Volume-3, Issue-9, Sept-2014 • ISSN No 2277 - 8160

Research Paper | Engineering

Model Analysis and Harmonic Analysis of Cantilever Beam by ANSYS

Chandradeep Kumar	Asst. Prof. Cambridge Institute of Technology, Tatisilway, Ranchi, Jharkhand, India
Anjani Kumar Singh	Ph.D. Scholar, Materials and metallurgical Engineering National Institute of Foundry and Forge, Technology, Hatia, Ranchi, Jharkhand, India
Nitesh Kumar	Western Central Railway,Bhopal,M.P.India
Ajit Kumar	Electrosteel Steels Limited,Bokaro, Jharkhand, India

ABSTRACT

Vibration measurement and analysis on engineeringstructures has begun to gain popularity in engineering field. Technologydevelopments have created an increasing requirement for reliable dynamic analysis. In civil and mechanical engineering field, the behaviour of a structure at "resonance" is a key aspectof structural dynamic analysis. The natural frequency of vibration of a structurecorresponds to that structure's resonant frequency. Modal analysis has become a major alternative to provide a helpfulcontribution in understanding control of many vibration phenomena whichencountered in practice. Determining the nature and extent of vibration responselevels and verifying theoretical models and prediction are both major objectives.

KEYWORDS : vibration,sdof,ansys,mdof,modal

INTRODUCTION

Vibration is an element which is hard to avoid in practice. Excitation ofresonant frequencies of some structural parts can occur with existence of vibrationeven it is only a small insignificant vibration. Then it can be amplified into majorvibration and noise sources. Vibration can be easily defined as an oscillation which is the analogous to the motion of the particles of a mass of air or the like, whose state of equilibrium has been disturbed. It exhibited a movement first in one direction and then back again in the opposite direction. The number of times that a complete motion takes place during the period of one second is called is called frequency which is measured in Hertz (Hz) [1].

Nowadays, vibration measurement and analysis on engineeringstructures has begun to gain popularity in engineering field. Technologydevelopments have created an increasing requirement for reliable dynamic analysis In civil and mechanical engineering field, the behaviour of a structure at "resonance" is a key aspectof structural dynamic analysis. The natural frequency of vibration of a structurecorresponds to that structure's resonant frequency. Maximum displacements areproduced if a structure is subjected to vibration at its natural frequency. Modal analysis has become a major alternative to provide a helpfulcontribution in understanding control of many vibration phenomena whichencountered in practice. Determining the nature and extent of vibration responselevels and verifying theoretical models and prediction are both major objectives.

Vibration is a mechanical oscillation about a reference position. Any system has certain characteristics to be fulfilled before it will vibrate. To putin simple words, every system has a stable position in which all forces are equivalent and when this equilibrium is disturbed, the system will try to regain its stable position To remain stable, structure exhibits vibration at different magnitude when excited, thedegree of vibration varies from point to point (node to node), due to the variation ofdynamic responses of the structure and the external forces applied Therefore, vibrationmay also be described as the physical manifestation of the interchange between kineticand potential energy [2].

Basically, the inertia force, damping force and stiffness force together with externally applied force will form the equation of equilibrium between them, which is called the 'equation of motion' that defines the dynamic behaviour of the structure.

It is in the form of:

Inertia force + damping force + stiffness force = external force

When in algebraic form, the equation becomes:

$$m\ddot{x} + c\dot{x} + kx = f$$

The equation of motion controls almost any structure's linear dynamic behaviour and the dynamic response can be found by solving this equation of motion. Values at the selected location, the degrees of freedom (DOF) will embody the displacement in a structure [3].

Single Degree of Freedom (S.D.O.F)

In the simplest case, a mechanical system can be modeled using methods of lumped parameter analysis whereby motion is described fully by only one time-dependent coordinate. Such a system is termed a 1-degree-of-freedom (DOF) vibration model.

