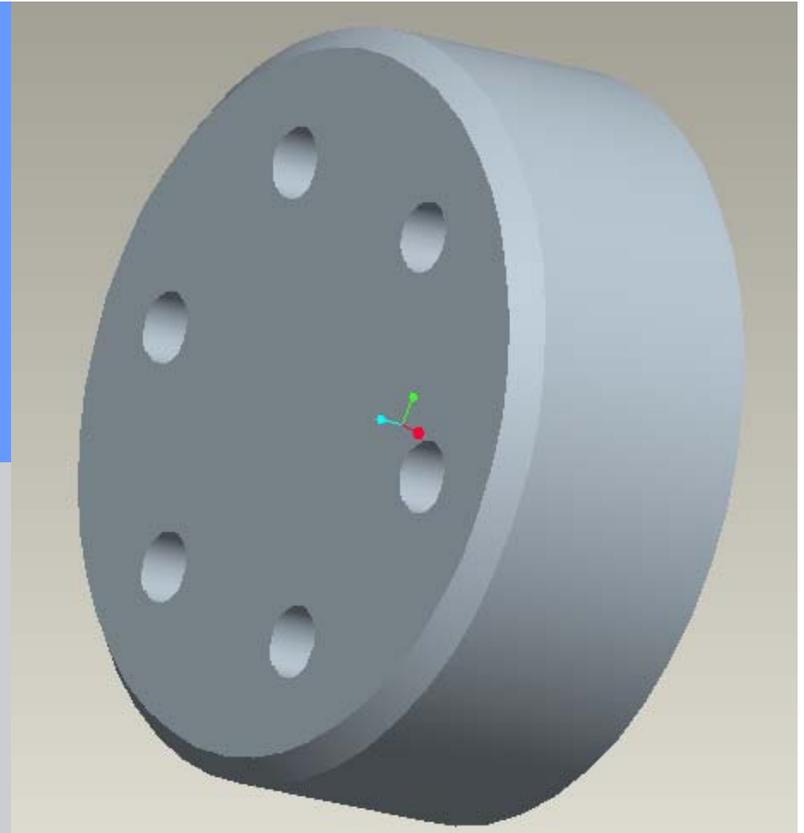
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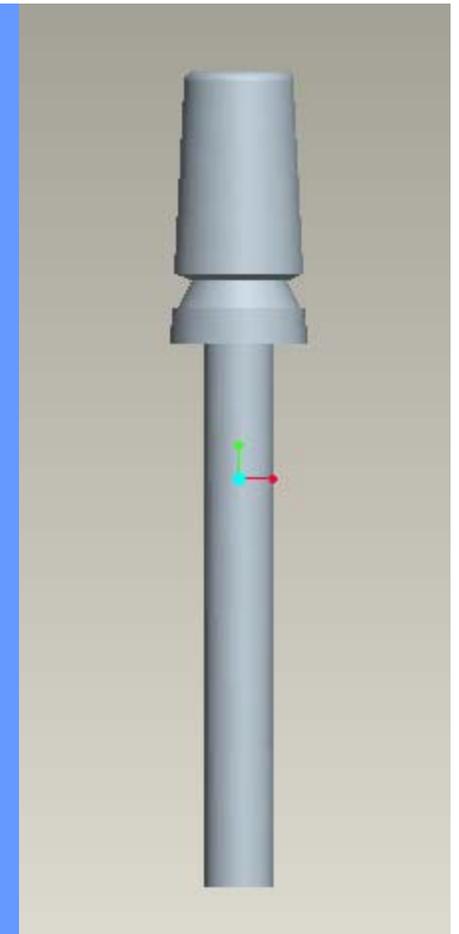
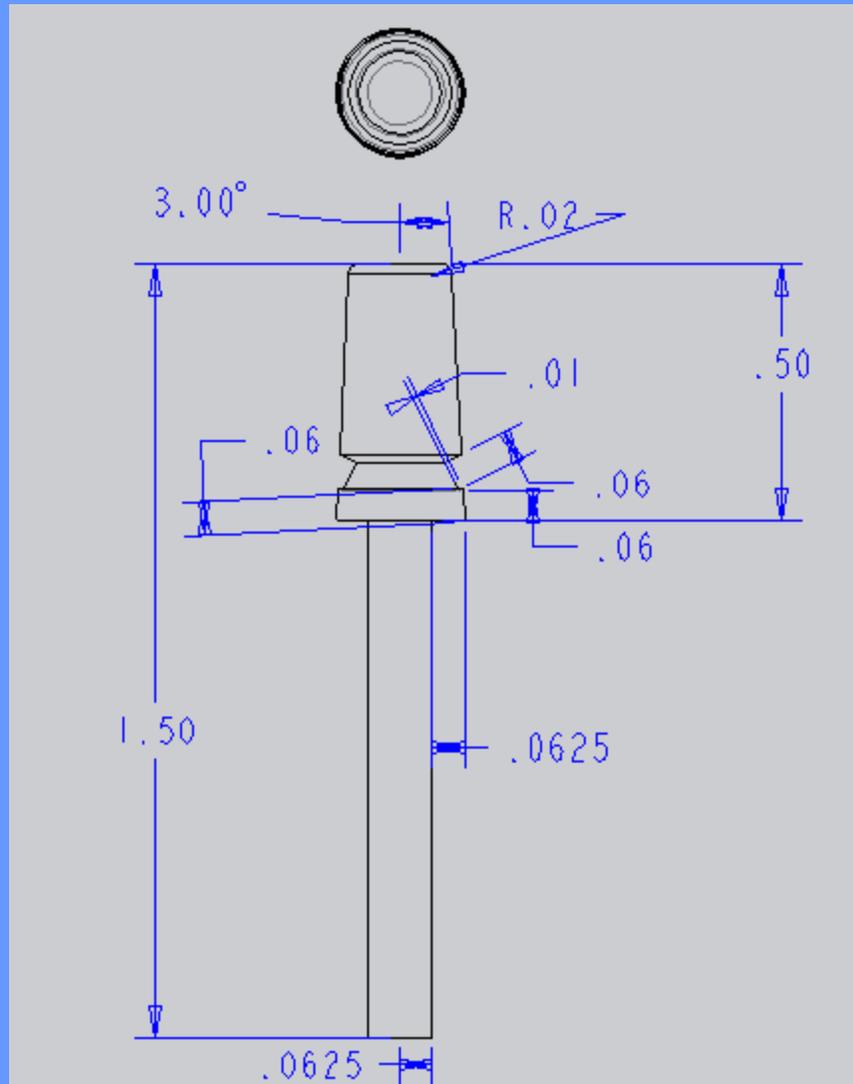
End-Sem-Paper 2007



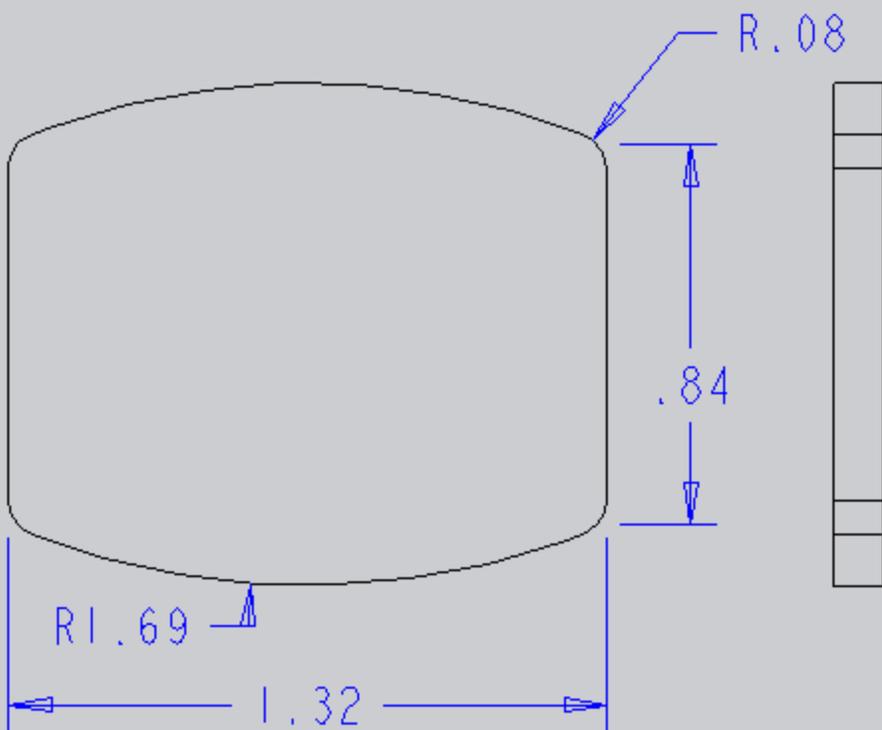
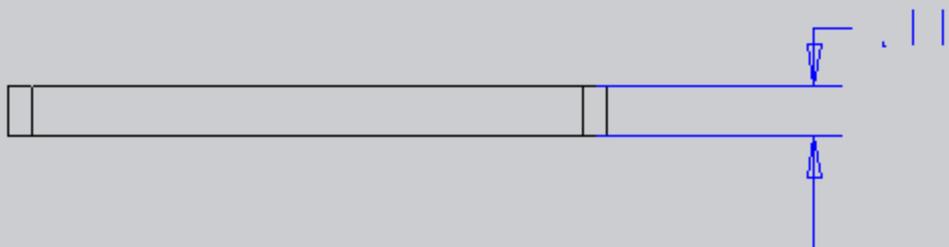
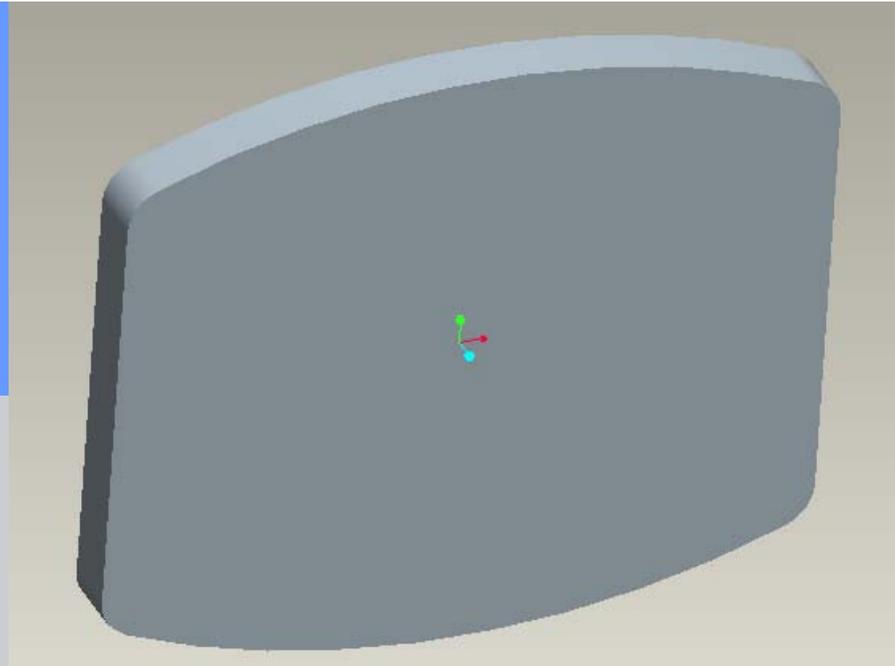
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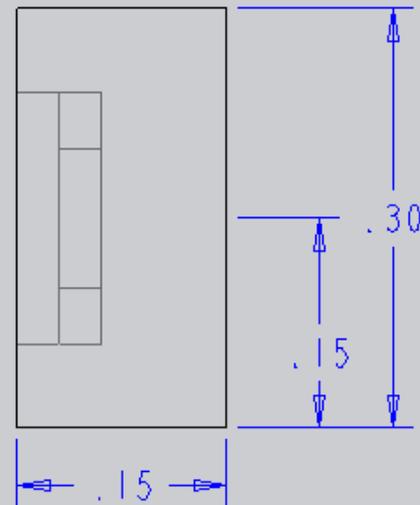
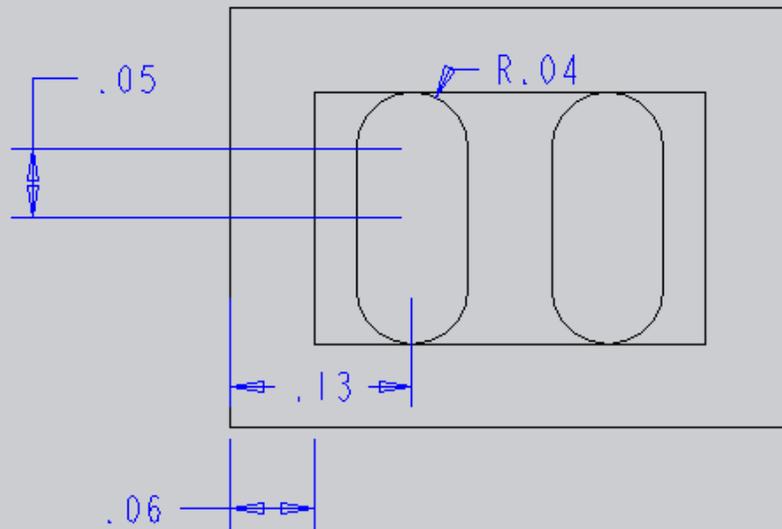
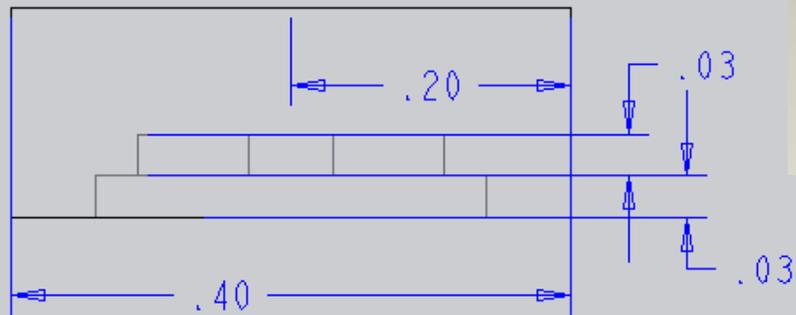
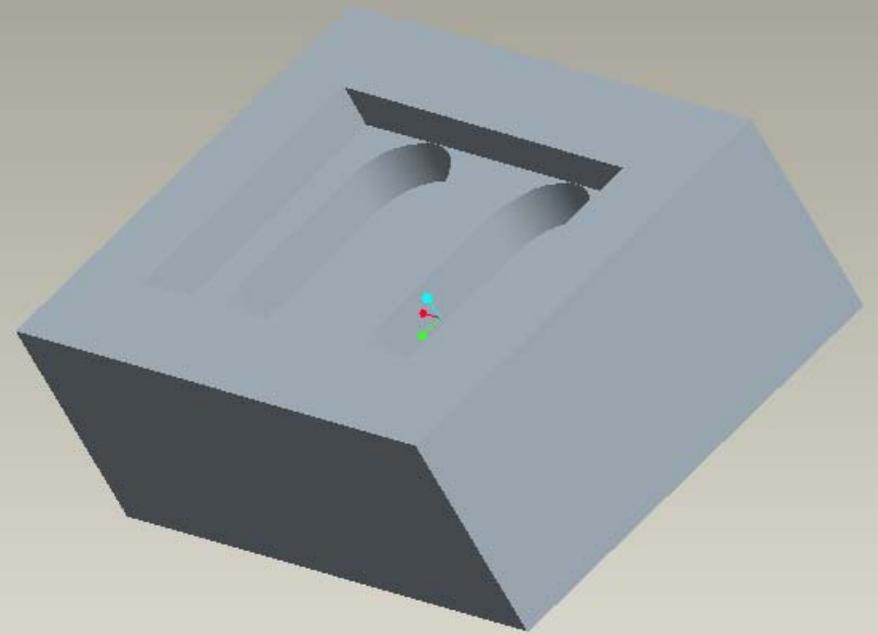
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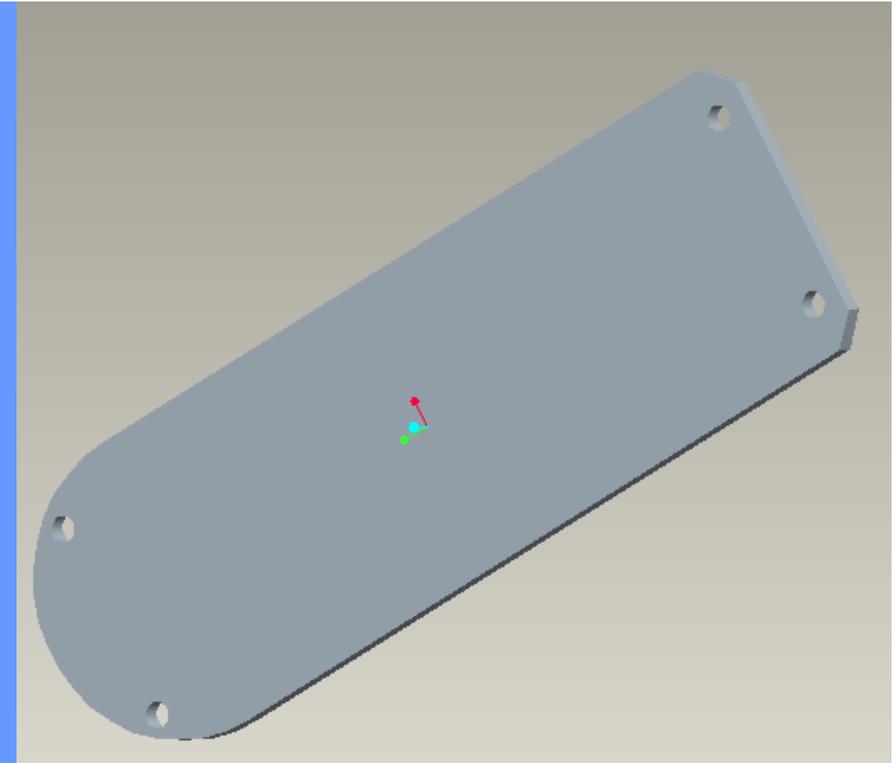
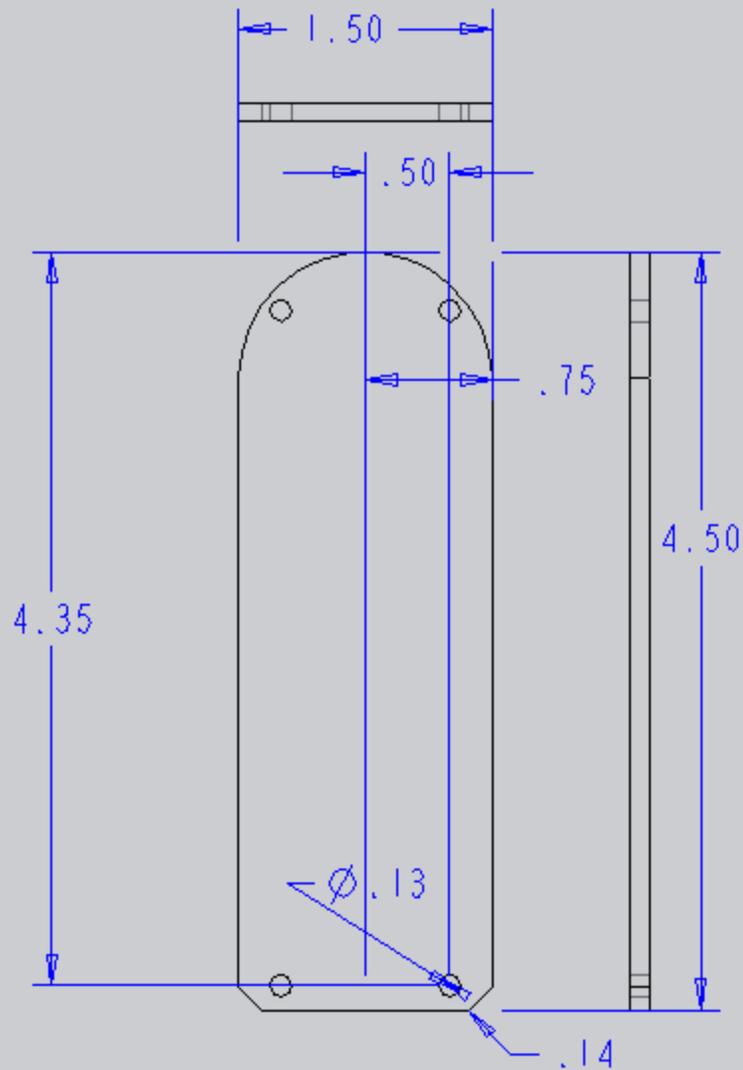
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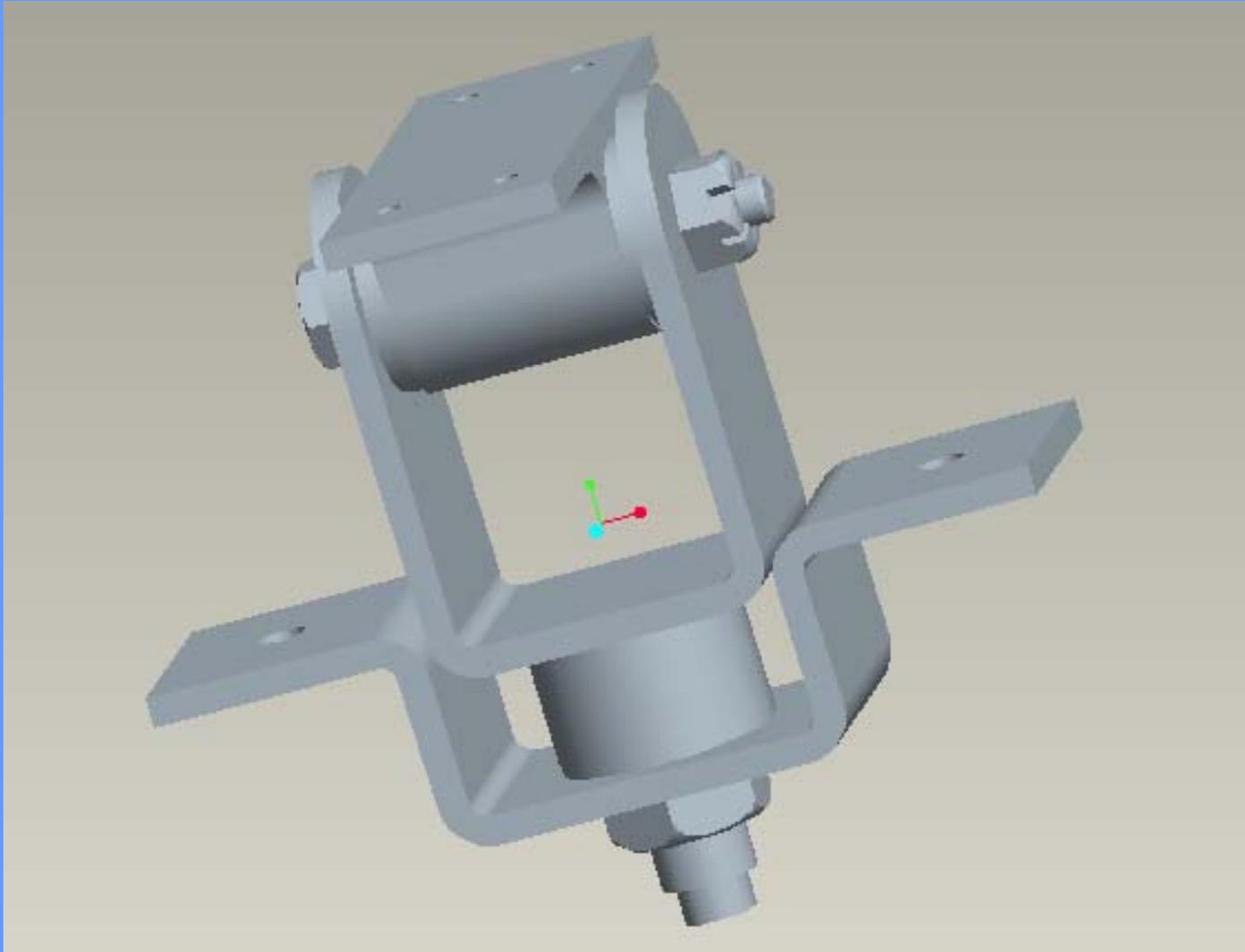
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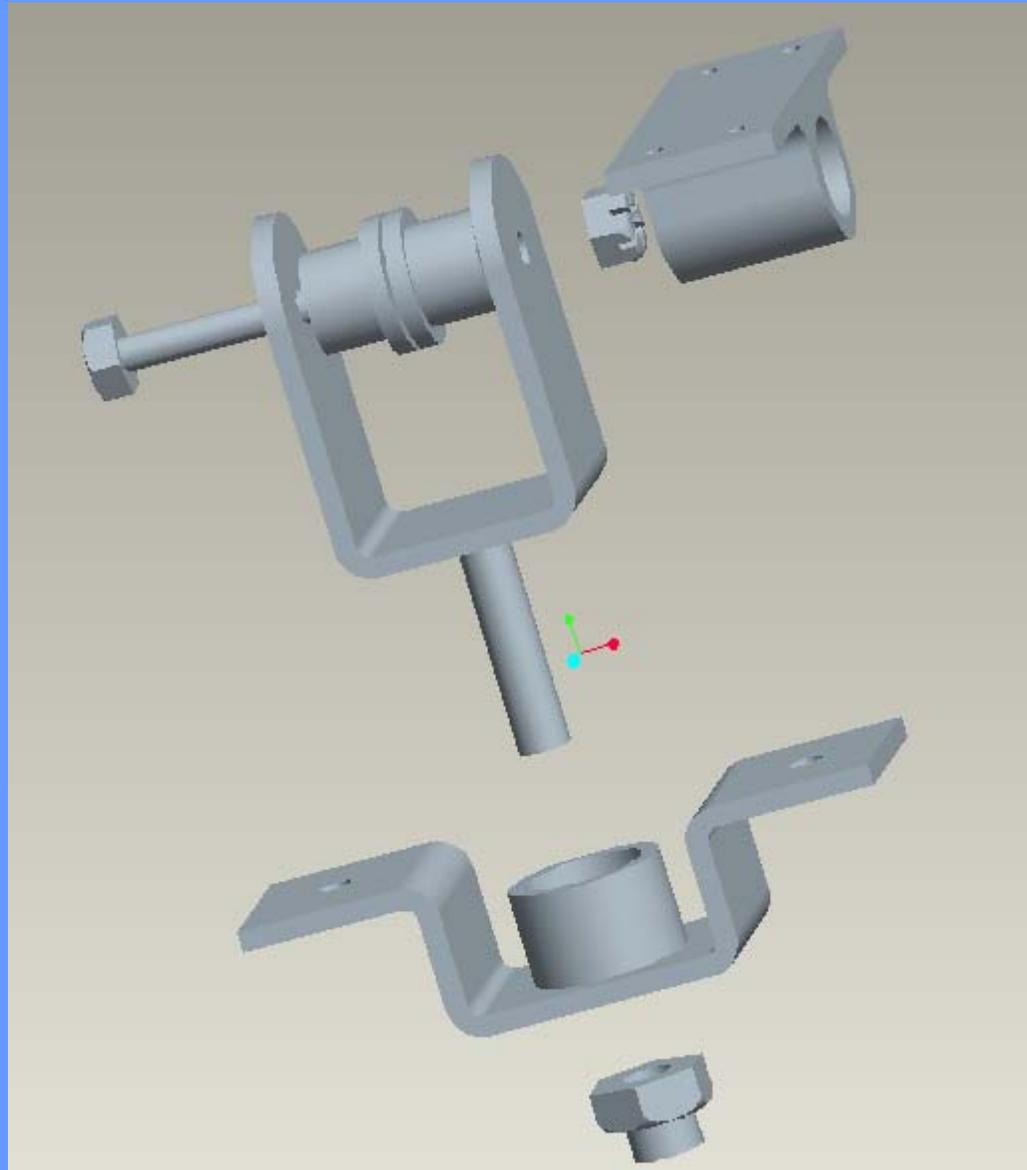
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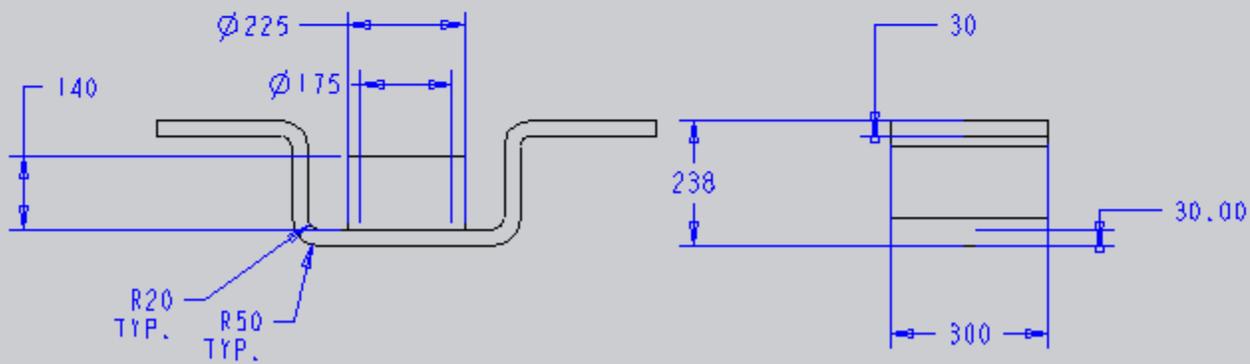
