Received MAY 4 2018; | Revised MAY 4 2018; | Accepted MAY 4 2018

DOI: xxx/xxxx

CLASS REPORT

Finite Element Analysis on one dimensional steel bar

Parth Thekdi[1] | Vinit Jagdale[2]

[1]Master of Science in Mechanical Engineering, UNCC, NC, USA
[2]Master of Science in Mechanical Engineering, UNCC, NC, USA

Summary

This study consider 1-D bar which is fixed at one end and the stress is provided at the other hand, using Finite Element Method and MATLAB one can find displacement and stress at each node of the 1-D bar. Finite Element Method divide any part in to finite number of nodes, depends on geometry the shape of the element can decide. In Finite Element Method governing equation need to deiced based on the problem and then convert strong form of the equation to the weak form by using weighting function and boundary condition. Using Galerkin approximation Method stiffness, force and mass matrix can form and Gauss quadrature integration method is used for the integration. Using this matrix we can do static, modal and dynamic analysis on the bar.

KEYWORDS:
Finite Element Method, Modal Analysis, Dynamic analysis, Galerkin Approach, Gauss quadrature integration.

1 System Description

The system is 1-D bar which is fixed at one end and other end is subjected to the stress.

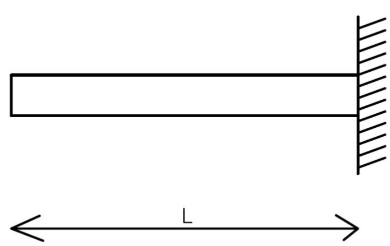

FIGURE 1 1-D bar fixed at one end

2 Intro to Finite Element Method

2.1 Governing Equation

$$\frac{d}{dx}(EA\frac{\partial u}{\partial x}) + Ab = 0 \tag{1}$$

Where E = Young's Modulus in N/m^2
A = cross-section area of bar in m^2
u = Displacement in m
b= body force in N
Left Boundary condition. Stress is prescribed

$$\sigma(0)A(0) = q_0 \tag{2}$$

Right Boundary condition. Displacement is prescribed

$$u(L) = g \tag{3}$$

2.2 Weak formulation

$$\int_0^L w' EAu' dx = \int_0^L W Abdx + w(0)q_o \tag{4}$$

Where, w is weight function depends on boundary condition, L is length of the bar in m.

2.3 Galerikn Approximation for forming matrix

Stiffness matrix

$$K_{ab} = \int_0^L N'_a EAN'_b dx \tag{5}$$

Force Vector

$$F_a = \int_0^L N_a q dx \tag{6}$$

Mass Matrix

$$M_{ab} = \int_0^L N_a \rho A N_b dx \tag{7}$$

Where ρ is density of the bar

2.4 Master element

To increase speed of computational we make one element master element and map them to other elements.
here, $x(\xi)$ ⇒x(-1)=x^e_1 ,$x(+1) = x^e_2$
N_a,N_b are linear shape functions

$$N_a = \frac{1 - \xi_1}{2} \tag{8}$$

$$N_b = \frac{1 + \xi_1}{2} \tag{9}$$

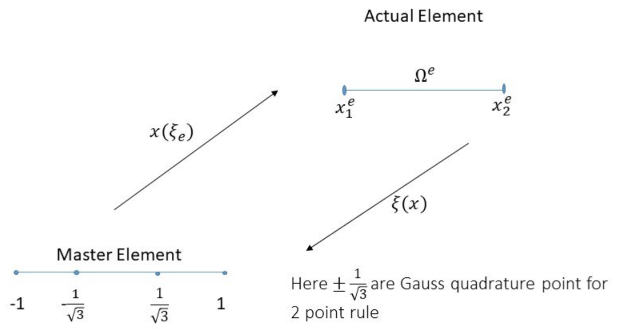

FIGURE 2 Mapping between master and actual element

2.5 Gauss quadrature

In numerical analysis, a quadrature rule is an approximation of the definite integral of a function, usually stated as a weighted sum of function values at specified points within the domain of integration.

$$I_{app} = \sum_{i=1}^{n} w_i f_i \tag{10}$$

1 Point rule - Weights w(1) = 2 integration point $\xi = 0$
2 Point rule- Weights w(1)= 1, W(2)=1 integration point $\xi = \pm\frac{1}{\sqrt{3}}$

2.6 Notation used for calculation

Notation	Name	unit
E	Young' s modulus	N/m^2
h_e	Element Length	m
q	distribution load	N/m
q_0	stress at left end	MPa
A	Cross-section Area	m^2
K	stiffness matrix	N/m
F	Force Vector	N
d	displacement	m
M	Mass matrix	kg
ρ	Mass density	kg/m^3
ω	Natural frequency	rad/s
σ	Stress	N/m^2
ϵ	strain	-

3 Static Analysis

3.1 Formation of K matrix

Here is a Element stiffness matrix

$$k^e = k_a^e \begin{vmatrix} 1 & -1 \\ -1 & 1 \end{vmatrix}$$

Where

$$k_a^e = \frac{2}{h_e}\Big[\sum_{l=1}^{n_{int}} w_l \Big(\sum_{i=1}^{n_{en}} E_i^e N_i \epsilon_l^{int}\Big)\Big(\sum_{i=1}^{n_{en}} A_i^e N_i \epsilon_l^{int}\Big) N_a(\epsilon_l)\Big] \tag{11}$$

Here stiffness matrix is obtain at the integration point using numerical integration.

Here we are doing 2 point Integration so $n_{int} = 2$ and for linear element $n_{el} = 2$

Shape matrix for Gauss quadrature.

$$N_i = \begin{vmatrix} \frac{1-\xi_1}{2} & \frac{1-\xi_2}{2} \\ \frac{1+\xi_1}{2} & \frac{1+\xi_2}{2} \end{vmatrix}$$

Where$\xi_1 = \frac{-1}{\sqrt(3)}$ $\xi_2 = \frac{1}{\sqrt(3)}$ Gauss quadrature points for 2 integration point. w(1)=1 w(2)=1 are weights.

For the assembly of the K matrix

$$K = \begin{bmatrix} \mathrm{k}_{11}^1 & \mathrm{k}_{12}^1 & 0 & 0 & ... & 0 \\ \mathrm{k}_{21}^1 & \mathrm{k}_{22}^1 + k_{11}^2 & k_{12}^2 & 0 & ... & 0 \\ 0 & \mathrm{k}_{21}^2 & \mathrm{k}_{22}^2 + k_{11}^3 & \mathrm{k}_{12}^3 & ... & 0 \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \end{bmatrix}$$

Here k^1 and k^2 are element stiffness matrix.

3.2 Formation of F vector

Here is a Element force matrix

$$f_a^e = \frac{he}{2}\Big[\sum_{l=1}^{n_{int}} w_l \Big(\sum_{i=1}^{n_{en}} q_i^e N_i \epsilon_l^{int}\Big)\Big(\sum_{i=1}^{n_{en}} A_i^e N_i \epsilon_l^{int}\Big) N_a(\epsilon_l)\Big] \tag{12}$$

Here force vector is obtain at the integration point using numerical integration. Assembly of the F vector

$$F = \begin{bmatrix} \mathrm{f}_1^1 \\ \mathrm{f}_2^1 + f_1^2 \\ \mathrm{f}_2^2 + f_1^3 \\ \vdots \end{bmatrix}$$

Here f^1 and f^2 are element stiffness matrix.

3.3 Displacement

Displacement at each node can solve using global stiffness matrix and global force vector

$$Kd = F,$$

$$d = K^{-1}F \tag{13}$$

3.4 Result of Static Analysis

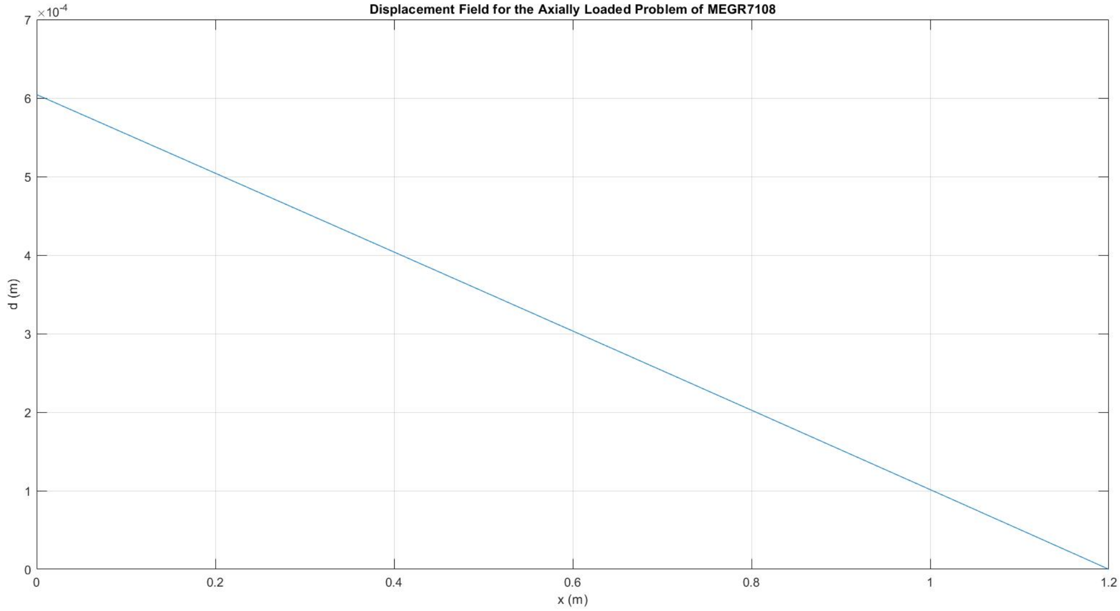

FIGURE 3 static Displacement Field for variable properties

Here We took $E = 200 * 10^9 + 100 * 10^9 x$ N/m^2,A=0.06 + 0.01x m^2, Length=1.2 m, Left BC (Stress)= $100 * 10^6$ MPa, Right BC (Displacement)= 0 m, Distributed load q=$10^6 + 5 * 10^5 x$ N/m

Where x is nodal point of the 1-D bar.

Here Stress is applied at left end so displacement is more at left end and keep decreasing while moving towards fix end.

4 Modal Analysis

4.1 Overview

Modal Analysis use stiffness and Mass matrix calculated at integration point and use this to find natural frequency and mode shape of the system.Stiffness matrix is same as as a static analysis and mass matrix is obtain at the integration point using numerical integration.

4.2 Formation of Lumped Mass matrix

Here is a Element Lumped Mass matrix

$$m^e = m_{ab}\begin{vmatrix} 1 & 0 \\ 0 & 1 \end{vmatrix}$$

$$m_{ab} = \frac{he}{2}\Big[\sum_{l=1}^{n_{int}} w_l \Big(\sum_{i=1}^{n_{en}} \rho_i^e N_i \epsilon_l^{int}\Big)\Big(\sum_{i=1}^{n_{en}} A_i^e N_i \epsilon_l^{int}\Big) N_a(\epsilon_l)\Big] \tag{14}$$

Interpolation of the Density and Area at integration point

For the assembly of the K matrix

$$M = \begin{bmatrix} \mathrm{m}_{11}^1 & 0 & 0 & 0 \; ... \; 0 \\ 0 & \mathrm{m}_{22}^1 + m_{11}^2 & 0 & 0 \; ... \; 0 \\ 0 & 0 & \mathrm{m}_{22}^2 + m_{11}^3 & 0 \; ... \; 0 \\ \vdots & \vdots & \vdots & \vdots \;\; \vdots \;\; \vdots \end{bmatrix}$$

4.3 Natural Frequency

For Modal analysis main governing equation is

$$M\ddot{d} + k\dot{d} = 0 \tag{15}$$

Let assume displacement as

$$d(t) = U sin(\omega t) \tag{16}$$

using equation 15 , 16 we get

$$(K - \omega^2 M)U = 0 \tag{17}$$

From Using MATLAB command [U eigen]=eig(M,K) can obtain Natural frequency ω^2.

Now, Natural Frequency = $\sqrt{\omega^2}$ in rad/s and U is mode shape vector.

4.4 Exact Solution

For our problem governing equation is

$$\frac{\partial^2 u}{\partial x^2} = \frac{1}{C_b^2}\frac{\partial^2 u}{\partial x^2} \tag{18}$$

where the bar-speed C_b is given by

$$C_b = \sqrt{\frac{E}{\rho}}(m/s) \tag{19}$$

Let us consider solution of the form

$$u(x,t) = A(x)sin(\omega t) \tag{20}$$

By substituting the above in the governing equation , we get that

$A,_{xx}(x) -^2 A(x) = 0$ where $= \frac{\omega}{C_b}$

$$A(x) = c_1 cos(x) + c_2 sin(x) \tag{21}$$

c_1 and c_2 are constants

with the boundary condition $u(L,t) = 0$ AND $u, x(0,t)$

we get values of λ_n are the modes of free vibration of the bar and the mode shapes given by

$$A_n(x) = cos\lambda_n x \tag{22}$$

4.5 Result of Modal Analysis

mode_no	Natural_frequency
1	6.6276e+03
2	1.9865e+04
3	3.3047e+04
4	4.6139e+04
5	5.9105e+04
6	7.1908e+04
7	8.4515e+04
8	9.6890e+04
9	1.0900e+05
10	1.2081e+05

FIGURE 4 First 10 Natural Frequency

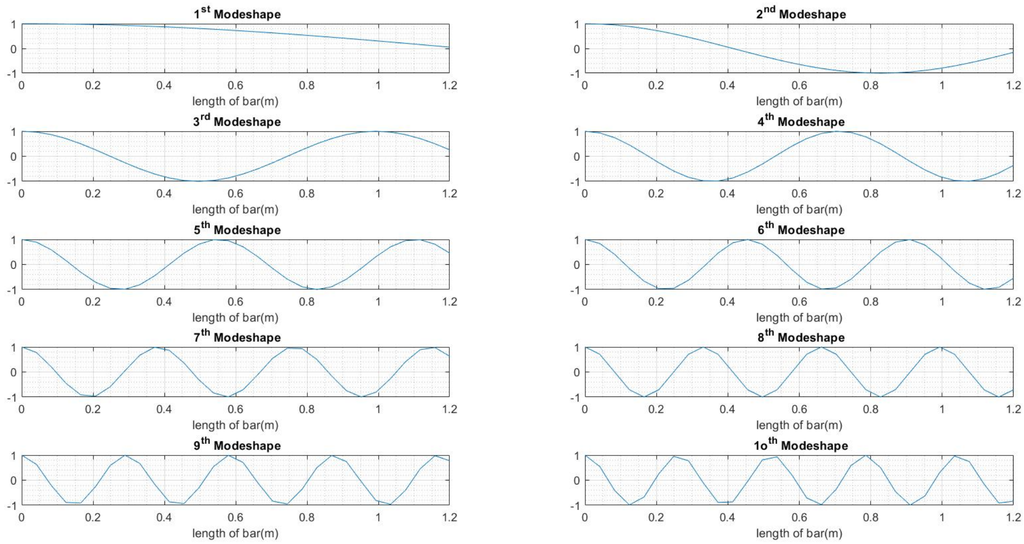

FIGURE 5 First 10 mode shape

From the figure, we can see that first 4-5 mode shapes have smooth curve. But for further modes, curve is not smooth. This is because, we are plotting this shapes for 30 elements only. Therefore, we do not have sufficient points to plot the curve for higher number of modes. If we increase the number of elements, we will get the smooth curve.

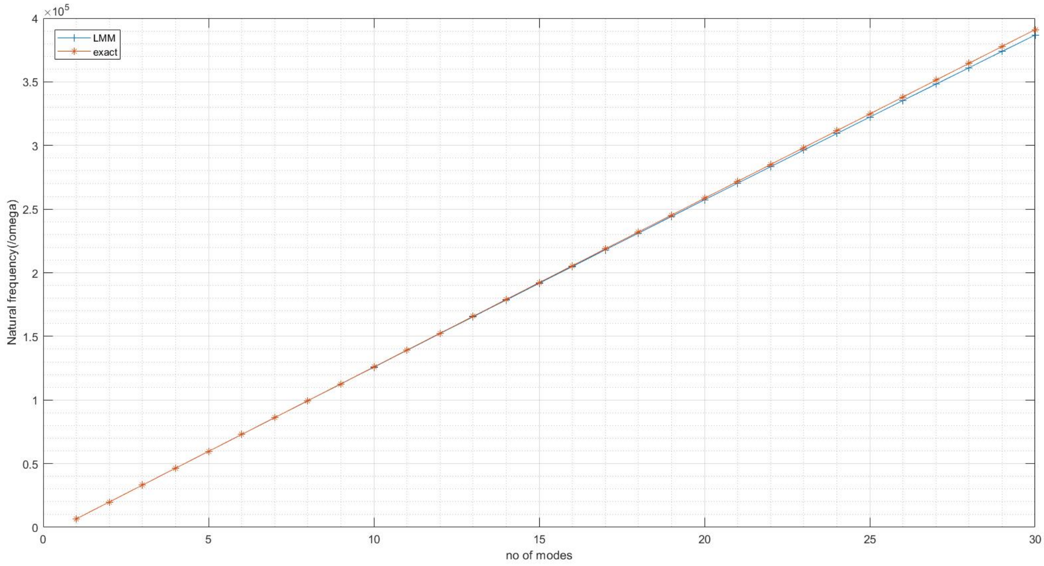

FIGURE 6 Comparison of Natural frequency For 30 Elements

If the user asks for first x number of natural frequencies, code will run for 7*x number of elements. This way the accuracy for number of frequencies user asked for will increase. From the plot, we can see that the error between the frequencies calculated by exact and FEM method are very close to each other.

5 Dynamic Analysis

5.1 Time Step

In this part of the project we are using an Explicit Method to solve for displacement and stress at different time.

Condition for the critical time

$$\frac{c_b \Delta t}{h} \leq \frac{1}{\sqrt(3)} \tag{23}$$

Here h is the smallest element length. Here we choose Δt as

$$\Delta t = \frac{h}{\sqrt{3} c_b} \tag{24}$$

c_b is wave speed

$$c_b = \sqrt{\frac{E}{\rho}} \tag{25}$$

5.2 Solving for displacement for all times

Let $a = \ddot{d}$ and $v = \dot{d}$.

So need to solve for explicit method of system of equation

$$M a_{n+1} + K d_{n+1} = F_{n+1}, \tag{26}$$

For every time interval F_{n+1} can find using body force and BC's so for acceleration

$$a = M^{-1}[F_{n+1} - K d_{n+1}] \tag{27}$$

Displacement at every time interval

$$d_{n+1} = d_n + (\Delta t) v_n + \frac{(\Delta t)^2}{2} a_n \tag{28}$$

For Velocity at every time interval

$$v_{n+1} = v_n + \frac{\Delta t}{2}[a_n + a_{n+1}] \tag{29}$$

5.3 Strain and Stress Calculation

Strain is calculated at element level based on nodal displacement of each linear element.This is linear interpolation of the displacement.

$$u_e^h(x) = N_e(x) * d_e + N_{e+1}(x) * d_{e+1} \tag{30}$$

strain is the slope of the displacement over length.

$$\epsilon = \frac{du}{dx}$$

For master element

$$u_\xi^h = \sum_{a=1}^{2} N_a(\xi) d_a^e$$

Strain

$$\epsilon = \frac{du}{d\xi}(x, \xi)^{-1}$$

$$\epsilon = \frac{2}{h_e}\left(\frac{d_{i+1} - d_i}{2}\right)$$

$$Strain(\epsilon) = \frac{d_{i+1} - d_i}{h_e} \tag{31}$$

where h_e is element length, i is global node number of element.

From the Hook's law ,

$$Stress(\sigma) = E * \epsilon \tag{32}$$

5.4 Input Data for the Dynamic Problem

E	200e9 PA
A	$6e-4m^2$
ρ	7800 kg/m^3
L	1.2 m
bias	1.0
u(x,0)	0
$\dot{u}(x,0)$	0
$q_0(t)$	$\Sigma[H(t)-H(t-t_c)]$

where $\Sigma = 100$ MPa
Here the H is Heaviside function which gives q as step function so on the left side of bar stress is prescribed as step function. Body force consider to be zero. so q_0 stress is only component for the global force vector.

5.5 Other Parameters calculated by code

Wave speed in m/s

$$c_b = \sqrt{\frac{E}{\rho}}, c_b = 5063.69m/s \tag{33}$$

Period

$$\tau = \frac{L}{c_b} \tag{34}$$

Critical Time in sec

$$t_c = \frac{\tau}{10} \tag{35}$$

Maximum time of interest

$$\tau_{max} = 3 * \tau \tag{36}$$

Total number of time increments

$$nt = \frac{\tau_{max}}{\Delta t} \tag{37}$$

5.6 Result

5.6.1 Displacement

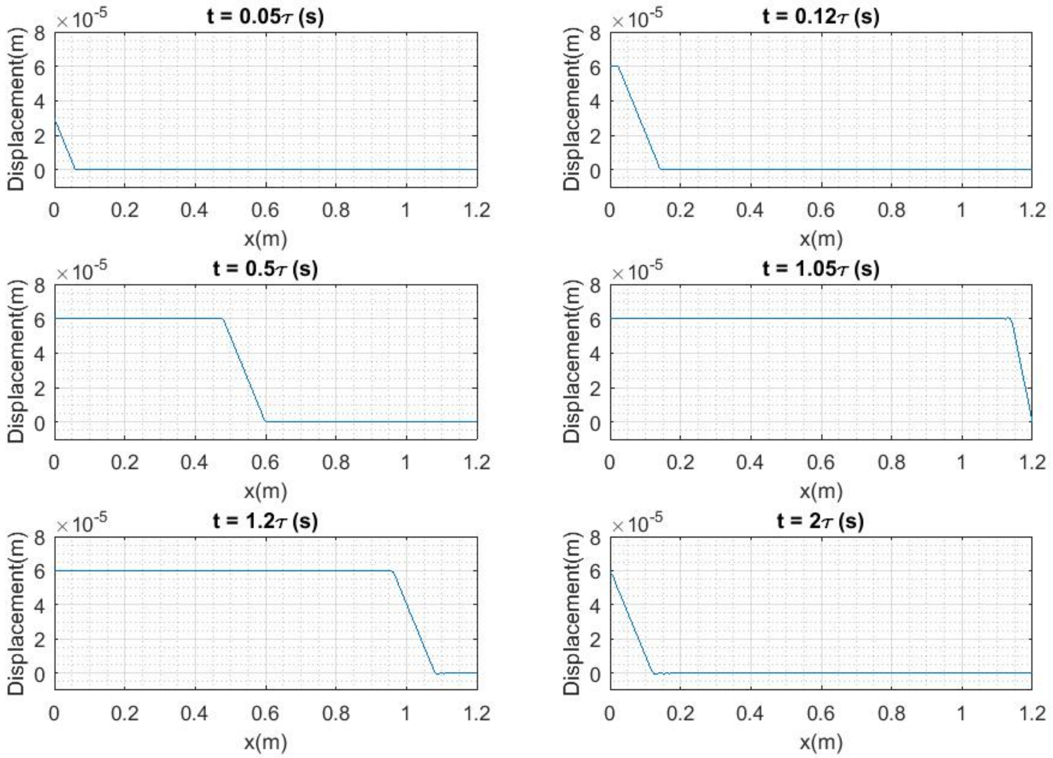

FIGURE 7 Displacement at given time

From the animation, displacement wave is propagating around the amplitude $6 * 10^-5$ meter up to the fixed end of the bar. Then, after reflecting from the fixed end, it propagates around 0 meter up to the free end of the bar. Again, it starts propagating in the opposite direction i.e. around $10 * 10^-6$ meter up to fixed end. This cycle continues until maximum number of time increment is achieved.

5.6.2 Stress

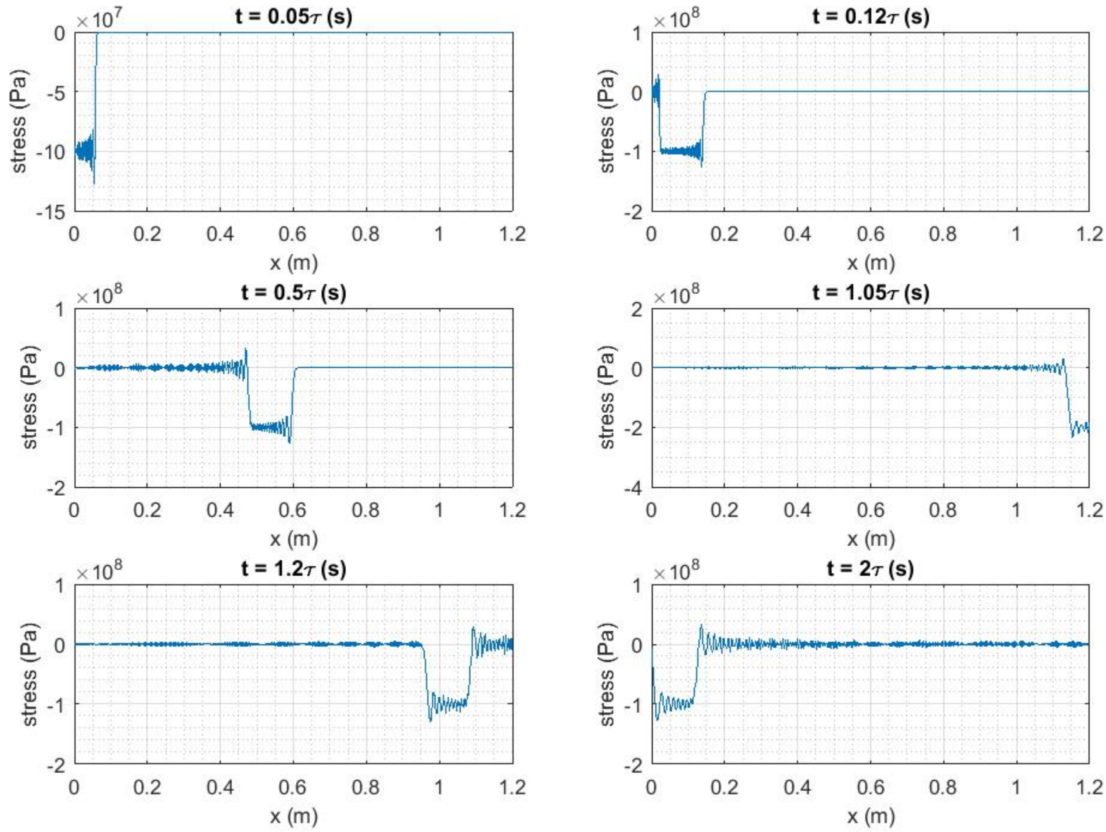

FIGURE 8 Stress at given time

From the animation, stress is propagating around the amplitude -100 MPa as step wave up to time of τ sec, after reflecting from the fixed end of the bar stress continue the same path, it start propagating in the opposite direction after 2τ sec. This result correlate the result of displacement which is propagating in opposite direction after 2τ sec.

5.6.3 Nodal displacement

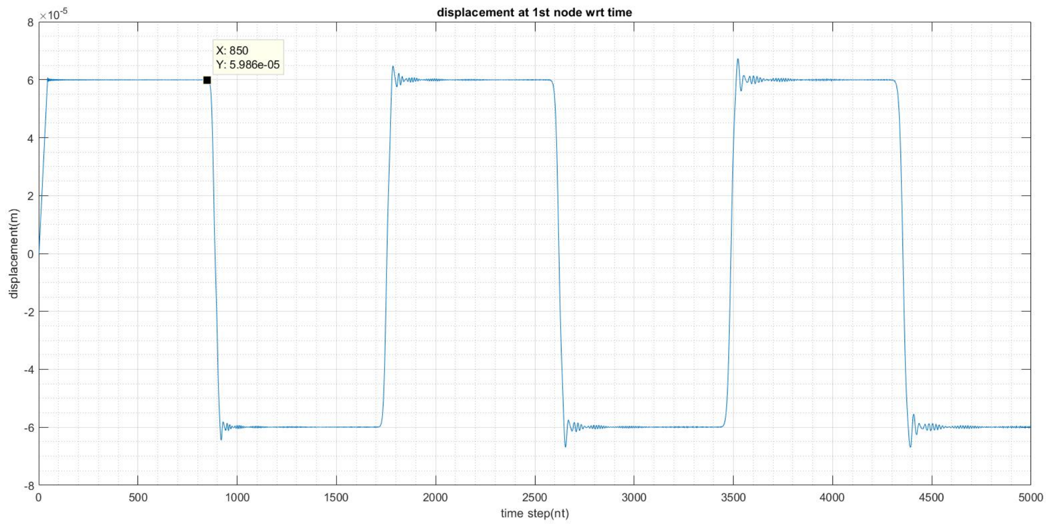

FIGURE 9 Displacement of 1^{st} node at all the time

The above plot shows displacement variation at the 1s node of the bar with respect to time increments. In the plot, from zero to 850^{th} time increment, the distance is $2 * \tau$ i.e. total time required for the wave to propagate from free boundary to fixed boundary and back to free boundary. We can see that after completion of every $2 * \tau$, the displacement amplitude keeps increasing little bit.

5.7 Verification of dynamic Analysis

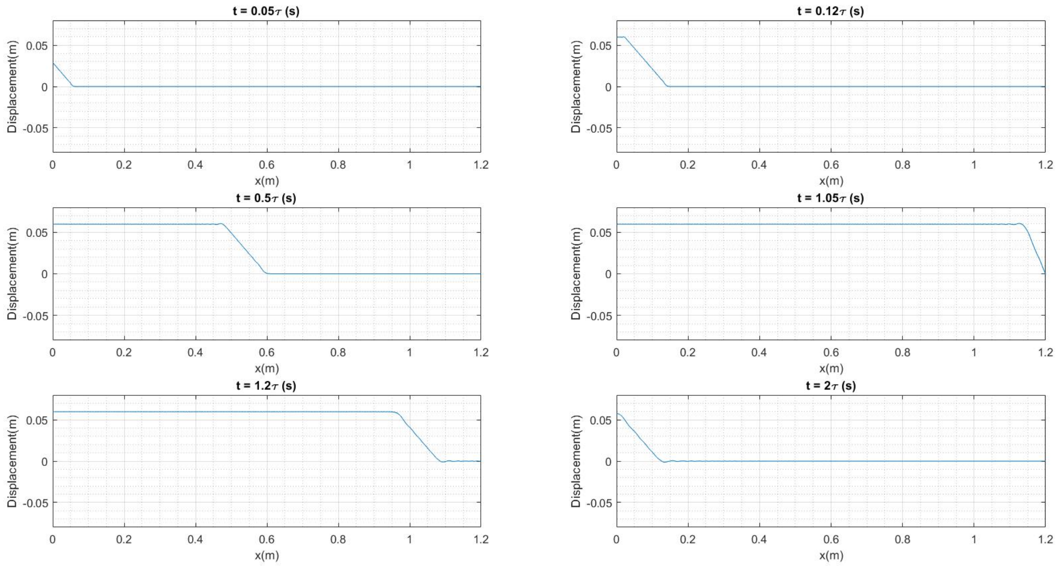

FIGURE 10 Displacement when stress applied $100 * 10^9$ MPa

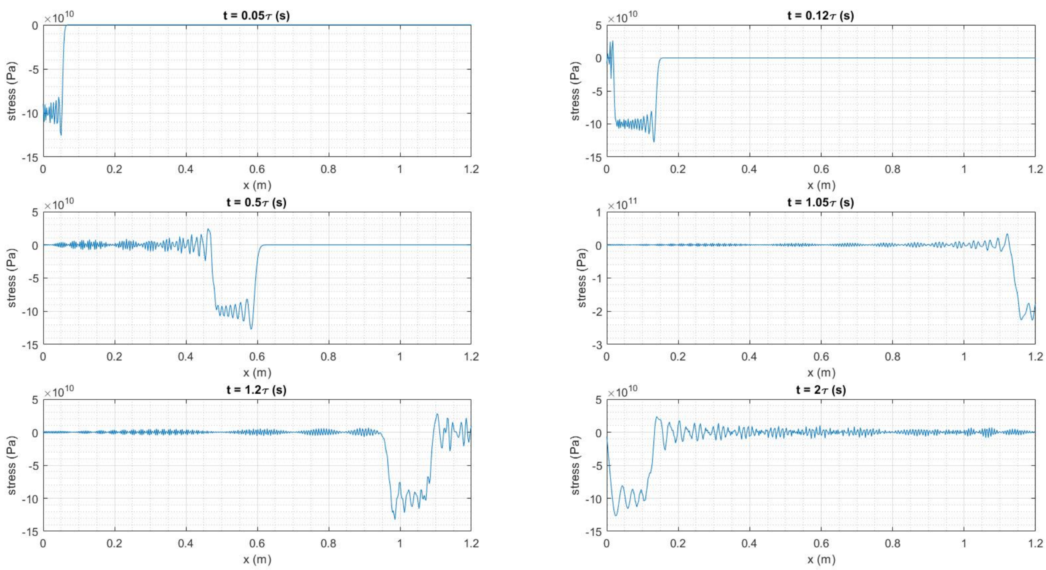

FIGURE 11 Stress when stress applied $100 * 10^9$ MPa

From figure, as we increase stress applied at left boundary, i.e. 100e9, keeping body force zero, we see the hike in the displacement plot as well as stress plot at given times. Hence, we can say that this code is running accurately when we keep body force zero and increase the stress at left boundary.

6 Main feature in MATLAB Code

Wrapperm.m is input file for modal analysis and wrapperdynamic.m is as input file for dynamic analysis.

To get accurate frequency we took element number be 7 time more than requested frequency.

Our code can work for both variable and constant property.

To use as constant value part.E =@(x) 200e9 + 0.*x, and to make variable property with x (nodal value) one assign value like part.E=@(x) 200e9 +100e9.*x.

Result provided over here is for dynamic analysis with zero body force which is $q_o = 0 + 0 * x + 0 * t$ to include body force user can change body force as function of x and t, i.e $5 + 100 * x + 500 * t$.

□