K.L.N. College of Engineering

(An Autonomous Institution Affiliated to Anna University, Chennai)

l by National Assessment and Accreditation Council (NAAC)
Pottapalayam – 630612.(11 km From Madurai City) TamilNadu, India.

Department of Mechanical Engineering

Accredited by NBA, New Delhi
Approved Research Center by Anna University, Chennai
Approved Nodal Center for e – YANTRA Lab

Regulations – 2020

Odd Semester

20ME5L2
CAD / CAM Laboratory

Laboratory Manual

Lab In charge

Mr. E.V. Ganesh Babu, Assistant Professor / Mech.

Prepared by

Mr. E.V. Ganesh Babu, Asst. Prof. / Mech.

Approved by

Dr. P. Udhayakumar
HOD / Mech. Engg.

DEPARTMENT OF MECHANICAL ENGINEERING

VISION

To become a Centre of excellence for Education and Research in Mechanical Engineering.

MISSION

- **Attaining academic excellence through effective teaching learning process and state of the art infrastructure.**
- **Providing research culture through academic and applied research.**
- **Inculcating social consciousness and ethical values through co-curricular and extra-curricular activities.**

PROGRAM EDUCATIONAL OBJECTIVES (PEOs)

PEO I	**Graduates will have successful career in Mechanical Engineering and service industries.**
PEO II	**Graduates will contribute towards technological development through academic research and industrial practices.**
PEO III	**Graduates will practice their profession with good communication, leadership, ethics and social responsibility.**
PEO IV	**Graduates will adapt to evolving technologies through lifelong learning.**

PROGRAM SPECIFIC OUTCOMES (PSOs)

PSO1	**Derive technical knowledge and skills in the design, develop, analyze and manufacture of mechanical systems with sustainable energy, by the use of modern tools and techniques and applying research based knowledge.**
PSO 2	**Acquire technical competency to face continuous technological changes in the field of mechanical engineering and provide creative, innovative and sustainable solutions to complex engineering problems.**
PSO 3	**Attain academic and professional skills for successful career and to serve the society needs in local and global environment.**

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GENERAL INSTRUCTIONS FOR LABORATORY CLASSES

- Students must attend the lab classes with **ID cards**
- Boys should **"TUCK IN"** the shirts
- Students should wear **uniform only**
- **LONG HAIR** should be protected
- Any other website **should not be operated** other than the prescribed one for that day.
- POWER SUPPLY to your test table should be obtained only through the **LAB TECHNICIAN**
- **Any damage** to any of the equipment/instrument/machine caused due to carelessness, the **cost** will be fully recovered from the individual (or) group of students.

20ME5L2 **CAD / CAM LABORATORY ` L T P C**
0 0 4 2

OBJECTIVES:

- To gain practical experience in handling 2D drafting and 3D modelling software systems.
- To study the features of CNC Machine Tool.
- To gain knowledge in modern control systems.
- To know the application of various CNC machines.
- To know the application of CAM packages.

PREREQUISITE:

Course code: 20GE201, 20ME402

Course Name: Engineering graphics, Manufacturing Technology

LIST OF EXPERIMENTS

3D GEOMETRIC MODELLING

1. Introduction of 3D Modelling software

2. Creation of 3D assembly model of following machine elements using 3D Modelling software

- Flange Coupling
- Plummer Block
- Screw Jack
- Universal Joint
- Geneva mechanism
- CAM and follower mechanism
- Quick return mechanism of shaping machine

3. Manual Part Programming.

(i) Part Programming - CNC Turning Centre

- Straight, Taper and Radius Turning.
- Thread Cutting.
- Rough and Finish Turning Cycle.
- Drilling and Tapping Cycle.

(ii) Part Programming - CNC Machining Centre

- Linear Cutting.
- Circular cutting.
- Cutter Radius Compensation.
- Canned Cycle Operations.
- Pocketing

4. Design and fabrication of a component using extrusion based additive manufacturing.

5. Perform machining operation in the given work piece using Wire cut EDM

TOTAL:60 PERIODS

20ME5L2 - CAD/CAM LABORATORY

LIST OF EXPERIMENTS

S.No.	Name of the Experiment	CO
1.	Draw 3D model using basic commands of 3D modelling software	1
2.	Draw 3D model using basic commands of 3D modelling software	1
3.	Draw 3D part model of flange coupling and assemble them using 3D modelling software	2
4.	Draw 3D part model of Screw Jack and assemble them using 3D modelling software	2
5.	Construct CNC part program for the given profile using box turning cycle and also check the simulation	3
6.	Construct CNC part program for the given profile using multiple turning cycle and also check the simulation	3
7.	Construct CNC part program for the given profile using grooving and thread cutting cycle and also check the simulation	3
8.	Construct CNC part program for the given profile using Drilling and boring cycle and also check the simulation	3
9.	Construct CNC part program for the given profile using Linear and circular interpolation and also check the simulation	4
10	Construct CNC part program for the given profile using circular pocketing and also check the simulation	4
11.	Construct CNC part program for the given profile using Rectangular pocketing and also check the simulation	4
12.	Construct CNC part program for the given profile using canned drilling cycle and also check the simulation	4
13.	Create 3D model using 3D printer (Demo only)	5
14.	Create cutting for the given profile using wire cut EDM	6

CONTENTS

S. No.	*Date of Experiment*	*Name of the Experiments*	*Page No.*	*Signature of the Staff*
1.				
2.				
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INTRODUCTION OF 3D MODELING SOFTWARE

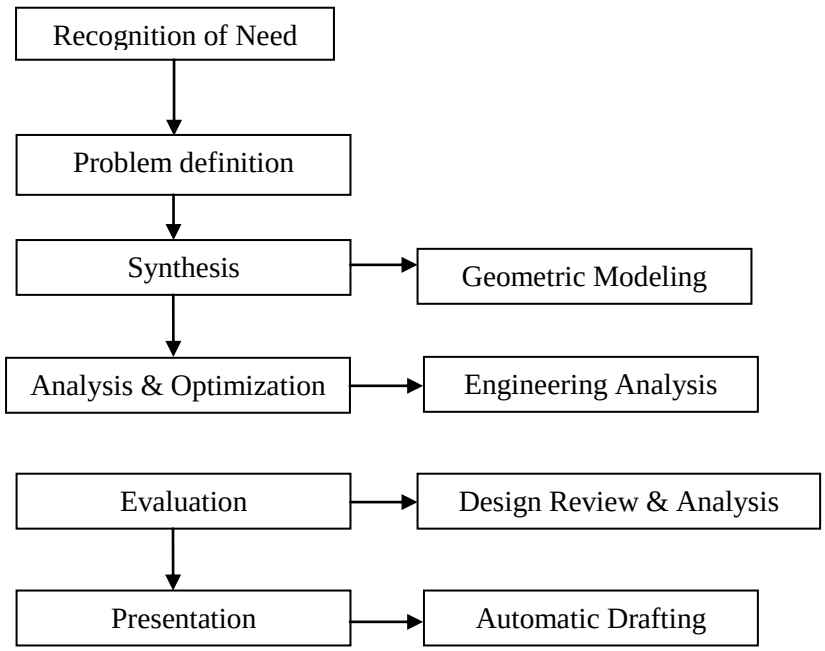
COMPUTER AIDED DESIGN

1.1 Introduction of CAD

Computer Aided Design is the process of developing and using computer assisted design tools in the design process. The advent of computers has contributed to significant advance in calculation, data handling and utilization applications. The ability to use the computers in these application areas enhances the capability of the design team significantly. Drafting and geometric modeling play significant roles in CAD. The module therefore concentrates on the geometric general design process with specific consideration to drafting and geometric modeling.

Role of computer in design

As manual design process has several risk factors including human fatigue and the evolution of design based on his previous experience. With the advent of computer and the development in the field of computer graphics, various design & manufacturing process takes place new faster rate with minimum or optimum error. The below figure shown the implementation of computer in design:



Implementation of computer in the design stage becomes the subset of design process. Once the conceptual design materializes in the designer mind the geometric model starts by the appropriate CAD software. The choice of geometric model to CAD is analogous to the choice. The various design related tasks which are performed by

a) modern computer-aided design system can be grouped into four functional areas :

1. Geometric Modeling
2. Engineering Analysis
3. Design review and evaluation
4. Automated drafting.

Geometric modeling:

It is concerned with the computer compatible mathematical description of the geometry of an object. The mathematical description allows the image of the object to be displayed & manipulated on a graphics terminal through signals from the CPU of the CAD system. The software that provides geometric modeling capabilities must be designed for efficient use both by the computer & the human designer. During the geometric modeling computer converts the command into a mathematical model, stores it in the computer data files, and display it as an image on the CRT screen. Object can be represented by geometric model by wireframe, surface model or solid model. Another feature of CAD system is color graphics capability. By means of color, it is possible to display more information on the graphics screen.

During this geometric modeling process the computer converts the commands into mathematical model, stores it in the computer data files and displays it as an image on the screen. The model can be subsequently being called from the data files for review, analysis or alteration. The most advanced method of geometric modeling is solid modeling in three dimensions. This method uses solid geometry shapes called primitives to construct the object.

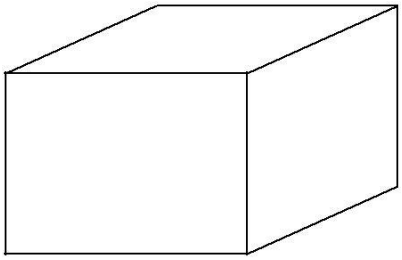
Basically there are three types of modeling, they are

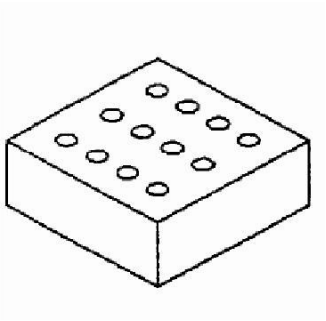
a. Wire Frame Modeling
b. Surface Modeling
c. Solid Modeling

Wire Frame Modeling:

This is the basic form of modeling; here the objects drawn will be simple but more verbose, geometric model that can be used to represent it mathematically in the computer. It is sometimes referred as a stick figure or an edge representation of the object. Typical CAD/CAM system provides users with possibly three modes to input coordinates: Cartesian, Cylindrical or Spherical. Each mode has explicit or implicit inputs. Explicit input could be absolute or incremental coordinates. Implicit input involves user digitizes...A wire frame model consists of points, lines, arcs, circles &curves. Early wire frame modeling techniques developed in 1960's were 2-dimensional. They are not

centralized &associative. Later in 1970's the centralized, associative database concepts enabled modeling of 3D objects as wire frame models that can be subject to 3-dimensional transformations.





WIRE FRAME ENTITIES

Wire frame Entities are divided into 2 types are:

a. Synthetic Entities---------- Splines & Curves

b. Analytic Entities---------- Points, lines, Circles, arcs, conics, fillet, chamfer

Applications:

1. Two-dimensional drafting.
2. Numerical control tool path generation.

Advantages:

1. It is simple to construct model.
2. Less computer memory to store the object.
3. CPU time to retrieve, edit or update a wireframe model is less.
4. Doesnot require extensive training.

Disadvantages:

1 It is ambiguous representation of real object.

2. It lack in visual coherence and information to determine the object.

3. User or terminal time needed to prepare & or input data increases with complexity of object.

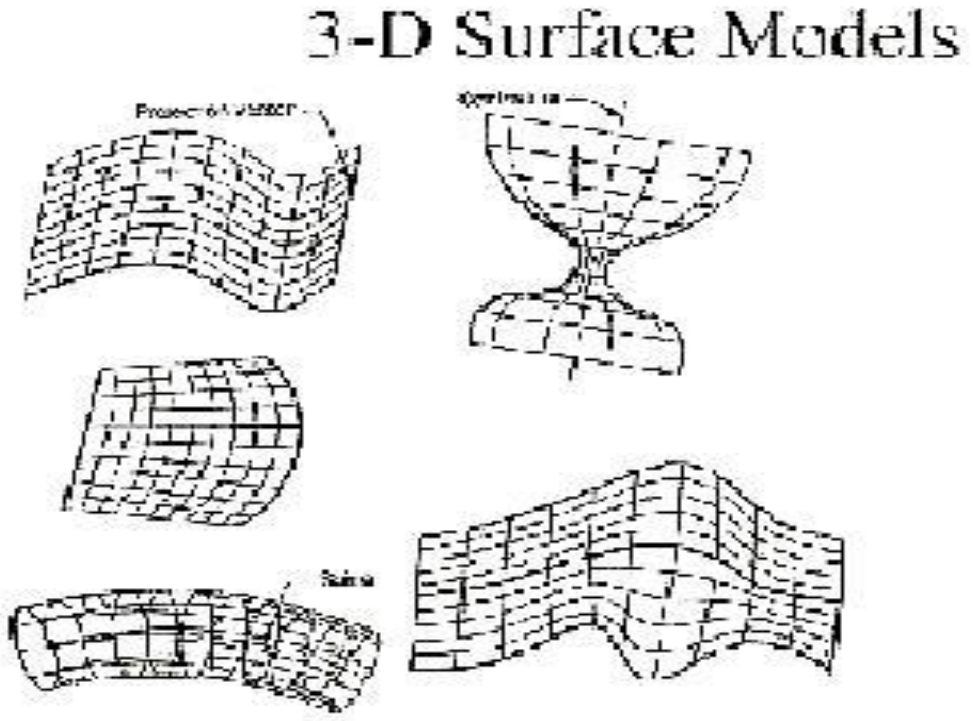
4. Inability to detect interference between components.

5. No facility for automatic shading.

Difficult in calculating Physical properties like Mass, surface area, Centre of gravity

Surface modeling:

A surface model of an object is more complete and less ambiguous representation than it wire frame model. It is also richer in associated geometric contents, which make it more suitable for engineering and design applications. Surface model takes one step beyond wire frame models by providing information on surfaces connecting the object edges. Creating a surface has some quantitative data such as point & tangents & some qualitative data like desired shape & smoothness. Choice of surface form depends on type of application.

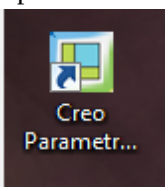


Solid modeling:

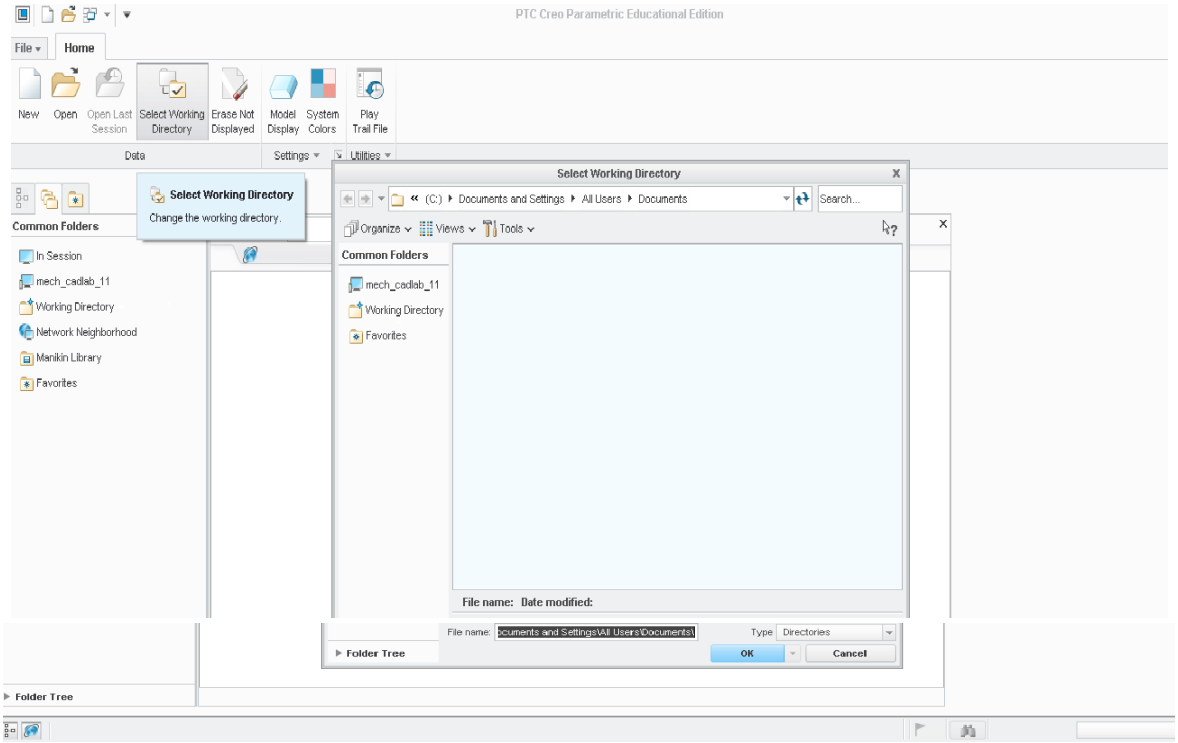
A solid model of an object is more complete representation than its surface model. It is unique from the surface model in topological information it stores which potentially permits functional automation and integration. Defining an object with the

solid model is the easiest of the available three modeling techniques. Solid model can be quickly created without having to define individual locations as with wire frames. The completeness and unambiguity of solid models are attributed to the information that is related database of these models stores **(Topology It determine the relational information between objects.)** To model an object completely we need both geometry & topological information. Geometry is visible, whereas topological information are stored in solid model database are not visible to user. Two or more primitives can be combined to form the desire solid. Primitives are combined by Boolean Operations

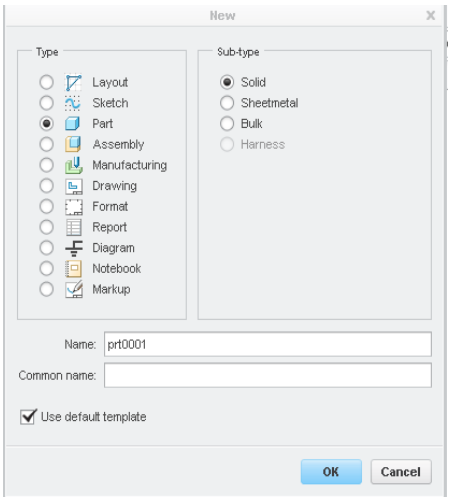
1. Open Creo Parametric 3.0



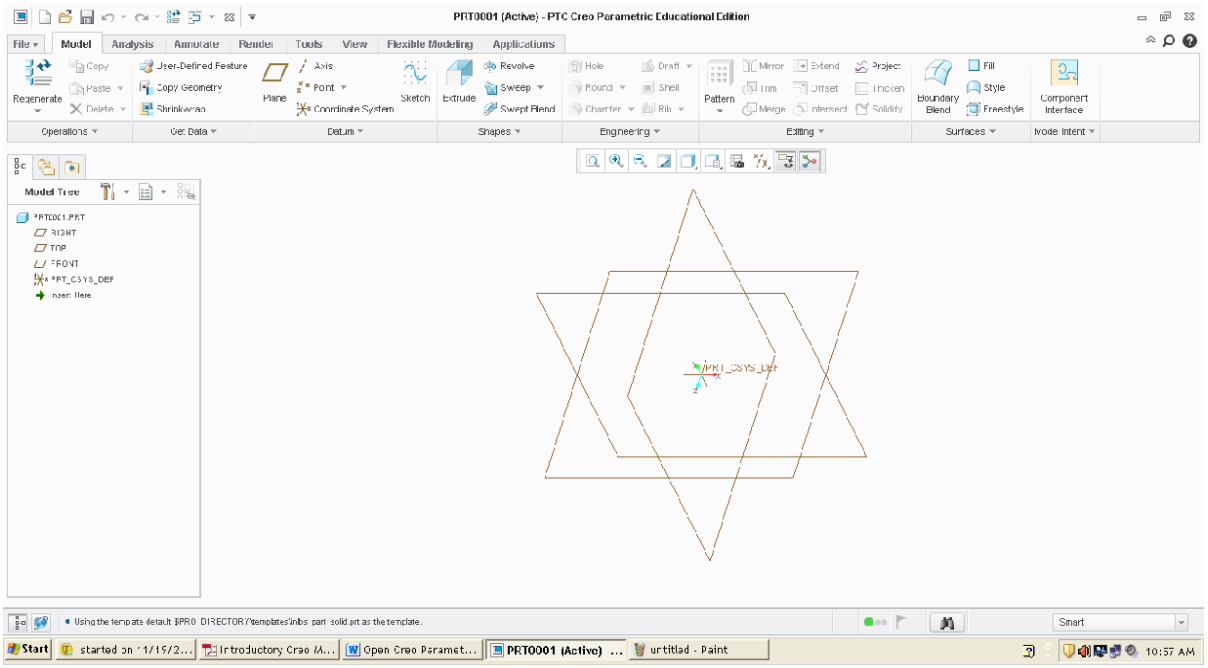
2. Hit Select Working Directory on the top bar and select whatever folder you want your new part to go into.



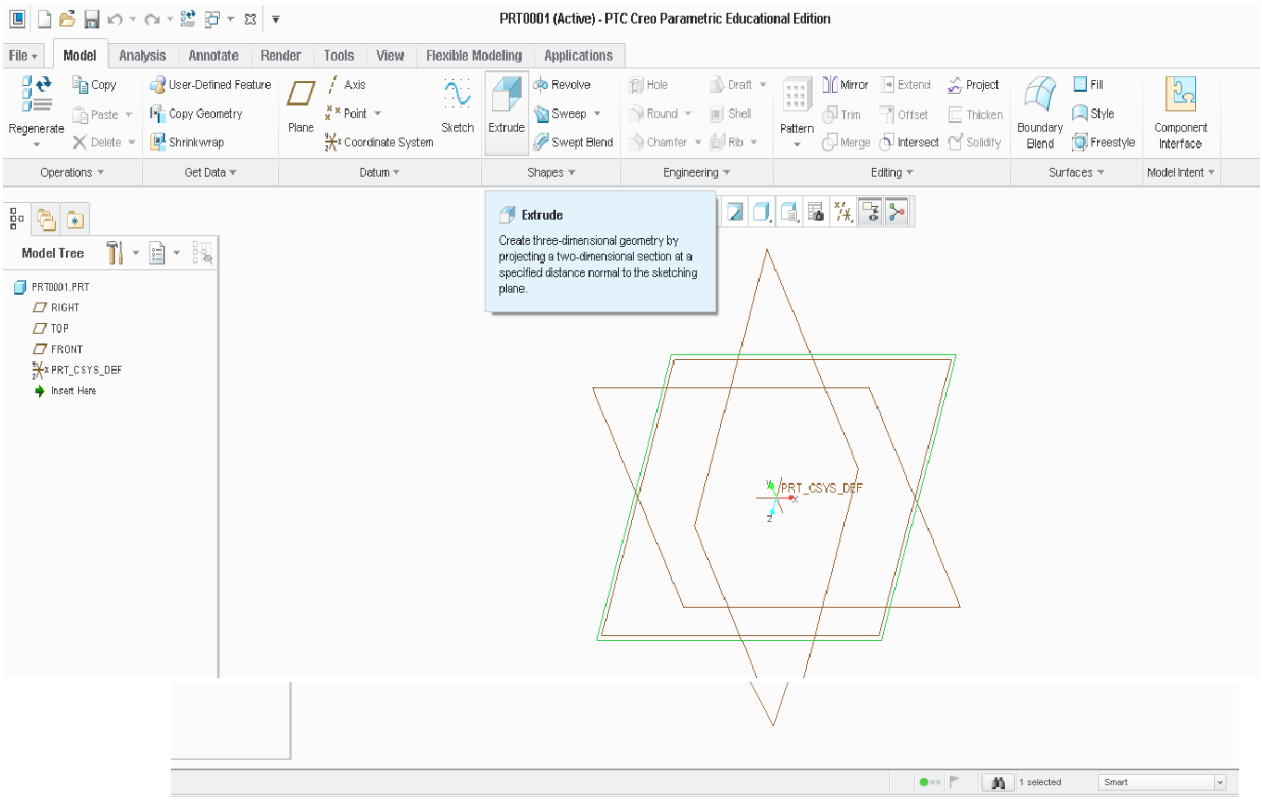
3. Next hit the New Button, make sure the type is set to part. Change the name to whatever you want to name your part and hit OK



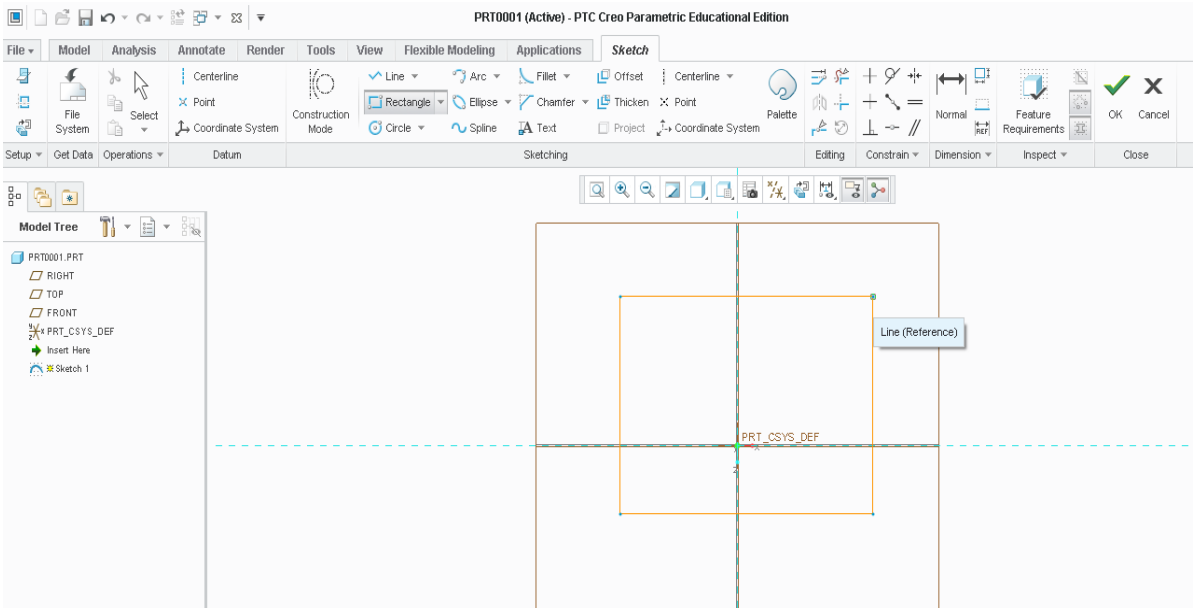
Your screen should now look like this. Hit the View tab along the top ribbon and hit the plane tag display button so you can see the names of the planes.



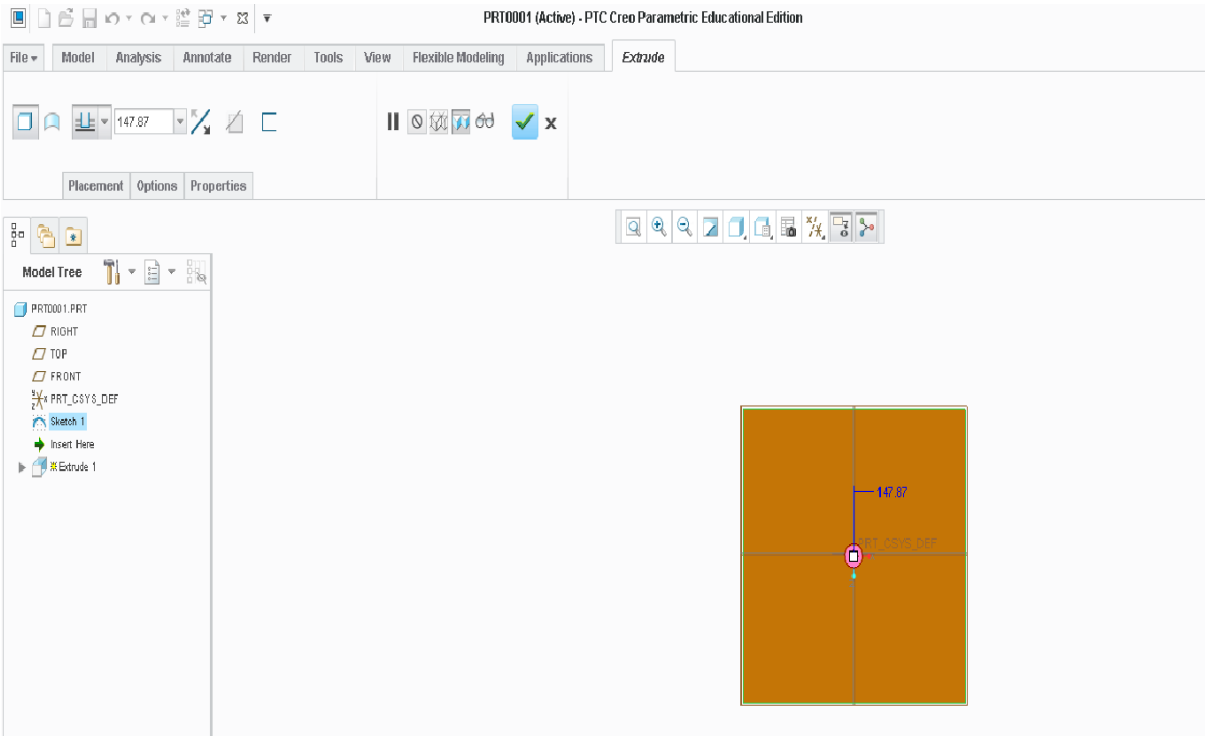
4. Next, select the Top plane, return to the model tab, and hit the extrude button

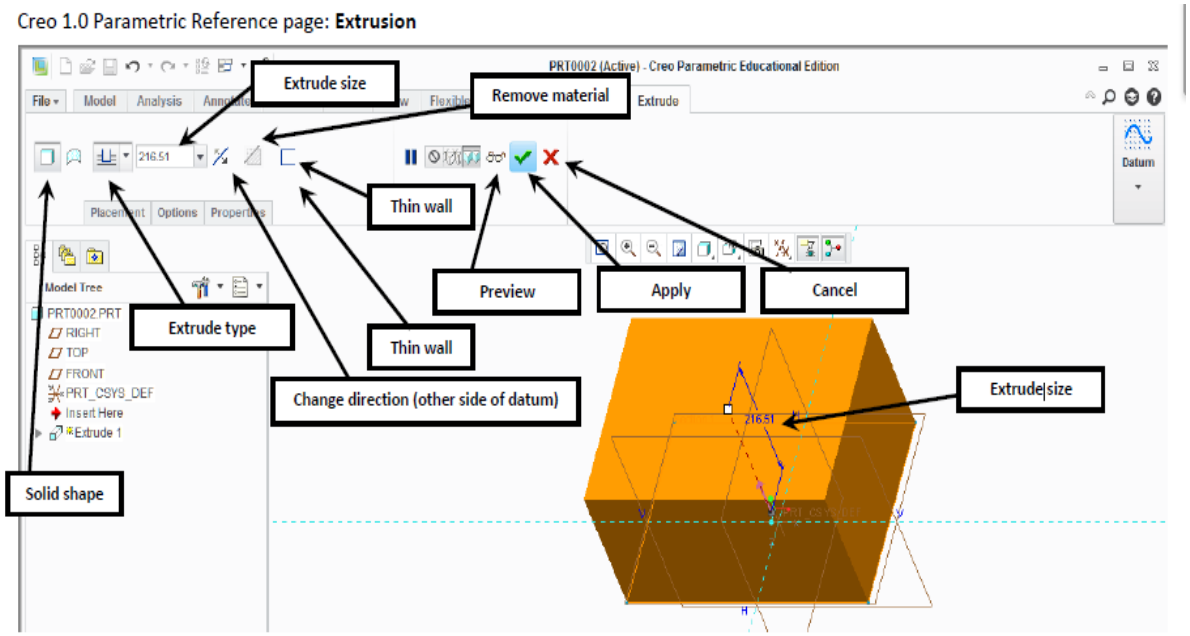


5. Then you should be in sketch mode. Hit the sketch view button on the small ribbon in the work area. So that you are viewing the top plane in 2D. Then select Rectangle in the sketch ribbon and click and drag a rectangle in your workspace .



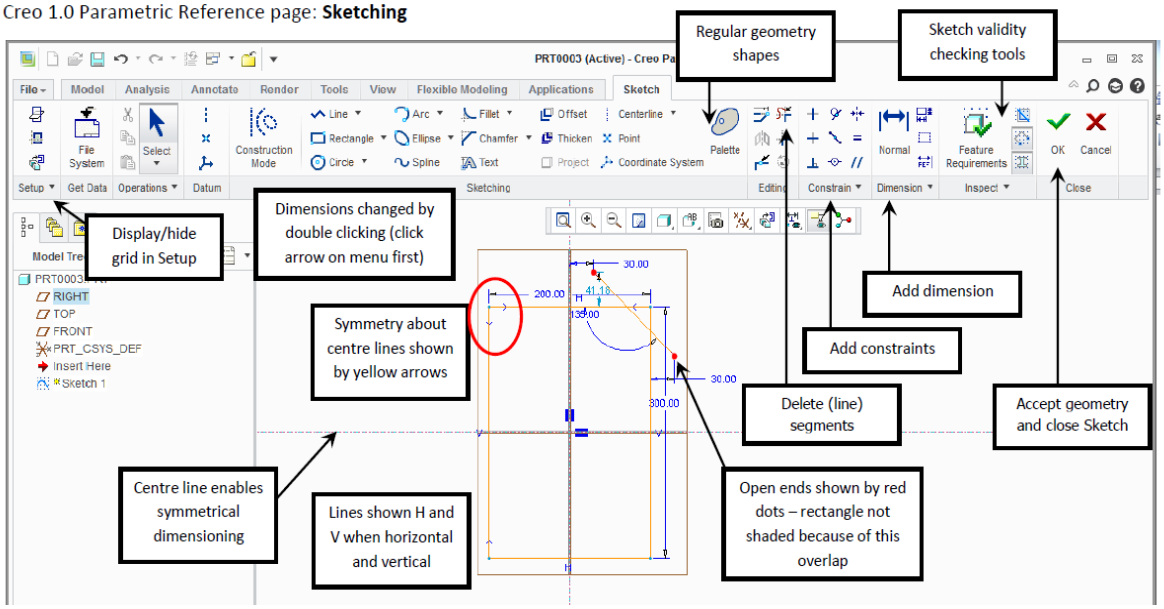
6. Once you click again and then click the select button you will see that inside the rectangle turns orange, and dimensions appear. Change the dimensions so that it has a length and width of 6.00 and the dimensions to the centerlines are 3.00. This is where you start to have freedom. If you want to make something smaller you can, but this is the maximum size you can have for your part. Once you are satisfied hit the check mark to finish the sketch.





1. Select Extrude from top menu (Shapes)
2. Click on required datum plane (e.g. TOP) or a flat surface on an existing model
3. Draw required shape (e.g. rectangle) – centre lines can be used for symmetrical shapes
4. Click green tick to exit from sketcher
5. Select extrusion type (e.g. Blind or Symmetric) – extrusion can be removed from existing solids
6. Drag extrusion size (or type value in Dashboard) and click green tick to apply
7. Save the part

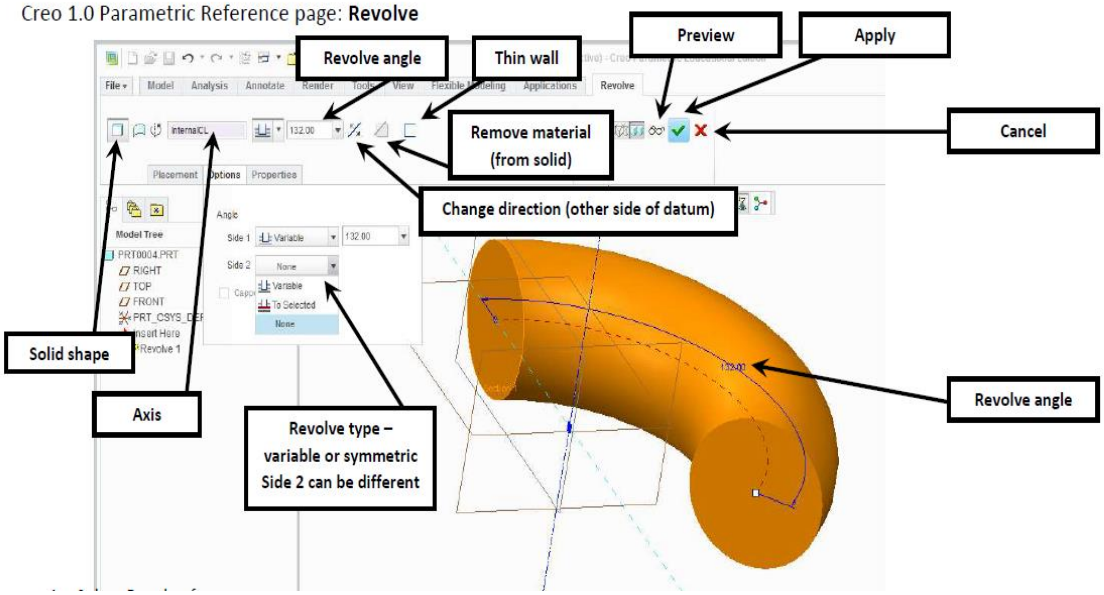
Creo 1.0 Parametric Reference page: **Sketching**



Delete line segments tool can be used in two ways:
(1) Click on line or part of line, (2) Drag the red line over all lines to delete

2. Inspect Sketch Tools show whether the sketch will work with 3D feature or not. Shape is color filled when OK.
3. Constraints (equal length etc.) can be added (or removed with the delete key)
4. For construction lines click Construction Mode before drawing
5. All the above applies to internal and external sketches.

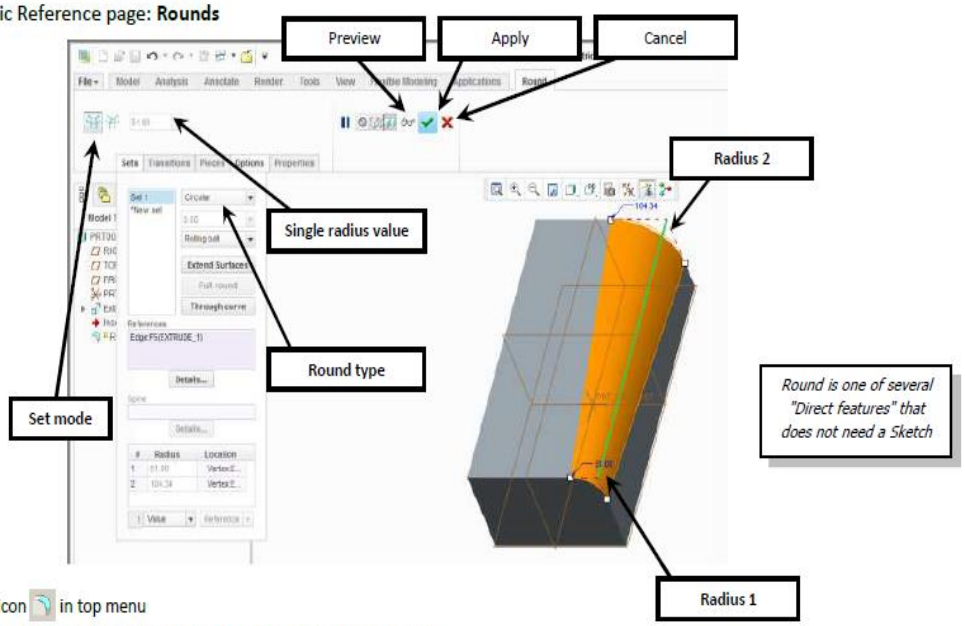
Creo 1.0 Parametric Reference page: **Revolve**



Select Revolve from top menu

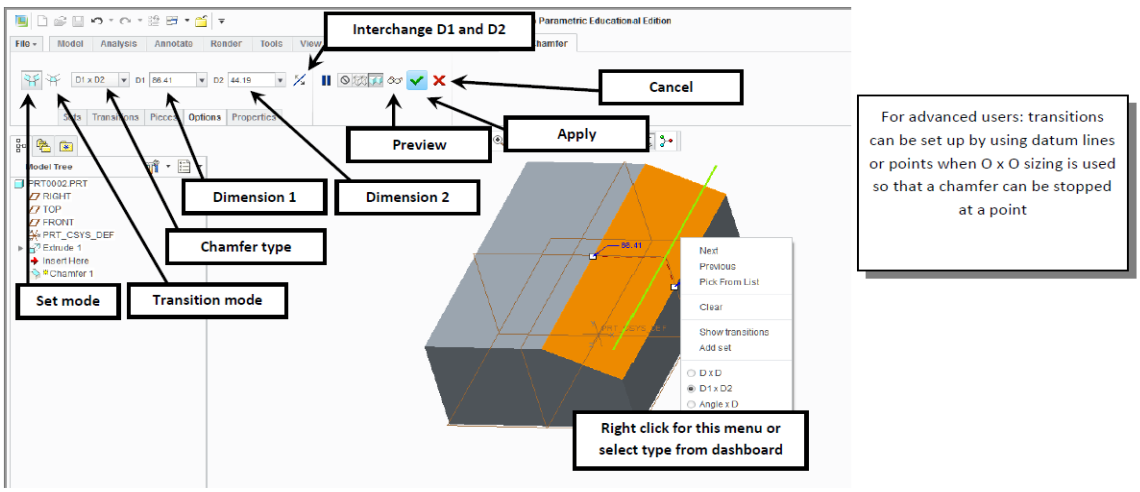
1. Click on required datum plane (e.g. RIGHT) or a plane surface of an existing solid
2. Draw required shape (e.g. circle) which must be closed - use Sketch Validity Tools to check
3. Add a Geometry centerline (or a datum axis must have been created first) - the existing edge of a solid can also be used as axis
4. Click green tick
5. Change the revolve angle (default is 360°) by entering value or drag (white) handle and click green tick to apply
    a. Save the part .

Creo 1.0 Parametric Reference page: **Rounds**



1. Select Round icon in top menu
2. Select an edge on your part - hold down CTRL to add further edges to the set
3. Drag white marker to radius required or enter value
4. To add second (or third) radius, right-mouse-click on the white drag handle and **Add Radius** – drag size of second radius
5. The position of third radius can be dragged along the edge to position required
6. To return to plain round, right-click on green line and **Make Constant** or open Sets tab to delete radius
7. Additional round sets (with single or multiple radii) can be added to other edges - click on **New set**, enter radius required
8. Click green tick to apply

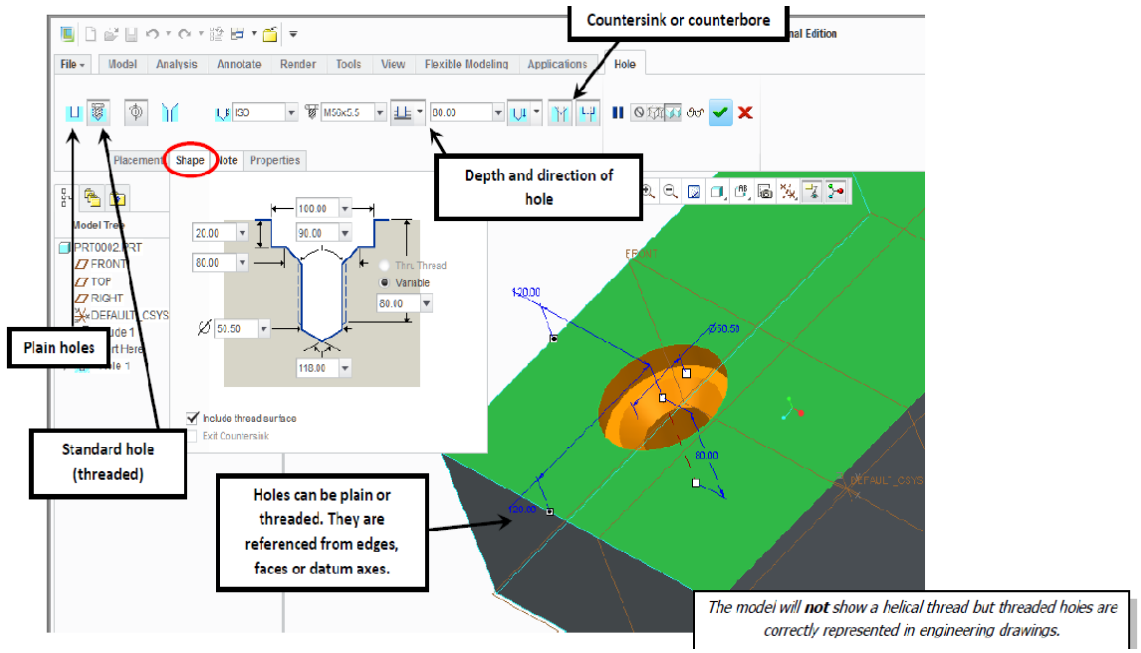
Creo 1.0 Parametric Reference page: **Chamfers**



1. Select the chamfer icon from the top menu
2. Select an edge on the solid part - hold CTRL to add further edges to the set

3. D x D is default for equal sizes of chamfer.
4. Use white drag handles to get the sizes D1 and D2 (or enter in the boxes above after selecting type)
5. Save the part

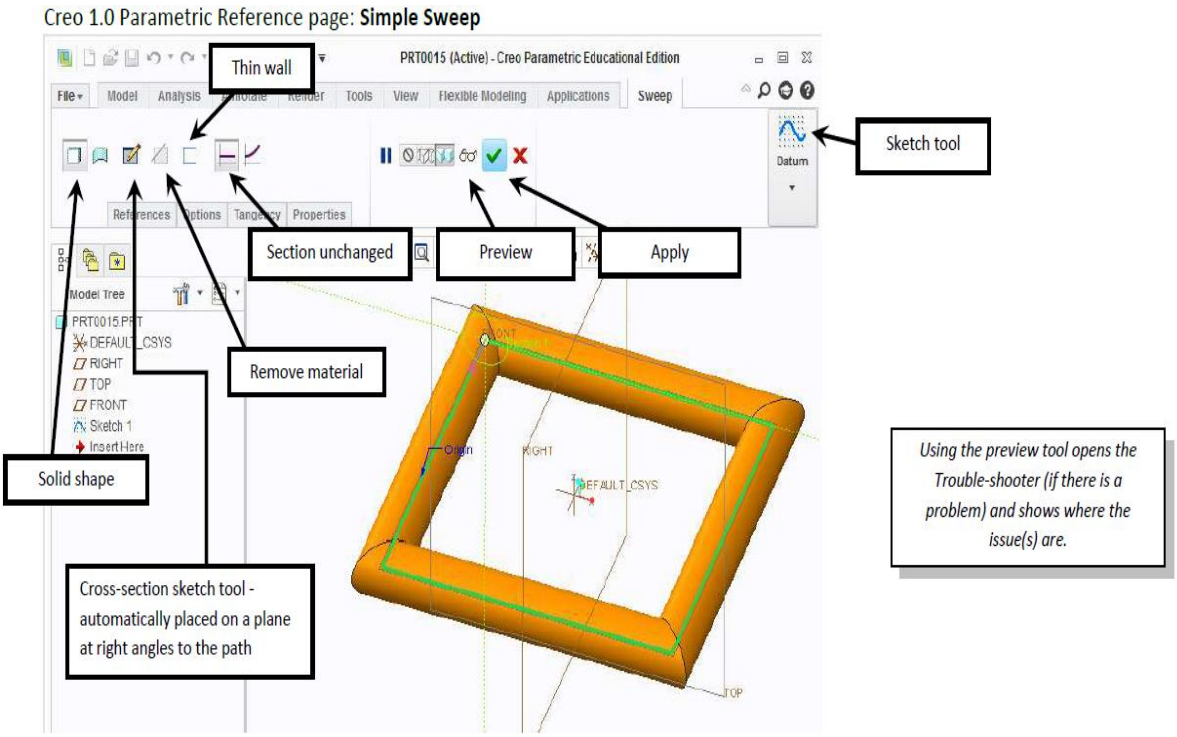
HOLE



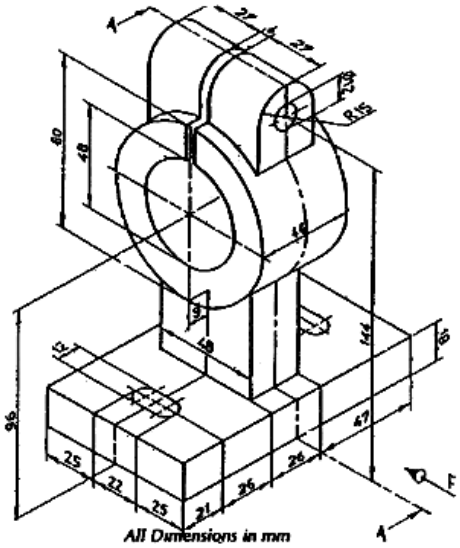
Plain holes can also be made using the Extrude feature. Holes can be made in any solid part.

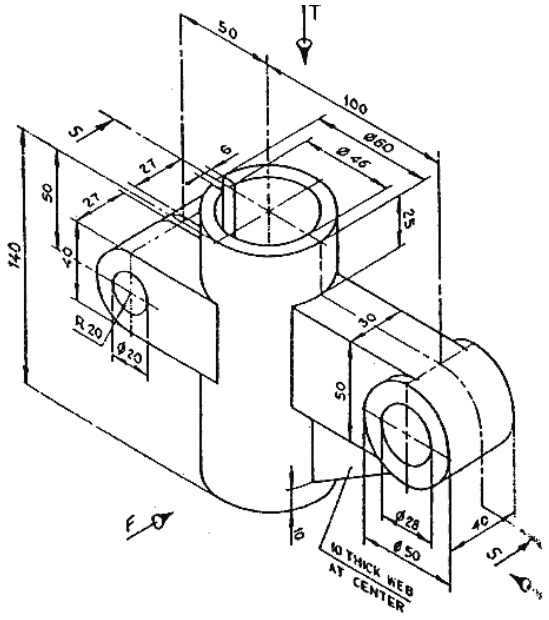
1. Select the Hole icon on the right hand menu
2. Select a face of your solid
3. Drag the green markers to faces (provides dimensional references), drag the white marker to place the hole centre
4. Drag or enter the hole diameter and hole depth
5. For threaded holes click the icon and select the required thread size and countersink, counterbore etc. – add measurements using the Shape tab.

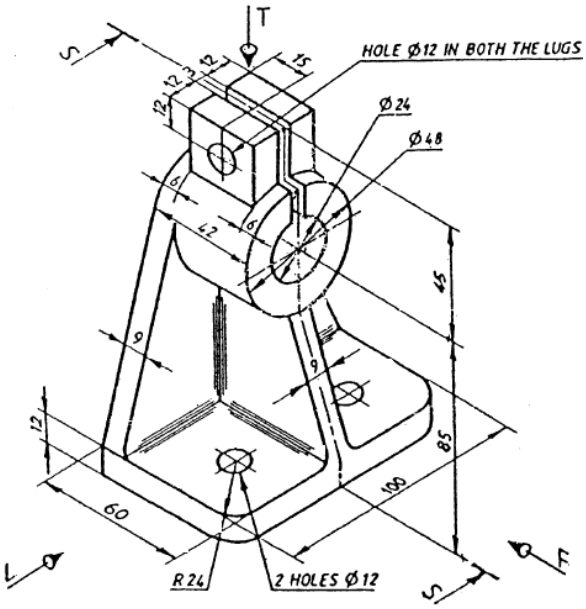
SWEEP

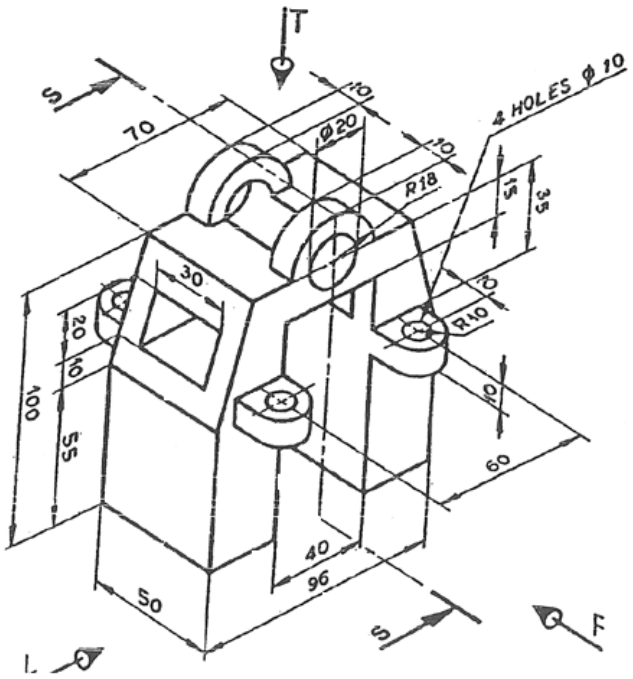


1. First draw the path as an **external sketch** on the required datum plane (e.g. TOP) or on a surface of an existing part - click green tick - leave the line highlighted (don't click on the window)
2. Select the Sweep feature in the top menu (Shapes)
3. For a simple sweep, keep Solid and Section remains unchanged (defaults)
4. Click on the **Create or edit sweep section tool** and draw the cross-section (e.g. a circle) at or near the end of the sketch line
5. Click green tick to accept the sketch and click green tick to apply
6. Save the part


All Dimensions in mm


All Dimensions in mm


HOLE Ø12 IN BOTH THE LUGS
Ø24
Ø48
R24
2 HOLES Ø12


4 HOLES ϕ10
Ø20
R18
R10

Ex. No : 1 Date :

EXERCISES ON 3D PART SOLID MODELING

AIM:

To create a given objects using the feature based parametric techniques as per the dimensions given.

HARDWARE REQUIRED:

1. CPU with Pentium IV processor.
2. Display with highest 32 bit color monitors with screen resolution 1024 x 768 pixels.
3. Scroll mouse

SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. PTC/Creo

PROCEDURE:

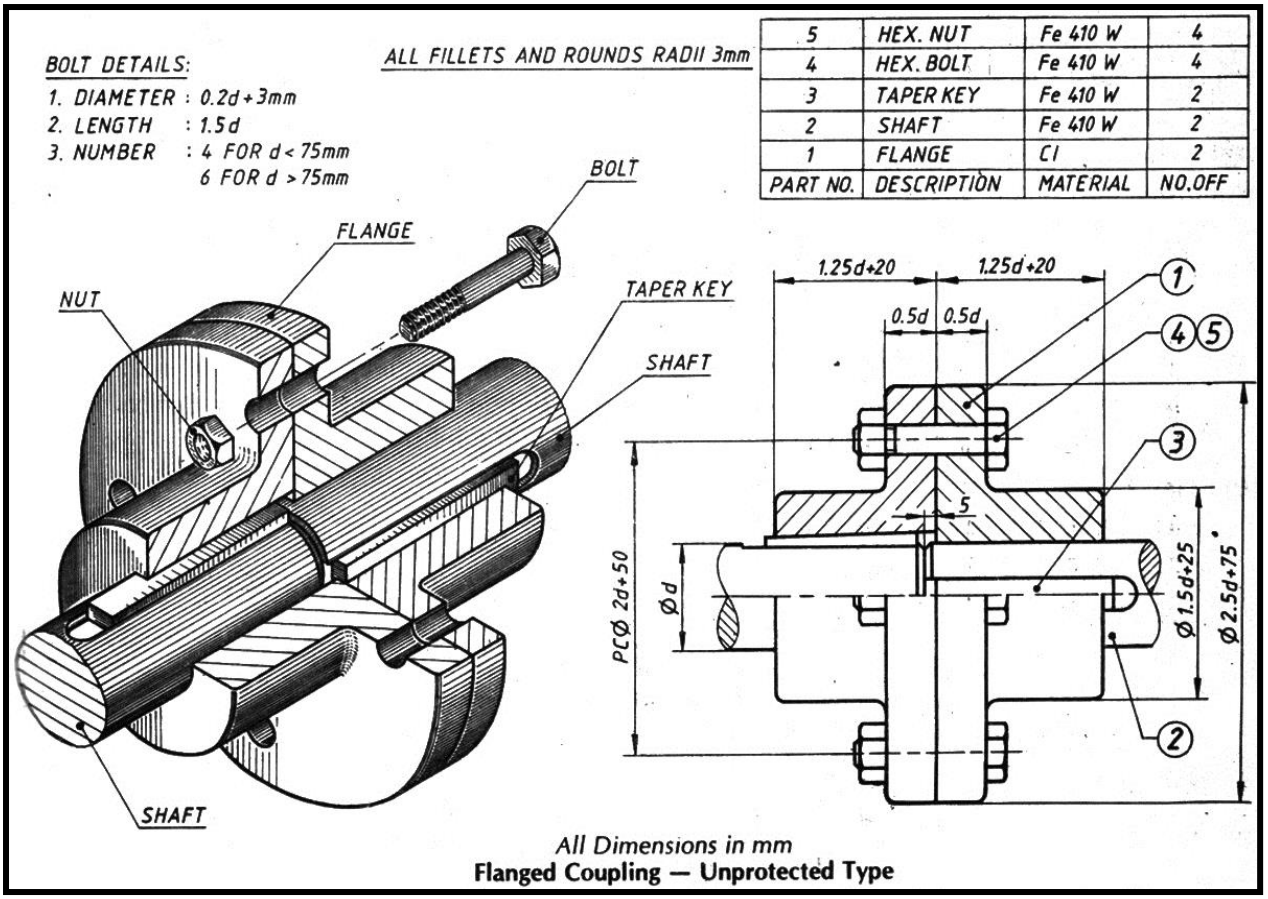
1. Identify various parts to be created.
2. First enter into part environment and create the given parts with an extension of *.prt using protrusion tool.
3. Select the sketch tool and then select the coincidental plane option and select any one of the standard 3 planes (i.e. front, right &top).
4. Create the cross-section profile as a closed one using the 2D commands available after completing the sketch, click open or return button and then click Done button.
5. For creating other parts, select sketch both parallel plane option or plane by 3points option and then select the required plane.
6. Do the protrusions by using protrusion command and the revolution by revolved protrusion command.
7. For constructing holes and cutout, used hole command and cutout command.
8. If we use hole command, change the diameter of the hole by using modify menu, resize hole option.
9. Use revolved cutout command whenever needed.
10. Use the distance between option to maintain accurate distance between one edge and other edge or between one edge and to center the hole.
11. After constructing each part save it as a separate part file with the extension of *. prt.
12. Then enter into the frame environment and create a frame with title block and save with an extension of *.frm
13. Then enter into the drawing environment with the created frame and creating all the models in the drawing sheet and save with an extension of *.drw

INFERENCE:

- Practical experience in making 2D sketches (using line , circle , radius , mirror , chamfer & Plane selection etc)
- Basic 3d commands like, revolve, extrude, pattern through the exercise.

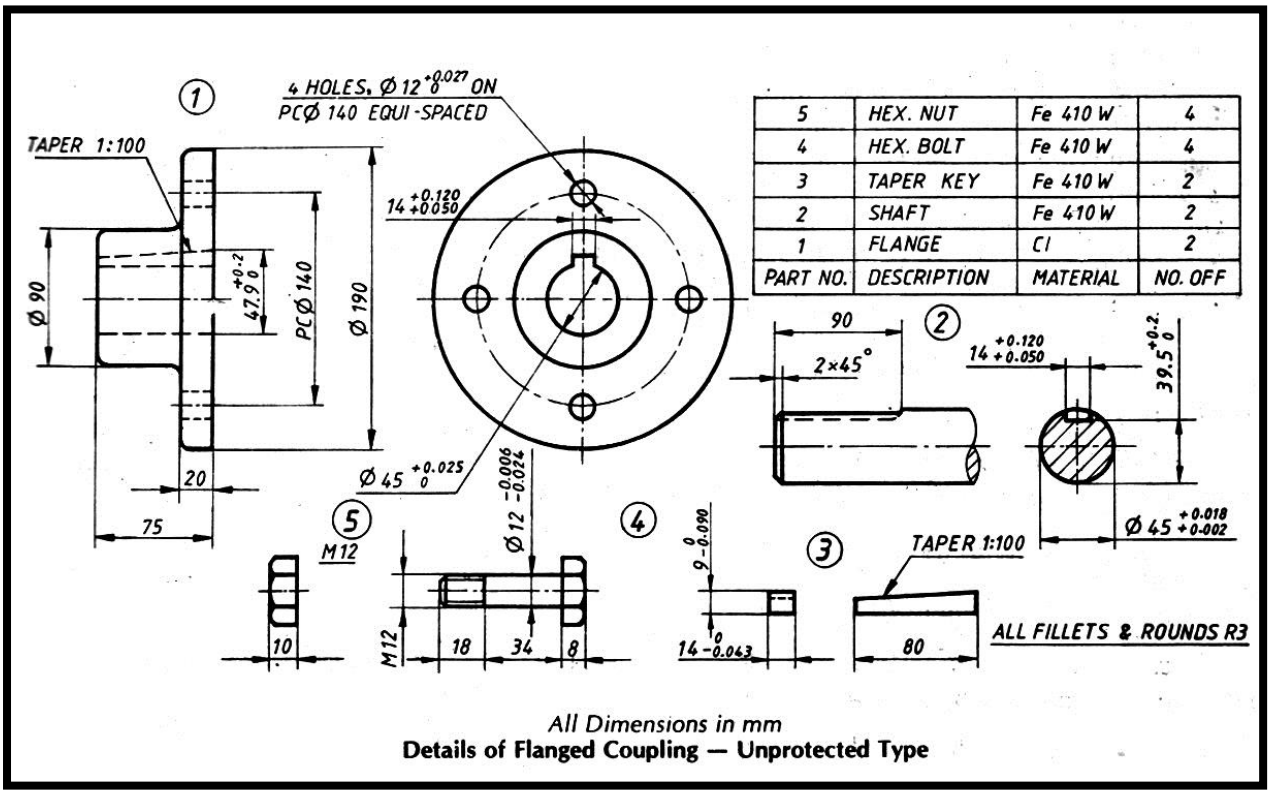
RESULT:

Thus the given 3D part models has been created by using the PTC/Creo software.



All Dimensions in mm

Flanged Coupling — Unprotected Type



All Dimensions in mm

Details of Flanged Coupling — Unprotected Type

Ex. No : 2 Date :

FLANGE COUPLING

AIM:

To create a Flange Coupling 2D part drawings and assembly as a 3D solid model using PTC/Creo Software

HARDWARE REQUIRED:

1. CPU with Pentium IV processor.
2. Display with highest 32 bit color monitors with screen resolution 1024 x 768 pixels.
3. Scroll mouse

SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. PTC/Creo

PROCEDURE:

1. Identify various parts to be created.
2. Enter into part environment and create the main part of the assembly.
3. Identify whether the main part or the first part environment is to be created by protrusion or by revolution.
4. Select the sketch tool, then coincidental plane option, from these select any one of the standard 3 planes (i.e. front, right &top).
5. Create the cross-section profile as a closed one by using the available 2D commands , then the click finish button for completing for completing the profile. .
6. Select sketch both parallel plane option or plane by 3points option for creating other parts and then select the required plane.
7. Do the protrusions by using protrusion command and the revolution by revolved protrusion command.
8. Use hole command and cutout command for constructing holes and cut out.
9. Change the diameter of the hole by using modify menu or resize hole option by using hole command.
10. Use revolve cutout command whenever in need.
11. Use the distance option to maintain accurate distance between one edge and other edge (or) one edge and center the hole.
12. Save it in separate part file with extension of *. prt. Similarly save it other parts.
13. Enter into the assembly environment, create the 3D model apply assembly constrains available like planer, design, mate, axial align, connect etc.
14. save with an extension of *.asm the various parts of drg.
15. Enter into the frame environment with title block and save with an extension of *.frm
16. Finally , Enter into the drawing environment with the created frame and all the models in the drawing sheet and save with an extension of *.drw

INFERENCE:

- Practical experience in making 2D flange coupling parts of flange, shaft, taper key, hexagonal bolts & nuts and 3D assemble model for flange coupling through the exercise.

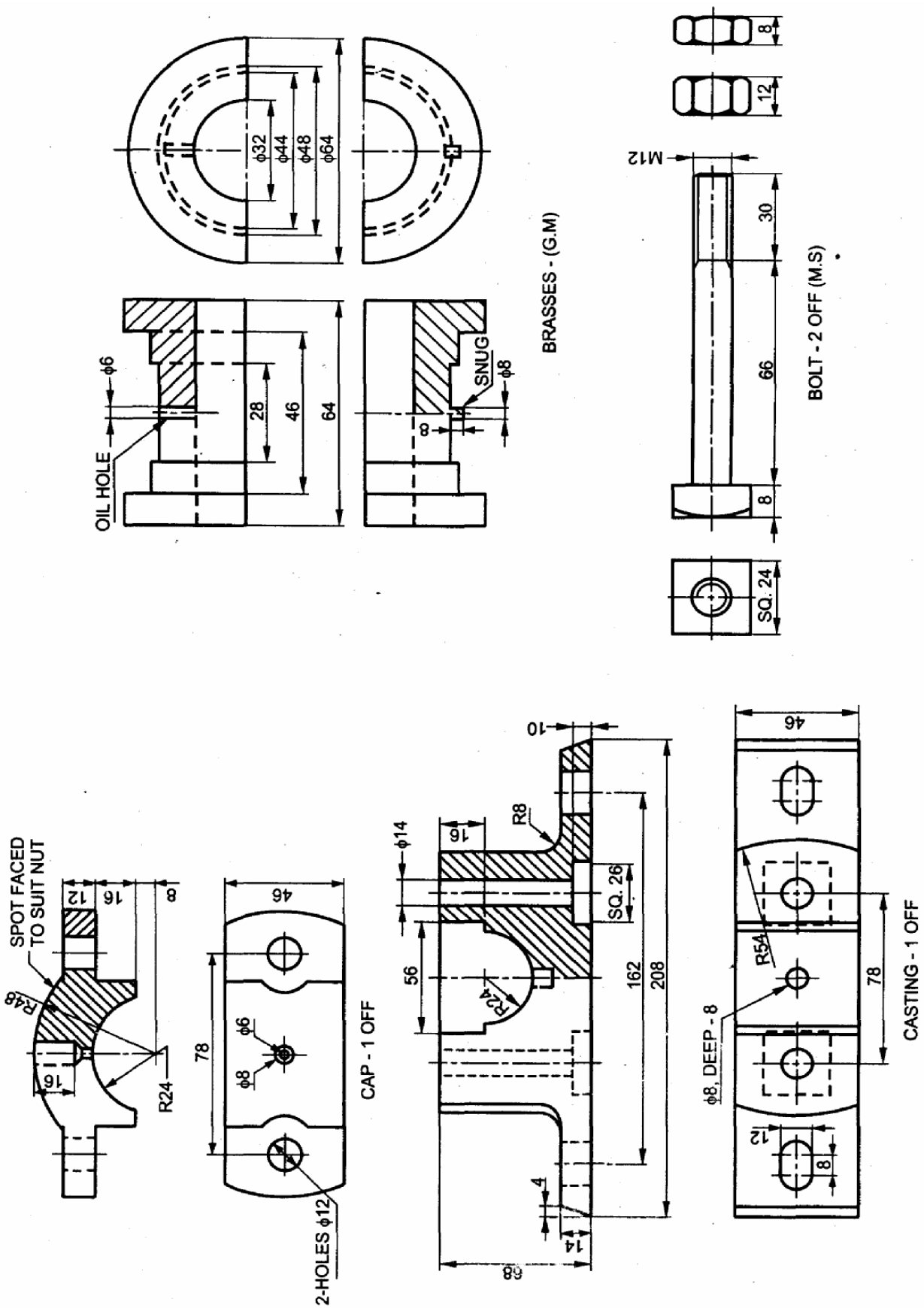
- Plane selection, revolve, blend cut, extrude, helical sweep, pattern, assembly features like mate, axial align, connect and move through the exercise.

APPLICATIONS:

- Design for heavy load and industrial equipment.
- Use in a drive shaft of a car and truck.
- Connect lengths of industrial pipe lines.
- Flexible hose connections.

RESULT:

Thus the 3D assembly of Flange Coupling has been created by using the PTC/Creo software.


BRASSES - (G.M)
OIL HOLE
SNUG
BOLT - 2 OFF (M.S)
SQ. 24
SPOT FACED TO SUIT NUT
CAP - 1 OFF
2-HOLES
CASTING - 1 OFF

Ex. No : 3 Date :

PLUMMER BLOCK

AIM:

To create a Plummer Block 2D part drawings and assembly as a 3D solid model using PTC/Creo Software

HARDWARE REQUIRED:

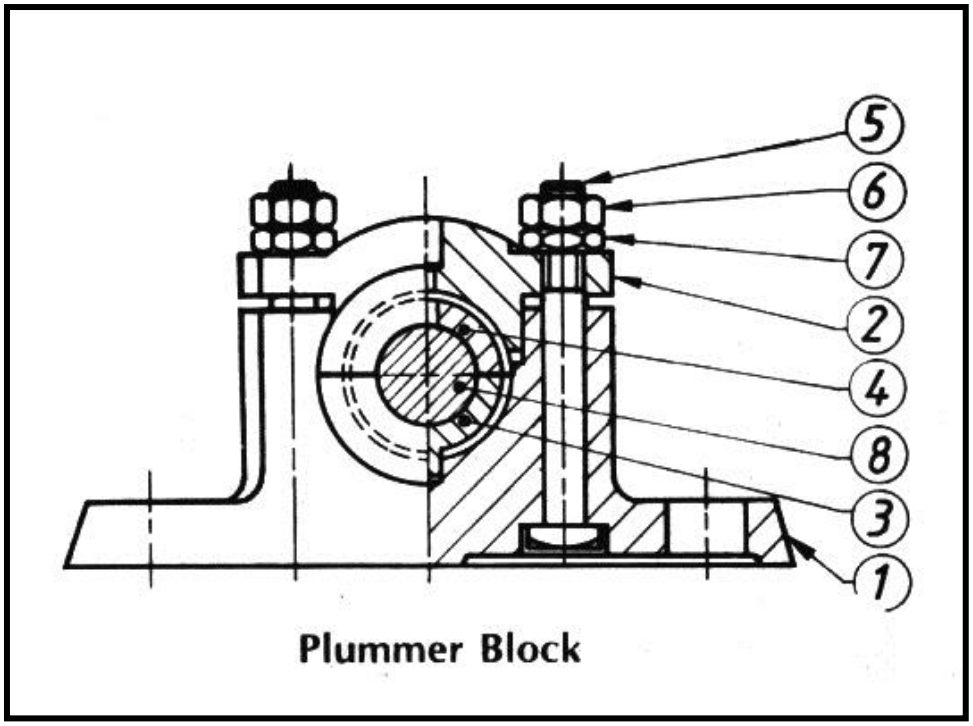
1. CPU with Pentium IV processor.
2. Display with highest 32 bit color monitors with screen resolution 1024 x 768 pixels.
3. Scroll mouse

SOFTWARE REQUIRED:

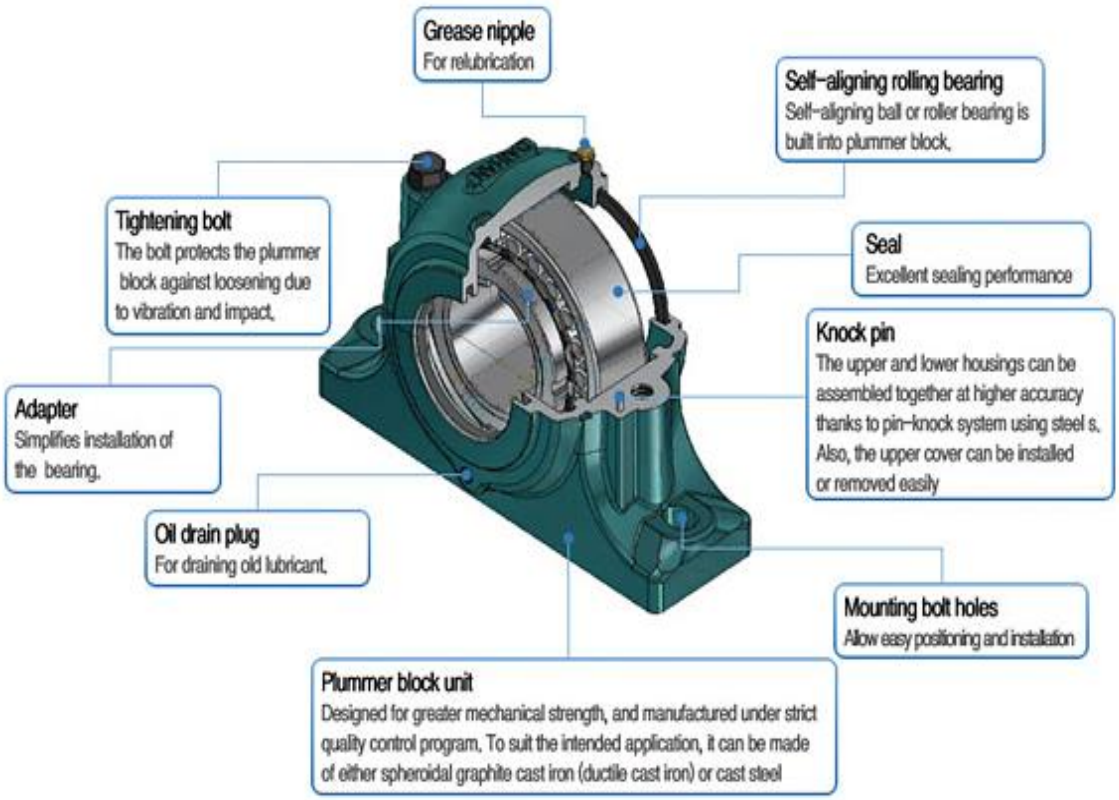
1. Windows XP or Windows 7 operating system
2. PTC/Creo

PROCEDURE:

1. Identify various parts to be created.
2. Enter into part environment and create the main part of the assembly.
3. Identify whether the main part or the first part environment is to be created by protrusion or by revolution.
4. Select the sketch tool, then coincidental plane option, from these select any one of the standard 3 planes (i.e. front, right &top).
5. Create the cross-section profile as a closed one by using the available 2D commands , then the click finish button for completing for completing the profile. .
6. Select sketch both parallel plane option or plane by 3points option for creating other parts and then select the required plane.
7. Do the protrusions by using protrusion command and the revolution by revolved protrusion command.
8. Use hole command and cutout command for constructing holes and cut out.
9. Change the diameter of the hole by using modify menu or resize hole option by using hole command.
10. Use revolve cutout command whenever in need.
11. Use the distance option to maintain accurate distance between one edge and other edge (or) one edge and center the hole.
12. Save it in separate part file with extension of *. prt. Similarly save it other parts.
13. Enter into the assembly environment, create the 3D model apply assembly constrains available like planer, design, mate, axial align, connect etc.
14. save with an extension of *.asm the various parts of drg.


5
6
7
2
4
8
3
1
Plummer Block

Structure


Grease nipple
For relubrication
Self-aligning rolling bearing
Self-aligning ball or roller bearing is built into plummer block.
Tightening bolt
The bolt protects the plummer block against loosening due to vibration and impact.
Seal
Excellent sealing performance
Knock pin
The upper and lower housings can be assembled together at higher accuracy thanks to pin-knock system using steel s. Also, the upper cover can be installed or removed easily
Adapter
Simplifies installation of the bearing.
Oil drain plug
For draining old lubricant.
Mounting bolt holes
Allow easy positioning and installation
Plummer block unit
Designed for greater mechanical strength, and manufactured under strict quality control program. To suit the intended application, it can be made of either spheroidal graphite cast iron (ductile cast iron) or cast steel

15. Enter into the frame environment with title block and save with an extension of *.frm
16. Finally , Enter into the drawing environment with the created frame and all the models in the drawing sheet and save with an extension of *.drw

INFERENCE:

- Practical experience in making 2D Plummer block parts of casting ,brass , cap , hexagonal bolts & nuts and 3D assemble model for plummer block through the exercise.

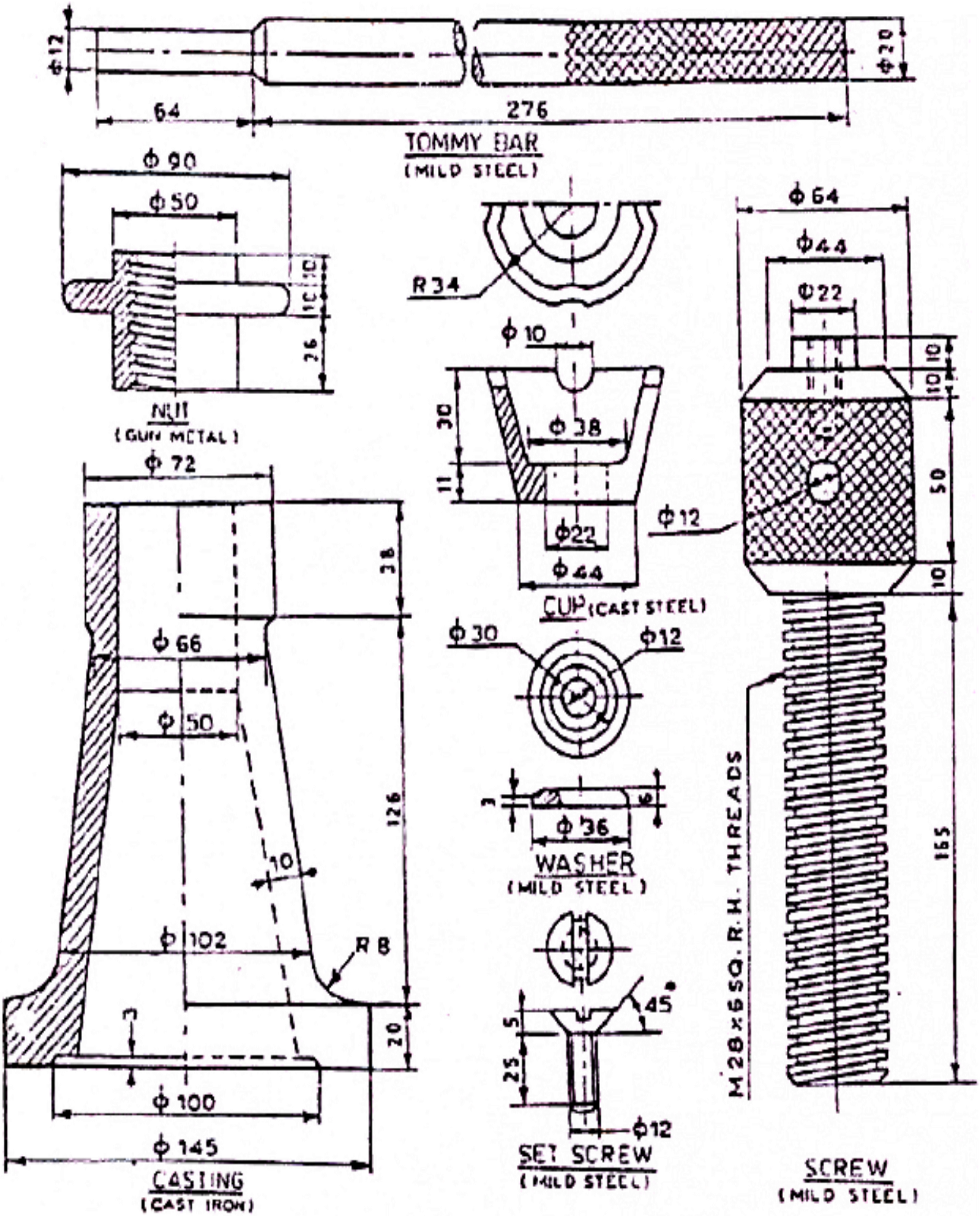
- Plane selection, revolve, blend cut, extrude, helical sweep, pattern, assembly features like mate, axial align, connect and move through the exercise.

APPLICATION:

- Provide support for a rotating shaft with the help of compatible bearings & various accessories. Housing material for a pillow block is typically made of cast iron or cast steel

RESULT:

Thus the 3D assembly of Plummer Block has been created by using the PTC/Creo software .


φ12
φ20
64
276
TOMMY BAR
(MILD STEEL)
φ90
φ50
10
10
26
NUT
(GUN METAL)
φ72
38
φ66
φ50
126
10
φ102
R8
3
20
φ100
φ145
CASTING
(CAST IRON)
R34
φ10
30
φ38
11
φ22
φ44
CUP (CAST STEEL)
φ30
φ12
3
φ36
WASHER
(MILD STEEL)
45°
5
25
φ12
SET SCREW
(MILD STEEL)
φ64
φ44
φ22
10
10
50
10
φ12
M28×6SQ. R.H. THREADS
165
SCREW
(MILD STEEL)

Ex. No : 4 Date :

SCREW JACK

AIM:

To create a Screw Jack 2D part drawings and assembly as a 3D solid model using PTC/Creo Software

HARDWARE REQUIRED:

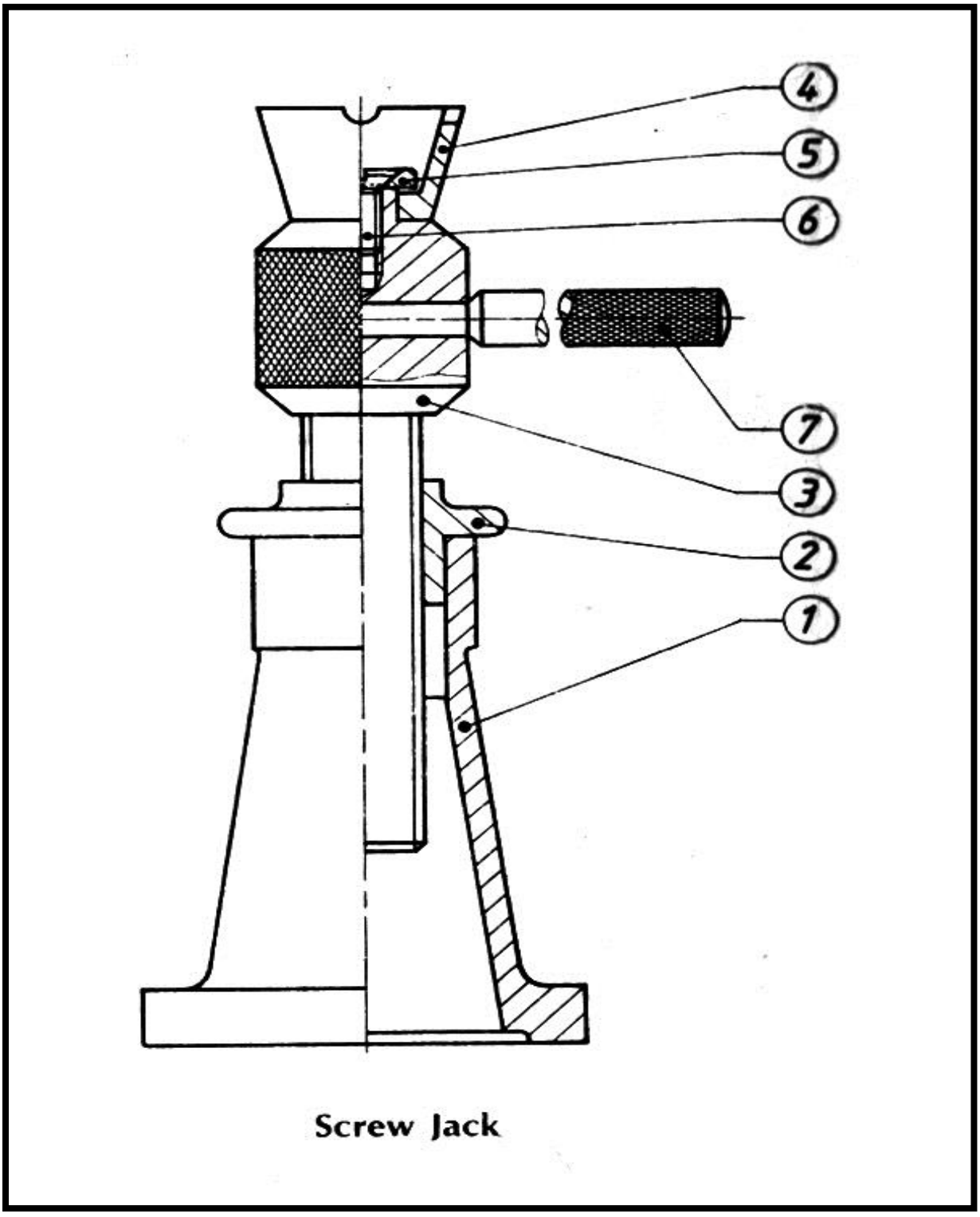
1. CPU with Pentium IV processor.
2. Display with highest 32 bit color monitors with screen resolution 1024 x 768 pixels.
3. Scroll mouse

SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. PTC/Creo

PROCEDURE:

1. Identify various parts to be created.
2. Enter into part environment and create the main part of the assembly.
3. Identify whether the main part or the first part environment is to be created by protrusion or by revolution.
4. Select the sketch tool, then coincidental plane option, from these select any one of the standard 3 planes (i.e. front, right &top).
5. Create the cross-section profile as a closed one by using the available 2D commands , then the click finish button for completing for completing the profile. .
6. Select sketch both parallel plane option or plane by 3points option for creating other parts and then select the required plane.
7. Do the protrusions by using protrusion command and the revolution by revolved protrusion command.
8. Use hole command and cutout command for constructing holes and cut out.
9. Change the diameter of the hole by using modify menu or resize hole option by using hole command.
10. Use revolve cutout command whenever in need.
11. Use the distance option to maintain accurate distance between one edge and other edge (or) one edge and center the hole.
12. Save it in separate part file with extension of *. prt. Similarly save it other parts.
13. Enter into the assembly environment, create the 3D model apply assembly constrains available like planer, design, mate, axial align, connect etc.
14. save with an extension of *.asm the various parts of drg.



Screw Jack

15. Enter into the frame environment with title block and save with an extension of *.frm
16. Finally , Enter into the drawing environment with the created frame and all the models in the drawing sheet and save with an extension of *.drw

INFERENCE:

- Practical experience in making 2D screw jack parts of casting ,gunmetal nut, tommy bar , screw ,cup , washer & set screw and 3D assemble model for screw jack through the exercise.

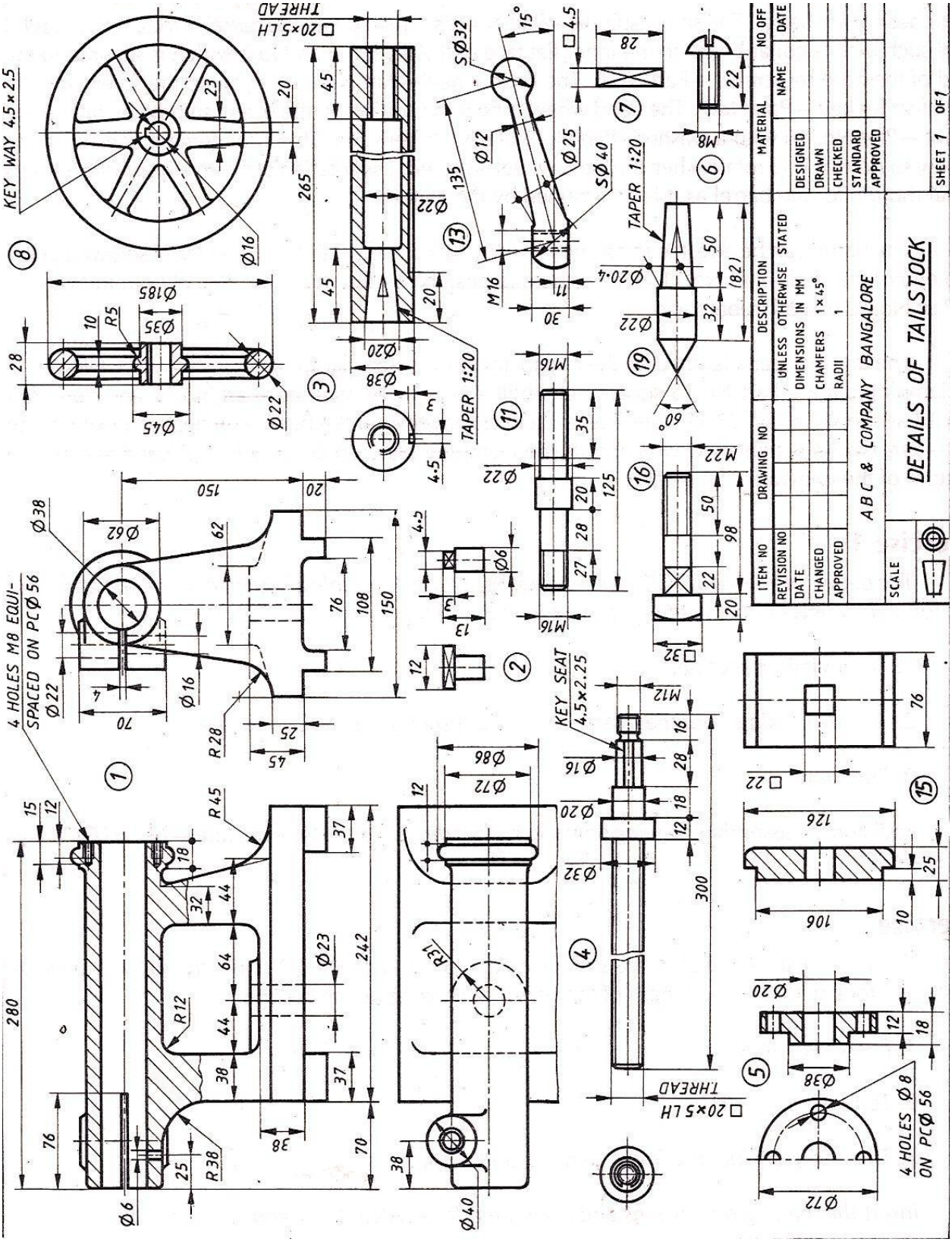
- Plane selection, revolve, blend cut, extrude, helical sweep, pattern, assembly features like mate, axial align, connect and move through the exercise.

APPLICATIONS:

- Large satellite dish antenna movement (x, y, z axis)
- Workplace table adjustments
- Drive wheel adjustment to change conveyor flow stops
- Conveyor lifts, diverters
- Knife blade filter drum skimmer
- Furnace combustion gun adjustment
- Mechanical clutch linkage
- Vacuum furnace lid lifters
- Roll lifts

RESULT:

Thus the 3D assembly of Screw Jack has been created by using the PTC / Creo software.


DETAILS OF TAILSTOCK
ABC & COMPANY BANGALORE
SHEET 1 OF 1
DESCRIPTION
MATERIAL
NO OFF
UNLESS OTHERWISE STATED
DIMENSIONS IN MM
CHAMFERS 1 × 45°
RADII 1
DESIGNED
DRAWN
CHECKED
STANDARD
APPROVED
SCALE
ITEM NO
DRAWING NO
REVISION NO
DATE
CHANGED
APPROVED
KEY WAY 4.5 × 2.5
4 HOLES M8 EQUI-SPACED ON PCØ56
KEY SEAT 4.5 × 2.25
TAPER 1:20
THREAD
4 HOLES Ø8 ON PCØ56

Ex. No : 5 Date :

LATHE TAILSTOCK

AIM:

To create a Lathe Tailstock 2D part drawings and assembly as a 3D solid model using PTC/Creo Software

HARDWARE REQUIRED:

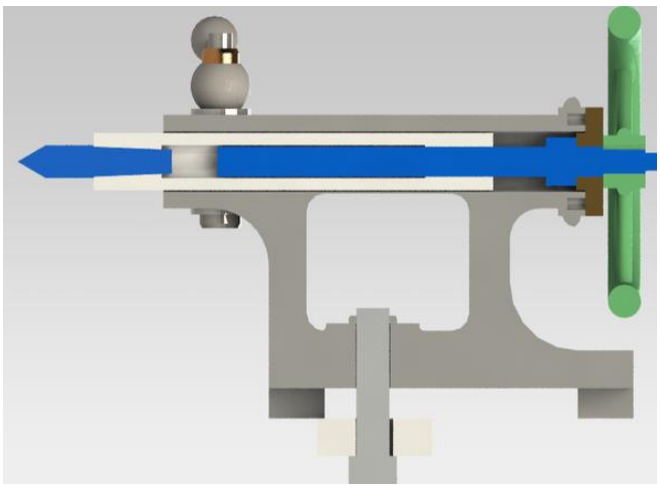
1. CPU with Pentium IV processor.
2. Display with highest 32 bit color monitors with screen resolution 1024 x 768 pixels.
3. Scroll mouse

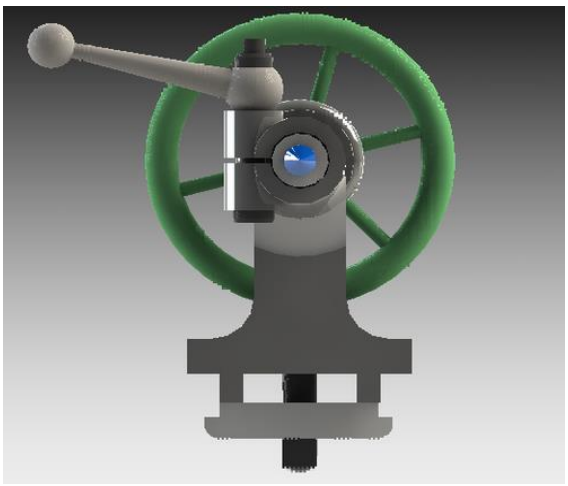
SOFTWARE REQUIRED:

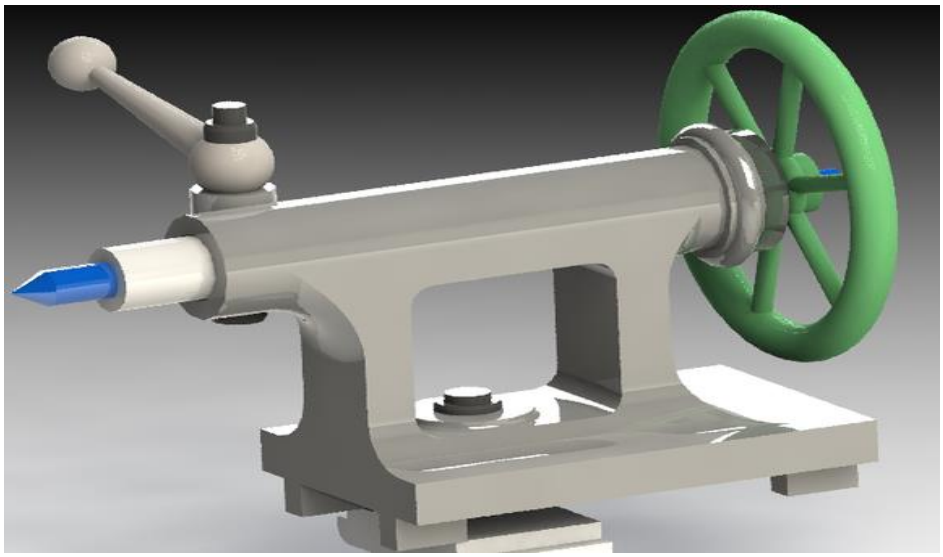
1. Windows XP or Windows 7 operating system
2. PTC/Creo

PROCEDURE:

1. Identify various parts to be created.
2. Enter into part environment and create the main part of the assembly.
3. Identify whether the main part or the first part environment is to be created by protrusion or by revolution.
4. Select the sketch tool, then coincidental plane option, from these select any one of the standard 3 planes (i.e. front, right &top).
5. Create the cross-section profile as a closed one by using the available 2D commands , then the click finish button for completing for completing the profile. .
6. Select sketch both parallel plane option or plane by 3points option for creating other parts and then select the required plane.
7. Do the protrusions by using protrusion command and the revolution by revolved protrusion command.
8. Use hole command and cutout command for constructing holes and cut out.
9. Change the diameter of the hole by using modify menu or resize hole option by using hole command.
10. Use revolve cutout command whenever in need.
11. Use the distance option to maintain accurate distance between one edge and other edge (or) one edge and center the hole.
12. Save it in separate part file with extension of *. prt. Similarly save it other parts.
13. Enter into the assembly environment, create the 3D model apply assembly constrains available like planer, design, mate, axial align, connect etc.
14. save with an extension of *.asm the various parts of drg.







15. Enter into the frame environment with title block and save with an extension of *.frm
16. Finally , Enter into the drawing environment with the created frame and all the models in the drawing sheet and save with an extension of *.drw

INFERENCE:

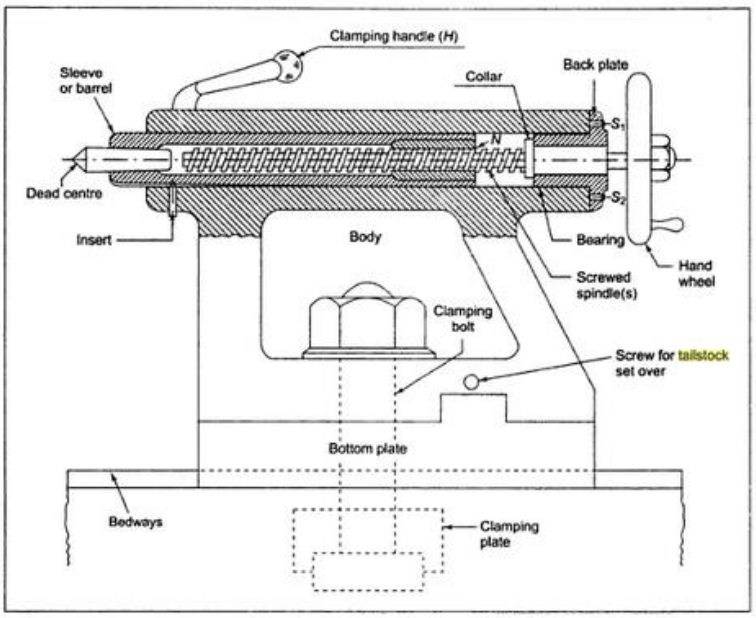
- Practical experience in making 2D tailstock parts of body , sleeve , dead center , back plate , collar , hand wheel ,clamping handle etc and 3D assemble model for tail stock through the exercise.

- Plane selection, revolve, blend cut, extrude, pattern, assembly features like mate, axial align, connect and move through the exercise.

APPLICATIONS:

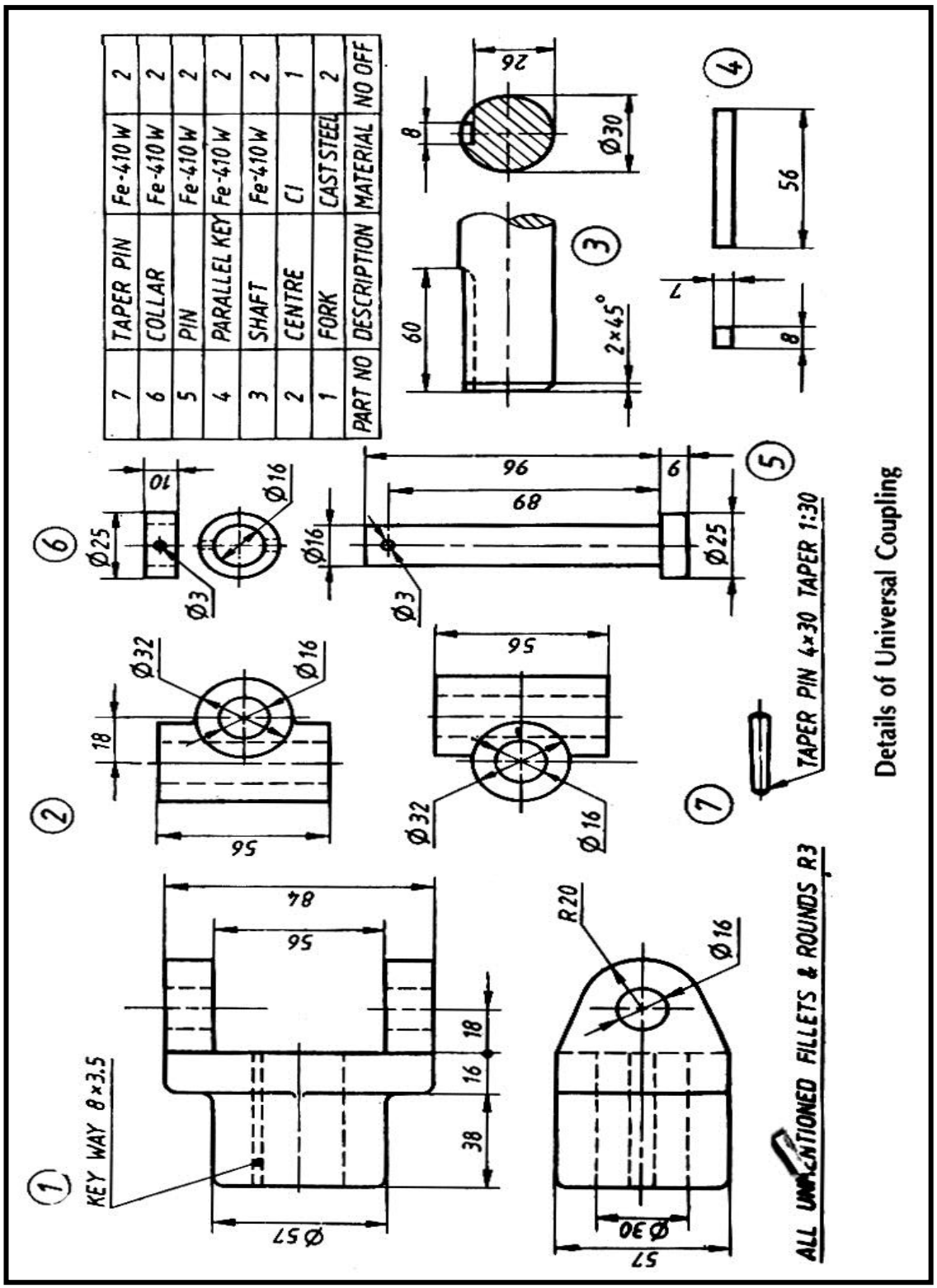
- When the work piece is relatively long and slender. Failing to use a tailstock can cause "chatter," where the work piece bends excessively while being cut.

- It is also used on a lathe to hold drilling or reaming tools for machining a hole in the work piece.



RESULT:

Thus the 3D assembly of Lathe Tailstock has been created by using the PTC/Creo software.


PART NO | DESCRIPTION | MATERIAL | NO OFF
7 | TAPER PIN | Fe-410 W | 2
6 | COLLAR | Fe-410 W | 2
5 | PIN | Fe-410 W | 2
4 | PARALLEL KEY | Fe-410 W | 2
3 | SHAFT | Fe-410 W | 2
2 | CENTRE | CI | 1
1 | FORK | CAST STEEL | 2
KEY WAY 8×3.5
ALL UNMENTIONED FILLETS & ROUNDS R3
TAPER PIN 4×30 TAPER 1:30
Details of Universal Coupling

Ex. No : 6 Date :

UNIVERSAL JOINT

AIM:

To create a Universal Joint of 2D part drawings and assemble 3D as solid model using PTC/Creo Software

HARDWARE REQUIRED:

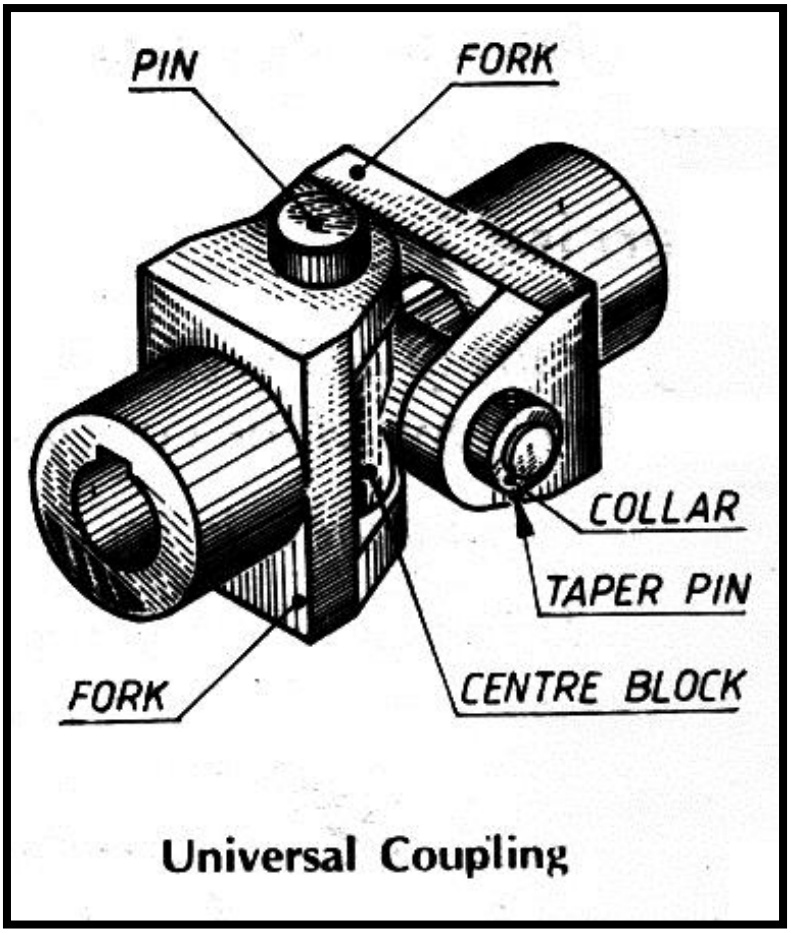
1. CPU with Pentium IV processor.
2. Display with highest 32 bit color monitors with screen resolution 1024 x 768 pixels.
3. Scroll mouse.

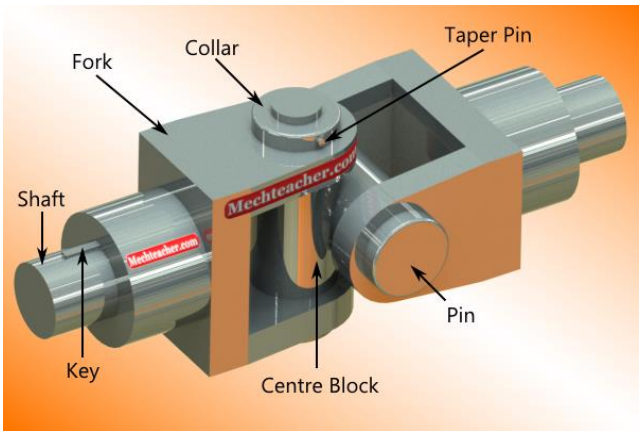
SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. PTC/Creo

PROCEDURE:

1. Identify various parts to be created.
2. Enter into part environment and create the main part of the assembly.
3. Identify whether the main part or the first part environment is to be created by protrusion or by revolution.
4. Select the sketch tool, then coincidental plane option, from these select any one of the standard 3 planes (i.e. front, right &top).
5. Create the cross-section profile as a closed one by using the available 2D commands , then the click finish button for completing for completing the profile. .
6. Select sketch both parallel plane option or plane by 3points option for creating other parts and then select the required plane.
7. Do the protrusions by using protrusion command and the revolution by revolved protrusion command.
8. Use hole command and cutout command for constructing holes and cut out.
9. Change the diameter of the hole by using modify menu or resize hole option by using hole command.
10. Use revolve cutout command whenever in need.
11. Use the distance option to maintain accurate distance between one edge and other edge (or) one edge and center the hole.
12. Save it in separate part file with extension of *. prt. Similarly save it other parts.


PIN
FORK
COLLAR
TAPER PIN
FORK
CENTRE BLOCK
Universal Coupling


Fork
Collar
Taper Pin
Shaft
Key
Centre Block
Pin
Mechteacher.com

13. Enter into the assembly environment, create the 3D model apply assembly constrains available like planer, design, mate, axial align, connect etc.
14. save with an extension of *.asm the various parts of drg.
15. Enter into the frame environment with title block and save with an extension of *.frm
16. Finally , Enter into the drawing environment with the created frame and all the models in the drawing sheet and save with an extension of *.drw

INFERENCE:

- Practical experience in making 2D Universal Joint parts of Fork, Eye, Pin, Taper Pin, Collar and 3D assemble model for Universal Joint through the exercise.
- Plane selection, revolve, blend cut, extrude, helical sweep, pattern, assembly features like mate, axial align, connect and move through the exercise.

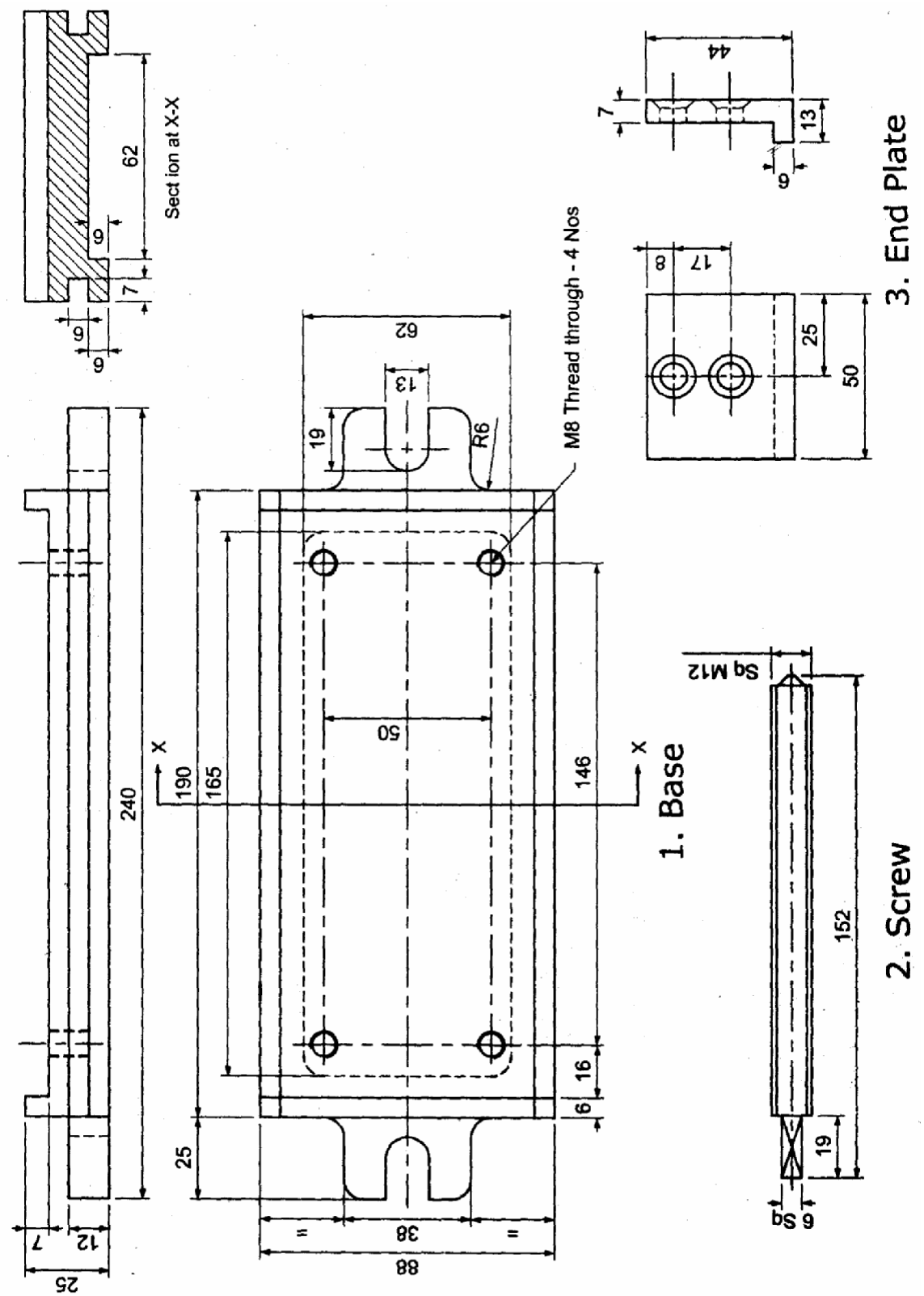
APPLICATION:

Universal joint has a wide range of applications. It is used in:

1. Driveshafts
2. Automobile propeller shafts
3. Stone crushers
4. Tapping machinery
5. Centrifugal blowers
6. Centrifugal fans and centrifugal pumps
7. Belt conveyors
8. Control mechanisms
9. Marine equipments
10. Metal forming machinery
11. Sockets

RESULT:

Thus, the 3D assembly of Universal Joint has been created by using the PTC/Creo software.


Section at X-X
62
6
9
7
6
6
44
7
13
6
8
17
25
50
3. End Plate
62
13
19
R6
M8 Thread through - 4 Nos
50
X
190
165
240
146
X
16
6
25
38
88
25
7
12
1. Base
Sq M12
152
19
6
Sq
2. Screw

Ex. No : 7 Date :

MACHINE VICE

AIM:

To create a Machine Vice of 2D part drawings and assemble 3D as solid model using PTC/Creo Software

HARDWARE REQUIRED:

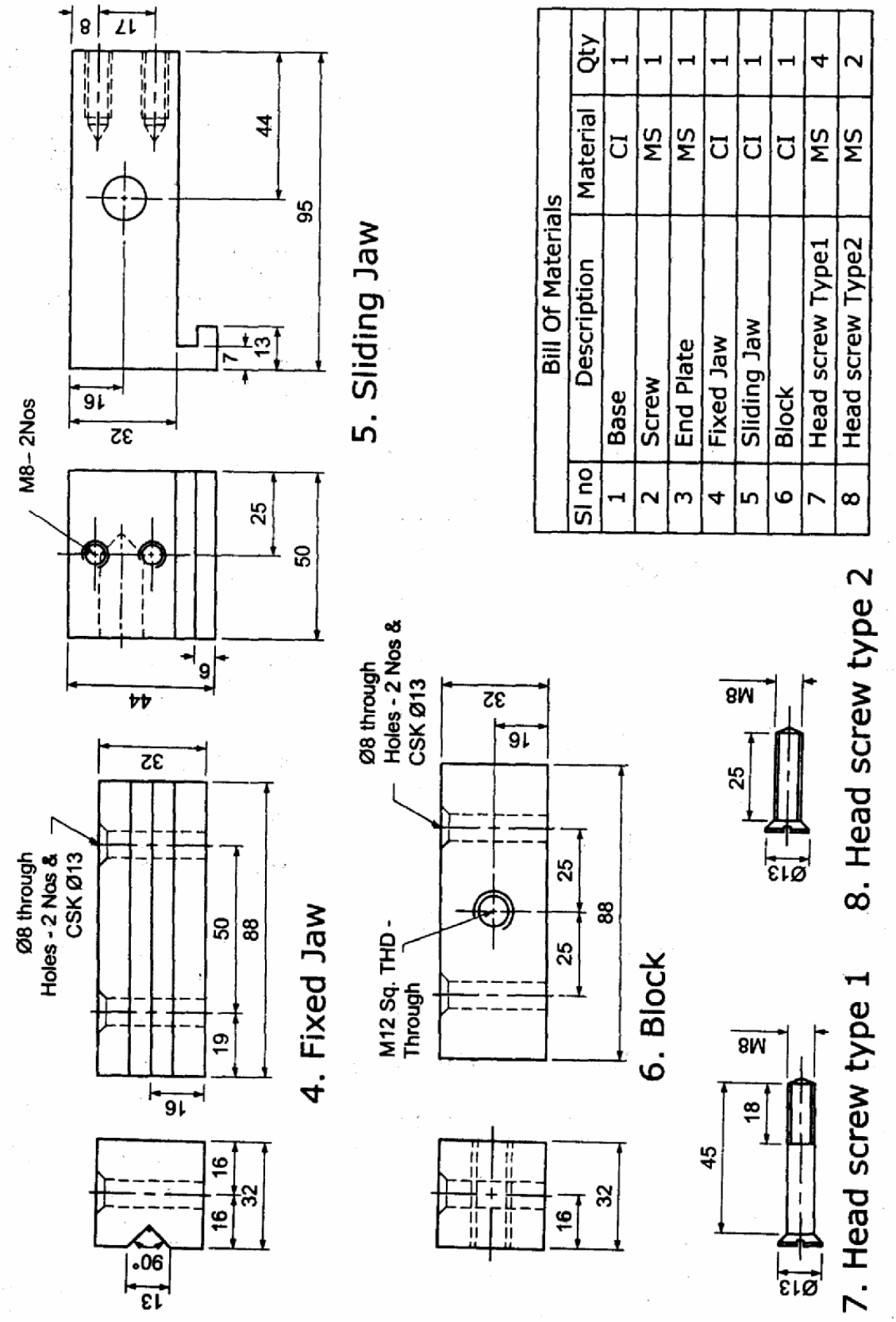
1. CPU with Pentium IV processor.
2. Display with highest 32 bit color monitors with screen resolution 1024 x 768 pixels.
3. Scroll mouse.

SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. PTC/Creo

PROCEDURE:

1. Identify various parts to be created.
2. Enter into part environment and create the main part of the assembly.
3. Identify whether the main part or the first part environment is to be created by protrusion or by revolution.
4. Select the sketch tool, then coincidental plane option, from these select any one of the standard 3 planes (i.e. front, right &top).
5. Create the cross-section profile as a closed one by using the available 2D commands , then the click finish button for completing for completing the profile. .
6. Select sketch both parallel plane option or plane by 3points option for creating other parts and then select the required plane.
7. Do the protrusions by using protrusion command and the revolution by revolved protrusion command.
8. Use hole command and cutout command for constructing holes and cut out.
9. Change the diameter of the hole by using modify menu or resize hole option by using hole command.
10. Use revolve cutout command whenever in need.
11. Use the distance option to maintain accurate distance between one edge and other edge (or) one edge and center the hole.
12. Save it in separate part file with extension of *. prt. Similarly save it other parts.
13. Enter into the assembly environment, create the 3D model apply assembly constrains available like planer, design, mate, axial align, connect etc.
14. save with an extension of *.asm the various parts of drg.
15. Enter into the frame environment with title block and save with an extension of *.frm



5. Sliding Jaw

Bill Of Materials			
Sl no	Description	Material	Qty
1	Base	CI	1
2	Screw	MS	1
3	End Plate	MS	1
4	Fixed Jaw	CI	1
5	Sliding Jaw	CI	1
6	Block	CI	1
7	Head screw Type1	MS	4
8	Head screw Type2	MS	2

4. Fixed Jaw

6. Block

7. Head screw type 1

8. Head screw type 2

16. Finally , Enter into the drawing environment with the created frame and all the models in the drawing sheet and save with an extension of *.drw

INFERENCE:

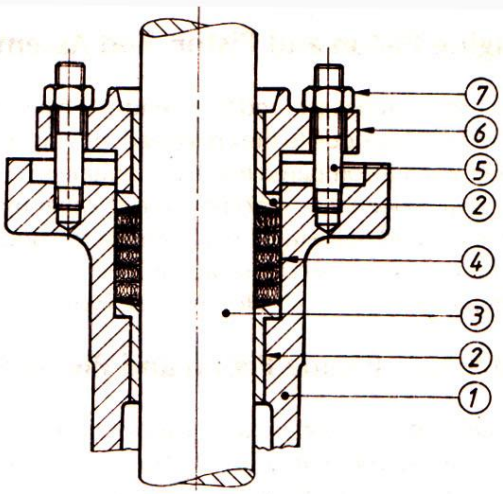
- Practical experience in making 2D Machine Vice parts of Base, Screw, End Plate, Fixed Jaw, Sliding Jaw, Block & Head Screws and 3D assemble model for Machine Vice through the exercise.
- Plane selection, revolve, blend cut, extrude, helical sweep, pattern, assembly features like mate, axial align, connect and move through the exercise.

APPLICATIONS:

It is used for clamping components on various machines

RESULT:

Thus, the 3D assembly of Machine Vice has been created by using the PTC/Creo software.