When more than one coordinate becomes necessary, the discrete system is said to have multiple degrees-of-freedom. Figure 1A illustrates the prototypical vibration model having lumped mass m, stiffness k, and damping c elements, in addition to the concentrated external force ft. The parameters m, c, and k are generally interpreted as being effective values that represent combinations of other, multiple, interconnected components. An example in that regard is several springs that are connected in series and/or parallel connections so as to form a single element of equivalent stiffness k [4].

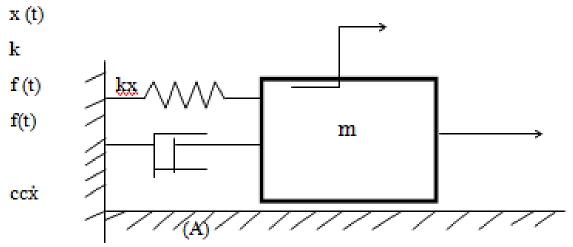


Figure 1 (A) Single-degree-of-freedom vibration model for a particle in translation.

(B)

Figure 1 (B) Free-body diagram of the inertia element [3].

In order to obtain the equation of motion, the free-body diagram of the inertia element is drawn as indicated in Figure 1.1B, and Newton's second law of motion, f = ma, for a particle is applied, wherein f represents the resultant force vector, and a is the particle's absolute acceleration. With displacement measured by the coordinate x(t), the particle's velocity and acceleration are denoted and respectively. The dot-superscript notation is used conventionally in vibration engineering to denote derivatives taken with respect to time.

With the positive sign convention directed rightward in Figure 1, application of the second law provides . In its standard form, the EOM is written with all terms involving displacement, velocity, and acceleration on one side, with one or more forcing terms grouped on the other. The final form of the equation of motion becomes:

$$m\ddot{x} + c\dot{x} + kx = f(t) \quad (1)$$

which is a second-order, linear, inhomogeneous, ordinary differential equation having constant coefficients.The solution is found subject to the initialconditions on displacement x(0) = x0 and velocity as evaluated at t = 0.

By convention, theEOM is also written:

$$\ddot{x} + 2\xi\omega_n\dot{x} + \omega_n^2 x + \frac{f(t)}{m} \quad (2)$$

In terms of parameters:

$$\omega_n = \sqrt{\frac{k}{m}} \quad \text{and} \quad \xi = \frac{c}{2\sqrt{mk}} \quad (3)$$

which are the undamped, circular, natural frequency (in units radians per second) and the dimensionless damping ratio.

Multi degree of freedom systems (MDOF)

For a discrete system that requires more than one coordinate to describe fully its configuration, a system of ordinary differential equations, one written for each coordinate or inertia element, arises. With N such coordinates, the system's overall mass, damping, and stiffness characteristics are embodied by matrices of dimension N×N.

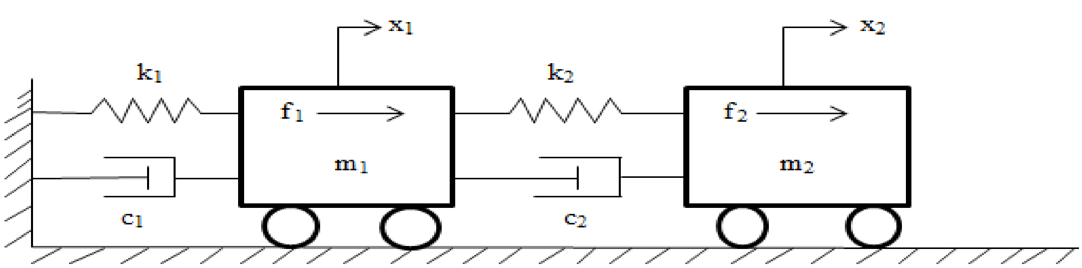


Figure 2 (A) Two-degrees-of-freedom Variation model [3]

