Elastic Analysis of Simply Supported Beam under Impact

Final Year Project (2019)

By

M. Hamza Khan (G.L) (NUST-2015-133906)

Mujtaba Usman (NUST-2015-129510)

Tauseef Maqsood (NUST-2015-127047)

Project Advisor: Dr. Azam Khan

NUST Institute of Civil Engineering (NICE)

School of Civil and Environmental Engineering (SCEE)

National University of Sciences and Technology (NUST)
Islamabad, Pakistan

(2019)

CERTIFICATION

This is to certify that thesis entitled

Elastic Analysis of Simply Supported Beam under Impact

By

M. Hamza Khan

Mujtaba Usman

Tauseef Maqsood

Has been accepted towards fulfillment of the requirements for

Bachelors in Civil Engineering

Dr. Azam Khan

Assistant Professor

NUST Institute of Civil Engineering (NICE)

School of Civil and Environmental Engineering (SCEE)

National University of Sciences and Technology (NUST)
Islamabad, Pakistan

DECLARATION

It is hereby solemnly and sincerely declared that the work referred to this thesis project has not been used by any other university or institute of learning as part of another qualification or degree. The research carried out and dissertation prepared was consistent with normal supervisory practice and all the external sources of information used have been acknowledged.

DEDICATION

This thesis is dedicated to all our beloved teachers, at home in the form of our parents and at university in the form of professors, lab engineers and other instructors. This work would not have been possible without their supervision, support and guidance throughout the term of this project.

ACKNOWLEDGMENT

We thank The Almighty Allah for giving us the strength and belief in ourselves for undertaking this final year project. We also take this opportunity to express our gratitude and respect to our parents, for it is only due to their prayers and wishes that enabled us to complete this work. It is utmost necessary to acknowledge and thank our advisor and mentor Dr. Azam Khan, for all his admirable guidance, assistance and motivation provided throughout our project. It is with his support that we have been successful in achieving our objectives. We would also like to appreciate the assistance and aid offered by our Head of Structures Department Dr. Rao Arsalan for his astounding support, which in fact became the crux in the progress of this project. We would also like to mention and thank all those departments and construction professionals who provided us with their time and assistance. Without their cooperation this project would not have seen the light of dawn.

ABSTRACT

This thesis presents numerical modelling of discretized simply supported beam under dynamic loading. The beam problem can be efficiently evaluated by devising an analytical problem and then by suitably selecting a computational algorithm for the numerical solution of dynamic problem. The simple beam is envisaged as an assembly of nodes with connected elements in which the mass is lumped at convenient nodes locations. Kinetic and kinematic structural laws are established under the restriction of small displacements in nodal forms. Impact loading, constituting high-intensity dynamic pressures and causing a global dynamic response, is considered.

Table of Contents

CHAPTER 1

INTRODUCTION

1.1 Practical motivation

Impact phenomena can be noticed in several aspects of life. Impact is an active and ever growing field of study it includes a wide variety of engineering problems such as military ballistic events, Impact resistance structural design against terrorist attack essential for the purpose of assessing the safety of nuclear, chemical and other industrial plants. Impact has a number of applications in the field of civil engineering, so design for impact is required. Highway structures, for example, may suffer damage due to vehicular impact, vibration of railways tracks due to cyclic impact loading causes damage to the railway track. Although, for numerous civil engineering structures, the risk of damage due to impact may be remote, but for sensitive structures impact must be considered during the design phase as the results of failure would be serious or possibly disastrous.

The ability to know the behavior of civil engineering structures under impact loads is needed for the purpose of assessing the safety. Significant attention has been given to civil engineering structures regarding their response under impact in recent years. Moreover, economic pressures result in lighter constructions, implying that the results of impact are likely to be crucial.

1.2 Historical background

The Second World War is viewed as a watershed in the development of structural impact. Before the end of WW2 in 1945, the scientific basis of the field unfolded by many intellectuals. The purpose of this section is to refer to various key contributions of the past upon which the principles of analytical mechanics were founded.

Even though the principles of statics had been correctly formulated by the scholars of ancient Greece, Aristotle's ideas (c. 350 BC), which distinguished between "natural motions" and "violent motions", were prevailed for a long time in the history of mechanics. Thus, a precise view of dynamics did not come into view until the end of the Middle Ages and the beginning of the modern era, when the field began to flourish.

Undoubtedly, most influential among the forefathers of dynamics is Newton. Proceeding of Galileo, Descartes and Huygens, Newton generalized the concept of force to include attraction of planets, actions of magnets and so forth; introduced the notion of mass; formulated the principle of the parallelogram of forces and the law of action and reaction, which have become inestimable vehicles for the writing down of equations of motion for dynamical systems. In his Principia (1686), he formalized the laws of motion, clearly establishing the vector nature and role of momentum as a measure for quantity of motion.

1.3 Thesis objective

Impact is a large field which encompasses a broad range of engineering problems including elastic dynamic behavior of structures, stress wave propagation, ballistics and crash impact test of vehicles. The impacted structural system may be devised for a variety of materials such as concrete, brittle and elastic metals, brick and composites.

In this thesis, attention has been focused upon the response of two dimensional simple beam to impulsive and short-term pulse loadings of intensity such that beam undergoes elastic deformation. This can be achieved by application of the laws of structural dynamics, mathematical modelling and laws of motion.

While the thesis above defines the general theme and strategy of the work, certain more specific objectives may be set down. They are:

1. To formulate analytical solution by employing nodal network concepts to set up the laws of kinetics (equations of motion) and of kinematics (geometry of motion) founded upon D'Alembert's principle in matrix form. To set up nodal network of lumped masses connected to one another by massless elements with stiffness lumped at nodes.
2. To develop a computer program on MATLAB that evaluates the dynamic response of an elastic beam under impact loading based on previously developed analytical equations and exact mathematical formulations for the numerical i.e. Runge Kutta method for solving n ordinary differential equations.
3. To model impact problem using commercial software (Abaqus) using the same perimeters i.e. impact mass, structural mass, initial conditions (initial velocity), boundary conditions (support conditions) as used for the MATLAB program.
4. To compare the results obtained from both MATLAB code and Abaqus model.

1.4 System modeling

The first stage involves formulation mathematical programming procedures for dynamic structural problems. Mathematical programming is a discrete variable form of optimization. It is based on principle that the model response can be effectively described by a finite number of variables. In this work, consideration is given exclusively to two dimensional elastic beam element but the same can be used for skeletal structures such as plane and spatial frames.

1.5 Structural modelling

The beam structure is modelled as a two dimensional planner beam with n number of nodes connected with mass less beam elements. This arrangement results in the nodal network system for beams. So nodes are positioned at:

1. At connections between two or more members
2. At the junction of a member with the support
3. At points where the geometrical or mechanical properties change
4. At the site of the structural masses
5. Where concentrated forces are applied

1.6 Mass modelling

The convenient and simple method for defining the mass properties of a structure is that of a lumped mass model, the actual distribution of mass is considered as a number of concentrated point masses connected by massless elements. For a lumped mass model to be adopted, the mass should be distributed as a finite number of concentrated point masses, however their magnitude and positioning requires sound judgment and experience. For the lumped mass model of the elastic beam it is assumed that the centroid of each concentrated mass is located at a node which is associated with a number of kinematically possible displacements, or degrees of freedom, two translations and one rotation. No consideration is given to the effects of damping because of its insignificance in regards to the class of problem investigated in the thesis.

1.7 Load modelling

Generally for impact problems structural systems subjected to short-term, high intensity dynamic pressures, usually known as pulse loads. But In the problem impacts is modelled by giving impact mass and initial velocity to a convenient node for the sake of simplicity.

CHAPTER 2

Literature review

2.1 Introduction

Theory of Structures deals with the behavior of structural elements like beams, columns, frames, and plate and shell elements subjected to loads or other forces which induce stress and deform the shape of structural elements. The analysis of structural elements involves the principles described by the Theory of Structure, under specified loading conditions and other factors like pressure variations temperature, or support movements. The bending moment diagram for the description of stress state of a beam based on structural analysis which involves the understanding of structural theory. The knowledge of Theory of Structure is necessary to relate the applied loads, reaction forces to the bending moment of the beam. The internal stress distribution in a member can be conveniently described in terms of 'stress resultants'. In a three dimensional analysis of a structure, there are two shear forces, two bending moments, a twisting moment and a thrust. Another aspect is Structural deformation it is not described throughout the continuous length of structural element, however the practice is to consider the value at selected points along the length of element. In structural analysis it is usually convenient to describe the state of stress or deformation in terms of forces and displacements at specific points known as 'nodes'. These are generally the critical points in the structure where there is maximum stress of deflection i.e. ends of members, or the joints. The information about the forces or displacements at the nodes of a structural element is enough to describe the state of stress within the element providing the relationships between forces and displacements are established. The Force-displacement concepts state that the structure must be in equilibrium as a whole and every part of it under the action of forces. For example, if a beam is isolated from a structure, it should be in equilibrium under the action of internal stress resultants and external forces. There are six conditions of equilibrium i.e. the sum of forces in three perpendicular axis must be zero and the summation of moments about three perpendicular axes must be zero. Another important principle is Compatibility principle, it states that the parts of a structural element must always deform in a conformable way. The parts of a structural element must set together in continuity. At certain points in a structure, the continuity of a member, can be interrupted by a 'release'. This release imposes a zero value on one of the stress resultants. A hinge is an example of a release. Releases are introduced as imaginary devices in a structure under analysis. The release will allow a discontinuity to develop, including a release will reduce the number of compatibility equations by one.

2.2 Force – Displacement relationship

In structural analysis the state of stress or deformation in terms of forces and displacements at 'nodes' and the structure should satisfy equilibrium and compatibility.

2.3 Static determinacy

When the structure nodal forces can be calculated directly from the equilibrium conditions the structure is statically determinate.

2.4 Kinematic determinacy

When the structure deflections can be calculated directly from the compatibility conditions the structure is kinematically determinate.

2.5 Kinematic indeterminacy

The degree of kinematic indeterminacy is the minimum number of movements (degree of freedom DOF) with which the kinematic configuration of the overall structure can be defined. For example for a planner beam there are 3 movements for each 'node'

For studying nearest possible behavior of a structure there should be multiple nodes (which means multiple degree of freedoms) and more complex system.

2.6 Multi degree-of-freedom system

When external forces act on a multiple -DOF -system, the system undergoes forced vibration. For a system with 'n' degrees of freedom, the governing equations of motion are a set of 'n' coupled ordinary differential equations of second order

In such cases, a more modal analysis can be used to solve the problem analysis

$$[m]\,\ddot{x} + [k]x \geq F \quad \text{(undamped system)}$$

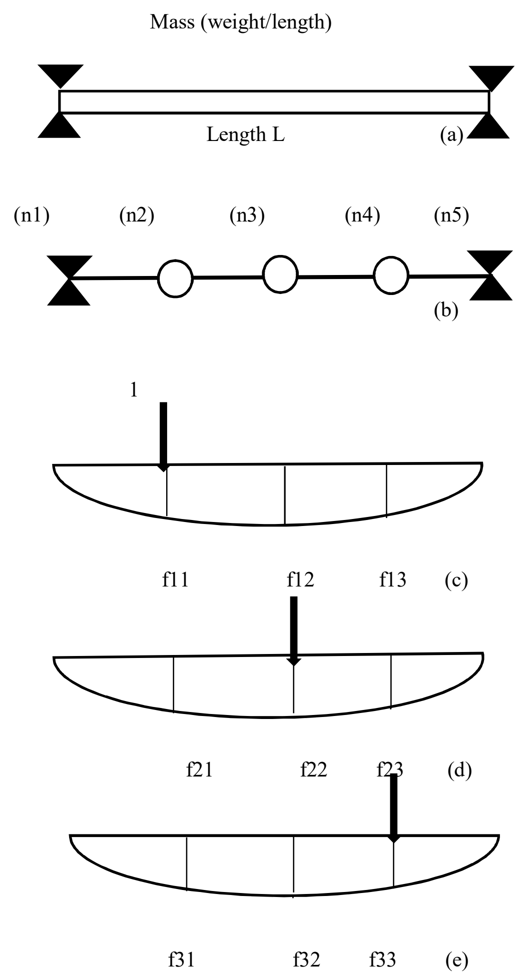
The convenient and simple method for defining the mass properties of a structure is that of a lumped mass model, the actual distribution of mass is considered as a number of concentrated point masses connected by massless elements. For a lumped mass model to be adopted, the mass should be distributed as a finite number of concentrated point masses, however their magnitude and positioning requires sound judgment and experience.

CHAPTER 3

METHODOLOGY

3.1 ANALYTICAL SOLUTION

3.1.1 Principle



To find out analytically the dynamic

Responsof a beam subjected to

Impact, one has to devise a

Analytical Solution. For this purpose,

We devised the a solution based

On the theory of free vibrations,

Laws of structural dynamics,

Flexibility influence coefficients

(For finding out flexibility and stiffness matrix)

And Runge Kutta method for solving

Ordinary Differential Equations of any order.

3.1.2 Analytical model introduction

The analytical solution follows a node – element model characterized by,

- A **node-element model** in which masses are lumped at nodes having certain values of stiffness calculated by flexibility influence co-efficient method.
- A description of structural network is obtained by series of massless elements connected with nodes, each node having specific mass (lumped mass system).
- Deformations (displacements and velocities) at nodes are obtained from stiffness or flexibility analysis.
- The accuracy of results is dependent on the number of nodes in model and their spacing.

3.1.3 Inputs

The analytical solution of beam problem requires following inputs:

- Length of beam
- Modulus of Elasticity
- Moment of inertia
- Initial displacement in meters
- Initial velocity
- Impactor mass

Units: Analytical solution requires the user to put all the data in consistent units.

3.1.4 Mass distribution

Consider figure (a) and (b), the beam has length "L" with "M" as the mass of the beam and mass per length unit of the beam is w/L. The overall mass of the beam is lumped at nodes. Each node carries a mass of wL/3 (total mass/ number of nodes). As the mass at supports does not take part in motion, the nodes at supports are not assigned any massso their mass is neglected in analysis.

m1= m2 = m3 = WL/3

$$\begin{bmatrix} m1 & 0 & 0 \\ 0 & m2 & 0 \\ 0 & 0 & m3 \end{bmatrix}$$

The impactor mass is added to the central node where "Mi" is the Impactor Mass, so the final mass matrix is

$$\begin{bmatrix} m1 & 0 & 0 \\ 0 & m2 + Mi & 0 \\ 0 & 0 & m3 \end{bmatrix}$$

3.1.5 Flexibility matrix

Flexibility Method (Flexibility Influence coefficients):

- The flexibility method provides a means of analysing statically indeterminate structures
- Flexibility coefficients are displacements calculated at specified positions, and directions, in a structure due to a single unit load replacing a redundant force in the structure.

Evaluation of flexibility coefficients:

- Referring to figures (c), (d), (e), flexibility influence co - efficient (displacements) by the application of Castigliano's theorem or use the principle of virtual work. In either case a convenient form is

$$f_i = \int M \partial M / \partial F_i \; ds/EI$$

Where

- f_i = displacement required
- M = function representing the bending moment distribution
- Fi is a force, real or virtual, applied at the position and in the direction designated by i.
- The flexibility coefficients f_{ij} provides the displacements at selected points in the structure due to unit values of the associated, redundant, forces.
- The evaluation of flexibility coefficients requires the integration of the product of two bending moment distributions over the complete structure (table of The product integrals can also be used see appendix A)

$$F = \begin{bmatrix} f11 & f12 & f13 \\ f21 & f22 & f23 \\ f31 & f32 & f33 \end{bmatrix}$$

3.1.6 Stiffness matrix

Stiffness matrix is obtained simply by taking inverse of the flexibility matrix.

$$K = F^{-1}$$

$$K = 1/\begin{bmatrix} f11 & f12 & f13 \\ f21 & f22 & f23 \\ f31 & f32 & f33 \end{bmatrix}$$

3.1.7 Free vibration response of multi degree of freedom system

A beam represented as a lumped mass system, as shown in figure (b) the governing system of equations emerges as a set of Ordinary differential Equations. The order of Ordinary Differentials Equations depends upon the degree of freedom.

$$[M]\, \ddot{x}\,(t) \;+\; [K]\, x\,(t) \;=\; F \qquad \text{(undamped system)}$$

For the beam problem refer to figure (a),

For free vibration response the governing equations are given by

F = 0

$$\begin{bmatrix} m1 & 0 & 0 \\ 0 & m2 + Mi & 0 \\ 0 & 0 & m3 \end{bmatrix} \begin{bmatrix} \ddot{x}1\,(t) \\ \ddot{x}2\,(t) \\ \ddot{x}3\,(t) \end{bmatrix} + \begin{bmatrix} k11 & k12 & k13 \\ k21 & k22 & k23 \\ k31 & k32 & k33 \end{bmatrix} \begin{bmatrix} x1\,(t) \\ x2\,(t) \\ x3\,(t) \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \end{bmatrix}$$

$m1\, \ddot{x}1\,(t) \quad + k11\, x1(t) \quad + k12\, x2\,(t) \quad + k13\, x3\,(t) \quad = 0$

$m2{+}Mi\, \ddot{x}2\,(t) \quad + k21\, x1(t) \quad + k22\, x2\,(t) \quad + k23\, x3\,(t) \quad = 0$

$m3\, \ddot{x}3\,(t) \quad + k31\, x1(t) \quad + k32\, x2\,(t) \quad + k33\, x3\,(t) \quad = 0$

RUNGE KUTTA METHOD FOR SOLVING SYSTEM OF ODE s:

The coupled Ordinary Differential Equations of second order, as written above the O D E has six variables (3 displacements and 3 accelerations). Since RK method only works for first order differential equations it is convenient to convert the second order differential equation in first order differential equations.

The system of equations above can be expressed as a set of six coupled first-order differential equations and then solved using RK method.

