DEPARTMENT OF MECHANICAL ENGINEERING

SEVENTH SEMESTER

LAB : MEP-72 COMPUTER AIDED ENGINEERING LAB

PREPARED BY

Mr. V. KARTHIKEYAN, M.Tech,
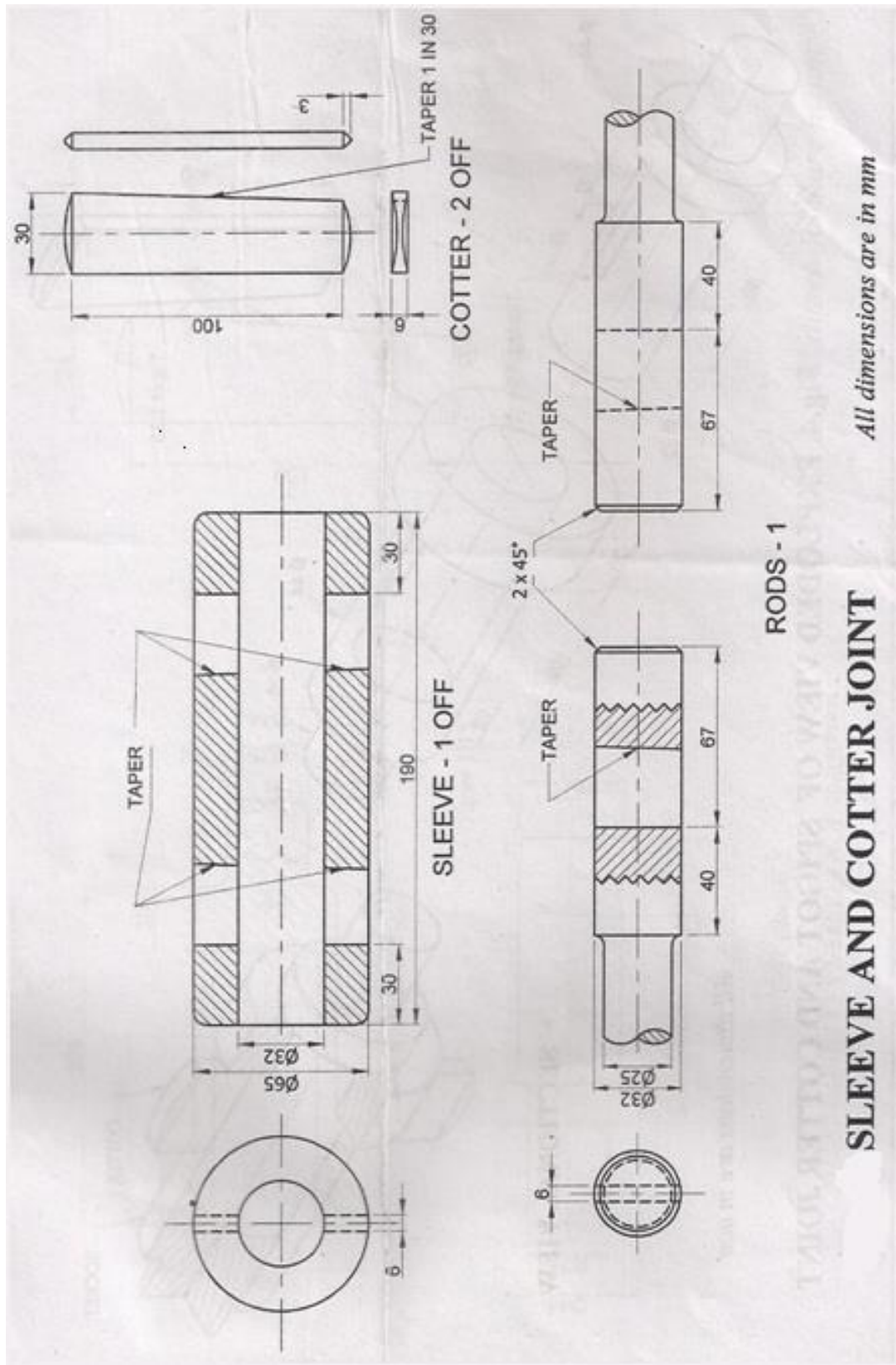**ASSISTANT PROFESSOR,**
Department of Mechanical Engineering

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DETAILED DRAWING:

SLEEVE AND COTTER JOINT

All dimensions are in mm

SLEEVE AND COTTER JOINT

AIM:

To model the given components of Sleeve And Cotter Joint and assemble the parts by using the Solidworks software and obtain its respective views.

COMMANDS USED:

Sketch, Extrusion, Revolve, Pattern, Mate, Align, Helical Sweep, Round, Chamfer etc.

DISCRIPTION:

A Sleeve and Cotter Joint consists of one sleeve, two cotters and two rods which are to be joined together. Cotters are flat wedge shaped pieces of steels of rectangular cross section. Cotters are uniform in thickness but tapering in width, generally on one side only. It is inserted at right angles to the axis of the rod to be connected. It is used to connect rigidly two rods subjected to tensile or compressive fore. The diameter of the rod ends are enlarged to compensate the weakening effect caused by the cotter joint. A clearance is allowed to provide to allow the tightening adjustment. It is generally up to 3mm when the cotter is in fixed position.

SOFTWARE USED:

- Operating system: Windows XP
- Solidworks 2016 edition

SYSTEM HARDWARE USED:

- Pentium IV processor
- 256MB RAM
- 40 GB Hard disk drive
- Keyboard and Mouse
- 15" Monitor and a Plotter

PROCEDURE:

- The total number of parts in the diagram are studied and they are as follows
- Sleeve and cotter joint consists of one sleeve, two cotters and two rods which are to be joined together.
- Open the Pro/Engineer 5.0 software and set the destination folder.
- Open the new part file in Pro/Engineer 5.0 and make sure that select the solid type and give the name of the part in the name box and click ok.
- By using operations such as extrude, revolve, sketching, mirror, and pattern, fillet, chamfer and hole to finish the part diagram and save it in the destination folder.
- Click new (ctrl+n) and select the assembly and make sure design is selected in the right side and give the assembly name as sleeve and cotter joint and click ok.
- In this add components and browse the part file in the destination folder to insert the part file and select the default location.
- Further add component as like above and use align, mate and offset reference to complete the assembly of sleeve and cotter joint.
- Check that all parts are assembled in fully constrained.
- Then new and select drawing for detailing sleeve and cotter joint.
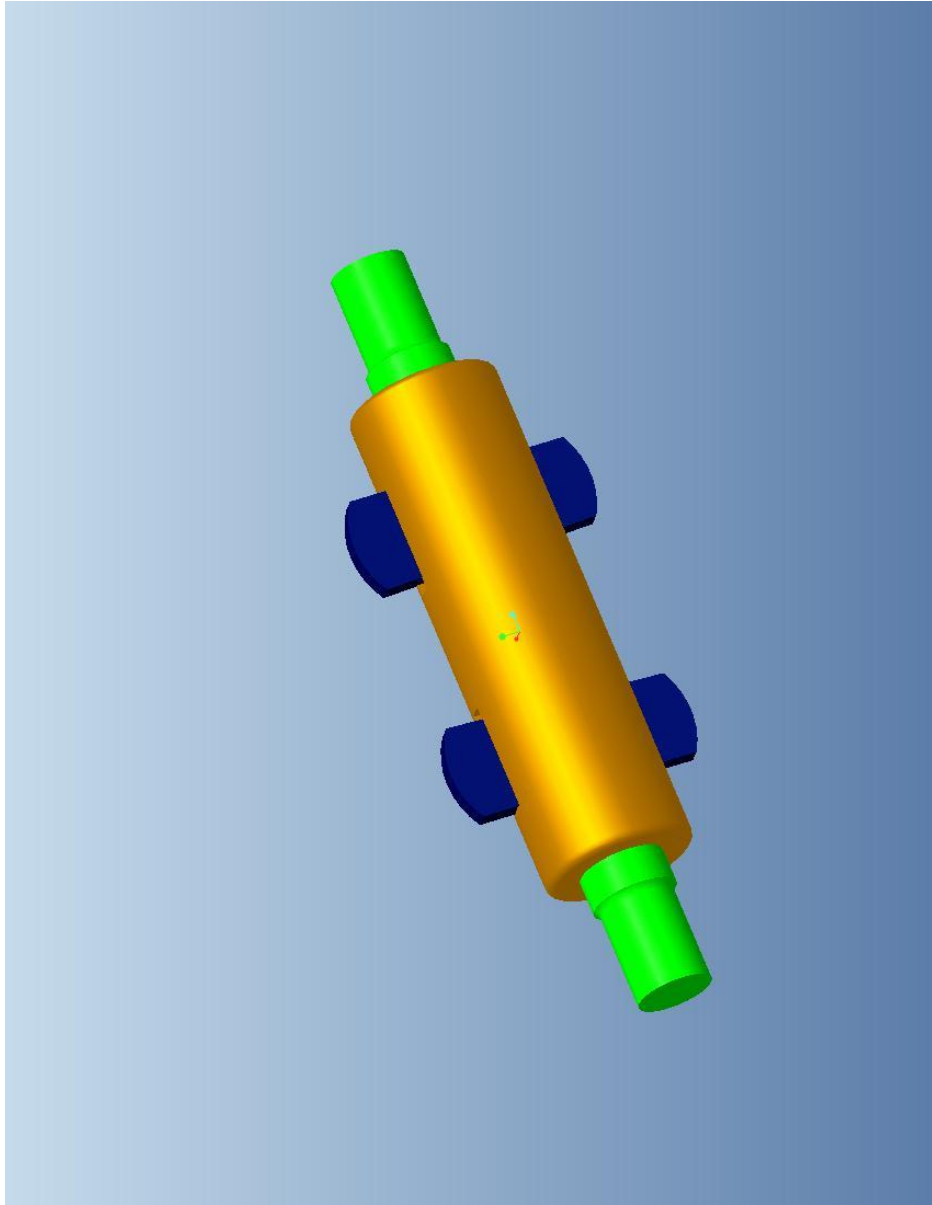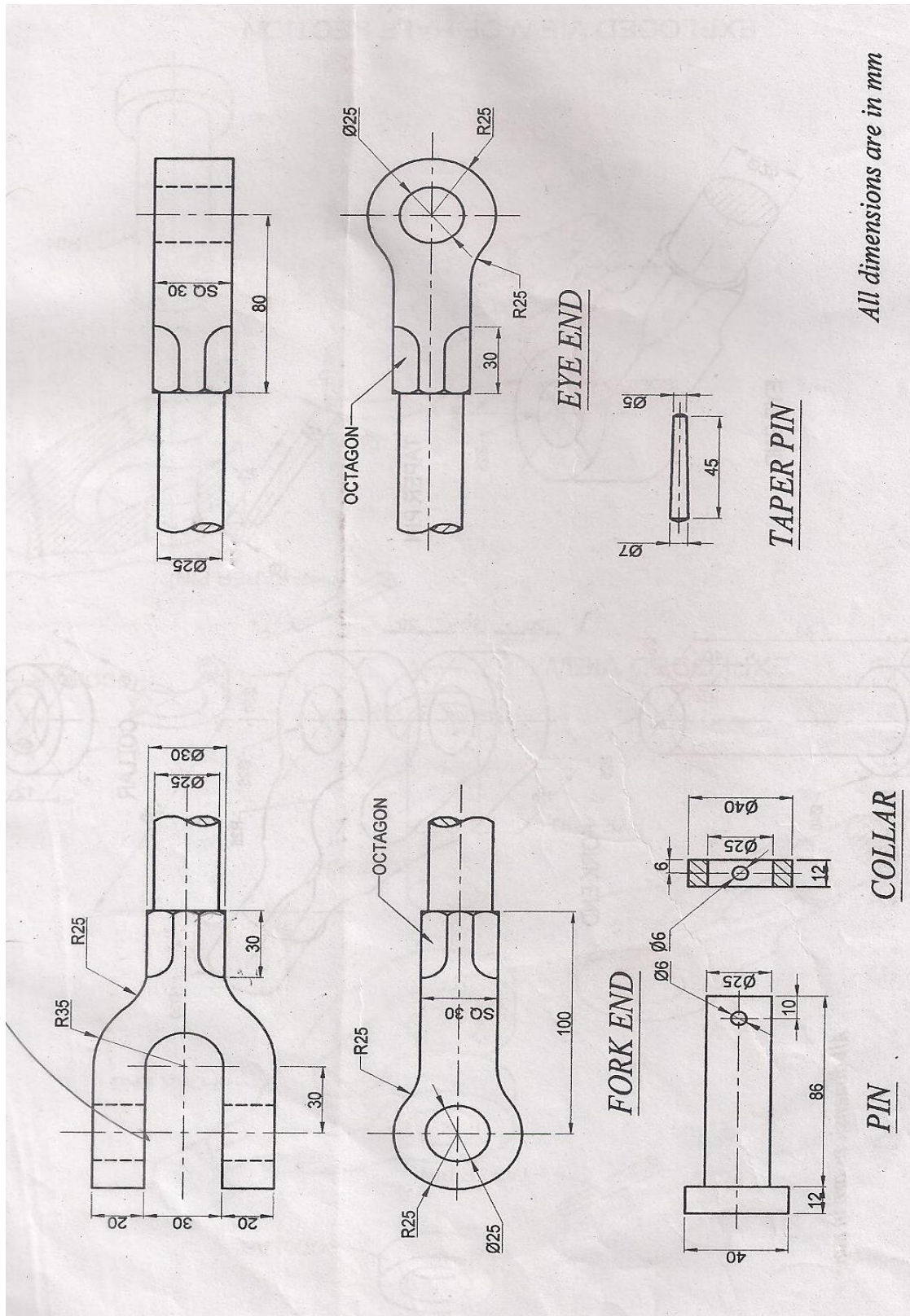
RESULT:

The components of sleeve and cotter joint are modelled and assembled using Solidworks software.

DETAILED DRAWING:

All dimensions are in mm

EYE END

TAPER PIN

FORK END

COLLAR

PIN

SQ 30

OCTAGON

R25

R35

OCTAGON

KNUCKLE JOINT

AIM:

      To model the givencomponents of knuckle joint and assemble the parts by using the Solidworks software and obtain its respective views.

COMMANDS USED:

      Sketch, Extrusion, Revolve, Pattern, Mate, Align, Helical Sweep, Round, Chamfer etc,

DISCRIPTION:

The rod ends connected by Knuckle pin is called Knuckle Joint. Knuckle joints are used when the two rods are required to have a small angular movement in one plane and can be used for rotary and transverse motions.

One end of the rod ends is forged into the shape of a 'fork' (double eye) and other in the shape of an eye. The eye end is inserted between the jaws, of the fork end. After alignment of holes, a Knuckle pin is passed into them. The knuckle pin has a head at one end and collar at the other end. The pin secured by a tape pin.

SOFTWARE USED:

- Operating system: Windows XP
- Solidworks 2016 edition

SYSTEM HARDWARE USED:

- Pentium IV processor
- 256MB RAM
- 40GB Hard disk drive
- Keyboard and Mouse
- 15" Monitor and a Plotter

PROCEDURE:

- The total number of parts in the diagram are studied and they are as follows
- Knuckle joint consists of eye end, fork end, pin, collar and taper pin which are to be joined together.
- Open the Pro/Engineer 5.0 software and set the destination folder.
- Open the new part file in Pro/Engineer 5.0 and make sure that select the solid type and give the name of the part in the name box and click ok.
- By using operations such as extrude, revolve, sketching, mirror, and pattern, fillet, chamfer and hole to finish the part diagram and save it in the destination folder.
- Click new (ctrl+n) and select the assembly and make sure design is selected in the right side and give the assembly name as knucklejoint and click ok.
- In this add components and browse the part file in the destination folder to insert the part file and select the default location.
- Further add component as like above and use align, mate and offset reference to complete the assembly of Knuckle joint.
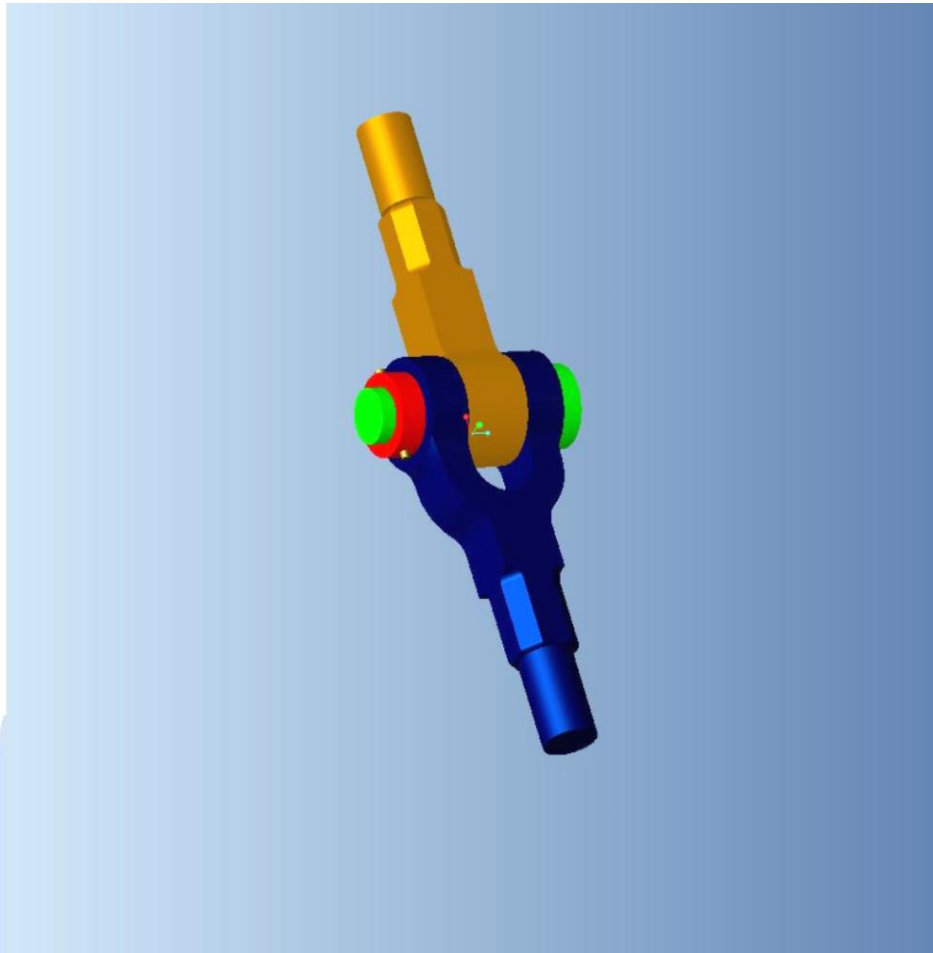
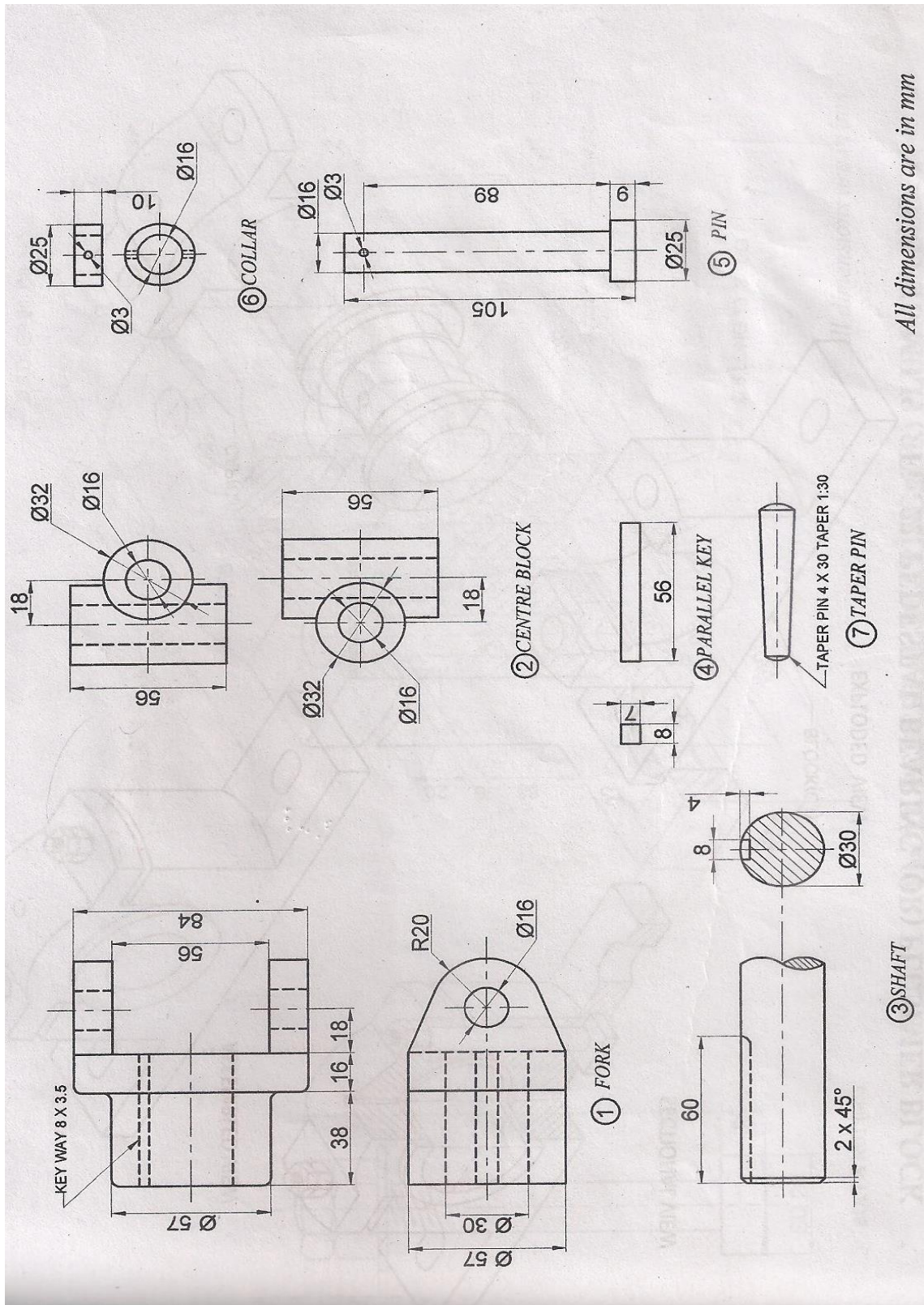**Assembled View of Sleeve and Cotter Joint**

- Check that all parts are assembled in fully constrained.

- Then new and select drawing for detailing Knuckle Joint.

RESULT:

The components of Knuckle Joint are modelled and assembled using Pro-E software

DETAILED DRAWING:

All dimensions are in mm

⑥ COLLAR

Ø25
Ø3
Ø16
10

⑤ PIN

Ø16
Ø3
68
6
Ø25
105

② CENTRE BLOCK

Ø32
Ø16
18
56
55
55
Ø32
Ø16
18

④ PARALLEL KEY

56
7
8

⑦ TAPER PIN

TAPER PIN 4 X 30 TAPER 1:30

③ SHAFT

4
8
Ø30
60
2 x 45°

① FORK

R20
Ø16
84
56
18
16
38
KEY WAY 8 X 3.5
Ø 57
Ø 30
Ø 57

UNIVERSAL COUPLING

AIM:

To model the givencomponents of Universal Coupling and assemble the parts by using the Solidworks software and obtain its respective views.

COMMANDS USED:

Sketch, Extrusion, Revolve, Pattern, Mate, Align, Helical Sweep, Round, Chamfer etc.

DISCRIPTION:

Universal Coupling is used to couple together two shafts whose axes intersect. The angle between the two shafts is usually less than 30˚. The angle between two shafts may be varied even they are in motion.

It consists of two forks, one central blocks, two cylinder pins with circular collars and two locking or taper pins. The forks are secured on the ends of the two shafts by means of keys. The central block is placed between these forks and pins are passed through the forks are central block holes.

SOFTWARE USED:

- Operating system: Windows XP

- Solidworks 2016 edition

SYSTEM HARDWARE USED:

- Pentium IV processor

- 256MB RAM

- 40GB Hard disk drive

- Keyboard and Mouse

- 15" Monitor and a Plotter

PROCEDURE:

- The total number of parts in the diagram are studied and they are as follows

- Universal Coupling consists of center-block, fork, collar, pin and taper pin which are to be joined together.

- Open the Pro/Engineer 5.0 software and set the destination folder.

- Open the new part file in Pro/Engineer 5.0 and make sure that select the solid type and give the name of the part in the name box and click ok.

- By using operations such as extrude, revolve, sketching, mirror, and pattern, fillet, chamfer and hole to finish the part diagram and save it in the destination folder.

- Click new (ctrl+n) and select the assembly and make sure design is selected in the right side and give the assembly name as Universal Coupling and click ok.

- In this add components and browse the part file in the destination folder to insert the part file and select the default location.

\
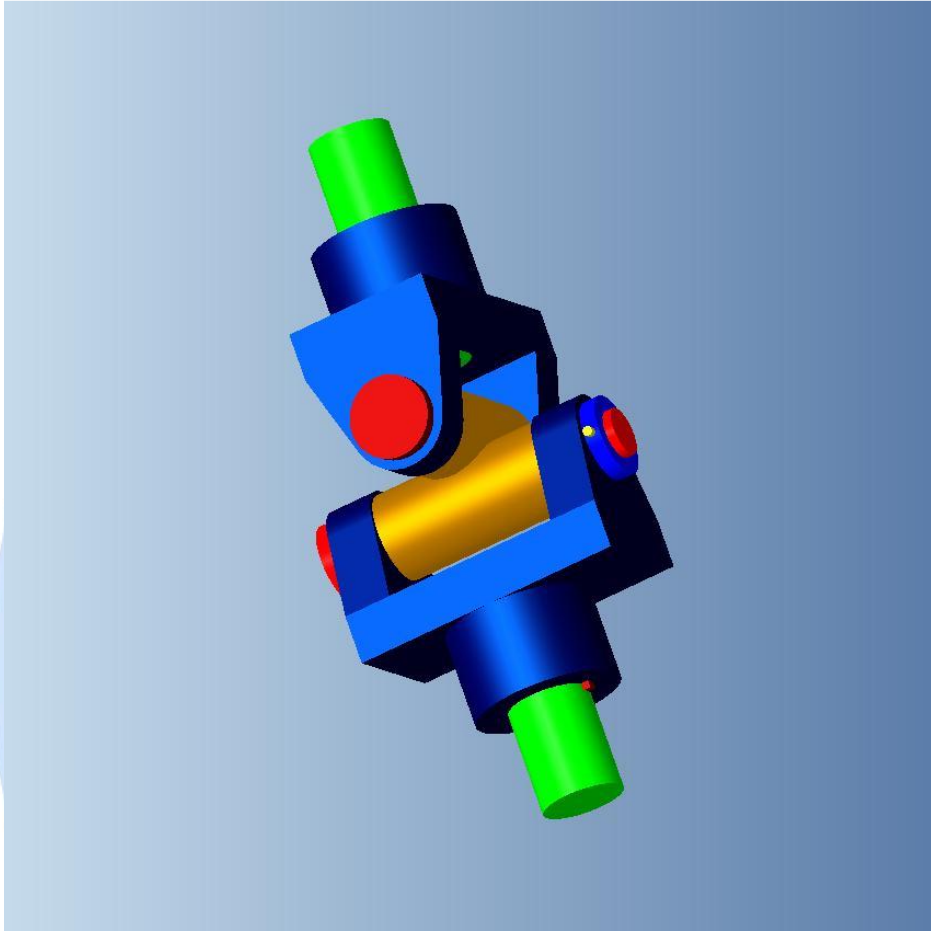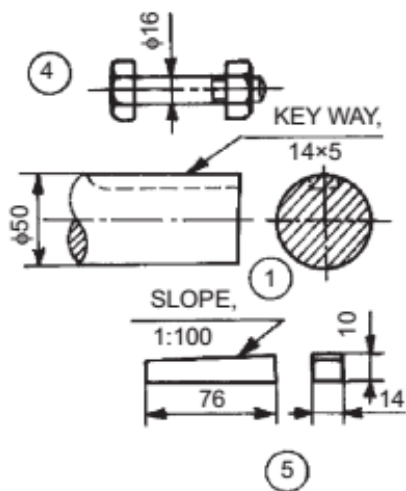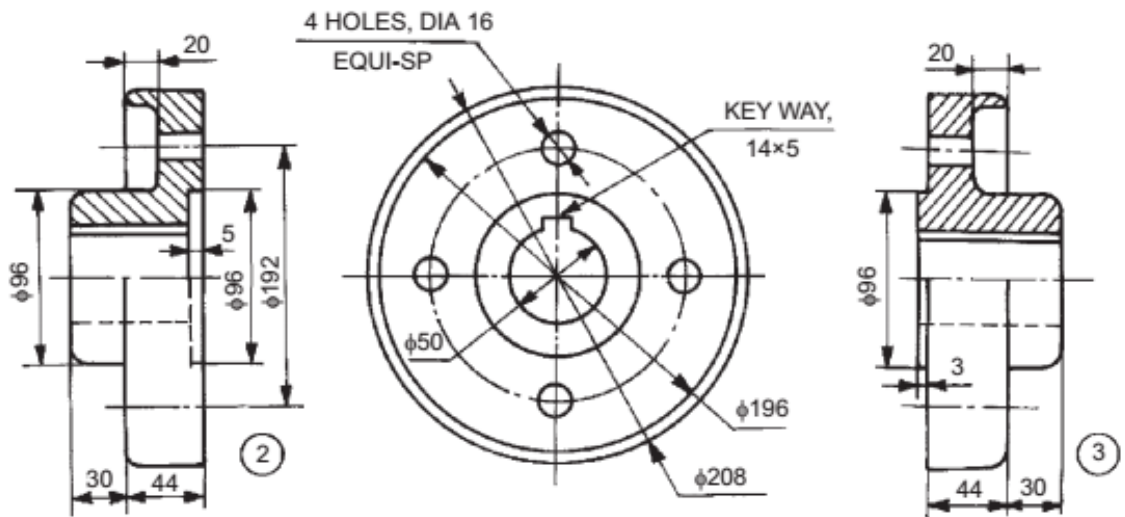

ASSEMBLED VIEW OF UNIVERSAL COUPLING

- Further add component as like above and use align, mate and offset reference to complete the assembly of Universal Coupling.
- Check that all parts are assembled in fully constrained.
- Then new and select drawing for detailing Universal Coupling.

RESULT:

The components of Universal Coupling are modeled and assembled using Solidworks software

DETAILED DRAWING:

4 HOLES, DIA 16
EQUI-SP

KEY WAY, 14×5

φ96   φ96   φ192

φ50   φ196   φ208

20   30   44   ②

20   φ96   3   44   30   ③

φ16   ④

KEY WAY, 14×5

φ50   ①

SLOPE, 1:100   10   76   14

⑤

Parts list

Sl. No.	Name	Matl.	Qty.
1	Shaft	MS	2
2	Flange	CI	1
3	Flange	CI	1
4	Bolt with nut	MS	4
5	Key	MS	2

PROTECTED FLANGED COUPLING

AIM:

      To model the given components of protected flanged coupling and assemble the parts by using the Solidworks software and obtain its respective views.

COMMANDS USED:

      Sketch, Extrusion, Revolve, Pattern, Mate, Align, Helical Sweep, Round, Chamfer etc,

DISCRIPTION:

Couplings are used for connecting shafts of two machines to function as a single unit. The protected type flange coupling is a standard form of coupling used for transmitting the rotating power from one shaft to another shaft. It consists of two flanges with rectangular keys and bolt with nuts. In this each flange is provided with a assembler projection. It shelters the bolt heads of the nuts. Hence it prevents them from catching clothes of workmen. To keys the axial alignment as concentric, by the circular projection on one flange and a corresponding recess is provide. The male extension fits into the recess leaving some clearance in between them for further adjustment.

SOFTWARE USED:

- Operating system: Windows XP
- Solidworks 2016 edition

SYSTEM HARDWARE USED:

- Pentium IV processor
- 256MB RAM
- 40GB Hard disk drive
- Keyboard and Mouse
- 15" Monitor and a Plotter

PROCEDURE:

- The total number of parts in the diagram are studied and they are as follows
- Protected Flange Coupling consists of male flange, female flange, bolt, nut and taper key which are to be joined together.
- Open the Pro/Engineer 5.0 software and set the destination folder.
- Open the new part file in Pro/Engineer 5.0 and make sure that select the solid type and give the name of the part in the name box and click ok.
- By using operations such as extrude, revolve, sketching, mirror, and pattern, fillet, chamfer and hole to finish the part diagram and save it in the destination folder.
- Click new (ctrl+n) and select the assembly and make sure design is selected in the right side and give the assembly name as protectedflangecoupling and click ok.
- In this add components and browse the part file in the destination folder to insert the part file and select the default location.
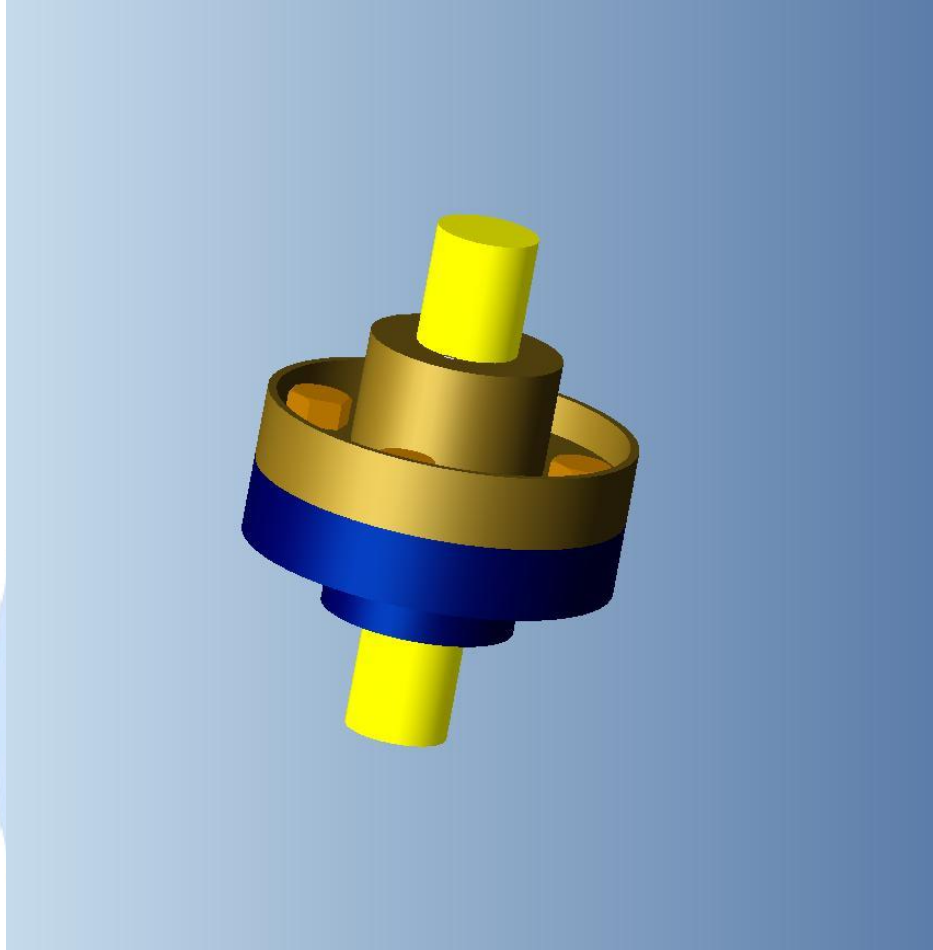
ASSEMBLED VIEW OF PROTECTED FLANGE COUPLING