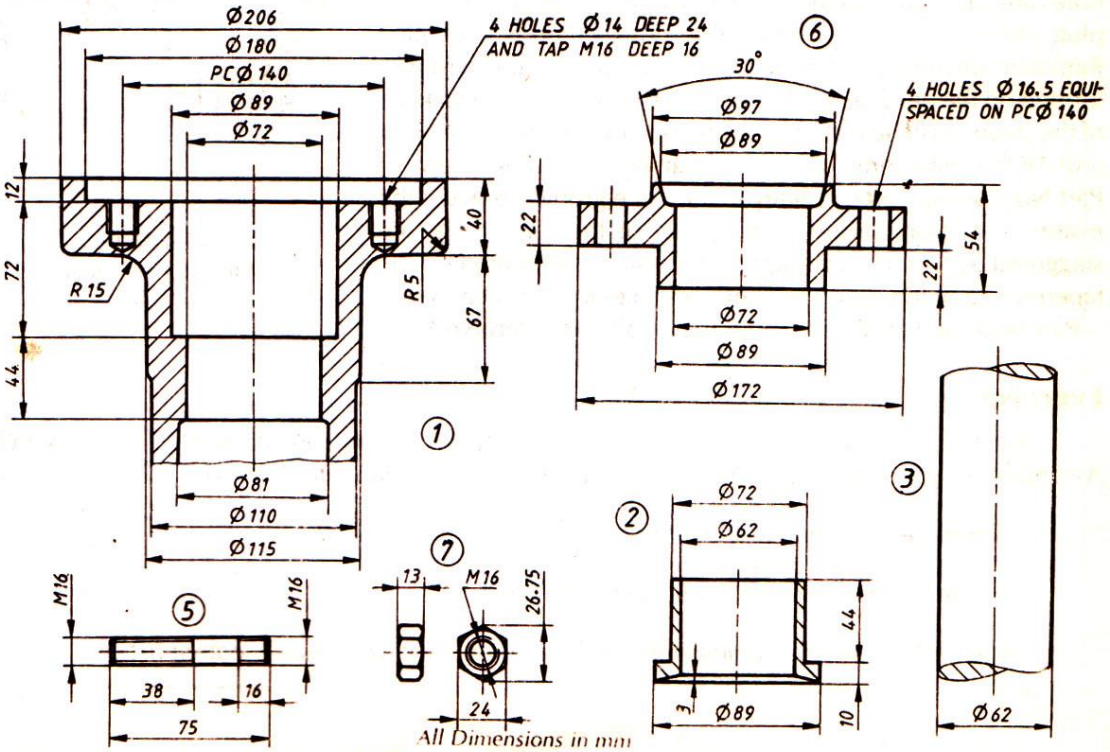


Gland and Stuffing Box

PART NO.	DESCRIPTION	MATERIAL	NO OFF
1	STUFFING BOX	CAST IRON	1
2	NECK BUSH	BRASS	2
3	PISTON ROD	Fe 410 W	1
4*	ASBESTOS	ASBESTOS	A.R.

PART NO.	DESCRIPTION	MATERIAL	NO OFF
5	STUD	Fe 410 W	4
6	GLAND	CAST IRON	1
7	HEX. NUT M16	Fe 410 W	4

*Commercial, AR = As required



All Dimensions in mm

Details of a Gland and Stuffing Box

Ex. No : 8 Date :

STUFFING BOX

AIM:

To create a Stuffing Box of 2D part drawings and assemble 3D as solid model using PTC/Creo Software

HARDWARE REQUIRED:

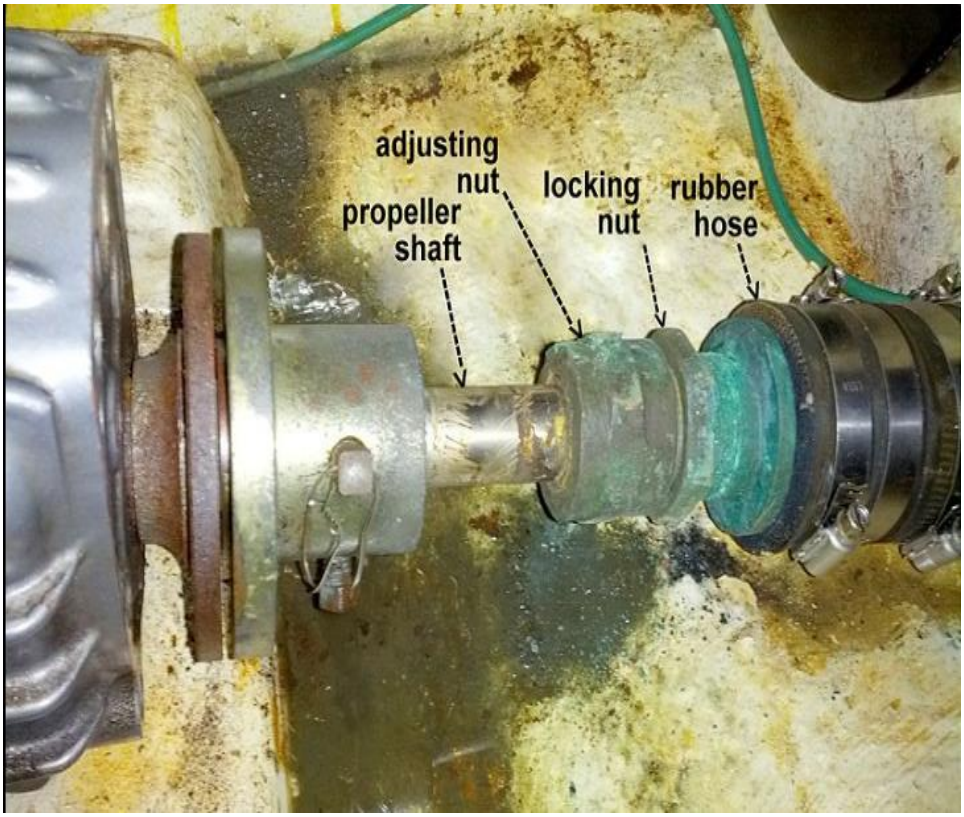
1. CPU with Pentium IV processor.
2. Display with highest 32 bit color monitors with screen resolution 1024 x 768 pixels.
3. Scroll mouse.

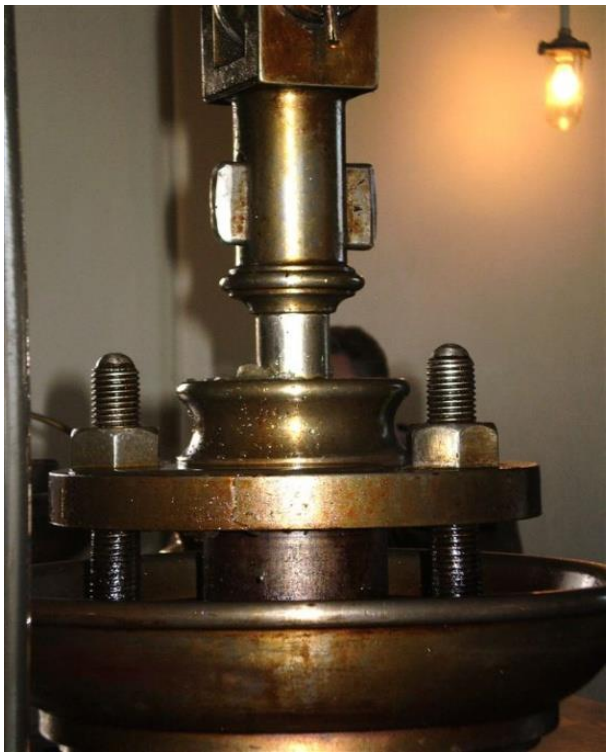
SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. PTC/Creo

PROCEDURE:

1. Identify various parts to be created.
2. Enter into part environment and create the main part of the assembly.
3. Identify whether the main part or the first part environment is to be created by protrusion or by revolution.
4. Select the sketch tool, then coincidental plane option, from these select any one of the standard 3 planes (i.e. front, right &top).
5. Create the cross-section profile as a closed one by using the available 2D commands , then the click finish button for completing for completing the profile. .
6. Select sketch both parallel plane option or plane by 3points option for creating other parts and then select the required plane.
7. Do the protrusions by using protrusion command and the revolution by revolved protrusion command.
8. Use hole command and cutout command for constructing holes and cut out.
9. Change the diameter of the hole by using modify menu or resize hole option by using hole command.
10. Use revolve cutout command whenever in need.
11. Use the distance option to maintain accurate distance between one edge and other edge (or) one edge and center the hole.
12. Save it in separate part file with extension of *. prt. Similarly save it other parts.
13. Enter into the assembly environment, create the 3D model apply assembly constrains available like planer, design, mate, axial align, connect etc.
14. save with an extension of *.asm the various parts of drg.
15. Enter into the frame environment with title block and save with an extension of *.frm


adjusting
nut
propeller
shaft
locking
nut
rubber
hose



16. Finally , Enter into the drawing environment with the created frame and all the models in the drawing sheet and save with an extension of *.drw

INFERENCE:

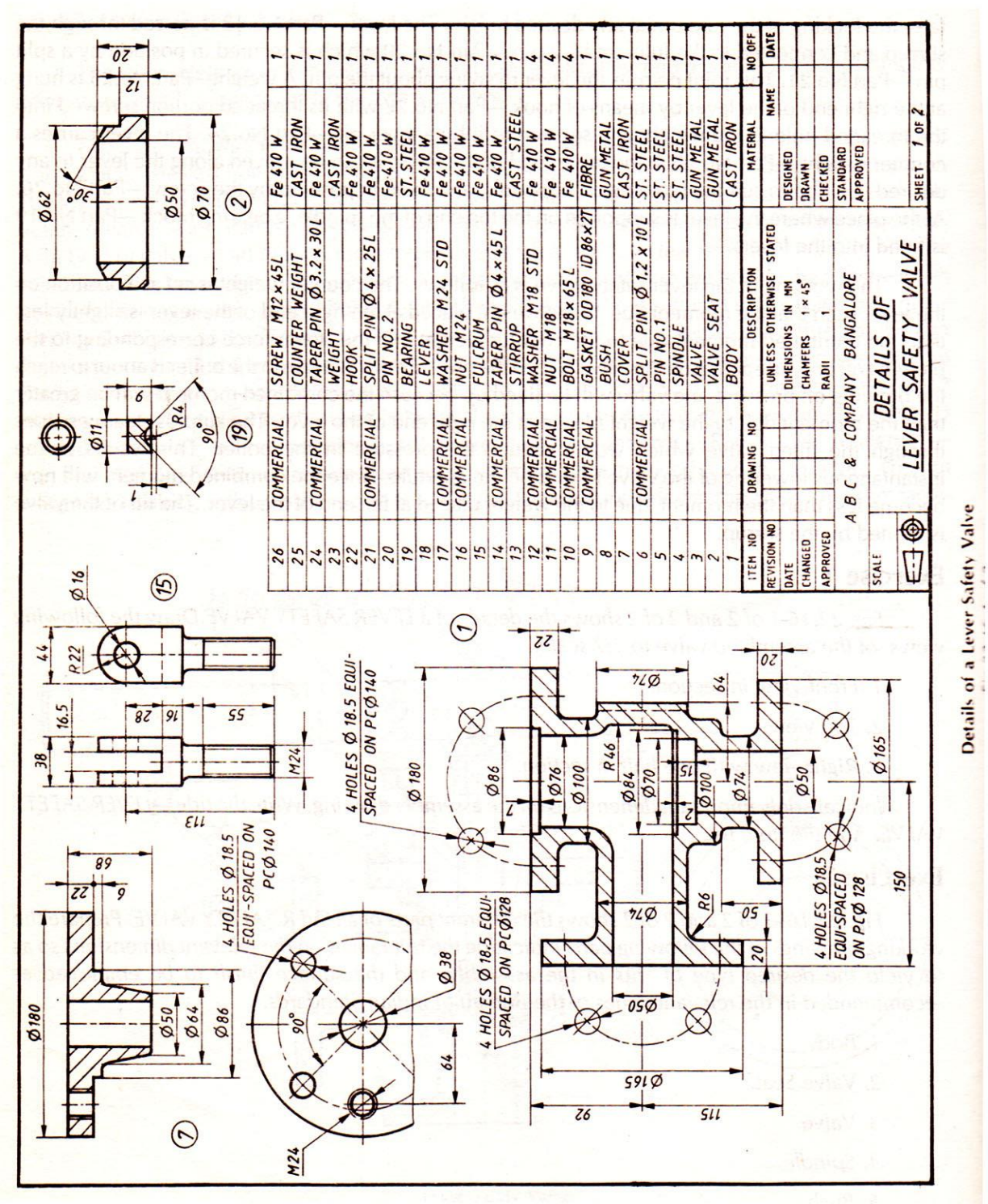
- Practical experience in making 2D Stuffing Box parts of Stuffing Box, Neck Bush, Piston Rod, Stud, Gland, Hexagonal nuts and 3D assemble model for Stuffing Box through the exercise.
- Plane selection, revolve, blend cut, extrude, helical sweep, pattern, assembly features like mate, axial align, connect and move through the exercise.

APPLICATIONS:

- Boats
  The stuffing box prevents sea water from entering the boat's hull
- Steam engines
  In a steam engine, where the piston rod reciprocates through the cylinder cover, a stuffing box provided in the cylinder cover prevents the leakage of steam from the cylinder

RESULT:

Thus, the 3D assembly of Stuffing Box has been created by using the PTC/Creo software.



Details of a Lever Safety Valve

Ex. No : 9 Date :

SAFETY VALVE

AIM:

To create a Safety Valve of 2D part drawings and assemble 3D as solid model using PTC/Creo Software

HARDWARE REQUIRED:

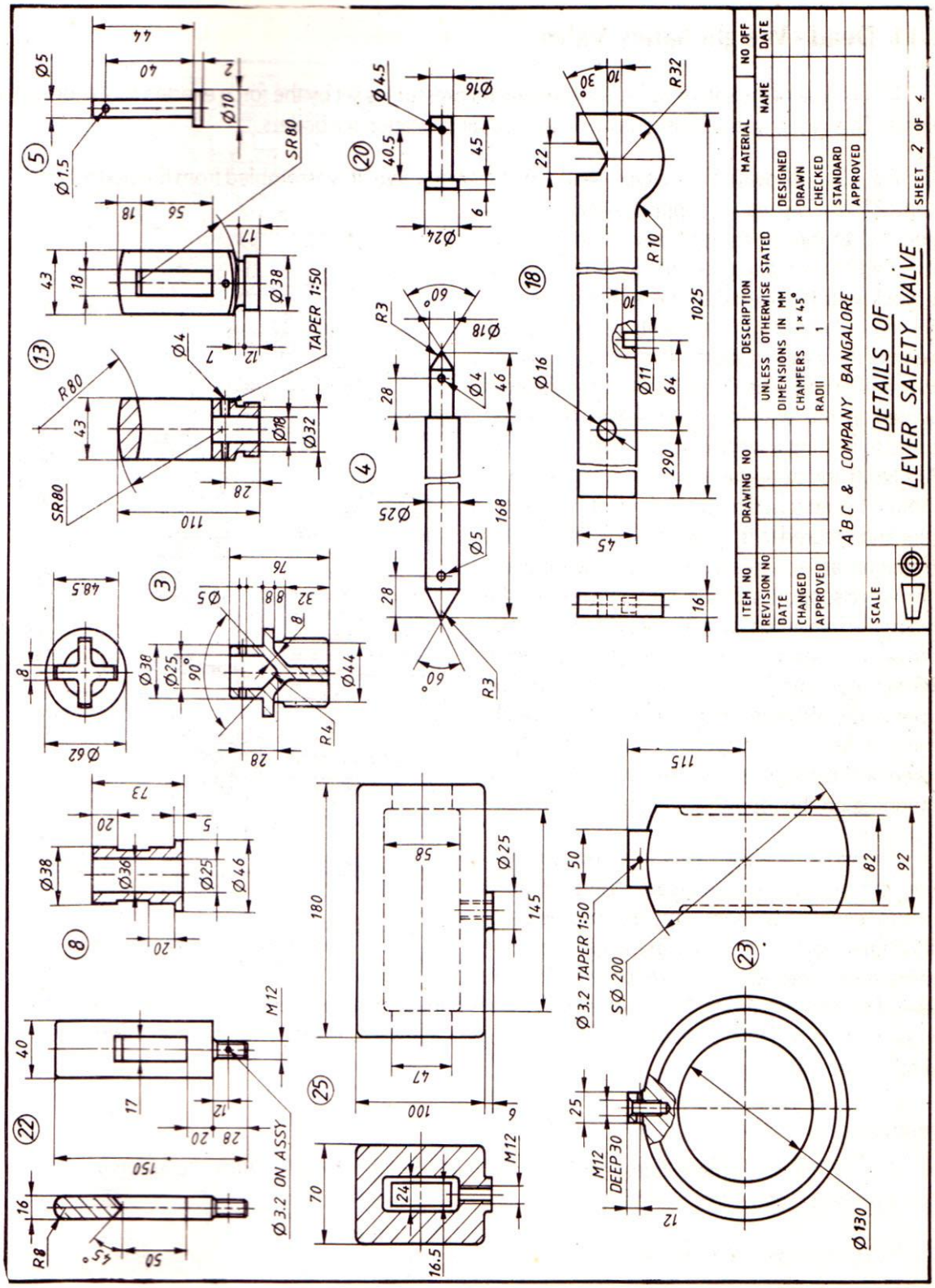
1. CPU with Pentium IV processor.
2. Display with highest 32 bit color monitors with screen resolution 1024 x 768 pixels.
3. Scroll mouse.

SOFTWARE REQUIRED:

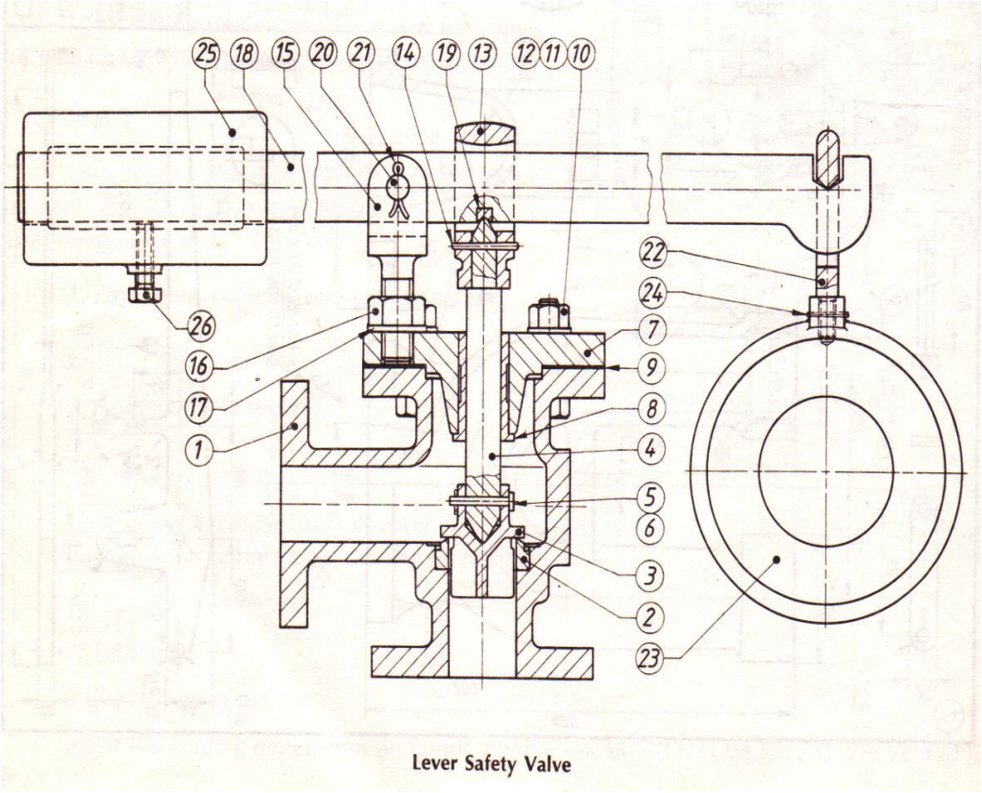
1. Windows XP or Windows 7 operating system
2. PTC/Creo

PROCEDURE:

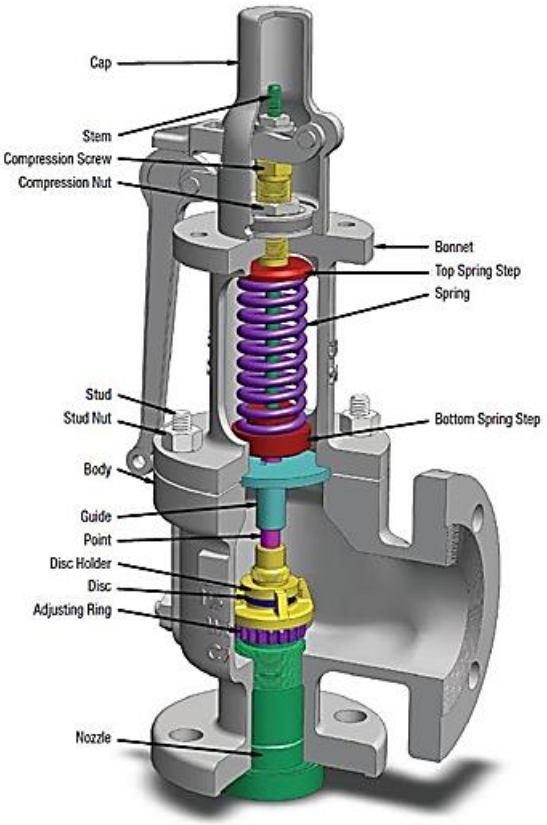
1. Identify various parts to be created.
2. Enter into part environment and create the main part of the assembly.
3. Identify whether the main part or the first part environment is to be created by protrusion or by revolution.
4. Select the sketch tool, then coincidental plane option, from these select any one of the standard 3 planes (i.e. front, right &top).
5. Create the cross-section profile as a closed one by using the available 2D commands , then the click finish button for completing for completing the profile. .
6. Select sketch both parallel plane option or plane by 3points option for creating other parts and then select the required plane.
7. Do the protrusions by using protrusion command and the revolution by revolved protrusion command.
8. Use hole command and cutout command for constructing holes and cut out.
9. Change the diameter of the hole by using modify menu or resize hole option by using hole command.
10. Use revolve cutout command whenever in need.
11. Use the distance option to maintain accurate distance between one edge and other edge (or) one edge and center the hole.
12. Save it in separate part file with extension of *. prt. Similarly save it other parts.
13. Enter into the assembly environment, create the 3D model apply assembly constrains available like planer, design, mate, axial align, connect etc.
14. save with an extension of *.asm the various parts of drg.
15. Enter into the frame environment with title block and save with an extension of *.frm



Details of a Lever Safety Valve



Lever Safety Valve



APPLICATIONS

Safety Valve for Chemical Process Industries

Safety Valve for Oil & Gas Industry

Safety Valve for Pharmaceutical Industry

Safety Valve for Textile Industry

Safety Valve for LNG/LPG

Safety Valve for Mining Industry

Safety Valve for Food & Beverage

Safety Valve for Power Industry

Safety Valve for Petrochemical Industry

Safety Valve for Air Compressor

16. Finally , Enter into the drawing environment with the created frame and all the models in the drawing sheet and save with an extension of *.drw

INFERENCE:

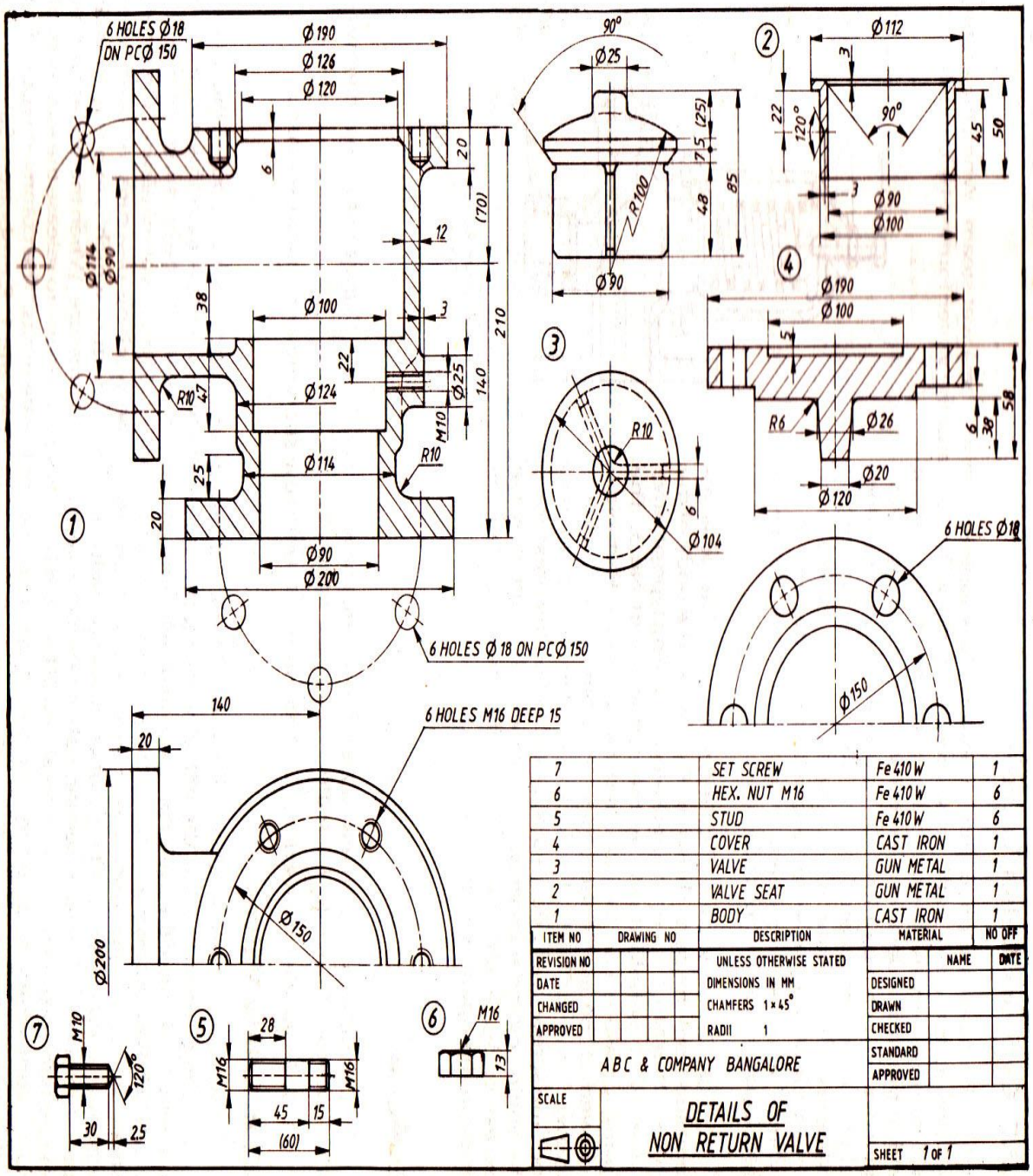
- Practical experience in making 2D Safety Valve parts of Body, Valve Seat, Spindle, Fulcrum and Bush, etc. and 3D assemble model for Safety Valve through the exercise.
- Plane selection, revolve, blend cut, extrude, helical sweep, pattern, assembly features like mate, axial align, connect and move through the exercise.

APPLICATIONS:

- Safety Valve for Chemical Process Industry
- Safety Valve for Oil and Gas Industry
- Safety Valve for Pharmaceutical Industry
- Safety Valve for Textile Industry
- Safety Valve for LNG / LPG
- Safety Valve for Mining Industry
- Safety Valve for Food and Beverage
- Safety Valve for Power Plants
- Safety Valve for Air Compressor

RESULT:

Thus, the 3D assembly of Safety Valve has been created by using the PTC/Creo software.



ITEM NO	DRAWING NO	DESCRIPTION	MATERIAL	NO OFF
7		SET SCREW	Fe 410 W	1
6		HEX. NUT M16	Fe 410 W	6
5		STUD	Fe 410 W	6
4		COVER	CAST IRON	1
3		VALVE	GUN METAL	1
2		VALVE SEAT	GUN METAL	1
1		BODY	CAST IRON	1

REVISION NO					NAME / DATE
DATE				DESIGNED	
CHANGED				DRAWN	
APPROVED				CHECKED	
				STANDARD	
				APPROVED	

UNLESS OTHERWISE STATED
DIMENSIONS IN MM
CHAMFERS 1 × 45°
RADII 1

ABC & COMPANY BANGALORE

SCALE

DETAILS OF NON RETURN VALVE

SHEET 1 OF 1

Details of a Non-Return Valve

Ex. No : 10 Date :

NON-RETURN VALVES

AIM:

To create a Non-Return Valve of 2D part drawings and assemble 3D as solid model using PTC/Creo Software

HARDWARE REQUIRED:

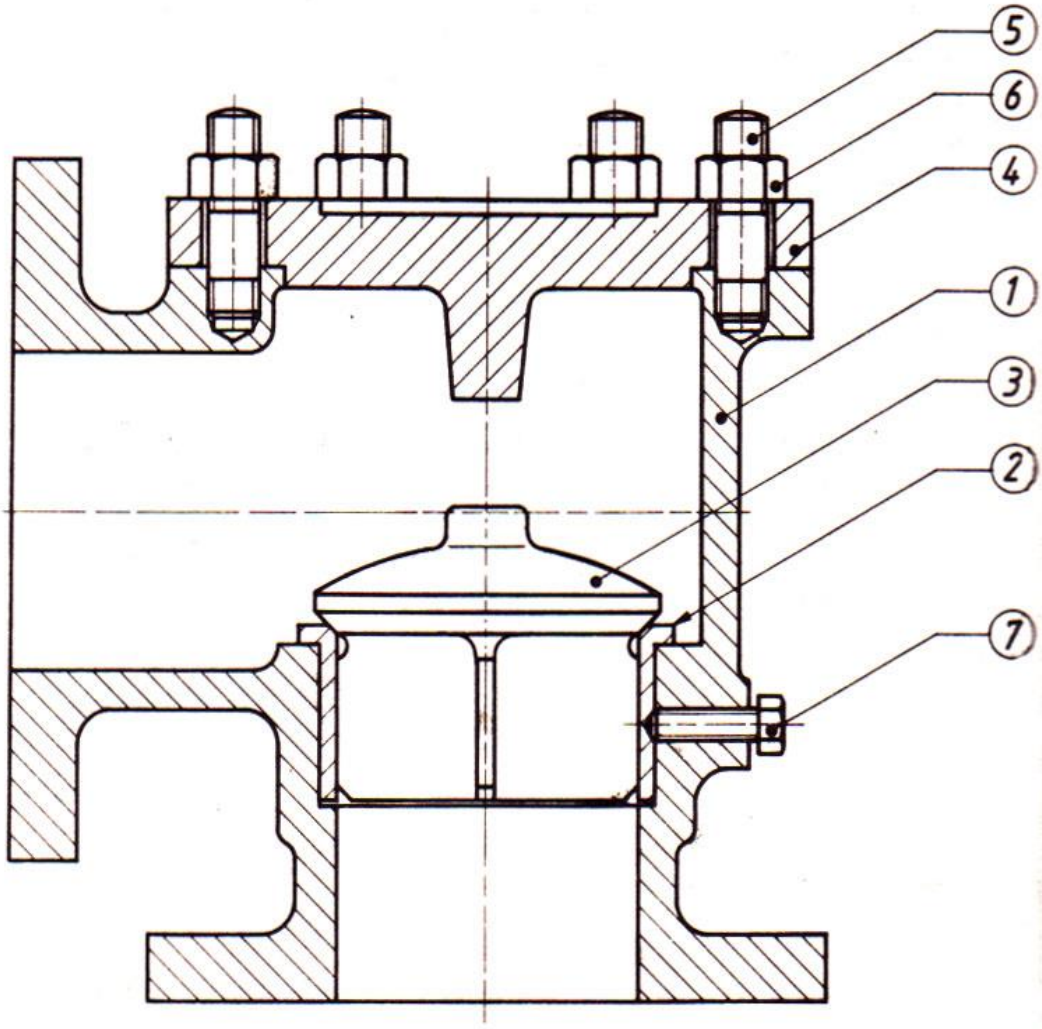
1. CPU with Pentium IV processor.
2. Display with highest 32 bit color monitors with screen resolution 1024 x 768 pixels.
3. Scroll mouse.

SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. PTC/Creo

PROCEDURE:

1. Identify various parts to be created.
2. Enter into part environment and create the main part of the assembly.
3. Identify whether the main part or the first part environment is to be created by protrusion or by revolution.
4. Select the sketch tool, then coincidental plane option, from these select any one of the standard 3 planes (i.e. front, right &top).
5. Create the cross-section profile as a closed one by using the available 2D commands , then the click finish button for completing for completing the profile. .
6. Select sketch both parallel plane option or plane by 3points option for creating other parts and then select the required plane.
7. Do the protrusions by using protrusion command and the revolution by revolved protrusion command.
8. Use hole command and cutout command for constructing holes and cut out.
9. Change the diameter of the hole by using modify menu or resize hole option by using hole command.
10. Use revolve cutout command whenever in need.
11. Use the distance option to maintain accurate distance between one edge and other edge (or) one edge and center the hole.
12. Save it in separate part file with extension of *. prt. Similarly save it other parts.
13. Enter into the assembly environment, create the 3D model apply assembly constrains available like planer, design, mate, axial align, connect etc.
14. save with an extension of *.asm the various parts of drg.



Non-Return Valve

15. Enter into the frame environment with title block and save with an extension of *.frm
16. Finally , Enter into the drawing environment with the created frame and all the models in the drawing sheet and save with an extension of *.drw

INFERENCE:

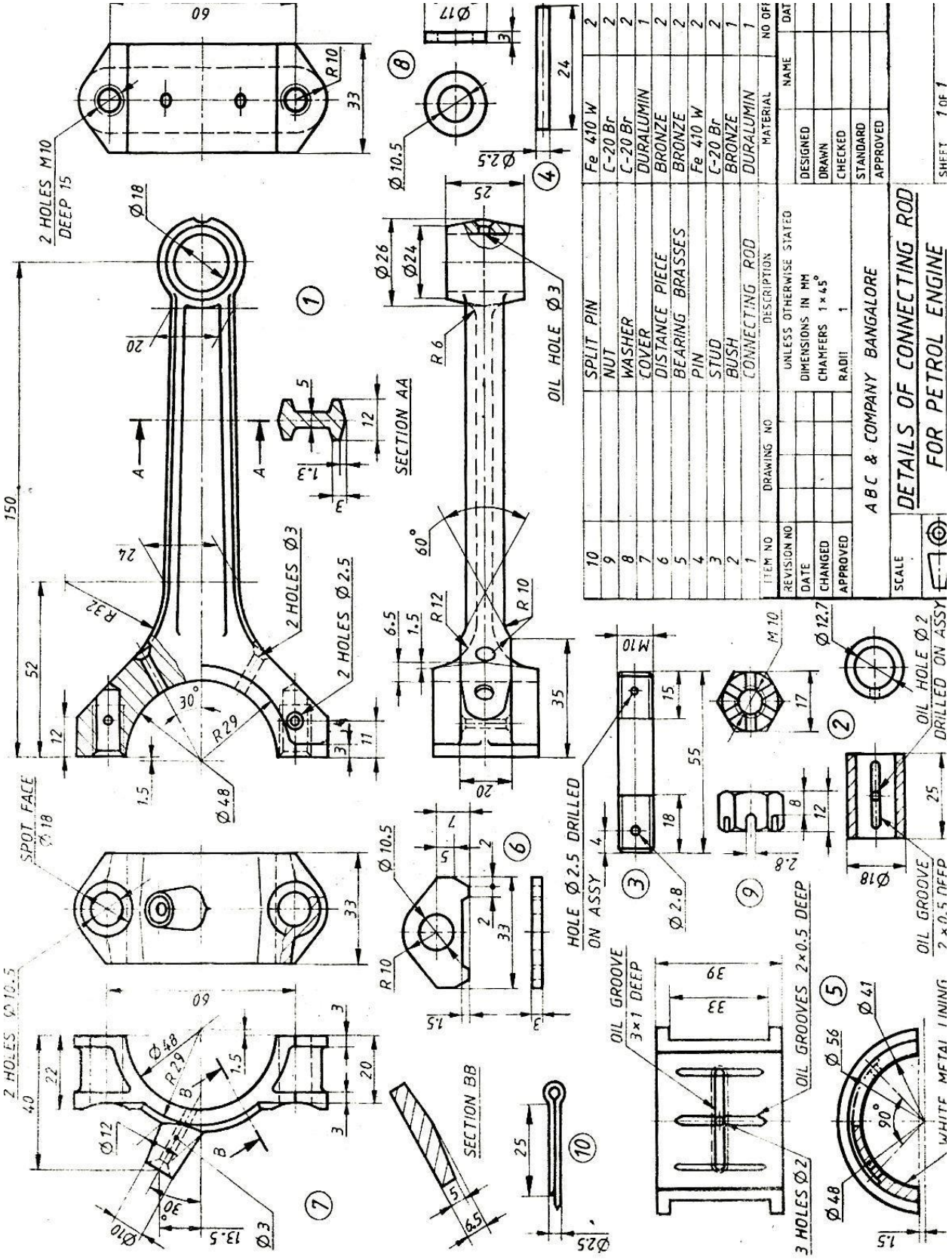
- Practical experience in making 2D Non-Return Valve parts of Body, Valve Seat, Valve, Cover, Studs, hexagonal nuts and 3D assemble model for Non-Return Valve through the exercise.
- Plane selection, revolve, blend cut, extrude, helical sweep, pattern, assembly features like mate, axial align, connect and move through the exercise.

APPLICATIONS:

- **A non-return valve allows a medium to flow in only one direction** , it is used in a number of applications such as oil, gas, Air, water, refining, steam and chemical applications.
- in waterworks piplines to reduce water hammer effect and Preventing back flow of water to the pump

RESULT:

Thus, the 3D assembly of Non-Return Valve has been created by using the PTC/Creo software.


DETAILS OF CONNECTING ROD
FOR PETROL ENGINE
ABC & COMPANY BANGALORE
SCALE
SHEET 1 OF 1
ITEM NO
DRAWING NO
DESCRIPTION
MATERIAL
NO OFF
1 CONNECTING ROD DURALUMIN 1
2 BUSH BRONZE 1
3 STUD C-20 Br 2
4 PIN Fe 410 W 2
5 BEARING BRASSES BRONZE 2
6 DISTANCE PIECE BRONZE 2
7 COVER DURALUMIN 1
8 WASHER C-20 Br 2
9 NUT C-20 Br 2
10 SPLIT PIN Fe 410 W 2
REVISION NO
DATE
CHANGED
APPROVED
UNLESS OTHERWISE STATED
DIMENSIONS IN MM
CHAMFERS 1 x 45°
RADII 1
NAME
DAT
DESIGNED
DRAWN
CHECKED
STANDARD
APPROVED
2 HOLES M10 DEEP 15
2 HOLES Ø3
2 HOLES Ø2.5
OIL HOLE Ø3
SECTION AA
SECTION BB
SPOT FACE Ø18
HOLE Ø2.5 DRILLED ON ASSY
OIL HOLE Ø2 DRILLED ON ASSY
OIL GROOVE 3×1 DEEP
OIL GROOVES 2×0.5 DEEP
OIL GROOVE 2×0.5 DEEP
3 HOLES Ø2
WHITE METAL LINING

Ex. No : 11 Date :

CONNECTING ROD

AIM:

To create a Connecting Rod of 2D part drawings and assemble 3D as solid model using PTC/Creo Software

HARDWARE REQUIRED:

1. CPU with Pentium IV processor.
2. Display with highest 32 bit color monitors with screen resolution 1024 x 768 pixels.
3. Scroll mouse.

SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. PTC/Creo

PROCEDURE:

1. Identify various parts to be created.
2. Enter into part environment and create the main part of the assembly.
3. Identify whether the main part or the first part environment is to be created by protrusion or by revolution.
4. Select the sketch tool, then coincidental plane option, from these select any one of the standard 3 planes (i.e. front, right &top).
5. Create the cross-section profile as a closed one by using the available 2D commands , then the click finish button for completing for completing the profile. .
6. Select sketch both parallel plane option or plane by 3points option for creating other parts and then select the required plane.
7. Do the protrusions by using protrusion command and the revolution by revolved protrusion command.
8. Use hole command and cutout command for constructing holes and cut out.
9. Change the diameter of the hole by using modify menu or resize hole option by using hole command.
10. Use revolve cutout command whenever in need.
11. Use the distance option to maintain accurate distance between one edge and other edge (or) one edge and center the hole.
12. Save it in separate part file with extension of *. prt. Similarly save it other parts.
13. Enter into the assembly environment, create the 3D model apply assembly constrains available like planer, design, mate, axial align, connect etc.
14. save with an extension of *.asm the various parts of drg.

15. Enter into the frame environment with title block and save with an extension of *.frm
16. Finally , Enter into the drawing environment with the created frame and all the models in the drawing sheet and save with an extension of *.drw

INFERENCE:

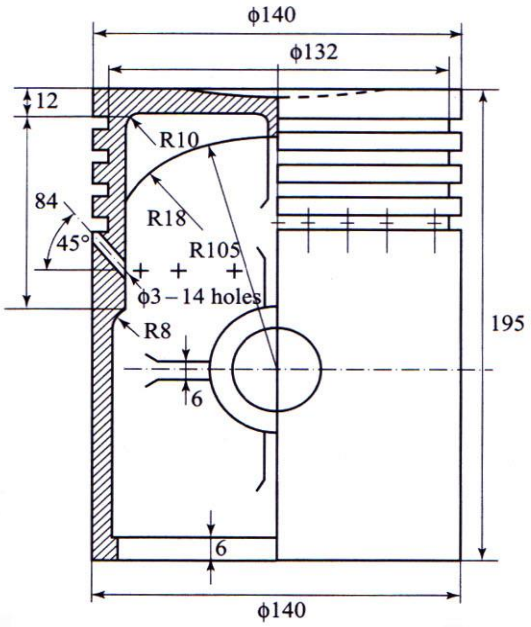
- Practical experience in making 2D Connecting Rod parts of Connecting Rod, Bush, Stud, Bearing Brasses, Distance Piece, Cover, hexagonal nuts and 3D assemble model for Connecting Rod through the exercise.
- Plane selection, revolve, blend cut, extrude, helical sweep, pattern, assembly features like mate, axial align, connect and move through the exercise.

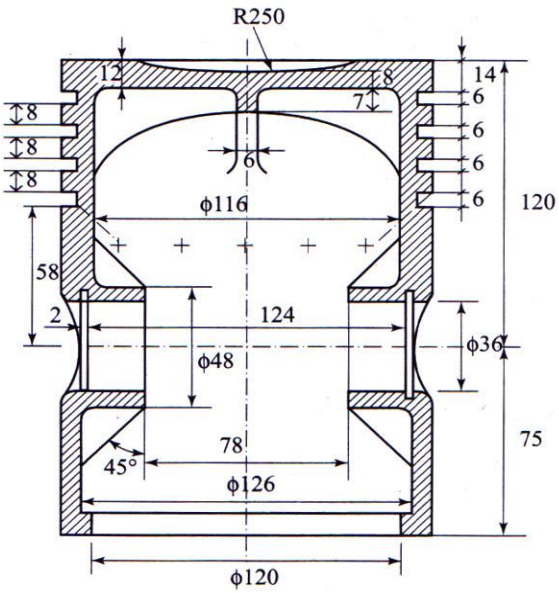
APPLICATIONS:

- Internal combustion engines.
- All cars and trucks that use this type of engine employ the use of connecting rods.
- Farm equipment like tractors and combines also use connecting rods.
- Even construction equipment like bulldozers use internal combustion engines,

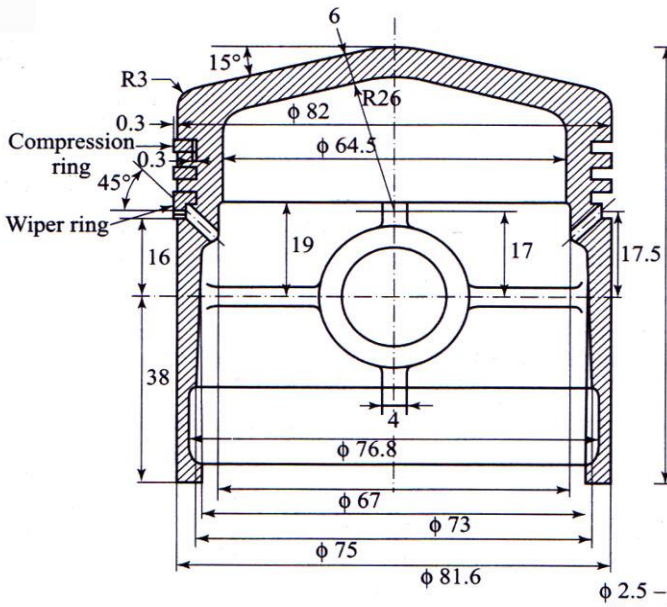
RESULT:

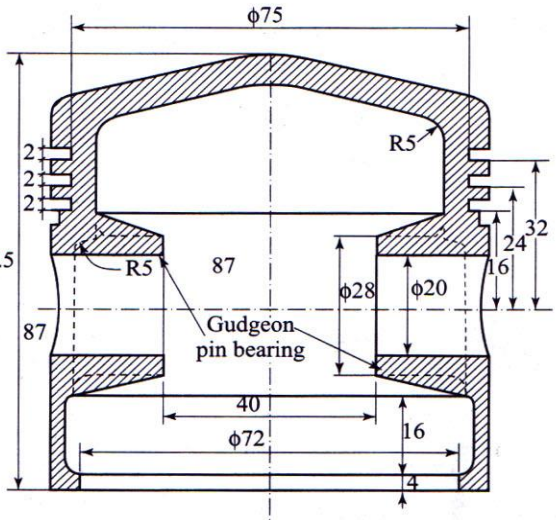
Thus, the 3D assembly of Connecting Rod has been created by using the PTC/Creo software

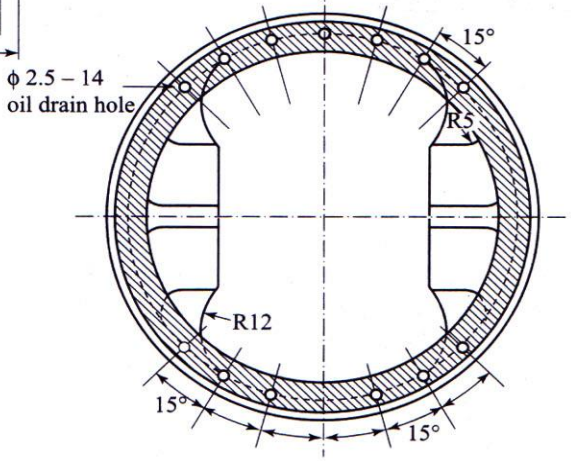




Details of a two-stroke engine







Details of a gasoline engine

Ex. No : 12 Date :

MANUAL PARTS DRAWING - PISTON

AIM:

To draw the 2D piston part view manually as shown in fig.

DRAWING INSTRUMENTS REQUIRED:

1. Drawing Board
2. A3 Drawing sheet
3. Mini drafter
4. Set – square
5. Scales
6. Pencils (H, 2H , HB)
7. Protractor
8. Eraser
9. Sharpener
10. Compass
11. Divider
12. French curves

PROCEDURE:

1. Detail study of piston views.
2. Draw margin line, trimmed line, folding mark and title block are drawn as per the dimensions in an A3 size drawing sheet.
3. Draw Assemble view of piston as per the dimensions in an A3 size drawing sheet by using drawing instruments.
4. Mark properly the notation of dimensioning like dimension lines, extension lines, arrow heads. Dimension figures, notes, symbols etc as per general rules for dimensioning.

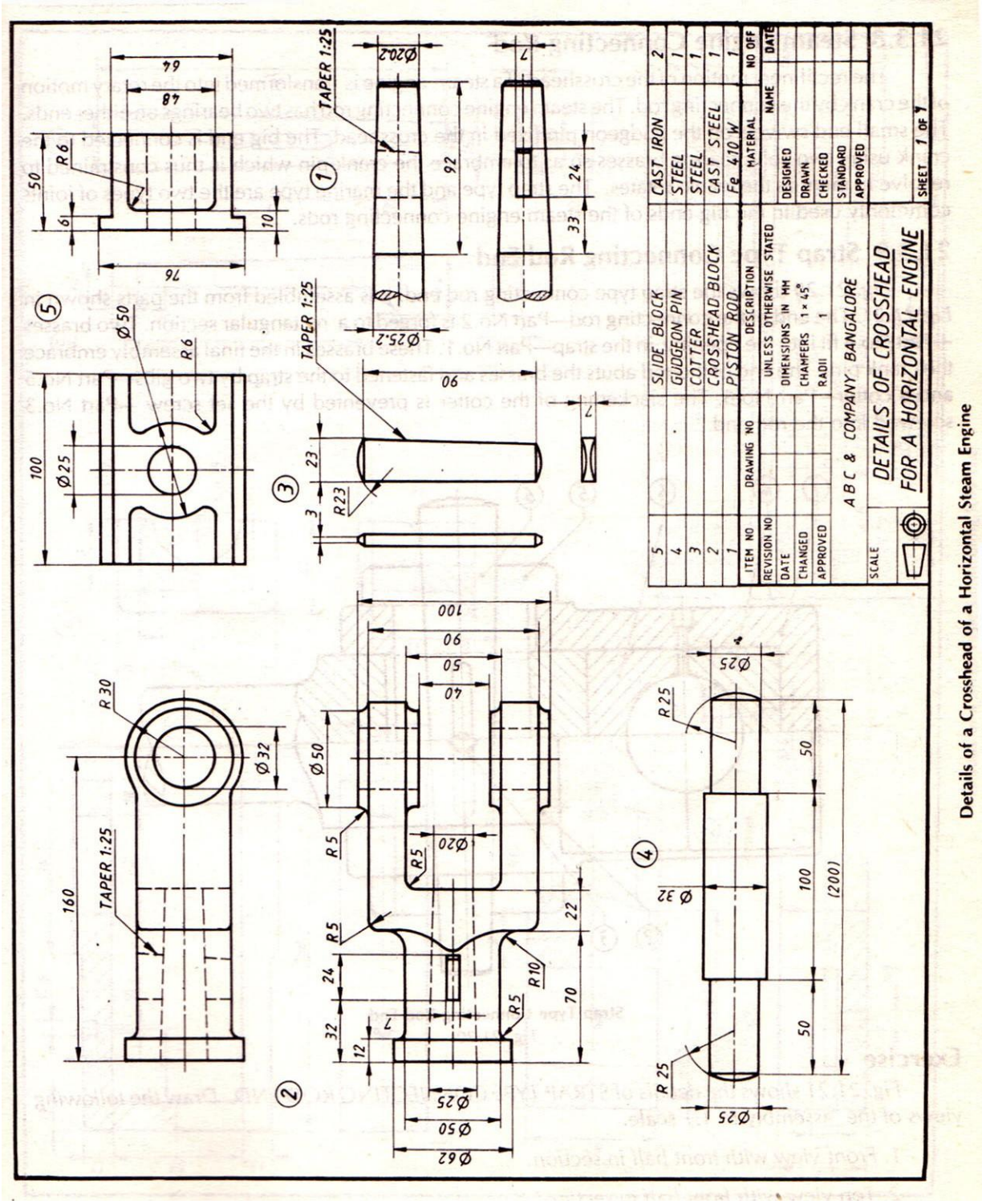
INFERENCE:

- I am familiar with symbols, perspectives, units of measurement, notation systems, visual styles, dimensioning, fits and tolerance etc .

- The drafter uses several technical drawing tools to draw curves and circles. Primary among these are the compasses, used for drawing simple arcs and circles, and the French curve, for drawing curves

RESULT:

Manual drawing of piston is drawn as per the specifications.



Details of a Crosshead of a Horizontal Steam Engine

Ex. No : 13 Date :

MANUAL ASSEMBLE DRAWING - CROSS HEAD

AIM:

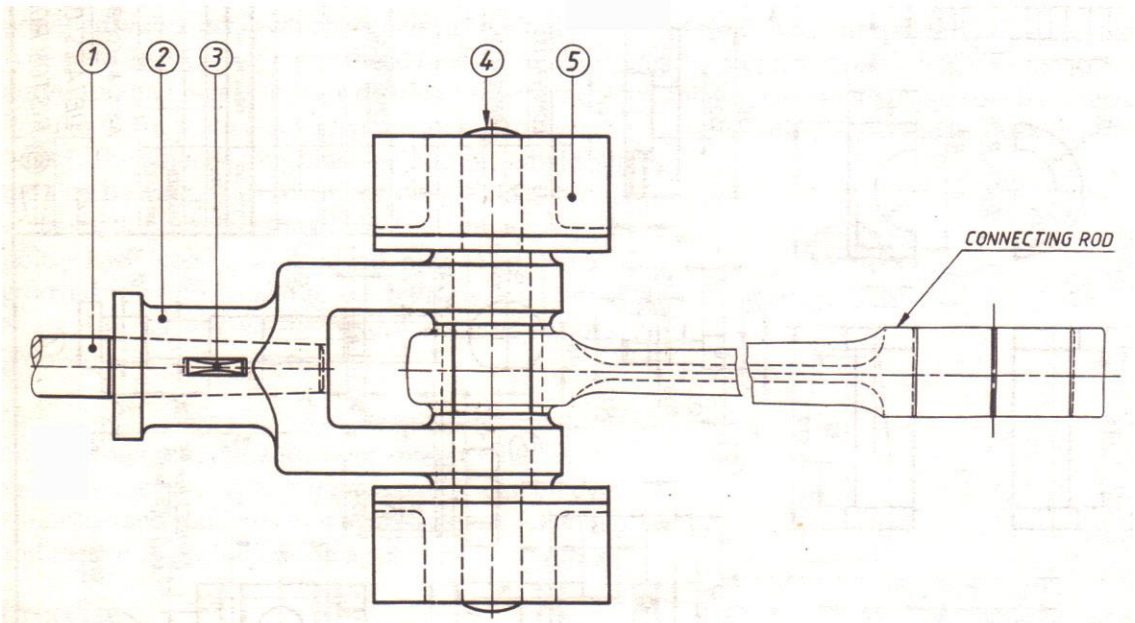
To draw the 2D cross head part view manually as shown in fig.

DRAWING INSTRUMENTS REQUIRED:

1. Drawing Board
2. A3 Drawing sheet
3. Mini drafter
4. Set – square
5. Scales
6. Pencils (H, 2H , HB)
7. Protractor
8. Eraser
9. Sharpener
10. Compass
11. Divider
12. French curves

PROCEDURE:

1. Detail study of cross head views.
2. Draw margin line, trimmed line, folding mark and title block are drawn as per the dimensions in an A3 size drawing sheet.
3. Draw Assemble view of cross head as per the dimensions in an A3 size drawing sheet by using drawing instruments.
4. Mark properly the notation of dimensioning like dimension lines, extension lines, arrow heads. Dimension figures, notes, symbols etc as per general rules for dimensioning.



Top view of a Crosshead of a Horizontal Steam Engine

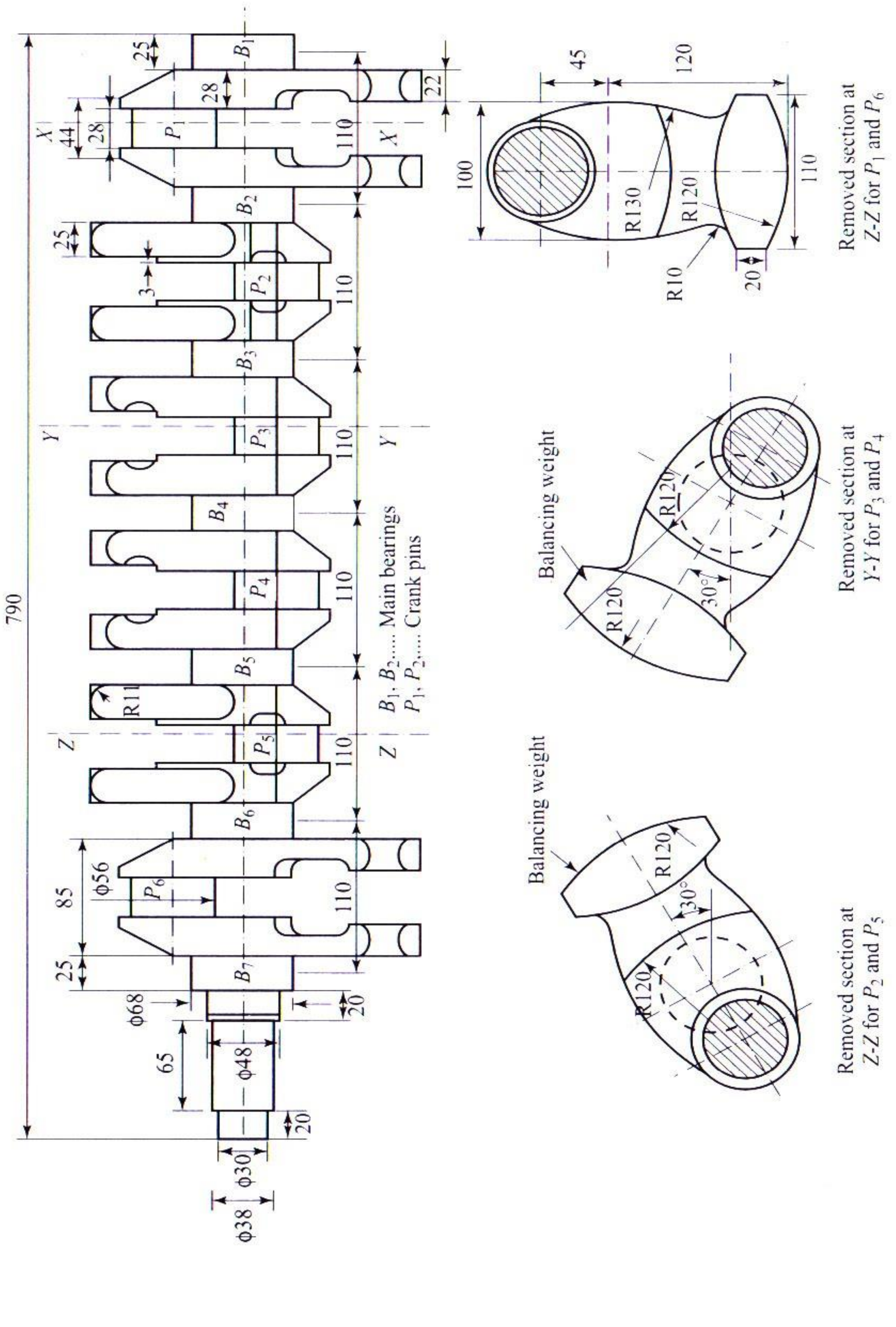
INFERENCE:

- I am familiar with symbols, perspectives, units of measurement, notation systems, visual styles, dimensioning, fits and tolerance etc .

- The drafter uses several technical drawing tools to draw curves and circles. Primary among these are the compasses, used for drawing simple arcs and circles, and the French curve, for drawing curves

RESULT:

Manual drawing of cross head is drawn as per the specifications.



Crankshaft of six cylinder diesel engine

Ex. No : 14 Date :

CRANK SHAFT

AIM:

To draw the 2D crank shaft assembly view manually as shown in fig.

DRAWING INSTRUMENTS REQUIRED:

1. Drawing Board
2. A3 Drawing sheet
3. Mini drafter
4. Set – square
5. Scales
6. Pencils (H, 2H , HB)
7. Protractor
8. Eraser
9. Sharpener
10. Compass
11. Divider
12. French curves

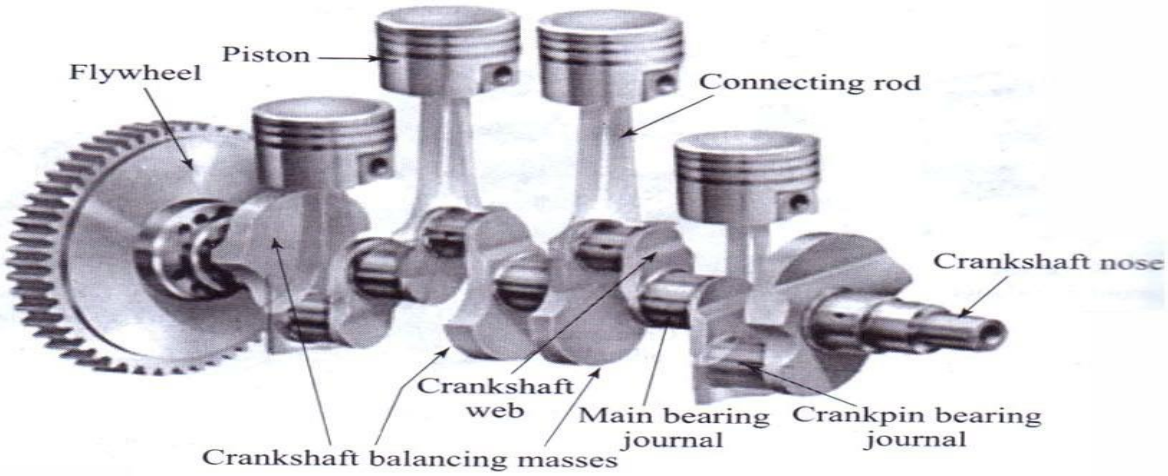
PROCEDURE:

1. Detail study of crank shaft views.
2. Draw margin line, trimmed line, folding mark and title block are drawn as per the dimensions in an A3 size drawing sheet.
3. Draw Assemble view of crank shaft as per the dimensions in an A3 size drawing sheet by using drawing instruments.
4. Mark properly the notation of dimensioning like dimension lines, extension lines, arrow heads. Dimension figures, notes, symbols etc as per general rules for dimensioning.

INFERENCE:

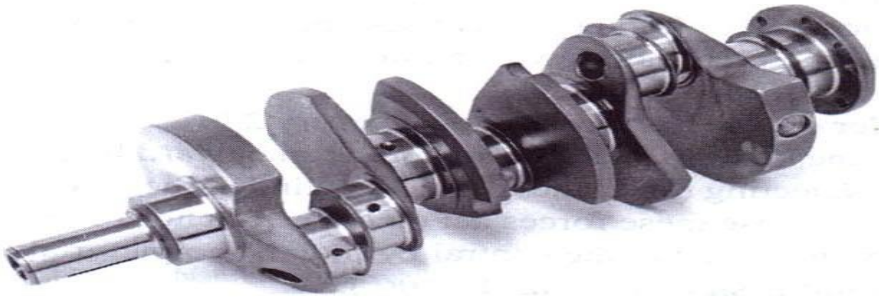
- Practical experience in making of crankshaft parts like flywheel mounting flange, crank pin journals, counter weight, crank, main journal bearing and crank nose.

- ISO standard in drawings and familiar with industrial drafting practices through this handling 2D drafting of crank shaft in assembly exercise.



Four cylinder crankshaft

Inline six cylinder crankshaft



V-6 cylinder crankshaft

V-8 cylinder crankshaft

APPLICATION:

The crankshaft is used everywhere that energy and power are transformed into rotational movements. It has a very wide variety of applications

- Chainsaws
- Compressors / chillers
- Power generators
- (Gas) lawnmowers
- Racecars
- Aircraft
- Rail vehicles
- Motor vehicles such as motorcycles, cars and trucks
- Marine engine

RESULT:

Manual drawing of crank shaft assembly is drawn as per the specifications.

CAM

MANUAL PART PROGRAMMING

Study of G Codes and M Codes to Write Manual Part Programming for Fanuc Control Systems.

AIM:

Study of G Codes and M Codes to Write Manual Part Programming for Fanuc Control Systems.

PREPARATORY FUNCTION (G CODES):

The preparatory functions are the commands, which prepare the machine for different modes of movement like contouring, thread cutting, positioning, peck drilling etc. These functions are also called as G codes. G codes are used as machining commands.

MISCELLANEOUS FUNCTION (M CODES):

The miscellaneous functions are the commands describing miscellaneous functions like subprogram call or exit., spindle clockwise or counter clockwise, coolant ON/ OFF, program end etc. these functions are also called as M codes. M codes are used as machine control commands.

SUBROUTINES:

Subroutines or subprograms are mini programs similar to macros but are used in manual part programming. It can be defined as a set of instructions that can be activated and used for repetitive applications in the main program. In Fanuc control systems, subprograms are written as separate programs with specific subprogram names. When a subprogram name is called in the main program, the set of instructions in the subprogram is executed. After execution of the program flow returns to the main program at a point immediately after the original call point.

CANNED CYCLE:

Canned cycles are built-in- functions that are available to perform specific tasks like thread cutting, grooving, pattern repeating, pocketing etc. A canned cycle simplifies the program using one or two blocks with specific G codes to specify the machining operations, instead of using several blocks.

Examples are,

G71 - Multiple turning cycle
G73 - Pattern turning cycle
G75 - Grooving cycle
G76 - Multiple thread cutting cycle
G170, G171 - Rectangular pocketing
G172, G173 - Circular pocketing

G - CODES

G codes are also called as Preparatory function codes. It always defines the movements or functioning of Tool.

G – Codes for Tuning Center

Denford Fanuc Language

G Code	Function
G00	Rapid Traverse
G01	Linear Interpolation
G02	Circular Interpolation for Clock wise
G03	Circular Interpolation for Counter Clock wise
G04	Dwell time
G20	Data Input in Inches – Inch Mode
G21	Data Input in Millimeters – Metric Mode
G28	Go to Reference Point
G40	Tool nose compensation Cancel
G41	Tool nose compensation at Right
G42	Tool nose compensation at Left
G50	Co-ordinate Setting
G96	Constant Surface Speed OFF – Variable speed
G97	Constant Surface Speed ON
G98	Feed (mm) per Minute
G99	Feed (mm) per Revolution

CANNED CYCLES	
G70	Finishing Cycle
G71	Stock Removal in Turning
G72	Multiple Facing
G73	Pattern Repeating
G74	Peck Drilling
G76	Multiple Threading
G81	Drilling Cycle
G90	Turning Cycle
G94	Facing Cycle

M - CODES

M codes are also called as Miscellaneous function codes. It always defines the functioning of Machine.

M – Codes for Tuning Center

Denford Fanuc Language

M Code	Function
M00	Program Stop
M01	Optional Stop
M02	Program End
M03	Spindle Rotation in Forward / Clock Wise
M04	Spindle Rotation in Reverse / Counter Clock wise
M05	Spindle Stop
M06	Tool Change
M08	Coolant ON
M09	Coolant OFF
M10	Chuck OPEN
M11	Chuck CLOSE
M30	Program End and Begin to Start Block
M98	Sub Program Call
M99	Sub Program Exit

NC Program

[BILLET X25.40 Z90.00…… FIRST LINE….. Express the Raw Material size

G21G98G40…….SECOND LINE…Input Method (Metric/feed rate/tool nose radius compensation cancel)

G28U0.00 W0.00 ….THIRD LINE……Move the Tool post to the M/C Reference point

M06 T0101 ……FOURTH LINE……Call the New / Required Tool

M03 S1500 …… FIFTH LINE……M/C spindle Rotation CW, Speed

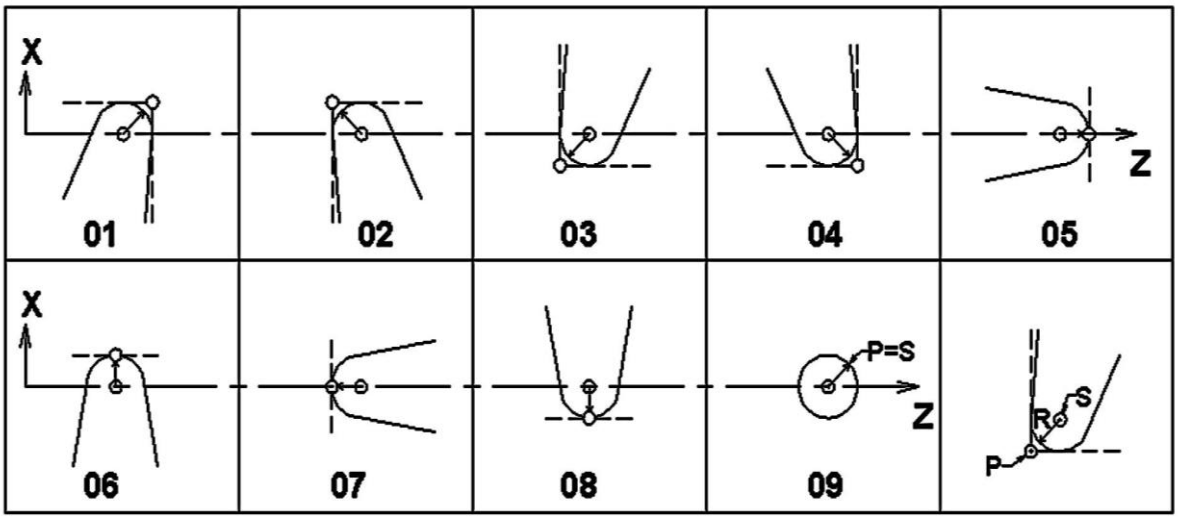
The above 5 basic lines are commonly used in all programs. And they always come at the beginning lines. (I.e. same as above format)

VARIOUS TOOL NOSE RADIUS

R - Tool Nose Radius (Tool Radius)

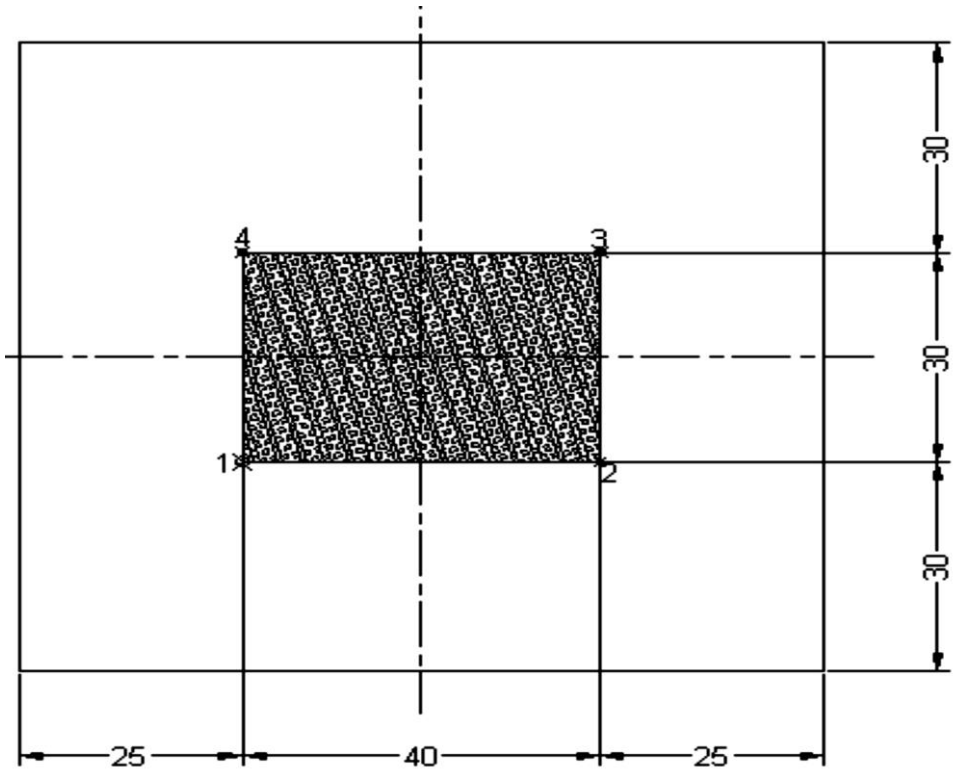
S - Position of Cutting Edge Center Point

P - Tool Tip (Cutting Edge)



RESULT:

Thus the G- Codes and M- Codes were studied.



Rectangular Pocketing : Billet size: - 90 x 90 x 10 mm; Depth of cut – 3 mm

Ex. No : 1 Date :

RECTANGULAR POCKETING

AIM :

To Write a CNC program for the profile given using rectangular pocketing syntax.

SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. Denford cam software.

Rectangular Pocketing Syntax:

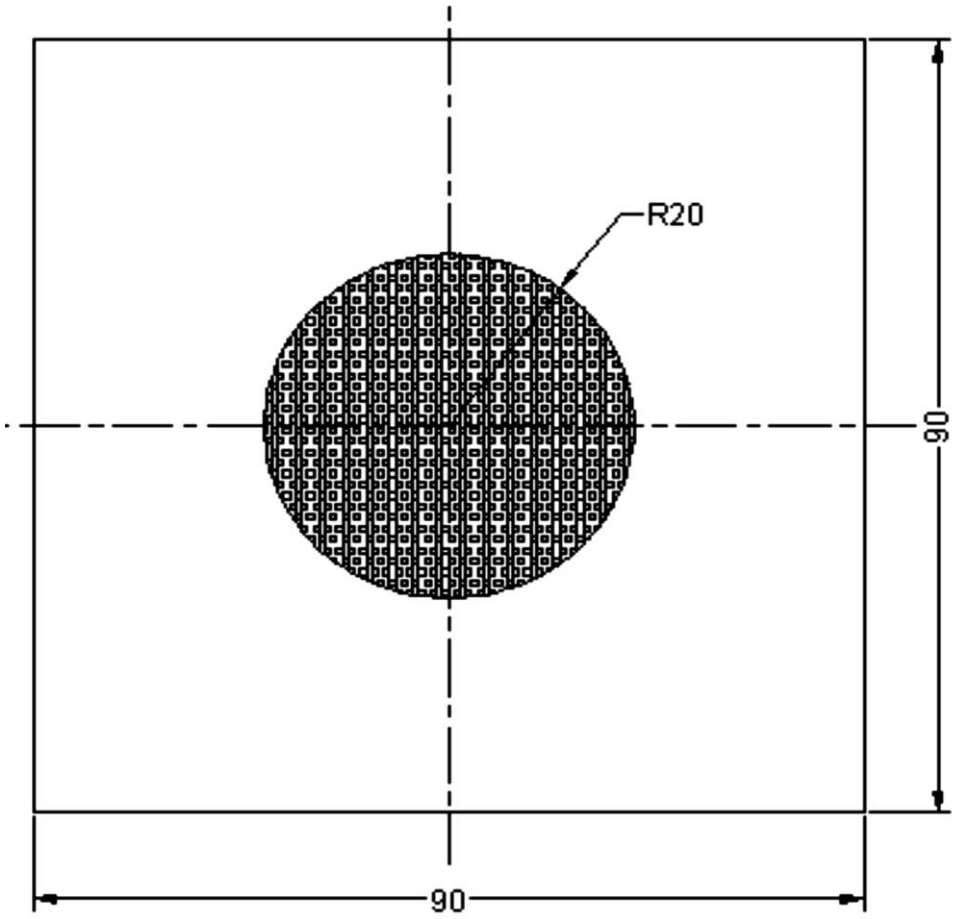
O1000 [BILLET X90 Y90 Z10 [TOOLDEF T1 D10 [EDGEMOVE X-45 Y-45 N10 G21 G40 G94 N20 G91 G28 Z0 N30 G28 X0 Y0 N40 M06 T0101 N50 M03 S2500 N60 G90 G00 X0 Y0 N70 Z5 N80 G172 I40 J30 K0 P0 Q0.5 R0 X-20 Y-15 Z-3 N90 G173 I0 K0 P50 T01 S1500 R15 F230 B2500 J15 Z-3 N100 G91 G28 Z0 N110 G28 X0 Y0 Z0 N120 M05 N130 M30	The syntax for **rectangular pocketing** is given below: **G172 I30 J50 K0 P0 Q.5 R0 X10 Y10 Z-5** **G173 I0 K0 P75 T1 S1000 R35 F45 B1500 J15 Z5** Where I,J - Length of pocket in X, Y directions K - Corner radius P - Roughing (0) / Finishing (1) Q - Depth of cut for each pass R - Absolute depth from the surface X & Y - Pocket corner coordinates Z - Base of pocket I,K - Pocket side and base finish allowance P - Cutter width percentage T - Tool number S - Roughing spindle speed R - Roughing feed in Z direction F - Roughing feed in XY directions B - Finishing spindle speed J - Finishing feed

INFERENCE:

Practiced with G codes and M codes for rectangular pocketing (**G172, G173**) through this exercise.

RESULT:

Thus, the part program for Rectangular Pocketing is developed and simulated in the software.



Circular Pocketing : Billet size: - 90 x 90 x 10 mm; Depth of cut – 3 mm

Ex. No : 2 Date :

CIRCULAR POCKETING

AIM :

To Write a CNC program for the profile given using Circular pocketing syntax.

SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. Denford cam software.

Circular Pocketing Syntax:

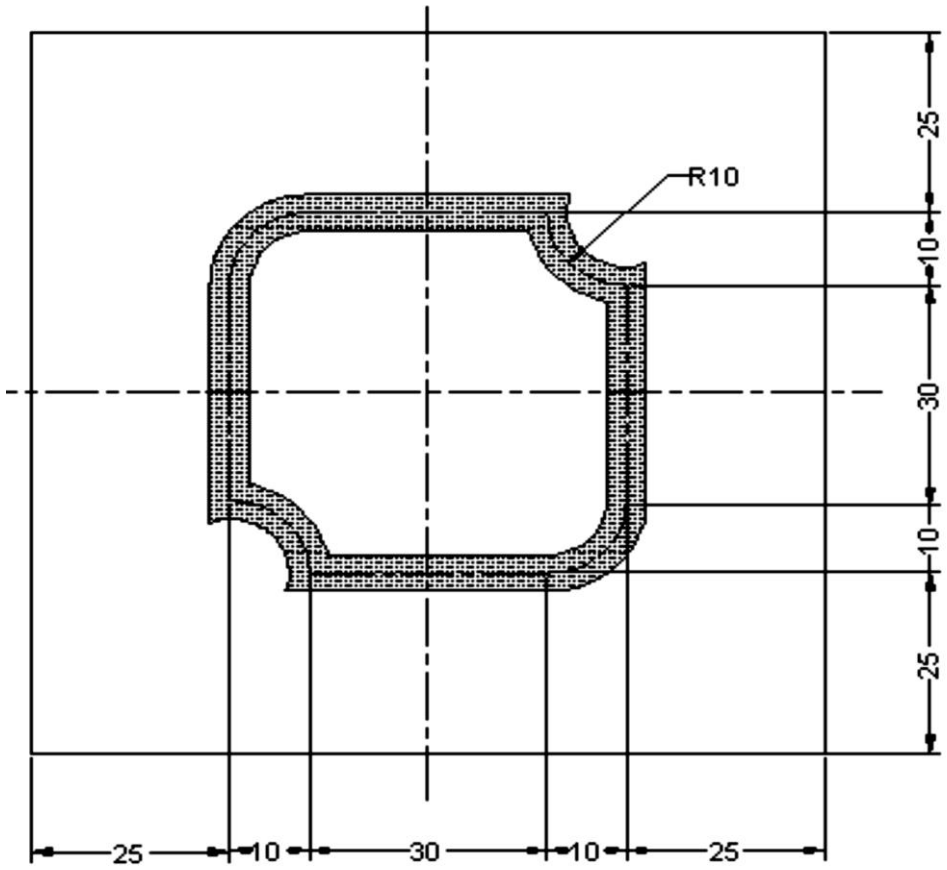
O2000 [BILLET X90 Y90 Z10 [TOOLDEF T1 D10 [EDGEMOVE X-45 Y-45 N10 G21 G40 G94 N20 G91 G28 Z0 N30 G28 X0 Y0 N40 M06 T0101 N50 M03 S2500 N60 G90 G00 X0 Y0 N70 Z5 N80 G170 R0 P0 Q0.5 R0 X0 Y0 Z-3 I0 J0 K20 N90 G171 P50 S1500 R15 F20 B2000 J15 Z3 N100 G91 G28 Z0 N110 G28 X0 Y0 Z0 N120 M05 N130 M30	The syntax for **circular pocketing** is given below. **G170 R0 P0 Q3 X10 Y10 Z - 5 I0 J0 K24** **G171 P75 S1000 R50 F45 B1500 J15** R - Position of tool to start cycle P - Roughing (0) / finishing (1) Q - Peck increment for each cut X,Y& Z - Coordinates of bottom center of the circular pocket. I & J - Finishing allowance for side and pocket base K - Radius of circular pocket P - Cutter movement percentage for next step S - Roughing spindle speed R - Roughing feed in Z direction F - Roughing feed in XY directions B - Finishing spindle speed J - Finishing feed

INFERENCE:

Practiced with G codes and M codes for Circular pocketing (**G170, G171**) through this exercise.

RESULT:

Thus, the part program for Circular Pocketing is developed and simulated in the software.



Simple Contour Milling: Billet size: - 100 x 100 x 10 mm; Depth of cut – 1 mm

Ex. No : 3 Date :

LINEAR AND CIRCULAR INTERPOLATION

AIM :

To Write a CNC program for the profile given using Linear and Circular interpolation syntax.

SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. Denford cam software.

Linear And Circular Interpolation Syntax:

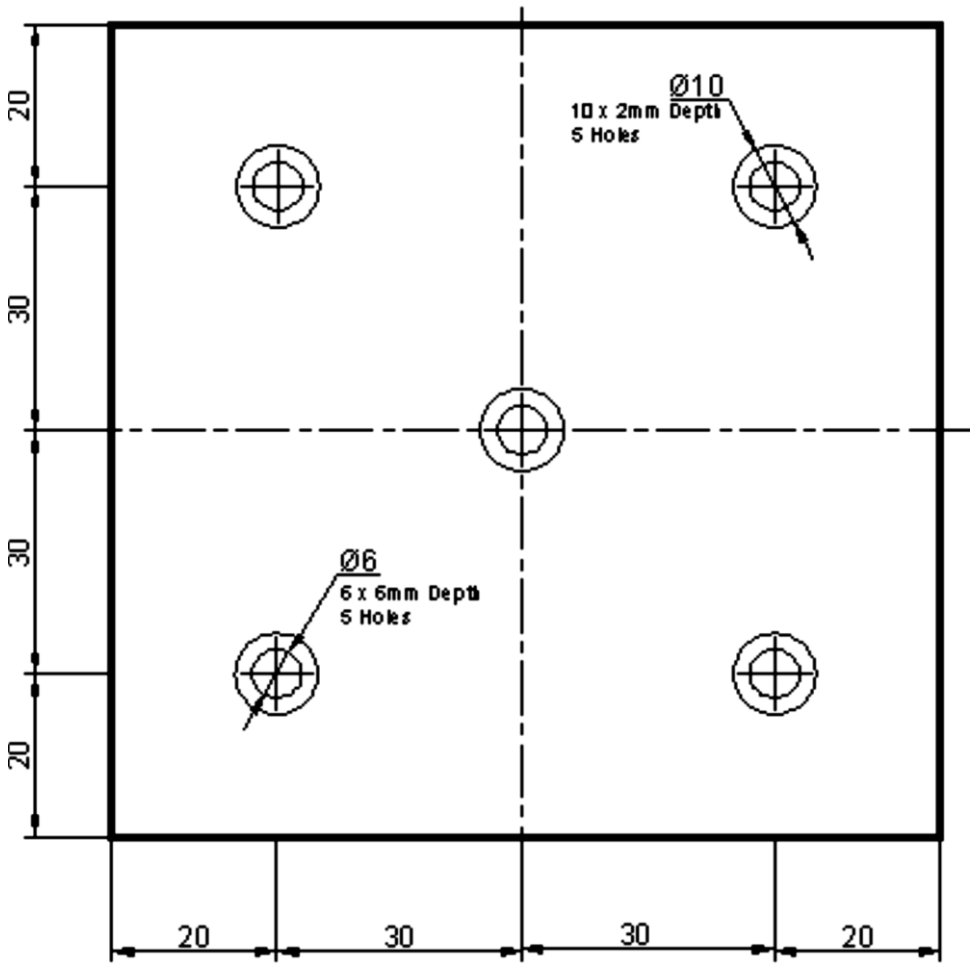
O3000 [BILLET X100 Y100 Z10 [TOOLDEF T1 D05 [EDGEMOVE X0 Y0 N10 G21 G40 G94 N20 G91 G28 Z0 N30 G28 X0 Y0 N40 M06 T0101 N50 M03 S2500 N60 G90 G00 X35.0 Y25.0 Z5.0 N70 G01 Z-1 F15 N80 X65 F30 N90 G03 X75 Y35 R10 F20 N100 G01 Y65 F30 N110 G02 X65 Y75 R10 F20 N120 G01 X35 F30 N130 G03 X25 Y65 R10 N140 G01 Y35 N150 G02 X35 Y25 R10 F20 N160 G00 Z1 N170 G91 G28 Z0 N180 X0 Y0 N190 M05 N200 M30 G00 - Rapid Traverse G90 – absolute mode G01 - Linear Interpolation G02 - Circular Interpolation for Clock wise G03 - Circular Interpolation for Counter Clock wise	[BILLET X100 Y100 Z10.00…… FIRST LINE….. Express the Raw Material size [TOOLDEF T1 D05 - SECOND LINE… Tool number , D05 – diameter of the tool [EDGEMOVE X0 Y0 - THIRD LINE…… set the datum point G21 G98 G40…….FOURTH LINE…Input Method (Metric/feed rate/tool nose radius compensation cancel) G28 U0.00 W0.00 ….FIFTH LINE……Move the Tool post to the M/C M06T0101 ……SIXTH LINE……Call the New / Required Tool M03S2500 …… SEVENTH LINE……M/C spindle Rotation CW, Speed The above 5 basic lines are commonly used in all programs. And they always come at the beginning lines. (I.e. same as above format)

INFERENCE:

Practiced with G codes and M codes for Linear and Circular interpolation (**G01, G02 and G03**) through this exercise.

RESULT:

Thus, the part program for Linear and Circular interpolation is developed and simulated in the software.



Canned Cycle : Billet size**:** 100 x 100 x 10 mm; Drilling Tool : **Ø** 6; Boring Tool : **Ø** 10

Ex. No : 4 Date :

CANNED CYCLES

AIM :

To Write a CNC program for the profile given using Canned Cycle syntax.

SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. Denford cam software.

Canned Cycle Syntax:

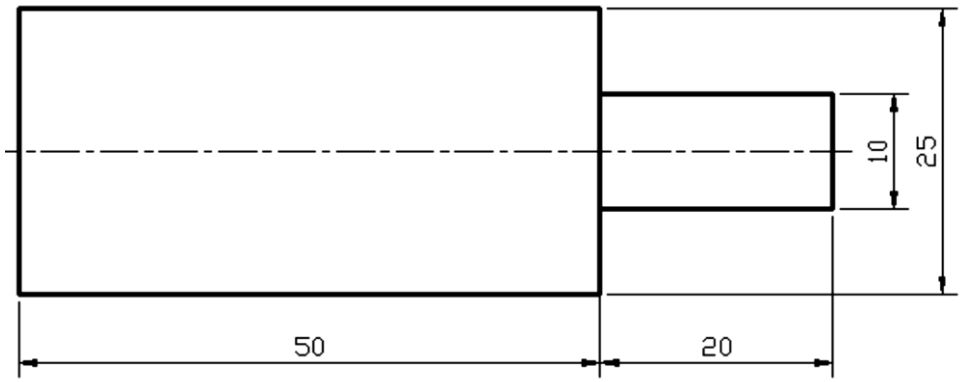
O6500 [BILLET X100.00 Y100.00 Z10.00; [TOOLDEF T01 D06; [TOOLDEF T02 D10; [EDGEMOVE X-50.00 Y-50.00; N10 G21 G94 G40; N20 G91 G28 Z0.00; N30 G28 X0.00 Y0.00; N40 M06 T0101; N50 M03 S2500; N60 G90 G00 X0.00 Y0.00; N70 Z5.00; N80 G83 X0 Y0 Z-6.00 Q1.5 R0 F25; N90 X-30 Y-30; N100 X30 Y-30; N110 X30 Y30; N120 X-30 Y30; N130 G80; N140 G91 G28 X0.00 Y0.00 Z0.00; N150 M01; N160 M06 T0202; N170 M03 S2500; N180 G90 G00 X0.00 Y0.00; N190 Z5.00; N200 G83 X0 Y0 Z-2.00 Q0.5 R0 F25; N210 X-30 Y-30; N220 X30 Y-30; N230 X30 Y30; N240 X-30 Y30; N250 G80; N260 G91 G28 X0.00 Y0.00 Z0.00; N270 M05; N280 M30;	***SYNTAX*** ***G83- CANNED CYCLES*** **G83 X0 Y0 Z-6.00 Q1.5 R0 F25**; **X , Y - Coordinates Of Bottom Center** **Z-6 Denotes The Depth Of The Drill** **Q1.5 Denotes The Depth Of Cut For Each Peck** **R –Retraction** **F - Feed** ***G80 – CANNED CYCLES OFF*.**

INFERENCE:

Practiced with G codes and M codes for Canned Cycle (**G83 and G80**) through this exercise.

RESULT:

Thus, the part program for Canned Cycle is developed and simulated in the software.



Turning Cycle : Billet size**:** Ø 25 x 70 mm

Part Programming - CNC - Turning Centre

Ex. No : 1 Date :

STEP TURNING

AIM :

To Write a CNC program for the profile given using Turning Cycle syntax.

SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. Denford cam software.

Turning Cycle Syntax:

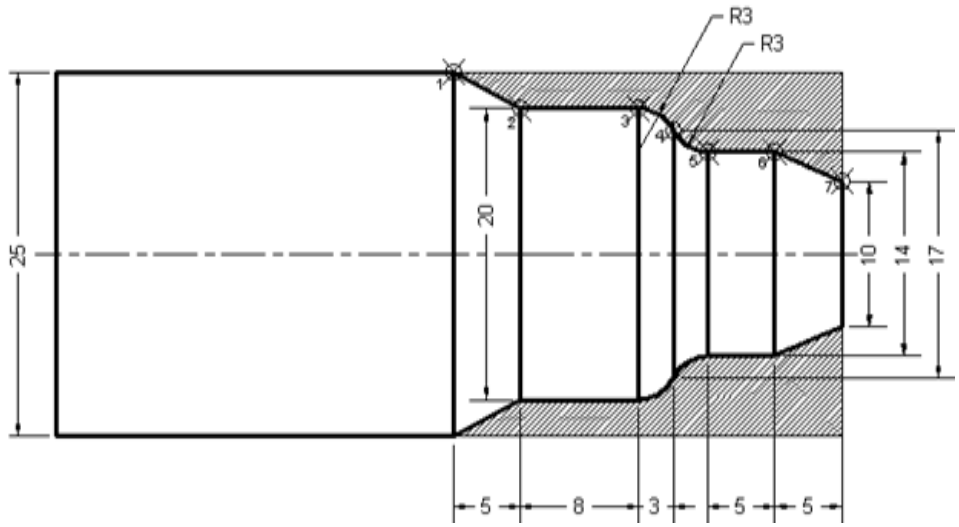
[BILLET X25.00 Z70.00 ; N10 G21 G98 G40 ; N20 G28 U0.00 W0.00 ; N30 M06 T0101 ; N40 M03 S1500 ; N50 G00 X25.00 Z1.00 ; N60 G90 X24.00 Z-20.00 F30 ; N70 X23.00; N80 X22.00 N90 X21.00; N100 X20.00; N110 X19.00; N120 X18.00; N130 X17.00; N140 X16.00; N150 X15.00; N160 X14.00; N170 X13.00; N180 X12.00; N190 X11.00; N200 X10.00; N 210 G28 U0.00 W0.00 ; M05; M30;	**Note :** [BILLET X20.00 Z70.00…… FIRST LINE….. Express the Raw Material size G21 G98 G40…….SECOND LINE…Input Method (Metric/feed rate/tool nose radius compensation cancel) G28 U0.00 W0.00 ….THIRD LINE……Move the Tool post to the M/C M06T0101 ……FOURTH LINE……Call the New / Required Tool M03S1500 …… FIFTH LINE……M/C spindle Rotation CW, Speed The above 5 basic lines are commonly used in all programs. And they always come at the beginning lines. (I.e. same as above format) G00 - Rapid Traverse G 90 --------- TURNING CYCLE F – INDICATES FEED RATE

INFERENCE:

Practiced with G codes and M codes for Turning Cycle (G**90**) through this exercise.

RESULT:

Thus, the part program for Turning Cycle is developed and simulated in the software.



Multiple Facing Cycle : Billet size**:** Ø 25 x 70 mm

Ex. No : 2 Date :

STRAIGHT, TAPER AND RADIUS TURNING

AIM :

To Write a CNC program for the profile given using Multiple Facing Cycle syntax.

SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. Denford cam software.

Multiple Facing Cycle Syntax:

```
O2014
[BILLET X25.00          Z70.00    ;
N10   G21      G98 G40 ;
N20   G28      U0.00     W0.00    ;
N30   M06      T0101     ;
N40   M03      S1500     ;
N50    G00     X25.00    Z0.00    ;
N60    G72    U 0.0  W0.10      R0.20;
N70    G72  P 80  Q140  U 0.0  W0.10  F20.;
N80 G01  Z-29.0;
N90 X20.00   Z -24.0;
N100    Z-16.0
N110    G02  X17.0  Z-13.0  R3.0 ;
N120    G03  X14.0  Z-10.0  R3.0;
N130    G01   Z-5.0;
N140    X10.00  Z 0.0;
N 150      G28      U0.00     W0.00    ;
N160     M05 ;
N170     M30  ;
```

G02 - Circular Interpolation for Clock wise
G03 - Circular Interpolation for counter clockwise

Note :

[BILLET X20.00 Z70.00…… FIRST LINE….. Express the Raw Material size

G21 G98 G40……..SECOND LINE…Input Method (Metric/feed rate/tool nose radius compensation cancel)

G28 U0.00 W0.00 ….THIRD LINE……Move the Tool post to the M/C

M06T0101 ……FOURTH LINE……Call the New / Required Tool

M03S1500 …… FIFTH LINE……M/C spindle Rotation CW, Speed

The above 5 basic lines are commonly used in all programs. And they always come at the beginning lines. (I.e. same as above format)

G00 - Rapid Traverse

G01 - Linear Interpolation

G72 – MUL;TIPLE FACING CYCLE

G72 U0.0 W0.0 R0.20;

U and **W** values indicate the cutting depth of both X and Z axis. And **R** indicates the Return value of Tool Retraction.

G72 P80 Q140 U0.0 W0.0 F20;

P indicates the Starting Block Number and **Q** indicate End Block Number of required Profile program.

U and **W** values indicate the Finishing Allowance of both X and Z axes to the finishing (last) pass.

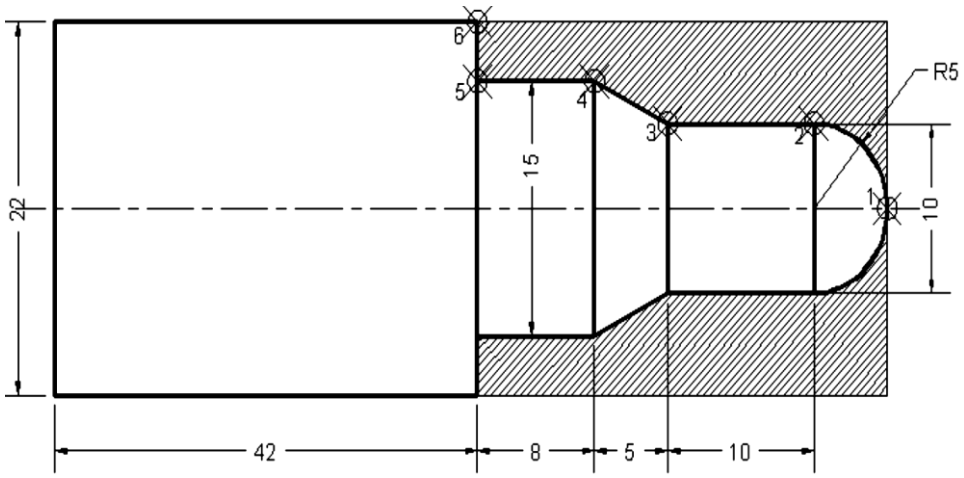
F - indicates the Feed Rate of profile program.

INFERENCE:

Practiced with G codes and M codes for Multiple Facing Cycle (G**72**) through this exercise.

RESULT:

Thus, the part program for Multiple Facing Cycle is developed and simulated in the software.



Multiple Turning Cycle : Billet size**:** Ø 22 x 65 mm

Ex. No : 3 Date :

MULTIPLE TURNING CYCLE

AIM :

To Write a CNC program for the profile given using Multiple Turning Cycle syntax.

SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. Denford cam software

Multiple Turning Cycle Syntax:

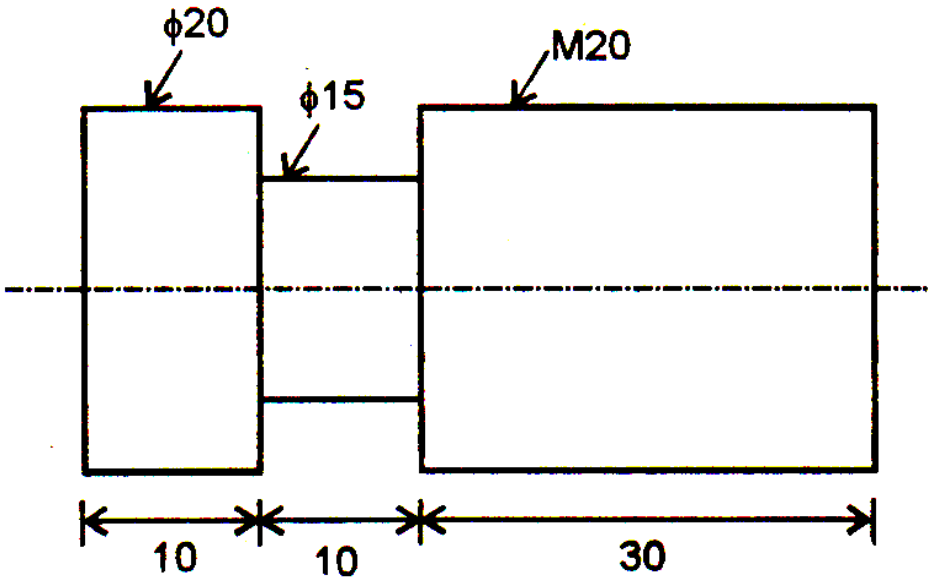
O5000 [BILLET X22.00 Z65.00 ; N10 G21 G98 G40 ; N20 G28 U0.00 W0.00 ; N30 M06 T0101 ; N40 M03 S1500 ; N50 G00 X22.00 Z1.00 ; N60 G71 U 0.20 W0.10 R0.40; N70 G71 P 80 Q140 U 0.0 W0.10 F20.; N80 G00 X0.0; N90 G01 Z0.0; N100 G03 X10.0 Z-5.0 R5.0; N110 G01 Z-15.0; N120 X15.00 Z-20.0; N130 Z-28.0; N140 X22.0; N 150 G28 U0.00 W0.00 ; N160 M05 ; N170 M30 ; G03 - Circular Interpolation for counter clockwise	**Note :** [BILLET X22.00 Z65.00…… FIRST LINE….. Express the Raw Material size G21 G98 G40…….SECOND LINE…Input Method (Metric/feed rate/tool nose radius compensation cancel) G28 U0.00 W0.00 ….THIRD LINE……Move the Tool post to the M/C M06T0101 ……FOURTH LINE……Call the New / Required Tool M03S1500 …… FIFTH LINE……M/C spindle Rotation CW, Speed The above 5 basic lines are commonly used in all programs. And they always come at the beginning lines. (I.e. same as above format) G00 - Rapid Traverse G01 - Linear Interpolation G71– MULTIPLE TURNING CYCLE G71 U0.0 W0.0 R0.20; **U** and **W** values indicate the cutting depth of both X and Z axis. And **R** indicates the Return value of Tool Retraction. G71 P80 Q140 U0.0 W0.0 F20; **P** indicates the Starting Block Number and **Q** indicate End Block Number of required Profile program. **U** and **W** values indicate the Finishing Allowance of both X and Z axes to the finishing (last) pass. **F** - indicates the Feed Rate of profile program.

INFERENCE:

Practiced with G codes and M codes for Multiple Turning Cycle (**G71**) through this exercise.

RESULT:

Thus, the part program for Multiple Turning Cycle is developed and simulated in the software.



Grooving and Thread Cutting Cycle : Billet size: Ø 20 x 50 mm

Ex. No : 4 Date :

GROOVING AND THREAD CUTTING

AIM :

To Write a CNC program for the profile given using Grooving and Thread Cutting Cycles syntax.

SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. Denford cam software

Grooving And Thread Cutting Cycle Syntax:

O2013; [BILLET X20 Z50; G21 G98 G40; G28 U0 W0; M06 T0101; M03 S750; G00 X22 Z-30; N10 G75 R1.0; N20 G75 X15 Z-40 W – 3 P250 Q2000 R0 F15; G28 U0 W0; N30 M06 T0202; M03 S300; G00 X20 Z01; G76 P031560 Q250 R0.15; N40 G76 X18.16 Z-30 P920 Q300 F1.5; G28 U0 W0; M05; M30;	***GROOVING*** Syntax of G75 is given below. G75 R0.1; R in the first block is used to define the relief amount G75 X15 W-10 P250 Q2500 R0.15 F15; X - Absolute depth position of the groove W - Incremental Width of the groove P - Depth of cut along X axis (microns) F - Feed in mm/min R0.15 - This defines the finishing allowance that should be left for finishing. ***THREADING*** **Syntax of G76 is given below.** G76 P031560 Q250 R0.15; G76 X18.16Z-30 P920Q300 F1.5; G76 requires two blocks to define all the P031560 - There are six digits after P. The first two digits (03) define the number of passes for finishing. The next two digits (15) define the chamfer amount. The last two digits (60) define the angle of thread in degrees. Q250 - This defines the minimum depth of cut during the thread cutting operations in microns. R0.15 - This defines the finishing allowance that should be left for finishing

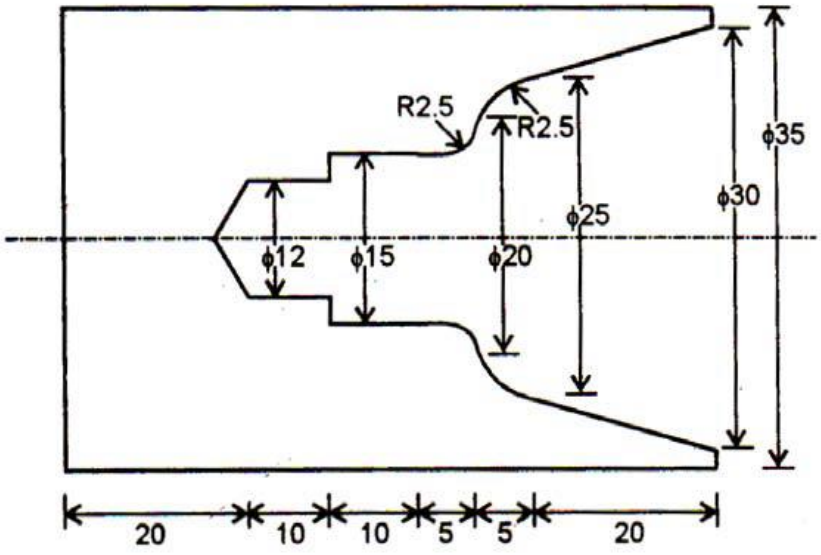
	X18.16- This denotes the core diameter for M20 thread in mm. (Refer Table 3.1) Z-40 - This denotes the length of the thread. P920 - This denotes the height of the thread in microns for M20 thread. Q300 - This denotes the depth of cut for the first pass in microns. F1.5 - This denotes the pitch in mm for M20 Thread.

INFERENCE:

Practiced with G codes and M codes for Grooving and Thread Cutting Cycles (**G75, G76**) through this exercise.

RESULT:

Thus, the part program for Grooving and Thread Cutting Cycle is developed and simulated in the software.



End Face Peck Drilling and Boring Cycle : Billet size**:** Ø 35 x 70 mm

Ex. No : 5 Date :

END FACE PECK DRILLING CYCLE AND BORING CYCLE

AIM :

To Write a CNC program for the profile given using End Face Peck Drilling and Boring Cycles syntax.

SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. Denford cam software

End Facing Peck Drilling And Boring Cycle Syntax:

O2013; [BILLET X35 Z70; G21 G98 G40; G28 U0 W0; M06 T0202; M03 S500; G00 X0 Z5; G74 R1; G74 X0 Z-50 Q500 F15; G28 U0 W0; M06 T0101; M03 S1000; G00 X12 Z01; G71 U0.5 R1; G71 P10 Q20 U0 W0 F15; N10 G01 Z0; X30; X25 Z-20; G03 X20 Z-25 R5; G02 X15 Z-30 R5; G01 Z-40; N20-X12;	G28 U0 W0; M05; M30; Syntax of G74 cycle is given below. G74 R1; G74 X0 Z-50 Q500 F15; R in first block defines the relief amount (tool retraction after each peck). X0 Z-50 denotes the coordinates of the tool at the bottom of the hole. Q500 - denotes the depth of cut during each peck in microns. F15 - Feed rate G74 cycle is used for drilling longitudinal holes in the work piece. After drilling, boring operations can be done by using G71 cycle with a suitable boring tool. In boring operation, using G71, the profile of the boring operation should be of decreasing type. A typical boring operation using G71 cycle for the above Fig. is explained in the Program before starting boring operation, a drill is made to enable boring using G74 cycle.

INFERENCE:

Practiced with G codes and M codes for End Face Peck Drilling and Boring Cycles (**G74, G71**) through this exercise.

RESULT:

Thus, the part program for End Face Peck Drilling and Boring Cycle is developed and simulated in the software.

COMPUTER AIDED PART PROGRAMMING

Study of NC code generation using EDGECAM software and Post processing for Fanuc CNC control

AIM:

To Study of NC code generation using EDGECAM software and Post processing for Fanuc CNC control

NC code generation using Pro E /EDGECAM software and Post processing for Fanuc CNC Control steps:

1. Create the model in part mode and save in working directory.
2. Assemble the Ref. model in manufacturing mode by default.
3. Create work piece by manual
4. Create coordinate system for work piece zero.
5. Go to steps – operation – give m/c details, work piece zero using coordinate system and select coordinate starting surface in z direction as 2 mm and tolerance as 0.1 mm
6. Go to turn profile icon – use cross section item – placement – select coordinate system define start and end points.
7. Select Area Turning icon- Tick tool, parameter and tool motion – give necessary informations for tool motions – select insert – choose the defined profile. Correct the start and end direction – click ok.
8. Play path – Screen play.
9. Save CL data and open with doc. file

RESULT:

Thus the NC code generation using Pro E / EDGECAM software steps and Post processing for Fanuc CNC control steps were studied.

Development of 3D component using 3D printer

AIM:

To develop the given 3D component using 3D printer.

System and Software Required

1. 3D modeling software
2. CURA slicing software

3D printing Definition

3D models are the foundation of 3D printing. Every print begins with a digital model designed using CAD software, which is then converted or "sliced" into instructions that can be understood by the 3D printer. The outcome of the print is a physical representation of the digital 3D model.

Steps to be followed to develop the given 3D component.

1. First Create given 3D model using any 3D modeling software like Creo, Solidworks etc.
2. Save the above 3D modeling file as .stl file.
3. Using Slicing software open the above stl file
4. It will shows how many time is required to develop the model and the material quantity is required
5. You can adjust the time using the scale command as per our convenient.
6. Select the material (PLA/ABS/PETG&TPU) for the machine available.
7. Run the machine and develop the model.

Free slicers for 3D printing

- **Cura**: Ultimaker's free-to-use and open-source slicer is popular with professionals and hobbyists alike. Offering a simple workflow and downloadable plugins, Cura divides the slicing process into three stages: Prepare, Preview, and Monitor. Helpful tools include print time and filament usage estimates.
- **Slic3r**: Developed and maintained by a group of RepRap community members, the open-source Slic3r is regularly updated by its active community of open-source developers. With features like auto-repair and multi-extruder slicing, Slic3r is a good tool for those interested in pushing the boundaries of their 3D printer.
- **OctoPrint**: Primarily a tool for remote 3D printer management (including webcam monitoring of prints), the free OctoPrint software also offers slicing features.

Materials of 3D printing

PLA - Polylactic Acid (Most popular material)

ABS - acrylonitrile butadiene styrene
PETG - Polyethylene terephthalate glycol

RESULT:

Thus the given 3D model is developed using 3D printer.

Cutting the given Profile using Wire Cut EDM Machine

AIM:

To cut the given profile using Wire cut EDM machine.

Wire cut EDM machine working principle:

Wire-Cut Electrical Discharge Machining (WEDM) is an electro thermal production process in which a thin metallic wire along with the de-ionized water allows the wire to cut through metal by the use of heat from electrical sparks. The wire is usually made of brass or stratifies copper and the size in between 0.02 and 0.33 mm in diameter.

Working Principle of Wire-Cut EDM:

Figure shows the working principle of wire cut electrical discharge machining. In this type of machining the wire is used as the form of tool electrode.

It is very thin wire of diameter 0.05mm to 0.25mm. The wire is connected to the negative terminal and work piece is connected to the positive terminal of the DC power supply same as the working of electro discharge machining. The dielectric is calculated to the storage of tank at high pressure pump. The spark can be generated due to current flowing from a wire and the process is done.

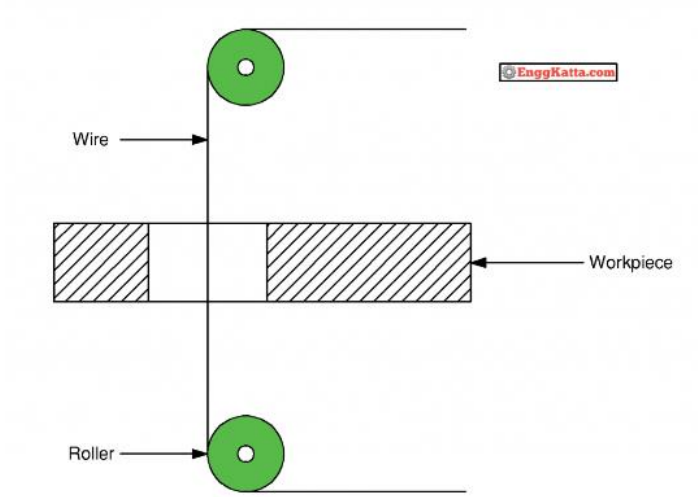


Figure: **Working Principle of Wire-Cut EDM**

Controlling Parameters of WEDM:

Following are the controlling parameters of wire-cut electrical discharge machining are as follows,

1. **Discharge Current:** The discharge current in between 20 to 30 amperes.
2. **Duration of pulse:** The high duration of pulse goes produce more removal rate and low duration of pulse produce low removal rate.
3. **Frequency of pulse:** The frequency of pulse in between 50KHz to 1MHz.
4. **Wire Diameter:** The diameter of thin wire is 0.05mm to 0.25mm.
5. **Wire Speed:** The speed of wire is about 5 to 200 mm/sec.

Advantages of WEDM:

Following are the advantages of wire-cut electrical discharge machining are as follows,

1. It gives high accuracy.
2. There is no need of storage of tool is required.
3. There is low operating skill is required.
4. Smooth complex surfaces can be possible to machining.
5. A good surface finish can be obtained.

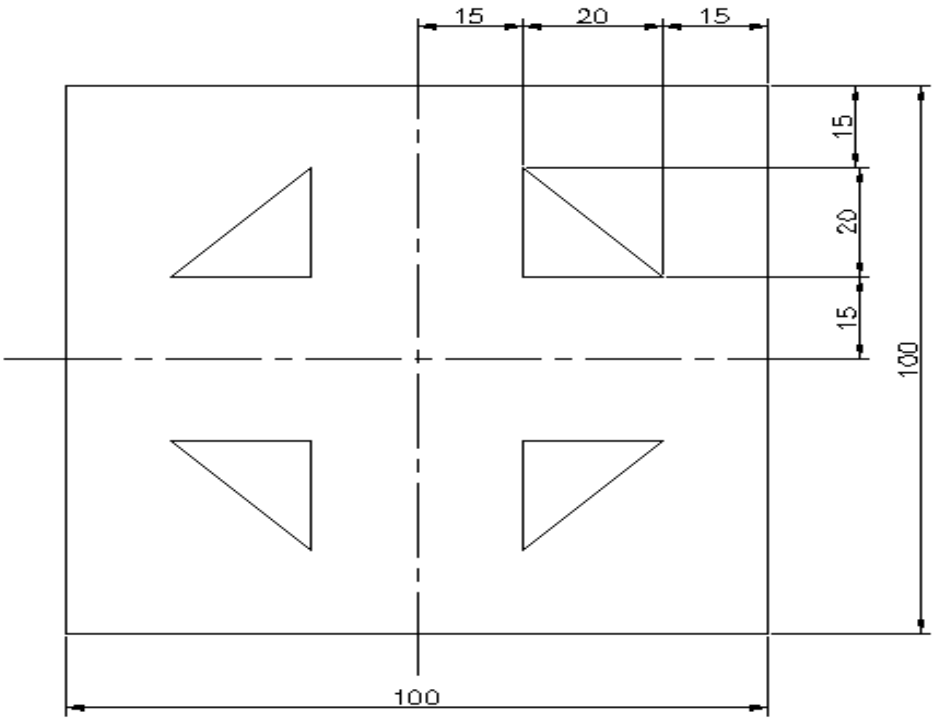
Applications of WEDM:

The following are the applications of wire-cut electrical discharge machining are as follows,

1. It works for production of prototypes.
2. It is used for making of stamping dies, drawings, extrusion tools, etc.
3. IT is used for making blanking dies, plastic molding dies, stamping dies, etc.
4. It is used for gauges, cam discs.
5. It is used to machining larger parts that need to hold accurate tolerances.
6. It is also used for making of small series of spare parts.

Result:

Thus the given 3d profile is cut using wire cut EDM machine.



Contour Milling using Mirroring cycle: Billet size: - 100 x 100 x 10 mm; Depth of cut – 1 mm

CONTENT BEYOND THE SYLLABUS

Ex. No : 1 Date :

MIRROR AND SUB PROGRAMME

AIM :

To Write a CNC program for the profile given using Mirror and Sub Routine syntax.

SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. Denford cam software

Contour Milling Using Mirroring Cycle Syntax:

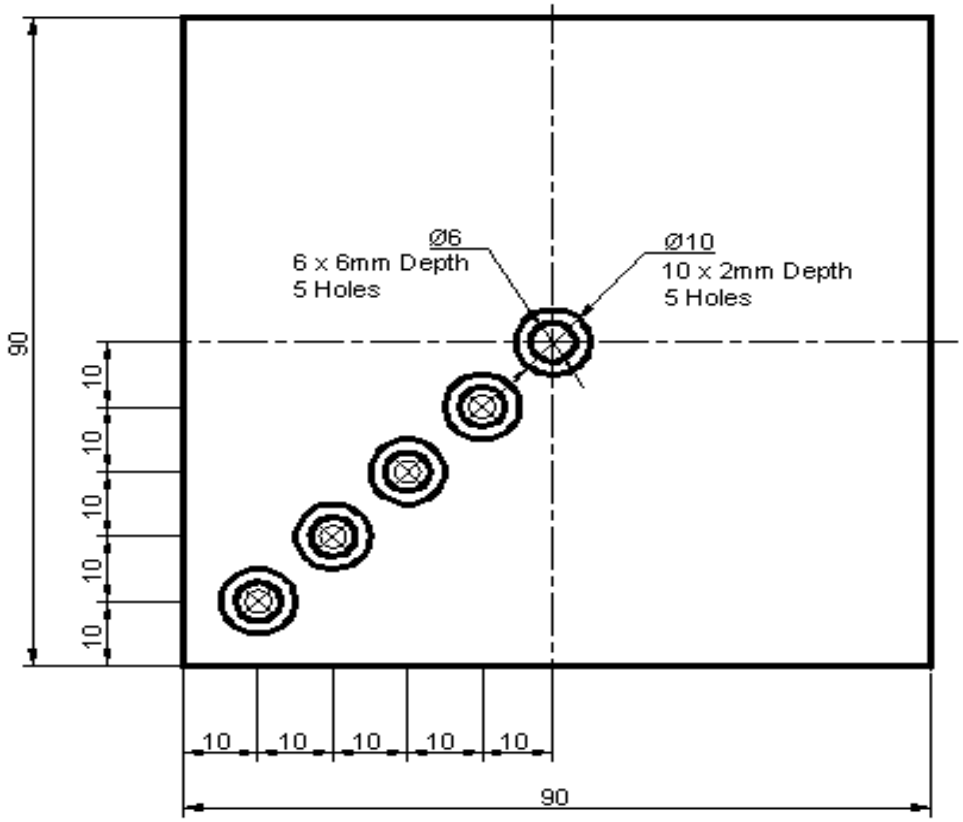
MAIN PROGRAM	
O5000 [BILLET X100.00 Y100.00 Z20.00; [TOOLDEF T01 D05; [EDGEMOVE X-50.00 Y-50.00; N10 G21 G94 G40; N20 G91 G28 Z0.00; N30 G28 X0.00 Y0.00; N40 M06 T0101; N50 M03 S2500; N60 G90 G00 X0.00 Y0.00 Z5.00; N70 G00 Z0.00; N80 M98 P0025001; N90 M70; N100 M98 P0025001; N110 M80; N120 M71; N130 M98 P0025001; N140 M81; N150 M70; N160 M71; N170 M98 P0025001; N180 M80; N190 M81; N200 G90 G00 Z5.00; N210 G91 G28 X0.00 Y0.00; N220 M05; N230 M30;	**SUB PROGRAM** O5001 N10 G90 G00 X15 Y15; N20 G01 Z-1 F25; N30 X35 Y15; N40 X15 Y35 ; N50 X15 Y15; N60 G0 Z1; N70 G00 X0 Y0; N80 M99; **The statement M98 P0029000 can be explained as follows:** **M 98** – For subprogram call. **P0025001** – 002 is the number of times the subprogram 5001 has to be called. (i.e. the subprogram5001has to be executed for 2 times.) **M70 – X MORROR ON** **M71 – Y MORROR ON** **M80 – X MORROR OFF** **M81 - YMORROR OFF**

INFERENCE:

Practiced with G codes and M codes for Contour Milling using Mirroring and Sub Routine Cycles **(M70, M71, M80, M81 and M98, M99)** through this exercise.

RESULT:

Thus, the part program for Contour Milling using Mirroring and Sub Routine Cycle is developed and simulated in the software.



Peck Drilling : Billet size: 90 x 90 x 10 mm;

Drilling Tool : Ø 6mm for 6mm depth & Ø 10mm for 2mm depth

Ex. No : 2 Date :

PECK DRILLING

AIM :

To Write a CNC program for the profile given using Peck Drilling syntax.

SOFTWARE REQUIRED:

1. Windows XP or Windows 7 operating system
2. Denford cam software

Peck Drilling Cycle Syntax:

O6600 [BILLET X90.00 Y90.00 Z10.00; [TOOLDEF T01 D06; [TOOLDEF T02 D10; [EDGEMOVE X0.0 Y0.0; N10 G21 G94 G40; N20 G91 G28 Z0.00; N30 G28 X0.00 Y0.00; N40 M06 T0101; N50 M03 S2500; N60 G90 G00 X0.00 Y0.00; N70 Z0.00; N80 G91 G73 X10 Y10 Z-6 Q1.5 R0.0 K5 F50; N90 G91 G28 X0.00 Y0.00 Z0.00; N100 M01; N110 M06 T0202; N120 M03 S2500; N130 G90 G00 X0.00 Y0.00; N140 Z0.00; N150 G91 G73 X10 Y10 Z-2 Q0.5 R0.0 K5 F50; N160 G91 G28 X0.00 Y0.00 Z0.00; N270 M05; N280 M30;	***Explanation*** ***G73- Peck drilling*** X10, Y10 denotes the incremental distance from its present position (Present position is defined in N10) Z-6 denotes the depth of the drill Q1.5 denotes the depth of cut for each peck K5 denotes that drilling has to be done five times at the given increment X & Y distances R - denotes the position of tool to start in Z direction Initially the tool is positioned at X0 and Y0 in block N60. From this position the tool takes a move of 10mm in X direction, which is point 1 and starts drilling. Then it moves to point 2 and starts drilling. So on, five drills at points 1,2,3,4 & 5 are drilled.

INFERENCE:

Practiced with G codes and M codes for Peck Drilling Cycle **(G73)** through this exercise.

RESULT:

Thus, the part program for Peck Drilling Cycle is developed and simulated in the software.

PROGRAM OUTCOMES (POs)

Mechanical Engineering Graduates will be able to

1	**Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals and an engineering specialization to solution of complex engineering problems.
2	**Problem analysis:** Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
3	**Design / development of solutions:** Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
4	**Conduct investigations of complex problems:** Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
5	**Modern tool usage:** Create, select and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
6	**The engineer and society:** Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
7	**Environment and sustainability:** Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
8	**Ethics:** Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
9	**Individual and team work:** Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
10	**Communication:** Communicate effectively on complex engineering activities withthe engineering community andwith society atlarge, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
11	**Project management and finance:** Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects in multidisciplinary environments.
12	**Life-long learning:** Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

K.L.N. COLLEGE OF ENGINEERING

VISION

To become a Centre of Excellence in Technical Education and Research in producing Competent and Ethical professionals to the Society.

MISSION

To impart Value and Need based curriculum to the students with enriched skill development in the field of Engineering, Technology, Management and Entrepreneurship and to nurture their character with social concern and to pursue their career in the areas of Research and Industry.

Principal **Secretary &Correspondent** **President**