IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
Volume 2, Issue 8, June 2022
Impact Factor: 6.252

Vibration Characteristics and Structural Behavior of Three-Wheeler Aluminum - Composite Square Chassis Using FEA and FFT Analyzers

Prof. Suresh Bhaisare[1], Mr. Rahul R Patil[2], Mr. Omkar A Patil[3], Mr. Saurabh R Patil[4], Mr. Mohseef Shaikh[5]

Assistant Professor, Mechanical Engineering, NBNSSOE, Pune, India[1]
UG Student, Mechanical Engineering, NBNSSOE, Pune, India[2,3,4,5]

Abstract: *Change in design of chassis, brings in change in vibration pattern and stress pattern will also change for impact loading conditions. This project to study the vibration characteristics and structural behavior of three-wheeler chassis configurations. To study the vibration aspects, with change in design, modal analysis is done. Front, side and rear impact simulations are carried out to obtain displacement and stress distribution patterns. Three-dimensional CAD model is designed using CATIA V5R20. Finite Element Analysis (FEA) software ANSYS version 19.0 is used to determine the Natural frequencies of three-wheeler chassis. Weight optimization of three-wheeler chassis by using aluminum - composite honeycomb structure. The conceptual design will primarily be manufactured using E-glass fiber and aluminum pipe. Modal analysis of modified three-wheeler composite - aluminum honeycomb chassis will be done using ANSYS workbench. Experimental validation of natural frequency of chassis will be done using FFT analyzer and impact hammer.*

Keywords: Chassis, Vibration Pattern and Stress Pattern, ANSYS, CATIA, etc.

I. INTRODUCTION

Automotive chassis is a skeletal frame on which various mechanical parts like engine, tires, axle assemblies, brakes, steering etc. are bolted. The chassis is considered to be the most significant component of an automobile. It is the most crucial element that gives strength and stability to the vehicle under different conditions. Automobile frames provide strength and flexibility to the automobile. The backbone of any automobile, it is the supporting frame to which the body of an engine, axle assemblies are affixed. Tie bars, that are essential parts of automotive frames, are fasteners that bind Different auto parts together. Automotive chassis is considered to be one of the significant structures of an automobile. It is usually made of a steel frame, which holds the body and motor of an automotive vehicle. The technology growth always enhances the existing practice and guide it to take a step ahead. This leads to innovation and new techniques evolving to solve complex problems much easier.

Automobiles have seen a tremendous improvement in its performance, safety equipment, styling, infotainment etc. Growth in technology is not only seen in the vehicles running on the road but also during design process. The entire product development cycle has employed number of comprehensive technologies to bring up an attractive product to market in much lesser time. One of such an advanced technique is Finite Element Method (FEM) which helps to analyse automotive components for its strength, durability and crashworthiness. Sophisticated hardware and software developments fetched path for advancement in Computer Aided Engineering (CAE). With accurate Finite Element (FE) modelling and defining properties it is very much possible to get results with good co-relation with experimental values. With advancement in FEM, it is becoming part of chassis design process for understanding system's behaviour in terms of strength, durability, vibration etc.

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
Volume 2, Issue 8, June 2022
Impact Factor: 6.252

II. LITERATURE REVIEW

1. In this research paper A Cardiff University student have used the freedom of the Formula Student rules to create an innovative chassis that combines a high performance with an efficient manufacturing process. All aluminum sandwich panels were pre-cut using computer numerical control routing, which is a rapid low-cost operation that produced highly accurate results. Assembling the monocoque consisted of folding and bonding panels along pre-routed lines and reinforcing the relevant joints. The chassis presented is a full monocoque, constructed from a 30mm aluminum sandwich panel throughout.
   The assembly from flat sheet as described here has a number of advantages: no specialized tooling is required; existing technological CNC equipment may be used; the material quality is assured by the supplier, and not the manufacturer. The chassis design is the sixth generation produced by Cardiff Racing. The design has been supported by extensive hand calculations, finite element modelling, and physical testing to prove that it is structurally equivalent to a space frame. The results show that, in certain load cases, the predicted failure load for the monocoque chassis is up to 3.4 times greater than that of a dimensionally equivalent space frame chassis.

