Preliminary Approach of Modal Analysis on MDOF System by using MATLAB for Experimental and Finite Element Validation

D.Kesavan Periyasamy
M.E Structural Engineering student
Department of Civil Engineering
Thiagarajar College of Engineering
Madurai,India
pdkesavan@gmail.com

Dr.R.Ponnudurai
Assistant Professor
Department of Civil Engineering
Thiagarajar College of Engineering
Madurai,India
rpdciv@tce.edu

Abstract— **In the past twenty years, modal analysis has become a significant technology rising and optimizing dynamic characteristics of engineering structures. Not solely has it been recognized in mechanical and aeronautical engineering, however modal analysis has also discovered profound applications for civil and building structures. In general Finite element method is used for calculating modal property of real time structure which is time consumed one also. The main aim of this paper creating a new method for calculating modal property. Then it may compared with experimental modal property for assessment of the structures. This methodology can greatly reduce the running time and iteration time of the finite element modelling, which is can consider as the preliminary approach of modal analysis for experimental validation with respect to any finite element method. In this paper we created analytical approach to proceed modal analysis for getting natural frequency and mode shape by using MATLAB 2014a for a free undamped system. Free undamped structure modal analysis used to calculate the natural frequency and mode shape of the complex structures such as irregular structures.**

Keywords—Free undamped vibration, MATLAB, Coding, Fundamental Frequency, Mode shape, Modal Analyis, Iteration.

I. INTRODUCTION

Modal analysis is the process of determining the inherent dynamic characteristics of a system in forms of natural frequencies, damping factors and mode shapes, and using them to formulate a mathematical model for its dynamic behaviour. The formulated mathematical model is referred to as the modal model of the system and the information for the characteristics are known as its modal data.

Modal analysis is an increasingly more important engineering tool that was first applied around 1940 in the search for a better understanding of aircraft dynamic behaviour. Till the end of the 60's developments were slow and experimental techniques were based on the use of expensive and cumbersome narrow band analogue spectrum analysers. The modern era of modal analysis can be taken as starting at the beginning of the 70's, based upon the commercial availability of Fast Fourier Transform (FFT) spectrum analysers, transfer function analysers (TFA) and discrete acquisition and analysis of data, together with the availability of increasingly smaller, less expensive and more powerful digital computers to process the data.

The natural modes of vibration are inherent to a dynamic system and are determined completely by its physical properties (mass, stiffness, damping) and their spatial distributions. Each mode is described in terms of its modal parameters: natural frequency, the modal damping factor and characteristic displacement pattern, namely mode shape. The mode shape may be real or complex. Each corresponds to a natural frequency. The degree of participation of each natural mode in the overall vibration is determined both by properties of the excitation source(s) and by the mode shapes of the system. Modal analysis embraces both theoretical and experimental techniques. The theoretical modal analysis anchors on a physical model of a dynamic system comprising its mass, stiffness and damping properties. These properties may be given in forms of partial differential equations.

The application of modal analysis is not limited to a specific engineering discipline. In recent years, extensive applications have been found in aerospace engineering, acoustical engineering, mechanical engineering, civil engineering, automotive engineering, ship building, mining, manufacturing, nuclear power plants, transportation, weapon systems and building engineering. Modal analysis and finite element analysis have become two pillars in modern structural dynamics.

Risk in RC structure is accidental torsional effect could occur in the seismic irregular structure. These effect make fail the structure well below the yield limit. Torsional irregularity is one of the major causes of severe damage and collapse of structures during an earthquake. It is due to that the irregular distribution of mass, stiffness and strengths may cause serious damage in asymmetry structural systems. During strong ground motion an asymmetrical building gets structural damage and failure due to the torsional behaviour. Structural Analysis software is used to conduct the modal analysis for getting the torsional modes of the seismic irregular structure. This project first concentrates in understanding the complex behavior of structure under asymmetric form and a study on the influence of the torsional effects on the behaviour of structure is done by using modal analysis. Torsional rotation of the mode shape has been found and then the torsional effect is reduced by two different ways such as Adding Shear wall, Adding Bracing to the irregular RC structure. Thus the final torsional rotation of the Reinforced concrete structure are again being modal analysis and the results of various methods are compared with the basic structure that without any retrofitting technique.

