© May 2018 | IJIRT | Volume 4 Issue 12 | ISSN: 2349-6002

Design Optimization and Analysis of Moped Silencer Guard

Raghavendra Reddy N V[1], Ramesh B R, [3]Madhu H T[2]
[1,2] Asst. Professor, Department of Mechanical Engineering, Jain Institute of Technology, Davangere – 577 003, Karnataka, India

Abstract- **Silencer guard is the component used to cover exhaust pipe in two wheeler vehicle such that the hot pipe should not affect the passenger. It is given to protect the exhaust pipe and as well as to protect the passenger from the hot pipe. Compact and contoured guard stays cool to the touch. It creates a barrier to prevent the rider from bumping into or resting his or her boot or leg along the back portion of the muffler. Generally silencer guard comes in different materials. In this, guard of Hero Pleasure is considered, as failure of the equipment is found in most of the case. The silencer guard of pleasure is made of poly propylene plastic. The failure of this equipment causes due to the load applied by passenger on it. In this project the force of about 60kg is applied on the guard component and analysis is carried out. For the existing component it was observed in the analysis results that stress induced is more than that of allowable limit and the displacement is maximum. Hence the optimization of design and analysis is carried out with additional stiffeners to overcome the failure and be implemented.**

Index Terms- **Silencer guard; FEA; Mesh; Strength, Stiffness; Analysis.**

INTRODUCTION

Silencer guard is the component used to cover exhaust pipe in two wheeler vehicle such that the hot pipe should not affect the passenger. It is given to protect the exhaust pipe and as well as to protect the passenger from the hot pipe. Compact and contoured guard stays cool to the touch. It creates a barrier to prevent the rider from bumping into or resting his or her boot or leg along the back portion of the muffler.

Silencer cover is manufactured from best quality Polypropylene plastic granules. The fitment of the product is as per the specification.

Figure 1: Silencer Guard of Hero Pleasure Scooter

Generally silencer guard comes in different materials. In this, guard of Hero Pleasure is considered, as failure of the equipment is found in most of the case. The silencer guard of pleasure is made of poly propylene plastic. The failure of this equipment causes due to the load applied by passenger on it.

A. History

Finite Element Analysis was used in Civil Engineering applications in early 1900's. Finite Element Analysis (FEA) was first developed in 1943 by R. Courant, who utilized the Ritz method of numerical analysis and minimization of variation in calculus. By the early 70's, FEA was limited to expensive mainframe computers generally owed by the aeronautics, automotive, defence, and nuclear industries. Since the rapid decline of cost of computers and the phenomenal increase in computing power, FEA has been developed to an incredible precision.

Advantages of Finite Element Analysis

- Visualization Of results is easy. Points of maximum stress and displacements can be easily visualized in structural analysis.
- Design cycle time is reduced.
- Number of prototypes for testing is reduced.
- Testing time and cost is reduced.
- Intricate geometry problems can be easily solved, which is difficult task if analytical methods are used.
- Approximate results are obtained close to exact solution.
- Optimization of design (physical or functional) can be done.

Applications

Linear, Non-linear, Buckling, Thermal, Dynamic and Fatigue analysis.

B. Introduction to Linear static analysis

Linear means straight line. In linear analysis, the FE solver will therefore always follow a straight line from base to deformed state.

Static: There are two conditions for static analysis:

- The force is static i.e. there is no variation with respect to time (dead weight).
- Equilibrium condition Σ forces (Fx, Fy, Fz) and ΣMoments (Mx, My, Mz) = 0.

In linear static analysis displacements, strains, stresses, and reaction forces under the effect of applied loads are calculated.

A series of assumptions are made with respect to a linear static analysis:

Small Deflection – Determine whether the deflections obtained or predicted are small relative to the size of the structure. For thin structures, a deflection that is less than the thickness would be considered a small deflection. The deflection between two supports should be only a small percent of the distance between supports. This is especially true if the deflection causes a differential stiffness effect such as mid-plane stretching of a clamped plate.

Small Rotations – In linear codes all rotations are assumed to be small. Any angle measured in radians should be small enough that the tangent is approximately equal to the angle. Using this assumption, a ten degree angle introduces an error of approximately one percent in all related calculations. A thirty degree angle results in approximately a 10 percent error in deflection due to rotations assumed linear.

Material Properties – Linear solvers assume that all material behaves in a linear elastic manner. Some materials have a non-linear elastic behavior, and although they do not necessarily yield, they still result in non-linear structural behavior and require non-linear codes for solution. If a structure is to be loaded beyond its yield point, non-linear analysis would also be required. See the figure below for a comparison of material behavior.

Constant Boundary Conditions – In order to correctly use a linear finite element program, the boundary conditions must not be dependent on the load application.

Linear Assumption Summary

- Deflections should be small relative to structure.
- Rotations should be less than 10 degrees to 15 degrees.
- Material should be linear elastic.
- Boundary conditions should be constant

C. Theories of Failure

When a material is subjected to one type of stress i.e., axial or bending or torsional, then it is very easy to predict when the failure is likely to occur. However if material is subjected to a complex stress system, then it is difficult to predict the failure of the material straight away. In order to predict the failure of the material under combined stresses, the following theories of failure have been formulated.

- Maximum normal stress theory or Rankine's theory
- Maximum shear stress theory or Guest's theory
- Maximum principal or Normal strain theory or Saint Venant's theory
- Maximum Strain energy theory or Haigh's theory
- Mohr's theory
- Maximum distortion energy theory or Von-Mises theory

To determine the stresses induced in the silencer guard, Von-Mises theory is used by the solver (NASTRAN). This theory gives the equivalent stresses induced in the component.

METHODOLOGY

A. Existing model of silencer guard

The existing model of the silencer guard and the place where it is mounted is shown in the figure.

Figure 2: Photograph of Existing Silencer Guard

The silencer guard is mounted on the mounting plate by using rubber washers, bolts & nuts. The mounting plate is shown in the figure, which is welded to the silencer pipe.

The load is applied on the silencer guard by the person sitting on the rear seat of the moped. The person tends to keep his or her leg on the silencer guard rather than keeping it on the foot rest. Due to this loading the component gets deformed and by regular loading the crack is formed as shown in the figure. This crack slowly propagates due loading and finally the component fails.

B. Drawings of Existing Model

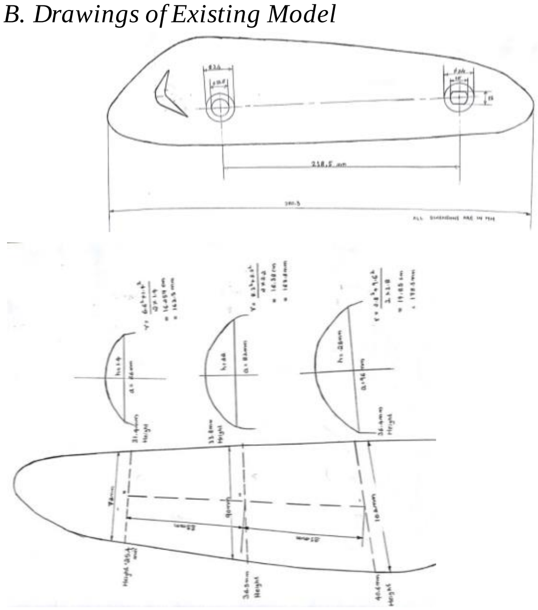


Figure 3: 2D drawing of Silencer Guard

The very first stage before analysis is the generation of CAD models on which analysis is done. The detailed information of the component can be described using the 2D drawings. The drawing of the part is shown below:

C. Silencer Guard Analysis - Objective

- To Study the existing design of the Silencer guard.
- To conduct structural analysis to understand the behavior of Silencer guard under static loading conditions.
- To propose new design and analyze it, to avoid the failure of the component.
- Finding out maximum displacement and stress induced.

D. Preparation of CAD Model of Existing Design using CATIA V5 R20

CAD Model is generated by using dimensions taken in the drafting process. Different views of the CAD Model are shown below.

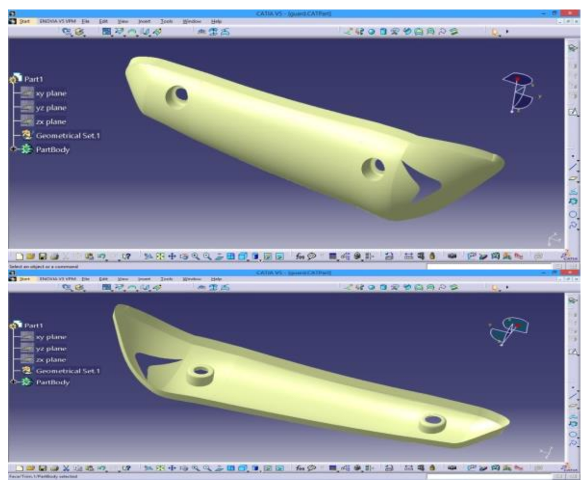
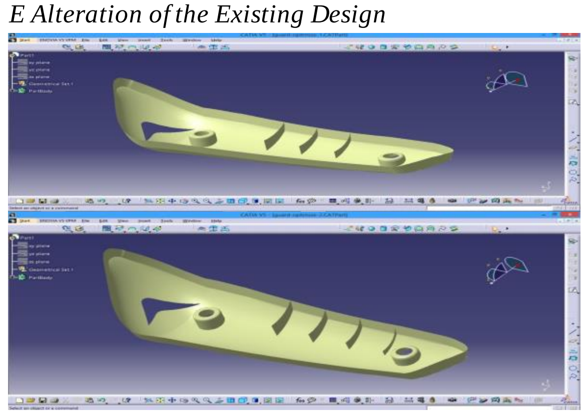
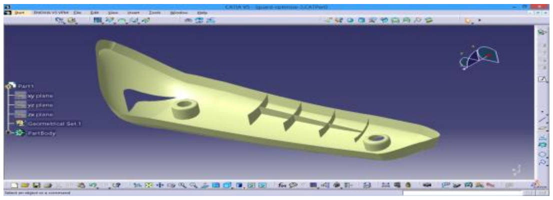


Figure 4: CAD Model of Existing Silencer Guard

E Alteration of the Existing Design



Case-I *Case-II*



Case-III

Figure 5: CAD Model of Modified Silencer Guard

Case-I – Three stiffeners have been added at high displacement zone to avoid the failure.

Case-II – Additional Stiffener is added to increase the efficiency

Case-III – Since the high stresses induced within the stiffeners, the horizontal stiffener is added as a support.

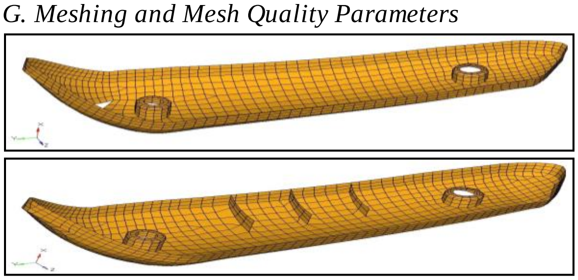
F. Procedure

The CAD model is imported into the HYPERMESH graphical interface. The geometry is first checked by visual inspection and then various features are identified. Now mid surface is extracted and geometry clean-up is carried out to ease the process of Discretization. Mesh the component according to the given quality parameters.

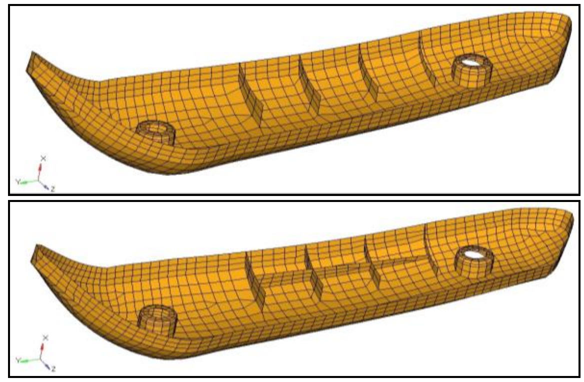
Once the mesh satisfies quality parameters, Deck preparation can be started. Now it is the time to apply all the boundary conditions and loads on the defined locations. After this all the input requirements for the solver (NASTRAN) are provided and deck is exported to the file format (*.dat) supported by NASTRAN. This file is fed to the solver to get required results.

The obtained results can be viewed in the form of plots using Post-processor (HYPERVIEW). This provides results with graphical images and the solution at any point can be easily determined. Thus obtained results are compared with the properties of material and standard conditions. If the component is failing, the design of failing part can be changed and required results can be obtained.

G. Meshing and Mesh Quality Parameters



Existing Case-II



Case-I I Case-III

Figure 6: Mesh Model of Silencer Guard

Mesh Type		Shell Mesh
Element Type		2D Linear Shell
No. of Elements	Existing model	1265
	Optimized model	
	Case-I	1303
	Case-II	1311
	Case-III	1354
Minimum Element Length		2mm
Average Element Length		5mm
Warpage		< 15deg
Jacobian		0.65 (should be > 0.6)
Skewness		< 60°
Aspect Ratio		4 (should be < 5)
Quad Angles		45° < quad < 135°
Tria Angles		20° <tria< 120°
Tria Percentage		1.5% (Should be < 5%)

Table 1: Meshing Parameters

H. Material of Silencer Guard

The material used for silencer guard is Polypropylene with 20% glass fibre (PP GF20).

Density	1.04 e^{-06} Kg/m^3
Poisson's Ratio	0.45
Tensile Strength	88 MPa
Young's Modulus	4.6 GPa
Thickness of material	2 mm

Table 2: Silencer Guard Material Properties

I. Deck Preparation

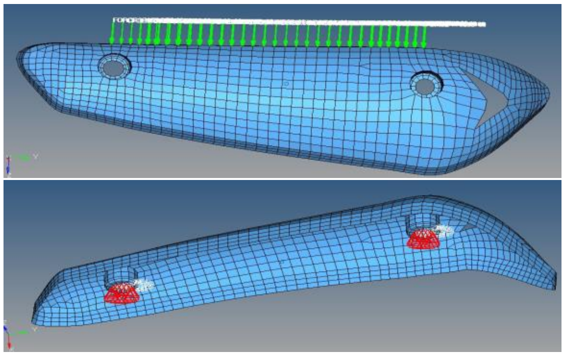


Figure 7: Deck Preparation of Existing Silencer Guard

The process of preparing the mesh data as per the input requirements of the solver is called as Deck Preparation. In the analysis of silencer guard, the meshed component is sent for deck preparation as per the input requirements of the solver (NASTRAN). In this process, loads, constraints are applied and along with this the Nastran codes for linear static analysis are given. The Loads and constraints applied are shown below

The loading area remains same on the existing and optimized model as shown in the figure above.

Constraints are applied at the bolt mounting locations.

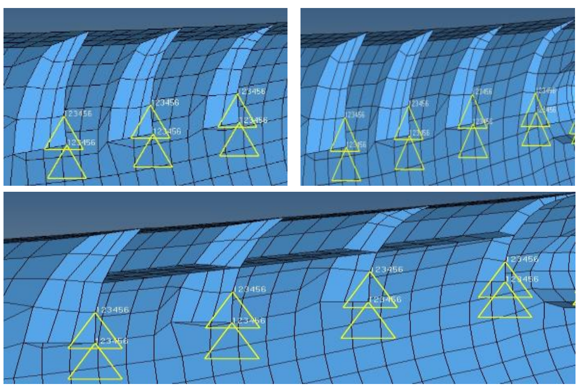


Figure 8: Silencer Guard with Constraints

The constraints are applied on the stiffeners because they are in contact with the mounting plate.

J. Validation

Before actually conducting the analysis of silencer guard it is important to know whether the software yields accurate results. To estimate this stepped bar problem is considered, its analysis done using Hyper Mesh pre-processor and Nastran as solver. For the same problem theoretical calculations have been done using Finite Element Method.

FEM Method

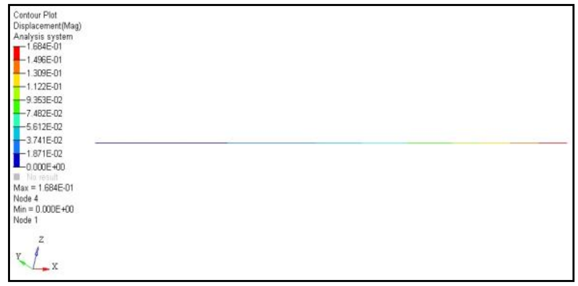


Figure 9: Displacement Distribution Graph

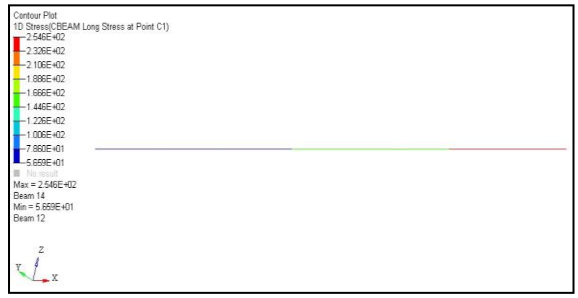


Figure 10: Stress Distribution Graph

The maximum displacement is 0.1684 mm.

The maximum stress is 254.6 MPa

Numerical Method

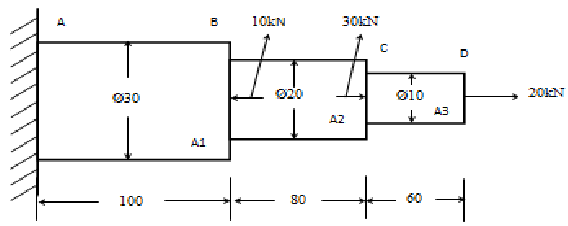


Figure 11: 2D Stepped Bar

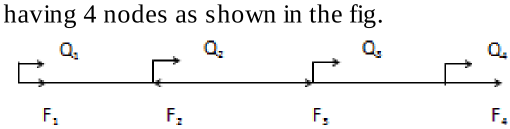
Data:

Young's Modulus E=200 GPa

Area of elements, A_1=706.858mm2, A_2=314.15mm2 and A_3=78.535mm2

Finite element model

Modelling the given bar using stepped bar element having 4 nodes as shown in the fig.



$F_1=0$, $F_2=-10kN$, $F_3=30kN$, $F_4=20kN$

Figure 12: Stepped Bar element with Nodes

Elemental stiffness matrix

The stiffness matrix for the bar element is given by,

$$\text{Element 1: } K_1 = (A_1 E/L_1) \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$K_1 = (706.858 * 2 * 10^6)/100 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$K_1 = 10^6 \begin{bmatrix} 1.413 & -1.413 \\ -1.413 & 1.413 \end{bmatrix}$$

Element 2: $K_2 = (A_2E/L_2) \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$

$$K_2 = (314.15 * 2 * 10^5)/ 80 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$K_2 = 10^6 \begin{bmatrix} 0.785 & -0.785 \\ -0.785 & 0.785 \end{bmatrix}$$

Element 3: $K_3 = (A_3E/L_3) \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$

$$K_3 = (78.539 * 2 * 10^5)/ 60 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$K_3 = 10^6 \begin{bmatrix} 0.261 & -0.261 \\ -0.261 & 0.261 \end{bmatrix}$$

Global stiffness Matrix

Since there are 3 elemental stiffness matrixes, the Global stiffness matrix can be obtained by adding these 3 stiffness matrix.

$$K = K_1 + K_2 + K_3$$

$$K = 10^6 \begin{bmatrix} 1.413 & -1.413 & 0 & 0 \\ -1.413 & 2.198 & -0.785 & 0 \\ 0 & -0.785 & 1.046 & -0.261 \\ 0 & 0 & -0.261 & 0.261 \end{bmatrix}$$

Global Nodal Displacement Vector

The Nodal Displacement Vector for the bar is given by,

$$\{Q\} = \begin{bmatrix} Q1 \\ Q2 \\ Q3 \\ Q4 \end{bmatrix}$$

Global Load Vector

The load vector for the bar is given by,

$$F = \begin{bmatrix} F1 \\ F2 \\ F3 \\ F4 \end{bmatrix} = \begin{bmatrix} 0 \\ -10000 \\ 30000 \\ 20000 \end{bmatrix}$$

Equilibrium Condition

The equilibrium conditions for the bar is given by,

$$\{k\}\{Q\} = \{F\}$$

$$\begin{bmatrix} 1.413 & -1.413 & 0 & 0 \\ -1.413 & 2.198 & -0.785 & 0 \\ 0 & -0.785 & 1.046 & -0.261 \\ 0 & 0 & -0.261 & 0.261 \end{bmatrix} 10^6 = \begin{bmatrix} Q1 \\ Q2 \\ Q3 \\ Q4 \end{bmatrix} \begin{bmatrix} 0 \\ -10000 \\ 30000 \\ 20000 \end{bmatrix}$$

Applying Boundary Conditions

From the fig boundary conditions are $Q_1 = 0$. Using elimination method of handling boundary condition, eliminating corresponding row and column with respect to the known boundary conditions i.e., $Q_1 = 0$ we get

$$2.198 \times 10^6 Q_2 - 0.785 \times 10^6 Q_3 + 0 Q_4 = -10 \times 10^3$$

$$-0785 \times 10^6 Q_2 + 1.046 \times 10^6 Q_3 - 0.261 \times 10^6 Q_4 = 30 \times 10^3$$

$$0 \times Q2 - 0.261 \times 10^6 Q_3 - 0.261 \times 10^6 Q_4 = 20 \times 10^3$$

The nodal displacement vector is

$$\{Q\} = \begin{bmatrix} 0 \\ 0.0283 \\ 0.0920 \\ 0.1686 \end{bmatrix} \text{mm}$$

To find the stresses induced.

For Equilibrium, Reaction force at fixed end

$$-R_A - (10 \text{ X } 10^3) + (30 \text{ X } 10^3) + (20 \text{ X } 10^3) = 0$$

$$R_A = 40 \text{ X } 10^3$$

Free Body Diagram

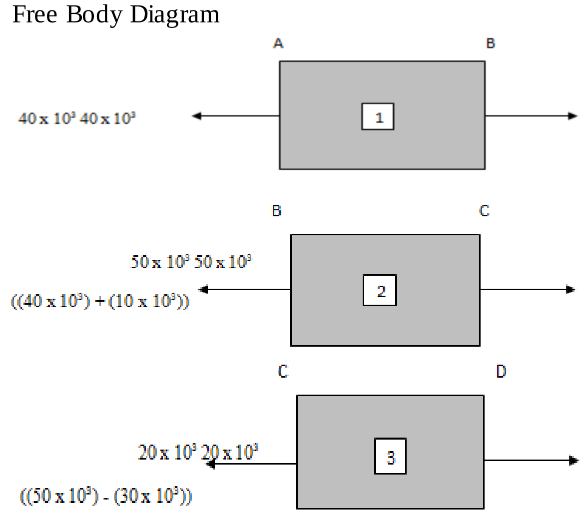


Figure 13: Free Body Diagrams

After conducting analysis it is found that the theoretical results and software results are matching. Hence the Hyper Mesh and Nastran is used to solve static analysis of silencer guard in further stage of the project.

	Theoretical results	Software results
Stress (MPa)	254.67	254.6
Displacement (mm)	0.1686	0.1684

Table 3: Comparison of theoretical and software result

RESULTS

NASTRAN after solving gives the result file in *.op2 format. To view thus obtained results HYPERVIEW post-processor is used. The input file of NASTRAN with *.dat extension and output file from the NASTRAN with *.op2 extension are the inputs to HYPERVIEW post-processor. Results in terms of displacement and stress can be viewed in the HYPERVIEW graphical environment.

The Results of Silencer Guard analysis in terms of displacement and stress are shown in the following pictures:

A. Displacement and Stress Plots

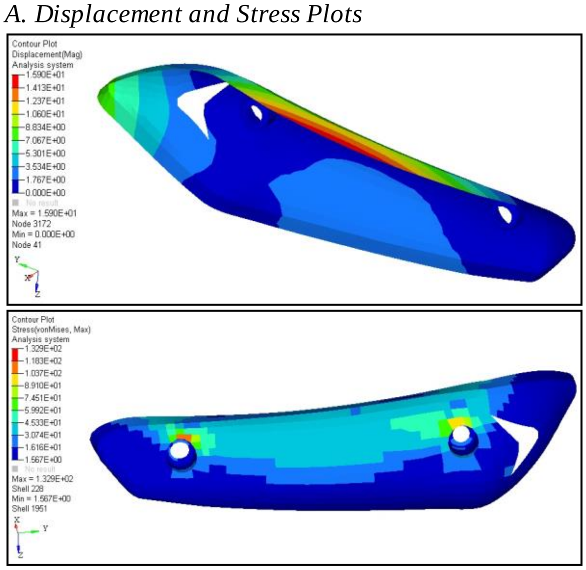


Figure 14: Displacement and Stress plots for Existing Model

It can be observed in the analysis that the

- Maximum displacement is 15.90 mm
- Maximum stress induced is 132.9 Mpa

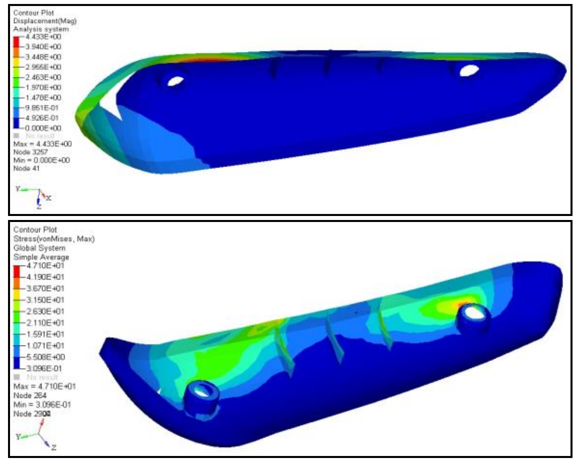


Figure 15: Displacement and Stress plots for Case-I

It can be observed in the analysis for case-I that the

- Maximum displacement is 4.433 mm
- Maximum stress induced is 47.10 Mpa

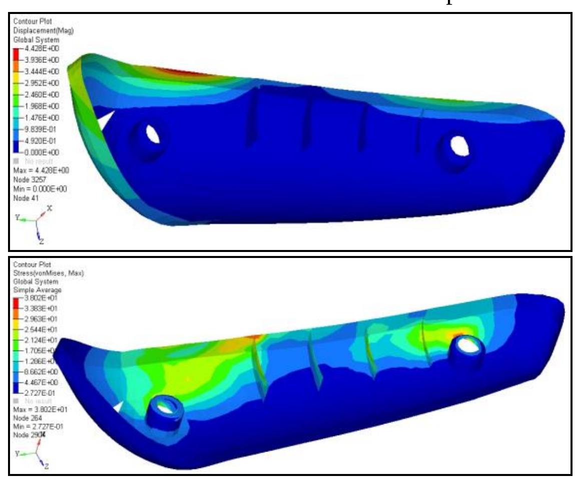


Figure 16: Displacement and Stress plots for Case-II

It can be observed in the analysis for case-II that the

- Maximum displacement is 4.428 mm
- Maximum stress induced is 38.02 MPa

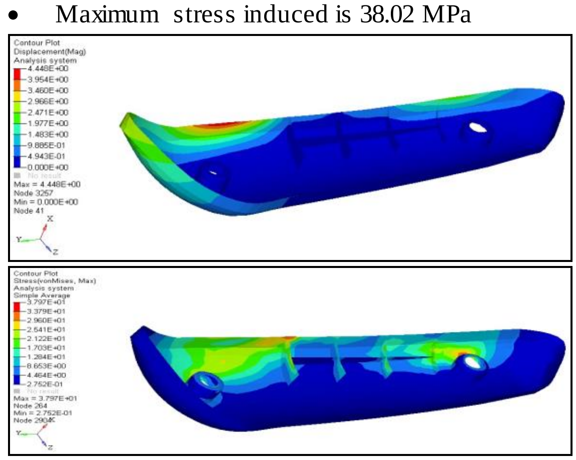


Figure 17: Displacement and Stress plots for Case-III

It can be observed in the analysis for case-II that the

- Maximum displacement is 4.448 mm
- Maximum stress induced is 37.97 Mpa

B. Result Comparison

The comparison between the stress induced in the Silencer Guard and the maximum allowable tensile strength as per the material properties of Silencer Guard is shown in table below

Model	Maximum Stress induced in Silencer Guard (MPa)	Tensile strength of the material of Silencer Guard (MPa)
Existing model	132.9	88
case-I optimized model	47.10	
case-II optimized model	38.02	
case-III optimized model	37.97	

Table 4: Comparison of stresses

The maximum stress induced in the existing model is greater than the tensile strength of the material of silencer guard, hence the component fails. To overcome the failure occurring in the silencer guard stiffeners are provided, which reduces the stress induced to approximately half the value of the tensile strength of material of silencer guard.

Model	Maximum Stress (MPa)	Maximum Displacement (mm)
Existing model	132.9	15.90
case-I optimized model	47.10	4.433
case-II optimized model	38.02	4.428
case-III optimized model	37.97	4.448

Table 5: Comparison of stresses and Displacements

The table below shows the maximum stresses and maximum displacements induced in the existing model and the optimized models of silencer guard.

CONCLUSION

The objective of silencer guard analysis was to find the maximum stress and displacements induced in the component and to optimize it for better performance. Considering the maximum loading conditions the analysis has been carried out using Nastran solver.

In this project the force of about 60kg is applied on the guard component and analysis is carried out. For the existing component it was observed in the analysis results that stress induced is more than that of allowable limit and the displacement is maximum. Hence the optimization of design was carried out.

Optimization was carried out in three stages, in first stage with three stiffeners in the model, in the second stage with four stiffeners and the final stage with an additional horizontal stiffener.

When analysis was carried out it was found that Case-1 stress and displacement is reduced in comparison with the existing model. Case-2 model has a stress value still reduced compared to case-1 optimized model. In case-3 there is a still reduction but not a drastic one. In comparison Case-2 will be the best one to finalize because case-3 has slighter reduction in stress and displacement but demands for an additional horizontal rib.

Hence, with all these observations it can be concluded that the Case-2 Altered model with four stiffeners is the optimized model and it is best to be implemented.

REFERENCES

[1] Mukesh Patil, Rohit Thakare, Aniket Bam., "Analysis of a Tanker Truck Chassis"

[2] Dr.R.Rajappan, M.Vivekanandhan,“ Static and Modal Analysis of Chassis by Using Fea” The International Journal Of Engineering And Science (IJES) Volume 2, Issue 2, Page 63-73 (2013).

[3] Vikrant C. Gaikwad, Dr. Suresh M. Sawant., "Free Free Run Analysis For 2d Meshing", E-ISSN2249–8974

[4] Mohd Nizam Sudin, Musthafah Mohd Tahir, Faiz Redza Ramli, Shamsul Anuar Shamsuddin, "Topology Optimization in Automotive Brake Pedal Redesign",ISSN : 0975-4024 Vol 6 No 1 Feb-Mar 2014

[5] Thangarasu Subramaniam, Dhandapani Velliangiri, Sureshkannan Gurusamy, A study of modeling and finite element analysis of Automotive vehicle wheel rim assembly for the Deformation and various stress distribution, E-ISSN 0976-3945

[6] Failure analysis of mudguard using “FEMFAT”