- Further add component as like above and use align, mate and offset reference to complete the assembly of protected flange coupling.
- Check that all parts are assembled in fully constrained.
- Then new and select drawing for detailing Protected Flange Coupling.
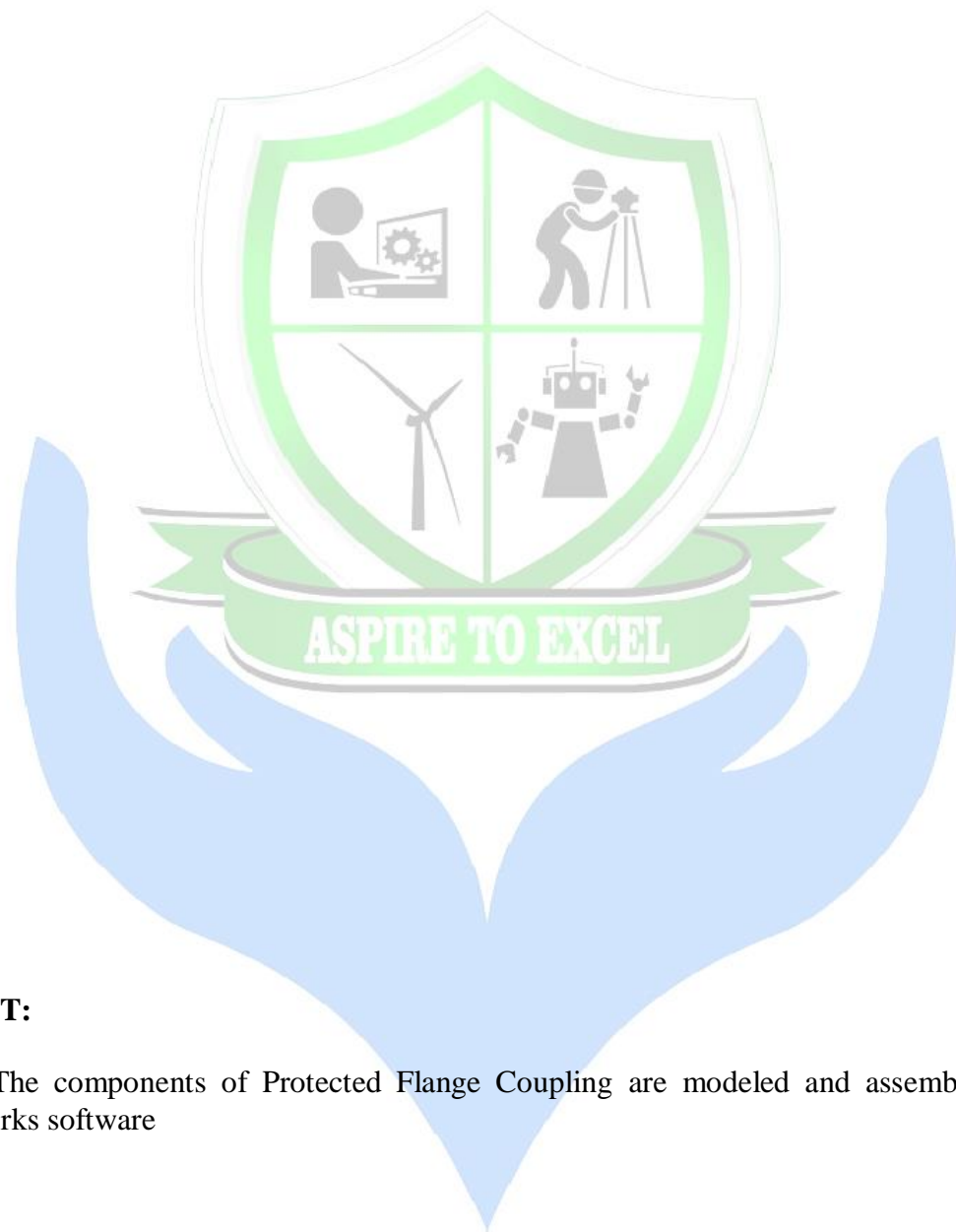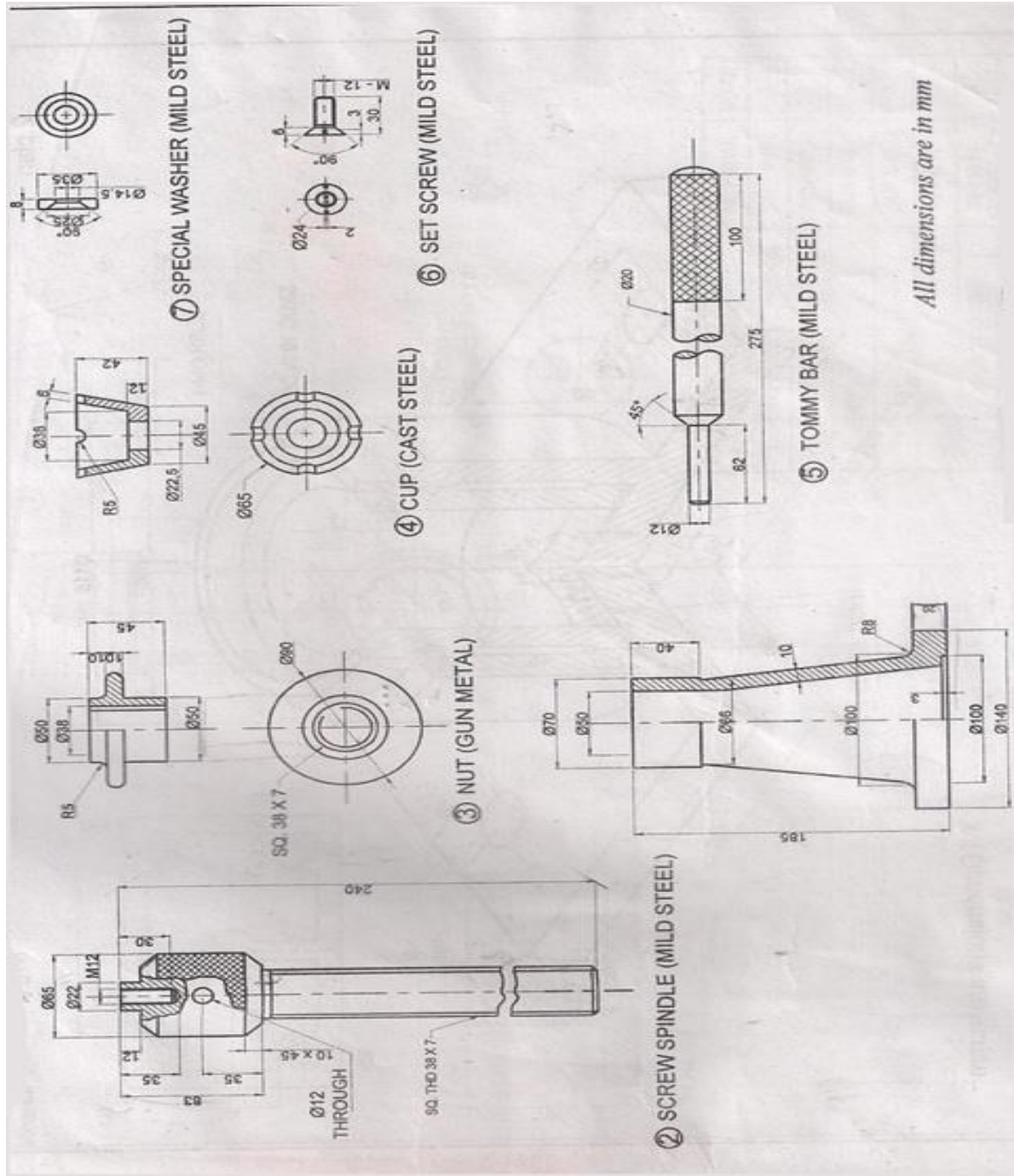
RESULT:

The components of Protected Flange Coupling are modeled and assembled using Solidworks software

DETAILED DRAWING:

① SPECIAL WASHER (MILD STEEL)

② SCREW SPINDLE (MILD STEEL)

③ NUT (GUN METAL)

④ CUP (CAST STEEL)

⑤ TOMMY BAR (MILD STEEL)

⑥ SET SCREW (MILD STEEL)

All dimensions are in mm

SCREW JACK

AIM:

      To model the given components of Screw Jack coupling and assemble the parts by using the Solidworks software and obtain its respective views.

COMMANDS USED:

      Sketch, Extrusion, Revolve, Pattern, Mate, Align, Helical Sweep, Round, Chamfer etc,

DISCRIPTION:

Screw jacks are used for raising heavy bodies through very small heights. In it, the screw works in a nut which is pressed fitted into the main body. A tommy bar is inserted into a hole through the enlarged head of a screw and when this is turned, the screw will move up or down, thereby raising or lowering the load.

A cup is placed at the top of the screw to carry the load conveniently. This is serrated to provide a better grip to the load. The cup, on assembly, is free to rotate independently of the screw.

SOFTWARE USED:

- Operating system: Windows XP

- Solidworks 2016 edition

SYSTEM HARDWARE USED:

- Pentium IV processor

- 256MB RAM

- 40GB Hard disk drive

- Keyboard and Mouse

- 15" Monitor and a Plotter

PROCEDURE:

- The total number of parts in the diagram are studied and they are as follows
- Screw jack consists of casting, nut, screw, cup, washer, setscrew and tommy bar which are to be joined together.
- Open the Pro/Engineer 5.0 software and set the destination folder.
- Open the new part file in Pro/Engineer 5.0 and make sure that select the solid type and give the name of the part in the name box and click ok.
- By using operations such as extrude, revolve, sketching, mirror, and pattern, fillet, chamfer and hole to finish the part diagram and save it in the destination folder.
- Click new (ctrl+n) and select the assembly and make sure design is selected in the right side and give the assembly name as screwjack and click ok.
- In this add components and browse the part file in the destination folder to insert the part file and select the default location.
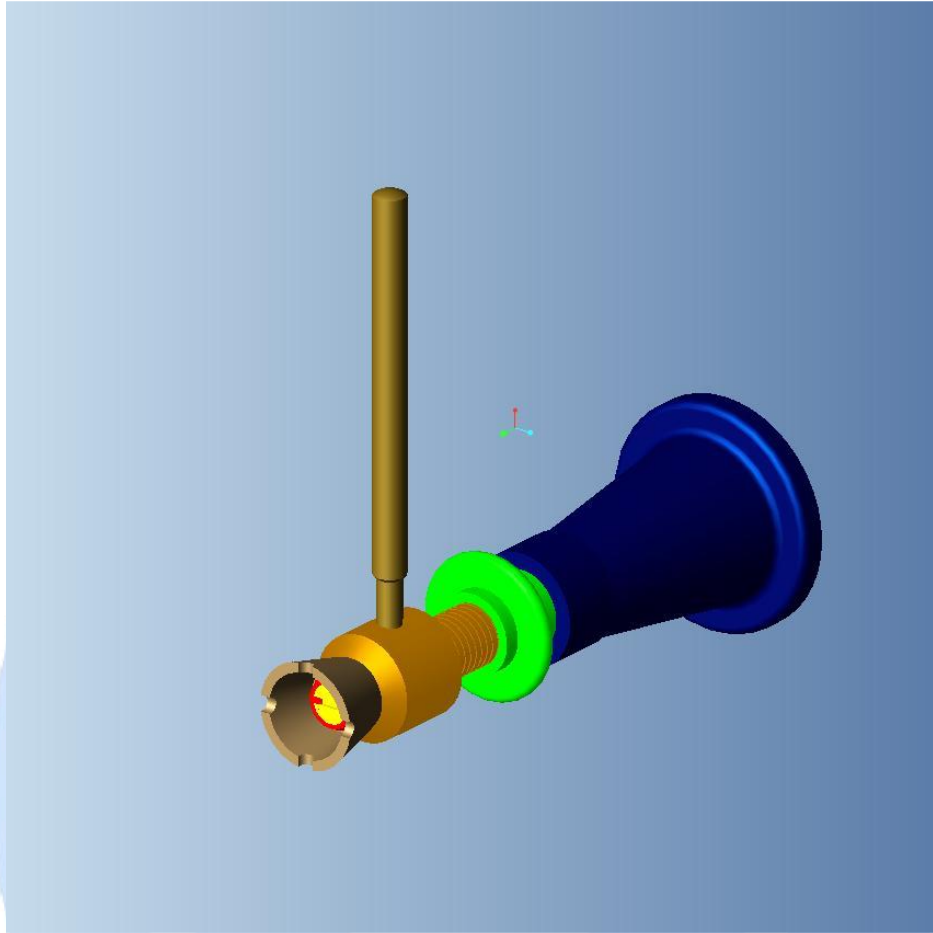
**ASSEMBLED VIEW OF SCREW JACK**

- Further add component as like above and use align, mate and offset reference to complete the assembly of screw jack.
- Check that all parts are assembled in fully constrained.
- Then new and select drawing for detailing Screw Jack.

RESULT:

The components of Screw Jackare modelled and assembled using Solidworks software

DETAILED DRAWING:

All dimensions are in mm
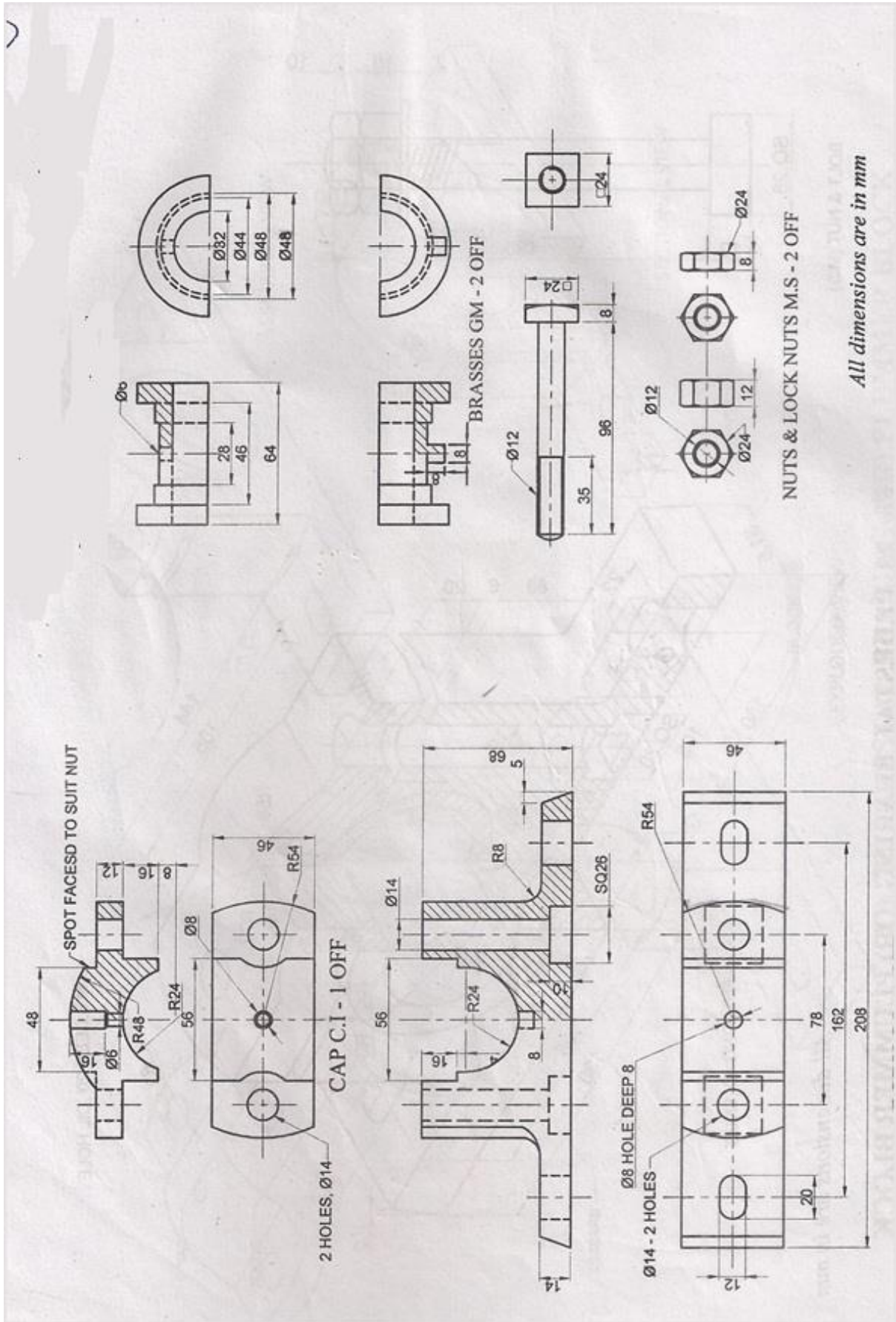
PLUMMER BLOCK

AIM:

To model the givencomponents of plummer block coupling and assemble the parts by using the Pro-E software and obtain its respective views.

COMMANDS USED:

Sketch, Extrusion, Revolve, Pattern, Mate, Align, Helical Sweep, Round, Chamfer etc,

DISCRIPTION:

A Plummer Block (Pedestal Bearing) consists of an iron block with a sole gun metal brasses or steps made in two halves, a cast iron cap and two mild steel bolts. It is a split of journal bearing to support horizontal shafts. It is a split type of journal bearing to support horizontal shafts. It is used for supporting the running shafts at high speed. The brasses are provided in between the block and cup and fastened by using two square headed bolts and lock nuts. The block has hole. The bottom brass has a sung. It seats on the sung hold provided in the block. This arrangement is to prevent the rotation of the bush, collars provided at the both end of the brasses to prevent axial movement of the brasses. It is used to support the shaft which rotated at high is the application of the Plummer Block.

SOFTWARE USED:
- Operating system: Windows XP
- Solidworks 2016 edition

SYSTEM HARDWARE USED:
- Pentium IV processor
- 256MB RAM
- 40GB Hard disk drive
- Keyboard and Mouse
- 15" Monitor and a Plotter

PROCEDURE:
- The total number of parts in the diagram are studied and they are as follows
- Plummer Block consists of body, cap, brasses, nuts and lock nuts which are to be joined together.
- Open the Pro/Engineer 5.0 software and set the destination folder.
- Open the new part file in Pro/Engineer 5.0 and make sure that select the solid type and give the name of the part in the name box and click ok.
- By using operations such as extrude, revolve, sketching, mirror, and pattern, fillet, chamfer and hole to finish the part diagram and save it in the destination folder.
- Click new (ctrl+n) and select the assembly and make sure design is selected in the right side and give the assembly name as plummerblock and click ok.
- In this add components and browse the part file in the destination folder to insert the part file and select the default location.
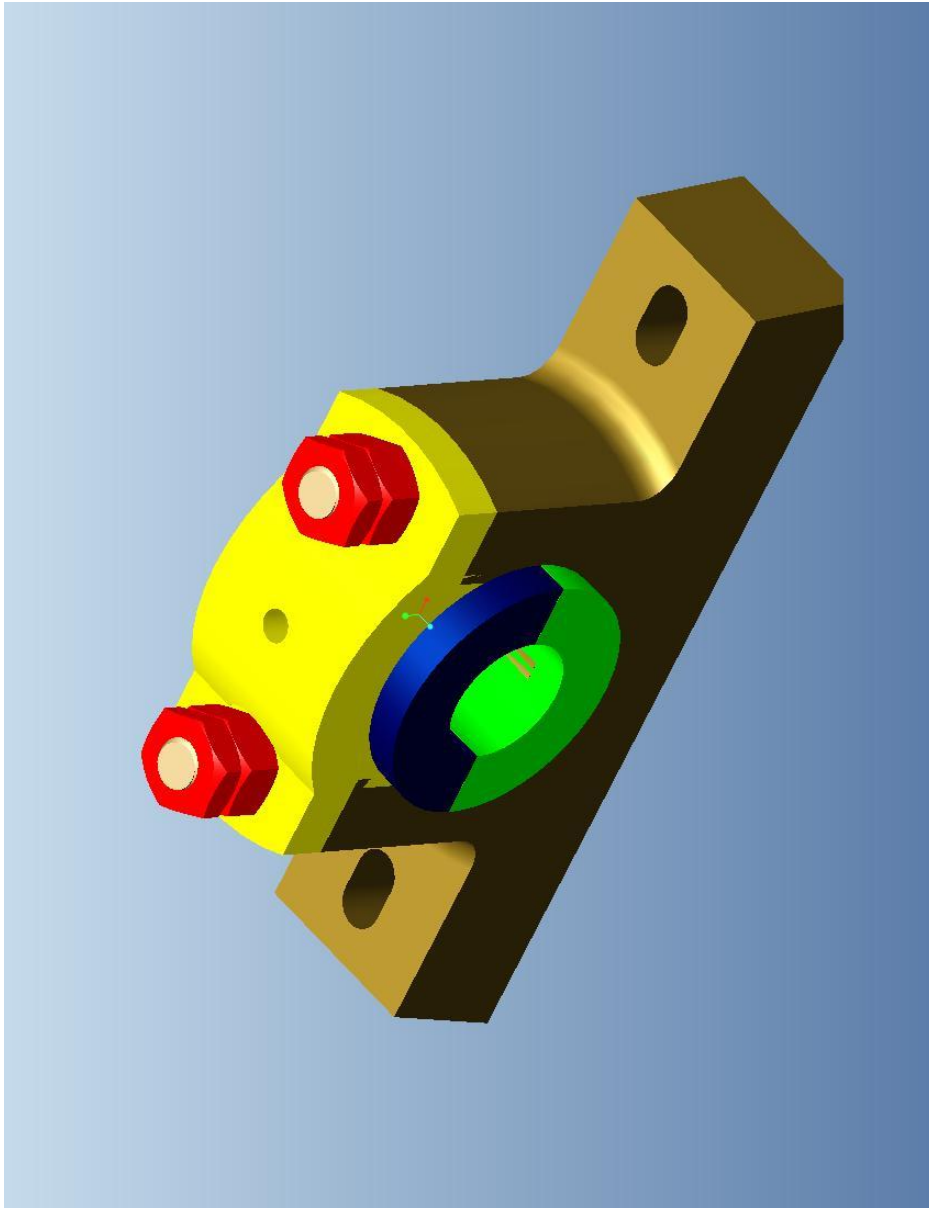
ASSEMBLED VIEW OF PLUMMER BLOCK
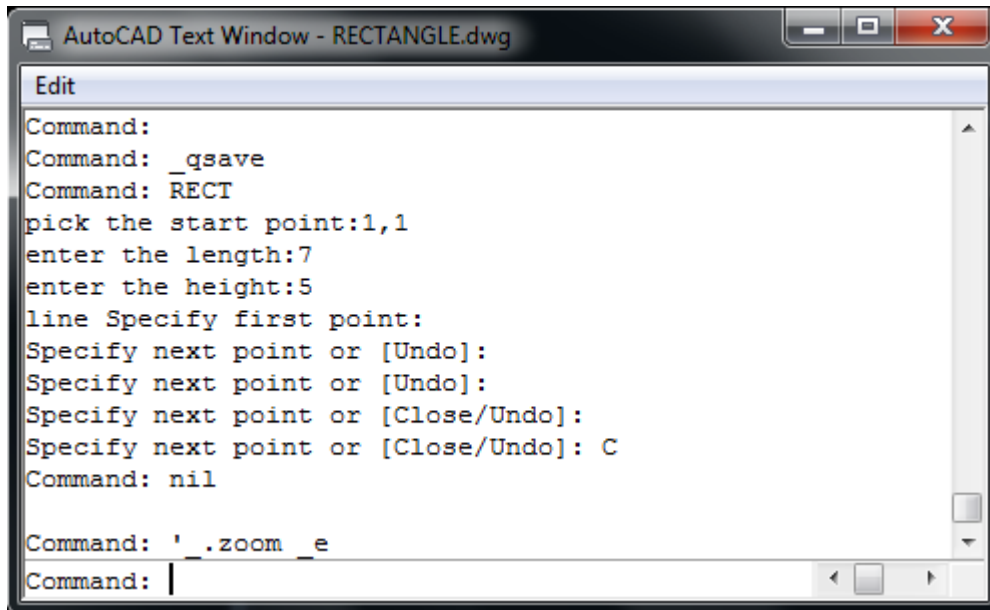
- Further add component as like above and use align, mate and offset reference to complete the assembly of Plummer Block.
- Check that all parts are assembled in fully constrained.
- Then new and select drawing for detailing Plummer Block.

RESULT:

      The components of Plummer block are modelled and assembled using Solidworks software

OUTPUT:

RECTANGLE USING AUTO LISP

RECTANGLE USING AUTO LISP

Aim

To draw rectangle with its length and height using Auto LISP

SOFTWARE USED:
- Operating system: Windows 7
- Auto CAD 2015

SYSTEM HARDWARE USED:
- Pentium IV processor
- 256MB RAM
- 40GB Hard disk drive
- Keyboard and Mouse
- 15" Monitor and a Plotter

PROGRAM:

```
(defun c:rect()
 (setq p1(getpoint "pick the start point:")
      L(getdist "enter the length:")
      h(getdist "enter the height:")
      p2(polar p1 (dtr 0)L)
      p3(polar p2 (dtr 90)h)
      p4(polar p3 (dtr 180)L)
      )
 (command "line" p1 p2 p3 p4 "C")
 )
(defun dtr(ang)
 (* (/ pi 180) ang)
 )
```

INPUT:

Pick the start point: 1, 1
Enter the length: 7
Enter the height: 5

RESULT:

Thus rectangle with its length and height is drawn using Auto LISP

OUTPUT:


```
AutoCAD Text Window - shaft1.dwg

Edit

Command:
Command:
Command: _qsave
Command: shaft

 enter the shear stress of a shaft:40

 enter the torque :20000

 specify the center of shaft: 10,10
circle Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
Specify radius of circle or [Diameter] <67.9435>: d Specify diameter of circle
<135.8870>: 13.62002980837264
Command: zoom
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: extents
Command: SHAFT Unknown command "SHAFT".  Press F1 for help.

Command: extrude
Current wire frame density:  ISOLINES=8
Select objects to extrude: nil

Select objects to extrude: 1 found

Select objects to extrude:

Specify height of extrusion or [Direction/Path/Taper angle] <50.0000>: 50

Command:
```


DESIGN OF SOLID SHAFT USING AUTO LISP

REG NO:	11TB2426

DESIGN OF SOLID SHAFT USING AUTO LISP

Aim

To design a solid shaft using Auto LISP

Problem description

The shearing stress of solid shaft is not to exceed 40 N/m^2 when the torque transmitted is 20000 N-m.

SOFTWARE USED:

- Operating system: Windows XP

- Auto CAD 2010

SYSTEM HARDWARE USED:

- Pentium IV processor

- 256MB RAM

- 40GB Hard disk drive

- Keyboard and Mouse

- 15" Monitor and a Plotter

PROGRAM:

```
(defun c:shaft()
 (setq ss(getreal "\n enter the shear stress of a shaft:")
      Tor(getreal "\n enter the torque :")
      v(/ (* 16 tor) (* pi ss))
      ds(expt v 0.333)
      p1 (getpoint "\n specify the center of shaft: ")
      )
 (command "circle" p1 "d" ds)
 (command "zoom" "extents" "")
 (command "extrude")
 )
```

INPUT:

Enter the shear stress of a shaft: 40
Enter the toque: 20000
Specify the center of shaft: 10, 10

RESULT:

Thus the shaft is designed for the given requirement using Auto LISP

OUTPUT:


```
AutoCAD Text Window - shaft2.dwg

Edit

Command:
Command: shaft1

 enter the power to be transmitted(w):300000

 speed of the shaft(rpm):100

 Max. shear stress:80

 specify center of shaft:0,0
circle Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
Specify radius of circle or [Diameter] <60.7958>: d Specify diameter of circle
<121.5917>: 121.5916995844352
Command: zoom
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: extents
Command: SHAFT1 Unknown command "SHAFT1".  Press F1 for help.

Command: extrude
Current wire frame density:  ISOLINES=8
Select objects to extrude: nil

Select objects to extrude: 1 found

Select objects to extrude:

Specify height of extrusion or [Direction/Path/Taper angle] <150.0000>: 150

Command:
```
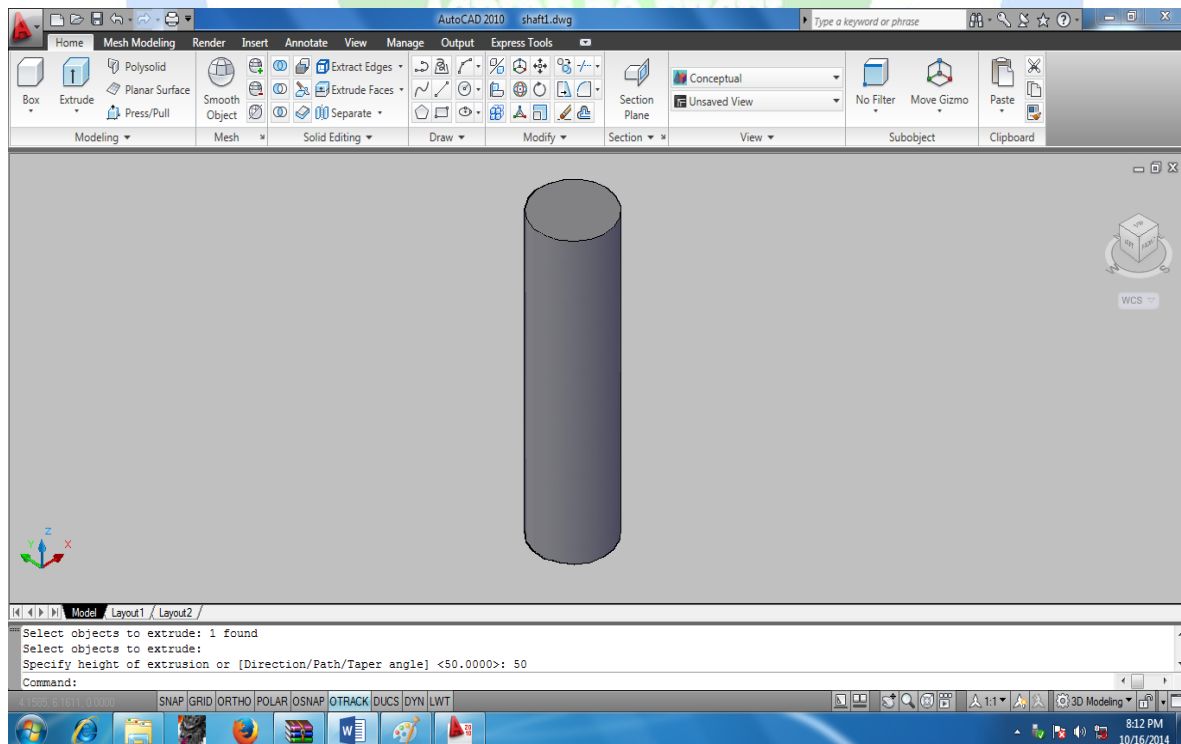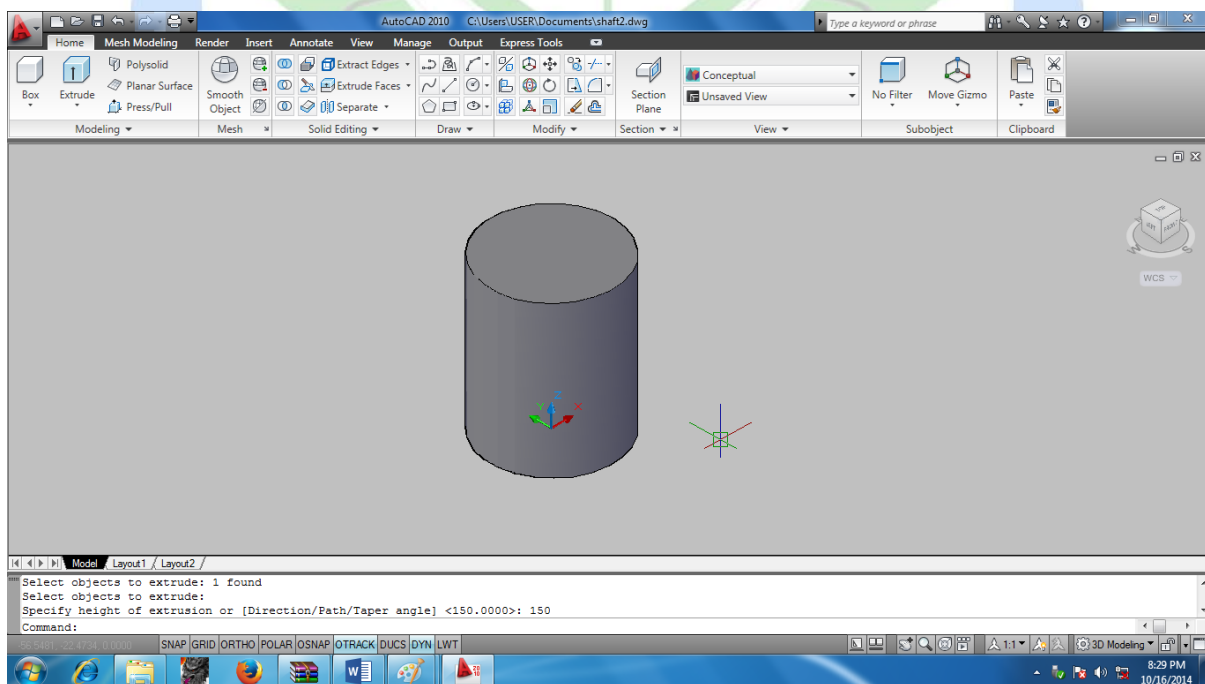

DESIGN OF SHAFT USING AUTO LISP

DESIGN A SOLID SHAFT USING AUTO LISP-II

Aim

    To design a solid shaft using Auto LISP

Problem description

    A solid cylindrical shaft is to transmit 300 kW power at 100 rpm, if the shear stress is not to exceed 80 N/m^2.

SOFTWARE USED:

- Operating system: Windows XP
- Auto CAD 2010

SYSTEM HARDWARE USED:

- Pentium IV processor
- 256MB RAM
- 40GB Hard disk drive
- Keyboard and Mouse
- 15" Monitor and a Plotter

PROGRAM:

```
(defun C:shaft1()
 (setq p(getreal "\n enter the power to be transmitted(w):")
      N(getreal "\n speed of the shaft(rpm):")
      ss(getreal "\n Max. shear stress:")
      p1(getpoint "\n specify center of shaft:")
      tor(/ (* p 60) (* 2 pi N))
      tormm(* tor 1000)
      ds( expt (/ (* 16 tormm) (* pi ss)) 0.333)
      )
 (Command "circle" p1 "d" ds)
 (Command "zoom" "extents" "")
  (Command "extrude")
  )
```

INPUT:

Enter the power to be transmitted (w): 3e5
Speed of the shaft (rpm):100
Max. Shear stress: 80
Specify center of shaft: 0,0

RESULT:

Thus the shaft is designed for the given requirement using Auto LISP.

STRESS ANALYSIS OF BEAM -CANTILEVER BEAM WITH POINT LOAD

AIM: To analysis the deflection and stresses at each nodal points of a cantilever beam with point load at free end using ANSYS software.

PROCEDURE:

Preprocessing:
Defining the Problem

1. Change jobname:

   File -> Change Jobname
   Enter "beam", and click on "OK".

2. Define element types:

   Preprocessor -> Element Type -> Add/Edit/Delete [Apply the BEAM3]

3. Define the real constants for the BEAM3 elements:

   Preprocessor > Real Constants > Add
   [Calculate and Apply suitable value in Area, Izz and Height]

4. Define Material Properties:

   Preprocessor -> Material Properties -> -Constant- Isotropic
   [Apply suitable value in young's modulus=2E06N/sq.m and passion ratio=.27]

5. Create nodes:

   Preprocessor -> -Modeling- Create -> Nodes -> In Active CS
   Preprocessor -> -Modeling- Create -> Nodes -> Fill between Nds.
   Utility Menu -> PlotCtrls -> Numbering. [Minimum 10 nodes]
   Preprocessor -> Create -> Elements ->-Auto Numbered-Thru Nodes

OUTPUT:


```
 PRNSOL  Command

File

   TIME=   1.0000     LOAD CASE=   0

   THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

     NODE       UX           UY           UZ           USUM
        1     0.0000       0.0000       0.0000       0.0000
        2     0.0000      -6.2085       0.0000       6.2085
        3     0.0000      -0.15691E-01  0.0000       0.15691E-01
        4     0.0000      -0.59870E-01  0.0000       0.59870E-01
        5     0.0000      -0.13135      0.0000       0.13135
        6     0.0000      -0.22894      0.0000       0.22894
        7     0.0000      -0.35146      0.0000       0.35146
        8     0.0000      -0.49772      0.0000       0.49772
        9     0.0000      -0.66654      0.0000       0.66654
       10     0.0000      -0.85672      0.0000       0.85672
       11     0.0000      -1.0671       0.0000       1.0671
       12     0.0000      -1.2964       0.0000       1.2964
       13     0.0000      -1.5436       0.0000       1.5436
       14     0.0000      -1.8074       0.0000       1.8074
       15     0.0000      -2.0866       0.0000       2.0866
       16     0.0000      -2.3800       0.0000       2.3800
       17     0.0000      -2.6865       0.0000       2.6865
       18     0.0000      -3.0049       0.0000       3.0049
       19     0.0000      -3.3339       0.0000       3.3339
       20     0.0000      -3.6725       0.0000       3.6725
       21     0.0000      -4.0193       0.0000       4.0193
       22     0.0000      -4.3733       0.0000       4.3733
       23     0.0000      -4.7333       0.0000       4.7333
       24     0.0000      -5.0979       0.0000       5.0979
       25     0.0000      -5.4661       0.0000       5.4661
       26     0.0000      -5.8367       0.0000       5.8367

   MAXIMUM ABSOLUTE VALUES
   NODE       0            2            0            2
   VALUE    0.0000      -6.2085       0.0000       6.2085
```


NODAL SOLUTION FOR STRESS ANALYSIS OF CANTILEVER BEAM

Solution Phase: Assigning Loads and Solving

6. Apply constraints and forces on the model:

To apply constraints:

      Solution -> -Loads- Apply -> -Structural- Displacement -> On Nodes
      [We will see the diagram and then apply suitable nodes]
      Solution -> -Loads- Apply -> -Structural- Force/Moment -> On Nodes
      [We will see the diagram and then apply suitable nodes]

7. Solve the problem:

      Solution -> -Solve- Current LS
      Post processing: Viewing the Results

8. Plot the deformed shape:

      General Postproc -> Plot Results -> Deformed Shape

9. List reaction forces:

      General Postproc -> List Results -> Reaction Solution

10. List nodal displacements:

      (a) General Postproc -> List Results -> Nodal Solution -> DOF Solution ->
           ALL DOFs

11. Define element table items for subsequent plotting and listing of various stress results.

12. List element table results. :
    1. General Postproc -> List Results -> Elem Table Data
    2. General Postproc -> Plot Results -> Line Elem Res
    3. General Postproc -> Plot Results -> Elem Table

13. Exit ANSYS. Toolbar: Quit ->Save Everything -> OK

RESULT:
      Thus the analysis the deflection and stresses at each nodal points of a cantilever beam
with point load at free end using ANSYS software.

