The International Journal Of Engineering And Science (IJES)
|| Volume || 3 || Issue || 4 || Pages || 70-76 || 2014 ||
ISSN (e): 2319 – 1813 ISSN (p): 2319 – 1805

Mathematical Modelling and Shape Optimisation of Front Damper Mount of Ashok Leyland 1612 Truck Using 3d Finite Element Method

[1,] Promit Choudhury, [2,] Chinmaya Hemanth Krishna
[1, 2,] *Department of Mechanical Engineering, SRM University*

ABSTRACT

A chassis serves the purpose of a framework for supporting the body and different parts of the vehicle. It consists of several mounting brackets. This paper focuses on the damper mount, which should be rigid enough to withstand the shock, twist, vibration and different stress conditions. Strength being the paramount concern, it should have adequate bending and torsional stiffness for better handling characteristics. The work performed towards the optimization of the damper bracket with constrains of stiffness, strength and natural frequency. The paper describes the mesh optimization with using finite element analysis technique to predict the higher stress and critical region on the component. The optimization is carried out to reduce the stress concentration and material. With using computer aided design (CAD), SOLIDWORKS software the structural model of a bracket is developed. Furthermore, the finite element analysis performed with using software ANSYS.

KEYWORDS – *Bending moment, Deformation, Neutral axis, Optimization, Von-Mises stress*

Date of Submission: 15 April 2014 Date of Publication: 30 April 2014

I. INTRODUCTION

Strength is an important consideration in the damper mount design and to increase the stiffness (bending and torsion) characteristics. Adequate torsional stiffness is required to have good handling characteristics. Generally brackets are designed on the basis of strength and stiffness. In the conventional design procedure the design is based on the strength and emphasis is then given to increase the stiffness of the mounts, with very little consideration to the weight. One such design procedure involves optimizing the periphery of bracket to the existing design to increase its torsional stiffness. As a result weight of the bracket decreases. This decrease in weight increases partly the fuel efficiency and helps in judicious use of material. The design of the bracket with adequate stiffness, strength and lower weight provides the scope for this project.

The goal of the structural design is to obtain minimum component weight and satisfying requirements of loads (stresses), stiffness, etc. The process of producing a best structure having optimum structural performance is termed as structural optimization.

II. DESIGN METHODOLOGY

Reduction of material has been one the critical aspects of any design along with reduction in deformation and stress factors, which increases the life of the product. Structural optimization tools like topology and shape optimization along with manufacturing simulation are becoming attractive tools in product design process. These tools also help to reduce product development time. Objective of this investigation is to reduce material and focuses on static analysis. Finite element analysis has been used to implement optimization and maintaining stress and deformation levels.

Validation of the product is of paramount importance and to verify the product the optimized model is compared with the initial model.

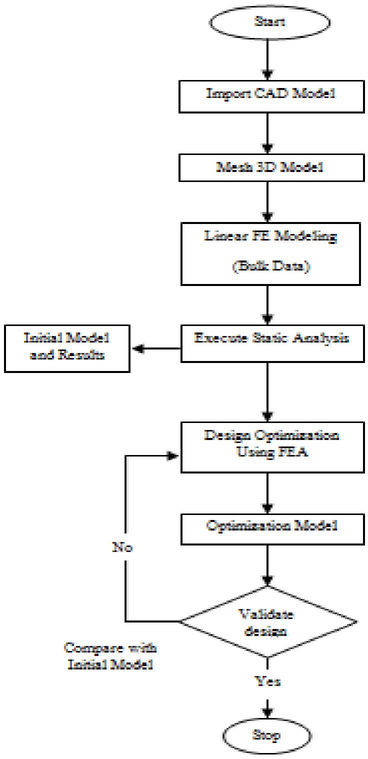


Fig1. Design optimization process flowchart

III. VEHICLE SPECIFICATION

TABLE1. SPECIFICATION DATASHEET

Product	1612
Type	Long Haul
Applications	Market Load, Parcel, Tankers, Agro Perishables
Engine	H Series 4CTI3K turbocharged intercooled with inline FIP
Power	120 Hp @ 2400 rpm
Torque	415 Nm @ 1500 rpm
Clutch	Single plate dry plate, 330 mm dia, F510 MCC lining
Transmission	ZF S636 OD gearbox; First Gear Ratio (FGR) 8.28:1
Suspension	Semi Elliptical multileaf
Cabin	All steel factory built sleeper cabin
Tire	10.00 x 20 - 16 PR - ply (Radial optional)
Maximum Speed	89 kmph
Gradeability	21.30 %
Gross Vehicle Weight	16200 kgs
Capacity	10 Ton
Wheelbase	4800
Loading Span	20 ft

IV. MODEL AND ANALYSIS

IV.I. MODELLING

CAD model of damper bracket is developed in 3D modeling software SOLIDWORKS, it consists of bolt holes for mounting, hole on surface for co-joining the damper strut. The periphery is designed such that the stress concentration can be reduced. The design depends on the diameter of the damper.

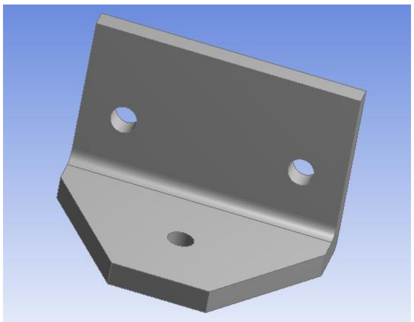


Fig.2 CAD model of bracket

IV.II. MESHING

CAD model of bracket is converted into Parasolid file and is imported into Ansys Workbench simulation. Geometry cleanup was performed prior to meshing of model. Finite element model was developed using Ansys Workbench Simulation. For better quality of mesh fine element size is selected with advanced featuring options.

IV.III. FORCE DISTRIBUTION ANALYSIS AND DEFINING CONSTRAINS

The boundary condition and loading condition were set following conditions for testing purpose in one automotive manufacturing company. The boundary conditions were defined by fixing all the bolt holes surfaces. The force is applied to the patch where the damper strut touches. It is solved in terms of total deformation, equivalent stress and equivalent strain.

IV.IV. CALCULATIONS

Capacity of chassis = 10 ton
Weight of the vehicle = 16200Kg
Total Load acting on the Mount = Capacity of the Chassis + Weight of the vehicle + 4000Kg (Tolerance)

Therefore,
Max load = 30000Kg.

The mount will act as a cantilever beam.

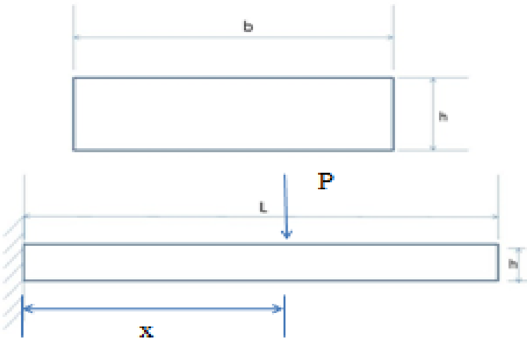


Fig.3 2D Sectional view of the bracket

Here P = 30000 x 9.81 = 294300N

b = 121.5mm, h = 10mm.

$Ixx = bh^3 / 12.$

Therefore,
Secondary moment of inertia:
$I_{xx} = 121.5*10^3/12$
$I_{xx} = 10125mm^4$.

Bending Moment (M) = P x X (where X = 34.75mm)

$M = 294300 \times 34.75 \times 10^{-3}$ Nm

Therefore,
M = 10,226.92 Nm

Section Modulus Z = I/y
Where I = Moment of Inertia about neutral axis.
y = Distance of the outermost layer from the neutral axis.

$Z_{xx} = 10125/ (h/2)$ Nmm^3
$Z_{xx} =$ 2025 Nmm^3

Maximum bending stress (τ) induced is given by the following equation:

$M = \tau_{max} \times Z$

Therefore,
$\tau_{max} = M/Z$
$= 5050.33$ N/mm^2

V. RESULTS

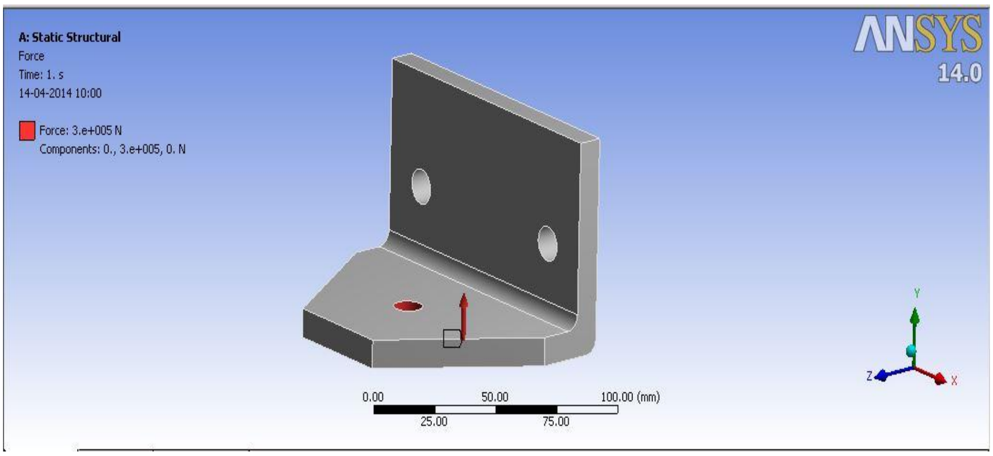


Fig.4 Constrains for structural analysis

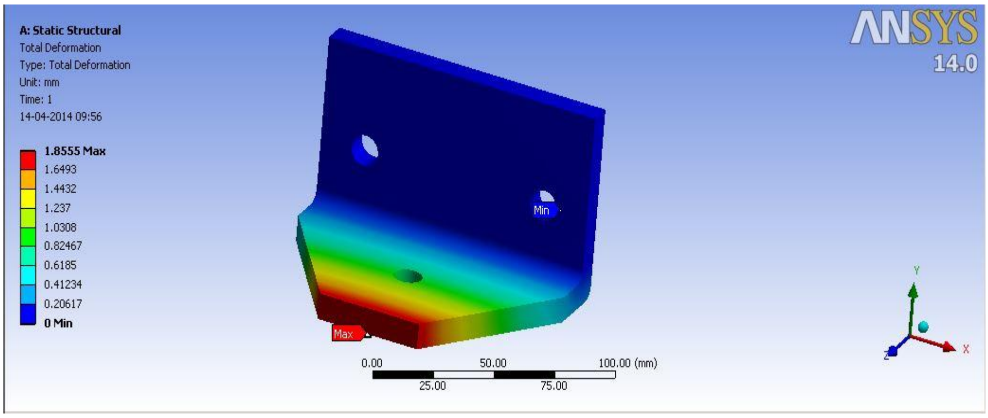


Fig.5 Total deformation for initial design

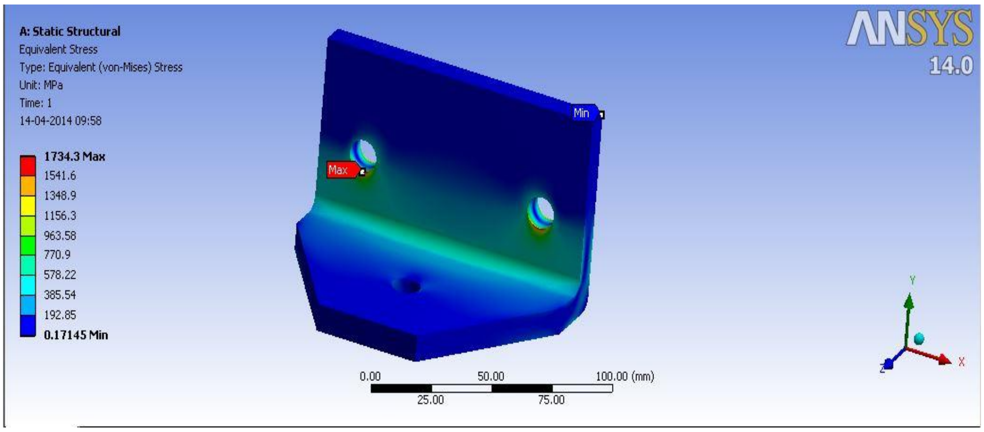


Fig.6 Von Mises stress for initial design

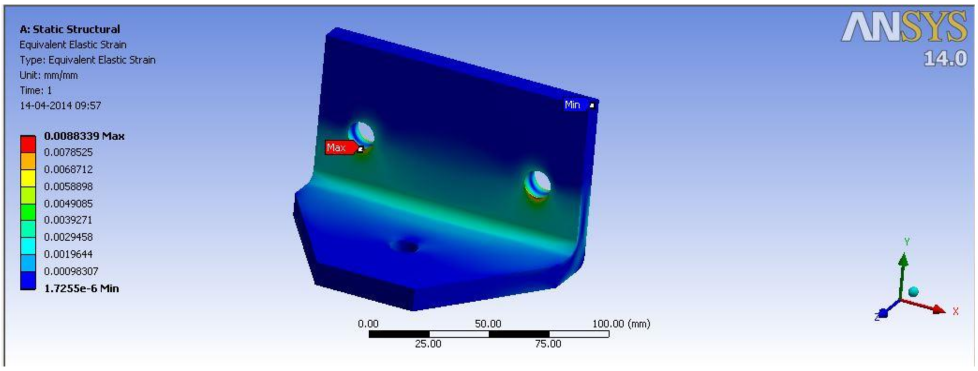


Fig.7 Equivalent elastic strain for initial design

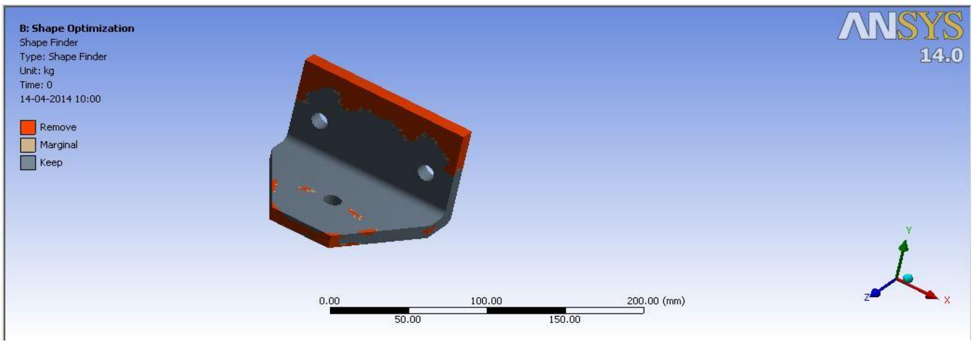


Fig.8 Shape optimization for material removal

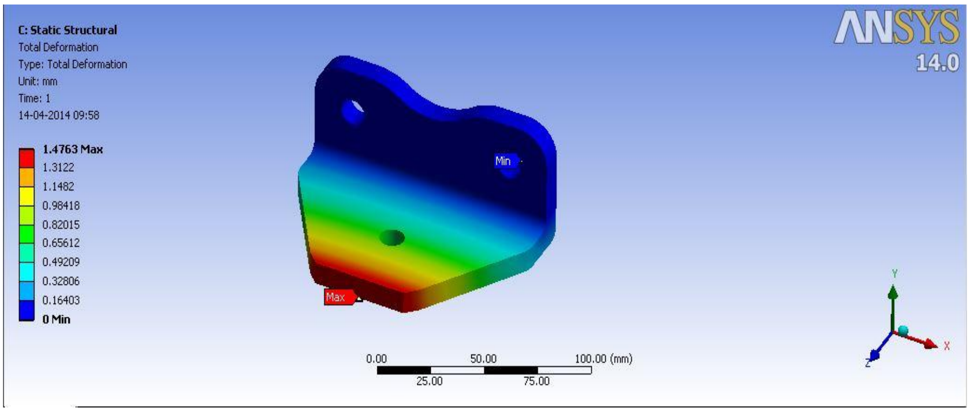


Fig.9 Total deformation for optimized design

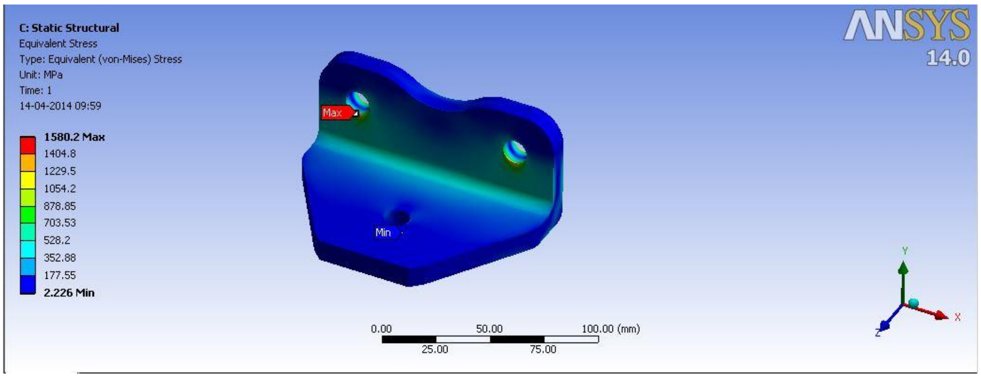


Fig.10 Von Mises stress for optimized design

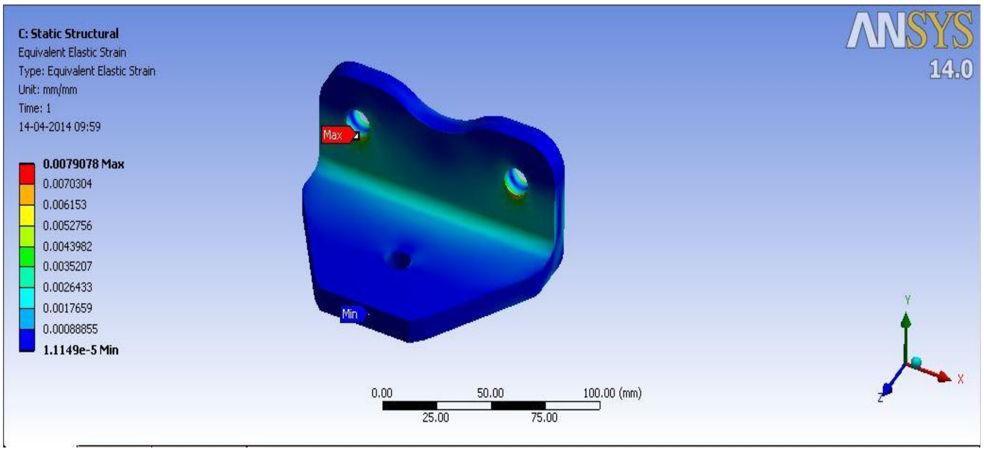


Fig.11 Equivalent elastic strain for optimized design

Table2. ANALYSIS RESULT

Serial Number	Material	Deformation (mm)	Von Mises Stress (MPa)	Equivalent Strain	Weight (kgs)
1.	Initial Design	1.8555	1734.3	0.0088	1.2435
2.	Optimized Design	1.4763	1580.2	0.0079	0.9804

VI. CONCLUSION

Validation is done through finite element solver with the initial model and checked that maximum stress and displacement are within control. This optimization process also gives small change on the displacement and stress. The displacement and stress has significantly reduced in the new optimized design proving to provide better life of the product and reduced mass of the bracket with a margin of 21.15% reduction in material.

REFERENCES

[1] Botkin M.E., Shape Design Modelling using Fully-Automatic 3-D Mesh Generation, *Proceedings of 31st AIAA SDM Conference,* 1990

[2] R. H. Hardin and N. J. A. Sloane, A New Approach to the Construction of Optimal Designs, *Journal of Statistical Planning and Inference, vol. 37,* 1993, 339-369

[3] Belegundu A.D., Rajan S.D., A shape optimization approach based on natural design variables and shape functions, *Computer Methods in Applied Mechanics and Engineering, 66,* 1988, 87–106

[4] Bennett J.A., Botkin M.E, Structural shape optimization adaptive mesh refinement, *AIAA Journal, 23,* 1985, 458–464

[5] Chang K.H., Choi K.K., Tsai C.S., Chen C.J., Choi B.S., Yu X., Design sensitivity analysis and optimization tool (DSO) for shape design applications, *Computing Systems in Engineering, 6,* 1995, 151–175

[6] Hong Suk Chang, A Study on the Analysis Method for Optimizing Mounting Brackets, *SAE International,* 2006, 01-1480

[7] Mahendra M. Gaikwad, Bracket Document 112978 (United Kingdom: Coventry Engineers)

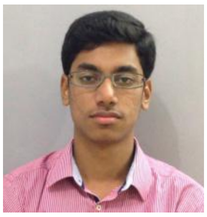
[8] Yao T.M., Choi K.K., 3-D shape optimal design and automatic finite element regridding, *International Journal for Numerical Methods and Engineering, 28,* 1989, 369–384

BIOGRAPHIES AND PHOTOGRAPHS

Promit Choudhury

Promit Choudhury is pursuing fourth year Mechanical Engineering, at SRM University, India. He will graduate in May 2014 with a Bachelor of Mechanical Engineering. He also underwent various courses on Computer Aided Engineering and Life cycle management Engineering.

He has 3 years of working experience as CAE and Power train Development engineer in a team and was the Technical Director of SRM University's Official BAJA SAE INDIA Team. With immense interest in power train development and having an integral role in the optimizing of an All-Terrain vehicle project, he gained a comprehensive overview of Vehicle dynamics, drive train systems and its design topology and optimization. Through his first experience in the field of motorsports he understood the challenges faced in designing and fabricating a Vehicle. His research interests includes trajectory mapping, shape optimization, FEA,, CFD.



Chinmaya Hemanth Krishna

Chinmaya Hemanth Krishna is currently a third year Automobile Engineering Student, at SRM University, India. He will graduate in May 2015 with a Bachelor of Automobile Engineering. He also underwent independent courses on Computer Aided Designing and Industrial Engineering.

Chinmaya Hemanth is currently working as CAD and Suspension designer in SRM University's Official BAJA SAE INDIA Team. With immense interest in Vehicle Dynamics and having an integral role in the designing of an All-Terrain vehicle project, he gained a comprehensive overview of Vehicle dynamics, suspension systems and its design considerations. Through his first experience in the field of motorsports he understood the challenges faced in designing and fabricating a Vehicle. He has also gained Industrial experience in complete Vehicle assembly by interning with Nissan Ashok Leyland, India.