
“DESIGN AND ANALYSIS OF FRONT FENDER OF THREE WHEELER VEHICLE”

¹**MR. ARPIT G. MAHURE**

**ME Scholar, Department of Mechanical, PRMIT & R Badnera, Amravati, India
arpitmah31@gmail.com**

²**DR. NITIN A. WANKHADE**

**Associate Professor, Department of Mechanical, PRMIT & R Badnera, Amravati, India
nitin19732001@rediffmail.com**

³**MR. C. G. JAWANJAL**

**Department of Mechanical, PRMIT & R Badnera, Amravati, India
jawanjal1993@gmail.com**

ABSTRACT: *Front fender design of three wheeler vehicle is very important with the focus on an improvement aspect in the automotive industry. The goals are to increase the performance and to find the solution to reduce the cost of the fender because of the amount of material used, hence able to reduce the production cost. In this paper Finite element models of the front fender of three wheeler vehicle were analyzed, using linear and nonlinear finite element analyses in CAE software. Experimental and analysis stress were compared to evaluate the validity of the FEA approaches. In this study, multi-stage topology and free-size optimization of front fender structure have been performed using Optistruct by Altair engineering, used to design a lightweight front fender when subjected to service loads. An existing commercially available front fender mudguard serves as the reference design space. The objective function for topology optimization is used during the development stage to determine placement of supporting ribs in fender component and confirm material reduction of an existing feasible design space. An overall reduction in weight of 1.04% is achieved over a reference commercially available front fender component.*

The study is focused on a simulation of low velocity impact by a moving object is carried on Existing & Optimized design of front fender mudguard of three wheeler will be tested by using the explicit nonlinear finite element code LS-DYNA solver for Energy absorption capability, Deflection, and stress distribution. Results of optimized front fender are compared against existing design of front fender for better and safe design utilization.

Keywords: Fender, Strain Gauges, Strain Indicator, CAD Modeling, CAE-Computer Aided Engineering, Hyper Mesh, Topology & Free-Size Optimization, Optistruct, Low-velocity impact, LS-DYNA solver.

1. INTRODUCTION

In recent years, with increased demand for low fuel consumption and low cost vehicles, the light weight designs are highest priority during product development stage in automobile industry. This paper deals with finite element stress analysis of front fender of three-wheeler vehicle, experimental validation of the stress data with analysis results and design optimization with the aim of reducing weight for specified loads, constrains and design space. Fenders provide sufficient housing for the wheels and suspension linkages. Various materials used for fender depend on the strength, expected life and suitability of manufacturing methods. Preferred materials are sheet metal, plastic and fiber reinforced plastic. Plastic is preferred because of its light weight characteristic but strength is a problem over sheet metal. While designing the fender following factors are

considered. It should provide sufficient cover to the wheel and suspension linkages, it should have sufficient strength to withstand loads and vibration under all operating conditions. Apart from normal loads the fender is subjected to different handling conditions during repair and maintenance of the vehicle. The manufacturer of the vehicle now came to know that the Mud-Guard design required to be modified for handling during repair and maintenance.

1.1. Problem Statement

Nowadays the part cost for front fender is still high and the cost for replacing is quite expensive especially if surrounding area or part also damaged. Manufacturing fender, involving a lot of material and

processes there for the manufacturer difficult to reduce the price. So optimization of the front fender should be done to improve its structural strength by use of CAE software's. The customer also blame the manufacture that the fender easily to damage although the collision was slow. The structural design of the fender should be analyzed to find the optimal design that can improve the toughness, structural strength.



Figure 1: Front Fender of Three Wheeler

1.2. Objectives of the Work

The main task is to investigate how and when structural optimization can be used for fender in the design process. The end result is a sensible methodology of when and how to perform an optimization using a commercial software suite. The front fender of three wheeler road vehicles are designed for low velocity impacts (8 km/hr.). It would be very desirable if vehicle fenders could absorb part of the crash impact to protect the occupants of the vehicle. There is therefore the need for a FE model of the front fender that is very responsive in the changes in the characteristics of the Existing fender to be used in investigating and proposing better structural design for this purpose.

