INTERNATIONAL JOURNAL OF CIVIL ENGINEERING RESEARCH AND DEVELOPMENT (IJCERD)

ISSN 2248 – 9428(Print)
ISSN 2248 – 9436(Online)

Volume 5, Number 1, January - April (2015), pp. 01-08

IJCERD

STRUCTURAL HEALTH MONITORING OF 2D FRAME STRUCTURE UNDER STATIC LOAD USING MATLAB

[1]Samik Banerjee

[2]Prof Priyabrata Guha

ABSTRACT

Human society spent many years after civilization and especially after invention of modern civil engineering. So, now a days structural health monitoring i.e. to check the health condition and tolerance of a structure after loss of area due to aging is very important.

In this modern era most of the buildings are designed in frame structure. So, this project deals with two dimensional analysis of plane frame under arbitrary size, support and load conditions. Finite Element method has been used for analysis here. Static response of the plane frame element was obtained by writing program on MATLAB. A study of nodal displacement at initial condition and after aging that is after area loss condition has been done with the written MATLAB code.

INTRODUCTION

Qualitative and non-continuous methods have long been used to evaluate structures for their capacity to serve their intended purpose. The process of implementing a damage detection and characterization strategy for engineering structures is referred to as **Structural Health Monitoring** (**SHM**).

In engineering design, the real behaviour of a structure is provided by determining geometrical connection model well. Thus is the importance of load or force analysis of framed structures under various conditions. The concept of finite element method is used for numerical analysis where in the model at hand is subdivided into components called finite elements and response of each element is expressed in terms of a finite number of degrees of freedom characterized as the value of an unknown function. We studied the various properties of a plane frame element and developed *MATLAB program for two dimensional numerical analysis of a plane frame under static load*. The analyses made have been done on the assumption that the joint connections are fully rigid.

Key Words: 2D Plane Frame, Finite Element Method, MATLAB.

OBJECTIVE AND SCOPE

The objective of the thesis work is *Structural Health Monitoring* of frame structure analytically. For this purpose a *MATLAB program will be developed* which is a common program for any kind of 2D plane frame of single bay and single storey under static loading. We will use *Finite Element Method* for the analysis.

Scope of this work is that we only need load and size data of plane frame structure to analyse. Horizontal and vertical displacements and reactions at each node, and bending moment, shear force, axial force values of each element will have been obtained for single bay and single storey in short time. For this paper work a plane frame structure with arbitrary value has been taken.

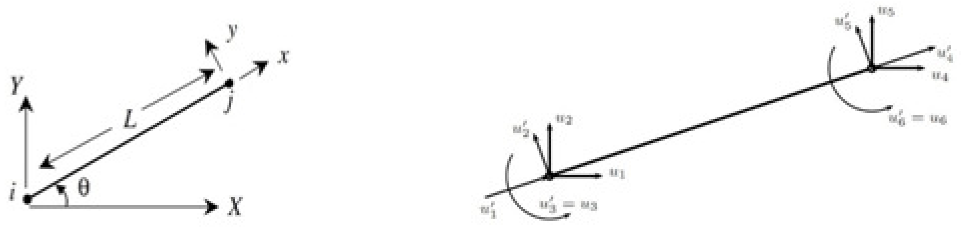
STRUCTURAL HEALTH MONITORING (SHM):

The process of implementing a damage detection and characterization strategy for engineering structures is referred to as **Structural Health Monitoring** (SHM). Here damage is defined as changes to the material and/or geometric properties of a structural system which includes changes to the boundary conditions and system connectivity and this will adversely affect the system's performance. For long term SHM, the output of this process is periodically updated information regarding the ability of the structure to perform its intended function in light of the inevitable aging and degradation resulting from operational environments. In other word Structural health monitoring (SHM) is a promising field with widespread application in civil engineering.

For this paper work we consider only the area reduction as bad health condition of a plane frame structure with arbitrary value and find out the changes of nodal displacement and changes of axial force, shear force, bending moment in each element to take necessary action for the structural good health.

