iJRASET
International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 5 **Issue:** VI **Month of publication:** June 2017

DOI:

www.ijraset.com

Call: 08813907089 | E-mail ID: ijraset@gmail.com

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Mathematical Modelling and Analysis of Automotive Chassis with Composites Materials Using Fem

Priyanka Ranu [1], Rajeev Singh Chauhan[2] Dr. V.N. Bartaria[3]
[1]*M. Tech. scholar Mechanical Eng. Department LNCT Bhopal*
[2]*Associate Prof. Mechanical Eng. Department LNCT Bhopal*
[3]*HOD Mechanical Eng. Department LNCT Bhopal*

Abstract: ***the automotive chassis serves as a frame work for supporting the body and different parts of the automobile. Also, it has to withstand the shock, twist, vibration and other stresses caused due to sudden braking, acceleration, shocking road condition, centrifugal force while cornering and forces induced by its components. The chassis acts as the backbone of a heavy vehicle which carries the maximum load for all designed operating conditions.***
Present study deals automotive chassis with i section and taken three different materials, steel as existing or already used materials and two others are composites materials (e-glass epoxy & carbon epoxy). By simulation results we found that carbon epoxy provides better strength as compare to e-glass epoxy and steel. Deformation and weight is reduced by efficient manner in carbon epoxy as compare to others.
Kew words: automotive chassis, fem, ansys, e-glass/epoxy, carbon/epoxy composites, weight reduction and strength.

I. INTRODUCTION

Chassis frame plays an most important role in every automobile it act as a skeleton for vehicles almost all the important parts are attached with chassis frame like engine, suspension system, steering system, tires, driveline and also hold almost all sprung and unsprung weight and chassis frame also heavy in weight so chassis frame must be strong enough to withstand shock, twist, vibration and other stresses. Chassis frame consist of side members along with supported cross member‘s, most important role of frame is to give strength and stability to automobile in different conditions. Automotive chassis s frame helps to keep up an automobile rigid, stiff and unbending. Automobile frames basically manufactured from aluminum and steel which make it heavy in weight and heavy weight directly effects the efficiency of vehicle so now a days the light weight material have gained a world wide popularity like composite materials – carbon fiber , epoxy glass etc. The composite materials have specialty that they are lighter in weight and many time in strength as compare to conventional steel used in chassis frame. Finite element analysis is one of the mostly used tool for stress analysis. Finite element analysis with required boundary conditions is used to determine the critical regions in chassis frame. Static analysis has been done to determine the maximum stress region.
Modal analysis is done to determine the natural frequency and different mode shapes on different frequency modal analysis helps to study the vibration characteristics of a body which helps in avoiding the resonance. the outcome of modal analysis can be used as a reference value to other analysis like random analysis, harmonic analysis etc. the main characteristics of each mode of structure can be figured out through the modal analysis and actual vibration’s can be anticipated.
In the era of globalization and tough competition the use of heavy vehicles is increasing for the transportation works, considerable attention has been focused on designing of the heavy vehicles. Thus it is very much necessary for the designers to provide not only equipment of maximum reliability but also of minimum weight and cost, keeping design safe under all loading conditions by careful stress analysis of the vehicles. Heavy Commercial Vehicles or alternatively Multi Axle Vehicles having gross vehicle weight upwards of 16.2 tons. Being a life line of the economy, these vehicles are an integral part of the commercial activity of any country and these vehicles are usually deployed in the long haul distance and in transportation of materials at the ports as also in the extraction of natural resources like Iron or Coal etc. In Modern vehicle, it is expected to fulfill the following functions:
i. Provide mounting points for the suspensions, the steering mechanism, the engine and gearbox, the final drive, the fuel tank and the seating for the occupants;
ii. Provide rigidity for accurate handling;

iii. Protect the occupants against external impact. While fulfilling these functions, the chassis should be light enough to reduce inertia and offer satisfactory performance. It should also be tough enough to resist fatigue loads that are produced due to interaction between the driver, engine, power transmission and road conditions.

To design and realize IDRA09, in the light of all different constructive solutions, a full unit body was considered first. A finite element analysis was carried out in order to understand if a full unit body solution could be applicable. Considering a shell body made of three different layers of bidirectional carbon fibre tissues, and a vertical load applied to the centre of gravity, which represents the vehicle weight with the driver, increased with a safety coefficient, the analysis showed the necessity of a structural frame. The body of IDRA09 was divided in two parts: a front body parts with the cockpit, and a rear engine compartment, Figure 4. These two parts were connected by bolts. The body was made by an internal frame, with a structural function, and by an external shell with an aerodynamic function. The internal frame was a simplified space frame structure made by structural shapes. Figure 5 shows the simplified finite element model made by one dimensional element. It can be considered as sandwich structure because the shapes were made by an internal core of structural foam and by an external shell of woven carbon fibre. The structural foam has a double function: it is used to obtain all the frame shapes, and it contributes to increase the stiffness of the vehicle body. The main contribution in terms of strength and stiffness is made of the carbon fibre that is used to cover all the structure.

II. LITERATURE

A. *Salvi Gauri Sanjay, Kulkarni Abhijeet, Gandhi Pratik Pradeep, Baskar P(2014),*

Chassis is the foremost component of an automobile that acts as the frame to support the vehicle body. Hence the frame ought to be very rigid and robust enough to resist shocks vibrations and stresses acting on a moving vehicle. Steel in its numerous forms is commonly used material for producing chassis and overtime aluminum has acquired its use. However, in this study traditional materials are replaced with ultra light weight carbon fiber materials. High strength and low weight of carbon fibers makes it ideal for manufacturing automotive chassis. This paper depicts the modal and static structural analysis of TATA 407 fire truck chassis frame for steel as well as carbon fibers. From the analyzed results, stress, strain and total deformation values were compared for both the materials. Since it is easy to analyze structural systems by finite element method, the chassis is modified using PRO-E and the Finite Element Analysis is performed on ANSYS workbench. 2. Archit Tomar & Dheer Singh(2016), In the case of vehicle the term frame means the part of automobile that holds all the important components all these components constitute together to from chassis. The chassis frame has to be robust enough to resist various forces due to undulation in surface of road or any other reason. Forces act on chassis frame like shock, twist vibration and also due to heavy weight of chassis fame add extra stress. Along with strength the most important in frame designing is to have sufficient bending stiffness. Natural frequency and also played most important role in chassis frame the excitation frequency and chassis frame frequency never match otherwise it create resonance and damage will incur in chassis frame. Now a day's lightweight material gained popularity worldwide due their high strength and less weight. This paper present the static structural analysis is done using FEA method, modal analysis of a chassis frame is done do determine natural frequency and corresponding Vibration mode shapes, and also design modification done to optimize weight of chassis frame to perform this work the chassis frame designed in CATIAV5R19 and analysis is done in ANSYS14.5 .Material used for chassis frame is steel 52 and carbon epoxy composite material. 3.A. Airale, M. Carello, A. Scattina (2011), Weight reduction and high mechanics performance are one of the challenges of the automotive industries future. A particular and interesting field is the application of carbon fibres in combination with structural foams. These allow to realize creative shapes for the car body but also strength and tough elements for the chassis, according to the necessity to combine style with design and engineering aspects. Nowadays, the problem of carbon fibre technology is the cost of production and the recyclability, so the major applications are in the racing field. One interesting application is the body of the prototype called IDRA, a low consumption vehicle which participated in the European Shell Eco-marathon. The chassis is a structural frame made with structural foam, the body is a part of the chassis and it is made of carbon fibre. The result is a well made car body integrated with all the sub-systems: steering, brakes, wheels, cockpit, electric wiring, controls, fuel cell, electric motor and transmission. The different design and production steps, from the concept phase and structural analysis made by means of finite element techniques to the production and the assembling of the monocoque, are discussed. 4.Divyesh N. Chaudhari, Prof. Vinod M. Makwana, Prof. Dharmeet J. Patel (2015), *In* this literature survey, we have found different technique to reduce the stress of heavy vehicle chassis. FEA can play important role in this analysis. The truck chassis is the main system of the vehicle and it is integrated with the main truck component systems such as the axles, suspension, power train, cab and body. The truck chassis has been offered to static, dynamic and also cyclic load. So analyze of chassis is crucial to avoid failure. Computer simulation techniques provides a great leverage in design optimization for weight