The following are the objectives of the study:

1. Literature review, collecting and gathering the information about existing three wheeler front fender mudguard in Indian market for possible design modifications.
2. To carry out finite element stress analysis of front fender of three-wheeler.
3. To carry out experimental validation of the stress distribution across the whole fender using strain gauges.
4. To perform Multi-Stage optimization the front fender to improve its structural strength by use of CAE software's.

5. For functional requirement validation is done by performing nonlinear finite element stress and stiffness design analysis.

6. To carry out impact analysis using CAE to benchmark the performance of the existing & optimized front fender.

7. To validate virtually the FE results of the optimized front mudguard against the existing front fender FE results in terms of deflection, strain and stress distribution.

2. FINITE ELEMENT MODELLING APPROACH

2.1.CAD modelling

Pro-Engineer is a parametric, feature-based modeling architecture incorporated into a single database philosophy with advanced rule-based design capabilities. The capabilities of the product can be split into the three main heading of Engineering Design, Analysis and Manufacturing. This data is then documented in a standard 2D production drawing or the 3D drawing standard ASME. Modeling of mudguard is done with help of Pro-e software. All geometric and dimensions of the mudguard system including ribs are plotted. The model which was developed in pro-e software required for FE modelling was acquired in Initial Graphical Exchange Specification (IGES) format that is supported in all the CAE pre – processing software. By Importing IGES file into hyper mesh software for Cleaning of geometry and preparing the file for mesh generation.

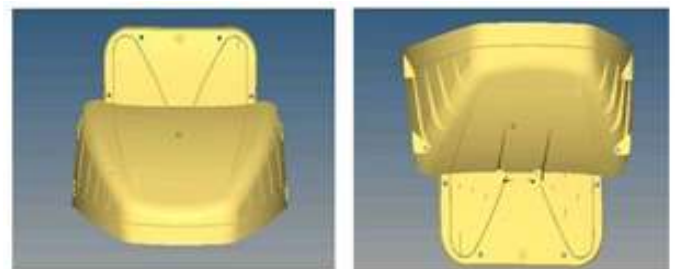


Figure 2 : Existing mudguard model

2.2. FE mesh generation using hyper mesh

Initially, the imported model as IGES, was Controlled for the corrupted and twisted Surfaces and also for the boundaries and Edges between surfaces where the gap in Between is beyond the acceptable range. The modifications are followed by suppressing the boundary patches to avoid the Undesirable layout of the mesh. There are two different 2D mesh types, namely quads and tries. The quads show Better results in

comparison with tries. Hence, each model's surfaces were meshed using 2D Mesh with a Combination of tries and quads. However, the Resulting mesh comprised mainly of quad Elements. Finer Mesh is required to show the nonlinear Behavior of the material and failure. In the meshing application, the aim was to keep the majority of the elements size around 4 mm which is finer/smaller than the mesh size-around 10 mm. But on the other hand, applying finer mesh raises the simulation time duration Due to the explicit nature of the solver for Dynamic analysis.

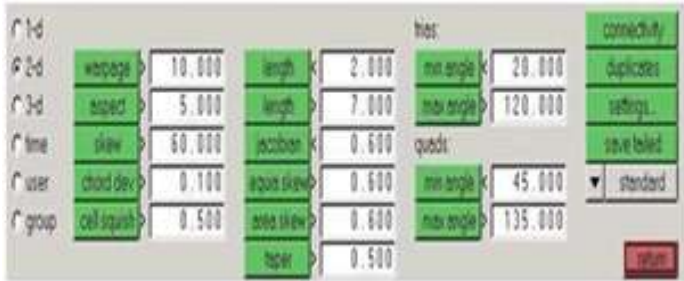


Figure 3: Mesh criteria

2.3. Material information

In the FEA, properties of material play an important role. The property of material is a basic input for structural analysis. Material used for front fender mudguard is normally polypropylene (PP).

Material properties for fender	Symbol (Unit)	Value
Young's modulus	E (MPa)	1100
Poisson's ratio	N	0.381
Density	ρ (ton/mm ³)	1.56 -9
Yield stress	σ_y (MPa)	30-34
Ultimate stress	σ_t (MPa)	42-50

Table 1: Mechanical properties of polypropylene.

2.4. Linear static structural analysis

Static analysis determines the displacements, stresses, strains, and forces in structures or components caused by lodes that do not induce significant inertia and damping effects. Steady loading and response conditions

are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time. During repair and maintenance of the vehicle is normally handled by servicemen, so upward lifting forces acting on fender are critical for which it is not designed and manufactured. In this paper the structural analysis performed for various loads ranges from 0-100 kg at interval of 10kg, respectively results are plotted.

Linear static analysis is carried out in stages as follows:

- Importing Part geometry in Hypermesh
- Meshing the surface
- Applying material and properties
- Applying loads and boundary conditions
- Solving in NASTRAN Solver (sol 101)
- Viewing results in Hyperview

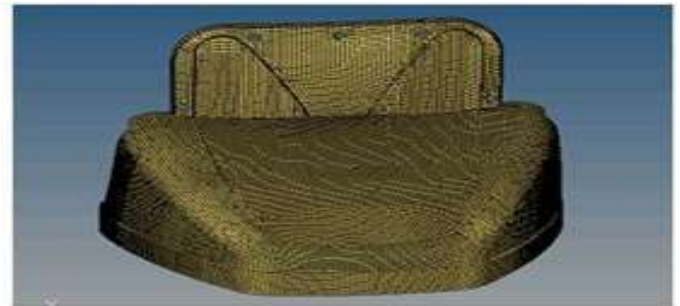


Figure 4: Meshed model of existing fender Critical load condition

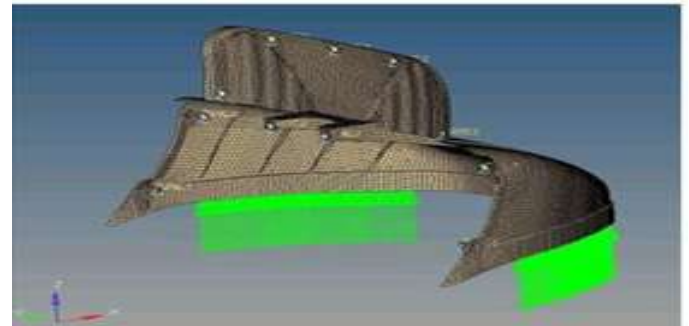


Figure 5: Applied loads & boundary condition on model Critical load condition

Load capacity acting on fender is maximum at 100kg relative to that results are plotted for maximum deflection, maximum stress values are checked for existing design. From the analysis, the fender is found to experience the largest stresses. Hence, the result of the maximum principle stresses is used for further topology optimization.

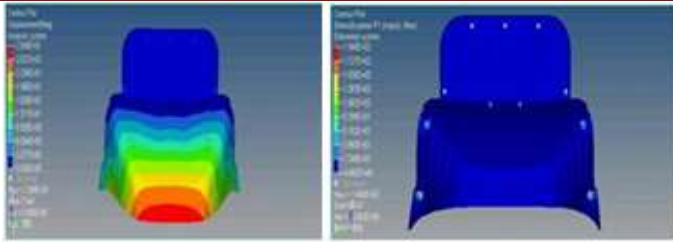
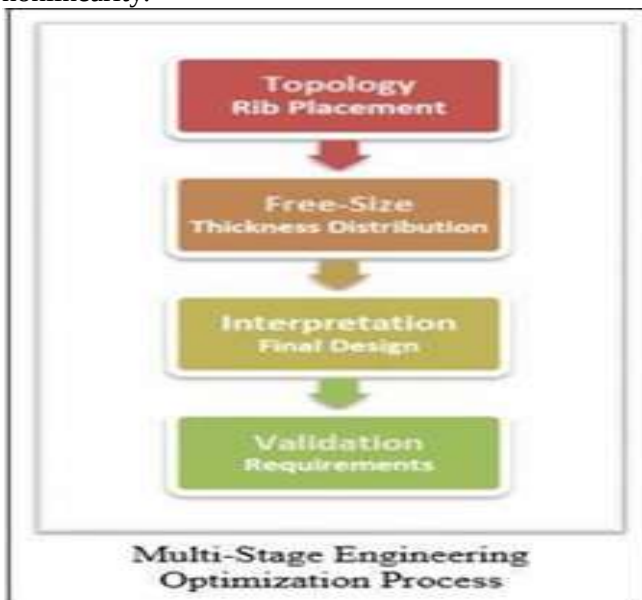


Figure 6 : Displacement & stresplots of the existing model.

3. MULTI STAGE OPTIMIZATION

The multi stage optimization is carried out in two main stages as follows:

The total mass of the reference fender was measured to be 1.48 kg and overall thickness of fender is 3mm. The reference frame is used as a basis for design optimization with the goal of reduced mass under constraints of stress. In the present work, both topology and free-size optimization are based on a linear finite element analysis in Optistruct solver, while the validation is performed using a nonlinear finite element model which accounts for both material and geometric nonlinearity.



3.1. Topology Optimization

It adopts Density approach for optimal material distribution. Available design space is defined with proper loading and boundary conditions with objective to minimize the global weight of the structure, subjected to stress constant.

The below mentioned criteria's are used for topology optimization.

Design Variable: Density of each element within design space

Design Constraint: Stress with specified limit

Objective: Minimize the mass

3.2. Free Size Optimization

The output design by topology optimization with introducing cutouts is set for size optimization to get the optimized thickness of all structural members.

The following criteria are defined for size optimization.

Design Variable: Thickness of the components

Design Objective: Minimize mass

Design Constraint: Stress with specified limit.

3.3. Interpretation Final Design

The results suggested from the free-size optimization of the shell model and topology optimization of the fender frame model are compared and used as a guide to create a final optimized design for the fender frame component with reduced weight. Since fully automated interpretation tools which account fully for manufacturing constraints are not currently available, the interpretation step must generally be done manually and the part redrawn. Based on the observations from topology and free-size optimization results of shell model, a new surface shell model of the fender frame is generated.

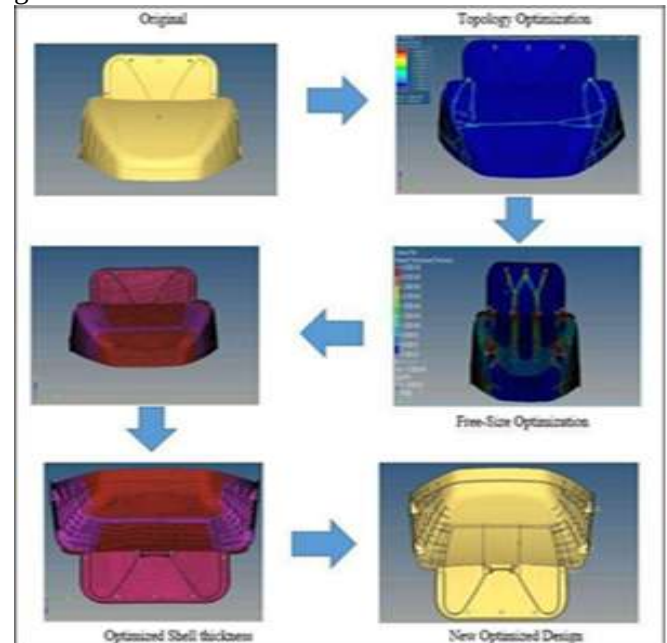


Figure 7: Multi-stage engineering optimization process for fender frame.

3.4. Validation of Optimized Design

To verify strength and deflection requirements, the final optimized fender model is subjected to nonlinear finite element analysis.

3.4.1. Nonlinear Static Structural Analysis

As material used for fender is polypropylene which is nonlinear elastic material. So nonlinear structural analysis is performed by considering material nonlinearity. Materials non-linear stress strain curves have been obtained experimentally by conducting uniaxial tests. These results are enough to carry out nonlinear analysis of homogeneous materials.

3.4.2. Design Validation by Nonlinear Analysis Results

Based on the material non-linearity, a non-linear static analysis is conducted. MSC.NASTRAN solver (sol 106) is used to solve for the stresses and strains. Result was obtained for all load cases upto 100kg but design point view stresses for nonlinear elastic material should be less than ultimate strength of material. Results obtained up to 70 kg are within design limits and above that are consider as failure, thus for comparison between exiting and optimize model results plots are shown for 70kg. The highest stress levels are observed in localized regions on the rear side of the frame at a support rib near the bolt locations.

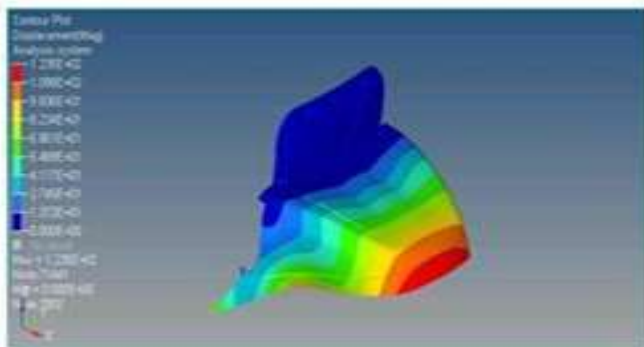


Figure 8(a): Stress & displacement plots of the existing model.

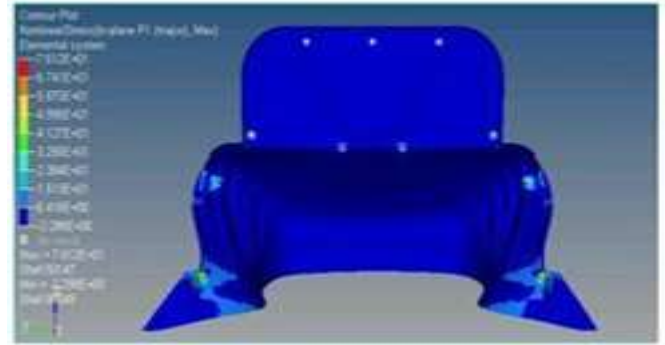


Figure 8(b): Stress & displacement plots of the existing model.

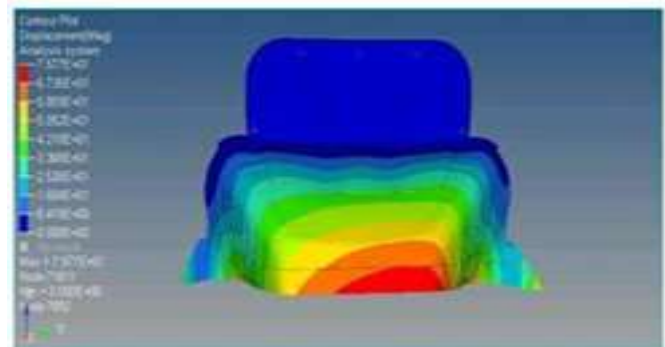


Figure 9(a): Stress & displacement plots of the optimized topology model.

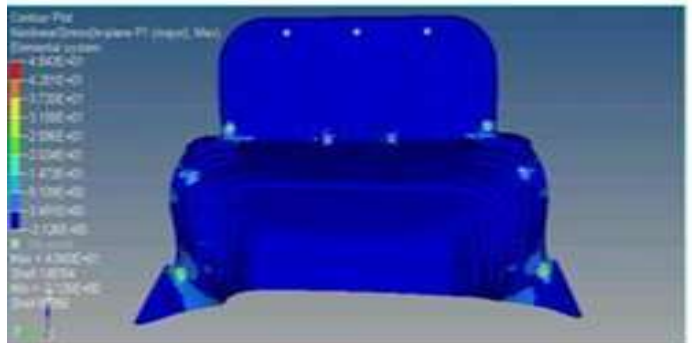


Figure 9(b): Stress & displacement plots of the optimized topology model.

The maximum principle stress is 48.43 MPa which is above the yield stress value for (PP), but below the ultimate strength for the material indicating local plastic yielding but no fracture has occurred. The maximum displacement for the optimized fender design predicted from nonlinear analysis occurs at the front of the fender with a value of 75.8 mm which is 38.7% below the value of 123.5 mm which was computed as the maximum value for the existing reference fender component.

4. SUMMARY AND CONCLUSIONS

Based on the results obtained in the present work, the following summary and main conclusions can be made:

1. The present work demonstrates how topology and free-size optimization with load requirements, stress and stiffness constraints, together with manufacturing constraints on die draw direction and symmetry, can be used to design a lightweight fender frame component. To use topology optimization as a tool for material placement is more systematic than to use more or less guesses based on experience.
2. The existing base design results for most critical loading condition are identified and corresponding stress & displacement plots. It is concluded that these regions are potential areas to remove weight. Moreover, the regions with lower factor of safety are also targeted for improvement using the optimization technique. By observing results obtained, the displacement and stress values are less in optimized fender compared with existing fender model. Using this engineering design optimization process, an overall reduction in weight of 1.04% is achieved over a reference commercially available fender frame component.
3. Crash simulation was performed using the LS-DYNA software package. The analysis has well established the method and parameters of the simulation on modeling and analysis software. It demonstrates the energy absorption pattern in mudguard. It can be seen from the plots that the optimized mudguard absorbs most of the energy during impact of the mudguard. The virtual simulation is a tool which can be used to avoid or reduce the physical testing of mechanical systems and components. The overall effect of this is reduction in development cost as compared to real time physical testing.
4. Impact analysis is done for different fender design models. By observing the Impact Analysis results like Stress, Displacement and strain, the stress values are less for optimized mudguard model than existing fender model. And effective plastic strain is also less for optimized than existing fender model. Thus optimized fender model is safe and better for utilization comparing original existing model.

5. RECOMMENDATIONS FOR FUTURE RESEARCH

There should be investigation on its fatigue life of fender component. Fatigue Failure must involve both vibrational fatigue as well as strain life theory (E-N) analysis. We were able to successfully model and simulate low impact crash using LS-Dyna and get various deformations and other results. Experimental validation of finite element models is a critical aspect which cannot be overlooked. Crash testing of instrumented, physical specimens should be performed as a validation and verification of the analytical results. So experimental validation of results provides much more insight into the problem, thereby, refining the model and hence more realistic results. More investigations may be required to further evaluate the best shape of the proposed fender and industry partner may be contacted to fabricate an optimized fender for experimental testing in the computational fluid dynamics laboratory. These results should be further validated for wind loads and other load combinations discussed. This would provide a more reliable relative displacement, acceleration, intrusion of passenger compartments values etc. which when standardized would serve as a guidelines for ensuring "Safety of Life" of those involved in vehicle crashes and development of an optimal design.

6. REFERENCES

- [1] Rafat Ali, "Finite Element Study of a Composite Material Sump Pan of an I.C. Engine", SAE Paper No.950942, 1995.
- [2] Basil Housari, Lian X. Yang, "Experimental Techniques for Strain Measurement and Validation of CAE Model", SAE Paper No.2005-01-0587, 2005.
- [3] Muniyasamy K., Govindrajan R., Jayram N., Ravi kharul, "Vibration Fatigue Analysis of Motorcycle Front Fender", SAE Paper No.2005-32-0030, 2005.
- [4] K. Bel Knani, P. Bologna, E. Duni, G.Villari, G.Armando, M Tortone, M Leghissa, S Borone, "CAE Methodologies for Virtual Prototyping of Cast Aluminum Suspension Components", SAE Paper no. 2002-01-0677, 2002.
- [5] Mohammed M Ansari, "Validation of Finite Element (FE) Model for All Radiator End Tank", SAE Paper No.2002-01-0951, 2002.

[6] "Ls-Dyna Keyword User's Manual", Livermore Software Technology Corporation.

7. AUTHOR PROFILE

	Mr. Arpit G. Mahure received the Bachelor of Engineering in Mechanical Engineering and pursuing Master Engineering in CAD-CAM from Prof. Ram Meghe Institute of Technology & Research, Badnera, Amravati, India. His research area design and analysis in CAD-CAM.
	Dr. N. A. Wankhade completed his Phd in Mechanical Engineering from Sant Gadge Baba Amravati University, Amravati, India. Currently working as a Associate Professor in Prof. Ram Meghe Institute of Technology & Research, Badnera, Amravati, India.
	Mr. C. G. Jawanjal received the Bachelor of Engineering in Mechanical Engineering and pursuing Master Engineering in CAD-CAM from Prof. Ram Meghe Institute of Technology & Research, Badnera, Amravati, India. His research area design and analysis in CAD-CAM.