iJRASET
International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 4 **Issue:** VIII **Month of publication:** August 2016

DOI:

www.ijraset.com

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

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Static Analysis of Alloy Wheel Using FEA

Praveen S.Ullagaddi[1], Sangangouda P.[2], Chandankumar J.[3], Dayanand H.[4], Karthik A.S.[5]
Sksvmacet-Laxmeshwar, Visveshwaraya Tecnological University

Abstract— Vital efforts have been made to identify the Finite Element Techniques for analyzing stress and displacement distribution in vehicle wheels subjected to inflation pressure and radial load. The model is done by using "CATIA V5" and the analysis is carried out by using "Ansys workbench" finite element package. The constitutive material model selected for the analysis is linear elastic with isotropic conditions. Various alloys of magnesium, aluminium, titanium are used to check the capacity of the wheel and finally the best is chosen by comparing the results.

Keywords— Alloy wheel, CATIA V5, Linear Isotropic Materials, ANSYS WORKBENCH 14.5

I. INTRODUCTION

Wheel is an important structural member of the vehicular suspension system that supports the static and dynamic loads encountered during vehicle operation. Since the rims, on which cars move, are the most vital elements in a vehicle, they must be designed careful-ly. Safety and economy are particularly of major concerns when designing a mechanical structure so that the people could use them safely and economically. Style, weight, manufacturability and performance are the four major technical issues related to the design of a new wheel and/or its optimization. The wheels are made of either steel or cast/forge Aluminum alloys. Aluminum is the metal with features of excellent light-ness, corrosion resistance, etc. In particular, the rims, which are made of Aluminum casting alloys, are more preferable because of their weight and cost.[1]

Automotive manufacturers have been developing safe, fuel efficient and lightweight vehicular components to meet governmental regulations and industry standards (Stearns, 2000). In the real service conditions, the determination of mechanical behavior of the wheel is important, but the testing and inspection of the wheels during their development process is time consuming and costly. For economic reasons, it is important to reduce the time spent during the development and testing phase of a new wheel. A 3–D stress analysis of Aluminium wheels of the car involves complicated geometry. Therefore, it is difficult to estimate the stresses by using elementary mechanical approximations.For this purpose, Finite Element Analysis(FEA) is generally used in the design stage of product development to investigate the mechanical performance of prototype designs. FEA simulation of the wheel tests can significantly reduce the time and cost required to finalize the wheel design. Thus, the design modifications could be conducted on a component to examine how the change would influence its performance, without making costly alteration to tooling and equipment in real production. .[1]

II. WHEEL NOMENCLATURE

A. Wheel

Wheel is generally composed of rim and disc.

B. Rim

This is a part where the tire is installed.

C. Disc

This is a part of the rim where it is fixed to the axle hub.

D. Offset

This is a distance between wheel mounting surface where it is bolted to hub and the centerline of rim.

E. Flange

The flange is a part of rim, which holds the both beads of the tire.

F. Bead Seat

Bead seat comes in contact with the bead face and is a part of rim, which holds the tire in a radial direction.

G. Hump

It is bump what was put on the bead seat for the bead to prevent the tire from sliding off the rim while the vehicle is moving.

H. Well

This is a part of rim with depth and width to facilitate tire mounting and removal from the rim.[2]

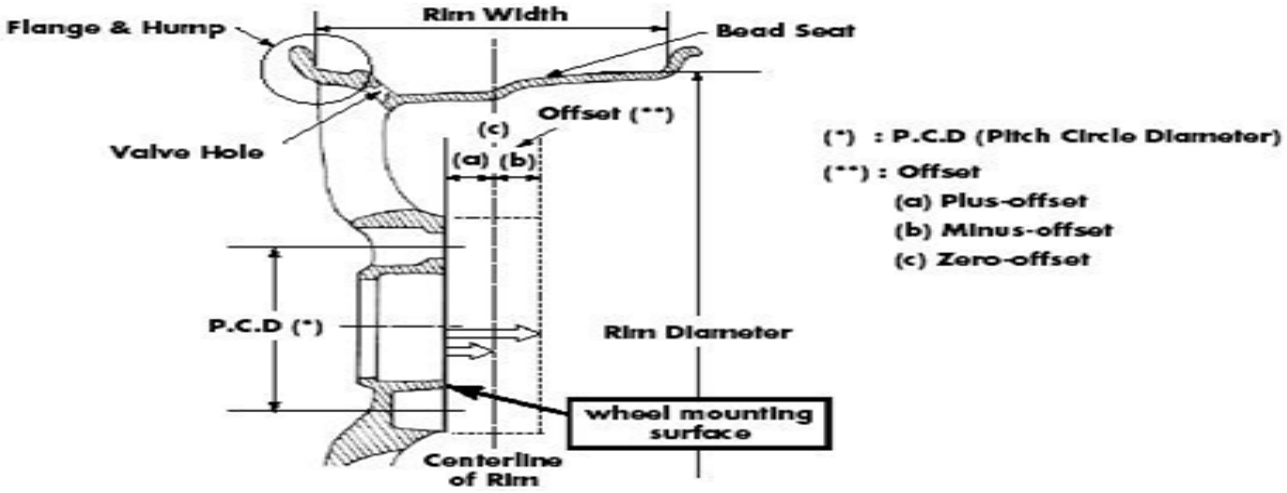


Fig.1 Wheel nomenclature[1]

III. MODELING WITH CATIA V5

CATIA is useful software for design analysis in mechanical engineering. CATIA is a design analysis automation application fully integrated with Solid Works. This software uses the Finite Element Method (FEM) to simulate the working conditions of your designs and predict their behaviour. FEM requires the solution of large systems of equations. Powered by fast solvers, CATIA makes it possible for designers to quickly check the integrity of their designs and search for the optimum solution. A product development cycle typically includes the following steps:[3]

A. Build your model in the Solid Works CAD system.
B. Prototype the design.
C. Test the prototype in the field.
D. Evaluate the results of the field tests[3]

IV. ANSYS 14.5

ANSYS is a general purpose finite element modeling package for numerically solving a wide variety of mechanical problems. These problems include: static/dynamic structural analysis (both linear and nonlinear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems .[5]

There are two main steps in a typical ANSYS analysis:

A. Model generation

1) Simplifications, idealizations.
2) Define materials/material properties.
3) Generate finite element model (mesh).

B. Solution

1) Specify boundary conditions.
2) Obtain the solutio

V. SPECIFICATIONS OF WHEEL

A. The typical chemical composition of the material for Aluminium alloy(%) is copper-0.25,maganese-0.35,silicon-6.5 to 7.5,

Volume 4 Issue VIII, August 2016
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

iron-0.6, zinc-0.35, others-0.05, aluminum-87 to 100.[4]

B. Magnesium alloy (%) is maganese-0.6 to 1.4, calcium-0.04, silicon-0.1, copper-0.05, nickel-0.005, iron-0.005, magnesium-85 to 100.[4]

C. Titanium alloy(%) is Aluminium-.03%,Vanadium-.06%,Chromium-.4%,Titanium 82-100.[4]

VI. BOUNDARY CONDITIONS

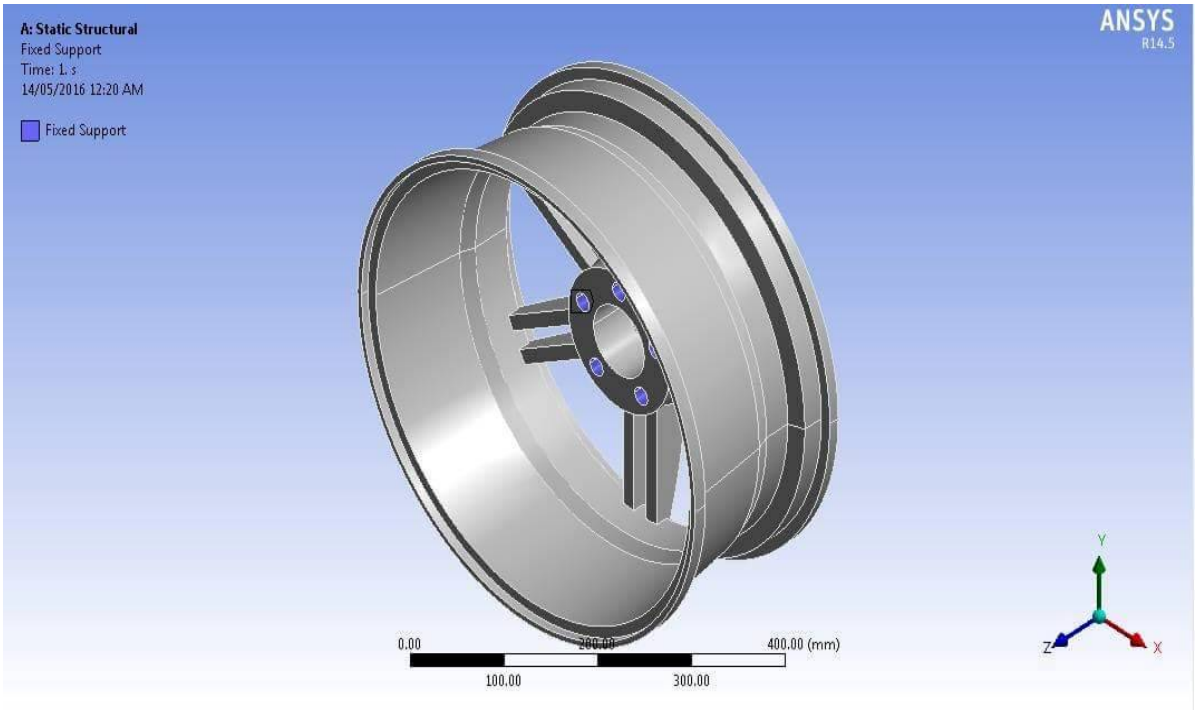


Fig 2.fixed support

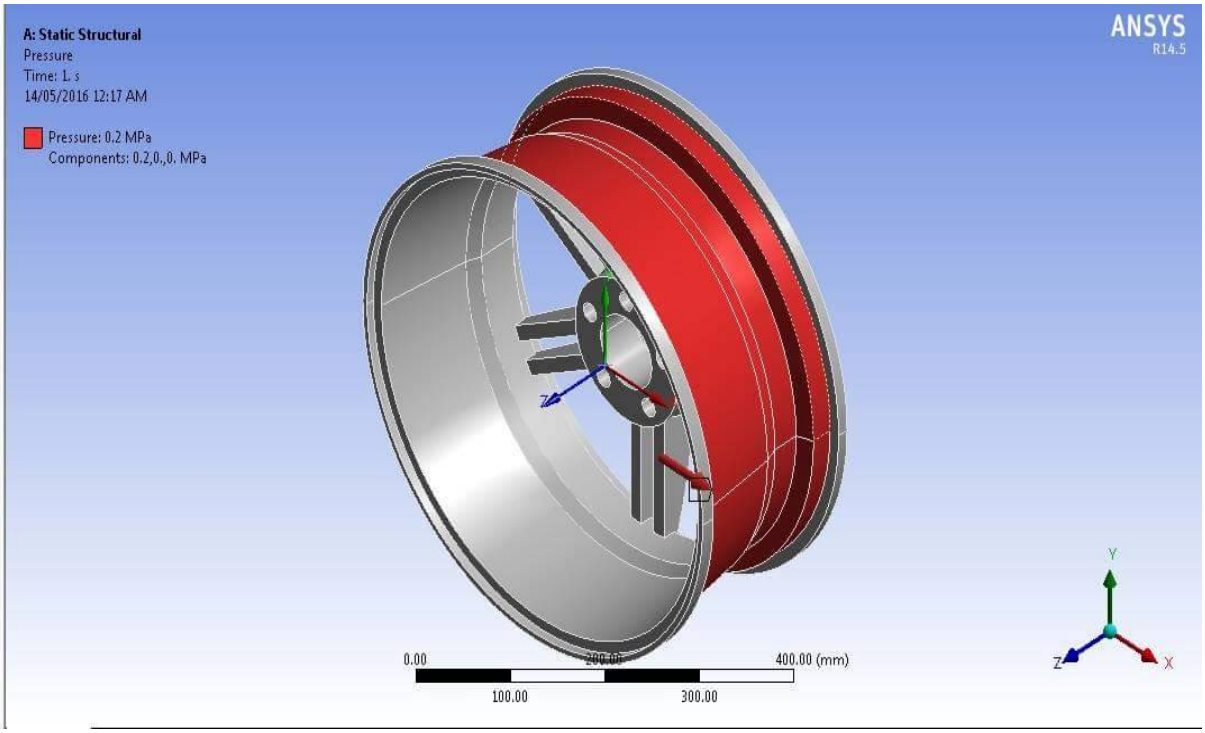


Fig 3.Pressue applied-210kpa[6]

Volume 4 Issue VIII, August 2016
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

VII. RESULTS AND DISCUSSIONS

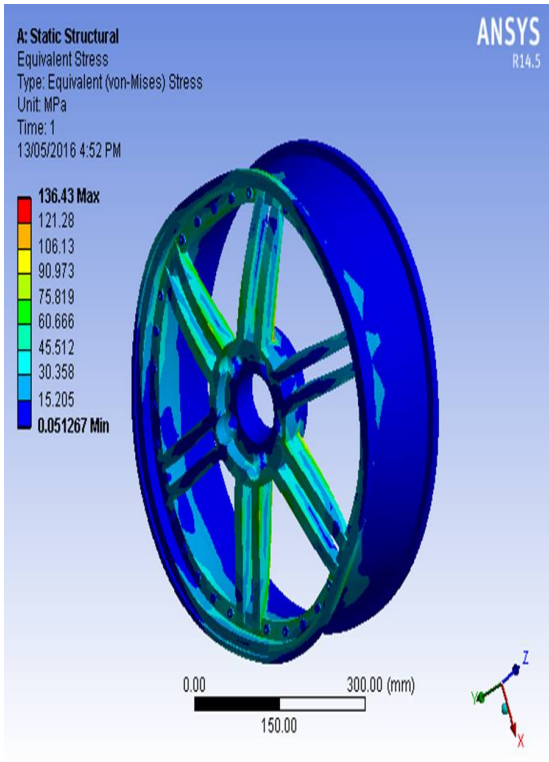


Fig. 4 Von-Mises Stress For Aluminium

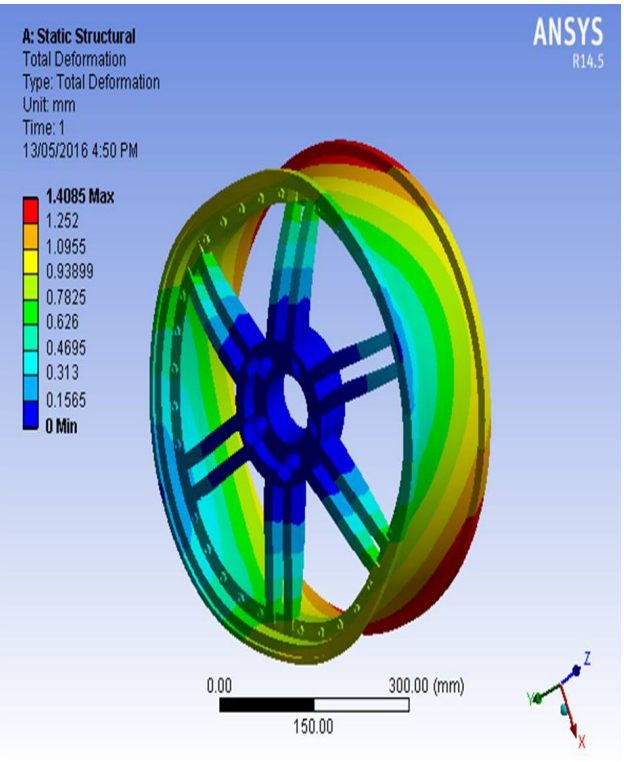


Fig. 5 Total Deformation For Aluminium

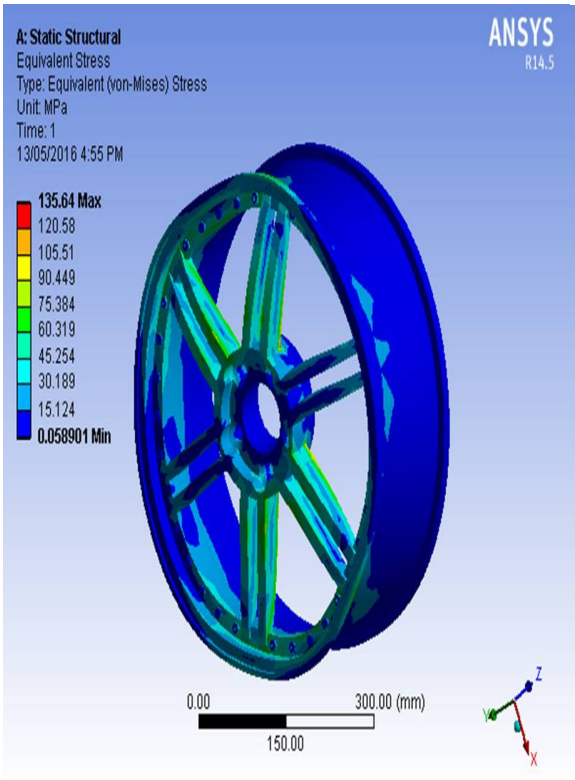


Fig.6 Von-Mises Stress For magnesium

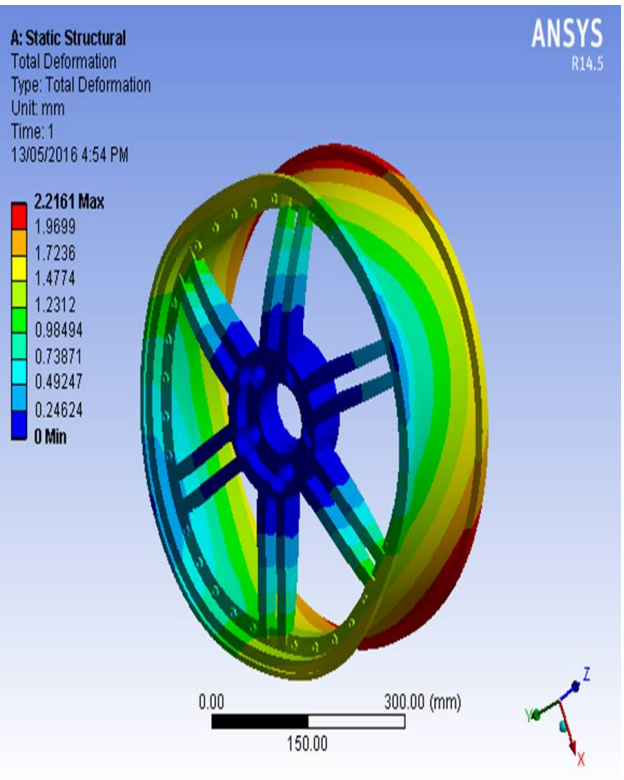


Fig. 7 Total Deformation For magnesium

Volume 4 Issue VIII, August 2016
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

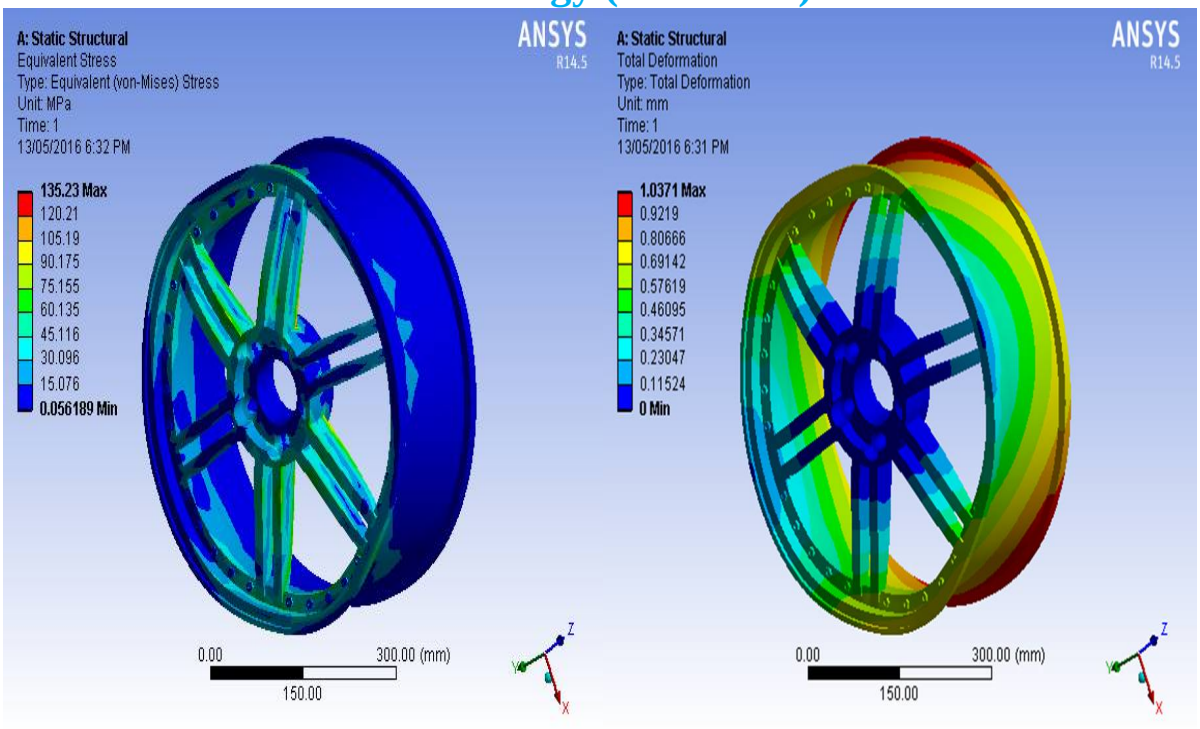


Fig.8 Von-Mises Stress For titanium

Fig. 9 Total Deformation For titanium

Table no.1 Comparison table

Material	Von-Mises Stress(Mpa)	Deformation(mm)
Aluminium	136.43	1.4085
Magnesium	135.64	2.2161
Titanium	135.23	1.0371

VIII. CONCLUSION

As in the case of an automobile wheel maximum load is applied on the alloy wheel. Analysis of the wheel plays an important role for the safety of the passenger cars. A pressure of 210Kpa is applied on the outer surface of the rim. The pitch circle holes are constrained in all degrees of freedom. The analysis is carried under these constraints . The equivalent stress of wheel maximum was 136.43Mpa for aluminium. The maximum deformation noted was for Magnesium 2.2161mm.As per the comparison table titanium has less equivalent stress of 135.23Mpa and less deformation of 1.031mm.Economically titanium is costly compared to magnesium and aluminium.But as per the life span is considered it is the best suited material for wheel .

IX. ACKNOWLEDGEMENT

It is immense pleasure to express my deepest sense of gratitude and sincere thanks to my respected Mr. Karthik A.S. Assistant Professor, to complete this paper. I would like to express thanks to Prof. R.D.Grampurohit and Dr.S.V.Gorbal without their encouragement love and blessings I would not have reached this level manuscript.

REFERENCES

[1] Sivakrishna. V and Bala bashker. J (2014), Impact Analysis of Aluminum Alloy Wheel,International Journal & Magazine of Engineering, Technology, Management and Research, Volume No: 1(2014), Issue No: 12 ,ISSN No: 2348-4845.

[2] Jaspreet Singh and Siddhartha Saha(2015), static analysis of alloy wheel using ansys15.0, IJRET: International Journal of Research in Engineering and Technology, Volume: 04 Issue: 07 | July-2015, eISSN: 2319-1163 | pISSN: 2321-7308

[3] Mr. Sasank Shekhar Panda, Mr. Dibya Narayan Behera, Mr. Satya Narayan Tripathy(2016), modeling and structural analysis of alloy wheel using ansys, international journal of engineering sciences & research, issn: 2277-9655

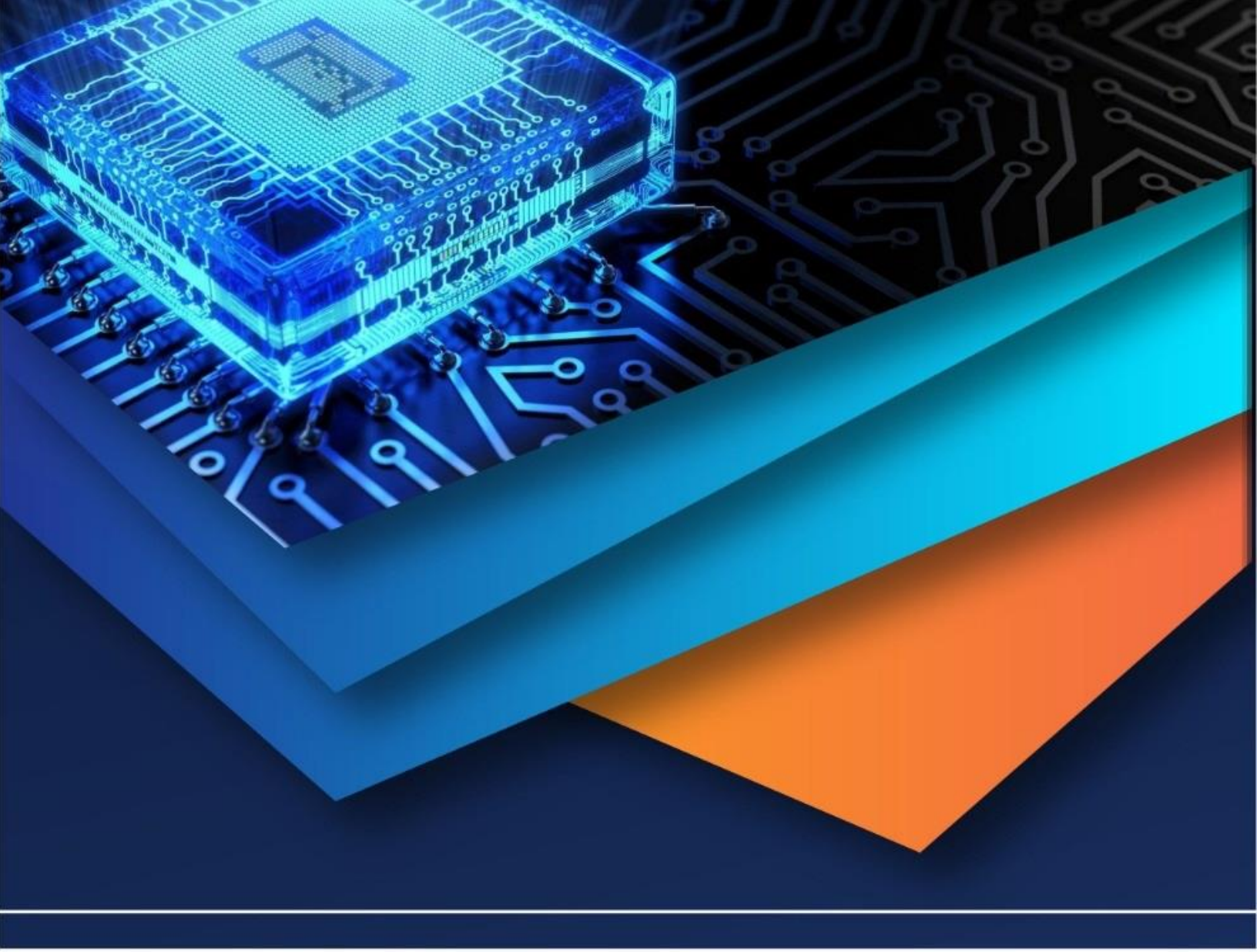
[4] S vikranth deepak, c naresh, and syed altaf hussain(2012), modelling and analysis of alloy wheel For four wheeler vehicle, school of mechanical engineering,

Volume 4 Issue VIII, August 2016
ISSN: 2321-9653

rgm college of engineering & technology, nandyal 518501, india.

[5] Ansysr multiphysics tm, release 14.5, help system, ansys workbench help, ansys, inc

[6] J. Janardhan, v. Ravi kumar, r. Lalitha narayana,(2014),radial fatigue analysis of an alloy wheel, intenatinal journal of engineering research and applicationsissn : 2248-9622, vol. 4, issue 12(part 6), december 2014, pp.253-258



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY