An Overview Of Modal Analysis Using Finite Element Method
Rucha S. Bhakte 1 A.P.Kulkarni 2 P.S.Purandare 3

1,2,3 Department Of Mechanical Engineering Vishwakarma Institute Of Information Technology.
S.No.2/3/4, Kondhawa Bk; Pune-411048. India

Abstract---In the recent years an application of the FEM(Finite Element Method) to the solution of the dynamic behavior of a structure has become more popular. Modal analysis technique is used to determine a structure’s vibration characteristics-natural frequencies and mode shapes. This paper explains simple steps of the modal analysis simulation using ANSYS®. The Cantilever beam is taken for study and frequencies and mode shapes are obtained in the graphical environment of ANSYS. Further Theoretical natural frequencies are obtained by hand calculations and compared with that obtained from FEM.

Keywords:- Modal Analysis; Frequency; Vibration, cantilever beam.

I. INTRODUCTION

Recently FEM is most popularly used to identify the dynamic behavior of the mechanical structures. The study of mode shapes is used as a simple and efficient means of describing resonant vibration. Currently, the evaluation of dynamic characteristics is done by both experimental and numerical (finite element) modal analysis. Experimental modal analysis can measure low-order modes of existing products very accurately, but it is not a cost-effective method for product optimization and development in the design phase. Numerical modal analysis, can be used to analyze and evaluate the dynamic characteristics of different designs, which allows the dynamic characteristics to be optimized and further decide which design can be selected for prototype building. Thus it can help to shorten the design cycle. Researchers are able to achieve good accuracy with numerical simulations on linear elastic materials and components using the FEM [1].

The cause for resonant vibration is due to an interaction between the inertial and elastic properties of the materials within a structure. The vibration related problems that occur in structures and operating machinery are caused due to the resonant effect.

For understanding any structural vibration problem, the resonances of a structure need to be identified and quantified. For this it is required to define the structure’s modal parameters. The combination of both forced and resonant vibration leads to vibration. Forced vibration can be due to the facts listed below:

− Internally generated forces.
− Unbalances.
− External loads.
− Excitation.

Resonant vibration occurs when one or more of the resonances or natural modes of vibration of a machine or structure is excited by some force. Resonant vibration amplifies the vibration response far beyond the level of deflection, stress and strain than that caused by static loading [2].

This paper deals with the dynamic behavior of a rectangular cantilever beam. Modal analysis using ANSYS is explained in simple steps. Mode shapes and frequencies are computed in ANSYS with numerical formulation of the direct solver including the block Lanczos method which is the default method selected by ANSYS. A simple formula for computation of the fundamental natural frequency of a cantilever beam vibration under certain boundary condition and loading are presented. The formulas presented in this paper are quite simple and the fundamental frequency could be obtained by hand calculation with an estimated error.

II. MODAL ANALYSIS

The advantages of using modal analysis are as stated below:

− The natural frequencies and the mode shapes of the model gives physical insight to the dynamic characteristics of the problem
− The set of equations and matrices are reduced to simplify the dynamic equation solution.

Generally, the dynamic behavior of a structure is determined by the equilibrium equation:

\[ m\ddot{U} + c\dot{U} + kU = F \]  

(1)

The mode of the mechanical vibration of a structure is determined by its shape and several physical constants. Assuming free undamped vibration, i.e., \( F=0 \), \( c=0 \), the equation of motion (1) becomes

\[ m\ddot{U} + kU = 0 \]

(2)

For a linear structure the displacements are harmonic of the form:

\[ U = U_0 \cos \omega t \]

(3)

Substituting for \( U \) and \( \ddot{U} \) into (2) yields

\[ (-\omega^2 m + k) U_0 = 0 \]

(4)

For non-trivial solutions the determinant of \( (k - \omega^2 m) \) must vanish:

\[ k - \omega^2 m = 0 \]

(5)

or

\[ k - \lambda m = 0 \]

(6)

where \( \lambda = \omega^2 \) and \( \omega = \sqrt{k}/m \). If \( n \) is the order of the matrices, the equation (5) has \( n \) roots: \( \lambda_1, \lambda_2, \ldots, \lambda_n \) which upon substituting into (4) yield: \( U_1, U_2, \ldots, U_n \) [3].

III. FINITE ELEMENT METHOD

FEM is a very powerful method by which we can create a geometrical model of a structure and perform a detailed dynamic analysis. FEM is widely used to predict the natural frequencies and associated mode shapes (modal
parameters), and the response of a structure to different forms of excitation.

During the past decade, the use of FEM in automotive structural design and analysis has increased extensively. This has been especially evident in the calculation of dynamic and vibration response for the vehicle structure as well as other structures. Because of the good reliability obtained by current designs, finite element modeling in the automotive industry is characterized by emphasis on both dynamic (stiffness) fidelity and stress analysis.

Modeling is often done with minute detail so that the result may be used for examining local behavior as well as overall dynamic properties. Therefore the first step is to construct a finite model. To create this mathematical model, all structural parameters like geometry and material properties should be known, but there is no need for an existing physical structure. It is very convenient to compare designs with different shapes and boundary conditions in any CAD software. The software like ANSYS is user oriented and has extensive documentation, facilitating application by non-specialist design engineers. ANSYS is employed here to create the finite element model for a cantilever beam and the dynamic analysis is performed [3].

IV. STRUCTURAL MODIFICATIONS

“What if” investigations can also be conducted using a mathematical model to determine how changes in the mass, stiffness or damping of the structure will affect its dynamic behavior (i.e. the modes of vibration). Using a finite element model, all types of investigations can be made before the first prototype structure is even built. This way any modifications in the design can be spotted out early in the design cycle where changes are less costly than in the latter stages. This capability is the single most important advantages of finite element methods.

A. Analytically Modeling Structures with FEM:
Just as with the experimentalist, the analyst also has the problem of how to intelligently modify the properties of the structure in order to correctly formulate the problem of interest. Even though the analyst has no actual hardware and thus cannot test and retest, he can do an analogous tasks with FEA model. That is, he may have to run and re-run his finite element model in order to investigate the effect of various design modifications till he reaches up-to the desired result.

B. Design Modifications:
Design modifications are most necessary in order to obtain a reliable and well functioning product. Several types of modifications are as below:

- Special attention to be highlighted to critical areas such as fillets, welds, corners.
- Increase the damping values where ever necessary:
  The amount of damping that can be added is so small that peak values of stress will be only slightly reduced.
- Modifications in Mass/ Stiffness.
- Modification of input load.

- Use of different material with better fatigue properties [4].

V. EVALUATION OF DYNAMIC PERFORMANCE

Some basic requirements must be met to ensure acceptable performance when the structure is in operating environment. The basic requirement is the fundamental natural frequency must be higher than the engine excitation frequency. The mode shape of the structure must be smooth and should not have sudden changes. After weight optimization with consideration of the working conditions and engine parameters the dynamic performance of the structure is evaluated. For example the following formula can be used to determine the engine excitation frequency,

\[ f = \frac{2\pi z}{60\tau} \]

Where
- \( z \) is the number of engine cylinders
- \( \tau \) is the number of strokes and
- \( n \) is the engine speed (rpm) [5].

Resonance situations are many times unavoidable in the case of variable speed main engines. By comparing the excitation and natural mode shape it is possible to find whether the excitation at a particular harmonic order is dangerous for the structure or not. The value of the frequency can be increased either by adding stiffness or decreased by adding the mass to the component. The main aim is to prevent resonance by moving the natural frequency of the component away from the harmful excitation frequency.

VI. MODAL ANALYSIS WITH ANSYS

Engineers need to understand the physical behavior of complex object. They also need to predict the dynamic behavior, performance, calculate the safety margin and accurately identify opportunities for improvement in the design phase. Thus by use of ANSYS this goal can be achieved in less time and at lower cost than with traditional prototyping. The steps for modal analysis using are given below:

- Geometric model
- Finite Element model
- Boundary Condition
- Mesh of Finite Elements
- Modal Analysis

A. Geometric Model:

The component is generated as 3D CAD model considering the design dimensions and working clearances as per the given data. Fig.1 shows the design model of rectangular beam created using Creo.2.0.

B. Finite Element Model:

The model created using Creo 2.0 was imported in Ansys. Some simplification is necessary to keep the model from becoming too large to solve, but it is also reasonable to

![Fig. 1: Model created in Creo.2.0](image)
neglect small geometric details that have little impact on
the component rigidity.

C. Boundary Conditions:
The material properties are assigned to the model and
boundary conditions are defined. The boundary conditions
are as shown in the Fig. 2. The beam cantilever beam thus
it is fixed at one end and other end is free.

D. Mesh of finite elements:
For better approximation of the solution a large number of
elements are provided, but in some cases more number of
elements may increase the error and solving time.
Therefore mesh should be adequately fine or coarse in the
appropriate regions. Different techniques are used for
mesh refinement like: i) Adaptive Meshing, ii) Mesh
Refinement Test within ANSYS, iii) Submodeling. Here
the mesh refinement test mesh within Ansys was used to
declare the adequate mesh in appropriate region. The
meshing of the model was done using tetrahedral solid
elements.

E. Modal Analysis:
The natural frequencies are extracted using the direct
solver which uses the block Lanczos method in ANSYS.
The first three mode shapes are shown in Fig. 5,
Fig. 6 and Fig. 7.

VII. THEORETICAL CALCULATIONS [6,7]
The general differential equation governing transverse
vibration of a beam is as given below:

\[ EI(\delta^2 y/\delta x^2) + \left(\frac{\rho A}{g}\right) (\delta^2 y/\delta t^2) = 0 \]  

(1)

Where,
E = modulus of elasticity of the beam, I=MI of the beam,
\( \rho \) = density of the beam, x = distance from one of the ends
of beam, y = amplitude of vibration, A = cross-section of
the beam.

Therefore, the equation is rewritten as,

\[ (\delta^4 y/\delta x^4) + (m\omega^2/EI)y = 0 \]  

(2)

Assume the solution of the equation in the form

\[ y = Ae^{px} \]  

(3)

And substituting this solution in equation, we get

\[ p^4 - m\omega^2/EI = 0 \]  

(4)

Putting, \( q^4 - m\omega^2/EI = 0 \)

(5)

where, \( m \) = mass per unit length of beam = \( \rho A/g \)

Equation (4) becomes, \( p^4 - q^4 = 0 \)

(6)

Four roots of this equation are,

\[ p_1 = q, \ p_2 = -q, \ p_3 = jq, \ p_4 = -jq \]

Using these values of \( p \) in the equation (3) the
general solution becomes as below,
y = A_1 e^{qx} + A_2 e^{-qx} + A_3 e^{jqx} + A_4 e^{-jqx} = 0
\tag{7}

e^{qx} = \cosh qx + \sinh qx, e^{-qx} = \cosh qx - \sinh qx, e^{jqx} = \cos qx + j\sin qx, e^{-jqx} = \cos qx - j\sin qx.

The solution can be written in the form as below,
\[ y = C_1 \sinh qx + C_2 \cosh qx + C_3 \sin qx + C_4 \cos qx \quad \tag{8} \]

This equation gives the transverse vibration of the beam and natural frequencies of transverse of the beam are given as follows,
\[ f_l = k_i \sqrt{ \frac{k_l}{m^2} } \quad \tag{9} \]

Where, \( f_l \) = natural frequency in Hz, \( k_i \) = constant changes with B.C., modes \( l \) = length of beam.

The values of constant \( k_i \) is given in Fig.9.

\[ \text{Fig. 7: Values of } \frac{k_i}{m^2} \text{ for different modes of vibration for beam.} \]

Density \( \rho = 7850 \text{ kg/m}^3 \). M.I of the beam = \( I = \frac{bd^3}{12} = 3.922 \times 10^{-9} \text{ m}^4 \), \( l = 415 \text{ mm}, b = 50 \text{ mm}, d = 9.80 \text{ mm} \), Material = MS, M.E. = \( 2.038 \times 10^{10} \text{ kg/m}^2 \), \( m = \rho A/g = 0.3921 \text{ kgf}^2/s^2 \).

VIII. RESULT

Natural frequencies are calculated both by FEA and Analytical hand calculations. The fundamental natural frequencies for the first three modes are listed below in Table 1. [8].

Percent Error should lie within permissible limit.

Percent error is given by:
\[ \%E = \frac{\text{abs} (f_{\text{theoretical}} - f_{\text{analysis}})}{f_{\text{theoretical}}} \times 100 \]

| TABLE I. FUNDAMENTAL NATURAL FREQUENCIES FOR FIRST THREE MODES |
|---|---|---|---|
| Modes | Frequency obtained from analytical calculations, Hz | Frequency obtained from FEA, Hz | % error |
| 1 | 46.92 | 46.71 | 0.44 |
| 2 | 290.97 | 291.91 | 0.32 |
| 3 | 814.052 | 814.64 | 0.07 |

九. CONCLUSIONS

- The step by step method for modal analysis using ANSYS was obtained.
- The values of frequencies obtained by FEA and analytical calculations were compared. The deviations in results obtained from both methods are due to the assumptions of boundary conditions made in ANSYS which are not close to the ideal ones.
- Thus the method to determine modal analysis of cantilever beam using ANSYS was validated by analytical hand calculations.

REFERENCES