In the multi-storey building or in any system having n degrees of freedom, if principal modes (normal modes) are used as generalized coordinates, the n dynamic equilibrium equations will be uncoupled. Hence each uncoupled equation can be solved independently as each equation contains one degree of freedom only. One can apply any one of the numerical methods to determine the response of a single-degree-of-freedom system. The response of a multipledegrees-of-freedom (MDOF) system is obtained by mode superposition by summing the response of the individual modes. This procedure of dynamic analysis is referred to as the mode superposition method, normal mode method or simply modal analysis.

II. NEED OF ALTERNATIVE APPROACH

A. Structural Finite Element Model Updating for OMA and EMA

Finite Element Modelling is the important parameters in Operational Modal Analysis and Experimental Modal Analysis. By updating the properties of the structure with experimental results gives the actual property of the real structure.

Bijaya Jaishi and Wei-Xin Ren made ambient vibration test on Concrete-Filled Steel Tubular Arch Bridge. That Beichuan River Bridge, located in Xining city, the Capital of Ningxia Province, China, is a half-through arch bridge, with the span of 90 m The cross-sectional dimension of two main concrete-filled steel tubular arch ribs is 650x10 mm. The remaining connecting tubes of superstructure are hollow steel tubes. There are 32 main suspenders of steel wire ropes that are vertically attached on the main arch rib and the floor system is suspended through it. The floor system consists of a 250 mm thick concrete slab supported by cross girders at a spacing of 5 m center-to-center. The typical rectangular cross section of the cross girder is 0.36x1.361 m. The main arch ribs are fixed at two abutments and connected by prestressed strands in the longitudinal direction.

They Modelled A three-dimensional linear elastic finite element bridge was constructed using ANSYS~ANSYS 1999. The arch members, cross girders, and bracing members were modeled by two-node beam elements. All suspenders were modeled by the truss elements. The slab of the bridge was modeled as shell elements. In order to simulate the behavior of connection between the cross girder and bridge deck in the transverse direction of the bridge, spring element is used. The value of spring stiffness is assumed as 500,000 N/m based on the previous experience on a similar bridge. Totally 3,120 nodes, 3,446 elements, and 14,060 active degrees of freedom were recognized in the FE model.

It is very hard to estimate the variation bound of the parameter during updating, it is assumed according to some engineering judgement. The changes of eigenvalues only after 70 iterations.

So that this alternative approach highly reduce the iteration numbers for finding the dynamic property of the structure.

B. Reducing Time Consumption

Finite Element Method heavily depend on the numerical integration. If the mesh is closely spaced then analysis takes a huge time to run. The basics assumption of this MATLAB coding is

- System is considered as the undamped system.
- Degree of freedom is limited to depends upon Number of storey.
- Rule of Mixture is consider for calculating effective Elastic modulus for composite of concrete and reinforcement.
- Translation modes along shorter, longer direction and rotational modes are calculate separately and combine all by American National Standard: "Vibration of Buildings – guidelines for the measurement of vibrations and their effects on buildings," ANSI S2.47-1990 (ASA 95-1990).

Further by consideration damping matrix into it accuracy may be highly improved. This method of analysis is less time consumed but the result is not accurate compare with Finite Element Modelling.

C. Implementation of FEMA 356 code

FEDERAL EMERGENCY MANAGEMENT AGENCY gives new methodology for calculating the column stiffness. Column Stiffness is greatly depend upon the Axial Rigidity, Flexural Rigidity and Shear Rigidity. One advantage by implementing the FEMA code is there is no need of mode merging. The following assumption been made up with implement of FEMA code

- System is considered as the undamped system.
- Degree of freedom is limited depends upon Number of storey.
- Length of the column does not considered.
- The effective stiffness calculated by using empirical equation in 'PRESTANDARD AND COMMENTARY FOR THE SEISMIC REHABILITATION OF BUILDINGS'.

Table 1. Effective Stiffness Value

Table 6-5 Effective Stiffness Values

Component	Flexural Rigidity	Shear Rigidity	Axial Rigidity
Beams—nonprestressed	$0.5E_cI_g$	$0.4E_cA_w$	—
Beams—prestressed	E_cI_g	$0.4E_cA_w$	—
Columns with compression due to design gravity loads $\geq 0.5\ A_gf'_c$	$0.7E_cI_g$	$0.4E_cA_w$	E_cA_g
Columns with compression due to design gravity loads $\leq 0.3\ A_gf'_c$ or with tension	$0.5E_cI_g$	$0.4E_cA_w$	E_sA_s
Walls—uncracked (on inspection)	$0.8E_cI_g$	$0.4E_cA_w$	E_cA_g
Walls—cracked	$0.5E_cI_g$	$0.4E_cA_w$	E_cA_g
Flat Slabs—nonprestressed	See Section 6.5.4.2	$0.4E_cA_g$	—
Flat Slabs—prestressed	See Section 6.5.4.2	$0.4E_cA_g$	—

Note: It shall be permitted to take I_g for T-beams as twice the value of I_g of the web alone. Otherwise, I_g shall be based on the effective width as defined in Section 6.4.1.3. For columns with axial compression falling between the limits provided, linear interpolation shall be permitted. Alternatively, the more conservative effective stiffnesses shall be used.

(source:FEMA356)

III. MATLAB CODING

A. Formation of Mass matrix

Mass Matrix is the diagonal matrix of the number of degree of freedom structure. For an MDoF structural system, usually the mass matrix is a full rank matrix.

$$[M] = \begin{bmatrix} m_{11} & m_{12} & \cdots & m_{1n} \\ m_{21} & m_{22} & \cdots & m_{2n} \\ \cdots & \cdots & \cdots & \cdots \\ m_{n1} & m_{n2} & \cdots & m_{nn} \end{bmatrix}$$

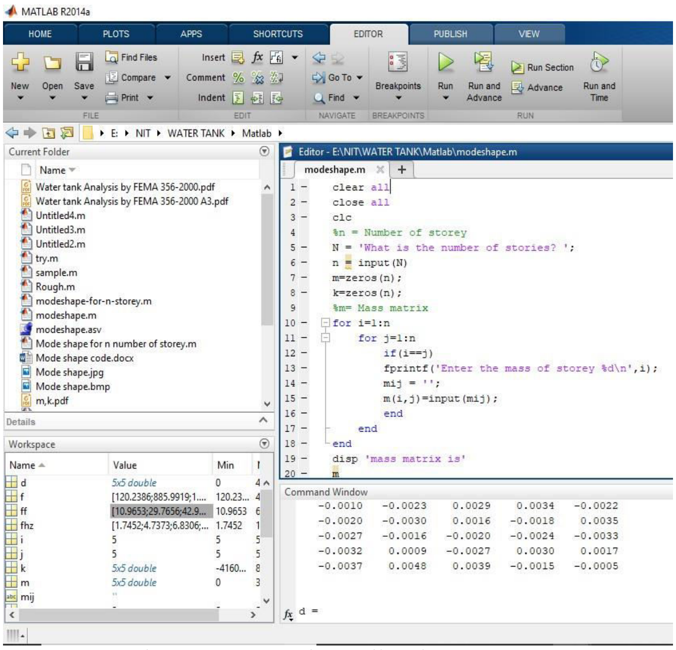

Fig.1 Mass Matrix Coding in MATLAB

Following coding was created in MATLAB for obtaining input and form a mass matrix

```
%m= Mass matrix
for i=1:n
        for j=1:n
                if(i==j)
                fprintf('Enter the mass of storey %d\n',i);
                mij = '';
                m(i,j)=input(mij);
                end
        end
end
disp 'mass matrix is'
m
```

B. *Formation of Stiffness Matrix*

The stiffness matrix of the system can be decomposed using the stiffness values, for example if consider three springs (k1,k2 and k3) connected in series then the stiffness matrix is

$$[K] = \begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_3 \end{bmatrix}$$

The mass matrix [M] and the stiffness matrix [K] are symmetric. The mass matrix is usually positive definite while the stiffness matrix may become semi-positive definite if the system possesses rigid body vibration modes. The orthogonality among incomplete data of an MDoF system is of interest because the dynamic stiffness matrix is usually sparse or banded. both the mass matrix [M] and the stiffness matrix [K] can be diagonalized