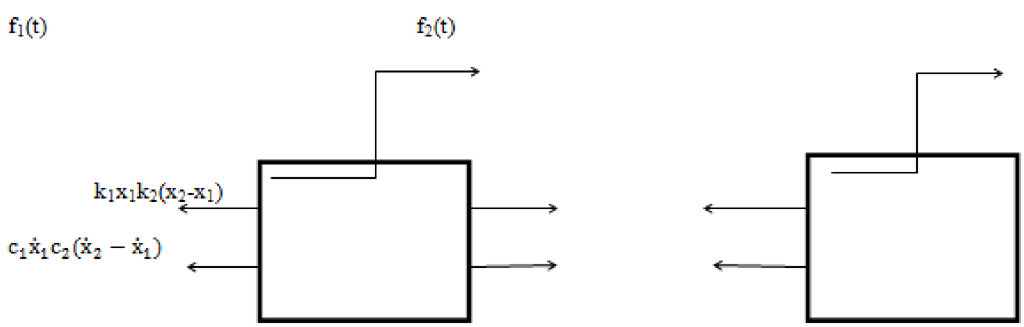


Figure 2 (B) Free-body diagrams of the inertia elements [3]

The major concepts of multiple degrees of freedom (MDOF) can be understood by looking at just a 2 degree of freedom model as shown in the figure 2(A).

In the lumped parameter system of Figure 2A, two masses vibrate with responses x1(t) and x2(t) under action of the impressed forces f1(t) and f2(t) and the indicated stiffness and damping elements. To establish the equations of motion, the second law is applied to m1 and m2 individually using the free-body diagrams of Figure 2(B), providing [4]:

$$m_1\ddot{x}_1 + (c_1 + c_2)\dot{x}_1 - c_2\dot{x}_2 + (k_1 + k_2)x_1 - k_2x_2 = f_1(t), \quad (4)$$

$$m_2\ddot{x}_2 - c_2\dot{x}_1 + c_2\dot{x}_2 - k_2x_1 + k_2x_2 = f_2(t) \quad (5)$$

The EOM are then conveniently written in the matrix vector form:

$$\begin{bmatrix} m_1 & 0 \\ 0 & m_2 \end{bmatrix}\begin{Bmatrix} \ddot{x}_1 \\ \ddot{x}_2 \end{Bmatrix} + \begin{bmatrix} c_1 + c_2 & -c_2 \\ -c_2 & c_2 \end{bmatrix}\begin{Bmatrix} \dot{x}_1 \\ \dot{x}_2 \end{Bmatrix} + \begin{bmatrix} k_1 + k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix}\begin{Bmatrix} x_1 \\ x_2 \end{Bmatrix} = \begin{Bmatrix} f_1(t) \\ f_2(t) \end{Bmatrix} \quad (6)$$

A more compact form of this matrix equation can be written as:

$$[M]\{\ddot{x}\} + [C]\{\dot{x}\} + [K]\{x\} = \{f\} \quad (7)$$

Where [M], [C], and [K] are symmetric matrices referred respectively as the mass, damping, and stiffness matrices. The matrices are N×N square matrices where N is the number of degrees of freedom of the system.

In case of no damping and no applied forces (i.e. free vibration), the matrix equation to represent an MDoF system takes the form as:

$$[M]\{\ddot{x}\} + [K]\{x\} = 0 \quad (8)$$

This differential equation can be solved by assuming the following type of solution:

$$\{x\} = \{X\}e^{i\omega t}$$

The equation then becomes:

$$[[-\omega^2[M] + [K]]\{X\}e^{i\omega t} = 0$$

Since cannot equal zero the equation reduces to the following:

$$[[K] - \omega^2[M]]\{X\} = 0 \quad (9)$$

This is referred to an eigenvalue problem in mathematics and can be put in the standard format by pre-multiplying the equation by $[M]^{-1}$

$$[[M]^{-1}[K] - \omega^2[M]^{-1}[M]]\{X\} = 0$$

Taking $[M]^{-1}[K] = [A]$ and, the equation can be written as:

$$[[A] - \lambda[I]]\{X\} = 0 \quad (10)$$

The solution to the problem results in N eigenvalues (i.e.), where N corresponds to the number of degrees of freedom. The eigenvalues provide the natural frequencies of the system. When these eigenvalues are substituted back into the original set of equations, the values of {X} that correspond to each eigenvalue are called the eigenvectors. These eigenvectors represent the mode shapes of the system. The eigenvalues and eigenvectors describe the modal of the system.

ANSYS

The use of finite element analysis as design tool has grown rapidly in the recent years. ANSYS has become a powerful and easy-to-use finite element program with comprehensive packages. ANSYS was released at 1971 for the first time. It contains over 100,000 lines of code and a lot of analysis can be performed through ANSYS. ANSYS has been a leading FEA program for over 20 years and now it has a completely new look and enhanced into program with multiple windows incorporating a graphical user interface (GUI) and other menus. Today, ANSYS is a vital tool in engineering field [5].

ANSYS enables engineer to perform the following task:

a) Construct computer models or transfer CAD models of structures, products, components or systems.
b) Study physical responses such as stress levels, temperature distributions or electromagnetic fields.
c) Apply operating loads or other design performance conditions.
d) Optimize a design early during the development process for the purpose of production costs reduction.
e) Do prototype testing in undesirable or impossible environments.

HARMONIC ANALYSIS USING ANSYS

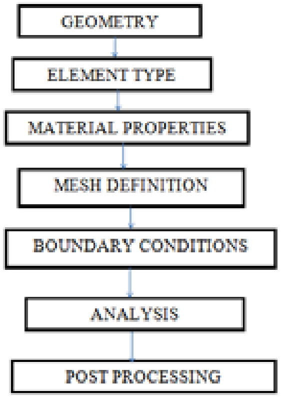
Any sustained cyclic load will produce a sustained cyclic response (a harmonic response) in a structural system. Harmonic response analysis gives the ability to predict the sustained dynamic behavior of structures, thus enabling to verify whether or not designs will successfully overcome resonance, fatigue, and other harmful effects of forced vibrations.

Harmonic response analysis is a technique used to determine the steady-state response of a linear structure to loads that vary sinusoidally (harmonically) with time. The idea is to calculate the structure's response at several frequencies and obtain a graph of some response quantity (usually displacements) versus frequency. "Peak" responses are then identified on the graph and stresses reviewed at those peak frequencies [6].

This analysis technique calculates only the steady-state, forced vibrations of a structure. The transient vibrations, which occur at the beginning of the excitation, are not accounted for in a harmonic response analysis. Three harmonic response analysis methods are available: full, reduced, and mode superposition. (A fourth, relatively expensive method is to do a transient dynamic analysis with the harmonic loads specified as time-history loading functions). The ANSYS/Linear Plus program allows only the mode superposition method [7].

MODAL ANALYSIS USING ANSYS

Vibrational analysis of a beam can be done on ANSYS by providing structural data and load conditions on different supports. Modal analysis in ANSYS is used to find the beam's natural frequencies. The modeling procedure is the following [6]:



RESULT FROM FEM SOLUTION USING ANSYS:

In this analysis, BEAM 3 and MASS 21 elements are considered to model beam and end tip mass. After assigning appropriate material properties and real constant values, meshing is done. Meshing is the process to divide the whole matrix in small-small parts. As a result the exact amount of force, displacement etc. for each small part can be find and the result becomes more accurate. After meshing, mass element is created at free end. As one portion of the beam remain fixed (there will be no displacement), degrees of freedom of that portion is made equal to zero. At the solution, three mode shapes of beam are obtained after performing modal analysis.

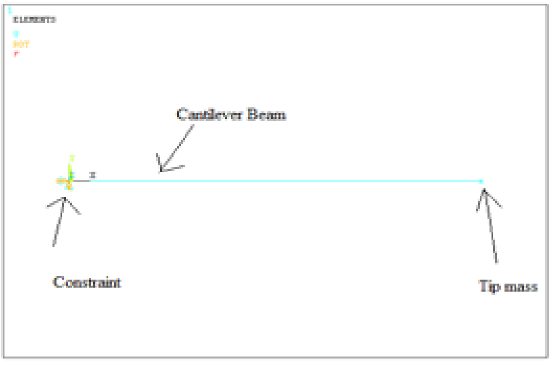


Figure 3 A cantilever beam with tip mass at free end in ANSYS

Figure 3 shows a cantilever beam modeled in ANSYS whose one end is fixed and other end is free. A tip mass weighting 20 gm is placed at its free end for further vibration analysis and calculating natural frequencies as well as mode shapes.