Volume 5 Issue VI, June 2017
ISSN: 2321-9653

reduction, better material utilization, shorter design cycles and elimination of major part of prototype testing. Static analysis of the chassis shows the equivalent stress and deformation contour. Aim of literature review is to find out high stress area under different load condition. After that we have tried to minimize the stress by considering design aspect. 5.Singarajan Nagammal Vijayan and Sathiavelu Sendhil kumar(2015), The automotive chassis serves as a frame work for supporting the body and different parts of the automobile. Also, it has to withstand the shock, twist, vibration and other stresses caused due to sudden braking, acceleration, shocking road condition, centrifugal force while cornering and forces induced by its components. The chassis acts as the backbone of a heavy vehicle which carries the maximum load for all designed operating conditions. This paper describes design and analysis of heavy vehicle chassis as the prime objective of any automobile industries in today's fast changing world considering weight reduction. In the present paper, the pertinent information of an existing heavy vehicle chassis of EICHER is considered for modeling and analysis for polymer composite materials namely, Carbon/Epoxy, and cross-sections like C, I and Box type subjected to the identical load as that of a steel chassis. The numerical results are validated with analytical calculation considering the stress distribution and deformation.

B. Objective of The Study

In earlier study different sections like C,I and box type is analysed with steel material and different loading condition is also taken in many cases. Based on the previous results it was inferred that steel with 'I' section has superior strength to withstand high load and induced low deformation and stress distribution when compared to other cross section.

In present study our main objective is to reduce chassis weight by replacing the rigid solid chassis with I-section chassis and also replacing conventional materials with composite materials Carbon fiber and E-glass epoxy. Numerical simulation techniques provides a great leverage in design optimization for weight reduction, better material utilization, shorter design cycles and elimination of major part of prototype testing. In this study the chassis frame will be modelled in CAD software and the further analysis in ANSYS by using the composite material like Carbon fiber and E-Glass Epoxy.

III. PROBLEM FORMULATION

Structural Analysis of Heavy Vehicle Chassis Dimensions of PCHVC (polymeric composite heavy vehicle chassis) are taken as that of the conventional SHVC (steel heavy vehicle chassis). Width of the chassis is 80mm and the properties of PCHVC vary with directions of fiber, a 3-D model of chassis is needed for analysis. The loading conditions are assumed to be static. The element has six degrees of freedom at each node translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. The finite element analysis is carried out on

Table 1 Specifications of heavy vehicle chassis.

Parameters	Value
Material of the chassis	Steel 52
Chemical composition	0.20% C, 0.50% Si, 0.9% Mn, 0.03% P and 0.025% S
Side bar of the chassis	200 mm × 76 mm × 6 mm
Cross bar of the chassis	180 mm × 75 mm × 4 mm
Front overhang (a)	935 mm
Wheel base (b)	3800 mm
Rear overhang (c)	1620 mm
Young's modulus E	2×10^5 N/mm^2
Poisson ratio	0.3
Radius of gyration R	100 mm

IV. METHODOLOGY

The finite element analysis (FEA) is a computing technique that is used to obtain approximate solutions to the boundary value problems in engineering. It uses a numerical technique called the finite element method (FEM) to solve boundary value problems.

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

FEA involves a computer model of a design that is loaded and analyzed for specific results. ANSYS is being used by designers across a broad spectrum of industries such as aerospace, automotive, manufacturing, nuclear, electronics, biomedical, and many more. ANSYS provides simulation solutions that enable designers to simulate design performance directly on desktop. In this way, it provides fast, efficient and cost-effective product development from design concept stage to performance validation stage of the product development cycle. ANSYS package help to accelerate and streamline the product development process by helping designers to resolve issues related to structural deformation, heat transfer, fluid flow, electromagnetic effects, a combination of these phenomena acting together, and so on. The finite element analysis is carried out on steel chassis as well as different types of polymeric composite heavy vehicle chassis. From the analysis the stress distribution (Von-mises stress) and deformations were carried out. The total load of chassis of magnitude 58860N is applied on each side of beam and the gravitational force of 9806.6N is also considered.