$y1 = x1 \qquad y2 = \dot{x}1 \qquad \dot{y}2 = \ddot{x}1$

$y3 = x2 \qquad y4 = \dot{x}2 \qquad \dot{y}4 = \ddot{x}2$

$y5 = x3 \qquad y6 = \dot{x}3 \qquad \dot{y}6 = \ddot{x}3$

$m1\ \dot{y}2\ (t) + k11\ y1\ (t) \quad + k12\ y3\ (t) \quad + k13\ y5\ (t) \quad = 0$

$m2{+}Mi\ \dot{y}4\ (t) + k21\ y1\ (t) \quad + k22\ y3\ (t) \quad + k23\ y5\ (t) \quad = 0$

$m3\ \dot{y}6\ (t) + k31\ y1\ (t) \quad + k32\ y3\ (t) \quad + k33\ y5\ (t) \quad = 0$

The system of first order ordinary differential equations written above can be solved for the initial condition

Initial displacement y (0) = 0

Initial velocity = ẏ (0) = 0.1 m/s (impact velocity at central node 3)

Time span = 10 s

Time step = 1s

$\dot{y}1\ (t) \quad = y2\ (t)$

$$\dot{y}2\ (t) \quad = -\frac{k11*y1\ (t)}{m1} - \frac{k1\ \ *y3\ (t)}{m1} - \frac{k13*y5\ (t)}{m1}$$

$\dot{y}3\ (t) \quad = y4\ (t)$

$$\dot{y}4 = -\frac{k21*y1\ (t)}{m2+Mi} - \frac{k22*y3\ (t)}{m2+Mi} - \frac{k23*y5\ (t)}{m2+Mi}$$

$\dot{y}5\ (t) \quad = y6\ (t)$

$$\dot{y}6\ (t) \quad = -\frac{k31*y1\ (t)}{m3} - \frac{k32*y3\ (t)}{m3} - \frac{k33*y5\ (t)}{m3}$$

The solution by RK method yields a resultant Row matrix with one row and six columns for each time step. The 1st and 2nd columns are the displacement and velocity at node 2, the 3rd and 4th columns are the displacement and velocity at node 3 (central node), the 5th and 6th columns are the displacement and velocity at node 4 respectively for a single time step. The number of results in the form of Row matrix depends upon the time span and time step i.e. for a time span of 0 to 10s the result would be 10 Row matrix,1 Row matrix for each 1s time step.

u2 v2 u3 v3 u4 v4

3.1.8 Eigen values and Eigen vector

For determining the mode of vibration and mode shapes of mass spring system Eigen Value Theorem can be conveniently applied.

$$[M]\,\ddot{x}\,(t) + [K]\,x\,(t) = 0$$

By assuming harmonic solution for the free vibration response

$X(t) = \Phi \sin(\omega t + \theta)$

Where, "θ" is phase angle and "ω" is the frequency of free vibration.

$\ddot{X}(t) = -\omega^2 \Phi \sin(\omega t + \theta)$

Substituting the solution into governing equation of motion

$-[M]\,\omega^2\,\Phi \sin(\omega t + \theta) + [k]\,\Phi \sin(\omega t + \theta)$

$[K]\,\Phi = [M]\,\omega^2\,\Phi$

$[K - M\,\omega^2]\;\Phi = 0$

By Crammers rule for a homogenous system of linear equations

MV = 0 (M is a given Matrix and V is an un-known vector) has a non-zero solution if and only if Det [M] = 0

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

The roots (solutions) of the above equation gives the frequencies i.e. $[\omega_1^2\ \omega_2^2\ \omega_3^2]$ of the possible mode shapes in the system.

3.1.9 Example:

Consider the beam as shown in figure (a) has the length of 4m, width x depth (50mm x 50 mm), the modulus of elasticity of steel beam is 210 GPa and density is 7850 kg/m^3. An impactor with a mass of 150 collides at the mid span of beam such that the initial velocity is 0.1 m/s.

MASS MATRIX:

As density = mass/volume

Mass = density x volume

Mass = 7850 x (4 x 0.05 x 0.05) = 78.5 kg

If we divide the beam into 4 elements, element length is equal to 1m.

Mass per unit length = 78.5/4

Mass per unit length = 19.625 kg

Mass at each node (wL/3) = (19.625 x 4)/3

Mass at each node = 26.167 kg

$$M = \begin{bmatrix} 26.167 & 0 & 0 \\ 0 & 26.167 & 0 \\ 0 & 0 & 26.167 \end{bmatrix}$$

Mass matrix considering impact

$$M = \begin{bmatrix} 26.167 & 0 & 0 \\ 0 & 26.167 + 150 & 0 \\ 0 & 0 & 26.167 \end{bmatrix}$$

FLEXIBILITY MATRIX:

The flexibility matrix is given by

$$F = \begin{bmatrix} f11 & f12 & f13 \\ f21 & f22 & f23 \\ f31 & f32 & f33 \end{bmatrix}$$

By the application of Castigliano's theorem (using table of product integrals see appendix A)

$f_i = \int M \partial M/\partial F_i \, ds/EI$

$$F = \begin{bmatrix} f11 & f12 & f13 \\ f21 & f22 & f23 \\ f31 & f32 & f33 \end{bmatrix}$$

$$= \begin{bmatrix} 6.86e-06 & 8.38e-06 & 5.33e-06 \\ 8.38e-06 & 1.22e-05 & 8.38e-06 \\ 5.33e-06 & 8.38e-06 & 6.86e-06 \end{bmatrix}$$

STIFFNESS MATRIX:

$K = F^{-1}$

$$K = 1/ \begin{bmatrix} f11 & f12 & f13 \\ f21 & f22 & f23 \\ f31 & f32 & f33 \end{bmatrix}$$

$$K = \begin{bmatrix} 1078056 & -1031184 & 421848 \\ -1031184 & 1499904 & -1031184 \\ 421848 & -1031184 & 1078056 \end{bmatrix}$$

FREE VIBRATION RESPONSE:

System of second order O D E s

$$\begin{bmatrix} m1 & 0 & 0 \\ 0 & m2+Mi & 0 \\ 0 & 0 & m3 \end{bmatrix} \begin{bmatrix} \ddot{x}1\ (t) \\ \ddot{x}2\ (t) \\ \ddot{x}3\ (t) \end{bmatrix} + \begin{bmatrix} k11 & k12 & k13 \\ k21 & k22 & k23 \\ k31 & k32 & k33 \end{bmatrix} \begin{bmatrix} x1\ (t) \\ x2\ (t) \\ x3\ (t) \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \end{bmatrix}$$

RUNGE KUTTA METHOD FOR SOLVING SYSTEM OF ODE s:

The system of equations above can be expressed as a set of six coupled first-order differential equations:

$$y1 = x1 \qquad y2 = \dot{x}1 \qquad \dot{y}2 = \ddot{x}1$$
$$y3 = x2 \qquad y4 = \dot{x}2 \qquad \dot{y}4 = \ddot{x}2$$
$$y5 = x3 \qquad y6 = \dot{x}3 \qquad \dot{y}6 = \ddot{x}3$$

$m1\ \dot{y}2\ (t) + k11\ y1\ (t) \quad + k12\ y3\ (t) \quad + k13\ y5\ (t) \quad = 0$

$m2+Mi\ \dot{y}4\ (t) + k21\ y1\ (t) \quad + k22\ y3\ (t) \quad + k23\ y5\ (t) \quad = 0$

$m3\ \dot{y}6\ (t) + k31\ y1\ (t) \quad + k32\ y3\ (t) \quad + k33\ y5\ (t) \quad = 0$

Solving boundary value problem

Initial displacement y (0) = 0

Initial velocity = ẏ (0) = 0.1 m/s (impact velocity at central node 3)

Time span = 10 s

Time step = 1s

$\dot{y}1\ (t) \quad = y2\ (t)$

$$\dot{y}2\ (t) \quad = -\frac{1078056 * y1\ (t)}{26.167} - \frac{-1031184 * y3\ (t)}{26.167} - \frac{421848 * y5\ (t)}{26.167}$$

$\dot{y}3\ (t) \quad = y4\ (t)$

$$\dot{y}4 = -\frac{-103118\ \ * y1\ (t)}{26.167+150} - \frac{1499904 * y3\ (t)}{26.167+1} - \frac{-1031184 * y5\ (t)}{26.167+1}$$

$\dot{y}5\ (t) \quad = y6\ (t)$

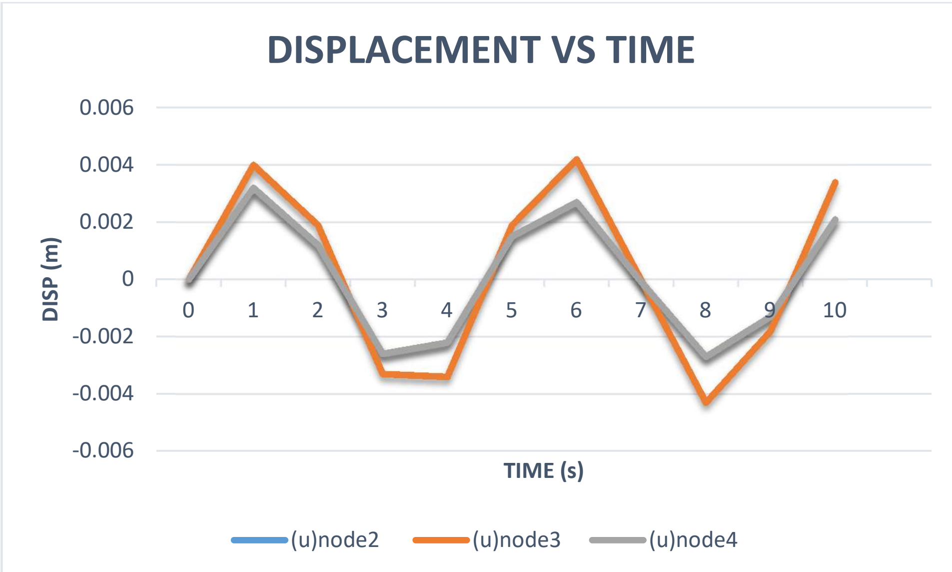
$$\dot{y}6\ (t) \quad = -\frac{421848 * y1\ (t)}{26.167} - \frac{-1031184 * y3\ (t)}{26.167} - \frac{1078056 * y5\ (t)}{26.167}$$

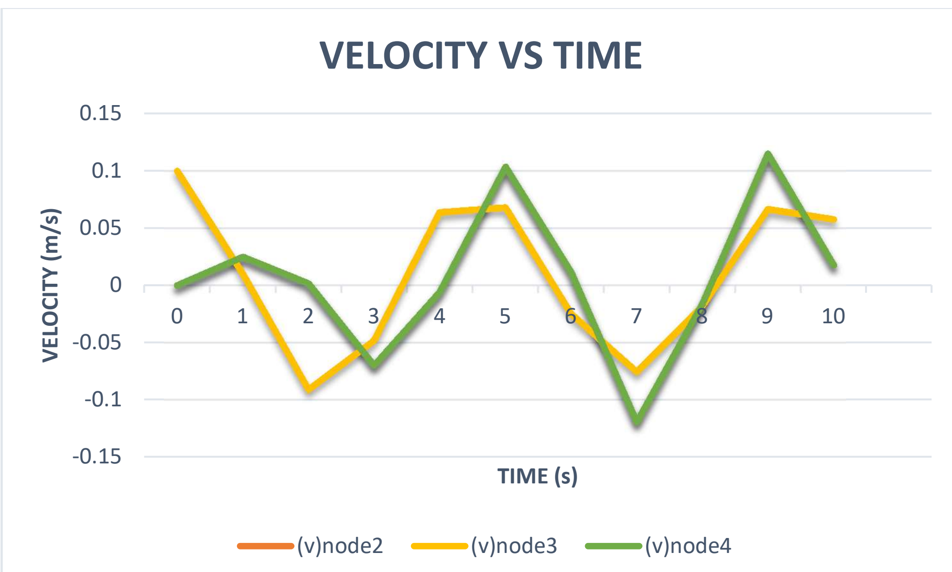
Solution:

T sec	U2 (n2) m	V2 (n2) m/s	U3 (n3)m	V3 (n3) m/s	U4 (n4) m	V4 (n4) m/s
0	0	0	0	0.1	0	0
1	0.0032	0.0250	0.004	0.01	0.0032	0.0250
2	0.0012	0.0017	0.0019	-0.0910	0.0012	0.0017
3	-0.0026	-0.0700	-0.0033	-0.0480	-0.0026	-0.0700
4	-0.0022	-0.0059	-0.0034	0.0640	-0.0022	-0.0059
5	0.0015	0.1037	0.0019	0.0680	0.0015	0.1030
6	0.0027	0.0116	0.0042	-0.025	0.0027	0.0117
7	-8.4e-05	-0.1195	-2.5e-05	-0.0754	-8.4e-05	-0.1195
8	-0.0027	-0.0163	-0.0043	-0.0179	-0.0027	-0.0164
9	-0.0013	0.1151	-0.0018	0.0668	-0.0013	0.1151
10	0.0021	0.01779	0.0034	0.0577	0.0021	0.01779

The table shows the velocities and displacement at each node (2, 3 and 4) during each time step of 1 second.

DISPLACEMENT AND VELOCITY PLOTS:





EIGEN VALUES AND EIGEN VECTOR:

By Eigen Value Theorem,

$[M]\ \ddot{x}(t) + [K]\ x(t) = 0$

$$M = \begin{bmatrix} 26.167 & 0 & 0 \\ 0 & 26.167 + 150 & 0 \\ 0 & 0 & 26.167 \end{bmatrix}$$

$$K = \begin{bmatrix} 1078056 & -1031184 & 421848 \\ -1031184 & 1499904 & -1031184 \\ 421848 & -1031184 & 1078056 \end{bmatrix}$$

$$[K- M\ \omega 2] = \begin{bmatrix} 1078056 & -1031184 & 421848 \\ -1031184 & 1499904 & -1031184 \\ 421848 & -1031184 & 1078056 \end{bmatrix} - \omega^2 \begin{bmatrix} 26.167 & 0 & 0 \\ 0 & 176.167 & 0 \\ 0 & 0 & 26.167 \end{bmatrix}$$

$$[K- M\ \omega^2] = 421848 \begin{bmatrix} 2.56 & -2.44 & 1 \\ -2.44 & 3.56 & -2.44 \\ 1 & -2.44 & 2.56 \end{bmatrix} - \omega^2 \begin{bmatrix} 26.167 & 0 & 0 \\ 0 & 176.167 & 0 \\ 0 & 0 & 26.167 \end{bmatrix}$$

$$= \begin{bmatrix} 2.56 - (\frac{\omega 2}{421848})26.167 & -2.44 & 1 \\ -2.44 & 3.56 - \left(\frac{\omega 2}{421848}\right)176.167 & -2.44 \\ 1 & -2.44 & 2.56 - (\frac{\omega 2}{421848})26.167 \end{bmatrix}$$

Now, Det $[K- M\ \omega 2] = 0$

The determinant will result in a polynomial equation and the roots (solutions) of the equation are the frequencies of modes of vibration.

$\omega_1 = 20.1972$ rad/s

$\omega_2 = 158.3594$ rad/s

$\omega_3 = 255.7863$ rad/s

For mode shapes

$[K - M\,\omega^2]\ \ \Phi = 0$

$$\begin{bmatrix} 2.56-(\frac{\omega 2}{421848})26.167 & -2.44 & 1 \\ -2.44 & 3.56-\left(\frac{\omega 2}{421848}\right)176.167 & -2.44 \\ 1 & -2.44 & 2.56-(\frac{\omega 2}{421848})26.167 \end{bmatrix} \Phi = 0$$

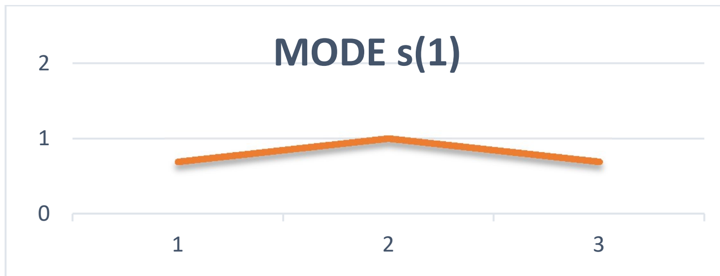
Where $\Phi 1=\begin{bmatrix}\Phi\ 11\\ \Phi\ 12\\ \Phi\ 13\end{bmatrix}$; $\Phi 2=\begin{bmatrix}\Phi\ 21\\ \Phi\ 22\\ \Phi\ 23\end{bmatrix}$; $\Phi 3=\begin{bmatrix}\Phi\ 31\\ \Phi\ 32\\ \Phi\ 33\end{bmatrix}$

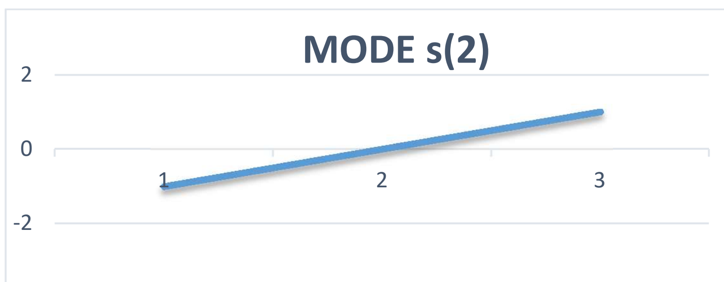
Mode shape 1 (Φ 1) = $\begin{bmatrix}0.4947\\ 0.7145\\ 0.4947\end{bmatrix}$; Normalized Mode shape 1 $\begin{bmatrix}0.6923\\ 1\\ 0.6923\end{bmatrix}$

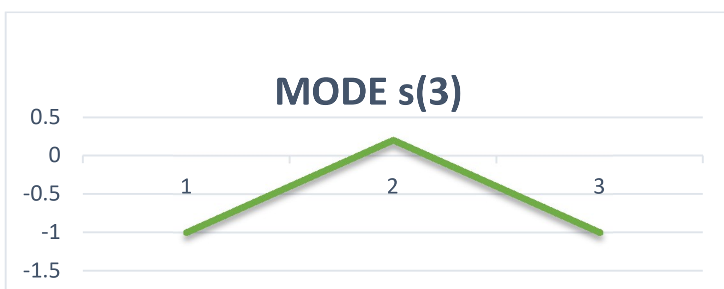
Mode shape 2 (Φ 2) = $\begin{bmatrix}-0.7071\\ 0.0000\\ 0.7071\end{bmatrix}$; Normalized Mode shape 2 $\begin{bmatrix}-1\\ 0\\ 1\end{bmatrix}$

Mode shape 3 (Φ 3) = $\begin{bmatrix}-0.6997\\ 0.1439\\ -0.6997\end{bmatrix}$; Normalized Mode shape 3 $\begin{bmatrix}-1\\ 0.2056\\ -1\end{bmatrix}$

MODES SHAPES PLOTS







3.2 MATLAB MODEL

3.2.1 MATLAB Introduction:

MATLAB is a numerical computing software with built in programming language. MATLAB allows implementation of algorithms, matrix manipulations, creation of user interphase, plotting of functions and interacting with programs written in other programming languages.

