Analysis and Optimization of Front Axle Beam of Light Commercial Vehicles

Amol B. Chandankar, PG Student(Mechanical)SRES College of Engineering, Kopargaon, India
Dr. A.G.Thakur Vice Principal and H.O.D (Mechanical) SRES College of Engineering, Kopargaon, India

***Abstract*—**The Front Axle Beam (FAB) is most important part in load carrying vehicle. The failure of FAB is serious concern to heavy vehicle and thus for human life. So it is necessary to analyze the axle beam's ability to withstand typical service loading which develops stress in the beam resulting into failure. Further the objective of analysis is to improve its product quality while reducing development time, material and manufacturing costs while maintaining the stress levels.

In this project, the front axle of Light Commercial Vehicle (LCV) model is used. For the finite element analysis we used Ansys14.0. The objective of the project is to analyze and optimize the beam for reducing weight. The FEA is been carried out for vertical loads due to total weight carried by vehicle. An inertial load due to acceleration and deceleration of vehicle, which is, causes twisting of king pin portion with respect of PAD centerline. And cornering load component arise due to centrifugal force acts on both king pin boss during turning. Lastly the modal analysis is carried out on FAB to find out mode shapes and natural frequencies. The model is optimized for weight reduction by using parametric approach.

***Index Terms*—Front Axle Beam (FAB); Finite Element Analysis (FEA),Light Commercial Vehicle (LCV),**

I. INTRODUCTION

Amol B. ChandankarAuthor, is currently with SRES College of Engg,Kopargaon,Maharashtra 423 603 India completing ME Design Engineering. Email-Amol.Chandankar@yahoo.co.in

PapalambrosPanos Y., Douglas J. Wilde [1] proposed engineering development process involving creation of 3d model and using it for analysis.

N. León *, O. Martínez, P. Orta C., P. Adaya [2] has given the case studies on front axle beam where he explains the complete procedure of analysis of front axle beam. Also explains how he reduced the weight of front axle beam by parametric optimization.

VeraN.S., author of “Meshing and Creating elements”, [3] MSC/Nastran Finiteelement Analysis software Help Manual, Vol. 2 gave detailed procedure of meshing elements.

Dr. N.K.Giri, is author of”Automobile Mechanics”[4]. He gave information of how load is acting on front axle beam at various conditions i.e. at static, at braking.

With this, I referred book of “Application of FEM tools in Engine Development Process”[5], for various application process of FEM tools.

I also referred S.S. Rao for book”FiniteElement Method In Engineerinh” [6], C. M. Rao for book “Finite element modeling and stress analysis of helical gear, teeth, computers and structures”[7], Ansys13.1 “Help Manual”[8] and book “Mechanics of Automobiles” by Barnacle H.E. [9]for the design calculation.

Today's Front Axles Beams is much difficult to reduce the weight of FAB for same stress/deflection ratio.

In the present analysis the structure of FAB is designed and analyzed using Finite Element Technique using Ansys analysis software. It is further optimized for reduction in mass. Static analysis of the FAB has been carried out to determine the deformation and stress pattern and to check whether the design is rigid and strong enough to with stand the forces coming during extreme working condition of the vehicle.

After studying the deformation and stress patterns, over and under designed areas are found out during post processing [1]. Information from this analysis is used to reduce/ increase the section changes wherever required after few iteration ideal, minimum weight, maximum strength FAB has been developed.

1.1 FRONT AXLE BEAM:

The Fig 1 showing the arrangement of the front axle beam. Front axles are subjected to two types of stresses bending and shear stresses. In the static condition, the axle may be considered as a beam supported vertically upward at the ends, i.e. at the center of the wheels and loaded vertically downward at the centers of the spring pads [2]. The vertical bending moment thus caused is zero at the point of support and rises linearly to a maximum at the point of loading and the remains constant.

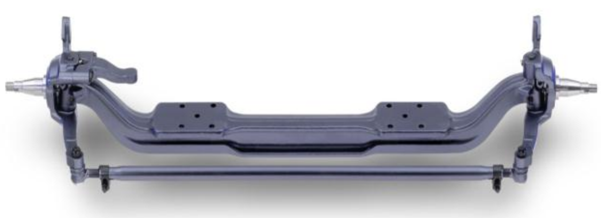


Fig.1 front axle beam.

II. OBJECTIVE

Front Axle Beam is analyzed to find out factor of safety for critically stressed regions due to vehicle load, braking load and cornering load under running condition. After that the model is optimize for weight.

III. METHODOLOGY

a. Select the product to be improved or developed
b. Create a 3D parametric product model as basis for a family of parts.
c. During this proposed engineering development process, import the geometry of the 3D CAD model is in the FEA software for stress and strain analysis.
d. Mesh the product geometry using analysis software.
e. Apply boundary conditions to be analyzed.
f. Post-process a product using software and send to design team for a thorough analysis.
g. The results of this analysis are then used to formulate new product geometry with the 3D Parametric CAD software, , which is analyzed until a satisfactory stress and strain map is obtained.
h. The obtained FEA results are verifies with the set of experimental test procedures.

IV. CREATION, MESHING DESIGN CALCULATIONS, EXPERIMENTAL ANALYSIS AND OPTIMIZATION OF FRONT AXLE BEAM

A. *Inputs required from the Vehicle:*

a. Data From Vehicle:

For the analysis purpose, we have chosen the front axle beam of LCV TATA SFC 407EX (light commercial vehicle). The following Fig.2 shows the position and various parameters of Front Axle Beam which are used in further design calculation.

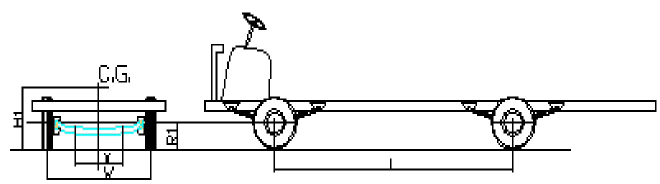


Fig.2 Position and various parameters of FAB

TABLE I. INPUT DATA FROM THE VEHICLE UNDER ANALYSIS

Parameter	**Value**
Total weight of vehicle	5300 Kg
Laden weight on front axle beam	1860 Kg
Laden weight on rear axle beam	3440 Kg
Height of C.G. of vehicle from ground, H1	1000 mm
Wheel base, L	2500 mm
KP eye to KP eye distance, W	1400 mm
Pad to pad distance, X	1342 mm
Spring width	115 mm
Spring pad length	70 mm
Radius of wheel, R1	203.2 mm
Bore diameter of KPB, D_i	25 mm
KPB outer diameter , D_o	47 mm

KPB height, H	62 mm
Velocity of vehicle	25 km/hr
Minimum turning radius,R	5300 mm

b. Material Description of The FAB:

1) Specification : S53C (AISI 1053)

2) Chemical Composition:

	C	Mn	Si	P	S
Min	0.48	0.7	0.1	-	-
Max	0.55	1.0	0.2	0.04	0.05

3) Hardness (Quench & Temper) : 248 - 302 BHN

4) Poisson's Ratio : 0.3

5) Young's Modulus : 21.1 E^5 (N/mm^2)

6) Yield Stress : 550 MPa (N/mm^2)

7) Allowable shear stress : Sy=467.5 MPa

B. Modeling:

The results of design calculations are used to formulate new product geometry with the 3D Parametric CAD software, which will be later analyzed until a satisfactory stress and strain map is obtained

C. Meshing procedure:

A 3D CAD model is imported in the meshing software and is meshed using the meshing elements available in Ansys[3]. As we are using a 3-D model for analysis, so for this we choose 3-D solid element:

1) SOLID 45 – 8 node first order element
2) SOLID 92 – 10 node second order element
3) SOLID 95 – 20 node second order element.

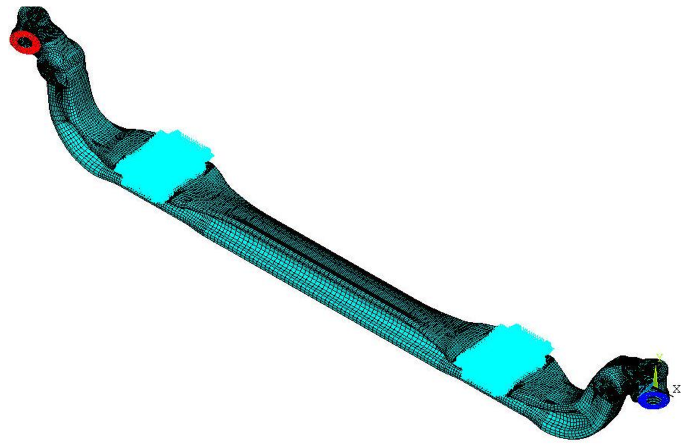


Fig.3 Meshed model

D. Design Calculations:

a. Vertical Load Due To Total Weight:

LOAD CALCULATIONS:

Total load on FAB = 1860 Kg

Load on one side pad = 1860/2 = 930 Kg

Taking factor for impact conditions = 2,

Load to be applied on one side pad (P) = 930 X 2

= 1860 Kg

= 18246.6 N

King pin boss area =3.142/4 * (D_o^2-D_i^2)

= 3.142/4 * (47^2-25^2)

= 1244.07 mm^2

Pressure on each KPB = 18246.6 / 1244.07

= 14.67 N/mm^2

b. Twisting Loads Due To Breaking Force Of Vehicle:

Similarly,

Calculating the Reaction force on area (R) = 19935.48 N

c. Corning Forces While Turning Of Vehicle On Curve Road:

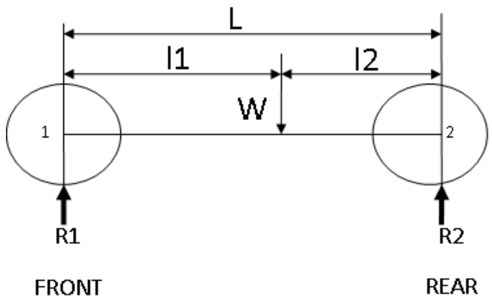


Fig.4

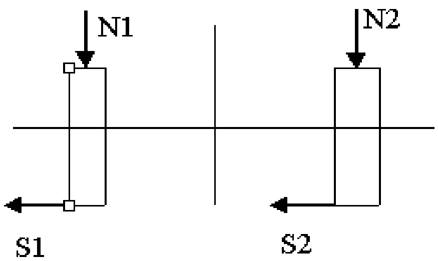


Fig.5

After calculations,

The total axial force on front beam S_1+S_2 = 9123.3 N

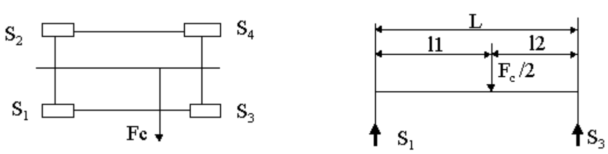


Fig.6

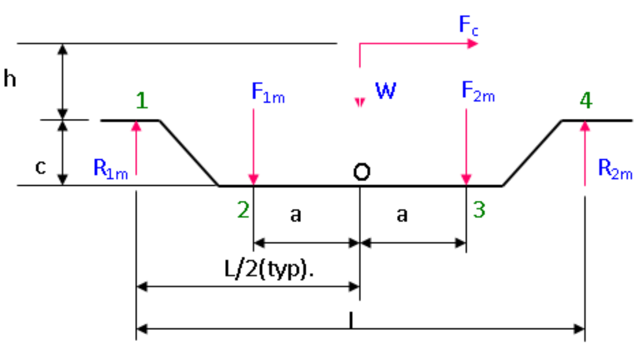


Fig.7

Centrifugal force, Fc = mv2/R

= 930* 5.0992/ 5.3

= 9124.45 N

l = 1.4 m, a= 0.6710 m,

h = 0.7968 m, c =0.1100 m

F1m=Fs1 + F1 ---1

(F1=Fc*(h+c)/2a=F2)

F2m =Fs2 + F2 ---2

Where,

Taking moment about point 3 we get,

F_1=F_c*(h+c)/2a.

Similarly taking moment about point 5 we get,

F_2 = F_1

Thus from equation 1& 2 we have

F_{1m} = F_{s1} - F_c(h+c)/2a

= 9123.3 - [9124.45 *(0.7968 + 0.1100)] / (2*0.671)

F_{1m} = 2957.84 N

F_{2m} = F_{s2}+F_c(h+c)/2a

= 9123.3 + [9124.45 *(0.7968 + 0.1100)] / (2*0.671)

F_{2m} = 15288.76 N

Reaction forces R1mand R2m are calculated by taking moments about 4 & 1.

R_{1m} = (F_{1m}*(l/2 + a) + F_{2m}*(l/2 - a))/

=[2957.84*((1.4/2)+ 0.671) + 15288.76*((1.4/2) - 0.671)]/1.4

R_{1m} = 3213.27 N

R_{2m}= (F_{2m}*(l/2 + a) + F_{1m}*(l/2 - a))/ l

=[15288.76*((1.4/2)+0.671)+2957.84*((1.4/2)-0.671)]/1.4

R_{2m}= 15033.33 N

E. Analysis:

a. Case 1: Vertical load due to total weight carried by vehicle:

Loads and boundary condition:

We calculate loads in terms of pressure, which is directly applied on KPB surface [5]. Similarly the pad is fixed to the spring, so we applied the displacement constraint on top surface of the pad in x, y and z direction.

Fig.8 showing the applied pressure on KPB area and displacement constraint on pad surface.

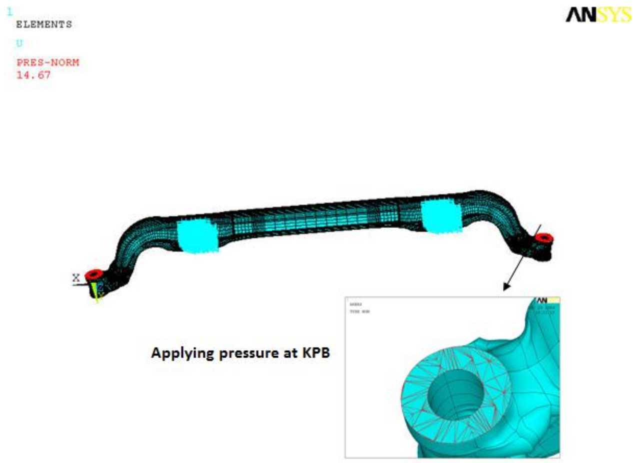


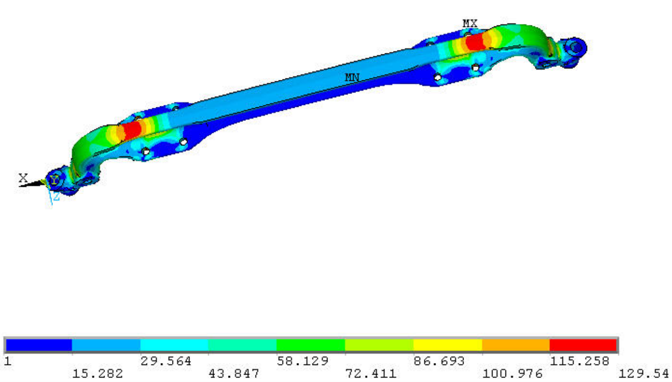
Fig.8 Loads and boundary conditions

Post Processing:

After solving the problem, we plot the von-mises stress contour and displacement contour form the Ansys post processing commands [6].

The Fig.9 von-mises stress contour showing that maximum stress developed is 129.54 N/MM^2. The maximum stress appears near to the spring's base.

The Fig.10 usum displacement contour showing the maximum deflection 0.9634 occurred.



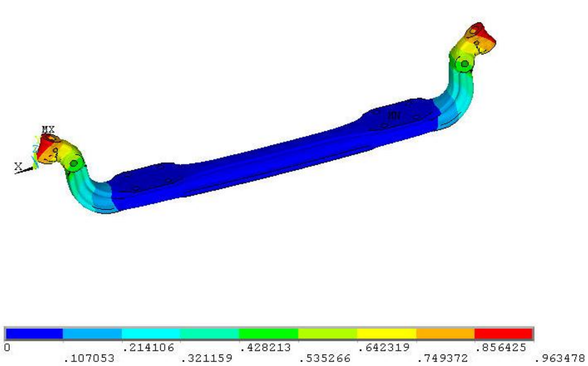


Fig.10 Displacement (usum) contour

Similarly Analysis is carried out for below conditions:

b. Case 2: Twisting Loads due to braking force of vehicle

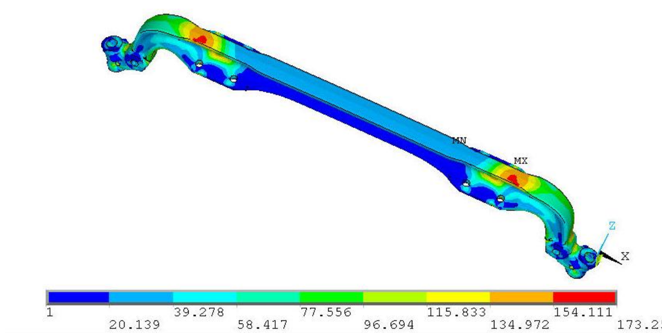


Fig.11 Von-mises stress Contour

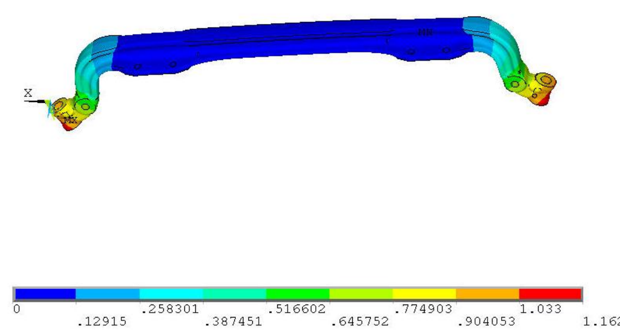


Fig.12 Displacement (usum) contour

c. Cornering forces while turning of vehicle on curve road

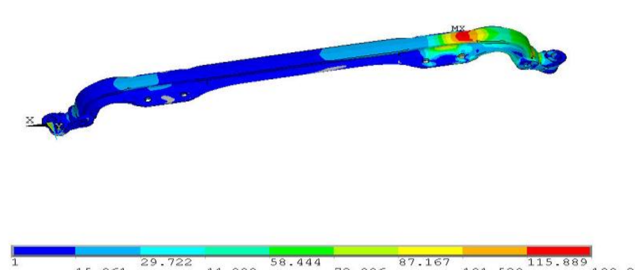


Fig.11 Von-mises stress Contour

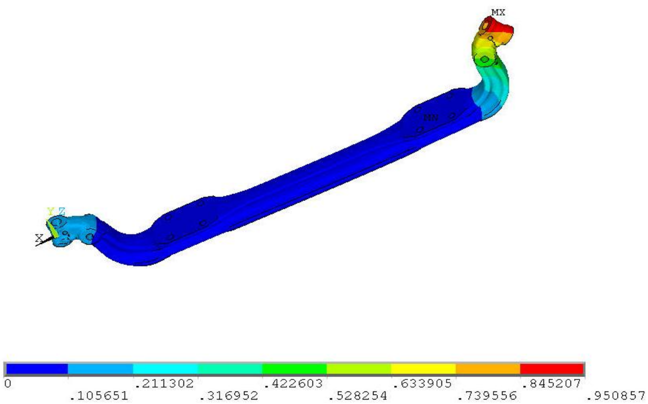


Fig.12 Displacement (usum) contour

d. Experimental Validation of the FEM results:
At this point the results obtained by finite elements were verified experimentally.

The test is carried out at Bharat Forge Fatigue Test Lab under project "2026 analysis and testing of front Axle beam" The stress measurement was carried out using photo stress. Photostress is a widely used technique for accurately measuring surface strains to determine the shear stress in a part of structure during static or dynamic.

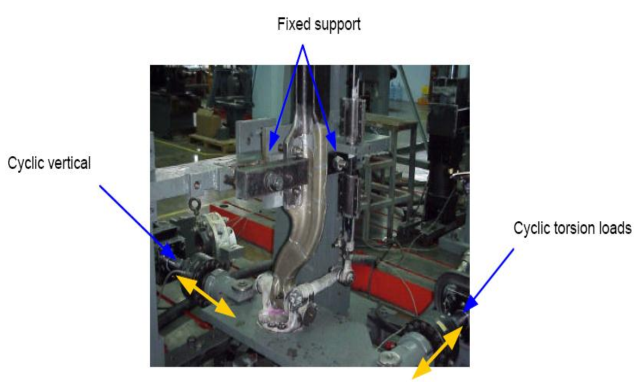


Fig.13View of the arrangement for experimental test procedure

Results obtain by experimentally and FEM analyses are tabulated as given in below table:

TABLE II. VON-MISES STRESSDISTRIBUTION RESULTS FROM EXPERIMENTAL ANALYSIS

Sr.No	Load condition	Von Mises Stress N/(mm)^2 FEM Analysis	Von Mises Stress N/(mm)^2 Experimental
1	Vertical Load condition	129.54	132.56
2	Braking Load condition	173.25	182.194
3	Turning Load condition	130.25	133.45

The results obtained with this experimental method and its quantitative comparison with values calculated with FEM show maximum variation of the 5.17 %. The variation depends on the measurement point.

F. Optimization:

From above it is clear that our first objective is over to analyze the model. In next we have to optimize the model for weight reduction.

We are using the methods which are explained in DIRONA'S case studies.

We use the parametric method for optimization of FAB [8].

Here, we have used the FAB models with 3 different the fillet radius of their respective I sections:

Model 1: 14 mm fillet radius

Model 2: 15 mm fillet radius

Model 3: 16 mm fillet radius

The obtained results with this optimization are given below in the results section.

V. RESULT & DISCUSSION

TABLE III- VON MISSES DISTRIBUTION RESULTS OF THE OPTIMIZED MODELS

Sr. No.	Load condition	Von Mises Stress N/(mm)^2 Model 1	Von Mises Stress N/(mm)^2 Model 2	Von Mises Stress N/(mm)^2 Model 3
1	Vertical Load Condition	129.54	130.24	136.12
2	Breaking Load Condition	173.25	175.23	176.24
3	Turning Load Condition	130.25	125.45	134.15

VI. CONCLUSION

In this project, the analysis of Front Axle Beam is done through Ansys analysis software. The results coming from these are reasonably (90 to 95%) same as experimental calculation hence it can conclude that software predict us best information and its results are trustable with time saving.

It concluded from stress analysis of FAB that the induced stress in FAB during full load condition are well below to the yield stress of Front Axle Beam material, hence FAB can sustained all coming loads during running.

From the optimization of Front Axle Beam, we found that the maximum reduction in the Front Axle Beam is to 3.42%. The modified model 2 (16 mm fillet radius) is least weight, which is 29.04 Kg and having factor of safety 2.64.

Hence it is concluded from the above analysis that the modified model 2 (16 mm fillet radius) is best satiated for the Front Axle Beam design.

VII. FUTURE SCOPE

Results obtained from analysis clearly state that FAB with a weight of 29.04 kg can sustain all loads in the running condition.
This FAB will be used to for future light commercial vehicles which have carrying capacity of 5300 – 5700 kg. In future the loading conditions and the factor of safety need to be evaluated for determining the total load carrying capacity of the FAB.

REFERENCES

[1]PapalambrosPanosY.Douglas J. Wilde.,Principles Of Optimal Design,Modeling And Computation, 49,Pp.253- 268, Cambridge University Press., 1988.

[2]N. León*, O. Martínez, P.Orta C., P. Adaya., Reducing the Weight of a Frontal Truck Axle Beam Using Experimental Test Procedures to Fine Tune FEA.,21, PP.13-30, Second Worldwide MSC Automotive Conference., Dearborn, Michigan; October 9-11, 2000.

[3] Vera N.S., "Meshing and Creating elements", MSC/Nastran Finite element Analysis software Help Manual, Vol. 2 Issue 7, Pp, 2608-2614,July 2003.

[4]Dr. N.K. Giri., "Automobile Mechanics"49, pp.1095-1106, 1993.

[5]Petrin, H. Wiesler, B. "Application of FEM-Tools in the Engine Development Process", FIRST Worldwide MSC Automotive Conference,Munich, Germany; September 20-22. 1999.

[6] S.S.Rao, "The Finite Element Method in Engineering", 2 nd Edition, pergamon press, 3,pp, 1055-1066, 1989

[7]Rao, C.M., and Muthuveerappan G., Finite Element Modeling and Stress Analysis of Helical Gear, Teeth, Computers & structures, 49, pp.1095 1106, 1993.

[8] ANSYS inc.U.S.A." Help Manual", ANSYS 13.1.

[9]BarnacleH.E.,"Mechanics of Autobobiles" by Pergamon Press Ltd, 21, pp.13-30.1995.

[10] Min Zhang, Lijun Li, "A research on fatigue life of front axle beam for heavy-duty truck" 52, pp.823-826.

[11] Siddarth Ray, P. Bhaskar, "Structural Analysis of Front Axle Beam of a Light Commercial Vehicle", International Journal of Engineering Trends and Technology, Vol.11 numer-5, 2014.

[12] Mahanty, K. D., Manohar, V., Khomane, B. S. and Nayak, S. 2001. Analysis and WeightReduction of a Tractor's FrontAxle.TataConsultancy Services, India, SwarupUdgata, International Auto Limited, India.

[13] Nanaware, G. K. and Pable, M. J. 2003. Failures of front axle shafts of 575 DI tractors.Engineering Failure Analysis 10:719-724.