V. RESULTS

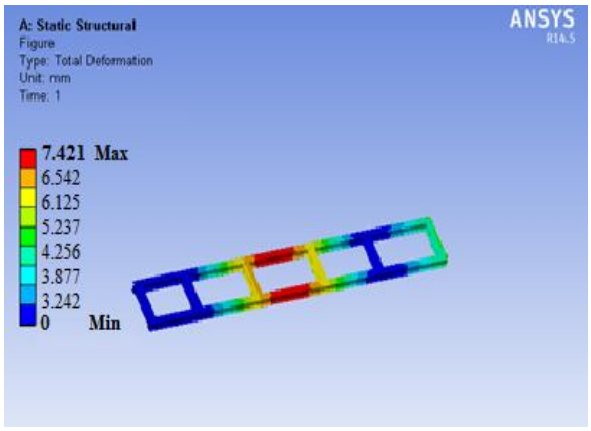


Figure 1 Contour of Deformation in steel

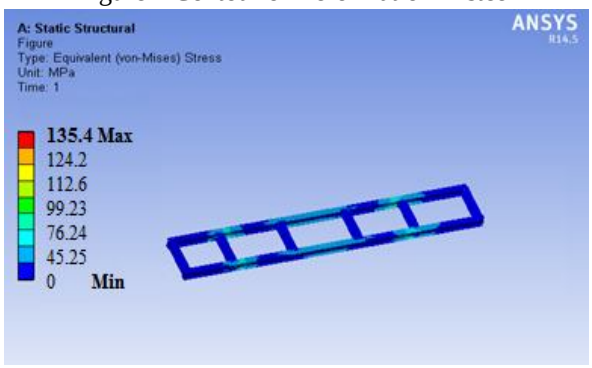


Figure: 2 Contour of Stress in steel

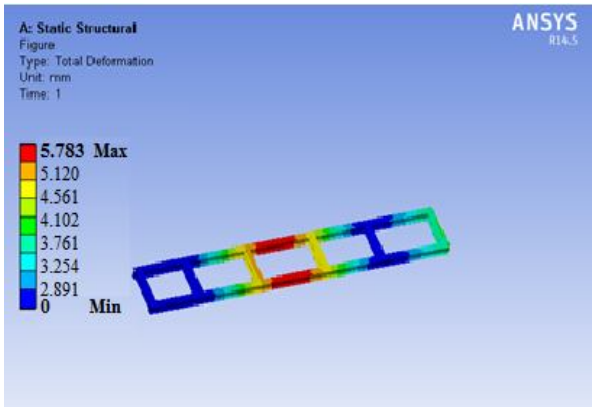


Figure: 3 Contour of Deformation in E-glass Epoxy

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

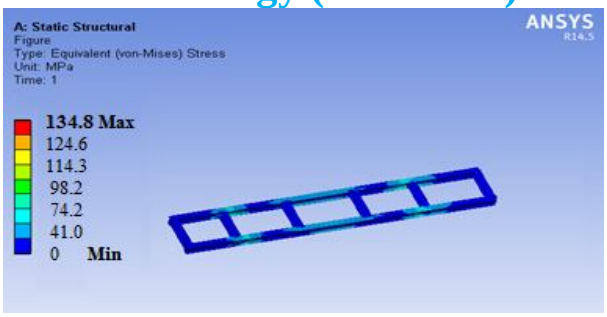


Figure: 4 Contour of Stress in E-glass Epoxy

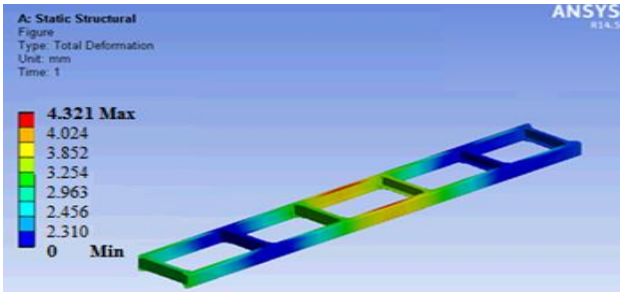


Figure: 5 Contour of Deformation in Carbon Epoxy

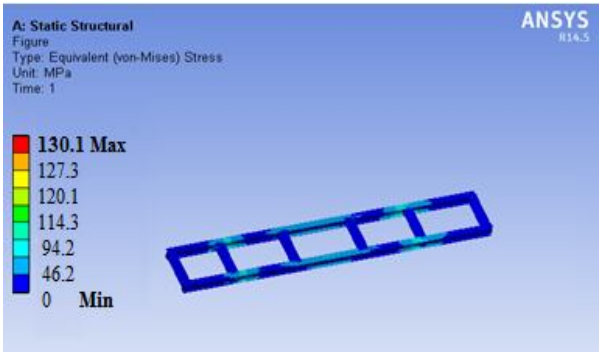


Figure: 6 Contour of Stress in Carbon Epoxy

Table no 1 Results Table

Type of Section	Material	Deformation (mm)	Stress (MPa)	Weight (Kg)
I-Type	Steel	7.42	135.4	640
I-Type	E-glass epoxy	5.78	134.8	470
I-Type	Carbon Epoxy	4.32	130	328

VI. CONCLUSION

Automotive Chassis structural design and analysis has been the focus of a number of previous works. The review of some of the previously conducted work related to vehicle structural design, analysis and optimization using Ansys software is surveyed. From FEM Simulation Results we found that I section of chassis frame is providing more strength to the structure in previous work. In

present work we compare the same geometrical section with different materials like Steel, E-glass Epoxy and Carbon Epoxy FEM using ANSYS. Simulation results shows that Carbon epoxy induces low deformation and stress distribution when compared to S-glass epoxy composite material and steel. It is inferred that by employing a carbon epoxy composites heavy vehicle chassis for same load carrying capacity, there is a reduction in weight when compared to steel. Carbon epoxy induces low deformation and stress distribution when compared to E-glass epoxy and steel.

VII. FUTURE SCOPE

A. The chassis strength can be improved by providing the stiffeners in 'I' channel section.

B. From linear static analysis, maximum deformation of the component and maximum stress can be known and from that the material can be changed if required to meet the loading condition.

C. The dynamic analysis can be done for finding the natural frequency and mode shape of chassis frame.

D. Finite Element Analysis can be used as a tool to redesign the component if it is already designed by classical design theory.

E. Feasibility of different design change & material change on stress & weight reduction can be done by FEA.

REFERENCES

[1] Salvi Gauri Sanjay, Kulkarni Abhijeet, Gandhi Pratik Pradeep, Baskar P – "Finite Element Analysis of Fire Truck Chassis for Steel and Carbon Fiber Materials" Salvi Gauri Sanjay et al Int. Journal of Engineering Research and Applications www.ijera.com ISSN : 2248-9622, Vol. 4, Issue 7(Version 2), July 2014, pp.69-7

[2] Archit Tomar1 & Dheer Singh – "Static Analysis, Modal Analysis and Design Modification in Chassis Frame to Optimize Weight by Using Composite Material" International Journal of Mechanical Engineering (IJME) ISSN(P): 2319-2240 ; ISSN(E): 2319-2259 Vol. 5, Issue 1, Dec – Jan 2016, 101-108 © IASE

[3] Airale, M. Carello, A. Scattina – "Carbon fiber monocoque for a hydrogen prototype for low consumption challenge" Volume - 42,Issue, 5 May2011 Pages 386-39

[4] Divyesh N. Chaudhari , Prof. Vinod M. Makwana, Prof. Dharmeet J. Patel – "A Review Paper on Strength and Rigidity analysis of Heavy Vehicle Chassis for different frame cross section by analytically and FEA under various loading condition" International Journal of Advance Research in Engineering, Science & Technology(IJAREST), ISSN(O):2393-9877, ISSN(P): 2394-2444, Volume 2,Issue 12, December-2015, Impact Factor: 2.12