BENEFITS:

- Built in algorithms
- Easy debug
- Use external libraries
- Extensive visualization and date analysis
- Easy and efficient computation
- Simple and easy to understand programming language

3.2.2 MATLAB model introduction

The elastic beam model (code) is based to the previously developed Analytical solution which means:

- A **node-element model** in which masses are lumped at nodes having certain values of stiffness calculated by flexibility influence co-efficient method.
- A description of structural network is obtained by series of massless elements connected with nodes, each node having specific mass (lumped mass system).
- Deformations (displacements and velocities) at nodes are obtained from stiffness or flexibility analysis.
- The accuracy of results is dependent on the number of nodes in model and their spacing.

3.2.3 Inputs

The following inputs are to be given when you run the program.

- Mass per unit length
- Length of beam in meters
- Number of elements
- Modulus of Elasticity
- Moment of inertia
- Initial displacement
- Initial velocity
- Impactor mass

Editor - F:\Final Code\DynamicAnalysis.m

DynamicAnalysis.m | DalembertEq.m

```

%mass = input('Enter value of m kg/m:>')
mass = 26.167
%Len = input('Enter Length of beam:>')
Len = 4
%Elem = input('Enter no. of elements:>')
Elem = 4
%E = input('Enter modulus of Elasticity N/m^2:>')
E = 2.1*10^11
%I = input('Moment of intertia m^4:>')
I = 5.208*10^-7
%uo= input('Initial displacement meters:>')
uo = 0
%uodot= input('Initial velocity m/s:>')
uodot = 0.1
%Impactor= input('Impactor mass kg:>')
Impactor = 150
```

3.2.4 Units

MATLAB code requires the user to put all the data in consistent units. In our MATLAB model we have used SI units (N, m, and kg).

3.2.5 Mass matrix

Diagonal matrix having values of lumped mass at each node and lumped mass + impact mass at central node.

Editor - F:\Final Code\DynamicAnalysis.m

DynamicAnalysis.m | DalembertEq.m

```
    %Mass on Standard node (except middle one)

    m = mass*Len/Elem;

    % 2-Generate mass matrix

    for i = 1:1:Elem-1
        for j = 1:1:Elem-1
            if (i==j)&(i~=Elem/2)
               Massmat(i,j) = m;
            elseif (i==j)&(i==Elem/2)
               Massmat(i,j) = m+Impactor;
            end

        end
    end

```

3.2.6 Flexibility matrix

The flexibility matrix is obtained by the application of the principal of virtual work (table of product integrals)

$f_i = \int M \partial M / \partial F_i \, ds/EI$

Editor - F:\Final Code\DynamicAnalysis.m

DynamicAnalysis.m | DalembertEq.m

```
    % 3-Generate Stiffness matrix

    for i = 1:1:Elem-1
        for j = 1:1:Elem-1
            if i==j
                a = i*(Len/Elem);
                b = Len - a;
                c = j*(Len/Elem);
                d = Len - c;
                M1 = 1*(b/Len)/(E*I)*a;
                M2 = 1*(c/Len)*d;
               Flexmat(i,j) = (1/3- (a-c)^2/(6*a*d))*(M1*M2*Len);
            elseif i>j
                a = i*(Len/Elem);
                b = Len - a;
                c = j*(Len/Elem);
                d = Len - c;
                M1 = 1*b/E/I/Len*a;
                M2 = 1*c/Len*d;
               Flexmat(i,j) = (1/3- (a-c)^2/(6*a*d))*(M1*M2*Len);
               Flexmat(j,i) = Flexmat(i,j);
            end

        end
    end

```

3.2.7 Eigen values And Eigen vectors

Stiffness matrix is obtained by directly inversing the flexibility matrix.

Eigen values and vectors are calculated by using a built-in Function “[V, D] = eig ( )”.

The V vector is a matrix containing Eigen Vectors.

The Eigen vectors are then normalized and plotted.

The D vector is a diagonal vector with diagonal elements as the squares of frequencies of modes of vibrations.

The modal frequencies are obtained by taking square root of the diagonal elements of vector “D”.

```
% Write Stiffness and flexibility matrices

Flexmat;
Stiffmat = inv(Flexmat);

% 4-Find Eigen values and eigen vectors

Dynmat = Flexmat*.Massmat;

[V,D] = eig(Dynmat);

% "V" is a matrix containing Eigen Vectors and "D" containing
%eigenvalues/natural frequencies

% separating eigen vectors
```

3.2.8 Solving system of O D E s for displacements:

Applying initial conditions

(Initial displacement = 0, initial velocity = 1)

Solving ODE to obtain displacement and velocity profiles for the specified timespan.

[t, y] = ode45 (odefun, tspan, y0);

Where tspan = [t0 tf],

Integrates the system of differential equations y′=f (t, y) from t0 to tf with initial conditions y0.

Each row in the solution array y corresponds to a value returned in column vector t.

```
% This subroutine is used to write
% system of first order differential equation
function f = DalembertEq(t,y)
global Elem Stiffmat Massmat
f = zeros(2*(Elem-1),1);.
j = 2;
k = 1;
m = 1;
n = 1;

    for i = 1:1:2*(Elem-1)
        f(i)=0;
        if (rem(i,2)~=0)
            f(i) = y(j);
            j = j +2;
        elseif (rem(i,2)==0)
            for l = 1:1:(Elem-1)
```

```
% Generating y matrix (dynamic) containing displacements and velocities

[t, y] = ode45 ('DalembertEq', tspan, yo);

% 6-Finding Shear Forces at Nodes and support Reactions
```

3.2.9 Results (graph plots)

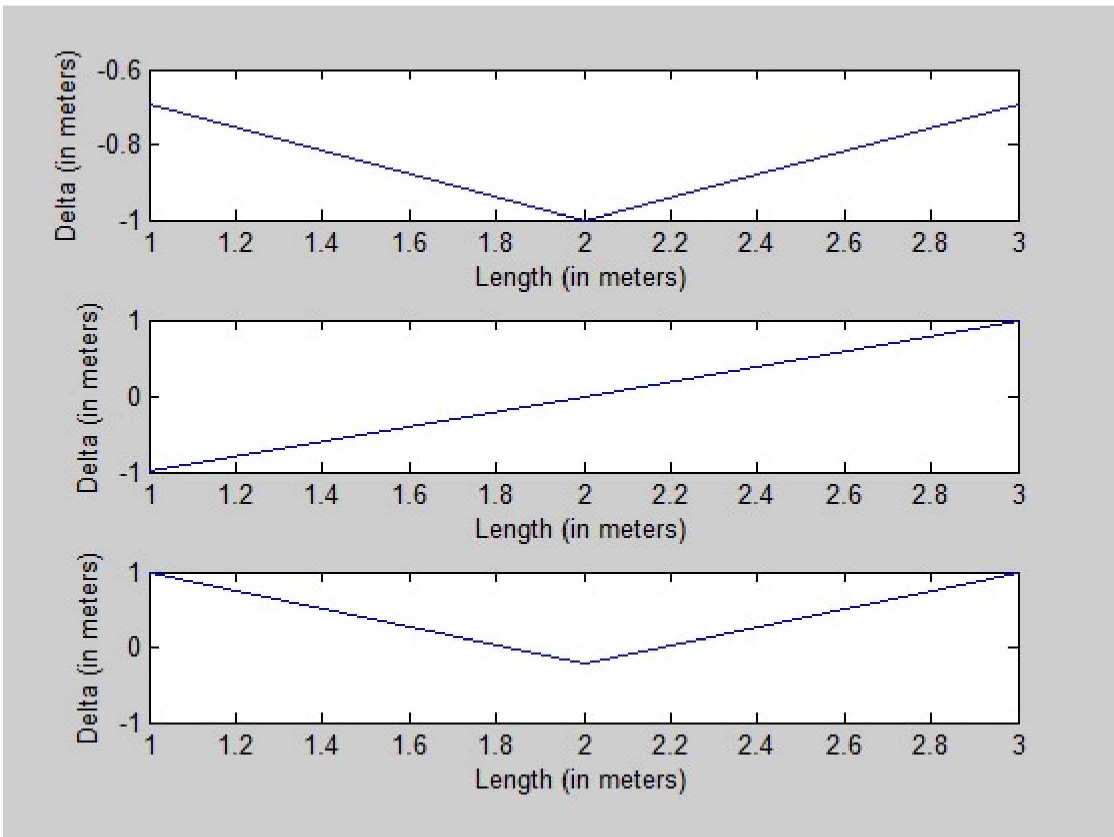


Figure 5.1 shows mode shapes

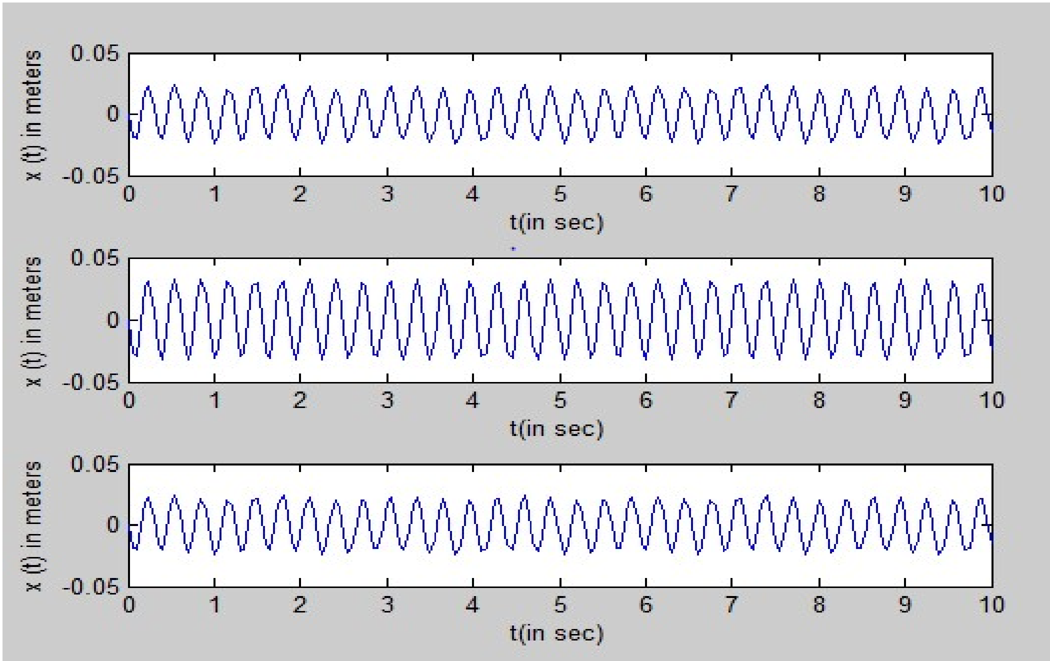


Figure 5.2 shows Displacements at 1, 2 and 3 Nodes

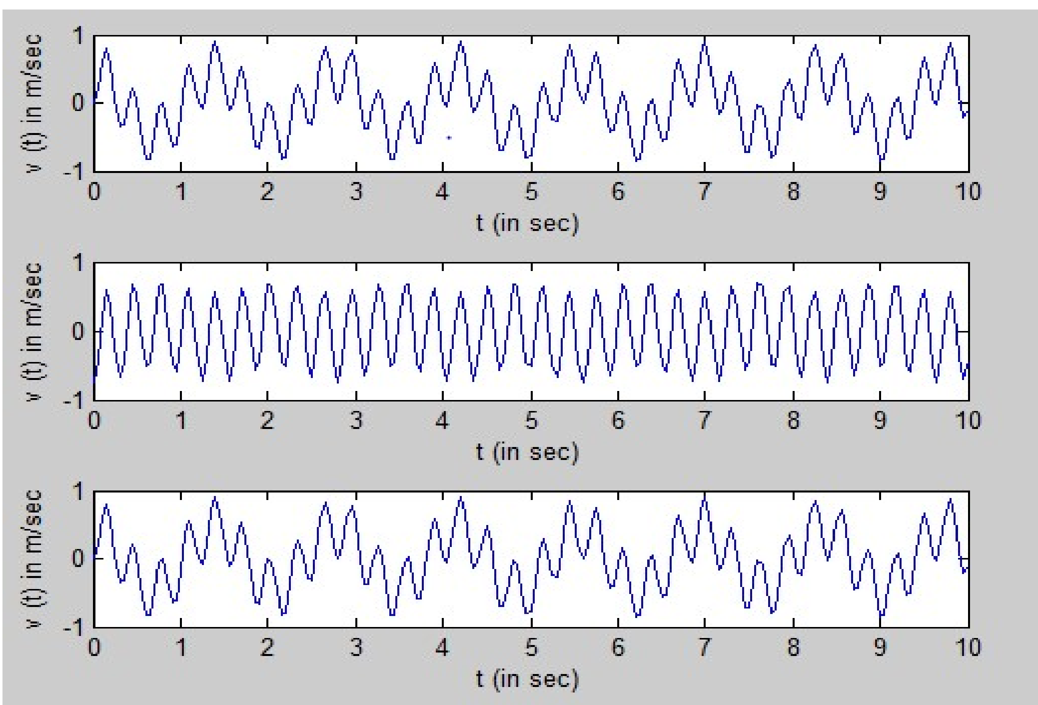


Figure 5.3 shows Velocities at Nodes 1, 2 and 3

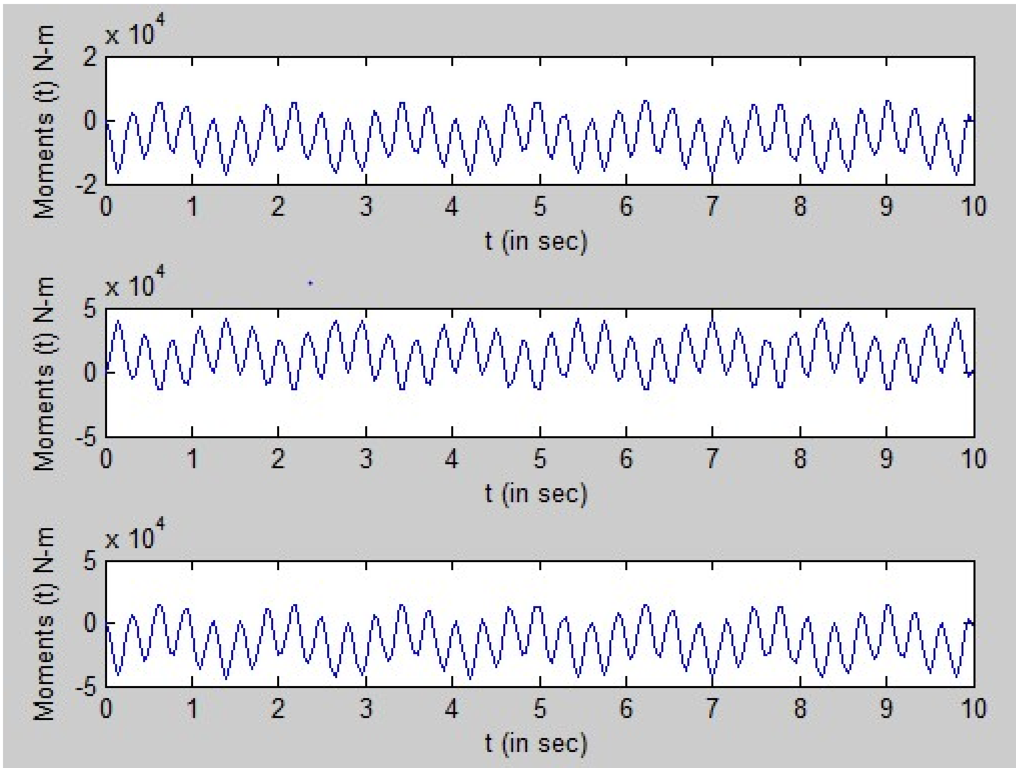


Figure 5.4 shows Moments at Nodes 1, 2 and 3

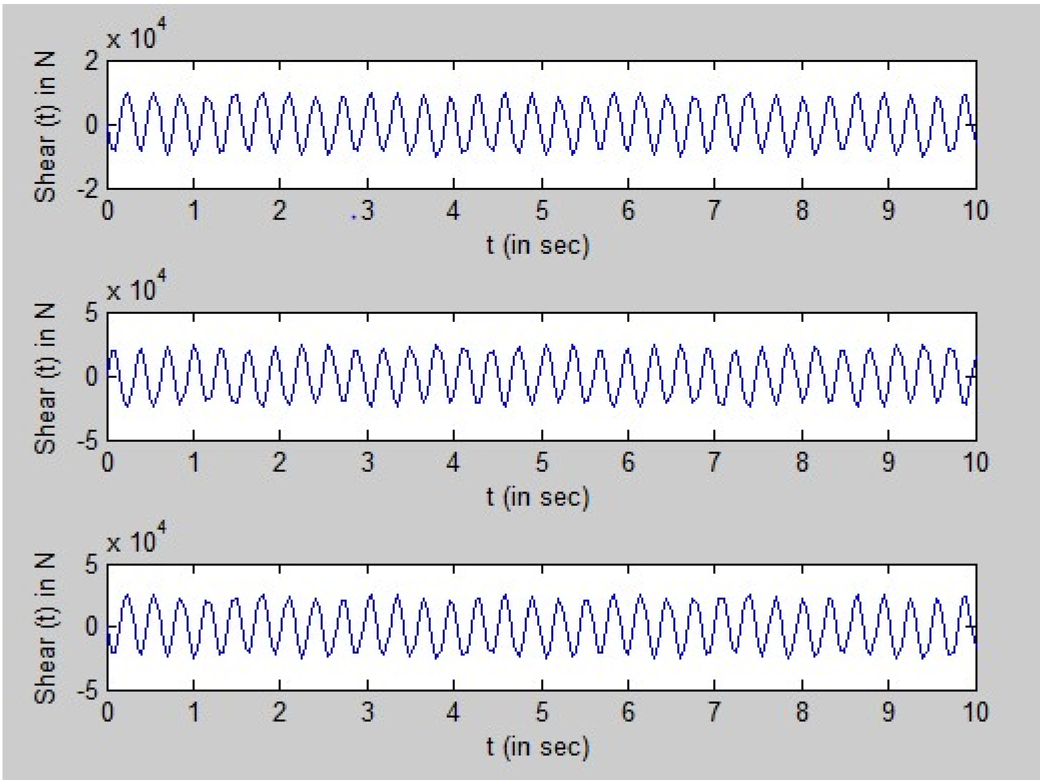


Figure 5.5 shows Shear Forces at Nodes 1, 2 and 3

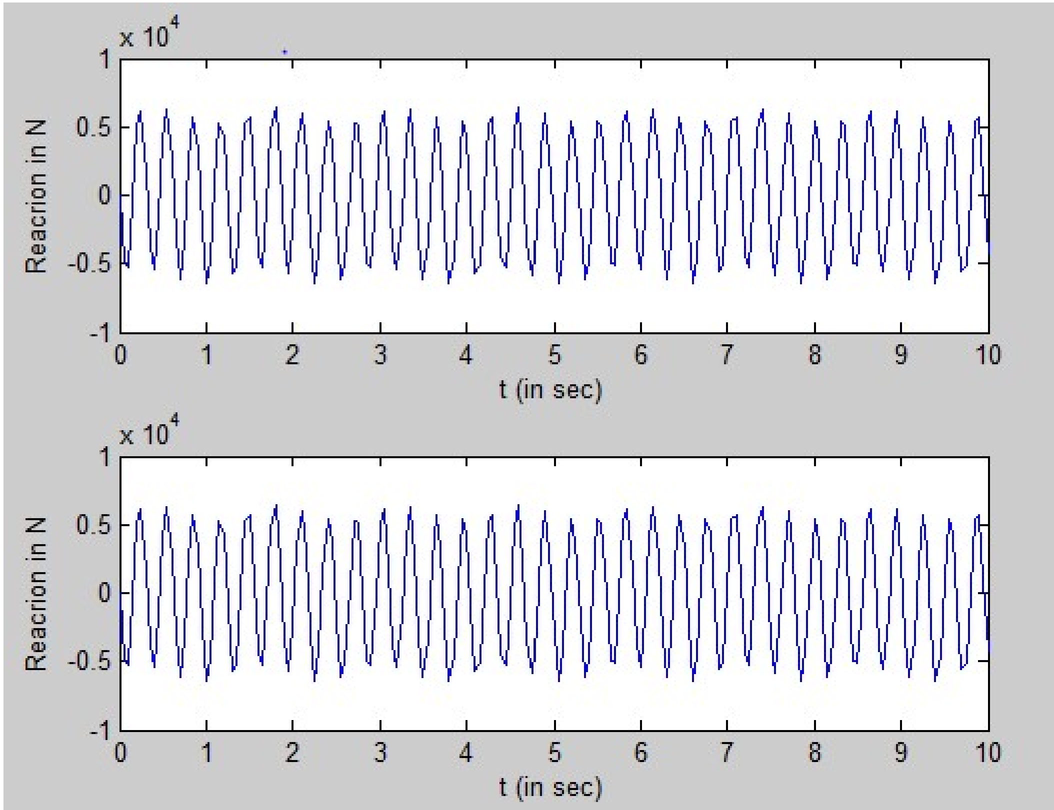


Figure 5.4 shows Reaction Forces at Nodes 1, 2 and 3

3.3 ABAQUS MODEL

3.3.1 Abaqus introduction

Abaqus is a complete and powerful environment for solving complex real life problems providing realistic solutions allowing the users to explore the real-world behavior of product, nature, and life.

