Structural Dynamics

Project Report

Under Guidance of

Univ.-Prof. Dr.- Ing. habil.Carolin Birk

Dr. rer. nat.Hauke Gravenkamp

University of Duisburg Essen
Computational Mechanics
PrudhviPriyanka Bhogavalli
Matr Nr ES03022760
March 20, 2016

Table of Contents

Objectivity:

- The objective of the project is to study behavior of gallery in a concert hall under forced vibration (Man Induced Vibration) without damping.
- To find the first three natural frequencies and natural mode shapes of the cantilever corresponding to deformations of the beam in the x-y-plane.
- Finding the natural frequency of the Gallery of two proposed designs and finding the best design based on their natural frequencies by comparing with exact natural frequencies of a uniform cantilever beam.
- To find the Transient Response for the given load at the tip of the Cantilever Beam.

Classification of vibration

• Free Vibration:

A system is left to vibrate on its own after an initial disturbance and no external force acts on the system.
E.g. simple pendulum

• Forced Vibration:

A system that is subjected to an external force.
E.g. vehicle travels on the road

Resonance occurs when the frequency of the external force coincides with one of the natural frequencies of the system

• **Undamped Vibration:**

When no energy is lost or dissipated in friction or other resistance during oscillations

• Damped Vibration:

When any energy is lost or dissipated in friction or other resistance during oscillations

• Linear Vibration:

When all basic components of a vibratory system, i.e. the spring, the mass and the damper behave linearly

• Nonlinear Vibration:

If *any* of the components behave nonlinearly

• Deterministic Vibration:

If the value or magnitude of the excitation (force or motion) acting on a vibratory system is known at any given time

• Nondeterministic or Random Vibration:

When the value of the excitation at a given time cannot be predicted:

Finite Element Method:

• It is one of the most important developments in applied mechanics
• Governing differential equations of distributed-mass systems are replaced by a system of ordinary differential equations
• Applicable to a wide range of problems
• Introduction with reference to one-dimensional structural finite elements

The basic model requirements

• Generation of the finite element mesh
• Selection of material model and input material parameters
• The boundary conditions specification

Units in FE Program

FE program does not have built-in unit systems.
Units must be consistent.

Several modelling issues

• Understand the physical process of the real world problem
• What are the important aspects to model (seepage, stress, consolidation, static/dynamic,..)
• To start, model only essential aspects. Your predication will not be closer to real world problem than your model.
• Simplify material behavior
• Linear elastic material model is often sufficient for evaluating serviceability
• Always perform a linear analysis before performing a nonlinear analysis
• Choose the simplest model based on understanding of the real world problem.
• Quick hand calculations to estimate the results
• Critically evaluate the results and debug the calculation.

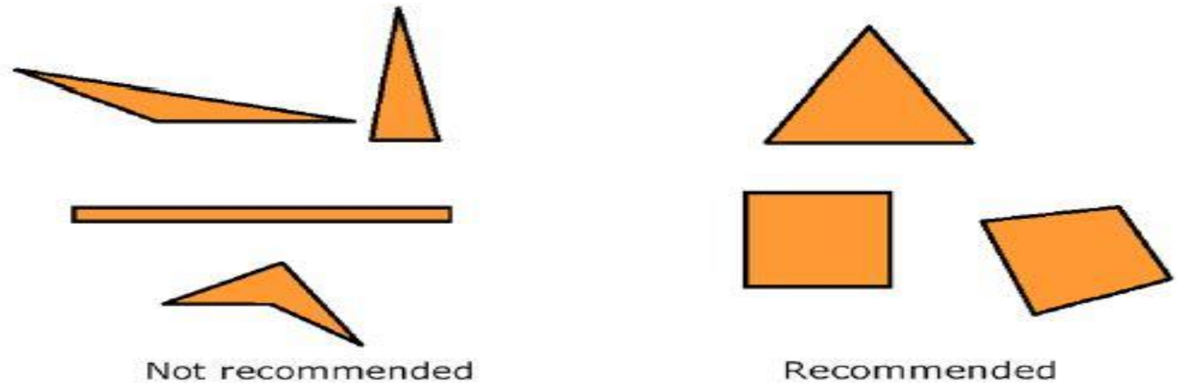
Finite element types

• ANSYS has more than 150 element types
• One-dimensional problems (2 node bar and beam)
• Two-dimensional problems (3-9 nodes elements)
• Three-dimensional problems (4-27 nodes elements)
• Plate elements
• Shell elements

Selection of element type

"Widely-accepted wisdom"
• Using higher order elements
• When bending behavior predominates
• At the location of stress concentration (footing corners, etc.)
• Higher order elements are usually more effective.
• Quadrilateral elements are more effective than corresponding triangular elements, but are more difficult to create.

Mesh quality
Quadrilateral elements are preferable to triangular elements due to accuracy consideration
• Keep triangular elements as equilateral and quadrilateral elements as square as possible.



Finite element mesh generation
Complex problem domains are divided into Regions
• Triangular and quadrilateral regions are commonly used.
• ANSYS: Using points to create surfaces/volumes
Regions are divided into elements according to a specific mesh pattern.

<u>Structured mesh:</u>
Consistent pattern, location of internal nodes and element sizes

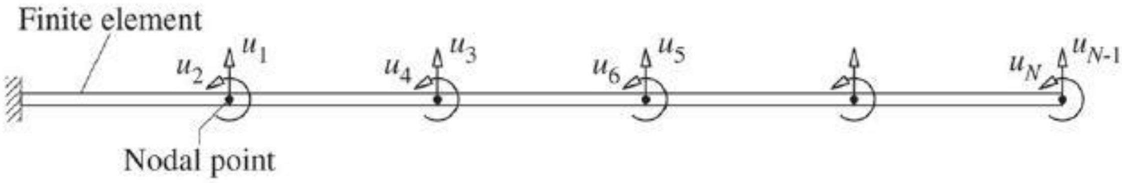
<u>Unstructured mesh:</u>
Location of boundary nodes are controlled

Boundary Condition (BC)
• Types of BC in mathematical model for stress analysis
• Displacement,
• Pressure or force, and
• Mixed (p=k*u

FEM for Structural Dynamics

• A dynamic finite element analysis requires considerably more understanding than a static analysis
• Number of modes = number of degrees of freedom
• Lower modes are modelled with higher accuracy
• Transient analysis = combination of modes

Finite Element Approximation



• Cantilever beam is subdivided into a number of segments = finite elements
• Elements are interconnected at nodes
• Beam: each nodes has 2 DOFs, transverse displacement and rotation.
Deflections are expressed as a linear combination of several trial functions

$$u(x) = \sum u_i \psi_i(x)$$

Properties of trail functions: unit value at the DOF, zero value at all other DOFs continuous with continuous first derivative

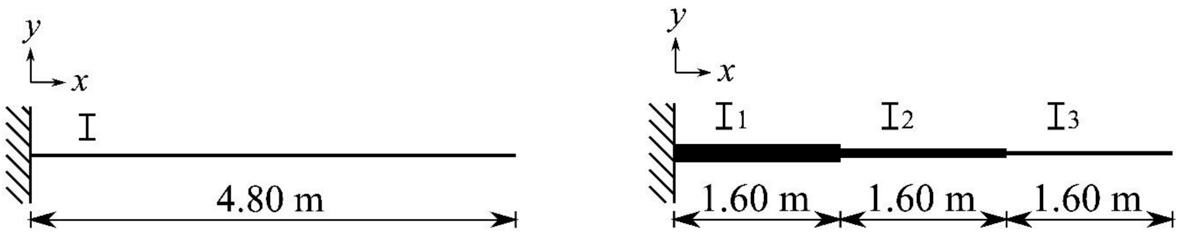
Analysis Procedure

• assemble the element matrices to determine the stiffness and mass matrices and applied force vector for the assemblage of FE
• Equations of motion for FE assemblage

$$\mathbf{m}\ddot{\mathbf{u}} + \mathbf{c}\dot{\mathbf{u}} + \mathbf{k}\mathbf{u} = \mathbf{p}(t)$$

• solve by methods developed previously
  - Classical modal analysis
  - Numerical Integration

Task1



Design (a)

Design (b)

Figure 1: Gallery in a concert hall

A gallery in a concert hall is to be designed. Two architectural designs have been proposed, which can both be idealized as cantilever systems. Design (a) is based on a constant cross- section as shown in Figure 1a), whereas in design (b) the cross-section is piece-wise constant, as shown in Figure 1b). It is your task to evaluate the two designs with respect to their dynamic properties.

Develop a numerical model of the cantilever system by following these steps:

1. Discretize the structure into 3 beam elements of length 1.60 m. Indicate the degrees of freedom of your structural model.
2. Using a programming language of your choice (or a spreadsheet), calculate the stiffness matrix k and mass matrix m of the system. The developed computer program (or spreadsheet) has to be explained and documented in your report.
3. For both designs, find the natural frequencies of the gallery

   by solving

$$E = 210 \times 10^9 \frac{N}{m^2}, \nu = 0.28, \rho = 7850 \frac{kg}{m^3}$$

4. Verify your computational model by comparing the natural frequencies computed for design (a) to the exact natural frequencies of a uniform cantilever beam. These can be found, for example, in A.K. Chopra, 'Dynamics of structures', 4th edition, Prentice Hall, 2012, Section 17.3.2

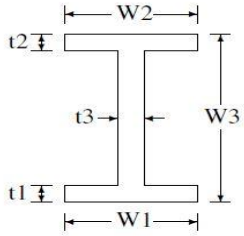


Figure 2: Beam cross-section

Cross-section	W_1 [mm]	W_2 [mm]	W_3 [mm]	t_1 [mm]	t_2 [mm]	t_3 [m]
Design (a)	160	160	330	8	8	11
Design (b), I1	160	160	300	8	8	10
Design (b), I2	140	140	260	7	7	10
Design (b), I3	120	120	220	6	6	10

Table 1: Cross-sectional dimensions in [mm]

Additional information:

All members have cross-sections as shown in Figure 2. The detailed cross-sectional

Dimensions are given in Table 1. All beams are steel members with

$$E = 210\times10^9 \frac{N}{m^2}, \nu = 0.28, \rho = 7850 \frac{kg}{m^3}$$

The cantilever beam carries a 70 mm thick concrete slab. The tributary width of the Cantilever beam is b = 3.0 m. The additional mass due to that concrete slab and any permanent fittings, such as seating is estimated at 500 kg/m. The concrete slab does not contribute to the bending stiffness.

When the gallery is in use, a total of 38 seats will fit on the contributory area of 3.0m ×4.80m. The average weight of one person is 75kg. Consider the additional mass of the spectators and work out how the natural frequencies of the gallery change if the concert hall is sold out. In this preliminary study it is sufficient to assume that the mass of all spectators is distributed uniformly along the length of the cantilever beam.

Note:

- Neglect axial deformations ($E A \rightarrow \infty$).

- For simplicity, assume that the neutral axis of each of the three different cross-sections of design (b) all coincide with the x-axis (i.e. no eccentricity).

- Neglect damping.

Description of Steps

Using FEM, we will find the natural frequency of the cantilever beam (continuous system). The basic procedure is outlined here.

1. In the first step, the geometry is divided into a number of small elements. The elements may be of different shapes and sizes.

2. Then elemental equations are obtained for each element.

3.In the third step the elemental equations are assembled to yield a system of global equations.

4. Then assembled equations are modified by applying the prescribed boundary conditions.

5. The last step is to find the solution from these modified equations.

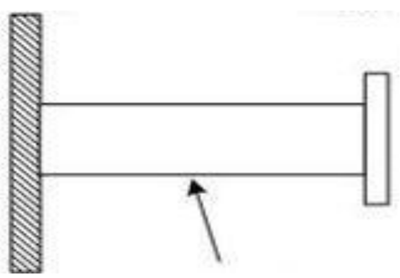


Fig. (a): A cantilever beam

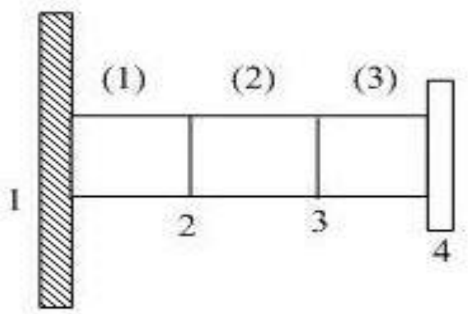


Fig. (b): Discretization of the beam into elements

For a uniform beam, the elemental stiffness matrix

$$[k]^e = \frac{EI}{l^3}\begin{bmatrix} 12 & 6l & -12 & 6l \\ 6l & 4l^2 & -6l & 2l^2 \\ -12 & -6l & 12 & -6l \\ 6l & 2l^2 & -6l & 4l^2 \end{bmatrix}$$

and the consistence mass matrix is given as

$$[M]^e = \frac{\rho A l}{420}\begin{bmatrix} 156 & 22l & 54 & -13l \\ 22l & 4l^2 & 13l & -3l^2 \\ 54 & 13l & 156 & -22l \\ -13l & -3l^2 & -22l & 4l^2 \end{bmatrix}$$

1. First step: Divide the given beam geometry into three small elements as shown in Fig (b). In Fig (b), numbers 1, 2, 3 and 4 represent the nodes.

2. Second step: Construct the elemental equations. For the first element, from equations above, we have

$$\frac{\rho A l}{420}\begin{bmatrix} 156 & 22l & 54 & -13l \\ 22l & 4l^2 & 13l & -3l^2 \\ 54 & 13l & 156 & -22l \\ -13l & -3l^2 & -22l & 4l^2 \end{bmatrix}\{\ddot{X}\} + \frac{EI}{l^3}\begin{bmatrix} 12 & 6l & -12 & 6l \\ 6l & 4l^2 & -6l & 2l^2 \\ -12 & -6l & 12 & -6l \\ 6l & 2l^2 & -6l & 4l^2 \end{bmatrix}\{X\} = \{F\}$$

Here $\{X\}$ represents the vector of nodal variables (i.e. at the first and second nodes). In this case node variables are displacement and slope, the right hand side vector $\{F\}$, represents the force

3. Third step:

Assemble all the elemental matrices to form a global matrix.

The length of the each element

$l = 1.6$ *m*

Total Length of the beam

$L=4.8$

Area of the beam

$A = 6014\ mm^2$

Mass density of the beam material

$\rho = 7850\ Kg/m^3$

Young's modulus of the beam $E = 2.1 \times 10^{11}\ N/m$.

After substituting values of the l, ρ, d, E, A we are able to find to find the global stiffness matrix of Design(a) and similarly for Design(b) can be found

and for free vibration

$$\{\ddot{X}\} = -\omega_{nf}^2 \{X\}$$

Hence, above equations can be written as

$$([M]\omega_{nf}^2 + [K])\{X\} = \{0\}$$

$$\text{Det } [K-\lambda M] = \omega^2\, 0$$

Where $\lambda = \omega^2$

Here it can solved like a Eigen value problem and natural frequencies can be found

Here the natural frequencies are find out by using MATLAB Program

MATLAB Code

```
% Moment of Inertia for Design (A)
E = 210 * 10^9;  % N/m^2
rho = 7850;  % kg/m^3
I  = 9.4751 * 10^(7) * 10^(-12); % m^4

% Moment of Inertia for Design (B)
I_b(1) = 73671200 * 10^(-12); % m^4
I_b(2) = 4.3778 * 10^7* 10^(-12) ;
I_b(3) =  2.3990e+07 * 10^(-12) ;

% Original Area of the beams
A = 6014; % mm^2

% Disceretize areas
A1 = 5400 ;
A2 = 4420;
A3 = 3520;

% Mass of the I Beams in Kg/m
ma = 47.21 ; % ^kg/m

% Discretize masses
m(1) = 42.39;
m(2) = 34.697;
m(3) = 27.632;
% Mass of the Concrete Slab + Fittings

Mcs = 500 ; % ^kg/m

% Mass of  the People
Mp = 593.75 ; % ^ kg/m

% Mass of the Design (a)

M_a = ma+Mcs+Mp;

% Mass of the Design B

for i = 1:3
    M_b(i) = m(i)+(Mcs+Mp);
end

% Mass Matrix

l= 1.6;
L = 4.8;
M_Facta = M_a*l/420;
K_Facta = (E*I)/l^3;

E = 210 * 10^9;
for i = 1:3
    M_Factb(i) =  M_b(i)*l/420;
    K_Factb(i) = (E*I_b(i))/l^3;
end
```

```
% Global Mass Matrix for Design (a)
prompt='Enter No of nodes from 1 to 4';
n=input(prompt)

M=zeros(4,4);

M      =      [156          22*l         54           -13*l  ;  ...,
               22*l         4*l*l        13*l         -3*l*l ;  ...,
               54           13*l         156          -22*l  ;  ...,
              -13*l        -3*l*l       -22*l          4*l*l ]  ;

K     =       [12            6*l        -12           6*l     ;  ...,
               6*l           4*l*l      -6*l          2*l*l   ;  ...,
              -12           -6*l         12          -6*l     ;  ...,
               6*l           2*l*l      -6*l          4*l*l   ];

   Ma=zeros(8,8);
   Mb=zeros(8,8);

   Ka=zeros(8,8);
   Kb=zeros(8,8);

   M1=zeros(8,8);
   M2=zeros(8,8);
   M3=zeros(8,8);

   K1=zeros(8,8);
   K2=zeros(8,8);
   K3=zeros(8,8);


   M1(1:4,1:4)=M;
   M2(3:6,3:6)=M;
   M3(5:8,5:8)=M;

   K1(1:4,1:4)=K;
   K2(3:6,3:6)=K;
   K3(5:8,5:8)=K;

         i=n;
            if(i==2)

                Ma   = M_Facta*M1+Ma;
                Mb   = M_Factb(i-1)*M1+Mb;

                Ka   = K_Facta*K1+Ka;
                Kb   = K_Factb(i-1)*K1+Kb;

                MGa  = Ma(1:4,1:4);
                MGb  = Mb(1:4,1:4);
                KGa  = Ka(1:4,1:4);
                KGb  = Kb(1:4,1:4);

            elseif(i==3)
```

```
                Ma   = Ma+M_Facta*M1+M_Facta*M2;
                Mb   = Mb+M_Factb(i-2)*M1+M_Factb(i-1)*M2;

                Ka   = Ka+K_Facta*K1+K_Facta*K2;
                Kb   = Kb+K_Factb(i-2)*K1+K_Factb(i-1)*K2;


                MGa = Ma(3:6,3:6);
                MGb = Mb(3:6,3:6);


                KGa = Ka(3:6,3:6);
                KGb = Kb(3:6,3:6);

            elseif(i==4)

                 Ma   = Ma+M_Facta*M1+M_Facta*M2+M_Facta*M3;
                 Mb   = Mb+M_Factb(i-3)*M1+M_Factb(i-2)*M2+M_Factb(i-1)*M3;

                 Ka   = Ka+K_Facta*K1+K_Facta*K2+K_Facta*K3;
                 Kb   = Kb+K_Factb(i-3)*K1+K_Factb(i-2)*K2+K_Factb(i-1)*K3;

                 MGa = Ma(3:8,3:8);
                 MGb = Mb(3:8,3:8);

                 KGa = Ka(3:8,3:8);
                 KGb = Kb(3:8,3:8);


            end
             % Calculating Eigen values and Eigen Vectors

%Eigenvectors and eigenvalues:
[Xa,lambdaA]=eig(MGa\KGa,'vector');
[Xb,lambdaB]=eig(MGb\KGb,'vector');


%Natural circular frequencies:
omegaA=sqrt(lambdaA);
omegaB=sqrt(lambdaB);

%  Natural frequencies:
fnA=omegaA/(2*pi);
fnB=omegaB/(2*pi);

FnA = real(fnA);
FnB = real(fnB);

% Normalizing the Eigen Vectors for both design (A) and (B)

XAnew = Xa;
XBnew = Xb;

for i = 1:6
    XAnew(:,i) = Xa(:,i)/min(Xa(:,i));

    XBnew(:,i) = Xb(:,i)/min(Xb(:,i));
end
```

```
% Length of the discretised beam

X = [0 1.6 3.2 4.8];

% Time Steps by using NEWMARK Time Discretization Method
for design = 1:2

for k = 1:6
ti = 0. ;
if design == 1
 tf = 4*2*pi/omegaA(k) ;
else
   tf = 4*2*pi/omegaB(k) ;
end
t = linspace(ti,tf) ;
dt = t(2)-t(1);

nstep = length(t) ;
n = length(MGa) ;

U = zeros(n,nstep) ;
V = zeros(n,nstep) ;
A = zeros(n,nstep) ;

% Initial Conditions
A(:,1) = zeros ;
V(:,1) = zeros ;

% Which Eigen vector should be used ?
% Normalized eigen vector or normal eigen vector

if design == 1
    U(:,1) = Xa(:,k);
else
    U(:,1) = Xb(:,k);
end


for i = 1:nstep-1

    if design == 1

        Numerator   = 4/dt^2 * MGa * U(:,i) + 4/dt * MGa * V(:,i) + MGa * A(:,i);
        Denominator = 4/dt^2 * MGa + KGa;
    else
        Numerator   = 4/dt^2 * MGb * U(:,i) + 4/dt * MGb * V(:,i) + MGb * A(:,i);
        Denominator = 4/dt^2 * MGa + KGb;
    end

    U(:,i+1) = (Denominator)\(Numerator) ;

    V(:,i+1) = 2/dt*(U(:,i+1)-U(:,i))-V(:,i);

    A(:,i+1) = 4/dt^2*(U(:,i+1)-U(:,i)) - 4/dt*V(:,i) - A(:,i);

end
```

```
u1=zeros(1,nstep);
Disp =[u1; U(1,:); U(3,:); U(5,:)];

if design == 1
    figure(k)
    title(['Mode ' k])
    plot(t,Disp)
    hold on
    xlabel('Time in sec') ;ylabel('U') ;
    legend('Node1','Node2','Node3','Node4')
else
    figure(k+9)
    title(['Mode for deisgn B ' k])
    plot(t,Disp)
    hold on
    xlabel('Time in sec') ;ylabel('U') ;
    legend('Node1','Node2','Node3','Node4')
end

end
end
```

MATLAB Results

```
>> My_Sd
Enter No of nodes from 1 to 44

n =

     4

>> MGa

MGa =

   1.0e+03 *

    1.3561         0    0.2347   -0.0904         0         0
         0    0.0890    0.0904   -0.0334         0         0
    0.2347    0.0904    1.3561         0    0.2347   -0.0904
   -0.0904   -0.0334         0    0.0890    0.0904   -0.0334
         0         0    0.2347    0.0904    0.6781   -0.1530
         0         0   -0.0904   -0.0334   -0.1530    0.0445

>> MGb

MGb =
```

```
   1.0e+03 *

    1.3458   -0.0010    0.2321   -0.0894         0         0
   -0.0010    0.0883    0.0894   -0.0330         0         0
    0.2321    0.0894    1.3370   -0.0009    0.2307   -0.0889
   -0.0894   -0.0330   -0.0009    0.0878    0.0889   -0.0328
         0         0    0.2307    0.0889    0.6664   -0.1504
         0         0   -0.0889   -0.0328   -0.1504    0.0437

>> KGa

KGa =

   1.0e+08 *

    1.1659         0   -0.5829    0.4664         0         0
         0    0.9949   -0.4664    0.2487         0         0
   -0.5829   -0.4664    1.1659         0   -0.5829    0.4664
    0.4664    0.2487         0    0.9949   -0.4664    0.2487
         0         0   -0.5829   -0.4664    0.5829   -0.4664
         0         0    0.4664    0.2487   -0.4664    0.4974

>> KGb

KGb =

   1.0e+07 *

    7.2259   -1.4713   -2.6934    2.1547         0         0
   -1.4713    6.1661   -2.1547    1.1492         0         0
   -2.6934   -2.1547    4.1693   -0.9739   -1.4759    1.1808
    2.1547    1.1492   -0.9739    3.5578   -1.1808    0.6297
         0         0   -1.4759   -1.1808    1.4759   -1.1808
         0         0    1.1808    0.6297   -1.1808    1.2595

>> Xa

Xa =
```

```
    0.0179   -0.0467   -0.0621   -0.3356    0.3385   -0.1343
    0.1145   -0.5434    0.4957    0.1562    0.2109   -0.1529
    0.0280    0.0822   -0.0287    0.2959    0.2430   -0.4438
    0.3217   -0.3574   -0.5079    0.1192   -0.3538   -0.2213
    0.1485    0.1768    0.2464   -0.4510   -0.5739   -0.8115
    0.9275    0.7326    0.6564   -0.7469   -0.5721   -0.2327

>> lambdaA

lambdaA =

   1.0e+06 *

    9.1517
    2.3026
    0.6501
    0.1282
    0.0161
    0.0004

>> Xb

Xb =

    0.0226    0.0203   -0.0969    0.2814    0.2688   -0.1144
    0.1755    0.4402    0.3064    0.0373    0.2128   -0.1305
    0.0208   -0.0766    0.0255   -0.2337    0.2715   -0.4172
    0.3660    0.1082   -0.5174   -0.3141   -0.2536   -0.2299
    0.1404   -0.1735    0.2323    0.4081   -0.5768   -0.8216
    0.9026   -0.8707    0.7579    0.7743   -0.6417   -0.2606

>> lambdaB

lambdaB =

   1.0e+06 *

    2.7096
```

```
    1.0453
    0.2972
    0.0548
    0.0079
    0.0003

>> fnA

fnA =

  481.4711
  241.5066
  128.3242
   56.9833
   20.1665
    3.2077

>> fnB

fnB =

  261.9825
  162.7202
   86.7633
   37.2506
   14.1149
    2.6743

>> XAnew

XAnew =

    1.0000    0.0859    0.1222    0.4493   -0.5899    0.1655
    6.4101    1.0000   -0.9760   -0.2091   -0.3674    0.1884
    1.5657   -0.1513    0.0565   -0.3962   -0.4235    0.5469
   18.0187    0.6578    1.0000   -0.1596    0.6165    0.2727
    8.3145   -0.3254   -0.4852    0.6039    1.0000    1.0000
   51.9472   -1.3481   -1.2924    1.0000    0.9969    0.2868
```

