# Pre-stress Modal Analysis of Space Frame Chassis Structure Using Finite Element Method

**M.A.B. Marzuki[1]\*, M.F.M. Azmi[2], A.R.N. Mohamed[3], M.A.A. Bakar[4], M.A.M. Nawi[5]**

*[1,2,3,4]Mechanical Engineering Department, Sultan Azlan Shah Polytechnic*
*[5]Faculty of Engineering Technology, Universiti Malaysia Perlis*
*\*Corresponding email: mohammad@psas.edu.my*

## Abstract

Modal analysis is a method to investigate the dynamic characteristic of a structure in the form of resonance frequencies, mode shapes and natural frequencies. A number of methods can be adopted to study the dynamic response of a structure, which include experimental modal analysis, operational modal analysis and operational deflection shapes. The modal analysis can be studied through experimentation, but with the advancement in computational field, dynamic characteristic of a structure is often analysed through finite element software package. The main objective of the modal analysis in this paper is to investigate the effect of pre-stressed space frame chassis using the finite element software – ANSYS APDL. ANSYS APDL is utilised to generate the chassis model, defining the material properties, element type and size, boundary condition and model constraint. The model is solved and plotted using the appropriate result plot in ANSYS APDL. The load factor $W_e/W_d$ is used to study the effect of different load distribution on the mode shape frequencies. The result shows that there are minimal effects of pre-stressed in comparison with unloaded structure. However, the first mode shape recorded the highest change in mode frequency in respect with increasing load factor. The significance of this study is diverse; in particular, it provides the foundation in determining pre-stressed effect on space frame chassis structure.

*Keywords: Modal analysis, pre-stress, space frame, FEA*

## 1. Introduction

Car chassis structure comprises of a combination of several small section which are welded together to complete a structure. Every section has different purposes, but collectively all of these sections provide load bearing capability when in operation. The chassis structure experiences a wide range of internal and external loads. The internal load is usually caused by the mass of the vehicle and payload, such as baggage and passengers. The engine and transmission also provide considerable amount of internal load to the structure of the chassis. External loads come from the interaction of wheel and ground surface, transferred through spring-damper system to the chassis structure and aerodynamically through wind loads acting on the body of the vehicle.

Car chassis structure can be categorised into three main of types, namely monocoque chassis, ladder frame and space frame. The monocoque is composed of stamped and welded sub-parts that form the body shell, the main part in monocoque chassis are the main pillar (A, B and C pillar for a four-door sedan body shell), roof, front section, rear section and floor. The ladder type chassis structure is often used in commercial vehicle (i.e. buses, trucks, and vans), delivering a cheaper, simpler and sturdier construction in comparison with monocoque chassis. The ladder frame, as the name suggests, is a ladder-shaped-like structure with the side members and cross members attached rigidly through a mechanical joining process. The side members act as the main beams to withstand load from engine weight, suspension load and

the passenger cabin, whereas the cross members are responsible in distributing loads from the suspension and powertrain.

Space frame chassis is a unique chassis structure, commonly used in a vehicular construction where weight and cost are critical components in the chassis construction process. This sees vehicles built using the space frame chassis being lighter and more cost-effective compared to the two previous chassis structure. However, it requires complex manufacturing process, thus it is used in specialised area of the car segment. The space frame chassis can be used in combination with other chassis design; for example, the space frame construction supports monocoque chassis through internal roll cage structure to provide additional rigidity and safety to the original chassis. Some motorsport competitions require the race car to be installed with space frame roll cage to enhance driver safety.

There are a number of testing methods to ensure that the design and development of chassis structure are performed according to the required standard. Historically, the testing of the chassis structure is performed experimentally, involving rigorous testing of the structure until destruction. The rapid development of computer and simulation technology allows the majority of the chassis structural integrity testing to be performed through simulation. In comparison with experimental analysis, the computational design and analysis provide faster result and lower cost. This is due to the fact that the design and analysis are performed and analysed through virtual simulation. Be that as it may, the result still needs to be validated through experimental analysis. Chassis structure can be analysed statically and dynamically. Basic static analysis involves the application of static load on the structure, in which the reaction produced from the simulation is analysed. Dynamic analysis is achieved by analysing

the characteristic of the chassis structure through modal or harmonic analysis, either done experimentally or computationally [1]. The outcome from the analysis can be used as the benchmark for the structure optimisation. The design optimisation loop can be developed to ensure that the final design is optimised against both static and dynamic loading conditions.

Every chassis structure developed through manufacturing process generated different sets of dynamic characteristic. This may be explained by the inconsistency in the quality of raw material and the deficiency in the manufacturing process. Both of these factors affirm the importance of analysing the dynamic characteristic of the structure. Some of the well-known procedures of obtaining structure's dynamic characteristic are Experimental Modal Analysis (EMA), Operational Modal Analysis (OMA) and Operational Deflection Shapes (ODS). These procedures measure the resonance frequency, natural frequencies and mode shapes of the structure. Chassis structure are dynamically excited both from the internal and external sources; internally from the reciprocating and rotating of internal components such as pistons, crank shaft in the engine and gearing mechanism in the gearbox. External sources are predominantly caused by the road surface profile, in addition to external wind loading factor that may contribute to external excitation loading to vehicle body shell. If the internal and external dynamic excitation frequency is the same as the chassis structure's natural frequency, this will cause a form of dynamic behaviour known as resonance. Resonance is the main cause of excessive structural deflection and failure as explained by [2]. The first natural frequency or the resonance frequency produced from any given structure is the main contributing factor for excessive noise and vibration. To acquire the dynamic characteristic of the chassis structure, EMA can be adopted to study the behaviour of vehicle chassis as performed by [3] and [4], with [4] extracted 11 mode shapes with the frequencies produced are below 150 Hz. Mode shape frequencies result is further validated using Modal Assurance Criteria (MAC) method. A two wheeler modal analysis conducted by [5] produced 6 mode shapes by utilising experimental ODS analysis, with which the outcome of the result is used as a basis to optimise the chassis to overcome excessive deflection.

Finite element method (FEM) has been utilised to extract the mode shapes, natural frequencies and resonance frequency from a structure in particularly chassis structure. There are several commercial finite element analysis (FEA) software packages that are available to analyse structure both statically and dynamically. As presented by [6], ANSYS Workbench is employed to extract the natural frequency and mode of vibration in a wind turbine at a certain speed, and further analysis is conducted to study the effect of stiffening to the value of natural frequencies and vibration mode. Block Lanczos and Iterative methods are utilised for both analyses, which resulted in an increase of natural frequency due to dynamic stiffening effect to the wind turbine rotor structure. The combination of both analytical and experimental modal analysis enables improved design of component and structures, as well as the enhancement in structure's dynamic properties.

Pre-stressed modal analysis is a study of loading effect on the dynamic response of a structure. The study by [7] shows the effect of pre-stressed load on structure could result in a noticeable variation on the extracted mode shape frequencies, especially on the first mode compared to the higher modes. Similarly, the mode shapes extracted using finite element package, ANSYS, shows that there is a linear relationship with the increase in the loading of pre-stress level [7]. Experimental and computational results can be compared to further validate the result. A study by [8] utilised finite element package ANSYS APDL to determine the natural frequencies and mode shape of a structure, with the result from finite element package is further validated experimentally. The result from [8] shows that the first natural frequency result derived both from ANSYS and experimental analysis are comparable, but the second frequency and above shows large disparity between the ANSYS and experimental result. There is a number of studies that

investigate the relationship between mode shape and natural frequencies of pre-stressed structure, both using experimental and computational methods, but none of these studies explore the dynamic characteristic of chassis structure on the space frame car chassis. Moreover, a significant amount of effort is directed into achieving the overall strength of the space frame structure, but not the pre-stressed dynamic characteristic. This could lead towards structural failure of the frame.

Given the limitation in the current body of knowledge and the importance of understanding the pre-stress characteristic of space frame chassis, this study seeks to analyse the effect of pre-stress on the space frame chassis structure. The analysis is carried out using the finite element software package, ANSYS APDL. This study will be the first step to develop a computational method of determining the mode shape and its frequencies through a pre-stress space frame chassis structure.

## 2. The Finite Element Solution

The Fast Fourier Transform (FFT) which was developed by J. W. Cooly and J.W. Tukey, is widely used in investigating the dynamic characteristic i.e. modal and natural frequencies for building and structure engineering. By using FFT, the dynamic response of a structure can be analysed from the measured input and the resultant responses data, as elaborated by [3]. The governing equation of motion for undamped system is formulated from;

$$[m]\{\ddot{q}\} + [k]\{q\} = 0 \qquad (1)$$

Where m, q and k in Equation (1) is the constant which define the mass, displacement and stiffness of a structure. Equation 1 is equal to 0 if there is no force is applied to the structure. Here, q is the displacement for a linear system in which the harmonic form will be;

$$\{q\} = \{Q_1\}\sin(w_i t) \qquad (2)$$

Where Q1 is the amplitude of mode shape at the $i^{th}$ natural frequency $w_i$, therefore;

$$-\omega^2[m] + [k] = 0 \qquad (3)$$

For the pre-stressed modal analysis, the stress is a result of constant load, which in theory may affect the natural frequency of the structure. The static analysis is analysed using the following:

$$[k]\{x\} = \{F\} \qquad (4)$$

The stress stiffness [S] in Equation (4) is used in the modal analysis and the initial modal equation with stress stiffness matrix is derived from the following;

$$-\omega^2[m] + [k + S] = 0 \qquad (5)$$

The mode shape frequencies extracted from the solution in Equation (5) can be utilised to provide information on pre-stressed dynamic characteristic of the chassis structure.

## 3. The Space Frame Chassis Structure

The space frame structure is divided into several main sections, namely the main hoop, front hoop, side impact protection and the frontal crush zone. The main function of these sections are mainly to provide protection the driver from in an event of collision. They also responsible for the ride and handling of the vehicle. The main section of the structure is illustrated in Figure 1.

The chassis structure is constructed using a hollowed tube with 27 mm in diameter and 2 mm in thickness for the main and front hoops construction. The other section used 25 mm diameter tube with 1.6 mm thickness. A thicker tube is chosen for the main and front hoop respectively as it provides stronger load bearing capability in the event where the vehicle rolls over. The chassis structure specification i.e. the tube diameter, thickness, length and height, is specified in the geometry data utility and modelled entirely in ANSYS Mechanical Parametric Design Language (ANSYS APDL).

The front section of the chassis structure houses various important vehicle sub-systems such as the braking system and the throttle input system. In addition, the front section also houses the push rod-operated independent suspension system which allowed the vehicle to take corners efficiently. The middle section is where the driver will operate the vehicle where the main component include driver's seat, steering, gear lever and switches are located. Engine and gearbox are positioned at the rear section of the vehicle which also housed critical subsystem namely fuel delivery system, power transmission system and rear suspension system. Fire wall are placed in between the driver's seat and the engine compartment which provide passive safety fire protection system in case of fire broke in the fuel delivery system.



**Figure** 1: Isometric view of space frame chassis structure main section

## 4. Finite Element Model of the Chassis Structure

The finite element method is a computational method which discretised geometric model into small element shapes, which are then calculated and simulated depending on type of analysis. Generally, ANSYS APDL application are divided into two stages: pre-processing, where the geometry, material properties element type and boundary condition are determined; and post processing, where the result is produced using suitable result plot depending on analysis type.

The geometric model is created using key points (KPs) function, in which these KPs are then connected by line function on the ANSYS APDL. It is important to represent the geometric model with suitable element shape and size for this study. Therefore, the model is meshed using PIPE288 element with 11000 and 10975 elements and nodes are created respectively as illustrated in Figure 2. PIPE288 element is a two-node element which offers six degree of freedom (d.o.f) translation in x, y, and z direction, and rotation in the x, y, and z direction in the ANSYS APDL local coordinate

system. PIPE288 is selected to represent space frame chassis structure because it is suitable for analysing thin to thick pipe structure which based on Timoshenko beam theory [9]. The element can withstand axial, shear and bending loading condition. The number of element created from PIPE288 are sufficient to be used in extracting mode shape frequencies from the chassis structure.The chassis structure is made from mild steel with the Elastic Modulus of 200 GPa, density of 7850kg/m3 and a Poisson's Ratio of 0.3.

The model is applied with loading condition at two positions namely driver position at mid-section and engine + gearbox position at the rear section of the vehicle. The two load position represent the weight of the driver at centre and engine + gearbox at rear of the chassis structure. Both loading condition are denoted with engine to driver factor (We/Wd) of 0.17, 0.22, 0.33, 0.42, 0.5, 0.58, 0.67, 0.75, 1.00 and 1.50 to analyse the effect of different weight combination against the outcome of the mode shape frequency. To ensure an accurate representation of the analysis, the model is applied with constraint which are accordance with the actual operating condition of the chassis structure. The mode shape is extracted using Block Lanczos Method which is the method utilised by [7] and [10], 10 mode shape with frequency extending from 1 Hz to 1000 Hz are the frequency range for the chassis structure to vibrate.



**Figure** 2: Various space frame chassis views in element plot

## 5. Finite Element Analysis Result

Two modal analyses are conducted with the first modal analysis is performed without any load applied on the chassis structure, while the second modal analysis is conducted with load applied on the space frame structure. The result of stress-free and pre-stressed modal analysis is presented in Table 1. Ten sets of loading conditions are being applied to the chassis structure to simulate the weight of the driver, engine and gearbox arrangement. The ten mode frequencies extracted using Block Lanczos method are all recorded below 1000 Hz, with the first mode frequency vibrates at 44.90 Hz and the final mode shape vibrates at 836.72 Hz for unloaded chassis structure. The chassis structure is loaded with $W_e/W_d$ load factor with the result tabulated in Table 1.The result shows that there is no significant impact of pre-stress on the chassis structure.

**Table** 1. Stress-Free And Pre-Stressed Modal Analysis Result

| Mode | Load Factor ($W_e/W_d$) | | | | | | | | | | |
| | Without Prestress | 0.17 | 0.22 | 0.33 | 0.42 | 0.5 | 0.58 | 0.67 | 0.75 | 1.00 | 1.50 |
| | Frequency (Hz) | | | | | | | | | | |
| 1 | 44.90 | 44.95 | 44.95 | 44.96 | 44.96 | 44.97 | 44.969 | 44.973 | 44.976 | 44.987 | 45.007 |
| 2 | 48.93 | 48.94 | 48.94 | 48.94 | 48.94 | 48.94 | 48.939 | 48.939 | 48.939 | 48.939 | 48.939 |
| 3 | 107.78 | 107.81 | 107.81 | 107.81 | 107.82 | 107.82 | 107.82 | 107.83 | 107.83 | 107.84 | 107.86 |
| 4 | 177.61 | 177.61 | 177.61 | 177.61 | 177.61 | 177.61 | 177.62 | 177.62 | 177.62 | 177.62 | 177.62 |
| 5 | 311.12 | 311.12 | 311.12 | 311.12 | 311.12 | 311.12 | 311.12 | 311.12 | 311.12 | 311.12 | 311.12 |
| 6 | 463.35 | 463.35 | 463.35 | 463.35 | 463.35 | 463.35 | 463.35 | 463.35 | 463.35 | 463.35 | 463.36 |
| 7 | 546.91 | 546.92 | 546.92 | 546.92 | 546.92 | 546.92 | 546.92 | 546.92 | 546.92 | 546.92 | 546.92 |
| 8 | 658.55 | 658.55 | 658.55 | 658.55 | 658.55 | 658.55 | 658.55 | 658.55 | 658.55 | 658.55 | 658.56 |

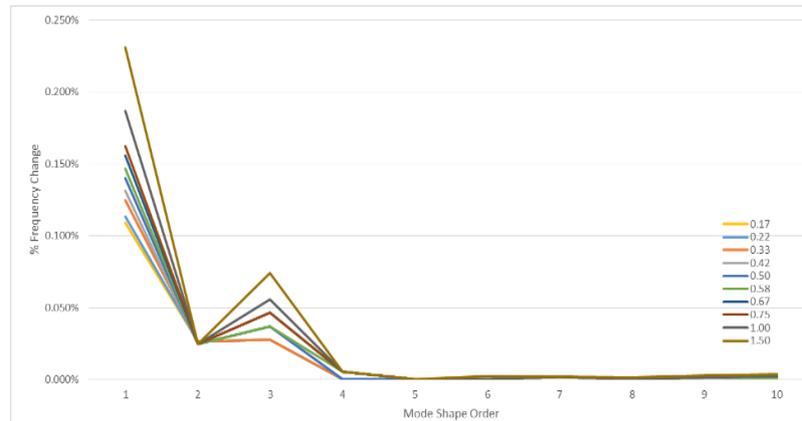| 9  | 691.33 | 691.34 | 691.34 | 691.34 | 691.34 | 691.34 | 691.34 | 691.34 | 691.34 | 691.34 | 691.35 |
| 10 | 836.72 | 836.73 | 836.73 | 836.73 | 836.73 | 836.73 | 836.73 | 836.74 | 836.74 | 836.74 | 836.75 |



**Figure** 3: % of frequency change versus mode shape order

The effect of load on the chassis structure is more obvious on the first mode where the difference between stress-free and pre-stress condition ranges from 0.1% to 0.2% higher, whereas on the mode 2 – 10 recorded minimal to no difference between the load-free and loaded analysis. The relationship between the change in frequency and mode shape order is illustrated in Figure 3.

The result indicates that mode 2 and 4 recorded approximately similar characteristic mode frequency for all weight factor. The highest % change is at mode one and followed by mode three, with a decreasing trend is evident with the increase in the number of mode. Although the difference is small, the increasing load to the chassis structure prior to the modal analysis resulted in the increase in frequency. The finding is consistent with that of [6] and [7]. The extent of load applied has minimal impact towards the overall natural frequency of the chassis structure. This is evident by the $W_e/W_d$ of 1.0 and 1.50 producing similar characteristic as per other load ratio.

# References

[1] Marzuki, M.A.B., Bakar, M.A.A. and Azmi, M.F.M., "Designing Space Frame Race Car Chassis Structure Using Natural Frequencies Data from ANSYS Mode Shape Analysis," *International Journal of Information System and Engineering*, Vol 1, no 1, pp. 1-10 (2015).

[2] Rao, S.S. and Yap, F.F., 2011. Mechanical vibrations (Vol. 4).

[3] Fu, Z.F. and He, J., *Modal Analysis*, (Butterworth-Heinemann, 2001)

[4] Wang, W., Cheng, S., Luo, M. and Zou, J., "Experimental modal test for structure refinement of an automotive chassis," *Measuring Technology and Mechatronics Automation (ICMTMA)*, Vol. 2, pp. 174-177 (2010).

[5] Hugar, N.L. and Venkatesh, P.R., "Shaker based Operating Deflection Shape (ODS) Testing of Two-Wheeler Chassis," *International Journal of Advance Engineering and Research Development*, Vol. 1, issue 6, pp 1-7 (2014).

[6] Xu, Z.Q., "Prestressed modal analysis of the wind turbine rotor based on ANSYS Workbench," *Applied Mechanics and Materials*, Vol. 327, pp. 301-305 (2013).

[7] Bedri, R. and Al-Nais, M.O., "Prestressed modal analysis using finite element package ANSYS," *International Conference on Numerical Analysis and Its Applications*, pp. 171-178 (2004).

[8] Li, R.G. and Zhang, Y.T., "Frequencies and Modals Analysis of Prestressed Concrete Beam by ANSYS," *Advanced Materials Research*, Vol. 243, pp. 769-773 (2011).

[9] Documentation, *ANSYS Mechanical APDL*, Release 13.0. (SAS IP, Vancouver, 2010)

[10] Hai-fei, W., Kun-kun, J. and Zi-Peng, G., "Random vibration analysis for the chassis frame of hydraulic truck based on ANSYS," *Journal of Chemical and Pharmaceutical Research*, 6(3), pp.849-852 (2014).