International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249-8958 (Online), Volume-2 Issue-4, April 2013

Modal Analysis of Chassis

Goutham Solasa, Nariganani SD Satadeep, T.Raghu Krishna Prasad, G.Suresh Babu

Abstract: project is aimed at finding the characteristics of mode (and vibrational) responses of heavy vehicle chassis at particular frequency inputs. The chassis dimensions are taken from an automobile workshop ,is developed in pro-E media and has been imported to ANSYS commercial software to perform the modal analysis. Modal analysis determines the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component during free vibration. The natural frequency and mode shapes are important parameters in the modal analysis of any component. Given suitable conditions with some excited input (frequency), mode shapes are obtained.We followed block lanzcos method in modal analysis. By analysing the mode shapes,we can able to detect the defects in the component.So changing the natural frequency or other parameters can fix the damage. So, using the 'Modal Analysis' feature of ANSYS we come to know of the mode shapes and their changes according to the frequencies. This is the way we get to know of unwanted vibrations and eliminate by further respective processes.In this paper we have shown only mode shapes and their results,in the next step we will do the analysis of mode shapes.

Keywords- ANSYS.

I. INTRODUCTION

A modal analysis determines the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component. It can also serve as a starting point for another, more detailed, dynamic analysis, such as a transient dynamic analysis, a harmonic response analysis, or a spectrum analysis. The natural frequencies and mode shapes are important parameters in the design of a structure for dynamic loading conditions. You can also perform a modal analysis on a prestressed structure, such as a spinning turbine blade.

Modal analysis is the study of the dynamic properties of structures under vibrational excitation.

Modal analysis is the field of measuring and analysing the dynamic response of structures and or fluids when excited by an input. Examples would include measuring the vibration of a car's body when it is attached to an electromagnetic, or the noise pattern in a room when excited by a loudspeaker.The animated display of the mode shape is very useful to NVH (noise, vibration, and harshness) engineers. The results can also be used to correlate with finite element analysis normal mode solutions.

Manuscript published on 30 April 2013.
* Correspondence Author (s)
Goutham Solasa*, KL University, Guntur (A.P.), India
Nariganani SD Satadeep, KL University, Guntur (A.P.), India
T.Raghu Krishna Prasad, KL University, Guntur (A.P.), India
G.Suresh Babu, KL University, Guntur (A.P.), India

Structures

In structural engineering, modal analysis uses a structure's overall mass and stiffness to find the various periods that it will naturally resonate at. These periods of vibration are very important to note in earthquake engineering, as it is imperative that a building's natural frequency does not match the frequency of expected earthquakes in the region in which the building is to be constructed. If a structure's natural frequency matches an earthquake's frequency, the structure could continue to resonate and experience structural damage.

Although modal analysis is usually carried out by computers, it is possible to hand-calculate the period of vibration of any high-rise building by idealizing it as a fixed-ended cantilever with lumped masses. For a more detailed explanation, see "Structural Analysis" by Ghali, Neville, and Brown, as it provides an easy-to-follow approach to idealizing and solving complex structures by hand.

Modal analysis using FEM

The goal of modal analysis in structural mechanics is to determine the natural mode shapes and frequencies of an object or structure during free vibration. It is common to use the finite element method (FEM) to perform this analysis because, like other calculations using the FEM, the object being analyzed can have arbitrary shape and the results of the calculations are acceptable. The types of equations which arise from modal analysis are those seen in eigensystems. The physical interpretation of the eigenvalues and eigenvectors which come from solving the system are that they represent the frequencies and corresponding mode shapes. Sometimes, the only desired modes are the lowest frequencies because they can be the most prominent modes at which the object will vibrate, dominating all the higher frequency modes.

- It is also possible to test a physical object to determine its natural frequencies and mode shapes. This is called an Experimental Modal Analysis. The results of the physical test can be used to calibrate a finite element model to determine if the underlying assumptions made were correct (for example, correct material properties and boundary conditions were used).

A **normal mode** of an oscillating system is a pattern of motion in which all parts of the system move sinusoidally with the same frequency and in phase. The frequencies of the normal modes of a system are known as its natural frequencies or resonant frequencies. A physical object, such as a building, bridge or molecule, has a set of normal modes that depend on its structure, materials and boundary conditions.]Mode numbers

A mode of vibration is characterized by a modal frequency and a mode shape, and is numbered according to the number of half waves in the vibration.

For example, if a vibrating beam with both ends pinned displayed a mode shape of half of a sine wave (one peak on the vibrating beam) it would be vibrating in mode 1. If it had a full sine wave (one peak and one valley) it would be vibrating in mode 2.
Each mode is entirely independent of all other modes. Thus all modes have different frequencies (with lower modes having lower frequencies) and different mode shapes.

Assumptions

- Damping is ignored in a modal analysis.
- Any applied loads are ignored.
- If the analysis is a pre-stress modal analysis and your model includes contact connections, their effects are calculated based on their status at the beginning of the static analysis.

Define Engineering Data

Due to the nature of modal analyses any nonlinearities in material behavior are ignored. Optionally, orthotropic and temperature-dependent material properties may be used. The critical requirement is to define stiffness as well as mass in some form. Stiffness may be specified using isotropic and orthotropic elastic material models (for example, Young's modulus and Poisson's ratio), using hyperelastic material models (they are linearized to an equivalent combination of initial bulk and shear moduli), or using spring constants, for example. Mass may derive from material density or from remote masses.

- Any nonlinear contact such as **Frictional** contact retains the initial status throughout the modal analysis. The stiffness contribution from the contact is based on the initial status and never changes.
- Joints are allowed in a modal analysis. They restrain degrees of freedom as defined by the joint definition.
- The stiffness of any spring is taken into account, however any damping specified is ignored.

Establish Analysis Settings

Number of Modes: You need to specify the number of frequencies of interest. The default is to extract the first 6 natural frequencies. The number of frequencies can be specified in two ways:

1. The first N frequencies (N > 0), or
2. The first N frequencies in a selected range of frequencies.

Output Controls: By default only mode shapes are calculated. You can request **Stress** and **Strain** results to be calculated but note that "stress" results only show the relative distribution of stress in the structure and are not real stress values.

Define Initial Conditions

You can point to a **Static Structural** analysis in the **Initial Condition** environment field if you want to include prestress effects. A typical example is the large tensile stress induced in a turbine blade under centrifugal load that can be captured by a static structural analysis. This causes significant stiffening of the blade. Including this prestress effect will result in much higher, realistic natural frequencies in a modal analysis.
When you perform a prestressed modal analysis, the support conditions from the static analysis are used in the modal analysis. You cannot apply any new supports in the modal analysis portion of a prestressed modal analysis.

Apply Loads and Supports

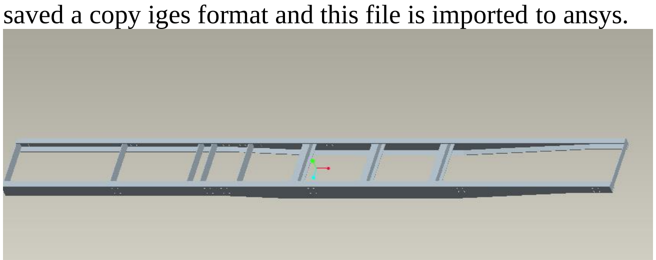
No loads are allowed in the modal analysis. All structural supports can be applied except the Non-zero Displacement and the Velocity boundary condition. Due to their nonlinear nature, compression only supports are not recommended in a modal analysis. Use of compression only supports may result in extraneous or missed natural frequencies.
Solution Information continuously updates any listing output from the solver and provides valuable information on the behavior of the structure during the analysis. Youngsmodulus,density,andpoiss ratio are given as loads.

Review Results

Highlight the **Solution** object in the tree to view a bar chart of the frequencies obtained in the modal analysis. A tabular data grid is also displayed that shows the list of frequencies.
You can choose to review the mode shapes corresponding to any of these natural frequencies by selecting the frequency from the bar chart or tabular data and using the context sensitive menu (right mouse click) to choose **Create Mode Shape Results**. You can also view a range of mode shapes.
Mode shape pictures are helpful in understanding how a part or an assembly vibrates, but do not represent actual displacements. If there are structural loads present in the environment, then the frequencies and mode shapes will depend on the loads and their magnitudes.

Chassis modeling

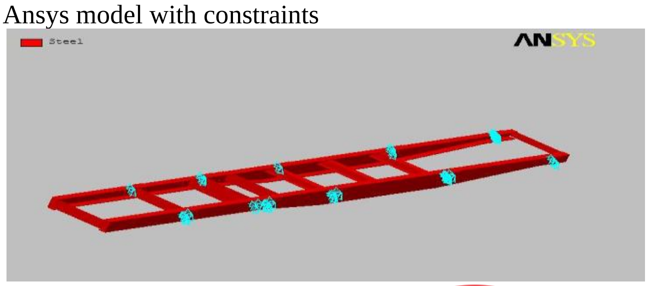
Length of the chassis=1052cm
Width of the chassis=90cm
Thickness of the chassis=1.2cm
There are six leaf springs to the whole beam, where the entire chassis is supported. Inside beams are assemeled and saved a copy iges format and this file is imported to ansys.



Leaf spring distance=126cm
Middle bar
Length=88.4cm
Thickness=1cm
Area=38cm^2

Modelling in Ansys

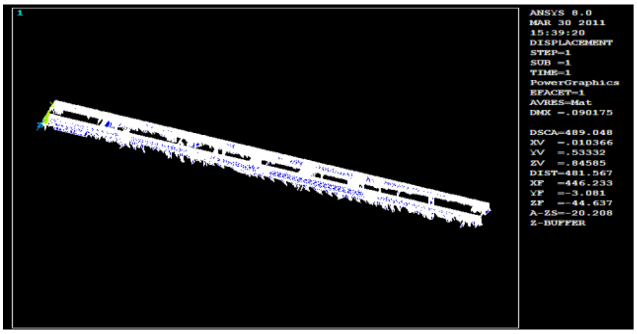
The pro-e model is imported to ansys. The material taken as steel ,youngs modulus, poisson ratio and density are given.
Youngs modulus=2.1*10^7
Poisson ratio=0.303
Density=7.85
Ansys model with constraints



Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249-8958 (Online), Volume-2 Issue-4, April 2013

Mode shape



This is line plot in ansys
The modal analysis is done by block lanzcos.
Material used: mild steel
Frequency range:0 to100

Conclusion
Natural frequencies and mode shapes of the chassis is obtained.
Future scope
After getting these mode shapes further analysis like harmonic analysis, transient analysis can de done.

Ansys report

Summary
This report documents a modal analysis of the part *fin_asm* which was imported from the file D:/Desktop/fin_asm.igs and subjected to the load environment *Environment 1* . The analysis was performed using the ANSYS 8.0 CAE software.
The part *fin_asm* was assigned properties of the material Steel and showed the following results:
The first three natural frequencies are 23.439 , 58.766 , and 72.311 Hz.
For details about the analysis, see Model Information, Analysis Information, and Results Information.

Model Information
The part *fin_asm* has a weight of 8.2126E+08 dyne (8.3802E+05g) and was imported from D:/Desktop/fin_asm.igs. Figure1 shows the model geometry and Figure 2 shows the finite element mesh. Table 1 lists the number of nodes and elements and Table 2 lists the properties of the material (Steel) used in the model.

Figure 2. Finite Element Mesh

Table 1. Details of the Finite Element Model

Entity	Number Defined	Description
SOLID92	78924	10-Noded Tetrahedron
Nodes	171610	

Table 2. Material Properties

Material Properties for Steel	
Modulus of Elasticity [dyn/cm^2]	1.9300E+12
Density [g/cm^3]	8.030
Poisson's Ratio	0.2900

Analysis Information

The part *fin_asm* was subjected to the load environment *Environment 1* (see Figure 3 and Table 3) and evaluated with a modal analysis.

Figure 3. Loads and Boundary Conditions

Table 3. Boundary Conditions for Environment: Environment 1

Constraints			
Type	**Entity**	**Direction**	**Coordinate System**
Constrained Translation	Area 31	XYZ	Global Cartesian
Constrained Translation	Area 32	XYZ	Global Cartesian
Constrained Translation	Area 33	XYZ	Global Cartesian

Constrained Translation	Area 34	XYZ	Global Cartesian
Constrained Translation	Area 42	XYZ	Global Cartesian
Constrained Translation	Area 43	XYZ	Global Cartesian
Constrained Translation	Area 44	XYZ	Global Cartesian
Constrained Translation	Area 45	XYZ	Global Cartesian
Constrained Translation	Area 139	XYZ	Global Cartesian
Constrained Translation	Area 140	XYZ	Global Cartesian
Constrained Translation	Area 141	XYZ	Global Cartesian
Constrained Translation	Area 142	XYZ	Global Cartesian
Constrained Translation	Area 143	XYZ	Global Cartesian
Constrained Translation	Area 144	XYZ	Global Cartesian
Constrained Translation	Area 145	XYZ	Global Cartesian
Constrained Translation	Area 146	XYZ	Global Cartesian
Constrained Translation	Area 147	XYZ	Global Cartesian
Constrained Translation	Area 148	XYZ	Global Cartesian
Constrained Translation	Area 149	XYZ	Global Cartesian
Constrained Translation	Area 150	XYZ	Global Cartesian
Constrained Translation	Area 151	XYZ	Global Cartesian
Constrained Translation	Area 152	XYZ	Global Cartesian
Constrained Translation	Area 153	XYZ	Global Cartesian
Constrained Translation	Area 154	XYZ	Global Cartesian
Constrained Translation	Area 155	XYZ	Global Cartesian
Constrained Translation	Area 156	XYZ	Global Cartesian
Constrained Translation	Area 157	XYZ	Global Cartesian
Constrained Translation	Area 158	XYZ	Global Cartesian
Constrained Translation	Area 159	XYZ	Global Cartesian
Constrained Translation	Area 160	XYZ	Global Cartesian
Constrained Translation	Area 161	XYZ	Global Cartesian
Constrained Translation	Area 162	XYZ	Global Cartesian
Constrained Translation	Area 163	XYZ	Global Cartesian

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

Constrained Translation	Area 164	XYZ	Global Cartesian
Constrained Translation	Area 165	XYZ	Global Cartesian
Constrained Translation	Area 166	XYZ	Global Cartesian
Constrained Translation	Area 167	XYZ	Global Cartesian
Constrained Translation	Area 168	XYZ	Global Cartesian
Constrained Translation	Area 169	XYZ	Global Cartesian
Constrained Translation	Area 170	XYZ	Global Cartesian
Constrained Translation	Area 171	XYZ	Global Cartesian
Constrained Translation	Area 172	XYZ	Global Cartesian
Constrained Translation	Area 173	XYZ	Global Cartesian
Constrained Translation	Area 174	XYZ	Global Cartesian
Constrained Translation	Area 175	XYZ	Global Cartesian
Constrained Translation	Area 176	XYZ	Global Cartesian
Constrained Translation	Area 177	XYZ	Global Cartesian
Constrained Translation	Area 178	XYZ	Global Cartesian
Constrained	Area	XYZ	Global
Translation	179		Cartesian
Constrained Translation	Area 180	XYZ	Global Cartesian
Constrained Translation	Area 181	XYZ	Global Cartesian
Constrained Translation	Area 182	XYZ	Global Cartesian
Constrained Translation	Area 183	XYZ	Global Cartesian
Constrained Translation	Area 184	XYZ	Global Cartesian
Constrained Translation	Area 185	XYZ	Global Cartesian
Constrained Translation	Area 186	XYZ	Global Cartesian
Constrained Translation	Area 187	XYZ	Global Cartesian
Constrained Translation	Area 188	XYZ	Global Cartesian
Constrained Translation	Area 189	XYZ	Global Cartesian
Constrained Translation	Area 190	XYZ	Global Cartesian
Constrained Translation	Area 191	XYZ	Global Cartesian
Constrained Translation	Area 192	XYZ	Global Cartesian
Constrained Translation	Area 193	XYZ	Global Cartesian

Published By:
Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP)

Constrained Translation	Area 194	XYZ	Global Cartesian
Constrained Translation	Area 195	XYZ	Global Cartesian
Constrained Translation	Area 196	XYZ	Global Cartesian
Constrained Translation	Area 197	XYZ	Global Cartesian
Constrained Translation	Area 198	XYZ	Global Cartesian
Constrained Translation	Area 199	XYZ	Global Cartesian
Constrained Translation	Area 200	XYZ	Global Cartesian
Constrained Translation	Area 201	XYZ	Global Cartesian
Constrained Translation	Area 202	XYZ	Global Cartesian
Constrained Translation	Area 203	XYZ	Global Cartesian
Constrained Translation	Area 204	XYZ	Global Cartesian
Constrained Translation	Area 205	XYZ	Global Cartesian
Constrained Translation	Area 206	XYZ	Global Cartesian
Constrained Translation	Area 207	XYZ	Global Cartesian
Constrained Translation	Area 208	XYZ	Global Cartesian
Constrained	Area	XYZ	Global

Translation	209		Cartesian
Constrained Translation	Area 210	XYZ	Global Cartesian
Constrained Translation	Area 211	XYZ	Global Cartesian
Constrained Translation	Area 212	XYZ	Global Cartesian
Constrained Translation	Area 213	XYZ	Global Cartesian
Constrained Translation	Area 214	XYZ	Global Cartesian
Constrained Translation	Area 215	XYZ	Global Cartesian
Constrained Translation	Area 216	XYZ	Global Cartesian
Constrained Translation	Area 217	XYZ	Global Cartesian
Constrained Translation	Area 218	XYZ	Global Cartesian
Constrained Translation	Area 219	XYZ	Global Cartesian
Constrained Translation	Area 220	XYZ	Global Cartesian
Constrained Translation	Area 221	XYZ	Global Cartesian
Constrained Translation	Area 222	XYZ	Global Cartesian
Constrained Translation	Area 223	XYZ	Global Cartesian

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

Constrained Translation	Area 224	XYZ	Global Cartesian
Constrained Translation	Area 225	XYZ	Global Cartesian
Constrained Translation	Area 226	XYZ	Global Cartesian

Results Information

The following figures and tables show the response of the part *fin_asm* to the load environment *Environment 1* . The first three natural frequencies are 23.439 , 58.766 , and 72.311 Hz.

Table 4. Natural Frequencies

Natural Frequencies	
Mode Number	**Frequency (Hz)**
1	23.439
2	58.766
3	72.311
4	81.170
5	82.361
6	82.364

Caution: *Do not accept or reject a design based solely on the results shown here. ANSYS, Inc. recommends that you also take into account experimental test data and/or prior experience with similar analyses when evaluating a design.*

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)