2. In this research study, honeycomb safety structures were designed, analysed for safe road transport application point of view and the use of materials in these structures were addressed. Honeycomb structure was constructed using honeycomb core, adhesive and aluminium panels and Compression test was performed on it. CAD model of the honeycomb structure was designed using PTC Cero parametric design software. Following which, ANSYS Workbench - Static Structural simulations are carried out on different material grades of aluminium on honeycomb structures.
   These compression test and simulations are carried out with the aim to compare and understand the impact load capacity on these materials of honeycomb structures. Apart from that, the energy absorption ability of honeycomb structure with the normal block structure has been analysed using ANSYS Workbench- Explicit Dynamics. This research study is carried out with the intention to investigate and understand the energy absorbing characteristics of different types of aluminium materials such as Aluminium 3003-H12, Aluminium 3004 - H112 and Aluminium 3102 - H112. In this research both theoretical analysis and also ANSYS experiments have been carried out to understand the energy absorption characteristics of aluminium honeycombs.
   This paper describes design and analysis of heavy vehicle chassis. Weight reduction is now the main issue in automobile industries. Traditionally most common material for manufacturing vehicle chassis has been mild steel, in various forms. Over time, other materials have come into use, the majority of which have been is steel and Aluminium. In this paper traditional materials are replaced with composite materials [S-glass epoxy and E-glass epoxy]. Using reverse engineering method. (Existing model, modified model, honey comb model). For validation the design is done by applying a single vertical load acting on the chassis. And then Structural and, fatigue analysis will be carried out on three models to all materials and select the best material Impact analysis can also be done for the selection material in all models Software's used in this work solid works for modelling ANSYS 14.5 for analysis.

3. In this paper, a methodology for chassis frame model construction and validation is explained. In present work, chassis frame model is validated in terms of modal parameters and also against static loading conditions. Existing chassis 3D Computer Aided Design (CAD) data was generated using scanning and cloud point data conversion technique. FE model was generated and validated through experimental measurements viz. modal testing, vertical bending, lateral bending, and torsional bending test. Loading and boundary conditions were replicated on the complete FE model in CAE domain and test validation was carried out using appropriate mesh biasing and weld modelling techniques.

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
Volume 2, Issue 8, June 2022
Impact Factor: 6.252

The chassis FE model was validated on the parameters of natural frequencies, mode shapes, and MAC correlation along with bending stiffness. The complete process gave an idea about the correlation environment between experimental and CAE domains. The correlated model was further used for Chassis optimization. The methodology proposed will not only gives a deep understanding and factors for real life correlation but also gives the insight of complete analysis procedure of ladder chassis frame.

4. This paper specifically talks about the ladder-fame chassis. This paper deals with the determination of exact volumetric composition of Carbon and Epoxy in Carbon Reinforced Epoxy based Polymeric Composites which would ultimately replace the existing steel alloys with the help of stimulation software, ANSYS. This paper also determines the exact cross-section of the chassis made with Carbon Reinforced Epoxy based Polymeric Composites which could eliminate the steel alloy chassis with existing cross-sections.

5. This paper aims to study the vibration characteristics and structural behaviour of three different Go-Kart chassis configurations. Chassis configurations are designed as per standards followed by major Go-Kart events organized in India and abroad. To study the vibration aspects, with change in design, modal analysis is done. Front, side and rear impact simulations are carried out to obtain displacement and stress distribution patterns. Three-dimensional CAD model is designed using SOLIDWORKS. Finite Element Analysis (FEA) software ANSYS version 16.0 is used to determine the Natural frequencies and impact testing. AISI 1018 material is taken for all the simulation cases. 4-G load is applied for front and rear impact while 2-G is applied for side impact test simulation. Results are extracted in terms of Natural Frequencies, von-Mises stress and deformation for all the simulation cases.

6. In this research paper, it is shown that the SVD method works with higher efficiency in comparison to the conventional SQP method since the conventional method takes longer time to calculate the optimum thickness for such a complex geometry like a truck chassis frame in comparison to a more simple geometry like a cantilever beam structure. As a result of this study, it is shown that multi-objective optimization of complex structures can be made using SVD in a fast and accurate manner, where it provides designers an advantage of conducting optimization analysis according to multiple objective functions to generate Pareto sets, which can be chosen according to demand.