PLANE FRAME ELEMENT

A plane frame element is a two-dimensional finite element. An element has two nodes and each node has three degrees of freedom, two displacement in axes and one rotation. So, each plane frame element has six degrees of freedom. It is expressed in both of the local and global coordinates. The angle between local and global axes is $\boldsymbol{\theta}$. In the case of plane frame, all the members lie in the same plane and are interconnected by rigid joints. At a cross-section, the internal stress resultants at a cross-section of a plane frame member consist of bending moment, shear force and an axial force. The significant deformations in the plane frame are only flexural and axial. Initially, the stiffness matrix of the plane frame member is derived in its local co-ordinate axes and then it is transformed to global co-ordinate system. For the case of plane frames, members are oriented in different directions and hence before forming the global stiffness matrix it is necessary to refer all the member stiffness matrices to the same set of axes. This is achieved by transformation of forces and displacements to global co-ordinate system.



(a) **(b)**

Fig.1 Plane Frame Element

In Fig.1(a) the L length element has two node i.e., i and j. the element lied in local axes x,y with respect to θ, the angel with global axes X,Y. In fig.1(b) the element shows three degrees of freedom at each node i.e., u_1, u_2 and u_4, u_5 are the displacement in local axes and $u_1^{'}$, $u_2^{'}$ and $u_4^{'}$, $u_5^{'}$ are the displacement in global axes and u_3, u_6 and $u_3^{'}$, $u_6^{'}$ are the rotation in local and global axes respectively.

ELEMENT STIFFNESS MATRIX

The plane frame element has modulus of elasticity *E*, moment of inertia *I*, cross-sectional area *A*, and length *L*. Each plane frame element has two nodes and is inclined with an angle θ measured counter clockwise from the positive global *X* axis as shown in Fig.8(a). Let $C = cos\theta$ and $S = sin\theta$ and k = element stiffness matrix. Then k is given as,

$$k = \frac{E}{L} \times \begin{bmatrix} AC^2 + \frac{12I}{L^2}S^2 & \left(A - \frac{12I}{L^2}\right)CS & -\frac{6I}{L}S & -\left(AC^2 - \frac{12I}{L^2}S^2\right) & -\left(A - \frac{12I}{L^2}\right)CS & -\frac{6I}{L}S \\ \left(A - \frac{12I}{L^2}\right)CS & AC^2 + \frac{12I}{L^2}C^2 & \frac{6I}{L}C & -\left(A - \frac{12I}{L^2}\right)CS & -\left(AS^2 - \frac{12I}{L^2}C^2\right) & \frac{6I}{L}C \\ -\frac{6I}{L}S & \frac{6I}{L}C & 4I & \frac{6I}{L}S & -\frac{6I}{L}C & 2I \\ -\left(AC^2 - \frac{12I}{L^2}S^2\right) & -\left(A - \frac{12I}{L^2}\right)CS & \frac{6I}{L}S & AC^2 + \frac{12I}{L^2}S^2 & \left(A - \frac{12I}{L^2}\right)CS & \frac{6I}{L}S \\ -\left(A - \frac{12I}{L^2}\right)CS & -\left(AS^2 - \frac{12I}{L^2}C^2\right) & -\frac{6I}{L}C & \left(A - \frac{12I}{L^2}\right)CS & AC^2 + \frac{12I}{L^2}S^2 & -\frac{6I}{L}C \\ -\frac{6I}{L}S & \frac{6I}{L}C & 2I & \frac{6I}{L}S & -\frac{6I}{L}C & 4I \end{bmatrix}$$

It is clear that the plane frame element has six degrees of freedom – three at each node (two displacements and a rotation). The sign convention which used is that displacements are positive if they point upwards and rotations are positive if they are counter clockwise.

For a frame structure with ***n***nodes, the global stiffness matrix ***K***will be of size **3*n*×3*n***(since we have three degrees of freedom at each node).