[5] Singarajan Nagammal Vijayan1 and Sathiavelu Sendhilkumar – "Structural Analysis of Automotive Chassis Considering Cross-Section and Material" Int. J. Mech. Eng. Autom. Volume 2, Number 8, 2015, pp. 370-376 Received: June 28, 2015; Published: August 25, 201

[6] Vijayan S N, S Karthik, K Maharaja – "Finite Element Analysis of Automotive Chassis with Various Cross-Section" ISBN 978-81-929866-6-1, VOL 02 Website icssccet.org email icssccet@asdf.res.in Received 25 – February – 2016 Accepted 10 - March – 2016 Article ID ICSSCCET037 eaid ICSSCCET.2016.03

[7] Archit Tomar1 & Dheer Singh – "Modelling and Analysis of a Chassis Frame by Using Carbon Fiber and E-Glass Epoxy as Composite Material: A Comparative Study" International Research Journal of Engineering and Technology (IRJET), e-ISSN: 2395 -0056, Volume: 03 Issue: 04 | April-2016, p-ISSN: 2395-007

[8] Vijayan.S.N, S.Sendhilkumar, M.Prakash – "Effect of Cross-Sections Considering Material and Design Aspects in Automotive Chassis" IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 12, Issue 5 Ver. I (Sep. - Oct. 2015), PP 86-9

[9] Seelam Sreekanth Reddy, G.Amarnath – "Modeling and Analysis of Automobile Chassis Using Honeycomb Sandwich Structure" International journal & magazine of engineering, technology, management and research ISSN 2348-4845, vol-3 (2016), Issue no-11 (November

[10] Jagbhooshan Patel; Veerendra Kumar – "Finite Element Analysis of Mono Composite Leaf Spring: A Review:" International Journal of Engineering Sciences & Research Technology (IJESRT) (Vol.3, No. 5), 30-05-20

[11] M. Z. A Rashid, M. F. Abdul Latif, M. Nur Othman & Marizan Sulaiman – "Design and Simulation Study of Small Four Wheel Vehicle Chassis for Single Driver" 2016MATEC Web of Conferences 74, 7400029© The Authors, published by EDP Sciences. This is an open access article distributed under the terms of the Creative Commons Attribution License 4.0 (http://creativecommons.org/licenses/by/4.0/). ICMER 201

[12] Carl Andersson, Eurenius Niklas Danielsson, Aneesh Khokar, Erik Krane, Martin Olofsson, Jacob Wass – "Analysis of Composite Chassis"(2016

[13] Robert L. West, Chair John M. Kennedy Scott W. Case – "Design and Optimization of Carbon-Fiber Chassis Panels" May 27, 2014 Blacksburg, Virgini

