K.L.N. College of Engineering

(An Autonomous Institution Affiliated to Anna University, Chennai)

l by National Assessment and Accreditation Council (NAAC)

Pottapalayam – 630612.(11 km From Madurai City) TamilNadu, India.

Department of Mechanical Engineering

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

Regulations – 2020

Even Semester

20ME6L1
Computer Aided Simulation and Analysis Laboratory

Laboratory Manual

Lab In charge

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

Prepared by

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

Approved by

Dr. P. Udhayakumar
HOD / Mech. Engg.

DEPARTMENT OF MECHANICAL ENGINEERING

VISION

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

MISSION

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

PROGRAM EDUCATIONAL OBJECTIVES (PEOs)

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

PROGRAM SPECIFIC OUTCOMES (PSOs)

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

K.L.N. College of Engineering

(An Autonomous Institution Affiliated to Anna University, Chennai)

l by National Assessment and Accreditation Council (NAAC)

Pottapalayam – 630612.(11 km From Madurai City) TamilNadu, India.

Department of Mechanical Engineering

Accredited by NBA, New Delhi

Approved Research Center by Anna University, Chennai

Approved Nodal Center for e – YANTRA Lab

Regulations – 2020

Even Semester

20ME6L1
Computer Aided Simulation and Analysis Laboratory

Laboratory Manual

Lab In charge

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

Prepared by	Approved by
Mr. E.V. Ganesh Babu, Asst. Prof. / Mech.	*Dr. P. Udhayakumar* *HOD / Mech. Engg.*

GENERAL INSTRUCTIONS FOR LABORATORY CLASSES

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

Name : ……………………………………………… Batch…………………………………………….
Roll No.:…………… Year ……..… Semester……..… Section : ……….....

Index

S. No.	Date	Name of the Experiment	Page	Marks	Staff Signature
1.					
2.					
3.					
4.					
5.					
6.					
7.					
8.					
9.					
10.					
11.					
12.					
13.					
14.					
15.					
16.					
17.					
18.					
19.					

S. No.	Date	Name of the Experiment	Page	Marks	Staff Signature
20.					
21.					
22.					
23.					
24.					
25.					
26.					
27.					
28.					
29.					
30.					
31.					
32.					
33.					
34.					
35.					
36.					
37.					
38.					
39.					

Completed date:

Average Mark: **Staff - in – charge**

COMPUTER AIDED SIMULATION AND ANALYSIS LABORATORY

20ME6L1 **L T P C**
0 0 3 1.5

OBJECTIVES:

- To understand the applications of various software tools for analysis
- To understand geometric modeling in analysis software.
- To find the stress and other related parameters of bars, beams loaded with loading conditions.
- To derive the output from the analysis software.
- To solve real time problems using these tools..

PREREQUISITE:
Course Code: 20ME301, 20ME304, 20ME502
Course Name: Strength of Materials, Thermal engineering, Dynamics of Machinery

LIST OF EXPERIMENTS
1. 1D application problems like composite walls/beams
2. 2D application problems like flat plates, simple shells, cylinder
3. Stress analysis of axi – symmetric components.
4. Modal analysis (Beams).
5. 3D modeling of pulley.
6. 3D analysis of rotating shaft.
7. Nonlinear analysis using contact elements.
8. Thermo mechanical analysis of plate.
9. Transient analysis of Fin.

TOTAL: 45PERIODS

LIST OF EQUIPMENT FOR A BATCH OF 30 STUDENTS

S. NO.	NAME OF THE EQUIPMENT	Qty.
1.	Computer work station	30
2.	Printer	1
3.	Ansys Software	30 licenses

Cycle A - Simulation

Ex No : 1 *Date :*

MATLAB basics, Dealing with matrices, Graphing-Functions of one variable and two variables

Aim: To the basics of MATLB dealing with matrices and to draw graph of one variable and two variables using MATLAB software.

Basic MATLAB Programming and matrix operations.

MATLAB is a matrix-based language. Since operations may be performed on each entry of a matrix, "for" loops can often be by passed by using this option. As a consequence, MATLAB programs are often much shorter and easier to read than programs written for instance in C or Fortran. Below, we mention basic MATLAB commands, which will allow a novice to start using this software.

1. Defining a row matrix and performing operations on it. Assume that you want to evaluate the function $f(x) = x^3 - 6x^2 + 3$ at different values of x. This can be accomplished with two lines of MATLAB code.

```
%Define the values of x
X= 0:0.1:1;
%Evaluate f
f = x.^3 – 6*x.^2 + 3;
```

In this example, x varies between o and 1 in steps of 0.01. Comments are preceded by a % sign. The symbols ^ and * stands for the power and multiplication operators respectively. The dot in front of ^n indicates that each entry of the row matrix x is raised to the power n. In the absence of this dot, MATLAB would try to take the nth power of x, and an error message would be produced since x is not a square matrix. A semicolon at the end of a command line indicates that the output should not be printed on the screen.

Exercises:

1. Type size (x) to find out what the size of x is
2. Evaluate the cosine and sine of x
3. Define a square matrix

A = (1 2 3 4) by typing A = [1 2; 3 4]. Then compute the square of A (type A^2) and compare the result to that obtained by typing A.^2.

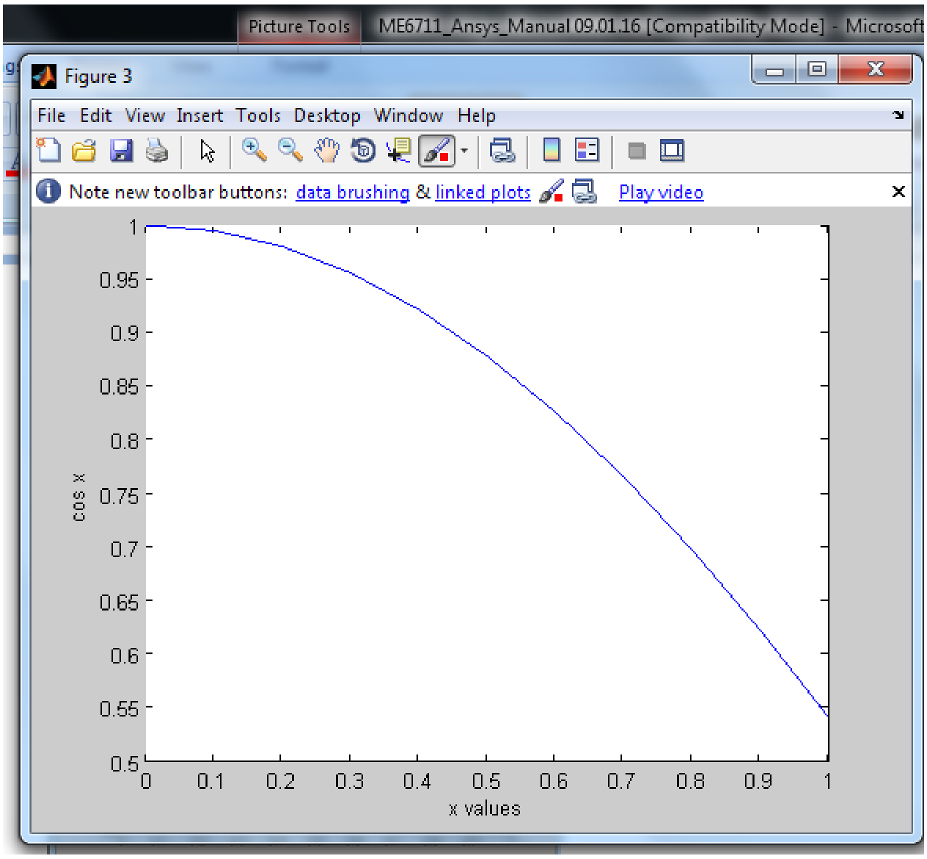
1. Define matrix A and B with any size and perform the value of A+B, A-B, A*B, A/B, A^2*B, A*A
2. Verify A*B is not equal to B*A
3. A*inv(A)= identity Matrix

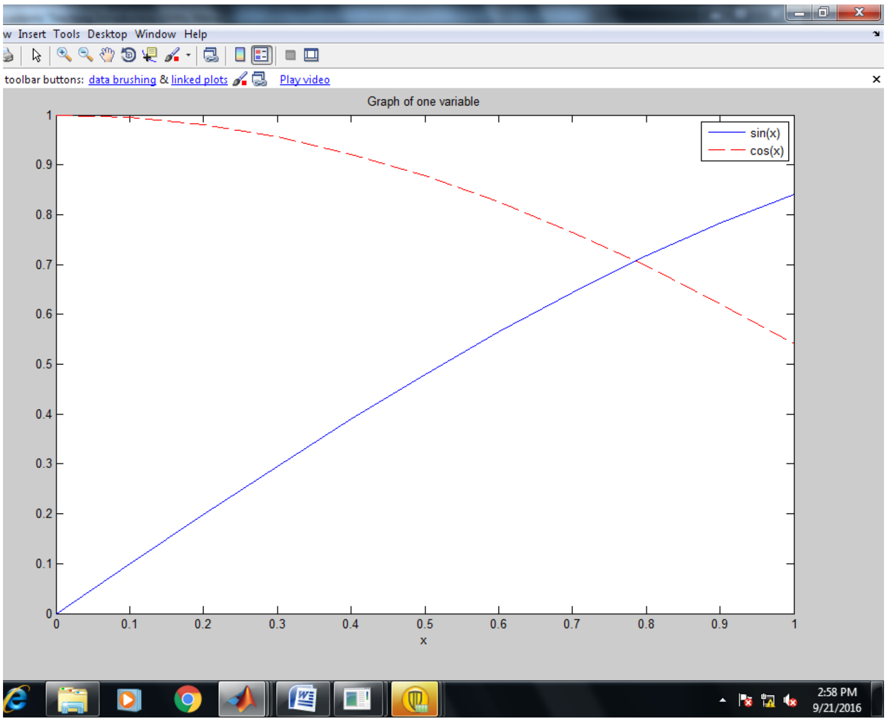
2. Plotting the graph of a function of one variable

The command plot(x,f) plots f as a function of x. The figure can be edited by hand to add labels, change the thickness of the line of the plot, add markers, change the axes etc. All of these attributes can also be specified as part of the plot command.

Exercises:

1. Plot (x, sin(x))
2. Plot (x, cos(x))
3. plot(x,sin(x),'-b',x,cos(x),'--r')
4. Plot the graph of exp(x) for x =(-5,10)




Graph of one variable
sin(x)
cos(x)
x

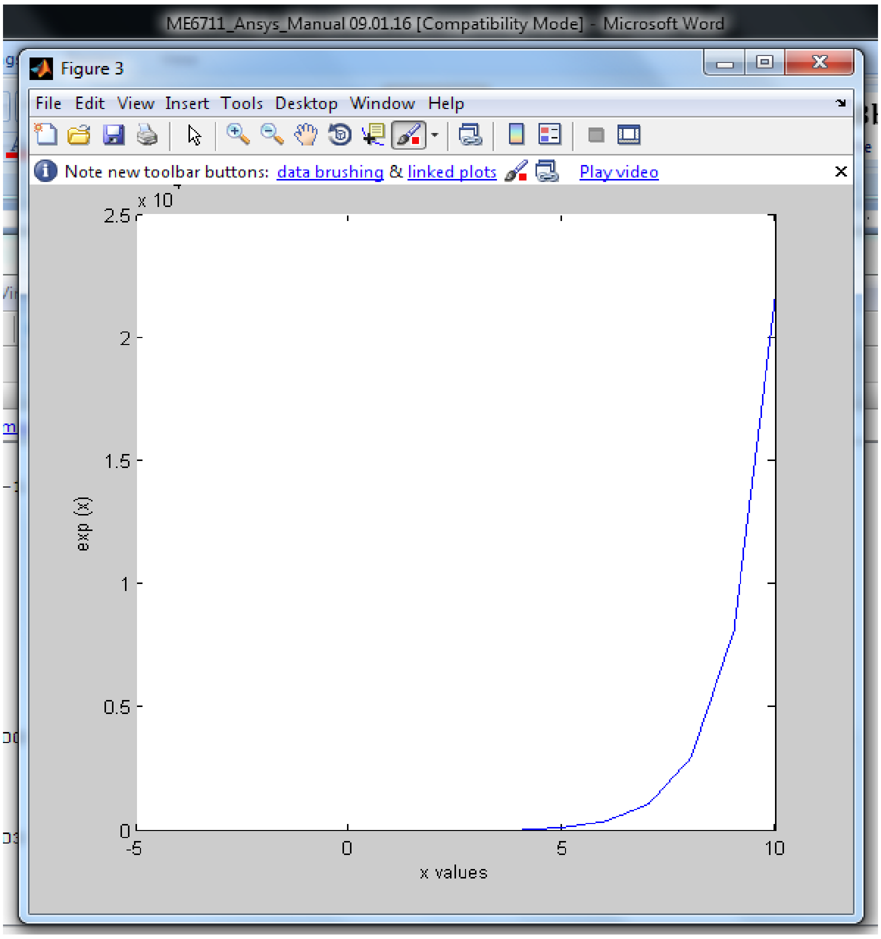


Figure 3
exp (x)
x values

3. Plotting the graph of a function of two variables

Assume that we want to use MATLAB to plot the graph of f(x,y) = $x^2 - 3y^2$

for x =-3:0.01:3

for y = -5:0.01:5

We first need to define a numerical grid where the function f will be evaluated. To this end, define the matrices x and y,

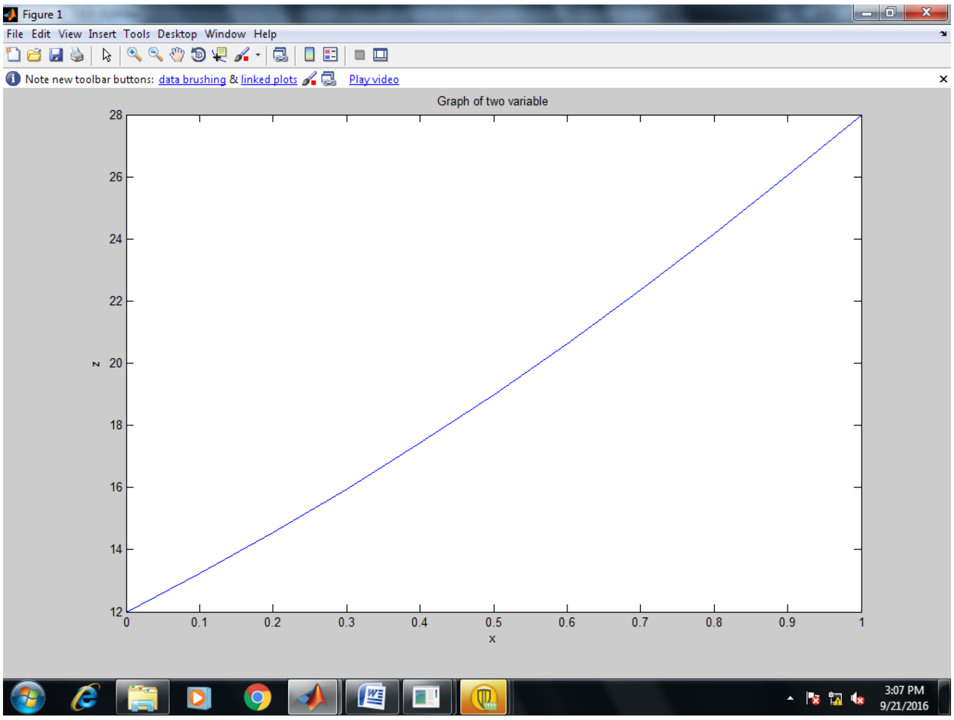
(x,y) = meshgrid(x,y)

Then, evaluate f at the points on the grid and put the result in a matrix Z

Z= x.^2-3*y.^2;

Finally, plot the graph of f with the following command surf (x,y,z), shading interp. The surface can be rotated by typing rotate 3D, or by clicking on the rotation icon on the figure.

Plot (x,z)

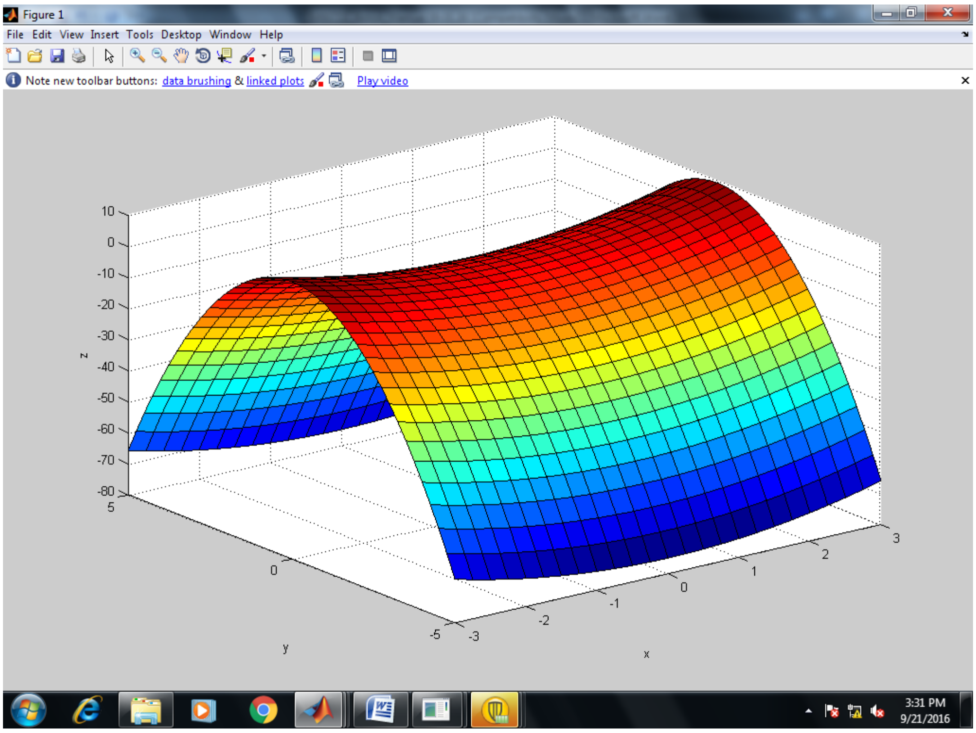


Exercises

1. Plot the graph of f(x) = exp(-2x^2-3y^2). Choose appropriate intervals for x and y.
2. Plot the graph of f(x) = cos(x) sin(y). Choose appropriate intervals for x and y.
3. Change the color map of one of the plots above by using the commands color map bone or colormap jet or colormap cool.

Matlab Program for Surface

```
x=-3:0.25:3;
y=-5:0.25:5;
[x,y] = meshgrid(x,y);
z=x.^2-(3*y.^2);
surf(x,y,z)
xlabel('x'); ylabel('y')
zlabel('z')
```



Result:

Thus the basics of MATLAB dealing with matrices are studied and the graph for one variable and two variables are drawn using MATLAB software.

****

Ex No : 2 *Date :*

Use of MATLAB to solve simple problems in vibration

Problem Definition

Spring Mass Damper System – Unforced Response

Solve for five cycles, the response of an unforced system given by the equation

$m\ddot{x} + c\dot{x} + kx =$ (1)

For c =2 ; m = 1 kg; k = 100 N/m; x (0) = 10 m; $\dot{x}(0) = -8$;

Aim:

To find the natural vibration of spring mass system using MATLAB software.

Solution

$X(t) = A\, e^{-\xi W_n t} \sin(W_d t + \Phi)$

A=(1/wd)*(sqrt(((vo+zeta*wn*xo)^2)+((xo*wd)^2)))

Φ = arctan((xo*wd)/(vo+zeta*wn*xo))

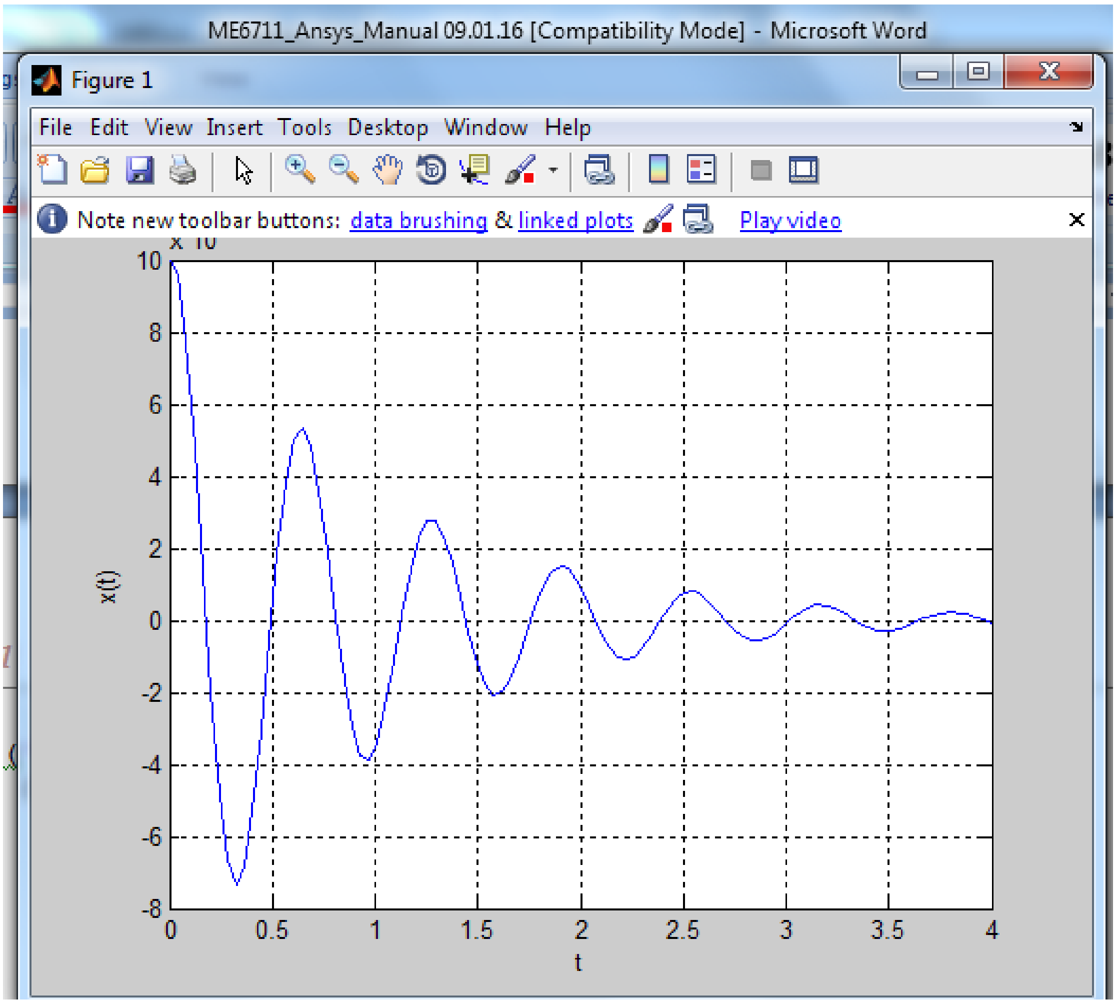
ξ=c/(2*m*wn)

Natural frequency wn=sqrt(k/m)

Mat Lab Program

```
m=1;
k=100;
c=2;
vo=1/1000;
xo=1/1000;
t=linspace(0,4,100);
wn=sqrt(k/m);
zeta=c/(2*m*wn);
wd=wn*sqrt(1-zeta^2);
A=(1/wd)*(sqrt(((vo+zeta*wn*xo)^2)+((xo*wd)^2)));
ang=atan((xo*wd)/(vo+zeta*wn*xo));
xt=A*sin(t*wd+ang).*exp(-zeta*t*wn);
display(c)
plot(t,xt)
```

```
grid on
ylabel('x(t)')
xlabel ('t')
```



Result:

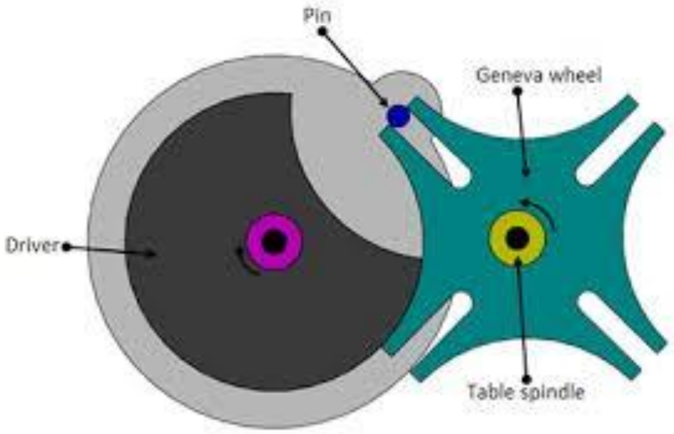
Thus the natural vibration of spring mass system is obtained using MATLAB software.

Ex No : 3 ***Date :***

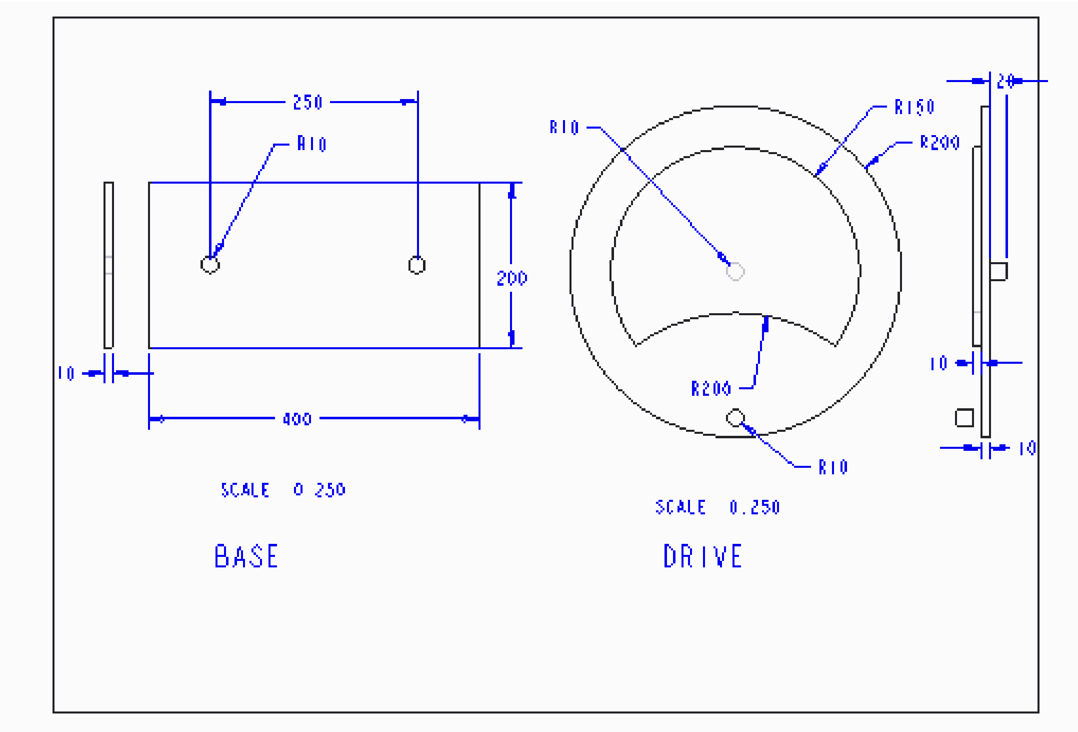
Mechanism Simulation using Multibody Dynamic software

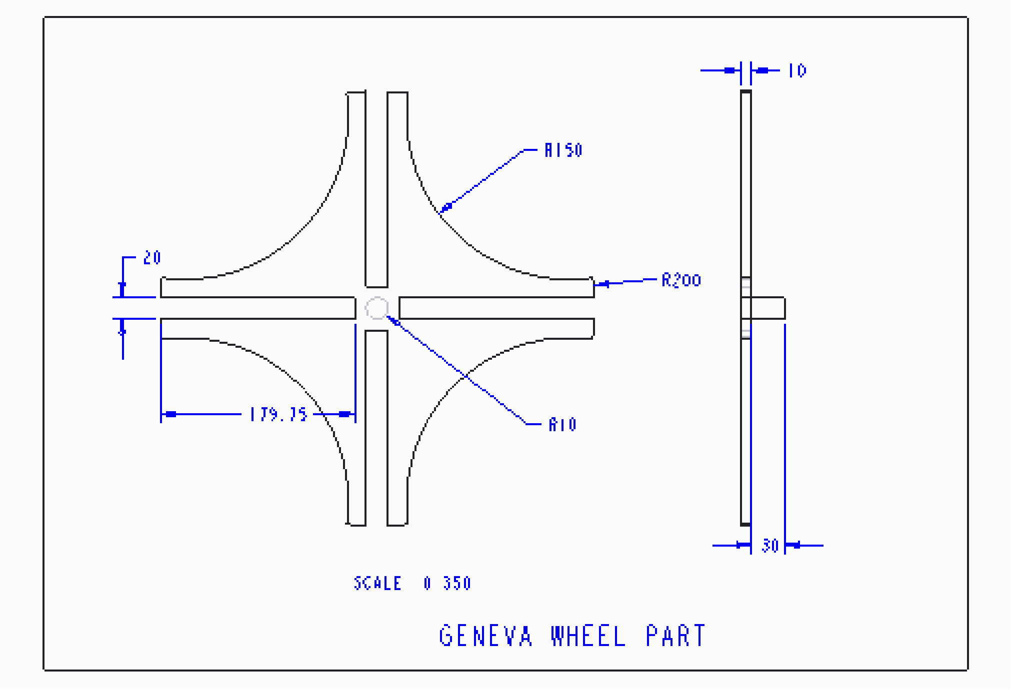
Problem Definition:

Assemble the given components of Geneva drive mechanism and simulate using multibody dynamic software.



Aim: To assemble the given components and simulate the Geneva drive mechanism using Creo 3.0 software.

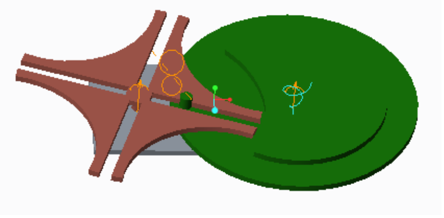




ALL DIMENSIONS ARE IN MM

Procedure:

1. Assemble the given components using user defined mating condition.
2. Attach the servo motor and give the velocity.
3. Go to the mechanism analysis and give the start time and end time.
4. Run the given mechanism.
5. Save the analysis definition file.



Result:

Thus the simulation of Geneva mechanism is done using Creo 3.0 software.

Cycle B - Analysis

Ex No : 1 ***Date :***

INTRODUCTION to FEA and ANSYS

What is FEA?

1. Finite Element analysis is a way to simulate loading conditions on a design and determine the design response to those conditions.
2. The design is modeled using discrete building blocks called elements.
3. Each element has exact equations that describe how it responds to a certain load.
4. The "Sum" of the response of all elements in the model gives the total response of the design.
5. The elements have a finite number of unknowns, hence the name finite elements.
6. The finite element model, which has a finite number of unknowns, can only approximate the response of the physical system which has infinite unknowns.

How good is the approximation?

Unfortunately, there is no easy answer to this question, it depends entirely on what you are simulating and the tools you use for the simulation.

Why is FEA needed?

1. To reduce the amount of prototype testing.
2. Computer Simulation allows multiple "what if"scenarios to be tested quickly and effectively.
3. To simulate designs those are not suitable for prototype testing. E.g. Surgical Implants such as an artificial knee.

About ANSYS:

- ANSYS is a complete FEA software package used by engineers worldwide in virtually all fields of engineering. ANSYS is a virtual Prototyping technique used to iterate various scenarios to optimize the product.

General Procedure of Finite Element Analysis:

1. Creation of geometry or continuum using preprocessor.
2. Discretization of geometry or continuum using preprocessor.
3. Checking for convergence of elements and nodes using preprocessor.
4. Applying loads and boundary conditions using preprocessor.
5. Solving or analyzing using solver
6. Viewing of Results using postprocessor.

Build Geometry:

Construct a two (or) three dimensional representation of the object to be modeled and tested using the work plane co-ordinate system in Ansys.

Define Material Properties:

Define the necessary material from the library that composes the object model which includes thermal and mechanical properties.

Generate Mesh:

Now define how the model system should be broken down into finite pieces.

Apply Loads:

- The last task in preprocessing is to restrict the system by constraining the displacement and physical loading.

Obtain Solution:

- The solution is obtained using solver available in ANSYS. The computer can understand easily if the problem is solved in matrices.

Present the Result:

- After the solution has been obtained there are many ways to present Ansys result either in graph or in plot.

Specific Capabilities of ANSYS Structural Analysis:

- Structural analysis is probably the most the common application of the finite element method such as piston, machine parts and tools.

Static Analysis:

- It is the used to determine displacement, stress etc. under static loading conditions. Ansys can compute linear and non-linear types (e.g. the large strain hyper elasticity and creep problems).

Transient Dynamic Analysis:

- It is used to determine the response of a structure to time varying loads.

Buckling Analysis:

- It is used to calculate buckling load and to determine the shape of the component after applying the buckling load. Both linear buckling and non – linear buckling analysis are possible.

Thermal Analysis:

The steady state analysis of any solid under thermal boundary conditions calculates the effect of steady thermal load on a system (or) component that includes the following.

a) Convection.
b) Radiation.
c) Heat flow rates.
d) Heat fluxes.
e) Heat generation rates.
f) Constant temperature boundaries.

Fluid Flow:

- The ANSYS CFD offers comprehensive tools for analysis of two-dimensional and three dimensional fluid flow fields.

Magnetic:

- Magnetic analysis is done using Ansys / Electromagnetic program. It can calculate the magnetic field in device such as power generators, electric motor etc. Interest in magnetic analysis is finding magnetic flux, magnetic density, power loss and magnetic forces.

Acoustic / Vibrations:

- Ansys is the capable of modeling and analyzing vibration system. Acoustic is the study of the generation, absorption and reflection of pressure waves in a fluid application.
- Few examples of acoustic applications are

a) Design of concert house, where an even distribution of sound pressure is possible.
b) Noise cancellation in automobile.
c) Underground water acoustics.
d) Noise minimization in machine shop.
e) Geophysical exploration.

Coupled Fields:

- A coupled field analysis is an analysis that takes into account the interation between two (or) more fields of engineering analysis. Pressure vessels, Induction heating and Micro electro mechanical systems are few examples.

Result:

Thus the basics of FEA and ANSYS are studied.

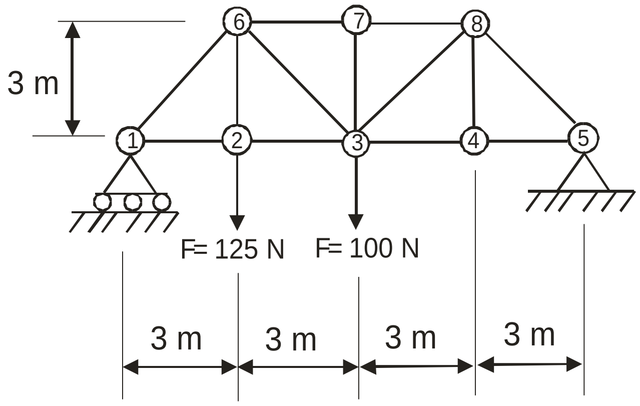
****

Ex No : 2

Force and stress analysis using Link elements in Trusses

Problem Specification:

Find the reaction forces, axial forces and stresses at each node for the following truss. Also find the deflection at each node.



Cross-sectional area of truss members = 3.0E-4 m^2;
Modulus of Elasticity = 2.07E11 N/m^2.
Circled numbers shown are node numbers.
Poisson's ratio = 0.3

Aim:

To find the reaction forces, axial forces and stresses at each node for the given truss and also find the deflection at each node using ANSYS software

Procedure:

1. Preferences

a. Click → Main Menu → Preferences → Click Structural → Click OK.

2. Element Type

a. Element Type → Add / Edit / Delete → Add → Link → 2D spar → OK

3. Real Constants

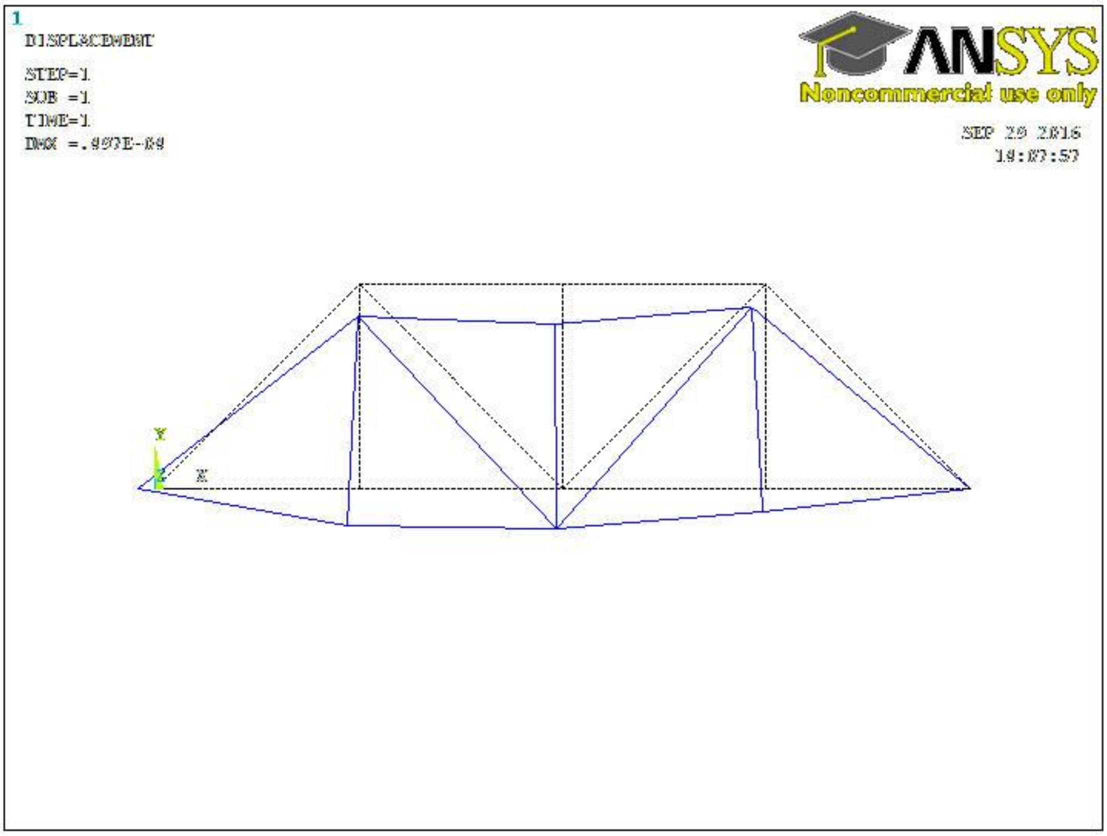
a. Real Constants → Add / Edit / Delete → Add → OK

b. Enter → Area = 3.0e-4 → Click OK.

4. Materials Props

a. Preprocessor → Materials Props → Materials Models → Structural → Linear → Elastic → Isotropic → **Ex: 2.07E11** → **Prxy : 0.3** → Ok

b. Materials → Exit



5. Modeling

a. Modeling → Create → nodes → In active CS → Enter point.

b. 1 → 0,0   2 → 3,0   3 → 6,0   4 → 9,0   5 → 12,0   6 → 3,3   7 → 6,3   8 → 9,3

c. Create → Elements → through auto numbered → Select nodes for element → 1 → 1,2   2 → 2,3   3 → 3,4   4 → 4,5   5 → 1,6   6 → 2,6   7 → 3,6   8 → 3,7   9 → 3,8   10 → 4,8   11 → 5,8   12 → 6,7   13 → 7,8 → Click OK.

7. Solution

a. Solution → Define Loads → Apply → Structural → Displacement → on nodes → Select node Point 5 → Click OK → Select All DOF Constrained. Select node Point 1 → Select (FY) → enter the value =0 Click OK

b. Force/ moment →on nodes → Select node point 2 → Select (FY) → Enter Value of -125 → Click OK → Select node point 3 → Select (FY) → Enter Value of -100 → Click OK

c. Solution → Solve → Current LS → OK.

8. General Post proc

a. General Post proc → Plot Results → Deformed Shape → Def+Undeformed → OK.

b. UtilityMenu → Plot Ctrls → Hardcopy → To file → pick JPEG → save to (enter file name)

c. List result → Reaction solution → ok→ save to file.

d. sList result → Element solution → ok → save to file.

e. General Postproc -> List Results -> Nodal Solution -> DOF Solution -> ALL DOFs → save to file.

Inference:

Analysis of truss is used mainly in civil engineering (Eg: Roof structures)

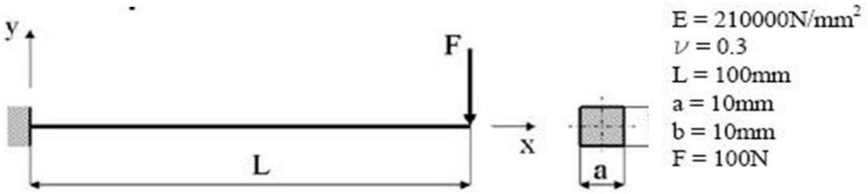
Result:

Thus the reaction forces, axial forces and stresses at each node for the given truss and also the deflection at each node is found using ANSYS software

Ex No 3(a) *Date :*

Stress Analysis of Cantilever Beam

Problem Specification:



E	=	2.1e5 N/mm^2	γ	=	0.3	Izz	=	833.33 mm^4
L	=	100mm	a	=	10mm	b	=	10mm
F	=	100 N						

Aim:

To find deformed shape, shear force diagram and bending moment diagram for the Cantilever beam using ANSYS software

Procedure:

1. Preferences

a. Click → Main Menu → Preferences → Click Structural → Click OK.

2. Element Type

a. Element Type → Add / Edit / Delete → Add → Beam → 2D Elastic 3 → OK

3. Real Constants

a. Real Constants → Add / Edit / Delete → Add → OK

b. Enter → Area = 100, Izz = 833.33 , Height = 10 → Click OK.

4. Materials Props

a. Preprocessor → Materials Props → Materials Models → Structural → Linear → Elastic → Isotropic → **Ex: 2.1e5** → **Prxy : 0.3** → Ok

b. Materials → Exit

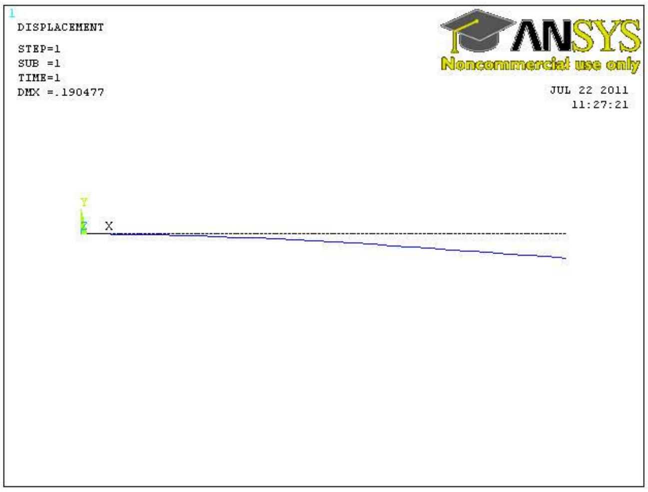
5. Modeling

a. Modeling → Create → Key points → In active CS → Enter Key point.

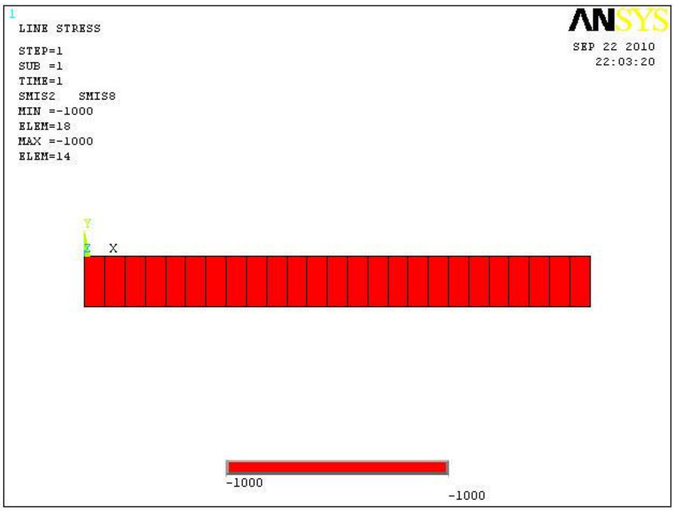
b. 1 → 0,0,0 2 → 100,0,0

c. Lines → Lines → Straight Lines → Select Key point 1&2 → Click OK.

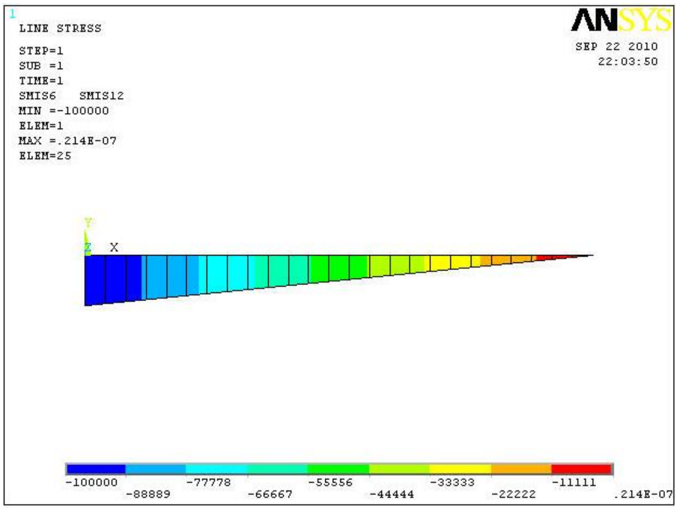
Deformed shape



Shear forces Diagram



Bending Moment Diagram



6. Meshing

a. Mesh Tool → Size control → Global → Size → Enter No. of Division = 25 → OK.
b. Mesh Tool → Mesh → (Select the line) → OK
c. Plot Cltrs (Main Menu) → Style → Size and shape → Display of Element (ON) → OK
d. Plot Cltrs (Main Menu) → Style → Size and shape → Display of Element (OFF) → OK

7. Solution

a. Solution → Define Loads → Apply → Structural → Displacement → on Key Points → Select key Points 1 → Click OK → Select All DOF Constrained.
b. Force/ moment → Select key point 2 → Select (FY) → Enter Value of -100 → Click OK.
c. Solution → Solve → Current LS → OK.

8. General Post proc

a. General Post proc → Plot Results → Deformed Shape → Def+Undeformed → OK.
b. UtilityMenu → Plot Ctrls → Hardcopy → To file → pick JPEG → save to (enter file name)
c. Element Table → Define Table → Add → Select By Sequence → SM1SC,2 → apply,
d. Select By Sequence → SM1SC,6 → apply
e. Select By Sequence → SM1SC,8 → apply
f. Select By Sequence → SM1SC,12 → apply → OK.

9. Plot Result

a. Plot Result → Contour Plot → Line Element Resolution →
b. Select SMISC2 + SMISC8 for S.F.D. → OK
c. UtilityMenu → Plot Ctrls → Hardcopy → To file → pick JPEG → save to (enter file name)
d. Select SMISC6 + SMISC12 for B.M.D. → OK
e. UtilityMenu → Plot Ctrls → Hardcopy → To file → pick JPEG → save to (enter file name)

Inference:

Analysis of beam is used mainly in civil engineering (Eg: Bridges, Roof structures)

Result:

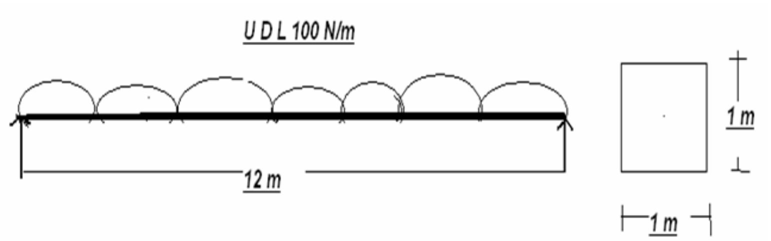
Thus the deformed shape , shear force diagram and Bending moment diagram are obtained for the Cantilever Beam using ANSYS software.

****

Ex No :3(b) ***Date :***

Stress Analysis of Simply Supported Beam with UDL

Problem Specification:



E = 2.1e11 N/m^2 ν = 0.3 Izz = 0.083333m^4

L = 12m a = 1m b = 1m

Aim:

To find deformed shape, shear force diagram and bending moment diagram for the simply supported beam with UDL using ANSYS software

1. Preferences

a. Click → Main Menu → Preferences → Click Structural → Click OK.

2. Element Type

a. Element Type → Add / Edit / Delete → Add → Beam → 2D Elastic 3 → OK

3. Real Constants

a. Real Constants → Add / Edit / Delete → Add → OK

b. Enter → Area = 1 ,Izz = 0.083333 , Height = 1 → Click OK.

4. Materials Props

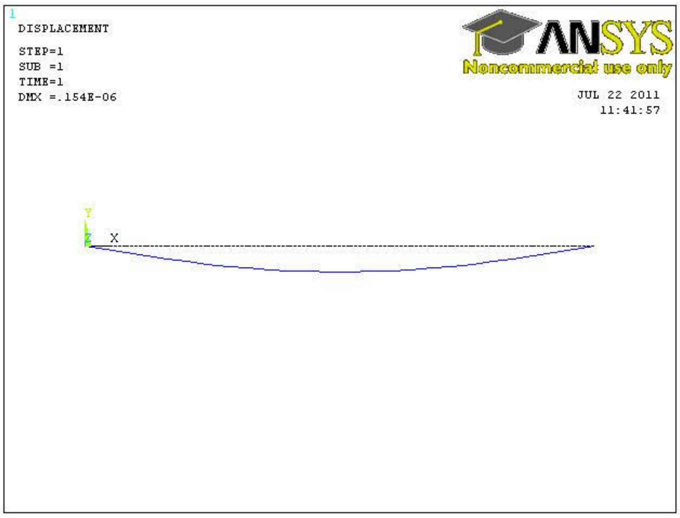
a. Preprocessor → Materials Props → Materials Models → Structural → Linear → Elastic → Isotropic → **Ex: 2.1e11** → **Prxy : 0.3** → Ok Materials → Exit

5. Modeling

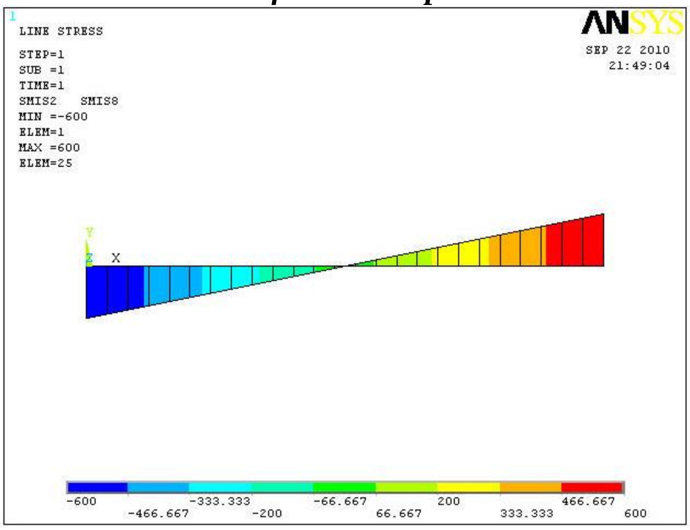
a. Modeling → Create → Key points → In active CS → Select Key point.

b. 1 → 0,0,0 2 → 12,0,0

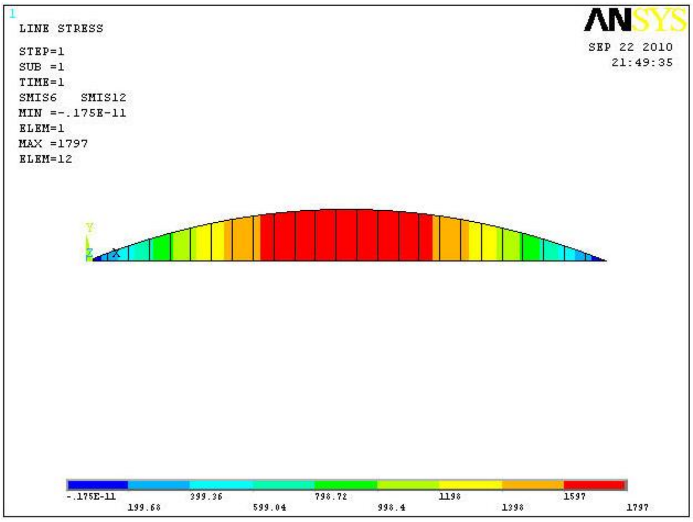
c. Lines → Lines → Straight Lines → Select Key point 1&2 → Click OK.



Deformed Shape



Shear force Diagram



Bending Moment Diagram

6. Meshing

a. Mesh Tool → Size control → Global → Size → Enter No. of Division → 25 → OK.
b. Mesh Tool → Mesh → (Select the line) → OK
c. Plot Cltrs (Main Menu) → Style → Size and shape → Display of Element (ON) → OK
d. Plot Cltrs (Main Menu) → Style → Size and shape → Display of Element (OFF) → OK

7. Solution

a. Solution → Define Loads → Apply Structural → Displacement → on Key Points → Select key Points 1 and 2 → Select (Ux , Uy)Constrained → OK.
b. Pressure → on Beam → Select Pick all → OK → enter the value = 100 → OK.
c. Solution → Solve → Current LS → OK.

8. General Post proc

a. General Post proc → Plot Results → Deformed Shape → Def + Undeformed → OK.
b. Utility Menu → Plot Ctrls → Hardcopy → To file → pick JPEG → save to (enter file name)
c. Element Table → Define Table → Add → Select By Sequence → SM1SC,2 → apply,
Select By Sequence → SM1SC,6 → apply
Select By Sequence → SM1SC,8 → apply
Select By Sequence → SM1SC,12 → apply → OK.

9. Plot Result

a. Plot Result → Contour Plot → Line Element Resolution → Select SMISC2 + SMISC8 for S.F.D. → OK
b. UtilityMenu → Plot Ctrls → Hardcopy → To file → pick JPEG → save to (enter file name)
c. Select SMISC6 + SMISC12 for B.M.D. → OK
d. UtilityMenu → Plot Ctrls → Hardcopy → To file → pick JPEG → save to (enter file name)

Inference:

Analysis of beam is used mainly in civil engineering (Eg: Bridges, Roof structures)

Result:

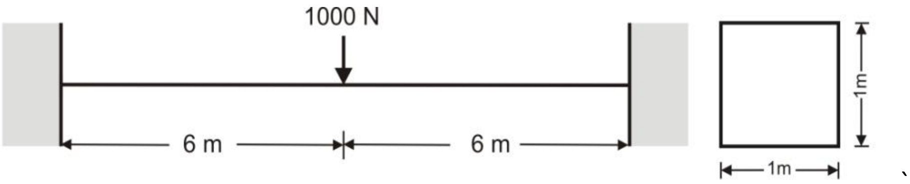
Thus the deformed shape, shear force diagram and Bending moment diagram are plotted for the simply supported beam with UDL using ANSYS software

****

Ex No : 3(c) *Date :*

Stress Analysis of Fixed End Beam with Point Load

Problem Specification:



E = 2.1e11 N/m^2 ν = 0.3 Izz = 0.0833m^4

L = 12m a = 1m

b = 1m

Aim:

To find the deformed shape, shear force diagram and bending moment diagram for Fixed end Beam with point load.

Procedure:

1. Preferences

a. Click → Main Menu → Preferences → Click Structural → Click OK.

2.Element Type

a. Element Type → Add / Edit / Delete → Add → Beam → 2D Elastic 3 → OK

3. Real Constants

a. Real Constants → Add / Edit / Delete → Add → OK

b. Enter → Area = 1 ,Izz = 0.083333 , Height = 1 → Click OK.

4. Materials Props

a. Preprocessor → Materials Props → Materials Models → Structural → Linear → Elastic → Isotropic → **Ex: 2.1e11** → **Prxy : 0.3** → Ok

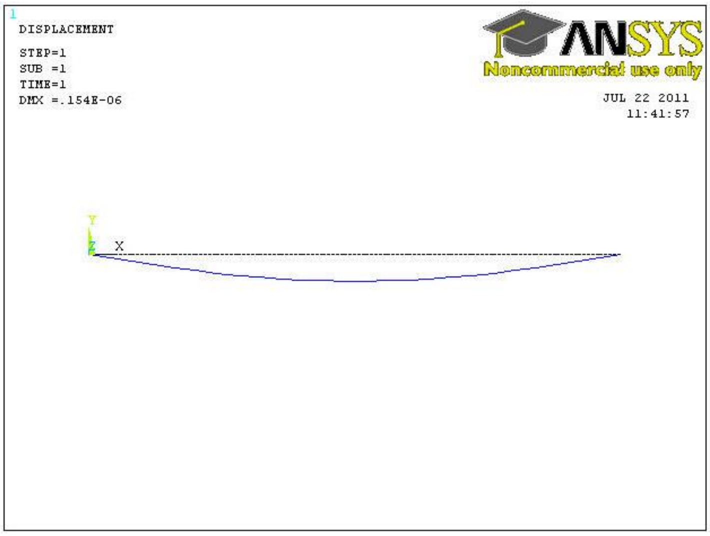
b. Materials → Exit

5. Modeling

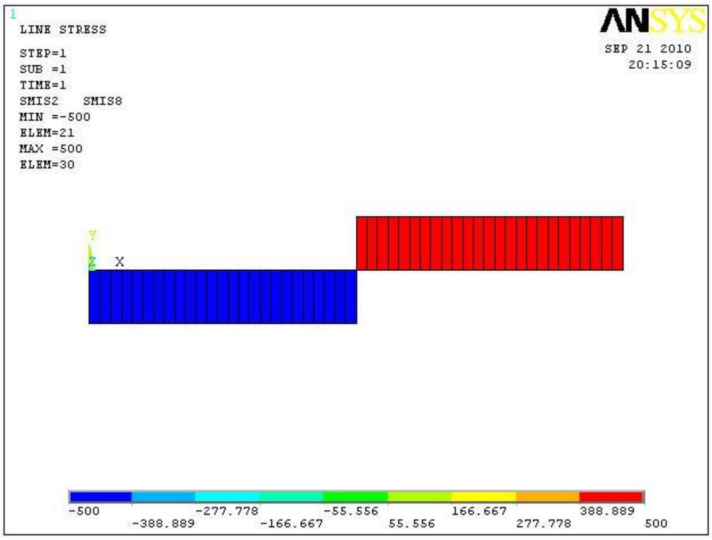
a. Modeling → Create → Key points → In active CS → Select Key point.

b. 1 → 0,0,0 2 → 6,0,0 3 → 12,0,0

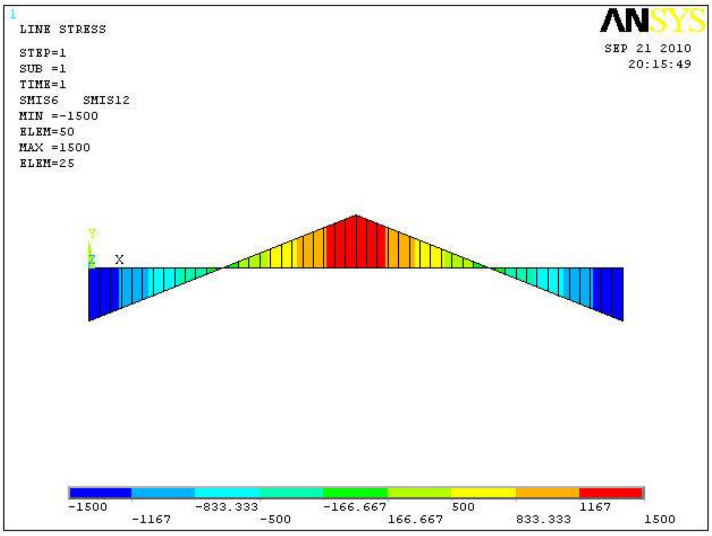
c. Lines → Lines → Straight Lines → Select Key point 1&2and 2&3 → Click OK.



Deformed Shape



Shear Force Diagram



Bending Moment Diagram

6. Meshing

a. Meshing → Size control → Global →Size → select the line → Enter No.of Divisions= 25 → OK.
b. Mesh Tool → Mesh → (Select the line) → OK
c. Plot Cltrs (Main Menu) → Style → Size and shape → Display of Element (ON) → OK
d. Plot Cltrs (Main Menu) → Style → Size and shape → Display of Element (OFF) → OK

7. Solution

a. Solution → Define Loads → Apply Structural → Displacement → on Key Points → Select key Points 1 and 3 → Select All DOF Constrained → Click OK.
b. Force moment → on key point 2 (Centre) → Select **(FY)** → Enter Value of - 1000 → Click OK.
c. Solution → Solve → Current LS → OK.

8. General Post proc

a. General Post proc → Plot Results → Deformed Shape → Def + Undeformed → OK.
b. Utility Menu → Plot Ctrls → Hardcopy → To file → pick JPEG → save to (enter file name)
c. Element Table → Define Table → Add → Select By Sequence → SM1SC,2 → apply,
d. Select By Sequence → SM1SC,6 → apply
e. Select By Sequence → SM1SC,8 → apply
f. Select By Sequence → SM1SC,12 → apply → OK.

9. Plot Result

a. Plot Result → Contour Plot → Line Element Resolution → Select SMISC2 + SMISC8 for S.F.D. → OK
b. Utility Menu → Plot Ctrls → Hardcopy → To file → pick JPEG → save to (enter file name)
c. Select SMISC6 + SMISC12 for B.M.D. → OK
d. Utility Menu → Plot Ctrls → Hardcopy → To file → pick JPEG → save to (enter file name)

Inference:

Analysis of beam is used mainly in civil engineering (Eg: Bridges, Roof structures)

Result:

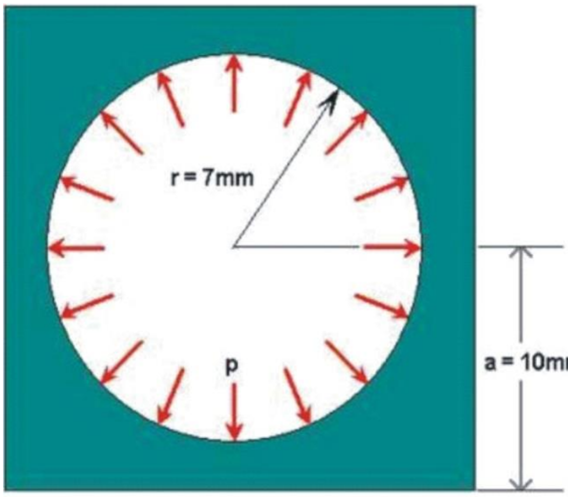
Thus the deformed shape, shear force diagram and Bending moment diagram are plotted for the Fixed End Beam with point load using ANSYS software.

****

Ex No : 4 ***Date :***

Stress Analysis of A Plate With A Circular Hole

Consider the square plate of uniform thickness with a circular hole with dimensions shown in thefigurebelow.Thethicknessoftheplateis1mm.TheYoung'smodulus $E=10^7$MPa and the Poissonratiois0.3. A uniform pressure = 1MPa act son the boundary of the hole. Assume that plane stress conditions prevail. Find stress and displacement fields using ANSYS software.



E = 1e13 N/m^2 γ = 0.3   P = 1e6 N/m^2

a = 10e-3 m   r = 7e-3 m

Aim:

To find the deformation shape and nodal stress analysis of rectangular component with hole.

1. Preferences

a. Main Menu → Preferences → Structural → OK.

2. Element Type

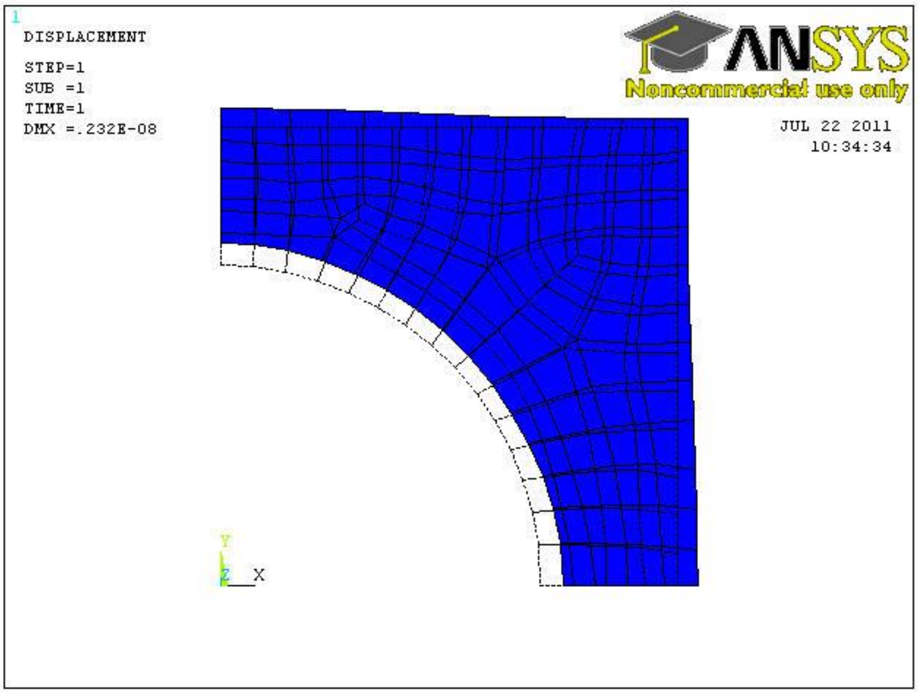
a. Element Type → Add / Edit / Delete → Add... → Pick Structural Solid → Quad4node42 → OK

3. Real Constants

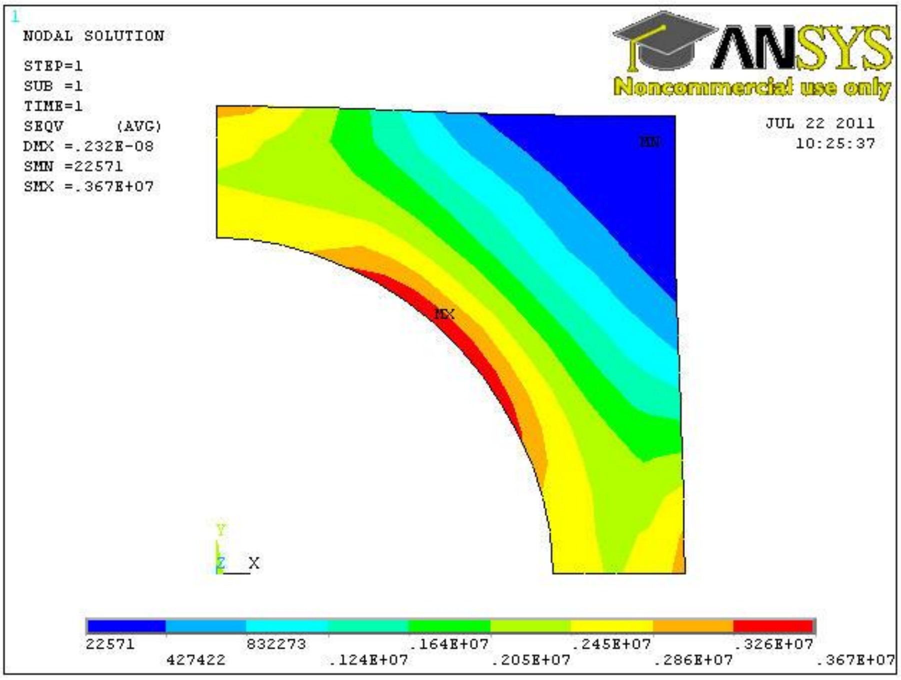
a. Real Constants → Add / Edit / Delete > Add → OK.

4. Materials Props

a. Material Props → Material Models.... → Structural, Linear, Elastic, and Isotropic → Enter **EX**, = 1e13 , **PRXY = 0.3** Click **OK**.



Deformed shape



5. Modeling

a. Modeling → Create → Areas → Rectangle → By 2 corners → X=0, Y = 0, width = 10e-3, height = 10e-3
b. Modeling → Create → Areas → Circle → solid circle → WP X = 0 , WP Y = 0 , Rad = 7e-3
c. Modeling → Operate → Booleans → Subtract → Areas → pick square area by using left mouse button → OK. → pick circular area → Click OK.

6. Meshing

a. Mesh Tool → click smart size → set size = 5 → free mesh → pick the area → OK.

7. Solution

a. Loads → Define Loads → Apply → Structural → Displacement → Symmetry B.C. → On Lines → Select left and bottom edges → OK.
b. Loads → Define Loads → Apply → Structural → Pressure → On Lines → Pressure Value = 1e6 → OK.
c. Solve → Current LS → OK.

8. General Post proc

a. General Post proc → Plot Results → Deformed Shape → Select(Def + undeformed) and click OK.
b. Utility Menu → Plot Ctrls → Hardcopy → To file → pick JPEG → save to (enter file name)
c. Utility Menu → Plot Ctrls → Animate → Deformed Shape... → Select Def + undeformed and click OK
d. General Post proc → Plot results → Contour Plot → Nodal Solu → Select Stress from the left list, vonmises from the right list and click OK
e. Utility Menu → Plot Ctrls → Hardcopy → To file → pick JPEG → save to (enter file name)

Inference:

Stress–strain analysis (or stress analysis) is an engineering discipline covering methods to determine the stresses and strains in materials or structures which is having a hole at the center and subjected to forces or loads.

Result:

Thus the deformation shape and nodal stress diagram of Rectangular plate with circular hole is plotted using ANSYS software.

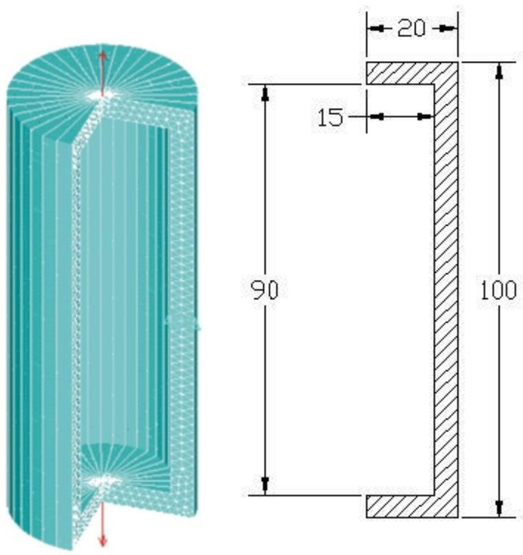
****

Ex No : 5 ***Date :***

Stress analysis of an axi – symmetric component

Problem definition:

The model will be that of a closed tube made from steel. Point loads will be applied at the center of the top and bottom plate to make an analytical verification simple to calculate. A 3/4 cross section view of the tube is shown below. As warning, point loads will created is continuities in your model near the point of application. If you chose to use these types of loads in your own modeling, be very careful and be sure to understand the theory of how the FEA package is Appling the load and the assumption it is making. In this case, we will only be concerned about the stress distribution far from the point of application, so the discontinuities will have a negligible effect.



E =2e5
γ =0.3

100N **All dimensions are in mm**

Aim:

To find out deformation shape and stress analysis of an axi-symmetric component using ANSYS software.

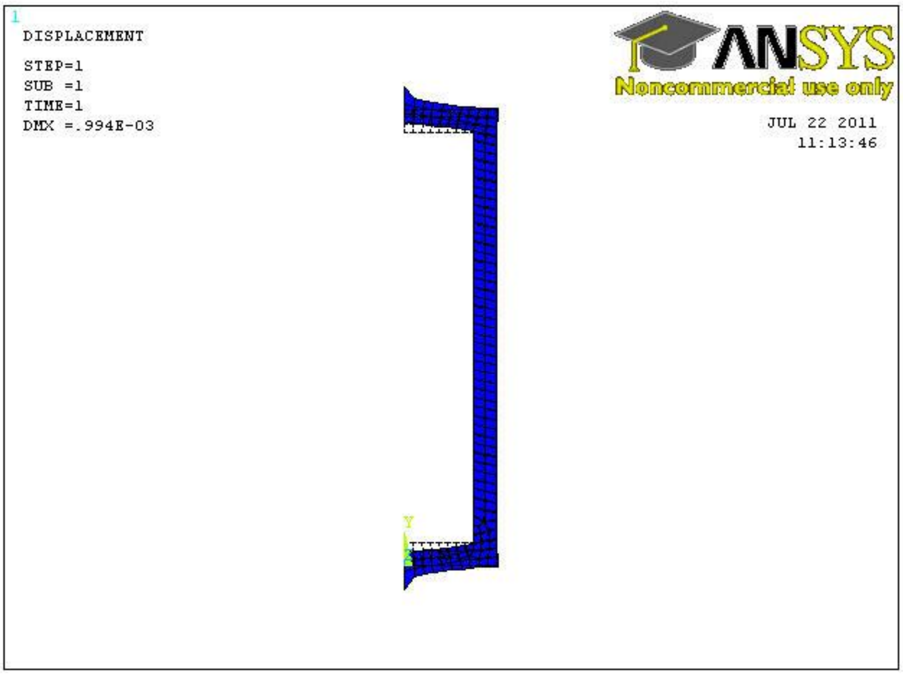
Steps :

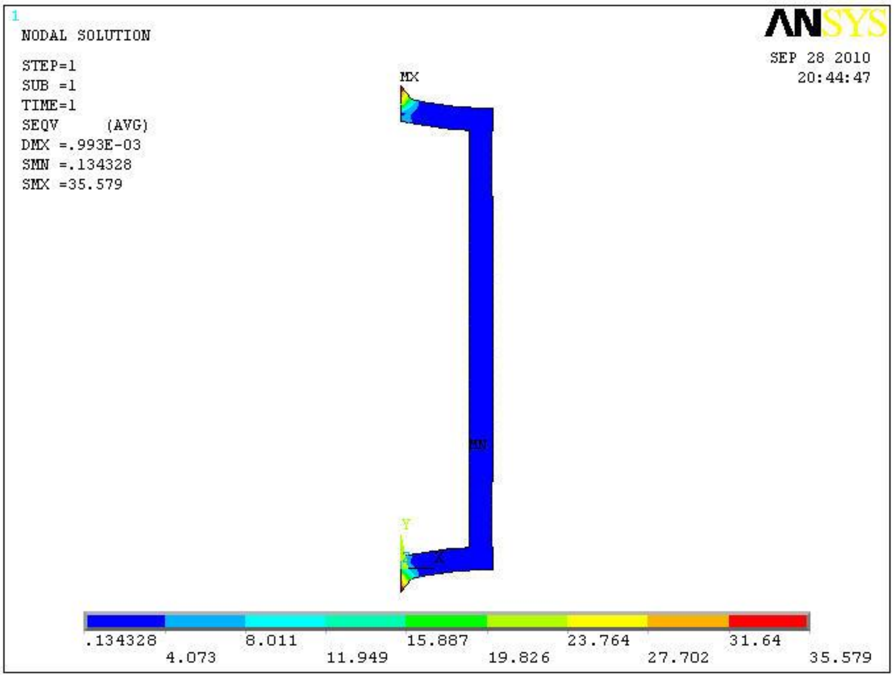
1. Preferences
2. Element Type
3. Real Constants
4. Materials Props
5. Modeling
6. Meshing
7. Solution
8. General Post proc

1. Preferences

a. Main Menu → Preferences → Structural → OK.

Deformed shape





2. Element Type

a. Element Type → Add/Edit/Delete → Add... → Pick Structural Solid → Quad 8 node 82 → OK

b. Element Type → Options → Select the Axisymmetric → OK

3. Real Constants

a. Real Constants → Add/Edit/Delete>Add → OK.

4. Materials Props

a. Material Props → Material Models.... → Structural, Linear, Elastic, and Isotropic → Enter **EX**, = 2e5 , **PRXY = 0.3** Click **OK**.

5. Modeling

a. Modeling → Create → Areas → Rectangle → By 2 corners → X = 0 ,Y = 0 ,W = 20 , H = 100 → OK.

b. Modeling → Create → Areas → Rectangle → By 2 corners → X = 0 ,Y = 5 , W = 15 , H = 90 → OK.

c. Modeling → Operate → Booleans → Subtract → Areas → pick First Rectangular area → OK. → pick Second Rectangular area → OK.

6. Meshing

a. Mesh Tool → set area → pick Area → OK → Element Size → 2 → OK.

b. Mesh Tool → Mesh → Free Mesh → pick Area → OK.

7. Solution

a. Solution → Define Loads → Apply → Structural → Displacement → Symmetry B.C. → On Lines → Rectangular Select left Top & Bottom edges → OK.

b. Utility Menu → Select → Entities → By Location → Y Co-ordinates → 50 → OK.

c. Define Load → Displacement → on node → Pick all → UY → Displacement Value = 0 → OK.

d. Utility Menu → Select → Entities → By Location → Select all → Cancel.

e. Define Load → apply → Structural → Force Moment → on Key point → pick Left Top Corner → enter the Value → FY → 100 → OK → pick the left bottom corner → enter the Value → FY → -100 → OK

f. Solve → Current L.S. → OK → Solution is done → Close

8. General Post proc

a. General Postproc → List Result → Nodal Solution → Stress → X – component → result copy word document past → OK.

b. General Postproc → Plot Results → Deformed Shape → Select Def + undeformed and click **OK**.

c. Utility Menu → Plot Ctrls → Hardcopy → To file → pick JPEG → save to (enter file name)

d. Utility Menu → Plot Ctrls → Animate → Deformed Shape... → Select Def + undeformed and click OK

e. General Postproc → Contour Plot → Nodal Solution → Stress → Vonmises Stress → OK

f. Utility Menu → Plot Ctrls → Hardcopy → To file → pick JPEG → save to (enter file name)

Inference:

With the help of modeling we can understand the use of axisymmetric component in engineering field (Eg: Boiler)

Using Stress–strain analysis (or stress analysis) is an engineering discipline covering methods to determine the stresses and strains in materials or structures subjected to forces or loads.

Result:

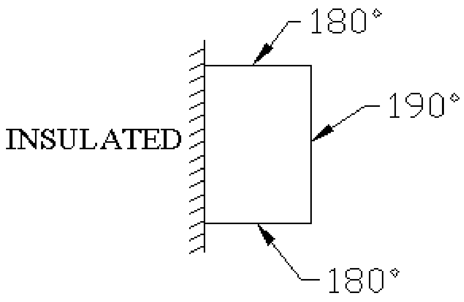
Thus the deformation shape and nodal stress diagram of an axisymmetric component is plotted using ANSYS software.

****

Ex No 6(a) *Date :*

Conductive heat transfer analysis of 2D component

Problem Specification:



Rectangular plate of Breadth = 0.4 m , height = 0.6 m. The upper and bottom edge of the plate temperature is taken as 180˚C. The right end of plate temperature is 190˚.C. Thermal conductivity = 1.5 w/mk.

Aim:

To find the temperature distribution on given plate

1. Preferences

a. Main Menu → Preferences → Thermal → OK.

2. Element Type

a. Element Type → Add/Edit/Delete → Add... → Pick Solid → Quad4node55 → OK

3. Materials Props

a. Material Props → Material Models.... → Thermal, Conductivity ,Isotropic → **KXX**, = 1.5Click **OK**. Material → Exit.

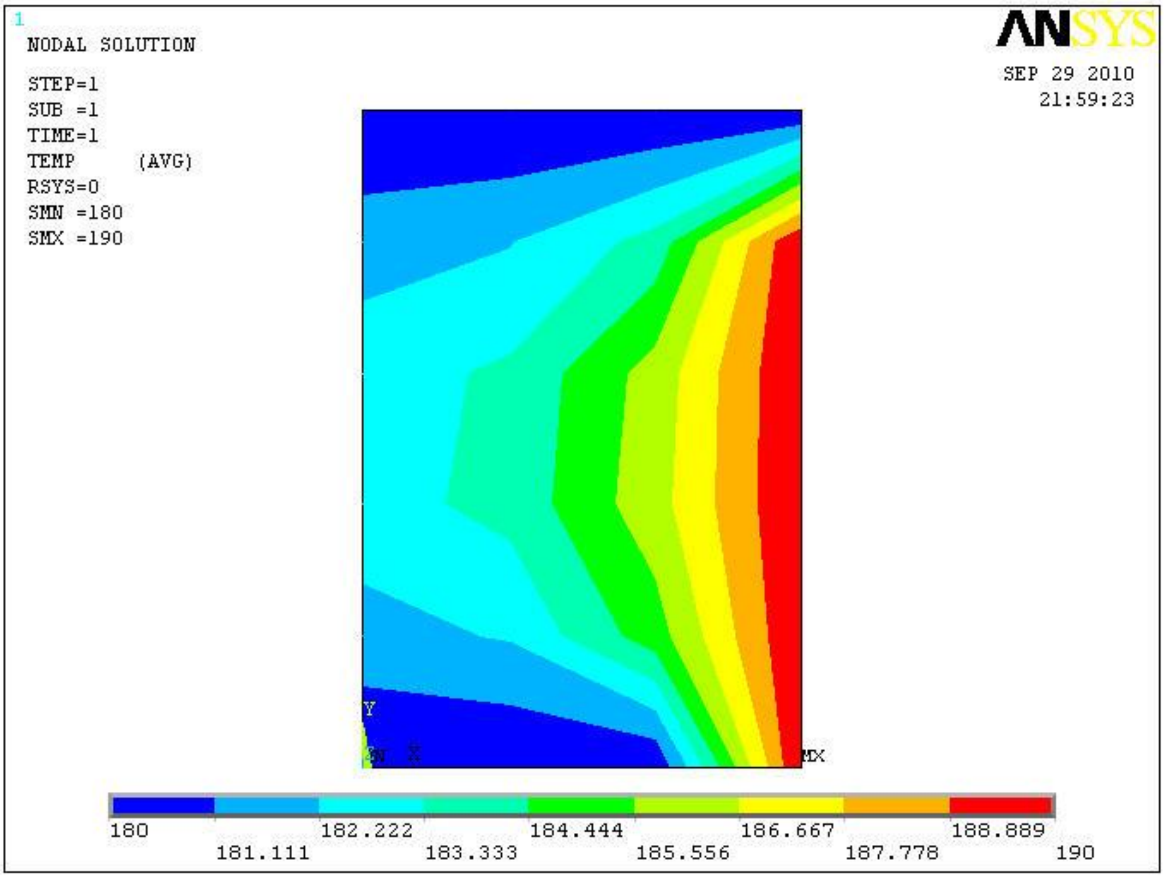
4. Modeling

a. Modeling → Create → Areas → Rectangle → By 2 Corner → X = 0,Y = 0 ,Width =0.4 , Height = 0.6 → OK.

5. Meshing

a. Meshing → Size Cntrls → Manual Size → Global → Size → 0.05 → OK

b. Mesh Tool → Mesh → Select Surface → OK


1
NODAL SOLUTION
STEP=1
SUB =1
TIME=1
TEMP (AVG)
RSYS=0
SMN =180
SMX =190
ANSYS
SEP 29 2010
21:59:23
Y
MN X
MX
180
181.111
182.222
183.333
184.444
185.556
186.667
187.778
188.889
190

6. Solution

a. Solution → Define Loads → Apply → Thermal → Heat Generate → On Lines → Select the Left Side of Rectangular Pick → OK → Value : 0 → OK.

b. Solution → Define Load → apply → Thermal → Temperature → Click the upper and bottom of the rectangular pick → OK → All DOF → Value : 180˚ → Right Side Pick → Value = 190˚ C → OK

c. Solution → Solve → Current L.S. → OK → Solution is done → Close

7. General Post proc

a. General Postproc → Plot results → Contour Plot → Nodal Solu → DOF Solution → Nodal Temperature → OK

Inference:

Thermal analysis is also often used as a term for the study of heat transfer through structures. Many of the basic engineering data for modeling such systems comes from measurements of heat capacity and thermal conductivity.

Result:

Thus the temperature distribution diagram is plotted using ANSYS software.

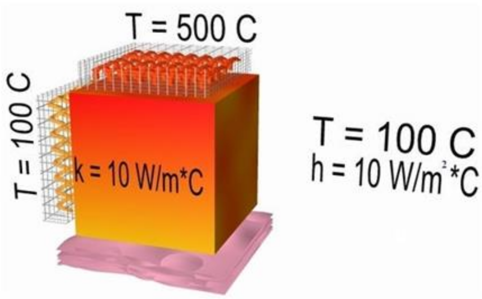
****

Ex No : 6(b) ***Date :***

Thermal analysis of 2D Plate (Conductive/Convective/Insulated)

Problem Definition

The Mixed Convection / Conduction / Insulated Boundary Conditions Example is constrained as shown in the following figure (Note that the section is assume to be in finitely long). Find the temperature distribution.



Aim:

To find the temperature distribution on the given plate using ANSYS software.

1.Preferences

a. Main Menu → Preferences → Thermal → OK.

2. Element Type

a. Element Type → Add/Edit/Delete → Add… → Pick Solid → Quad4node55 → OK

3. Materials Props

a.Material Props → Material Models…. → Thermal, Conductivity ,Isotropic → **KXX**, = 10Click **OK**. Material → Exit.

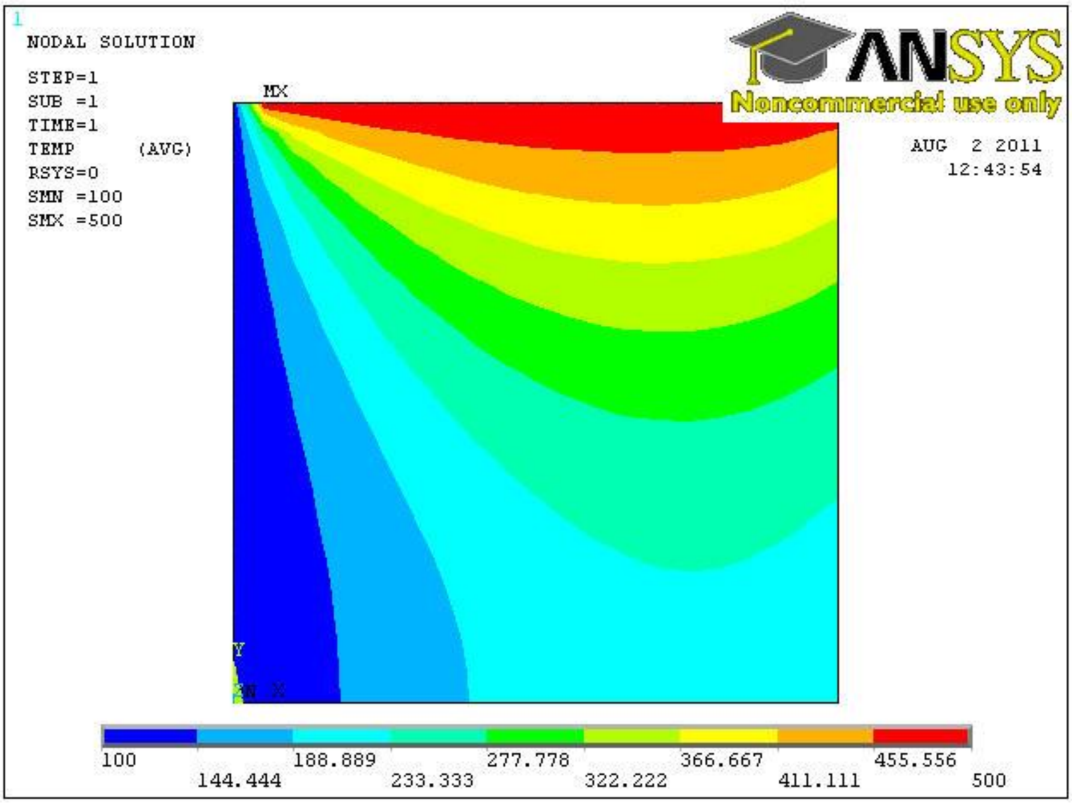
4. Modeling

a. Modeling → Create → Areas → Rectangle → By 2 Corner → X = 0,Y = 0 ,Width =1 , Height = 1 → OK.

5. Meshing

a. Meshing → Size Controls → Manual Size → Global → Size → Area Controls → length: 0.05 → OK

b. Mesh Tool → Mesh → Select Surface → OK


1
NODAL SOLUTION
STEP=1
SUB =1
TIME=1
TEMP (AVG)
RSYS=0
SMN =100
SMX =500
MX
ANSYS
Noncommercial use only
AUG 2 2011
12:43:54
Y
X
100
144.444
188.889
233.333
277.778
322.222
366.667
411.111
455.556
500

6. Solution

a. Solution → Define Loads → Apply → Thermal → On Lines → Select the Top Edge Rectangular Pick → OK → All DOF,Value:500 → OK.

b. Solution → Define Loads → Apply → Thermal → On Lines → Select the left Edge Rectangular Pick → OK → All DOF, Value: 100 → OK.

c. Solution → Define Load → apply → Convection → on lines → Select the Right Side Pick → OK → Film Coefficient : 10 , Bulk Temperature : 100 → OK.

d. Solution → Define Load → apply → Convection → on lines → Select the Bottom Side Pick → OK → Value:0,Value: 0 → OK.

e. Solution → Solve → Current L.S. → OK → Solution is done → close

7. General Post proc

a. General Postproc → Plot results → Contour Plot → Nodal Solu → DOF Solution → Nodal Temperature → OK

Inference:

Thermal analysis is also often used as a term for the study of heat transfer through structures. Many of the basic engineering data for modeling such systems comes from measurements of heat capacity and thermal conductivity.

Result:

Thus the temperature distribution diagram is plotted using ANSYS software.

****

Ex No : 7 ***Date :***

Thermal stress analysis of cylindrical shells

Problem Specification:

compute the temperature distribution in a long steel cylinder with inner radius 5 inches and outer radius 10 inches. The interior of the cylinder is kept at 75 deg F, and heat is lost on the exterior by convection to a fluid whose temperature is 40 deg F. The convection coefficient is 0.56 BTU/hr-sq.in-F and the thermal conductivity for steel is 0.69 BTU/hr-in-F.

EX = 3.E7 (psi)
DENS = 7.36E-4 (lb sec^2/in^4)
ALPHAX = 6.5E-6
PRXY = 0.3
KXX = 0.69 (BTU/hr-in-F)

Aim:

To find the temperature distribution on the given cylindrical shells using ANSYS software.

1.Preferences

a. Main Menu → Preferences → Thermal → OK.

2. Element Type

a. Element Type → Add/Edit/Delete → Add… → Thermal Solid → solid 8 node 77 → OK → options → elements behavior → Axi symmetric → OK

3. Materials Props

a. Material Props → Material Models…. → Thermal, Conductivity ,Isotropic → **KXX**, = 0.69 Density → 7.36e-4 → OK →

b. Favorites → Linear static → Linear isotropic → EX=3e7 → PRXY =0.3 → Thermal expansion → ALPHAX=6.5E-6 → Click **OK**. Material → Exit.

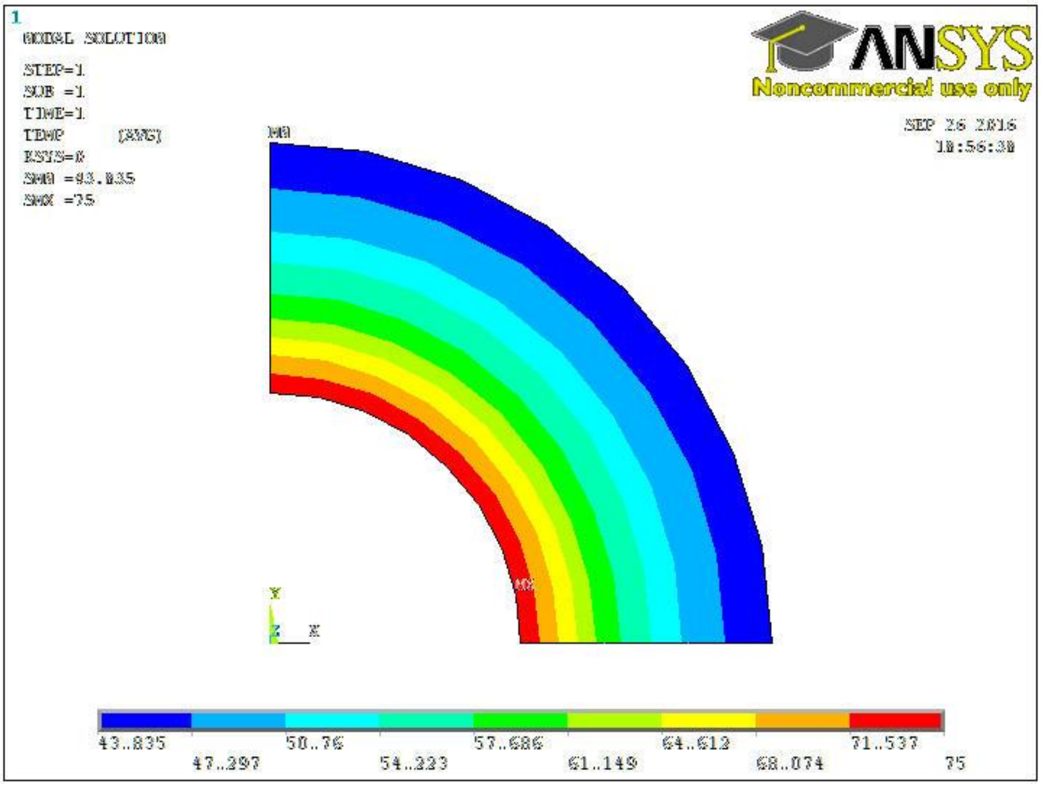
4. Modeling

a. Modeling → Create → Areas → circle → Partial Annulas→ X = 0,Y = 0 rad1 =5 → theta1 = 0 → rad2=10 → theta2 = 90 → OK.

5. Meshing

a. Meshing → Size Controls → Manual Size → Global → Size → Edge length =2 → → OK

b. Mesh Tool → Mesh →mapped → Select Surface → OK


1
NODAL SOLUTION
STEP=1
SUB =1
TIME=1
TEMP (AVG)
RSYS=0
SMN =43.835
SMX =75
MN
MX
ANSYS
Noncommercial use only
SEP 26 2016
10:56:30
Y
Z X
43.835
47.297
50.76
54.223
57.686
61.149
64.612
68.074
71.537
75

6. Solution

a. Solution → Define Loads → Apply → Thermal → on nodes → Select inner circular nodes → temperature =75 → OK →

b. Solution → Define Loads → Apply → Thermal → convection → On Lines → Select outer circular line → OK → Film Coefficient : 0.56 , Bulk Temperature : 40 → OK.

c. Solution → Define Load → apply → Convection → on lines → Select vertical and horizontal line → OK → Film Coefficient : 0 , Bulk Temperature : 0 → OK.

d. Solution → Solve → Current L.S. → OK → Solution is done → close

7. General Post proc

a. General Postproc → Plot results → Contour Plot → Nodal Solution → DOF Solution → Nodal Temperature → OK

Inference:

Thermal analysis is also often used as a term for the study of heat transfer through structures. Many of the basic engineering data for modeling such systems comes from measurements of heat capacity and thermal conductivity.

Result:

Thus the temperature distribution diagram is plotted using ANSYS software.

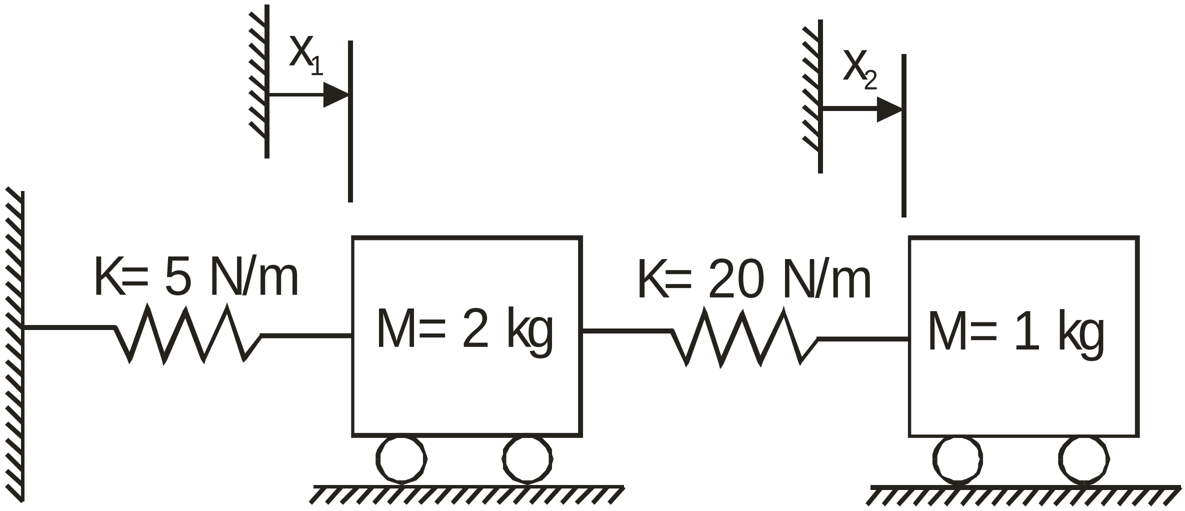
****

Ex No : 8 ***Date :***

Vibration analysis of spring mass system

Problem Specification:

Find the 2 mode frequency analysis of spring mass system



Notes: ANSYS has an option that will allow for the masses to be modeled as point masses. Using this option, no dimensions or material properties would be necessary for the masses. However, so that the mode shapes are easier to understand when they are animated, in this case, the masses will be modeled as blocks, 1 m x 1 m, with unit thickness. So, they have a volume of 1 m^3. The densities will be specified to produce the correct masses. Also, the free lengths of the springs are irrelevant in this analysis, as only the stiffnesses matter. But, they will both be assumed to have a length of 5 m.

1. Preferences

a. Click → Main Menu → Preferences → Click Structural → Click OK.

2. Element Type

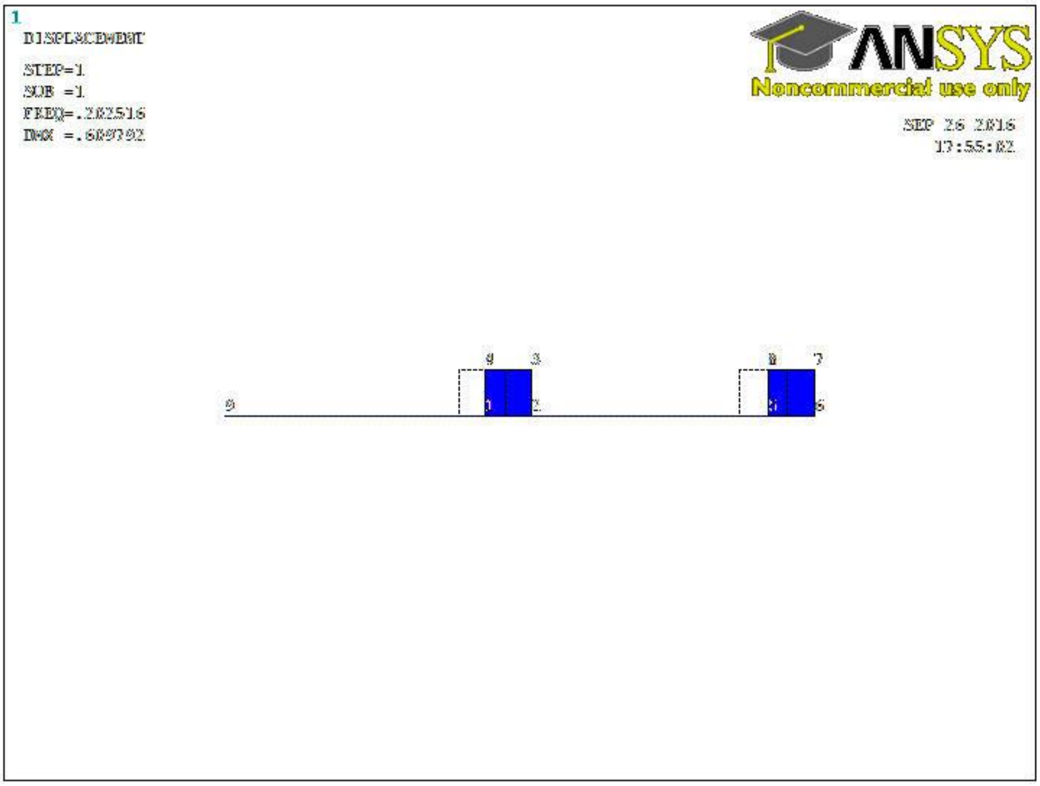
a. Element Type → Add / Edit / Delete → Add → Solid → Quad 4 node 42 → apply → Combination → Spring damper 14 → OK

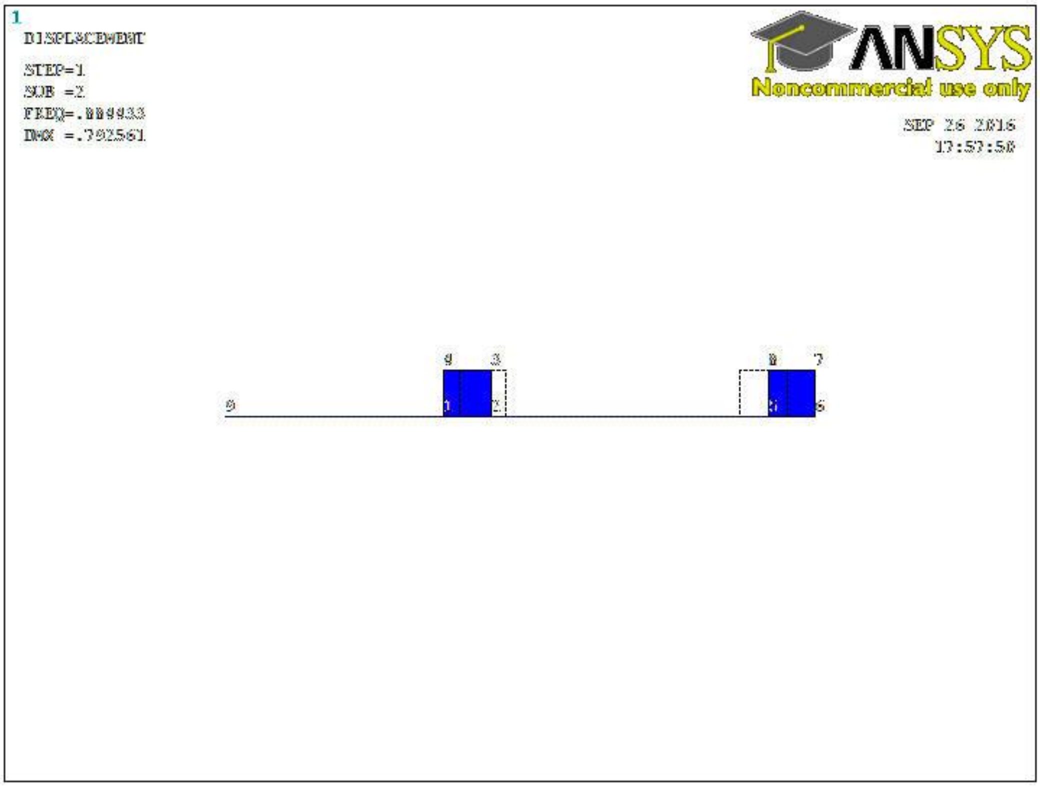
3. Real Constants

a. Real Constants → select element 2 → Add / Edit / Delete → Add → set no 1→K =5 →OK →Add → for set no 2→K=20→ OK

```
 ****  INDEX OF DATA SETS ON RESULTS FILE  *****

 SET   TIME/FREQ    LOAD STEP   SUBSTEP  CUMULATIVE
   1 0.20252             1          1          1
   2 0.88443             1          2          2
```





4. Materials Props

a. Preprocessor → Materials Props → Materials Models → Structural → Linear → Elastic → Isotropic → **Ex: 1** → **Prxy : 0.27** → Ok → density =2 → material→new model→ Structural → Linear → Elastic → Isotropic → **Ex: 1** → **Prxy : 0.27** → density =1 →Ok

Materials → Exit

5. Modeling

a. Modeling → Create → Areas →Rectangle →by 2 corners →enter X=5, Y=0, Width =1, Height =1→apply→ enter X=11, Y=0, Width =1, Height =1 → OK

6. Meshing

a. Meshing → Size Cntrls → Manual Size → Global → Size → Enter NDIV=1 → OK.

b. Mesh Tool → Mesh area → mapped →select left side area → OK

c. Modeling → create → element → element attributes →Material No.= 2

d. Mesh Tool → Mesh area → mapped →select right side area → OK

e. Plot controls → Window controls → window options → triad → not shown.

f. Plot → nodes.

g. Modeling → Create → Nodes → in active co systems → Node no 9 → OK

h. Modeling → create → element → element attributes →Element type 2 → Material No.1→real constant no.1→ OK

i. Modeling → create → element →Auto numbered →through nodes→pick node 9 and1→OK

j. Modeling → create → element → element attributes →Element type 2 → Material No.1→ real constant no.1→ → OK

k. Modeling → create → element →Auto numbered →through nodes→pick node 2 and 5→ OK

7. Solution

a. Solution → Define Loads → Apply → Structural → Displacement → on nodes → Select all nodes → select UY and UZ → OK

b. On nodes → select node 9 → select all DOF → OK

c. Preprocessor → coupling/ceqn → couple DOFS → select nodes1, 2, 3 ,4 → set UX NSET = 1→ OK

d. Couple DOFS → select nodes 5,6,7,8→ set UX NSET =2→ OK Ansys Type → New Ansys → Modal → OK

e. Ansys Options → PCG Lanczous → No. of Modes to extract 2 → Expand 2 → OK → OK.

f. Solve → current L.S→ OK.

8. General Post proc

a. General Post proc → Results Summery → the result copy Word past → OK.

b. Read Result → Select First Set → Plot Result → Deformed Shape → Click Def+ undeformed → OK

c. Plot Ctrls → Hard copy → to file JPG → OK

d. Read Result → Select Last Set → Plot Resutl → Deformed Shape → Click Def+ undeformed → OK

e. Plot Ctrls → Hard copy → to file JPG → OK

f. Plot Ctrls → Animate → Deformed Shape → Click Def + Undeformed → OK.

Inference:

Model analysis is used for finding the mode of natural frequency of a given member.

We can understand that the frequency of vibration depends on mass, geometry value and stiffness of that element.

Result:

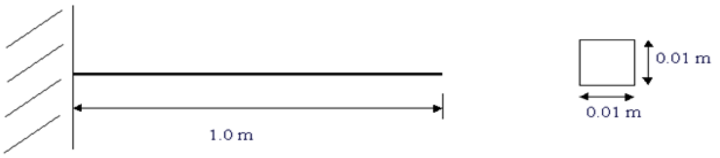
Thus we plotted the 2 mode frequency analysis of Spring Mass system using ANSYS software.

****

Ex No : 9(a) ***Date :***

Modal Analysis of Cantilever Beam

Problem Specification:



Modulus of Elasticity (E) = 206800(10^6) N/m²

Density = 7830 kg/m³

Find the 5 mode shape analysis of given Cantilever Beam.

E = 206800e6 N/m^2 ν = 0.27 Izz = 8.33e-10 m^4

Density σ = 7830 kg/m^3 a = 0.01m b = 0.01m

Area = 0.0001m^2

Aim:

To find 5 mode frequency analysis of Cantilever Beam using ANSYS software.

1.Preferences

a. Click → Main Menu → Preferences → Click Structural → Click OK.

2. Element Type

b. Element Type → Add / Edit / Delete → Add → Beam → 2D Elastic 3 → OK

3. Real Constants

a. Real Constants → Add / Edit / Delete → Add → OK

b. Enter → Area = 0.0001 ,Izz = 8.33e^{-10} , Height = 0.01 → Click OK.

4. Materials Props

a. Preprocessor → Materials Props → Materials Models → Structural → Linear → Elastic → Isotropic → **Ex: 206800e^6** → **Prxy : 0.27** → Ok

b. Density → 7830 → OK.

c. Materials → Exit

5. Modeling

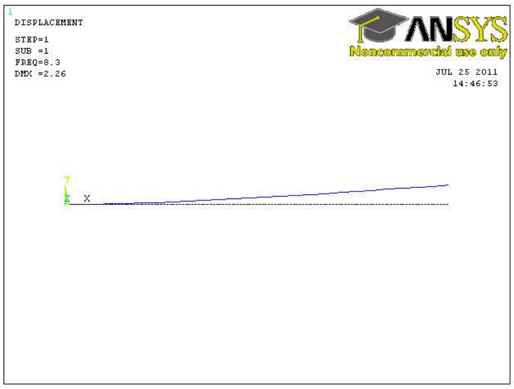
a. Modeling → Create → Key points → In active CS → Select Key point.

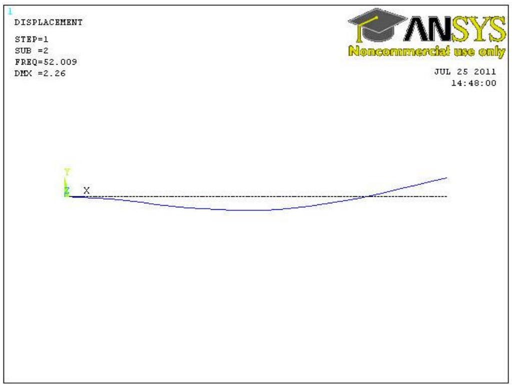
b. 1 → 0,0,0 2 → 1,0,0

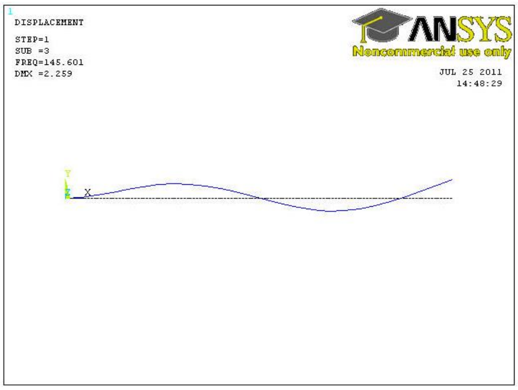
c. Lines → Lines → Straight Lines → Select Key point 1,2 → Click OK.

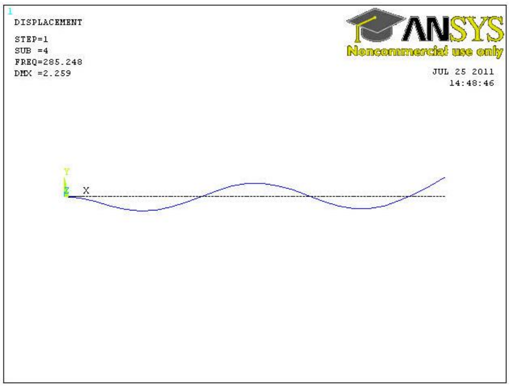
***** INDEX OF DATA SETS ON RESULTS FILE *****

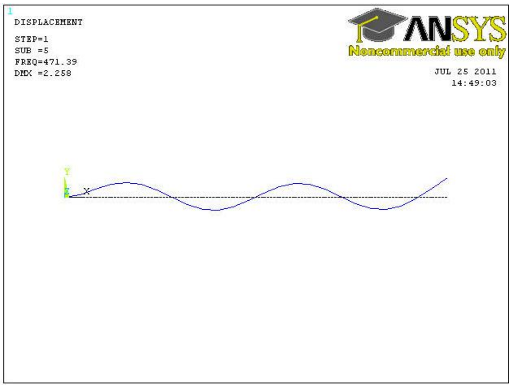
SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	8.2996	1	1	1
2	52.009	1	2	2
3	145.60	1	3	3
4	285.25	1	4	4
5	471.39	1	5	5











6. Meshing

a. Meshing → Size Cntrls → Manual Size → Global → Size → Enter the Element Division → 25 → OK.

b. Mesh Tool → Mesh → (Select the line) → OK

c. Plot Cltrs (Main Menu) → Style → Size and shape → Display of Element (ON).

7. Solution

a. Ansys Type → New Ansys → Model → OK

b. Ansys Options → PCG Lanczous → No. of Modes to extract 5 → Expand 5 → OK → OK.

c. Solution → Define Loads → Apply → Structural → Displacement → on Key Points → Select key Points 1 → Left Side Click OK → Select All DOF Constrained.

d. Solution → Solve → Current LS → OK.

8. General Post proc

a. General Post proc → Results Summery → the result copy Word past → OK.
b. Read Result → Select First Set → Plot Result → Deformed Shape → Click Def+ undeformed → OK
c. Plot Ctrls → Hard copy → to file JPG → OK
d. Read Result → Select Next Set → Plot Result → Deformed Shape → Click Def+ undeformed → OK
e. Plot Ctrls → Hard copy → to file JPG → OK
f. Read Result → Select Next Set → Plot Result → Deformed Shape → Click Def+ undeformed → OK
g. Plot Ctrls → Hard copy → to file JPG → OK
h. Read Result → Select Next Set → Plot Result → Deformed Shape → Click Def+ undeformed → OK
i. Plot Ctrls → Hard copy → to file JPG → OK
j. Read Result → Select Next Set → Plot Result → Deformed Shape → Click Def+ undeformed → OK
k. Plot Ctrls → Hard copy → to file JPG → OK
l. Plot Ctrls → Animate → Deformed Shape → Click Def + Undeformed → OK.

Inference:

Model analysis is used for finding the mode of natural frequency of a given member.

We can understand that the frequency of vibration depends on mass, geometry value and stiffness of that element.

Result:

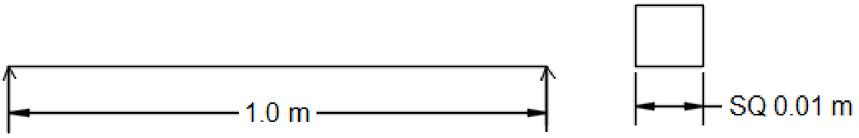
Thus we plotted the 5 mode shape analysis of Cantilever Beam using ANSYS software.

****

Ex No : 9 (b) ***Date :***

Model Analysis of Simply Supported Beam

Problem Specification:



Modulus of Elasticity (E) = 206800(10^6) N/m^2

Density = 7830 kg/m^3

E = 206800e6 N/m^2 ν = 0.27 Izz = 8.33e-10

Density σ = 7830 kg/m^3 a = 0.01m

b = 0.01m Area = 0.0001m^2

Aim:

To find 5 mode frequency analysis of simply supported beam using ANSYS software.

1. Preferences

a. Click → Main Menu → Preferences → Click Structural → Click OK.

2. Element Type

a. Element Type → Add / Edit / Delete → Add → Beam → 2D Elastic 3 → OK

3. Real Constants

a. Real Constants → Add / Edit / Delete → Add → OK

b. Enter → Area = 0.0001m ,Izz = 8.33e^{-10}m^4 , Height = 0.01m → Click OK.

4. Materials Props

a. Preprocessor → Materials Props → Materials Models → Structural → Linear → Elastic → Isotropic → **Ex: 206800e^6** → **Prxy : 0.27** → Ok

b. Density → 7830 → OK.

c. Materials → Exit

5. Modeling

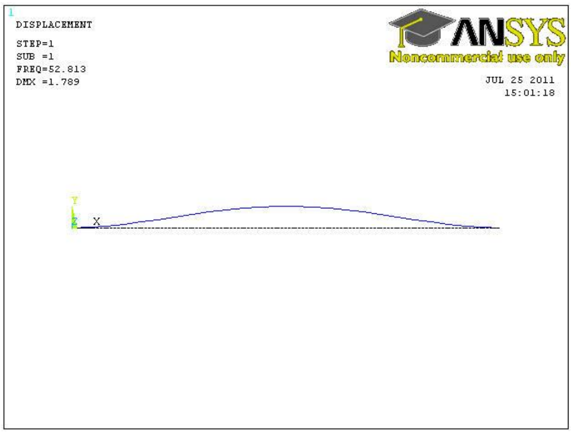
a. Modeling → Create → Key points → In active CS → Select Key point.

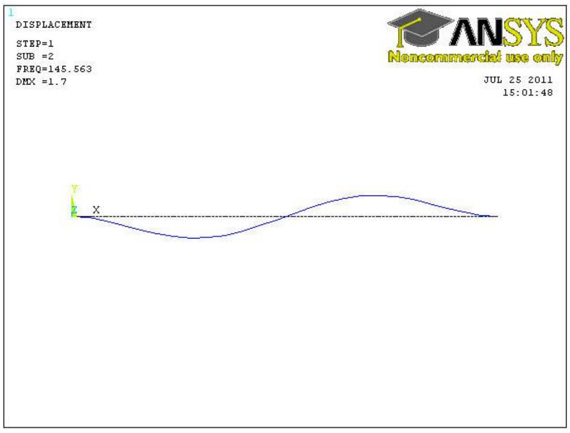
b. 1 → 0,0,0 2 → 1,0,0

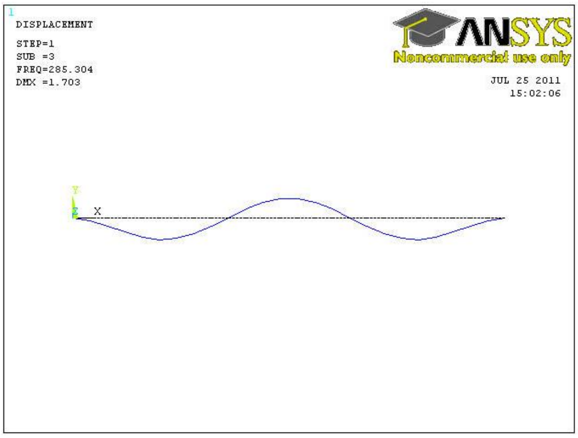
c. Lines → Lines → Straight Lines → Select Key point 1,2 → Click OK.

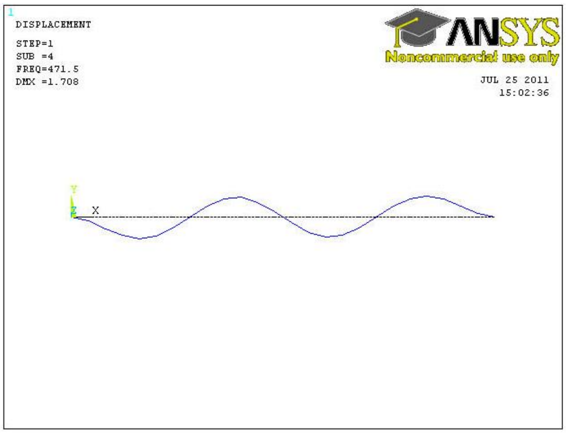
***** INDEX OF DATA SETS ON RESULTS FILE *****

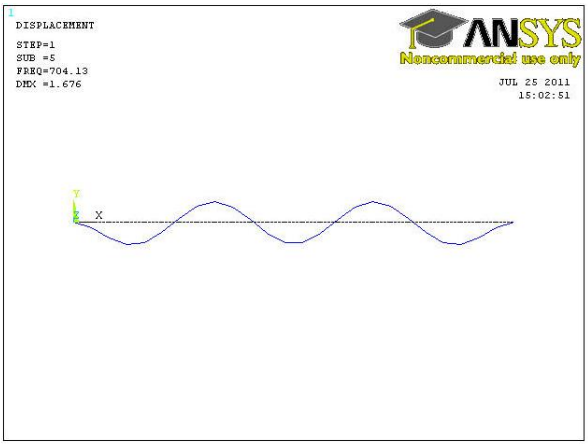
SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	52.813	1	1	1
2	145.56	1	2	2
3	285.30	1	3	3
4	471.50	1	4	4
5	704.13	1	5	5











6. Meshing

a. Meshing → Size Cntrls → Manual Size → Global → Size → Enter the Element Division → 25 → OK.

b. Mesh Tool → Mesh → (Select the line) → OK

c. Plot Cltrs (Main Menu) → Style → Size and shape → Display of Element (ON).

7. Solution

a. Ansys Type → New Ansys → Model → OK

b. Ansys Options → PCG Lanczous → No. of Modes to extract 5 → Expant 5 → OK → OK.

c. Solution → Define Loads → Apply → Structural → Displacement → on Key Points → Select key Points 1 → Left Side & 2 → Right Side Click OK → Select UX,UY Constrained.

d. Solution → Solve → Current LS → OK.

8. General Post proc

a. General Post proc → Results Summery → the result copy Word past → OK.
b. Read Result → Select First Set → Plot Resutl → Deformed Shape → Click Def+ undeformed → OK
c. Plot Ctrls → Hard copy → to file JPG → OK
d. Read Result → Select Next Set → Plot Resutl → Deformed Shape → Click Def+ undeformed → OK
e. Plot Ctrls → Hard copy → to file JPG → OK
f. Read Result → Select Next Set → Plot Resutl → Deformed Shape → Click Def+ undeformed → OK
g. Plot Ctrls → Hard copy → to file JPG → OK
h. Read Result → Select Next Set → Plot Resutl → Deformed Shape → Click Def+ undeformed → OK
i. Plot Ctrls → Hard copy → to file JPG → OK
j. Read Result → Select Next Set → Plot Resutl → Deformed Shape → Click Def+ undeformed → OK
k. Plot Ctrls → Hard copy → to file JPG → OK
l. Plot Ctrls → Animate → Deformed Shape → Click Def + Undeformed → OK.

Inference:

Model analysis is used for finding the mode of natural frequency of a given member.

We can understand that the frequency of vibration depends on mass, geometry value and stiffness of that element.

Result:

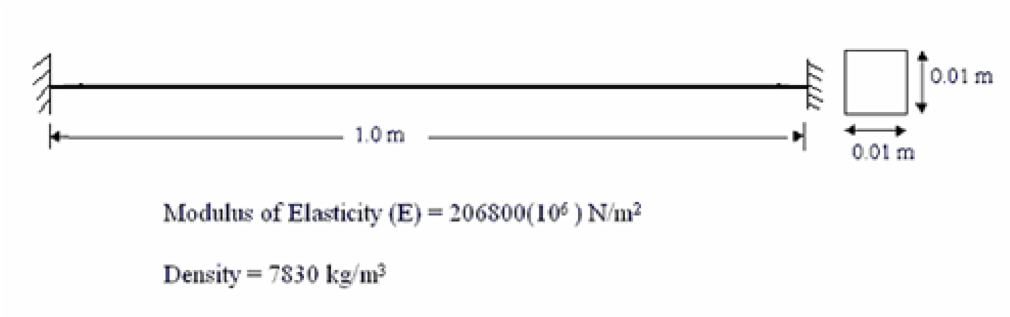
Thus we plotted the 5 mode frequency analysis of simply supported Beam using ANSYS software.

****

Ex No : 9(c) ***Date :***

Model Analysis of Fixed End Beam

Problem Specification:



Find the 5 mode frequency analysis of Fixed End Beam

E = 206800e6 N/m^2	ν = 0.27	Izz = 8.33e-10
Density σ = 7830 kg/m^3	a = 0.01m	
b = 0.01m	Area = 0.0001m^2	

Aim:

To find 5 mode frequency analysis of Fixed End Beam using ANSYS software.

1. Preferences

a. Click → Main Menu → Preferences → Click Structural → Click OK.

2. Element Type

a. Element Type → Add / Edit / Delete → Add → Beam → 2D Elastic 3 → OK

3. Real Constants

a. Real Constants → Add / Edit / Delete → Add → OK

b. Enter → Area = 0.0001 ,Izz = 8.33e^{-10} , Height = 0.01 → Click OK.

4. Materials Props

a. Preprocessor → Materials Props → Materials Models → Structural → Linear → Elastic → Isotropic → **Ex: 206800e6** → **Prxy : 0.27** → Ok

b. Density → 7830 → OK.

c. Materials → Exit

5. Modeling

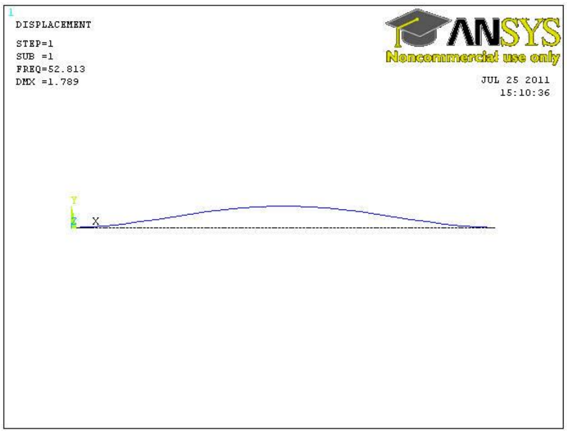
b. Modeling → Create → Key points → In active CS → Select Key point.

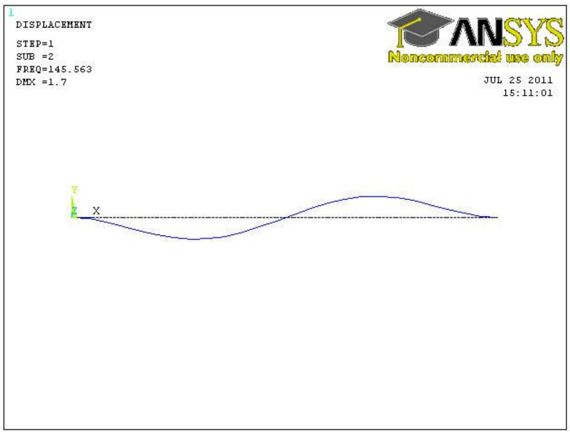
c. 1 → 0,0,0 2 → 1,0,0

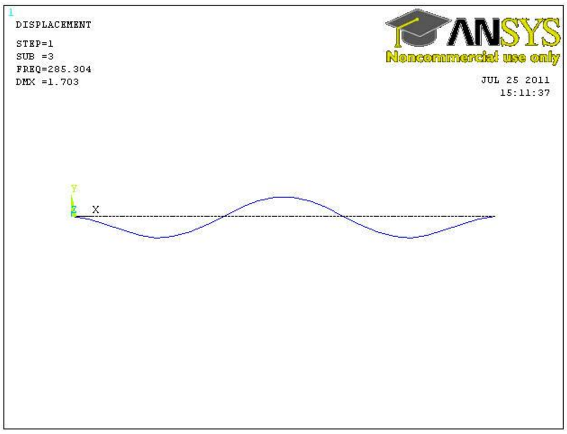
d. Lines → Lines → Straight Lines → Select Key point 1,2 → Click OK.

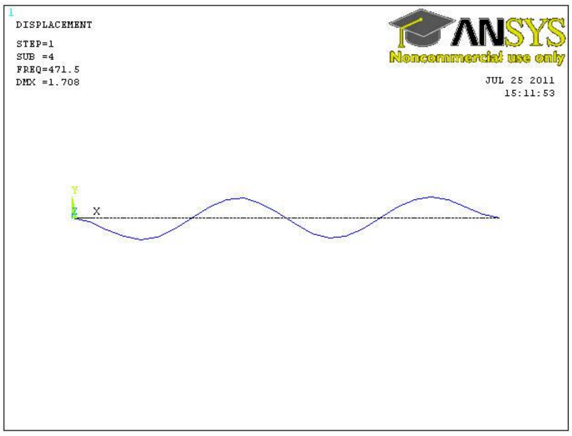
***** INDEX OF DATA SETS ON RESULTS FILE *****

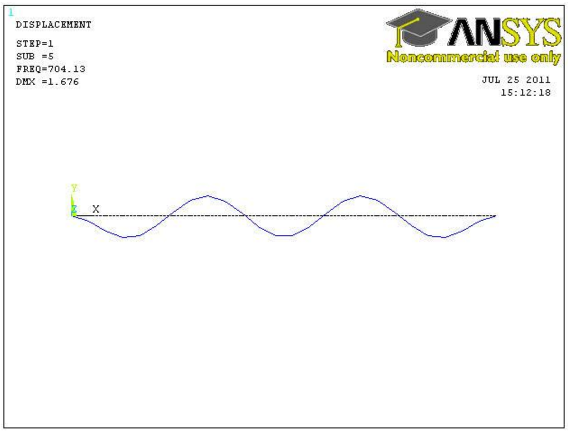
SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	52.813	1	1	1
2	145.56	1	2	2
3	285.30	1	3	3
4	471.50	1	4	4
5	704.13	1	5	5











6. Meshing

l. Meshing → Size Cntrls → Manual Size → Global → Size → Enter the Element Division → 25 → OK.

m. Mesh Tool → Mesh → (Select the line) → OK

n. Plot Cltrs (Main Menu) → Style → Size and shape → Display of Element (ON).

7. Solution

a. Ansys Type → New Ansys → Model → OK
b. Ansys Options → PCG Lanczous → No. of Modes to extract 5 → Expant 5 → OK → OK.
c. Solution → Define Loads → Apply → Structural → Displacement → on Key Points → Select key Points 1 → Left Side & 2 → Right Side Click OK → Select All DOF Constrained.
d. Solution → Solve → Current LS → OK.

8. General Post proc

a. General Post proc → Results Summery → the result copy Word past → OK.
b. Read Result → Select First Set → Plot Resutl → Deformed Shape → Click Def+ undeformed → OK
c. Plot Ctrls → Hard copy → to file JPG → OK
d. Read Result → Select Next Set → Plot Resutl → Deformed Shape → Click Def+ undeformed → OK
e. Plot Ctrls → Hard copy → to file JPG → OK
f. Read Result → Select Next Set → Plot Resutl → Deformed Shape → Click Def+ undeformed → OK
g. Plot Ctrls → Hard copy → to file JPG → OK
h. Read Result → Select Next Set → Plot Resutl → Deformed Shape → Click Def+ undeformed → OK
i. Plot Ctrls → Hard copy → to file JPG → OK
j. Read Result → Select Next Set → Plot Resutl → Deformed Shape → Click Def+ undeformed → OK
k. Plot Ctrls → Hard copy → to file JPG → OK
l. Plot Ctrls → Animate → Deformed Shape → Click Def + Undeformed → OK.

Inference:

Model analysis is used for finding the mode of natural frequency of a given member.

We can understand that the frequency of vibration depends on mass, geometry value and stiffness of that element.

Result:

Thus we plotted the 5 mode frequency analysis of Fixed End Beam using ANSYS software.

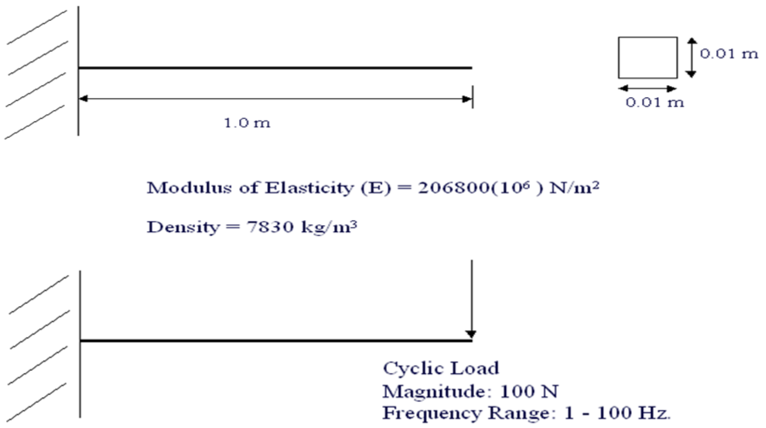
****

Ex No : 10 ***Date :***

Harmonic Analysis Of Cantilever Beam

Problem Specification:

We will now conduct a harmonic forced response test by applying a cyclic load (harmonic) at the end of the beam. The frequency of the load will be varied from 1-100Hz.The figure below depicts the beam with the application of the load.



E = $206800e^{6}$ N/m^2 ν = 0.27 Izz = 833e-12

D = 7830Kg/m^3 a = 0.0001m^2 h = 0.01 m

Aim:

To find the harmonic analysis of Cantilever Beam using ANSYS software.

Procedure:

1. Preferences

a. Click → Main Menu → Preferences → Click Structural → Click OK.

2. Materials Props

a. Preprocessor → Materials Props → Materials Models → Structural → Linear → Elastic → Isotropic → **Ex: 206800e6** → **Prxy : 0.27** → Ok

b. Density → 7830 → OK.

c. Materials → Exit

3. Element Type

a. Element Type → Add / Edit / Delete → Add → Beam → 2D Elastic 3 → OK

4. Real Constants

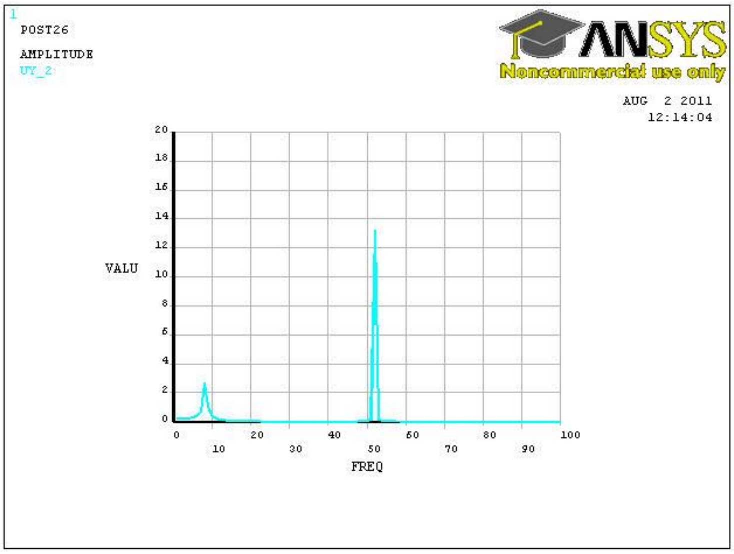
a. Real Constants → Add / Edit / Delete → Add → OK

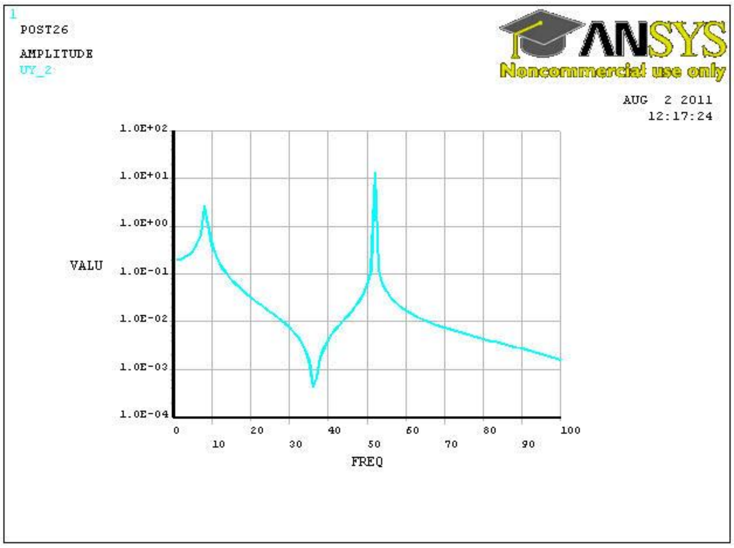
b. Enter → Area = 0.0001m^2 ,Izz = 833e-12 , Height = 0.01m → Click OK.

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

TIME	2 UY	
	UY_2	
	AMPLITUDE	PHASE
1.0000	0.196269	0.00000
2.0000	0.205086	0.00000
3.0000	0.221743	0.00000
4.0000	0.250351	0.00000
5.0000	0.300534	0.00000
6.0000	0.399155	0.00000
7.0000	0.656309	0.00000
8.0000	2.65172	0.00000
9.0000	1.06273	180.000
10.000	0.410080	180.000
11.000	0.242418	180.000
12.000	0.166333	180.000
13.000	0.123260	180.000
14.000	0.957484E-01	180.000
15.000	0.767763E-01	180.000
16.000	0.629764E-01	180.000
17.000	0.525351E-01	180.000
18.000	0.443899E-01	180.000
19.000	0.378782E-01	180.000

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





5. Modeling

a. Modeling → Create → Key points → In active CS → Select Key point.

b. 1 → 0, 0, 0   2 → 1, 0, 0

c. Lines → Lines → Straight Lines → Select Key point 1, 2 → Click OK.

6. Meshing

a. Mesh Tool → Set line → Enter the Element Division → 25 → OK.

b. Mesh Tool → Mesh → (Select the line) → OK

7. Solution

a. Solution → Analysis Type → New Analysis → Harmonic ANTYPE, 3 → OK.

b. Solution → Analysis Type → Analysis Options. → OK → OK

c. Solution → Define Loads → Apply → Structural → Displacement → on key point → Click Left Side keypoint → OK → Select All DOF Constrained → OK.

d. Select Solution → Define Loads → Apply → Structural → Force/Moment → on key point → Click Right Side key Point → OK → Direction of Force – FY → Real Part of Force → 100 ; Image Part of Force – 0 → OK

e. Solution → Load Step Opts → Time/Frequency → Freq and Substps... → Harmonice Freq Range – 0 – 100; Number of Sub step – 100 → Select the Stepped → OK.

f. Solution → Solve → Current LS SOLVE → Solution done.

8. TimeHistPostpro

a. SelectTimeHistPostpro → Click Add Data → Click DOF Solution → Click Y – Component of displacement → OK → Cantilever Beam Right Side Point Click → OK

b. Select the List Data → Data List → Close

c. Graph Data → Displace Hard Copy → to file JPG → OK

d. UtilityMenu → PlotCtrls → Style → Graphs → ModifyAxis → Select the – LOGY Y – axis Scale = Linear Change the Logarithmic → OK

e. UtilityMenu>Plot>Replot

Inference:

Harmonic analysis is used for finding the mode of natural frequency of a given member in dynamic and/ or for dynamic loading condition.

We can understand that the frequency of vibration depends on mass, geometry value and stiffness of that element.

Result:

Thus the harmonic analysis is conducted on cantilever beam using ANSYS software and the results are plotted.

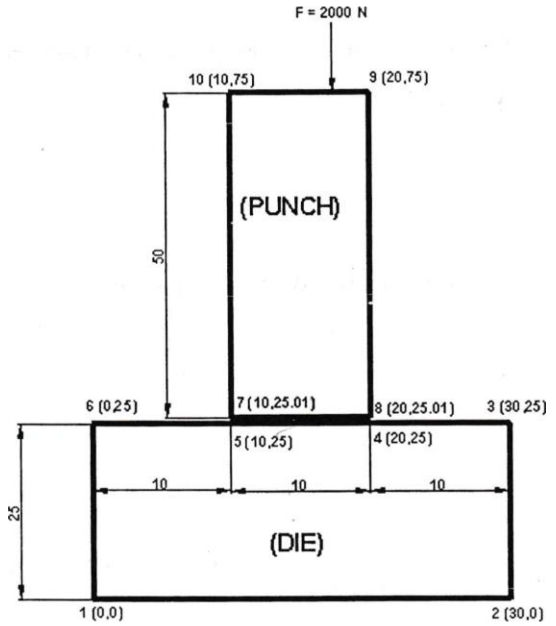
****

Ex No : ***Date :***

Non-Linear Analysis Using Contact Element

Problem:

A set of punch and die with dimensions is as shown in the figure. The thickness for both punch and die is 0.1 mm. A force of 2000 N is applied at top of the punch. Analyze the stress distribution in punch as well die using contact elements.



GUI Solution:

1. Starting:

Click → start → ANSYS → ANSYS product launcher

Launch →ANSYS Multiphysics

File management → browse the directory for saving and retrieving the files. Click → run

2. Preferences → tick → structural → select h-method → ok

<u>**Title:**</u>

Utility menu → File → Change title → “Machine part” ok.

Utility menu →-> plot →> replot

4. Elements:

Main menu → Preprocessor → element type → add → add → Structural → Solid → quad 4node (PLANE42) → ok

Set options → pane stress with thickness → ok

Main menu → Preprocessor → element type → add → add → Contact → 2D target (TARGE 169) → apply → 2 nd surf 171 (CONTA171) → ok

5. <u>Real Constant:</u> (Contact material only needs Real Constants)

(Punch is contact Die is Target)

Main menu → Preprocessor → real constant → add → Type 1 → ok → enter thickness as 0.1 → ok

→ add → Type 2 → ok → enter RI = 0.01 & R2 = 0.01 → ok

→ add → Type 3 → ok → enter RI = 0.01 & R2 = 0.01 → ok → close

6. Material Properties:

(Main material only needs material properties)

Main menu → Preprocessor → Material prop → Material models → Material model no 1 → Structural → Linear -» Elastic -> isotropic -> -> enter EX = 2.0e+05; NUXY = 0.3 -> ok.

<u>7.Modeling</u>:

Main menu → Preprocessor → create → keypoints → in active CS →

enter KPI = (0,0),

KP2 = (30,0),

KP3 = (30,25),

KP4 = (20,25),

KP5 = (10,25),

KP6 = (0,25),

KP7 = (10,25.01),

KP8 = (20,25.01),

KP9 = (20,75),

KP10 = (10,75) → ok

Main menu → Preprocessor → create → lines → st. lines → select KPI & KP2 → apply →

select KP2 & KP3 → apply →

select KP3 & KP4 → apply →

select KP4 & KP5 → apply →

select KP5 & KP6 → apply →

select KP6 & KPI → apply →

select KP7 & KP8 → apply →

select KP8 & KP9 → apply →

select KP9 & KP10 → apply →

select KP10 & KP7 → ok

Utility menu → plotctrls → numbering → line no → on

Utility menu → plot → lines

Main menu → Preprocessor → modeling create → areas → arbitrary → by lines →

pick LI, L2, L3, L4, L5, L6 → apply →

pick L7, L8, L9, L10 → ok.

Utility menu → plotctrls → numbering → line no → on

Utility menu → select → everything

Utility menu → plot → area

Main menu → Preprocessor → meshtool → set → lines →

select L3, L4, L7, L5, L9 → apply →

enter no. of division = 8 → apply

select LI, L2, L6 → apply → enter no. of division = 10 → apply

select L8, L10 → apply → enter no. of division = 15 → apply → ok

Main menu → Preprocessor → meshtool → mesh → area → pickall → ok.

8. Creating Contact Elements:

Utility menu → select → entities → lines → by num/pick → apply → select L7 (bottom line of the punch) → apply → ok

Utility menu → select → entities → nodes → attached to → lines, all → apply →> ok

Utility menu → Utility menu → plot → nodes

Main menu → Preprocessor → modeling → create → elements →

elements attributes select type 2 ;

mat no 1 ;

real constant set no 2 → ok

Main menu → Preprocessor → modeling → create → elements → Auto numbered → through nodes → pick adjacent two nodes at time → apply

Utility menu → select → entities → lines → by num/pick → apply → select L4 (line on die below bottom line of the punch) → apply → ok.

Utility menu → select → entities → nodes → attached to → lines, all → apply → ok

Utility menu → plot → nodes

Main menu → Preprocessor → modeling → create → elements → elements attributes → select element type 3 ; mat no 1 ; real constant set no 3 → ok.

Main menu → Preprocessor → modeling →create → elements → Auto numbered → through nodes → pick adjacent two nodes at time → apply

Utility menu → select → everything

Utility menu → plot multiplot

Utility menu → select → entities → lines → by num/pick → apply → select L7 & L4 together → apply → ok.

Utility menu → select → entities → nodes → attached to → lines, all → apply → ok

Utility menu plot → nodes

Preprocessor → Coupling/Ceqn → Couple DOF → pickall → set reference no 1 and select Uy → ok **(Contact Manag we can add friction)**

Utility menu → select → everything

Utility menu → plot → multiplot

9. Boundary Conditions:

Utility menu → select → entities → lines → by num/pick → apply → select bottom most line → apply → ok.

Utility menu → select → entities → nodes → attached to → lines, all → apply → ok

Utility menu → plot → nodes

Main menu → Solution → loads → apply → displacement → on nodes → select pickall → ok → select all DOF ok.

Utility menu → Select → everything

Utility menu → plot → multiplot

Main menu → Solution → loads → apply → force → on nodes → select last but one node from right on the top of the punch → ok → enter Fy = -2000 → ok

10. Non Linear Settings:

Main menu → solution → analysis type → new analysis → transient → full → ok

Main menu → solution → analysis type → solun contrl → basic →

Analysis option → large displacement transient

Time control:

Time at the end of load step → 1

Automatic time stepping → on

Time increment → enter 0.01, 0.001, 0.25

Write items to result file→ All solution items

Frequency → for every substep

Main menu → solution analysis type → solun contrl → Non-Linear → equilibrium iterations = 25.

Main menu → Solution → solve → current LS

11. Post Processing:

Gen. post processor → plot results nodal solutions → stress → von-mises → ok

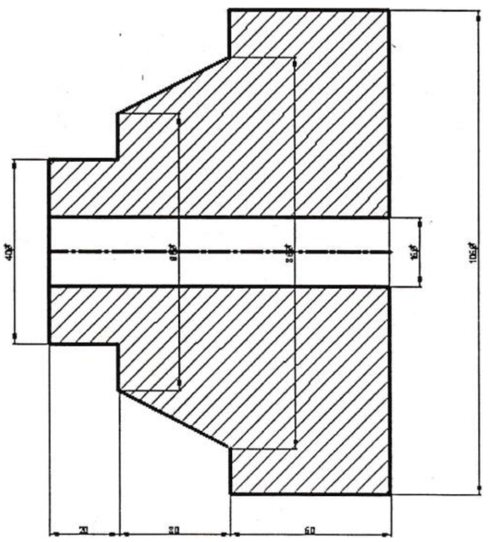
Ex No : ***Date :***

3D Static Analysis of a Pulley (Applying Angular Velocity and Torque)

Applying Torque by Mass Element

Problem:

A pulley is rotating at 1000 rpm and require to transmit power of 5 kW. The dimensions of the pulley are shown in the figure. The material properties are young's modulus is 210 GPa (2.1X103 N/mm2) and Poisson's ratio is 0.3 and density is 7800 kg/m3. Analyze for the von-mises stress.



GUI Solution:

1. Starting:

Click → start →ANSYS → ANSYS product launcher

Launch → ANSYS Multiphysics

File management → browse the directory for saving and retrieving the files. Click → run

2. Preferences → tick → structural → select h-method → ok

Title:

Utility menu → File → Change title → "Machine part" → ok.

Utility menu → plot replo

3. Elements:

Preprocessor → element type → add → add → structural solid → qudra 8 noded (PLANE42) → apply → add → brick with rotate (SOLID73) (not available in ANSYS v9 directly) → apply → add → 3D mass (MASS21) → apply

3.Real Constants:

Preprocessor → real const → add → Qdd → select MASS 21 → ok → enter set no = 3,

MASSX = 0.0001,

MASSY = 0.0001, MASSZ = 0.0001,

IXX = 0.0001,

IYY = 0.0001,

IZZ = 0.0001 → ok

6. Material Properties:

Preprocessor → material prop → constant → isotropic →enter the material no as 1 → ok → enter the following values:

EX = 2.1e+05;

NUXY = 0.3 → DENS = 7800e-09 → ok

7. Modeling:

Material ID2 ;

Ex = 1 ;

NUXY =0.3 ;

DENS 0.0001-> ok

a. Main menu → Preprocessor → Modeling → create → keypoints → in active CS →Enter the

   KPI = (0,0),

   KP2 = (100,0),

   KP3 = (0,7.5),

   KP4 = (0,20),

   KP5 = (20,20),

   KP6 = (20,30),

   KP7 = (50,42.5),

   KP8 = (50,52.5),

   KP9 = (100,52.5),

   KP10 = (100,7.5) → C ok.

b. Main menu → Preprocessor → Modeling → create → lines → st. lines → pick the keypoints by sequence from KP3 to KP10 → ok.

c. Main menu → Preprocessor → Modeling → create → areas → arbitrary → pick all the lines → ok.

d. Main menu → Preprocessor → meshtool → size cntrl → manual size → size → enter SIZE = 2 → ok.
e. Main menu → Preprocessor → attributes → define → default attributes → set Element type → 1, Mat. ID → 1 → ok.
f. Main menu → Preprocessor → Mesh tool → mesh → area → pick all → ok. NO' visions
g. Main menu → Preprocessor → meshtool → set global → No. of divisions = 8 → ok.
h. Main menu → Preprocessor → attributes → define → default attributes → set Element type→ 2, SOLID73, Mat. ID → 1 ok.
i. Main menu → Preprocessor → Modeling → operate → extrude → area → about axis → pick all → enter the key points on the axis about which it is revolved 360, 4 **3D object will be created.**
j. Main menu → Preprocessor → attributes → define → default attributes → set Element type → 1 Mat. ID → 1 → ok
k. Utility menu → select → elements → by attributes → Elem num type → enter min, max = 1 → from, full → apply → ok.
l. Utility menu → plot → elements
m. Main menu → Preprocessor → Meshtool → select areas → clear → Select area corresponding 2D element → ok
n. Main menu → Preprocessor → Modeling → create → nodes → in active CS → enter node no as 50000 (more than the maximum number of node) and x = 100, y = 0, z = 0 →ok
o. Main menu → Preprocessor → modeling → create → element →element attributes → set element as 3, Mat ID as 1, Real cont as 1 → ok
p. Main menu → create → element → through → auto numbered → through nodes → pick 50000th node → ok.

8. Application of Boundary conditions:

Menu bar → select → entities areas → by num/pick → apply → select small end of the shaft which is fixed → apply → ok.

Select → entities → nodes → attached to → areas, all → apply → ok.

Menu bar → plot → nodes.

Solution → loads → apply → displacement → on nodes → pickall → apply all DOF except ROTX (about which shaft rotate) → ok.

Menu bar → Select everything.

Menu bar → plot → multiplot.

9. Application of Angular velocity:

Solution → loads → apply → others → angular velocity → enter the value of angular velocity (ω_x) = (2x π x 1000/60 = 104 r/s) → ok.

10.Application of Torque:

Torque is applied at the other end of the shaft at 52.5 mm radius

Calculation of Torque:

$$p = \frac{2\pi NT}{60000} ; \frac{P * 6000}{2\pi N} = \frac{5 * 60000}{2\pi * 1000} = 47.7\ Nm = 47746.5\ Nmm$$

Coupling the Nodes:

Utility menu → select → entities → node → by num/pick → apply → enter 11)000 → ok.

Utility menu → plot → nodes.

Utility menu → plot → lines

Utility menu → select → entities → lines → by num/pick → apply → select front end circle lines → ok.

Utility menu → select → entities → node → attached to → lines, all → also select apply → ok.

Utility menu → plot → nodes

Main menu → Preprocessor → coupling/Ceqn → Rigid region → enter master node as 50000 → apply select the circumferential nodes on the circle of the shaft → ok.

In constraint equation for regid region table → enter Ld of -> all applicable, Ldof2 → slave rotate ROTZ, LdofB → slave rotate ROTZ, Ldof4 → slave rotate ROTZ

(the direction in which axis shaft rotates) → ok

Solution → solve → current LS

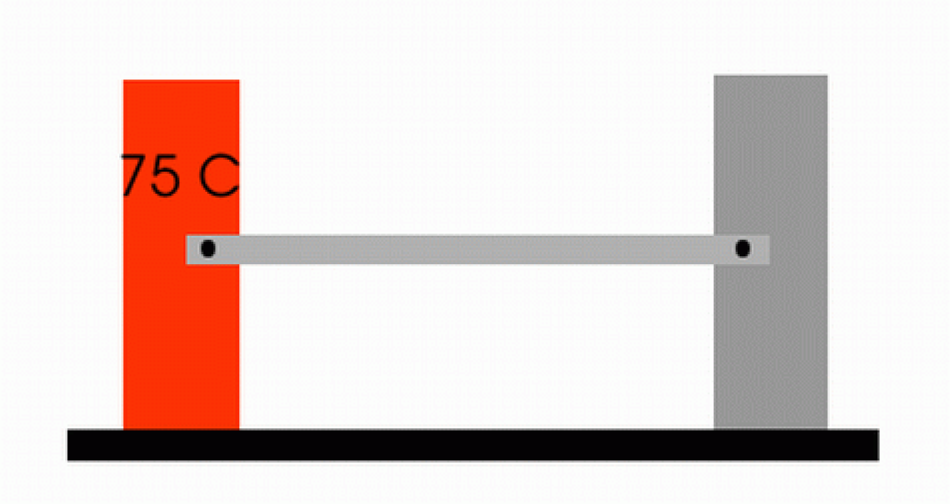
11. General Post processing:

Gen. post processor → plot results → nodal solution → stress → von-mises →

Ex No : 11 (CBS) ***Date :***

Coupled Structural/Thermal Analysis

Problem Specification:



Length of the link = 1m

Area = 4e-4 m

Element = Link 33(Thermal Mass Link 3D conduction)

KXX = 60.5

Mesh Edge length = 0.1 m

Element = Link 8

EX : 200e9

PRXY : 0.3

ALPX : 12e-6

When the input of one physics analysis depends on the results from another analysis, the analyses are coupled."

Thus, each different physics environment must be constructed seperately so they can be used to determine the coupled physics solution. However, it is important to note that a single set of nodes will exist for the entire model. By creating the geometry in the first physical environment, and using it with any following coupled environments, the geometry is kept constant. For our case, we will create the geometry in the Thermal Environment, where the thermal effects will be applied.

Although the geometry must remain constant, the element types can change. For instance, thermal elements are required for a thermal analysis while structural elements are required to deterime the stress in the link. It is important to note, however that only certain combinations of elements can be used for a coupled physics analysis.

Thermal Environment - Create Geometry and Define Thermal Properties

1. Give example a Title

Utility Menu > File > Change Title ... `/title, Thermal Stress Example`

2. Open preprocessor menu

ANSYS Main Menu > Preprocessor `/PREP7`

3. Define Keypoints

Preprocessor > Modeling > Create > Keypoints > In Active CS... `K,#,x,y,z`

We are going to define 2 keypoints for this link as given in the following table:

Key point	Coordinates (x,y,z)
1	(0,0)
2	(1,0)

4. Create Lines

Preprocessor > Modeling > Create > Lines > Lines > In Active Coord `L,1,2`

Create a line joining Keypoints 1 and 2, representing a link 1 meter long.

5. Define the Type of Element

Preprocessor > Element Type > Add/Edit/Delete...

For this problem we will use the LINK33 (Thermal Mass Link 3D conduction) element. This element is a uniaxial element with the ability to conduct heat between its nodes.

6. Define Real Constants

Preprocessor > Real Constants... > Add...

In the 'Real Constants for LINK33' window, enter the following geometric properties:

i. Cross-sectional area AREA: 4e-4

This defines a beam with a cross-sectional area of 2 cm X 2 cm.

7. Define Element Material Properties

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

In the window that appears, enter the following geometric properties for steel:

i. KXX: 60.5

8. Define Mesh Size

Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > All Lines...

For this example we will use an element edge length of 0.1 meters.

9. Mesh the frame

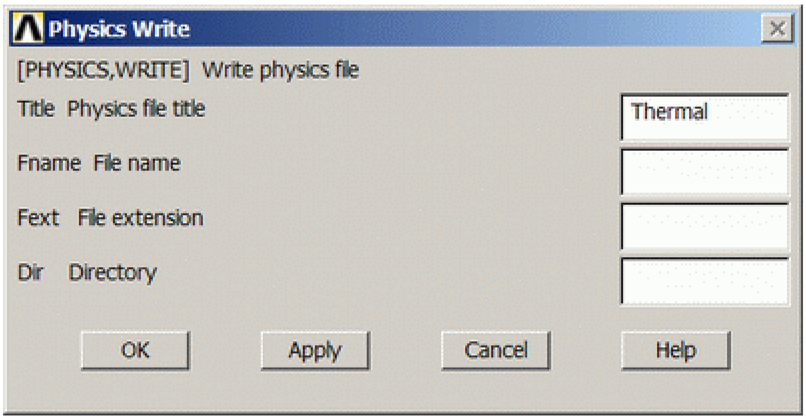
Preprocessor > Meshing > Mesh > Lines > click 'Pick All'

10. Write Environment

The thermal environment (the geometry and thermal properties) is now fully described and can be written to memory to be used at a later time.

Preprocessor > Physics > Environment > Write

In the window that appears, enter the TITLE **Thermal** and click OK.



11. Clear Environment

Preprocessor > Physics > Environment > Clear > OK

Doing this clears all the information prescribed for the geometry, such as the element type, material properties, etc. It does not clear the geometry however, so it can be used in the next stage, which is defining the structural environment.

Structural Environment - Define Physical Properties

Since the geometry of the problem has already been defined in the previous steps, all that is required is to detail the structural variables.

1.Switch Element Type

Preprocessor > Element Type > Switch Elem Type

Choose **Thermal to Struc** from the scoll down list.

This will switch to the complimentary structural element automatically. In this case it is LINK 8. For more information on this element, see the help file. A warning saying you should modify the new element as necessary will pop up. In this case, only the material properties need to be modified as the geometry is staying the same.

2.Define Element Material Properties

Preprocessor > Material Props > Material Models > Structural > Linear > Elastic > Isotropic

In the window that appears, enter the following geometric properties for steel:

i. Young's Modulus EX: 200e9
ii. Poisson's Ratio PRXY: 0.3

Preprocessor > Material Props > Material Models > Structural > Thermal Expansion secant Coefficient > Isotropic

i. ALPX: 12e-6

3.Write Environment

The structural environment is now fully described.
Preprocessor > Physics > Environment > Write

In the window that appears, enter the TITLE **Struct**

Solution Phase: Assigning Loads and Solving

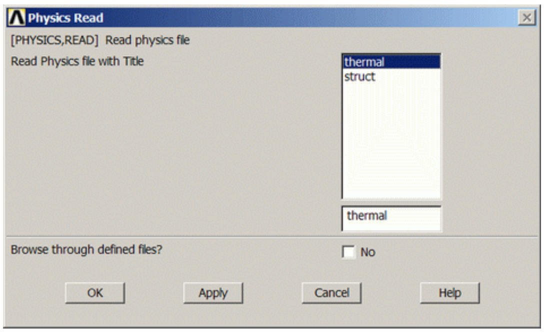
1. Define Analysis Type

Solution > Analysis Type > New Analysis > Static `ANTYPE,0`

2. Read in the Thermal Environment

Solution > Physics > Environment > Read

Choose **thermal** and click OK.



If the Physics option is not available under Solution, click **Unabridged Menu** at the bottom of the Solution menu. This should make it visible.

1. Apply Constraints

Solution > Define Loads > Apply > Thermal > Temperature > On Keypoints

Set the temperature of Keypoint 1, the left-most point, to 348 Kelvin.

2. Solve the System

Solution > Solve > Current LS > `SOLVE`

3. Close the Solution Menu

Main Menu > Finish It is very important to click **Finish** as it closes that environment and allows a new one to be opened without contamination. If this is not done, you will get error messages.

The thermal solution has now been obtained. If you plot the steady-state temperature on the link, you will see it is a uniform 348 K, as expected. This information is saved in a file labelled `Jobname.rth`, were .rth is the thermal results file. Since the jobname wasn't changed at the beginning of the analysis, this data can be found as **file.rth**. We will use these results in determing the structural effects.

1. Read in the Structural Environment

Solution > Physics > Environment > Read

Choose **struct** and click OK.

2. Apply Constraints

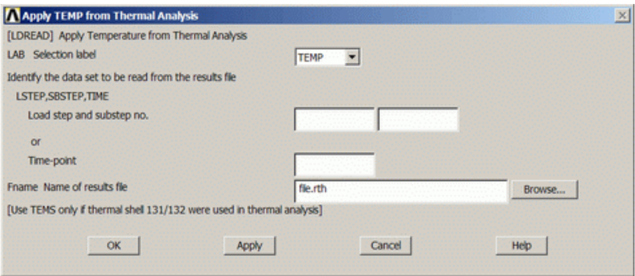
Solution > Define Loads > Apply > Structural > Displacement > On Keypoints

Fix Keypoint 1 for all DOF's and Keypoint 2 in the UX direction.

3. Include Thermal Effects

Solution > Define Loads > Apply > Structural > Temperature > From Therm Analy

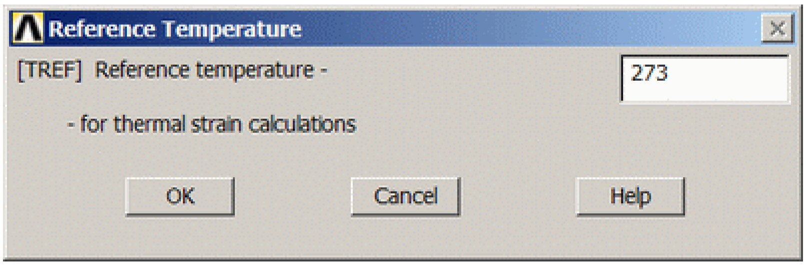
As shown below, enter the file name `File.rth`. This couples the results from the solution of the thermal environment to the information prescribed in the structural environment and uses it during the analysis.



4. Define Reference Temperature

Preprocessor > Loads > Define Loads > Settings > Reference Temp

For this example set the reference temperature to 273 degrees Kelvin.



5. Solve the System

Solution > Solve > Current LS

```
SOLVE
```

Postprocessing: Viewing the Results

1. Hand Calculations

Hand calculations were performed to verify the solution found using ANSYS:

Expansion due to thermal stress in a link can be calculated using:

$$\delta = \alpha \Delta T L$$

Expansion due to structural forces can be determined using:

$$\delta = \frac{PL}{EA}$$

Soving for the structural forces due to the thermal expansion,

$$P = \alpha \Delta T E A$$

Or

$$\sigma = \frac{F}{A} = \alpha \Delta T E$$

Therefore, in this example

$$\sigma = (0.000012/\mathrm{K})(348\ \mathrm{K} - 273\ \mathrm{K})(200e3\ \mathrm{MPa}) = 180\ \mathrm{MPa}$$

As shown, the stress in the link should be a uniform 180 MPa in compression.

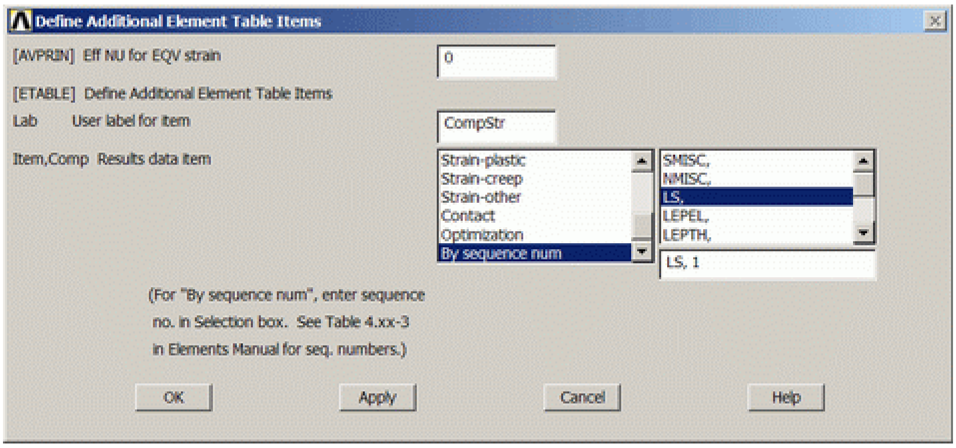
2. Get Stress Data

Since the element is only a line, the stress can't be listed in the normal way. Instead, an element table must be created first.

General Postproc > Element Table > Define Table > Add

Fill in the window as shown below. [CompStr > By Sequence Num > LS > LS,1

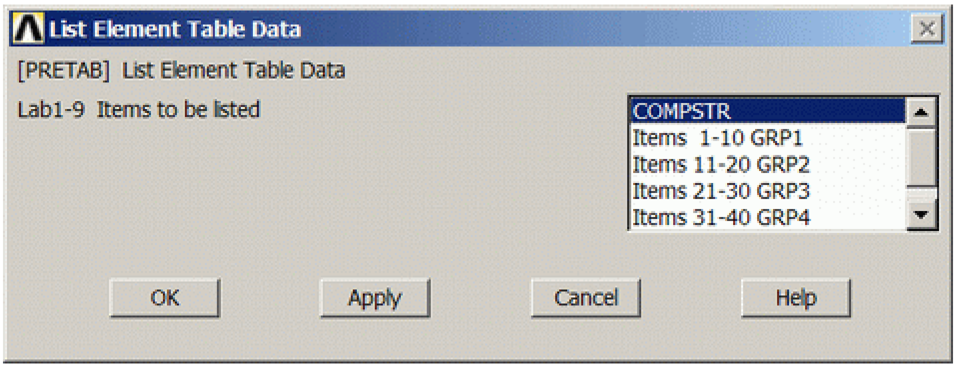
```
ETABLE,CompStress,LS,1
```



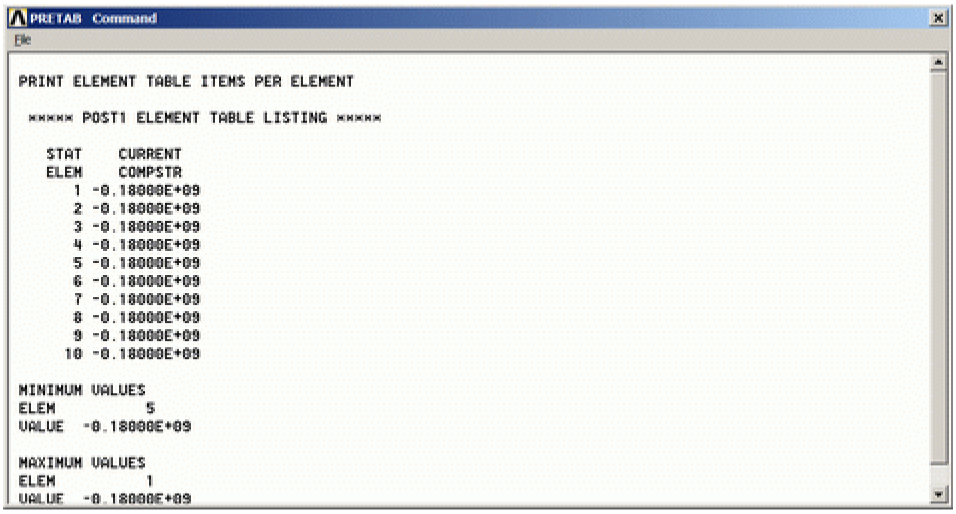
3. List the Stress Data

General Postproc > Element Table > List Elem Table > COMPSTR > OK

```
PRETAB,CompStr
```



The following list should appear. Note the stress in each element: -0.180e9 Pa, or 180 MPa in compression as expected.



Result:

Thus the coupled analysis for thermal and structural is conducted using ANSYS software and the results are plotted.

****

PROGRAM OUTCOMES (POs)

Mechanical Engineering Graduates will be able to

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

K.L.N. COLLEGE OF ENGINEERING

VISION

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

MISSION

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

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