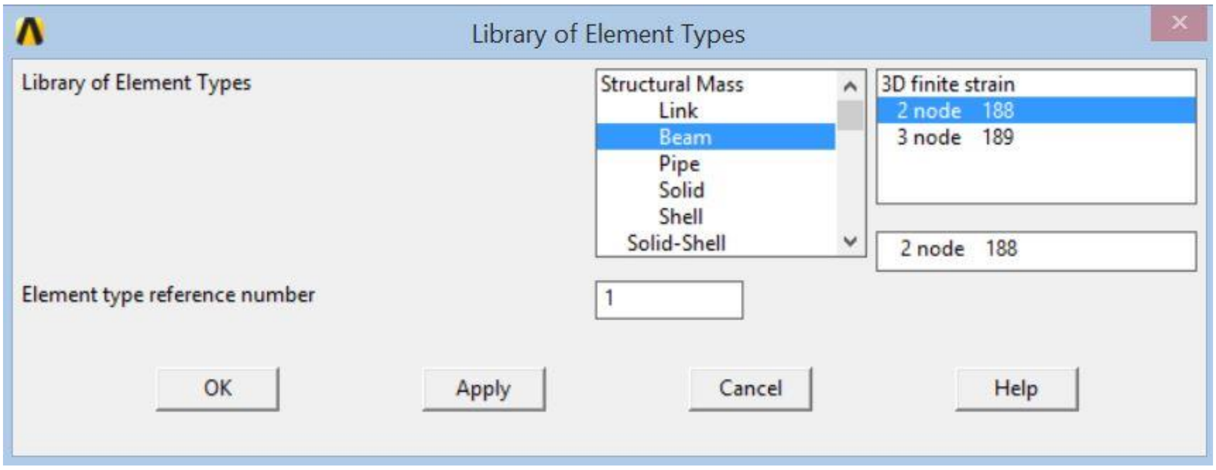
```
>>  XBnew

XBnew =

    1.0886   -0.0233    0.1873   -0.8960   -0.4189    0.1393
    8.4500   -0.5056   -0.5923   -0.1189   -0.3316    0.1588
    1.0000    0.0879   -0.0492    0.7440   -0.4231    0.5078
   17.6169   -0.1243    1.0000    1.0000    0.3952    0.2798
    6.7564    0.1993   -0.4489   -1.2994    0.8988    1.0000
   43.4481    1.0000   -1.4650   -2.4651    1.0000    0.3172

>>
```

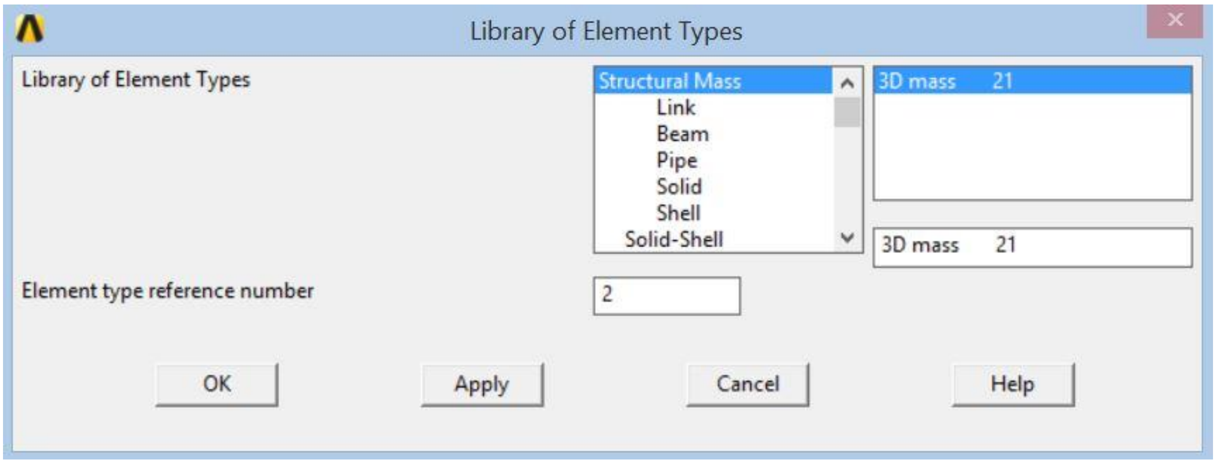
Modal Analysis Procedure Ansys

1. Open preprocessor menu
2. Give example a Title
   Utility Menu > File > Change Title ...
3. Give example a Jobname
   Utility Menu > File > Change Jobname ...
   Enter ’Cantilever_dynamic’ for the jobname

4.Element type >Add/edit >popup >Add >Beam >2node188 >apply

Structural Mass 3Dmass >ok >close





5.Real Constants >Add/Edit >popup >Add >Mass2l >popup

Set1 8*75

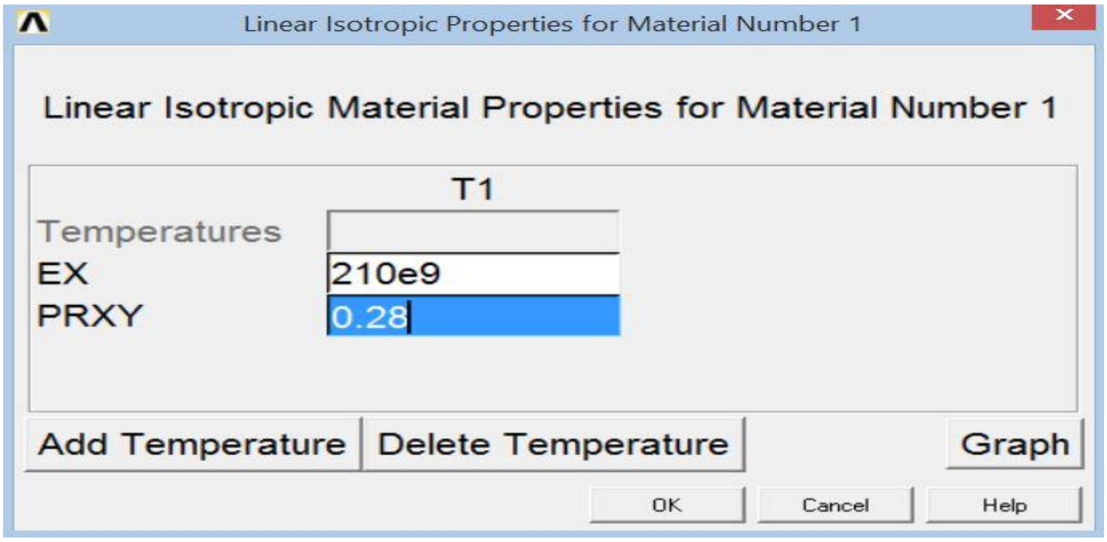
Set2 7*75 close

6.Material Properties >Material Models >Structural >Linear >Elastic >isotropic

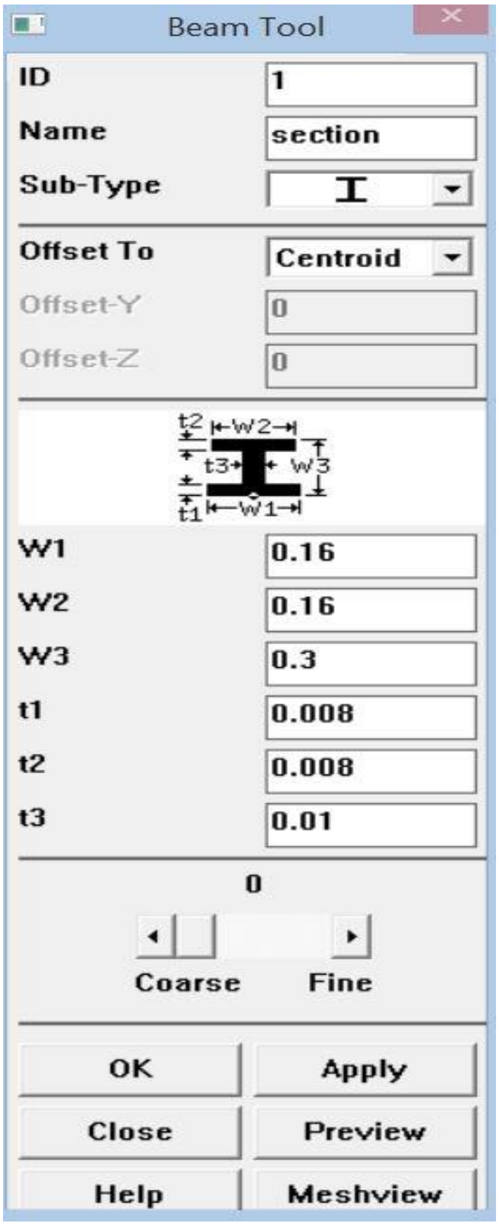
E=210e09

ϑ=0.28

Density=7850



7.sections >Beam >Common Sections



Give values as shown for other sections

8.Sections >Beam >Section Control >popup

Sec1 Add mass/length=500

Sec2 Add mass/length=500

Sec3 Add mass/length=500

keypoints	coordinatinates(x,y)
1	(0.8*0,0)
2	(0.8*1,0)
3	(0.8*2,0)
4	(0.8*3,0)
5	(0.8*4,0)
6	(0.8*5,0)
7	(0.8*6,0)

9.Modelling>Create >Lines>popup

Straight Line join with arrow ok

10.Meshing>Size Controls>Manual Size>Lines>All Lines>pop Up>

No of Element Divisions 1> ok

11.Meshing>Meshing Attributes>Picked Keypoints

2,3,4 ok>popup

Real Set1 >Element type> Mass 2l>ok

Meshing>Meshing Attributes>Picked Keypoints

5,6 ok>popup

Reals Set2>Element Type> Mass 2l>ok

12..Meshing>Meshing Attributes>

Picked Lines

1,2>Beam 188 >sec1>ok

Picked Lines

3,4>Beam 188 >sec2>ok

Picked Lines

1,2>Beam 188 >sec3>ok

13. Meshing>Mesh>Key Points

Select all middle Keypoints>ok

Meshing>Mesh>Lines>Pick All>Ok

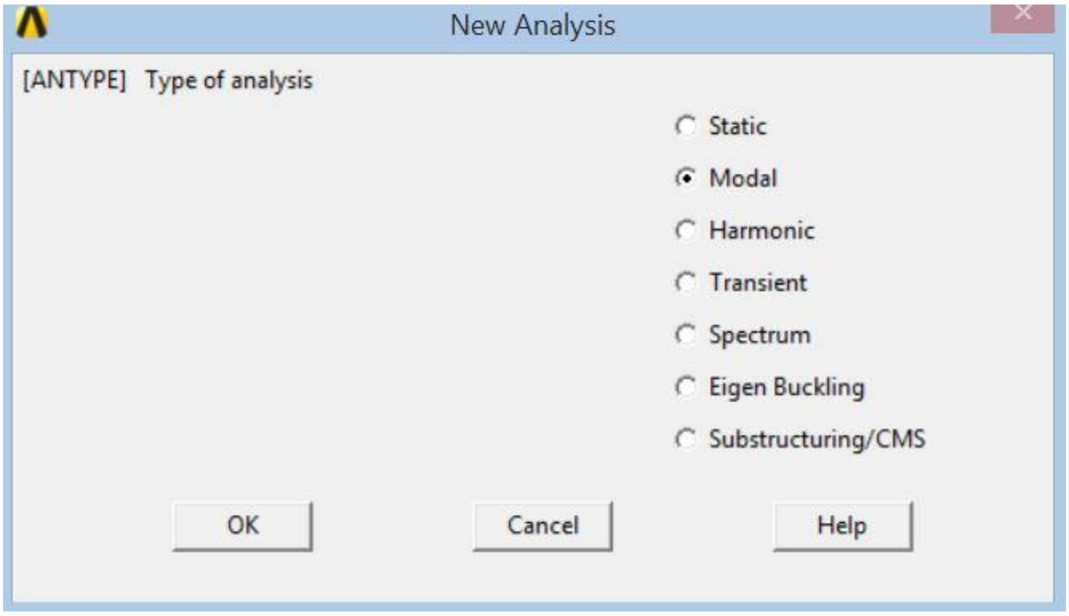
14.Plot Controls>Style>Size and Shape>pop up >Display of Element >on>ok

15.Soluton>Analysis Type>New Analysis>pop up>Modal>ok

Solution>Analysis>Analysis Options

No of Modes to Extract 6

No of Modes to Expand 6 No of Modes to Extract 6 >ok



16.Solution>Define Loads>Apply>Structural>Displacement>On Nodes

Select 1st Node>ok>popup>

Select All Dof>ok

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

Select All other Nodes>ok>popup>

Select Ux,Uy,Rotx,Rotz>ok

17.Solve>Current LS>ok

18.General Post Proc>Results Summary>popup>Results

19. General Post Proc> Read Results>First set

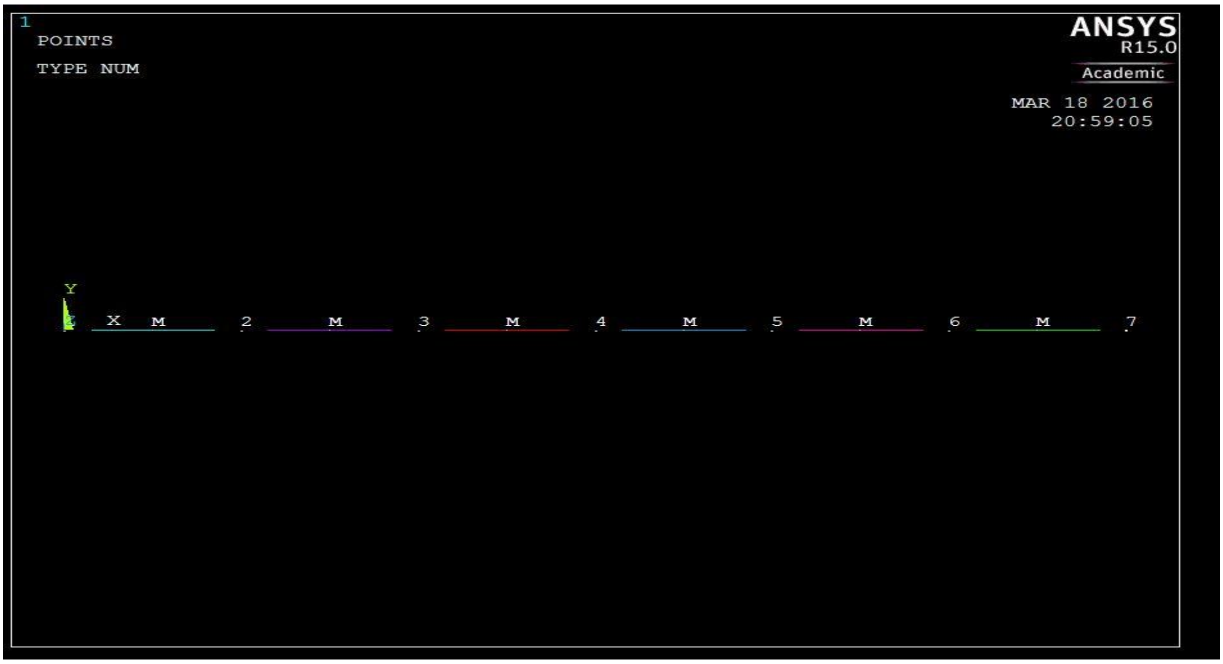
>Plot Results>Deformed Shape>pop up>Def+UnDef edge>Ok

20. General Post Proc> Read Results>Next set

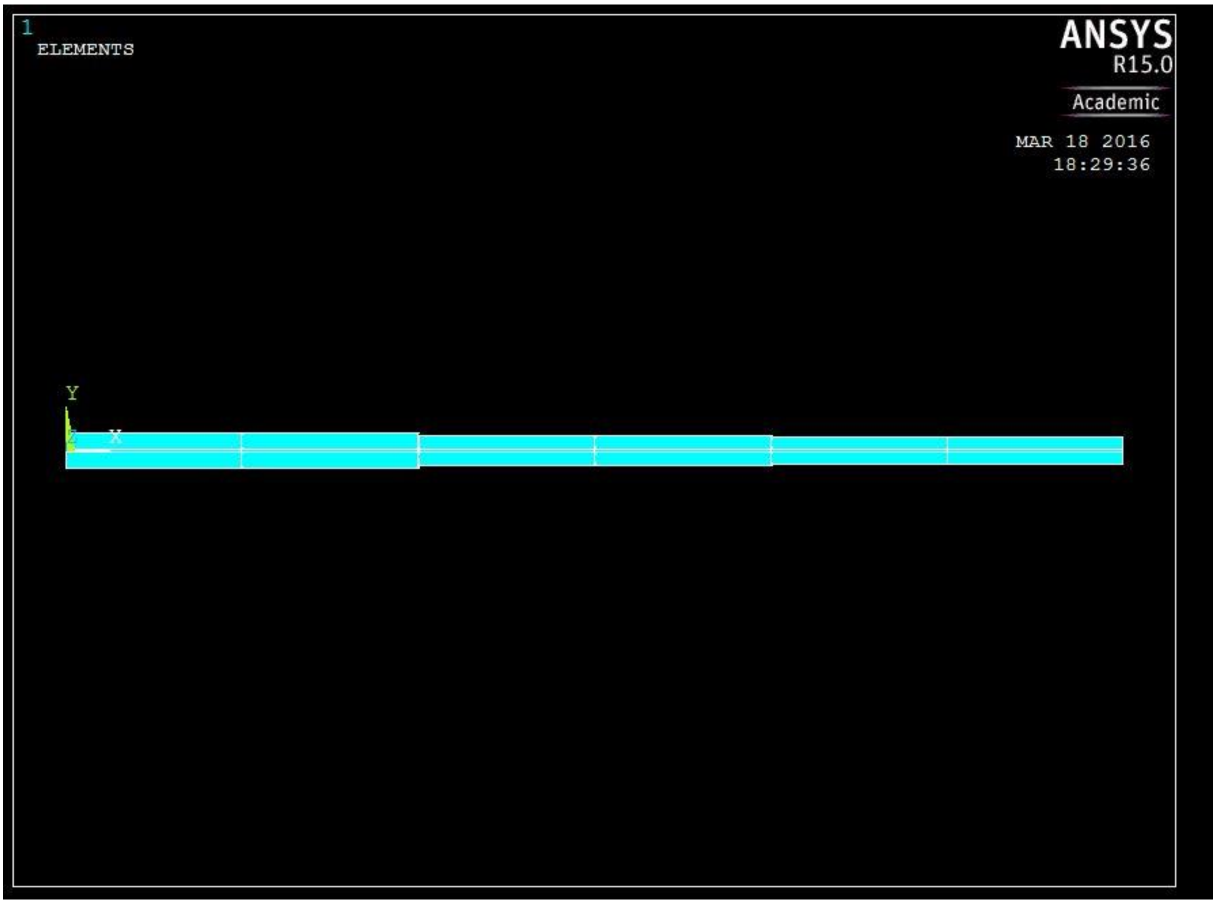
>Plot Results>Deformed Shape>pop up>Def+UnDef edge>Ok

