DESIGN AND ANALYSIS OF BUS BODY SUPERSTRUCTURE

D.Purushottam Reddy[1], S.Sardar ali [2], D.Srikanth[3], JV.Mohanachari[4]

[1]PG Student, *Intell Engineering College ,Anantapur,A.P*
[2]*Asst professor &HOD, Intell Engineering College ,Anantapur,A.P*
[3] *Associate Professor, Intell Engineering College ,Anantapur,A.P*
[4,] *Assistant Professor ,Gates Institute Of Technology,Gooty,Anantapur,A.P.*
Corresponding Author: E-Mail:pr546@yahoo.co.in

ABSTRACT

Buses are the major mode of public transportation. The design of the bus body depends largely upon the performance requirements under various types of loading, operating conditions and the road conditions. The structural design also varies on the basis of the passenger capacity, the types of usage, and the nature of operation: urban/rural, long run/city run etc. Based on above requirements the structural materials, the size and shape of the structural member, the joint designs etc., are selected. In India, most of the buses are designed and fabricated on the basis of past experience . The 3D model of bus body superstructure is prepared by using Pro E software and structure and modal analysis carried out by Ansys work bench. At present evaluating the strength and vibration in Bus Body Superstructure at different locations with normal static load. Finite Element modelling of a Bus body structure described for finding the analysis of vibration/stress levels at different locations of the vehicle. The stress, deformation are obtained for different materials of bus superstructure at cant rail and waist rail.

Key words: Pro E, ANSYS, Composite Material , structural and modal analysis

1 INTRODUCTION

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. The CMVR - Technical Standing Committee, addressed the problem areas and the whole exercise was aimed towards standardizing the essential aspects involved in the construction of the bus body considering the minimum requirements of Safety and Comfort for a passenger. The finalized " Code of Practice for Bus Body Design and Approval " was submitted to the Ministry for further necessary action. The standard AIS-052 was published in September 2001.It was noticed that the OE vehicle manufacturers sell their products in the form of drive away chassis and the body design and building is being done by way side body builders who employ poor design, poor quality products, spurious materials and parts, with no uniformity in the construction, resulting in large amounts of fatal accidents. The existing bus body structures are hardly design optimal and safe. The cabin and seats have cramped designs which do not provide safety and comfort to the driver. Body designs offer extreme heat, vibration, noise, poor comfort and protection. Wood is being used in the construction of the body to a large extent. Of late some reputed body builders have brought in improved bus designs in the market but still a lot has to be done.

2 LITERATURE SURVEY

Prasana Priya .Chinta has designed bus body structure in such away that to reduce the number of elements from the structure as well as thickness towards weight reduction of bus structure. Optimization of mechanical response of automotive and body designs are increasingly relies on new models. Generally in international market for passenger's buses design processes can rely on supercomputing facilities. Nowadays for the passenger buses have many local producers which construct vehicles based on local needs. In the competitive to stay these producers comply with the same requirements and weight reduction of their international counterparts without access to latest computation facilities. This paper proposes a new method for designing a bus body structure is designed and modeled in 3D modeling software Pro/Engineer. The original body is redesigned by changing the thickness and reducing the number of elements so that the total weight of the bus is reduced. The present used material for structure is steel. It is replaced with composite materials Kevlar and S 2 Glass Epoxy. The density of steel is more than that of composite materials, so by replacing with composites, the weight of the structure is reduced. Structural and Dynamic analysis is done on both the structures using three materials to determine the strength of the structure. Analysis is done in Ansys.[1]

Rajesh S.Rayakar - Buses are the foremost mode of road transportation. The design of the bus body depends mainly leading the performance constraint under various types of loading and operating circumstances besides those of the road conditions. In India the majority of the buses are designed and fabricated on the basis ancient time experience. The bus body design parameter essentially consists of shape, stability purpose and strength is carried out at different operating circumstance such as quasi static load and braking loads. Here we analyze two different carline, state transport utility passenger vehicle is compared with new developed prototype carline. Applied quasi static loading & different loading conditions using yield strength of materials 240 Mpa and 380 Mpa respectively, Test procedures followed were as per AIS-052 (Revision 1) and AIS-031 results analyzed by FE model for strength analysis . done his research work on the determination of Strength Analysis of Bus Body Carline through, objective of his work is concerned about the strength analysis of bus carline,analyse the carline of bus existing state transport utility passenger vehicle is compared with newly developed carline applies quasi static loading and different loading conditions using yield strength of the material 240 Mpa to 380 Mpa respectively, result analyzed by FE Model for strength and stability analysis.[2]

Santosh B.Beluru done his research work was critical analysis of passenger Bus body structure Design using FEA, Buses are the major mode of public transportation. The design of the bus body depends largely upon the performance requirements under various types of loading and operating conditions besides those of the road conditions. The structural design also varies on the basis of the passenger capacity, the types of usage, and the nature of operation: urban/rural. long run/city bus etc. Based on above requirements the structural materials, the size and shape of the structural member. the joint designs etc. are selected. In India, most of the buses are designed and fabricated on the basis of past experience and not on any adequate scientific consideration. Their structural strength, stability, crash worthiness, etc. are hardly evaluated, resulting in reduced passenger safety with increased possibility of damages. Hence, there is a need to design and critically evaluate.bus body-structural design using scientific techniques. In this paper the criticality analysis of the bus body structure is presented under different operating conditions; such as normal static load. braking loads. loads due to: speed breakers. single wheel road bump, single wheel in. road pot hole and dynamic loads caused by: frontal impact. Finite Element modeling of a Bus is described for finding the Eigen pairs and analysis of vibration/stress levels at different locations of the vehicle. In this work an attempt is made to investigate the effect of stiffness, strength and vibration in Bus design. on the predicted stress distribution. In the given research work it is found that. the aluminum structure is. effective than M.S. structure by considering strength to weight ratio., his objective was to design and develop an Aluminum bus body structure of city bus, to optimize bus body structure design for strength and stiffness under static loadings, and to optimize bus body structure for front impact.[3]

Mahesh Haldankar,A.M.Shirahatti- Road transport is the most popular transport which mostly uses buses. Local and medium enterprises take the bus body building work. The smaller companies do not have much funding towards the FEA to develop the design. To remain competitive in the market, the continuous improvement of the design is needed and it is easier to do in finite element analysis. During the normal operation, the bus body is subjected to several loads, external loads from the road (i.e. crossing over a speed bump). Moreover, there is a substantial possibility that these loads may lead to a structural failure. Hence, it is necessary to determine stresses occurred in the bus body to ensure its integrity under these driving scenarios. This project deals with the modelling, analyzing and optimization of important section of the bus body for the standing gravity load and for the bump case and it is further tried to validate the results with the help of theoretical calculations. he has done their research work was the FE Analysis and Optimization of Commercial Bus Body Structure his main objective is to analyze and optimize the bus body for standing load and bump case and optimization the floor section.[4]

Kalmikov B. Yu., Ovchinnikov N. A-Reducing the total impact energy bus is possible due to reduction of the mass of the bus, the acceleration of free fall or drop height of center of gravity. The most promising at the moment is to reduce the height of the fall of the center of gravity of the bus. To reduce the height of the fall is possible to use the author's device to prevent a rollover accident. This paper proposes a method for determining the total impact energy bus rollover bus. The method is applicable to conditions determined by the UNECE Regulation No66, intended for the certification of passenger motor vehicles of category M3, taking into account the possible application of the device to prevent tipping. [5]

3. DESCRIPTION OF THE PROBLEM

The Preparation of 3D Model of Bus Body Super structure as follows

1.Design data has taken from Tata Bus 40 seats

2.The parameters which considered are the dimensions of actual bus given below table

Specification Parameters	Dimensions (mm)
Wheel Base	4975
Front over hang	1798
Rear over hang	3343
Max width of the bus	2390
Overall length	10108
Overall height in laden condition	3250

Table: Specification Parameters of Bus are in mm

Load Calculation

For cant rail: Weight has taken the roof structure of the bus is 21000 N, Mass is considered the 7 roof arch members as given the data in Table 5.1and Table 5.2, it is shared by two cant rails of the bus.

For Waist Rail: Weight has taken the one side either L.H.S or R.H.S is as follows.

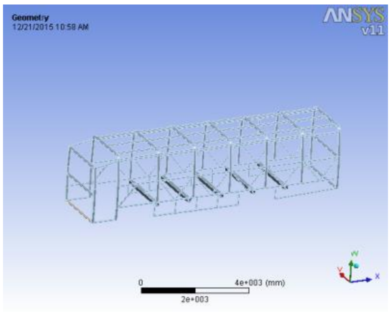
1.Half of the roof structure weight.

2. 7 Pillars weight, cant rail, window rail ,waist rail, 12 ribs weight and 40 passengers

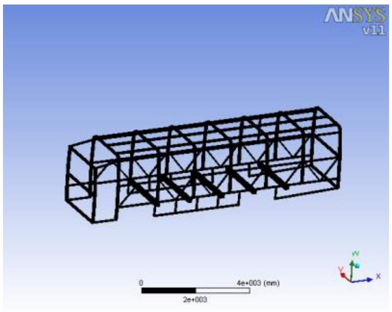
Load Applied on parts of Bus	Weight(N)		
	Total Load	L.H.S	R.H.S
Cant Rail	21000	10500	10500
Waist Rail	98550	49275	49275

Table: Load Application Table

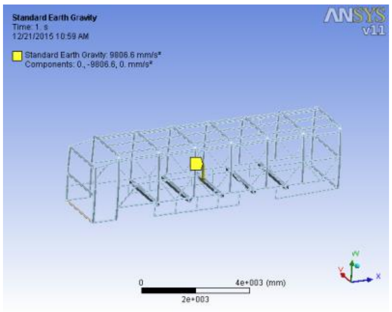
4. ANALYSIS OF BUS BODY SUPERRUCTURE



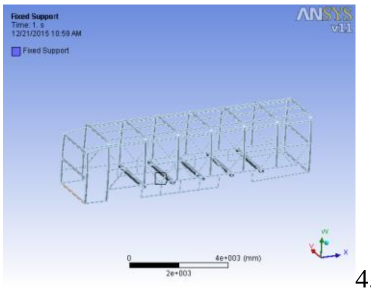
4.1Geometry of the Bus Body Super Structure



4.2 mesh generation



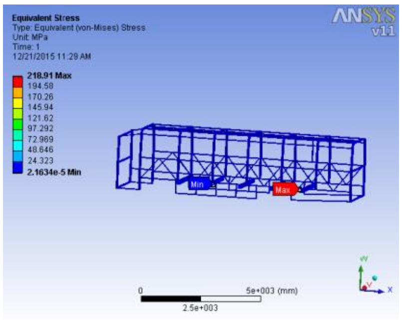
4.3 Stand by earth gravity

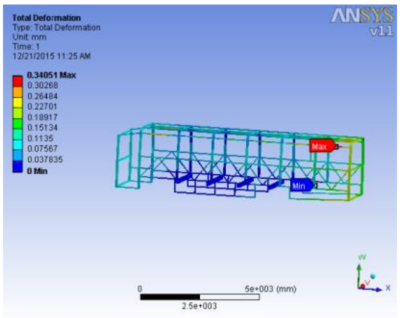


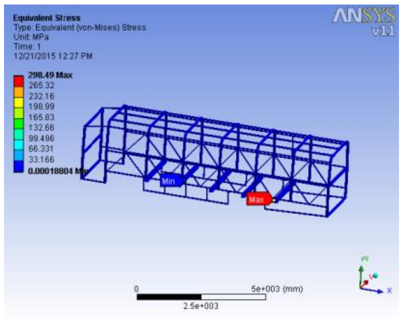
4.4 Cross bearers are taken as Fixed support

CARBON FIBRE

Young's Modules : $3.88X10^5$ Mpa
Positions Ratio : 0.358
Density :$1.6X10^{-006}$Kg/mm^3



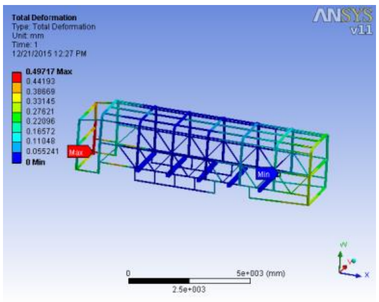


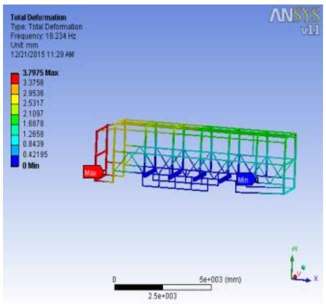


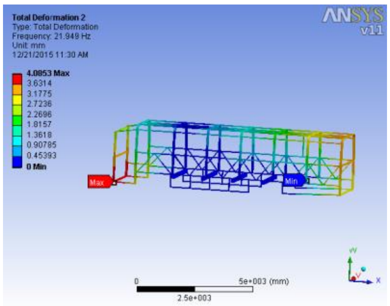
4.6 Equivalent stress developed is 218.91 Mpa at 10500 N load applied On cant rail

4.7 Max deformation is 0.34051mm when 10500N load is applied on cant rail

4.8 Equivalent stress developed is 752.85 Mpa at 49275 N load on waist rail



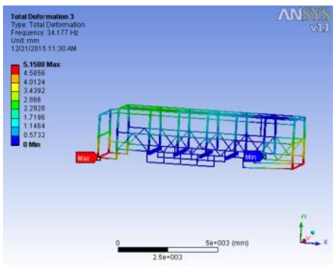


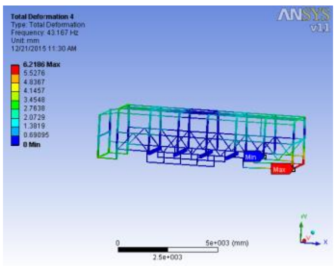


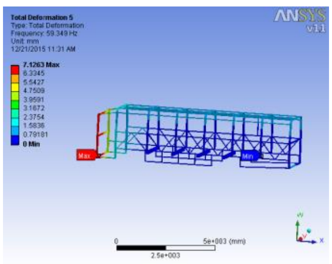
4.9 Max Deformation is 1.2438 mm when load is 49275N applied on waist rail

4.10 mode 1

4.11 mode 2



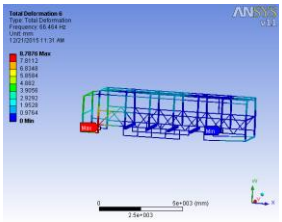




4.12:mode 3

4.13: mode 4

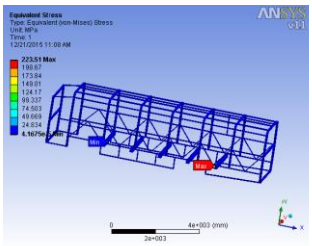
4.14: mode 5



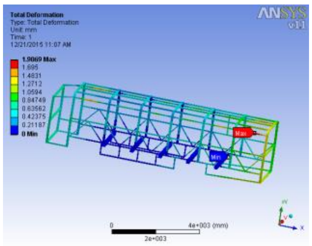
4.15: mode 6

E-GLASS

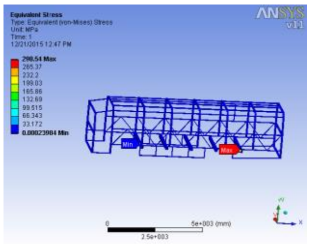
Young's Modules : 72500Mpa
Positions Ratio : 0.28
Density : 2.58 X 10^{-006} kg/mm^3



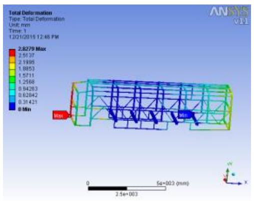
4.16: Equivalent stress developed is 223.51 Mpa at 10500 N load applied On cant rail



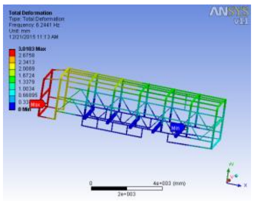
4.17: Max deformation is 1.9069 mm at 10500 N load applied on cant rail



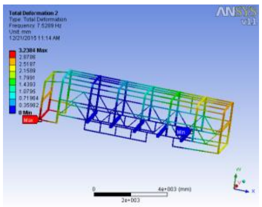
4.18: Equivalent stress developed is 731.02 Mpa at 49275 N Load applied on waist rail



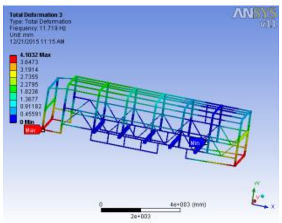
4.19: Max Deformation is 6.8314 mm at 49275N load applied on waist rail



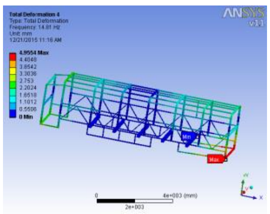
4.20:mode 1



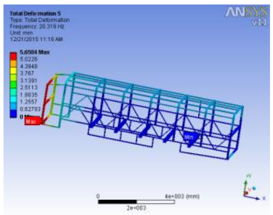
4.21: mode 2



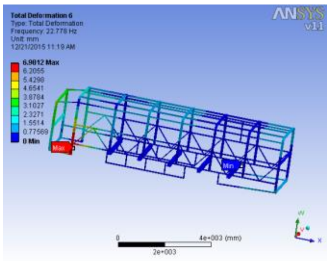
4.22: mode 3



4.23: mode 4



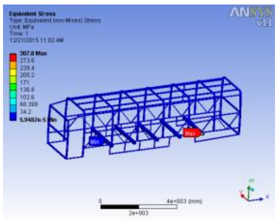
4.24: mode 5



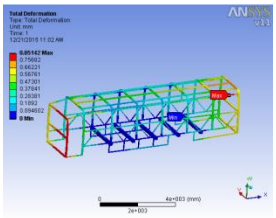
4.25: mode 6

Structural steel

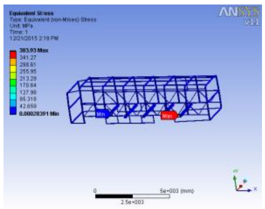
Young's Modules : $1.95X10^5$ Mpa
Positions Ratio : 0.31
Density $7.750X10^{-006}$ Kg/mm^3



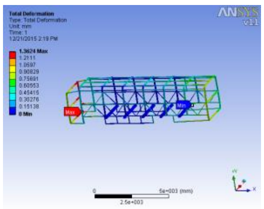
4.26: Equivalent stress developed is 307.8 Mpa at 10500 N load applied On cant rail



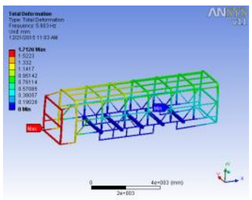
4.27: Max deformation is 0.85142 mm when load is 10500N applied on cat rail



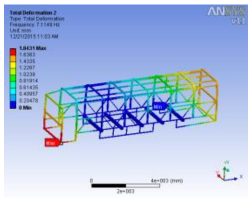
4.28: Equivalent stress developed is 822.02 Mpa at 49275 N load applied On waist rail



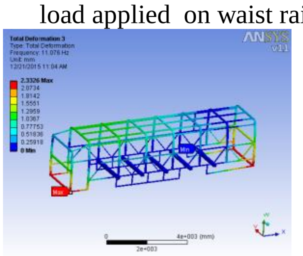
4.29: Max deformation is 2.7925 mm at 49275N load applied on waist rail



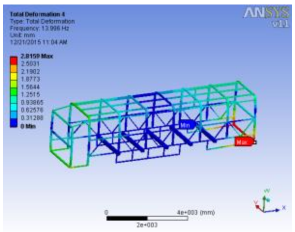
4.30: Mode1:Max deflection is 1.726mm when the frequency is 5.903Hz



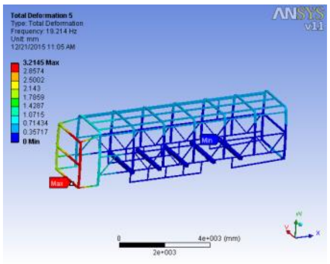
4.31: Model 2: Max deflection is 1.8431mm when the frequency is 7.1149Hz



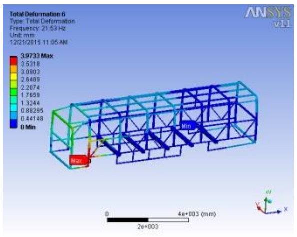
4.32: Mode 3: Max deflection is 2.3326 mm when the frequency is 11.076 Hz



4.33: Mode4: Max deflection is 2.81590 mm when the frequency is 13.996 Hz



4.34: Mode 5: Max deflection is 3.2145 mm when the frequency is 19.214 Hz



4.35: Mode 6: Max deflection is 3.9733 mm when the frequency is 21.53 Hz

5 RESULTS AND DISCUSSIONS

The structural Analysis is performed for Bus body super structure , the model is discritized in to 241728 elements and 474933 nodes.

Based on the Analysis results obtained by performing analysis on different locations(Cant rail, waist rail) of the Bus body super structure .The following discussions are written for different materials are Structural steel, carbon fiber, and E-Glass.

- When the load 10500 N is applied on cant rail of Bus body super structure, the following result are obtained.

  For Structural steel material the max stress developed is 307.8Mpa and max deflection is 0.85142 mm.

  For carbon fiber material the max stress developed is 218.91Mpa and max deflection is 0.34051 mm.

  For E-Glass material the max stress developed is 223.51Mpa and max deflection is 1.9069. mm.

- When the load 49275 N is applied on waist rail of Bus body super structure, the following result are obtained.

  For Structural steel material the max stress developed is 822.02 Mpa and max deflection is 2.7925 mm. For carbon fiber material the max stress developed is 752.85 Mpa and max deflection is 1.2438 mm. For E-Glass material the max stress developed is 731.02 Mpa and max deflection is 6.8314 mm.

  Based on above result and discussion the Carbon fiber material is suitable for bus body super structure construction due to development of less stress and deformation.

The curb weights of the Bus body super structure of different materials are as follows

  Structural steel material is 1494 kg

  Carbon fiber material is 308.78 kg

  E-glass martial 490.2kg

- The model analysis is carried out for three different materials which are Structural Steel, carbon fiber and E-Glass.

Table :Stress and Deformation Comparison of different materials

MATERIAL	VON-MISES STRESS (Mpa)		DEFORMATION(mm)	
	10500N	49275N	10500N	49275N
STRUCTURAL STEEL	307.8	822.02	0.85142	2.7925
CARBAN FIBRE	218.91	752.85	0.34051	1.2438
E- GLASS	223.51	731.02	1.9069	6.8314

Table :Frequency and Total Deformation Comparison of Different Material

Material	Mode1		Mode2		Mode3		Mode4		Mode5		Mode6	
	Fn	T.D	Fn	T.D	Fn	T.D	Fn	T.D	Fn	T.D	Fn	T.D
CarbonFiber	18.234	3.797	21.949	4.0853	34.177	5.1588	43.167	6.218	59.349	7.126	66.46	8.787
E-Glass	6.244	3.010	7.5289	3.2384	11.719	4.1032	14.81	4.955	20.319	5.650	22.77	6.9812
Structural Steel	5.936	1.72	7.1149	1.8431	11.076	2.3326	13.996	2.815	19.214	3.214	21.53	3.9733

CONCLUSION:

The bus body Super structure is analyzed by using Ansys for different materials like Carbon Fiber-Glass and Structural Steel ,as per the result the maximum deformation occurred for Structural steel and E-Glass are more compared to carbon fiber and E-Glass, so this project to recommends the carbon fiber for Construction of Bus body Super Structure.

The curb weight of the bus structure is less for Carbon Fiber is less when compared to Structural Steel and E-Glass materials we can reduce the weight 78.2% when the structural steel is replacing by carbon fiber ,so this project work can prefer the carbon fiber for bus structure construction and also improve the mileage of vehicle.

REFERENCES:

[1] Prasannapriya. Chinta, Dr.L.V.Venugopal Rao, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 11, Issue 5 Ver. I (Sep- Oct. 2014), PP 39-47

[2]Rajesh S.Rayakar - INTERNATIONAL JOURNA L FOR RES EARCH IN AP PL I ED SC IENC E AND ENGINEERING TECHNOLO GY (I JRAS ET) Vol. 2 Issue VII, July 2014
ISSN: 2321-9653.

[3] Satosh.B PRECEDING OF COMPUTIME Sep 16-17-2015,Natinal conference on Computational Methods in Mechanical Engineering,Deparartment of Mechanical Engineering Osmania University Hyderabad.

[4] Mahesh Haldankar, A. M. Shirahatti- International Journal of Engineering and Technical Research (IJETR) ISSN: 2321-0869, Volume-2, Issue-12, December 2014

[5] Kalmikov B. Yu., Ovchinnikov N. A- ARPN Journal of Engineering and Applied Sciences
VOL. 10, NO. 8, MAY 2015 ISSN 1819-6608 .