"STRUCTURAL ANALYSIS OF CAR UNDER STATIC AND DYNAMIC CONDITION BY USING ANSYS SOFTWARE"

Rohit Walia
[1](Department of Mechanical Engineering, Swami Vivekananda College of Engineering, Indore. (M.P))

Abstract—*This paper presents the static and dynamic behavior of car chassis using ANSYS. The application of static & dynamic technique together with Finite Element Tools had been utilized in order to verify the simulation and analysis of the chassis. Results were used in conjunction with the finite element to predict the dynamic characteristic of car chassis such as the natural frequency and corresponding mode shape. Damage can occur if the car chassis is excited at resonance during operation. Therefore, based on the result gained from the finite element analysis, further enhancement of the current chassis had been done through the chassis FE model in order to improve its stiffness as well as reduce the vibration level. In Finite Element Analysis, the model of car chassis was created using 10 nodes tetrahedral elements. Normal mode analysis was performed and free-free boundary condition was applied in the analysis to determine the first seven natural frequencies and mode shapes. It was noticed that the first seven natural frequencies of the car chassis were below 100 Hz and vary from 47.423 to 79.531 Hz. This problem was resolved by carrying out the Finite Element Model updating, by adding sub-structure.*

1. INTRODUCTION

Chassis is one of the major components of a vehicle because it can consider effected to the performance of a car. This can see when it be subjected to mechanical shocks or and vibrations that may result in failures some component and after some limit, it can also be major problem to the car such as the car can be collapse while it in running that cause from resonant. The resonant happen 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. Finite element analysis is a computer simulation technique for modeling and analyzing the effect of mechanical loads applied to a part or material that use in the system. This also tool to identifying the areas of stress concentration that are susceptible to the mechanical or thermal failure before manufacturing and test. It is the new method to define the parameter in save and short time because no waste sample material will produce and the result better and accurate. Validation of computational is important to make the result from both method is acceptable. Through experimental method, it can define the properties of the structure using modal analysis. Therefore; the prediction of the dynamic properties of the chassis is of great significance. In this project, the finite element analysis using 3D modeling issues regarding the experimental analysis of car chassis is addressed. The modeling approach is investigated extensively using computational modal analysis. A comparison of the modal parameters from both experiment and simulation shows the validity of the proposed approach. Then perform the computational stress analysis with linear material type analysis to find the stress concentration point in the car chassis. The point that come from the stress analysis can be use to determine the structure ability to withstand the load, force and the vibration.

2. LITERATURE REVIEW

Dave Anderson and Greg Schade[1] developed a Multi-Body Dynamic Model of the Tractor Semitrailer for ride quality prediction. The studies involved representing the distributed mass and elasticity of the vehicle structures e.g. frame ladder, the non-linear behavior of shock absorbers, reproduce the fundamental system dynamics that influence ride and provide output of the acceleration, velocity and displacement measures needed to compute ride quality. I.M. Ibrahim, et.al. [2] had conducted a study on the effect of frame flexibility on the ride vibration of trucks. The aim of the study was to analyze the vehicle dynamic responses to external factors. The spectral analysis technique was used in the problem study. Other than that, the driver acceleration response has been weighted according to the ISO ride comfort techniques. Another case study was presented by Romulo Rossi Pinto Filho [3], who analyzed on the Automotive Frame Optimization. The objective of his study was basically to obtain an optimized chassis design for an off-road vehicle with the appropriate dynamic and structural behavior. The studies were consisted of three main steps. Firstly, the modeling of the chassis used in a commercial off-road vehicle using commercial software based on the finite elements method (FEM). Secondly, a series of testing were conducted to obtain information for modeling and validation. Finally, the validated model allowed the optimization of the structure seeking for higher torsion stiffness and maintenance of the total structure mass. In the finite element model, the author has developed the chassis by using steel with closed rectangular profile longitudinal rails and tubular section cross member. Izzudin B. Zaman [4] has conducted a study on the application of dynamic correlation and model updating techniques. These techniques were used to develop a better refinement model of existing truck chassis with approximately 1 tone and also for verification of the FEA models of truck chassis. The dynamic characteristics of truck chassis such as natural frequency and mode shape were determined using finite element method. Lonny L. Thomson, et. al., [5] had presented his paper on the twist fixture which can measure directly the torsion stiffness of the truck chassis. The fixture was relatively lightweight and portable with the ability to be transported and set-up by one person. The extensive testing has been carried out to check on the accuracy of the fixture and was found to be within 6% accuracy. Using the twist fixture design, the author has managed to test on several chassis of different manufacturers. These tests were performed to compare the stiffness values of the different chassis. The results show that the uncertainty and standard error were below 5%. Murali M.R. Krishna [6] has presented his study on the Chassis Cross- Member Design Using Shape Optimization. The problem with the original chassis was that the fundamental frequency was only marginally higher than the maximum operating frequency of the transmission and drive shaft, which were mounted on these cross-members. The aim of this testing was to raise the cross-member frequency as high as possible (up to 190-200 Hz) so that there was no resonance and resulting fatigue damaged.