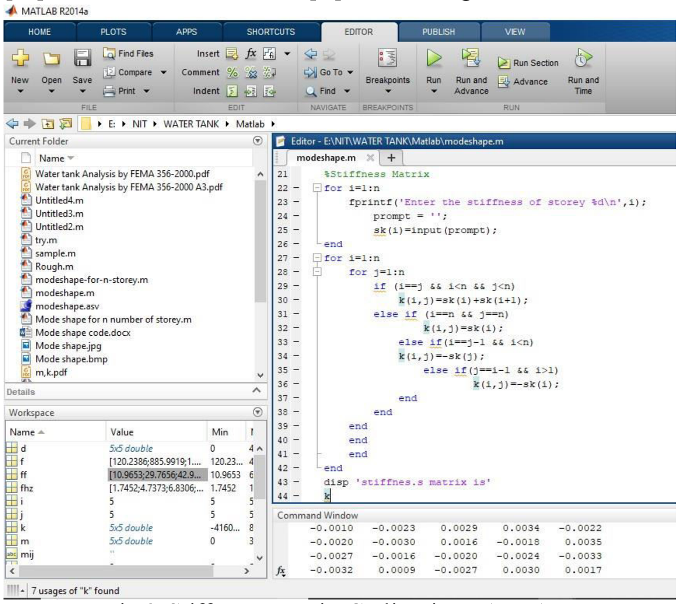

Fig.2 Stiffness Matrix Coding in MATLAB

Following coding was created in MATLAB for obtaining input and form a stiffness matrix

```
%Stiffness Matrix
for i=1:n
    fprintf('Enter the stiffness of storey %d\n',i);
    prompt = '';
    sk(i)=input(prompt);
end
for i=1:n
        for j=1:n
                if (i==j && i<n && j<n)
                k(i,j)=sk(i)+sk(i+1);
        else if (i==n && j==n)
                k(i,j)=sk(i);
        else if(i==j-1 && i<n)
        k(i,j)=-sk(j);
                else if(j==i-1 && i>1)
                k(i,j)=-sk(i);
                end
        end
    end
    end
    end
end
disp 'stiffnes.s matrix is'
k
```

C. *Formation of Eigen value and Eigen vector*

MATLAB has automatic solver option for calculating Eigen Values and Eigen Vectors by using Mass and Stiffness Matrix

Commonly from the equation of motion

$$([K]-\omega^2[M])\{X\} = \{0\}$$

For this equation to have non-zero solution {X}, matrix $([K] - \omega^2[M])$ has to be singular so that:

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

This is the characteristic equation of the system. The solutions of this equation are its natural frequencies. Where ω^2 is the eigenvalue and {X} the eigenvector. The eigenvalue is actually the square of the natural frequency of the systemand the eigenvectorthe mode shape.

IV. MODAL ANALYSISPROBLEM BY USINGMATLAB CODING

Modal analysis is used to find the natural frequency and corresponding deflection shape of the structure.

Consider 5 storey structure where
m1=m2=m3=m4=32500kg
m5=18500kg
k1=k2=k3=k4= 41.6 x 10^6 N/m
k5= 20 x 10^6 N/m

Commonly 5x5 matrix is very difficult to analyse by manual. By using this coding MATLAB automatically generate mass matrix and stiffness matrix by using of these eigen values and eigen vectors calculated. After that the separate coding has been used for plot mode shape in terms of number of storeys and deflection.

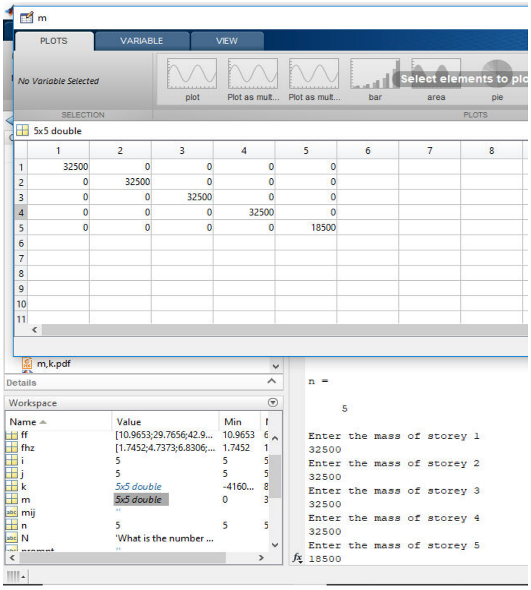

Fig.3 Formation of Mass Matrix in MATLAB

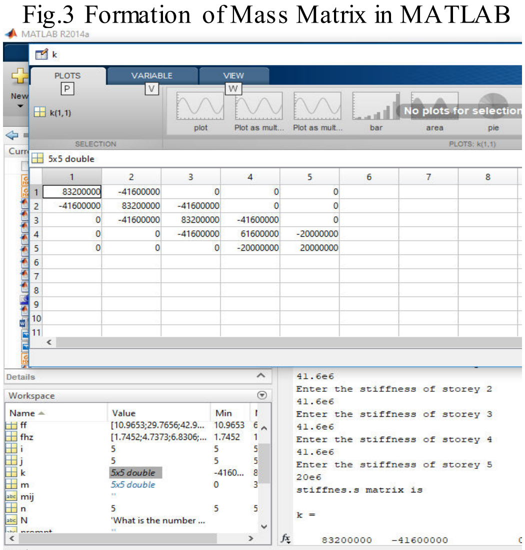

Fig.4 Formation of Stiffness Matrix in MATLAB

Eigen values and Eigen vectors are represent as the v and d in the matlab coding

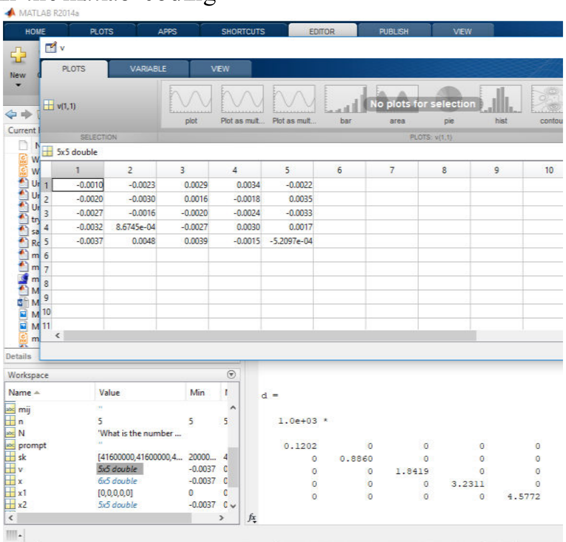

Fig.5 Eigen Value and Eigen Vector Formation

By using the Eigen Values the natural frequency calculated in terms of rad/sec. Further some codings are used to convert into Hz.

Tab.1 Dynamic Property

Natural Frequency	(rad/sec)	10.96534	29.76562	42.91786	56.84247	67.6552
	(Cycles/ Sec)	1.745188	4.737345	6.830589	9.04676	10.76766
Storey Number		Mode Shape				
1		-0.00104	-0.00229	0.002882	0.003353	-0.00221
2		-0.00199	-0.003	0.001617	-0.00176	0.003484
3		-0.00275	-0.00163	-0.00197	-0.00243	-0.00328
4		-0.00325	0.000867	-0.00272	0.003033	0.001685
5		-0.00366	0.004807	0.003871	-0.00152	-0.00052

Mode Shape of the above problem plot by separate coding by using the eigen vectors