7. This paper gives an insight about design, simulation and manufacturing aspects of honeycomb sandwich structures. A summary of honeycomb sandwich panel terminologies, characteristics and properties are described. Various configurations of honeycomb cells and types of hexagonal honeycomb cores with reinforcements are studied and illustrated. To avoid modelling of actual hexagonal cells in CAD software, easier approach to model equivalent homogenized structure for FEA is described. Manufacturing processes for Honeycomb core and sandwich structures are explained. Honeycomb sandwich panels are widely used in automobile, aerospace and space structures due to unique characteristics like high strength to weight ratio and High stiffness.
Honeycomb sandwich panels consist of honeycomb core made of either metal or thin paper like materials. Core is sandwiched with metallic or composite face sheets. Core gives high compressive strength in T direction whereas face sheet gives shear strength in T and W direction. Compressive strength of sandwich panel depends on foil thickness of honeycomb core, cell size, thickness of core in T direction and face sheet thickness. For nearly same weight honeycomb sandwich panels can give up to 30 time's higher stiffness than metallic sheets. Face sheets of the sandwich structure facilitate mounting of the instruments as core has very high-volume fraction of cavity and hence cannot hold fasteners alone. Modelling of Honeycomb sandwich structures with actual cell configuration is difficult and time consuming. Hence sandwich structure is generally modelled as equivalent homogeneous structure.

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
Volume 2, Issue 8, June 2022
Impact Factor: 6.252

III. IMPLEMENTATION

Initially research paper relevant to the topic is gathered and after going through research papers, conventional design of chassis. Prepared 3-D model will be transferred to ANSYS software and proper meshing will be created on the model for further analysis. • For determining the natural frequency, modal analysis will be performed on ANSYS. For studying vibration analysis will be performed on the square chassis model. Redesigning of model will be done to reduce the stresses and to increase the natural frequency of square chassis. A prototype of the model will be manufactured. FFT analysis will be performed on the prototype. Vibrations and natural frequency will be calculated by using FFT results. Comparison of ANSYS and FFT results will be carried out.

IV. CONCLUSION AND DISCUSSION

- It was observed that natural frequency obtained by FEA and natural frequency obtained using FFT was close enough to compare, and not only the frequency was maintained but also the weight was optimized in the prototyped chassis.
- Previous weight of the chassis which was of structural steel was 2.877kg and the weight after using aluminium composite material was about 1.019kg.
- There was total 2.877- 1.019= 1.858 kg reduction in weight.
- So final total weight of the chassis is near about 1.019±0.300 kg.
- Considering failure, stresses in chassis and bending in chassis, we have decided to design such a chassis that it can carry high load weight light weight n less failure. It also sustains high vibration occurred due to bad road condition n internal vibration of engine. As it is made up of aluminium composite material it will definitely reduce its weight and by reinforcing it will provide enough strength to the vehicle.

REFERENCES

[1] "Design Manufacturing and Vibration Analysis of Worm and Worm Wheel Gear Box" by Prof. R.K. Nanwatkar[2] Sushmita Kamble[1], IJRASET, volume, Issue 11, 2019/11.
[2] "Design, Development, and Manufacture of an Aluminum Honeycomb Sandwich Panel Monologue Chassis for Formula Student Competition." By H C Davies, M Bryant, M Hope, And C Miller (2011).
[3] "Honeycomb Safety Structure: Design, Analysis and Applications in Safe Road Transport." by Pankaj Prakash Andes (2012).
[4] "Structural Analysis of Heavy Vehicle Chassis Using Honey Comb Structure." By Indu Gadagott And M V Mallikarjun Issn 2278 – 0149 Www.Ijmerr.Com Vol. 4, No. 1, January 2015.
[5] "LCV Chassis Frame Optimization Using Combined Simulation and Experimental Approach" By H. Rajesh, Mahesh Anand Patwardhan, Nagesh Voderahobli Karat, Mandar Bhatkhande, R Ramkumar, Ninad Pachhapurkar, And M Saraf (2009).
[6] "Efficacy Of Vehicle Chassis of Polymeric Composite" By Nipun Sinha and Kaushik Kumar (2010).
[7] "Numerical Investigation on Vibration Characteristics and Structural Behavior of Different Go-Kart Chassis Con (2010).
[8] "Multi-Objective Structural Optimization of a Chassis Frame." By Ahmet Turan.
[9] "Design, Modelling And Manufacturing Aspects Of Honeycomb Sandwich Structures: A Review" By Shubham V. Rupani, Shivang S. Jani, G.D.Acharya (2016).
[10] "A Study on Composite Honeycomb Sandwich Panel Structure" By Meifeng He, Wenbin Hu (2015).
[11] "Design And Fabrication of Aluminium Honeycomb Structures Based on Origami Technology" By Lijun Wang, Kazuya Saito, You Gotou and Yoji Okabe.
[12] "Analysis Of Honeycomb Structure" By Ms. Poonam Joshilkar, R. D. Deshpande, R. B. Kulkarni (2014)
[13] "Failures Analysis Of E-Glass Fibre Reinforced Pipes in Oil and Gas Industry: A Review" By Sujith Bobba, Z. Lemana, E.S. Zainuddina and S.M. Sapuana, (2016).