ANSYS Results

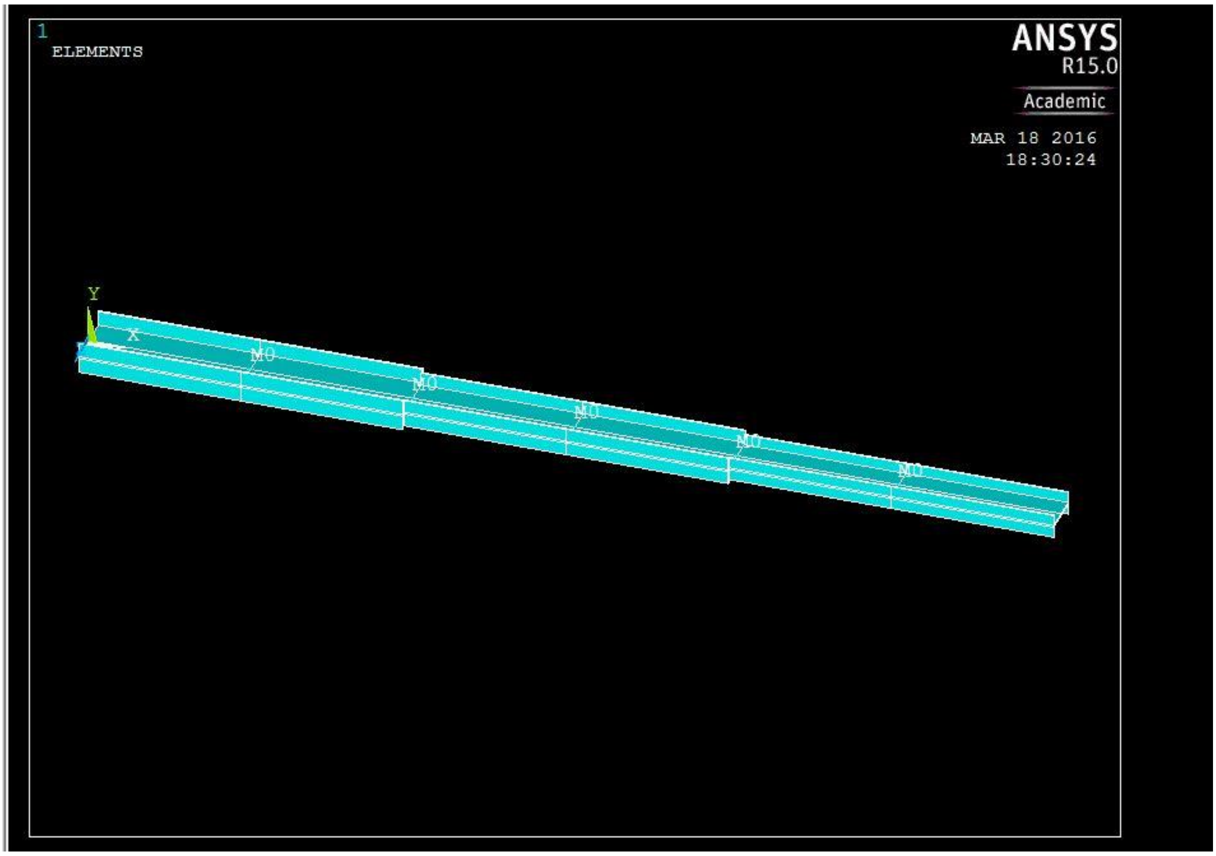
BEAM WITH 1 DIVISION OF ELEMENT



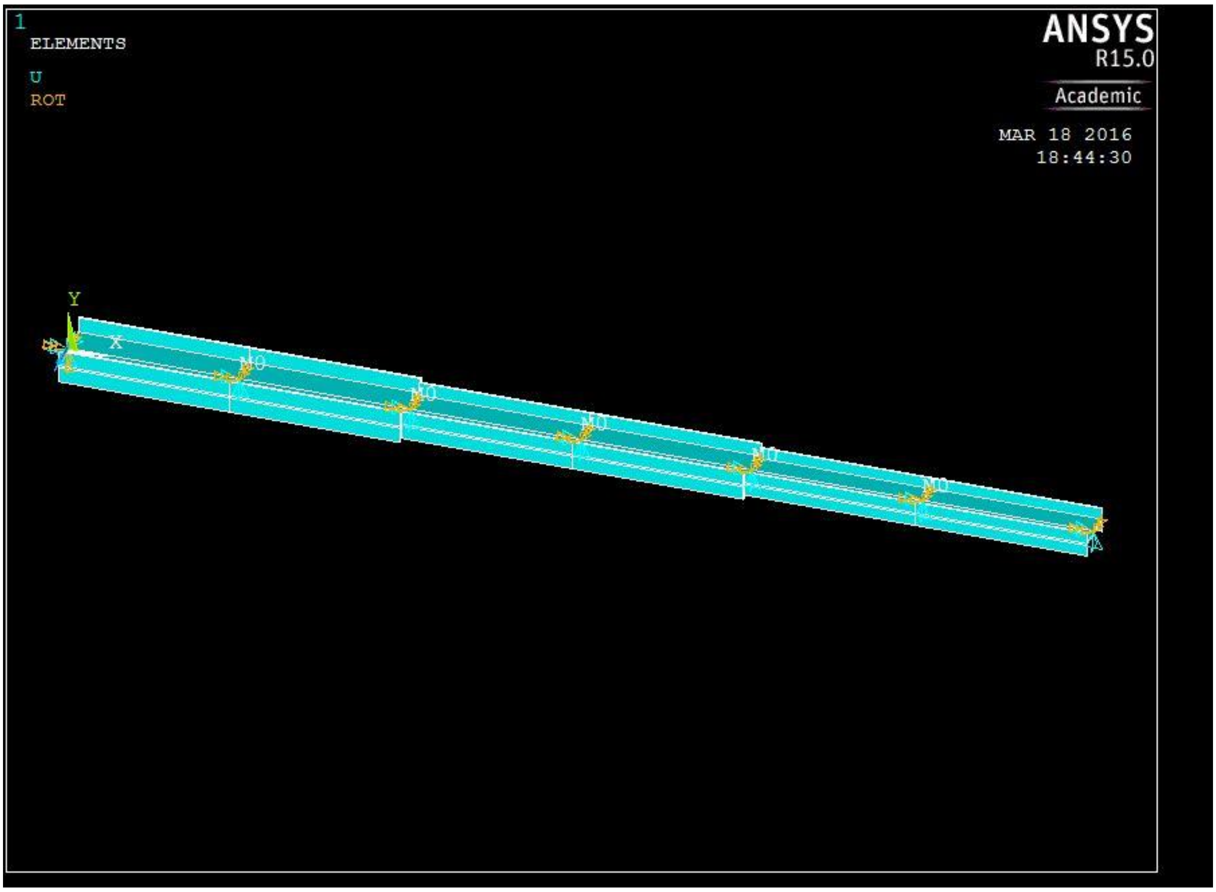
MESHED BEAM FRONT VIEW



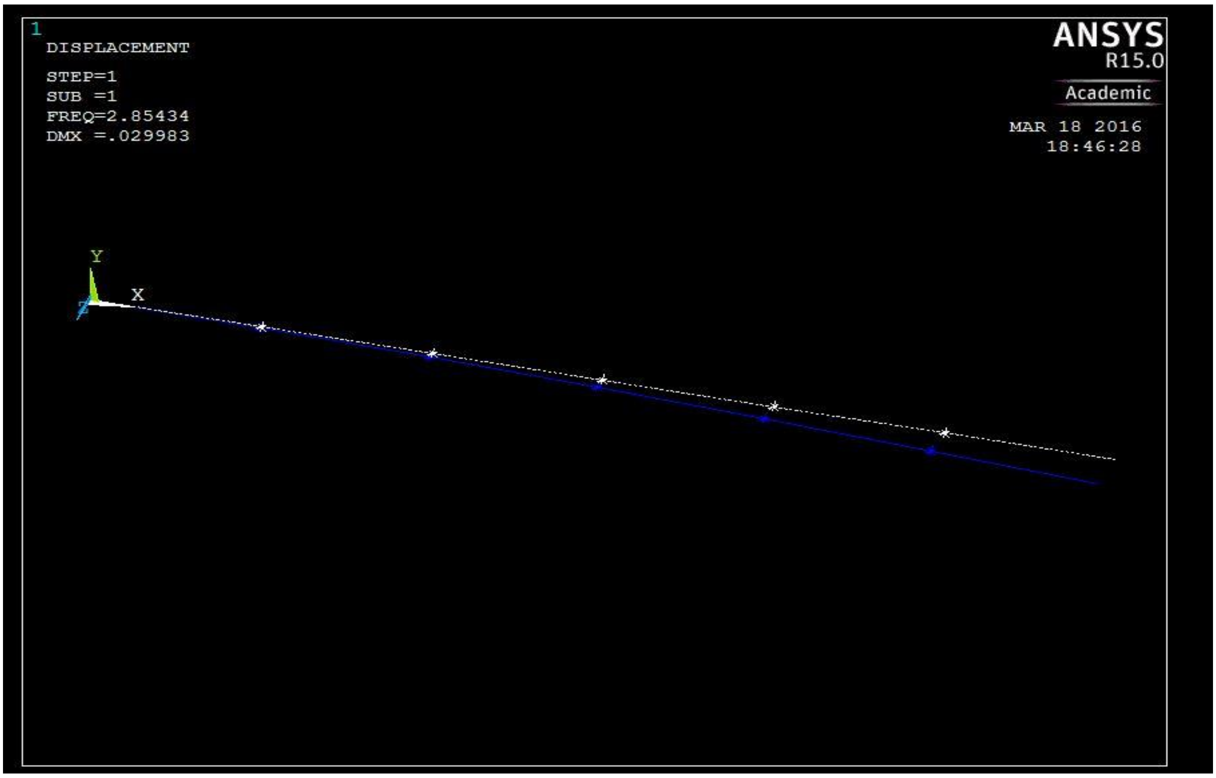
MESHED BEAM OBLIQUE VIEW



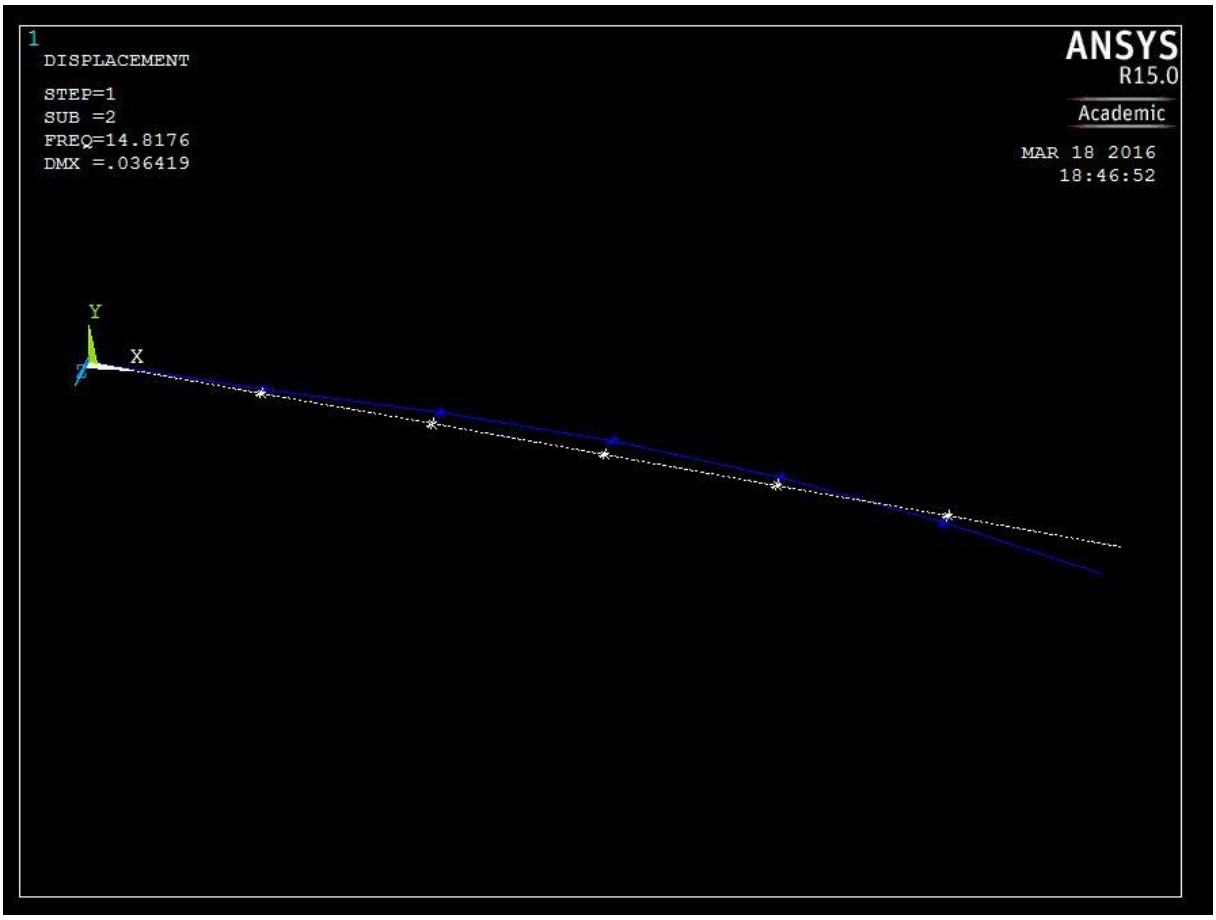
BEAM WITH LOADS



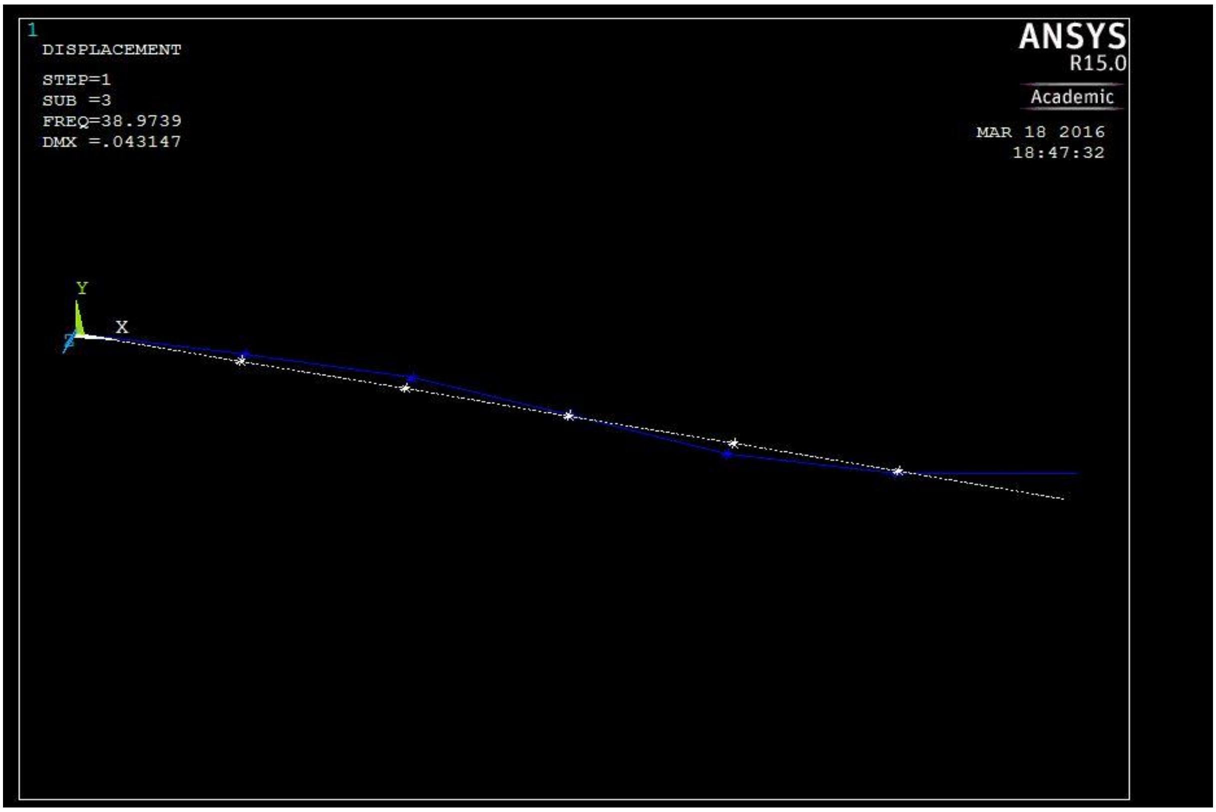
Mode1


1
DISPLACEMENT
STEP=1
SUB =1
FREQ=2.85434
DMX =.029983
ANSYS
R15.0
Academic
MAR 18 2016
18:46:28
Y
X
Z

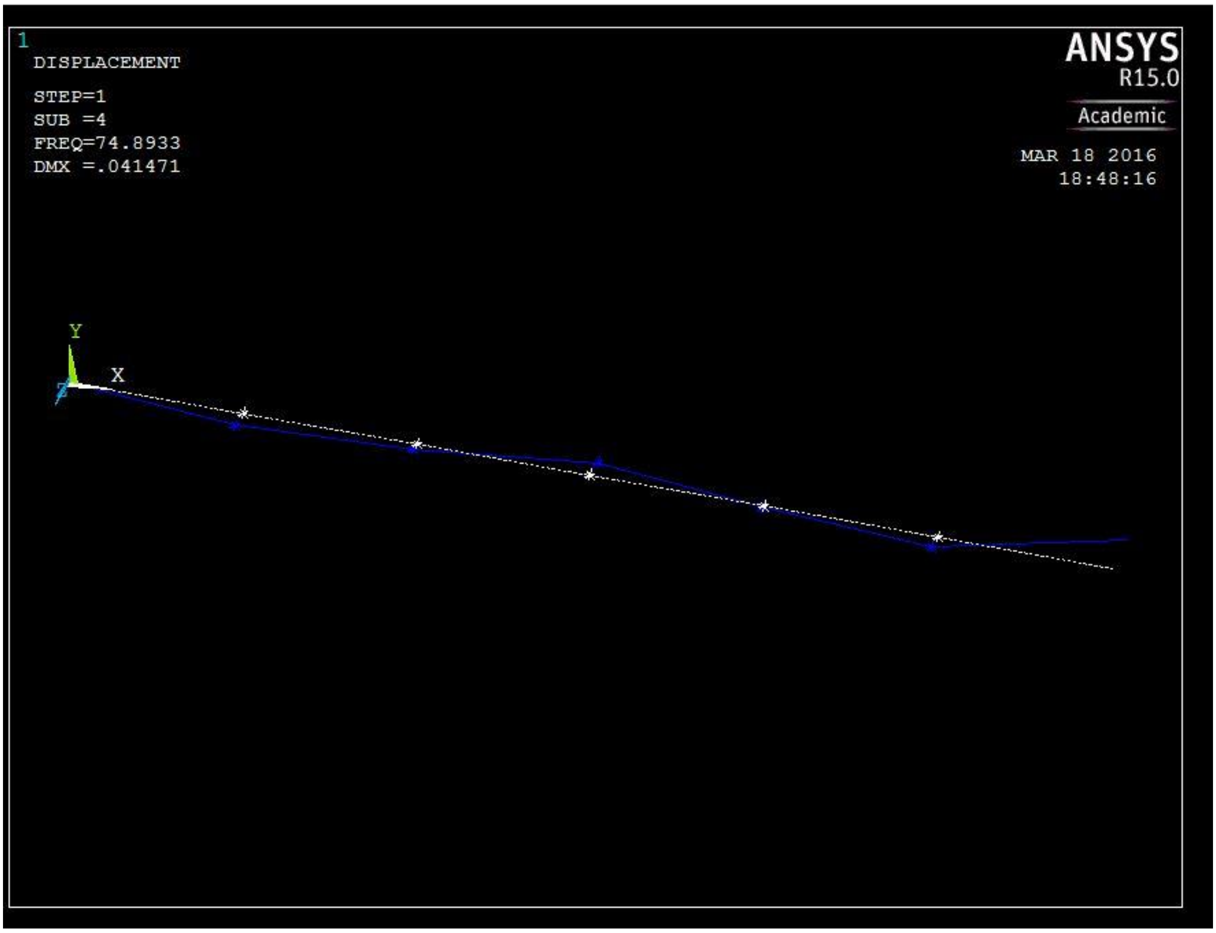
Mode2


1
DISPLACEMENT
STEP=1
SUB =2
FREQ=14.8176
DMX =.036419
ANSYS
R15.0
Academic
MAR 18 2016
18:46:52
Y
X
Z

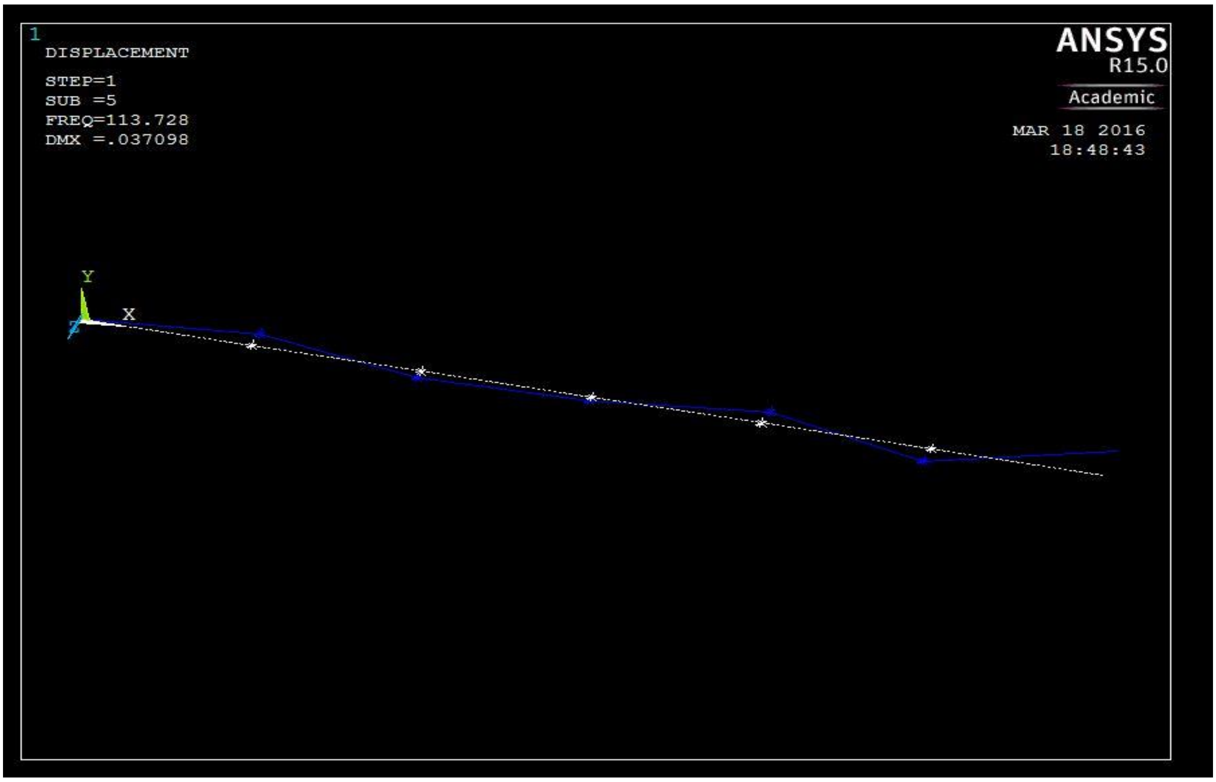
Mode3


1
DISPLACEMENT
STEP=1
SUB =3
FREQ=38.9739
DMX =.043147
ANSYS
R15.0
Academic
MAR 18 2016
18:47:32
Y
X

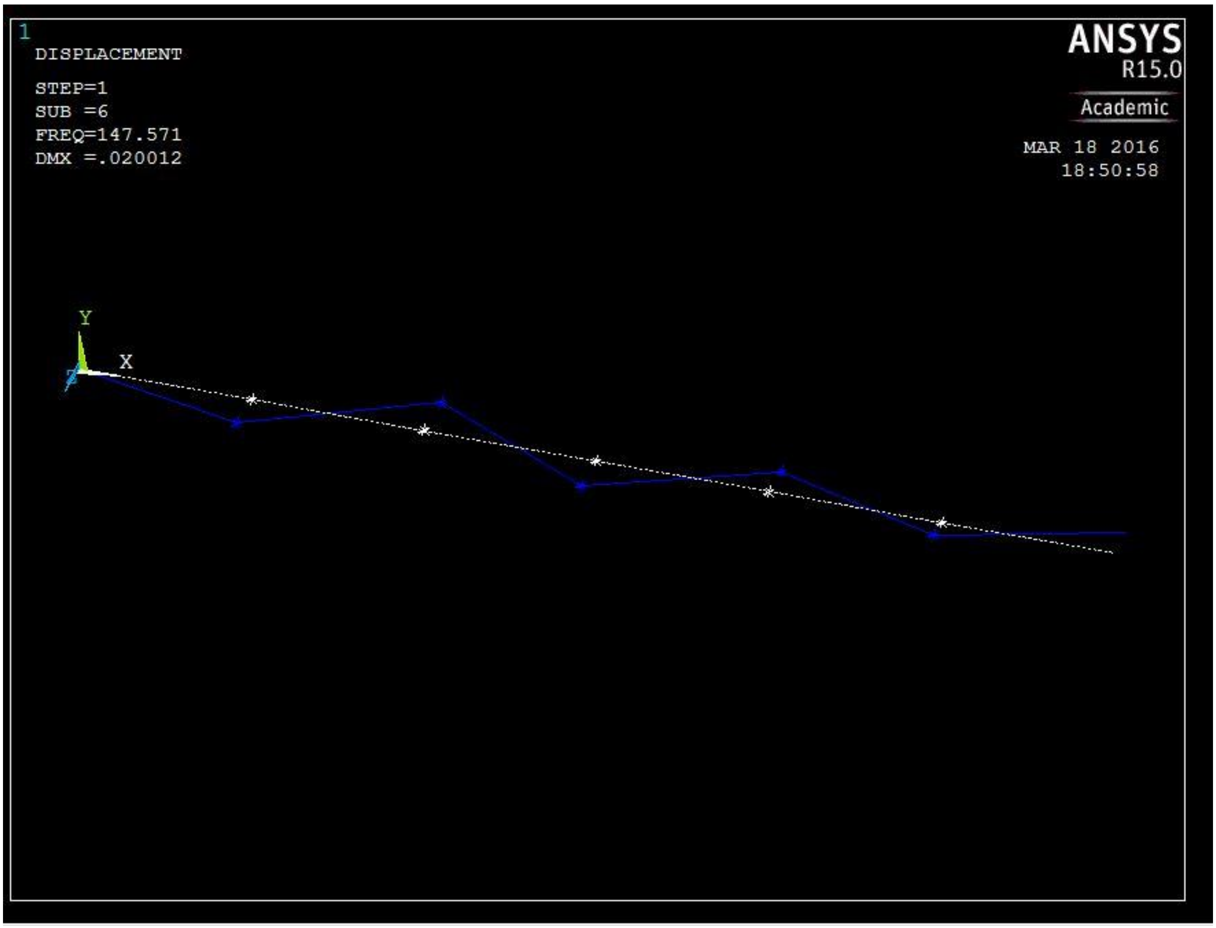
Mode4


1
DISPLACEMENT
STEP=1
SUB =4
FREQ=74.8933
DMX =.041471
ANSYS
R15.0
Academic
MAR 18 2016
18:48:16
Y
X

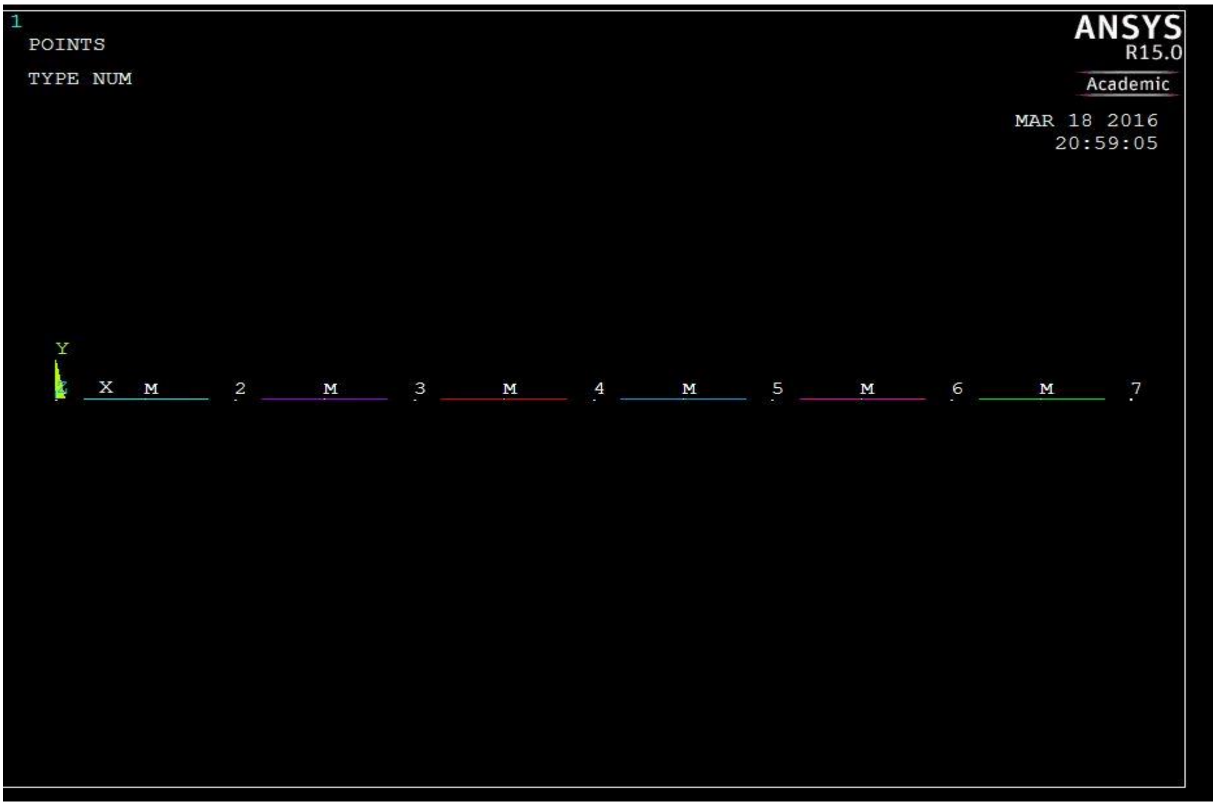
Mode5


1
DISPLACEMENT
STEP=1
SUB =5
FREQ=113.728
DMX =.037098
ANSYS
R15.0
Academic
MAR 18 2016
18:48:43
Y
X

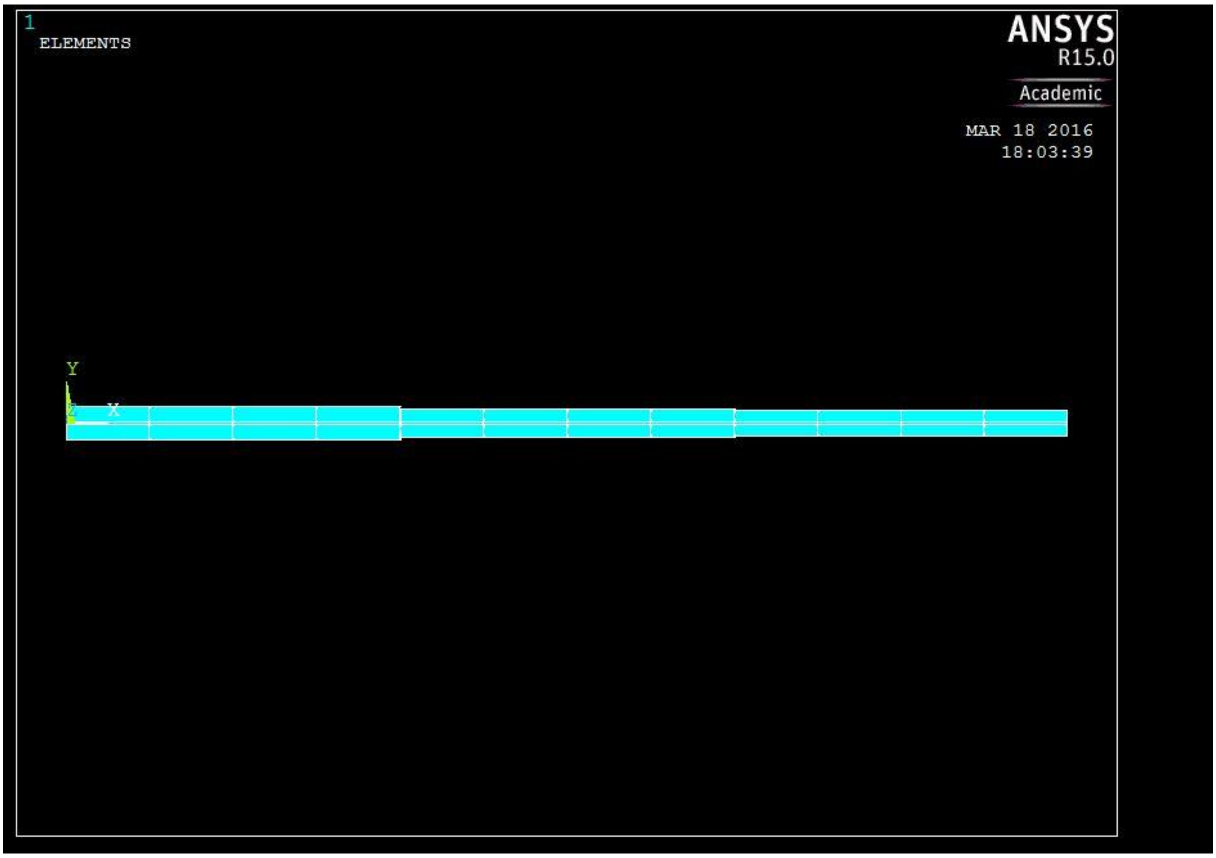
Mode6


1
DISPLACEMENT
STEP=1
SUB =6
FREQ=147.571
DMX =.020012
ANSYS
R15.0
Academic
MAR 18 2016
18:50:58
Y
X

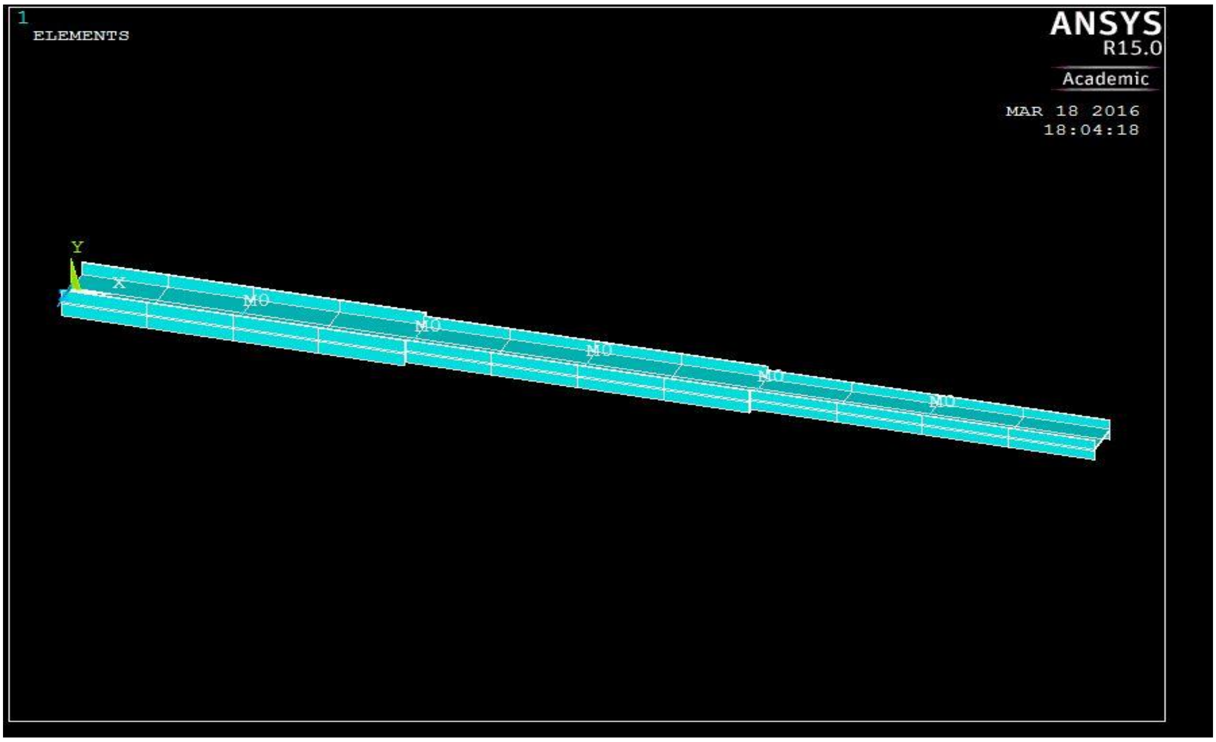
Meshed beam with 2 divisions of elements



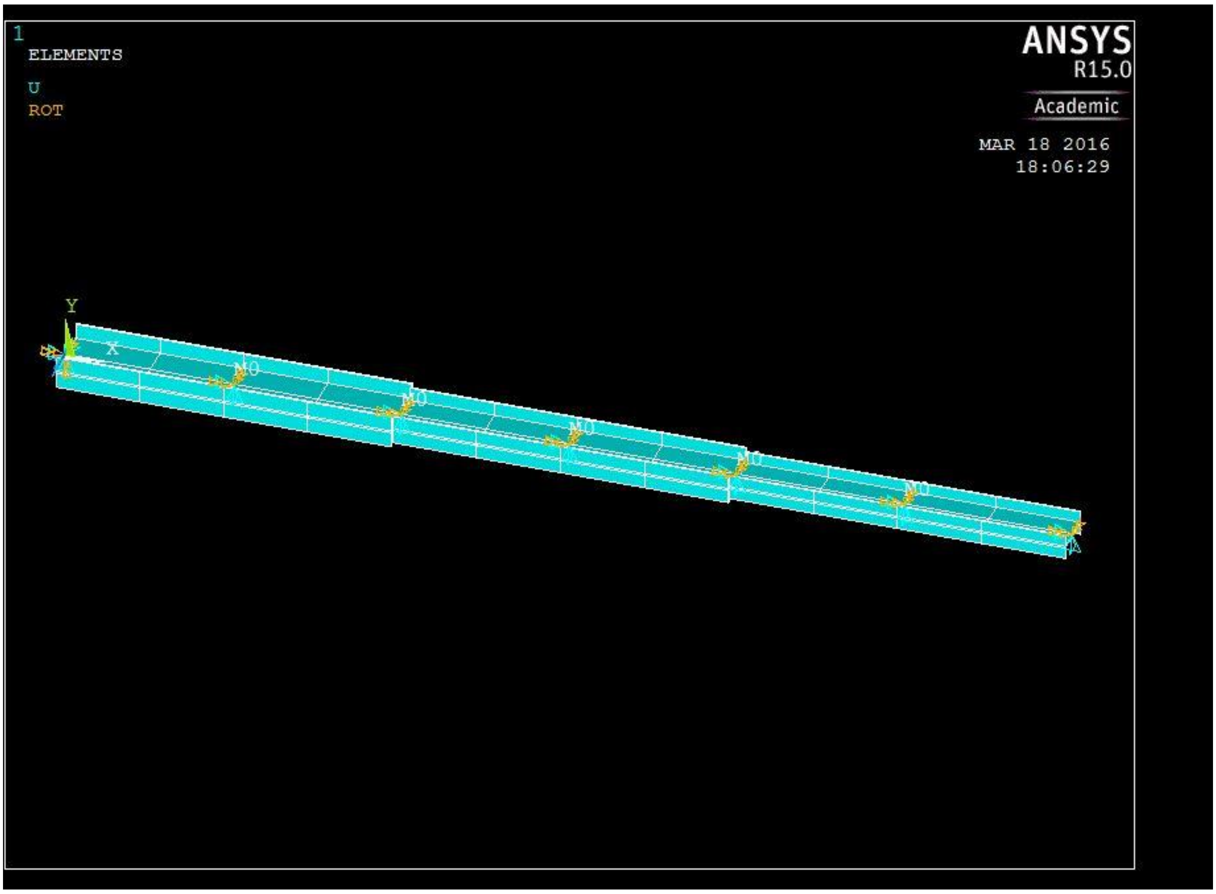
Meshed Beam front view



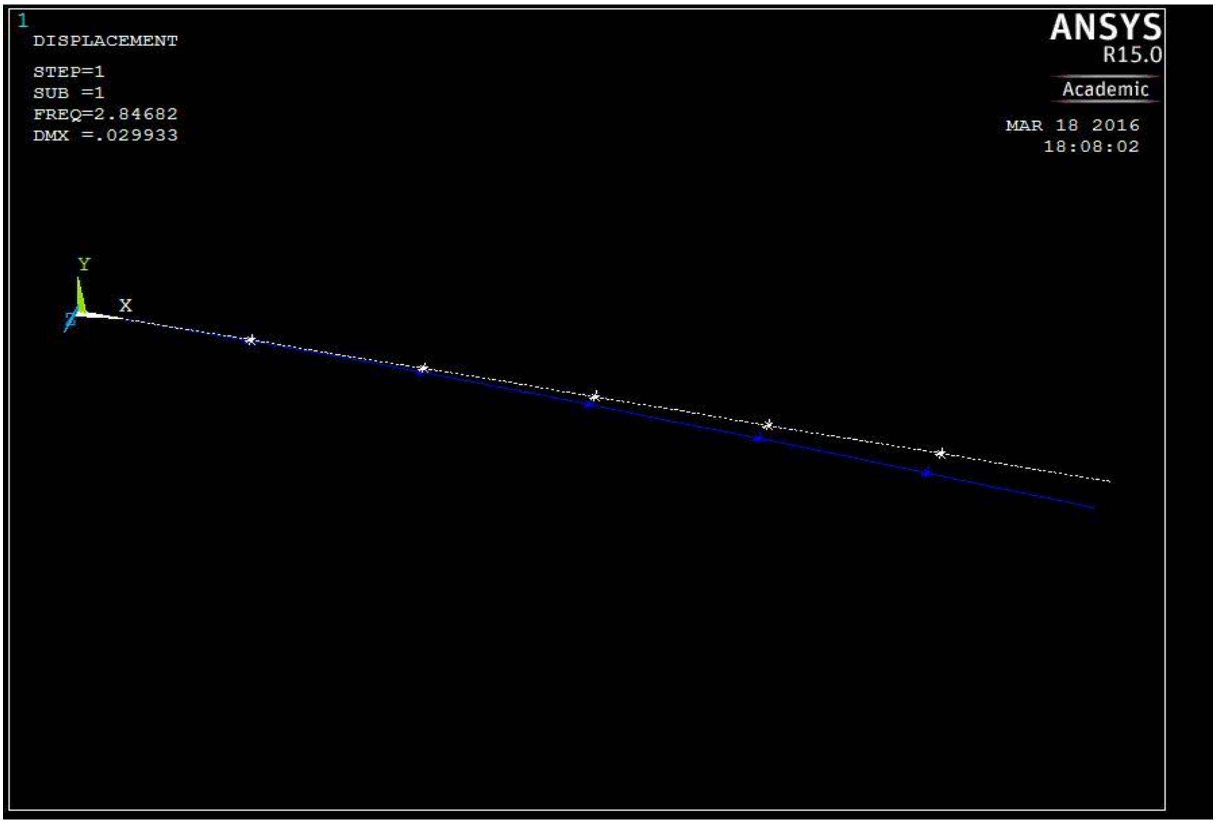
Meshed Beam Oblique View



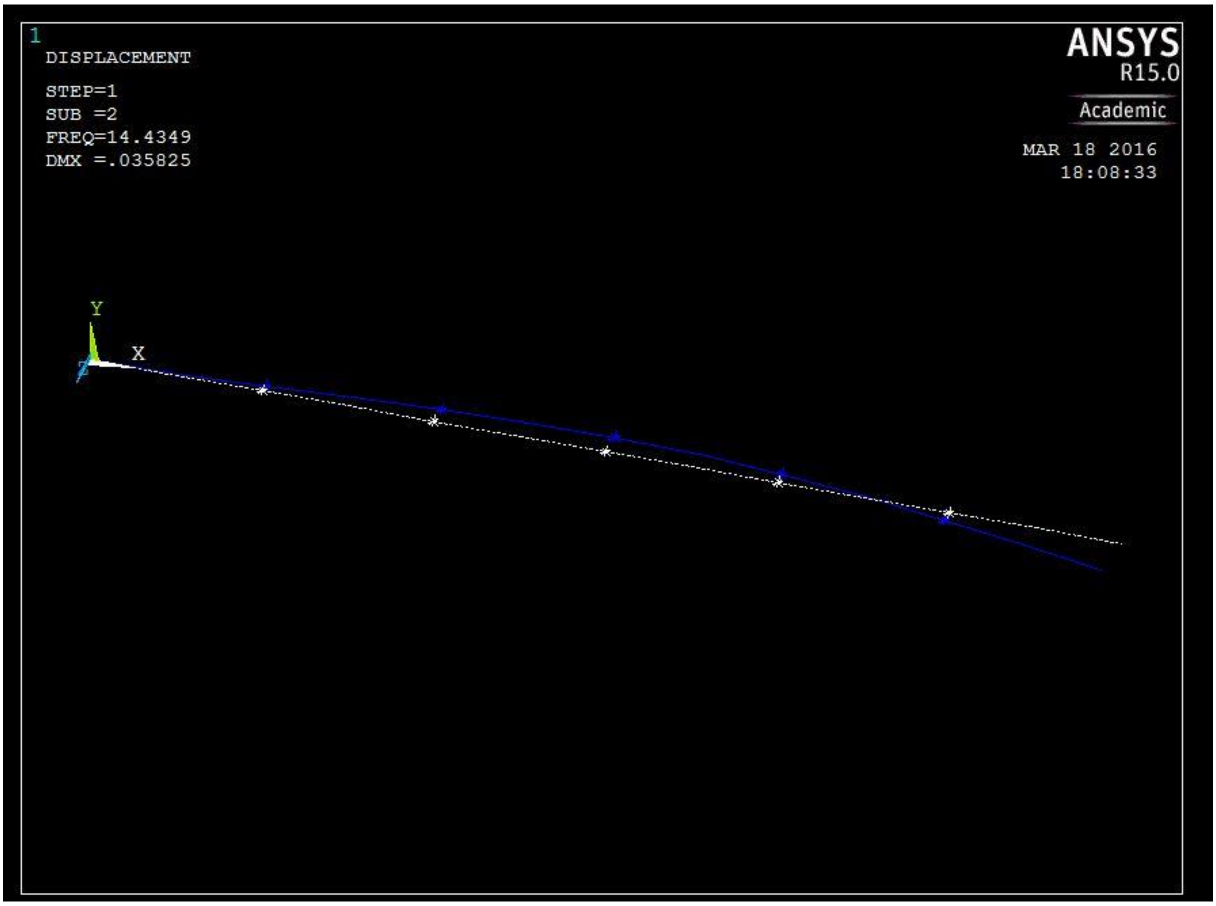
Meshed Beam with Loads:



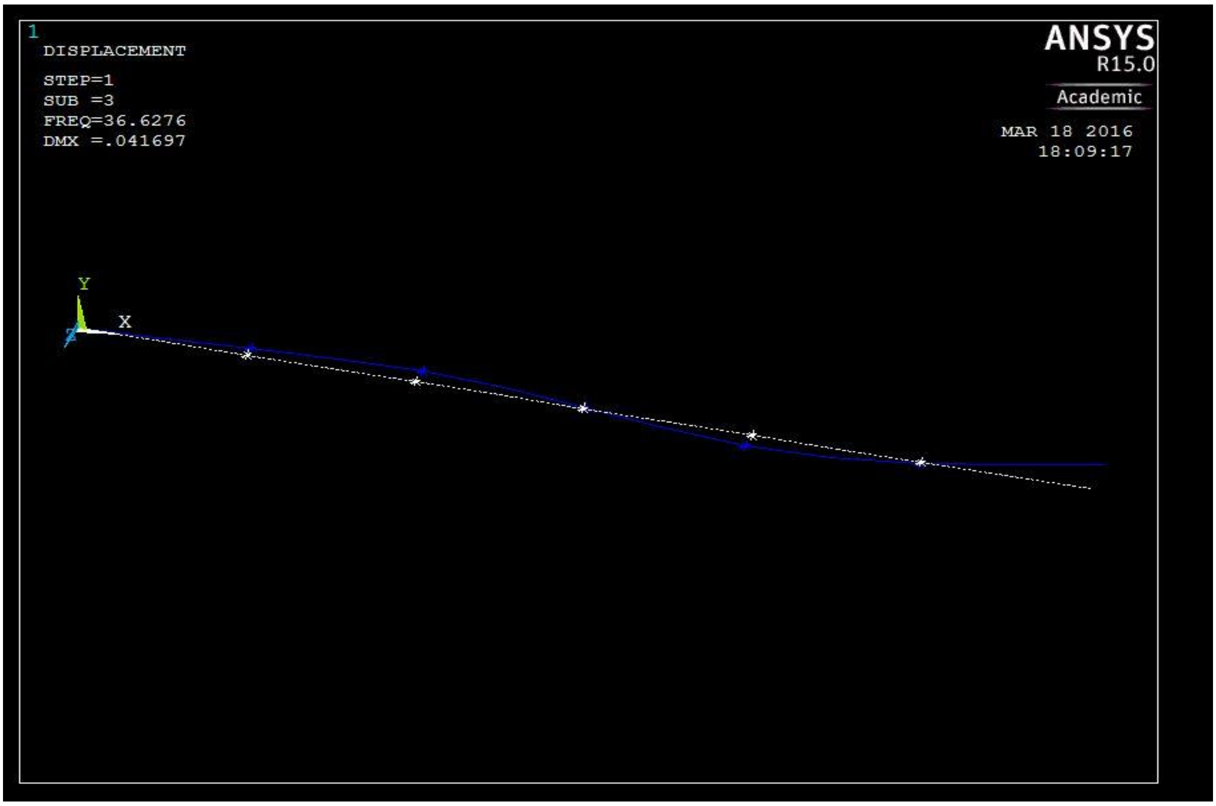
Mode1


1
DISPLACEMENT
STEP=1
SUB =1
FREQ=2.84682
DMX =.029933
ANSYS
R15.0
Academic
MAR 18 2016
18:08:02
Y
X
Z

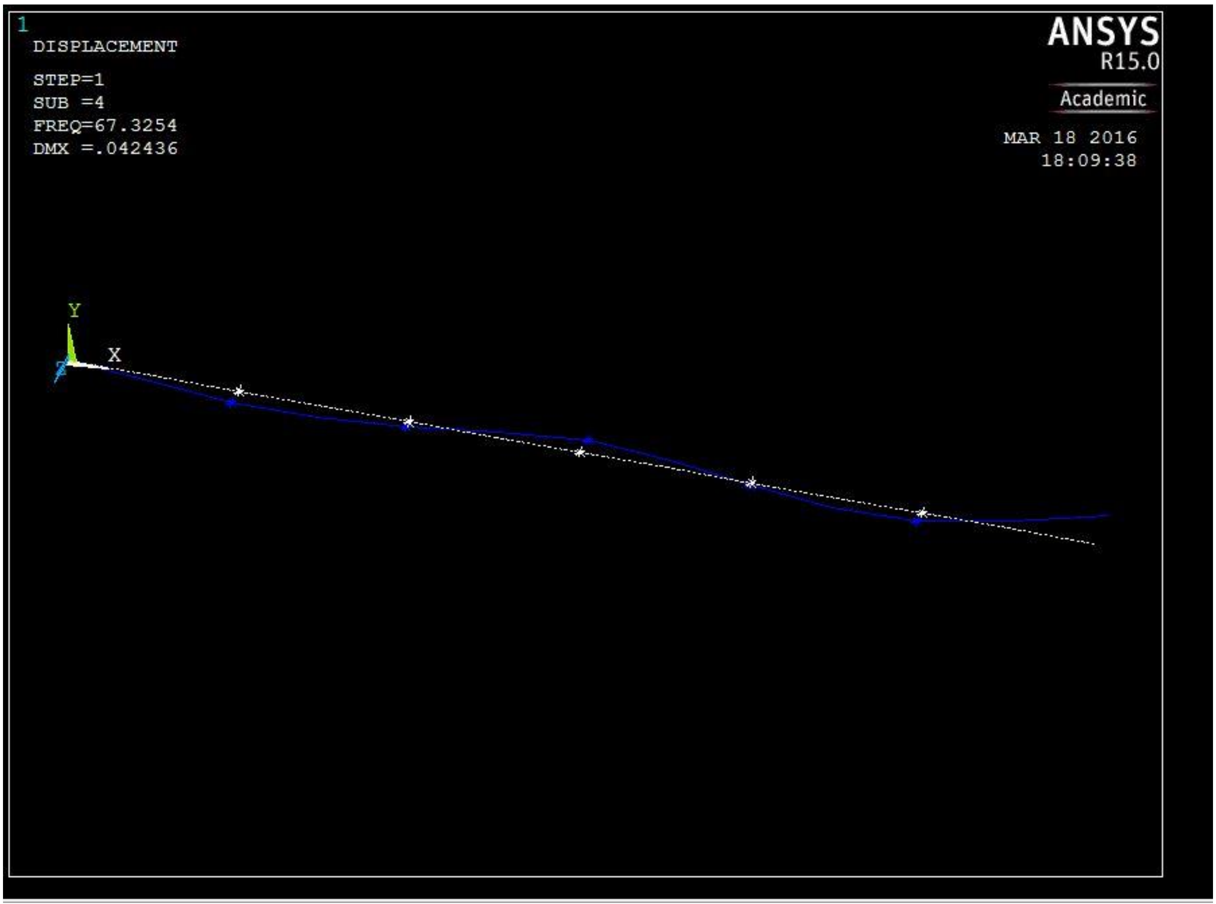
Mode2


1
DISPLACEMENT
STEP=1
SUB =2
FREQ=14.4349
DMX =.035825
ANSYS
R15.0
Academic
MAR 18 2016
18:08:33
Y
X
Z

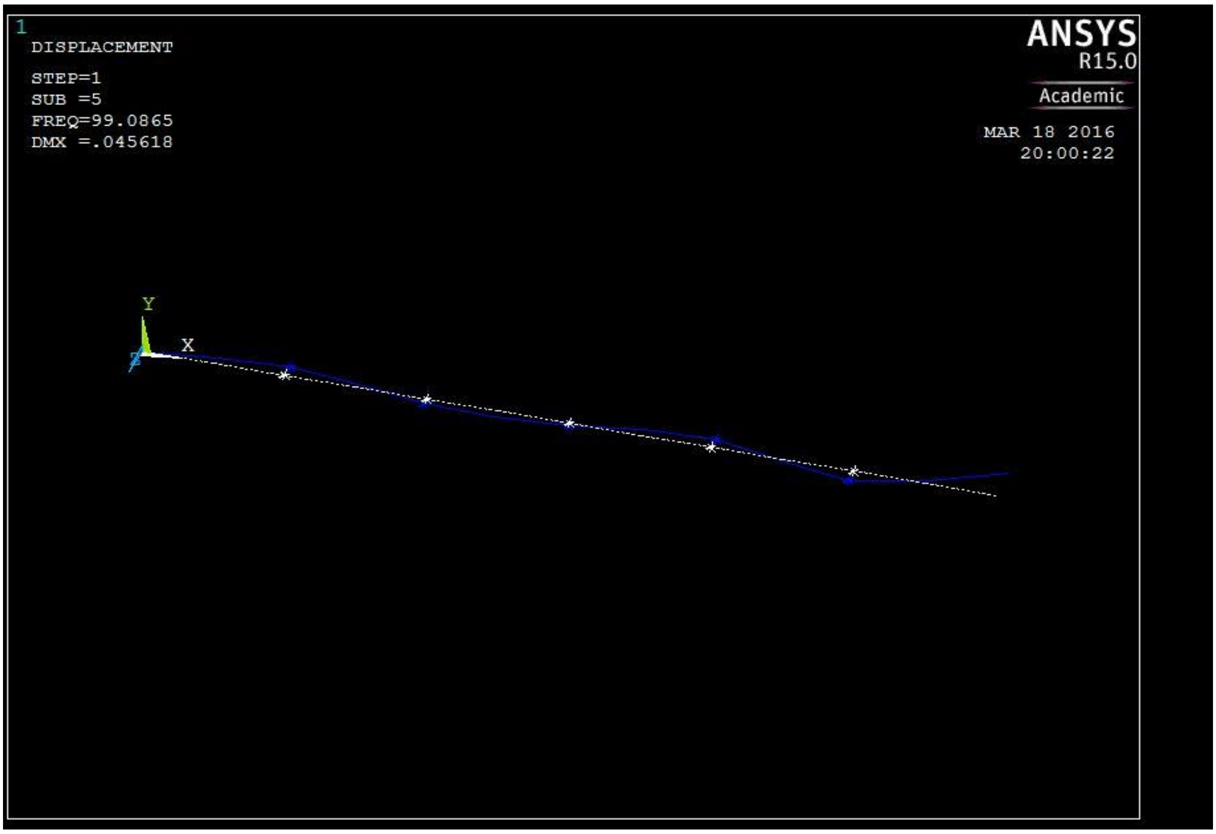
Mode3


1
DISPLACEMENT
STEP=1
SUB =3
FREQ=36.6276
DMX =.041697
ANSYS
R15.0
Academic
MAR 18 2016
18:09:17
Y
X

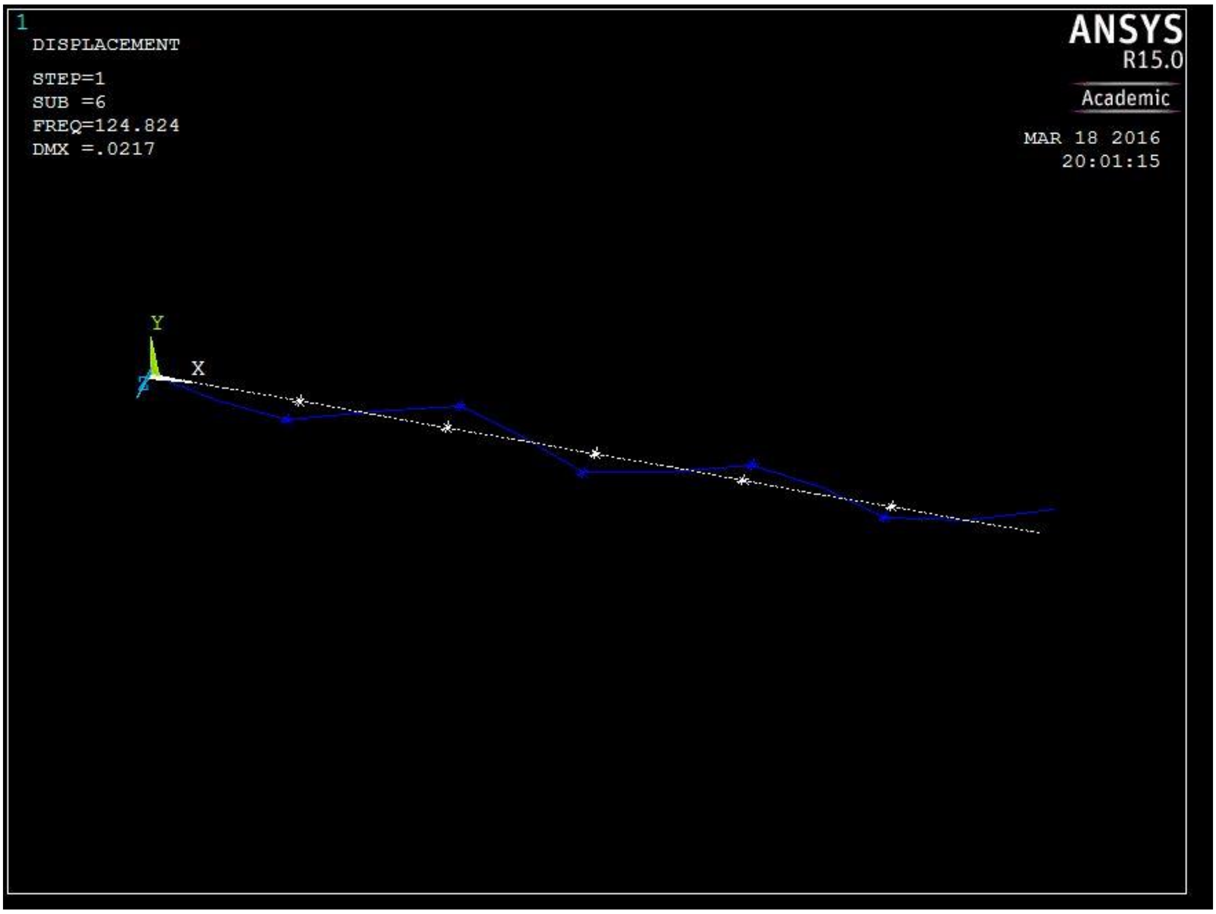
Mode4


1
DISPLACEMENT
STEP=1
SUB =4
FREQ=67.3254
DMX =.042436
ANSYS
R15.0
Academic
MAR 18 2016
18:09:38
Y
X

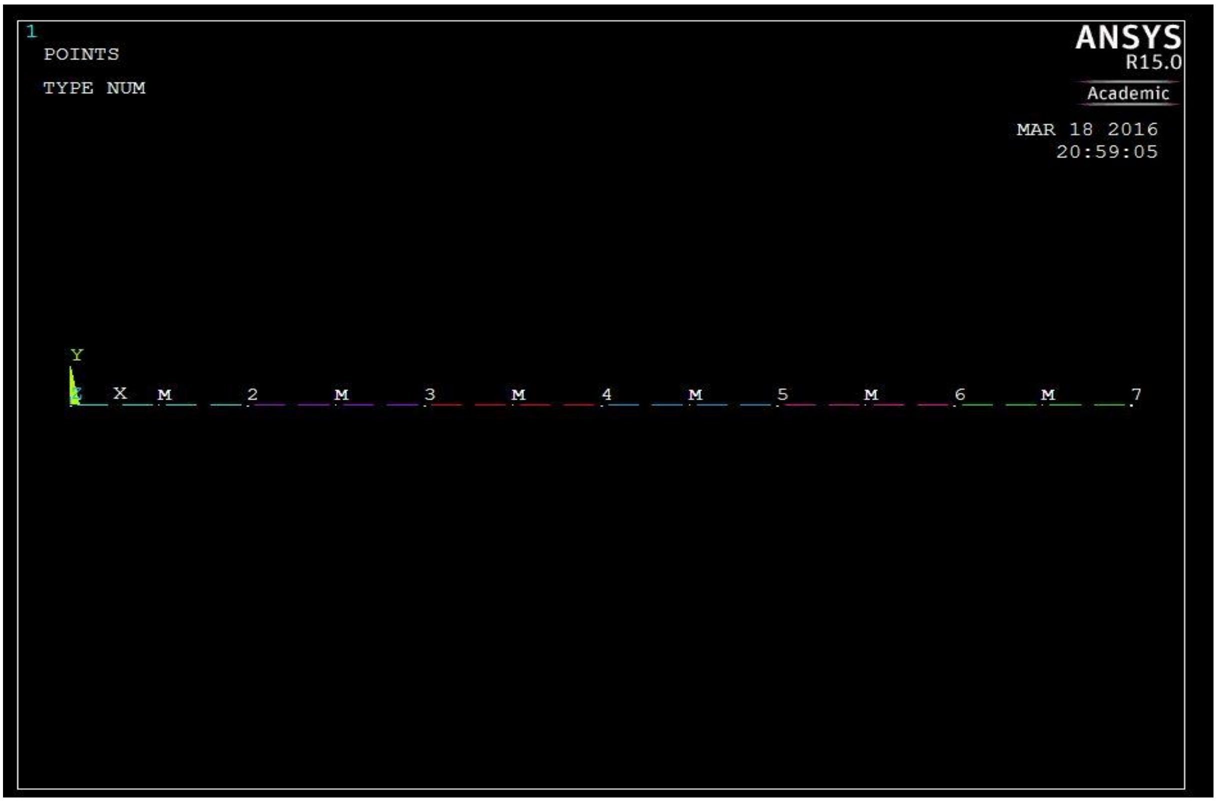
Mode5


1
DISPLACEMENT
STEP=1
SUB =5
FREQ=99.0865
DMX =.045618
ANSYS
R15.0
Academic
MAR 18 2016
20:00:22
Y
X
Z

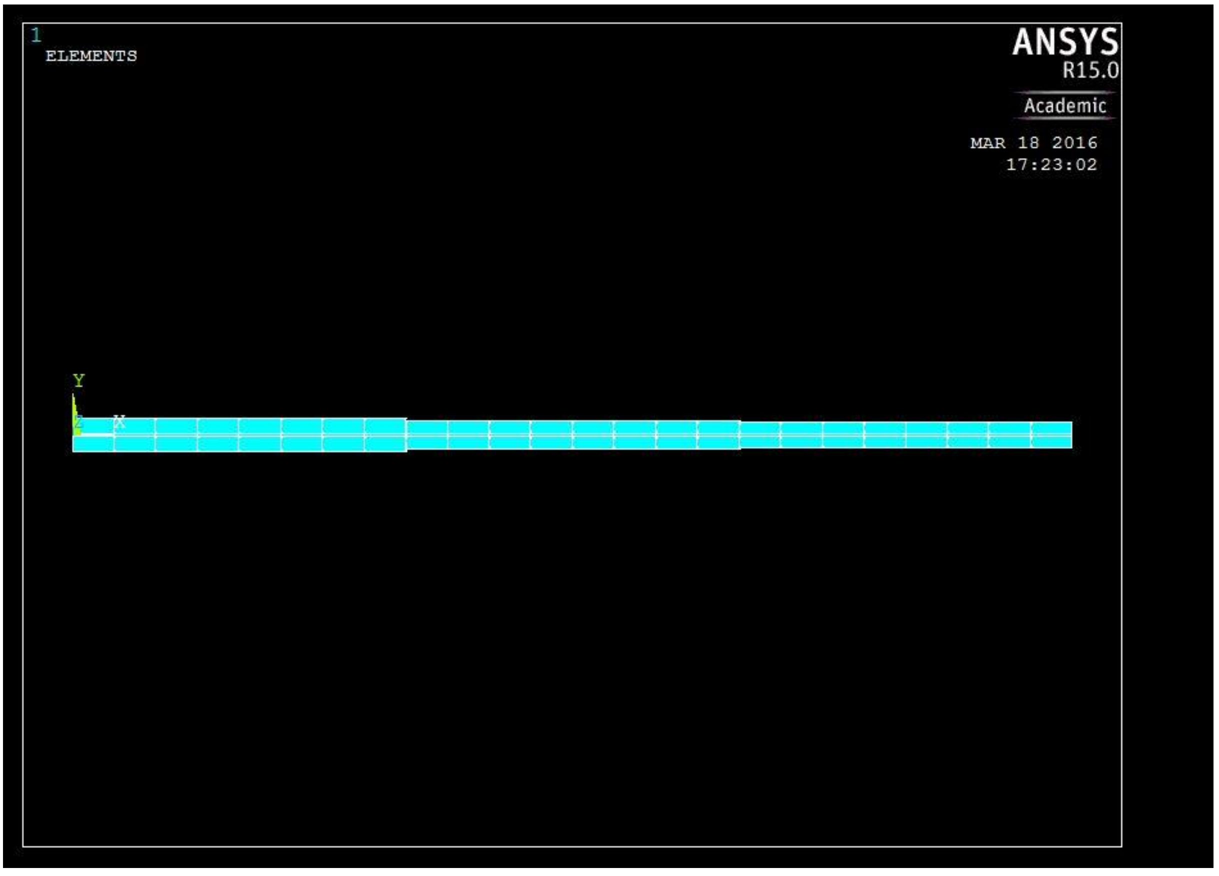
Mode6


1
DISPLACEMENT
STEP=1
SUB =6
FREQ=124.824
DMX =.0217
ANSYS
R15.0
Academic
MAR 18 2016
20:01:15
Y
X
Z

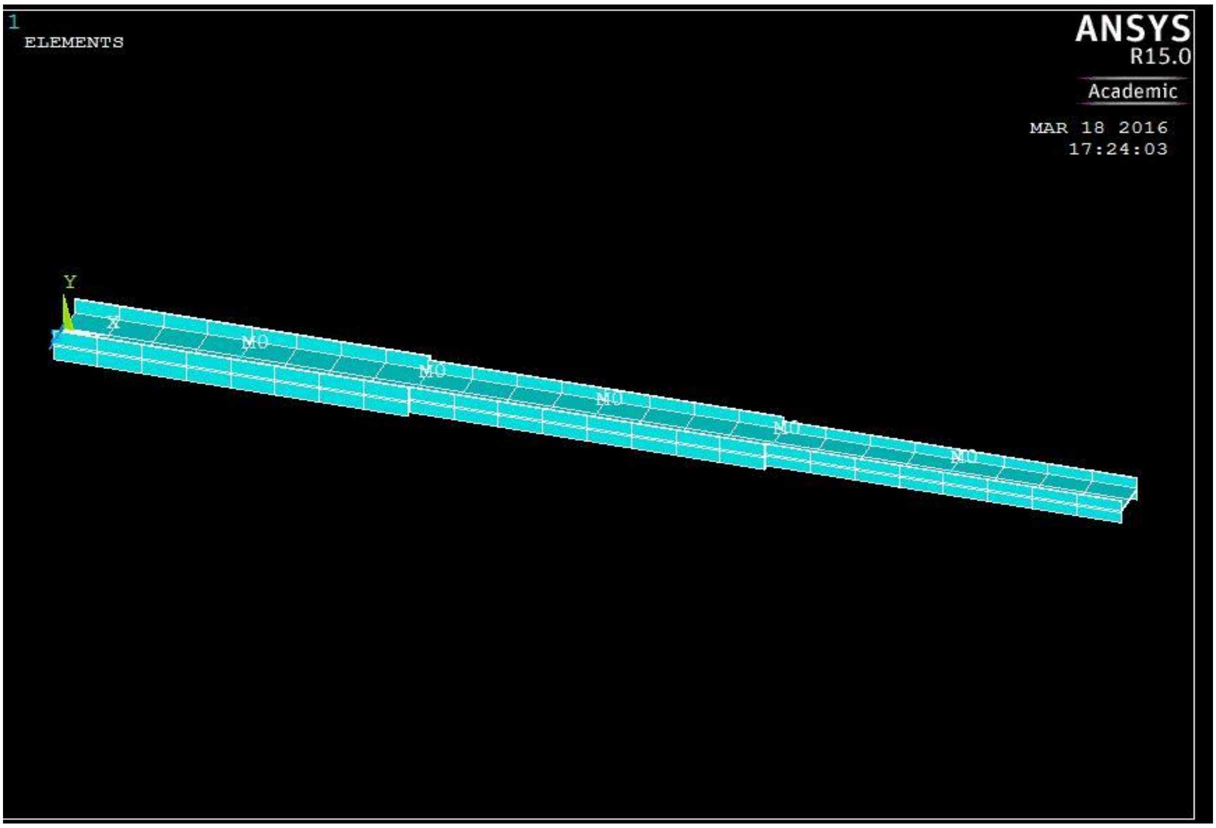
Beam with 4 divisions



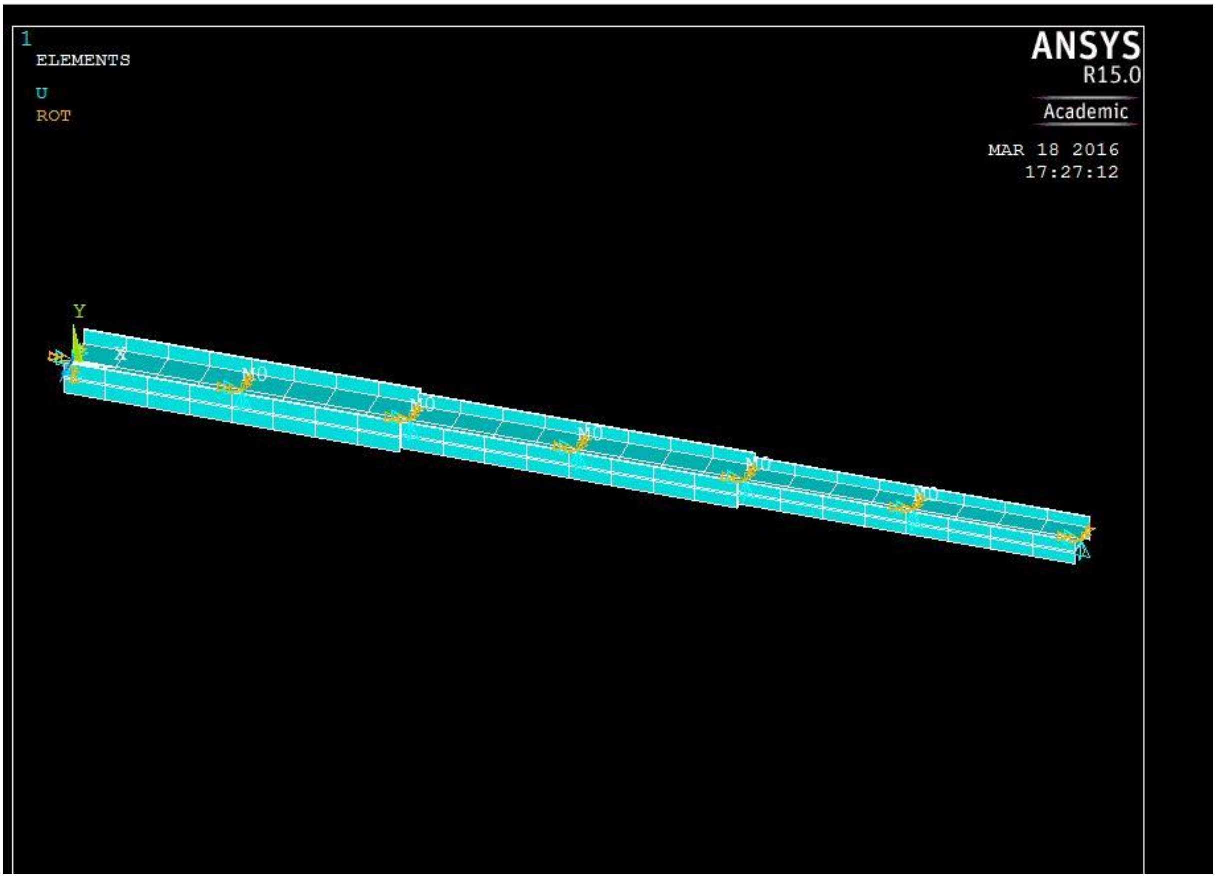
Meshed Beam with Front view



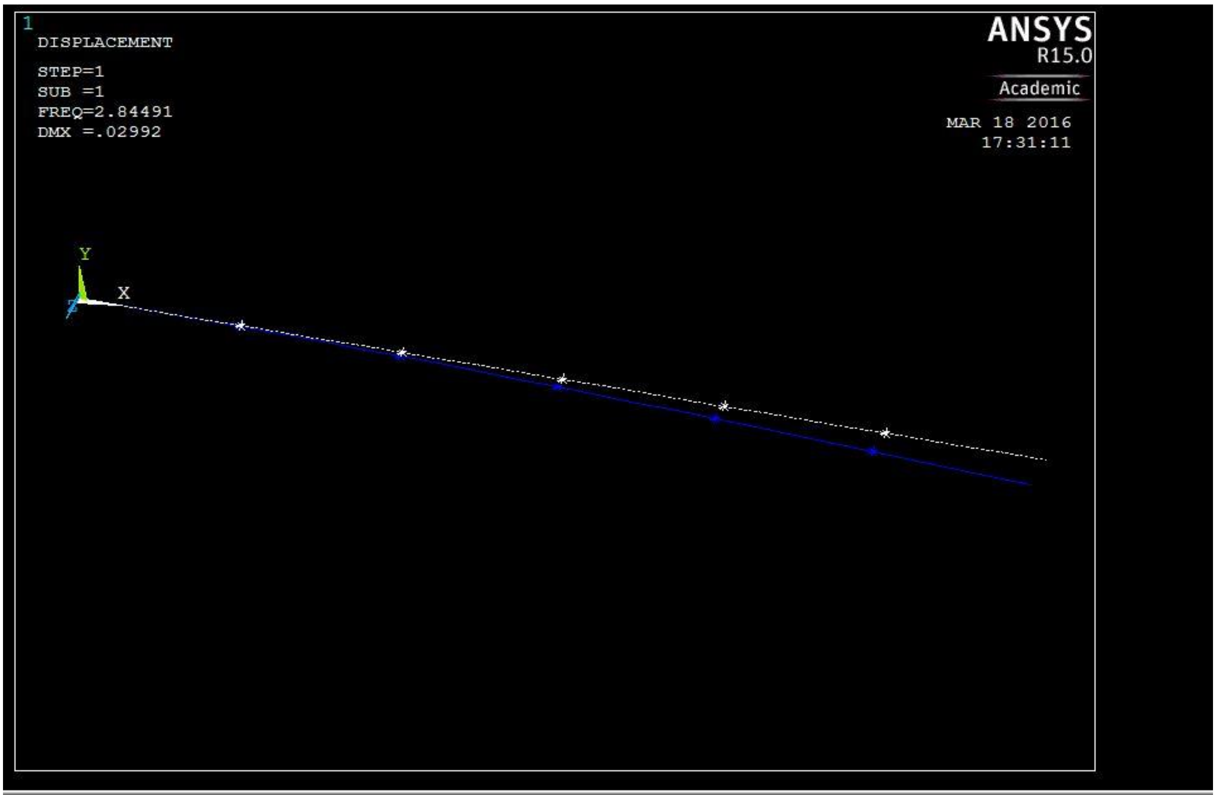
Meshed Beam with oblique view



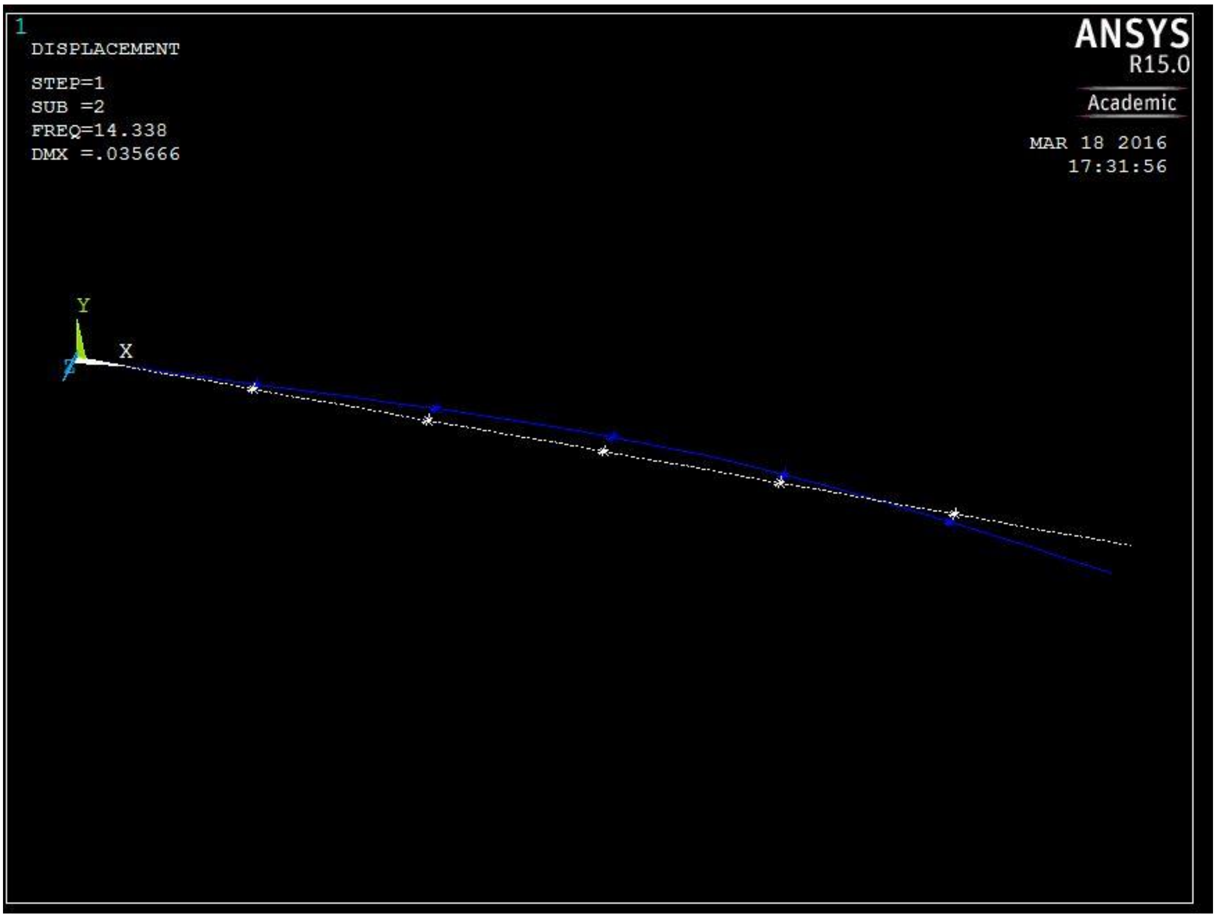
Meshed Beam with loads



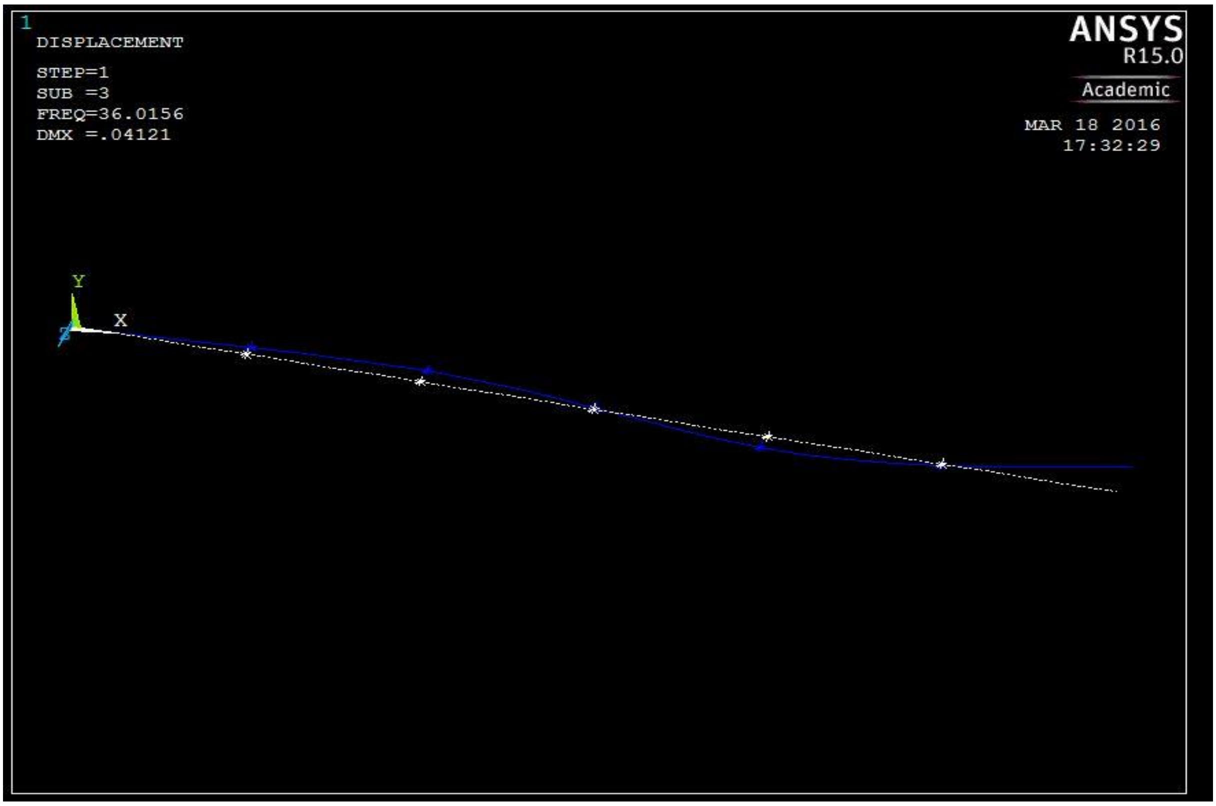
Mode1


1
DISPLACEMENT
STEP=1
SUB =1
FREQ=2.84491
DMX =.02992
ANSYS
R15.0
Academic
MAR 18 2016
17:31:11
Y
X

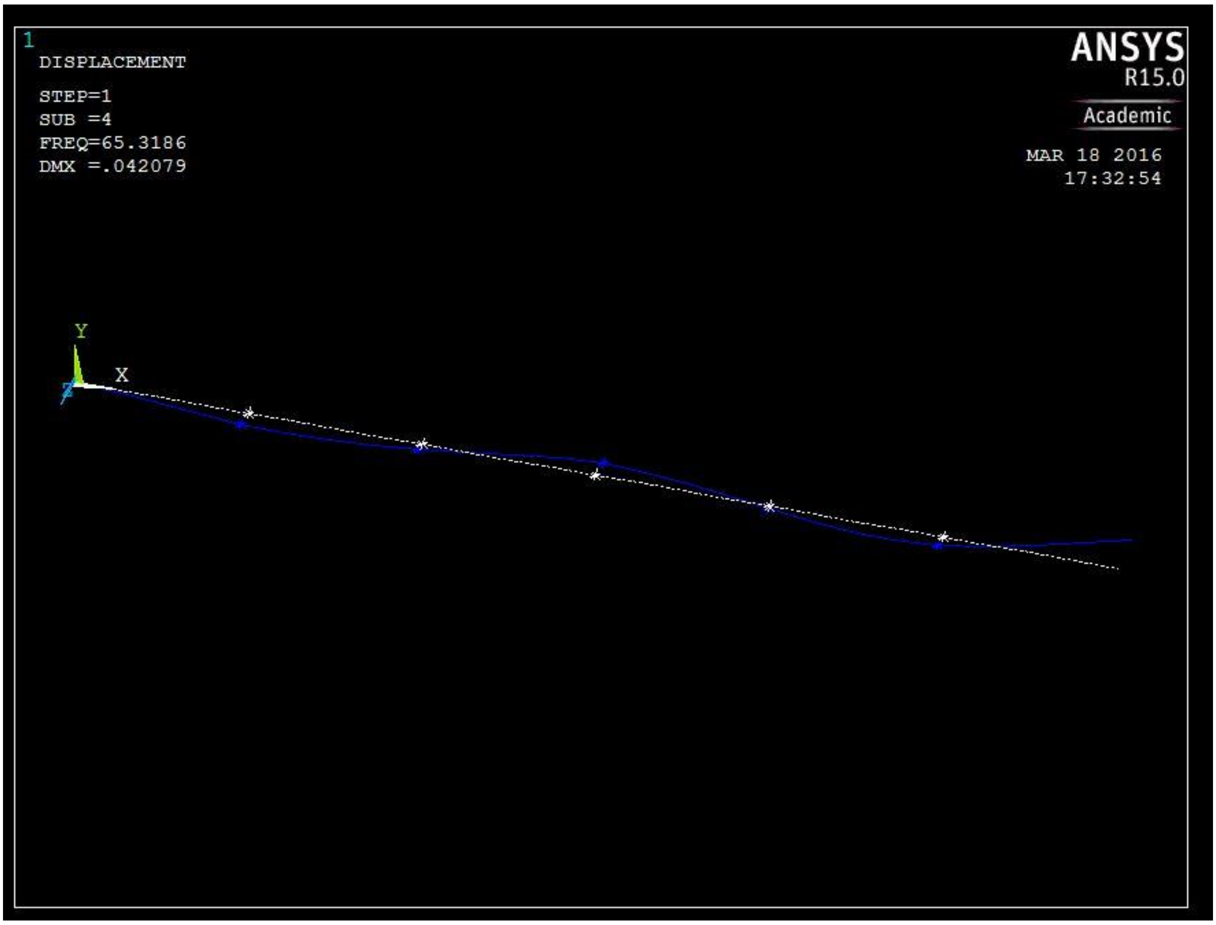
Mode2


1
DISPLACEMENT
STEP=1
SUB =2
FREQ=14.338
DMX =.035666
ANSYS
R15.0
Academic
MAR 18 2016
17:31:56
Y
X

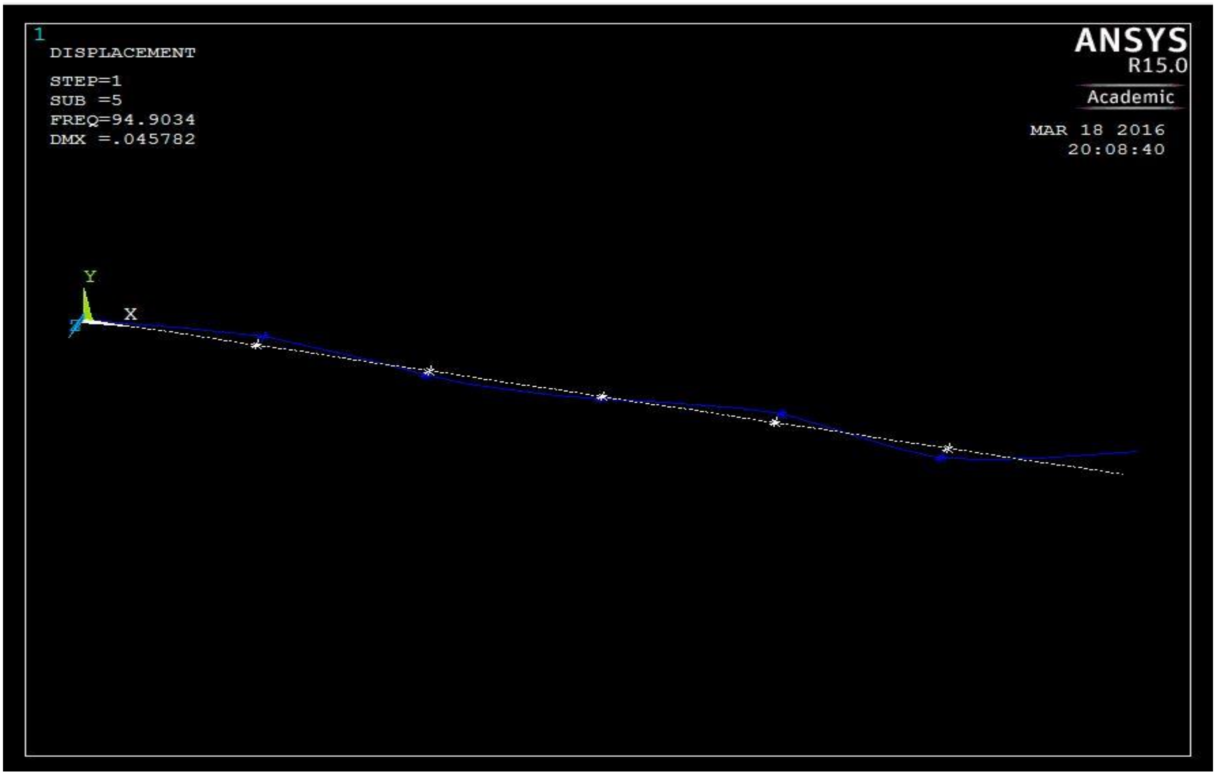
Mode3


1
DISPLACEMENT
STEP=1
SUB =3
FREQ=36.0156
DMX =.04121
ANSYS
R15.0
Academic
MAR 18 2016
17:32:29
Y
X
Z

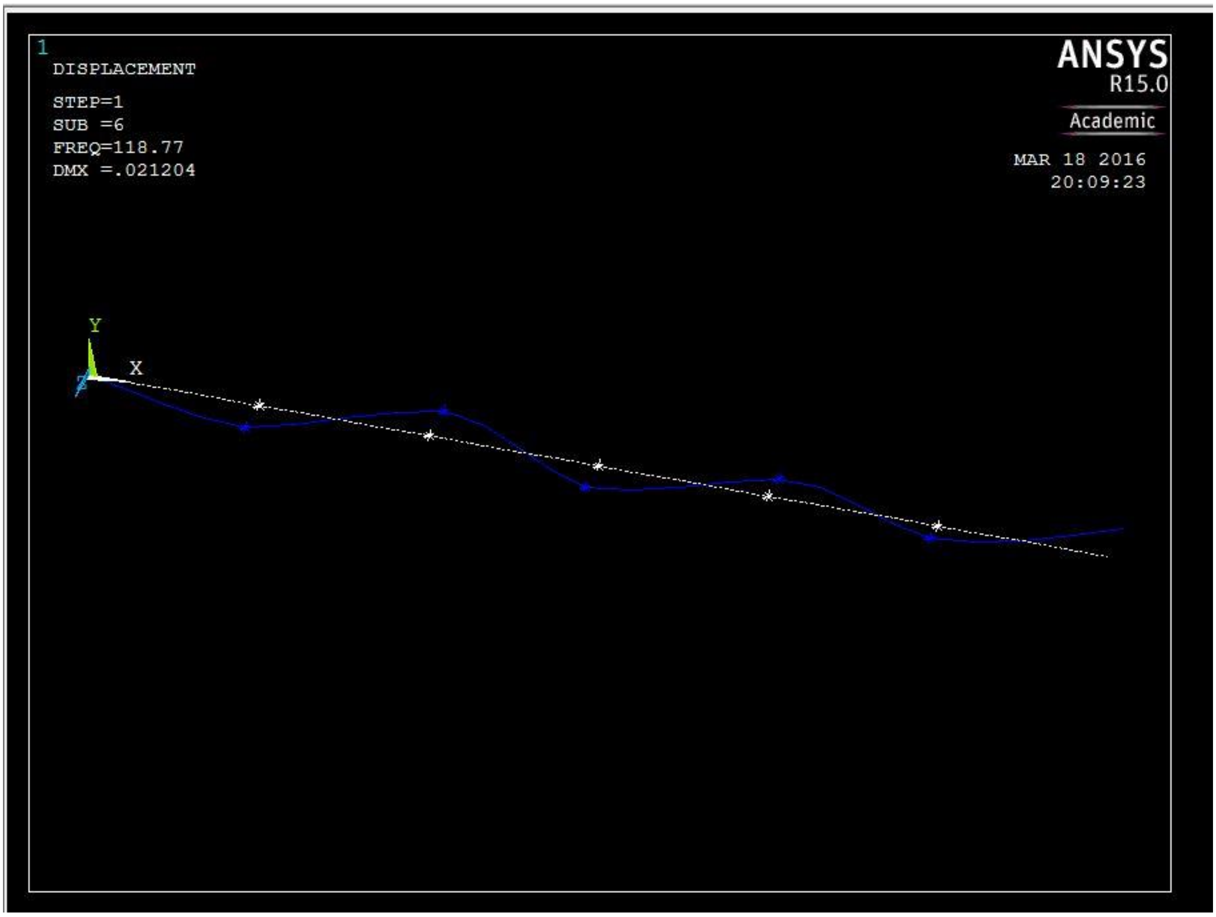
Mode4


1
DISPLACEMENT
STEP=1
SUB =4
FREQ=65.3186
DMX =.042079
ANSYS
R15.0
Academic
MAR 18 2016
17:32:54
Y
X
Z

Mode5


1
DISPLACEMENT
STEP=1
SUB =5
FREQ=94.9034
DMX =.045782
ANSYS
R15.0
Academic
MAR 18 2016
20:08:40
Y
X

Mode6


1
DISPLACEMENT
STEP=1
SUB =6
FREQ=118.77
DMX =.021204
ANSYS
R15.0
Academic
MAR 18 2016
20:09:23
Y
X

Task2

For architectural reasons, design (b) is chosen. A more detailed dynamic analysis of the system is now to be carried out using finite element software. The displacement response of the gallery to human action is to be evaluated. The critical load case is that all seats are occupied and all spectators are stomping and clapping their hands synchronously during a percussion

performance. This will cause a time-dependent load on the cantilever. The time-dependence is given as

$$p(t) = m_s g \left(1 + \alpha_1 \sin(2\pi f t) + \alpha_2 \sin(4\pi f t) + \alpha_3 \sin(6\pi f t)\right),$$

f [Hz]	α_1	α_2	α_3
2.4	0.38	0.12	0.02

The symbols m_S and g denote the total mass of the spectators at a particular point and the acceleration of gravity, respectively. The seats are arranged in 5 rows as indicated in Figure 3. There are 8 seats in each of the three back rows and 7 seats in each of the two front rows.

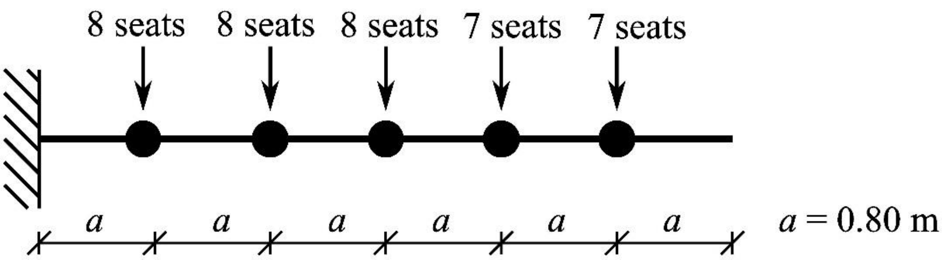


Figure 3: Additional masses / nodal forces due to spectators

Tasks

You are required to use the commercial software ANSYS to perform the following two tasks:

1. Find the first three natural frequencies and natural mode shapes of the cantilever
   corresponding to deformations of the beam in the x-y-plane. Compare these values to the results obtained in Part 1 of the assignment. Perform a

convergence study and show that your results for these natural frequencies are within an error tolerance of 2%.

2. Neglecting damping, determine the vertical displacement of the cantilever tip due to the human-induced load as a function of time. Consider 3 s of vibration. It is required that at least the first two natural modes of the gallery corresponding to deformations in the $x-y$-plane are captured in the dynamic response.

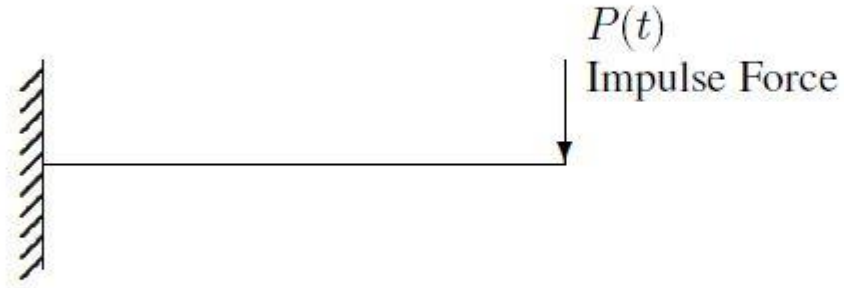
You may need to consult the user's manual to find out how to

- define and assign different cross-sections to individual beam members,
- Model additional distributed mass on beam members.
- Model additional lumped masses.
- define an arbitrary time variation of a load using the function tool.

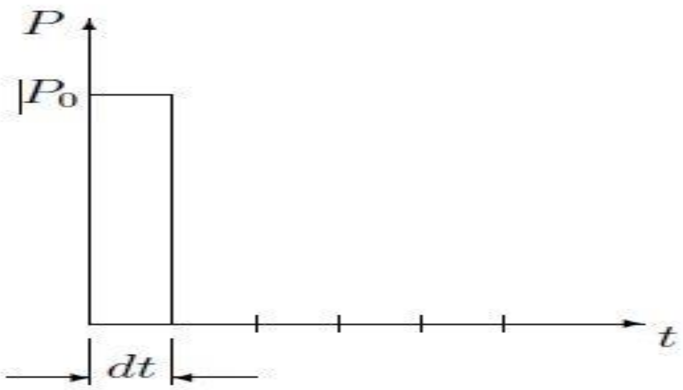
Transient Analysis of a Cantilever Beam:

Transient dynamic analysis is a technique used to determine the dynamic response of a structure under a time-varying load. The time frame for this type of analysis is such that inertia or damping effects of the structure are considered to be important. Cases where such effects play a major role are under step or impulse loading conditions, for example, where there is a sharp load change in a fraction of time.

If inertia effects are negligible for the loading conditions being considered, a static analysis may be used instead. For our case, we will impact the end of the beam with an impulse force and view the response at the location of impact.



Since an ideal impulse force excites all modes of a structure, the response of the beam should contain all mode frequencies. However, we cannot produce an ideal impulse force numerically. We have to apply a load over a discrete amount of time dt.



1. After the application of the load, we track the response of the beam at discrete time points for as long as we like (depending on what it is that we are looking for in the response).The size of the time step is governed by the maximum natural frequency of the structure we wish to capture. The smaller the time step, the higher the natural frequency we will capture.

The rule of thumb in ANSYS is time step = 1 / 20f

2.Where f is the highest natural frequency we wish to capture. In other words, we must resolve our step size such that we will have 20 discrete points per period of the highest natural frequency.

3. It should be noted that a transient analysis is more involved than a static or harmonic analysis. It requires a good understanding of the dynamic behavior of a structure.

Therefore, a modal analysis of the structure should be initially performed to provide information about the structure's dynamic behavior.
In ANSYS, transient dynamic analysis can be carried out using two methods.

The Full Method:

This is the easiest method to use. All types of non-linearity's are allowed. It is however very CPU intensive to go this route as full system matrices are used.

The Mode Superposition Method:

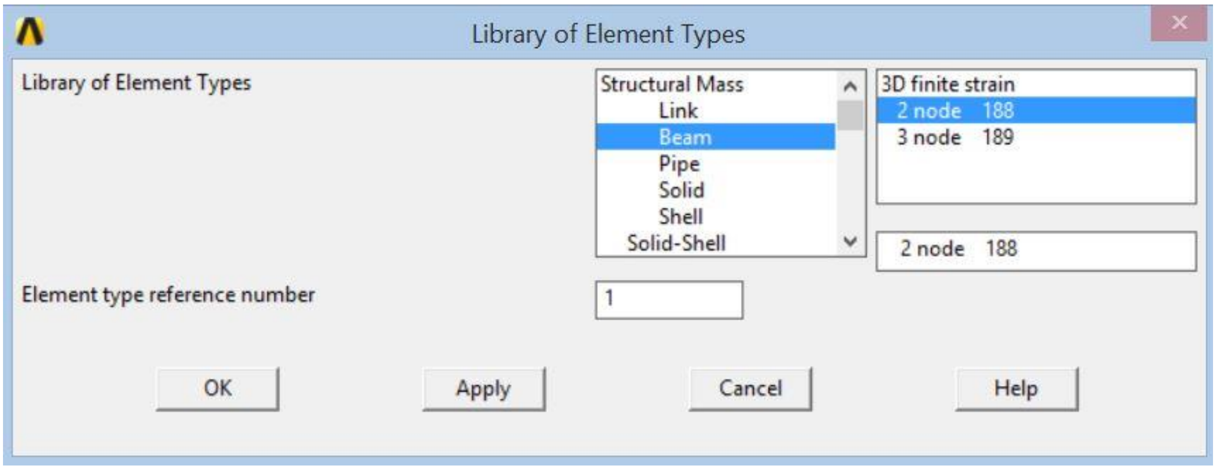
This method requires a preliminary modal analysis, as factored mode shapes are summed to calculate the structure's response. It is the quickest of the three methods, but it requires a good deal of understanding of the problem at hand.
We will use the Full Method for conducting our transient analysis.

By following Transient analysis procedure we will apply our load at the end of the cantilever beam in intervals of 0.01 sec up to 3 secs
We will plot the transient analysis graph for every sub step and the response will be as shown in figure for mode1

Transient Analysis Procedure Ansys

1. Open preprocessor menu
2. Give example a Title
   Utility Menu > File > Change Title ...
3. Give example a Jobname
   Utility Menu > File > Change Jobname ...
   Enter ’Cantilever_dynamic’ for the jobname

4.Element type >Add/edit >popup >Add >Beam >2node188 >apply

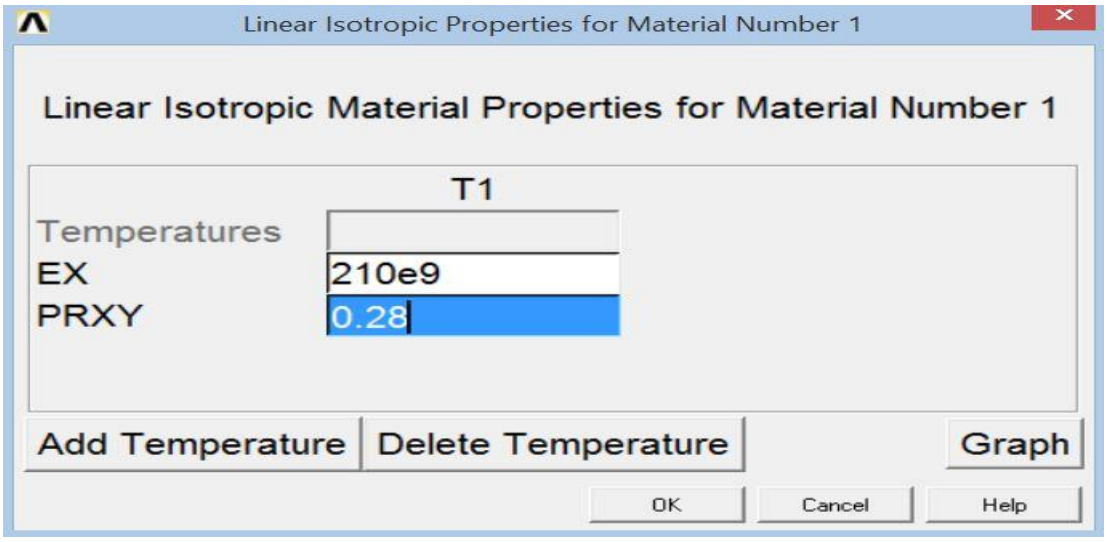


5.Material Properties >Material Models >Structural >Linear >Elastic >isotropic

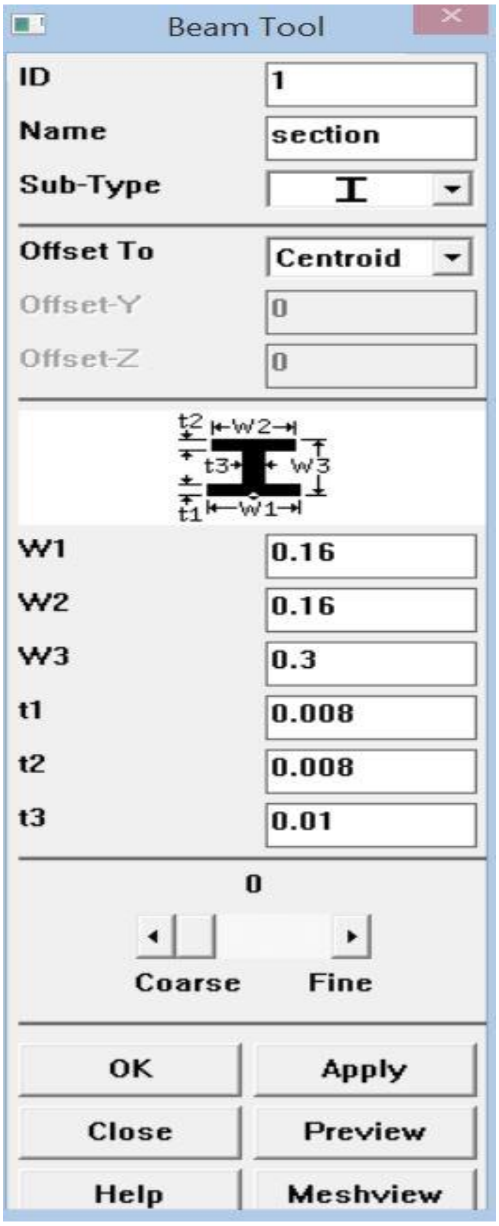
E=210e09

ϑ=0.28

Density=7850



6.sections >Beam >Common Sections



Give values as shown for other sections

7.Sections >Beam >Section Control >popup

Sec1 Add mass/length=500

Sec2 Add mass/length=500

Sec3 Add mass/length=500

keypoints	coordinatinates(x,y)
1	(0.8*0,0)
2	(0.8*1,0)
3	(0.8*2,0)
4	(0.8*3,0)
5	(0.8*4,0)
6	(0.8*5,0)
7	(0.8*6,0)

8.Modelling>Create >Lines>popup

Straight Line join with arrow ok

9.Meshing>Size Controls>Manual Size>Lines>All Lines>pop Up>

No of Element Divisions 1> ok

10.Meshing>Meshing Attributes>Picked Keypoints

2,3,4 ok>popup

Real Set1 >Element type> Mass 2l>ok

Meshing>Meshing Attributes>Picked Keypoints

5,6 ok>popup

11.Meshing>Meshing Attributes>

Picked Lines

1,2>Beam 188 >sec1>ok

Picked Lines

3,4>Beam 188 >sec2>ok

Picked Lines

1,2>Beam 188 >sec3>ok

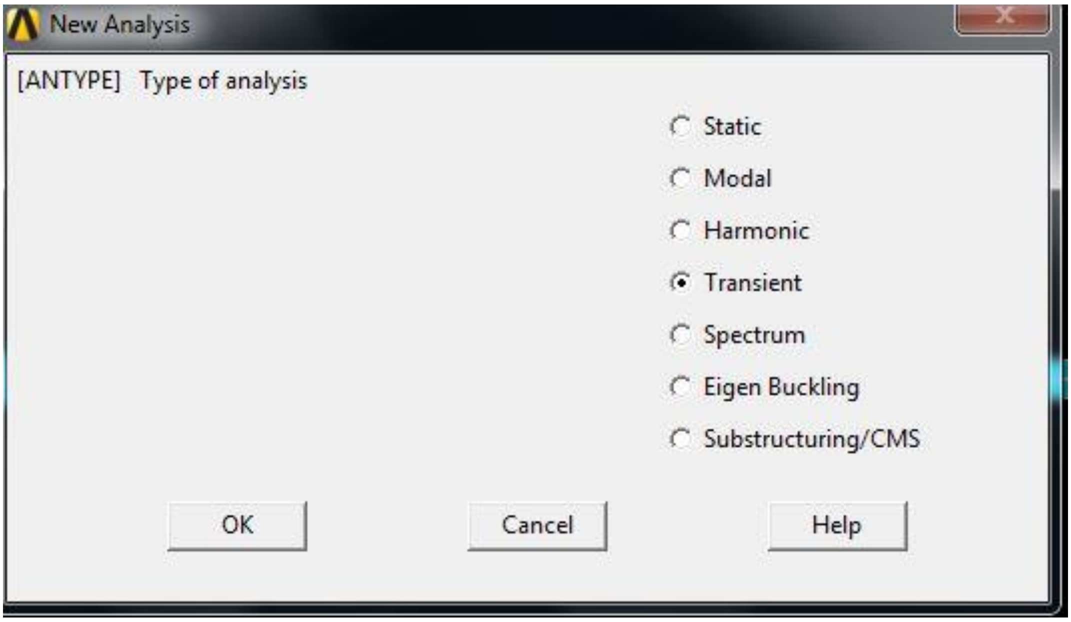
12. Meshing>Mesh>Key Points

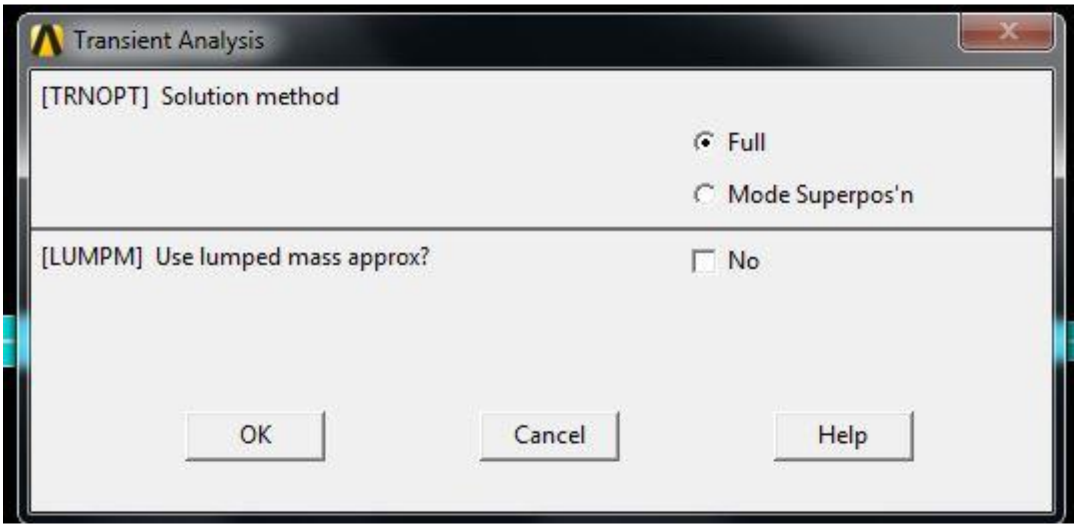
Select all middle Keypoints>ok

Meshing>Mesh>Lines>Pick All>Ok

13.Plot Controls>Style>Size and Shape>pop up >Display of Element >on>ok

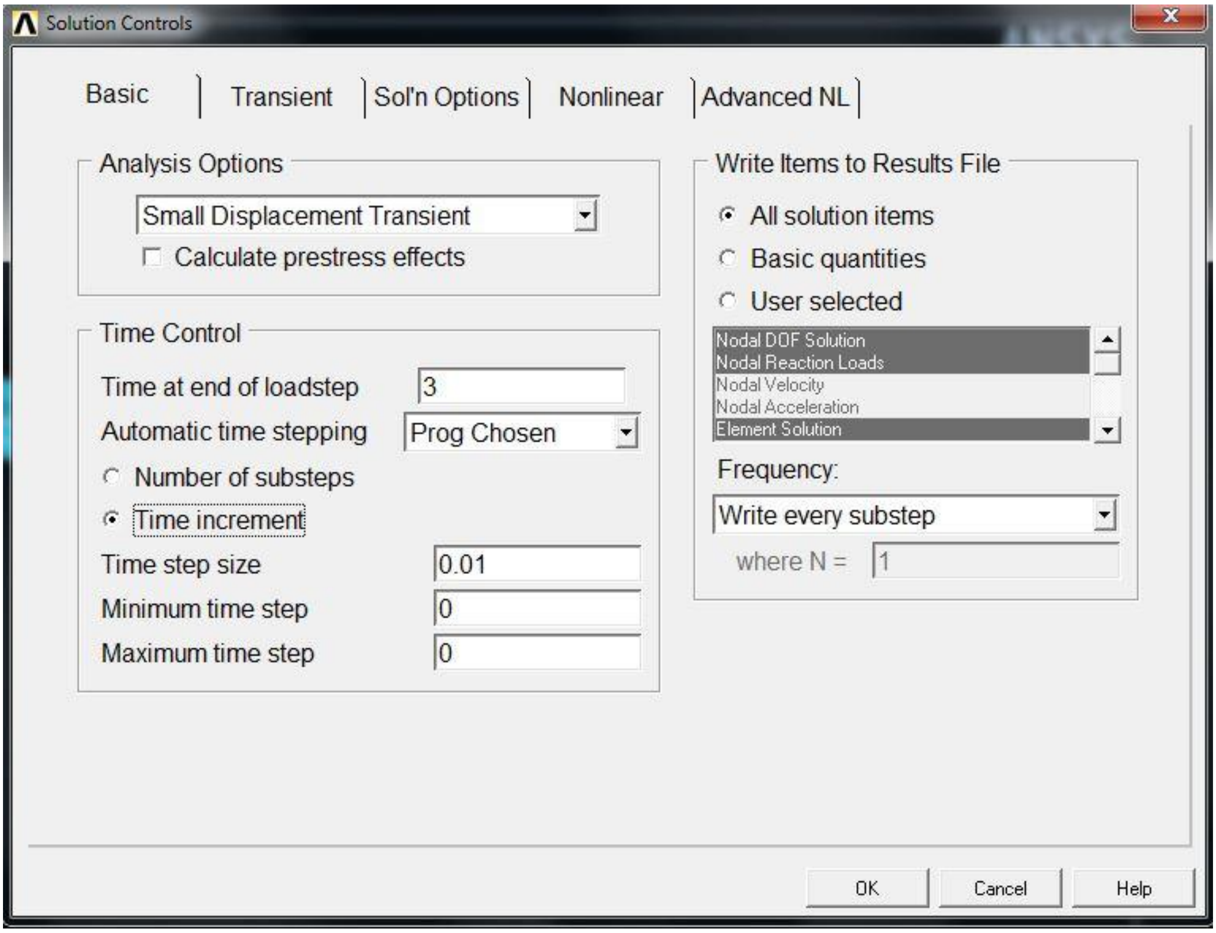
14.Solution>Analysis Type>New Analysis>pop up>Transient>ok>popup>Full>Ok



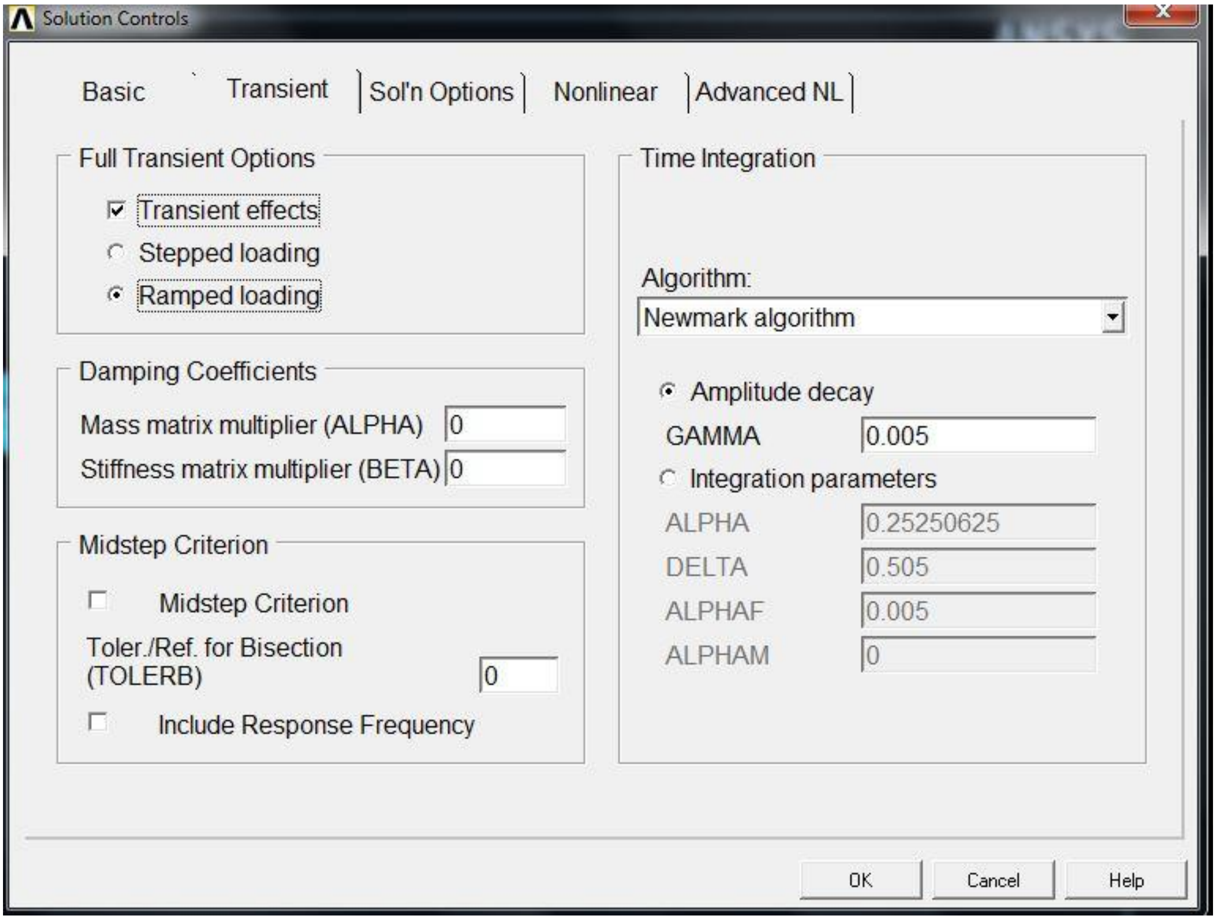


15. Solution>Analysis Type>Sol’n Controls Solution>ok

Give the Values as shown



Solution Controls
Basic
Transient
Sol'n Options
Nonlinear
Advanced NL
Analysis Options
Small Displacement Transient
Calculate prestress effects
Time Control
Time at end of loadstep
3
Automatic time stepping
Prog Chosen
Number of substeps
Time increment
Time step size
0.01
Minimum time step
0
Maximum time step
0
Write Items to Results File
All solution items
Basic quantities
User selected
Nodal DOF Solution
Nodal Reaction Loads
Nodal Velocity
Nodal Acceleration
Element Solution
Frequency:
Write every substep
where N =
1
OK
Cancel
Help



Solution Controls
Basic
Transient
Sol'n Options
Nonlinear
Advanced NL
Full Transient Options
Transient effects
Stepped loading
Ramped loading
Damping Coefficients
Mass matrix multiplier (ALPHA)
0
Stiffness matrix multiplier (BETA)
0
Midstep Criterion
Midstep Criterion
Toler./Ref. for Bisection (TOLERB)
0
Include Response Frequency
Time Integration
Algorithm:
Newmark algorithm
Amplitude decay
GAMMA
0.005
Integration parameters
ALPHA
0.25250625
DELTA
0.505
ALPHAF
0.005
ALPHAM
0
OK
Cancel
Help

16.Solution>Define Loads>Apply>Structural>Displacement On Nodes

Select 1st Node>ok>popup>

Select All Dof>ok

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

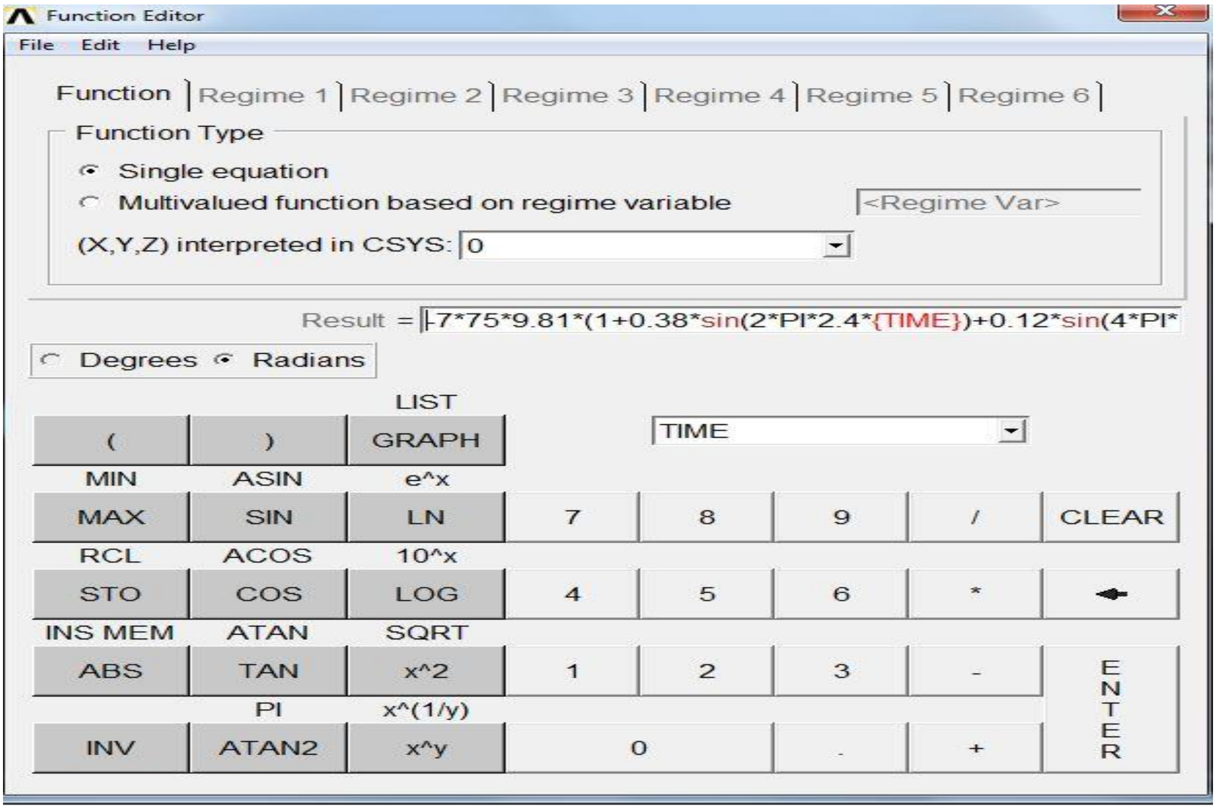
Select All other Nodes>ok>popup>

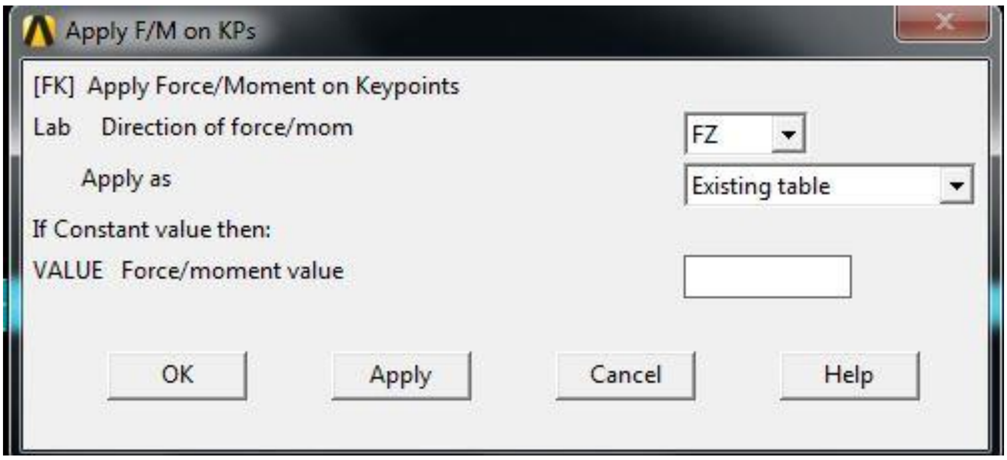
Select Ux,Uy,Rotx,Rotz>ok

17. Main Menu>Parameters>Functions>Define/Edit

Here give the value of load as shown

And save them as functions





18. Main Menu>Parameters>Read From file>

Load previously defined loads

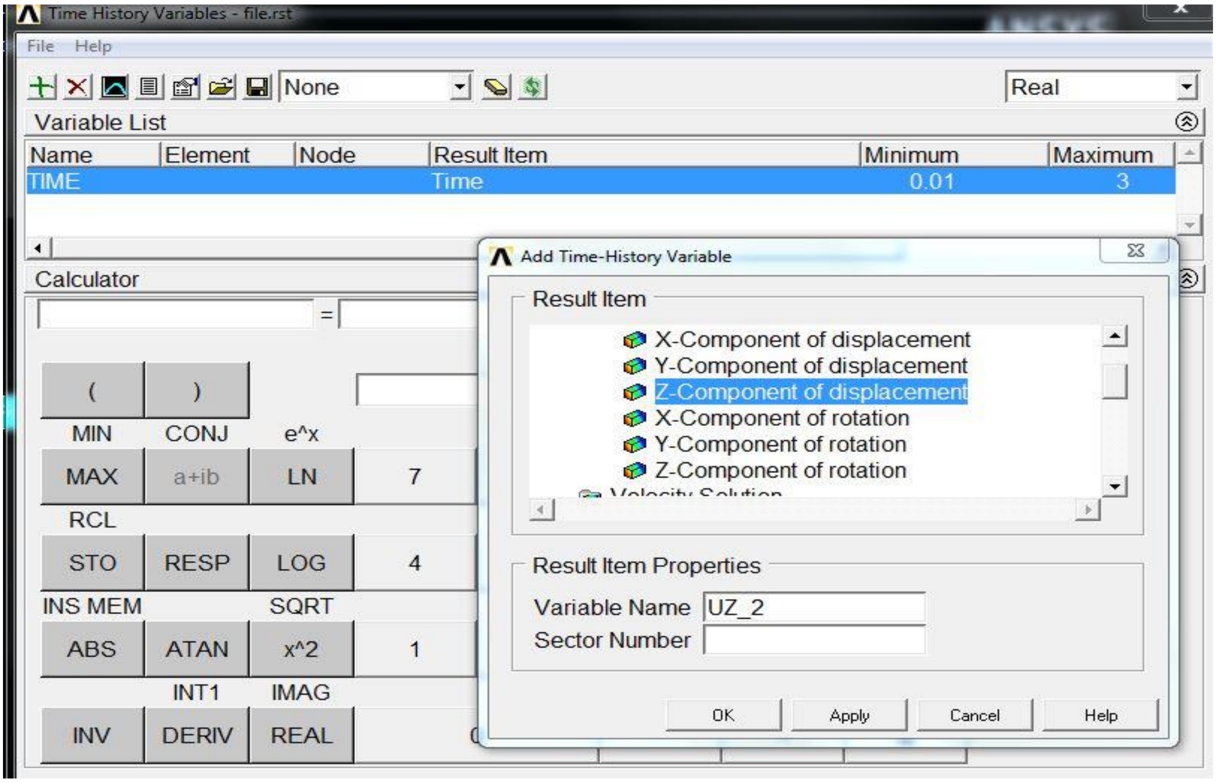
19. Solve>Current LS>ok

20.Time History PostPro>PopUp>

Press plus symbol

For mode 1 Dof Z Direction Of Displacement

For Mode2 Dof Y Direction Of Rotation

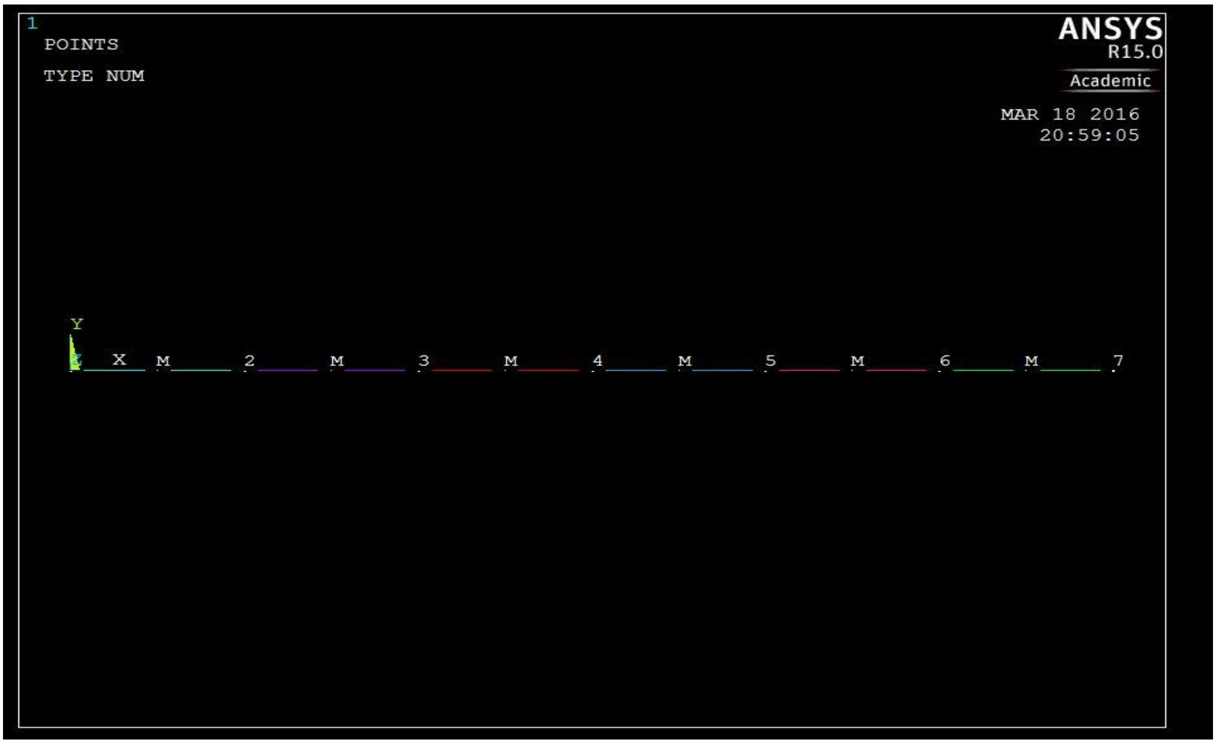


21.Press Graph symbol in the pop up menu

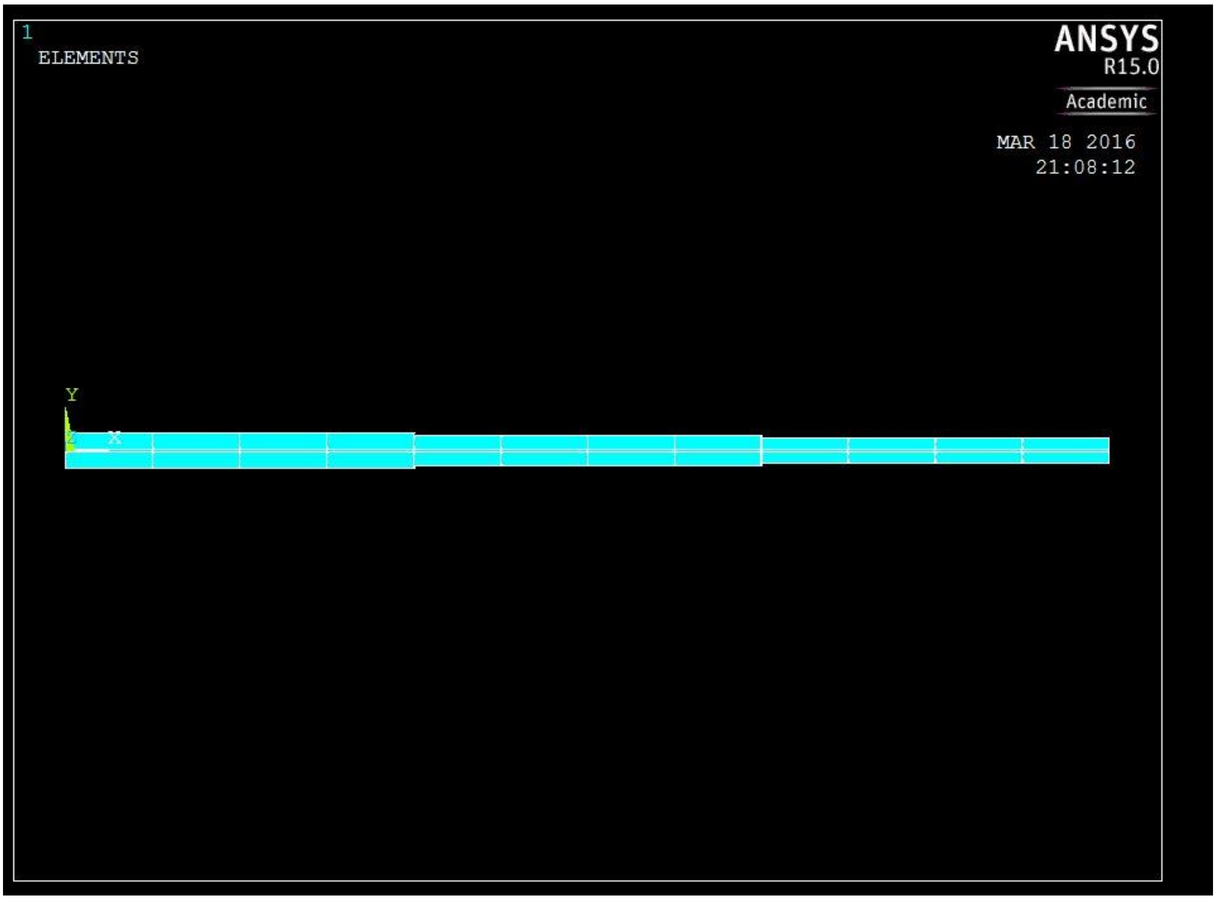
We get a transpose response of mode1 & mode2 at tipoff the cantilever

Transient Analysis Results

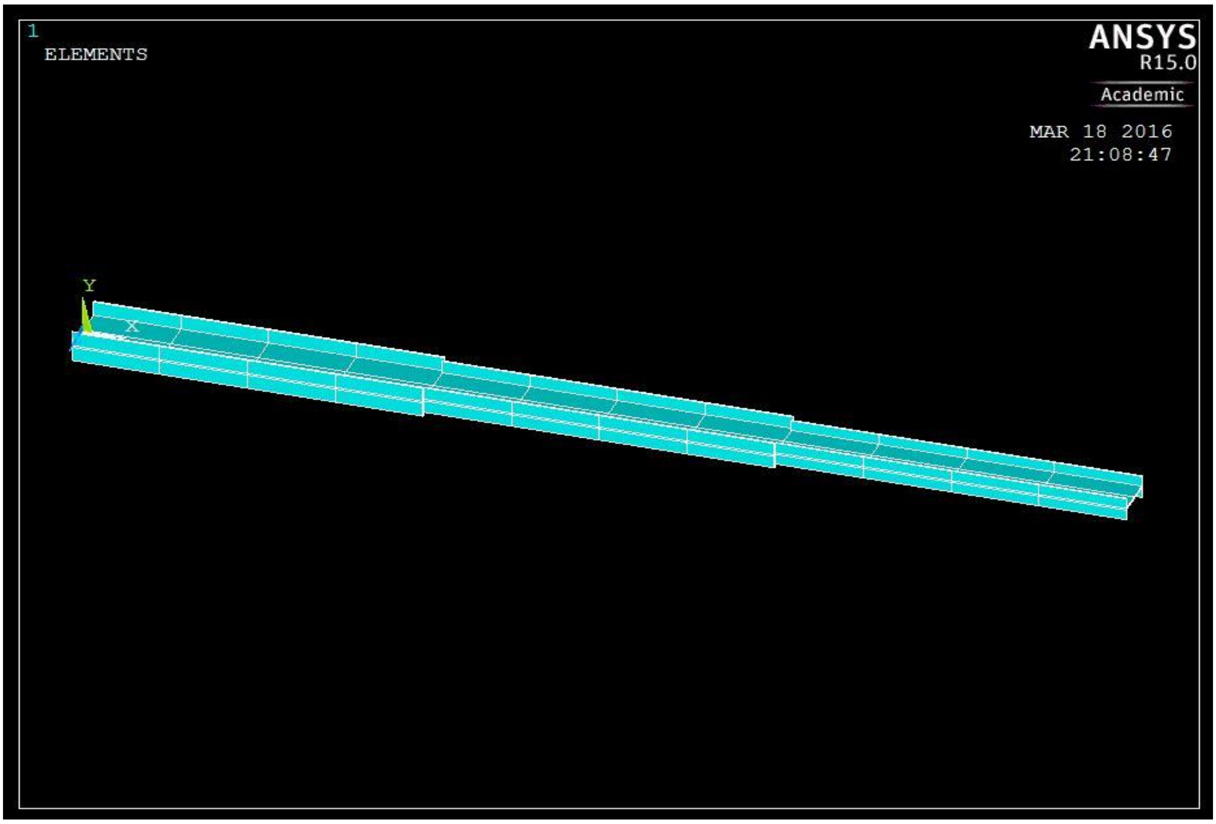
Transient Response



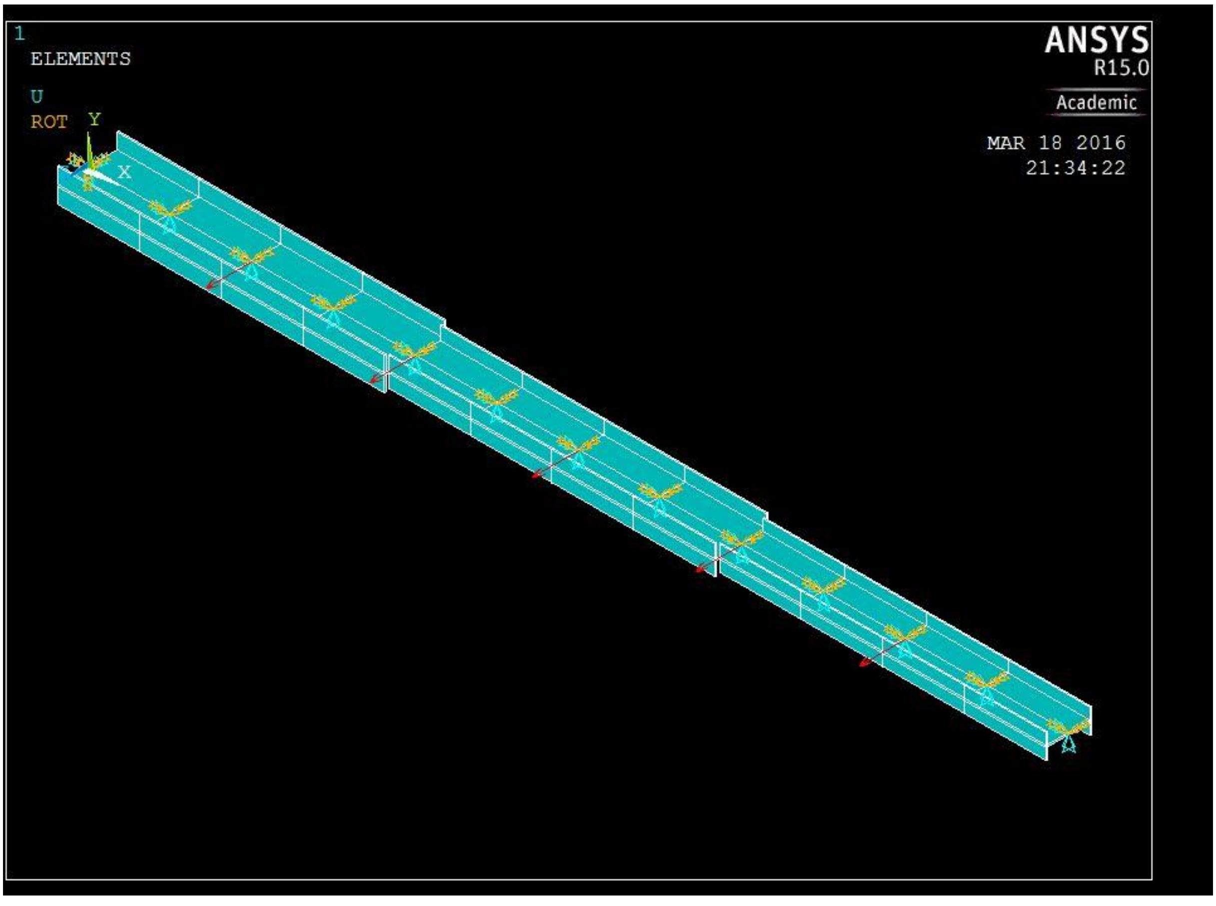
Meshed Beam Front View



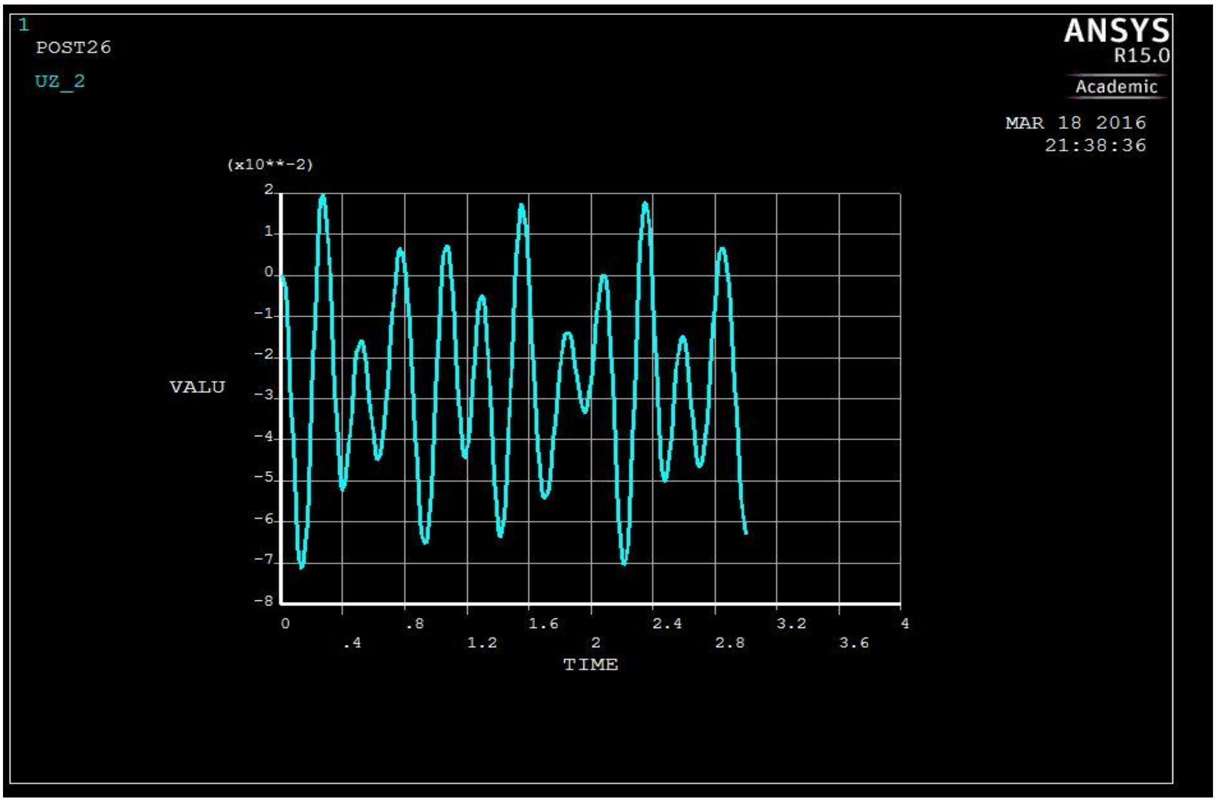
Meshed beam with oblique view



Meshed beam with applied load

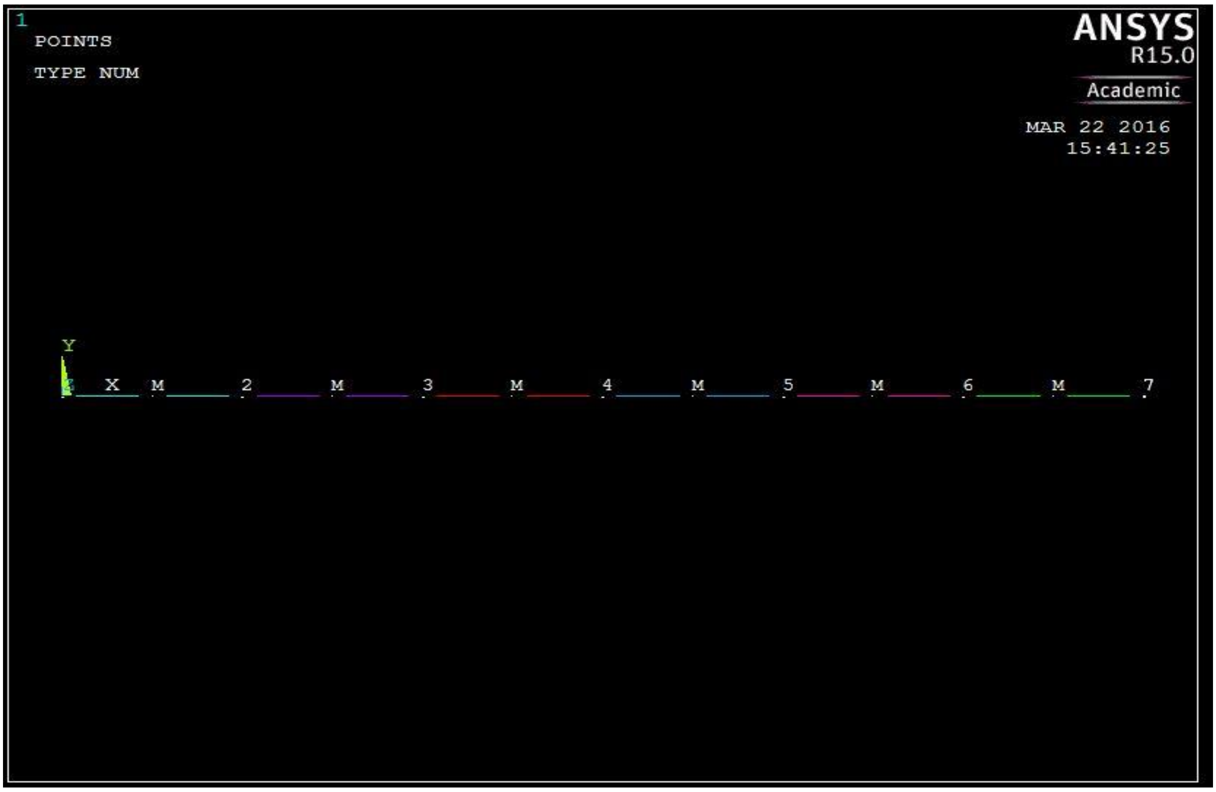


Transient Time History Graph for mode1

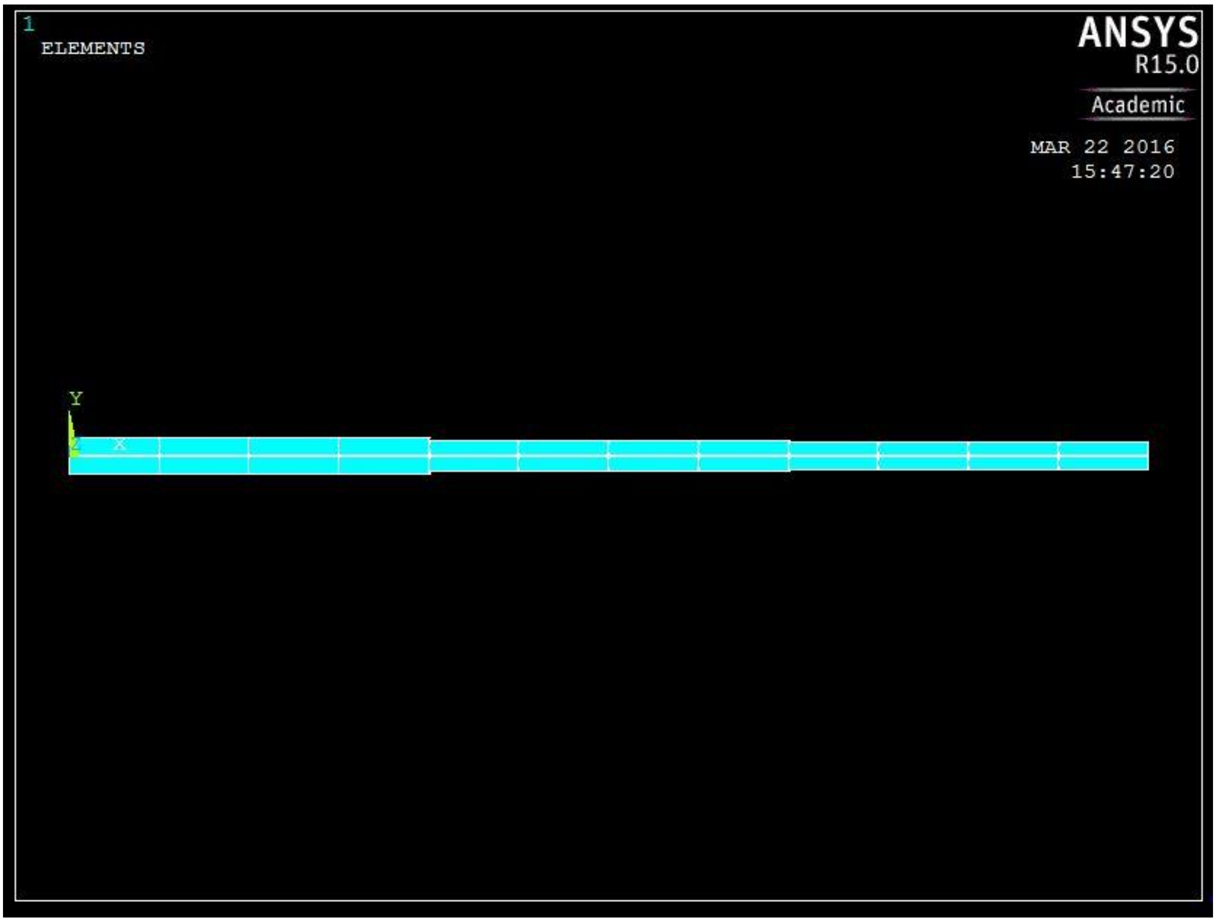


Transient Analysis for mode2

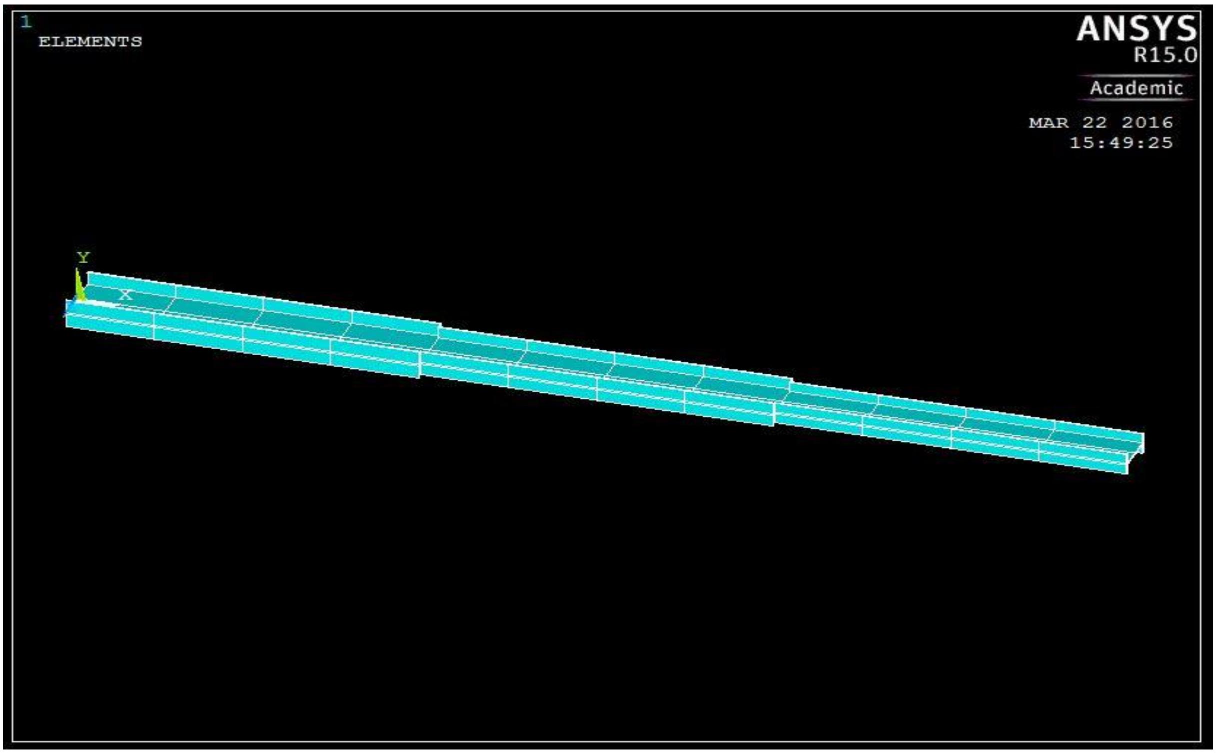
2 divison of meshed beam



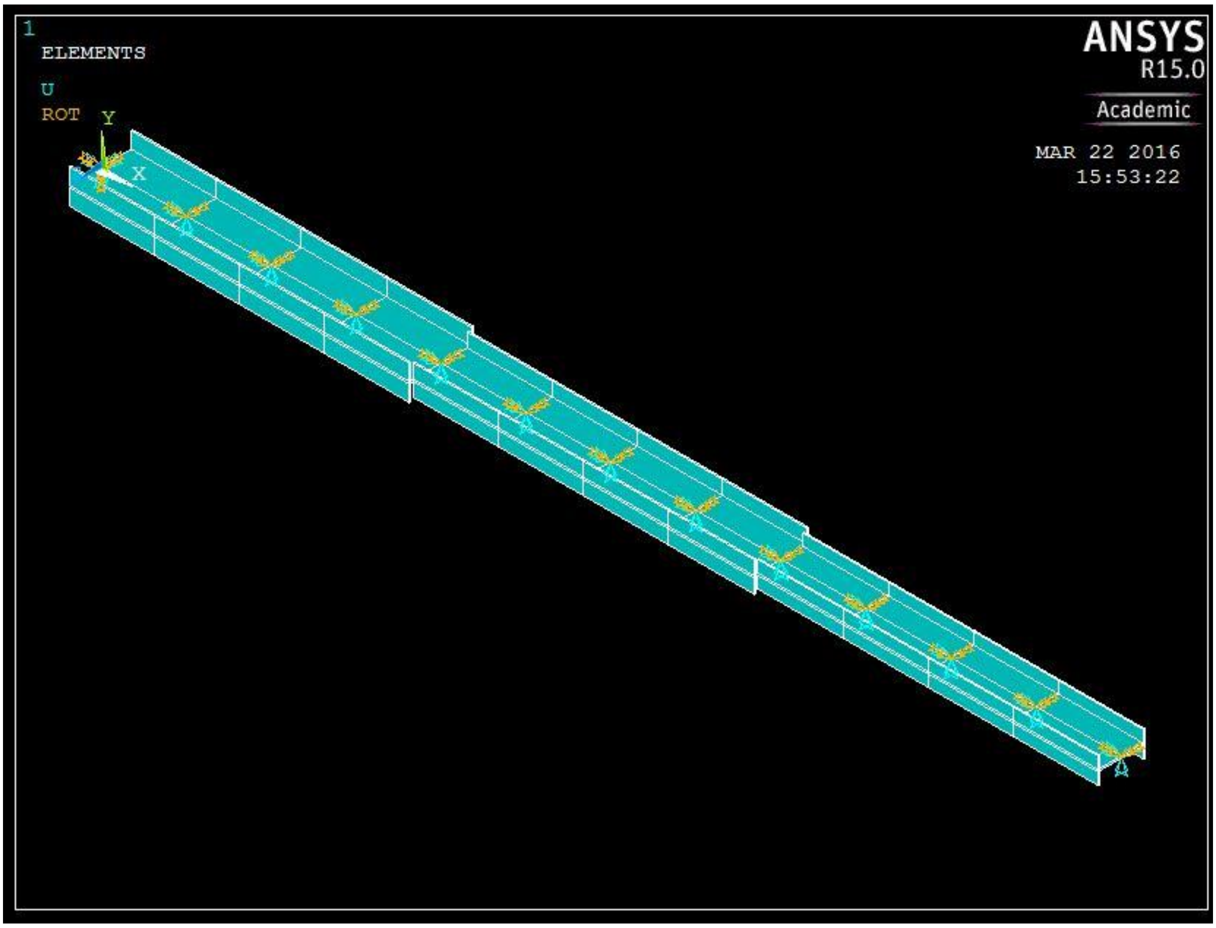
Front view of meshed beam



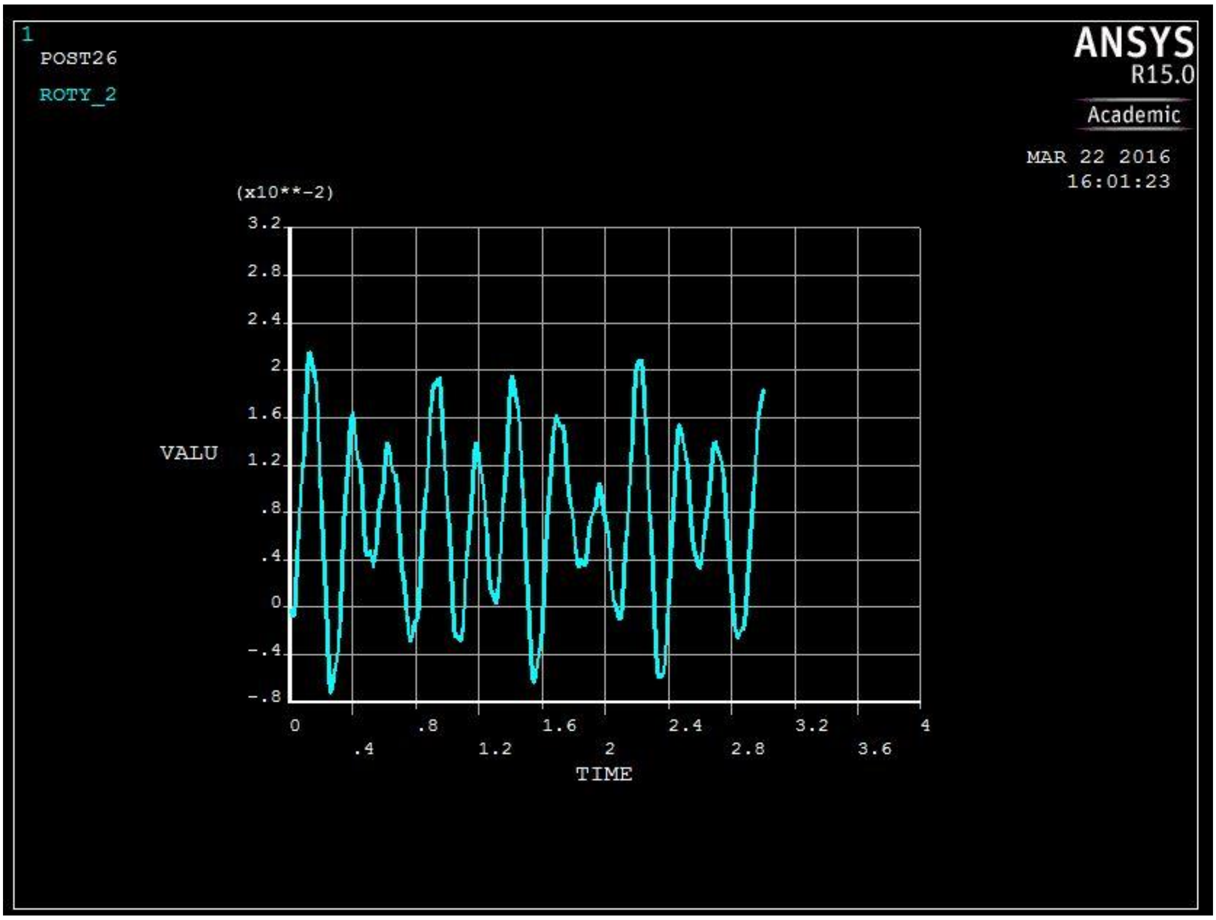
Meshed beam with oblique view



Meshed beam with loads



Transient Response Mode2



Results & Conclusion:

The analytical solution for the natural frequencies of a simply supported uniform Euler-Bernoulli beam From Ray W. Clough and J. Penzien, Dynamics of Structures, McGRAW-HILL, 1975 is given by

$$\omega_1 = (1.875)^2 \sqrt{\frac{EI}{mL^4}}$$

$$\omega_2 = (4.694)^2 \sqrt{\frac{EI}{mL^4}}$$

$$\omega_3 = (7.855)^2 \sqrt{\frac{EI}{mL^4}}$$

Here for Design (a)

$$E = 210 \times 10^9 N/m^2$$
$$I = 9.4751 \times 10^{-5} m^4$$
$$m_e = 1141 \times 1.6 Kg/(ElementalMass)$$
$$L = 4.8$$

Total Mass $m = 3 * m_e$

Substituting all the values we have we get

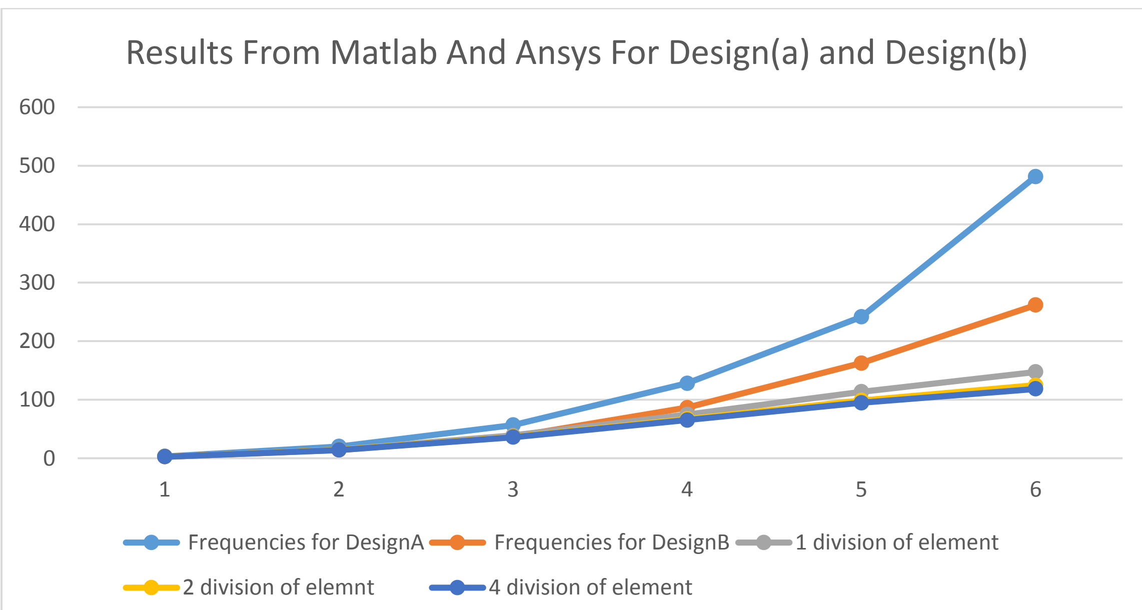
	Frequencies from Euler - Bernoli	Matlab Results Design(a)	Matlab Results Design(b)
Frequency 1	3.207	3.2077	2.6743
Frequency2	20.0994	20.1665	14.1149
Frequency3	56.2845	56.9833	37.2506

From above Graph, we can say that Matlab results (by FEM procedure) and Euler_bernoli Results for Design (a) are almost same for first three natural frequencies. We can also say that Design (b) is better estimation than Design (a) because Design (b) is having lesser frequencies than the Euler_Bernoli Estimation for Cantilever Beam and the design of Cantilever Beam for Design (b) is helpful in reducing the frequency of Vibration. By this chances of Resonance occurrence is low because the system could not be excited lower natural frequencies than the natural easily. When the frequencies of the system is low, then displacement of the system is also low. Therefore, Design (b) is better design than Design (a).

The results from Matlab and Ansys are as shown

In Ansys analysis is carried out using different division of elements as shown

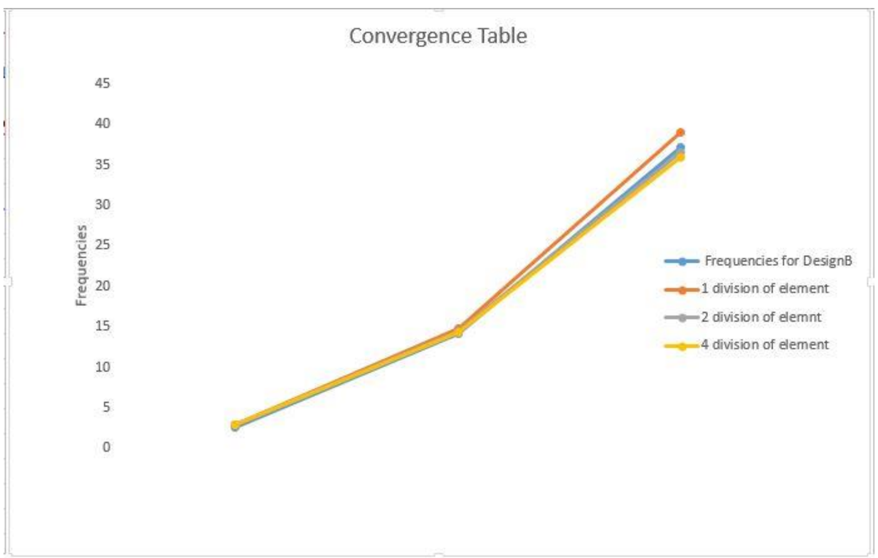
Results from Matlab		Results From Ansys		
Frequencies for DesignA	Frequencies for DesignB	1 division of element	2 division of elemnt	4 division of element
3.2077	2.6743	2.8543	2.8468	2.8449
20.1665	14.1149	14.818	14.435	14.338
56.9833	37.2506	38.974	36.628	36.016
128.3242	86.7633	74.893	67.325	65.319
241.5066	162.7202	113.73	99.086	94.903
481.4711	261.9825	147.57	124.82	118.77



From this Graph, it is clear that the results are converging by increasing the number of division of elements (Decreasing the size of Mesh)

Convergence Study:

	Results From Matlab	Results From Ansys		
	Frequencies for DesignB	1 division of element	2 division of elemnt	4 division of element
1	2.6743	2.8543	2.8468	2.8449
2	14.1149	14.818	14.435	14.338
3	37.2506	38.974	36.628	36.016



- As it can be seen from the Graph that Matlab results and Ansys Results for Design(b) are almost Same
- By Increasing Mesh size, results that are more accurate can be obtained.

Error Estimation for Design (b)

	Results From Matlab	Results From Ansys					
	Frequencies for DesignB	1 division of element	2 division of elemnt	4 division of element	percentage Errors for 1 Division Of Element	percentage Errors for 2 Division Of Element	percentage Errors for 4 Division Of Element
1	2.6743	2.8543	2.8468	2.8449	6.7	6.4	6.3
2	14.1149	14.818	14.435	14.338	4.98	2.23	1.58
3	37.2506	38.974	36.628	36.016	4.6	1.67	3.3

From this, we can say that error is decreasing by increasing number of divisions of element and is less than two percent for second frequency.

From all this results the frequencies with the four divisions of elements is considered for designing Gallery in a concert hall. So that the frequencies of the Gallery is given as

2.8449
14.338
36.016
65.319
94.903
118.77

For Vibrational analysis and design of the Cantilever beam, the above frequencies can be considered.