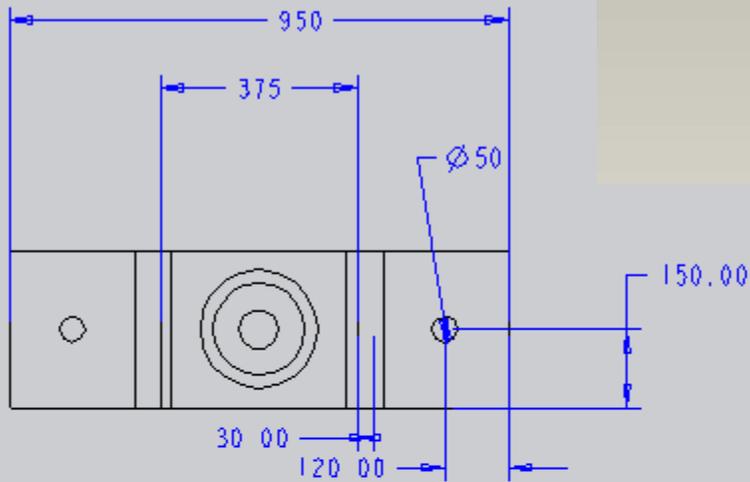
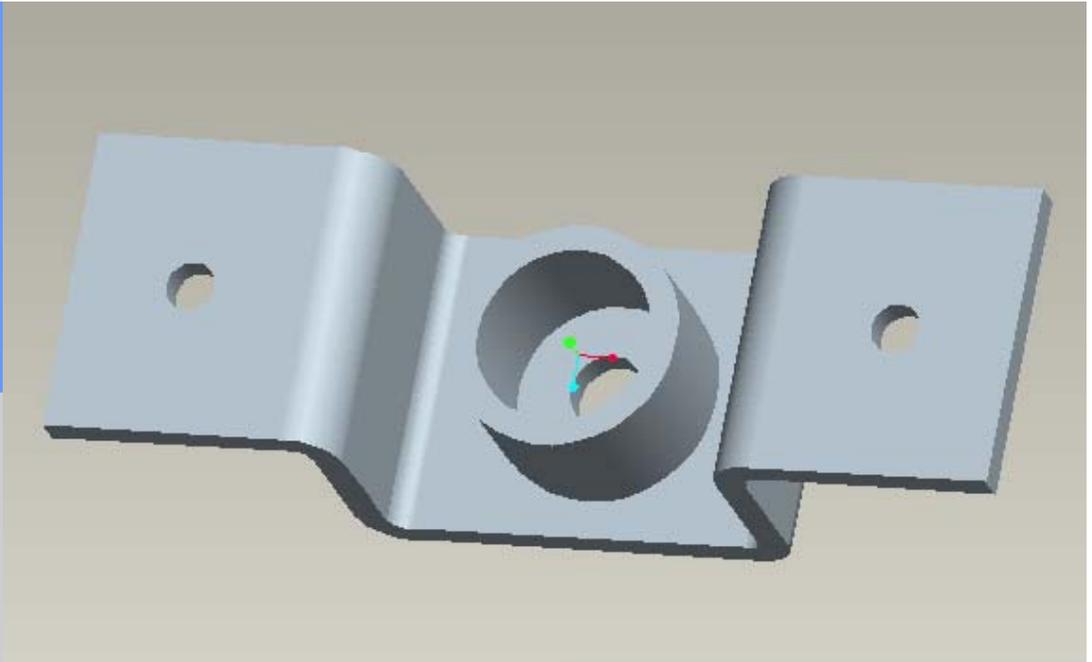
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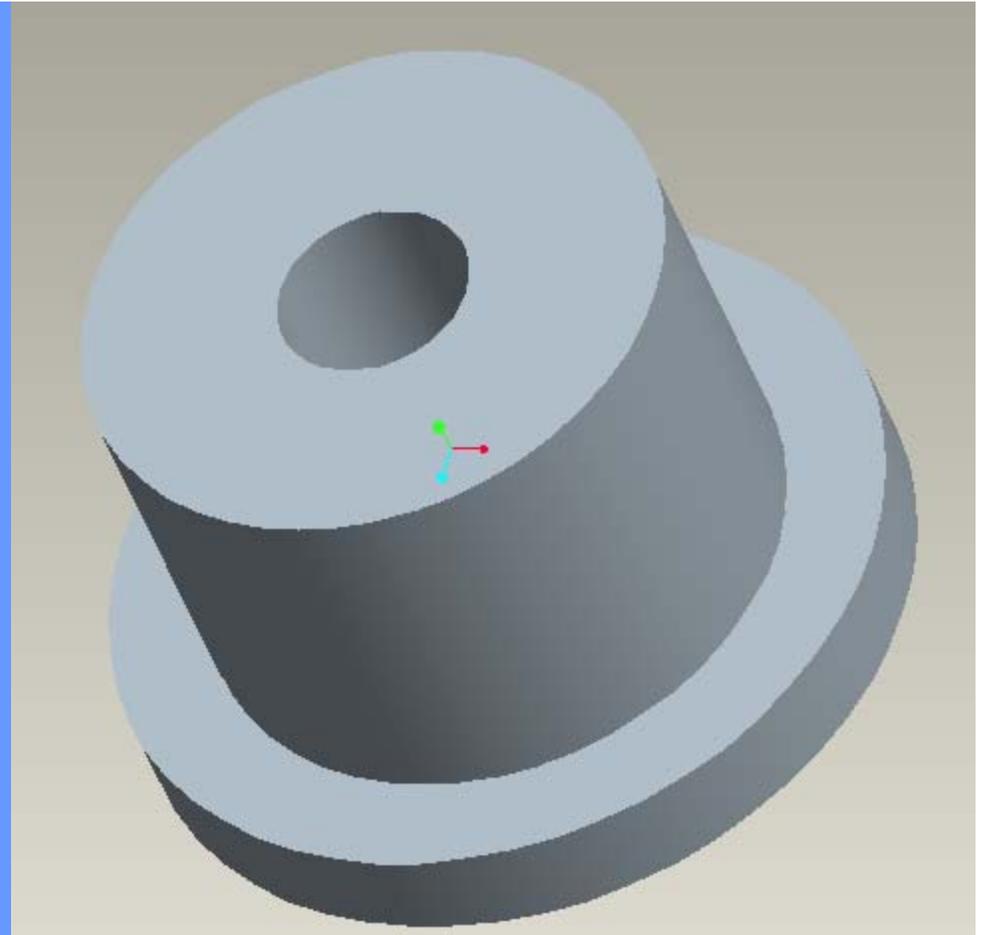
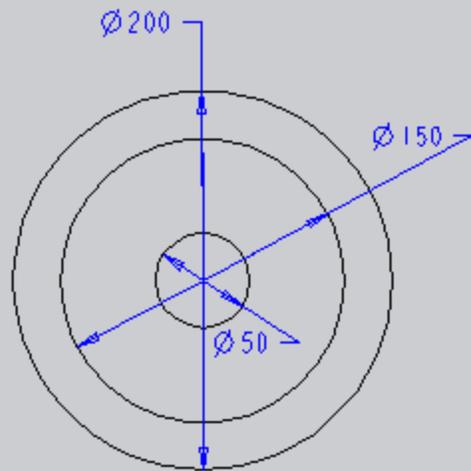
Exploded View of Assembly



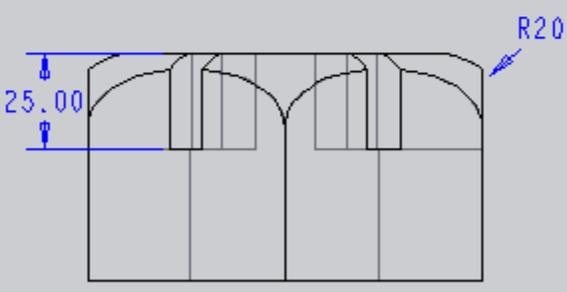
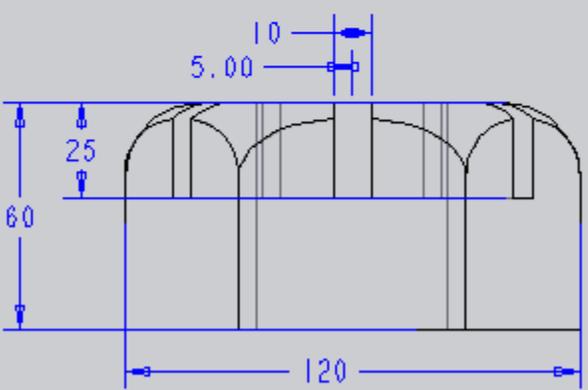
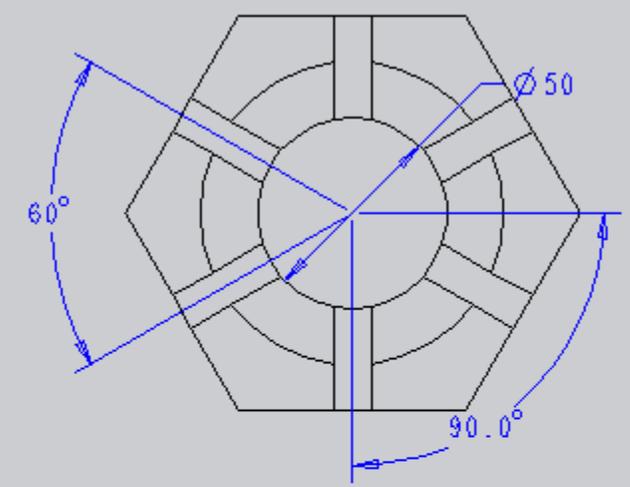
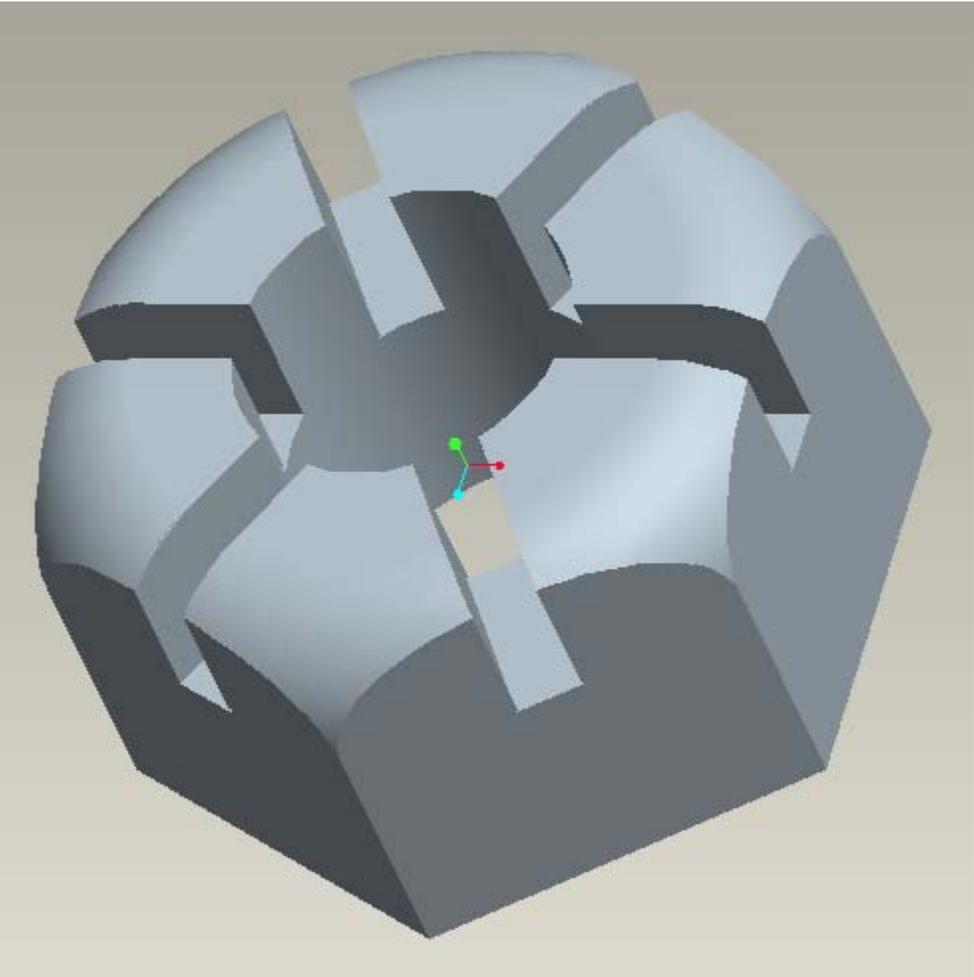
Bracket _ Assembly



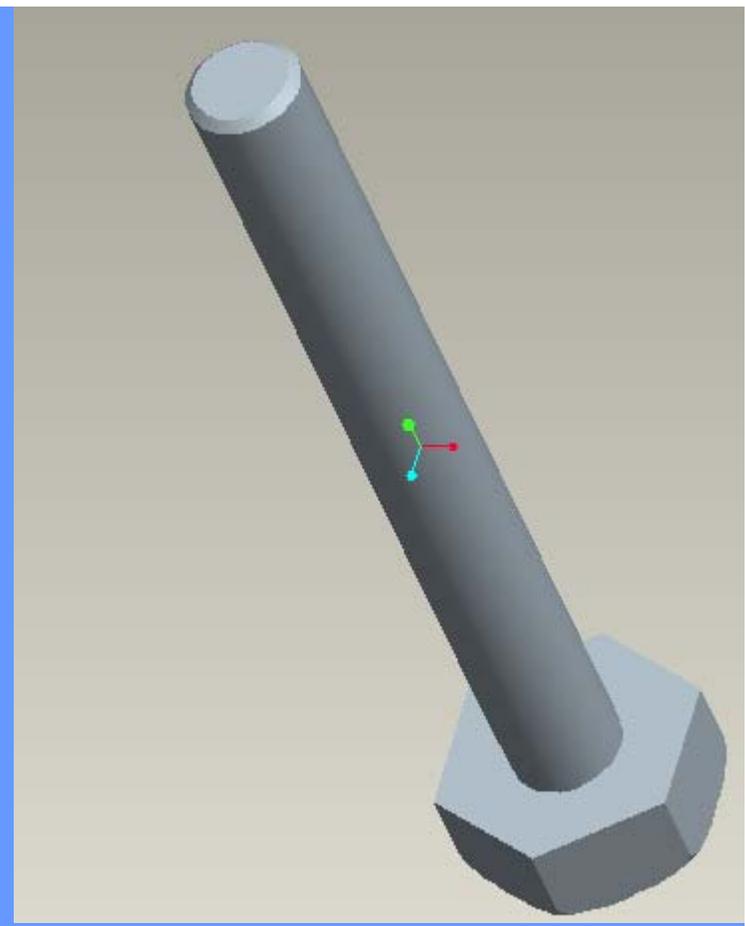
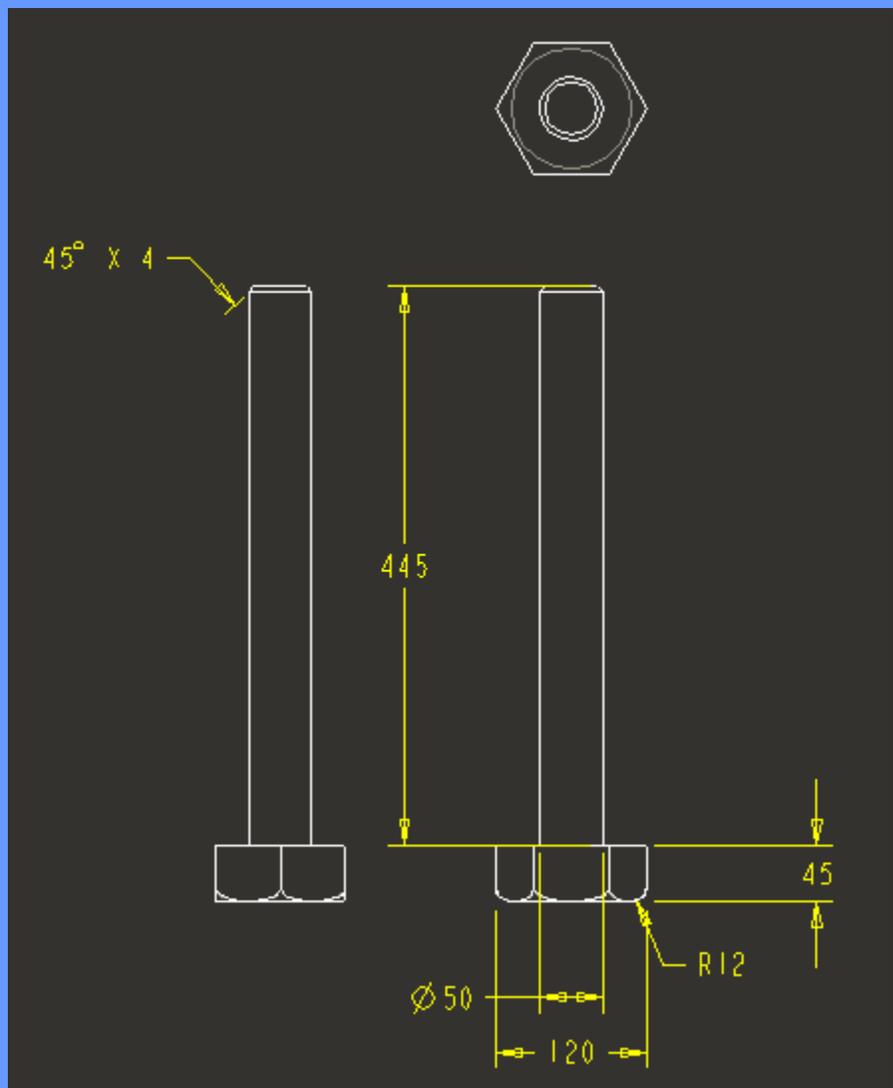
Bushing_Assembly



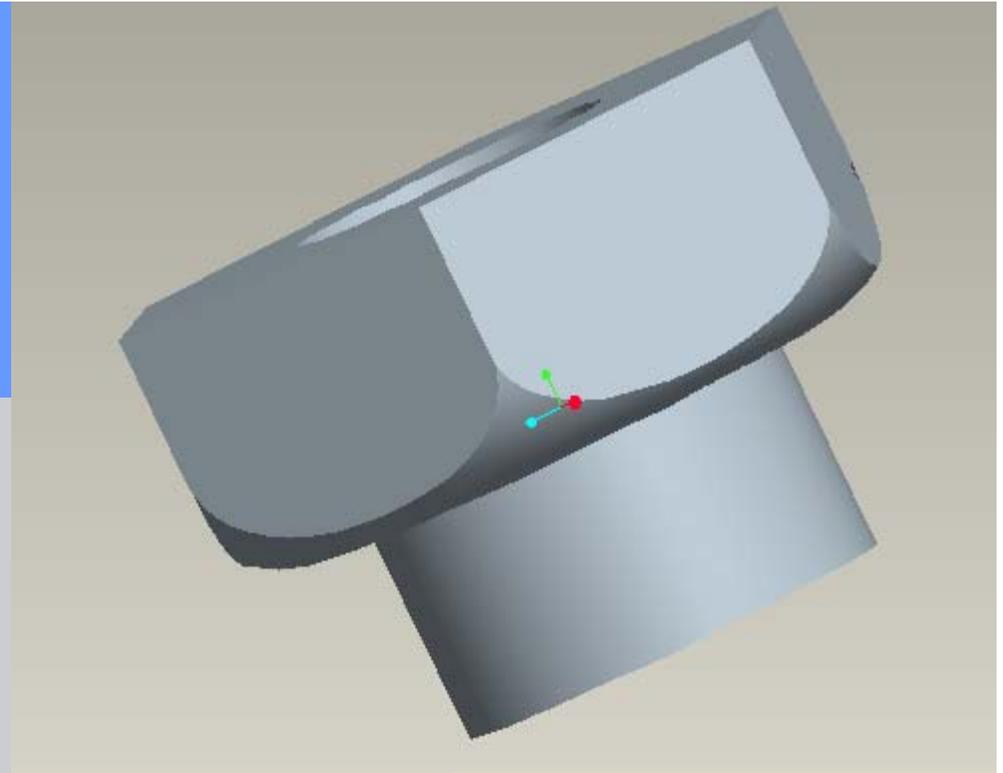
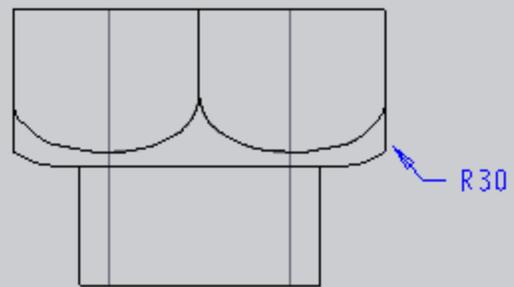
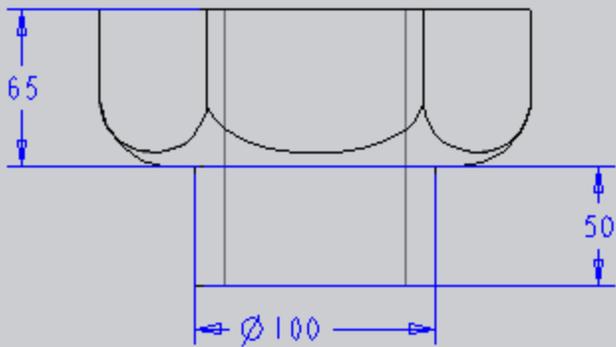
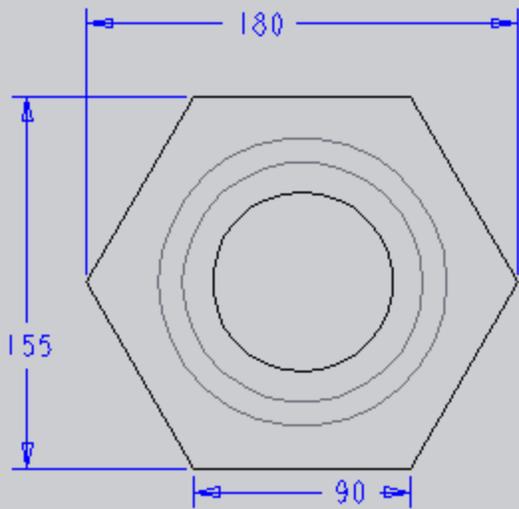
Castle Nut_Assembly



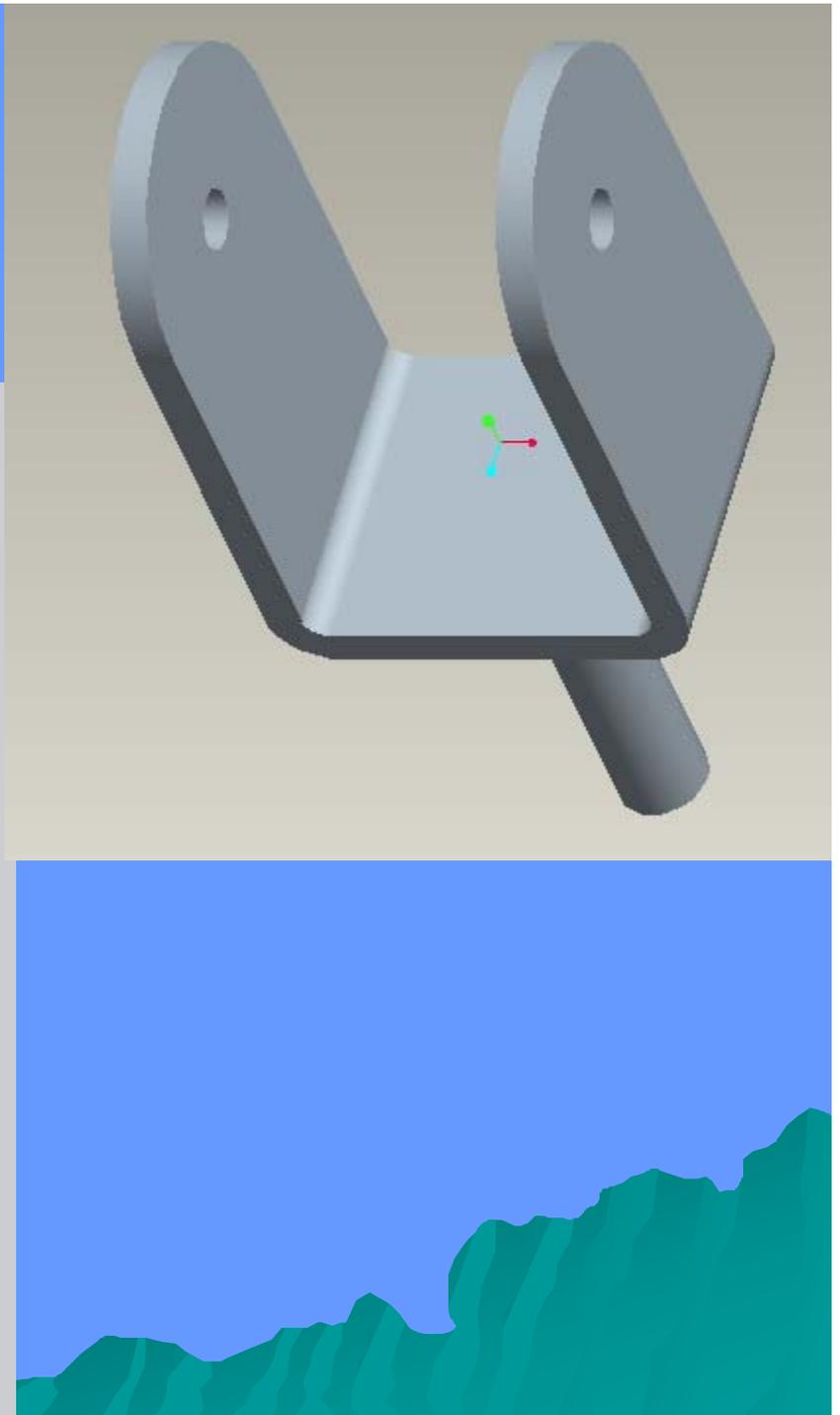
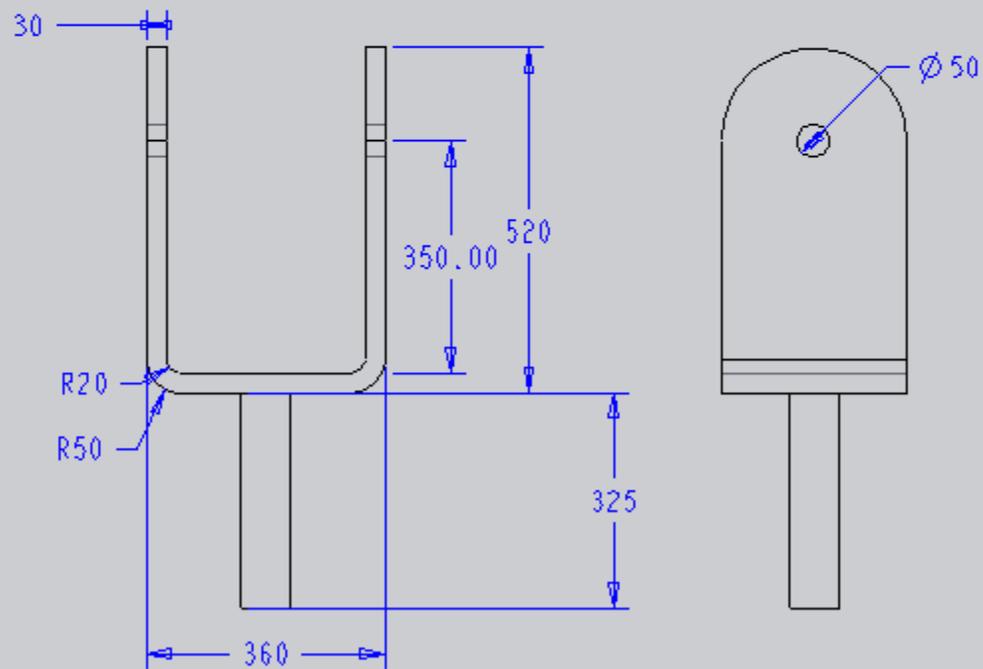
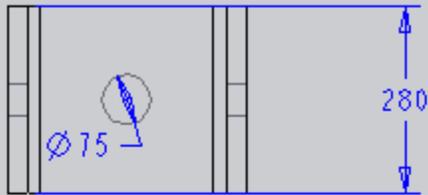
Hexagonalbolt_Assembly



Lockingnut_Assembly



U_Support_Assembly



Washer_Assembly