After obtaining the global stiffness matrix ***K,*** we have the following structural equation:

[K]{U} = {F}

where *U* is the global nodal displacement vector and *F* is the global nodal force vector. At this step the boundary conditions are applied to the vectors *U* and *F*. Then the matrix is solved by partitioning and Gaussian elimination. Finally the unknown displacements and reactions are found. Then nodal force vector is obtained for each element as follows:

{f} = $[k^{/}][R]$ *{u}*,

where *{f}*is the 6 × 1nodal force vector in the element and $u$is the 6 × 1 element displacement vector. The matrices $[k^{/}]$and $[R]$ are given by the following:

$$[k^{/}] = \begin{bmatrix} \frac{EA}{L} & 0 & 0 & -\frac{EA}{L} & 0 & 0 \\ 0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{4EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{2EI}{L} \\ -\frac{EA}{L} & 0 & 0 & \frac{EA}{L} & 0 & 0 \\ 0 & -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & 0 & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{2EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix} \qquad [R] = \begin{bmatrix} C & S & 0 & 0 & 0 & 0 \\ -S & C & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & C & S & 0 \\ 0 & 0 & 0 & -S & C & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

Where C=cosθ; S=sinθ

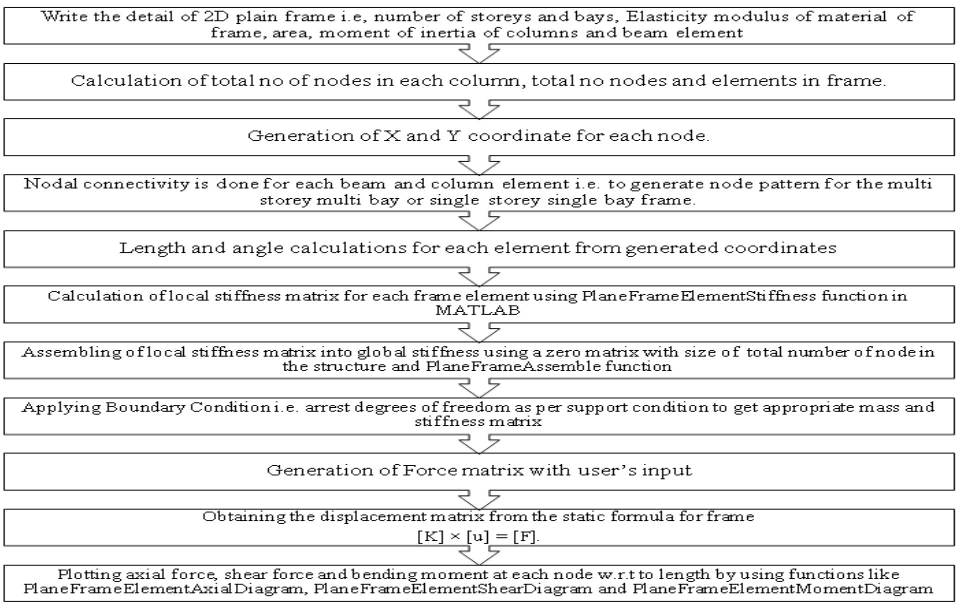
The first and second elements in each vector {u} are the two displacements while the third element is the rotation, respectively, at the first node, while the fourth and fifth elements in each vector are the two displacements while the sixth element is the rotation, respectively at the second node.

MATLAB

The MATLAB programming language is useful in exhibiting how to program the finite element method due to the fact it allows one to very quickly code numerical methods and has a vast predefined mathematical library and also due to the fact that matrix (sparse and dense), vector and many of linear algebra tools are already defined and the developer can focus entirely on the implementation of the algorithm not defining these data structures.

For this thesis work all the various steps of *Finite Element Method* analysis i.e., writing the element stiffness matrices, assembling global stiffness matrix, applying boundary condition, solving the equation, post processing are performed by using MATLAB code.

STEPS OF FRAME ANALYSIS INCLUDING *MATLAB*



SHM OF ARBITRARY 2D FRAME USING MATLAB

This section deals with arbitrary 2D plane frame analysis and its health monitoring. First of all it has been considered a 2D plane frame with arbitrary dimension load and support condition. Then the frame has been analysed with *Finite Element Method* (i.e. *Direct Stiffness Method*) using MATLAB and got nodal displacement of each element. After analysing the frame, the area has been reduced, due to the bad health condition of the structure. In the same condition the frame has been again analysed and again the nodal displacement of each element takes place. Then the graph of those 2different types of values has been plotted.
A MATLAB program has been written which will give this kind of analysis results. It will help to get the nodal displacements and plotting axial force, bending moment and shear force at each node w.r.t length when user input the data of frame like each element length, vertical and horizontal loads and rotation at each nodes.

SINGLE BAY SINGLE STOREY ARBITRARY FRAME

An arbitrary concrete frame has been considered with 3 element of 4 meter length each and 2 ends fixed. Loads are shown in Fig.2 below.
Assume M-25 concrete, so the characteristic strength of concrete (f_{ck}) is 25 $N/_{mm^2}$ or 25000 $KN/_{m^2}$

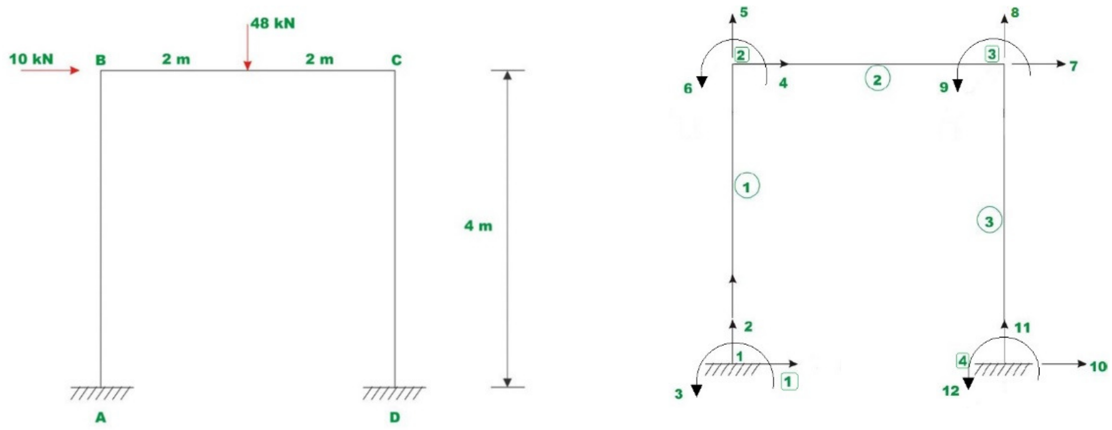


Fig. 2(a). Single bay Single Storey Plane Frame **Fig.2(b).** Node and Member Numbering

The plane frame is divided into three beam elements as shown in Fig.2. The numbering of joints and members are also shown. The possible degrees of freedom (**DOF**) at nodes are also shown in the figure.

Now the element stiffness matrix has been formulated in local co-ordinate system and then transformed it to global coordinate system. In the present case three degrees of freedom are considered at each node. The origin of the global co- ordinate system has been taken as the direction of DOF's direction at$A$(node 1).

Length, start node – end node and angle of the member (**θ**) w.r.t global co-ordinate system is given in Table.1 below

Table. 1. Details of Each Element

ELEMENT No.	ELEMENT LENGTH (in m.)	START NODE No.	END NODE No.	ANGLE θ
1	4	1	2	90^0
2	4	2	3	0^0
3	4	3	4	270^0

Fixed end action due to *static load* and equivalent joint loads at nodes are shown below in Fig.3(a) and Fig.3(b).

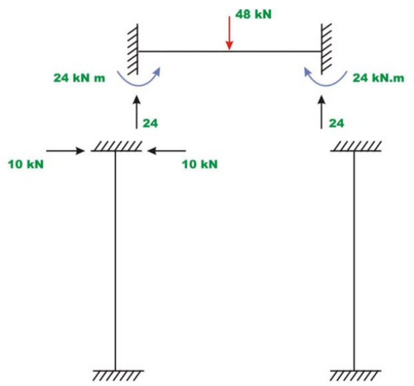


Fig. 3(a). Fixed end action due to external static loads

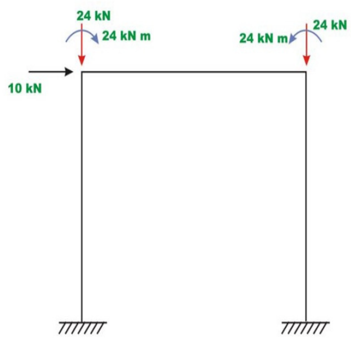


Fig. 3(b). Equivalent joint loads

Fig.3(b) shows the vertical and horizontal loads and rotation at nodes. This values are used as input for the analysis in MATLAB code.

CASE – 1

Width b = 250 mm, depth d = 300 mm for each element.

Result

This header is dealing with the results which obtained through running the MATLAB program by inputting the values of the single bay-single storey arbitrary frame structure. Obtained nodal displacements of each element are listed below in Table. 2.

Table. 2. Obtained Nodal Displacement (in m.)

ELEMENT NO.	START NODE NO.	END NODE NO.	START NODE DISPLACEMENTS			END NODE DISPLACEMENTS		
			X Axis(Dx)	Y Axis(Dy)	Rotation(θ)	X Axis(Dx)	Y Axis(Dy)	Rotation (θ)
1	1	2	0	0	0	0.2642	-0.0032	-0.0722
2	2	3	0.2642	-0.0032	-0.0722	0.2641	-0.0050	0.0627
3	3	4	0.2641	-0.0050	0.0627	0	0	0

Obtained nodal forces of each element are listed below in Table. 3.

Table. 2. Obtained Nodal Forces (in KN)

ELEMENT NO.	START NODE NO.	END NODE NO.	START NODE FORCES			END NODE FORCES		
			X Axis (Fx)	Y Axis (Fy)	Moment	X Axis (Fx)	Y Axis (Fy)	Moment
1	1	2	1798	7.5	23.1	-1798	-7.5	7.1
2	2	3	1.5644	-1.4302	-17.8549	-1.5644	1.4302	12.1342
3	3	4	3867	13.9	34.8	-3876	-13.9	20.9

CASE – 2

In this case, the area of each element has been reduced for the same structure but the external static loads are same. It has been placed to monitor the health of the structure. To check the nodal displacement with same load as case-1 with reduced area.
Reduced width b = 200 mm, reduced depth d = 250 mm for each element

RESULT

This header is dealing with the results which obtained through running the MATLAB program by inputting the values of the single bay-single storey arbitrary frame structure. Obtained nodal displacements of each element are listed below in Table. 2.

Table. 3. Obtained Nodal Displacement (in m.)

ELEMENT NO.	START NODE NO.	END NODE NO.	START NODE DISPLACEMENTS			END NODE DISPLACEMENTS		
			X Axis (Dx)	Y Axis (Dy)	Rotation (θ)	X Axis (Dx)	Y Axis (Dy)	Rotation (θ)
1	1	2	0	0	0	0.5705	-0.0049	-0.1557
2	2	3	0.5705	-0.0049	-0.1557	0.5703	-0.0074	0.1356
3	3	4	0.5703	-0.0074	0.1356	0	0	0

Obtained nodal forces of each element are listed below in Table. 3.

Table. 4. Obtained Nodal Forces (in KN)

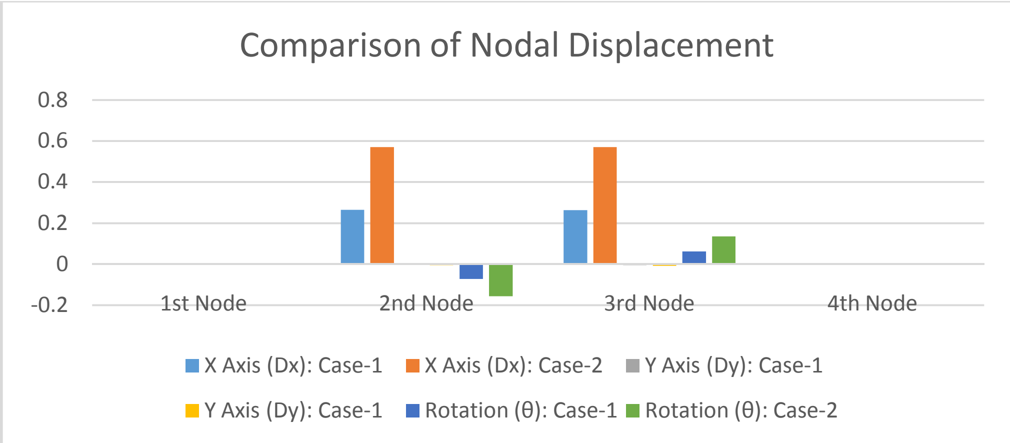
ELEMENT NO.	START NODE NO.	END NODE NO.	START NODE FORCES			END NODE FORCES		
			X Axis (Fx)	Y Axis (Fy)	Moment	X Axis (Fx)	Y Axis (Fy)	Moment
1	1	2	2569	7.6	23.2	-2569	-7.6	7.1
2	2	3	1.5448	-1.4514	-17.8991	-1.5448	1.4514	12.0934
3	3	4	5561	14.1	35.1	-5561	1.4514	-14.1

INFERENCE

Using the MATLAB code, in the static analysis for 2D plane frame, the obtained result is

- Displacements in x-direction and y-direction for each node of element
- Rotation for each node of an element
- Forces in x- direction and y-direction for each node of an element
- Moment for each node of an element
- Graph of axial forces, bending moment and shear forces of each element

From result of case 1, the nodal displacement of each element of the single bay-single storey plane frame structure has been obtained. From the result of case 2, the nodal displacement for the same frame with reduced area and same loads has been secured. So with those values a graph has been plotted for each node to monitor the health of the structure.



CONCLUSION

From the above study and results it has been concluded that SHM is very important field to work for civil or structural engineers. Because after aging, the damage assessment of a structure is very important. SHM by field test method with the help of sensor is a generic method used on regular basis. Not only this, through analytical process with the help of Finite Element Method (FEM) by using computer language MATLAB is also possible and finite element method is very useful in analysis of frame structures. A program has been developed in MATLAB for analysis of two dimension frames under arbitrary loading. The only necessary ingredients are - the data that is area of the elements and the external loads on 2D frame structure, the obtained result is the static properties of two dimensional frame structures such as nodal displacement, nodal force for the each element of the structure, axial force, shear force, bending moment at any storey of frame structure. Only two data is needed, one is initial data of the structure and another one is the damage data that is the decreased area of the structure. Then run these obtained data in the MATLAB program for two times to get the results.

REFERENCES

1. Hoon Sohn, Charles R. Farrar, Francois Hemez and Jerry Czarnecki, “A Review of Structural Health Monitoring Literature 1996 – 2001”. (LA-UR-02-2095)
2. Jack Chessa (2002),Programing the Finite Element Method with Matlab.
3. G. Aditya and S. Chakraborty (2008), “Sensitivity Based Health Monitoring of Structures with Static Response”. (Scientia Iranica, Vol. 15, No. 3, pp 267-274)
4. http://nptel.iitm.ac.in/courses/Webcoursecontents/IIT%20Kharagpur/Structural%20A na lysis/pdf/m4l30.pdf , ‘Direct Stiffness Method: Plane Frames’
5. MATLAB Codes for Finite Element Analysis by A.J.M. Ferreira
6. Plane Frame FEA Solution via Matlab by J.E. Akin