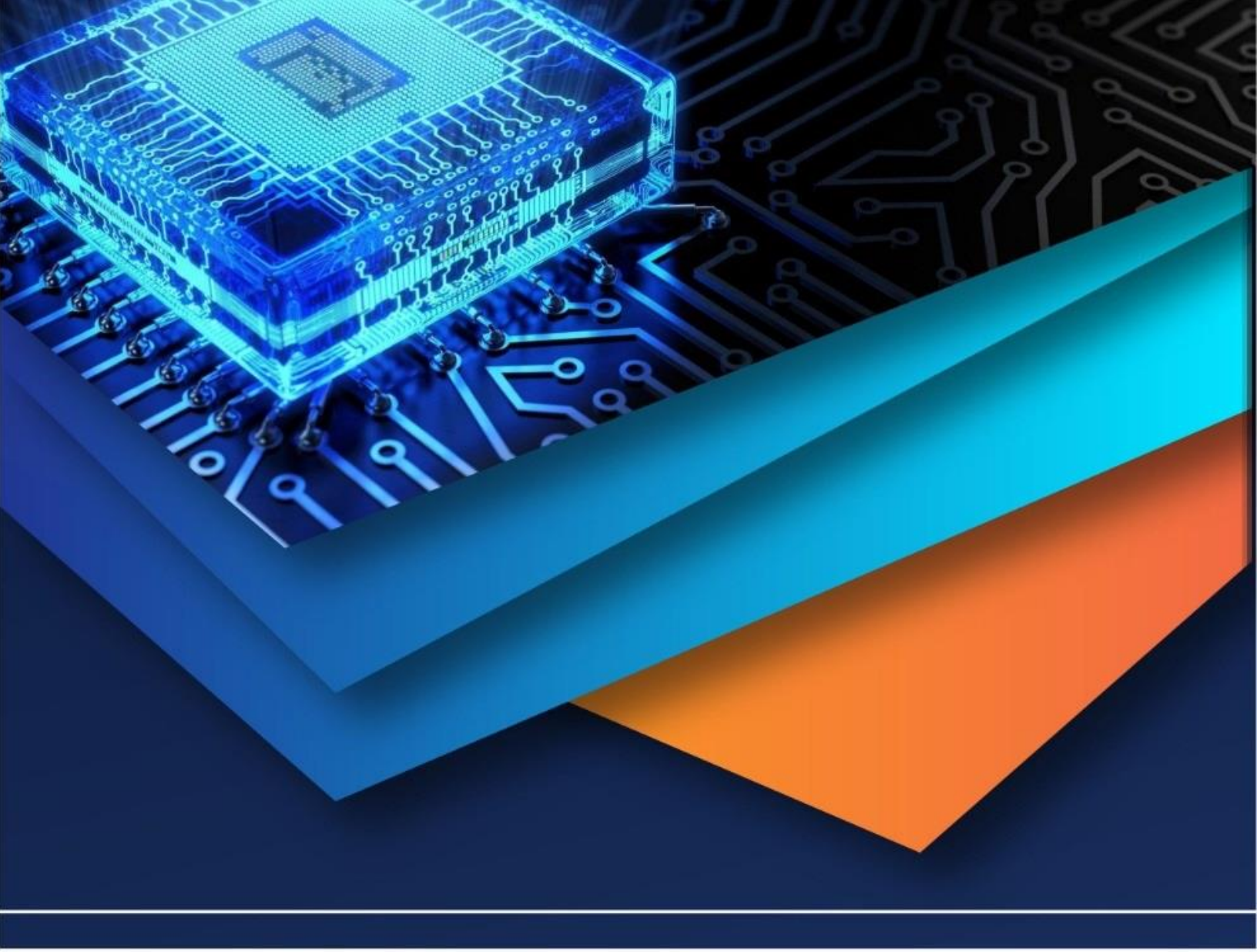
[14] Pizhong Qiao, Mijia Yang, and Florin Bobaru – "Impact Mechanics and High-Energy Absorbing Materials: Review" Published in Journal of Aerospace Engineering 21:4 (October 1, 2008), pp. 235–248; doi 10.1061/(ASCE)0893-1321(2008)21:4(235) Copyright © 2008 ASCE. Used by permission

[15] Hemant B.Patil, Sharad D.Kachave, Eknath R.Deore – "Stress Analysis of Automotive Chassis with Various Thicknesses" IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684 Volume 6, Issue 1 (Mar. - Apr. 2013), PP 44-49 www.iosrjournals.or

[16] S. Heimbs a, P. Middendorf b, M. Maier – "Honeycomb Sandwich Material Modeling for Dynamic Simulations of Aircraft Interior Components" Conference Paper-June 20

[17] .A.Hari Kumar, V. Deepanjali – "Design & Analysis of Automobile Chassis" ISSN: 2319-5967 ISO 9001:2008 Certified International Journal of Engineering Science and Innovative Technology (IJESIT) Volume 5, Issue 1, January 201

[18] Tandra Naveen kumar, N.Jeevan Kumar – "Design and Analysis of Army Vehicle Chassis" International Journal of Engineering Trends and Technology (IJETT) – Volume 9 Number 3 - Mar 2014



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY