CAD/CAM

LABORATORY MANUAL

B.TECH
(IV YEAR – I SEM)
(2019-20)

Department of Mechanical Engineering

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(Autonomous Institution – UGC, Govt. of India)

Recognized under 2(f) and 12 (B) of UGC ACT 1956
Affiliated to JNTUH, Hyderabad, Approved by AICTE - Accredited by NBA & NAAC – 'A' Grade - ISO 9001:2015 Certified)
Maisammaguda, Dhulapally (Post Via. Kompally), Secunderabad – 500100, Telangana State, India

(R15A0390) COMPUTER AIDED DESIGN AND COMPUTER AIDED MANUFACTURING LAB

OBJECTIVES

The students will learn to:

- Explain basic concepts of CIM systems
- Develop machining programs for CNC equipment
- Develop PLC-based control systems for manufacturing cells

LIST OF EXPERIMENTS:

1) Determination of deflection and stresses in 2D and 3D trusses and beams.
2) Determination of deflections component and principal and Von-mises stresses in . plane stress, plane strain and Axisymmetric components.
3) Determination of stresses in 3D and shell structures (at least one example in each .case)
4) Estimation of natural frequencies and mode shapes, Harmonic response of 2D beam.
5) Steady state heat transfer Analysis of plane and Axisymmetric components.
6) Development of process sheets for various components based on tooling Machines.
7) Development of manufacturing and tool management systems.
8) Study of various post processors used in NC Machines.
9) Development of NC code for free form and sculptured surfaces using CAM packages.
10) Machining of simple components on NC lathe by transferring NC Code from a CAM package.

Any Three Software Packages from the following:

Use of Auto CAD, Micro Station, CATIA, Pro-E, I-DEAS, ANSYS, NISA, CAEFEM,Gibbs CAM,Master CAM etc.

Open source Softwares :

MAT LAB, NASTRON, HYPERMESH, PATRAN

OUTCOMES:

This course primarily contributes to Mechanical Engineering program outcomes:

- An ability to apply knowledge of mathematics, science, and engineering
- An ability to design a system, component, or process to meet desired needs within realistic constraints.

LIST OF EXPERIMENTS

DRAFTING:

1. Development of part drawings for various components in the form of orthographic and isometric.

PART MODELING:

1. Generation of various 3D Models through pad, shaft, shell sweep.
2. Feature based and Boolean based modelling surface and Assembly Modelling. Design simple components.

ANSYS:

1. Determination of deflection and stresses in 2D and 3D trusses and beams.
2. Determination of deflections component and principal and VonMises stress, plane strain and Axi-symmetric components.
3. Determination of stresses in 3D and shell structures.
4. Estimation of natural frequencies and mode shapes Harmonic response of 2D beam.
5. Steady state heat transfer Analysis of plane and Axi-symmetric components.

CAM :

1. To manufacture the given model using facing operation.
2. To manufacture the given model using parting-off operation.
3. To manufacture the given model using step turning operation.
4. To manufacture the given model using thread cutting operation.

INSTRUCTIONS TO STUDENTS

1. Students are required to remove their footwear outside the centre and keep it in the box provided for the same.
2. Students should leave their belongings outside the lab except their observation note book, the concerned books/manuals and calculators.
3. Students are requested not to place their legs on the wall or on the table.
4. Students should refrain from leaning on the table and sitting on it.
5. Before logging in to a particular terminal, if there is something wrong in the terminal, the student should report the same immediately to the concerned staff.
6. Students should not use any disks brought from outside without prior permission from the concerned staff.
7. Students can get the required manual or disks from the staff after signing in the appropriate register
8. Students should collect their printouts before leaving the lab for that particular session.
9. Before leaving the Terminal, the students should logout properly and leave their chairs in position.
10. Students are not allowed to take any manual outside the centre.
11. Edibles are strictly prohibited in the centre.
12. No internet browsing allowed during the lab hours.

******

IMPORTANT INSTRUCTIONS TO HANDLE CNC MACHINES

1. Get permission from the concerned staff before switch ON the CNC machines.
2. Ensure the proper power supply for the system and machine.
3. Handle the CNC machines very carefully.
4. If any problem occurs in the system or machine immediately inform to the concerned staff.
5. Your batch is responsible for the CNC machine and its system while doing the lab exercise assigned to your batch.

What is CAD/CAM?

CAD/CAM (computer-aided design and computer-aided manufacturing) refers to computer software that is used to both design and manufacture products.

CAD is the use of computer technology for design and design documentation. CAD/CAM applications are used to both design a product and program manufacturing processes, specifically, CNC machining. CAM software uses the models and assemblies created in CAD software to generate tool paths that drive the machines that turn the designs into physical parts. CAD/CAM software is most often used for machining of prototypes and finished parts.

Why CAD/CAM?

Computer Aided Design and Computer Aided Manufacture is the way things are made these days. Without this technology we wouldn't have the range and quality of products available or, at least, they wouldn't be available at a price most of us can afford. Hand-building and manual techniques still very much have their place and Design Education needs to treasure and foster these skills so that future generations will have the hands-on‟ skills to understand the man-made world and provide the next generation of engineers, designers and technicians. All of these professionals will be using CAD/CAM techniques or CAD/CAM products in their work, alongside practical hands-on skill. Design and Technology education has to reflect modern practice so it is crucial that students have the opportunity to use real CAD/CAM tools in their designing and making.

DESIGN PROCESS AND ROLE OF CAD:

1. Recognition of need
2. Definition of problem
3. Synthesis
4. Analysis and optimization
5. Evaluation
6. Presentation

SOLID MODELLING

A solid modeling system is usually an interactive computer graphics system that is intended to create true three-dimensional components and assemblies. Recent advances in CAD software, computers, and graphical displays have made it possible to use solid representations of components being considered in the design process. These solid models can be employed in numerous ways.

Advantages of Solid Modeling:

A realistic visual display: By producing a shaded visible surface image of the solid, solid modelling allows a designer to see exactly what has been created.

Easy to deal with different views: Once a part has been created, we have the ability to rotate, shade, section, or produce almost any view required by a designer.

Single associated model database: The solid modeler provides the only database suitable for all CAD operations. Almost all information needed for part generation is contained in the solid model. The algorithm should be able to ensure that it represents physically possible shape that is complete and unambiguous applications. e.g., automatic generation of a mesh for a finite element analysis.

REQUIREMENTS FOR MODELING ASSEMBLING:

1. **Part modelling and analysis:**

   The part analysis includes the material type, mass and inertial properties, functional properties of the faces, etc.

2. **Hierarchical relationships:**

   An assemble tree and assemble sequence must be given.

3. **Mating conditions:**

   There are two methods for specifying mating conditions:

Specify the location and orientation of each part in the assembly, together with the representation of the part itself, by providing a 4 x 4 homogeneous transformation matrix. (i.e., transformation from MCS to WCS).

Specify the spatial relationships between its individual parts as mating conditions.

CAD/CAE/CAM Data Exchange

Computer databases are now replacing paper blueprints in defining product geometry and non-geometry for all phases of product design, analysis, and manufacturing. It becomes increasingly important to find effective procedures for transferring data among CAD/CAE/CAM systems.

The need to exchange modeling data is directly motivated by the need to integrate and automate the design and manufacturing process to obtain the maximum benefits from CAD/CAE/CAM systems.

Four Types of Modelling Data to be transferred:

(1)Shape

(2)Non-Shape

(3)Design

(4)Manufacturing

1. **Shape** data consists of both geometrical and topological information as well as part features. Entity attributes such as font, colour, and layer as well as annotation is considered part of the entity geometrical information. Topological information applies only to products described via solid modelling. Features allow high-level concept communication about parts.

Examples are hole, flange, web, pocket, chamfer, etc.

2. **Non-Shape** data includes graphics data such as shaded images, and model global data as measuring units of the database and the resolution of storing the database numerical values.

3. **Design** data has to do with the information that designers generate from geometric models for analysis purposes. e.g., mass property and finite element mesh data.

4. **Manufacturing** data consists of information such as tooling, NC tool paths, tolerance, process planning, tool design, and bill of materials.

Commonly Used CAD Data Exchange Format:

IGES (Initial Graphics Exchange Specification)

PDES (Product Data Exchange Using STEP)

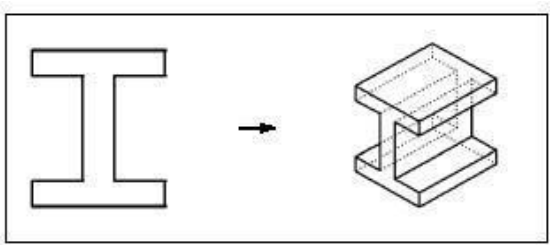
IGES is focused on CAD-to-CAD exchange where primarily shape and non-shape data were to be transferred from one system to another.

PDES is previous called Product Data Exchange Standard. It is for the exchange of complete product descriptions which covers the four types of modelling data (i.e., shape, non-shape, design and manufacturing).

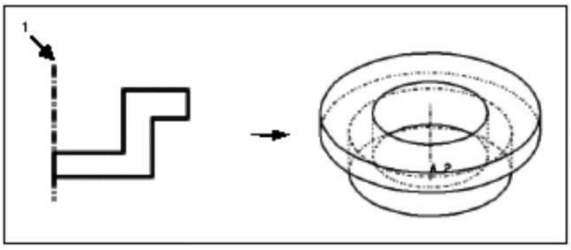
Other data exchange interfaces include: STL, Neutral, SET, ECAD, VDA, STEP, PDGS, CATIA, Render, CGM, VRML, PATRAN, TIFF, etc.

Base Features in Part Modelling:

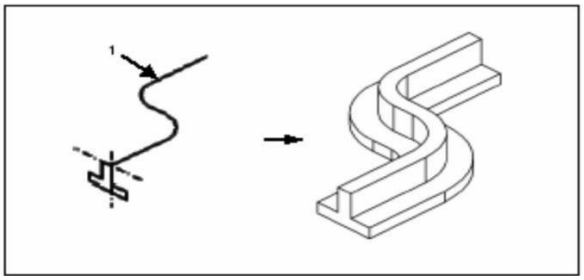
Pad: Pad in third axis of the profile



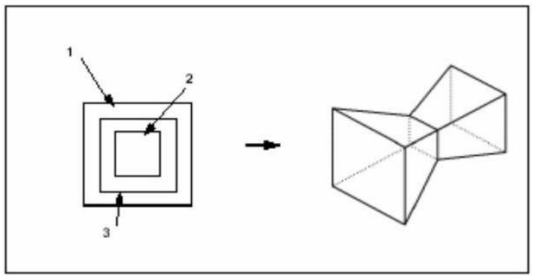
Shaft: Shaft the profile about axis of symmetry



Sweep: Pad of a cross section along a path



Blend / Loft: Blending of different cross sections along a path



Editing & Engineering Features in Part Modelling:

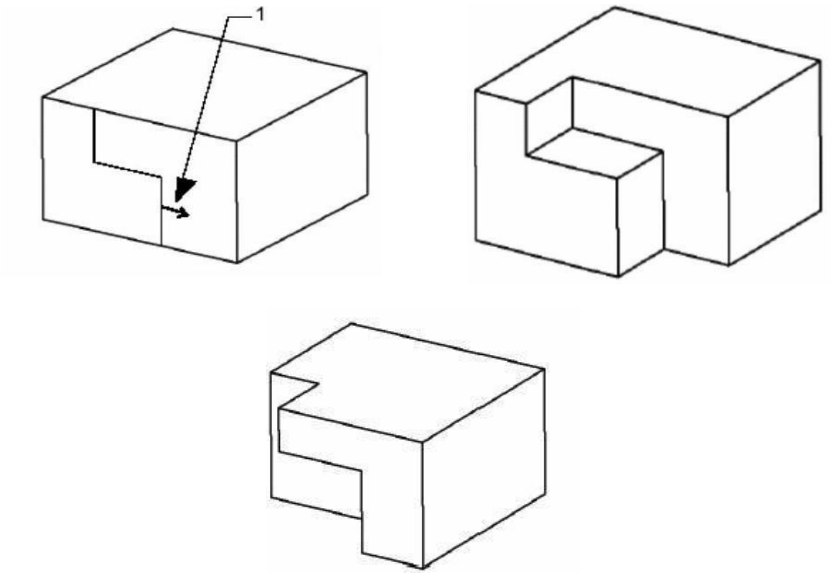
Round: Modify the sharp edge to curved edge

Chamfer: Modify the sharp edge to flat edge

Shell: Removes a surface or surfaces from the solid then hollows out the inside of the solid, leaving a shell of a specified wall thickness.

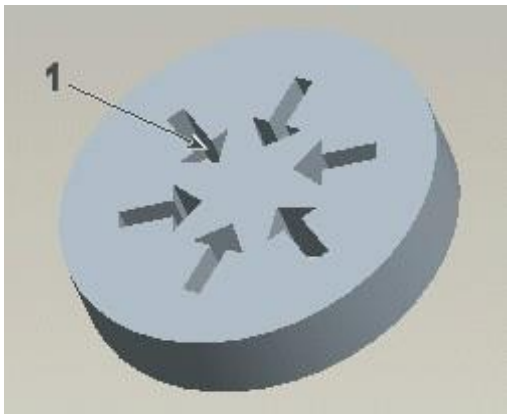
Rib: Special type of protrusion to create a thin fin or web

Cut: Remove the undesirable portion from the basic part



Hole: Remove cylindrical portion from the basic part

Pattern: Create instances of the selected feature by varying some specified dimensions



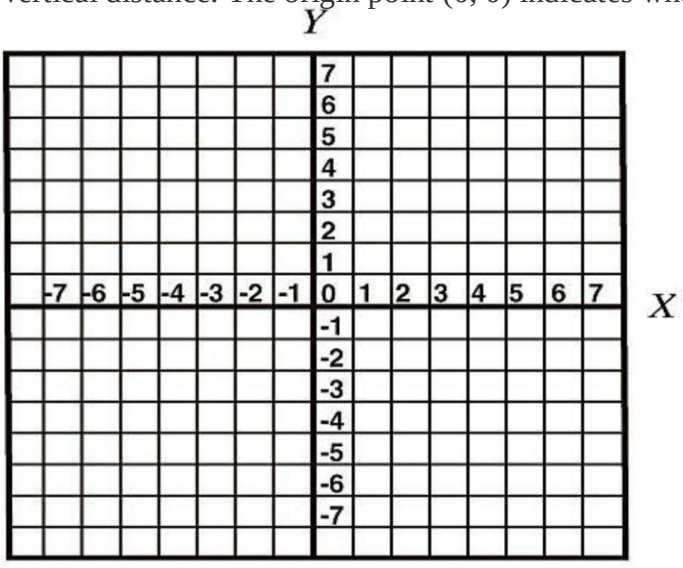
COORDINATES:

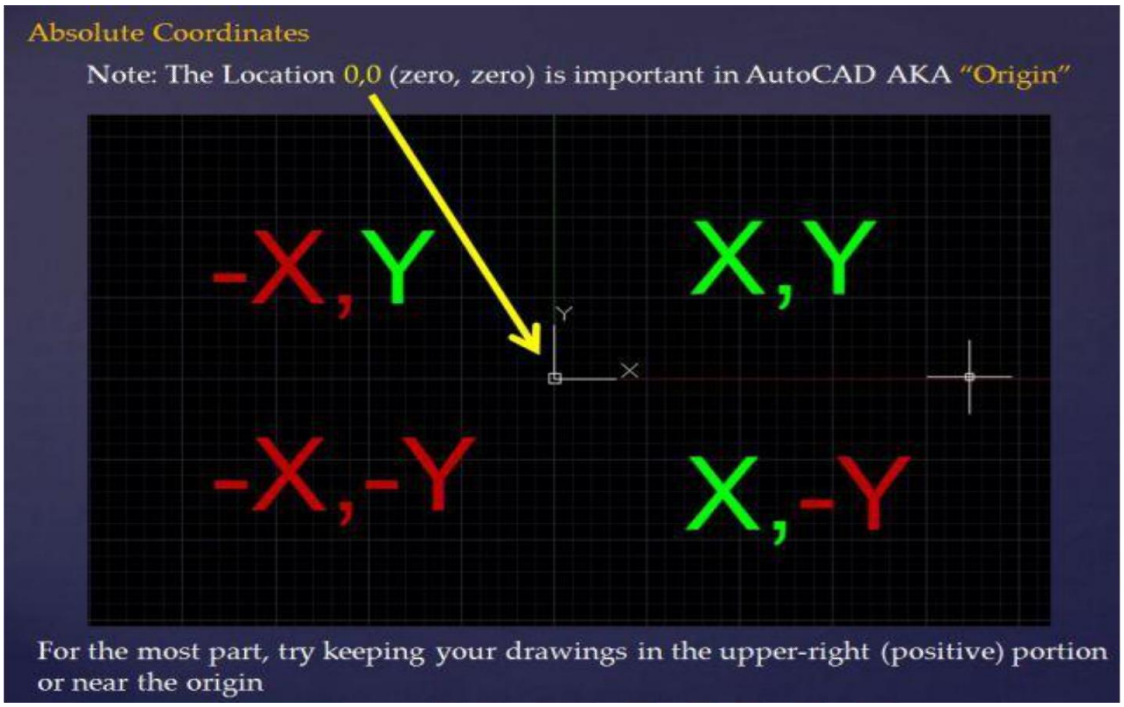
"When a command prompts you for a point, you can use the pointing device to specify a point, or you can enter a value on the command line. You can enter two-dimensional coordinates as either Cartesian (X, Y) or polar coordinates.

Cartesian and Polar Coordinate:

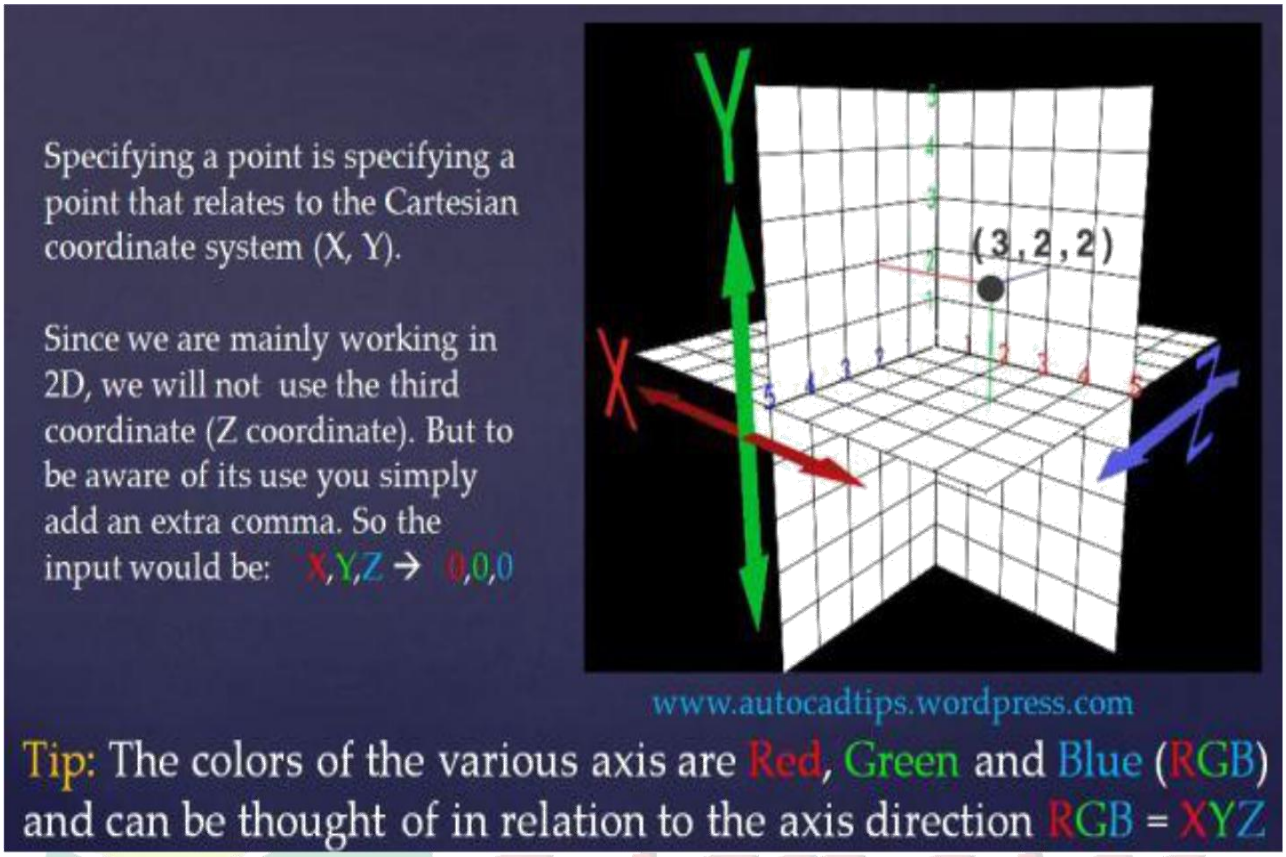
A Cartesian coordinate system has three axes X, Y, and Z. When you enter coordinate values, you indicate a point's distance (in units) and its direction (- or +) along the X, Y and Z axes relative to the coordinate system origin (0, 0, 0).

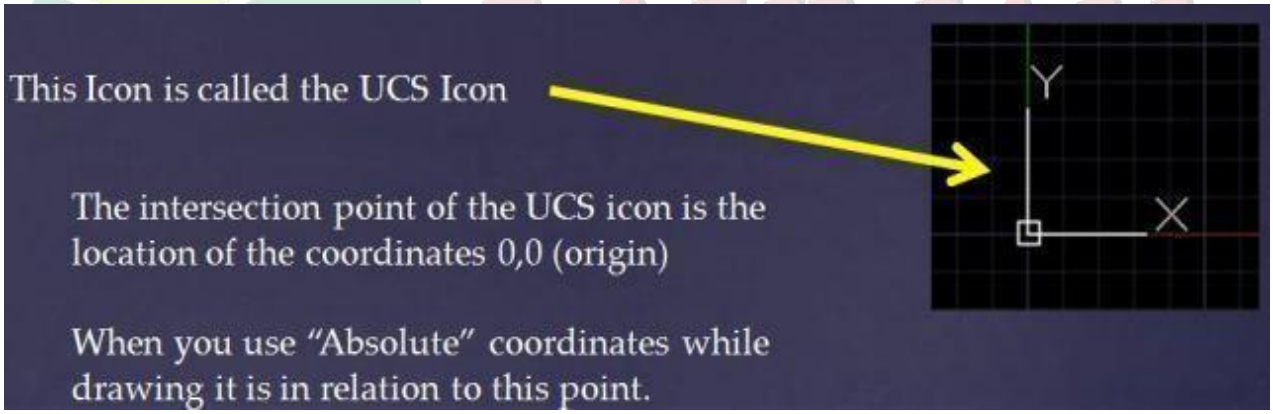
In 2D, you specify points on the XY plane, also called the construction plane. The construction plane is similar to a flat sheet of grid paper. The X value of the Cartesian coordinate specifies the horizontal distance, and the Y value specifies the vertical distance. The origin point (0, 0) indicates where the two axes intersect.





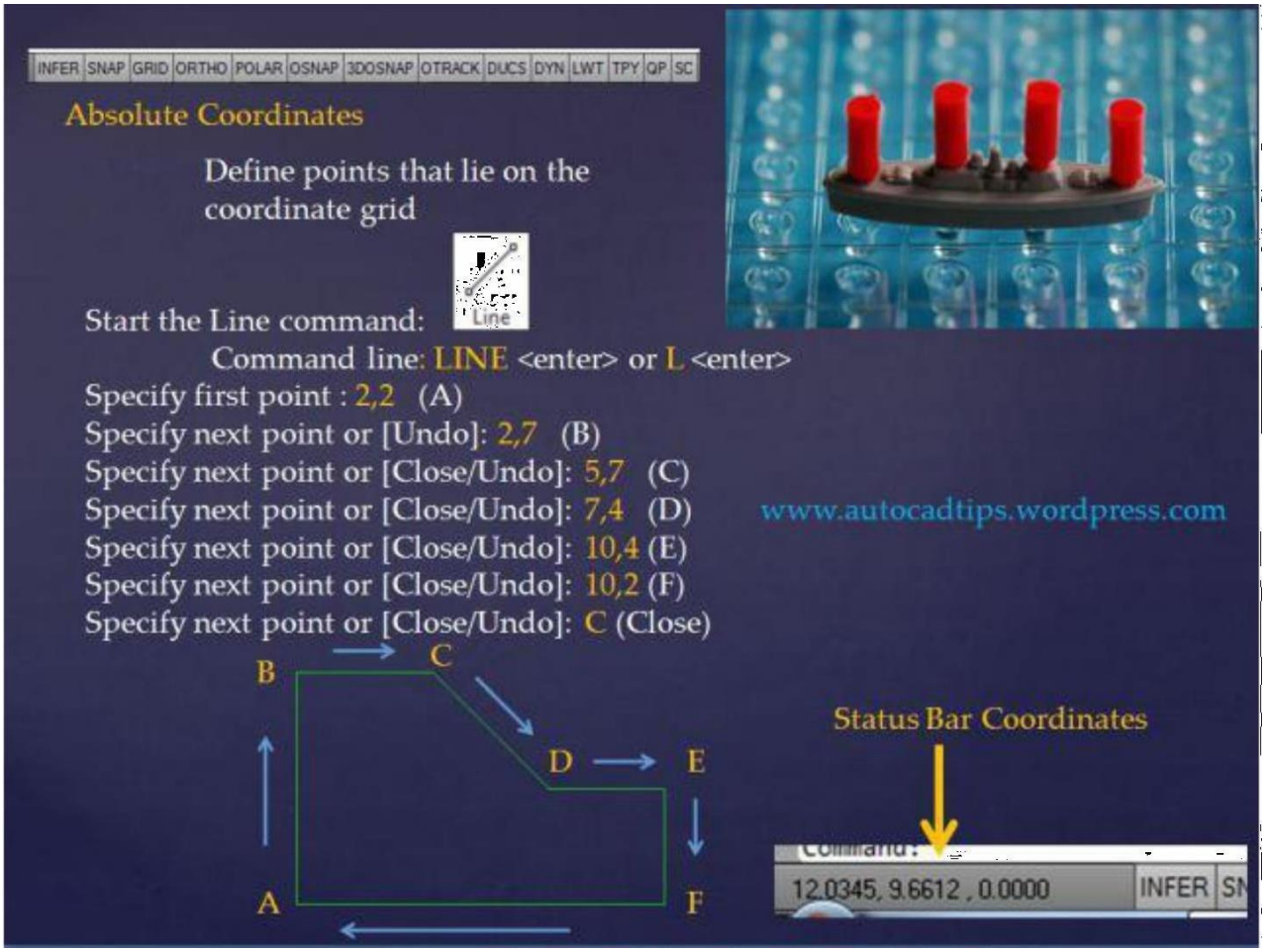
The origin is marked by an object known as the UCS icon UCS = User Coordinate System. This icon can be manipulated (moved and rotated) such that the origin point moves. But for the purposes of learning the basics of coordinates, this post will treat the UCS icon as a static object in a permanent position. Oddly enough, the UCS icon does have permanent position that it can always be re-positioned to. This position is called the "World coordinates."





Absolute Coordinates let you type in a specific X and Y location on the construction plane.

Use the picture below to work through the use of using absolute coordinates to create the shape that is shown. Note that the coordinates that are shown in the lower left of the screen (status bar) displays the coordinates of your cursor.

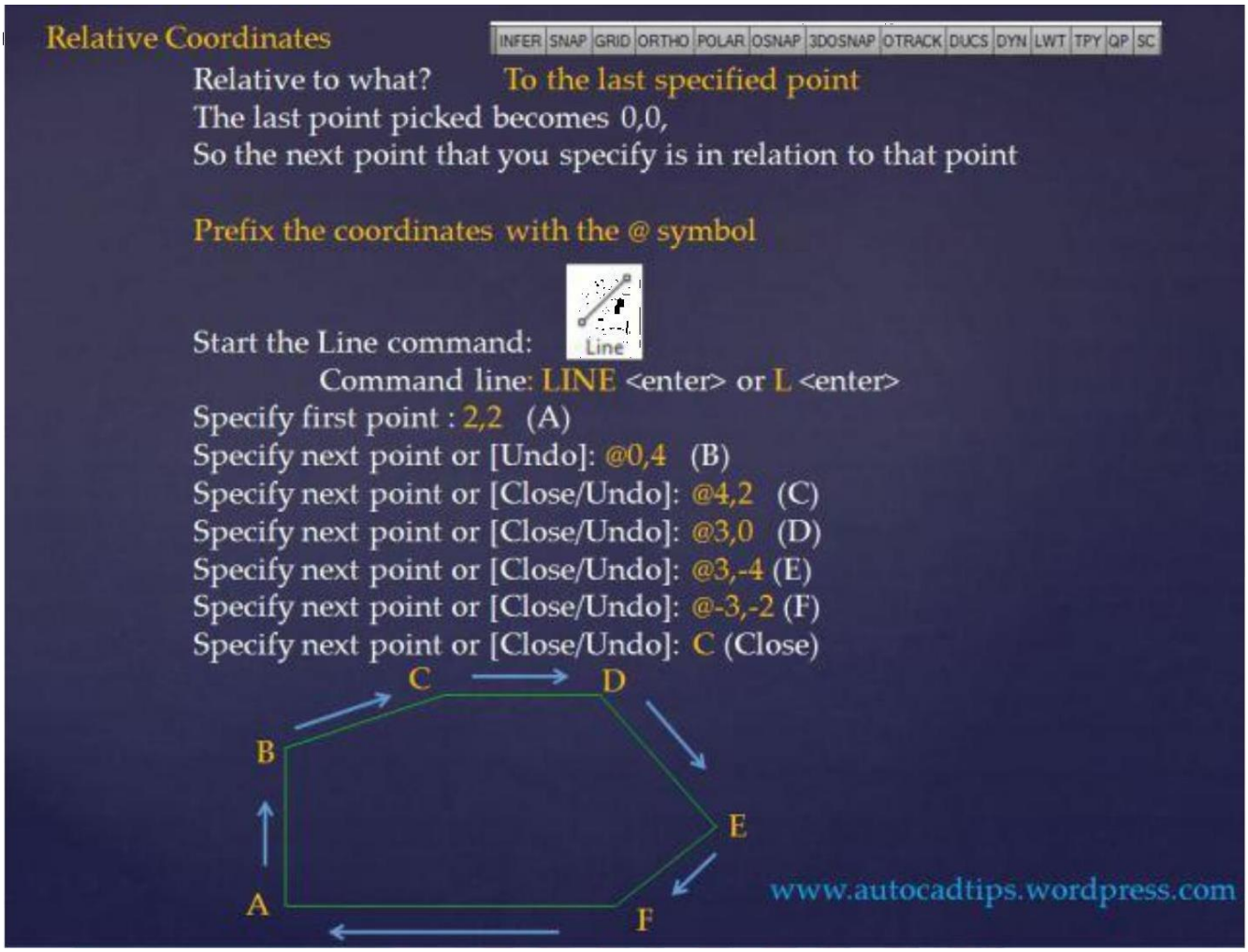


Note the all of the Drafting Setting toggles are turned off. this is because when DYN (dynamic input) is turned on it changes how absolute and relative coordinates are entered.

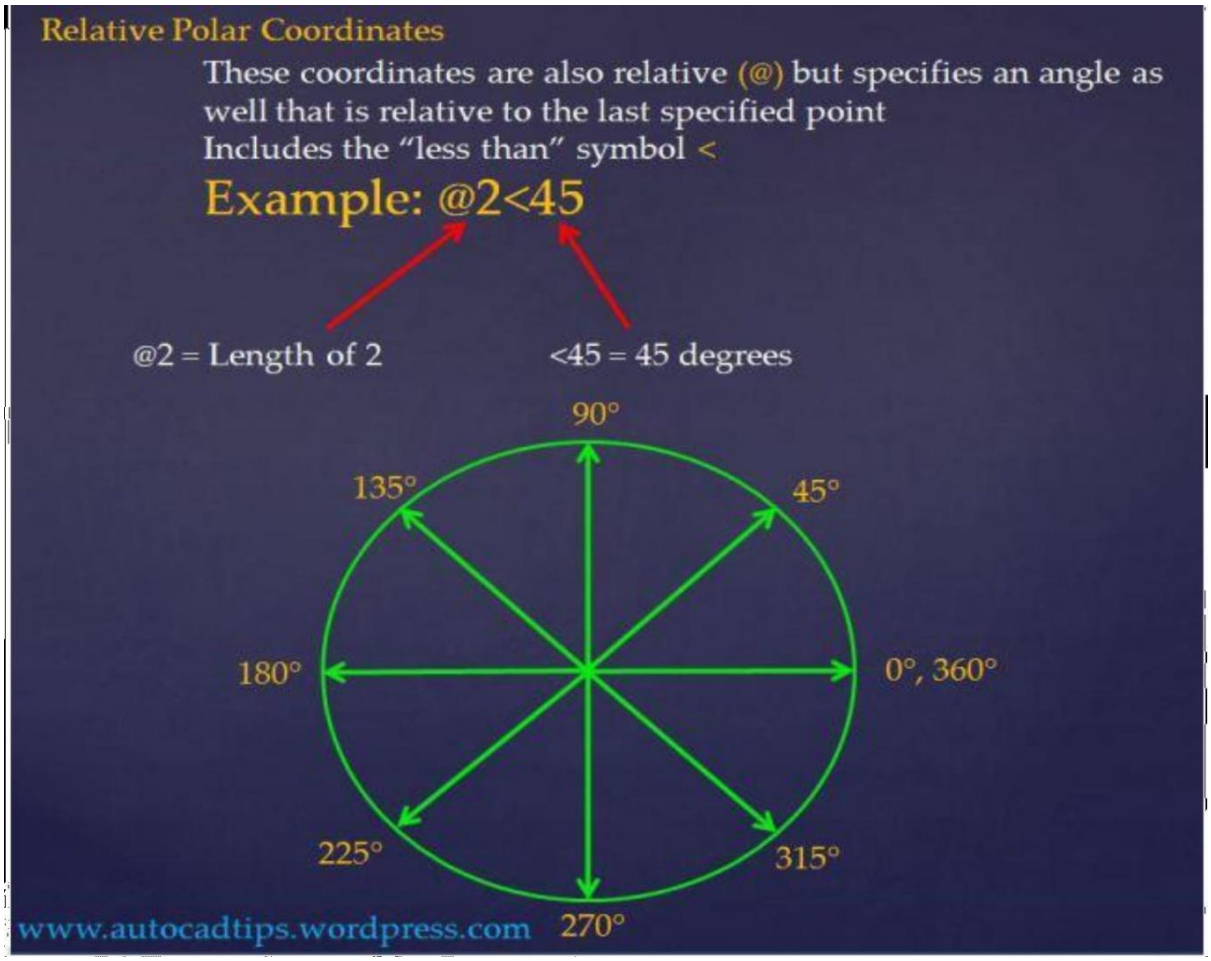
Relative Coordinate:

Relative Coordinate entry is simply specifying the next point as it relates to your previously defined point whether you picked a point with the cursor or you entered a specific coordinate. Think of this method as the last point you specified being equal to the coordinate 0, 0. The way that you tell AutoCAD that you want to use relative coordinates is that you prefix the coordinate with the @ symbol. Note that the first point that is specified is an absolute coordinate since there is not a specified previous point.

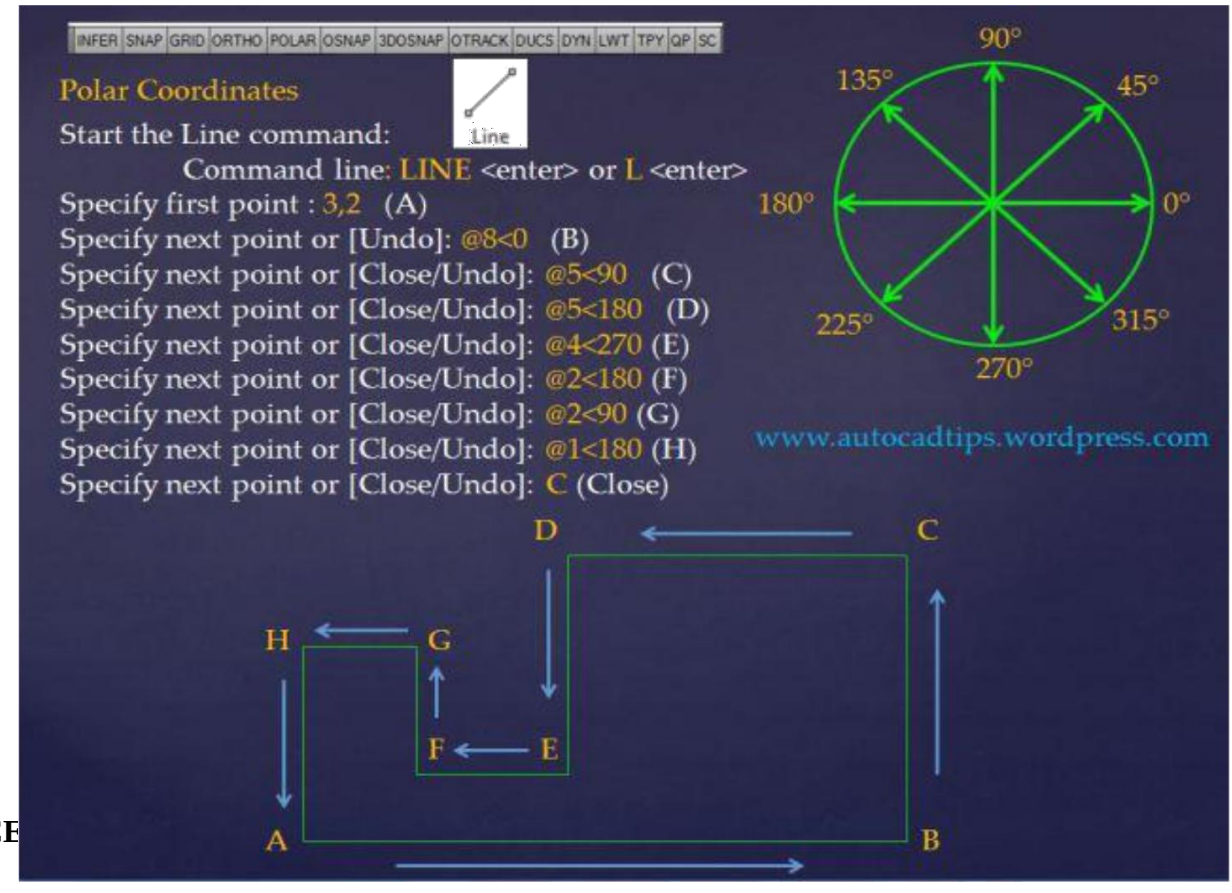
Note the all of the Drafting Setting toggles are turned off this is because when DYN (dynamic input) is turned on it changes how absolute and relative coordinates are entered.



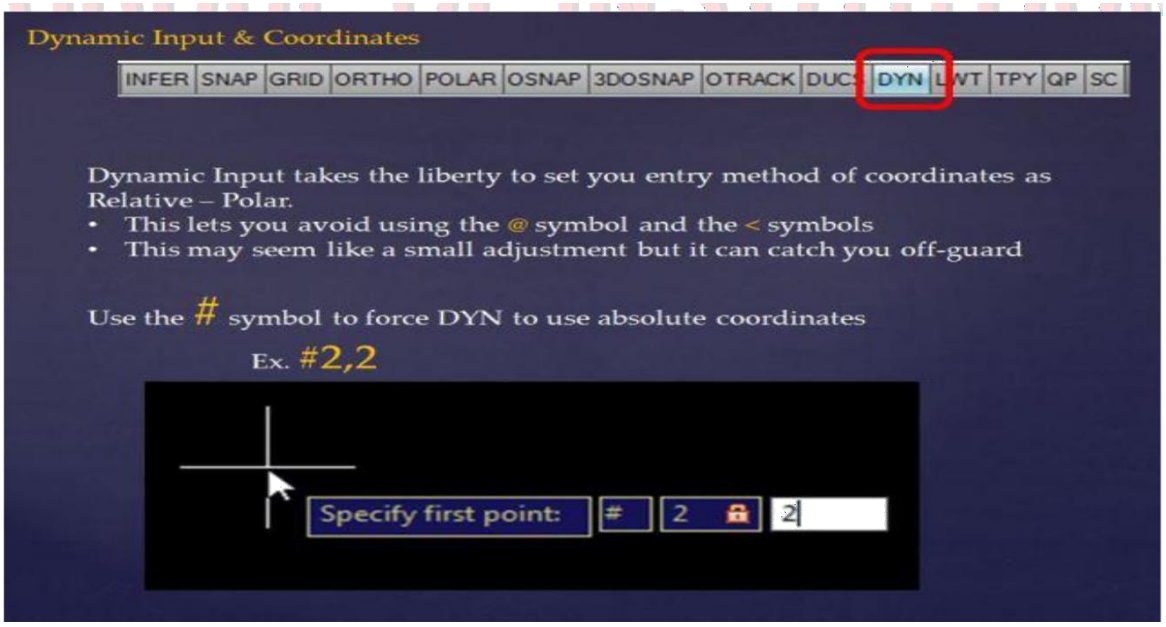
Relative Polar Coordinate entry is very useful but for some people it is easily forgotten. As shown above, Relative coordinates use the @ symbol when specifying a point. As shown in the picture below, relative polar uses the # symbol to specify the LENGTH of the object followed by the < symbol to specify the angle.



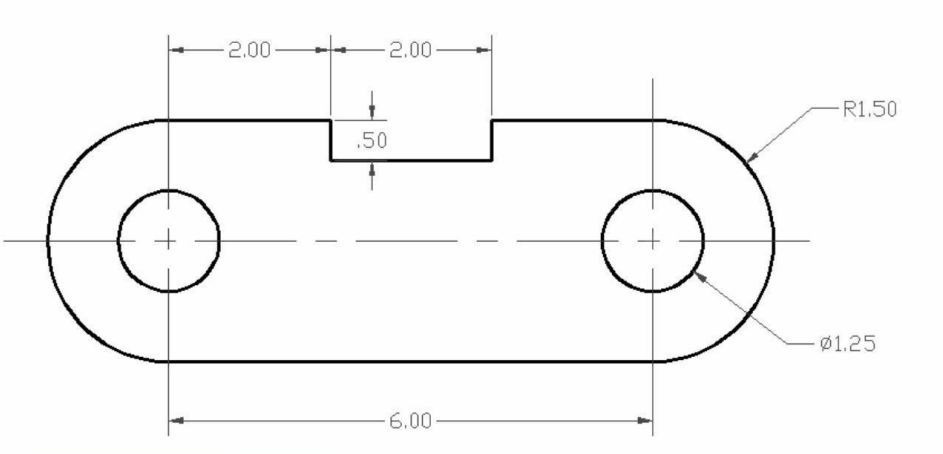
Use the below picture to create the shape that is shown using Relative Polar Coordinate entry. Turn off all toggles in the drafting settings.



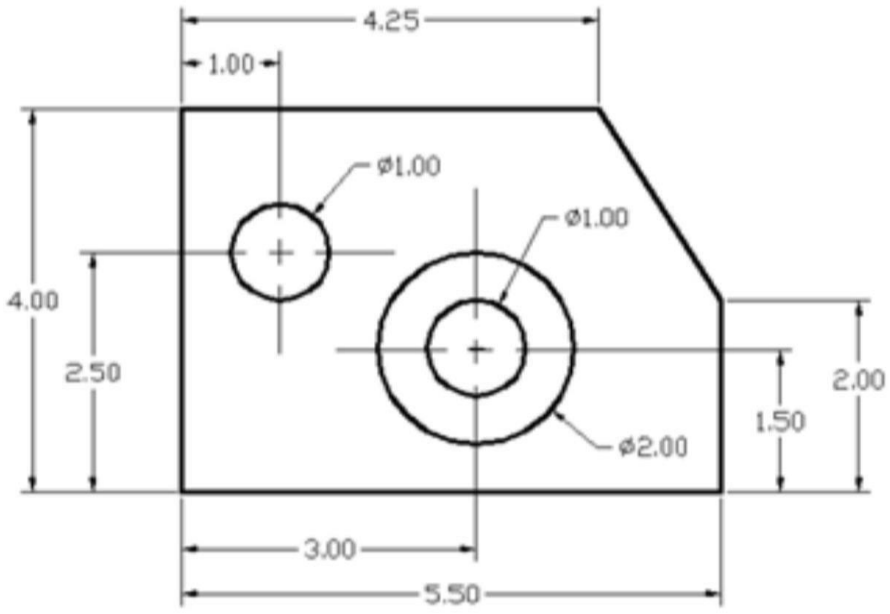
As mentioned before each exercise, Dynamic Input when enabled turns your entry method to an automatic Relative Polar method. This is confusing because when DYN is turned on and say that you want to specify the coordinate of 0, 0 you will not see anything happen. So in order to force Dynamic input to use Absolute coordinates use the # sign before the coordinates.



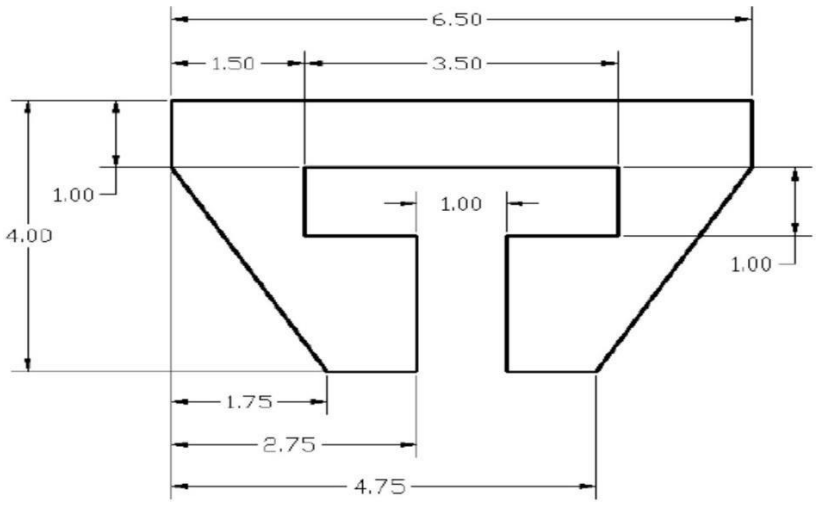
To draw the given diagram by Using AUTOCAD/CATIA



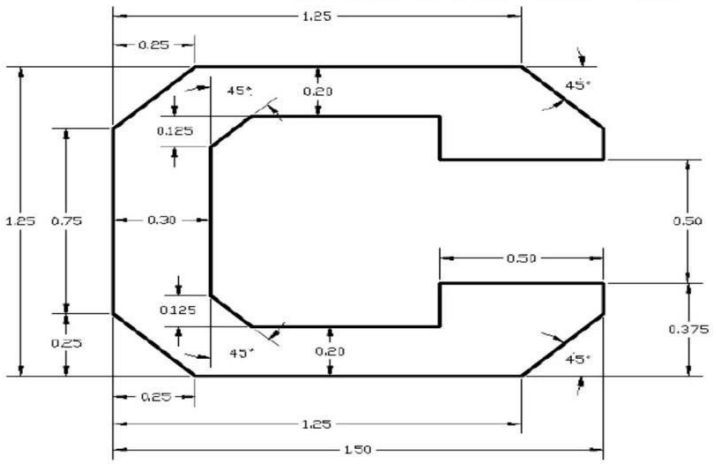
To draw the given diagram by Using AUTOCAD/CATIA



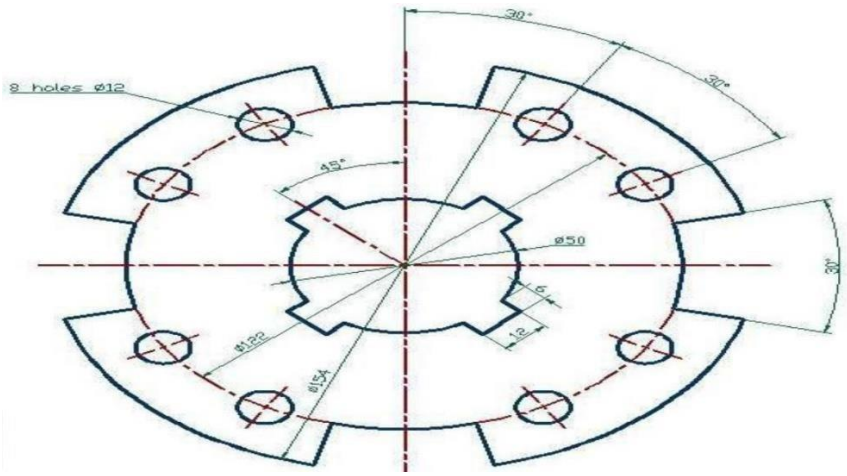
To draw the given diagram by Using AUTOCAD/CATIA



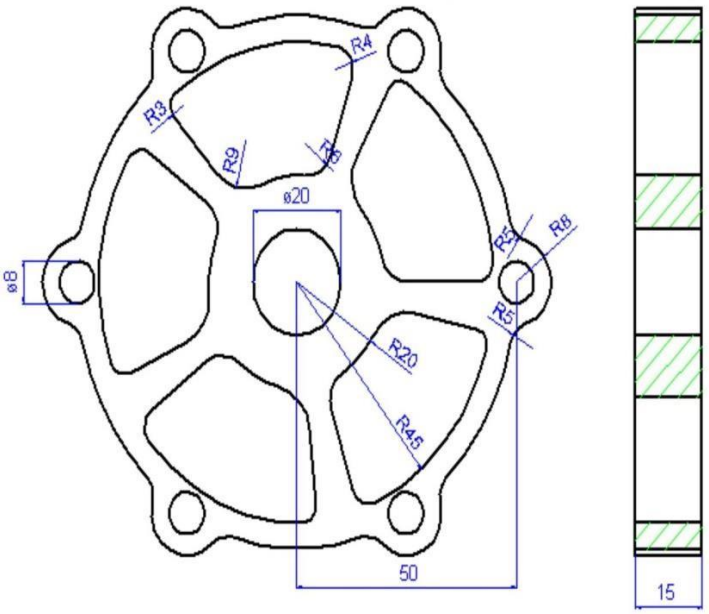
To draw the given diagram by Using AUTOCAD/CATIA



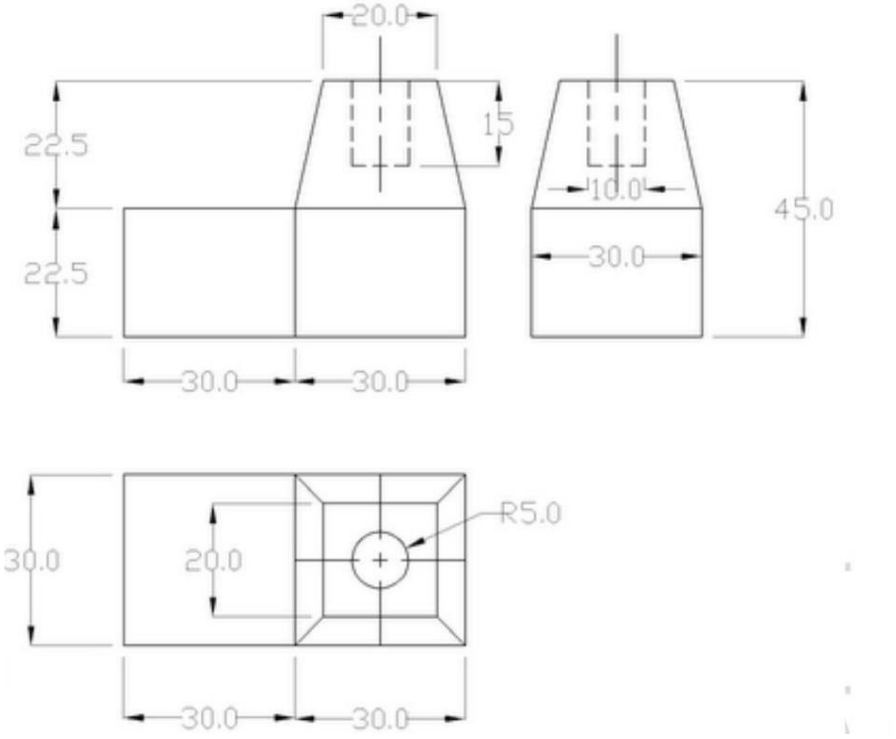
To draw the given diagram by Using AUTOCAD/CATIA



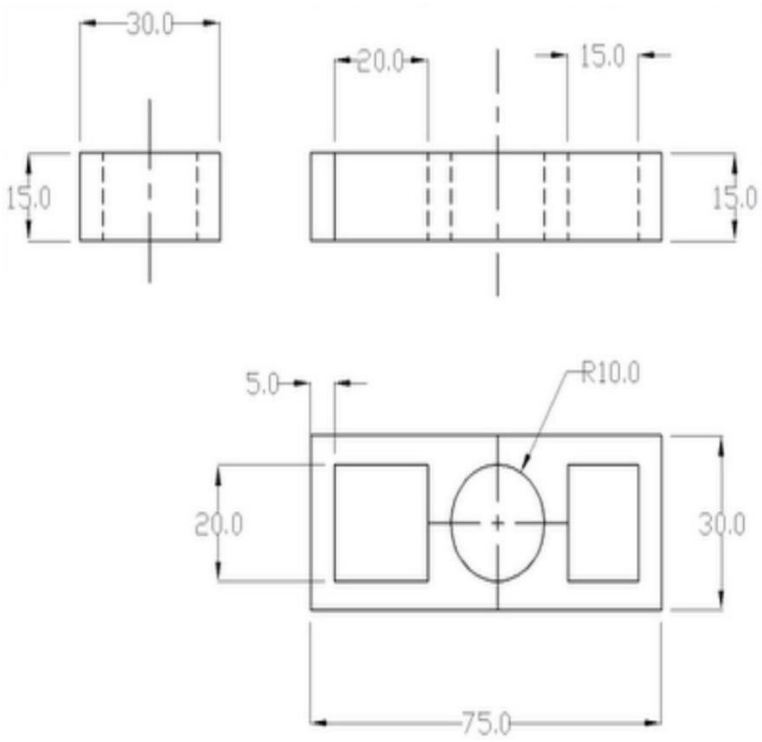
To draw the given diagram by Using AUTOCAD/CATIA



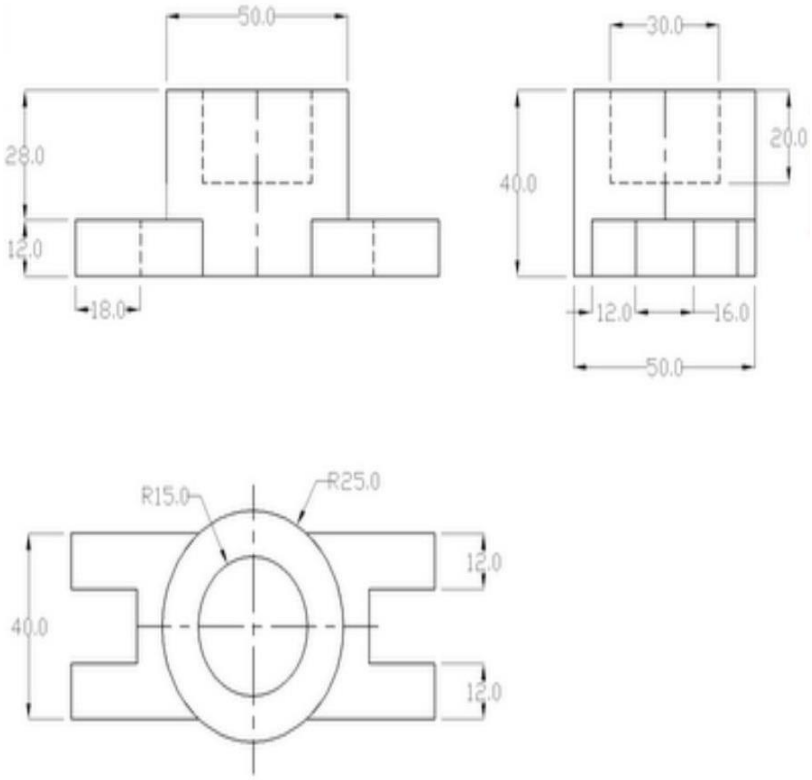
To draw the given diagram by Using AUTOCAD/CATIA



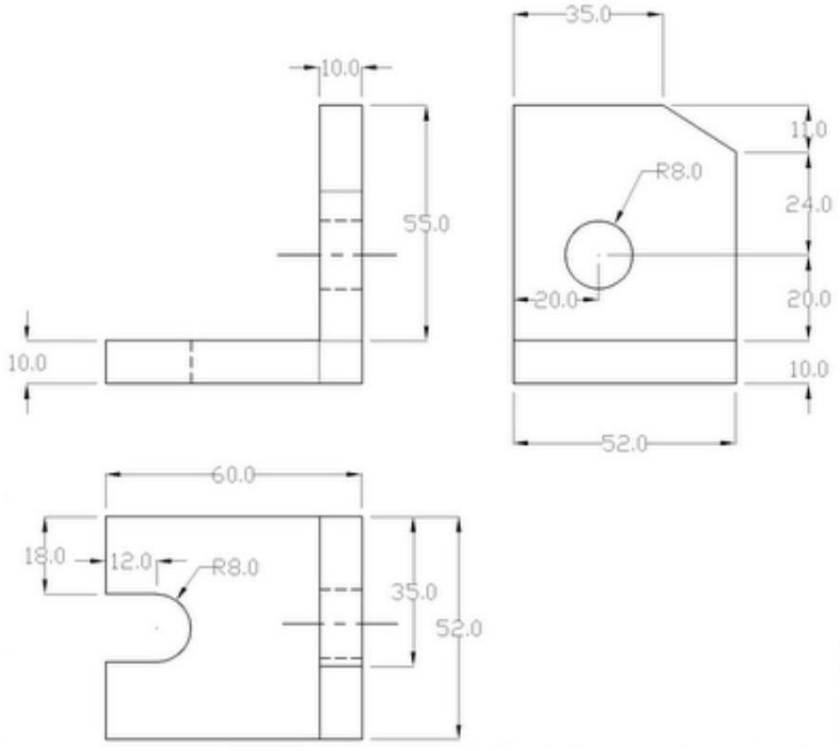
To draw the given diagram by Using AUTOCAD/CATIA



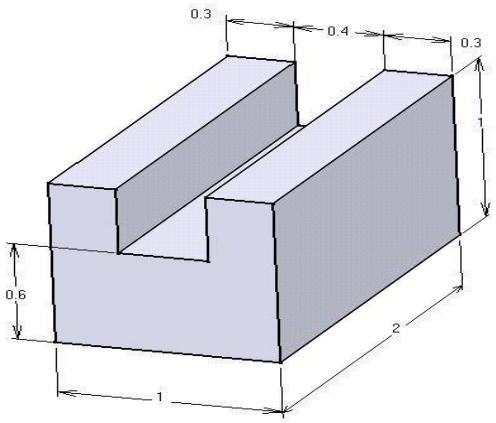
To draw the given diagram by Using AUTOCAD/CATIA



To draw the given diagram by Using AUTOCAD/CATIA



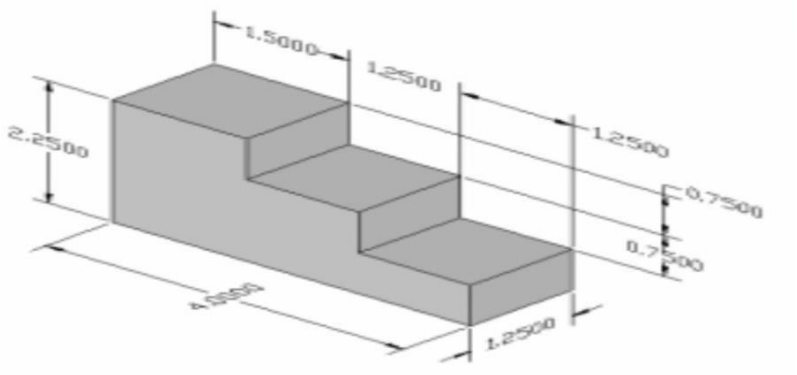
To create a given model by Using the AUTOCAD/CATIA/CATIA



Suggested Steps:

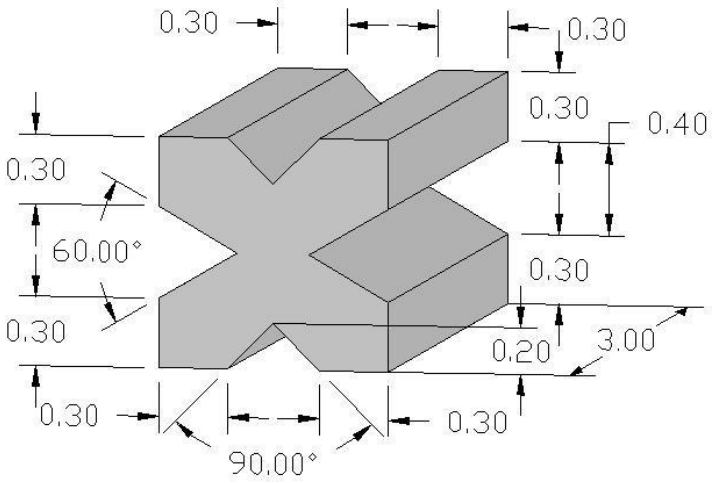
1. Select the XY plane (the plane the profile will be sketched on).
   Reference Step 3 for information on selecting planes.
2. Enter the **Sketcher Work Bench**. Reference Step 4.
3. Sketch the profile of the part.
   Hint: use the **Profile** tool.
4. Anchor the lower left hand corner of the sketch.
5. Constrain the profile to match the dimensions shown above.
   Reference Step 18 for constraining a profile.
6. Exit the **Sketcher Work Bench**, return to the **Part Design Work Bench** (the 3D environment).
   Reference Step 21 for exiting the **Sketcher Work Bench** and entering the **Part Design Work Bench**.
7. Once in the **Part Design Work Bench** pad the profile to the dimension has shown (2").
   Reference Step 22 for padding a profile.
8. Save the part.

To create a given model by Using the AUTOCAD/CATIA



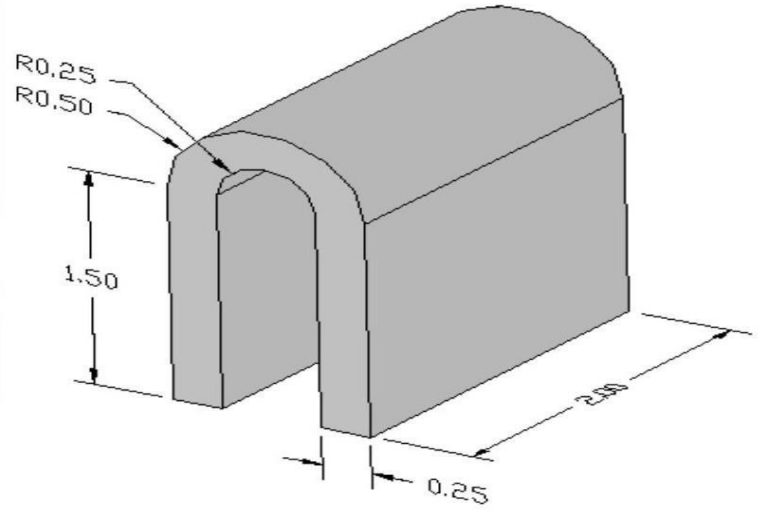
Hint: To help make it easier to sketch this part set the grid **Primary Spacing** to 1and the **Graduations** to 4. This will put the grid lines in the **Sketcher** screen to.25 inch spacing. With that spacing all you have to do is snap to the intersections of the grid to sketch the part.

To create a given model by Using the AUTOCAD/CATIA



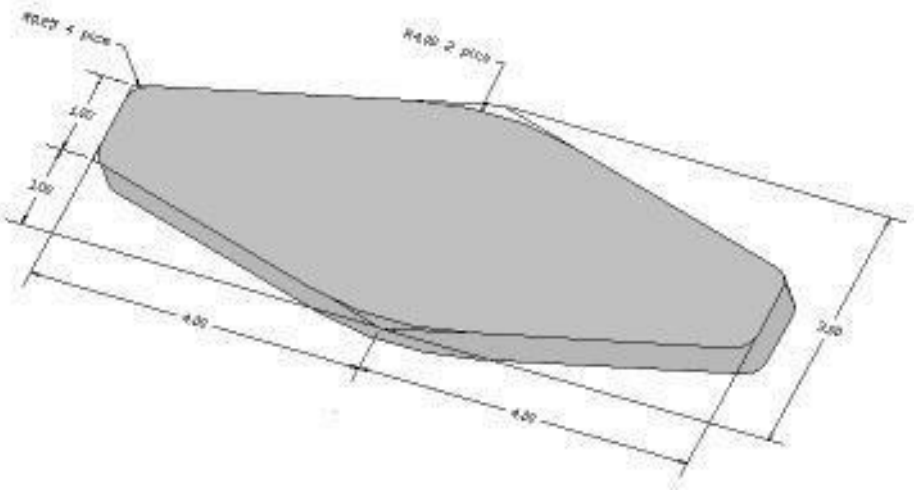
Hint: It is not as complicated as it looks. If your grids Graduations are set to 10 just snap to the intersections for the beginning and ending points of your lines. To set the constraint for the angles select the angled lines and the angle constraint will appear. Reference Step 19 for modifying the angle value.

To create a given model by Using the AUTOCAD/CATIA



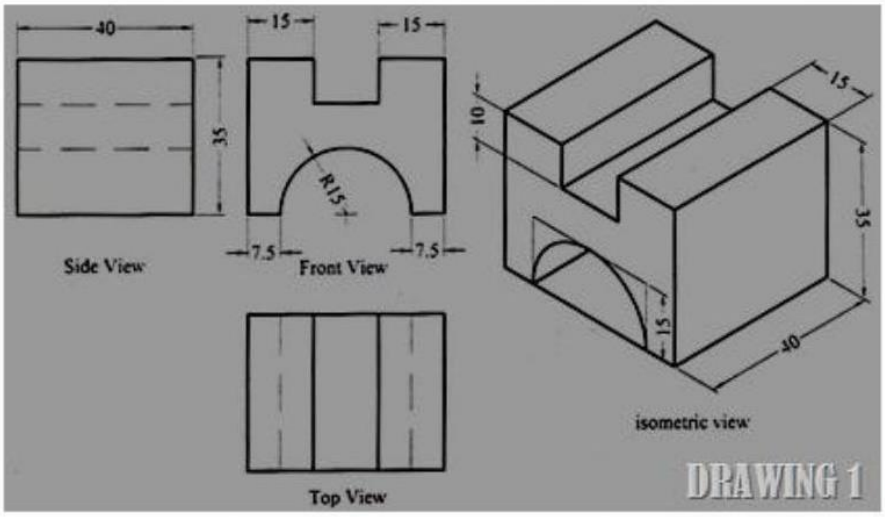
Hint: This part can be done using the radius option in the profile command.

To create a given model by Using the AUTOCAD/CATIA

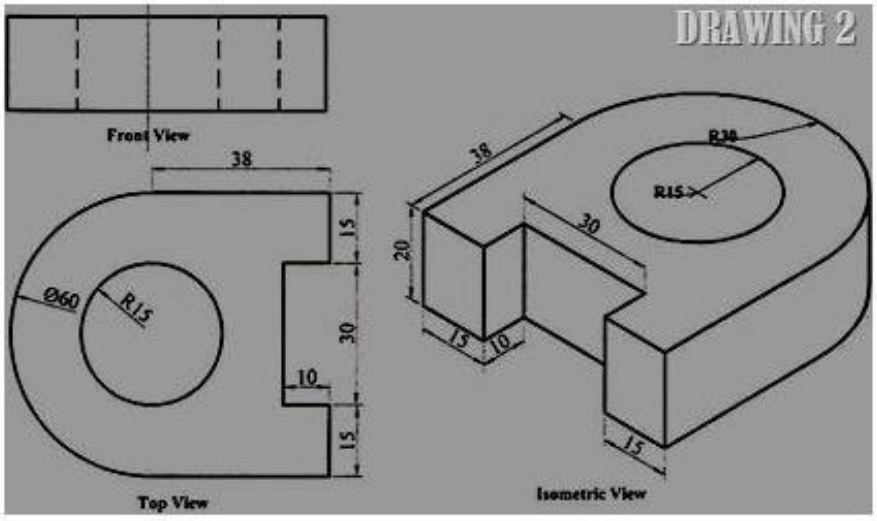


Hint: Use the **Line** or **Profile** icon first to sketch the profile using sharp corners (no radius). Once it is constrained to the dimensions above, go back and add in the radiuses using the **Corner** icon.

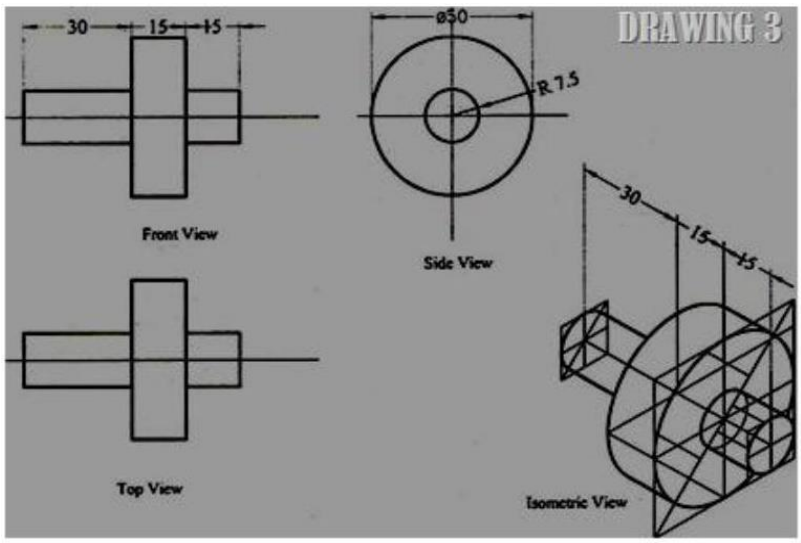
To create a given model by Using the AUTOCAD/CATIA



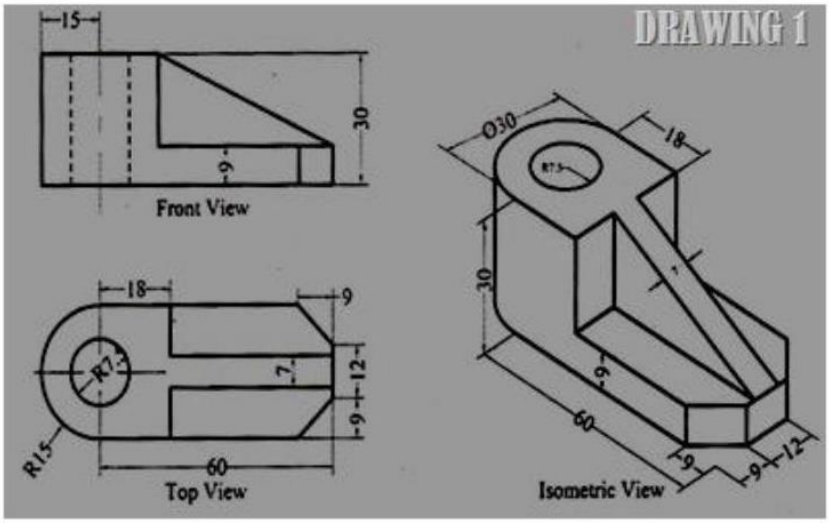
To create a given model by Using the AUTOCAD/CATIA



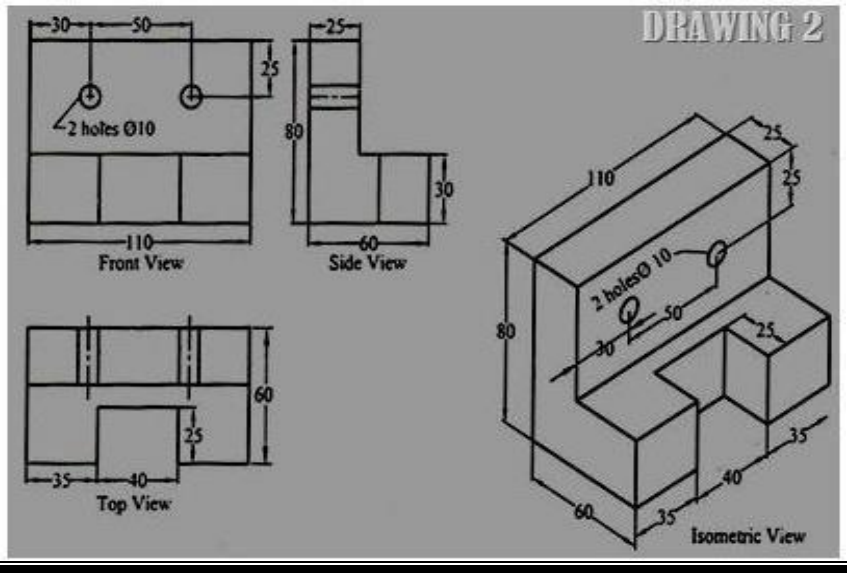
To create a given model by Using the AUTOCAD/CATIA



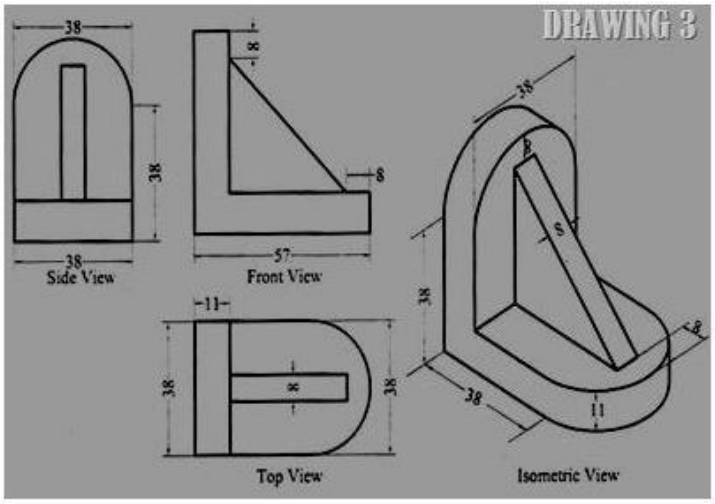
To create a given model by Using the AUTOCAD/CATIA



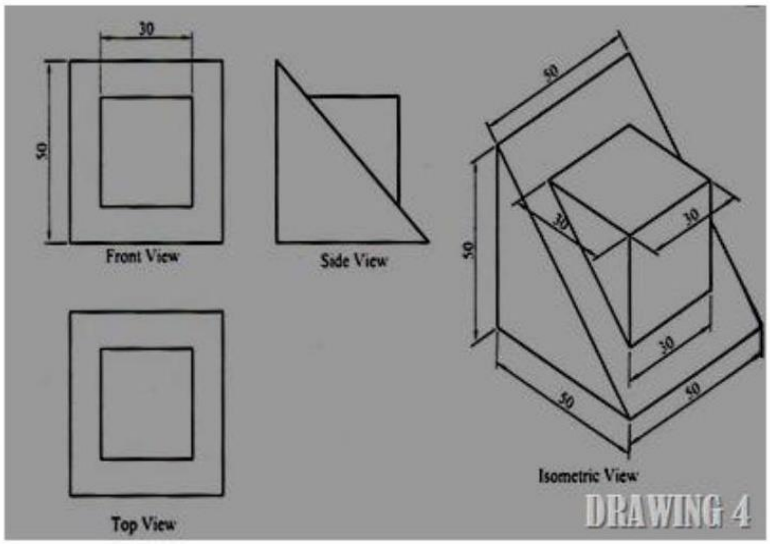
To create a given model by Using the AUTOCAD/CATIA



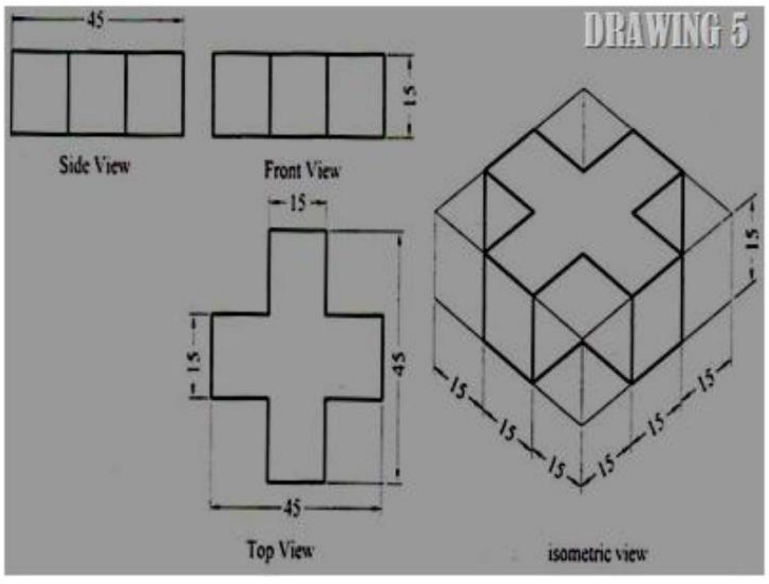
To create a given model by Using the AUTOCAD/CATIA



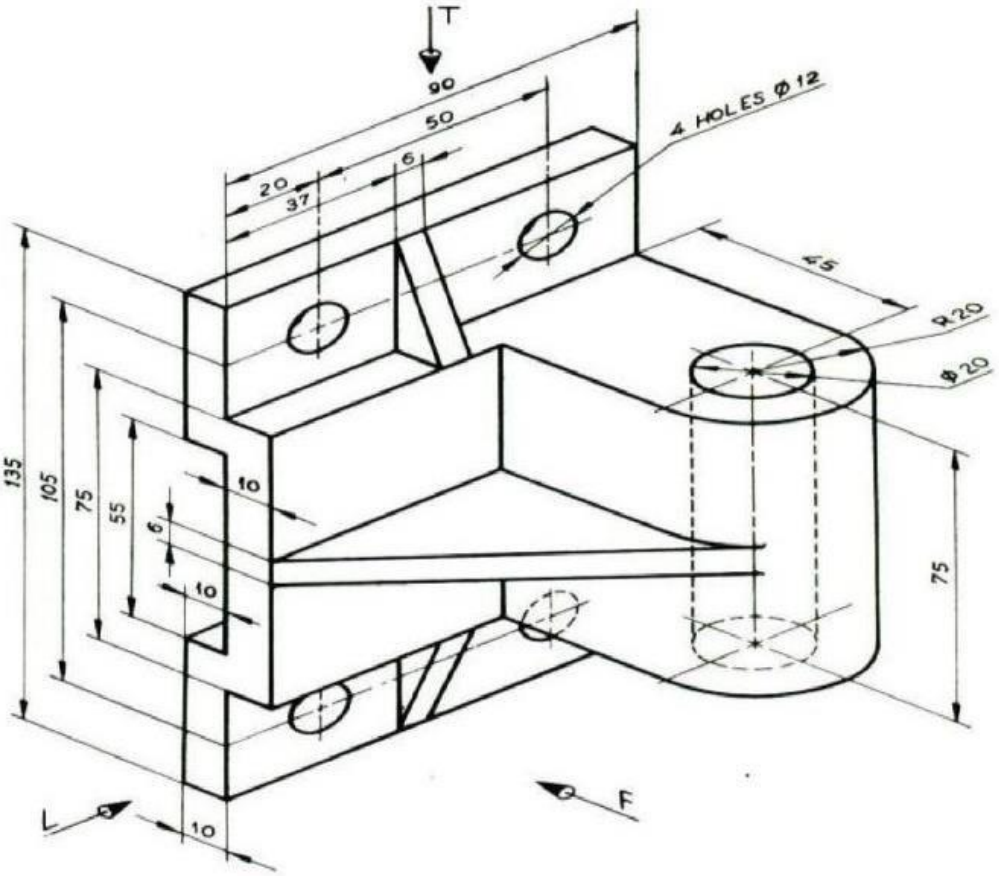
To create a given model by Using the AUTOCAD/CATIA



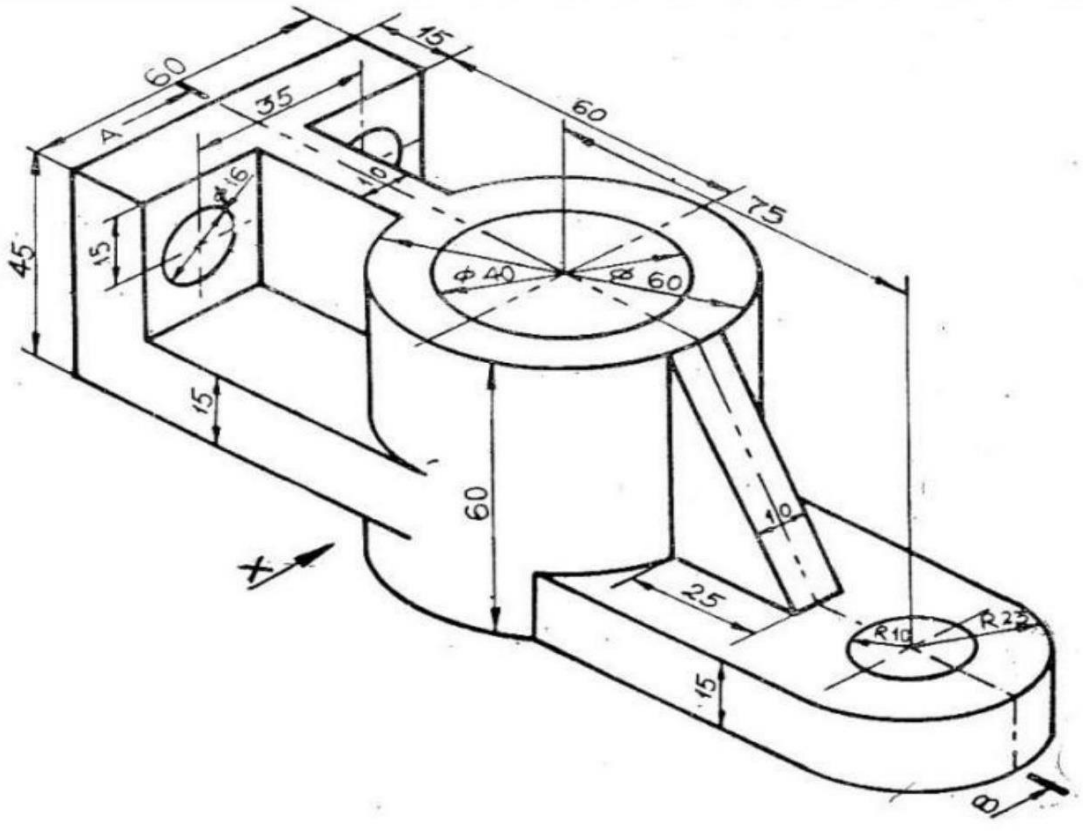
To create a given model by Using the AUTOCAD/CATIA



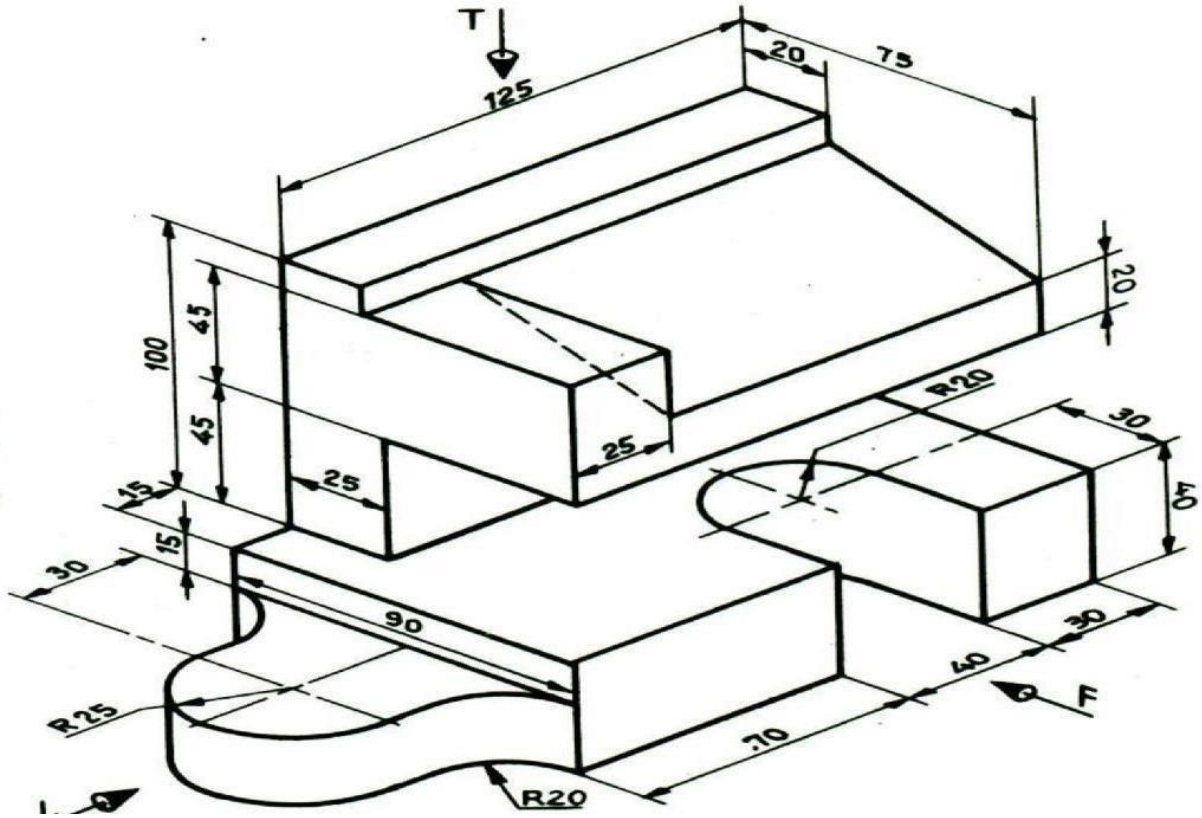
To create a given model by Using the AUTOCAD/CATIA



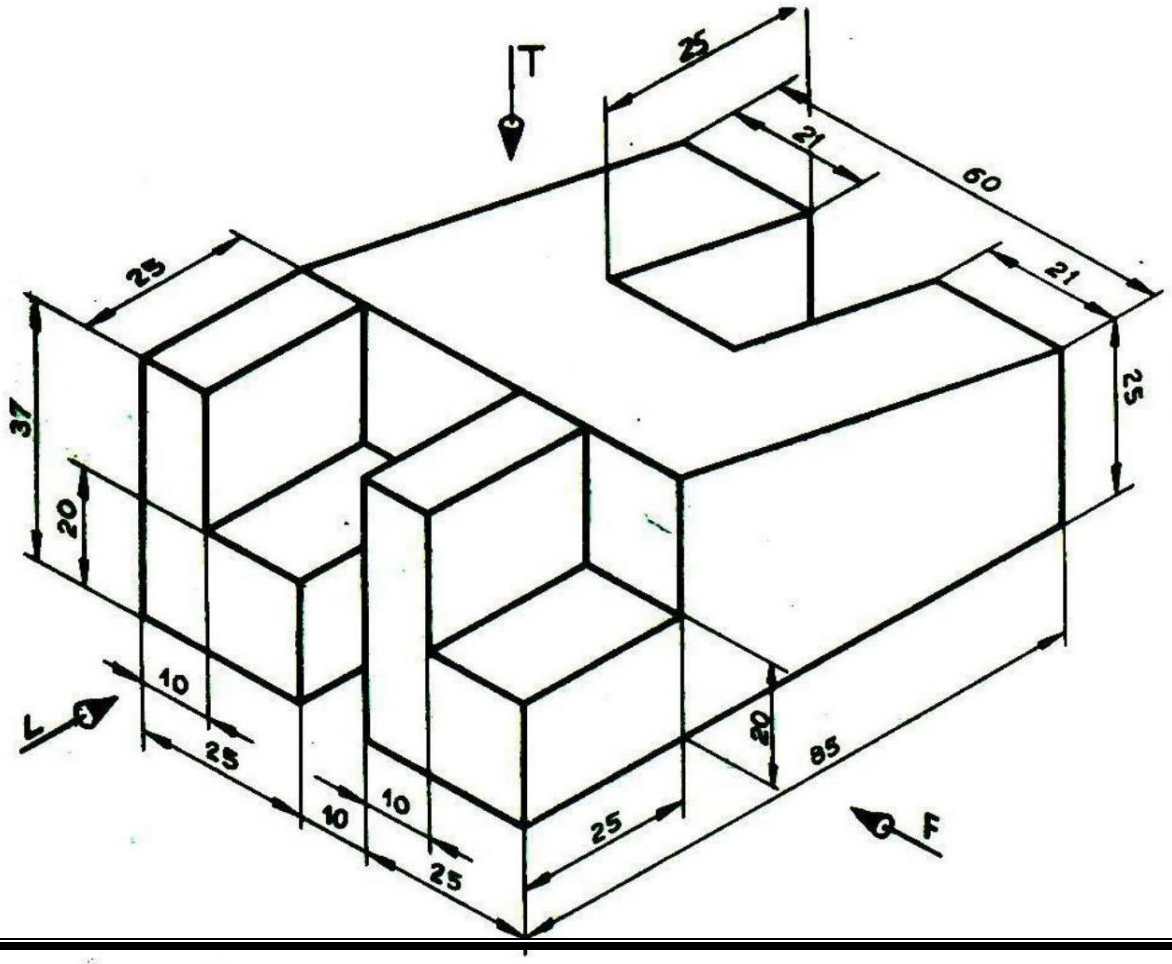
To create a given model by Using the AUTOCAD/CATIA



To create a given model by Using the AUTOCAD/CATIA



To create a given model by Using the AUTOCAD/CATIA



(FLANGE COUPLING)

AIM:

1. To draw the detail view of the flange coupling and assemble the parts by using the CATIA V5 R20.0 software and obtain its respective views.
2. To find the mass properties of the final assembly.

COMMANDS USED: Sketch, pad, shaft, Pattern, Mate, Align, Helical Sweep, Round, Chamfer etc,.

PROCEDURE: PART DRAWING: FLANGE:

- Draw the sectional view of the flange in the sketcher mode.
- Draw the middle axis line for the purpose of using shaft command and make the flange.
- Make the keyway and holes by using pad material remove command.
- Round the sharp edges of the flange by using round tool command.

SHAFT AND KEY:

- Use pad command to make the shaft and the keyway.
- Use pad command to make the key.

BOLT AND NUT:

- Use the pad command makes the bolt head and shank of the bolt.
- Use the helical sweep command makes the thread in bolt shank.
- Use the pad and helical sweep command make the nut with thread.

ASSEMBLY:

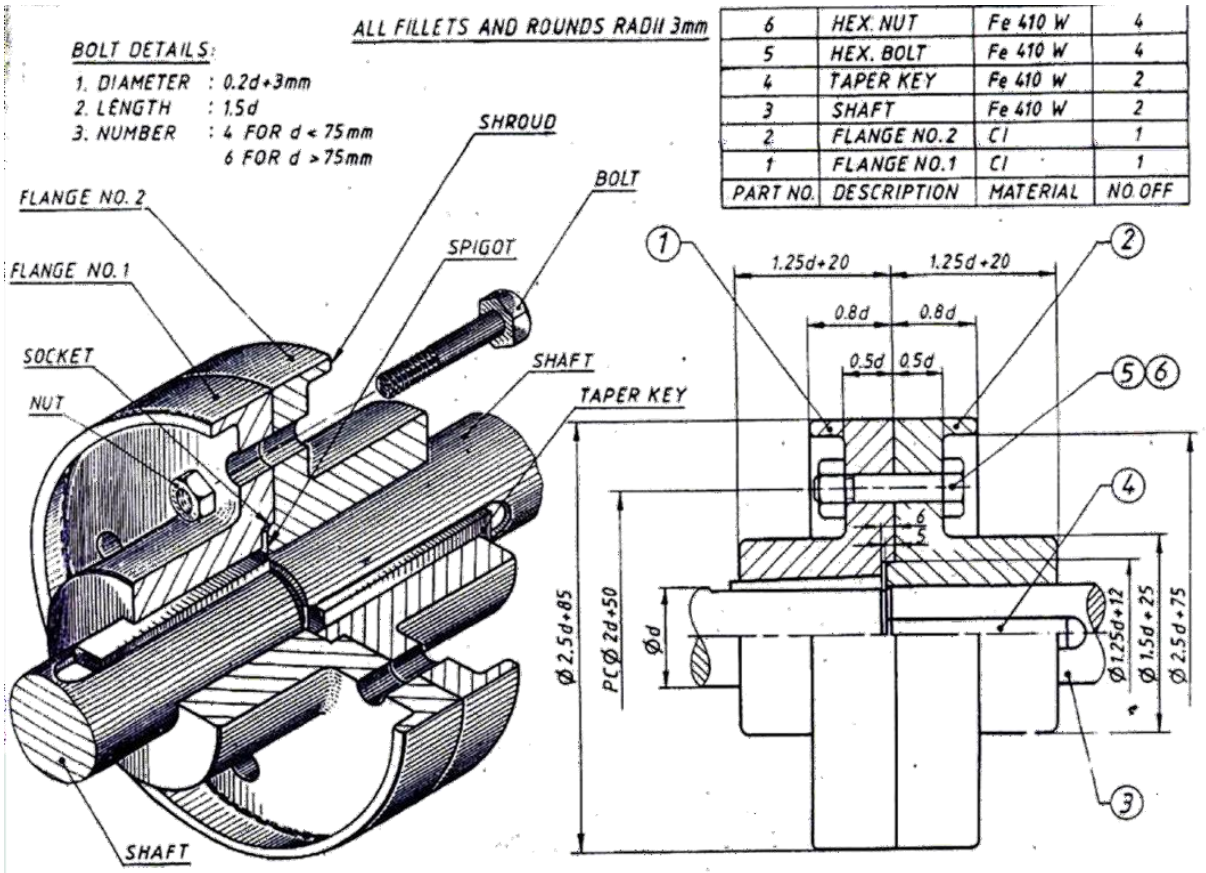
- Use the mate, align, insert and pattern commands to assemble the flange coupling.

DETAILED DRAWING:

- Use the drawing mode makes the respective views and bill of materials.

RESULT:

Thus the Detail View of the Flange Coupling along with its respective views and mass properties has been found.

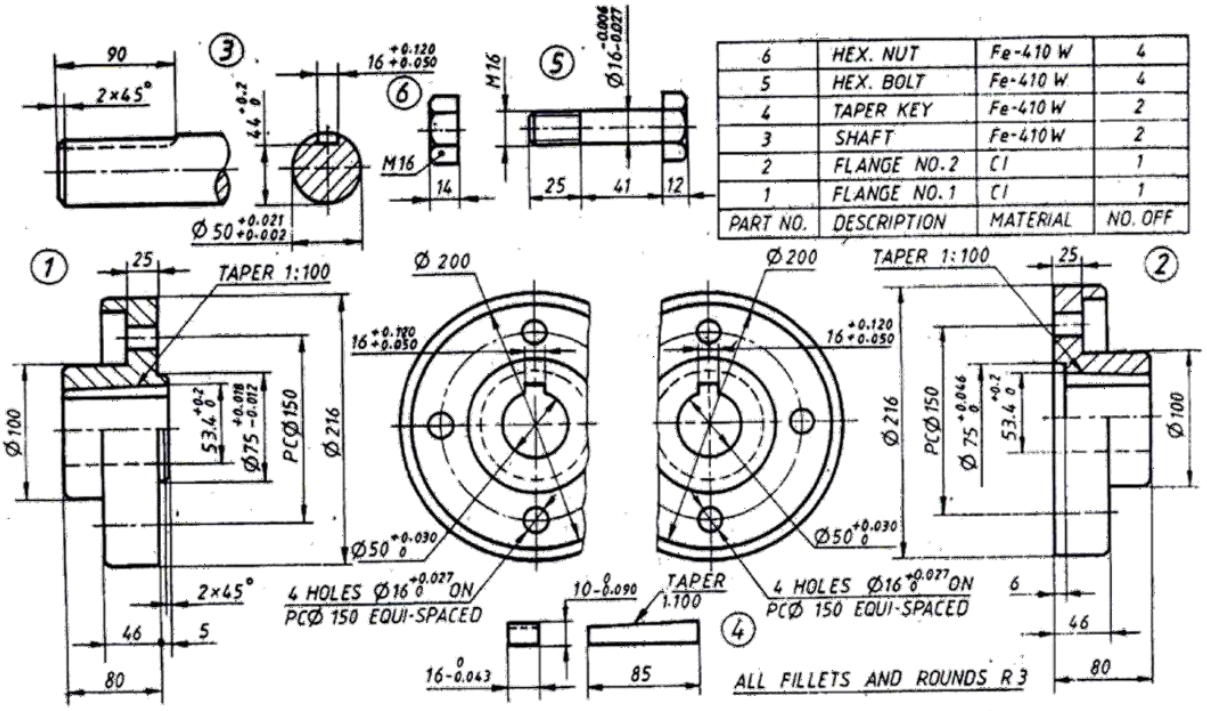


PART NO.	DESCRIPTION	MATERIAL	NO OFF
6	HEX. NUT	Fe 410 W	4
5	HEX. BOLT	Fe 410 W	4
4	TAPER KEY	Fe 410 W	2
3	SHAFT	Fe 410 W	2
2	FLANGE NO.2	CI	1
1	FLANGE NO.1	CI	1

All Dimensions in mm

Flanged Coupling — Protected Type

Fig. 16.9



PART NO.	DESCRIPTION	MATERIAL	NO. OFF
6	HEX. NUT	Fe-410 W	4
5	HEX. BOLT	Fe-410 W	4
4	TAPER KEY	Fe-410 W	2
3	SHAFT	Fe-410 W	2
2	FLANGE NO.2	CI	1
1	FLANGE NO.1	CI	1

To develop the given model by Using CATIA

(NON RETURN VALVE)

AIM:

To draw the detail view of the Universal Coupling and assemble the parts by using the CATIA V5 R20.0 software and obtain its respective views.

COMMANDS USED: Sketch, pad, shaft, Mate, Align, Round, Chamfer etc,.

PROCEDURE: PART DRAWING: BODY:

- Draw the cross section of the body and revolve it.
- Draw the concentric circles of the fork and remove materials.
- Draw the flange part of the body and pad it.

Valve Seat:

- Use the shaft command makes the Valve Seat of the Non Return Valve.

Valve:

- Use the shaft command makes the Valve Seat of the Non Return Valve.

ASSEMBLY:

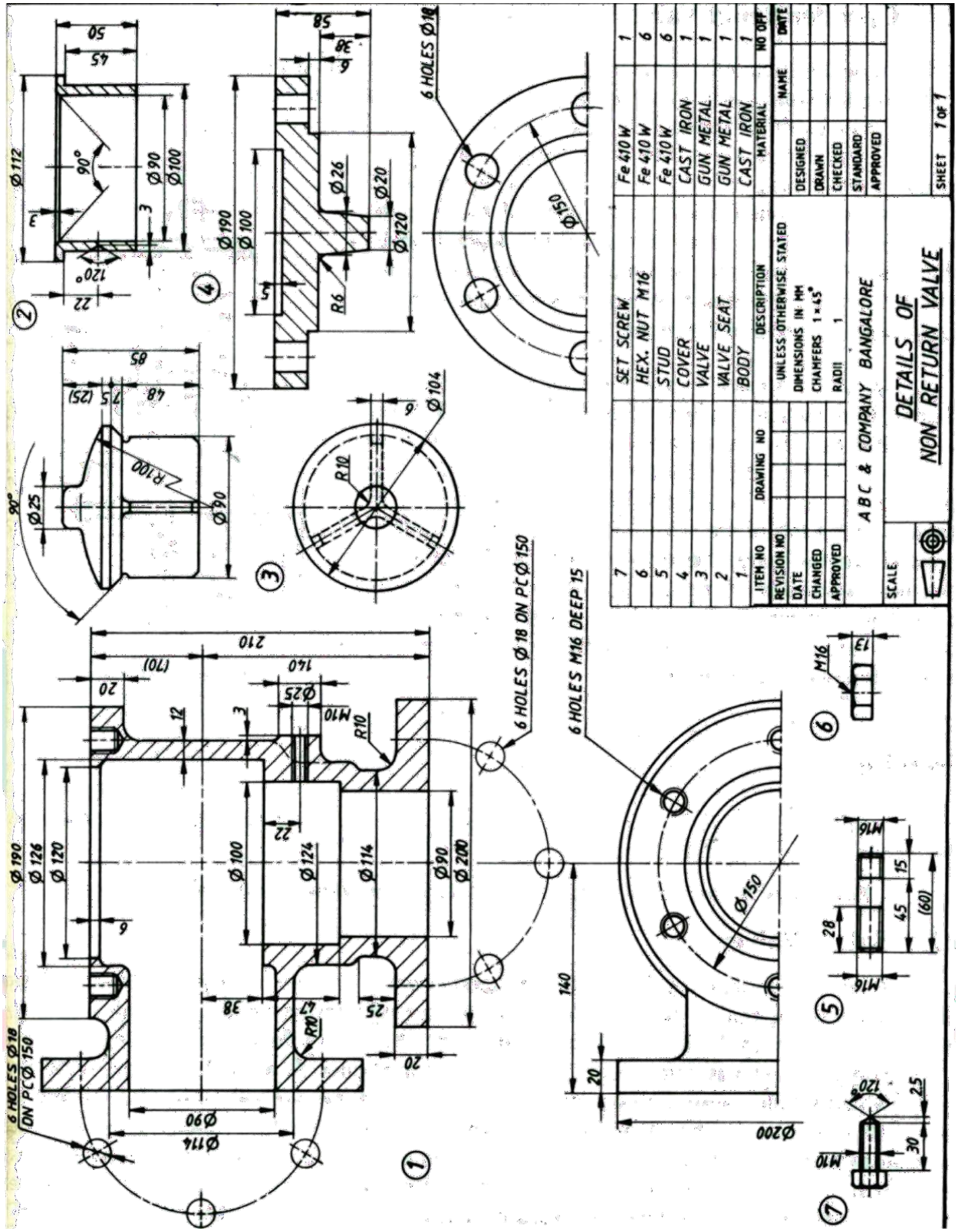
- Use the mate, align, insert and pattern commands to assemble the Non Return Valve.

DETAILED DRAWING:

- Use the drawing mode makes the respective views and bill of materials.

RESULT:

Thus the Detail View of the Non Return Valve and then its respective views have been drawn.


DETAILS OF
NON RETURN VALVE
ABC & COMPANY BANGALORE
SCALE
SHEET 1 OF 1
ITEM NO
DRAWING NO
DESCRIPTION
MATERIAL
NO OFF
SET SCREW
HEX. NUT M16
STUD
COVER
VALVE
VALVE SEAT
BODY
Fe 410 W
CAST IRON
GUN METAL
CAST IRON
UNLESS OTHERWISE STATED
DIMENSIONS IN MM
CHAMFERS 1×45°
RADII 1
REVISION NO
DATE
CHANGED
APPROVED
NAME
DATE
DESIGNED
DRAWN
CHECKED
STANDARD
APPROVED
6 HOLES Ø18 ON PCØ 150
6 HOLES M16 DEEP 15
Ø150
Ø200
Ø114
Ø90
Ø100
Ø124
Ø120
Ø126
Ø190
Ø112
Ø104
Ø25
Ø26
Ø20
M10
M16
R10
R100
R6
120°
90°

To develop the given model by Using CATIA

(PIPE VICE)

AIM: To draw the detail view of the Pipe Vice and assemble the parts by using the CATIA software and obtain its respective views.

COMMANDS USED: Sketch, pad, Shaft, Pattern, Mate, Align, Helical Sweep, Round, Chamfer etc,

PROCEDURE: PART DRAWING: PIPE BASE:

- Using Pad command the Pipe Base of the Pipe Vice has been drawn.

MOVABLE JAW:

- Using Pad command the Movable Jaw of the Pipe Vice has been drawn.

SET SCREW:

- Using Shaft and Pad Set Screw of the Pipe Vice has been drawn.

HANDLE BAR:

- Using Shaft command the Handle Bar has been drawn.

HANDLE BAR CAP:

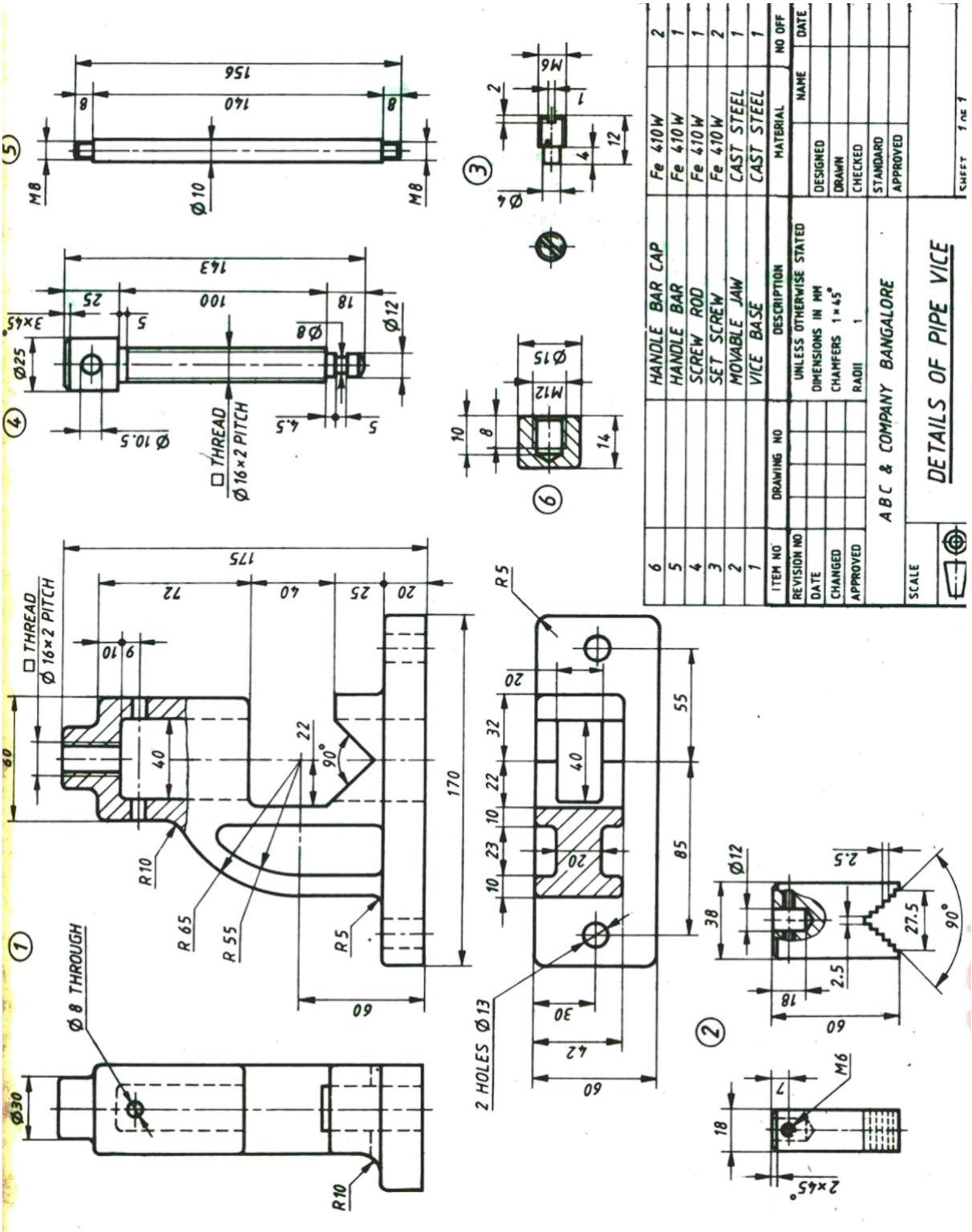
- Using Shaft command the Handle Bar Cap has been drawn.

ASSEMBLY AND DETAILED DRAWING:

- Using the Assembly and Drawing mode to make the respective views and bill of materials.

RESULT:

Thus the Detail View of the Pipe Vice and its respective views have been drawn.


DETAILS OF PIPE VICE
ABC & COMPANY BANGALORE
HANDLE BAR CAP
HANDLE BAR
SCREW ROD
SET SCREW
MOVABLE JAW
VICE BASE
Fe 410 W
CAST STEEL
THREAD Ø 16×2 PITCH
Ø 8 THROUGH
2 HOLES Ø13
UNLESS OTHERWISE STATED
DIMENSIONS IN MM
CHAMFERS 1×45°
RADII 1

To develop the given model by Using CATIA

(STUFFING BOX)

AIM: To draw the detail view of the Stuffing Box and assemble the parts by using the pro-e software and obtain its respective views.

COMMANDS USED: Sketch, pad , Shaft, Pattern, Mate, Align, Helical Sweep, Round, Chamfer etc,

PROCEDURE: PART DRAWING: CYLINDER:

- Using Pad, Cut and Round Commands the cylinder has been drawn

NUT:

- Using Pad, Cut and Round Commands the nut has been drawn.

GLAND BUSH:

- Using Pad and Cut Commands the gland bush has been drawn.

PISTON ROD:

- Using Pad and Cut Commands the piston rod has been drawn.

PACKING:

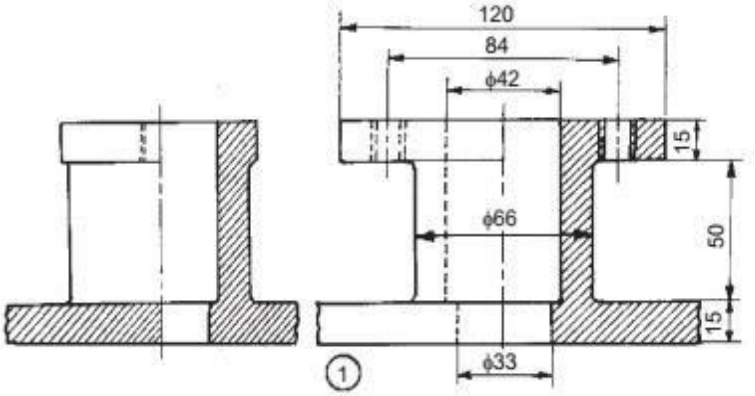
- Using Shaft command the packing has been drawn.

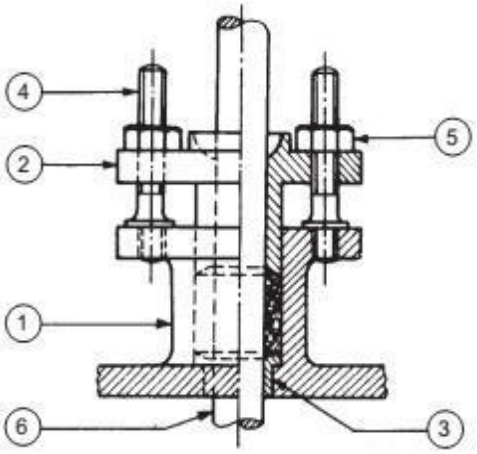
ASSEMBLY AND DETAILED DRAWING:

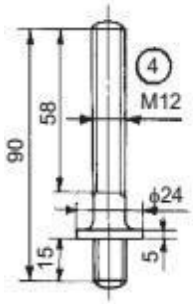
- Using the Assembly and Drawing mode to make the respective views and bill of materials.

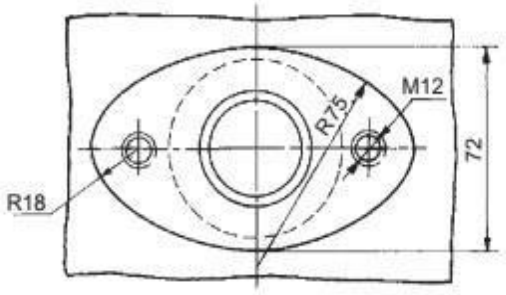
RESULT:

Thus the Detail View of the Stuffing Box and then its respective views have been drawn.



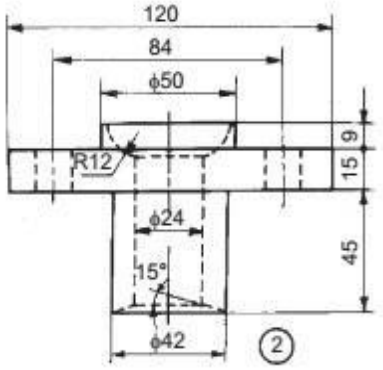


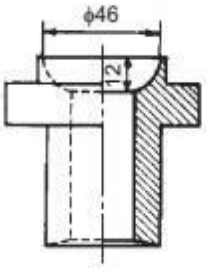


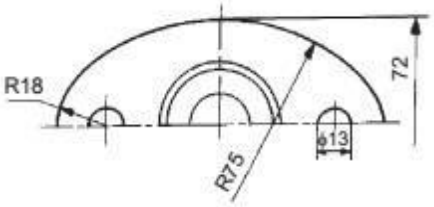


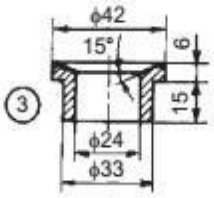
Parts list

Part No.	Name	Matl	Qty
1	Body	CI	1
2	Gland	Brass	1
3	Bush	Brass	1
4	Stud	MS	2
5	Nut, M12	MS	2





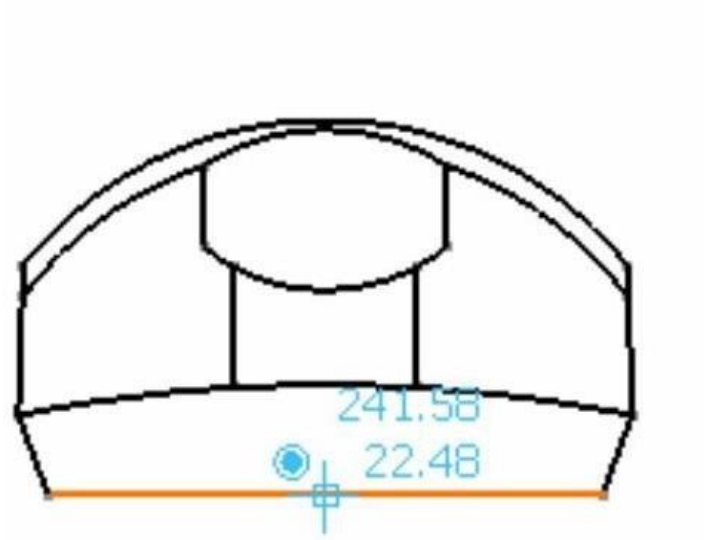


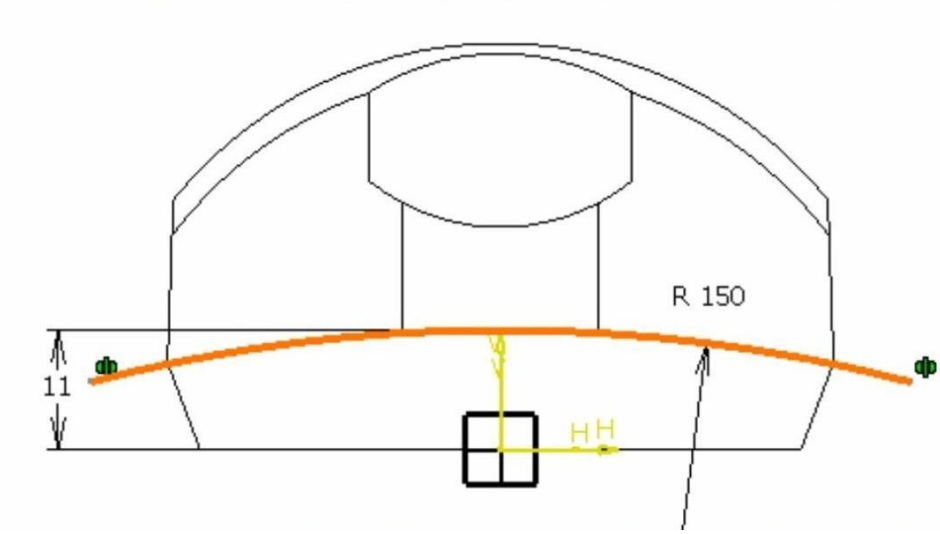


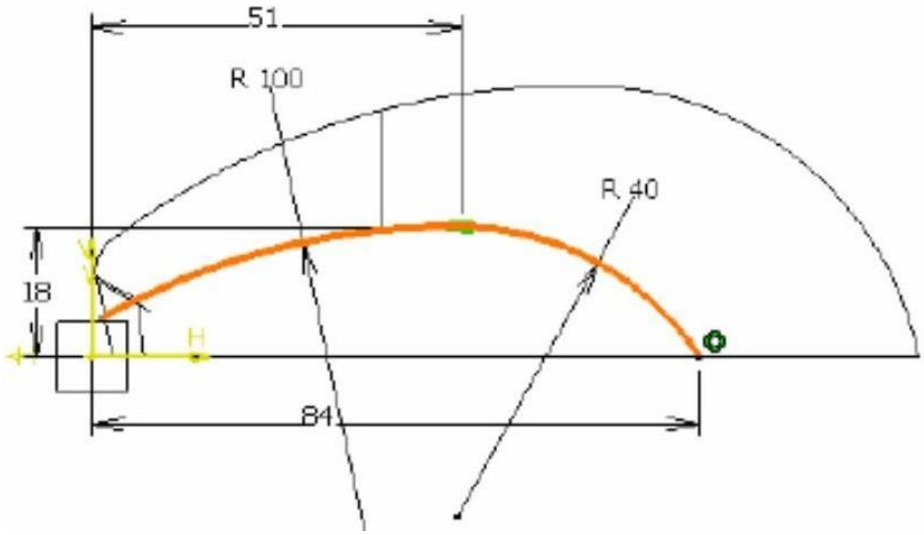
To develop the given model by Using CATIA

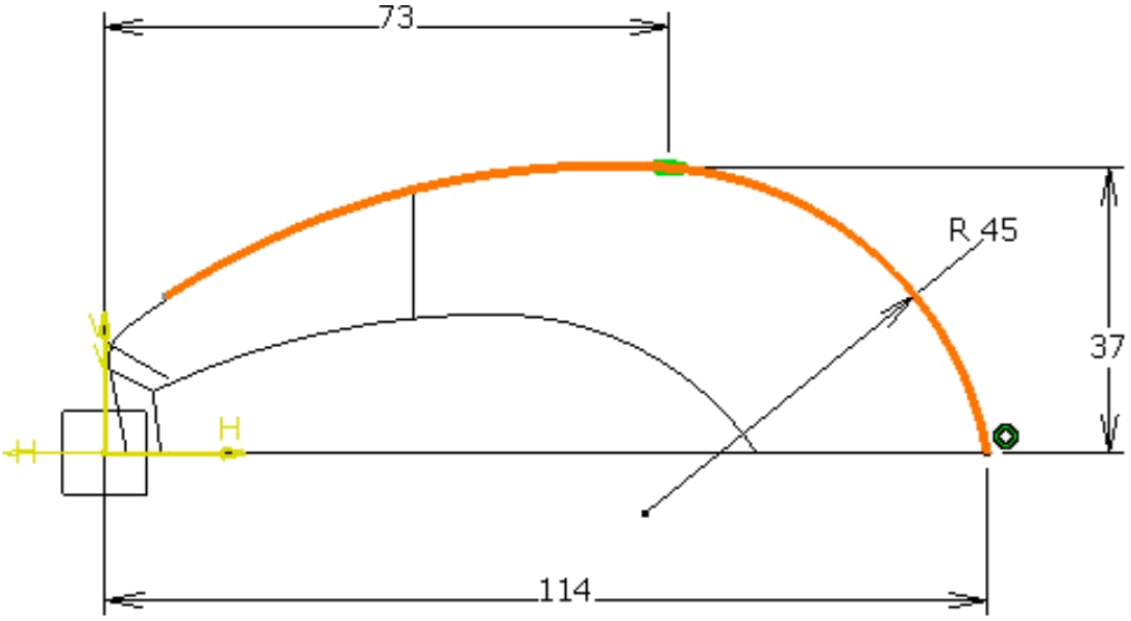
SURFACE MODELING

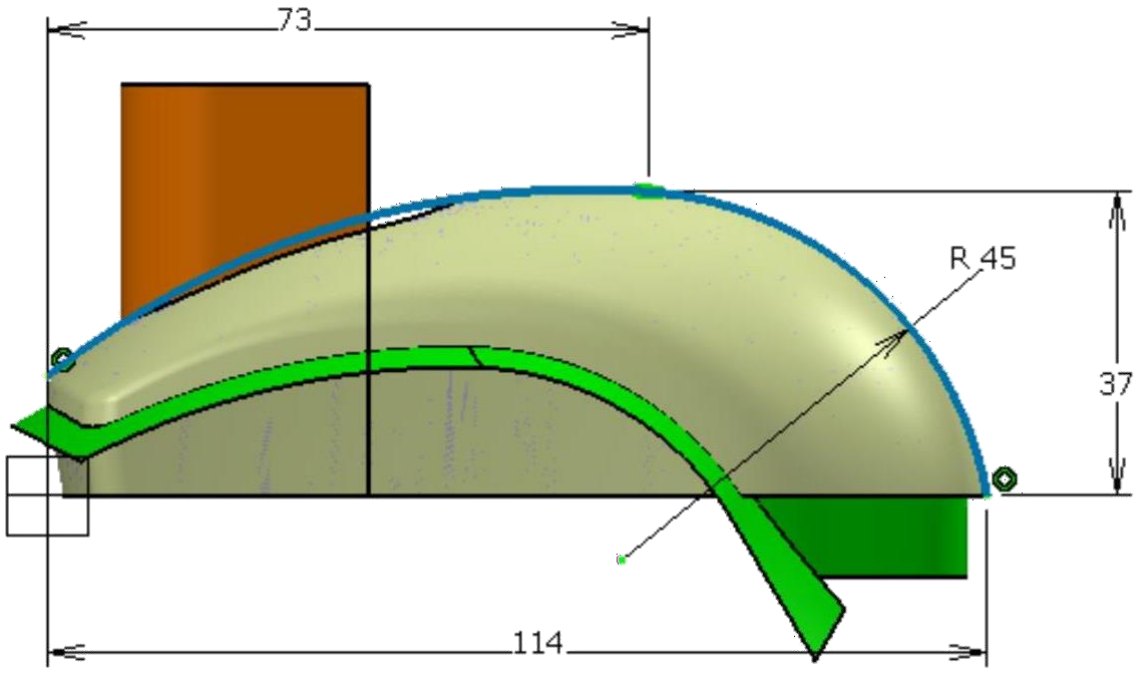
(MOUSE)








73
R 45
37
V
H
114


73
R 45
37
114

Expand CATIAV5?

Computer Aided Three Dimensional Interactive Applications.

What is the save pad of sketcher file?

CAT Part

Does CATIA V5 work on UNIX Platform?

Yes

Is it possible to increase the size of plane boundary representation & how?

Yes, go for Tools-Options- Infrastructure-Part structure-Display

Is It Possible to directly enter in to Sketcher Workbench?

No, it is not possible to enter in to sketcher workbench directly. We have to go for any workbench &form there we can enter the sketcher workbench..

Which is the tool used to exit from sketcher workbench to part design Workbench?

Exit Sketcher.

What is the use of construction elements?

Construction elements assist in sketching the required profile in sketcher.

What are the default units of LMT (Length, Mass and Time) mm, Kg,
Second.

Is it possible to hide specification tree?

Yes, with help of F3 button, but the option in Tools command must be checked to allow this.

What is SHOW/HIDE option?

Show mode enables us to see all the components presently opened and in Hide mode we can hide the desired elements from the view for time being.

What is the meaning of true dimension?

True dimension is the dimension desired after the machining. In other words, this is the value that should be attained after the machining.

What are different conic sections?

Ellipse, Hyperbola and Parabola

STRUCTURAL ANALYSIS

EXPERIMENT: 1

TWO DIMENSIONAL STATIC LINEAR ANALYSIS OF A CANTILEVER BEAM

AIM:

To determine the stresses acting on a cantilever beam with a point load of 750 N acting at one of its ends and perpendicular to the axis of the beam.

- Young's modulus = 2e5
- Poisson's ratio = 0.3
- Length of the beam = 2 m = 2000 mm
- Breadth of the beam = 80 mm
- Height of the beam = 40 mm

PROCEDURE:

PRE PROCESSING

STEP 1: From the Main menu select preferences Select structural and press OK

STEP 2: From the main menu select **Pre-processor**

Element type Add / edit/Delete Add BEAM – 2D Elastic 3 Apply Close.
Material properties material models Structural Linear Elastic Isotropic EX = 2e5; PRXY = 0.3

STEP 3: From the main menu select Pre-processor

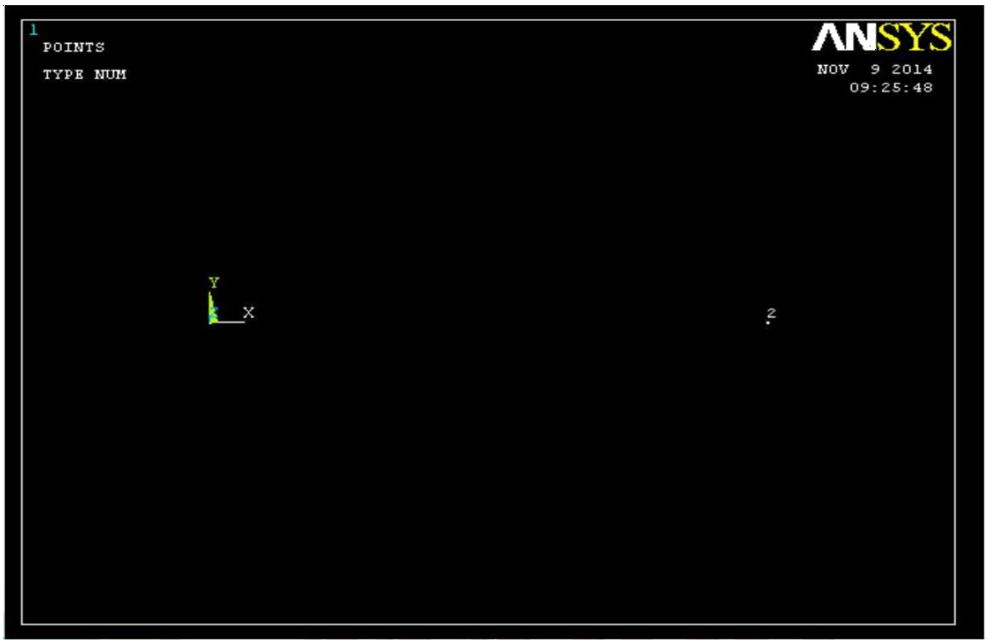
Sections Beam Common Sections Select subtype as Rectangular section Enter B = 100, H = 50 Apply Preview

Real constants Add Add Ok Geometric Properties Area = 5000, I_{zz} = 4170000, Height = 40 Ok Close

STEP 4: From the main menu select Pre-processor Modelling

- Create the key points in the Workspace
  Create Key points in active CS

X	0	2000
Y	0	0



Click APPLY to all the points and for the last point click OK

- Create LINES using the Key points

Create Lines Lines Straight Line Click on Key points to generate lines Select Plot controls from menu bar Capture image file save as and save your file.

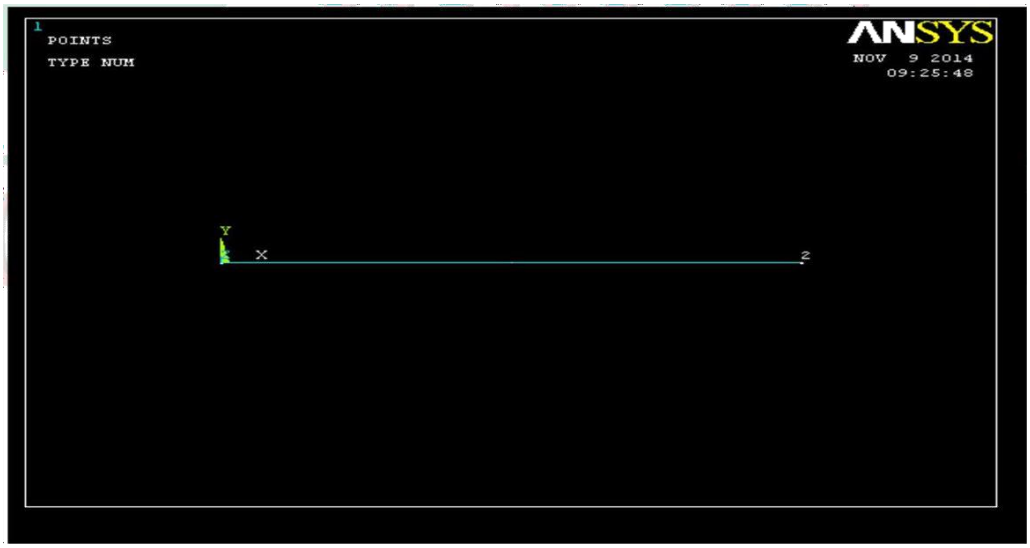


Figure: Model

STEP 4: Meshing the Geometry

From the main menu select **Meshing**

Meshing Size controls Manual size Lines All lines – Number of element divisions = **20** Click OK

Meshing Mesh Lines – pick all

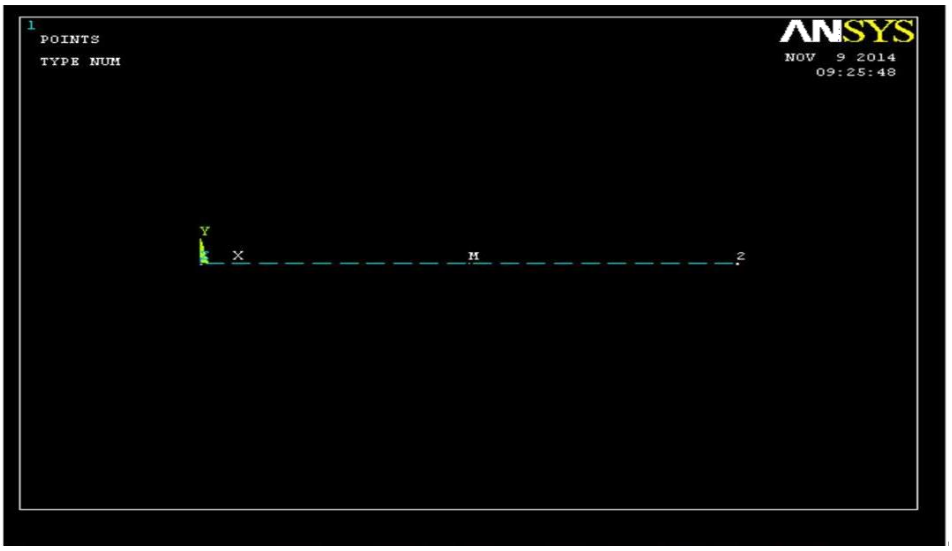


Fig. MESHED MODEL

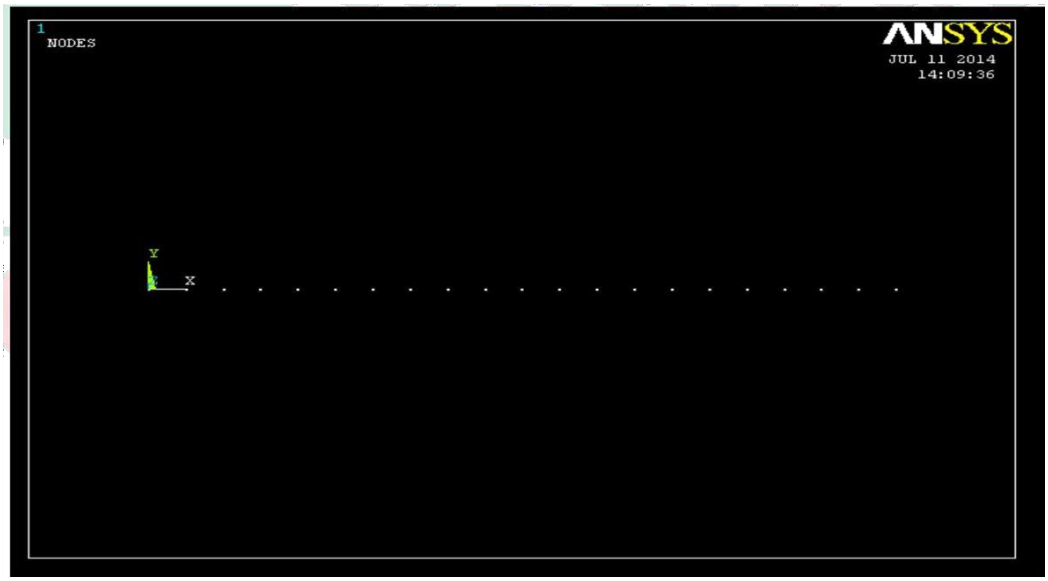


Figure: Meshed Model with nodes

SOLUTION PHASE: ASSIGNING LOADS AND SOLVING

STEP 5: From the ANSYS main menu open **Solution** solution Analysis type new analysis – Static

STEP 6: Defining loads at the Key points

Solution Define Loads Apply Structural Displacement On key points Left end – ALL DOF arrested

Solution Define loads Apply Structural Force/moment On key Points

Right end – Apply a load of FY = -750 N

Select Plot controls from menu bar Capture image file save as and save your file

Figure: Model with boundary conditions

STEP 7: Solving the system

Solution ⌞Solve ⌷ Current LS

POSTPROCESSING: VIEWING THE RESULTS

1. **Deformation**

From the main menu select **General post processing**

General post processing ⌷ Plot Results ⌷ Deformed Shape

Select 'Def + undef edge' and click 'OK' to view both the deformed and the undeformed object.

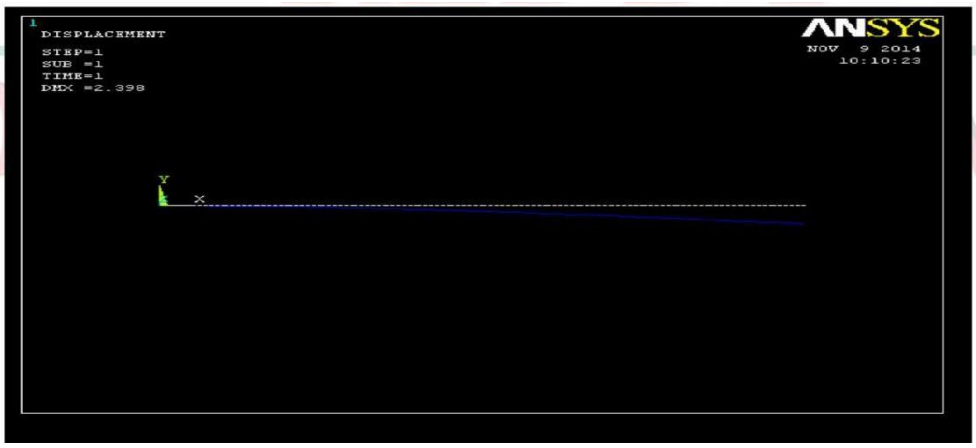


Figure: Deformed and undeformed Model

- **Nodal solution**

From the Utility menu select PLOT

PLOT Results Contour plot Nodal solution – DOF solution – Y component of displacement – OK

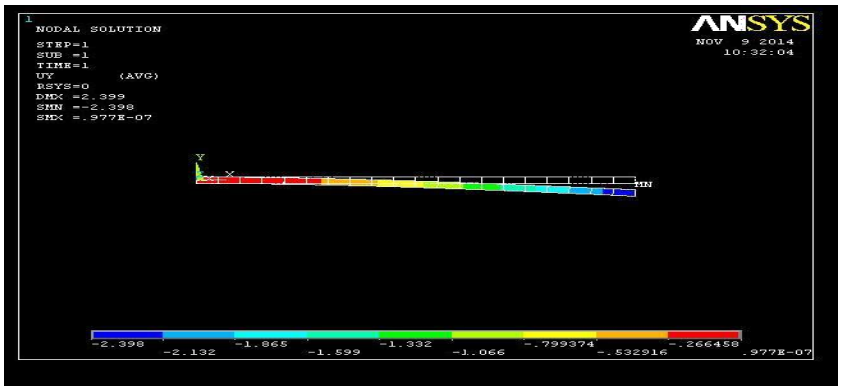


Figure: Y-Component displacement of the Model

RESULT:

DMX =

SMN =

SMX =

VIVA QUESTIONS

1. If a cantilever beam has a uniformly distributed load, will the bending moment diagram be quadratic or cubic?
2. Name the element type used for beams?
3. Define Analysis and its Purpose?
4. What are the modules in Ansys Programming?
5. What are the Real Constants & Material Properties in Ansys? Explain?

EXPERIMENT: 2

2-D STATIC LINEAR ANALYSIS OF A TRUSS STRUCTURE

AIM:

To determine the nodal deflections, reaction forces, and stress of the indeterminate truss system when it is subjected to a load of 2000 N.

APPARATUS: Ansys 12.0

GIVEN DATA:

- Young‟s modulus = 2e5

A = 3250mm^2

Poisson‟s ratio = 0.3

PROCEDURE:

PREPROCESSING

STEP 1: From the Main menu select preferences Select structural and press OK

STEP 2: From the main menu select **Preprocessor**

Element type Add / edit/Delete Add Link – 2D spar 8 ok close Real constants Add Geometric Properties Area = 3250

Material properties material models Structural Linear Elastic Isotropic EX = 2e5; PRXY = 0.3

STEP 3: From the main menu select Pre-processor **Modelling**

- Create the key points in the Workspace

 Pre-processor Modelling Create Nodes In active CS

X	Y	Z
0	0	0
5	0	0
10	0	0
15	0	0
2.5	2.5	0
7.5	2.5	0
12.5	2.5	0

 Click APPLY to all the points and for the last point click OK

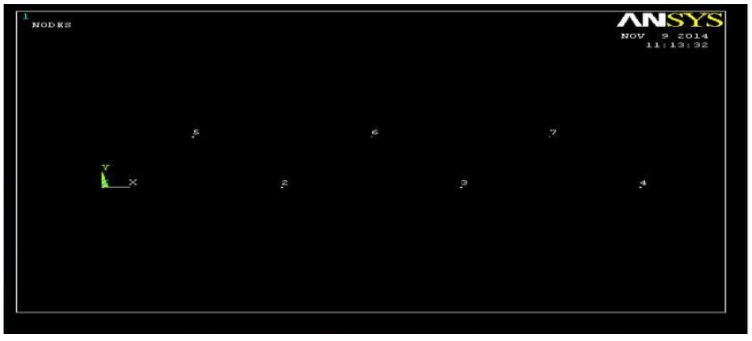


FIG: TRUSS (KEY POINTS MODEL)

- Create LINES using the Elements

Pre-processor Modelling Create Elements Auto numbered through nodes select node 1&2 apply 2&3 apply 3&4 apply 1&5 apply 5&2 apply 2&6 apply 6&3 apply 3&7 apply 7&4 apply 5&6 apply 6&7 ok close.

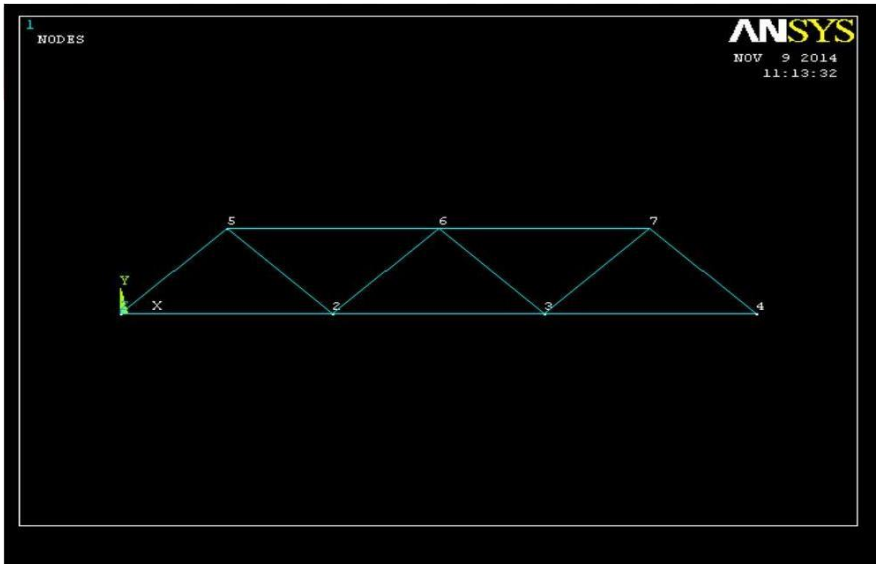


FIG: TRUSS (LINE MODEL)

SOLUTION PHASE: ASSIGNING LOADS AND SOLVING

STEP 5: From the ANSYS main menu open **Solution** Solution
Analysis type new analysis – Static

STEP 6: Defining loads at the Key points

Solution Define Loads Apply Structural Displacement On nodes select node 1&4 ok select All DOF ok

Left end – ALL DOF arrested

Solution Define loads Apply Structural Force/moment On nodes

Select node 2&3 ok FY direction Give force value as 2000 N ok close

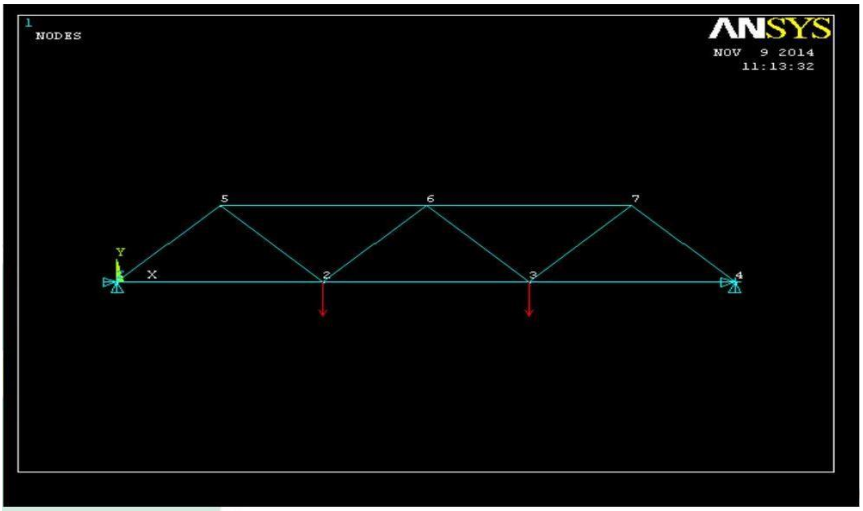


Figure: Model with boundary conditions

STEP 7: Solving the system

Solution Solve Current LS

POSTPROCESSING: VIEWING THE RESULTS

1. Deformation

From the main menu select **General post processing**

General post processing Plot Results Deformed Shape

Select 'Def + undef edge' and click 'OK' to view both the deformed and the undeformed object.

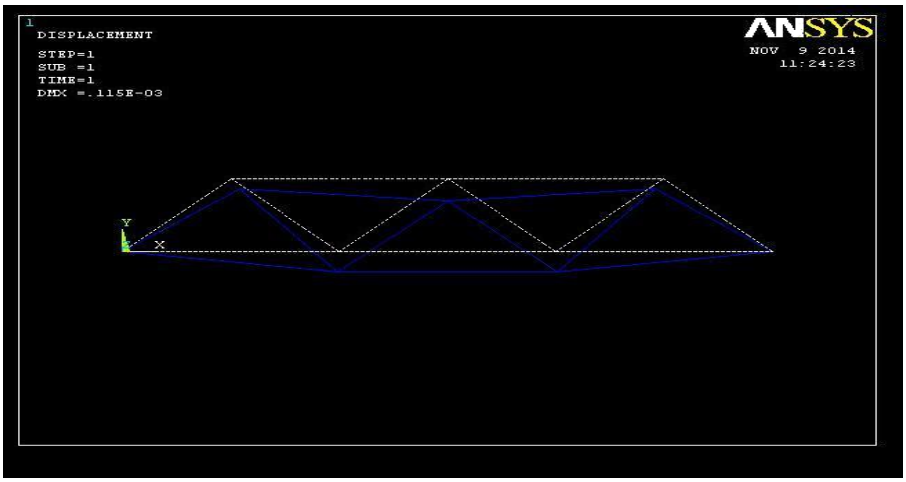


Figure: Deformed and undeformed Model

Nodal solution

From the Utility menu select PLOT

PLOT Results Contour plot Nodal solution DOF solution Y component of displacement OK

RESULT:

DMX =

SMN =

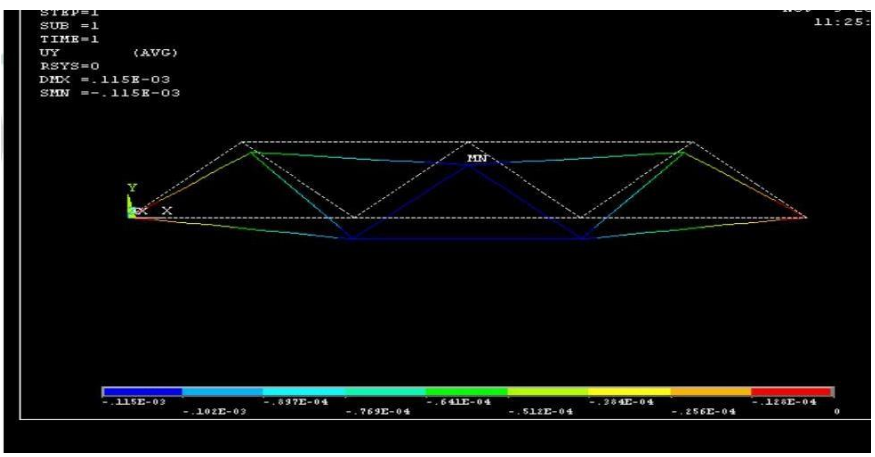


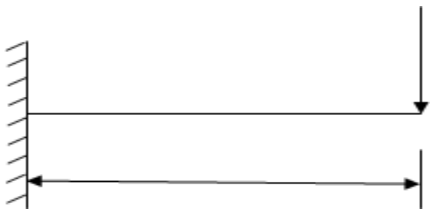
Figure: Y-Component displacement of the Model

VIVA QUESTIONS

1. Ansys needs the final element model(FEM) for its final solution.(T/F)
2. Element attributes must be set before meshing the solid model. (T/F)
3. In a plane strain, the strain in the direction of thickness is assumed to be zero.(T/F)
4. The_____elements are used for in-plane bending problems.
5. Which one of the following elements is required to define the thickness as a real constant?

a. Beam
b. Shell
c. Solid
d. None

To find the displacement, maximum, minimum stresses induced in a given cantilever beam and draw the shear force and bending moment diagrams by using ANSYS tool, also list the results according to the given loads.



Procedure:

Preferences:

Main menu > preferences > check structural

Pre-processor:

1. Main menu > preprocessor > element type > add/edit/delete > Add > select Beam -2D elastic3 > click ok > close.
2. Main menu > pre-processor > Real constraints > Add/Edit/Delete > Add > Click ok > Enter the following values Area=1, Area moment of Inertia (IZZ) =1/12, Height=1, > click ok > close.
3. Main menu > pre-processor > Material properties > Material model > structural > linear > elastic > isotropic > enter Ex as 2E5 & PRXY as 0.3 > click ok > close.
4. Main menu > pre-processor > modelling > create > nodes > Inactive CS > (Enter X<Y<Z location

values) as shown in the figure starting from node-1 i.e., 0, 0, 0 and 10,0,0 thus two nodes are created.

5. Main menu > pre-processor > modelling > create > element > auto numbered > through nodes (select node by node and middle click) with this elements are created.

Solution:

1. Solution > define loads > Apply > structural > displacement > pick on nodes (Pick the node which are to be constrained with direction) > middle click > select DOF or UX or UY or UZ as required > click ok.

Pick node-1 > middle click > select all DOF since to constrain the beam.

2. Solution > define loads > Apply > structural > force/moment > on nodes > select node > middle click > select FX and FY or MZ depending on the problem > click ok. In the above problem load acting downwards i.e. in negative Y-direction at node -2 of about 10N.

General post processor:

1) General post processor > element table > define table > add select by sequence number > enter SMISC , 2,4,6,8,12 one after the other > click ok > close.

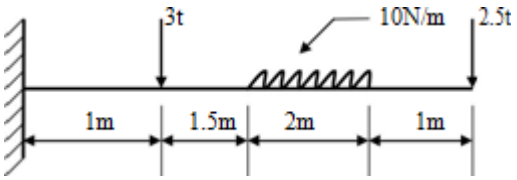
2) General post processor > plot results > deformed shape > click ok.
3) General post processor > plot results > contour plot > nodal solution > click ok.
4) General post processor > plot results > contour plot > element solution > click ok.
5) General post processor > plot results > contour plot > line element results > use the
6) Combination 2,8 for shear force diagram and 6,12 for bending moment diagram.
7) General post processor > plot results > vector plot > predefined > click ok.
8) General post processor > list results > nodal solution > click ok.
9) General post processor > list results > element solution > click ok.

Result: Thus the analysis on the given beam is performed using ANSYS tool, Shear force ,bending moment diagrams are captured and values are noted.

Viva Questions:

1. What is ANSYS?
2. Why ANSYS is used?
3. What is Structural Analysis?
4. What is Thermal Analysis?
5. What is the difference between Structural Analysis and Thermal Analysis?
6. What is node?
7. What is element?
8. What is difference between node and element?
9. What are the minimum requirements to develop a drawing ANSYS?
10. What is general post processor?

To find the displacement, maximum, minimum stresses induced in a given cantilever beam with uniformly distributed load and point loads and draw the shear force and bending moment diagrams by using ANSYS tool, also list the results according to the given loads.



Procedure:

Preferences:

Main menu > preferences > check structural

Pre-processor:

1) Main menu > pre-processor > element type > add/edit/delete > Add > select Beam -2D elastic3 > click ok > close.

2) Main menu > pre-processor > Real constraints > Add/Edit/Delete > Add > Click ok > Enter the following values Area=1, Area moment of Inertia (IZZ) =1/12, Height=1, > click ok > close.

3) Main menu > pre-processor > Material properties > Material model > structural > linear > elastic > isotropic > enter Ex as 2E5 & PRXY as 0.3 > click ok > close.

4) Main menu > pre-processor > modelling > create > nodes > Inactive CS > (Enter X<Y<Z location values) as shown in the figure starting from node-1 i.e., 0,0,0 and 1,0,0, 2.5,0,0, 4.5,0,0, 5,0,0 thus five nodes are created.

Main menu > pre-processor > modelling > create > element > auto numbered > through nodes (select node by node and middle click) with this elements are created.

Solution:

1. Solution > define loads > Apply > structural > displacement > pick on nodes (Pick the node which are to be constrained with direction) > middle click > select DOF or UX or UY or UZ as required > click ok.

   Pick node-1 > middle click > select all DOF since to constrain the beam.

2. Solution > define loads > Apply > structural > force/moment > on nodes > select node > middle click > select FX and FY or MZ depending on the problem > click ok. In the above problem load acting

downwards i.e. in negative Y-direction at node -2 and 5 of about 3tones and 2.5 tones.

3. Solution > define loads > apply > structural > pressure (applied or uniformly distributed load) > on beams > select beam middle click(Enter pressure value at the both ends) i.e., at I & J , enter 1 and 1 > click ok

   Solution > solve > click ok.

General post processor:

1. General post processor > element table > define table > add select by sequence number > enter SMISC, 2, 4, 6,8,12 one after the other > click ok > close.
2. General post processor > plot results > deformed shape > click ok.
3. General post processor > plot results > contour plot > nodal solution > click ok.
4. General post processor > plot results > contour plot > element solution > click ok.
5. General post processor > plot results > contour plot > line element results > use the
6. Combination 2,8 for shear force diagram and 6,12 for bending moment diagram.
7. General post processor > plot results > vector plot > predefined > click ok.
8. General post processor > list results > nodal solution > click ok.
9. General post processor > list results > element solution > click ok.

Result: Thus the analysis on the given beam is performed using ANSYS tool, Shear force, bending moment diagrams are captured and values are noted.

Viva Questions:

1. What is nodal solution?
2. What is pre processor?
3. What is post processor?
4. What is modelling?
5. What type of options we use in preferences?
6. ANSYS is it a tool or software?
7. What is the difference between Animation and Modelling?
8. What is DOF?
9. How to create a Node?
10. How to create an Element?

CAM LAB

CNC TURNING LATHE

SPECIFICATIONS:

Make: HMT Model:

CNC Tutor Control system:

Hinumeric Axis: 2(X, Y,)

Year of purchase: 2015

Cost: Rs.12, 92,399.80

Machine Specifications:

1. Height of centres: 70mm
2. Distance between centres: 310mm
3. Swing over bed: 100 mm
4. Swing over cross slide: 60mm
5. Traverse of cross slide: 55mm

Head Stock:

1. Spindle Taper MT 2
2. Hole through work spindle 16mm
3. Spindle range 50-3200 RPM
4. Output power 0.3 KW

Tail Stock:

1. Spindle Diameter 22mm
2. Spindle Taper MT 2
3. Sleeve stroke 35mm

Feed Rate:

1. Rapid traverse 700mm/min
2. Auto feed rate 1-699mm/min

CNC System Features:

- Standard preparatory and miscellaneous codes Linear & Circular Interpolations
- 200 blocks of storable part program Program editing facility Inch/Metric Mode of input
- Absolute/incremental method of programming Feed programmable in per minute & per revolution Position preset possible
- 10 tool offset possible Subroutine programming Threading cutting operation
- Canned Turning & Threading Cycles Digital display of spindle speed
- Alarm messages during fault conditions Display of programs and axes positions RS 232 C Serial Port
- Home Cycle

Optional Features

1. Alphanumeric CRT display
2. DNC interface
3. Graphics with DNC interface
4. Magnetic cassette recorder for multiprogramming storage

Main Specifications:

1. Table Size: 420x125mm
2. Max. Job weight: 20Kg.

Axes Travel Ranges:

1. Longitudinal (Table): 200mm
2. Cross (Saddle): 125mm
3. Vertical: (Spindle): 200mm
4. Distance from spindle nose To Table top: 25-225mm
5. Distance from spindle centre line to column: 155mm
6. Spindle Centre to table top (Horizontal mode) 114-314mm

Spindle:

1. Spindle nose: ISO 30
2. Spindle motor: 0.44 Kw
3. Speed Range: 200-2000 RPM (Infinitely Variable)
4. Tool change: Manual with CNC prompting
5. No. of tools: 8 No.
6. Axis drive: Through stepper motor 1.8 step angles, on all the 3 axes.
7. Machine weight including CNC system: 200 Kg
8. Overall machine dimensions with control system: 1800x1000x800mm

EXPERIMENT 1

Aim:

To develop the given model by using CNC machine.

TOOLS REQUIRED:

1. Tool
2. Mild Steel shaft
3. Micro meter
4. Vernier

PROCEDURE:

1. For the given dimensions of the work piece to be machined write the program using G codes and M codes

2. Using the simulation software or by running the machine in test mode check the Program and if there is any error make the correction in the program.

3. Fix the work piece on the chucks.

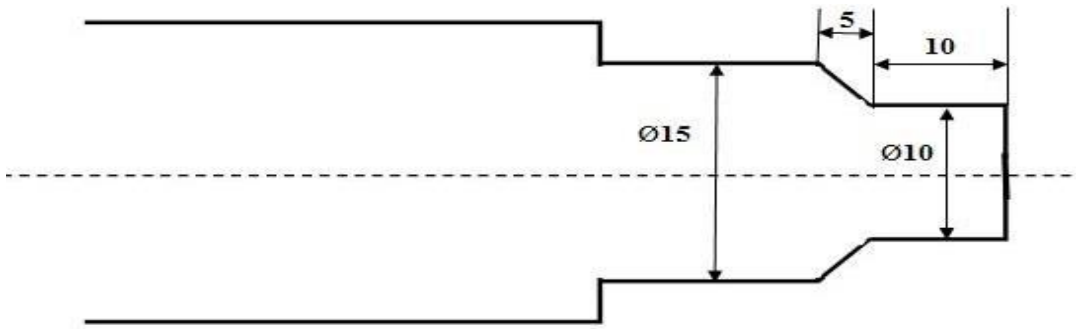
4. Move the tool to the start point of the work piece by manual mode.

5. Reset the Machine.

6. Change the machine from manual mode to single block mode or auto mode.

7. Execute the program to get the required shape of the work piece.

8. Remove the machined work piece from the chuck



Procedure: O0055;

N1;

G21G40G0;

G28 U0 W0;

T0101;

G92S3000;

G96S300M03;

G00 X31 Z1

G71 U0.5 R1

G71 P1 Q2 U0.2 W0.1 F50

N1G01 X10. 20

G01Z-10.0

N2G01X15.0 Z-15.0

G70 P1 Q2 S2000 F0.3

G28 U0 W0

M05

G97;

M30;

Result:

Thus the given job is manufactured.

Viva Question:

1. What is CAM?
2. What is CIM?
3. What is P code?
4. What is G code?
5. What are the advantages using Computer Aided Lathe Machine over Conventional Lathe?
6. What are the advantages of CAD?
7. What is mean by G21?
8. What is mean by G00?
9. What is mean by G71?
10. What is mean by G98?

EXPERIMENT 2

Aim: To develop the given model by using CNC machine.

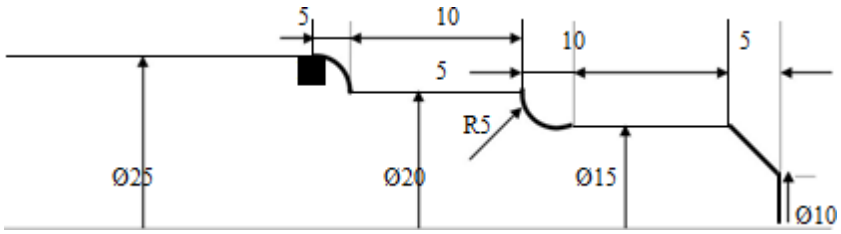
TOOLS REQUIRED:

1. Tool
2. Mild Steel shaft
3. Micro meter
4. Vernier

PROCEDURE:

1. For the given dimensions of the work piece to be machined write the program using G codes and M codes
2. Using the simulation software or by running the machine in test mode check the Program and if there is any error make the correction in the program.
3. Fix the work piece on the chucks.
4. Move the tool to the start point of the work piece by manual mode.
5. Reset the Machine.
6. Change the machine from manual mode to single block mode or auto mode.
7. Execute the program to get the required shape of the work piece.

8. Remove the machined work piece from the chuck.



Procedure:O00066;

N1;

G21G40G0;

G28 U0 W0;

M06 T0101;

G92S3000;

G96S300

G00G42 X29.0

G71 U0.5 R1.0

G71 P1 Q2 U0.3 W0.1 F0.2;

N1 G01 X10 20

G01 X15 2-5

G01 X15 2-15

G02 X15 2-20 R5

G01 X20 2-30

N2 G03 X25 2-35 R5

G70 P1 Q2 U0.0W0.0F0.1;

G40G0;

G28 U0 W0

G97;

M05

M30;

Result:

Thus the given job is manufactured.

Viva Questions:

1. What is mean by M05?
2. What is mean by U00?
3. What is mean by W00?
4. What is mean by M30?
5. What is N series?
6. What is the difference between Manufacturing and Production?
7. What is Counter?
8. What is ASP?
9. What is F30 command?
10. How you fix the tool?