3. RESEARCH METHODOLOGY

In this project, Modal Analysis, Finite Element analysis was used to determine the characteristics of the car chassis. The combination of all the analysis results were used to develop virtual model created using FEM tools and the model was updated based on the correlation process. Further analysis and modification were then executed to the existing car chassis design For the purpose of this study, the car chassis was modeled using Solid works software according to the original size of structures. The model was then imported into Finite Element software (ANSYS). The purpose was to determine the natural frequencies and mode shapes. For the meshing analysis, 4-noded c quad and 3noded tria shell elements were chosen to model the solid chassis.

TABLE 1 FEA RESULTS OF TORSION TEST BEFORE MODIFICATION

Nomenclature		Length in mm	Values From Analysis
Transverse Dist Between two Front wheels	Lf (m)	550.00	0.55
Traverse distance between the force application points	Ls (m)	1247.55	1.25
Transverse Dist Between two rear wheels	Lt (m)	542.57	0.84
Force in the Right front end of the structure	Fd (N)	9810.00	98.10
Force in the Left front end of the structure	Fe (N)	9810.00	98.10
Front torsion angle of the structure	θf (radians)		0.070854545
Rear Torsion angle of the structure	θt (radians)		0.03083416
Torque	T (N.mm)		12238.49493
Theta	θ (radians)		0.040020385
Vertical disp by Fd	δd (m)		0.01299
Vertical disp by Fe	δe (m)		0.01299
Ttal Deflection	δ (m)		0.02598
Global Stiffness	K (kNm/rad.)		305806.5244
Torsional Stiffness of the Chassis		305806. 5244	3.06E+05

TABLE 2 FEA RESULTS OF TORSION TEST AFTER MODIFICATION

Nomenclature		Length in mm	Values From Analysis
Transverse Dist Between two Front wheels	Lf (m)	550.00	0.55
Traverse distance between the force application points	Ls (m)	1247.55	1.25
Transverse Dist Between two rear wheels	Lt (m)	842.57	0.84
Force in the Right front end of the structure	Fd (N)	9810.00	9810
Force in the Left front end of the structure	Fe (N)		0.07085454 5
Front torsion angle of the structure	θf (radians)		0.03083416
Rear Torsion angle of the structure	θt (radians)		12238.4949 3
Torque	T (N.mm)		12238.4949 3
Theta	θ (radians)		0.04002038 5
Vertical disp by Fd	δd (m)	8.46	0.00846
Vertical disp by Fe	δe (m)	8.46	0.00846

total Deflection	δ (m)		0.01692
Global Stiffness	K(kNm/rad.)		469553.9896
Torsional Stiffness of the Chassis		**4639553.9896**	**4.7E+5**
%ichange in stiffness w.r.t. baseline model.	**53.55%**		

TABLE 3 RESULT COMPARISONS BETWEEN BEFORE & AFTER MODIFICATION

Analysis	**Torsion stiff. (kNm/rad.)**	**Max. Def. (mm)**	**Torsional stiff. %**
FEA Analysis before modification	305806.5244	12.991	**53.55**
FEA Analysis after modification	469553.9896	8.46	

TABLE 4 RESULTS COMPARISON OF CHASSIS BEFORE & AFTER THE MODIFICATION

Analysis	**FEA of the chassis Before modification**	**FEA of the chassis After modification**	**Final results**
Natural Frequency (Hz)	29.54	38.16	30% Increased
Torsional stiffness (kNm/rad.)	305806.5244	469553.9896	54% Increased
Total deflection (mm)	12.991	8.66	34% Reduced
Von-mises stress (Mpa)	672.16	434.77	36% Reduced
Overall weight of new chassis	119.42	134.92	12% Increased

4. CONCLUSION

As conclusion, this study has achieved its core objectives. The Static & Dynamic characteristic such as the natural frequencies and corresponding mode shapes of the car chassis were determined by finite element analysis with Ansys software. The results were used to validate a finite element model representing the real structure. The overall modifications resulted in the natural frequency shifted by 30 % higher than the original value, increased the torsion stiffness by 54 % and reduced the total deflection by 34 %. The overall weight of the new car chassis was increased by 12%.

REFERENCES

[1] Dave Anderson and Grey Schede. Development of a Multi- Body Dynamic Modal of a Tractor.Semi trailer for Ride Quality Prediction. International Truck and Engine Corp. 2001.

[2] I.M. Ibrahim, D.A.Crolla and D.C. Barton. Effect Of Frame Flexibility On The Ride Vibration Of Trucks. Department of Mechanical Engineering, University of Leeds LS2 9JT, U.K. August 1994.

[3] Pomulo Rossi Pinto Filho. Automotive Frame Optimization. Universidade Federal de Uberlandia. November 2003.

[4] Zaman@Bujang, Izzudin and Abd. Rahman, Roslan (2006). Application of Dynamic Correlation Technique and Model Updating on Truck Chassis. 1st Regional Conference on Vehicle Engineering & Technology, July 2006.

[5] Lonny L. Thomson, Jon K. Lampert and E. Harry Law. Design of a Twist Fixture to measure the Torsional Stiffness of a Winston Cup Chassis. Department of Mechanical Engineering, Clemson Univ.1998.

[6] Murali M.R. Krisna. Chassis Cross-Member Design Using Shape Optimization. International Congress and Exposition Detroit, Michigan. February 23-26,1998.