International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 4 Apr. 2021, pp: 429-459 www.ijaem.net ISSN: 2395-5252

A Finite Element Model for the Analysis of Rigid-Jointed Plane Truss Structures

[1]Engr. A. I. Ebu and [2]Engr. Dr. M. E. Onyia
[1,2] Department Of Civil Engineering, University Of Nigeria, Nsukka.

Submitted: 15-04-2021 Revised: 28-04-2021 Accepted: 30-04-2021

ABSTRACT: It has become a norm to perform the analysis of plane truss structures based on the assumptions that (1) members are connected at joints by frictionless pins and (2) loads are applied at joints only. However, it is a fact that practical trusses are always constructed by connecting members to gusset plates using welds, rivets, or high-strength bolts and loads may not necessarily be applied only at the nodes. By the nature of practical nodes connections, joints in trusses are rigid and not frictionless pins and as such analyzing plane trusses as rigidly connected nodes yields more precise results. This work proposes a finite element model for plane truss analysis with rigid joints as compared to frictionless pin joints in both static and dynamic (modal) forms. Computer programs using MATLAB codes were developed for the analysis and the commercial software ROBOT Structural Analysis Software was used to cross check results obtained using the developed MATLAB solution. The results showed that trusses where pin joints were assumed resonate faster than those where rigid joints were assumed, and axial stresses were found to be smaller in members with rigid joints than those with pin joints.

Keywords: Finite element analysis, Trusses, MATLAB computer program, ROBOT software

I. INTRODUCTION

Fundamentally, the behavior of all types of structures – framework, plates, shells or trusses is described by means of differential equations. In practice, the writing of differential equations for truss structures is rarely necessary. It has been long established that such structures may be treated as assemblage of one-dimensional members. Exact or approximate solutions to the ordinary differential equations for each member are well-known. These solutions can be cast in the form of relationships between the forces and the displacements at the ends of the members. Proper combinations of these relationships with the equations of equilibrium and compatibility at the joints and supports yield a system of algebraic equations that describes the behavior of the structure.

Structures consisting of two- or three-dimensional components- plates, trusses, membrane shells, solids are more complicated in that rarely do exact solutions exist for applicable partial differential equations as said before. One approach to obtaining practical, numerical solutions is the finite element method. The basic concept of the method is that a continuum (the total structure) can be modeled analytically by its subdivision into regions (the finite elements) [20], 'each of the behavior is described by a set of assumed functions representing the stresses or displacements in that region'. This permits the problem formulation to be altered to one of the establishment of a system of algebraic equations.

Therefore, the high speed precise computing and increased memory of the computers have made it possible to solve complex models. Finite element method and matrix methods are the two methods which show great compatibility for computing processes and have become the most powerful tools in many engineering branches. Therefore, analysis that was considered cumbersome and consequently avoided can now be carried out easily using finite element method which has a great potential of being easily programmed especially with such computing language like MATLAB (used in this work) that has codes which can easily manipulate and solve mathematical problems. This has made actual structures like plane trusses to be analyzed based on its true service conditions (nodes connections and loadings) without assuming conditions (frictionless pins connection and loads only acting at the joints) for the purpose of simplifying the analysis that this can lead to results that are not precise or uneconomical in designs.

This work, analysis of rigid-jointed plane truss structures using finite element analysis technique tries to understudy analysis of plane trusses using the assumptions of frictionless pins joints and loads acting only at the joints to develop

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 4 Apr. 2021, pp: 429-459 www.ijaem.net ISSN: 2395-5252

plane truss analysis with rigid joints and no restriction of loading conditions. It modeled rigid jointed plane truss as plane frames, then uses stiffness method to formulate the stiffness and mass matrices. Finite element interpolation functions for truss element formulation was used to obtain formula for strain and stresses and equivalent nodal loads in the case of uniformly distributed loads and point loads. The work was carried out in two phases namely: (1) static analysis and (2) modal analysis.

1.2 STATIC ANALYSIS

Static analysis of plane truss structures using finite element analysis technique is achieved by converting nodal coordinates of the truss structure from local coordinates to global coordinates system [2]. The stiffness matrix of individual members were calculated and then transformed into the global stiffness matrix using conversions outlined in the next section for both frictionless pin joints and rigid joints. The elements by elements global stiffness matrix are then assembled into the structure stiffness matrix by direct combination procedure[8]. This was then followed by providing constraints to the finite element equation of the plane truss problem

KU = F 1.1

Where K is the stiffness matrix, U is the displacement vector and F is the force vector.

Displacements are first sort as the primary variables and the values are substituted back into the equation1.1 to find secondary variables such as supports reactions, strains and stresses. The procedure is the same for both assumptions of pin and rigid joints, only that in addition to axial stresses bending stresses were also evaluated in the case of rigid joints.

1.3 MODAL ANALYSIS

Modal analysis is the study of the dynamic properties of structures under vibrational excitation [24].When a structure undergoes an external excitation, its dynamic responses are measured and analyzed. This process of measuring and analyzing is called modal analysis. Modal analysis can be used to measure the response of a car's body to a vibration when vibration of an electromagnetic shaker is attached or the pattern created by noise of a loudspeaker which acts as excitation [27]

In structural engineering, modal analysis is applied to find the various periods that the structure will naturally resonate at, by using the structural overall mass and stiffness. The modal analysis is very important in earthquake engineering, because the periods of vibration evaluated helps in checking that a building's natural frequency does not coincide with the frequency of earthquakes prone region where the building is to be constructed. In case a structure's natural frequency coincidentally equals an earthquake's frequency, the structure suffers severe structural damage due to resonance [36]

The frequency and mode shape of a model is determined by modal analysis. When the models are subjected to cyclic or vibration loads, the dynamic response of structures due to these external loads acting, which include resonance frequencies (natural frequencies), mode shape and damping,are estimated.

2.1 FORMULATION OF FLEXURE BEAM ELEMENT

Using the elementary beam theory, the 2-D beam or flexure element is now developed with the aid of the first theorem of Castigliano. The assumptions and restrictions underlying the development are the same as those of elementary beam theory with the addition of

1. The element is of length L and has two nodes, one at each end.
2. The element is connected to other elements only at the nodes.
3. Element loading occurs only at the nodes.

Recalling that the basic premise of finite element formulation is to express the continuously varying field variable in terms of a finite number of values evaluated at element nodes, we note that, for the flexure element, the field variable of interest is the transverse displacement v(x) of the neutral surface away from its straight, undeflected position. As depicted in Figure 4.3a and 4.3b, transverse deflection of a beam is such that the variation of deflection along the length is not adequately described by displacement of the end points only. The end deflections can be identical, as illustrated, while the deflected shape of the two cases is quite different. Therefore, the flexure element formulation must take into account the slope (rotation) of the beam as well as end-point displacement. In addition to avoiding the potential ambiguity of displacements, inclusion of beam elements, thus precluding the physically unacceptable discontinuity depicted in Figure 3.1c.

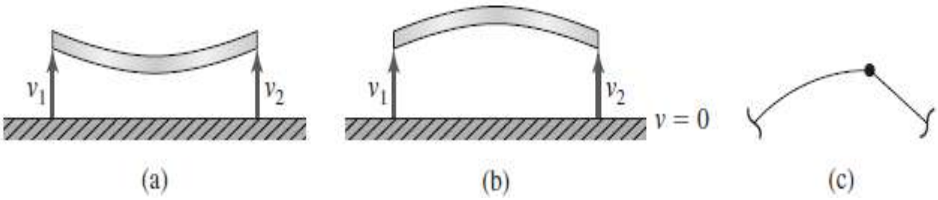


Figure 2.1 (a) and (b) Beam elements with identical end deflections but quite different deflection characteristics. (c) Physically unacceptable discontinuity at the connecting node.

In light of these observations regarding rotations, the nodal variables to be associated with a flexure element are as depicted in Figure 3.2. Element nodes 1 and 2 are located at the ends of the element, and the nodal variables are the transverse displacements v1 and v2 at the nodes and the slopes (rotations) _1 and _2. The nodal variables as shown are in the positive direction, and it is to be noted that the slopes are to be specified in radians. For convenience, the superscript (e) indicating element properties is not used at this point, as it is understood in context that the current discussion applies to a single element. When multiple elements are involved in examples to follow, the superscript notation is restored.

The displacement function v(x) is to be discretized such that nodal displacement

$$v(x) = f(v_1, v_2, \theta_1, \theta_2, x) \qquad \text{2.1}$$

subject to the boundary conditions

$$v(x = x_1) = v_1 \qquad \text{2.2}$$

$$v(x = x_2) = v_2 \qquad \text{2.3}$$

$$\left.\frac{dv}{dx}\right|_{x=x_1} = \theta_1 \qquad \text{2.4}$$

$$\left.\frac{dv}{dx}\right|_{x=x_2} = \theta_2 \qquad \text{2.5}$$

Considering the four boundary conditions and the one-dimensional nature ofthe problem in terms of the independent variable, we assume the displacement function in the form

$$v(x) = a_0 + a_1 x + a_2 x^2 + a_3 x^3 \qquad \text{2.6}$$

Application of the boundary conditions 3.2–3.5 in succession yields

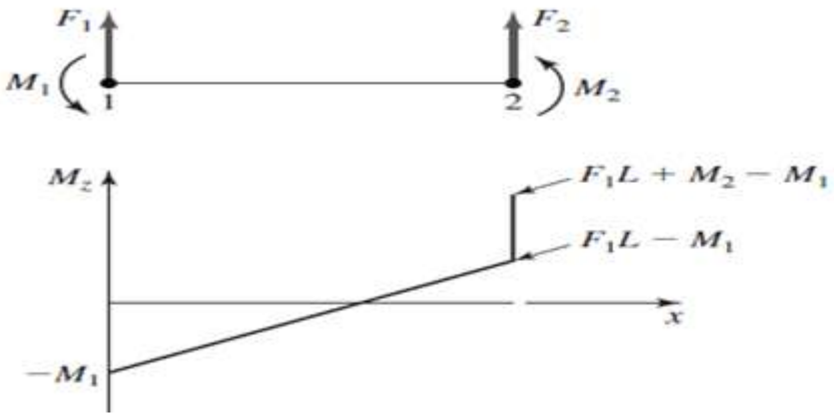
$$v(x = 0) = v_1 = a_0 \qquad \text{2.7}$$

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 4 Apr. 2021, pp: 429-459 www.ijaem.net ISSN: 2395-5252

$$v(x = L) = v_2 = a_0 + a_1 L + a_2 L^2 + a_3 L^3$$

$$\left.\frac{dv}{dx}\right|_{x=0} = \theta_1 = a_1 \quad ....2.8$$

$$\left.\frac{dv}{dx}\right|_{x=L} = \theta_2 = a_1 + 2a_2 L + 3a_3 L^2 \quad ....2.9$$



………2.10

Figure 2.2Bending moment diagram for a flexure element. Sign convention per the strength of materials theory.

Equations 3.7–3.10 are solved simultaneously to obtain the coefficients in terms of the nodal variables as

$$a_0 = v_1 \quad . \; . \; . \; . \; . \; . \; 2.11$$

$$a_1 = \theta_1 \quad . \; . \; . \; . \; . \; . \; 2.12$$

$$a_2 = \frac{3}{L^2}(v_2 - v_1) - \frac{1}{L}(2\theta_1 + \theta_2) \quad . \; . \; . \; . \; . \; . \; 2.13$$

$$a_3 = \frac{2}{L^3}(v_1 - v_2) + \frac{1}{L^2}(\theta_1 + \theta_2) \quad ………………………………2.14$$

Substituting Equations 3.11–3.14 into Equation 4.17 and collecting the coefficients of the nodal variables results in the expression:

$$v(x) = \left(1 - \frac{3x^2}{L^2} + \frac{2x^3}{L^3}\right)v_1 + \left(x - \frac{2x^2}{L} + \frac{x^3}{L^2}\right)\theta_1 + \left(\frac{3x^2}{L^2} - \frac{2x^3}{L^3}\right)v_2 + \left(\frac{x^3}{L^2} - \frac{x^2}{L}\right)\theta_2 \quad . \; . \; . \; . \; . \; . \; 2.15$$

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 4 Apr. 2021, pp: 429-459 www.ijaem.net ISSN: 2395-5252

which is of the form

$$v(x) = N_1(x)v_1 + N_2(x)\theta_1 + N_3(x)v_2 + N_4(x)\theta_2 \qquad \text{. . . .. 2.16}$$

or, in matrix notation,

$$v(x) = [N_1 \quad N_2 \quad N_3 \quad N_4] \begin{Bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{Bmatrix} = [N]\{\delta\} \qquad \text{. . . . . 2.17}$$

whereN1, N2, N3, and N4 are the interpolation functions that describe the distribution of displacement in terms of nodal values in the nodal displacement vector

$\{\delta\}$.

For the flexure element, it is convenient to introduce the dimensionlesslength coordinate

$$\xi = \frac{x}{L} \qquad \text{. . . . . 2.18}$$

so that Equation 3.26 becomes

$$v(x) = (1 - 3\xi^2 + 2\xi^3)v_1 + L(\xi - 2\xi^2 + \xi^3)\theta_1 + (3\xi^2 - 2\xi^3)v_2 + L\xi^2(\xi - 1)\theta_2 \qquad \text{. . . 2.19}$$

Using Equation 2.11 in conjunction with Equation 3.17, the normal stress distribution on a cross section located at axial position x is given by

$$\sigma_x(x, y) = -yE\frac{d^2[N]}{dx^2}\{\delta\} \qquad \text{..........2.20}$$

Since the normal stress varies linearly on a cross section, the maximum and minimum values on any cross section occur at the outer surfaces of the element, where distance y from the neutral surface is largest. As is customary, we take the maximum stress to be the largest tensile (positive) value and the minimum to be the largest compressive (negative) value. Hence, we rewrite Equation 3.30 as

$$\sigma_x(x) = y_{max}E\frac{d^2[N]}{dx^2}\{\delta\} \qquad \text{. . . . . 2.21}$$

$$\sigma_x(x) = y_{max}E\left[\left(\frac{12x}{L^3} - \frac{6}{L^2}\right)v_1 + \left(\frac{6x}{L^2} - \frac{4}{L}\right)\theta_1 + \left(\frac{6}{L^2} - \frac{12x}{L^3}\right)v_2 + \left(\frac{6x}{L^2} - \frac{2}{L}\right)\theta_2\right]$$

. . .3.22

Observing that Equation 3.21 indicates a linear variation of normal stress along the length of the element and since, once the displacement solution is obtained, the nodal values are known constants, we need calculate only the stress values at the cross sections corresponding to the nodes; that is, at x = 0 and x = L. The stress values at the nodal sections are given by

$$\sigma_x(x=0) = y_{\max} E\left[\frac{6}{L^2}(v_2 - v_1) - \frac{2}{L}(2\theta_1 + \theta_2)\right] \quad \text{. . . . .} \quad 2.23$$

$$\sigma_x(x=L) = y_{\max} E\left[\frac{6}{L^2}(v_1 - v_2) + \frac{2}{L}(2\theta_2 + \theta_1)\right] \quad \text{. . . . .} \quad 2.24$$

2.2 FLEXURE ELEMENT STIFFNESS MATRIX

The total strain energy is expressed as

$$U_e = \frac{1}{2}\int_V \sigma_x \varepsilon_x \, \mathrm{d}V \quad \ldots \quad 2.25$$

whereV is total volume of the element. Substituting for the stress and strain per Equations 3.5 and 3.6, into equation 3.25

$$U_e = \frac{E}{2}\int_V y^2 \left(\frac{\mathrm{d}^2 v}{\mathrm{d}x^2}\right)^2 \mathrm{d}V \quad \ldots \quad 2.26$$

which can be written as:

$$U_e = \frac{E}{2}\int_0^L \left(\frac{\mathrm{d}^2 v}{\mathrm{d}x^2}\right)^2 \left(\int_A y^2 \, \mathrm{d}A\right) \mathrm{d}x \quad \ldots \quad 2.27$$

Again recognizing the area integral as the moment of inertia Izabout the centroidal axis perpendicular to the plane of bending, we have

$$U_e = \frac{EI_z}{2}\int_0^L \left(\frac{\mathrm{d}^2 v}{\mathrm{d}x^2}\right)^2 \mathrm{d}x \quad \ldots \quad 2.28$$

Equation 2.38 represents the strain energy of bending for any constant cross-section beam that obeys the assumptions of elementary beam theory. For the strain energy of the finite element being developed, we substitute the discretized displacement relation of Equation 3.27 to obtain

$$U_e = \frac{EI_z}{2}\int_0^L \left(\frac{\mathrm{d}^2 N_1}{\mathrm{d}x^2} v_1 + \frac{\mathrm{d}^2 N_2}{\mathrm{d}x^2}\theta_1 + \frac{\mathrm{d}^2 N_3}{\mathrm{d}x^2} v_2 + \frac{\mathrm{d}^2 N_4}{\mathrm{d}x^2}\theta_2\right)^2 \mathrm{d}x \quad \ldots 2.29$$

as the approximation to the strain energy.

Applying the first theorem of Castigliano to the strain energy function with respect to nodal displacement v_1 gives the transverse force at node 1 as

$$\frac{\partial U_e}{\partial v_1} = F_1 = EI_z \int_0^L \left(\frac{d^2N_1}{dx^2} v_1 + \frac{d^2N_2}{dx^2} \theta_1 + \frac{d^2N_3}{dx^2} v_2 + \frac{d^2N_4}{dx^2} \theta_2 \right) \frac{d^2N_1}{dx^2} dx$$

..2.30

while application of the theorem with respect to the rotational displacement gives the moment as

$$\frac{\partial U_e}{\partial \theta_1} = M_1 = EI_z \int_0^L \left(\frac{d^2N_1}{dx^2} v_1 + \frac{d^2N_2}{dx^2} \theta_1 + \frac{d^2N_3}{dx^2} v_2 + \frac{d^2N_4}{dx^2} \theta_2 \right) \frac{d^2N_2}{dx^2} dx$$

.2.31

For node 2, the results are

$$\frac{\partial U_e}{\partial v_2} = F_2 = EI_z \int_0^L \left(\frac{d^2N_1}{dx^2} v_1 + \frac{d^2N_2}{dx^2} \theta_1 + \frac{d^2N_3}{dx^2} v_2 + \frac{d^2N_4}{dx^2} \theta_2 \right) \frac{d^2N_3}{dx^2} dx$$

..2.32

$$\frac{\partial U_e}{\partial \theta_2} = M_2 = EI_z \int_0^L \left(\frac{d^2N_1}{dx^2} v_1 + \frac{d^2N_2}{dx^2} \theta_1 + \frac{d^2N_3}{dx^2} v_2 + \frac{d^2N_4}{dx^2} \theta_2 \right) \frac{d^2N_4}{dx^2} dx$$

..2.33

Equations 3.29–3.32 algebraically relate the four nodal displacement values to the four applied nodal forces (here we use force in the general sense to include applied moments) and are of the form

$$\begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} \\ k_{21} & k_{22} & k_{23} & k_{24} \\ k_{31} & k_{32} & k_{33} & k_{34} \\ k_{41} & k_{42} & k_{43} & k_{44} \end{bmatrix} \begin{Bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ M_1 \\ F_2 \\ M_2 \end{Bmatrix}$$

. . . . 2.34

wherekmn, m, n = 1, 4 are the coefficients of the element stiffness matrix. By comparison of Equations 3.40–3.43 with the algebraic equations represented by matrix Eqution 4.44, it is seen that

$$k_{mn} = k_{nm} = EI_z \int_0^L \frac{d^2N_m}{dx^2} \frac{d^2N_n}{dx^2} dx \qquad m, n = 1, 4$$

. . . . 2.35

and the element stiffness matrix is symmetric, as expected for a linearly elastic element.
Prior to computing the stiffness coefficients, it is convenient to convert the integration to the dimensionless length variable ξ = x/L by noting

$$\int_0^L f(x)\, dx = \int_0^1 f(\xi) L\, d\xi$$

. . . . 2.36

$$\frac{d}{dx} = \frac{1}{L} \frac{d}{d\xi}$$

. . … 2.37

so the integrations of Equation 2.36 become

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 4 Apr. 2021, pp: 429-459 www.ijaem.net ISSN: 2395-5252

$$k_{11} = \frac{EI_z}{L^3}\int_0^1 (12\xi - 6)^2\, d\xi = \frac{36EI_z}{L^3}\int_0^1 (4\xi^2 - 4\xi + 1)\, d\xi$$
$$= \frac{36EI_z}{L^3}\left(\frac{4}{3} - 2 + 1\right) = \frac{12EI_z}{L^3}$$
$$k_{12} = k_{21} = \frac{EI_z}{L^3}\int_0^1 (12\xi - 6)(6\xi - 4)L\, d\xi = \frac{6EI_z}{L^2}$$
$$k_{13} = k_{31} = \frac{EI_z}{L^3}\int_0^1 (12\xi - 6)(6 - 12\xi)\, d\xi = -\frac{12EI_z}{L^3} \qquad ..2.38$$
$$k_{14} = k_{41} = \frac{EI_z}{L^3}\int_0^1 (12\xi - 6)(6\xi - 2)L\, d\xi = \frac{6EI_z}{L^2}$$

$$k_{mn} = k_{nm} = EI_z\int_0^L \frac{d^2N_m}{dx^2}\frac{d^2N_n}{dx^2}dx = \frac{EI_z}{L^3}\int_0^1 \frac{d^2N_m}{d\xi^2}\frac{d^2N_n}{d\xi^2}d\xi \qquad m,n = 1,4 \qquad ..\ 2.39$$

The stiffness coefficients are then evaluated as follows:

$$k_{22} = \frac{4EI_z}{L}$$
$$k_{23} = k_{32} = -\frac{6EI_z}{L^2} \qquad 2.40$$
$$k_{24} = k_{42} = \frac{2EI_z}{L}$$
$$k_{33} = \frac{12EI_z}{L^3}$$
$$k_{34} = k_{43} = -\frac{6EI_z}{L^3}$$
$$k_{44} = \frac{4EI_z}{L}$$

The complete stiffness matrix for the flexure element is then written as

$$[k_e] = \frac{EI_z}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \qquad ...\ .....\ 2.41$$

2.3 STIFFNESS MATRIX FOR RIGID JOINTED PLANE TRUSS MEMBER

Due to fixity of the joints, this class of element is modeled as a flexure element with axial loading as shown on fig.3.4. It is seen that in addition to the nodal transverse deflections and rotations, there are displacements in the nodes. This means that, the total degrees of freedom for each element are six with each node having three global degrees of freedom, two displacements in global axis and one rotation.

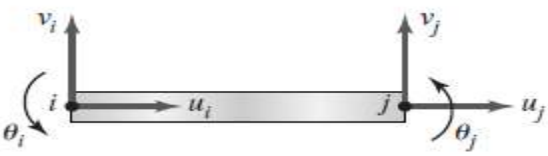


Figure 2.3:Nodal displacements of rigid jointed plane truss member

This being the case, we can simply add the spartial element stiffness matrix to the flexure element stiffness matrix to obtain the 6 × 6 element stiffness matrix for a rigid jointed plane truss element as follows:

$$[k_e] = \begin{bmatrix} \frac{AE}{L} & \frac{-AE}{L} & 0 & 0 & 0 & 0 \\ \frac{-AE}{L} & \frac{AE}{L} & 0 & 0 & 0 & 0 \\ 0 & 0 & \frac{12EI_z}{L^3} & \frac{6EI_z}{L^2} & \frac{-12EI_z}{L^3} & \frac{6EI_z}{L^2} \\ 0 & 0 & \frac{6EI_z}{L^2} & \frac{4EI_z}{L} & \frac{-6EI_z}{L^2} & \frac{2EI_z}{L} \\ 0 & 0 & \frac{-12EI_z}{L^3} & \frac{-6EI_z}{L^2} & \frac{12EI_z}{L^3} & \frac{-6EI_z}{L^2} \\ 0 & 0 & \frac{6EI_z}{L^2} & \frac{2EI_z}{L} & \frac{-6EI_z}{L^2} & \frac{4EI_z}{L} \end{bmatrix}$$ **2.42**

which is seen to be simply

$$[k_e] = \begin{bmatrix} [k_{\text{axial}}] & [0] \\ [0] & [k_{\text{flexure}}] \end{bmatrix}$$ 2.43

and is a non-coupled superposition of axial and bending stiffnesses.

For plane truss structures, orientation of the element in the global coordinate system must be considered. Fig 2.5a depicts an element oriented at an arbitrary angle from the X axis of a global reference frame and shows the element nodal displacements. Before proceeding, note that it is convenient here to reorder the element stiffness matrix given by Equation 3.43 so that the element displacement vector in the element reference frame is given as

$$\{\delta\} = \begin{Bmatrix} u_1 \\ v_1 \\ \theta_1 \\ u_2 \\ v_2 \\ \theta_2 \end{Bmatrix}$$ 2.44

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 4 Apr. 2021, pp: 429-459 www.ijaem.net ISSN: 2395-5252

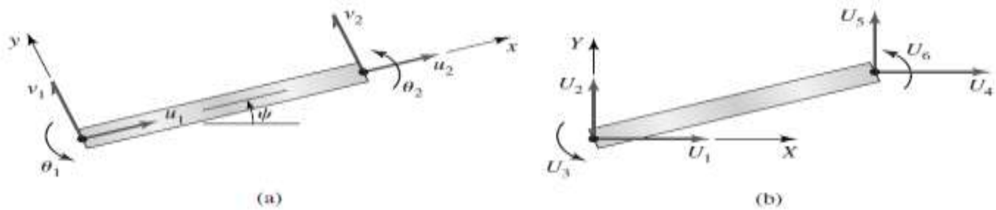


Figure 2.4: (a) Nodal displacements in the element coordinate system. (b) Nodal displacements in the global coordinate system.

And the element stiffness matrix becomes:

$$[k_e] = \begin{bmatrix} \frac{AE}{L} & 0 & 0 & \frac{-AE}{L} & 0 & 0 \\ 0 & \frac{12EI_z}{L^3} & \frac{6EI_z}{L^2} & 0 & \frac{-12EI_z}{L^3} & \frac{6EI_z}{L^2} \\ 0 & \frac{6EI_z}{L^2} & \frac{4EI_z}{L} & 0 & -\frac{6EI_z}{L^2} & \frac{2EI_z}{L} \\ \frac{-AE}{L} & 0 & 0 & \frac{AE}{L} & 0 & 0 \\ 0 & \frac{-12EI_z}{L^3} & \frac{-6EI_z}{L^2} & 0 & \frac{12EI_z}{L^3} & \frac{-6EI_z}{L^2} \\ 0 & \frac{6EI_z}{L^2} & \frac{2EI_z}{L} & 0 & \frac{-6EI_z}{L^2} & \frac{4EI_z}{L} \end{bmatrix}$$

……………. 2.45

Using Fig 3.4 the element displacements are written in terms of the global displacemen

$$\begin{aligned} u_1 &= U_1 \cos\psi + U_2 \sin\psi \\ v_1 &= -U_1 \sin\psi + U_2 \cos\psi \\ \theta_1 &= U_3 \\ u_2 &= U_4 \cos\psi + U_5 \sin\psi \\ v_2 &= -U_4 \sin\psi + U_5 \cos\psi \\ \theta_2 &= U_6 \end{aligned}$$

Equations 2.46 can be written in matrix form as

$$\begin{Bmatrix} u_1 \\ v_1 \\ \theta_1 \\ u_2 \\ v_2 \\ \theta_2 \end{Bmatrix} = \begin{bmatrix} \cos\psi & \sin\psi & 0 & 0 & 0 & 0 \\ -\sin\psi & \cos\psi & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos\psi & \sin\psi & 0 \\ 0 & 0 & 0 & -\sin\psi & \cos\psi & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \\ U_5 \\ U_6 \end{Bmatrix} = [R]\{U\}$$

. . 2.47 2.47

where [R] is the transformation matrix that relates element displacements to global displacements. Therefore, the 6 × 6 element stiffness matrix in the global system is given by

$$[K_e] = [R]^T [k_e][R]$$

. . . . 2.48

Carrying out matrix multiplication gives:

$$K = \begin{pmatrix} a_3 & a_4 & a_5 & -a_3 & -a_4 & a_5 \\ a_4 & a_6 & a_7 & -a_4 & -a_6 & a_7 \\ a_5 & a_7 & a_1 & -a_5 & -a_7 & a_2 \\ -a_3 & -a_4 & -a_5 & a_3 & a_4 & -a_5 \\ -a_4 & -a_6 & -a_7 & a_4 & a_6 & -a_7 \\ a_5 & a_7 & a_2 & -a_5 & -a_7 & a_1 \end{pmatrix}$$

. . . . 2.49

Where,

$$\left.\begin{aligned} a_1 &= 4EI/L \\ a_2 &= 2EI/L \\ a_3 &= (AE/L)c^2 + (12EI/L^3)s^2 \\ a_4 &= (AE/L)cs - (12EI/L^3)cs \\ a_5 &= -(6EI/L^2)s \\ a_6 &= (AE/L)s^2 + (12EI/L^3)c^2 \\ a_7 &= (6EI/L^2)c \end{aligned}\right\} \qquad 2.50$$

and, A=cross sectional area of element in m^2, E is modulus of elasticity in kn/m^2, L is the length of element in metre(m)

$c = \cos\psi$

$s = \sin\psi$

2.4 EQUIVALENT NODAL LOAD VECTOR OF RIGID JOINTED PLANE TRUSS SUBJECTED TO OUT OF JOINT TRANVERSE LOADING

In the derivation of bending stress of a flexure element loads were restricted to be applied at the nodes. But in some real cases loads may not only act at the nodes since we can have point loads at any section of the element or distributed loads in the case of dead loads. In this section, we derive work equivalent nodal load vectors that will actually give the same impact of the applied load at the nodes. Having in mind that the rigid jointed plane truss elements are modeled as a flexure element, the following derivation holds:

1. A point load on the rigid jointed plane truss bar. This is modeled as a flexure element with fixed ends. The truss element is divided into two elements with a node at the point of application of the force as shown below.

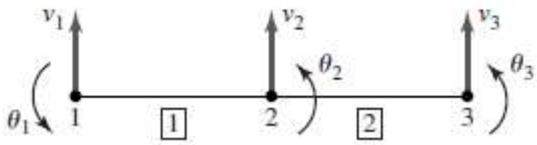


Figure 2.6: Flexure element with node at the point of application of a point load along the element

Recalling stiffness matrix of a flexure element as:

$$[k_e] = \frac{EI_z}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \qquad 2.51$$

Considering element 1, stiffness matrix for element 1 is as follows:

$$K^1 = EI/x^3 \begin{bmatrix} 12 & 6x & -12 & 6x \\ 6x & 4x^2 & -6x & 2x^2 \\ -12 & -6x & 12 & -6x \\ 6x & 2x^2 & -6x & 4x^2 \end{bmatrix} \qquad 2.52$$

Considering element 2, stiffness matrix for element 2 is as follows:

$$K^2 = EI/(L-x)^3 \begin{bmatrix} 12 & 6(L-x) & -12 & 6(L-x) \\ 6(L-x) & 4(L-x)^2 & -6(L-x) & 2(L-x)^2 \\ -12 & -6(L-x) & 12 & -6(L-x) \\ 6(L-x) & 2(L-x)^2 & -6(L-x) & 4(L-x)^2 \end{bmatrix} \qquad 2.53$$

Assembling the global stiffness matrix, the resultant components of the global stiffness matrix are.

$$
\left.\begin{aligned}
&K_{11} = K_{11}^{(1)} = 12EI/x^3\\
&K_{12} = K_{12}^{(1)} = 6EIx/x^3\\
&K_{13} = K_{13}^{(1)} = -12EI/x^3\\
&K_{14} = K_{14}^{(1)} = 6EIx/x^3\\
&K_{15} = 0\\
&K_{16} = 0\\
&K_{21} = K_{21}^{(1)} = 6EIx/x^3\\
&K_{22} = K_{22}^{(1)} = 6EIx/x^3\\
&K_{23} = K_{23}^{(1)} = -6EIx/x^3\\
&K_{24} = K_{24}^{(1)} = 2EIx^2/x^3\\
&K_{25} = K_{25}^{(1)} = 0\\
&K_{26} = K_{26}^{(1)} = 0\\
&K_{12} = K_{12}^{(1)} = 6EIx/x^3\\
&K_{31} = K_{31}^{(1)} = -12EI/x^3\\
&K_{32} = K_{32}^{(1)} = -6EIx/x^3\\
&K_{33} = K_{33}^{(1)} + K_{33}^{(2)} = 12EI/x^3 + 12EI/(L-x)^3\\
&K_{34} = K_{34}^{(1)} + K_{34}^{(2)} = -6EIx/x^3 + 12EI(L-x)/(L-x)^3\\
&K_{35} = K_{35}^{(2)} = -12EI/(L-x)^3\\
&K_{36} = K_{36}^{(2)} = 6EI(L-x)/(L-x)^3\\
&K_{41} = K_{41}^{(1)} = 6EIx/x^3\\
&K_{42} = K_{42}^{(1)} = 2EIx^2/x^3\\
&K_{43} = K_{43}^{(1)} + K_{43}^{(2)} = -6EIx/x^3 + 6EI(L-x)/(L-x)^3\\
&K_{44} = K_{44}^{(1)} + K_{44}^{(2)} = 4EIx^2/x^3 + 4EI(L-x)^2/(L-x)^3\\
&K_{45} = K_{45}^{(2)} = -6EI(L-x)/(L-x)^3
\end{aligned}\right\} \qquad 2.54
$$

$$K_{46} = K_{46}^{(2)} = 2EI(L-x)^2/(L-x)^3$$
$$K_{51} = 0$$
$$K_{52} = 0$$
$$K_{53} = K_{53}^{(2)} = -12EI/(L-x)^3$$
$$K_{54} = K_{54}^{(2)} = -6EI(L-x)/(L-x)^3$$
$$K_{55} = K_{55}^{(2)} = 12EI/(L-x)^3$$
$$K_{56} = K_{33}^{(2)} = -6EI(L-x)/(L-x)^3$$
$$K_{61} = 0$$
$$K_{62} = 0$$
$$K_{63} = K_{63}^{(2)} = 6EI(L-x)/(L-x)^3$$
$$K_{64} = K_{64}^{(2)} = 2EI(L-x)^2/(L-x)^3$$
$$K_{65} = K_{33}^{(2)} = -6EI(L-x)/(L-x)^3$$
$$K_{66} = K_{66}^{(2)} = 4EI(L-x)^2/(L-x)^3$$

Therefore the global stiffness matrix is as follows:

$$K = \begin{pmatrix} K_{11} & K_{12} & K_{13} & K_{14} & K_{15} & K_{16}\\ K_{21} & K_{22} & K_{23} & K_{24} & K_{25} & K_{26}\\ K_{31} & K_{32} & K_{33} & K_{34} & K_{35} & K_{36}\\ K_{41} & K_{42} & K_{43} & K_{44} & K_{45} & K_{46}\\ K_{51} & K_{52} & K_{53} & K_{54} & K_{55} & K_{56}\\ K_{61} & K_{62} & K_{63} & K_{64} & K_{65} & K_{66} \end{pmatrix} \qquad 2.55$$

From the general form,

$$KU = F \qquad 2.56$$

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 4 Apr. 2021, pp: 429-459 www.ijaem.net ISSN: 2395-5252

Where,

$$\{U\} = \begin{Bmatrix} v_1 \\ \Theta_1 \\ v_2 \\ \Theta_2 \\ v_3 \\ \Theta_3 \end{Bmatrix} \quad \ldots\ldots\ldots\ldots \quad 2.57$$

$$\{F\} = \begin{Bmatrix} F_1 \\ M_1 \\ F_2 \\ M_2 \\ F_3 \\ M_3 \end{Bmatrix} \quad \ldots\ldots\ldots\ldots \quad 2.58$$

$$\begin{bmatrix} K_{11} & K_{12} & K_{13} & K_{14} & K_{15} & K_{16} \\ K_{21} & K_{22} & K_{23} & K_{24} & K_{25} & K_{26} \\ K_{31} & K_{32} & K_{33} & K_{34} & K_{35} & K_{36} \\ K_{41} & K_{42} & K_{43} & K_{44} & K_{45} & K_{46} \\ K_{51} & K_{52} & K_{53} & K_{54} & K_{55} & K_{56} \\ K_{61} & K_{62} & K_{63} & K_{64} & K_{65} & K_{66} \end{bmatrix} \begin{Bmatrix} v_1 \\ \Theta_1 \\ v_2 \\ \Theta_2 \\ v_3 \\ \Theta_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ M_1 \\ F_2 \\ M_2 \\ F_3 \\ M_3 \end{Bmatrix} \quad \ldots \quad 2.59$$

Consider a fixed ended truss member with the following support conditions
Applying boundary conditions:

$$v_1 = v_3 = \Theta_1 = \Theta_3 = 0 \quad \ldots\ldots\ldots\ldots \quad 2.60$$

Therefore,

$$\begin{bmatrix} K_{33} & K_{34} \\ K_{43} & K_{44} \end{bmatrix} \begin{Bmatrix} v_2 \\ \Theta_2 \end{Bmatrix} = \begin{Bmatrix} F_2 \\ M_2 \end{Bmatrix} \quad \ldots\ldots\ldots \quad 2.61$$

But, $F_2 = -P$ and $M_2 = 0$;
Substituting the expressions for K_{33}, K_{34}, K_{43} and K_{44} in the above equation we obtain:

$$\begin{bmatrix} 12EI/x^3 + 12EI/(L-x)^3 & -6EIx/x^3 + 12EI(L-x)/(L-x)^3 \\ -6EIx/x^3 + 6EI(L-x)/(L-x)^3 & 4EIx^2/x^3 + 4EI(L-x)^2/(L-x)^3 \end{bmatrix} \begin{Bmatrix} v_2 \\ \Theta_2 \end{Bmatrix} = P. \begin{Bmatrix} . \\ 0 \end{Bmatrix} \quad 2.62$$

At x = L/2,

$$\left. \begin{aligned} V_2 &= -PL^3/(192EI) \\ \Theta_2 &= 0 \end{aligned} \right\} \quad \ldots\ldots\ldots\ldots \quad 2.63$$

Substituting the nodal displacement values into the constraints equation gives:

$$\left. \begin{aligned} F_1 &= P/2 \\ M_1 &= PL/8 \\ M_2 &= 0 \\ F_2 &= -P \\ M_3 &= -PL/8 \\ F_3 &= P/2 \end{aligned} \right\} \quad \ldots\ldots\ldots\ldots \quad 2.64$$

Therefore, the equivalent force vector, for a rigid truss member subjected to point load at the centre is as follows:

$$\{F\} = \begin{Bmatrix} F_1 \\ M_1 \\ F_3 \\ M_3 \end{Bmatrix} = \begin{Bmatrix} P/2 \\ PL/8 \\ -P/2 \\ PL/8 \end{Bmatrix} \quad . \quad . \quad . \quad . \quad . \quad . \quad . \quad . \qquad 2.65$$

Uniformly distributed load on a fixed ended plane truss member.

The restriction that loads be applied only at element nodes for the flexure element must be dealt with if a distributed load is present. The usual approach is to replace the distributed load with nodal forces and moments such that the mechanical work done by the nodal load system is equivalent to that done by the distributed load. Referring to Figure 3.7, the mechanical work performed by the distributed load can be expressed as:

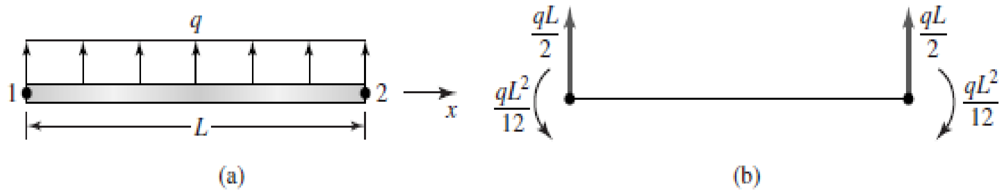


Figure 2.7: work equivalent nodal forces and moment for a uniformly distributed load

$$W = \int_0^L q(x)v(x)\,dx \quad . \quad . \quad . \quad . \quad . \quad . \quad . \quad . \quad . \qquad 2.66$$

The objective here is to determine the equivalent nodal loads so that the workexpressed in Equation 3.66 is the same as:

$$W = \int_0^L q(x)v(x)\,dx = F_{1q}v_1 + M_{1q}\theta_1 + F_{2q}v_2 + M_{2q}\theta_2 \quad . \quad . \quad . \qquad \mathbf{2.67}$$

whereF1q , F2q are the equivalent forces at nodes 1 and 2, respectively, and M1q and M2q are the equivalent nodal moments. Substituting the discretized displacement function given by Equation 3.16, the work integral becomes:

$$W = \int_0^L q(x)[N_1(x)v_1 + N_2(x)\theta_1 + N_3(x)v_2 + N_4(x)\theta_2]\,dx \quad . \quad . \quad . \qquad 2.68$$

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 4 Apr. 2021, pp: 429-459 www.ijaem.net ISSN: 2395-5252

Comparison of Equations 3.67 and 3.68 shows that

$$\left.\begin{aligned} F_{1q} &= \int_0^L q(x)N_1(x)\,dx \\ M_{1q} &= \int_0^L q(x)N_2(x)\,dx \\ F_{2q} &= \int_0^L q(x)N_3(x)\,dx \\ M_{2q} &= \int_0^L q(x)N_4(x)\,dx \end{aligned}\right\} \quad 2.69$$

Hence, the nodal force vector representing a distributed load on the basis of work equivalence is given by Equations 3.69. For example, for a uniform load q(x)= q = constant, and also substituting the interpolation functions

$$\left.\begin{aligned} N_1 &= 1 - 3x^2/L^2 + 2x^3/L^3 \\ N_2 &= x - 2x^2/L + x^3/L^2 \\ N_3 &= 3x^2/L^2 - 2x^3/L^3 \\ N_4 &= x^3/L^2 - x^2/L \end{aligned}\right\} \quad 2.70$$

and integrating them within specified limits yields:

$$\{F\} = \begin{Bmatrix} F_{1q} \\ M_{1q} \\ F_{2q} \\ M_{2q} \end{Bmatrix} = \begin{Bmatrix} qL/2 \\ -qL^2/12 \\ qL/2 \\ qL^2/12 \end{Bmatrix} \quad 2.71$$

Which is the nodal equivalent loads on the element.

2.5 BENDING STRESSES IN RIGID JOINTED PLANE TRUSS MEMBER

For plane truss structures, orientation of the element in the global coordinate system must be considered. Fig 2.4a and b depicts an element oriented at an arbitrary angle from the X axis of a global reference frame and shows the element nodal displacements in both local and global coordinates. This orientation of the element was not considered in deriving bending stresses of equations 3.22 and 3.23, so re-arranging the equations considering the orientation of the element in global reference frame yields as follows: From eq. 2.23,

$$\sigma_x(x=0) = y_{max}E\left[\frac{6}{L^2}(v_2 - v_1) - \frac{2}{L}(2\theta_1 + \theta_2)\right] \quad 2.72$$

$\sigma_x(x=0) = y_{max}E[\,6v_2/L^2 - 6v_1/L^2 - 4\Theta_1/L - 2\Theta_2/L]$ . . . 2.73

re-arranging in an ascending order gives:

$\sigma_x(x=0) = y_{max}E[\,6v_1/L^2 - 4\Theta_1/L + 6v_2/L^2 - 2\Theta_2/L]$ . . . 2.74

substituting eq. 3.46 for $v_1, \Theta_1, v_2, \Theta_2$ in eq. 3.74 gives:

$\sigma_x(x=0) = y_{max}E[\,6sU_1/L^2 - 6cU_2/L^2 - 4U_3/L - 6s\,U_4/L^2 + 6c\,U_5/L^2 + 2\,U_6/L]$ . . . 2.75

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 4 Apr. 2021, pp: 429-459 www.ijaem.net ISSN: 2395-5252

in matrix form gives:

$$\sigma_x(x=0) = y_{max}E[\ 6s/L^2\ \ -6c/L^2\ \ {}^{-}4/L\ -6s/L^2\ \ 6c/L^2\ \ 2/L] \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \\ U_5 \\ U_6 \end{Bmatrix} \quad . \quad . \quad . \quad . \quad 2.76$$

Similarly, eq. 3.24 is re-arranged as follows:

$$\sigma_x(x=L) = y_{max}E[\ -6s/L^2\ \ 6c/L^2\ \ 2/L\ 6s/L^2\ -6c/L^2\ 4/L] \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \\ U_5 \\ U_6 \end{Bmatrix} \quad . \quad . \quad . \quad . \quad . \quad 2.77$$

2.6 AXIAL STRESSES IN RIGID JOINTED PLANE TRUSS MEMBER

From, $\sigma_x = E\varepsilon$ 2.78

where,

$\varepsilon = (u_2 - u_1)/L$ 2.79

$\sigma_x = E(u_2 - u_1)/L$ 2.80

substituting, expressions for u_1, u_2 in eq. 3.46 Into eq. 3.80 Gives:

$\sigma_x = E[\ -cU_1 - sU_2 + cU_4 + sU_5]$ 2.81

converting the above equation into matrix form gives

$$\sigma_x = E[\ -c\ -s\ \ 0\ \ c\ \ s\ \ 0\ ] \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \\ U_5 \\ U_6 \end{Bmatrix} \quad . \quad . \quad . \quad . \quad . \quad . \quad . \quad 2.82$$

2.7 MODAL ANALYSIS (NATURAL FREQUENCY)

2.7.1 FORMATION OF LUMP MASS MATRIX

By taking the flexural effects of the members into consideration and by using the lump mass method, the mass influence coefficient for axial effects of rigid jointed truss element is found out. Combining the mass matrix for flexural effects with the matrix for axial effects we obtain the lump mass matrix for a uniform element of a plane truss with rigid joint in reference to the modal coordinates. [34]

Lump mass matrix is given the following equation:

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \\ F_6 \end{Bmatrix} = \varrho AL/2 \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \\ U_5 \\ U_6 \end{Bmatrix} \quad .. \quad . \quad . \quad .2.83$$

or in condensed notation,

$\{F\} = [Me]\ \{U\}$ 2.84

In which [M] is the lump mass matrix for the element of a rigidly jointed plane truss.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 4 Apr. 2021, pp: 429-459 www.ijaem.net ISSN: 2395-5252

2.7.2 TRANSFORMATION FROM LOCAL TO GLOBAL CO-ORDINATE SYSTEM

Repeating the procedure of transformation as applied to stiffness matrix , for the lump mass matrix we obtain in the similar manner

{F} = [M] {U} ……………………………………. 2.85

In which,

$\{M\} = [R]^T[M_e][R]$ …………………………… 2.86

2.7.3 CALCULATION OF EIGEN VALUE AND EIGEN VECTOR

The structure is not excited externally in free vibration mode that is no force or support motion acts on it. So, under condition of free motion, dynamic analysis can be carried out and the important properties like natural frequencies mode shapes corresponding to the natural frequency can be obtained.

2.7.4 NATURAL FREQUENCIES

Since free vibration mode is considered, the structure is not under influence of any external force. Hence, the force vector in stiffness equation or flexible equations is taken as zero.

By taking the above condition into consideration, the stiffness equation can be represented as:

$$[M]\{\ddot{U}\} + [K]\{U\} = \{0\} \quad \text{……………………………………… 2.87}$$

The solution of the above equation for undamped structure is in the form,

$U_i = a_i + \sin(\omega t\text{-}a)$ i = 1,2,3…….n …………… 2.88

When represented as vector,

$\{U\} = \{a\}\sin(\omega t\text{-}a)$ ………………………………………… 2.89

Where a_i is the amplitude of motion of nth coordinate and is the and n is the number of degree of freedom. Substituting equation 3.89 in equation 3.87, we get

$-\omega^2[M]\{a\}\sin(\omega t\text{-}a) + [K]\{a\}\sin(\omega t\text{-}a) = 0$ …………………… 2.90

Or

$[[K] - \omega^2[M]]\,\{a\} = \{0\}$ …………………………………… 2.91

The mathematical problem for the formulation of the above equation is called eigenproblem. As the right hand side of the equation is equal to zero. It can be considered as a set of n number of homogeneous linear equations with n unknown displacements a_i and ω^2 as the unknown parameter, amplitude ‘a’ cannot be zero. Its solution is non-trivial. So,

$\left|[[K] - \omega^2[M]]\right| = 0$ ……………………………… 2.92

The polynomial equation of degree n in ω^2 obtained is the characteristic equation and the values of ω are the natural frequency of the structure. The values that satisfy equation 2.92can be substituted in equation 2.91 to obtain the amplitudes in terms of arbitrary constant.

2.8 ORTHOGONALITY OF PRINCIPAL MODE

The principal modes of vibration of systems with multiple degrees of freedom share a fundamental mathematical property known as orthogonality. The free-vibration response of a multiple degrees-of-freedom system is described by Equation below:

$$[M]\{\ddot{q}\} + [K]\{q\} = \{0\} \quad \text{……………………………. 2.93}$$

Assuming that we have solved for the natural circular frequencies and the modal amplitude vectors via the assumed solution form qi (t)= Ai sin(ωt + ω), substitution of a particular frequency ωi into Equation 3.93 gives

$$-\omega_i^2[M]\{A^{(i)}\} + [K]\{A^{(i)}\} = 0 \quad \text{………………………….. 2.94}$$

and for any other frequency ωj

Multiplying Equation 3.93 by $\{A(j)\}^T$and Equation 3.94by $\{A(i)\}^T$gives

. .. . . 2.95

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 4 Apr. 2021, pp: 429-459 www.ijaem.net ISSN: 2395-5252

$$-\omega_i^2\{A^{(j)}\}^T[M]\{A^{(i)}\} + \{A^{(j)}\}^T[K]\{A^{(i)}\} = 0 \quad \text{……………..} \quad 2.96$$
$$-\omega_j^2\{A^{(i)}\}^T[M]\{A^{(j)}\} + \{A^{(i)}\}^T[K]\{A^{(j)}\} = 0$$

Subtracting Equation 2.96from Equation 2.95, we have

$$\{A^{(j)}\}^T[M]\{A^{(i)}\}(\omega_i^2 - \omega_j^2) = 0 \qquad i \neq j \quad \text{…………………} \quad 2.97$$

In arriving at the result represented by Equation 3.97, we utilize the fact from matrix algebra that $[A]^T[B][C] = [C]^T[B][A]$, where [A], [B], [C] are any three matrices for which the triple product is defined. As the two circular frequencies in Equation 10.104 are distinct, we conclude that

$$\{A^{(j)}\}^T[M]\{A^{(i)}\} = 0 \qquad i \neq j \quad \text{…………………………..} \quad 2.98$$

Equation 3.98 is the mathematical statement of orthogonality of the principal modes of vibration.
For a system exhibiting P degrees of freedom, we define the modal matrix as a P × P matrix in which the columns are the amplitude vectors for each natural mode of vibration; that is,

$$[A] = \left[\{A^{(1)}\}\{A^{(2)}\}\ldots\{A^{(P)}\}\right] \quad \text{………………………} \quad 2.99$$

and consider the matrix triple product [S] = [A]T [M][A] . Per the orthogonality condition, Equation 2.99, each off-diagonal term of the matrix represented by the triple product is zero; hence, the matrix [S] = [A]T[M][A] is a diagonalmatrix. The diagonal (nonzero) terms of the matrix have magnitude

$$S_{ii} = \{A^{(i)}\}^T[M]\{A^{(i)}\} \qquad i = 1, P \quad \text{………………….} \quad 2.100$$

As each modal amplitude vector is known only within a constant multiple the modal amplitude vectors
can be manipulated such that the diagonal terms described by Equation 3.100 can be made to assume any desired numerical value. In particular, if the value is selected as unity, so that

$$S_{ii} = \{A^{(i)}\}^T[M]\{A^{(i)}\} = 1 \qquad i = 1, P \quad \text{……………………} \quad 2.101$$

where [I] is the P × P identity matrix.
Amplitude vectors are then normalized as follows:

The corresponding diagonal term of the modal matrixis

$$\sum_{j=1}^{P}\sum_{k=1}^{P} m_{jk}A_j^{(i)}A_k^{(i)} = S_{ii} = \text{constant} \quad \text{………………..} \quad 2.102$$

If we redefine the terms of the modal amplitude vector so that

$$A_j^{(i)} = \frac{A_j^{(i)}}{\sqrt{S_{ii}}} = \frac{A_j^{(i)}}{\sqrt{\sum_{j=1}^{P}\sum_{k=1}^{P} m_{jk}A_j^{(i)}A_k^{(i)}}} \qquad i = 1, P \quad \text{………………….} \quad 2.103$$

the matrix described by Equation 3.103 is indeed the identity matrix.
Having established the orthogonality concept and normalized the modal matrix, we return to the general problem described by Equation 3.93, in which the force vector is no longer assumed to be zero. For reasons that will become apparent, we introduce the change of variables

$$\{q\} = [A]\{p\} \quad \text{…………………………} \quad 2.104$$

Where,{ p} is the column matrix of generalized displacements, which are linear combinations of the actual nodal displacements {q}, and [A] is the normalized modal matrix. Equation 3.93 then becomes

$$[M][A]\{\ddot{p}\} + [K][A]\{p\} = \{F\} \quad \text{…………………….} \quad 2.105$$

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 4 Apr. 2021, pp: 429-459 www.ijaem.net ISSN: 2395-5252

Premultiplying by $[A]^T$, we obtain

$$[A]^T[M][A]\{\ddot{p}\} + [A]^T[K][A]\{p\} = [A]^T\{F\} \quad \ldots\ldots\ldots\ldots\ldots\ldots \quad 2.106$$

Now we must examine the stiffness effects as represented by $[A]^T[K][A]$. Given that [K] is a symmetric matrix, the triple product $[A]^T[K][A]$ is also a symmetric matrix. Following the previous development of orthogonality of the principal modes, the triple product [A]T[K][A] is also easily shown to be a diagonal matrix. The values of the diagonal terms are found by multiplying Equation 3.106 by $A(i)^T$ to obtain

$$-\omega_i^2\{A^{(i)}\}^T[M]\{A^{(i)}\} + \{A^{(i)}\}^T[K]\{A^{(i)}\} = 0 \qquad i = 1, P \quad \ldots\ldots\ldots\ldots\ldots\ldots \quad 2.107$$

If the modal amplitude vectors have been normalized as described previously, Equation 3.107 is

$$\{A^{(i)}\}^T[K]\{A^{(i)}\} = \omega_i^2 \qquad i = 1, P \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad 2.108$$

hence, the matrix triple product [A]T [K][A] produces a diagonal matrix having diagonal terms equal to the squares of the natural circular frequencies of the principal modes of vibration; that is,

$$[A]^T[K][A] = \begin{bmatrix} \omega_1^2 & 0 & \cdot & \cdot & \cdot & 0 \\ 0 & \omega_2^2 & & & & \cdot \\ \cdot & & \cdot & & & \cdot \\ \cdot & & & \cdot & & \cdot \\ \cdot & & & & \cdot & \cdot \\ 0 & \cdot & \cdot & \cdot & & \omega_P^2 \end{bmatrix} \quad \ldots\ldots\ldots\ldots\ldots\ldots \quad 2.109$$

Finally, Equation 10.115 becomes

$$[I]\{\ddot{p}\} + [\omega^2]\{p\} = [A]^T\{F\} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad 2.110$$

with matrix $[\omega^2]$ representing the diagonal matrix defined in Equation 1.111.
where A is the mode shape(eigen vector).

3.1 ANALYSIS

In this section, static and dynamic analyses of rigid jointed and pin jointed plane truss structures are considered. Other forms of loadings other than the conventional loading (load must act at the joint) such as uniformly distributed loads and point loads are also considered. Equations and derivations emphasized in the previous sections using finite element analysis techniques are used in the analysis.

Parameters such as stresses in members, joints displacements, and reactions at the supports are sorted in the case of static analysis. While, natural frequencies and mode shapes are sort in the case of dynamic (modal) analysis. Both rigid and pin jointed plane truss structures are analyzed for these parameters and their results are compared.

Robot structural analysis software, a commercial structural analysis software is used to cross check the results obtained from the computer program written in this project using MATLAB codes to analyze the plane truss structures. The program is based on the finite element method of structural analysis and is developed using MATLAB. Care was taken during input process as it has a great impact on the results. The input data in this program are: unit area of material, density of material, modulus of elasticity of material, nodal coordinates, prescribed degree of freedom of the system(PrescribedDof), and element nodes connections.

Numerical example 1(pin jointed plane truss structures)

Consider a modified warren truss for Railway Bridge shown in/ figure 3.1. The dimensions, nodal numberings, supports and loadings are as also shown on fig. 3.1. The elements are composed of 50 x 50 x 6 equal angle iron with the foll0wing properties: modulus of elasticity(E)= 205EKN/m^2, density(ϱ)=7850Kg/m^3, area(A) =0.000569m^2.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 4 Apr. 2021, pp: 429-459 www.ijaem.net ISSN: 2395-5252

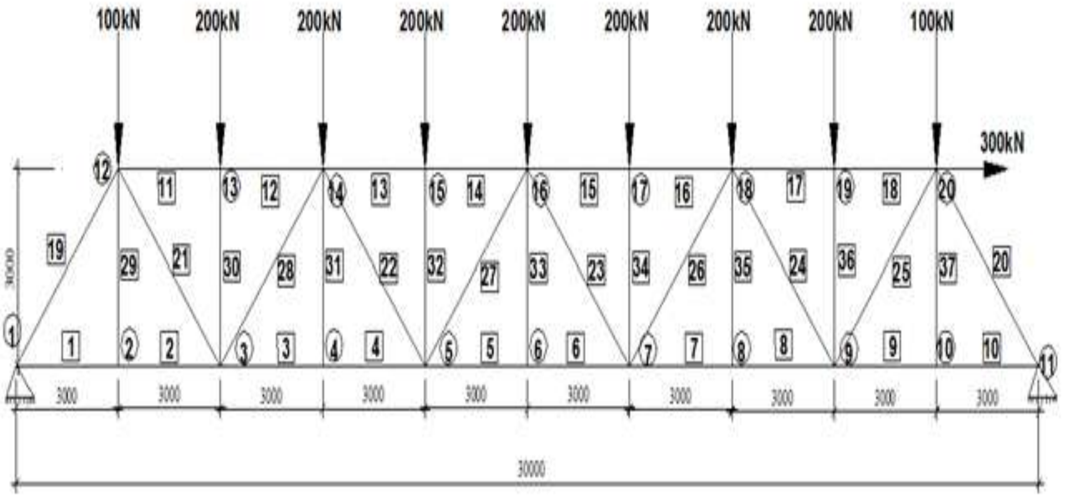


Figure 3.1 Warren truss for Railway Bridge.

The nodal coordinates and elements numbers are as follows:

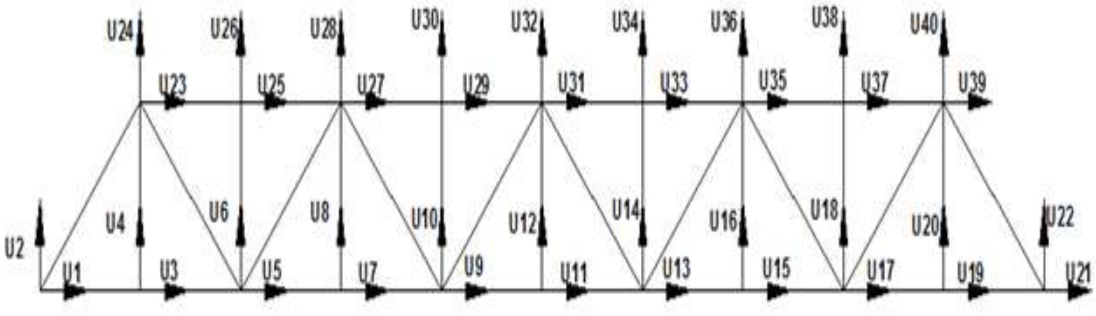


Figure 3.2 Nodal coordinates and element numbering.

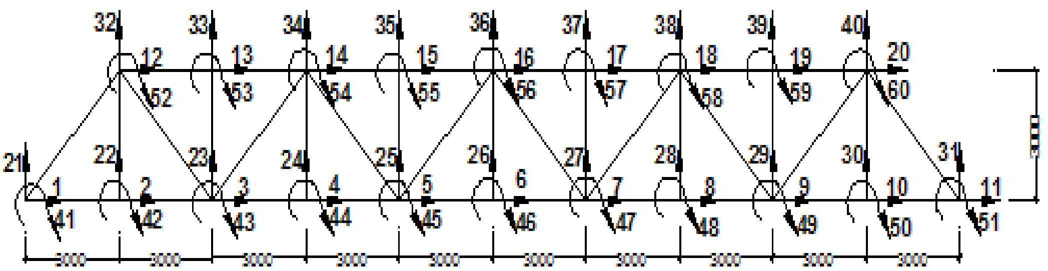
Nodes 1 and 11 are constrained by their support conditions. Therefore, $U_1 = U_2 = U_{21} = U_{22} = 0$. The above equation is then reduced to 36 x 36 matrix equation. The unknowns U_s which are global displacements are then sort by solving the equation using MaTlab as follows:

NUMERICAL EXAMPLE2: RIGID JOINTED PLANE TRUSS STRUCTURES

Considering figure 3.1 as a plane truss structure whose joints are replaced with rigid joints and the loadings and support conditions remains the same.

SOLUTION 2

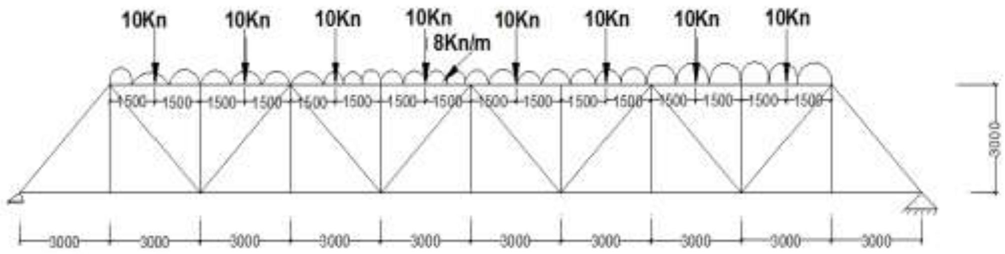
The free body diagram of the structure is as follows:



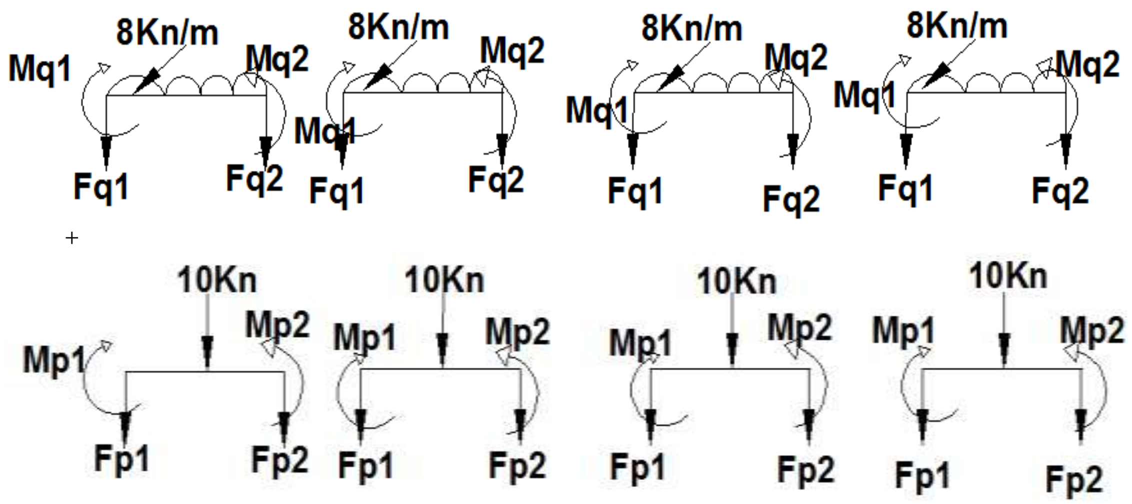
Tutorial 3(Uniformly distributed loads and out – of – joints point load)

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 4 Apr. 2021, pp: 429-459 www.ijaem.net ISSN: 2395-5252

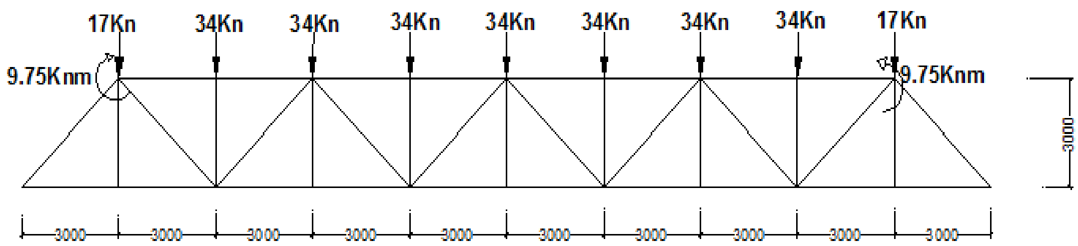
Consider the truss structure shown below, all the joints in the structure are rigid jointed, its members have the same sectional properties as the structure in tutorial 2, the loadings are also as shown and all dimensions are in metres.



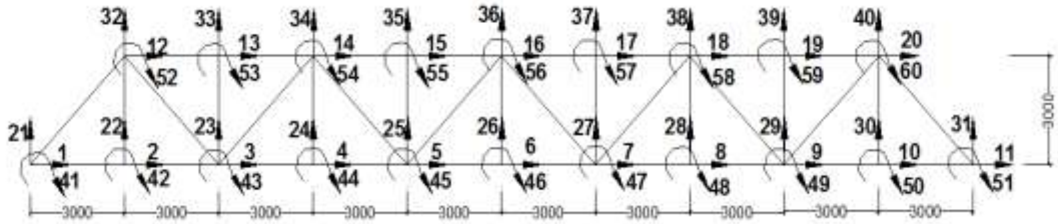
Solution:
Work equivalent nodal loads are calculated using equations 3. And 3. As follows:
Consider the top cord which is loaded as follows:



But,
$Mq1 = ql^2/12 = 8 \times 3^2 / 12 = 6Knm = - Mq2$
$Mp1= PL/8 = 10 \times 3/8 = 3.75Knm = - Mp2$
$Fq1 =-qL/2 = -8 \times 3/2 =- 12Kn = -Fq2$
$Fp1 = -P/2 = -10/2 = -5Kn = -Fp2$
Moment at the first external node = Mq1 + Mp1= 9.75Knm
Moment at the second external node = Mq2 + Mp2 = - 9.75Knm
Moment at the internal nodes = Mq1 + Mq2 + Mp1 + Mp2 = 0Knm
Force at the external nodes = Fq1 + Fp1 = -17Kn
Force at the internal nodes = Fq1 + Fp1 + Fq2 + Fp2 = 34Kn
Therefore the work equivalent nodal loads on the structure are shown below:



The structure universal coordinate system is the same as the one for tutorial 2 as shown below:



Since the structure geometry and member properties are the same with that of tutorial 2, the members' stiffness matrices are the same as and the global structure stiffness matrix is the same too.

SOLTION BY MATLAB PROGRAM
M - FILES

a. FormStiffness2Dtruss.m –

This m.file calculates stiffness matrix for each element and assemble them for the entire structure. The script is written as follows:

```
function [stiffness]=...
formStiffness2Dtruss(GDof,numberElements,...
elementNodes,numberNodes,nodeCoordinates,xx,y
y,EA);
stiffness=zeros(GDof);
for e=1:numberElements;
formatlong
indice=elementNodes(e,:) ;
elementDof=[ indice(1)*2-1 indice(1)*2
indice(2)*2-1 indice(2)*2] ;
xa=xx(indice(2))-xx(indice(1));
ya=yy(indice(2))-yy(indice(1));
length_element=sqrt(xa*xa+ya*ya);
C=xa/length_element;
S=ya/length_element;
k1=EA/length_element*[C*C C*S -C*C -C*S;
C*S S*S -C*S -S*S;-C*C -C*S C*C C*S;-C*S -
S*S C*S S*S];
stiffness(elementDof,elementDof)=stiffness(eleme
ntDof,elementDof)+k1
end
```

b. FormMass2Dtruss.m –

This m.file calculate mass matrix for each element and assemble them for the entire structure. The script is written as follows:

```
function
[mass]=formMass2Dtruss(GDof,numberElements,.
..
elementNodes,numberNodes,nodeCoordinates,xx,y
y,rhoA);
mass=zeros(GDof);
for e=1:numberElements;
indice=elementNodes(e,:) ;
elementDof=[ indice(1)*2-1 indice(1)*2
indice(2)*2-1 indice(2)*2] ;
xa=xx(indice(2))-xx(indice(1));
ya=yy(indice(2))-yy(indice(1));
leng=sqrt(xa*xa+ya*ya);
m1=(rhoA*leng/6)*[2 0 1 0;0 2 0 1;1 0 2 0;0 1 0
2];
mass(elementDof,elementDof)=mass(elementDof,e
lementDof)+m1;
end
```

c. DisplacementsReactions-

This computes and displays displacements and reaction outputs and the script is program as follows.

```
%.............................................................
functionDisplacementsReactions...
(displacements,stiffness,GDof,prescribedDof)
formatlong
disp('Displacements')
jj=1:GDof; format
[jj' displacements]
F=stiffness*displacements;
reactions=F(prescribedDof);
disp('reactions')
[prescribedDof reactions]
```

d. Stress2Dtruss.m-

This computes and displays stresses outputs and the script is programmed as follows:

```
function
stresses2Dtruss(numberElements,elementNodes,...
xx,yy,displacements,E)
% stresses at elements
formatlong
for e=1:numberElements;
indice=elementNodes(e,:);
elementDof=[ indice(1)*2-1 indice(1)*2
indice(2)*2-1 indice(2)*2] ;
xa=xx(indice(2))-xx(indice(1));
ya=yy(indice(2))-yy(indice(1));
length_element=sqrt(xa*xa+ya*ya);
C=xa/length_element;
S=ya/length_element;
sigma(e)=E/length_element* ...
[-C -S C S]*displacements(elementDof);
end
sigma1=sigma/1000;
num=1:numberElements;
stresses=[num' sigma1']
```

e. massStiffness2DRigidJointedTruss

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 4 Apr. 2021, pp: 429-459 www.ijaem.net ISSN: 2395-5252

```
function [mass]=...
massStiffness2DRigidJointedTruss(GDof,numberE
lements,...
elementNodes,numberNodes,xx,yy,rhoA);
mass=zeros(GDof);
% computation of the system stiffness matrix
for e=1:numberElements;
% elementDof: element degrees of freedom (Dof)
indice=elementNodes(e,:) ;
elementDof=[ indiceindice+numberNodes
indice+2*numberNodes];
nn=length(indice);
xa=xx(indice(2))-xx(indice(1));
ya=yy(indice(2))-yy(indice(1));
length_element=sqrt(xa*xa+ya*ya);
cosa=xa/length_element;
sena=ya/length_element;
ll=length_element;
L= [cosa*eye(2) sena*eye(2) zeros(2);
-sena*eye(2) cosa*eye(2) zeros(2);
zeros(2,4) eye(2)];
oneu=[1 -1;-1 1];
oneu2=[1 -1;1 -1];
oneu3=[1 1;-1 -1];
oneu4=[4 2;2 4];
M1=zeros(6,6);
mm = rhoA*ll/420;
ma=rhoA*ll/6;
M1=[2*ma 0 0 ma 0 0;...
  0 156*mm 22*ll*mm 0 54*mm -13*ll*mm;...
  0 22*ll*mm 4*ll*ll*mm 0 13*ll*mm -
3*ll*ll*mm;...
ma 0 0 2*ma 0 0;...
  0 54*mm 13*ll*mm 0 156*mm -22*ll*mm;...
  0 -13*ll*mm -3*ll*ll*mm 0 -22*ll*mm
4*ll*ll*mm]
mass(elementDof,elementDof)=...
mass(elementDof,elementDof)+L'*M1*L;
end
```

f. formStiffness2DRigidJointedTruss

```
function [stiffness]=...
formStiffness2DRigidJointedTruss(GDof,numberE
lements,...
elementNodes,numberNodes,xx,yy,EI,EA);
stiffness=zeros(GDof);
% computation of the system stiffness matrix
for e=1:numberElements;
% elementDof: element degrees of freedom (Dof)
indice=elementNodes(e,:) ;
elementDof=[ indiceindice+numberNodes
indice+2*numberNodes] ;
nn=length(indice);
xa=xx(indice(2))-xx(indice(1));
ya=yy(indice(2))-yy(indice(1));
length_element=sqrt(xa*xa+ya*ya);
cosa=xa/length_element;
sena=ya/length_element;
ll=length_element;
L= [cosa*eye(2) sena*eye(2) zeros(2);
-sena*eye(2) cosa*eye(2) zeros(2);
zeros(2,4) eye(2)];
oneu=[1 -1;-1 1];
oneu2=[1 -1;1 -1];
oneu3=[1 1;-1 -1];
oneu4=[4 2;2 4];
k1=[EA/ll*oneu zeros(2,4);
zeros(2) 12*EI/ll^3*oneu 6*EI/ll^2*oneu3;
zeros(2) 6*EI/ll^2*oneu2 EI/ll*oneu4];
stiffness(elementDof,elementDof)=...
stiffness(elementDof,elementDof)+L'*k1*L;
end
```

g. bendingstresses2Dtruss

```
function
bendingstresses2Dtruss(numberElements,elementN
odes,numberNodes,...
xx,yy,displacements,E,Ymax)
% stresses at elements
format
for e=1:numberElements;
indice=elementNodes(e,:);
elementDof=[ indiceindice+numberNodes
indice+2*numberNodes] ;
nn=length(indice);
xa=xx(indice(2))-xx(indice(1));
ya=yy(indice(2))-yy(indice(1));
length_element=sqrt(xa*xa+ya*ya);
L=length_element;
C=xa/length_element;
S=ya/length_element;
sigmax(e)=E*Ymax* ...
[(-6*S)/(L*L) (-6*S)/(L*L) (-6*C)/(L*L)
(6*C)/(L*L) (-4)/(L)
2/L]*displacements(elementDof);
sigmal(e)=E*Ymax* ...
[(-6*S)/(L*L) (6*S)/(L*L) (6*C)/(L*L) (-
6*C)/(L*L) (2)/(L)
4/L]*displacements(elementDof);
end
sigma1=-sigmax/1000;
sigma2=-sigmal/1000;
num=1:numberElements;
Bendingstresses=[num' sigma1' sigma2']
```

h. femodal

```
function[omega,Vnorm,Modf]=femodal(mass,stiffn
ess,F)
%variable
[n,n]=size(mass);
[n,m]=size(F);
[V,D]=eig(stiffness,mass);
[Lamda,k]=sort(diag(D));V=V(:,k);
Factor=diag(V'*mass*V);
```

```
Vnorm=V*inv(sqrt(diag(Factor)));
omega=diag(sqrt(Vnorm'*stiffness*Vnorm));
Modf=Vnorm'*F;
```

MATLAB PROGRAM FOR THE ANALYSIS PIN JOINTED PLANE TRUSS STRUCTURES (STATIC AND MODAL ANALYSIS)

```
%Ebu's Plane Truss Program (EPTP)
format
%..............................................................
% static / dynamic analysis of plane trusses
% clear memory
clearall
% E; modulus of elasticity
% A: area of cross section
% L: length of bar
E=205e9; A=0.000569; EA=E*A;
rho=7850;rhoA=rho*A;
% generation of coordinates and connectivities
elementNodes=[1 2;2 3;3 4;4 5;5 6;6 7;7 8;8 9;9 10;10 11;12 13;13 14;14 15;15 16;16 17;17 18;18 19;19 20; 1 12;11 20;3 12;5 14;7 16;9 18;9 20;7 18;5 16;3 14;2 12;3 13;4 14;5 15;6 16;7 17;8 18;9 19;10 20];
nodeCoordinates=[0 0;3 0;6 0;9 0;12 0;15 0;18 0;21 0;24 0;27 0;30 0;3 3;6 3;9 3;12 3;15 3;18 3;21 3;24 3;27 3];
numberElements=size(elementNodes,1);
numberNodes=size(nodeCoordinates,1);
xx=nodeCoordinates(:,1);
yy=nodeCoordinates(:,2);
% for structure:
% displacements: displacement vector
% force : force vector
% stiffness: stiffness matrix
GDof=2*numberNodes; % GDof: total number of degrees of freedom
U=zeros(GDof,1);
force=zeros(GDof,1);
% applied load at node 2
force(24)=-100.0;
force(26)=-200.0;
force(28)=-200.0;
force(30)=-200.0;
force(32)=-200.0;
force(34)=-200.0;
force(36)=-200.0;
force(38)=-200.0;
force(40)=-100.0;
force(39)=300.0;
% computation of the system stiffness matrix
[stiffness]=...
formStiffness2Dtruss(GDof,numberElements,...
elementNodes,numberNodes,nodeCoordinates,xx,yy,EA);
[mass]=...
formMass2Dtruss(GDof,numberElements,...
elementNodes,numberNodes,nodeCoordinates,xx,yy,rhoA);
% boundary conditions and solution
prescribedDof=[1 2 21 22]';
% solution
displacements=solution(GDof,prescribedDof,stiffness,force);
% drawing displacements
clf
holdon
% stresses at elements
stresses2Dtruss(numberElements,elementNodes,xx,yy,displacements,E);
% output displacements/reactions
DisplacementsReactions(displacements,stiffness,GDof,prescribedDof);
%apply constraints
[stiffness,mass]=feaplycsm(stiffness,mass,prescribedDof);
fsol=eig(stiffness,mass);
fsol2=sqrt(fsol);
fsol1=0.159171*fsol2;
%print fem solutions
num=1:1:GDof;
frequency=[num' fsol1]
```

MATLAB PROGRAM FOR THE ANALYSIS OF RIGID JOINTED PLANE TRUSS STRUCTURES (STATIC AND MODAL ANALYSIS)

```
clearall
% E; modulus of elasticity
% I: second moment of area
% L: length of bar
format
E=205e9; A=0.000569; EA=E*A;
rho=7850;rhoA=rho*A; I=0.00000841; EI=E*I;
Ymax=0.0205;
% generation of coordinates and connectivities
elementNodes=[1 2;2 3;3 4;4 5;5 6;6 7;7 8;8 9;9 10;10 11;12 13;13 14;14 15;15 16;16 17;17 18;18 19;19 20; 1 12;11 20;3 12;5 14;7 16;9 18;9 20;7 18;5 16;3 14;2 12;3 13;4 14;5 15;6 16;7 17;8 18;9 19;10 20];
nodeCoordinates=[0 0;3 0;6 0;9 0;12 0;15 0;18 0;21 0;24 0;27 0;30 0;3 3;6 3;9 3;12 3;15 3;18 3;21 3;24 3;27 3];
numberElements=size(elementNodes,1);
numberNodes=size(nodeCoordinates,1);
xx=nodeCoordinates(:,1);
yy=nodeCoordinates(:,2);
% for structure:
% displacements: displacement vector
% force : force vector
% stiffness: stiffness matrix
```

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 4 Apr. 2021, pp: 429-459 www.ijaem.net ISSN: 2395-5252

```
% GDof: global number of degrees of freedom
GDof=3*numberNodes;
U=zeros(GDof,1);
force=zeros(GDof,1);
%force vector
force(32)=-100.0;
force(33)=-200.0;
force(34)=-200.0;
force(35)=-200.0;
force(36)=-200.0;
force(37)=-200.0;
force(38)=-200.0;
force(39)=-200.0;
force(40)=-100.0;
force(20)=300.0;
F=[0 0 0 0 1 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
0 0 0 0 0 0]';
% stiffness matrix
[stiffness]=...
formStiffness2DRigidJointedTruss(GDof,numberE
lements,...
elementNodes,numberNodes,xx,yy,EI,EA);
[mass]=...
massStiffness2DRigidJointedTruss(GDof,numberE
lements,...
elementNodes,numberNodes,xx,yy,rhoA);
% boundary conditions and solution
prescribedDof=[1 31 2 32]';
% solution
displacements=solution(GDof,prescribedDof,stiffn
ess,force);
% output displacements/reactions
DisplacementsReactions(displacements,stiffness,...
GDof,prescribedDof)
bendingstresses2Dtruss(numberElements,elementN
odes,numberNodes,...
xx,yy,displacements,E,Ymax)
axialstresses2Dtruss(numberElements,elementNod
es,numberNodes,...
xx,yy,displacements,E)
[stiffness,mass]=feaplycsm1(stiffness,mass,prescri
bedDof);
fsol=eig(stiffness,mass);
fsol2=sqrt(fsol)
fsol1=0.159171*fsol2;
%print fem solutions
num=1:1:GDof;
frequency=[num' fsol1]
[omega,Vnorm,Modf]=femodal(mass,stiffness,F)
% drawing undeformed and deformed meshes
U=displacements;
clf
drawingMesh(nodeCoordinates+500*[U(1:number
Nodes)...
U(numberNodes+1:2*numberNodes)],elementNod
es,'L2','k.-');
drawingMesh(nodeCoordinates,elementNodes,'L2','
k--');
```

TABLE 3.1 REACTION FORCES

NO.	REACTION(KN) PIN JOINT	REACTION(KN) RIGID JOINT
F1	1450	1459.5
F2	700	770
F21	-1750	-1759.5
F22	830	830

TABLE 3.2 AXIAL AND BENDING STRESSES

	PIN JOINT	RIGID JOINT	BENDING STRESSES FOR RIGID JOINT (Mpa)	
ELEMENT NO.	AXIAL STRESS(Mpa)	AXIAL STRESS (Mpa)	At x=0	x=L
1	-1195.08	1206.3	-15.24	483.96
2	-1195.013	1157.6	-117.3	488.29
3	808.37	-786.8	-23.98	267.91
4	808.4	8116.9	-102.83	178.05
5	1405.98	-1.3925	-3.40	-86.77
6	1405.98	-1390.5	-70.12	-173.97
7	598	-604.9	36.50	-411.56
8	597.54	-575.7	-19.01	-419.86
9	-1616.87	1573.2	99.86	-549.02
10	-1687.87	1625.2	33.56	-396.42
11	-2530.76	2460.9	-129.03	515.62

12	-2530.76	2500.2	-11.94	245.71
13	-3831.28	3764.5	-116.17	196.70
14	-3821.28	3776.0	11.12	-101.78
15	-3725.84	3673.5	-85.85	-162.58
16	-3725.84	3657.9	53.45	-419.33
17	-2214.41	2188.6	-37.07	-415.73
18	-2214.41	2145.2	119.94	-549.77
19	-1913.79	1909.9	38.22	284.81
20	-2062.91	2061.3	-45.93	-263.80
21	1665.24	-1502.1	-27.71	291.43
22	671.07	-559.6	-8.61	242.87
23	-323.11	387.2	-6.45	112.93
24	-1317.28	1334.8	-18.01	-68.27
25	1814.37	-1643.3	31.52	-245.38
26	820.19	-701.6	10.09	-211.29
27	-173.98	245.2	5.90	-91.46
28	-1168.16	1192.7	15.41	83.79
29	0	-38.4	98.56	243.7
30	-351.49	286.2	83.37	132.79
31	0	-32.5	51	108.77
32	-351.49	287.9	24.56	64.52
33	0	-32.5	-4.14	45.60
34	-351.49	287.9	-33.22	28.28
35	0	-32.8	-59.31	-23.30
36	-351.49	286.0	-92.16	51.27
37	0	-41.7	-105.53	-176.54

TABLE 3.3 NATURAL FREQUENCIES

MODES	NATURAL FREQUENCIES (HETZ) FOR PIN JOINT	NATURAL FREQUENCIES (HETZ) FOR RIGID JOINT
1	10.53	14.2
2	27.05	35.23
3	49.3	62.43
4	53.91	66.14
5	81.29	74.89
6	94.34	80.97
7	110.16	83.29
8	123.34	86.72
9	157.41	88.95
10	158.95	92.5
11	189.64	97.75
12	189.64	98.95
13	197.22	101.65
14	218.08	104.39
15	245.23	107.26
16	261.83	111.32
17	300.37	118.05
18	305.41	127.88
19	373.05	130.31
20	374.07	140.95
21	377.43	144.52
22	377.60	160.20
23	379.11	162.85
24	381.67	172.04

25	385.5	185.48
26	395.32	195.40
27	396.69	202.28
28	396.83	209.61
29	407.4	214.54
30	438.82	227.82
31	465.14	236.04
32	482.51	265.20
33	517.5	274.19
34	519.48	301.87
35	539.95	306.26
36	556.86	324.83
37		345.65
38		379.04
39		385.49
40		387.29
41		400.85
42		412.11
43		423.57
44		428.41
45		438.17
46		456.01
47		458.68
48		464.09
49		472.53
50		477.45
51		491.43
52		502.20
53		504.27
54		552.88
55		594.02
56		620.51

3.3 DISCUSSION OF RESULTS

The analysis of rigid jointed plane truss structures using finite element analysis procedure carried out in this work was done in two phases. The first phase took into account the static analysis of plane truss structure with pin joints and that with rigid joints. The static analysis parameters are displacements, support reactions, axial stresses and bending stresses in terms of rigid jointed plane truss structure. The second phase dealt with modal analysis of both pin jointed and rigid jointed trusses. In this phase, parameters such as natural frequency mode shapes(amplitudes of vibration) of the structure were sort.

ROBOT structural analysis software was used to model and analyze the same plane truss analyzed in this work with the assumption of pin joints and the results obtained from manual analysis with the aid of MATLAB program agreed with insignificant difference.

Lastly, finite element analysis formulations were used to analyze plane trusses with uniform loading and out of joint loading.

3.31 STATIC ANALYSIS

3.311 DISPLACEMENTS

The displacements obtained at each joint for pin jointed plane truss were close to those obtained using rigid joints except for the rotation displacement which is only applicable to rigid joints and not in pin joints. For example, in numerical example 1, node 2 has horizontal displacement (U_5) =-34.98mm and vertical displacement (U_6) =-522.6mm while for the same node in numerical example 2 (rigid jointed), horizontal displacement (U_2) =-17.65, vertical displacement (U_{23}) = -510.66mm and rotation(U_{43}) = -71.328

3.312 AXIAL STRESSES

Axial stresses for both pin and rigid joints were approximately the same. The axial forces are

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 4 Apr. 2021, pp: 429-459 www.ijaem.net ISSN: 2395-5252

slightly smaller in most members when rigid joints were assumed because a portion of the load is transmitted by shear and bending. This can be verified from results from numerical example 1 and 2. For example, member 2 has axial stress of 1195mpa when pin joint was assumed and 1157.6mpa when rigid joint was used.

3.313 BENDING STRESSES

It is shown from the results that bending stresses which is significant when plane trusses are analyzed in its true state of rigid joints are smaller than the axial stresses for the same member and the same condition of loadings. For example member in numerical example 2 when rigid joint was assume has bending stress of 488mpa while axial stress for that of pin jointed is 1195mpa.

3.314 PLANE TRUSSES WITH UNIFORMLY DISTRIBUTED LOADS AND OUT OF JOINT LOADINGS

Tutorial 3 showed how finite element formulations can be used to analyze plane trusses with loadings apart from the usual assumption of point loads only at the joints. The results obtained for displacements, supports reactions, axial stresses and bending stresses were highly reliable.

3.32 MODAL ANALYSIS

In modal analysis, the natural frequency and modes shapes of the structure were sort. The fundamental natural frequency of the structure when pin joints were assumed is 10.53cycles/seconds and 14.1997cycles/seconds when rigid joints were assumed for the same structure geometry and composition. It therefore means that for plane trusses of the same geometry and composition, resonance will deem to have occurred faster in the one with pin joints than the one with rigid joints. Figures 3.3 and 4.4 also show the difference in the fundamental mode shapes which entails that figure 3.4 shows a sharp resonating frequency within a very shorter time while figure 3.4 resonate senosoidally.

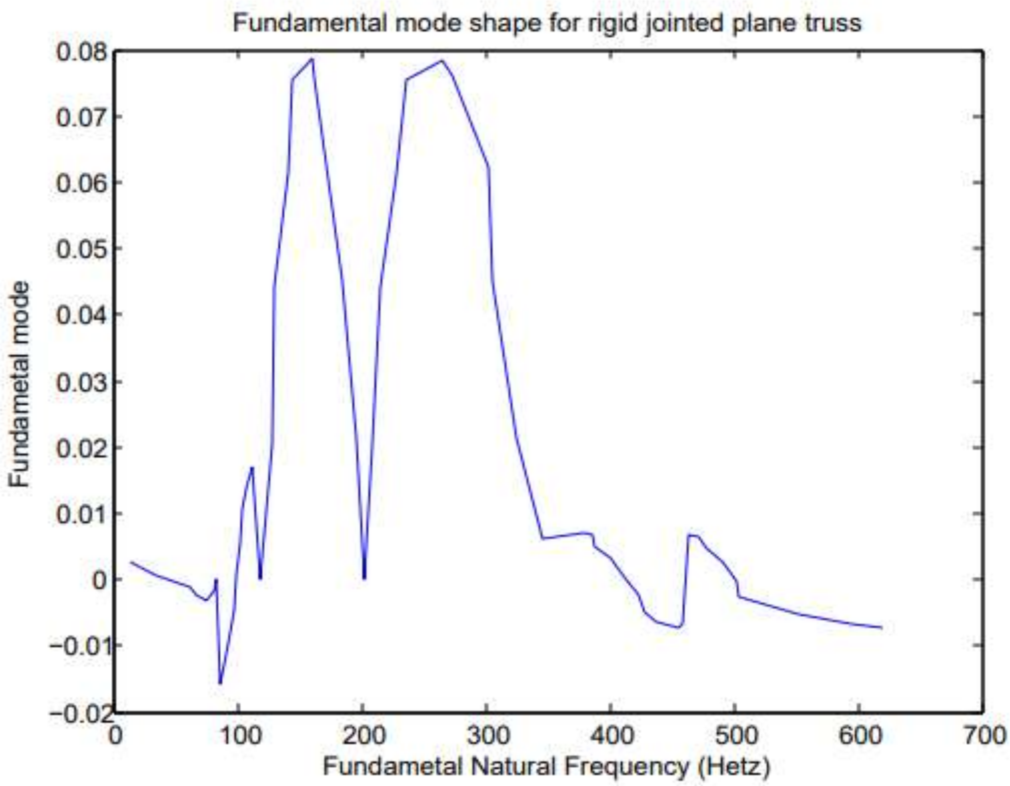


FIG. 3.3 RIGID JOINTED TRUSS

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 4 Apr. 2021, pp: 429-459 www.ijaem.net ISSN: 2395-5252

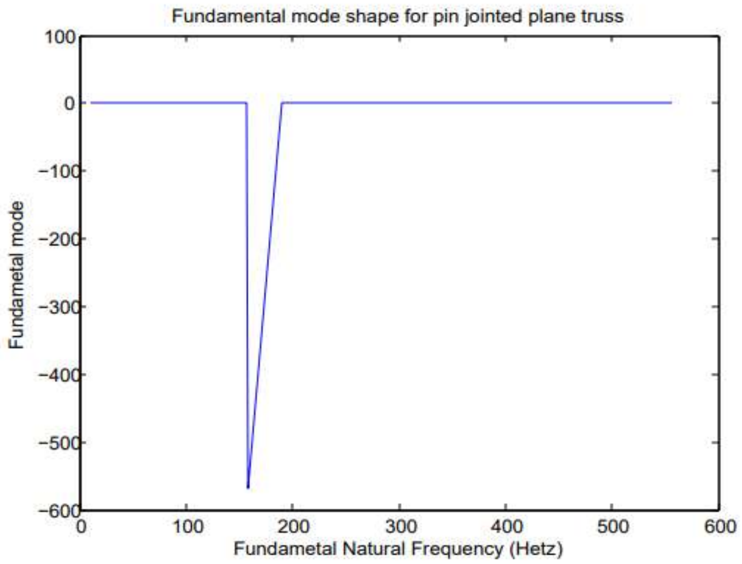


FIG.4.4 PIN JOINTED TRUSS

II. CONCLUSIONS AND RECOMMENDATIONS

The results of the analysis and methodology of work show that;

(a) There was clear understanding in the use of finite element analysis techniques in analysis of plane truss structures..

(b) Comparison of results were achieved in both cases of pin and fixed - joints. Smaller axial stresses were obtained for the same members when fixed joint was used in the analysis. In the case of dynamic analysis, it is seen from the results that the structure where frictionless pin joints are assumed resonate faster than those with rigid joints, therefore making the structure not economical during damping and also not given a true mode shape of the structure.

(c) Finite element analysis techniques were properly used in formulating equations of strain and stresses in rigid jointed plane trusses.

(d) .Finite element analysis techniques interpolation functions were successfully used in deriving equivalent nodal forces for plane trusses subjected to uniformly distributed loads and point loads acting at any portion of the truss member other than the joint.

(e) MATLAB program was developed and run for static and dynamic analysis for both pin and rigid joint connections.

(f) It is also concluded that plane truss structures can be best analyzed in its true state of loading other than assuming loads to only act at joints so that results obtained can represent the true service condition.

Therefore, the following recommendations are made;

(a) Plane truss structures should be analyzed in its true state rather than the assumptions of frictionless pin joints and loads acting only at the joint for the purpose of simplifying the analysis so as to achieve more reliable results.

(b) Writing of simple MATLAB programs should be encouraged in analyzing plane truss structures so that analysis can be simplified in its true state rather than used simplified assumptions.

(c) Plane truss structures should be analyzed in it true state of loading so as to have true deformation of the structure.

Finally, this work has been able to contribute the following to knowledge:

1. Development of finite element model for the analysis of fixed jointed plane trusses.
2. Showing the importance of analyzing plane trusses in its true joint condition other than assuming simplified conditions by comparing the results between pin and rigid jointed plane trusses.
3. By using finite element analysis techniques interpolation to develop equivalent nodal forces for all kinds of loadings on the truss members.

REFERENCES

[1] Archer, J. "Consistent Mass Matrix Formations for Structural Analysis using Finite Element Techniques" Journal of the American Institute of Aeronautics and Astronautics 3, no. 10, 1965.

[2] Bathe, K. J., Finite Element Procedures, Englewood Cliff, Prentice Hall, NJ, 1996.

[3] Beer, F. P. and Johnson, E. R., Vectors Mechanics for Engineers, Static and Dynamics, 6th Ed., McGraw-Hill: New York, 1997.

[4] Budynas, R., Advanced Strength and Applied Stress Analysis; 2nd ed., McGraw-Hill, New York, 1998.

[5] Chana, S., Vibration and Modal Analysis of Steel frames and Semi-rigid Connections, Engineering Structures. 16, (2003). Pp.25-31

[6] Cheng, F. K., Effects of Shearing Deformation on Natural Frequencies of Rigid Jointed Structures, McGraw-Hill, 1975. Pp. 9-10

[7] Clough, R. W. "The Finite Element Method in plane Stress Analysis" Proceedings, American Society of Civil Engineers, Second Conference on Electronic Computation, Pittsburgh,1990.

[8] Cook, R. D., Malkas, D. S., Plesha, M. E. and Witt, R. J., Concepts and Application of Finite Element Analysis 4th ed. John Wiley and Sons, 2002.

[9] Crisfield, M. A., Finite Element and Solution Procedures for Structural Analysis, 1: Linear Analysis, Pineridge Press, 1986.

[10] Dadeppo, D., Introduction to Structural Mechanics and Analysis; Prentice Hall, NJ, 1999.

[11] Dawe, D. J. Matrix and Finite Element Displacement Analysis of Structures, Oxford University Press, New York, 1984.

[12] Ekere, J. T. and Osadebe, N.N. (Project Supervisor), Determination of Stiffness for Timoshenko Beams with infinitely Rigid Ends and Their Applications in Analysis of Frames, M. Eng. (Structural Engineering) Project, UNN, 1998.

[13] Gallagher, R. H., Finite Element Analysis Fundamentals; Prentice-Hall, NJ, 1975.

[14] Gao, W., Interval natural Frequency and Mode Shape Analysis for Truss Structures with Interval Parameters, Finite Element in Analysis and Design. 42, (2006): pp 471-477

[15] Ginsberg, J. H., Advanced Engineering Dynamics, 2nd ed., Cambridge University Press, New York, 1985.

[16] Hinton, E. and Owen, D. R., Finite Element Programming; Academic Press, 1979.

[17] Hughes, T. J., Finite Element Method linear Static and dynamic Finite element Analysis, Prentice-hall, NJ, 1987.

[18] Hutton, V. D., Fundamentals of Finite Element Analysis; McGraw Hill, New York, 2004.

[19] Irons, B. M. and Ahmad, S., Techniques of finite elements; Elis Harwood, Chichester, 1980.

[20] Inman, D. J., Engineering Vibration, 2nd ed., Prentice-Hall, NJ, 2001.

[21] Kiusalaas, J., Numerical methods in Engineering with MATLAB; Cambridge University Press, New York, 2005.

[22] Kreyszig, E., Advanced Engineering Mathematics, 8th ed., John Wiley and Sons, Inc., New York, 2005.

[23] Mau, T. S., Fundamentals of Structural Analysis, the Library of Congress, New York, 2003.

[24] Maurinia C., Porfirib M. and Pougeta J., Numerical Methods for Modal Analysis of stepped piezoelectric beams, Journal of sound and vibration, 298, (2006): pp 918-933

[25] MATLAB Reference Manual 2nd ed.; MathWork Inc., Natrick, 1997.

[26] Onate, E., Structural analysis with the Finite Element Method Linear Statics vol. 1; International center for Numerical Methods, Barcelona, 2009.

[27] Paz, M., Structural Dynamics: Theory and Computation; Van Nostrand Reinhold Company, New York, 1980.

[28] Rajput, R. K., Strength of Materials: Mechanic of Solids S. Chand and Company Ltd, New Delhi, 2006.

[29] Stein, E., Ruter, M and Ohninomus, S., Adaptive Finite Element Analysis and Modelling of Solids and Structures: Findings, Problems and Trends; Int. J. Num. Meth. Engrg., 60, 103-138, 2004.

[30] Taig, O. C., Structural Analysis by the Matrix Displacement Method, Electric aviation Report, No. 5017, 1961.

[31] Timoshenko, S. P., and Young, D. M., Theory of Structures 2nd Ed. McGraw-Hill, New York, 1965.

[32] Topping, B. H. and Khan, A. I. Parallel Finite Element Computations; Saxe-loburg Publications, edingburg, 1996.

[33] Utku, S., " Explicit Expressions for Triangular Torus Element Stiffness Matrix" AIAA Journal, 1174-1176, 1968.

[34] Weaver, W.(Jr) and Johnson, P. R., Finite Elements for Structural analysis; Prentice-Hall, NJ, 1984.
[35] Wilkinson, J. H. and Reinsch, C., Linear Algebra: Hand book for Automatic Computation, vol. 2; Springer- Verlag, Berlin, 1971.
[36] Xu, Y. and Zhang, W., Modal Analysis and Seismic response of Steel Frames with connection Dampers, Engineering structures, (2001): pp385-396