STRESS ANALYSIS OF BEAM-SIMPLY SUPPORT BEAM
## WITH DISTRIBUTED LOADS

AIM:

To analysis the stress and deflection in a Distributed load of 1000 N/m (1 N/mm) will be applied to a solid steel beam with a rectangular cross section as shown in the figure below. The cross -section of the beam is 10mm x 10mm while the modulus of elasticity of the steel is 200GPa.
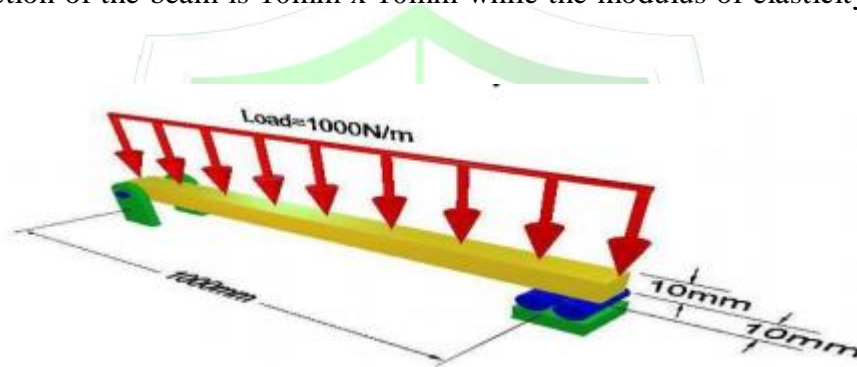

PROCEDURE:

Preprocessing: Defining the Problem

1. Open preprocessor menu

2. Give example a Title
   Utility Menu > File > Change Title .../title, Distributed Loading

3. Create Keypoints
   Preprocessor > Modeling > Create > Keypoints > In Active CS

4. Define Lines
   Preprocessor > Modeling > Create > Lines > Lines > Straight Line

5. Define Element Types
   Preprocessor > Element Type > Add/Edit/Delete... [Apply the BEAM3]

6. Define Real Constants
   Preprocessor > Real Constants... > Add..
   [Calculate and Apply suitable value in Area, Izz and Height.

7. Define Element Material Properties
   Preprocessor > Material Props > Material Models > Structural > Linear > Elastic > Isotropic

8. Define Mesh Size
   Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > All Lines...
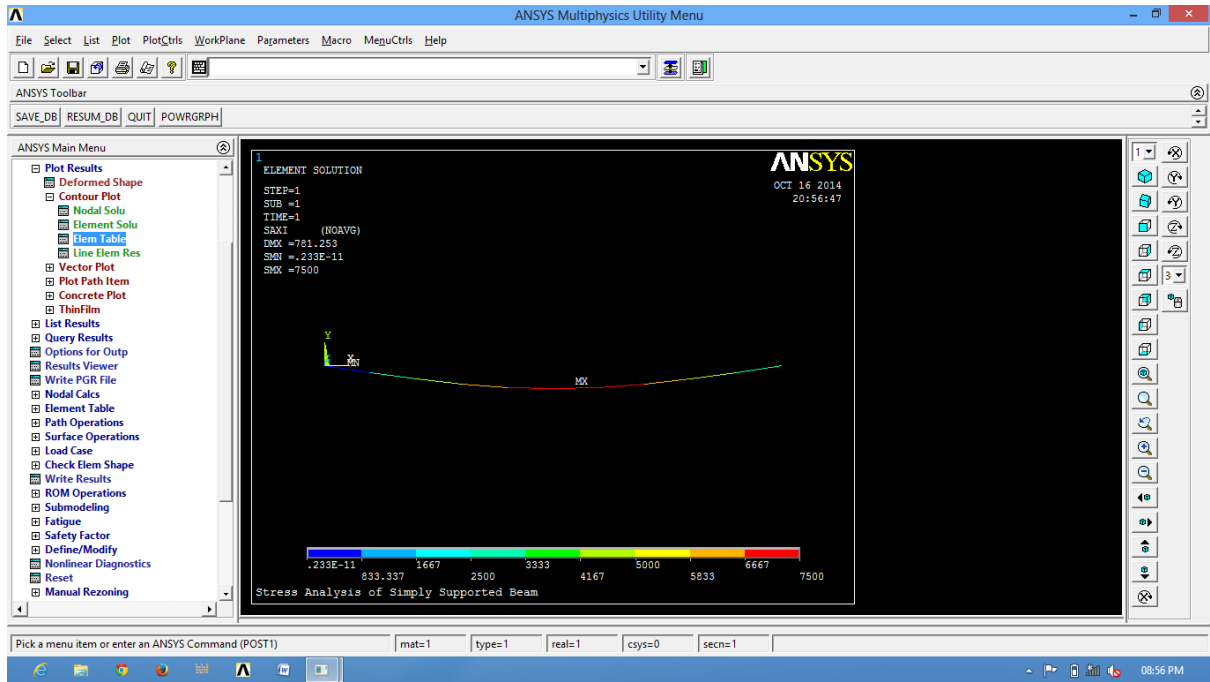
NODAL SOLUTION OF DEFORMED SHAPE
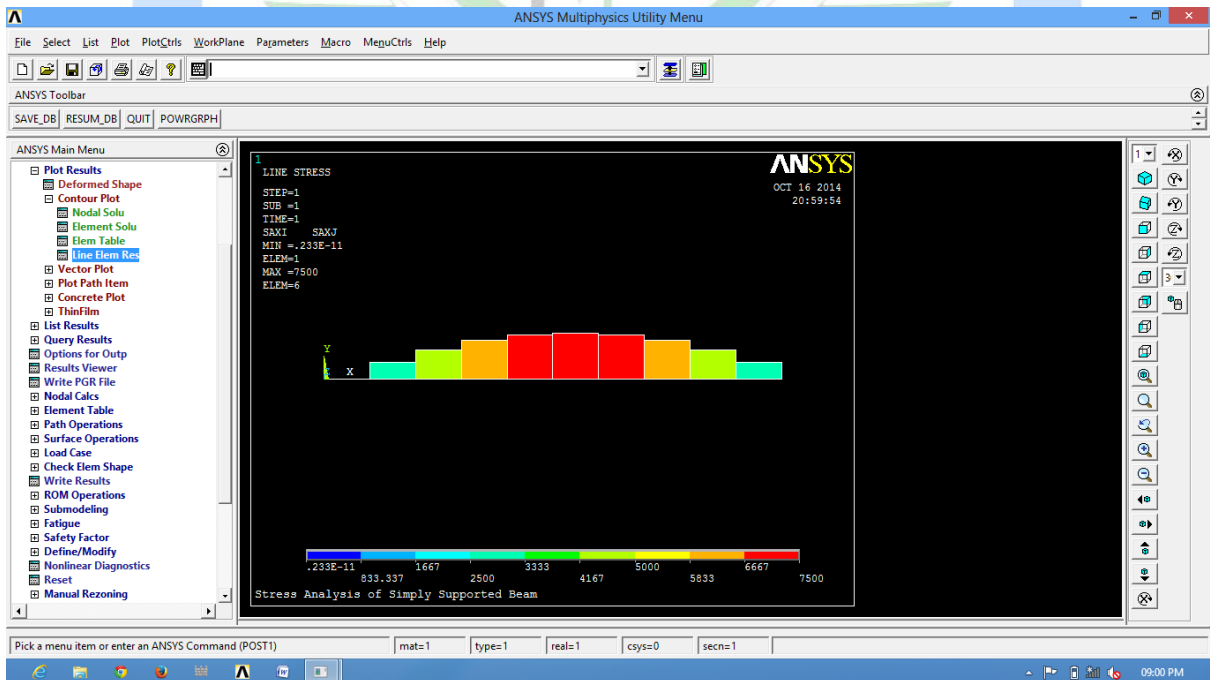
NODAL SOLUTION FOR MAXIMUM STRESS

9. Mesh the frame
        Preprocessor > Meshing > Mesh > Lines > click 'Pick All'

10. Plot Elements
        Utility Menu > Plot > Elements

Solution Phase: Assigning Loads and Solving

1. Define Analysis Type
        Solution > Analysis Type > New Analysis > Static

2. Apply Constraints
        Solution > Define Loads > Apply > Structural > Displacement > On
    Keypoints

        Apply Loads
        Select Solution > Define Loads > Apply > Structural > Pressure > On Beams

3. Solve the System
        Solution > Solve > Current LS

Post processing: Viewing the Results

1. Plot Deformed Shape
        General Postproc > Plot Results > Deformed Shape

2. Plot Principle stress distribution
As shown previously, we need to use element tables to obtain principle stresses for line
elements.

        1. Select General Postproc > Element Table > Define Table

RESULT:
        Thus the analysis of stress and deflection in a Simply Supported Beam is done using
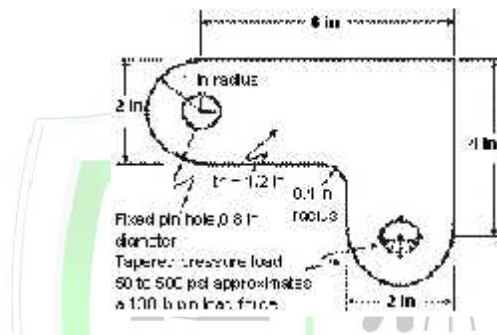ANSYS software.

STRESS ANALYSIS OF A PLATE WITH CIRCULAR HOLE

AIM

To analysis the given problem to be modeled in this example is a simple bracket shown in the following fig. This bracket is to be built from a 20mm thick steel plate is shown below this plate will be fixed at the two small holes on the left and have a load applied to the larger hole on the right.
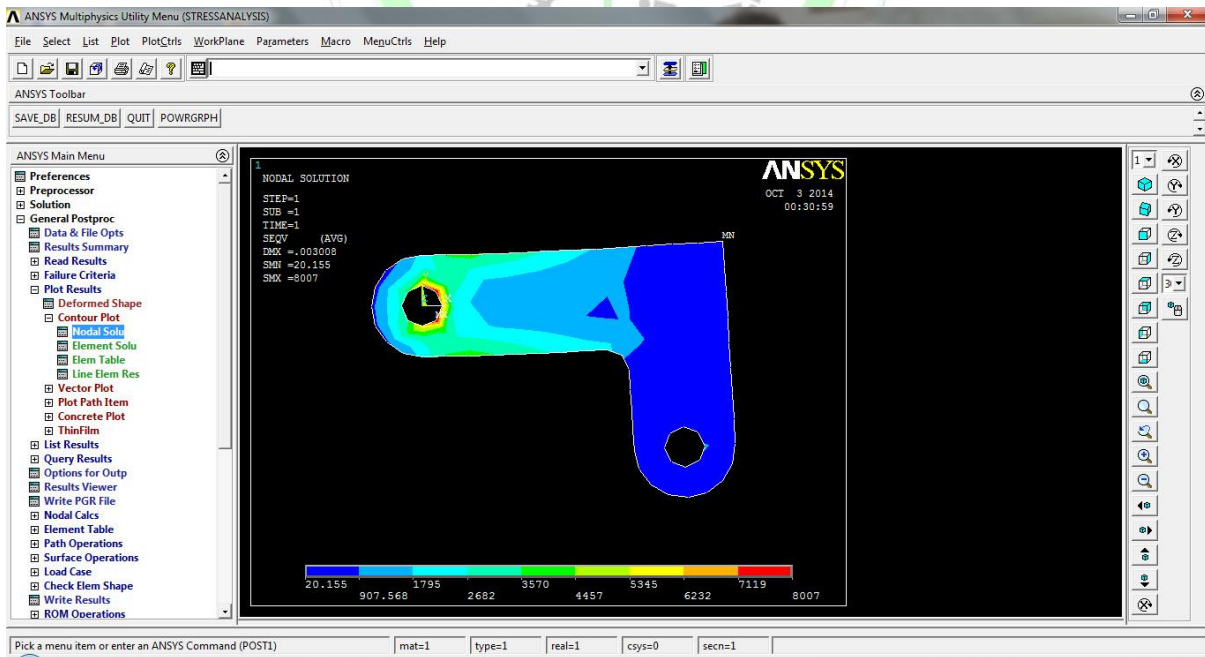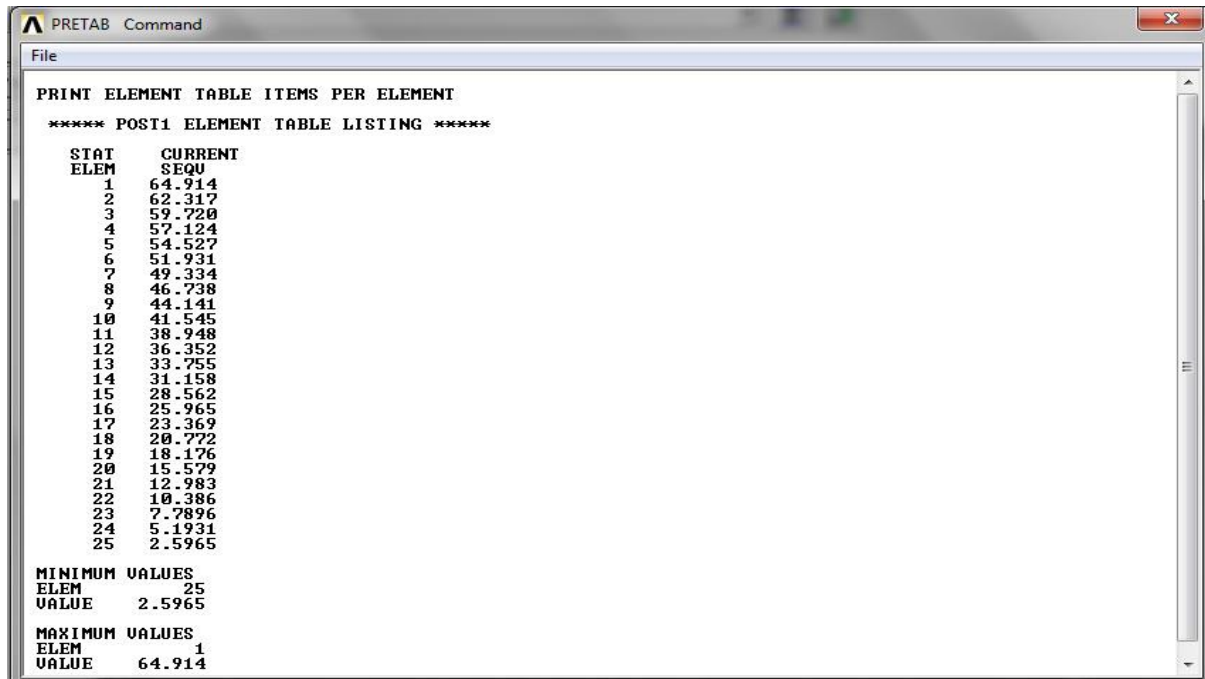

PROCEDURE:

Preprocessing: Defining the Problem

1. Give the Simplified Version a Title\
   Utility Menu > File > Change Title

2. Create the main rectangular shape
   Preprocessor > Modeling > Create > Areas > Rectangle >By Dimension

Rectangle 1				Rectangle 2			
X1	X2	Y1	Y2	X1	X2	Y1	Y2
0	6	-1	1	4	6	-1	-3

3. Create the circle
   Preprocessor > Modeling > Create > Areas > Circle > Solid Circle
   This will create a circle where the center has the coordinates 100,50,0 (the center of the rectangle) and the radius of the circle is 20 mm.

4. Subtraction
   Modeling > Operate > Booleans > Subtract > Areas.

5. Define the Type of Element
   Preprocessor Menu > Element Type > Add/Edit/Delete
   [Add the element: PLANE82]

6. Define Geometric Properties
   Preprocessor menu > Real Constants > Add/Edit/Delete [Enter a thickness of 20mm]

7. Element Material Properties
   Preprocessor > Material Props > Material models > Structural > Linear > Elastic > Isotropic

OUTPUT:


```
PRINT ELEMENT TABLE ITEMS PER ELEMENT

 ***** POST1 ELEMENT TABLE LISTING *****

   STAT    CURRENT
   ELEM    SEQU
      1    64.914
      2    62.317
      3    59.720
      4    57.124
      5    54.527
      6    51.931
      7    49.334
      8    46.738
      9    44.141
     10    41.545
     11    38.948
     12    36.352
     13    33.755
     14    31.158
     15    28.562
     16    25.965
     17    23.369
     18    20.772
     19    18.176
     20    15.579
     21    12.983
     22    10.386
     23    7.7896
     24    5.1931
     25    2.5965


 MINIMUM VALUES
 ELEM         25
 VALUE    2.5965

 MAXIMUM VALUES
 ELEM          1
 VALUE    64.914
```


NODAL SOLUTION OF DEFORMED RESULT

We are going to give the properties of Steel. Enter the following when prompted:

EX	200000
PRXY	0.3

8. Mesh Size

    Preprocessor > Meshing > Size Cntrls > Manual Size > Areas > All Areas
    [Element minimum edge length of 20]

9. Mesh

    Meshing > Mesh > Areas > Free

10. Saving Your Job

    Utility Menu > File > Save as...

Define Solution Phase: Assigning Loads and Solving

1. Define Analysis Type

    Solution > Analysis Type > New Analysis

2. Apply Constraints

    Solution > Define Loads > Apply > Structural > Displacement > On Lines
    [This location is fixed which means that all DOF's are constrained]

3. Apply Loads

    Solution > Define Loads > Apply > Structural > Pressure > On Lines
    [There is a load of 20N/mm distributed]

4. Solving the System

    Solution > Solve > Current LS

Post processing: Viewing the Results

1. Deflection

    General Postproc > Plot Results > Nodal Solution... Then select DOF solution,
USUM in the window

2. Stresses

    General Postproc > Plot Results > Nodal Solution... Then select Stress, von
    Mises in the
    window.
    You can list the von Mises stresses to verify the results at certain nodes
    General Postproc > List Results. Select Stress, Principals SPRIN
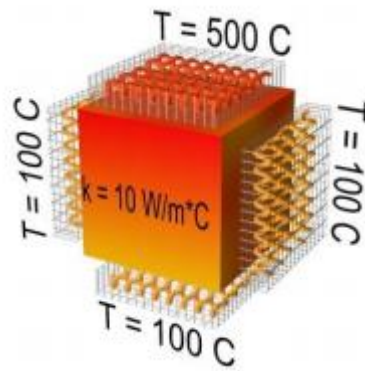   General Postproc > List Results. Select Stress, Principals SPRIN

RESULT:

    Thus the Stress analysis of Bracket is done using ANSYS software.

SIMPLE CONDUCTION EXAMPLE

AIM:

To analysis in the Simple Conduction Example is constrained as shown in the following figure. Thermal conductivity (k) of the material is 10 W/m*C and the block is assumed to be infinitely long.


PROCEDURE:

Preprocessing: Defining the Problem

1. Give example a Title

2. Open preprocessor menu

    ANSYS Main Menu > Preprocessor

3. Create geometry

    Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners > X=0, Y=0,

Width=1, Height=1

4. Define the Type of Element

    Preprocessor > Element Type > Add/Edit/Delete... > click 'Add' > Select

    Thermal Mass Solid, Quad 4Node 55

5. Element Material Properties

    Preprocessor > Material Props > Material Models > Thermal > Conductivity >

    Isotropic > KXX = 10 (Thermal conductivity)

OUTPUT:
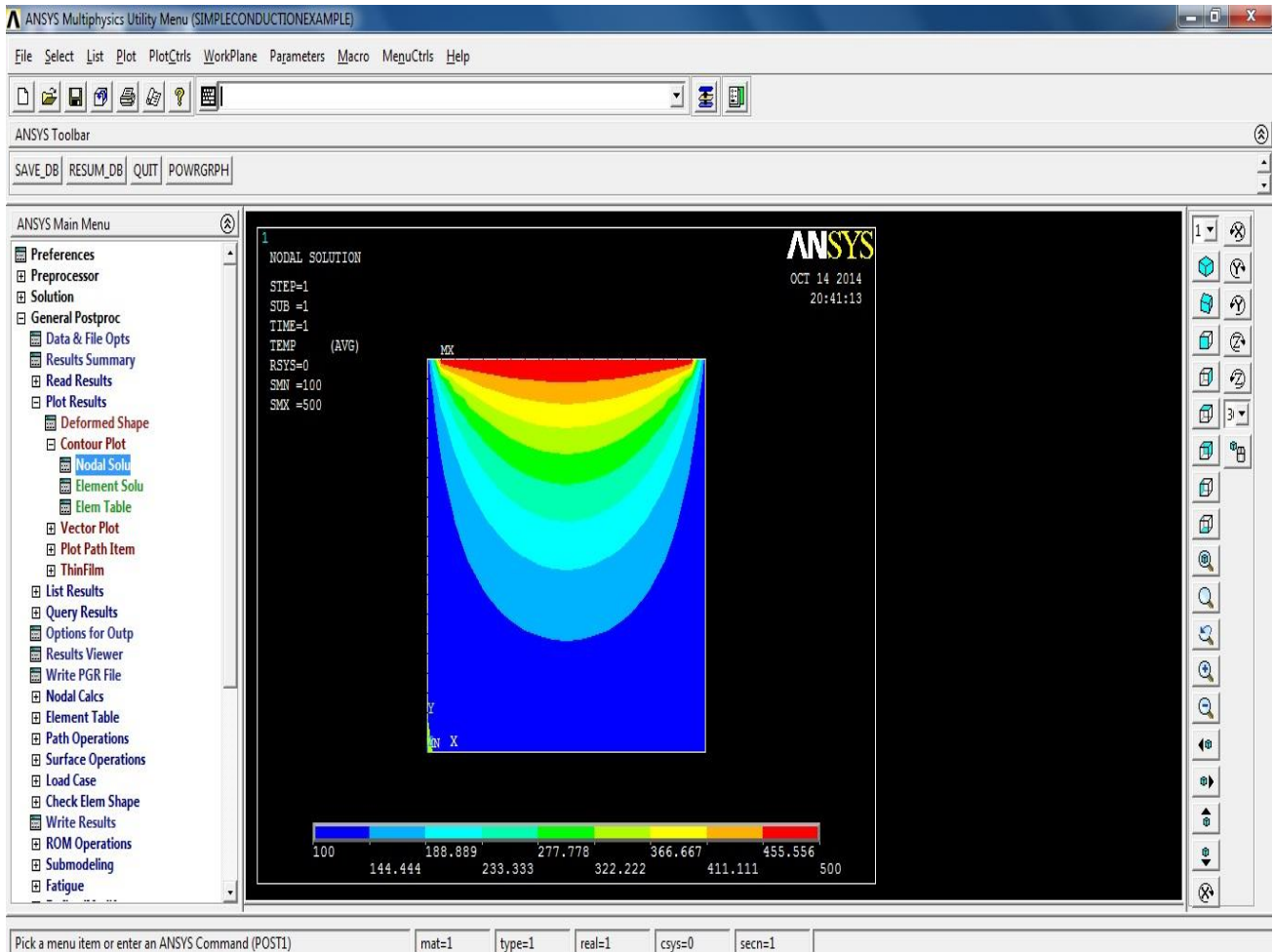
NODAL SOLUTION FOR THERMAL ANALYIS

6. Mesh Size

   Preprocessor > Meshing > Size Cntrls > ManualSize > Areas > All Areas >
   0.05

7. Mesh

   Preprocessor > Meshing > Mesh > Areas > Free > Pick All

Solution Phase: Assigning Loads and Solving

1. Define Analysis Type

   Solution > Analysis Type > New Analysis > Steady-State

2. Apply Constraints

   o Solution > Define Loads > Apply Thermal > Temperature > On Nodes

   o Click the Box option (shown below) and draw a box around the nodes on
   the top line.

   o Fill the window in as shown to constrain the side to a constant temperature
   of 500

   o Using the same method, constrain the remaining 3 sides to a constant value
   of 100

3. Solve the System

   Solution > Solve > Current LS

Post processing: Viewing the Results

1. Results Using ANSYS

   Plot Temperature

   General Postproc > Plot Results > Contour Plot > Nodal Solu ... > DOF
   solution, Temperature

RESULT:

   Thus the analysis in the Simple Conduction for the Rectangular Solid Model using

ANSYS software.