International Conference on Advance Mechanical Engineering (ICAME09)
22~25 Jun, Concorde Hotel, Shah Alam, Selangor, 2009

STRESS ANALYSIS AND MODAL TRANSIENT RESPONSE OF CAR CHASSIS

M.S.M.Sani, M.T.Arbain, M.M.Noor, G.L.Ming, M.H.Zohari, C.K.E Nizwan & T.T.Mon

Faculty of Mechanical Engineering, Universiti Malaysia Pahang,Tun Razak Highway
26300 UMP, Kuantan Pahang. Phone: 09-5492223, Fax: 09-5492244
e-mail: mshahrir@ump.edu.my / muhamad@ump.edu.my

Abstract

This paper discusses the computational modal transient response and stress analysis of car chassis. The prediction of the dynamic properties of the chassis is great significant to determine the natural frequencies of the structure. In order to avoid resonance, value operating frequency must lower than natural frequency of the chassis. Stress analysis was carried out by using Algor FEMPRO Software to determine the stress distribution on the chassis structure when load been applied. Result shows that bending mode observe at first mode frequency, torsion mode at second mode frequency , mixed bending and torsion at third mode frequency and bending mode at fourth mode frequency. Range natural frequency from 50-99 Hz was examined on this analysis. Maximum stress, 45 MPa was determined at each corner at pillar joint and this value is under the allowable stress for steel which is 300 MPa. The stress and modal analysis techniques are significant essential for automotive chassis structure design.

Keywords: natural frequencies, mode shape, stress analysis, Algor FEMPRO, modal analysis

Introduction

Automotive manufacturer companies are focusing on the processes of delivering high quality cars to the market faster and cheaper. Automotive body structure is very complex and must have enough stiffness ensure assemble and make use of [1]. Chassis is one of the major body components of a vehicle because it is subjected to mechanical shocks or vibrations that may result such as the car can be collapse while it in running that cause from resonant. Due to various dynamic excitations, chassis will tend to vibrate and can lead to ride discomfort, ride safety and stability problems [2]. Structure resonant occurred when the excitation same to the natural frequency of the chassis and important to determine the natural frequency of the structure to avoid this situation. Resonant vibration is mainly caused by an interaction between the inertial and elastic properties of the materials within a structure. Resonance is often the cause of or at least a contributing factor too many of the vibration and noise related problems that occur in structures and operating machinery [3].Therefore, investigation on dynamic characteristic is very significant in order to control vibration and noise.

Finite element analysis is a computer simulation technique for modeling and analyzing the effect of the part or model. This tool is very important to identifying the structure failure before manufacturing and test. Efficient, large, general-purpose computer codes now exist with appropriate matrix assembler routines and equation solvers for calculation of the following structural properties [4]: static displacement and static stress, natural frequencies and mode shapes, forced harmonic response amplitude and dynamic stress, transient dynamic response and transient stress, random forced response and dynamic stress. Ying et al. [5] stated that finite element analysis using MSC Nastran is very useful tool to identify dynamic characteristics such as natural frequencies and mode shapes. Go-kart chassis structure was determined using finite element analysis to compare eigenvalues and eigenvectors of dynamic parameters to compare with experimental results [6]. Finite element analysis of car body in white modal performance has been investigated to assessing design capability [7]. Finite element model was developed to predict noise and vibration on drive train [8]. Schedlinski et al. [9] setup the model validation of MSC Nastran software is to predict the structural dynamics of the body in white.

A finite element stress analysis is carried out at the failure region to determine the stress distribution and possible design improvement [10]. Longitudinal stringer was modeled and stress analysis was performed using Hypermesh software. Martinson et al. [11] were performed 3D ABAQUS finite element simulations for the Brezel, C-weld and

Ring-weld geometries to determine the residual stresses. Finite element analysis was performed to investigate the magnitude and direction of the stresses and strains in the different geometries in order to understand failure [12]. However, the finite element results are extremely sensitive to mesh refinement, especially with respect to singularities at the ends of the joint. Elastoplastic FEM is adopted to understand the characteristics of the stress distribution in a weld-bonded joint, the dimensions and shape of the weld-bonded specimen [13]. Skopinsky [14] was investigated stress analysis of nonradial intersections of cylindrical shells subjected to external loadings using thin shell theory and finite element method. Numerical approach have been devoted to the stress analysis of pipe and nozzle connections subjected to different external loadings [15].