This sophisticated Abaqus product provides efficient solutions for most challenging and sophisticated engineering problems. Its applications cover a broad spectrum of industrial applications. Its high robustness, accuracy, and performance are amalgamate with easy to use pre and post processor.

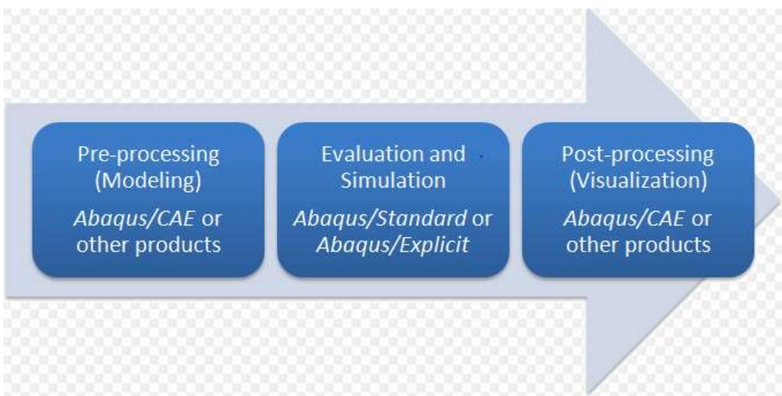


Figure 3.1 shows Abaqus methodology

BENEFITS:

<u>Allows linear and non-linear modelling</u>

Allows modelling using diverse materials like rubbers, thermo-plastics, powder metals, human tissue, soil, composites, and more.

Allows modelling complex assemblies' joint behavior, Flexible multi-body dynamics.

Allows efficient modelling of problems related to Crack, impact, and crash events.

Allows fast and efficient complex and large-scale analyses.

3.3.2 Model introduction

Abaqus uses a finite element model for solving real life, complex engineering problems requiring large-scale analyses. The finite element model is characterized by:

- A Finite-element model in which meshing procedure creates a network of line elements connected with nodes in material continuum.
- Mass of the model is a continuous mass which depends upon density, and geometry of model.
- Deformations (displacements and velocities) at nodes are obtained from finite element method.
- The accuracy of results depends upon how fine or coarse is the applied mesh.

3.3.3 Inputs

For beams, Abaqus finite element analysis requires the following data:

- Beam dimensions and geometry
- Elements lengths, connecting nodes and location of nodal points
- Mass characteristics
- Boundary conditions
- Loading conditions or initial conditions
- Analysis type options

3.3.4 Units

Abaqus does not have any built-in system of units. It requires the user to put all the data in consistent units. In our Abaqus model we have used SI units (N, m, and kg).

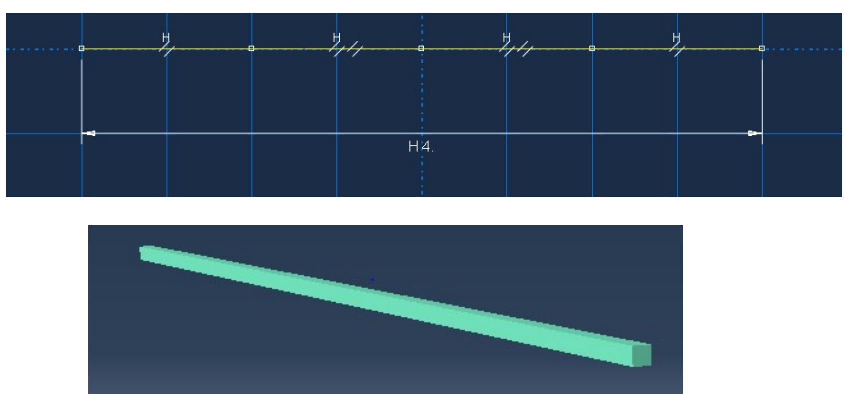


Figure 3.2 shows ELASTIC BEAM MODEL (ABAQUS)

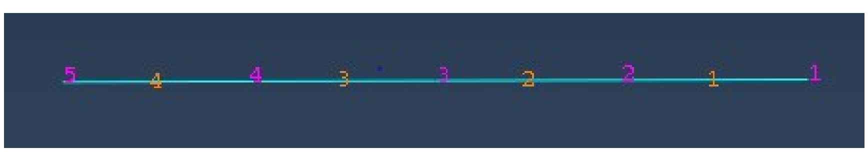
3.3.5 Beam geometry

The first step for finite element analysis of beam is defining its geometry, to make Abaqus problem conformable with the MATLAB model. We modelled the beam as a 2-Dimensions planner element with deformable wire element as a base feature as we had to keep beam in elastic range. The length of beam is 4 m and its width and height are 0.05 m each.

3.3.6 Mesh (element length and nodes)

The meshing procedure creates a network of line elements connected with nodes. The number of elements and nodes in a model depends upon how fineness or coarseness of the applied mesh. For simple problems without changing geometry including thickness or change in boundary conditions for individual elements a coarse mesh is preferred because it provides efficient results within lesser computational time. Fine mesh is used for more complex system as it provides accurate results but requires more computation and time. For this problem a coarse mesh is applied with global seed size of 1 m (element length 1 m) which divides the beam into 4 element of 1 m and the elements are connected to each other with nodes. There are 5 nodes in total for this model 3 in between members and 2 connection the elements with supports.

- Purple colour: nodes
- Orange colour: elements



3.3.7 Mass properties

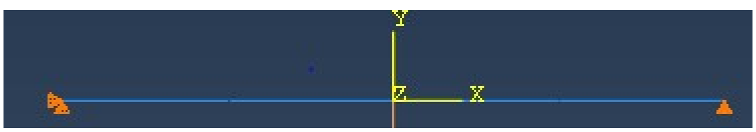
Abaqus assigns a same continues mass to all the elements of beam based on the density specified to the model. In this example density of steel ρ = 7850 kg/m^3 is used. Beam dimensions are already provided (4 x 0.05 x 0.05) m the software will calculate and assign mass to the beam. As we have to carry out elastic analysis therefore beam is modelled as elastic beam with elasticity is E = 2.1 x 10^{11} N/m^2 (210GPa) and 0.3 Poisons ratio.

m = ρ x v

m = 78.5 kg/m

3.3.8 Boundary conditions

The boundary conditions are applied by deploying simply supported beam conditions. The left support is setup as pinned support and the right support is setup as roller support.



3.3.9 Initial conditions

Abaqus model requires boundary conditions depending upon the type of problem. Generally impact problem is modelled on Abaqus by defining an impactor (with a particular mass and initial velocity) colliding with a stationary body with its own mass and boundary conditions. But for this example to make it comparable to the MATLAB the central node of beam was given initial velocity of 0.15 m/s in negative y direction and a point mass of 150 kg added to the central node equal to the impactor mass. The added mass acts as a structural mass at central node.

3.3.10 Analysis type options

The final step is to specify the type of analysis procedure to be carried out. For this problem dynamic explicit analysis was adopted because it is computationally efficient for analysis of impact problems with short time period for dynamic response. For the elastic beam impact analysis the time period was selected as 10 sec with 0.05 as the time step increment.

3.3.11 Results

The figure below shows the result for elastic beam model for Test no 1:

Test no 1 M =150 kg
V = 0.75 m/s
ρ =7850 kg/m^3
E = 2.1 x 10^11 N/m sq.
L = 4 m
L x W (0.05 x 0.05) m sq.

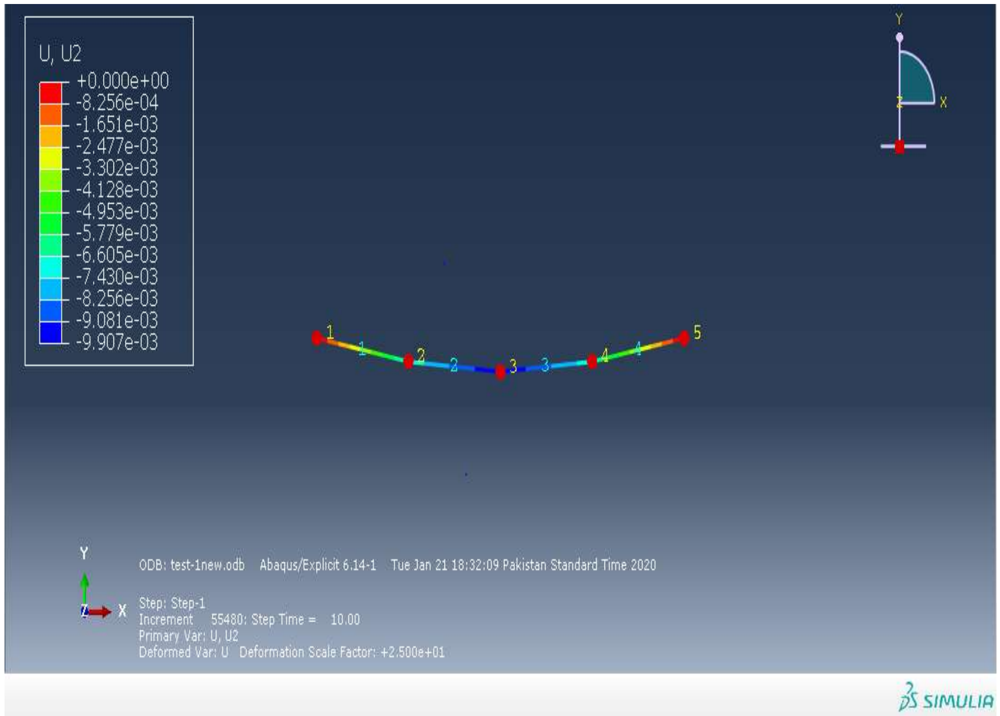


Figure 3.3 shows Test no.1 performed with Abaqus

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 Test data

Three tests are conducted on both Abaqus and MATLAB by increasing the impact mass and velocity successively, while keeping other perimeters constant (length, moment of inertia, elasticity and density). The impact mass is increased by 50 kg during each succession and the impact velocity is increased by 0.5 m/s. The impact mass and velocity was decided such that the beam remains in elastic zone

The test data is given below in tabular form.

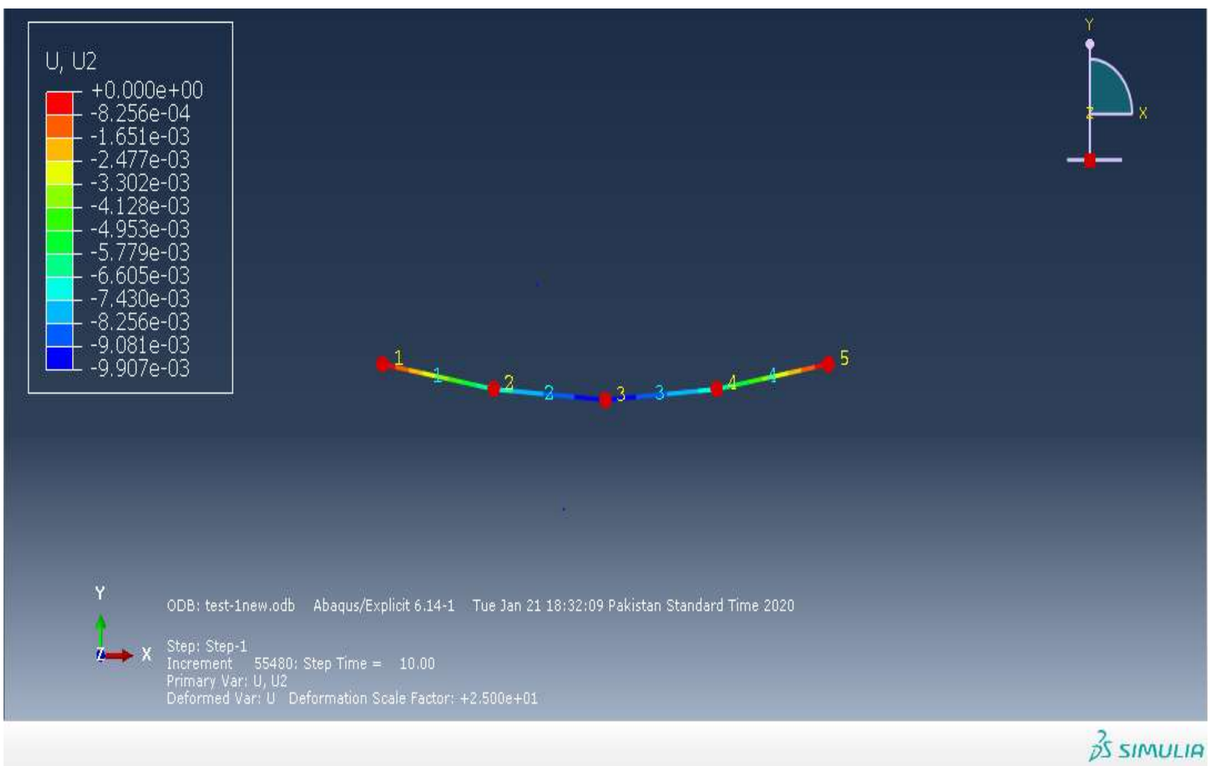
Inputs	Test 1	Test 2	Test 3
Mass/length	26.167 kg/m	26.167 kg/m	26.167 kg/m
Total Length	4 m	4 m	4 m
No. of elements	4	4	4
Elasticity modulus	2.1 x 10^11 N/m sq.	2.1 x 10^11 N/m sq.	2.1 x 10^11 N/m sq.
Moment of Inertia	5.208 x 10^-7 m^4	5.208 x 10^-7 m^4	5.208 x 10^-7 m^4
Impact mass	150 kg	200 kg	250 kg
Initial Velocity	0.75 m/s	1 m/s	1.5 m/s

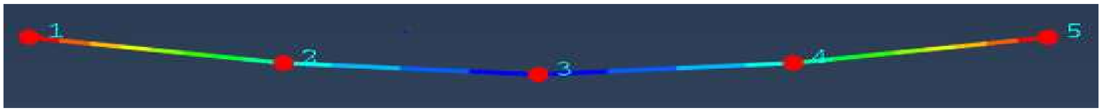
The time span for analysis is 10 seconds and time step is 0.05 seconds for both Abaqus model and MATLAB program, meaning the analysis will results in 200 values (displacements or velocities) for each analysis. The time period is kept comparable on purpose.

In both Abaqus and MATLAB models the node location is kept identical so that the displacements and velocities at these nodes can be compared. In both models node 1 is located at the extreme left and then node 2 is located at a distance of 1m from node 1, node 3 is located at a distance of 1m from node 2, node 4 is located at a distance of 1m from node 3, node 5 is located at the extreme right ta distance of 1m from node 4.

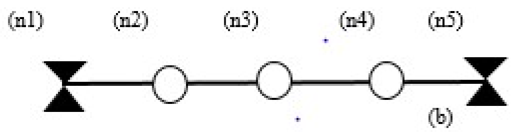
ABAQUS MODEL:

Test 1: Impact mass = 150 kg; Initial velocity = 0.75 m/s





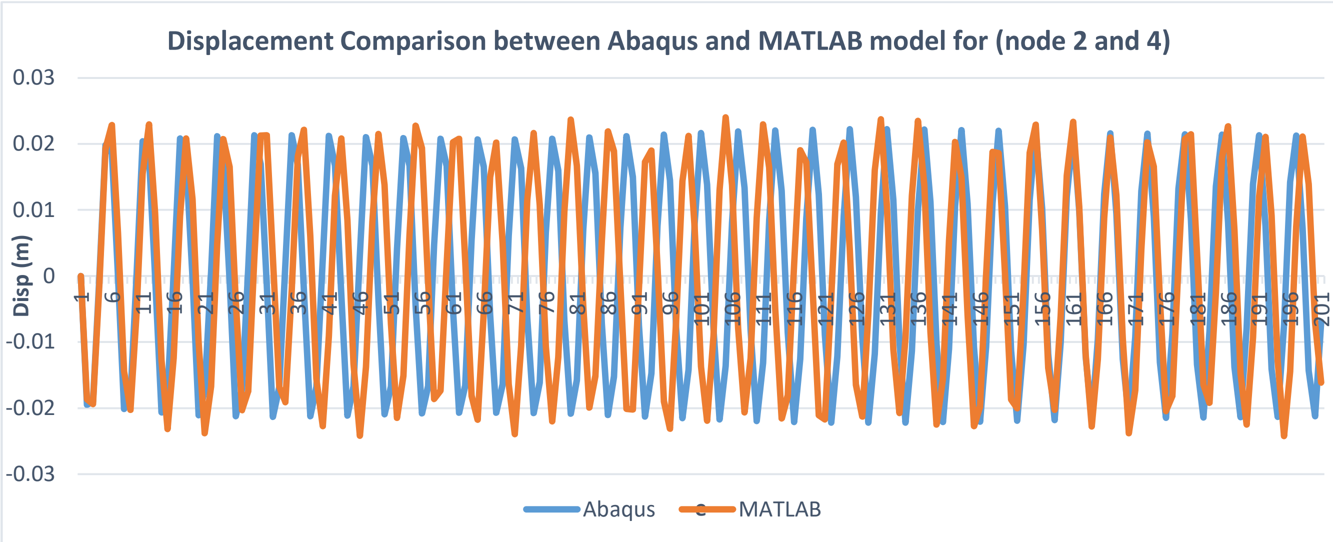
MATLAB MODEL:



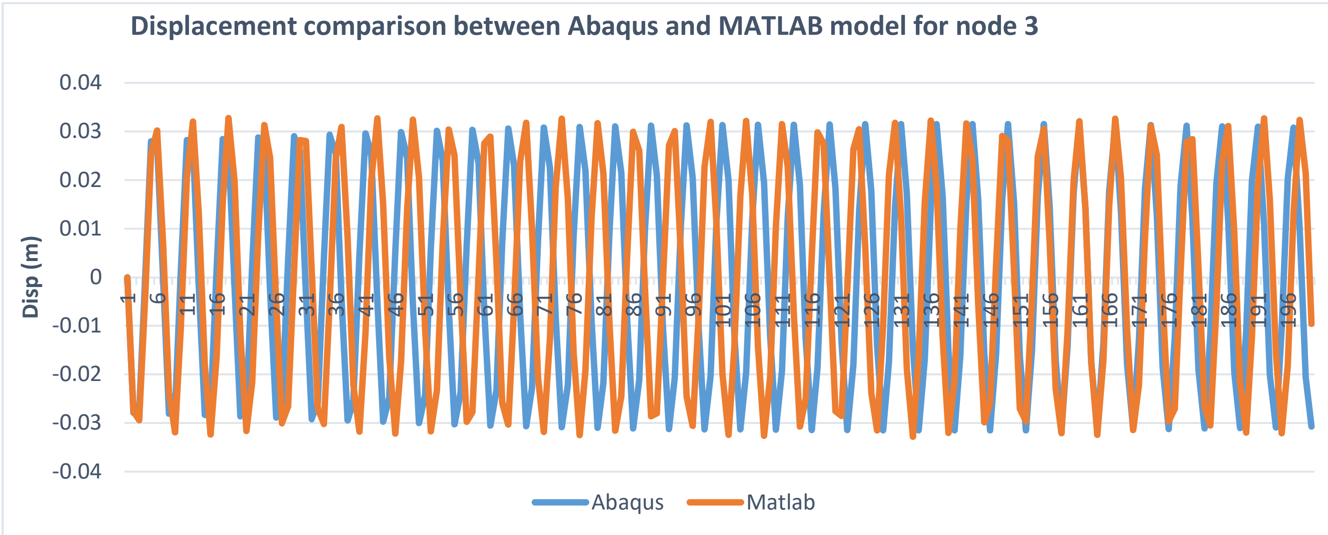
4.2 Comparison of results

Test 1: Impact mass = 150 kg; Initial velocity = 0.75 m/s

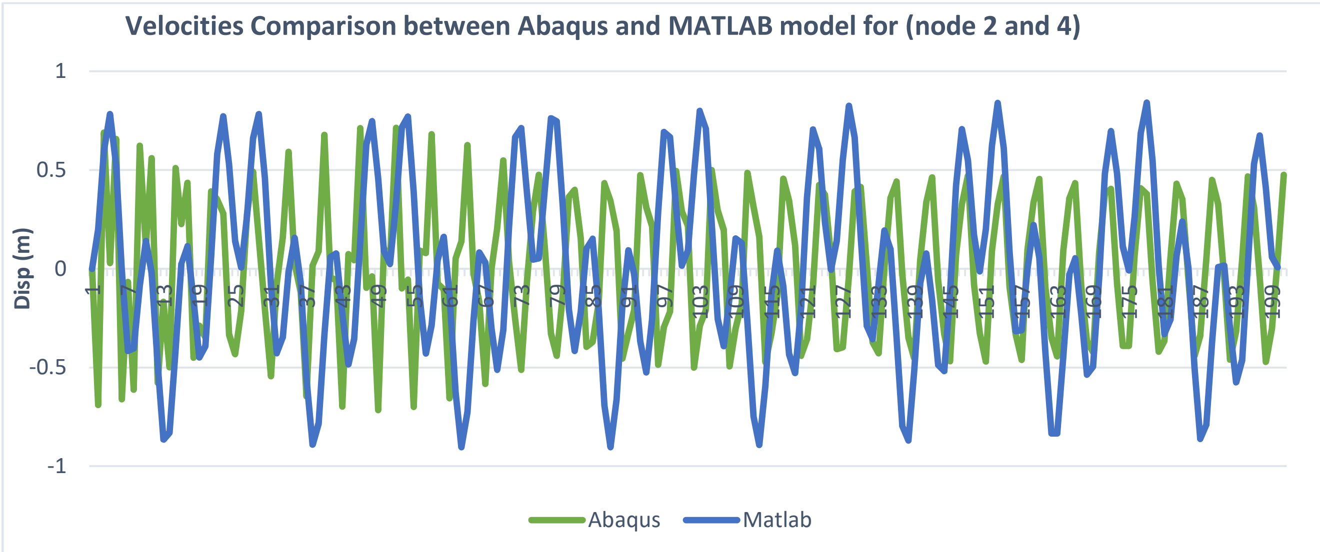
The results are shown for test 1 only as the results for other tests shows similar trends but with higher magnitude.



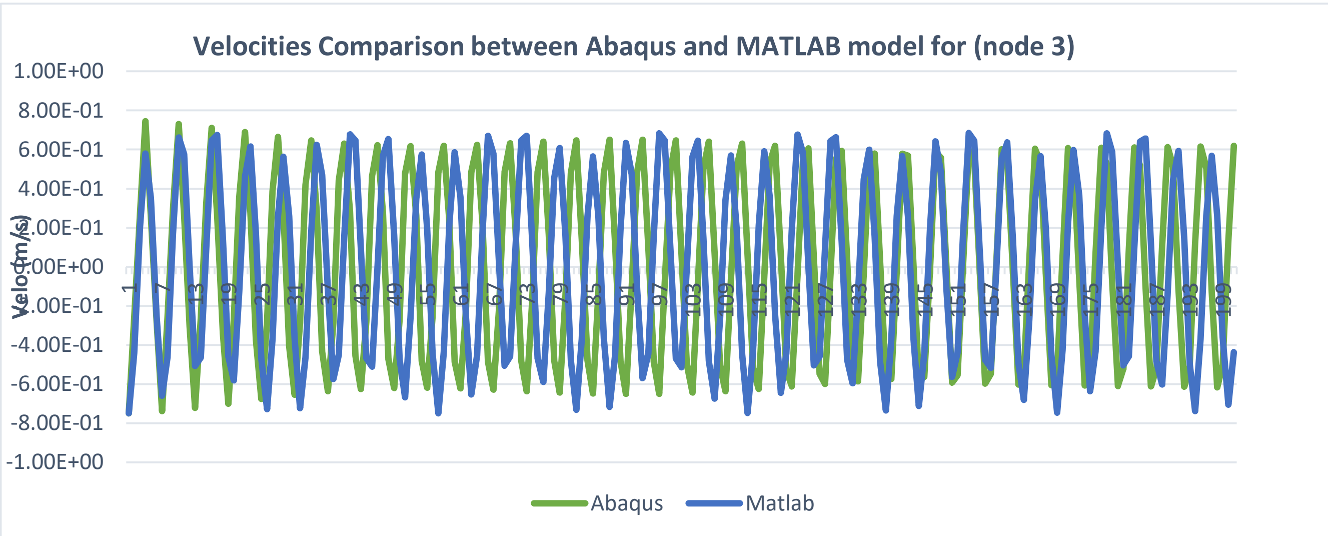
Model	**Max Displacement (m)**	**Max Disp (cm)**	**At time step (s)**	**% Error**
Abaqus	0.0222095	2.220	6.20013	5.72
MATLAB	0.0234706984662625	2.347	6.75005	



Model	Max Displacement (m)	Max Disp (cm)	At time step (s)	% Error
Abaqus	0.0314882	3.155	6.20013	2.18
MATLAB	0.0322486412939452	3.224	6.75005	

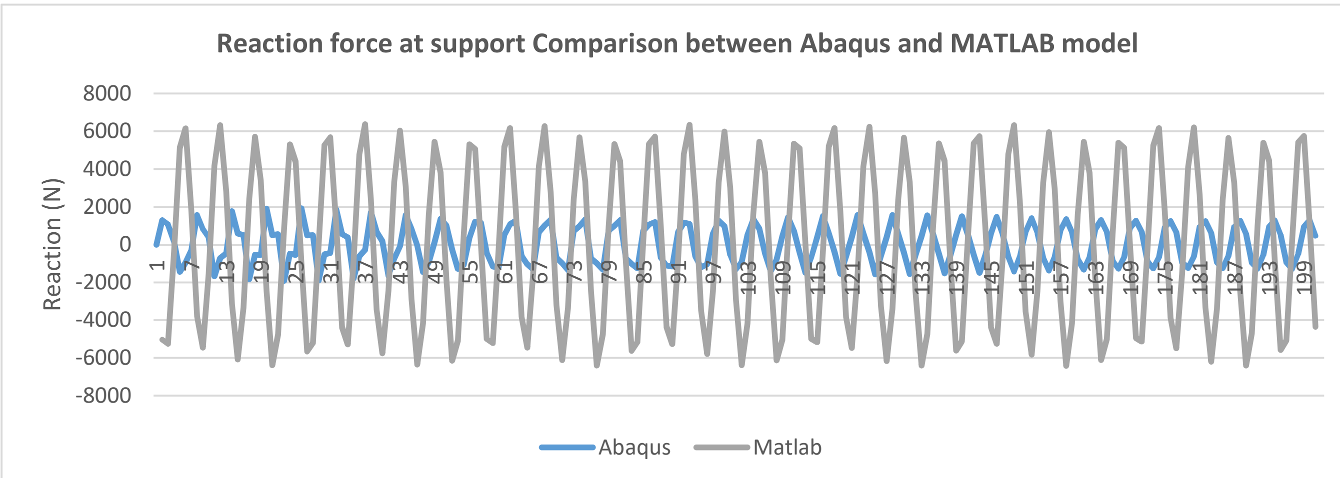


Model	Max Velocity (m/s)	Max Velo (cm)	At time step (s)	% Error
Abaqus	0.712076	71.2076	2.25007	8.18
MATLAB	0.770372643571549	77.0372	2.65003	



The maximum velocity is the initial velocity = - 0.75 m/s at t = 0 seconds. The table shows the maximums velocity at some time step when the analysis runs for a specific time span.

Model	Max Velocity (m/s)	Max Velo (cm)	At time step (s)	% Error
Abaqus	0.744984	74.4984	0.150164	1.77
MATLAB	0.731793232291336	73.1793	4.05017	



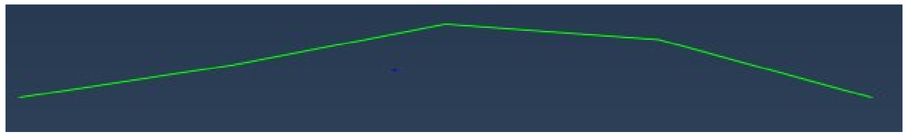
As we can see from the graph results of supports reactions using Abaqus and MATLAB show significant differences. Also the results for shear force, moments show significant differences, and so there is needs for re-evaluation.

Abaqus results

For modal analysis

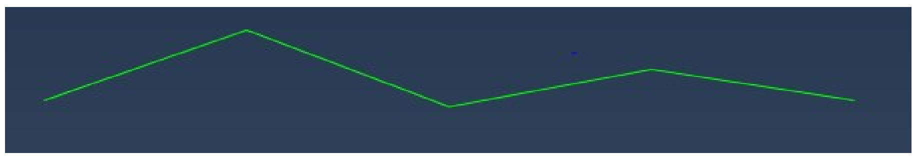
Index	Description
0	Increment 0: Base State
1	Mode 1: Value = 1049.5 Freq = 5.1560 (cycles/time)
2	Mode 2: Value = 50886. Freq = 35.902 (cycles/time)
3	Mode 3: Value = 1.52980E+05 Freq = 62.250 (cycles/time)

Mode 1



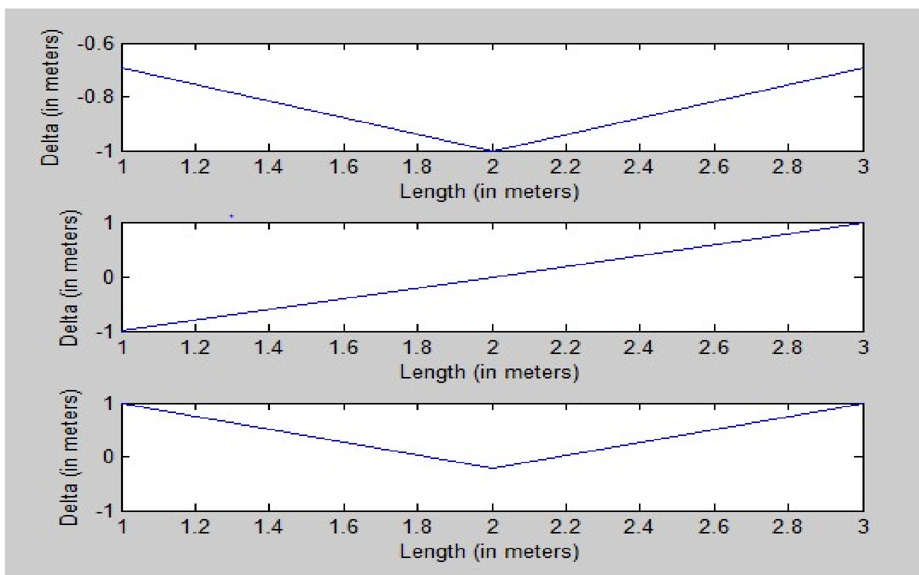
Mode 2

Mode 3



MATLAB results for modal analysis

```
nat_freq_1 =

   20.1972

nat_freq_2 =

  158.3594

nat_freq_3 =

  255.7863
```



The results of modal analysis from Abaqus and MATLAB show significant differences. So it needs to be re-evaluated.

CHAPTER 5

CONCLUSIONS

5.1 Conclusions

- The comparison shows that the results for velocities and displacements from both procedures i.e. Abaqus simulation and MATLAB model are comparable. The slight differences are possibly due to the fact that the Abaqus model is based on finite element model in which mass of the beam is continuous mass. Whereas in case of MATLAB node element model the mass of the beam is lumped at nodes.

- The results for support reactions, shear force, moments, Eigen values and modes shapes are not comparable, they show significant difference, this implies that MATLAB code requires revision concerning these results.

5.2 Future prospects

The next phase would be to fix the issues regarding support reactions, shear forces, moments, Eigen values and modes shapes and to verify the results by experimentation using some standardize equipment. The MATLAB program we coded for elastic simply supported beam can be extended to elastic-plastic behavior of the beam. Also it can be improved to account multi degree of freedom systems such membrane, shell and 3D elements. It can be extended to cater for more complex loading conditions i.e. seismic loading. It can be extended to study vibration phenomenon such as the vibration of railways tracks due to cyclic impact loading.

REFERENCES

[1] Jenkins, W. M (1982) Structural mechanics and analysis level IV7 V. Thomas Nelson/Van Nostrand Reinhold, London.

[2] Mechanical Vibrations, Sixth Edition in SI Units, by Singiresu S. Rao (1986) 612-630

[3] Investigation of rigid-plastic beams subjected to impact using linear complementarity Azam Khan, David Lloyd Smith, Bassam A Izzuddin (2013)

[4] Mathematical programming methods for dynamically loaded rigid-plastic framed str uctures by carmen lucia de me squita Sahlit (1992)

[5] Cheung, Y. K. and Yeo, M. F. (1979) A practical introduction to finite element analysis. Pitman, London.

[6] Dawe, D. J. (1984) Matrix and finite element displacement analysis of structures. Clarendon Press, Oxford.

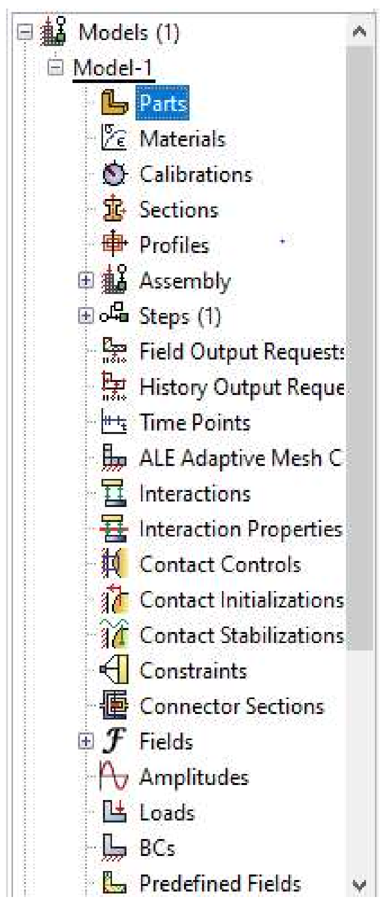
[7] Graves Smith, T. R. (1983) linear analysis of frameworks. Ellis Horwood, Chichester.

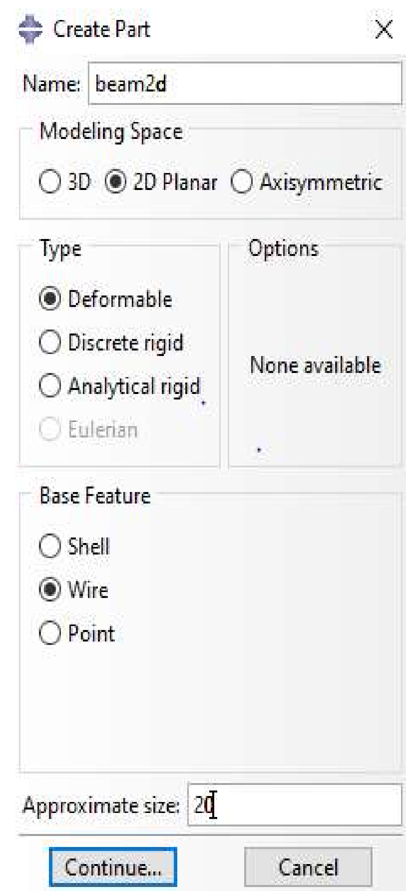
[8] Book - Dynamics of Structures by Anil K. Chopra, 8^{th} edition, 2012.

APPENDIX A

BEAM MODEL IN ABAQUS

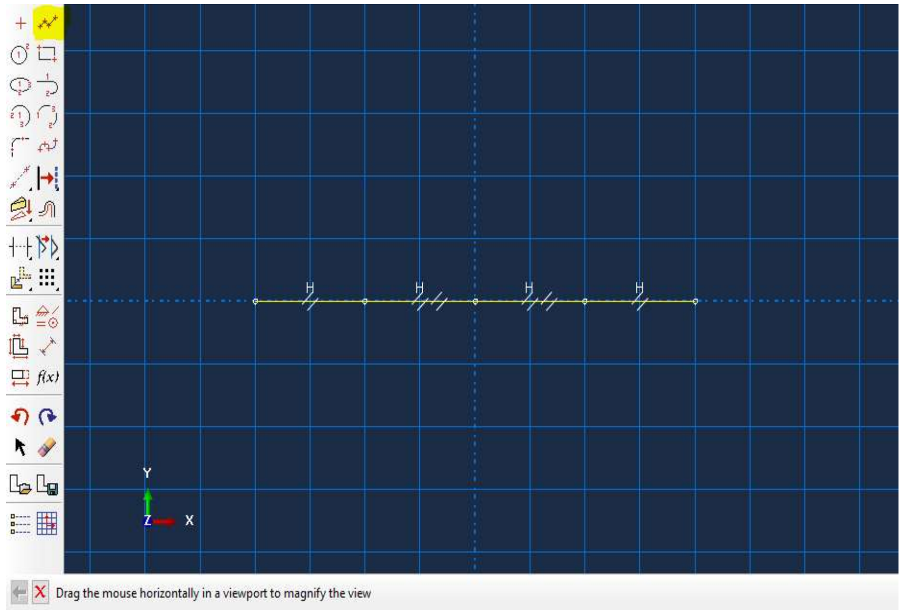
1. Open Abaqus CAE and create a Model Database by selecting the highlighted option (With Standard/Explicit Model)



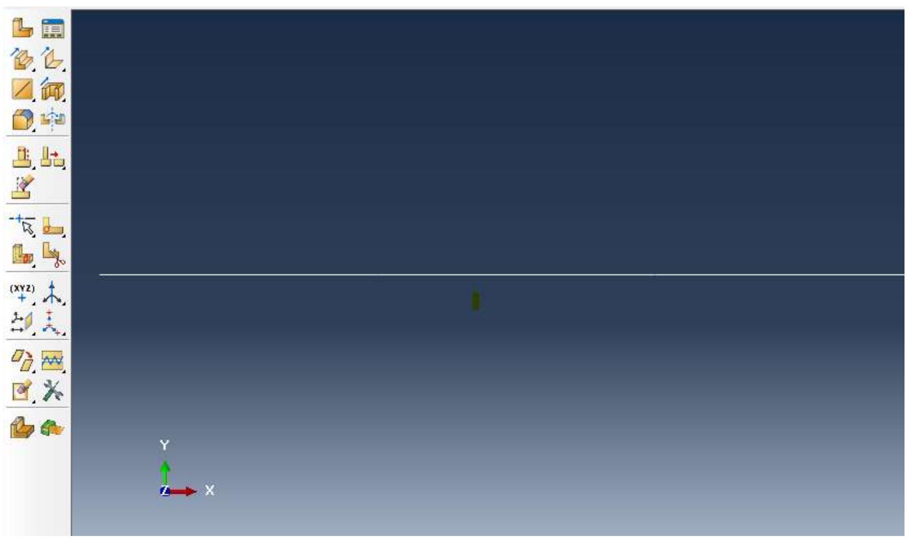


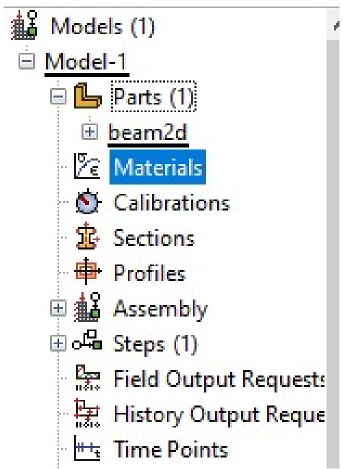
2. Select "Part" module from the model tree. Click create and "Create Part" table will appear name the part i.e. "beam2d" in this case. Select the "Modelling Space" depending upon the type of problem you want to model i.e"2D planner" in our case. Select the type of beam based on its behaviour i.e. "Deformable". Select the approximate size.

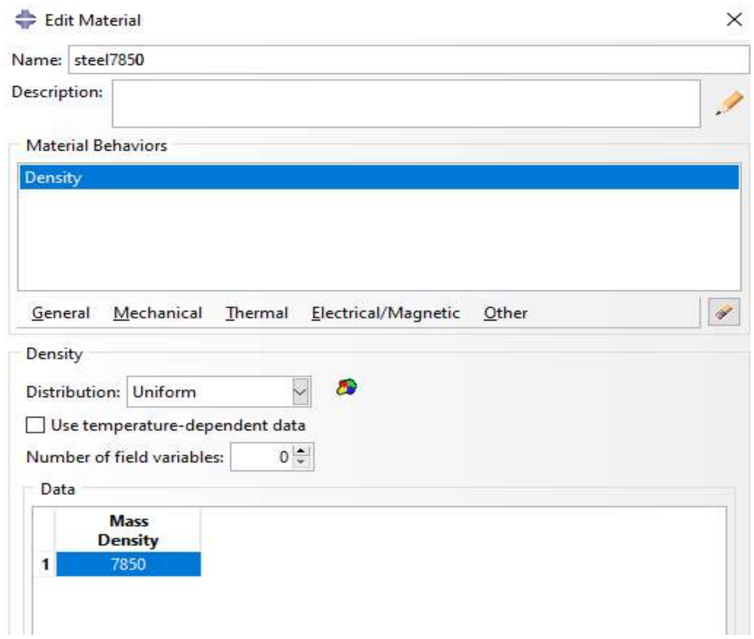
3. Select the highlighted line feature and sketch the beam of required length i.e. 4m in our case.



Press "Done" and beam is formed in the workspace.

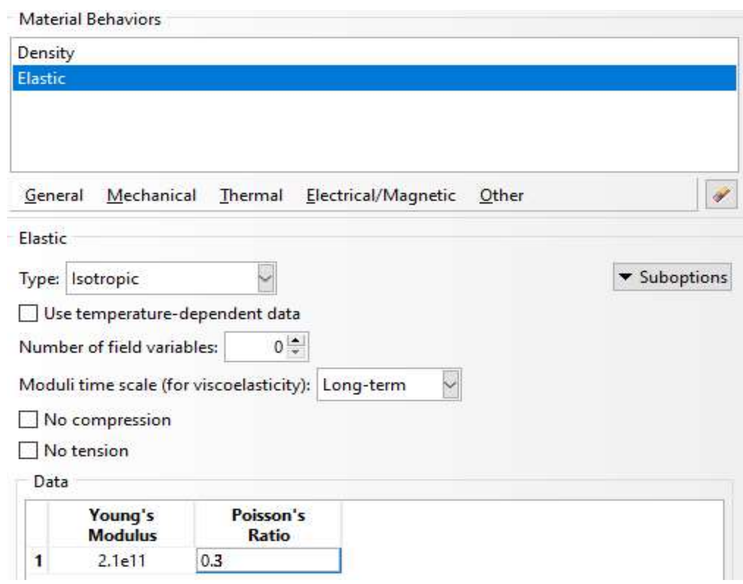




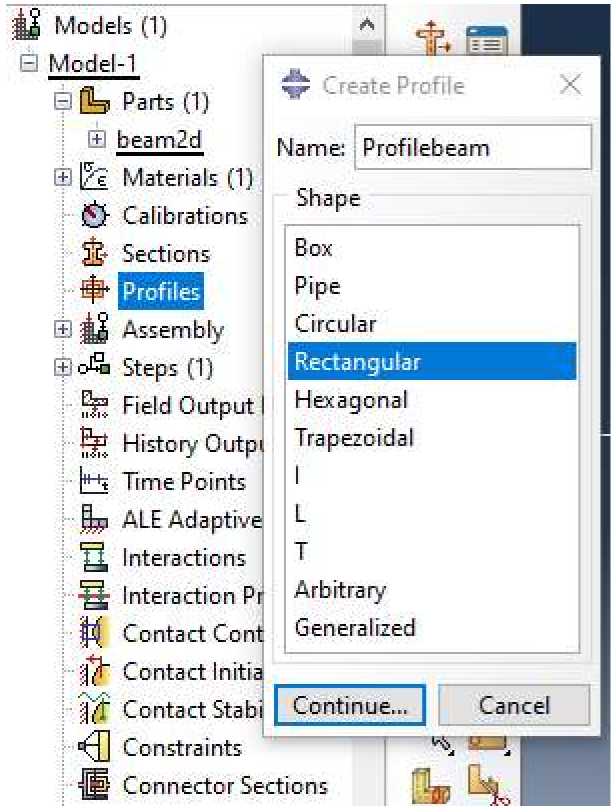


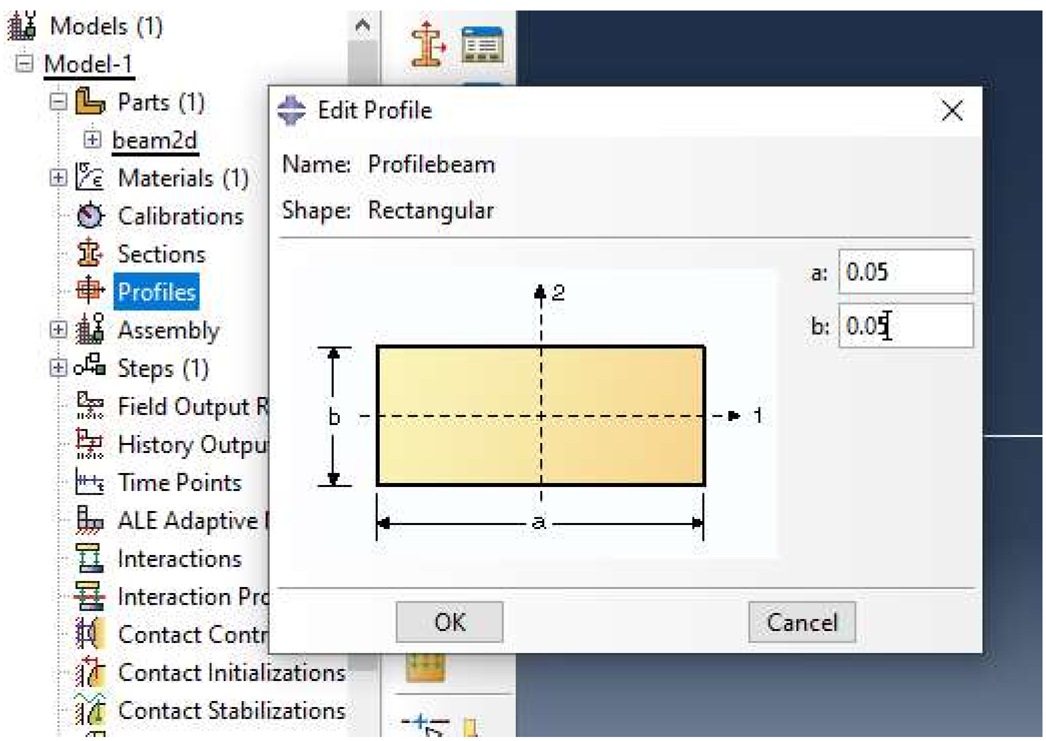
4. Select "Material" module from the model tree. Click create and "Create" to create material of beam. Define the density by clicking "General" tab and put the value of density in table.

Define the elasticity and Poisons ration by clicking "Mechanical" tab and put the values in table.

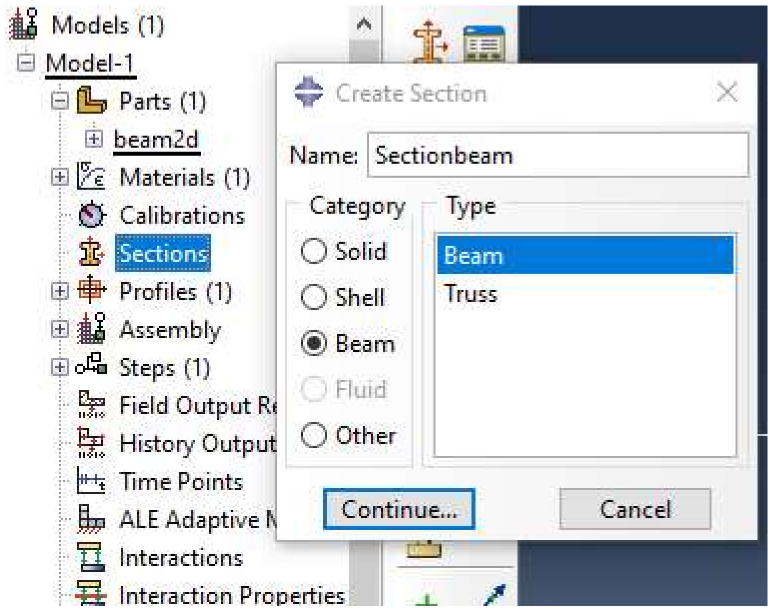


5. Select "Profile" module from the model tree. Click create and "Create Profile" table will appear name the profile i.e. "profile beam" in this case. Then select the shape of beam. After selecting the shape of beam "Edit Profile" table will appear put the values of shorter dimensions of the beam i.e. width and height.

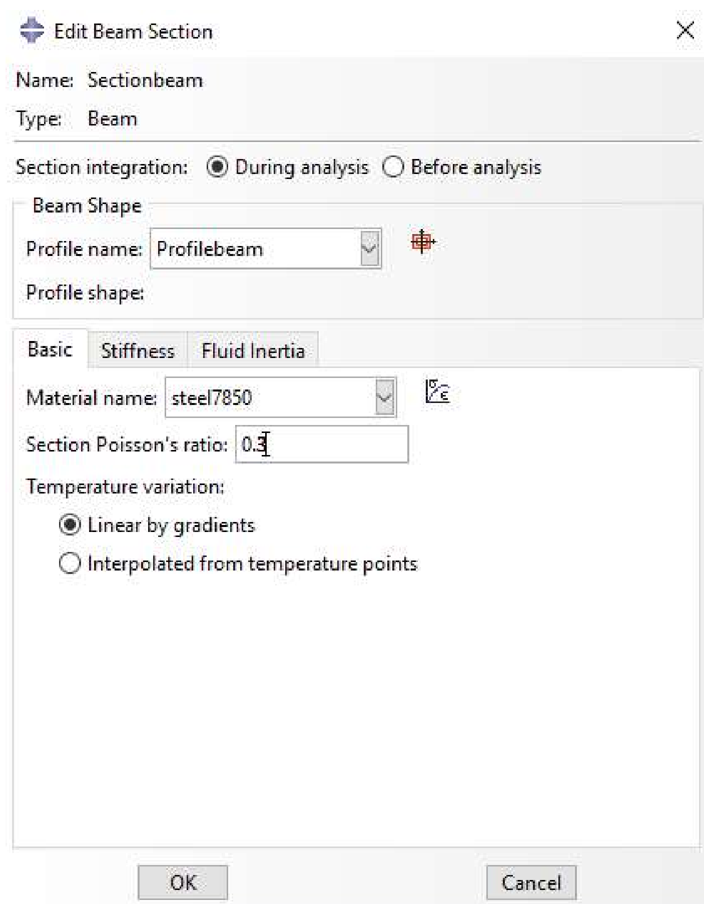




6. Select “Section” module from the model tree. Click create and “Create section” table will appear name the section i.e. “section beam” in this case. Check beam as a category and continue.

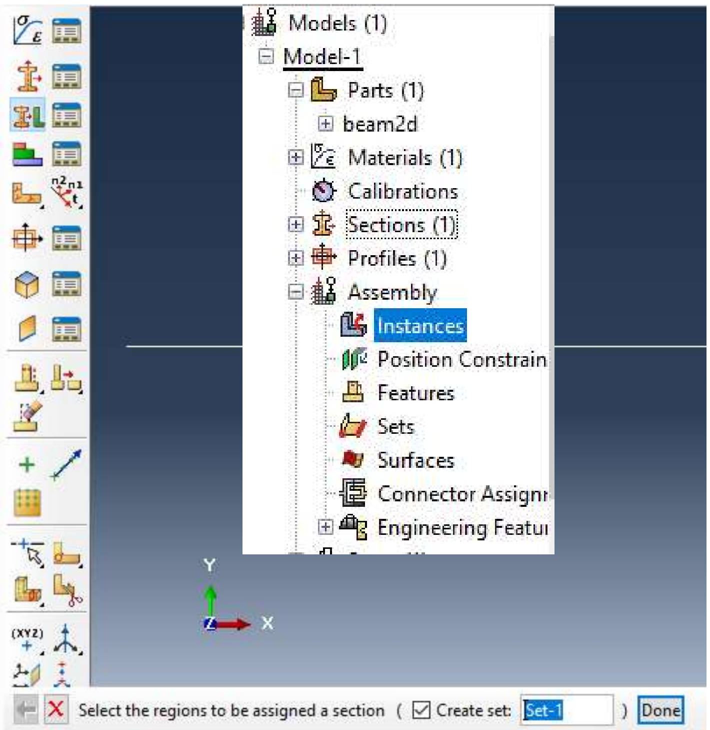


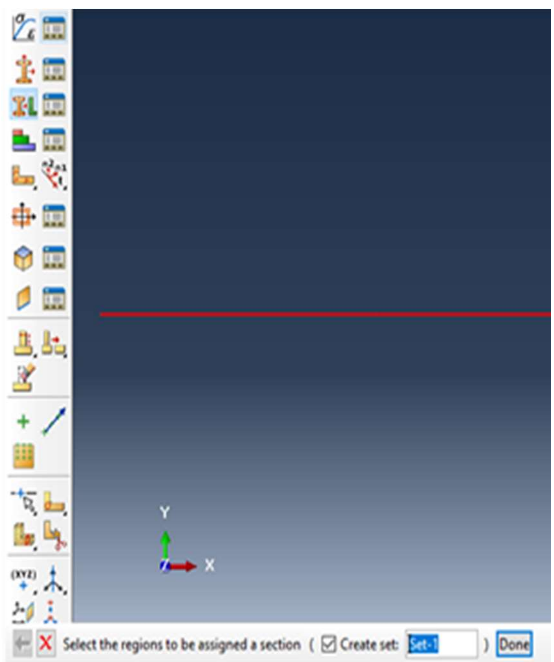
“Edit Beam Section” Table will appear press ok.

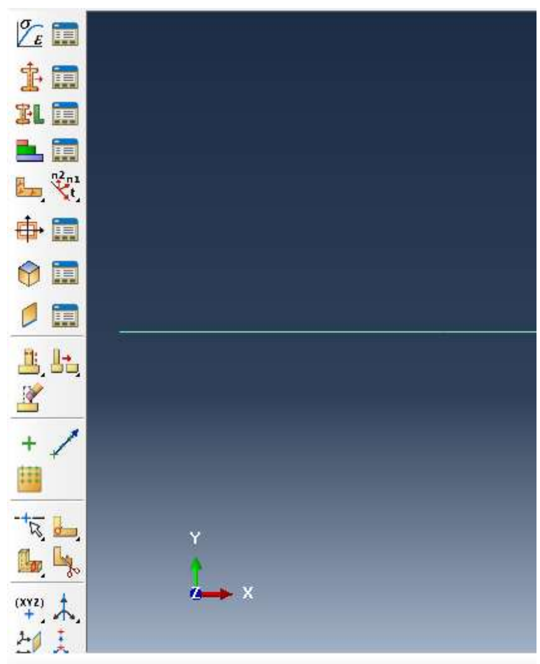


7. Select the "section assignment" tool as highlighted in the figure. Select the beam in the working space the beam will turn "red" and press done.

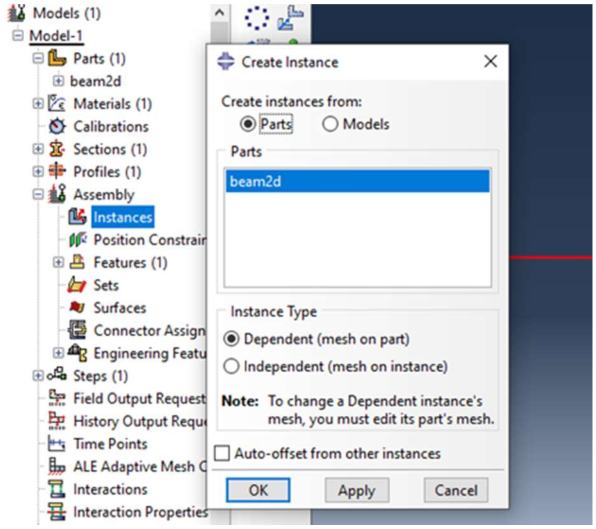
   Finally the beam will turn "green" meaning beam has been assigned section with density and elasticity defined in previous steps.

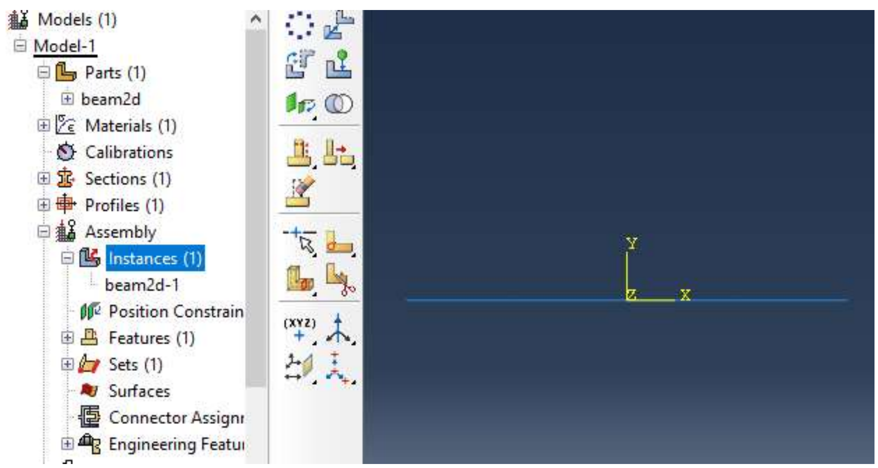






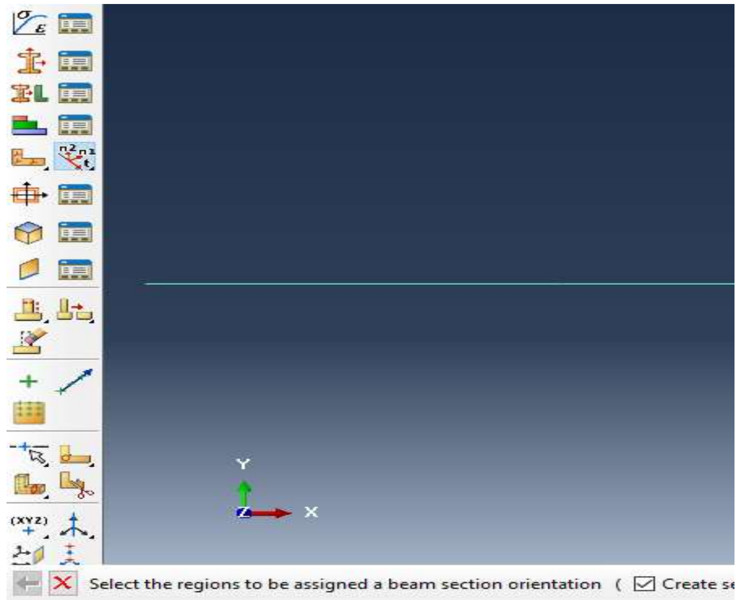
8. Select the “Instances” From “Assembly Module” and click Create and “Create Instance” Table will appear check “Dependent (mesh on part) “ And press ok beam instance Would be created.

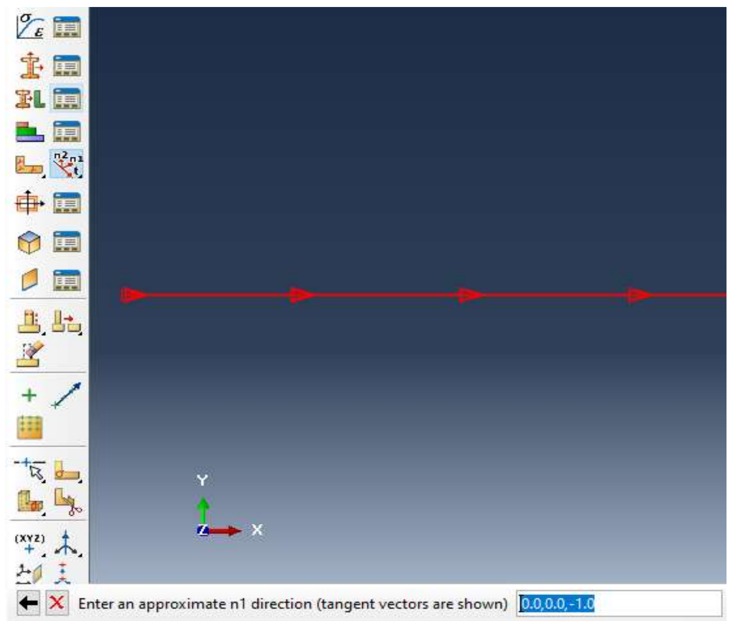


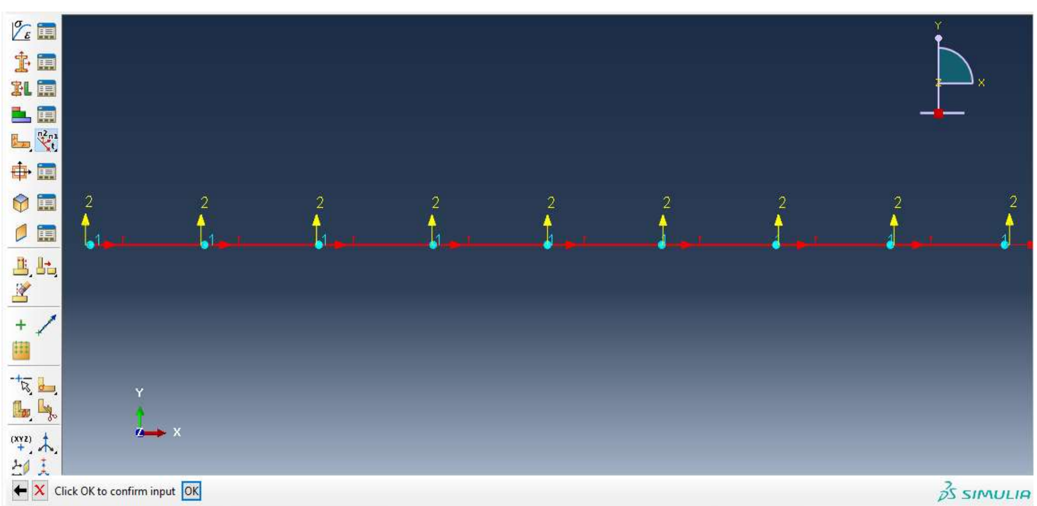


9. Select the “Assign Beam Orientation” tool as highlighted in the figure. Select the beam in the working space the beam will turn “red” and press done.

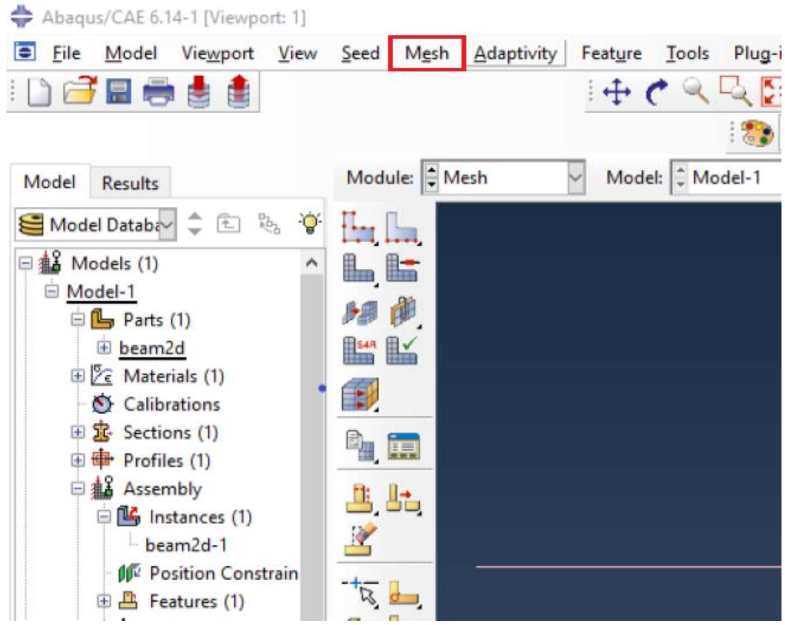
   The beam will be assigned orientation.

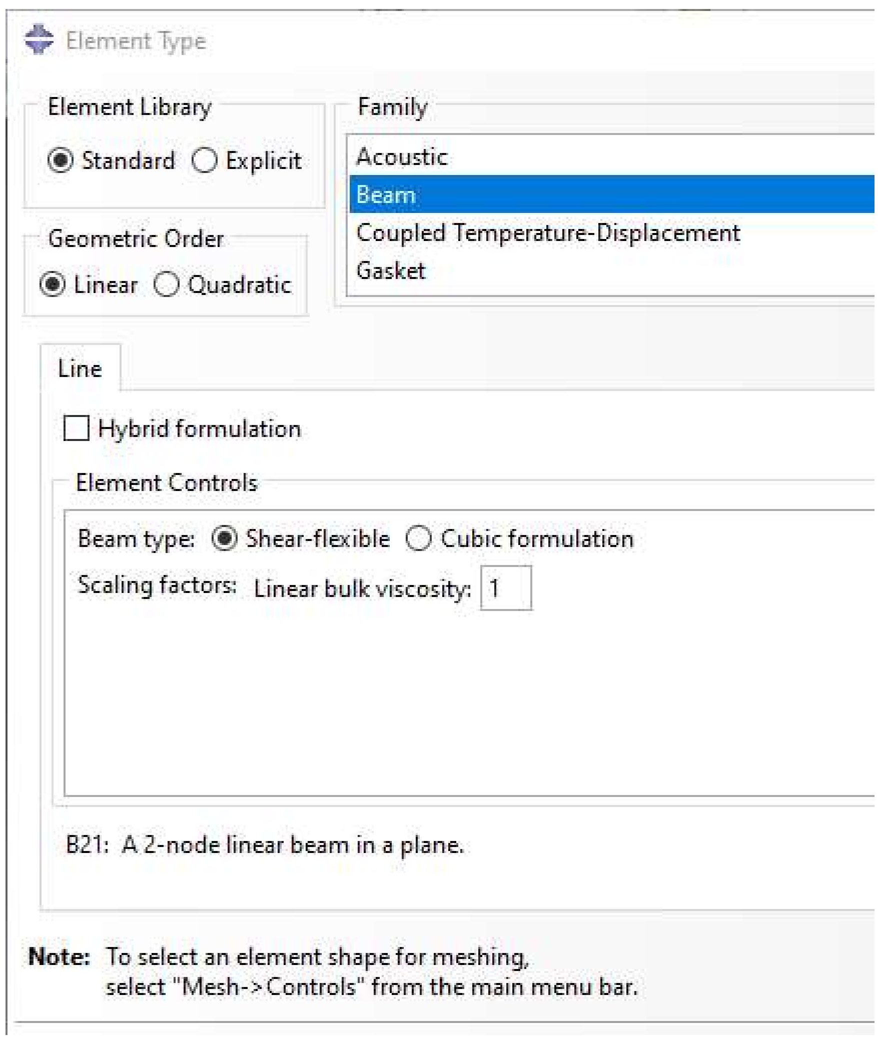




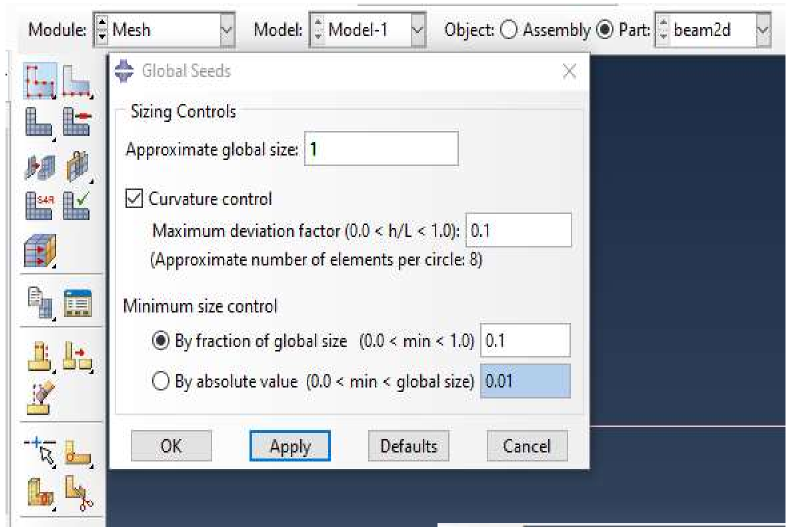


10. From the main tool bar select the "mesh" option as highlighted by "red box". Then from drop down menu select the "element type option" Element type table will appear check "standard" and "linear" options in the family of beam, and click ok.

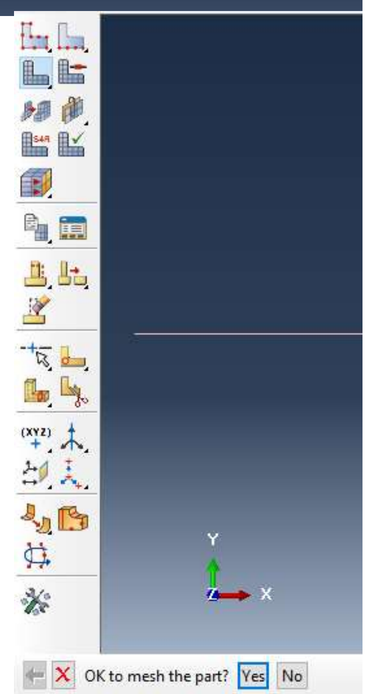




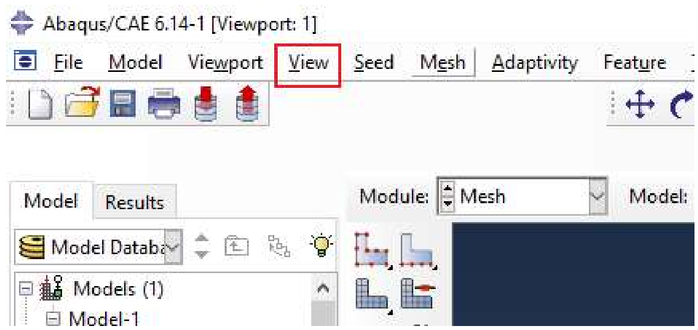
11. Select the “Assign Beam Orientation” tool as highlighted in the figure. Global seeds table will appear select the “Approximate global size” i.e. 1 selected in this case means 1m meaning beam Of 4m is divided into 4 Elements of 1m each.

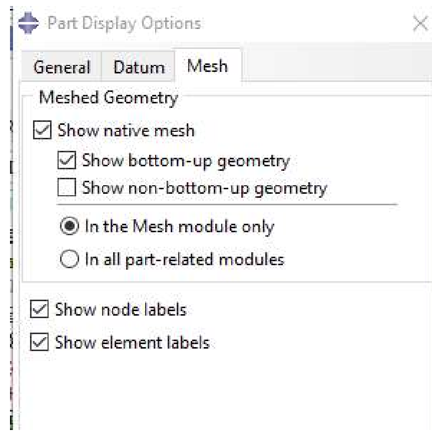


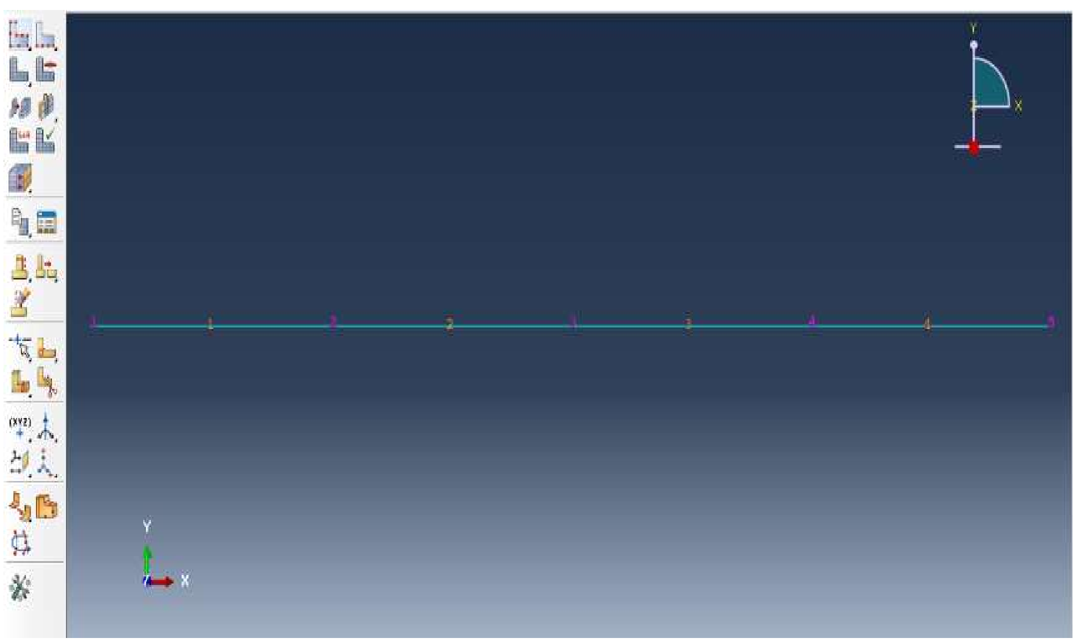
Select the mesh part option and click ok.



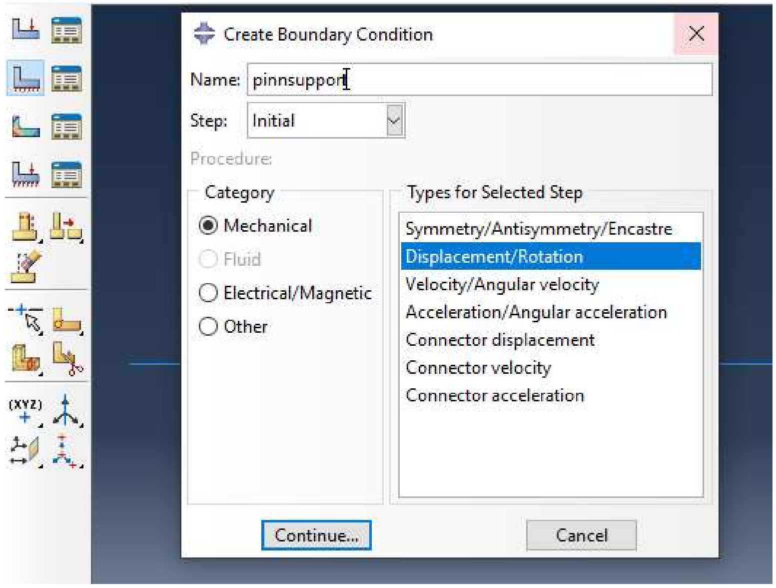
12. From the main tool bar select the "view" option as highlighted by "red box". Then from drop down menu select the "Part display option" table will appear in the mesh tab check "Show node label" and "show element label options" and press ok.



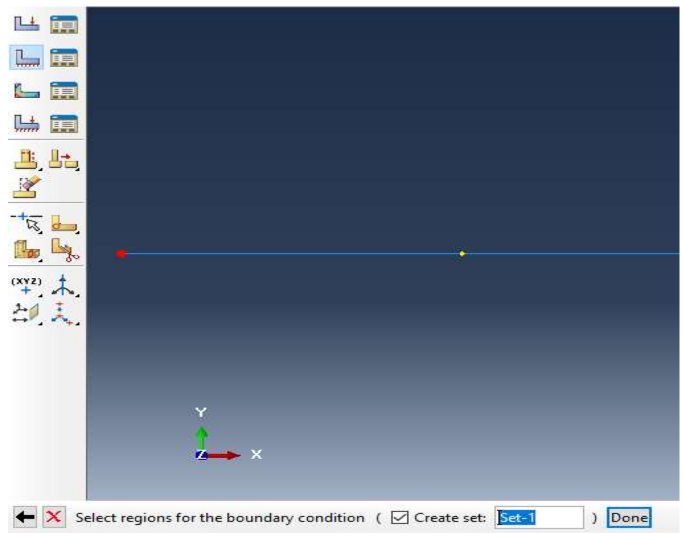




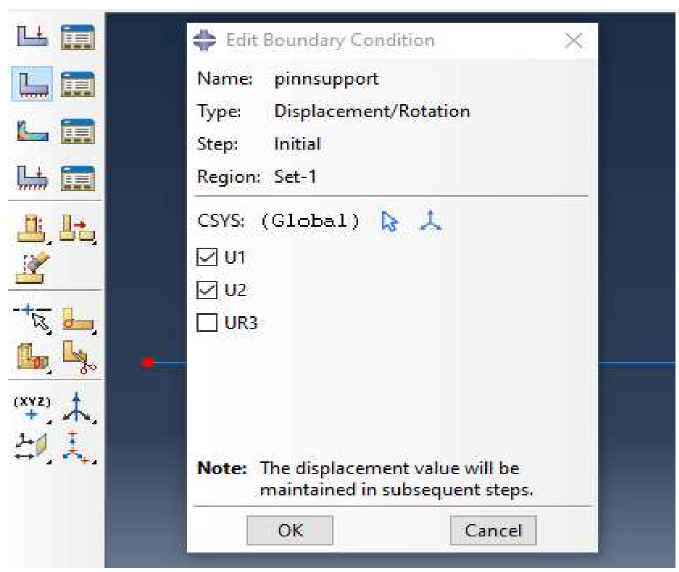
13. Select the “Assign boundary condition” tool as highlighted in the figure. Create “Boundary condition table” will appear name the type of support i.e. pin support in this case.



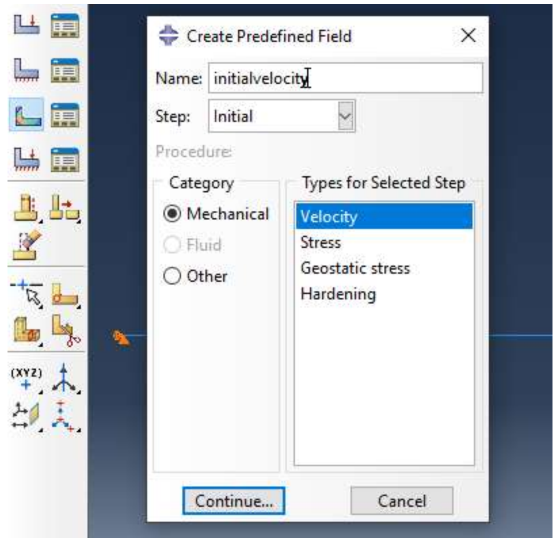
Check “mechanical” as a category and “displacement rotation” as type of selected step and continue. Then select the node you want to assign support condition the selected point will become “red” and click done. “Edit Boundary Condition” manager will Appear check U1 and U2 to Constraint those translations And click ok

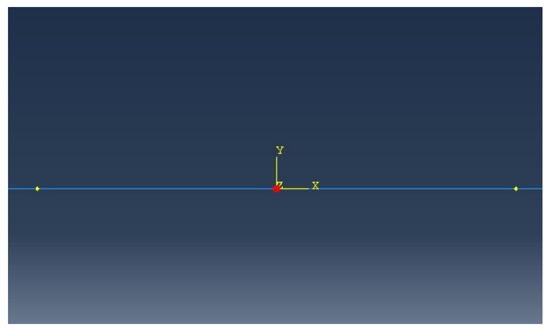


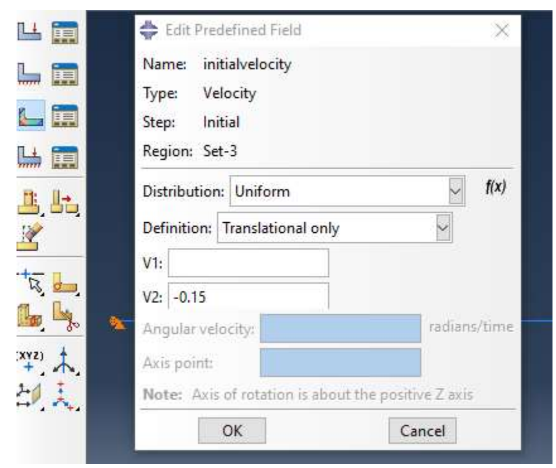
Pin support is created at the Node.

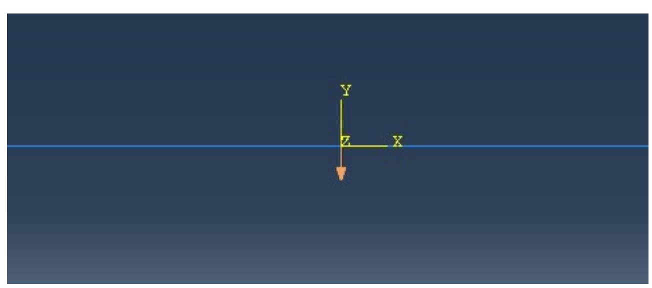


14. Select the “Create predefined field” tool as highlighted in the figure. Create “Create predefined field” will appear name the type of field i.e. initial velocity in this case. Check “mechanical” as a category and “velocity” as type of selected step and continue. Then select the node you want to assign initial velocity, then selected point will become “red” and click done. “Edit Predefined Field Condition” manager will Appear put the value of velocity in downward direction and click ok Initial velocity will be assigned to the Node.









15. Select “Inertia” from “Engineering Features” module of the model tree.

Click create and “Create inertia”

Table will appear name the inertia

Mass i.e. “Inertia-1” in this case.

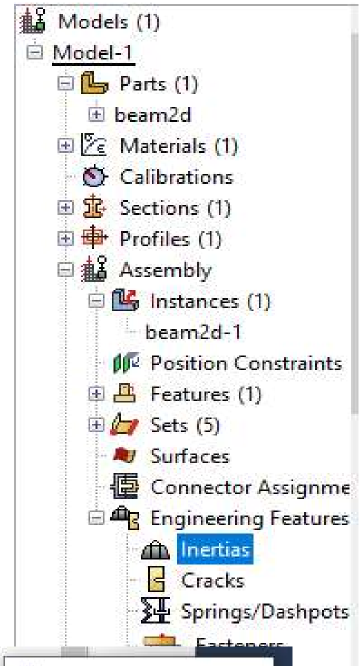
then select the node to which

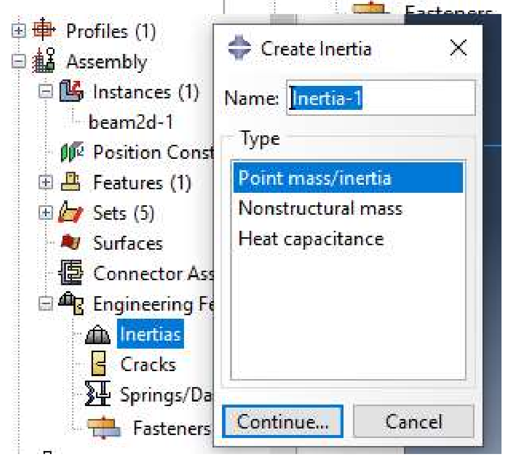
you want to assign impact mass.

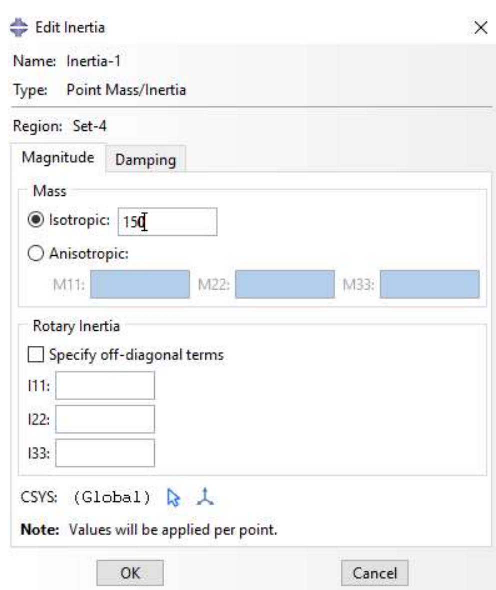
Then “Edit Inertia” table will

appear input the impact mass

and click ok.







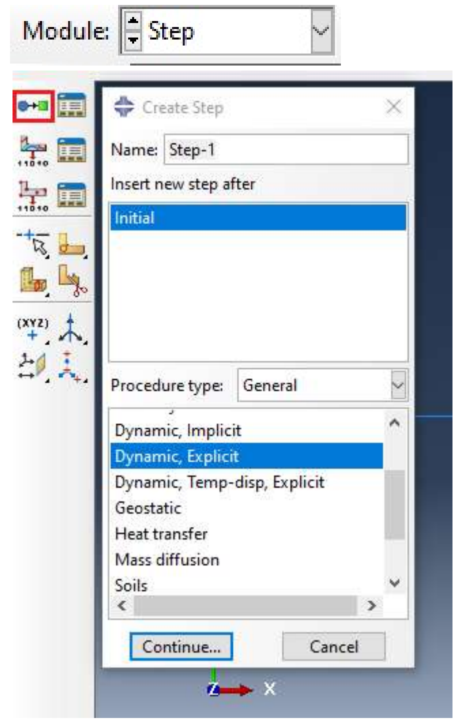
16. In “step module” select the

“Create step” tool select

The type of analysis procedure

“General” and “dynamic explicit”

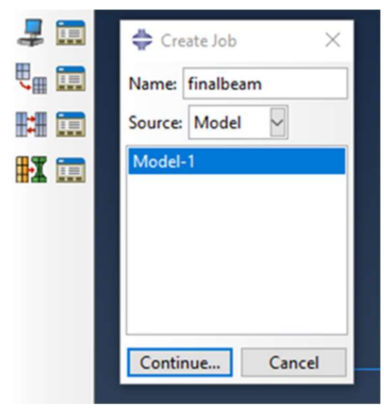
And continue.



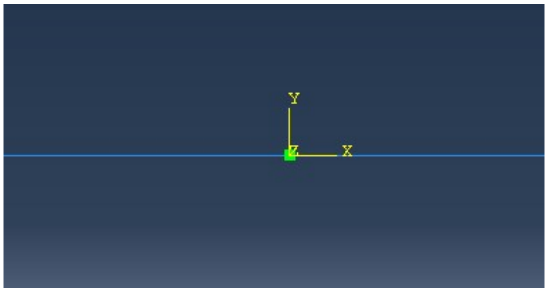
17. In “Job module” select the

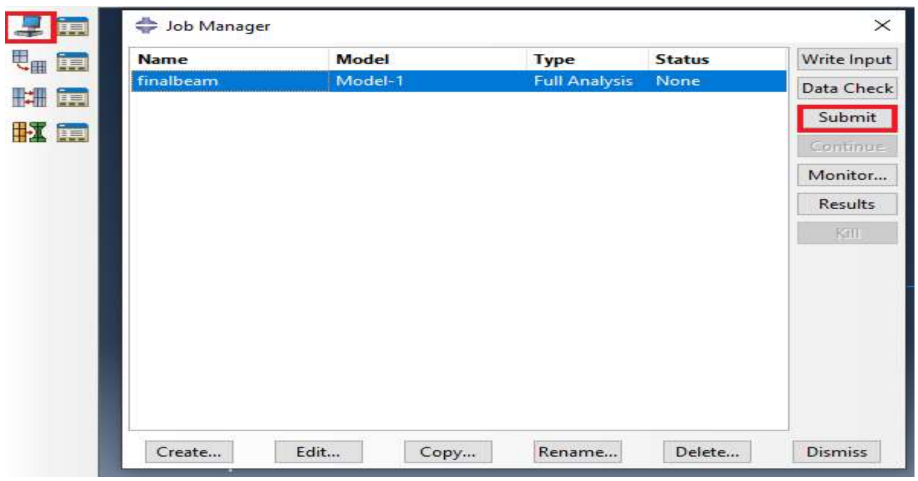
“Create Job” tool select name the job and finally submit

for analysis

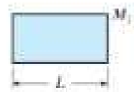
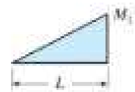
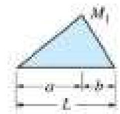
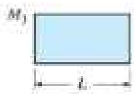
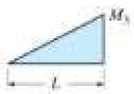
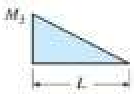
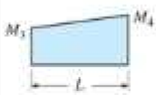
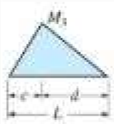
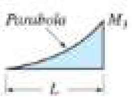


18. Final results are shown after successful analysis.





$$\int_{x=0}^{x=L} \left(M_P M_Q \right) dx$$

M_Q / M_P				
	M_1M_3L	$\frac{1}{2}M_1M_3L$	$\frac{1}{2}(M_1 + M_2)M_3L$	$\frac{1}{2}M_1M_3L$
	$\frac{1}{2}M_1M_3L$	$\frac{1}{3}M_1M_3L$	$\frac{1}{6}(M_1 + 2M_2)M_3L$	$\frac{1}{6}M_1M_3(L + a)$
	$\frac{1}{2}M_1M_3L$	$\frac{1}{6}M_1M_3L$	$\frac{1}{6}(2M_1 + M_2)M_3L$	$\frac{1}{6}M_1M_3(L + b)$
	$\frac{1}{2}M_1(M_3 + M_4)L$	$\frac{1}{6}M_1(M_3 + 2M_4)L$	$\frac{1}{6}M_1(2M_3 + M_4)L$ $+ \frac{1}{6}M_2(M_3 + 2M_4)L$	$\frac{1}{6}M_1M_3(L + b)$ $+ \frac{1}{6}M_1M_4(L + a)$
	$\frac{1}{2}M_1M_3L$	$\frac{1}{6}M_1M_3(L + c)$	$\frac{1}{6}M_1M_3(L + d)$ $+ \frac{1}{6}M_2M_3(L + c)$	for $c \leq a$: $\left(\frac{1}{3} - \frac{(a-c)^2}{6ad}\right)M_1M_3L$
	$\frac{2}{3}M_1M_3L$	$\frac{1}{3}M_1M_3L$	$\frac{1}{3}(M_1 + M_2)M_3L$	$\frac{1}{3}M_1M_3\left(L + \frac{ab}{L}\right)$
	$\frac{1}{3}M_1M_3L$	$\frac{1}{4}M_1M_3L$	$\frac{1}{12}(M_1 + 3M_2)M_3L$	$\frac{1}{12}M_1M_3\left(3a + \frac{a^2}{L}\right)$

Blank Page