International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 6 **Issue:** II **Month of publication:** February 2018

DOI:

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue II, February 2018- Available at www.ijraset.com

Design and Analysis of Body Structure: a Case of Locally Built Isuzu Passenger Bus

Dawit Tafesse [1], Ramesh Babu Nallamothu [2], Seshu Kishan Nallamothu[3], Anantha Kamal Nallamothu[4]
[1] School of Mechanical and Electro-Mechanical Engg, Institute of Technology, Hawassa University, Ethiopia.
[2] Mechanical and Vehicle Engineering Department, Adama science and Technology Universit PoBoX: 1888, Adama, Ethiopia
[3]Automobile Engineering Department, SRM University, Chennai, India.
[4]Mechanical Engineering Department, Vellore Institute of Technology, Vellore, India.

Abstract: Bus is a popular and common means of transport in Ethiopia. The structural strength and speed is a fundamental concern. Unnecessary weight of the bus structure leads to reduction of speed, high fuel consumption and reducing the overall performance of bus. The primary objective of this research is to investigate the bus structure strength. The improvement is carried out by analysing the structure for two main loading cases, bending load and torsional stiffness. This study examines the stresses and deformation responses of a typical bus structure during application of the load in service. A bus structure was modelled using computer aided design (CAD) CATIA V5R19 software and exported to finite element analysis ANSYS work bench15 software and simulated by applying boundary conditions. The structure weight improvement of the bus is made by reducing the size and thickness of metal used in the structure. In this analysis a new model of the bus was made and analysed. Two loading cases were incorporated in the analysis. Bending load case was used to simulate the bus travelling on smooth road and twisting moment is used to simulate a bus travelling on a bumpy road. This re-modelled bus is again simulated by finite element analysis software and found to be structurally safe. The structural strength analysis was investigated Total deformation was 0.83903mm, equivalent elastic strain 0.1282mm/m, equivalent vonmisses stress was 2.5213x×107pa,normal elastic strain was 3.509×105mm/m, maximum shear elastic strain was 0.1717mm/m, maximum shear stress was 1.3208×107pa .
Keywords: Carrying Capacity, Structure, Weight.

I. INTRODUCTION

The design of the internal bus skeleton structure is the basis of various bus developments in the bus industries. It contains the framework of tubes with various cross sections are arranged within specified shapes based on the design philosophy. The bus body structure have six main components the left and right frame side, the front and back frame side, the top and bottom frame side. In that the top frame side is sometime called roof frame side. The bottom frame side is also called floor frame side [1].

The stakeholders on the nowadays automotive market have started to tackle with a design process that takes advantage of various non-ferrous materials available on the market, exploiting their specific properties. The main outcome of a multi-material design is a significant mass Reduction and enhancements in structural mechanical properties e.g. increased stiffness of Crash worthiness. The motivation for weight minimization comes from the fact that the mass of new vehicles has been growing steadily for the last four decades, exhibiting 1.1% of an annual increase. Despite the fact that load bearing components are the subject of structural optimization approaches, the mass increase takes place due to the intensive use of auxiliary systems like air-conditioning, electronics or additional gas tanks. Moreover, from the economical point of view, lower curb weight result in decrease of operational costs. This is particularly important in the case of the mass transport (e.g. buses or coaches), for which the annual mileage is very high. These mechanical performances along with the weight minimization were the subject of many Optimization attempts.

The same authors in presented a study on enhancing the bus rollover resistance. They identified the components that exhibited high internal energy level and applied on them Structural modifications that were obtained by means of a simple one-variable regression based Optimization process, Similar studies were presented by Su et al. in in which the authors [2].

II. LITERATURE REVIEWS

A bus in any country is a kind of industries which is connected directly to the prosperity and maintains the stability of this state. After two years of completion of the security and sociality

Stability of the welfare states, it stopped immediately in case of any defects in the stability. The design of the internal bus skeleton structure is the basis of various bus developments in the bus industries. It contains the framework of tubes with various cross sections are arranged within specified shapes based on the design [3].

Our society's increasing requirements for mobility with simultaneously growing environmental sensitivity is a big challenge for the traffic policy makers and the transport corporations including private fleet operators. Consequently, it is also indispensable for the manufacturers of light and heavy passenger vehicles and the body builders to adapt to the ecologically motivated requirements, which becomes more and more important without compromising on basic minimum requirements of safety and comfort.

It is found that the young's modulus, which is a measure of stiffness, is high for structural steel. Also the load bearing capacity is higher for structural steel. Hence structural steel is selected as the material for analysis. The behaviour of the structure under different load conditions like static, cornering etc. are also analyzed. The scope of this project is to provide a light weight design which will reduce the weight as well as improve the stability under all the driving conditions. Cornering

In case of cornering, the major part of cornering load is acting on the side frames. When the vehicle turns a left turn most of the weight shifts to the right side in lateral direction. Hence a portion of load is applied on to the right side frame and Von-Mises stresses are evaluated [4].With the growing concerns on energy conservation and vehicle emission it is increasing pressure on vehicle manufacturers internationally to provide lighter weight components for vehicles. In order to minimize energy consumption, lightweight has become a critical issue because, weight is a key factor for the fuel consumption. The lighter the vehicle more will be the fuel efficiency [5].

III. MODELLING OF BUS STRUCTURE

The modelling of the Isuzu passenger bus structure is made with data collected from Adama Ethiopia Endalk garage Isuzu passenger bus body builder. Before measuring the structure, drawing of the bus structure is made. This helps for easy placement of the measured length on the corresponding members on the drawing. During the data collection of the structure, all members of the structure are measured with length measuring tape. The modelling of the bus structure is made with CATIA V5R19 software. The bus structure is made with steel beams of rectangular hollow section, with different size. This model Isuzu passenger bus has a capacity of 29 passengers.

Table1: Material property of rectangular hollow section used [6].

Material Property	Structural steel(st 37.2)
Tensile yield strength	360 MPa
Compressive yield strength	360 MPa
yield stress	220 MPa
Density	7850 kg/m3
Young's modulus	200 GPa
Elongation in 50mm gauge $5.65\sqrt{a^x}$ length	25

Before going to the finite element analysis, the modelling of the Isuzu bus structure should be made.

The figure given below shows the model of the Isuzu passenger bus made in CATIA V5R19 window. The structure parts are made with the real profile and dimension used in Endalk garage Adama Ethiopia. but its front and roof parts are modified

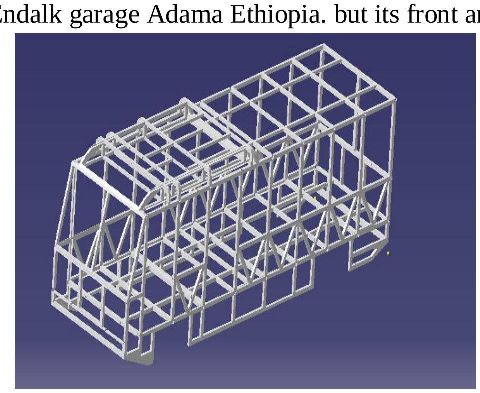


Figure1: Model Structure modified Isuzu passenger bus

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue II, February 2018- Available at www.ijraset.com

A. Structural strength

FEA has become a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material and allowing designers to see all of the theoretical stresses within. This method of product design and testing is far superior to the manufacturing costs which would accrue if each sample was actually built and tested. In practice, a finite element analysis usually consists of three principal steps:

1) *Pre-processing:* The user constructs a model of the part to be analyzed in which the geometry is divided into a number of discrete sub regions, or elements," connected at discrete points called nodes." Certain of these nodes will have fixed displacements, and others will have prescribed loads. These models can be extremely time consuming to prepare, and commercial codes vie with one another to have the most user-friendly graphical "pre-processor" to assist in this rather tedious chore. Some of these pre-processors can overlay a mesh on a pre-existing CAD file, so that finite element analysis can be done conveniently as part of the computerized drafting-and-design process.
2) Analysis: The dataset prepared by the pre-processor is used as input to the finite element code itself, which constructs and solves a system of linear or nonlinear algebraic equations KijUj = fi Where u and f are the displacements and externally applied forces at the nodal points. The formation of the K matrix is dependent on the type of problem being attacked, and this module will outline the approach for truss and linear elastic stress analyses. Commercial codes may have very large element libraries, with elements appropriate to a wide range of problem types. One of FEA's principal advantages is that many problem types can be addressed with the same code, merely by specifying the appropriate element types from the library.
3) *Post processing:* In the earlier days of finite element analysis, the user would pore through reams of numbers generated by the code, listing displacements and stresses at discrete positions within the model. It is easy to miss important trends and hot spots this way, and modern codes use graphical displays to assist in visualizing the results. A typical postprocessor display overlays colored contours representing stress levels on the model, showing a full field picture similar to that of photo elastic or moiré experimental results. Solving any problem analytically, you need to define [1].

You then solve the problem and present the results. In numerical methods, the main difference is an extra step called mesh generation. This is the step that divides the complex model into small elements that become solvable in an otherwise too complex situation. Below describes the processes in terminology slightly more attune to the software.

Finite element modelling Finite element modelling is made with ANSYS 15 Workbench. The structure model made by CATIA V5R19 software is imported to ANSYS 15 workbench. In order to import the CATIA made model to ANSYS 15 window, both software run at the same time. The model in CATIA is opened. Then with the import option found in ANSYS workbench, the modelling is imported. Figure shows the model of the bus structure in ANSYS workbench window

B. Build Geometry

Construct a three dimensional representation of the Isuzu passenger body structure to be modeled and tested using the work plane coordinate system within ANSYS.

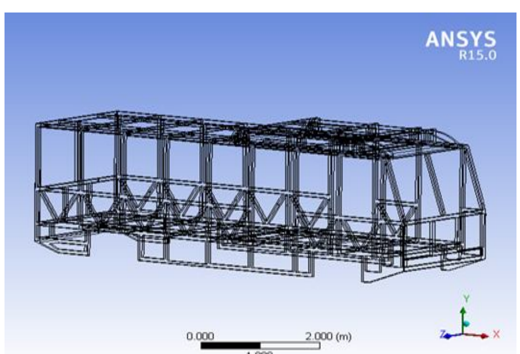


Figure2: Frame structure for Modified Isuzu passenger bus ansys window

Finite element meshing of the bus Finite element meshing is made with ANSYS 15 workbench. Meshing is an integral part of the computer-aided engineering (CAE) simulation process. The mesh influences the accuracy, convergence and speed of the solution. Furthermore, the time it takes to create a mesh model is often a significant portion of the time it takes to get results from a CAE solution. Tetrahedral mesh elements are used in meshing of the bus structure

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue II, February 2018- Available at www.ijraset.com

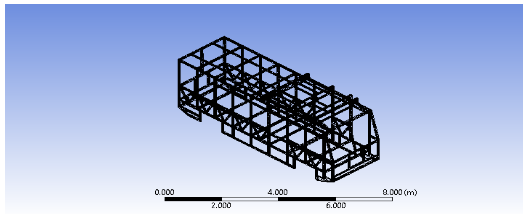


Figure3. Generate Mesh Inputs for Finite Element Analysis

The inputs for the finite element analysis are all the loads carried by the bus. These loads Carried by the bus during operation are self-weight of bus=17tone, passenger weight=65kg assumption times the number of passengers, fuel weight is 200kg. There are different loading cases that act on the bus structure such as aerodynamic load 15,852.6N. But the two main loading cases are bending load due to the weight and torsional stiffness due to the relative vertical movement of the wheels.

C. Structural Analysis Of The Bus

The structure model constitutes assemblies of individual members to form the whole structure. When the model is imported in to ANSYS 15 workbench, the ANSYS 15 workbench automatically recognizes the members in contact. The contact surfaces in reality are the places where welds are found. But to in incorporate the weld effects in to the analysis, we need to use ANSYS 15 workbench weld representative element. Therefore, in ANSYS 15workbench, there are two ways of considering welded assemblies. The first one is contact element where it is used

When we have a solid model, the other one called spot-weld connection is used when we have model of surface bodies assembled together [7].

Figure 4: structural analysis of model 3 Isuzu passenger bus

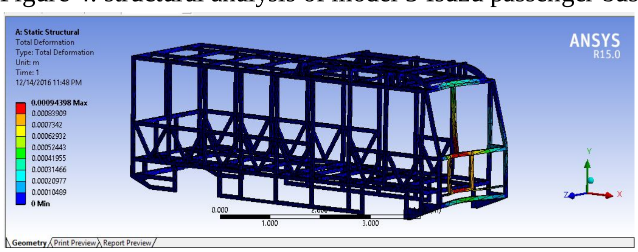


Figure 5: Deformation

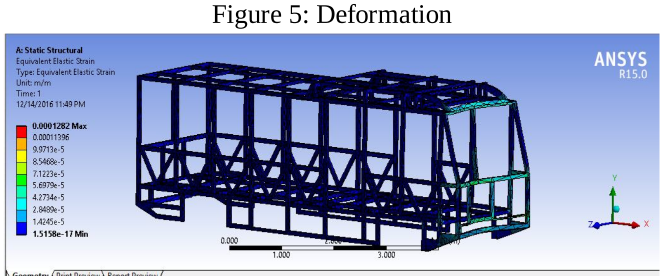


Figure 6: equivalent elastic strain

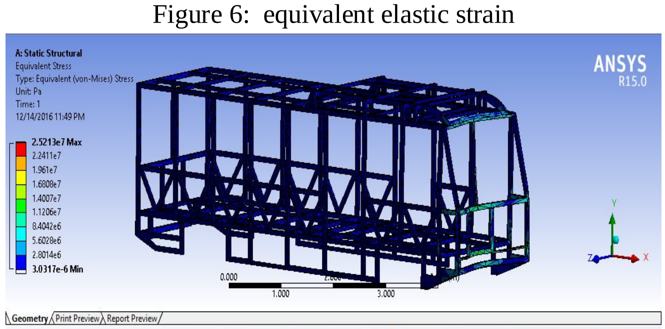


Figure 7: von- mises stress

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue II, February 2018- Available at www.ijraset.com

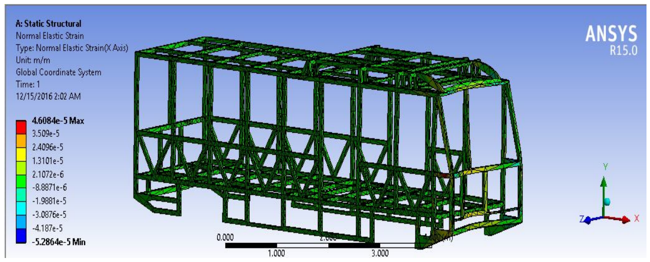


Figure 8: Normal elastic strain

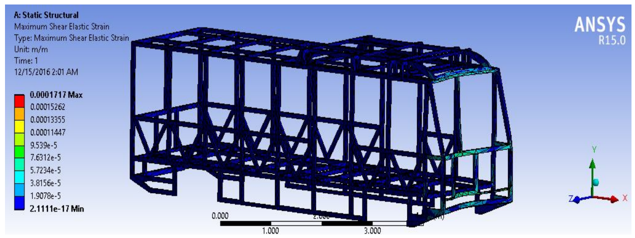
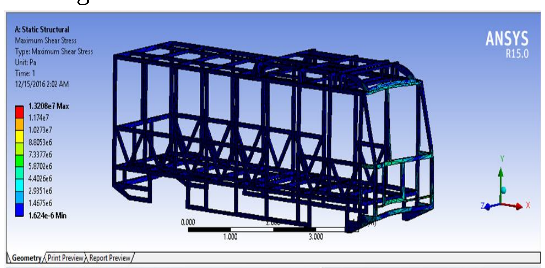


Figure 9: shear elastic strain



Objective was to investigate the structural strength of modified Isuzu passenger bus. In this project, a model 3 Isuzu passenger bus body structure is designed and modelled in 3D modelling software CATIA VR19 and ansys work bench15. The dimensions of the body structure are taken from Endalk garage Adama Ethiopia. The original body is as it is but at the front small modification and at roof air conditioning unit. The structural strength analysis Total deformation is 0.83903mm, equivalent elastic strain 0.1282mm/m, equivalent (Von-mises) stress is 2.5213×107pa, normal elastic strain was 3.509×105mm/m, maximum shear elastic strain is 0.1717mm/m, maximum shear stress was 1.3208×107pa.the results are shown in table 4.1.5

Table 2: The result of FEM analysis

	St 37.2
Total deformation	0.83903mm max
Equivalent elastic strain	0.1282mm/m max
Equivalent (von- mises) stress	2.5213×10^{7}pa max
Normal elastic strain	3.509×10^{5}mm/m max
Maximum shear elastic strain	0.1717mm/m max
Max shear stress	1.3208×10^{7}pa max

IV. CONCLUSION

The structural strength of modified Isuzu passenger bus was investigated. In this thesis work, a model 3 Isuzu passenger bus body structure is designed and modeled in 3D modeling software CATIA VR19 and ansys work bench15. The dimensions of the body structure are taken from Endalk garage Adama Ethiopia. The structural analysis was made on body structure after making slight modification in front and roof air conditioning unit. The structural strength analysis results are as follows: total deformation is 0.83903mm, equivalent elastic strain 0.1282mm/m, equivalent von-mises stress is 2.5213×107pa, normal elastic strain was 3.509×105mm/m, maximum shear elastic strain is 0.1717mm/m, maximum shear stress is 1.3208×107pa.

REFERENCES

[1] Prasannapriya. Chinta , Dr.L.V.Venugopal Rao . A New Design and Analysis of BUS Body Structure. IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)Volume 11, Issue 5 Ver. I (Sep- Oct. 2014),

[2] Jakub Korta, Tadeusz Uhl.MULTI-MATERIAL DESIGN OPTIMIZATION OF A BUS BODY STRUCTURE. Journal of KONES Powertrain and Transport, Vol. 20, No. 1 2013

[3] Swapnil Khatavkar1, Sanjay R. Kumbhar etal. Structural analysis of passenger bus body using fea by changing joint structure for improving strength. international journal of current engineering and scientific research (ijcesr).VOLUME-3, ISSUE-6, 2016

[4] SREENATH S,K Kamalakkannan..design and analysis of a bus body side frame. sreenath351@gmail.com, Proceedings of 36th IRF International Conference, 14th February, 2016, Chennai, India, ISBN: 978-93-85973-37-6. 14th February, 2016.

[5] Mahesh Suresh Sabale, 2N.Vivekanandan etal. Design and analysis of automotive roof by using modern materials forms like plastics as an effective alternative. Sabale et al, International Journal of Advanced Engineering Research and Studies E-ISSN2249–8974July-Sept.,2014/38-44.

[6] The McGraw-Hill Companies, MACHINE DESIGN DATABOOK, (www.digitalengineeringlibrary.com), 2004.

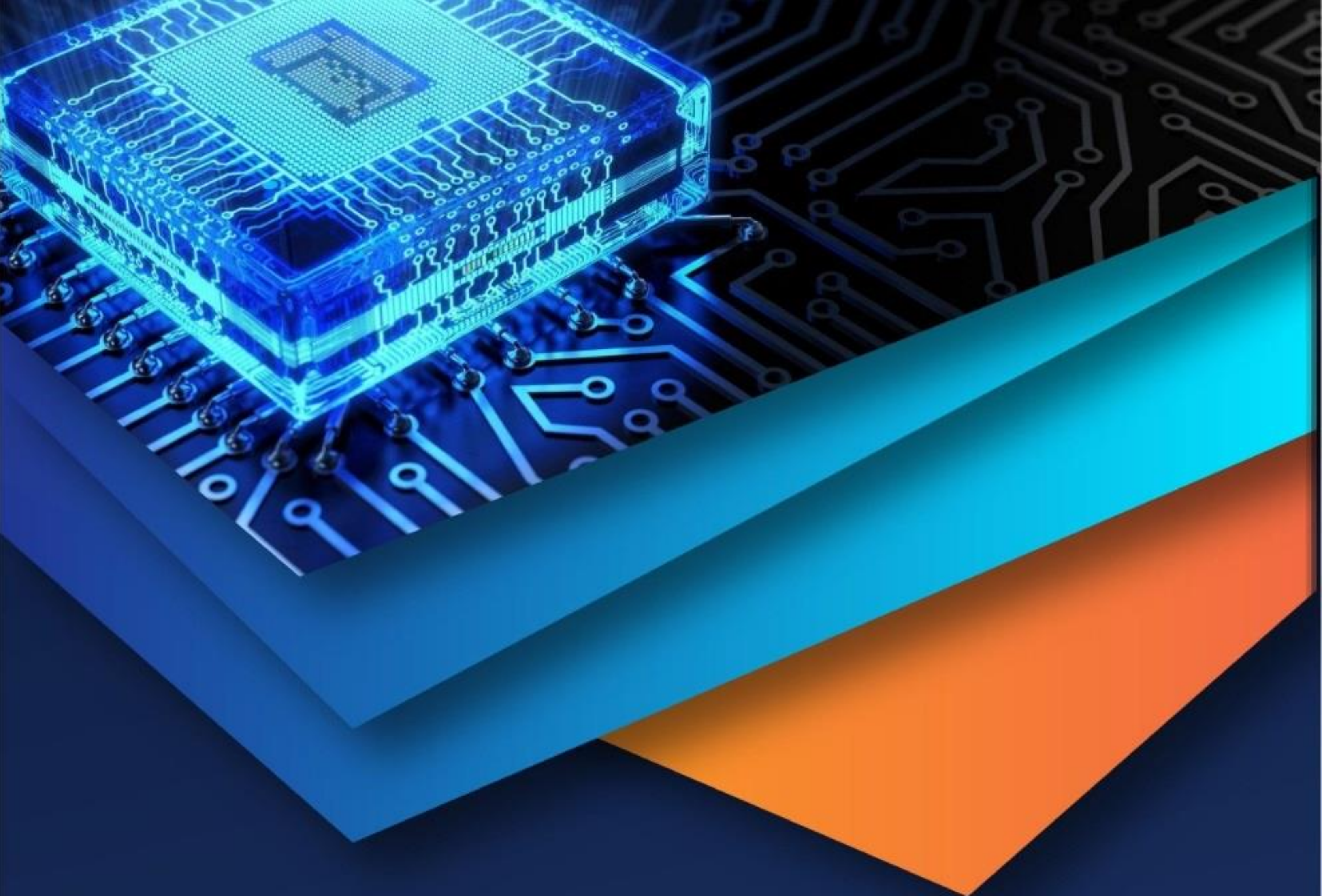
[7] Finite element analysis and concepts," Guide for the Use and Applicability of Workbench Simulation Tools from ANSYS, Inc."2009.

[8] Lemi Abebe, Ramesh Babu Nallamothu, K.H.S. Subrahmanyam, Seshu Kishan Nallamothu, Anantha Kamal Nallamothu " Thermal Analysis of Disc Brake Made of Different Materials" SSRG International Journal of Mechanical Engineering (SSRG-IJME), V3(6),5-9 June 2016, ISSN: 2348 – 8360. Doi> 10.14445/23488360/IJME-V316P102

[9] Ermias Feliche, Ramesh Babu Nallamothu et al " A Study on cause of rollover of sugar cane haulage semitrailer truck in Ethiopia Sugar Estate: case of Wonji Sugar Factory", International journal of Engineering trends and Technology (IJETT), V43(4),205-211 January 2017. ISSN:2231-5381. www.ijettjournal.org. published by seventh sense research group. Doi: 10.14445/22315381/IJETT-V43P235

[10] H. Yesfalgn Damissie , N. Ramesh Babu "Aerodynamics drag reduction on locally built bus body using computational fluid dynamics (CFD): a case study at Bishoftu automotive industry" International Journal of Engineering Research & Technology (IJERT), ISSN: 2278-0181, Vol. 6 Issue 11, November – 2017.

[11] Girmachew Ashagrie , Ramesh Babu Nallamothu, Anantha Kamal Nallamothu, Seshu Kishan Nallamothu "A Study on Driving Stability of Bus Using Computational Fluid Dynamics (CFD)" International Journal for research in Applied Science and Engineering Technology,Volume : Volume 5, Issue XI, November 2017.



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY