International Journal For Technological Research In Engineering
Volume 3, Issue 9, May-2016 *ISSN (Online): 2347 - 4718*

FINITE ELEMENT ANALYSIS PROGRAM OF FRAMES

Monica V. Pathak[1], Asst. Prof. G. B. Bhaskar[2]
[1]Student, M.Tech, [2]Associate Professor
Department of Civil Engineering, G. H. Raisoni Academy of Engineering & Technology, Nagpur, Maharashtra, India.

Abstract: The present project involves discovering how to develop a Matlab program to analyze a frame structure. The goal is to use finite element method in Matlab and solve frame structures which are used on a large scale. Finite element method is extensively used for analysis of various structures. Using Matlab defining codes, program for frame analysis is written to obtain desired results. The results obtained through frame analysis in developed Matlab program were compared with another structural analysis software, Staad –pro. Thus using the designed program analysis of frames can be done with ease and expeditiously.

Keywords: Finite element method, frame structure analysis, MATrix LABoratory(MATLAB), stiffness, dispalcements, reactions.

I. INTRODUCTION

With the rapid growth of urban population in both the developing and the industrialized countries, framed structures have become a material of choice for residential and commercial construction because of unfortunate scarcity of land. Frame structures are the structures having the combination of beam, column and slab to resist the lateral and gravity loads. These structures are usually used to overcome the large moments developing due to the applied loading. A frame structure has three degree of freedom at each node. Therefore, the total number of degree of freedom is three times the number of nodes. When it comes to analyze the frame for applied loadings, solving it analytically, can be tedious and inaccurate especially when a large number of degree of freedom are acting on the frame structure. Hence, an effective accurate and rapid method to solve such frames was the reason to develop this program. To solve such a high degree of freedom frame, finite element analysis method can be one of the best suited, widely used methods. The ease to compute complicated steps in the solution, is the reason it is used for both analytical methods as well as different programming languages. The finite element method is a tool to solve one dimensional, two- dimensional and three - dimensional structures with approximation instead of solving complicated partial differential equations. Major benefit is, it can efficiently optimizes and validates each design step using fast-solving and derivate the structure in small definable elements. Finite element method uses matrix calculations for solving such small elements which gives rise to complexity to the solution and Matlab is quite efficient in solving such complicated matrices. MATLAB is a high-level language specially designed for dealing with matrices, making it particularly suited for programming the finite element method. It introduces the theory of the finite element method while maintaining a balanced approach between its mathematical formulation, programming in implementation and as its application using commercial software. The computer implementation is carried out using MATLAB for finite element analysis of frames. Till date, various attempts were made to write programs in Matlab using FEM to carry out the analysis particular frame structure. Whereas, this paper put on record a generalized program which can solve any frame structure excluding the inclined member frame structure. This Matlab generated program provides the accessibility to use it with frame structures and numerous examples can be solved by using this program.

II. METHODOLOGY

The method used to analyze the frame is a finite element method. The analysis of a structure by the Finite Element Method can be divided into several distinctive steps. Here a theoretical approach to the method is presented with its different steps.

A. Discretization:

Discretization is the process where the frame structure is being divided into several small elements, connected with nodes. All elements and nodes are numbered to set up a matrix of connectivity. It is vital to remember that the order of nodes and elements are numbered with precision as it affects the computing time. This is because we get a symmetrical, ordered stiffness matrix, whose order depends on the difference in the node numbers for each element. This matrix order is directly connected with the number of calculations which are tackled by programme. Matlab-FEM program has internal numbering that optimizes the matrix order to a minimum by undertaking some internal renumbering of nodes, if they are not optimal.

B. Element properties:

Element properties, Material Properties and Units used in analysis of the frame elements are determined viz. the area (A), element length(L), nodes coordinates, forces at nodes, Young's modulus (E) , moment of inertia (I).

Element length for a frame element is calculated by following equation

$$L = \sqrt{(x_2 - x_1) + (y_2 - y_1)} \qquad (1)$$

C. Assembling:

This process includes the assembly of the overall or global stiffness matrix for the entire frame from the individual element stiffness matrices. The global forces or load matrix

 Copyright 2016.All rights reserved.

[q] from the element nodal vectors is assembled. The most common technique used is called as the direct stiffness matrix [K].
The stiffness matrix equation for each element is computed in this step as

$$k = \begin{bmatrix} \frac{AE}{L} & 0 & 0 & -\frac{AE}{L} & 0 & 0 \\ 0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{4EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{4EI}{L} \\ -\frac{AE}{L} & 0 & 0 & \frac{AE}{L} & 0 & 0 \\ 0 & -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & 0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{4EI}{L} & 0 & \frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix} \quad (2)$$

D. Application of Boundary Conditions:
The boundary conditions according to the support of the frame can be expressed in terms of the displacement vector. These are introduced by setting nodal displacements to known values. They reflect the known values for certain primary unknowns. Imposing the boundary conditions modifies the global equations to get reduced stiffness matrix. Any boundary condition is satisfied by deleting the rows and columns corresponding to the node that has zero deflection. In a frame structure, a node has three degrees of freedom (DOF), which includes two translations and one rotation.

E. Solve for Primary displacements:
The modified global equations are solved for the primary unknowns at the nodes to get displacements using the reduced stiffness matrix.

$$[K][d] = [F] \quad (3)$$

Where:
F = external loading.
K = reduced stiffness matrix.
d = displacement matrix.

F. Computation of reactions at the nodes.
Additional calculations of reaction are carried as follows
The reaction is calculated as

$$R = [K][d] - [q] \quad (4)$$

III. MATLAB PROCEDURE

A. Preprocessing:
In the preprocessing phase, the input file (m.file), which defines material properties, load vector and matrices initializations, essential and natural conditions, nodal loads, the required output, is to be defined by the user.

B. Processing:
The processing section is where the finite element objects i.e. stiffness matrices, force vectors etc. are computed, boundary conditions are enforced and the system is solved.
The stiffness matrix is computed by the following function

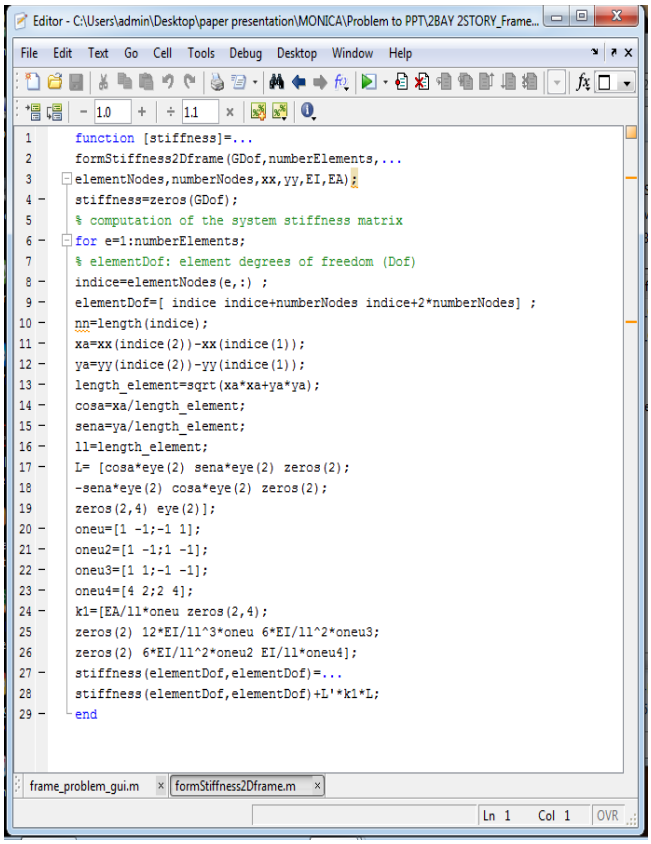


Fig.1 Stiffness matrix function.

Calculation of displacements is carried out with the help of following function.

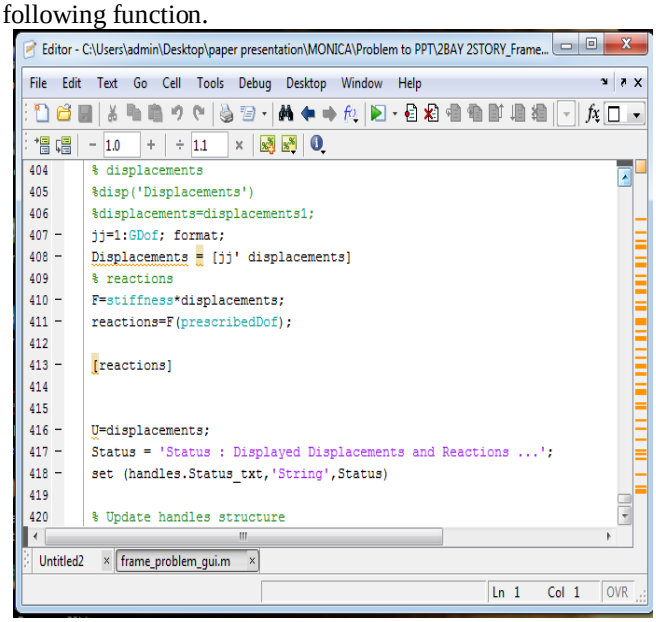


Fig.2 Displacements and reaction function.

C. The post-processing:
In this section the results from the processing section are analyzed.

 Copyright 2016.All rights reserved.

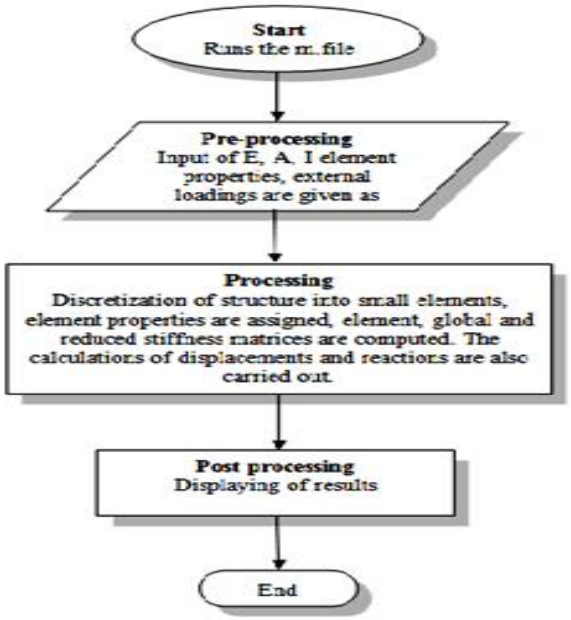


Fig.3 Flowchart of Matlab program execution of frame structure analysis.

IV. PLANE FRAME STRUCTURE TO BE ANALYSED USING DEVELOPED PROGRAM

The frame structures analyzed in this paper are of two types (a) sway frame (b) non-sway frame.

Example 1: Sway frame

The proposed frame structure is 12m wide with 11.5m height. Material considered is steel with modulus of elasticity of steel $2x10^5$ N/mm^2. The moment of inertia (I) is $2x10^8$ mm^4. The area for all members is 4000 mm^2.The structural configuration of proposed structure is depicted in the following figure.

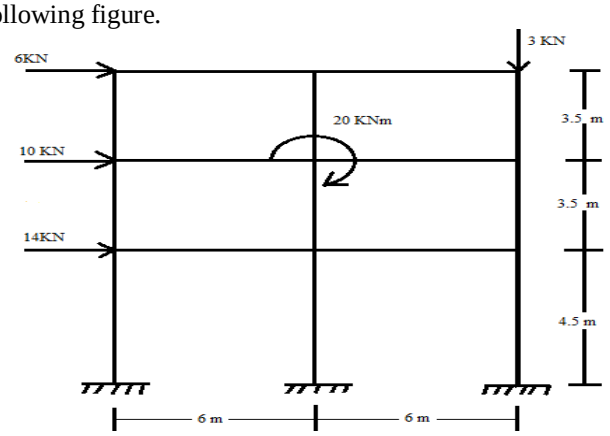


Fig.4 Frame structure to be analyzed.

The inputs are given as joint and support data, material property data, member and nodal load data. Joint data consists of the total number of joints which is 12, and the global coordinates of each joint. The position of each joint is specified by means of the global coordinates of the joint. For a plane frame, the number of degree of freedom of a joint is three giving the calculation of global degree of freedom. The program itself calculates number of nodes and the number of elements. The numbering of the elements is carried out as all the horizontal reactions are numbered serially. After all the horizontal reactions are numbered all vertical reactions are numbered. And lastly all the rotations are numbered. Material property data are defined by storing modulus of elasticity as E. In cross-sectional property data, the cross-sectional areas are stored as variable A and moment of inertia I.

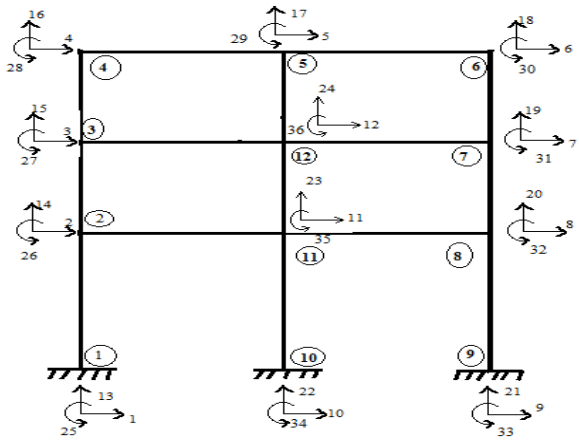


Fig.5 Analytical model of frame structure.

External loads, element nodes, node co-ordinates were assigned with the specific variables. The joint numbers and their corresponding directions of restraints are defined and stored. The degrees of freedom which are restrained are assigned to prescribedDof and the degrees of freedom which are non-restrained are assigned as activeDof. Preparing all the input data required for the analysis of the plane frame in the form of m-file, these files are attached to the program for processing. The analysis of the frame is carried out. The execution and results are as given below. In processing all member stiffness matrices were calculated leading to the compilation of global stiffness matrix. After application of boundary conditions reduced stiffness matrix was obtained by following function as shown in figure. (All input data are taken in N and mm units for processing.

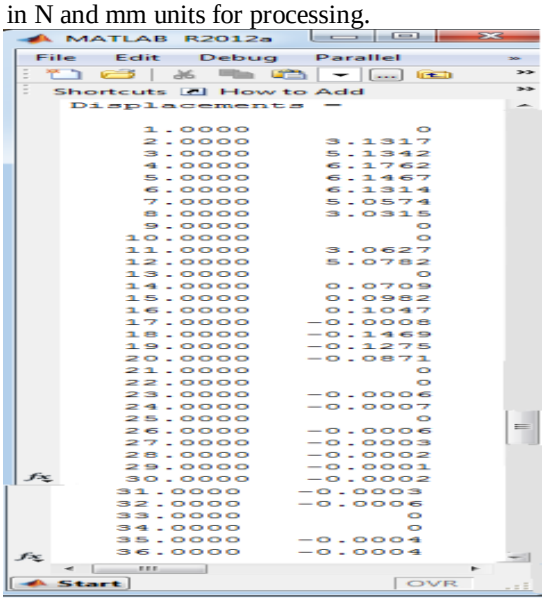


Fig.6 MATLAB scribe file shows the results of displacement.

 Copyright 2016.All rights reserved.

The same frame was analyzed with other software and the results were compared to that of the Matlab developed program. The following table shows the comparison of displacements by the developed Matlab program and Staad-pro. All the values are in millimeter (mm).

TABLE I: COMPARISON OF DISPLACEMENTS

Node number	Displacements (mm)		
	Matlab		Staad -pro
1	X	0	0
	Y	0	0
2	X	3.1317	3.132
	Y	0.0709	0.071
3	X	5.1342	5.134
	Y	0.0928	0.098
4	X	6.1762	6.176
	Y	0.1047	0.105
5	X	6.1467	6.147
	Y	-0.0008	-0.001
6	X	6.1314	6.131
	Y	-0.1469	-0.147
7	X	5.0574	5.057
	Y	-0.1275	-0.127
8	X	3.0351	3.032
	Y	-0.0871	-0.087
9	X	0	0
	Y	0	0
10	X	0	0
	Y	0	0
11	X	3.0627	3.063
	Y	-0.0006	-0.001
12	X	5.0782	5.078
	Y	-0.0007	-0.001

The following table shows the comparison of rotational displacements by the developed Matlab program and Staad-pro.

TABLE II: COMPARISON OF ROTATIONAL MOMENTS

Node number	Rotational displacements (radian)	
	Matlab	Staad-pro
1	0	0
2	-0.0006	-0.001
3	-0.0003	-0.000
4	-0.0002	-0.000
5	-0.0001	-0.000
6	-0.0002	-0.000
7	-0.0003	-0.000
8	-0.0006	-0.001
9	0	0.000
10	0	0.000
11	-0.0004	-0.000
12	-0.0004	-0.000

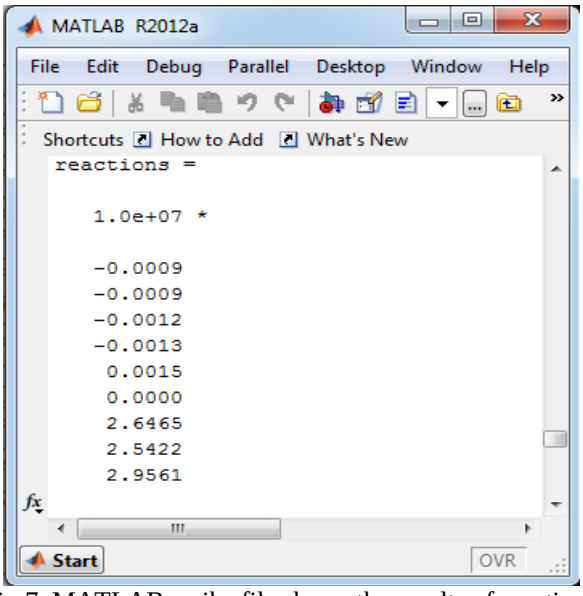


Fig.7. MATLAB scribe file shows the results of reactions

The following table shows the comparison of reactions by the developed Matlab program and Staad-pro. All the horizontal and vertical reactions are in Newton (N) and rotational values are in radian.

TABLE III: COMPARISON OF NODAL REACTIONS

Node number		Reactions	
		Matlab	Staad-pro
1	Horizontal	-9000	-9395.501
	Vertical	-13000	-12600.237
	Rotational	26465000	26465000
2	Horizontal	-9000	-8963.472
	Vertical	15000	15491.735
	Rotational	25722000	25422000
3	Horizontal	-12000	-11641.030
	Vertical	107	108.502
	Rotational	29561000	29561000

Example 2: Non-sway frame

The frame structure is 12m wide with 6m height. Material considered is steel with modulus of elasticity of steel 2.1x105 N/mm2. The moment of inertia (I) is 2x108 mm4. The area for all members is 4000 mm2.The structural configuration of proposed structure is depicted in the following figure.

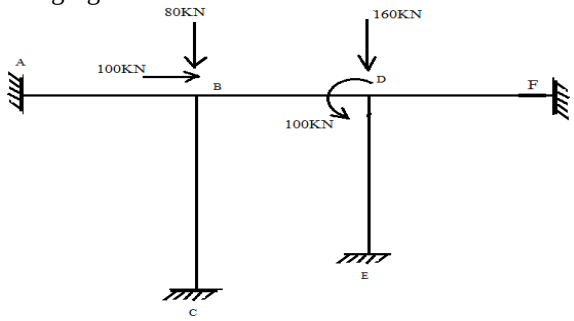


Fig.7 Non-sway frame example

 Copyright 2016.All rights reserved.

Analytical model of this frame was given as:

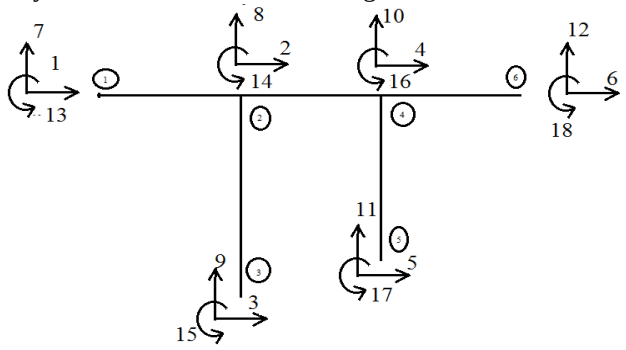


Fig.8 Analytical model of frame structure.

The following table shows the comparison of rotational displacements by the developed Matlab program and Staad-pro.

TABLE IV: COMPARISON OF DISPLACEMENTS

Node number	Displacements (mm)		
		Matlab	Staad -pro
1	X	0	0
	Y	0	0
2	X	0.2977	0.297
	Y	-0.6436	-0.646
3	X	0	0
	Y	0	0
4	X	0.1126	0.113
	Y	-0.7518	-0.752
5	X	0	0
	Y	0	0
6	X	0	0
	Y	0	0

TABLE V: COMPARISON OF ROTATIONAL DISPLACEMENTS

Node number	Rotational displacements (radian)	
	Matlab	Staad-pro
1	0	0
2	-0.0003	-0.000
3	0	0
4	0.0009	0.001
5	0	0.000
6	0	-0.000

The following table shows the comparison of reactions by the developed Matlab program and Staad-pro. All the horizontal and vertical reactions are in Newton (N) and rotational values are in radian.

TABLE VI: COMPARISON OF NODAL REACTIONS

Node numbers	direction	Nodal reactions	
		Matlab	Staad-pro
1	Horizontal	-63000	-623510.601
	Vertical	401	425.377
	Rotational	3946000	3946000
3	Horizontal	1300	1368.950
	Vertical	90099	90102.753
	Rotational	-2043000	-2043000
5	Horizontal	-15150	-15206.108
	Vertical	158000	157870.88
	Rotational	20866000	20866000
6	Horizontal	-24000	-23652.248
	Vertical	-8350	-8398.991
	Rotational	7252000	7252000

V. CONCLUSIONS

We find Matlab a convenient platform for development of computational algorithms, and with the visualization functions of Matlab. This is supported by a general purpose interactive (programming) language. This paper attempts the programming language adoption of FEM for frame analysis. In this paper development of analyzing program of frame structure using FEM in Matlab is achieved. Observing the results obtained from the proposed frame, it shows this Matlab generated program analyses frames having different number of bays and floors. By comparing the results for displacements, rotational moments and reactions, it is observed that the results are acceptable. It can be concluded that designed analysis program with Matlab codes can be implemented to various frame structures to analyze them with meticulousness, with less efforts and in short period of time.

REFERENCES

[1] H. Rahemi, Ph. D. S. Baksh, "Finite difference impulsive response analysis of a frame structure – a matlab computational project-based learning", 2010.
[2] H. Rahami, Ph. D. "Matrix structural analysis (MSA)", University of Tehran-Iran, March 2010.
[3] U. Draz Ahmad, "Development of a Structural analysis program (SABSM) using MATLAB featuring graphical user interface", London university, September 2013.
[4] Yasser B, "Structure analysis program", 2012.
[5] A.J.M.Ferreira, "MATLAB Codes for Finite Element Analysis", Solids mechanics and its application, vol.157., university of Porto, Portugal, 2000
[6] J. Chessa "Programming the finite element method with Matlab", Northwestern University, October 2002.
[7] A. Khennane. "Introduction to finite element analysis using Matlab® and abaqus", University of New South Wales, Australia, 2013.
[8] M. W. Minett, C. Perera, "Programming for civil and building engineers using Matlab," June 2006.
[9] Y. W. Kwon, H.Bang, "The Finite Element Method Using MATLAB", Second Edition Duke University. J.E. Akin, "Plane Frame FEA Solution via Matlab" April 2006.

 Copyright 2016.All rights reserved.