In this paper, finite element was performed to indentify modal parameter and stress analysis of car chassis using ALGOR Fempro software.

Modeling

Figure 1 shows the actual car chassis hang up using bungee cord. Modeling is important stage to perform analysis in finite element. SolidWorks as the commercial CAD software package has been use to create 3D model of car chassis. The important in the exact and accurate measure of the car chassis and transfer that to create the 3d model of car chassis. Figure 2 shows the CAD modeling of car chassis.

Figure 1: Actual Car Chassis

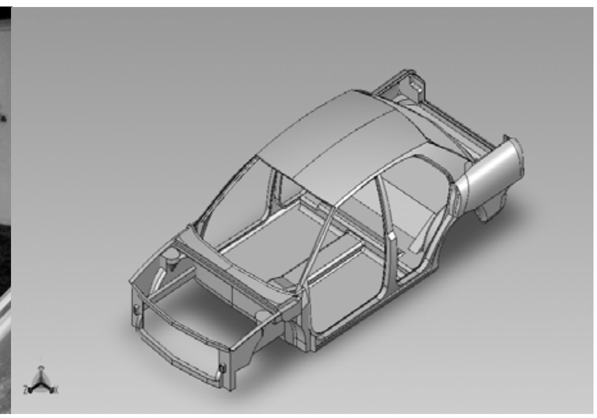

Figure 2: CAD Modeling of Car Chassis

The mesh generation is an important procedure to subdivide the solid geometry into elements. Fine mesh requires considerable computing time and memory space. On the other side, coarse mesh reduces the computing time and memory space, but will cause inaccurate results. In this work, proper meshing is determined based on the convergence regarding the model's element types, element constants, material properties, and the model geometries, which is used in this work. The model surface have many rectangle that indicate the element size choose at the mesh setting. The from the convergence test, the optimum mesh percentages was 70 percent and the number element that produce with this mesh percentages was 6626 element as shown in Figure 3. In this project brick has been choose as the element type for the model and AISI 1005 steel was select for element material.

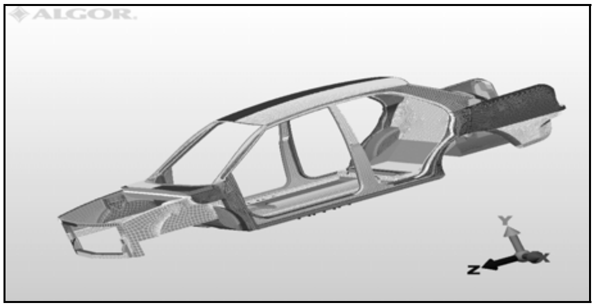

Figure 3: Meshing model

Table 1: AISI 1005 steel properties

Mass density	7872 kg/m^3
Modulus of elasticity	200000 MPa
Poisson's ratio	0.29
Thermal coefficient of expansion	0.0000126(1/^{0}C)
Shear modulus of elasticity	80000 MPa

Result and Discussion

The modal analysis was performed to determine the mode shapes and the natural frequencies of the car chassis. Figure 4 show the first natural frequency was 50.56 Hz. This mode shape was bending with vertical bending type. This can see with the axis of the bending mode shape in the vertical direction. Front and A-pillar joint to the passengers' compartment have maximum displacement in this mode shape condition. The maximum displacement for first mode shape was 1.79mm. The second mode shape of the chassis is twisting or also known as torsion mode shape as shown in Figure 5. The value natural frequency of this mode is 62.10 Hz and the most displacement occurs at the engine bay of the chassis. The rear seat area and the wall of passenger compartment to engine bay were the axis of the twisting. This can see the displacement is lower at this area with the blue color. The maximum displacement for this mode shape was 2.37 mm.

