WEST GODAVARI INSTITUTE OF SCIENCE & ENGINEERING

(Approved by AICTE, New Delhi and Affiliated to JNTU, Kakinada)

An ISO 9001-2015 Certified College

AVAPADU, **PRAKASARAOPALEM** – 534 112, W.G.Dist., A.P

CAE & CAM LAB MANUAL-R20

DEPARTMENT OF MECHANICAL ENGINEERING

III B.TECH II SEMESTER

JAWAHARLAL NEHRU TECHNOLOGICAL UNIVERSITY KAKINADA
KAKINADA–533003, Andhra Pradesh, India

2022-23

INTRODUCTION

Ansys is one of the Mechanical finite element analysis software is used to simulate computer models of structures, electronics, or machine components for analyzing the strength, toughness, elasticity, temperature distribution, electromagnetism, fluid flow, and other attributes. Ansys is used to determine how a product will function with different specifications, without building test products or conducting crash tests.

Most Ansys simulations are performed using the Ansys Workbench system, which is one of the company's main products, Ansys users break down larger structures into small components that are each modeled and tested individually. A user may start by defining the dimensions of an object, and then adding weight, pressure, temperature and other physical properties. Finally, the Ansys software simulates and analyzes movement, fatigue, fractures, fluid flow, temperature distribution, electromagnetic efficiency and other effects over time.

Ansys also develops software for data management and backup, academic research and teaching. Ansys software is sold on an annual subscription basis.

The finite element method (FEM) is a popular method for numerically solving differential equations arising in engineering and mathematical modeling. Typical problem areas of interest include the traditional fields of structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential

.**FEA Applications :**

- Mechanical/Aerospace/Civil/Automotive Engineering
- Structural/Stress Analysis
    - Static/Dynamic
    - Linear/Nonlinear
- Fluid Flow
- Heat Transfer
- Electromagnetic Fields
- Soil Mechanics
- Acoustics
- Biomechanics

<u>The common steps involved in finite element analysis are</u>

Step1:Modelling

Step 2: Material definition. The material properties are defined in this step.Step

3: Definition of loads.

Step 4: Boundary conditions.Step

5: Meshing.

Step 6: Solution.

Step 7: Post-processing

Step 1: Modelling

The part is modeled omitting complicated geometrical features. This is the first and the most crucial step in any analysis. This will provide you the insights to remove insignificant features from our geometry eventually saving some computational time and unwanted complexity. Always remember, simpler, the better.

Step 2: Material definition

The material properties are defined in this step. These material properties depend on the type of analysis that needs to be carried out. Literature survey for similar problems could help you a lot in tackling this step.

Step 3: Definition of loads

This step is about the definition of external forces acting on the part or the body force by virtue of the weight of the component. We have to be careful about the force definition type in order avoid encountering problems like singularities.

Step 4: Boundary conditions

This step is mainly done to reduce the complexity of the problem from an engineering sense. For example, in some problems, the user might be knowing some initial conditions before starting an analysis (like displacement of a point in the geometry). Such cases fall into the "Initial value problem" category.

Step 5: Meshing

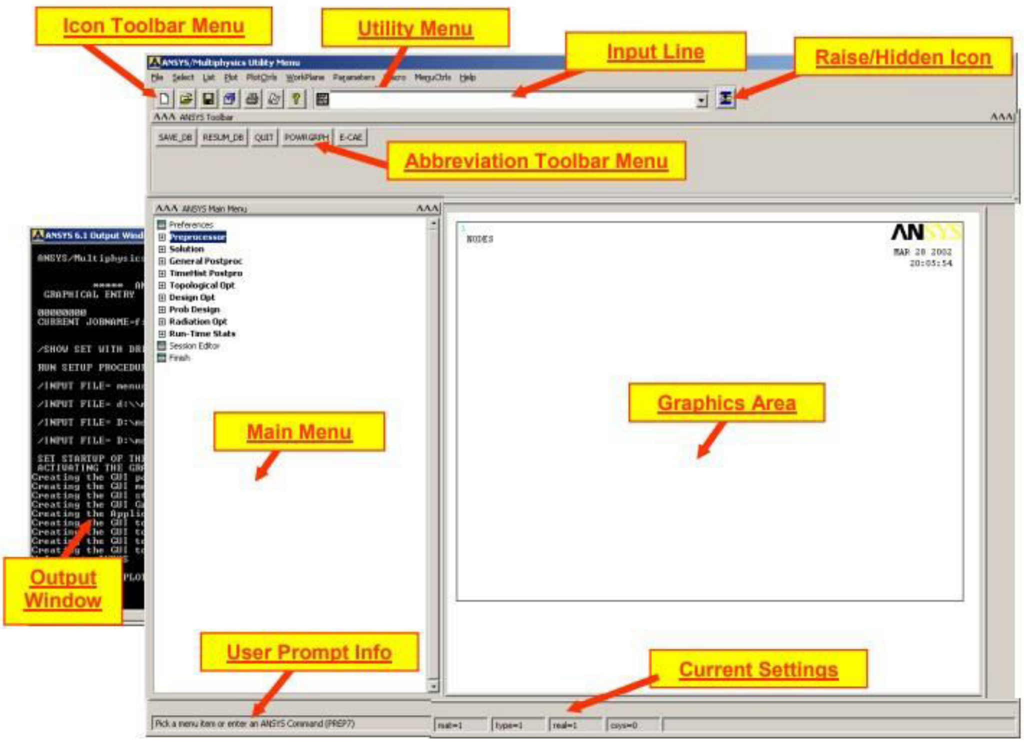
Our geometry is divided into smaller and simpler shapes called as finite elements. And then bind in to a single element

Step 6: Solution

This step works after the definition of the simulation properties. It means that the partial differential equations are converted into algebraic equations. Doing this helps the code to represent equations in terms of matrices. Matrices of individual elements are assembled into global matrices for the entire geometry which is then solved by solvers for unknown variables.

Step 7: Post-processing

Any FEM software will have some form of an indicator to show the user if the solution has been completed successfully. Once the solver gives this message, the variables (Displacement, von Mises Stress etc.) that have been calculated are presented in terms of contour plots or graphs.



Utility Menu Contains functions which are available throughout the ANSYS session, such as file controls, selecting, graphics controls, parameters, and exiting.

Toolbar Menu Contains push buttons for executing commonly used ANSYS commands and functions.Customized buttons can be created.

Graphics Area Displays graphics created in ANSYS or imported into ANSYS.

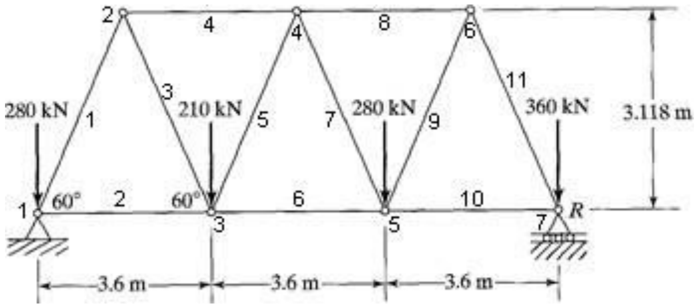
Input Line Displays program prompt messages and a text field for typing commands. All previously typedcommands appear for easy reference and access.

Main Menu Contains the primary ANSYS functions, organized by processors (preprocessor, solution,general postprocessor, etc.)

Output Displays text output from the program. It is usually positioned behind the other windows andcan be raised to the front when necessary.

EXPERIMENT -1(a) ANALYSIS ON TRUSS ELEMENT

AIM: Determine the nodal deflections and stress for the 2D and 3D truss system shown below (E = 200GPa, A = 3250 mm^2).



Procedure:

The main steps to be involved are

1. Preferences
2. Pre Processor
3. Solution
4. General Post Processor

To Give Title for the Experiment and some Basic Requirements

- Utility menu bar > File > Change Title> Analysis on 2D Beam element
- Utility menu bar > File > Save as> Give file name and Location
- Utility Menu > Plot > Replot (For quick visibility of Title and all)

STEP 1: Preferences

- Preferences > Structural>OK

STEP 2: Pre Processor

Step 2(a): Define the Type of Element

- Preprocessor >Element Type > Add/Edit/Delete >Add >Link > 3d finit stn 180 > OK > Close

Step 2(b): Define Geometric Properties

- Preprocessor > Real Constants > Add/Edit/Delete > Add >Ok > Cross Sectional area-3250 mm^2 > Ok > Close

Step 2(c) : Element Material Properties

- Preprocessor >Material Props > Material Models > Double click on Structural > Linear > Elastic > Isotropic > EX(Young's Modulus)- 2e5 >PRXY(Poissons Ratio)-0.3 > Ok > Close

Step 2(d) : To Model Element

- Preprocessor > Modeling > Create >Key points> In Active CS

keypoint	coordinate	
	x	y
1	0	0
2	1800	3118
3	3600	0
4	5400	3118
5	7200	0
6	9000	3118
7	10800	0

- Preprocessor > Modeling > Create > Lines > Lines > In Active Coord. > Select all key points to get truss element

Step 2(e) : To Mesh Truss Element

- Preprocessor >Meshing > Size Cntrls>ManualSize> Lines > All Lines > No of element divisions :1> Ok
- Preprocessor >Meshing > Mesh > Lines >Pick All > Ok
- Utility Menu > PlotCtrls > Numbering > Node Numbers –ON > OK

Step 2(f): Assigning Loads and Constraints on element

- Preprocessor > Loads >Analysis Type > New Analysis > Static >Ok
- Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Keypoint > Click on Keypoint 1 >Ok > All DOF > Ok
- Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Keypoint > Click on Keypoint 7 >Ok > UY > Ok
- Preprocessor > Loads > Define Loads > Apply > Structural > Force/Moment > On Keypoint > Click on Keypoint 1 > Ok > Select FY (-280000) > Ok
- Preprocessor > Loads > Define Loads > Apply > Structural > Force/Moment > On Keypoint > Click on Keypoint 3 > Ok > Select FY (-210000) > Ok
- Preprocessor > Loads > Define Loads > Apply > Structural > Force/Moment > On Keypoint > Click on Keypoint 5 > Ok > Select FY (-280000) > Ok
- Preprocessor > Loads > Define Loads > Apply > Structural > Force/Moment > On Keypoint > Click on Keypoint 7 > Ok > Select FY (-360000) > Ok

STEP 3: Solution

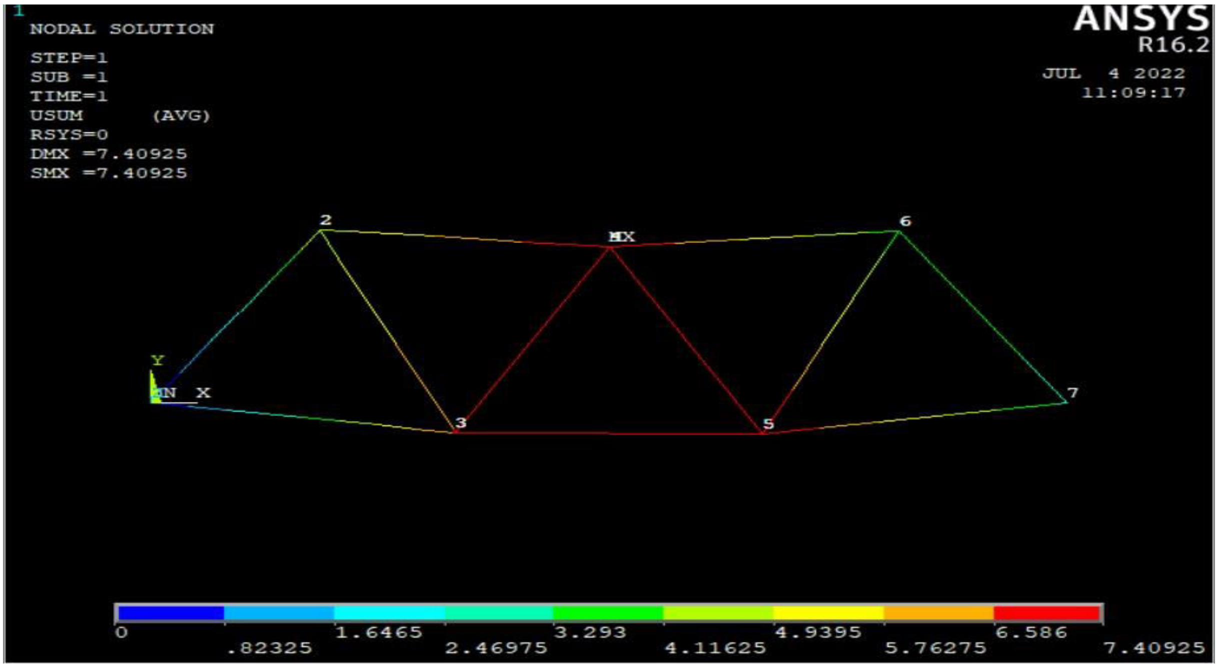
Solution >Solve > Current LS> Ok

STEP 4: General Postprocessor

Step 4(a): To View Nodal Deflection of Truss Element

- General Postprocessor > Plot results > Contour Plot > Nodal Solution > Nodal Solution > DOF solution > Displacement Vector sum
- General Postprocessor > List results > Nodal Solution > Nodal Solution> DOF solution >

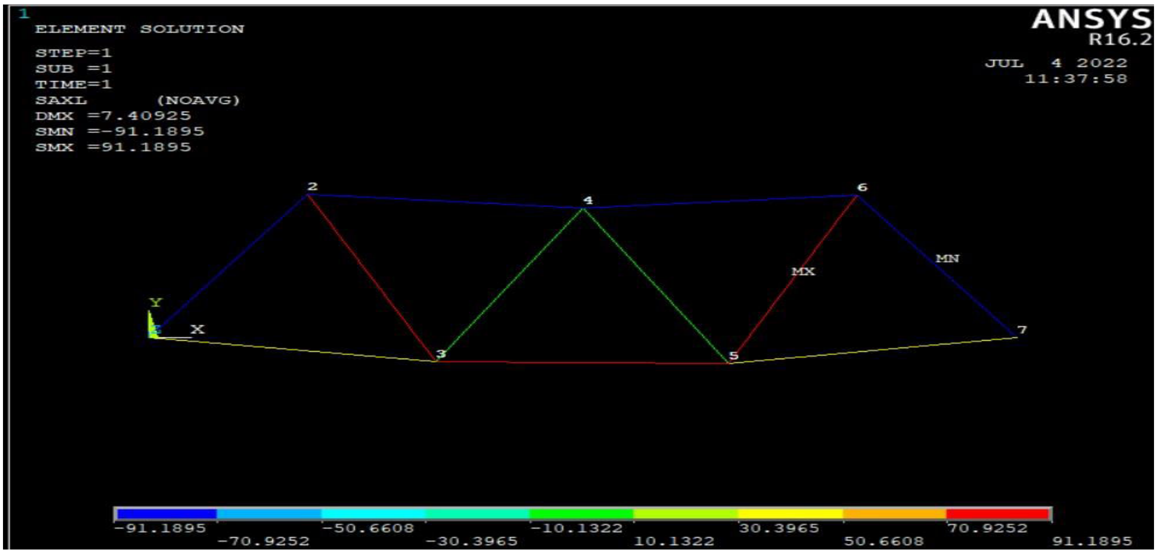
Displacement Vector sum




```
PRINT U     NODAL SOLUTION PER NODE

 ***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

 LOAD STEP=      1  SUBSTEP=      1
  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    3.0836     -3.5033      0.0000      4.6671
      3   0.74604     -6.5759      0.0000      6.6181
      4    1.5916     -7.2363      0.0000      7.4093
      5    2.3127     -6.9923      0.0000      7.3648
      6  -0.49736E-01 -3.7330     0.0000      3.7333
      7    3.1334      0.0000      0.0000      3.1334

MAXIMUM ABSOLUTE VALUES
NODE          7           4           0           4
VALUE    3.1334     -7.2363      0.0000      7.4093
```

Step 4(b): To View Stresses produced in Truss Element

- General Postprocessor > Element Table > Define Table> Add>User Label for item :SAXL> click on By sequence Number > Click on LS > Enter LS,1> Ok > close
- General Postprocessor > Element Table > Plot Element table > Select SAXL > Ok
- General Postprocessor > Element Table > List Element table Select SAXL > Ok



PRINT ELEMENT TABLE ITEMS PER ELEMENT

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

STAT ELEM	CURRENT SXAL
1	-82.900
2	82.900
3	-8.2900
4	8.2900
5	91.189
6	-91.189
7	45.591
8	87.038
9	41.447
10	-82.893
11	-91.183

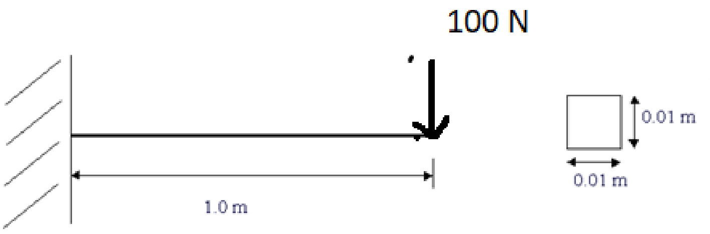
MINIMUM VALUES
ELEM 6
VALUE -91.189

MAXIMUM VALUES
ELEM 5

Result: Nodal deflections and stresses are determined for the 2D truss element

EXPERIMENT -1(b) ANALYSIS ON BEAM ELEMENT

AIM: Determine the nodal deflections and stress for the 2D Beam as shown below (E = 200GPa, Density=7800 kg/m^3).



Procedure:

The main steps to be involved are

1. Preferences
2. Pre Processor
3. Solution
4. General Post Processor

To Give Title for the Experiment and some Basic Requirements

- Utility menu bar > File > Change Title> Analysis on 2D Beam element
- Utility menu bar > File > Save as> Give file name and Location
- Utility Menu > Plot > Replot (For quick visibility of Title and all)

STEP 1: Preferences

- Preferences > Structural>OK

STEP 2: Pre Processor

Step 2(a): Define the Type of Element

- Preprocessor >Element Type > Add/Edit/Delete >Add > BEAM > 2 Node 188 > Ok.

Step 2(b) : Element Material Properties

- Preprocessor >Material Props > Material Models > Double click on Structural > Linear > Elastic > Isotropic > EX(Young's Modulus)- 2e11 >PRXY(Poissons Ratio)-0.3 > Ok
- Preprocessor >Material Props > Material Models > Double click on Structural>Density > 7800 > ok > close

Step 2(c) : To Select type of Section

- Preprocessor Sections > Beam > Common Sections > enter B- 0.01 and H- 0.01 > ok

Step 2(d): To Model Element

- Preprocessor > Modeling > Create >Key points> In Active CS

Keypoint	Coordinates (x,y)
1	(0,0)
2	(1,0)

- Preprocessor > Modeling > Create > Lines > Lines > In Active Coord. > Select all key points to get Beam element

Step 2(e) : To Mesh Truss Element

- Preprocessor >Meshing > Size Cntrls>ManualSize> Lines > All Lines > No of element divisions :10> Ok
- Preprocessor >Meshing > Mesh > Lines >Pick All > Ok
- Utility Menu > PlotCtrls > Numbering > Node Numbers –ON > OK

Step 2(f): Assigning Loads and Constraints on element

- Preprocessor > Loads >Analysis Type > New Analysis > Static >Ok
- Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Keypoint > Click on Keypoint 1 >Ok > All DOF > Ok
- Preprocessor > Loads > Define Loads > Apply > Structural > Force/Moment > On Keypoint > Click on Keypoint 2 > Ok > Select FY (-100) > Ok

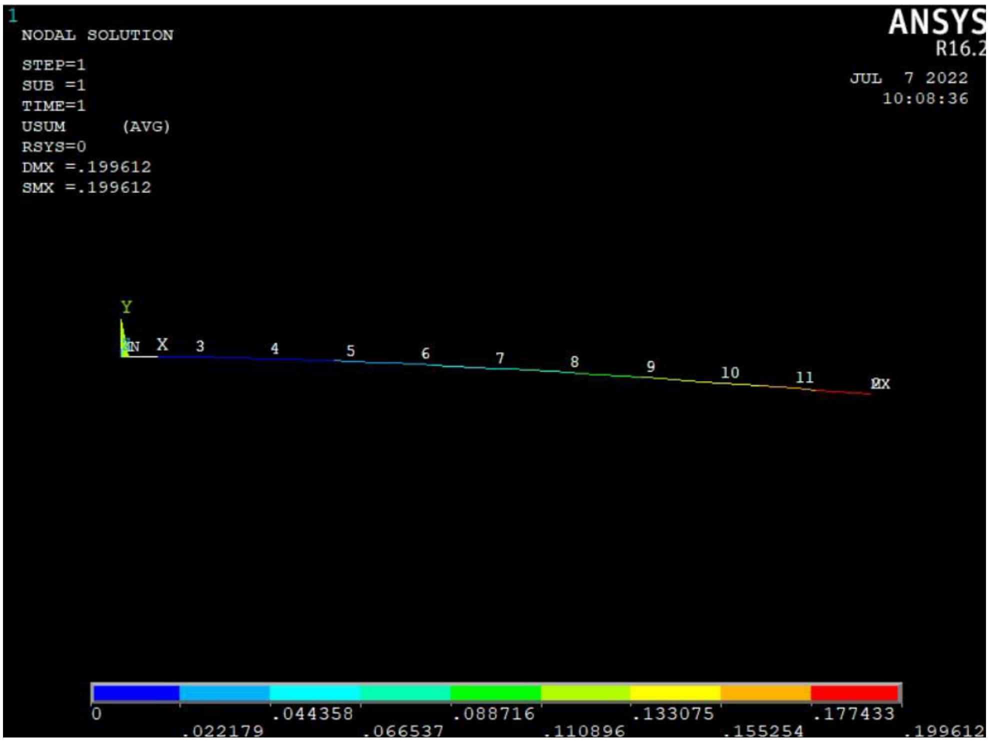
STEP 3: Solution

Solution >Solve > Current LS> Ok

STEP 4: General Postprocessor

Step 4(a): To View Nodal Deflection of Truss Element

- General Postprocessor > Plot results > Contour Plot > Nodal Solution > Nodal Solution > DOF solution > Displacement Vector sum
- General Postprocessor > List results > Nodal Solution > Nodal Solution> DOF solution > Displacement Vector sum



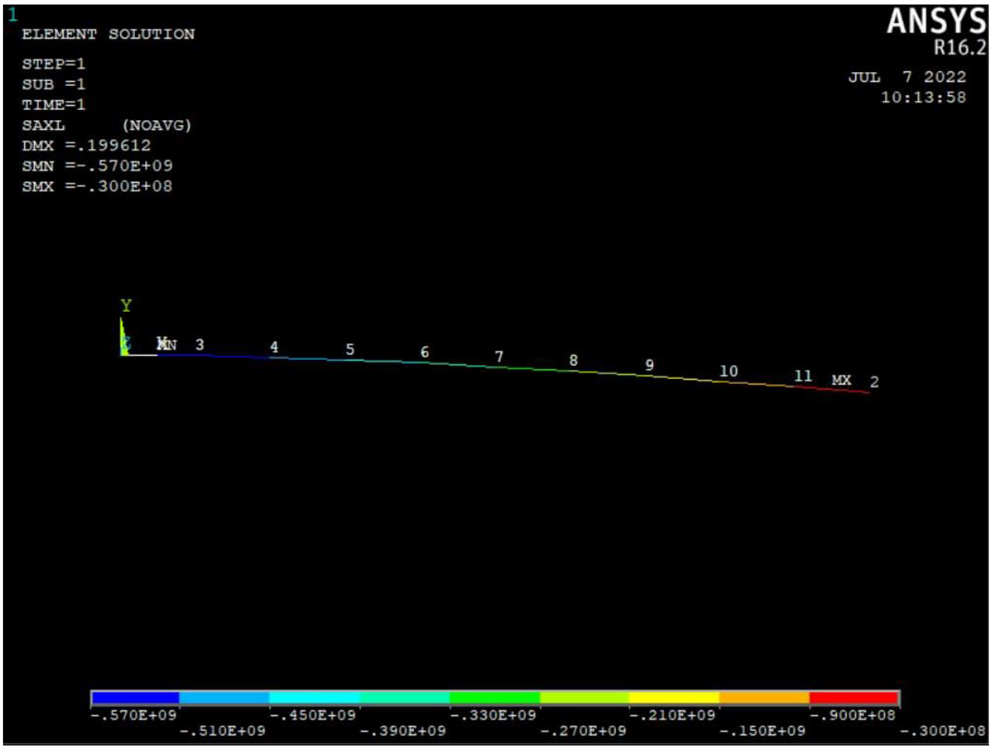
PRNSOL Command

File

```
***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

LOAD STEP=     1  SUBSTEP=     1
 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     -0.19961     0.0000     0.19961
      3    0.0000     -0.28612E-02 0.0000     0.28612E-02
      4    0.0000     -0.11122E-01 0.0000     0.11122E-01
      5    0.0000     -0.24184E-01 0.0000     0.24184E-01
      6    0.0000     -0.41445E-01 0.0000     0.41445E-01
      7    0.0000     -0.62306E-01 0.0000     0.62306E-01
      8    0.0000     -0.86167E-01 0.0000     0.86167E-01
      9    0.0000     -0.11243     0.0000     0.11243
     10    0.0000     -0.14049     0.0000     0.14049
     11    0.0000     -0.16975     0.0000     0.16975

MAXIMUM ABSOLUTE VALUES
NODE          0           2           0           2
VALUE    0.0000     -0.19961      0.0000     0.19961
```

Step 4(b): To View Stresses produced in Truss Element

- General Postprocessor > Element Table > Define Table> Add>User Label for item :SAXL> click on By sequence Number > Click on LS > Enter LS,1> Ok > close
- General Postprocessor > Element Table > Plot Element table > Select SXAL > Ok
  General Postprocessor > Element Table > List Element table >Select SXAL > Ok



PRETAB Command

File

```
PRINT ELEMENT TABLE ITEMS PER ELEMENT

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

    STAT     CURRENT
    ELEM     SAXL
       1 -0.57000E+09
       2 -0.51000E+09
       3 -0.45000E+09
       4 -0.39000E+09
       5 -0.33000E+09
       6 -0.27000E+09
       7 -0.21000E+09
       8 -0.15000E+09
       9 -0.90000E+08
      10 -0.30000E+08

MINIMUM VALUES
ELEM          1
VALUE  -0.57000E+09

MAXIMUM VALUES
ELEM         10
VALUE  -0.30000E+08
```

Result: Nodal deflections and stresses are determined for the 2D Beam element,

EXPERIMENT -2

ANALYSIS ON RECTANGULAR PLATE WITH HOLE

AIM: Determination of deflections, principal and Von-mises stresses in plane Stress, plane Strain and Axi-symmetric components.

Procedure:

The main steps to be involved are

1. Preferences
2. Pre Processor
3. Solution
4. General Post Processor

To Give Title for the Experiment and some Basic Requirements

- Utility menu bar > File > Change Title> Analysis on Rectangular plate with hole element
- Utility menu bar > File > Save as> Give file name and Location
- Utility Menu > Plot > Replot (For quick visibility of Title and all)

STEP 1: Preferences

- Preferences > Structural>OK

STEP 2: Pre Processor

Step 2(a): Define the Type of Element

- Preprocessor >Element Type > Add/Edit/Delete >Add > Solid > 8 node 183 > ok >Options > Elememt Behaviour(k3)- Plane Stress > ok > close.

Step 2(b) : Element Material Properties

- Preprocessor >Material Props > Material Models > Double click on Structural > Linear > Elastic > Isotropic > EX(Young's Modulus)- 2e5 >PRXY(Poissons Ratio)-0.3 > Ok > Close

Step 2(c): To Model Element

- Preprocessor > Modeling > Create >Ares >Rectangle > by 2 Corners

WP X	**0**
WP Y	0
WIDTH	200
HEIGH T	100

- Preprocessor > Modeling > Create >Ares >Circle > Solid circle

WP X	**100**

WP Y	50
RADIUS	20

- Preprocessor > Modeling > Operate > Booleans > Subtract > areas > Select total Rectangle > Ok in New Open end Box > Ok > Double click on circle > Ok> Close.

Step 2(d) : To Mesh Truss Element

- Preprocessor >Meshing > Size Cntrls> Manual Size> areas > all areas > element edge length > 10 > Ok
- Preprocessor >Meshing > Mesh >areas > free > Pick All > Ok
- Utility Menu > PlotCtrls > Numbering > Node Numbers –ON > OK

Step 2(e): Assigning Loads and Constraints on the elemnt

- Preprocessor >Loads >Analysis type > New Analysis > Static >Ok
- Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On lines > Click on left side vertical line > Ok > All DOF > Ok
- Preprocessor > Loads > Define Loads > Apply > Structural > Pressure> On Lines > Click on Right side vertical line > Ok > VALUE Load PRES Value as -20 N/mm^2> Ok

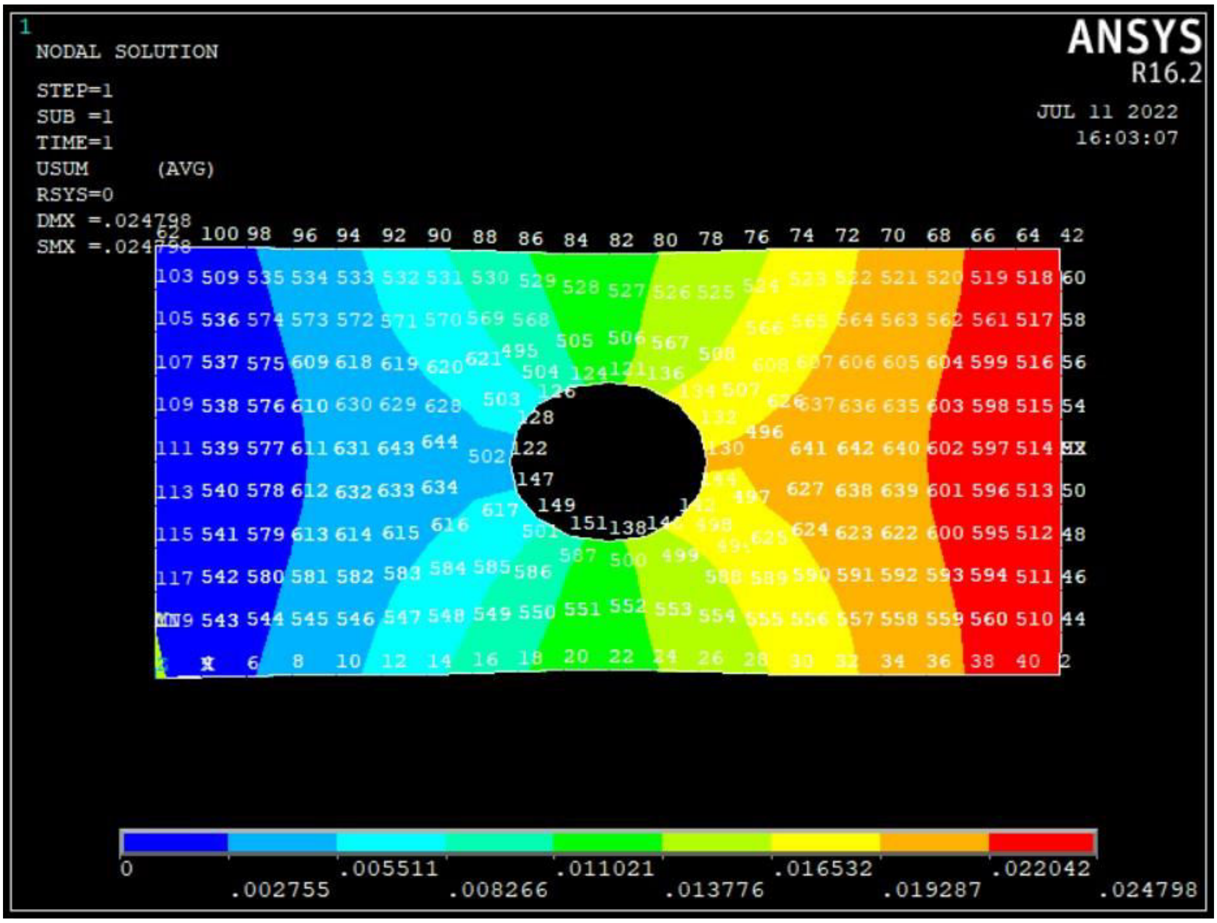
STEP 3: Solution

Solution >Solve > Current LS> Ok

STEP 4: General Postprocessor

Step 4(a): To View Nodal Deflection in Element

- General Postprocessor > Plot results > Contour Plot > Nodal Solution > Nodal Solution > DOF solution > Displacement Vector sum
- General Postprocessor > List results > Nodal Solution > Nodal Solution> DOF solution > Displacement Vector sum



PRNSOL Command

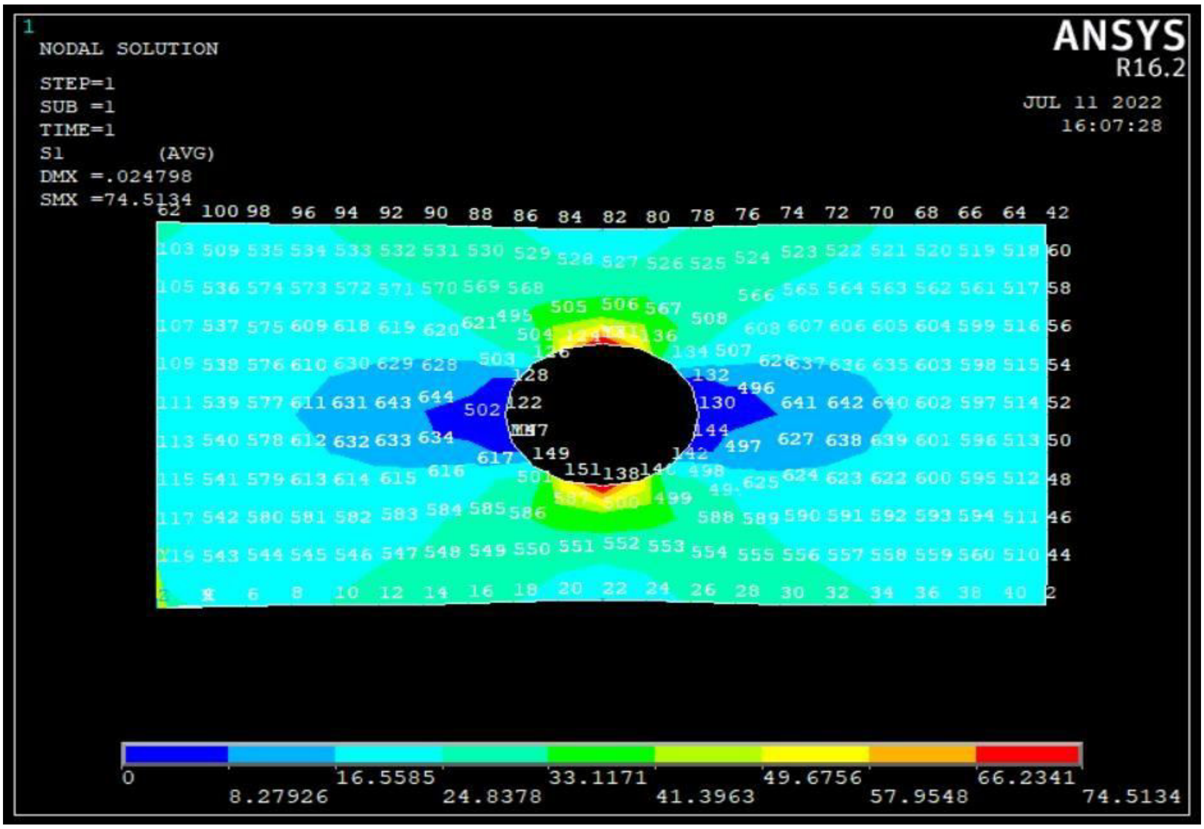
File

```
 TIME=     1.0000      LOAD CASE=   0

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

   NODE       UX          UY          UZ          USUM
    630  0.32947E-02-0.93537E-04  0.0000      0.32960E-02
    631  0.31720E-02 0.41263E-07  0.0000      0.31720E-02
    632  0.32945E-02 0.94026E-04  0.0000      0.32959E-02
    633  0.40251E-02 0.99569E-04  0.0000      0.40264E-02
    634  0.46339E-02 0.11685E-03  0.0000      0.46354E-02
    635  0.21132E-01-0.13800E-03  0.0000      0.21132E-01
    636  0.20379E-01-0.12535E-03  0.0000      0.20379E-01
    637  0.19736E-01-0.14535E-03  0.0000      0.19736E-01
    638  0.20333E-01 0.12494E-03  0.0000      0.20334E-01
    639  0.21111E-01 0.13750E-03  0.0000      0.21111E-01
    640  0.21235E-01 0.70291E-06  0.0000      0.21235E-01
    641  0.19979E-01 0.44016E-05  0.0000      0.19979E-01
    642  0.20525E-01 0.12576E-06  0.0000      0.20525E-01
    643  0.38483E-02-0.13258E-05  0.0000      0.38483E-02
    644  0.43887E-02-0.15524E-04  0.0000      0.43887E-02

 MAXIMUM ABSOLUTE VALUES
 NODE          52          22           0            52
 VALUE   0.24798E-01 0.42534E-02  0.0000      0.24798E-01
```

Step 4(b): To View Principal Stresses in Element

- ➤ General Postprocessor > Plot results > Contour Plot > Nodal Solution > Nodal Solution > Stress> 1St Principal Stresses > Ok

- ➤ General Postprocessor > List results > Nodal Solution > Nodal Solution> Stress> 1St Principal Stresses > Ok



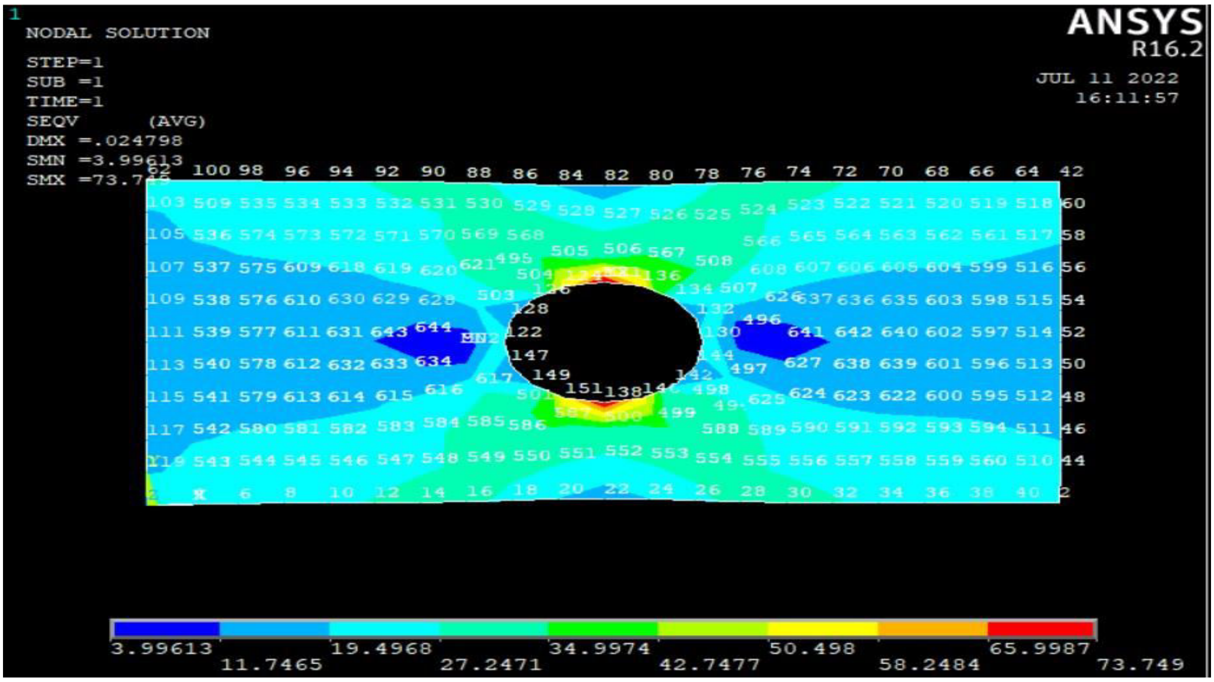
PRNSOL Command

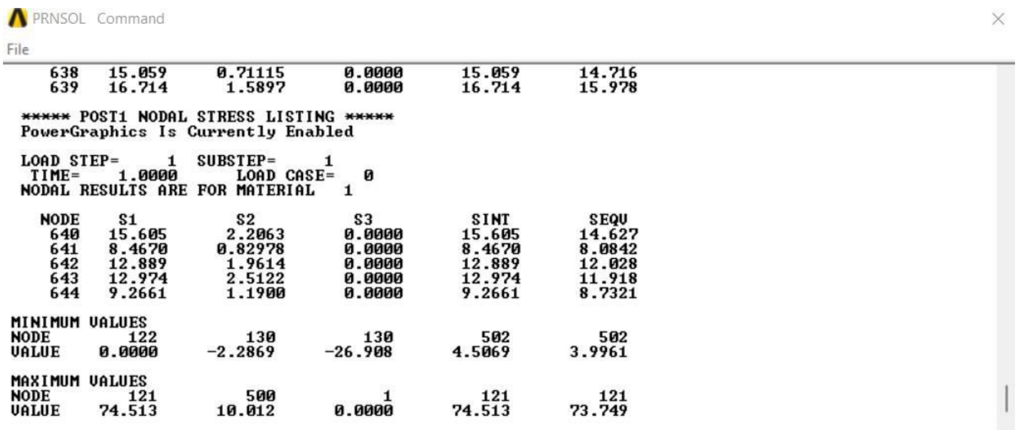
File

```
   638    15.059      0.71115      0.0000      15.059      14.716
   639    16.714       1.5897      0.0000      16.714      15.978

 ***** POST1 NODAL STRESS LISTING *****
 PowerGraphics Is Currently Enabled

 LOAD STEP=      1  SUBSTEP=      1
  TIME=    1.0000      LOAD CASE=   0
 NODAL RESULTS ARE FOR MATERIAL    1

   NODE     S1          S2          S3          SINT        SEQV
    640    15.605      2.2063      0.0000      15.605      14.627
    641    8.4670     0.82978      0.0000      8.4670      8.0842
    642    12.889      1.9614      0.0000      12.889      12.028
    643    12.974      2.5122      0.0000      12.974      11.918
    644    9.2661      1.1900      0.0000      9.2661      8.7321

 MINIMUM VALUES
 NODE        122         130         130         502         502
 VALUE    0.0000     -2.2869     -26.908      4.5069      3.9961

 MAXIMUM VALUES
 NODE        121         500           1         121         121
 VALUE    74.513      10.012      0.0000      74.513      73.749
```

Step 4(b): To View Von Mises Stresses in Element

- General Postprocessor > Plot results > Contour Plot > Nodal Solution > Nodal Solution > Stress> Von Mises Stress> Ok
- General Postprocessor > List results > Contour Plot > Nodal Solution > Nodal Solution > Stress> Von Mises Stress> Ok





PRNSOL Command

File

638	15.059	0.71115	0.0000	15.059	14.716
639	16.714	1.5897	0.0000	16.714	15.978

***** POST1 NODAL STRESS LISTING *****
PowerGraphics Is Currently Enabled

LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0
NODAL RESULTS ARE FOR MATERIAL 1

NODE	S1	S2	S3	SINT	SEQU
640	15.605	2.2063	0.0000	15.605	14.627
641	8.4670	0.82978	0.0000	8.4670	8.0842
642	12.889	1.9614	0.0000	12.889	12.028
643	12.974	2.5122	0.0000	12.974	11.918
644	9.2661	1.1900	0.0000	9.2661	8.7321

MINIMUM VALUES

NODE	122	130	130	502	502
VALUE	0.0000	-2.2869	-26.908	4.5069	3.9961

MAXIMUM VALUES

NODE	121	500	1	121	121
VALUE	74.513	10.012	0.0000	74.513	73.749

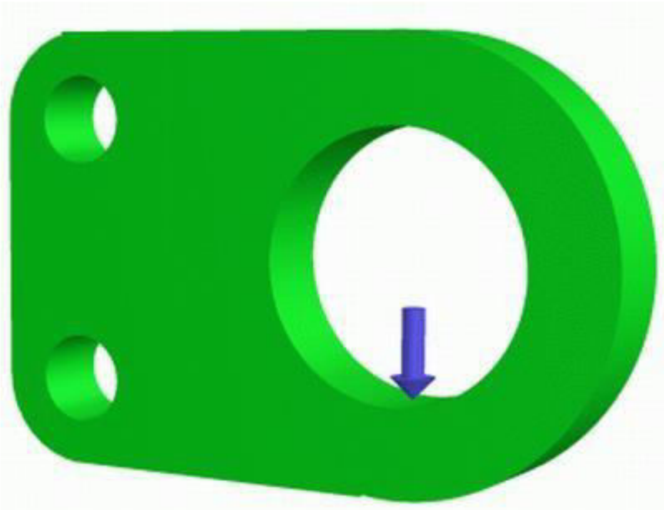
Note: Similarly deflections, principal and Von-mises stresses for plane Strain and Axi-symmetric components can be obtained using same process except by changing the element behaviour as plane strain or axi symeetric,

Result: Deflections, principal and Von-mises stresses in plane Stress/ plane Strain /Axi-symmetric components are determined.

EXPERIMENT -3

ANALYSIS ON 3D BRACKET

AIM: Determination of stresses in 3D Bracket



G=200 GPa

t=20 mm

Procedure:

The main steps to be involved are

1. Preferences
2. Pre Processor
3. Solution
4. General Post Processor

To Give Title for the Experiment and some Basic Requirements

- Utility menu bar > File > Change Title> Analysis on Bracket
- Utility menu bar > File > Save as> Give file name and Location
- Utility Menu > Plot > Replot (For quick visibility of Title and all)

STEP 1: Preferences

- Preferences > Structural>OK

STEP 2: Pre Processor

Step 2(a): Define the Type of Element

- Preprocessor >Element Type > Add/Edit/Delete >Add > Solid > 8 node 183 > ok >Options > Element Behavior(k3)- Plane Stress with thickness > ok > Close.

Step 2(b): Define Geometric Properties

- Preprocessor > Real Constants > Add/Edit/Delete > Add > thickness- 20 mm >ok

Step 2(c) : Element Material Properties

- Preprocessor >Material Props > Material Models > Double click on Structural > Linear > Elastic > Isotropic > EX(Young's Modulus)- 2e5 >PRXY(Poissons Ratio)-0.3 > Ok > Close

Step 2(d): To Model Element

For Rectangle 1 :

- Preprocessor > Modeling > Create >Ares >Rectangle > by 2 Corners

WP X	**0**
WP Y	0
WIDTH	80
HEIGHT	100

For Big Circle 1:

- Preprocessor > Modeling > Create >Areas > Circle > Solid circle

WP X	**80**
WP Y	50
RADIUS	50

For small Circles 2 &3 :

- Preprocessor > Modeling > Create >Ares >Circle > Solid circle

Parameter	**Circle 2**	**Circle3**
WP X	**0**	**0**
WP Y	20	80
RADIUS	20	20

For Rectangle 2 :

- Preprocessor > Modeling > Create >Ares >Rectangle > by 2 Corners

WP X	**-20**
WP Y	20
WIDTH	20
HEIGHT	60

- Preprocessor > Modeling > Operate > Booleans > add > areas > Pick all > Ok

For Bolt Holes:

- Preprocessor > Create > Areas > Circle > Solid Circle

Parameter	**Circle 1**	**Circle2**	**Circle 3**
WP X	**80**	**0**	**0**
WP Y	50	20	80
RADIUS	30	10	10

- Preprocessor > Modeling > Operate > Booleans > Subtract > areas > Select total Rectangle > Ok in New Openend Box > Ok > Ok > Double click on bolt circles > Ok >Close

Step 2(e) : To Mesh Element

- Preprocessor >Meshing > Size Cntrls> Manual Size> areas > all areas > element edge length :10 > Ok
- Preprocessor >Meshing > Mesh >areas > free > Pick All > Ok

- Utility Menu > PlotCtrls > Numbering > Node Numbers –ON > OK

Step 2(e): Assigning Loads and Constraints on the elemnt

- Preprocessor >Loads >Analysis type > New Analysis > Static >Ok
- Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On nodes > select circle option in window > select small top circle > Apply > all DOF > Ok > close
- Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On nodes > select circle option in window > select small bottom circle > Apply > all DOF > Ok > close

- Preprocessor > Loads > Define Loads > Apply > Structural > force/ Moment > On Key points > Click on bottom side of big circle > Select FY : -1000 > Ok

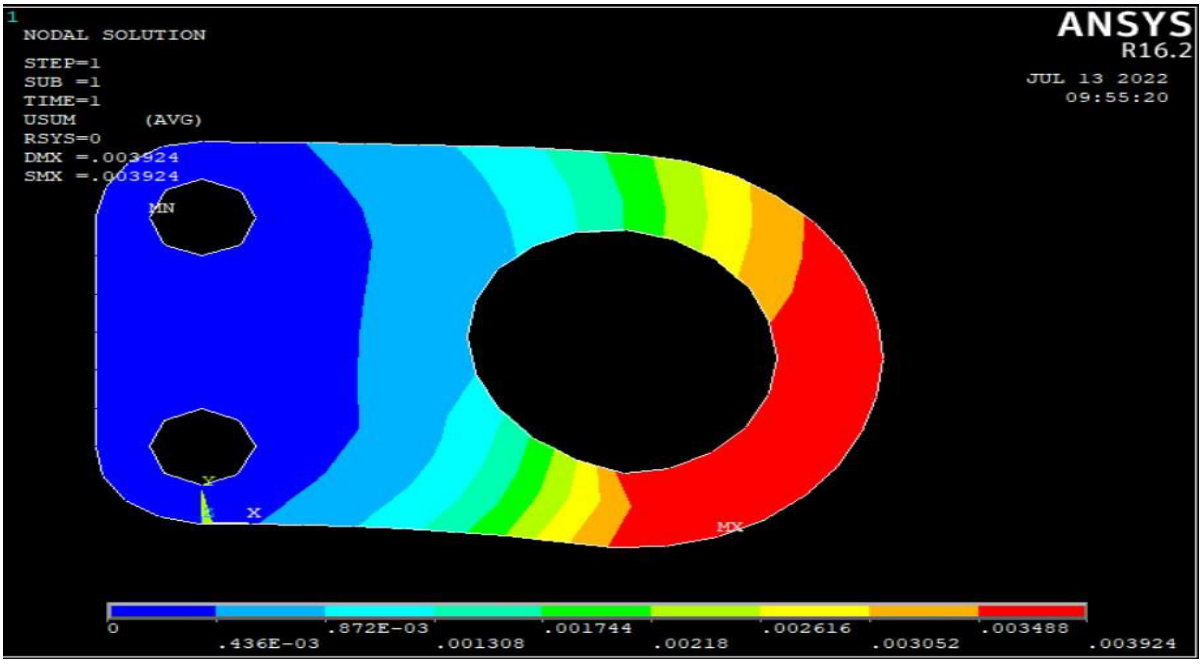
STEP 3: Solution

Solution >Solve > Current LS> Ok

STEP 4: General Postprocessor

Step 4(a): To View Nodal Deflection in Element

- General Postprocessor > Plot results > Contour Plot > Nodal Solution > Nodal Solution > DOF solution > Displacement Vector sum
- General Postprocessor > List results > Nodal Solution > Nodal Solution> DOF solution > Displacement Vector sum



PRNSOL Command

File

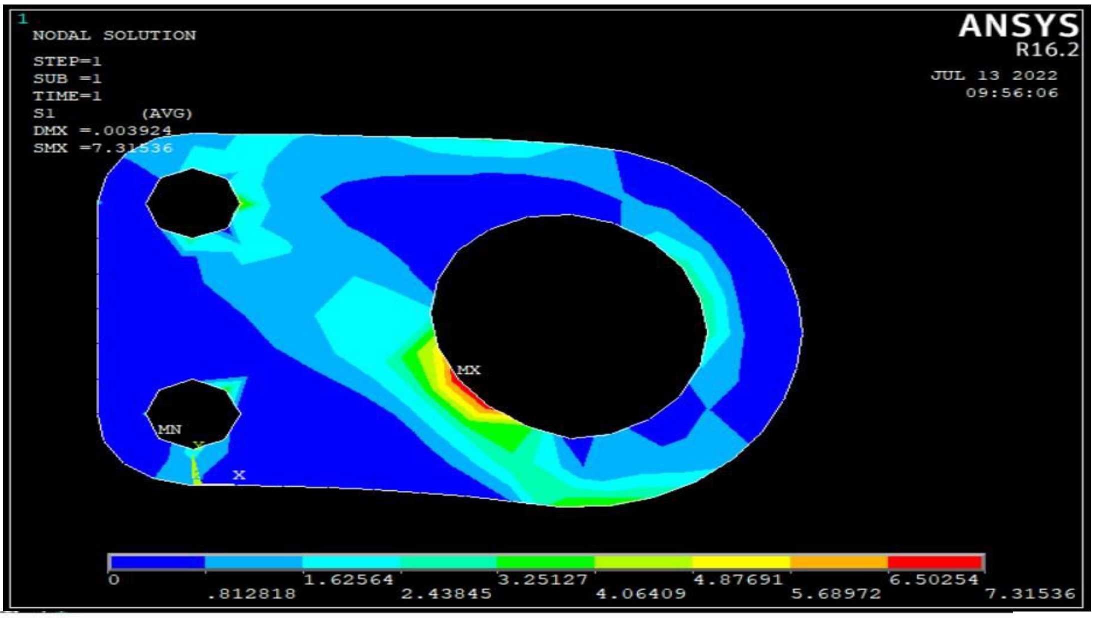
NODE	UX	UY	UZ	USUM
408	0.10342E-03	-0.66500E-03	0.0000	0.67299E-03
409	0.11319E-04	-0.94590E-03	0.0000	0.94597E-03
410	-0.25216E-03	-0.14250E-02	0.0000	0.14472E-02
411	0.19146E-03	-0.28830E-03	0.0000	0.34609E-03
412	-0.20753E-03	-0.74527E-03	0.0000	0.77363E-03
413	-0.40456E-03	-0.10826E-02	0.0000	0.11557E-02
414	0.23738E-03	-0.39252E-03	0.0000	0.45872E-03
415	0.26312E-03	-0.63890E-03	0.0000	0.69096E-03
416	0.12607E-03	-0.34919E-03	0.0000	0.37125E-03
417	0.10172E-03	-0.24757E-03	0.0000	0.26766E-03
418	0.14197E-03	-0.58825E-03	0.0000	0.60514E-03
419	-0.10096E-03	-0.51986E-03	0.0000	0.52957E-03
420	0.66065E-04	-0.36348E-03	0.0000	0.36944E-03
421	0.10074E-03	-0.52298E-03	0.0000	0.53259E-03
422	0.98891E-04	-0.44320E-03	0.0000	0.45410E-03
423	0.38067E-04	-0.15611E-03	0.0000	0.16068E-03
424	0.90875E-04	-0.29406E-03	0.0000	0.30778E-03
425	0.91292E-04	-0.35682E-03	0.0000	0.36831E-03

MAXIMUM ABSOLUTE VALUES

NODE	91	7	0	6
VALUE	-0.14238E-02	-0.38453E-02	0.0000	0.39244E-02

Step 4(b): To View Principal Stresses in Element

- General Postprocessor > Plot results > Contour Plot > Nodal Solution > Nodal Solution > Stress> 1St Principal Stresses > Ok
- General Postprocessor > List results > Nodal Solution > Nodal Solution> Stress> 1St Principal Stresses > Ok



-

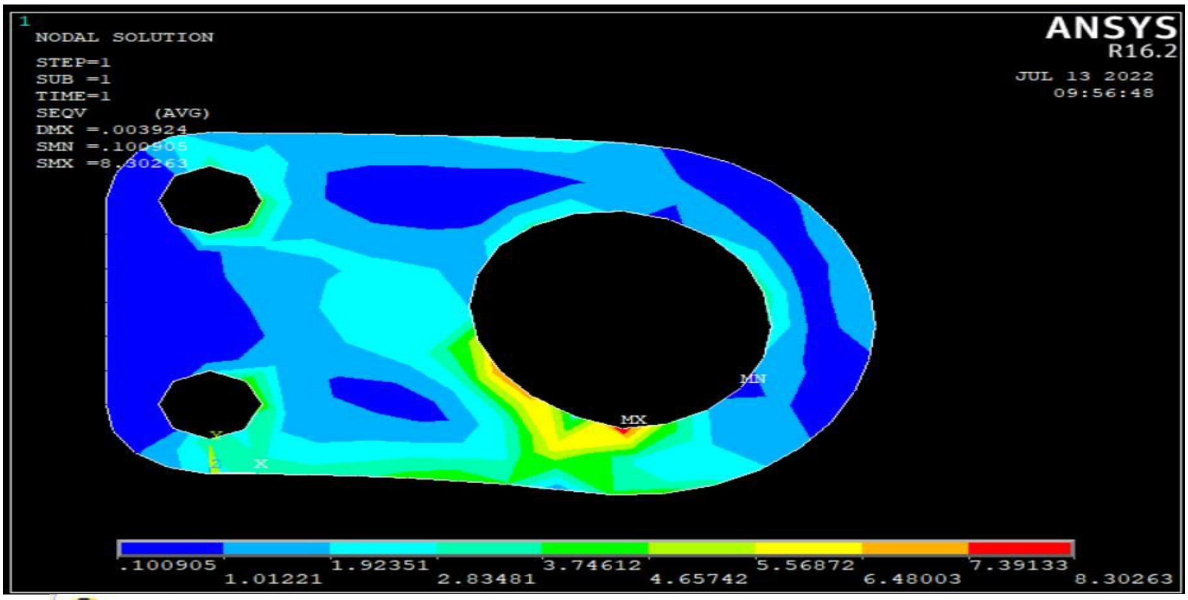
PRNSOL Command

File

```
     419   0.34100        0.0000      -0.66926E-01  0.40792        0.37892
     420    1.3413        0.0000      -0.86243       2.2038         1.9235

 ***** POST1 NODAL STRESS LISTING *****
 PowerGraphics Is Currently Enabled

 LOAD STEP=      1   SUBSTEP=      1
  TIME=    1.0000        LOAD CASE=   0
 NODAL RESULTS ARE FOR MATERIAL    1

   NODE     S1           S2           S3           SINT         SEQU
    421   0.94960       0.0000      -0.80583       1.7554        1.5219
    422    1.4535       0.0000      -0.86889       2.3224        2.0324
    423   0.54112       0.0000      -0.73464       1.2758        1.1091
    424    1.3806       0.0000      -0.75209       2.1327        1.8735
    425    1.6179       0.0000      -0.92042       2.5383        2.2257

 MINIMUM VALUES
 NODE          22          114          114          120          120
 VALUE     0.0000      -7.8607      -8.6833      0.10915      0.10091

 MAXIMUM VALUES
 NODE         127          131            1          114          114
 VALUE     7.3154       1.6732       0.0000       8.6833       8.3026
```

Step 4(b): To View Von Mises Stresses in Element

- General Postprocessor > Plot results > Contour Plot > Nodal Solution > Nodal Solution > Stress> Von Mises Stress> Ok
- General Postprocessor > List results > Contour Plot > Nodal Solution > Nodal Solution > Stress> Von Mises Stress> Ok



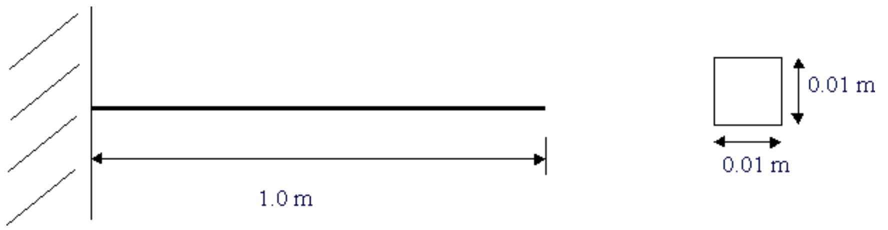
PRNSOL Command

File

```
    419  0.34100       0.0000      -0.66926E-01 0.40792      0.37892
    420   1.3413       0.0000      -0.86243      2.2038       1.9235

 ***** POST1 NODAL STRESS LISTING *****
 PowerGraphics Is Currently Enabled

 LOAD STEP=     1  SUBSTEP=     1
  TIME=    1.0000      LOAD CASE=   0
 NODAL RESULTS ARE FOR MATERIAL    1

   NODE     S1          S2          S3          SINT        SEQV
    421  0.94960       0.0000      -0.80583      1.7554       1.5219
    422   1.4535       0.0000      -0.86889      2.3224       2.0324
    423  0.54112       0.0000      -0.73464      1.2758       1.1091
    424   1.3806       0.0000      -0.75209      2.1327       1.8735
    425   1.6179       0.0000      -0.92042      2.5383       2.2257

 MINIMUM VALUES
 NODE         22         114         114         120         120
 VALUE    0.0000     -7.8607     -8.6833     0.10915     0.10091

 MAXIMUM VALUES
 NODE        127         131           1         114         114
 VALUE    7.3154      1.6732      0.0000      8.6833      8.3026
```

EXPERIMENT -4(a) MODAL ANALYSIS ON 2-D BEAM

AIM: Estimation of natural frequencies and mode shapes of 2-D Beam



$E=200*10^9$ N/m^2

Density=7800 kg/m^3

Frequency=0 to 10000Hz

Procedure:

The main steps to be involved are

1. Preferences
2. Pre Processor
3. Solution
4. General Post Processor

To Give Title for the Experiment and some Basic Requirements

- Utility menu bar > File > Change Title> Modal Analysis of 2D beam
- Utility menu bar > File > Save as> Give file name and Location
- Utility Menu > Plot > Replot (For quick visibility of Title and all)

STEP 1: Preferences

- Preferences > Structural>OK

STEP 2: Pre Processor

Step 2(a): Define the Type of Element

- Preprocessor >Element Type > Add/Edit/Delete >Add > BEAM > 2 Node 188 > Ok.

Step 2(b) : Element Material Properties

- Preprocessor >Material Props > Material Models > Double click on Structural > Linear > Elastic > Isotropic > EX(Young's Modulus)- **2.068e11** >PRXY(Poissons Ratio) **0.3** > Ok
- Preprocessor >Material Props > Material Models > Double click on Structural>Density > **7800** > ok > close

Step 2(c) : To Select type of Section

- Preprocessor Sections > Beam > Common Sections > enter B- 0.01 and H- 0.01 > ok

Step 2(d): To Model Element

- Preprocessor > Modeling > Create >Key points> In Active CS

Keypoint	Coordinates (x,y)
1	(0,0)
2	(1,0)

- Preprocessor > Modeling > Create > Lines > Lines > In Active Coord. > Select all key points to get Beam element

Step 2(e): To Mesh Truss Element

- Preprocessor >Meshing > Size Cntrls>ManualSize> Lines > All Lines > No of element divisions :10> Ok
- Preprocessor >Meshing > Mesh > Lines >Pick All > Ok
- Utility Menu > PlotCtrls > Numbering > Node Numbers –ON > OK

Step 2(f): Assigning Loads and Constraints on element

- Preprocessor > Loads >Analysis Type > New Analysis > Modal >Ok
- Preprocessor > Loads > Define Loads > Analysis Options > ok> Block Lanczos >No of modes to extract :5 > No of modes to Expand :5 > Ok > Start frequency : 0> End frequency: 10000> Ok
- Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Keypoint > Click on Keypoint 1 >Ok > All DOF > Ok

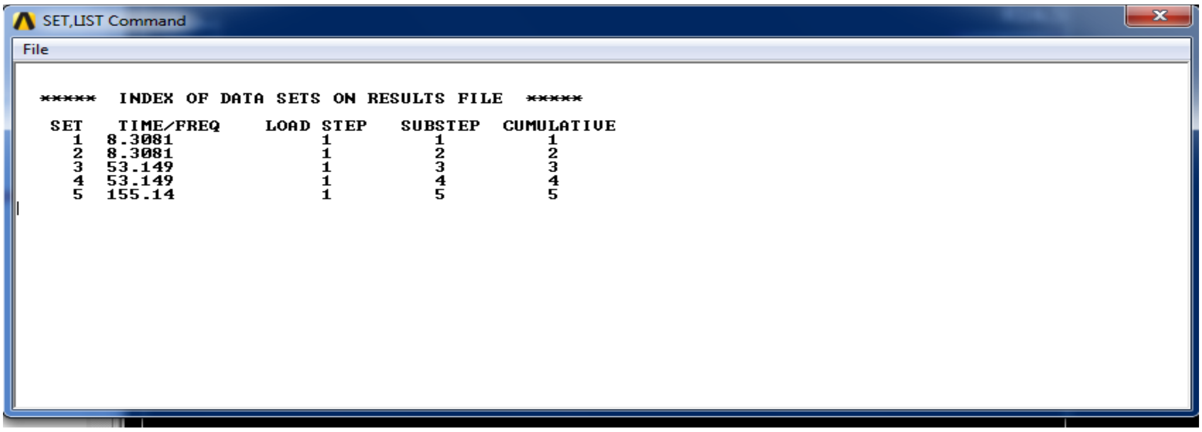
STEP 3: Solution

Step 3(a): Solution >Solve > Current LS> Ok

STEP 4: General Postprocessor

Step 4(a): To View frequencies

- General Postproc> Results Summary



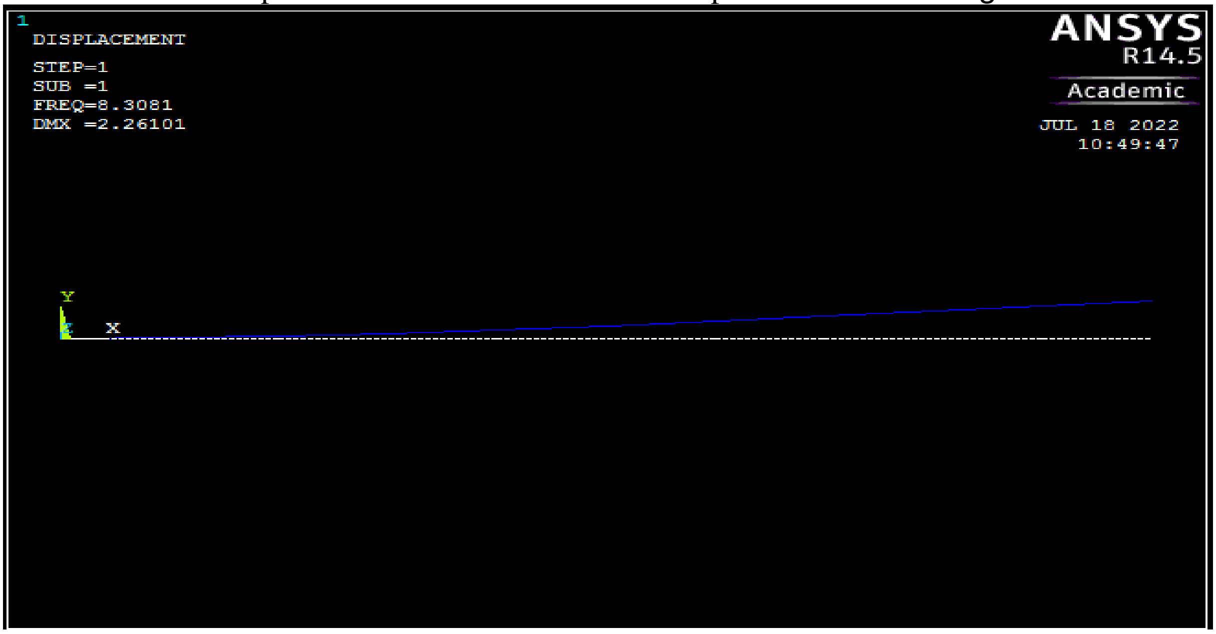
SET,LIST Command

File

***** INDEX OF DATA SETS ON RESULTS FILE *****				
SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	8.3081	1	1	1
2	8.3081	1	2	2
3	53.149	1	3	3
4	53.149	1	4	4
5	155.14	1	5	5

Step 4(b): To View First Mode shape

- General Postproc> Read Results > First Set
- General Postproc> Plot Results > Deformed shape >Def + undef edge

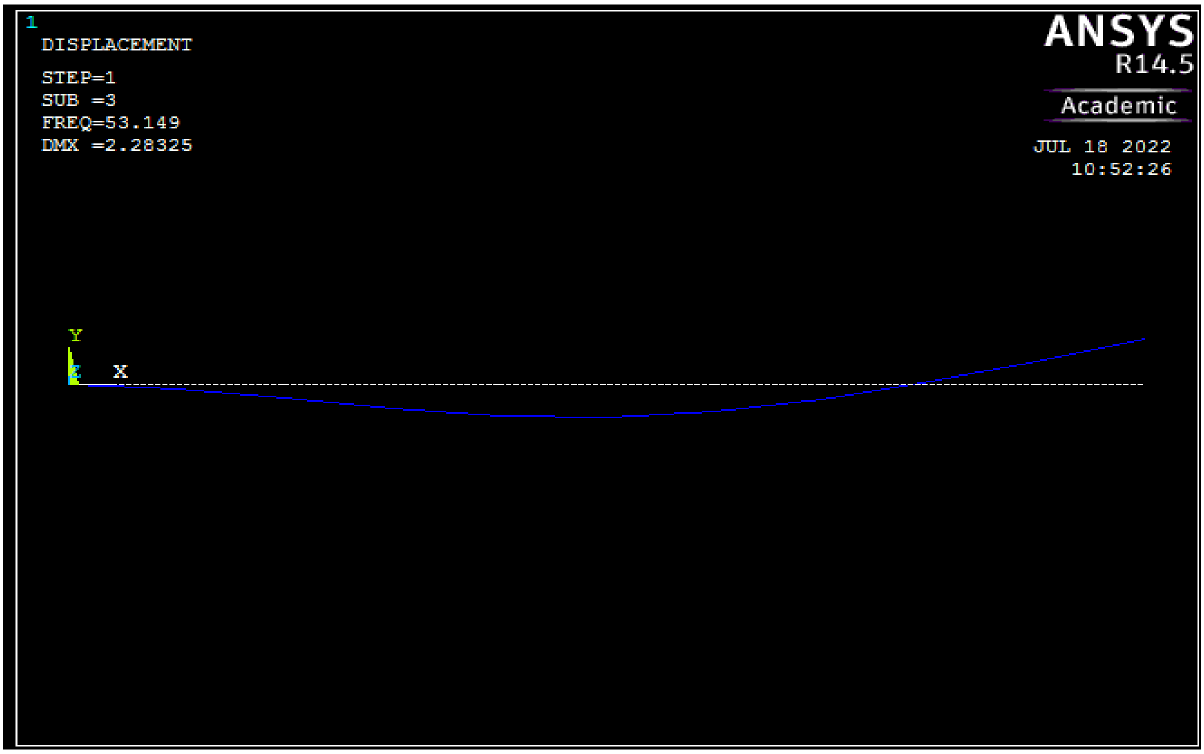


To View second Mode shape

- General Postproc> Read Results > Next Set
- General Postproc> Plot Results > Deformed shape >Def + undef edge

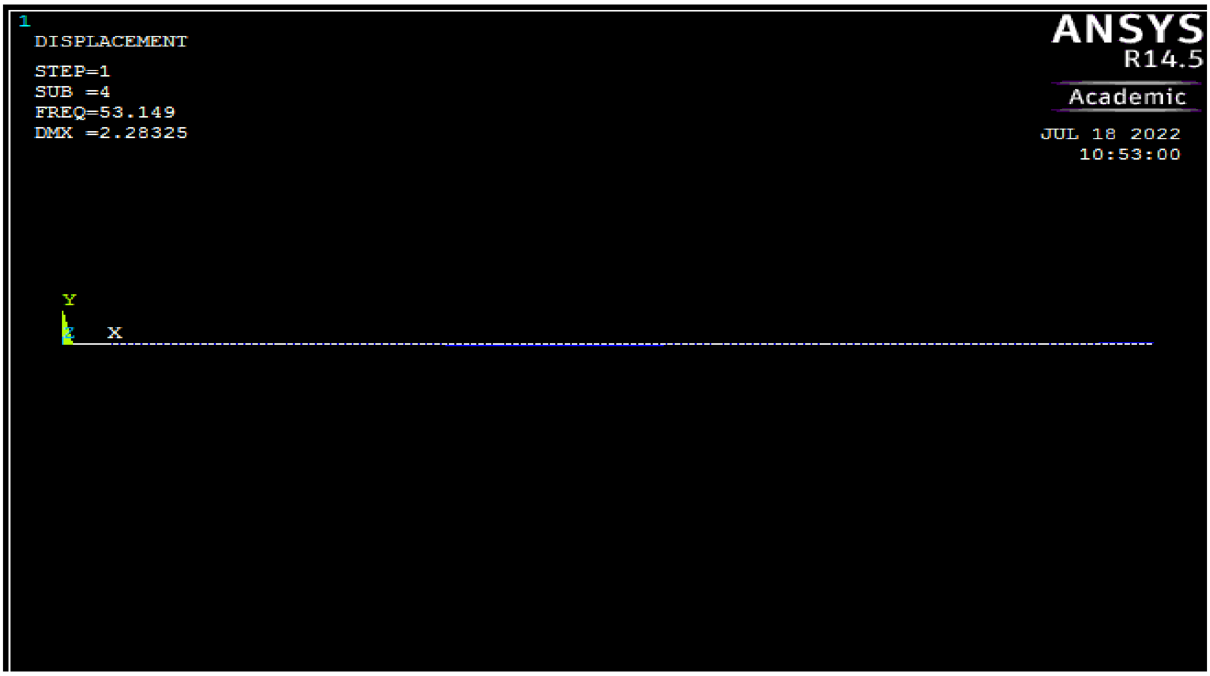
To View Third Mode shape

- General Postproc> Read Results > Next Set
- General Postproc> Plot Results > Deformed shape >Def + undef edge



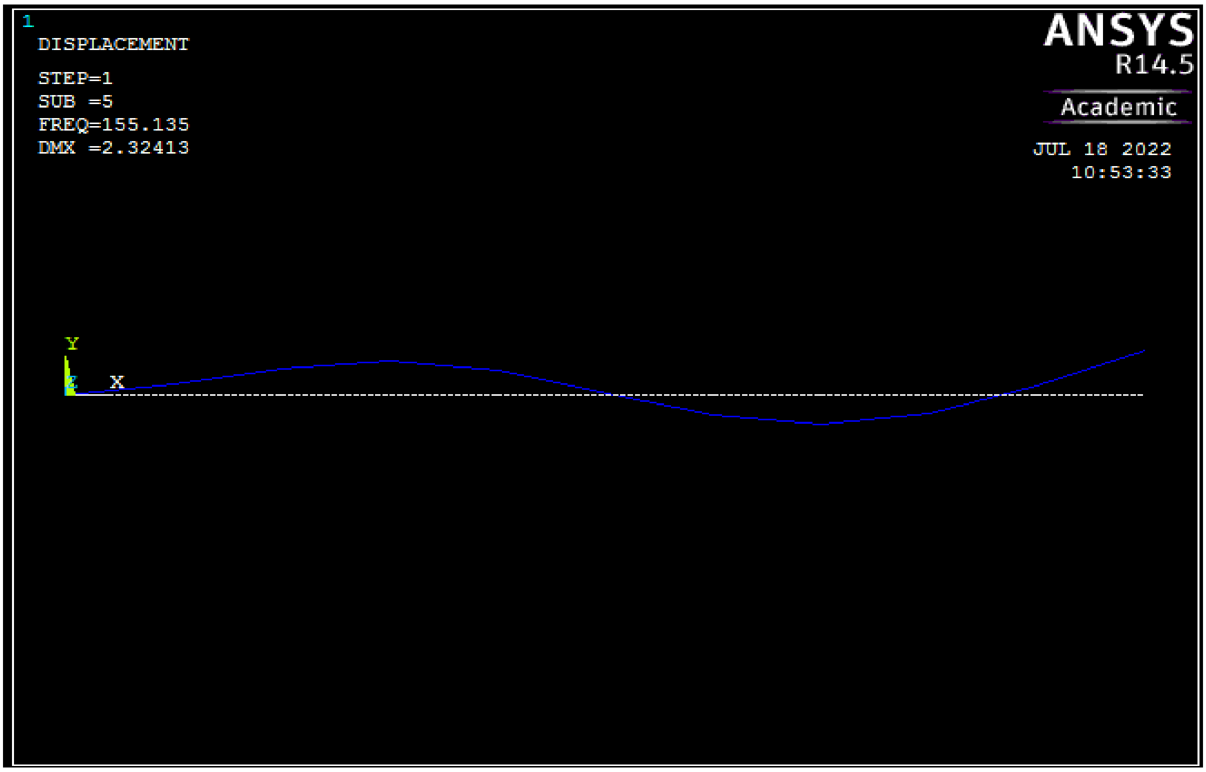
To View Fourth Mode shape

- General Postproc> Read Results > Next Set
- General Postproc> Plot Results > Deformed shape >Def + undef edge



To View last Mode shape

- General Postproc> Read Results > last Set
- General Postproc> Plot Results > Deformed shape >Def + undef edge

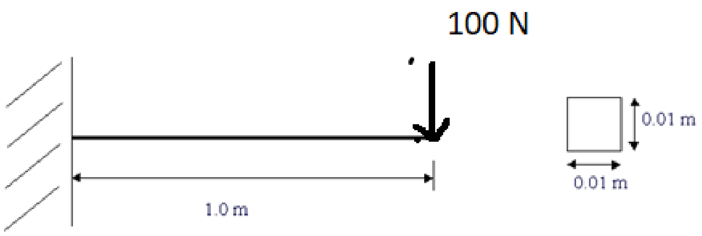


Result: Natural frequencies and mode shapes of 2-D Beam are done.

EXPERIMENT -4(b)

HARMONIC ANALYSIS ON 2-D BEAM

AIM: Estimation of natural frequencies & Harmonic response of 2D beam



E=200*10^9 N/m^2

Density=7800 kg/m^3

Force:100 N

Procedure:

The main steps to be involved are

1. Preferences
2. Pre Processor
3. Solution
4. General Post Processor

To Give Title for the Experiment and some Basic Requirements

- Utility menu bar > File > Change Title> Harmonic response of 2D beam
- Utility menu bar > File > Save as> Give file name and Location
- Utility Menu > Plot > Replot (For quick visibility of Title and all)

STEP 1: Preferences

- Preferences > Structural>OK

STEP 2: Pre Processor

Step 2(a): Define the Type of Element

- Preprocessor >Element Type > Add/Edit/Delete >Add > BEAM > 2 Node 188 > Ok.

Step 2(b): Element Material Properties

- Preprocessor >Material Props > Material Models > Double click on Structural > Linear > Elastic > Isotropic > EX(Young's Modulus)- **2e11** >PRXY(Poissons Ratio) **0.3** > Ok
- Preprocessor >Material Props > Material Models > Double click on Structural>Density > **7800** > ok > close

Step 2(c): To Select type of Section

- Preprocessor Sections > Beam > Common Sections > enter B- 0.01 and H- 0.01 > ok

Step 2(d): To Model Element

- Preprocessor > Modeling > Create >Key points> In Active CS

Keypoint	Coordinates (x,y)
1	(0,0)
2	(1,0)

- Preprocessor > Modeling > Create > Lines > Lines > In Active Coord. > Select all key points to get Beam element

Step 2(e): To Mesh Truss Element

- Preprocessor >Meshing > Size Cntrls>ManualSize> Lines > All Lines > No of element divisions :10> Ok
- Preprocessor >Meshing > Mesh > Lines >Pick All > Ok
- Utility Menu > PlotCtrls > Numbering > Node Numbers –ON > OK

Step 2(f): Assigning Loads and Constraints on element

- Preprocessor > Loads >Analysis Type > New Analysis > Harmonic >Ok
- Preprocessor > Loads > Define Loads > Analysis Options > Solution Method : Full >Dof Printout format:Real+imaginary> ok> Equation solver: sparse solver > Tolerance: 1e-008>ok
- Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Keypoint > Click on Keypoint 1 >Ok > All DOF > Ok
- Preprocessor > Loads > Define Loads > Apply > Structural > Force/Moment > On Keypoint > Click on Keypoint 2 > Ok > Select FY (-100) > Ok
- Preprocessor > Loads > Load Step Opts > Time/Frequency >Freq and Substps>**harmonic frequency range: 0 to 100 > number of sub steps :100 > Ramped > ok**

STEP 3: Solution

Step 3(a): Solution >Solve > Current LS> Ok

STEP 4: TimeHist Processor

Step 4(a): To View Harmo nic Response

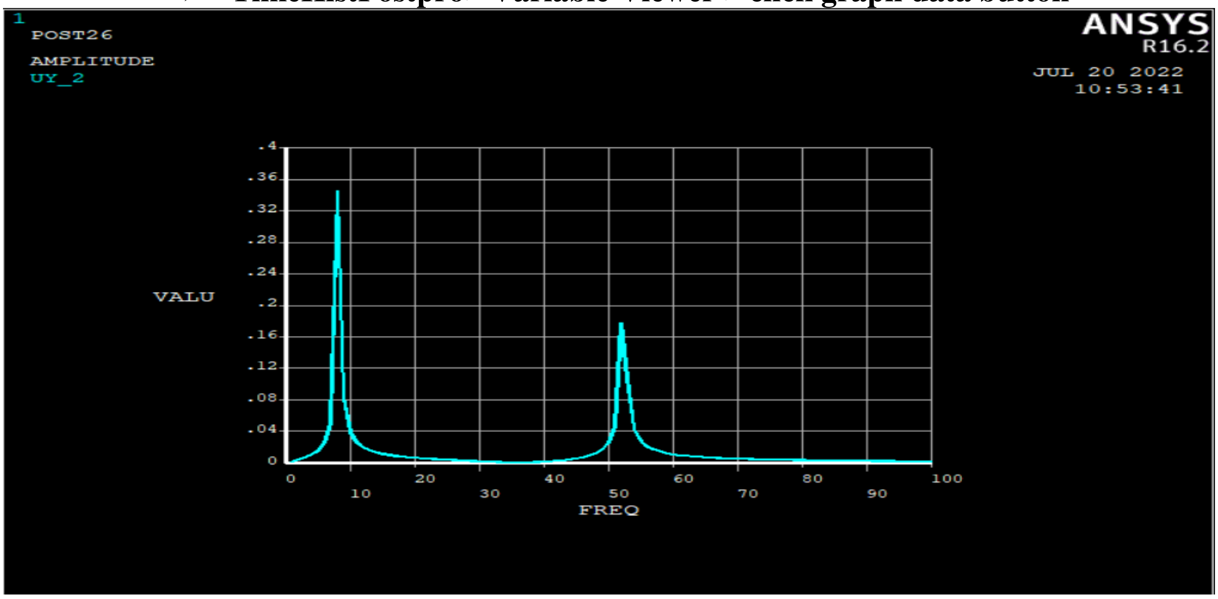
- **TimeHistPostpro> Variable Viewer > select add(+) > nodal solution > DOF solution > Y-component of displacement > ok > enter node number 2 in Box > Ok**

- **TimeHistPostpro> Variable Viewer > click list data button**

```
PRVAR   Command
File

        ***** ANSYS POST26 VARIABLE LISTING *****

 FREQ                2 UY
                  UY_2
               AMPLITUDE      PHASE
 1.0000        0.202552E-02   180.000
 2.0000        0.423866E-02   180.000
 3.0000        0.689168E-02   180.000
 4.0000        0.104194E-01   180.000
 5.0000        0.157545E-01   180.000
 6.0000        0.254918E-01   180.000
 7.0000        0.509196E-01   180.000
 8.0000        0.345757       180.000
 9.0000        0.831155E-01   0.00000
 10.000        0.388237E-01   0.00000
 11.000        0.258413E-01   0.00000
 12.000        0.195533E-01   0.00000
 13.000        0.157957E-01   0.00000
 14.000        0.132679E-01   0.00000
 15.000        0.114318E-01   0.00000
 16.000        0.100238E-01   0.00000
 17.000        0.889971E-02   0.00000
 18.000        0.797346E-02   0.00000
```

```
PRVAR   Command
File
 84.000        0.325224E-02   0.00000
 85.000        0.314741E-02   0.00000
 86.000        0.304645E-02   0.00000
 87.000        0.294900E-02   0.00000
 88.000        0.285473E-02   0.00000
 89.000        0.276334E-02   0.00000
 90.000        0.267455E-02   0.00000
 91.000        0.258812E-02   0.00000
 92.000        0.250381E-02   0.00000
 93.000        0.242142E-02   0.00000
 94.000        0.234075E-02   0.00000
 95.000        0.226160E-02   0.00000
        ***** ANSYS POST26 VARIABLE LISTING *****

 FREQ                2 UY
                  UY_2
               AMPLITUDE      PHASE
 96.000        0.218382E-02   0.00000
 97.000        0.210723E-02   0.00000
 98.000        0.203167E-02   0.00000
 99.000        0.195701E-02   0.00000
 100.00        0.188308E-02   0.00000
```

- **TimeHistPostpro> Variable Viewer > click graph data button**



- Utility Menu >PlotCtrls> Style > Graphs > Modify Axis
- Thickness of axes: single > Number of y-Axis : single y-Axis > Y-axis scale :

Logarithmic > Ok
- Utility Menu > Plot >Replot

Result: Natural frequencies & Harmonic response of 2D beam are observed.

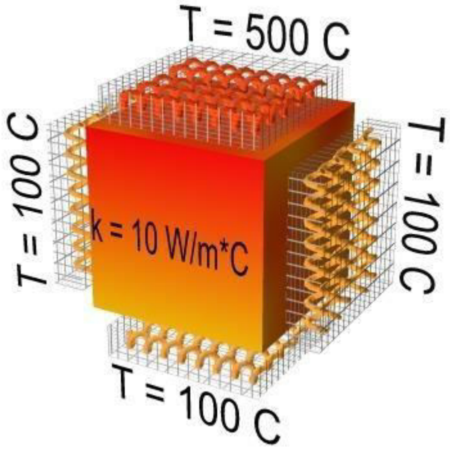
EXPERIMENT -5

STEADY STATE HEAT TRANSFER ANALYSIS

AIM: To perform Steady state heat transfer Analysis of plane and Axisymmetric components

Problem Description:

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:

The main steps to be involved are

1. Preferences
2. Pre Processor
3. Solution
4. General Post Processor

To Give Title for the Experiment and some Basic Requirements

- Utility menu bar > File > Change Title> steady state heat transfer analysis
- Utility menu bar > File > Save as> Give file name and Location
- Utility Menu > Plot > Replot (For quick visibility of Title and all)

STEP 1: Preferences

- Preferences > Thermal >OK

STEP 2: Pre Processor

Step 2(a): Define the Type of Element

- Preprocessor >Element Type > Add/Edit/Delete >Add > Solid >Quad 4 Node 55 > Ok.

Step 2(b): Element Material Properties

- Preprocessor >Material Props > Material Models > Thermal > Conductivity > Isotropic > KXX = 10(Thermal conductivity)

Step 2(c): Create geometry

- Preprocessor > Modeling > Create > Areas > Rectangle > By 2 Corners > X=0, Y=0, Width=1,Height=1.

Step 2(d): To Mesh Truss Element

- Preprocessor > Meshing > Size Cntrls>ManualSize> Areas > All Areas > 0.05
- Preprocessor >Meshing > Mesh > Areas> Free >Pick All > Ok
- Utility Menu > PlotCtrls > Numbering > Node Numbers –ON > OK

Step 2(f): Assigning Loads and Constraints on element

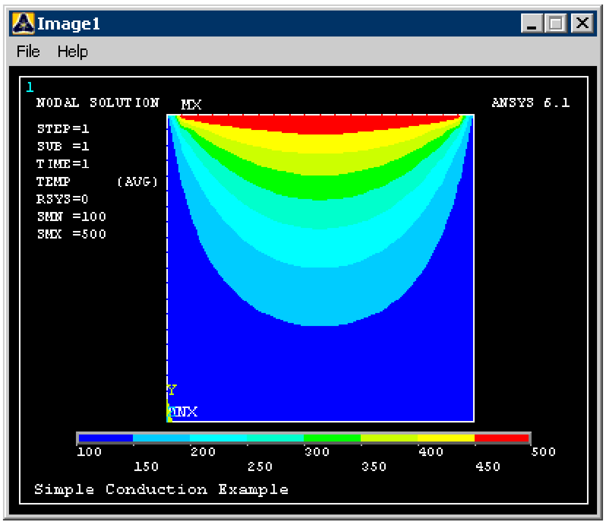
- Preprocessor > Loads >Analysis Type > New Analysis > Steady-State >Ok
- Preprocessor > Loads > Define Loads > Apply > Thermal > Temperature > On Nodes >Click the Box option and draw a box around the nodes on the top line>ok>Temp: 500>ok
- Preprocessor > Loads > Define Loads > Apply > Thermal > Temperature > On Nodes >Click the Box option and draw a box around the nodes on the left line>ok>Temp: 100>ok
- Preprocessor > Loads > Define Loads > Apply > Thermal > Temperature > On Nodes >Click the Box option and draw a box around the nodes on the right line>ok>Temp: 100>ok
- Preprocessor > Loads > Define Loads > Apply > Thermal > Temperature > On Nodes > Click the Box option and draw a box around the nodes on the bottom line>ok>Temp:100>ok

STEP 3: Solution

Step 3(a): Solution >Solve > Current LS> Ok

STEP 4: General Postprocessor

- General Post proc > Plot Results > Contour Plot > Nodal Solu > DOF solution > Nodal Temperature



Result: Steady state heat transfer Analysis on Axisymmetric component is done.

DEFINITION OF CAD/CAM/CAE

Computer Aided Design – CAD

CAD is technology concerned with using computer systems to assist in the creation, modification, analysis, and optimization of a design. Any computer program that embodies computer graphics and an application program facilitating engineering functions in design process can be classified as CAD software.

The most basic role of CAD is to define the geometry of design – a mechanical part, a product assembly, an architectural structure, an electronic circuit, a building layout, etc. The greatest benefits of CAD systems are that they can save considerable time and reduce errors caused by otherwise having to redefine the geometry of the design from scratch every time it is needed.

Computer Aided Manufacturing – CAM

CAM technology involves computer systems that plan, manage, and control the manufacturing operations through computer interface with the plant's production resources.

One of the most important areas of CAM is numerical control (NC). This is the technique of using programmed instructions to control a machine tool, which cuts, mills, grinds, punches or turns raw stock into a finished part. Another significant CAM function is in the programming of robots. Process planning is also a target of computer automation.

Computer Aided Engineering – CAE

CAE technology uses a computer system to analyze the functions of a CAD-created product, allowing designers to simulate and study how the product will behave so that the design can be refined and optimized.

CAE tools are available for a number of different types of analyses. For example, kinematic analysis programs can be used to determine motion paths and linkage velocities in mechanisms. Dynamic analysis programs can be used to determine loads and displacements in complex assemblies such as automobiles. One of the most popular methods of analyses is using a Finite Element Method (FEM). This approach can be used to determine stress, deformation, heat transfer, magnetic field distribution, fluid flow, and other continuous field problems that are often too tough to solve with any other approach.

CNC PROGRAMMING

Basic CNC input data:

It is used to identify each block with the CNC program and provides a means by which CNC commands may be rapidly located. Some control units requires that sequence numbers be input in ascending order where has other system allow any three digit numbers appear after symbol N eg: N05 Y25 Z0

Coordinate function:

The coordinates of the tooltip are programmed for generating a given component geometry. The coordinates are specified by using word addresses X,Y,Z,U,V,W etc..

Feed function:

The feed state for slider displacement or spindle field rate is expressed in mm/min and a threedigit number prefixed by a letter F that indicates feed rate.

Once the feed rate is programmed in a block it remains enforce in all the subsequent blocks till it is replaced by another F value.

Speed function:

The spindle is expressed in Rev/minute and is a three digit number prefixed by letter for example S1000 indicates the spindle speed is 1000 rpm

Tool function:

The tool function is used in conjunction with the miscellaneous function for tool change (M06) and has a means of addressing the new tool

Preparatory functions:

The preparatory functions are represented by a two digit number prefixed by the letter G. The purpose of the preparatory function is to command the machine tool to perform function represented by the selected code number.

G code	Description
G00	Rapid traverse
G01	Linear interpolation
G02	Circular interpolation CW
G03	Circular interpolation CCW
G04	Dwell
G17	X Y plane selection
G18	Z X plane selection
G19	Y Z plane selection
G28	Return to reference position

G30	2nd, 3rd and 4th reference position return
G40	Cutter compensation cancel
G41	Cutter compensation left
G42	Cutter compensation right
G43	Tool length compensation + direction
G44	Tool length compensation – direction
G49	Tool length compensation cancel
G53	Machine coordinate system selection
G54	Workpiece coordinate system 1 selection
G55	Workpiece coordinate system 2 selection
G56	Workpiece coordinate system 3 selection
G57	Workpiece coordinate system 4 selection
G58	Workpiece coordinate system 5 selection
G59	Workpiece coordinate system 6 selection
G68	Coordinate rotation
G69	Coordinate rotation cancel
G73	Peck drilling cycle
G74	Left-spiral cutting circle
G76	Fine boring cycle
G80	Canned cycle cancel
G81	Drilling cycle, spot boring cycle
G82	Drilling cycle or counter boring cycle
G83	Peck drilling cycle
G84	Tapping cycle
G85	Boring cycle
G86	Boring cycle
G87	Back boring cycle
G88	Boring cycle
G89	Boring cycle
G90	Absolute command
G91	Increment command
G92	Setting for work coordinate system or clamp at maximum spindle speed
G98	Return to initial point in canned cycle

G99	Return to R point in canned cycle

Miscellaneous functions:

Miscellaneous functions involve actions that are necessary for machining i.e (spindle on/off and coolant on/off). These are used to designate a particular mode of operation for a CNC machine tool.

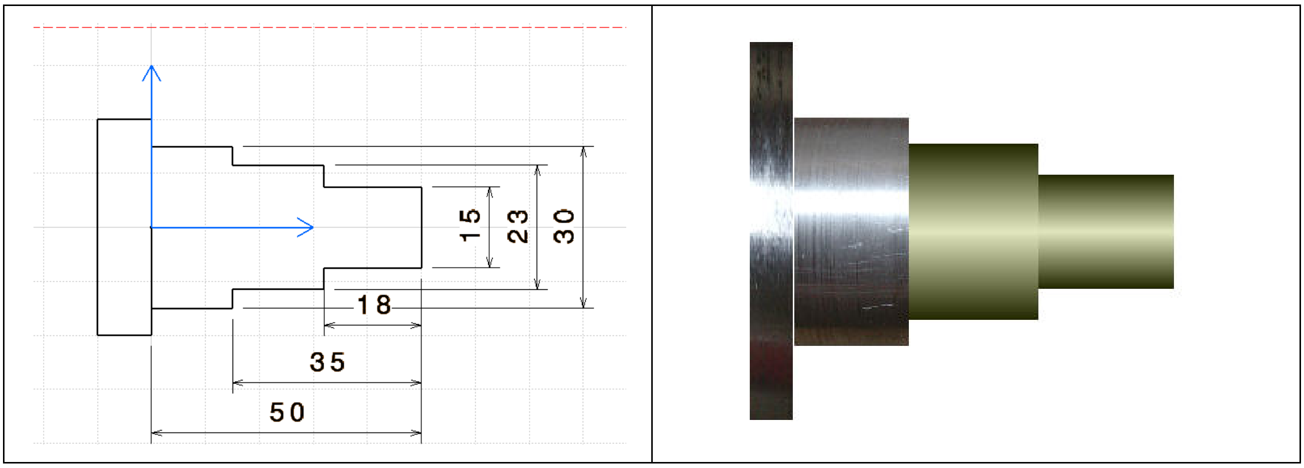
M code	Description
M00	Program stop
M01	Optional program stop
M02	End of program
M03	Spindle start forward CW
M04	Spindle start reverse CCW
M05	Spindle stop
M06	Tool change
M07	Coolant ON – Mist coolant/Coolant thru spindle
M08	Coolant ON – Flood coolant
M09	Coolant OFF
M19	Spindle orientation
M28	Return to origin
M29	Rigid tap
M30	End of program (Reset)
M41	Low gear select
M42	High gear select
M94	Cancel mirrorimage
M95	Mirrorimage of X axis
M96	Mirrorimage of Y axis
M98	Subprogram call
M99	End of subprogram

Program number:

The symbol used for the program number is 0 or: followed by number. For example 0123 or :123 the program does not interfere with the execution of a CNC program.

Exp:-1 **Step Turning** (CNC TURNING)

Aim: - To write a program to simulate **step turning** operation on a work piece as shown in the figure on a TURNING MACHINE. (Length-50mm, diameter -30mm)



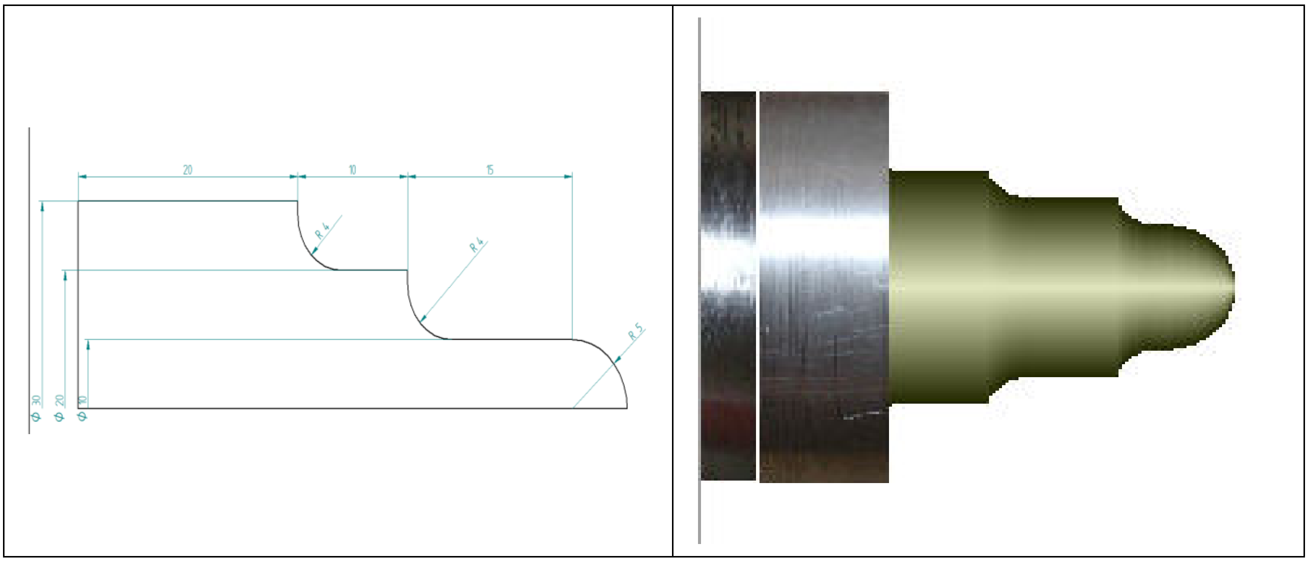
Code:

```
G21 G98
G28 U0 W0
M06 T02
M03 S1500
G00 X30 Z2
G90 X29 Z-35 F100
X28
X27
X26
X25
X24
X23
X22 Z-18
X21
X20
X19
X18
X17
X16
X15
G28 U0 W0
M05
M30
```

Result: - Simulation for step turning operation on a TURNING MACHINE completed

Exp:-2 **Circular Interpolation** (CNC TURNING)

Aim: - To write a program to simulate required **circular interpolation** operation on a work piece as shown in the figure on a TURNING MACHINE. (Length-50mm, diameter -30mm)



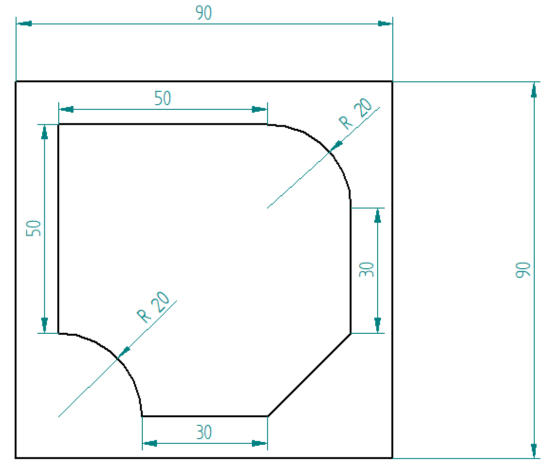
Code:

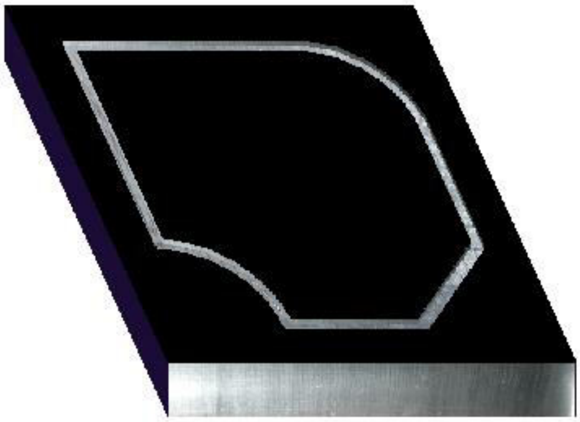
```
G21 G98
G28 U0 W0
M06 T02
M03 S1500
G00 X30 Z3
G71 U0.5 R0.5
G71 P1Q2 U0.1 W0.1 F100
N1 G01 X0 Z0
G03 X16 Z-8 R8
G02 X20 Z-12 R4
G01 X23
G01 Z-22
G02 X30 Z-26 R4
N2 G01 Z-36
G70 P1 Q2
G28 U0 W0
M05
M30
```

Result: - Simulation for circular interpolation operation on a TURNING MACHINE completed

Exp:-3 Profile (CNC MILLING)

Aim: - Write a Milling program to simulate & machine a Following-Section on top face of a billet of size 90*90*12 mm.





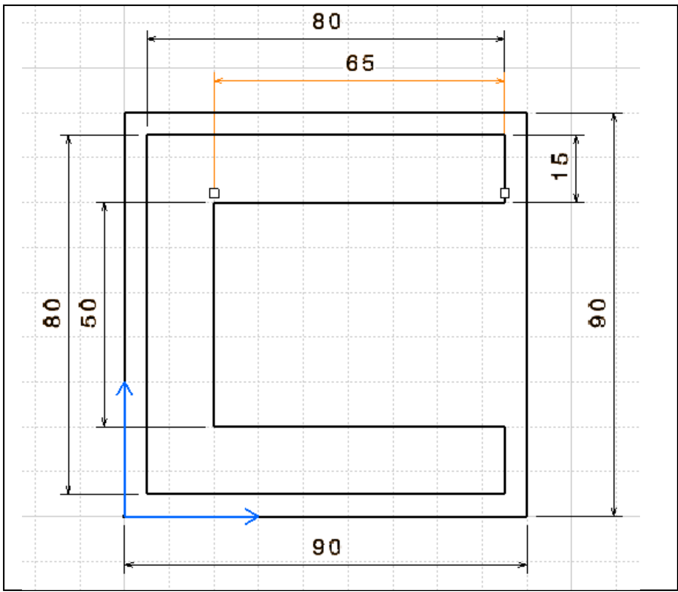
Code:

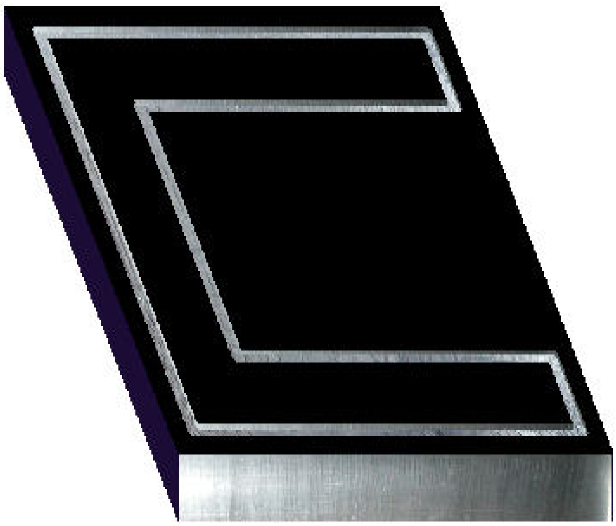
```
N00 T02 D2 M06
N01 G00 X0 Y0 Z2
N02 M03 S500
N03 G01 X30 Y10
N04 G01 Z-1 F50
N05 G01 X60 Y10
N06 G01 X80 Y30
N07 G01 X80 Y60
N08 G03 X60 Y80 R20 F30
N09 G01 X10 Y80
N10G01 X10 Y30
N11 G02 X30 Y10 R20 F30
N12 GO1 Z10
N13 G28 X0 Y0
N13 M05
N14 M30
```

Result: - For a closed section Simulation and machining completed on milling machine.

Exp:-4 **C-Section** (CNC MILLING)

Aim: - Write a Milling program to simulate & machine a C-Section on top face of a billet of size 90*90*12 mm.





Code:

```
N10 T02 D2 M06
N20 M03 S1000
N30 G00 Z100
N40 X5 Y5
N50 G01 Z-1 F30
N60 X85 Y5
N70 X85 Y20
N80 X20 Y20
N90 X20 Y70
N100 X85 Y70
N110 X85 Y85
N120 X5 Y85
N130 X5 Y5
N140 G40 G80
N150 G00 X5 Y5 Z2
N160 G28 X0 Y0
N170 M05
N180 M30
```

Result: - For a C- section Simulation and machining completed on milling machine.

Exp:-5

Simulation of Fused Deposition Modelling (FDM) process

Aim: To simulate the Fused Deposition Modelling (FDM) process.

Theory:

In FDM process, thermoplastic material in the form of filament is unwound from a spool and is fed into a extruder assembly where it is melted in liquefier and this semi-liquid material is laid down on the build platform by extrusion process through a nozzle according to computer-controlled paths, where it cools and solidifies.

In this manner a cross section of an object is 3d printed each layer at a time. The solid portion of the incoming filament serves as a “plunger” to extrude the material through a nozzle. The extrusion nozzle or the 3d printed object (or both) are moved along 3 axis by a computer-controlled mechanism.

Stepper motors are employed for all these movements, as well as for pushing the filament into the extruder. Layer height determines the quality of the 3D print.

Some FDM 3D printers can have two or more print heads that can print in multiple different colors and use support for overhanging areas of a complex 3D print.

Medical sciences has made many breakthroughs with the support of FDM technology, today we are capable of replicating functional human organs and implant artificially.

Procedure:

1) Click on 'Base'. Base and Build Platform will be displayed on the left side of the screen.
2) Click on 'Extruder' then extruder and extruder nozzle will be displayed.
3) Click on 'Material Spool' then material spool will be shown.
4) Click on 'Filament' then filament will be added.
5) Click on 'Start Process' to begin the process.
6) After the process is complete click on 'Stop Process' to view product generated using FDM process.

Applications:

1) Education
2) Modelling & Prototyping
3) Medical
4) Space Technology

Advantages:

1) FDM machines are safe, reliable, easy to use, and office friendly.
2) Minimum material wastage.
3) Varieties of engineering polymers are available commercially with different strengths and mechanical properties.

Limitations:

1) FDM material needs to be made in filament form of required diameter.

2) As the parts are built by depositing extruded rasters, the mechanical properties of the parts are not the same in all directions.
3) To change extruder nozzle, it is require to Disassemble complete extruder assembly. Thus, it is very difficult to change extruder nozzle in FDM.

Result: To studied the Fused Deposition Modelling (FDM) 3d printing process.