iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 6 **Issue:** VIII **Month of publication:** August 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 VIII, August 2018- Available at www.ijraset.com

Analysis of Stress Induced in Connecting Rod of Two Wheeler Engine

[1]Mithilesh Kumar, Jitendra Jadon[2]
[1]M.Tech Scholar, [2]Assistant Professor, Department of Mechanical Engineering, Shobhit University, Meerut

Abstract: A connecting Road mainly connect crankshaft and piston during combustion from the combustion chamber. A connecting rod is made of different material like carbon steel or any alloy. My aim of this project is to design a connecting rod of standard dimension using PRO E software and to analyze the design by using analysis software ANSYS like stress, strain, factor of safety. Different forces to be calculated by using analysis software. KEYWORDS ansys, connecting rod, static analysis, carbon steel, aluminium

I. INTRODUCTION

A connecting rod transfer power to the crank during combustion to generate rotational energy. A connecting rod may be of two different type, H type of I type or both used according to their field. An I-beam is light in weight and strong but it has limitation to handle the load due to type of material used where H-type is better than I type which handle more stress so, H-Beam type can be used in heavy engines.

1) Why Aluminum ?

I have selected aluminium alloy for this project. Aluminium is light in weight, easily available , and low cost. So, our choice is ALUMNIUM 2024 the chemical compositon of Al 2024 is tabulated below in Table.

AL	Chromium, Cr	Copper, Cu	Iron, Fe	Magnesium, Mg	Manganese, Mn	Silicon, Si	Titanium, Ti	Zinc, Zn
90.4 - 95 %	0.10 %	3.9 - 5.0 %	0.70 %	0.20 - 0.80 %	0.40 - 1.2 %	0.50 - 1.2 %	0.15 %	0.25 %

ENGINE SPECIFICATIONS

Here i have used A Bajaj Pulsar 150 engine which is a four stroke engine with 149 cc of displacement.
Below the specification of Engine:
Bore*Stroke=58.00 mm* 56.00 mm
The compression ratio of the engine = 9.5:1
Maximum power= 15.06 Ps@9000 rpm
Maximum Torque= 12.28 Ps@6500 rpm
Density of petrol= 737.22 kg/m3
Temperature=60F=288.855 K
Displacement= 149 cc
Mass of petrol=Density*Volume
=737.22*E^-9 * 149*10^3
=0.11 kg
Molecular weight of petrol=114.228 g/mole
Gas equation,
PV=MrT {R=R*/Mw=8.3143/0.114228=72.786}
P=0.11*72.786*288.855/149E^3=15.521 Mpa

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 VIII, August 2018- Available at www.ijraset.com

2) Designing Process

Pro E software is used to design the connecting rod of two wheeler Engine according to the standard dimensions. Thereafter the design is imported into Ansys to analyse the stress, and displacement.

II. LITERATURE SURVEY

There are many literature related to Finite Element Analysis of connecting rod. Many research publications, journals, reference manuals. books are available of national and international editions dealing with basic concepts of FEA. For the current study it is necessary to study about the stress analysis of connecting rod. A connecting rod should be durable , due to this connecting rod is an Important topic in the field of research.

1) *Webster et al.* IN (1983) performed 3D finite element analysis of a high diesel engine connecting rod. For this analysis they used the maximum compressive load which was measured experimentally, and the maximum tensile load which is essentially the inertia load of the piston assembly mass.[1] The load distributions on the piston pin end and crank end were determined experimentally. They modeled the connecting rod cap separately, and also modeled the bolt pretension using beam elements and multi point consequations. [2].
2) *Hitesh Kumar* [3] et al. did analysis of connecting rod stress producing compressive loading. And then give idea about weight reduction opportunities in connecting rod of an I.C. engine by examining two materials, AISI 1040 carbon steel and AISI 4340 alloy steel. Therefore, this study has deal with two subjects, first, static load and stress analysis of the connecting rod and second optimization for weight reduction and shape. With the use of FEA von-misses stress, strain, shear stress, deformat ion, and weight reduction etc, were calculated for a part circular loading conditions using FEA Software ANSYS WORKBENCH 14.0.Compared to the former material the new material found to have less weight, stress reduction and better stiffness. The connecting rod failed at the location indicated by the fea. An axi-symmetric model was initially used to obtain the stress concentration factors 9 at the thread root.
3) *Shahrukh Shamim* et al studied finite element analysis of connecting rod used in single cylinder four stroke petrol engines. Static stress analysis is conducted on connecting rod made up of two different materials viz. E-glass/Epoxy and Aluminum composite reinforced with Carbon nano tubes. Modeling and comparative analysis of connecting rod is carried out in commercially used FEM software ANSYS 14.0. Static structural analysis was done by fixing the piston end and applying load at the crank end of the connecting rod. Output parameters in static stress analysis are von-Mises stress, Shear stress, total deformation and equivalent elastic strain for the given loading conditions. [4]

III. PROBLEM STATEMENT

The design of the Connecting rod starts with the definition of the connecting rod geometry using 3D CAD program. The idea behind finite analysis is to divide a model into a constant finite number of elements. The system software generates and forecasts the complete stiffness of the total rod. evaluating the data is possible forecast how the connecting rod will behave in a real loading condition and allows the engineer to see where the FEA is carried out by using the ANSYS programmable software. Based on the analysis of optimal result, the stress concentrates on the model has become evaluate, which provides a better hint for rebuilt of connecting rod.

A. Problem Formulation

The objective of present study is carried out the analysis of a two wheeler connecting rod of different materials and making a meaningful comparison among results of analysis, which can be helpful for getting suitable material for the manufacturing of connecting rod. Study also incorporates the fabrication of material by changing the chemical constituents to find the new material. In the present study 3D model of connecting rod is developed using CAE software (PRO-E SOFTWARE) and the static analysis is done by ANSYS WORKBENCH 13.0. The results of analysis are noted for making helpful comparison.

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 VIII, August 2018- Available at www.ijraset.com

IV. MODELING & ANALYSIS

PRO E is Used to create the Model according to the standard dimension. First 2D model is developed and then 3d Model is generated.

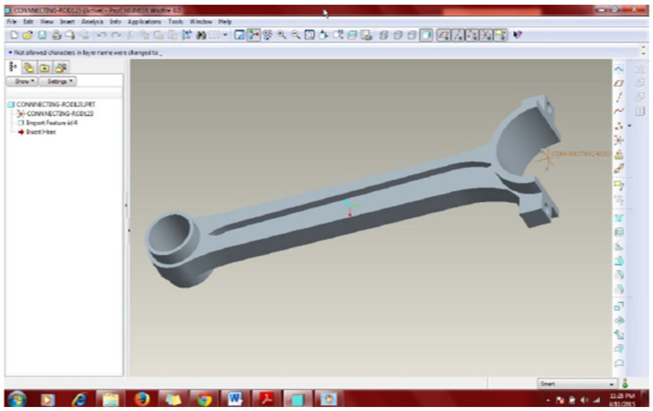

Figure 1: Model of connecting Rod

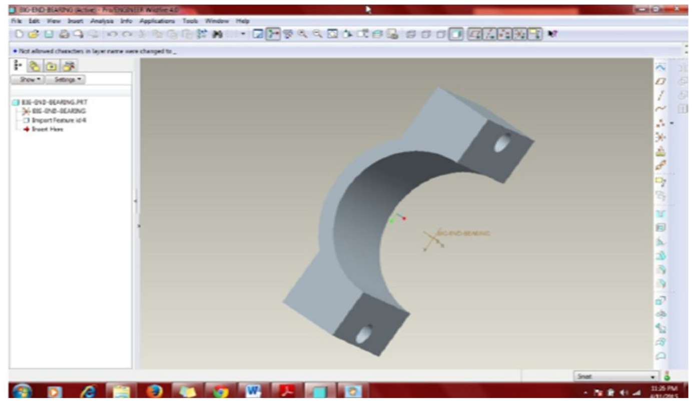

Figure 2 : Model of Crank end Bearing Lower half

Finite Element Analysis Using Ansys

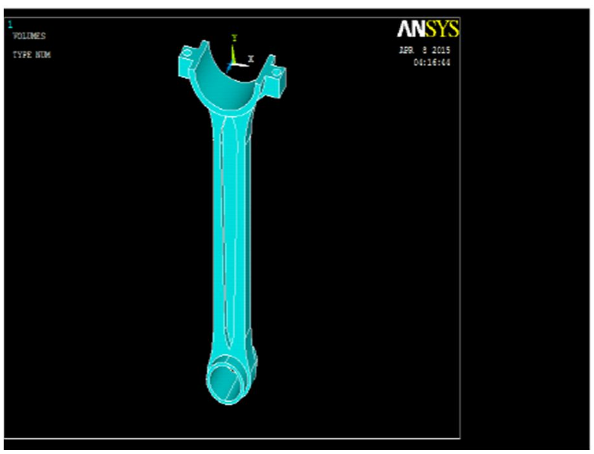

Figure 3 :Model of Connecting Rod

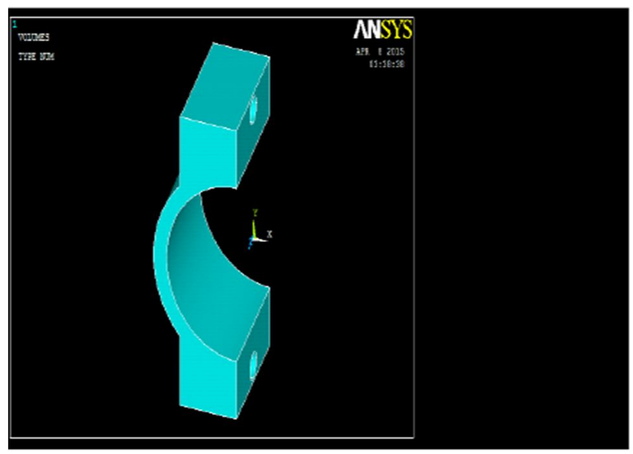

Figure 4 :model of the Lower Bearing

The meshed models of the connecting rod and lower are as shown in above figure.

Once meshing is done the boundary conditions i.e. DOF constrains, forces, loads are to be applied on the model. As shown in above figure the pressure is applied on the Crank end bearing of the connecting rod, while keeping piston end fixed.The pressure of 15 Mpa is used.

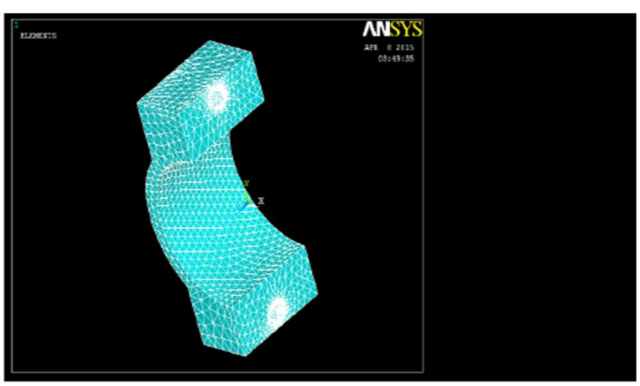

Figure 5 Meshed Model of Connecting Rod

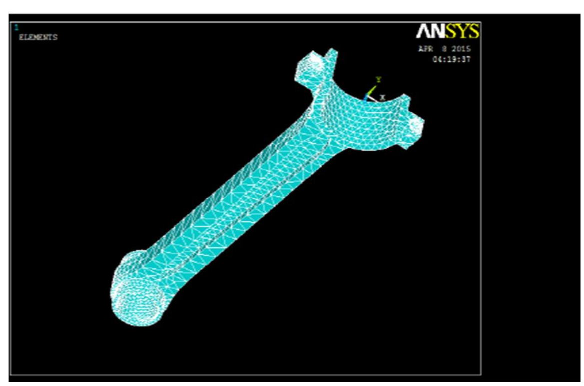

Figure 6 : Meshed model of Lower Bearing

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 VIII, August 2018- Available at www.ijraset.com

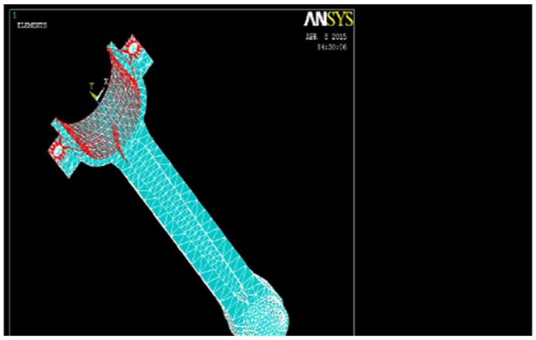

Figure 7 Load constrained section of connecting rod

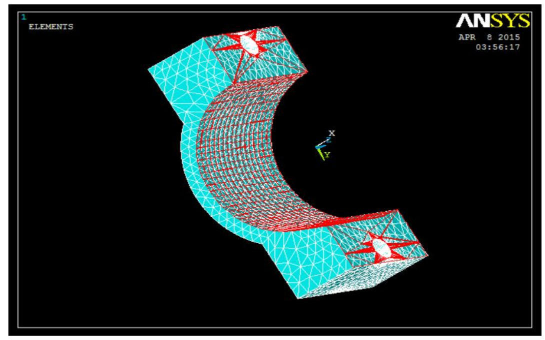

Figure 8 :Load Constrained Section of Lower Bearing

V. RESULTS & DISCUSSION

The Analysis is done for two different materials, Al 2024 and Stailness Steel at 15 Mpa pressure.

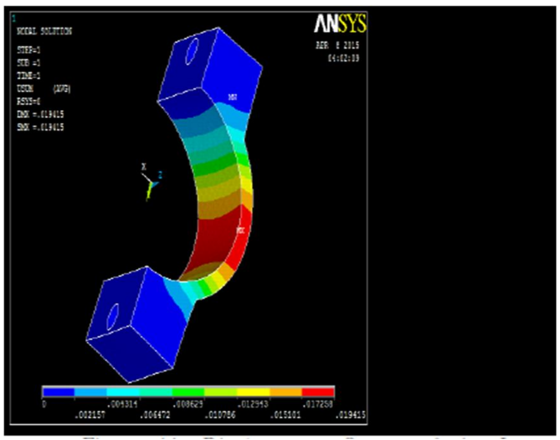

Figure 9 : Displacement Output of the Lower Bearing of Stainless Steel

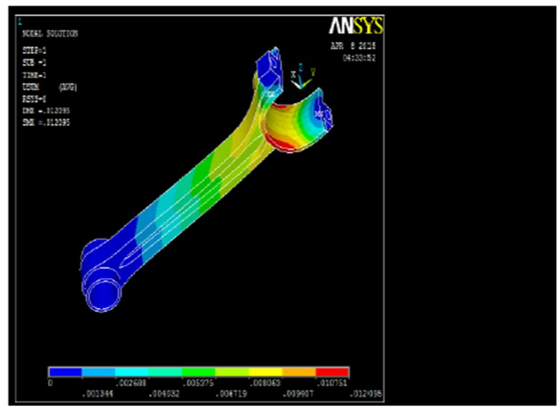

Figure 10 : Displacement Output of the Connecting Rod

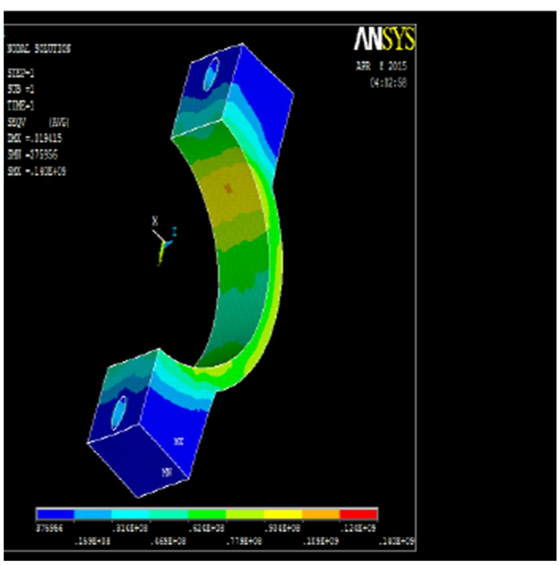

Figure 11 : Von-Misses Stress Output of the Lower Bearing of Stainless Steel

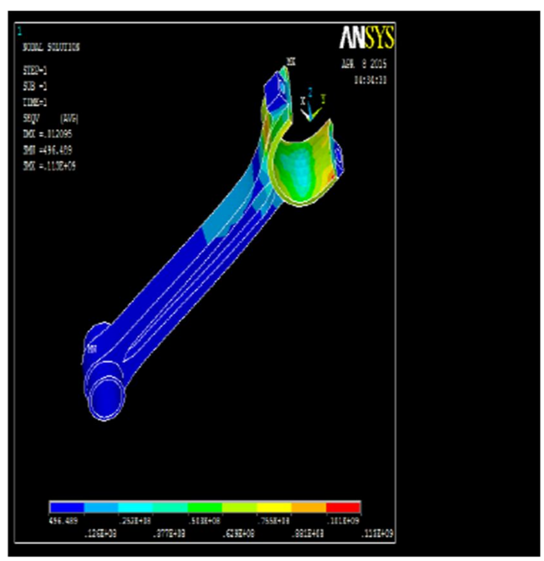

Figure 12 :Von-Misses Stress Output of the Stainless Steel Connecting Rod

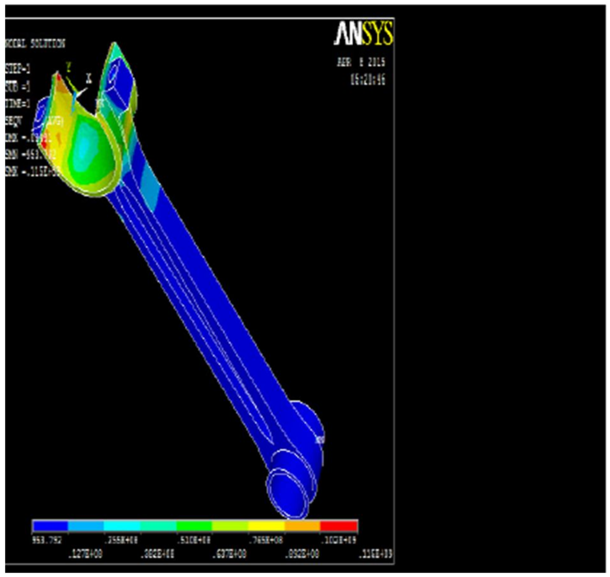

Figure 13 : Displacement Output of the Aluminium Alloy Connecting Rod

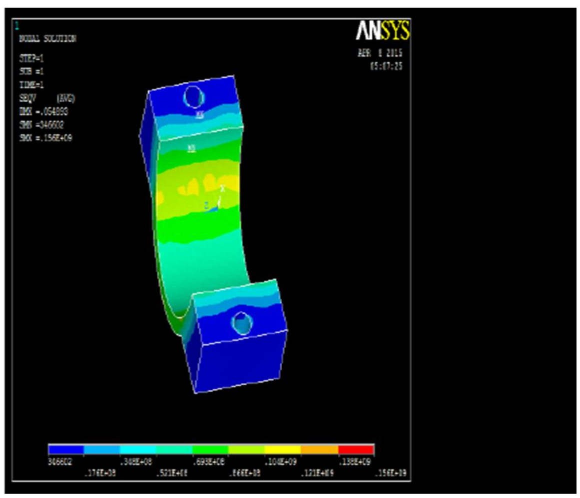

Figure 14 :Displacement Output of the Aluminium Alloy Connecting Rod

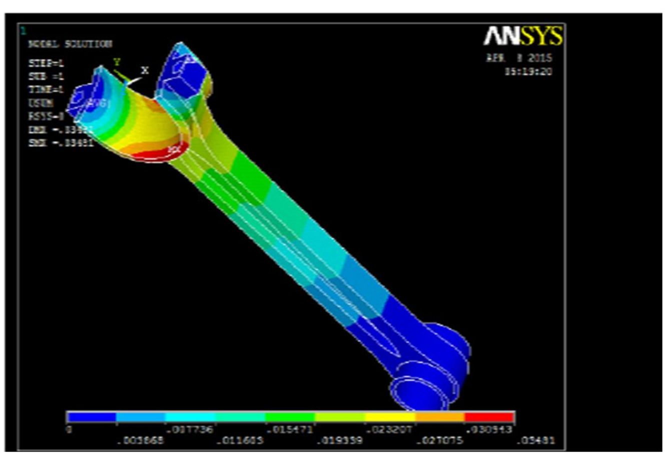

Figure 15 :Von-Misses Stress Output of the Aluminium Alloy Connecting Rod

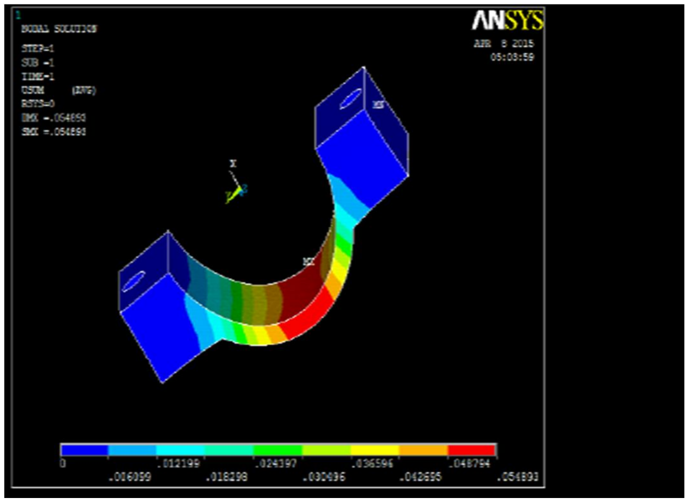

Figure 16 : Von-Misses Stress Output of the Lower Bearing of Aluminium Alloy

According to the results obtained by the ANSYS software for the displacement, Von-Misses Stress at the pressure of 15 MPa the values for the output obtained can be represented as shown in the tables below.

FOR 15 MPA PRESSURE	Stainless Steel	Aluminium Alloy 2024
Displacement	0.0194161	0.024894
Von-Misses Stress	0.149E+09	0.151E+09

FOR 15 MPA PRESSURE	Stainless Steel	Aluminium Alloy 2024
Displacement	0.012094	0.03381
Von-Misses Stress	0.123E+09	0.115E+09

VI. CONCLUSION

The results or conclusion thus that can made on the bases of the output results by ANSYS can be as followed: The analysis was completed and we got the Strong and weak points of the design upon which improvements can be done. Other materials can be used to compare with Al 2024 and further weight and cost reduction can be done to optimize the design. The design similar to existing ones and much research needs to be done to make a ground-breaking design that is very efficient as compared to existing ones. The areas of the design which buckle most under stress need require more attention to obtain a more efficient design.

REFERENCES

[1] Achudhan M., Prem Jayakumar M.,Mathematical modeling and control of an electrically heated catalyst, International Journal of Applied Engineering Research, v-9, i-23, pp-23013-, 2014.

[2] Kumar J., Sathish Kumar K., Dayakar P., Effect of microsilica on high strength concrete, International Journal of Applied Engineering Research, v-9, i-22, pp-5427-5432, 2014

[3] Hitesh Kumar, 2, Vijay Kumar Sharma, Department of Mechanical Engineering Stress Analysis and Optimization of I. C. Engine Connecting Rod by Finite Element Analysis ,IJERT Vol. 4 Issue 04, April-2015\

[4] Shahrukh Shamim,Design and Comparat ive Analysis of Connecting Rod usin g Finite Element Method,IJERT, Vol. 3 Issue 9, September- 201

[5] Pushpendra Kumar Sharma1, Borse Rajendra R, " t ue n lys s n opt m s t on o connect n ro us n n te element n lys s",Intern t on l Journ l O v nce rese rc n Sc ence n En neer n , Vol. No.1, Issue No. I, pp-3367-3371, September 2012

[6] Udaya kumar R., Kaliyamurthie K.P., Khanaa, Thooyamani K.P., Data mining a boon: Predictive system for university topper women in academia, World Applied Sciences Journal, v-29, i-14, pp-86-90, 2014

[7] Praveen R., Achudhan M., Optimization of jute composite as a noise retardant material, International Journal of Applied Engineering Research, v-9, i-22, pp-7627-7632, 2014.