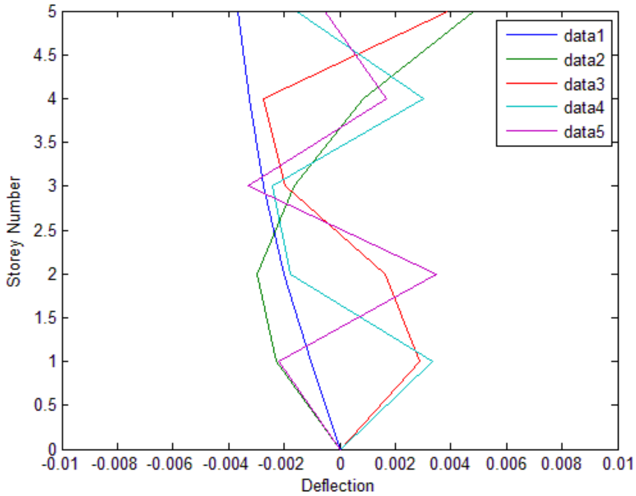

Fig.6 Mode shape

In real time problem the above methodology can be adopted in two ways

1. Conventional approach

In conventional approach there are two stiffness to be considered that is Flexural stiffness and torsional stiffness. Flexural stiffness is directly proportional to the inertia of the column so it vary with respect to both two direction. Torsional stiffness is calculated by using of shear modulus and polar moment of inertia. So that totally three times of above approach can adopted for common rectangular cross sectional column. When center of mass is coincide with the center of rigidity torsional stiffness can be neglected. After calculation of all modal property it should merge with respect to the natural frequency by considering all kind of stiffness.

2. FEMA 356 Stiffness approach

FEMA 356 Stiffness approach is simplified method of conventional approach .In this method Flexural,Shear and Axial rigidity connected. Emprical Equations are used for calculating the Effective Stiffness.Then we adopt the same methodology as previously this paper followed. The major limitation of this method is Direction of the bending is we cannot determined.

V. CONCLUSION

1. In this paper explained a new preliminary approach for modal analysis formed for validating the Finite Element software.
2. The iteration of experimental to finite element modelling have greatly reduce reduced.
3. By using this approach we can reduce time consumed for finite element model validation.
4. This approach can be used for calculating the dynamic property of Undamped Vibration system.

REFERENCES

[1] Anil.K.Chopra (2012), "Dynamics of structures: theory and applications to earthquake engineering", prentice-hall.

[2] ATC-40. (1996). "Seismic Evaluation and Retrofit of Concrete Buildings".Applied Technology Council, 555. Twin Dolphin Drive, Suite 550 Redwood City, California.

[3] Bijaya Jaishi and Wei-Xin Ren (2005), "Structural Finite Element Model Updating Using Ambient Vibration Test Results"in JOURNAL OF STRUCTURAL ENGINEERING ASCE April 2005.

[4] Bohle, K., and Fritzen, C. P. ~2003!. "Results obtained by minimising natural frequency and MAC value errors of a plate model." Mech. Syst. Signal Process., 17(1), 55–64.

[5] D. Kesavan Periyasamy, Dr.R. Ponnudurai (2018), "Mode Mitigation of Seismic Irregular Structure", International Research Journal of Engineering and Technology (IRJET), p-ISSN: 2395-0072,e-ISSN: 2395-0056 Volume:05 Issue: 11, Nov 2018, Page 54-58.

[6] Datta, T. K. (2010), "Seismic Analysis of Structures", John Wiley & Sons (Asia) Pte Ltd, Singapore.

[7] Duggal S K (2010) , "Earthquake Resistance Design of Structure", Fourth Edition, Oxford University Press, New Delhi.

[8] FEMA 356. (2000). "Pre-standard and Commentary on the Guidelines for the Seismic Rehabilitation of Buildings", American Society of Civil Engineers, USA.

[9] FEMA 440. (2005). "Improvement of Nonlinear Static Seismic Analysis Procedures". Department of Homeland Security, Federal Emergency Management Agency, Washington, D.C., U.S.A.

[10] IS 1893 (Part 1) (2002). "Indian Standard Criteria for Earthquake Resistant Design of Structures", (fifth revision). Indian Standard Institute, New Delhi, India.

[11] Jimin He and Zhi-Fang Fu (2001)," Modal Analysis".

[12] Zhang, Q. W., Chang, C. C., and Chang, T. Y. P. (2000). "Finite element model updating for structures with parametric constraints." Earthquake Eng. Struct. Dyn., 29, 927–944.

[13] Zhang, Q. W., Chang, C. C., and Chang, T. Y. P. (2001). "Finite element model updating for the Kap Shui Mun cable-stayed bridge." J. Bridge Eng., 6(4), 285–293.

[14] Zhao, J., and DeWolf, J. T. (2002). "Dynamic monitoring of steel girder highway bridge." J. Bridge Eng., 7(6), 350–356.