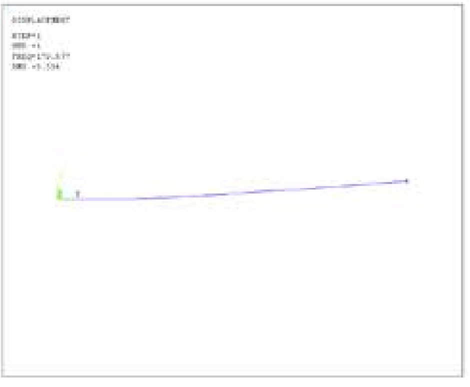


Figure 4 Cantilever beam with tip mass - STEP 1

Step =1
Sub = 1
Frequency =179.577 Hz

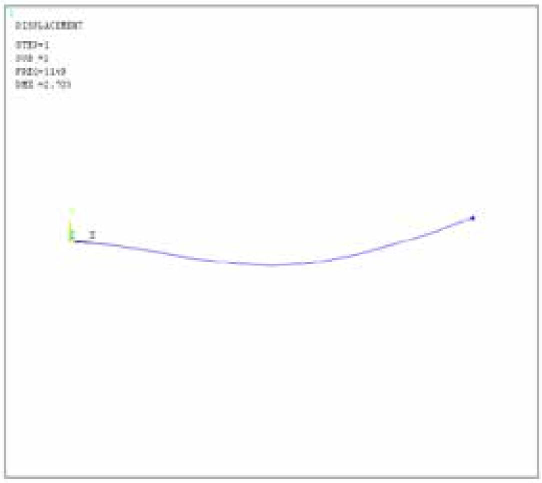


Figure 5 Cantilever beam with tip mass - STEP 2
Step =1
Sub = 2
Frequency =1149 Hz

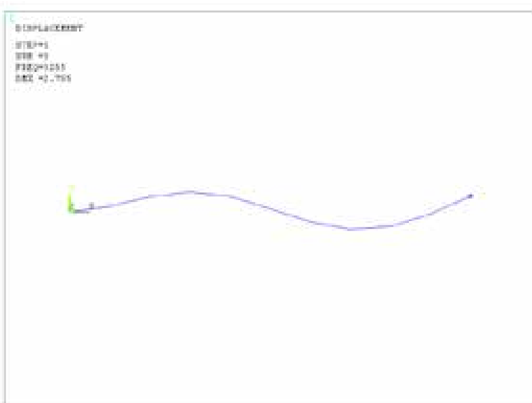


Figure 6 Cantilever beam with tip mass - STEP 3
Step =1
Sub = 3
Frequency =3255 Hz

RESULT FROM HARMONIC ANALYSIS USING ANSYS:

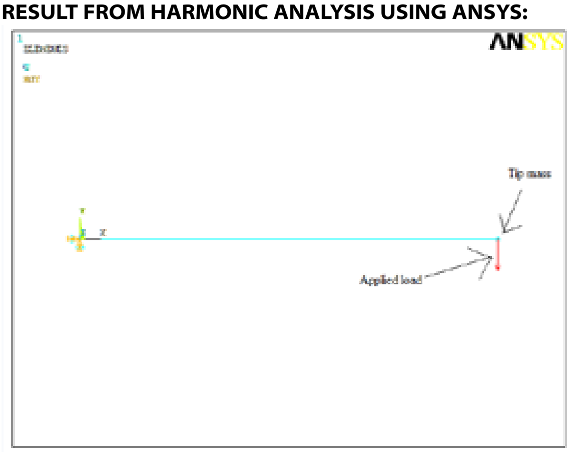


Figure 7 Cantilever beam carrying mass and load at the free end

During harmonic analysis, a force of 50 N is applied in the downward direction at free end to determine the resonant frequency of the beam-mass system. The response obtained from harmonic loading is shown in fig.8

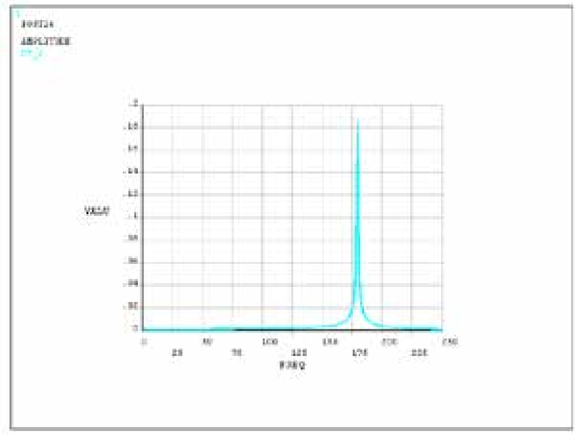


Figure 8 Harmonic Response of cantilever beam with tip mass and load

The frequency response function represents the ratio of output over input signals. The frequency response is a function of frequency and reaches its maximum value at natural frequency. It is clear from the figure that resonant occurs at frequency 180 Hz.

TABLE 1: Results obtained using different approach

Natural frequency obtained by analytical method (Hz)	Naural frequency obtained from finite element coding in MATLAB(Hz)	Natural frequency obtained from ANSYS (Hz)	Resonant frequency from Harmonic Analysis (Hz)
182	179.93	179.577	180
	1158	1149	
	3307	3255	

The value of fundamental natural frequency obtained from FEM solution through MATLAB program is 179.93 which is very much close to 179.577, the value of natural frequency obtained from FEM solution through another approach, ANSYS. From the table 5.1, it is clear that the fundamental natural frequency obtained from finite element program is also comparable to the natural frequency obtained through theoretical approach.

CONCLUSION

Finite element analysis of a cantilever beam with tip mass at its free end was used to demonstrate and investigate aspects of vibration theory. A direct solution by finite element code was developed in MATLAB capable of solving beam flexure problems. The code was validated against analytical solutions and a commercial finite element analysis using ANSYS. The solutions compare favorably to theoretical and results obtained using ANSYS method. Lastly when harmonic analysis is performed, it also favours the result obtained using different approaches and confirms that resonance occurs at fundamental natural frequency.

Good agreement of the natural frequency obtained from MATLAB programming with the ANSYS result and theoretically calculated one is found. The present theoretical calculation is based on the cantilever beam end conditions (i.e., one end is fixed end), in actual practice it may not be always the case because of flexibility in support that may affect the natural frequency.

REFERENCES

1.Hadman M. N.andJubran B, A., "Free and Forced Vibrations of a Restrained Cantilever", Beam Carryinga Concentrated Mass, Eng. Sci., vol. 3, pp. 71-83, 1991. | 2. Veletsos A. S. and Newmark N. M., "A method for calculating the natural frequencies of continuousbeams", engineering studies, Structural research, series no. 38, University of Illinois, Urbana, January,1953. | 3.Fralich, R. W., "Response of long, flexible cantilever beams applied to root motions",Advan. in Eng.Sci., Vol. 2; p 491-499,JAN 1, 1976. | 4.Yeselcei Y., Demirdag O., Catali S., "Free vibrations of a multi-span Timoshenko beam carryingmultiple spring-mass systems",Sadhana,Vol. 33, Part 4, pp. 385–401, August, 2008. | 5. Wanga D., Friswellb M.I., Leic Y., "Maximizing the natural frequency of a beam with an intermediateelastic support", Journal of Sound and Vibration, Volume 291, Page No. 1229–1238, June 23, 2006. | 6. Sofian M., Hazry D., Saifullah K., Tasyrif M.,Salleh K.,Ishak I., "A study of Vibration Analysis forGearbox Casing Using Finite Element Analysis",Proceedings of International Conference on Applicationsand Design in Mechanical Engineering, Penang, Malaysia,10E-1, October 11 – 13, 2009. | 7. Kien N. D., "Free vibration of prestress Timoshenko beams resting on elastic foundation", VietnamJournal of Mechanics, Vast, Vol. 29, No. 1, Page No. 1 – 12, 2007. |