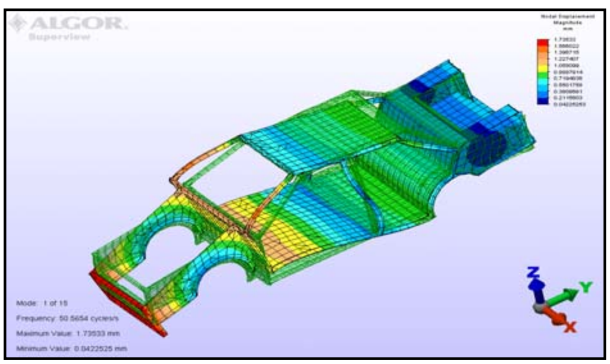

Figure 4 First mode shape

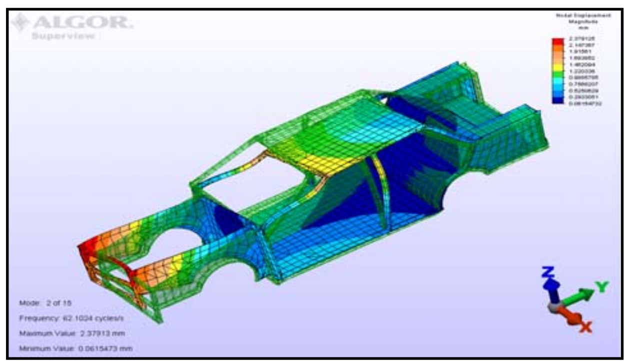

Figure 5 Second mode shape

The natural frequency of the third mode shape of chassis is 83.25 Hz with the mixed mode shape as shown in Figure 6. The mode shapes in this mode were mixed due the combination both bending and twisting mode in one mode. The twisting axis are at the rear area with the roof and the floor of the chassis do the twisting meanwhile the engine bay or the front panel receive the lateral bending. The maximum displacement for this mode shape was 2.417 mm. Bending are the forth mode shape of this car chassis and the natural frequency are 91.98 Hz as shown in Figure 7. The lateral bending was the bending type and the axis is at the side of the chassis floor. That was the reason the middle of the floor got high displacement differ to the side of the floor. The maximum displacement is 2.0 mm and front panel is the maximum place was occurring.

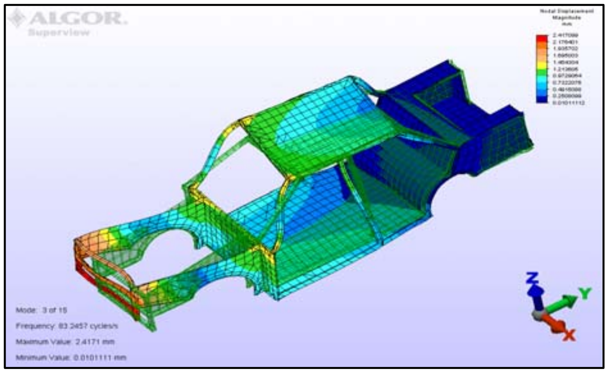

Figure 6: Third mode shape

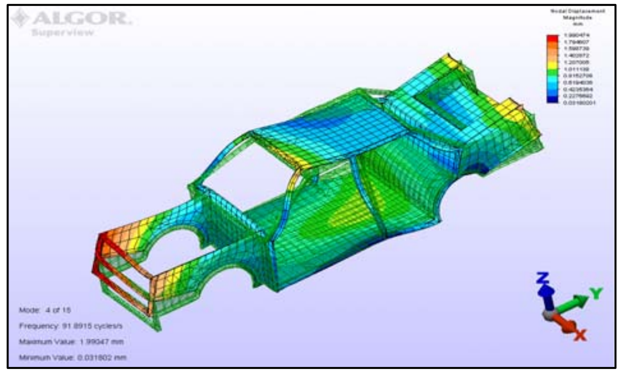

Figure 7: Fourth mode shape

Stress analysis is used to determine the stress point when the chassis structure receives the loads. The analysis will perform by the numbers of passengers to determine the reaction each load to the chassis for each case. First case for indicate one passenger, second case for indicate two passengers, third case indicate for three passengers and forth case indicate for four passengers in the car. Each group of nodes indicates the force load each passengers like in the Figure 10 and each group indicates 637.65 N per group. Every node group contains 12 nodes. One passenger can define as only driver weight as the load for case 1. Two passengers can indicate as the both front seat receive the load for case 2 and for three passengers was addition front passengers and one rear passengers for case 3. The last was the four passengers was seat at the car seat for case 4. Figure 8 shows the stress distribution in the chassis and the clearly on the figure that the higher value of Von Misses stress was 11.7 MPa. The figure clearly shows that each corner of pillar joint has higher stress concentration in when it receive the load or force. B-pillar have high stress due the nearest to the source of load. Figure 9 shows the stress distribution in the chassis and the clearly on the figure that the higher value for Von Misses stress was 23.49 MPa. The figure clearly shows that each corner of pillar joint has higher stress concentration in when it receive the load or force. B-pillar has less stress but both A-pillars have same stress concentration due to both loads at front seat position.

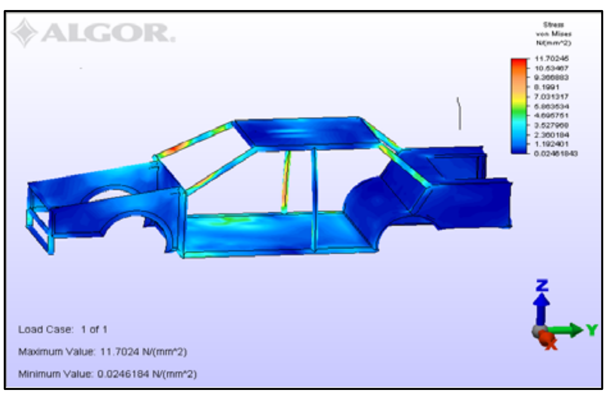

Figure 8: Stress Von Misses of the chassis for case 1

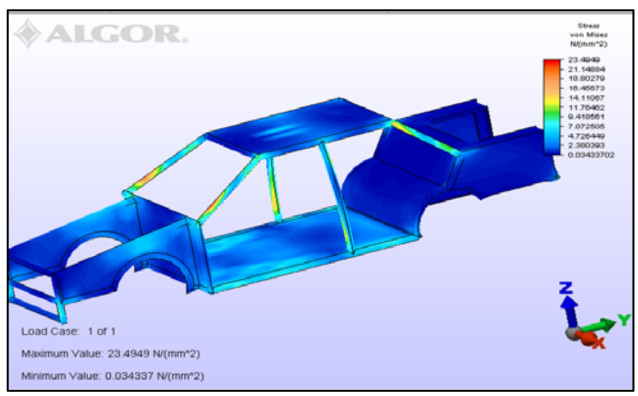

Figure 9: Stress Von Misses of the chassis for case 2

Figure 10 shows the stress distribution in the chassis and the clearly on the figure that the higher value of Von Misses stress was 32.95 MPa. The figure clearly shows that each corner of pillar joint has higher stress concentration in when it receive the load or force. Figure 11 shows the stress distribution in the chassis and the clearly on the figure that the higher value of Von Misses stress was 35.88 MPa. The figure clearly shows that each corner of pillar joint has higher stress concentration in when it receive the load or force.

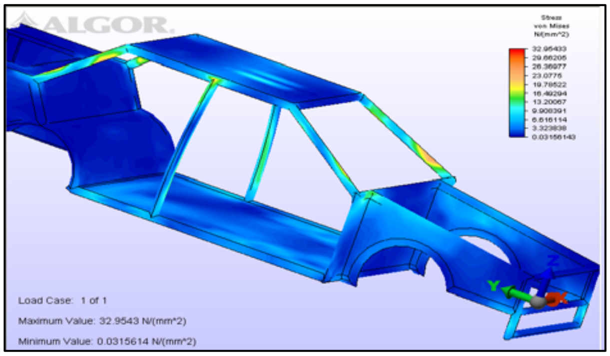

Figure 10: Stress Von Misses of the chassis for case 3

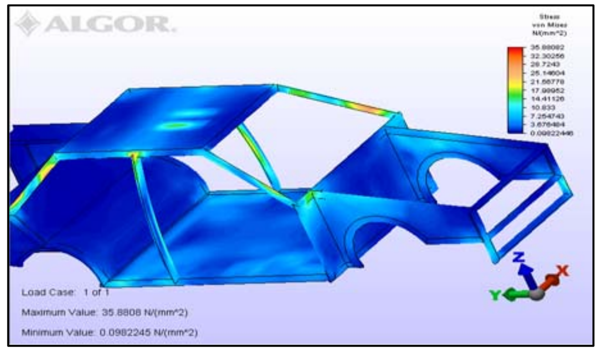

Figure 11: Stress Von Misses of the chassis for case 4

Figure 8 until Figure 11 shows the reaction for all cases which the car receive the load forces. Each figure indicates the condition when increasing the numbers of passengers in the car. The boundary condition was fixed at each corner of the model. Only one node chooses at each corner to perform this analysis. This was the condition on the boundary condition to make bending analysis to the structure. Through bending analysis can carry out the stress analysis on the structure. From all the figure, the stress concentration was appears at the joints. Pillar was play the main enroll to withstand the stress distribution. This can see on the above figure that each of each of it will appears that pillar have the stress concentration and it place at the joint the pillar to the body such as the lower part of the chassis or the roof of the chassis.

Conclusion

Static stress with linear material models analysis and the linear natural frequency (modal analysis) were perform on the 3D model of car chassis to determine stress distributions, natural frequencies, and mode shape of the car chassis by using FEMPRO Algor. Result shows that bending mode observe at first mode frequency, torsion mode at second mode frequency , mixed bending and torsion at third mode frequency and bending mode at fourth mode frequency. Range natural frequency from 50-99 Hz was examined on this analysis. Computational stress analysis was carried out to determine the stress distribution on the chassis structure when load been applied. Maximum stress, 45 MPa was determined at each corner at pillar joint and this value is under the allowable stress for steel which is 300 MPa. The stress and modal analysis techniques are significant essential for automotive chassis structure design. It is important to study the dynamic characteristic of the car chassis so that resonance and structure failure does not occur on the chassis in working condition.

Acknowledgement

The authors would like to express their deep gratitude to Universiti Malaysia Pahang for provided the financial support.

References

[1] Y.M.Jin (2003), "Analysis and Evaluation of Minivan Body Structure Finite Element Methods", Technical Paper Technology Research Institute, Beiqi Futian Vehicle Co.,Ltd
[2] R.N. Brady (1997), "Heavy Duty Trucks: Powertrain, Systems and Service", Prentice Hall, New Jersey
[3] P. Guillaume (2000), "Modal Analysis", Technical Notes Research Groups Acoustics and Vibration, *Vrije* Universiteit Brussel, Pleinlaan 2, B-1050 Brussel, Belgium.
[4] N. F. Rieger (2003), "The Relationship between Finite Element Analysis and Modal Analysis", Technical Paper Stress Technology Incorporated, Rochester, New York
[5] J.Zhanwang, C.Zongyu (2002), "Dynamic Response Analysis of Minicar Changan Star 6350", Technical Paper , Changan Automobile (Group) Liability Corp., LTD.
[6] M.S.M.Sani, G.L. Ming, M.M.Rahman, A.S.Sulaiman, T.T.Mon, K.Kadirgama, M.M.Noor (2008), "Dynamic Correlation Technique and Model Updating on Go Kart Chassis Structure", Proceedings of EnCon2008,2nd

Engineering Conference on Sustainable Engineering Infrastructures Development & Management, December 18 -19, 2008, Kuching, Sarawak, Malaysia.

[7] M.J. Moeller, R.S.Thomas, H.Maruvada, N.S.Chandra, M.Zebrowski (2001), “An Assessment of FEA NVH CAE Body Model for Design Capability”, SAE Proceeding 2001-01-1401, pp 147-153

[8] K.D. Wang, K.Bairavasu (2000), “Noise and Vibration Analysis by Finite Elements”, Technical Notes LUK Incorporated.

[9] C.Schedlinski, , F.Wagner, , K.Bohnent, , J.Frappier, , A.Irrgang, R.Lehmann, , A. Muller, (2004), Experimental Modal Analysis and Computational Model Updating of a Car Body in White, International Conference on Noise and Vibration Engineering (ISMA), Leuven, Belgium.

[10] V.Veloso,H.S. Magalhães, G.I. Bicalho, E.S.Palma (2009), “Failure investigation and stress analysis of a longitudinal stringer of an automobile chassis”, Journal of Engineering Failure Analysis 16, pp1696–1702

[11] P. Martinson, S. Daneshpour, M. Koçak, S. Riekehr, P. Staron (2009), “Residual Stress Analysis of Laser Spot Welding of Steel Sheets, Journal of Materials and Design (2009), doi: 10.1016/j.matdes.2009.03.041

[12] L.D.R. Grant, R.D.Adams, L.F.M.Silva (2009), “Experimental and numerical analysis of single-lap joints for the automotive industry”, International Journal of Adhesion & Adhesives 29, pp 405–413

[13] B. Chang, Y. Shia, S. Dong (2000), “Studies on a computational model and the stress ®eld characteristics of weld-bonded joints for a car body steel sheet”, Journal of Materials Processing Technology 100, pp 171-178

[14] V. N. Skopinsky (1996), “Stress analysis of nonradial cylindrical shell intersections subjected to external loading”, International Journal Pressure Vessel & Piping 67, pp 145-153

[15] Steele, C. R. , Steele, M.-L.(1983), “Stress analysis of nozzles in cylindrical vessels with external load”, Tran. ASME, J. Pres. Ves. Techn., 105 (3), pp 191-200