International Journal of Engineering Research And Management (IJERM)
ISSN: 2349- 2058, Volume-03, Issue-11, November 2016

A Review on Design and Analysis of Front Fender of Three Wheeler Vehicle

Dr.Prof. Nitin A. Wankhade, Mr.Arpit G. Mahure

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 analysed, using linear and nonlinear finite element analyses in CAE software.**

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.

Index Terms— **Fender, Strain Gauges, Strain Indicator, CAD Modeling, CAE-Computer Aided Engineering, Hyper Mesh, Topology & Free-Size Optimization, Optistruct, Low-velocity impact, LS-DYNA solver**

I. 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.

Manuscript received Nov 24, 2016

Dr.Prof. Nitin A. Wankhade, Associate Professor, Mechanical Department, PRMIT & R Badnera, Amravati

Mr.Arpit G. Mahure, ME Student, Mechanical Department, PRMIT & R Badnera, Amravati

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.

Fig.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.

1.3.LITERATURE SURVEY

The following are the objectives of the study:

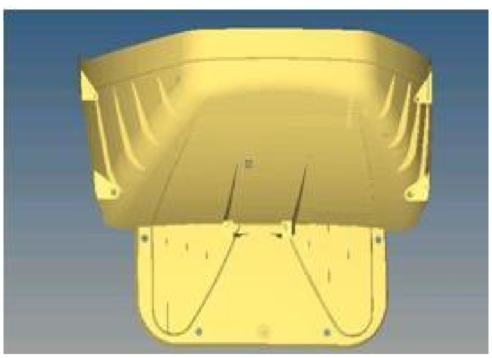
1. Literature review, collecting and gathering the information about existing three wheeler front fender mudguard in Indian market for possible design modifications.
2. To carry out finite element stress analysis of front fender of three-wheeler.
3. To carry out experimental validation of the stress 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.

II. Finite Element Modelling Approach

2.1 CAD modeling

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.



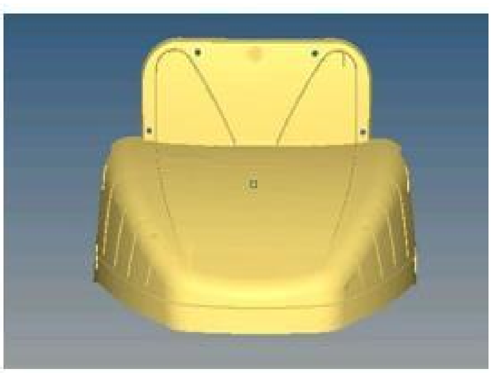


Figure 2 (i): Existing mudguard model
Figure 2 (ii): Existing mudguard model

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.

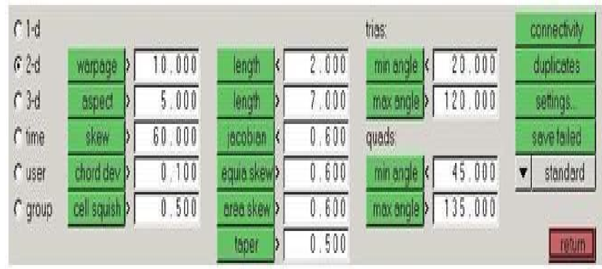


Figure 3: Mesh criteria

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).

Table 1: Mechanical properties of polypropylene.

Material properties for fender	Symbol (Unit)	Value
Young's modulus	E (MPa)	1100
Poisson's ratio	N	0.381
Density	ρ (ton/mm³)	1.56 −9
Yield stress	σy (MPa)	30-34
Ultimate stress	σt (MPa)	42-50

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

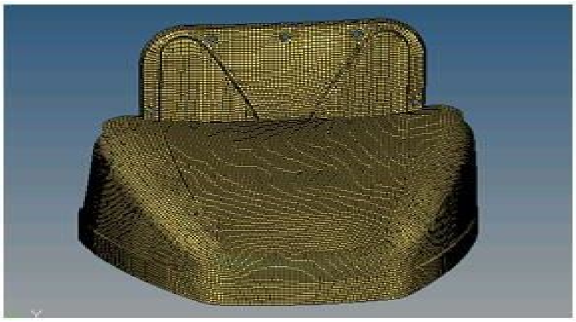


Figure 4: Meshed model of existing fender

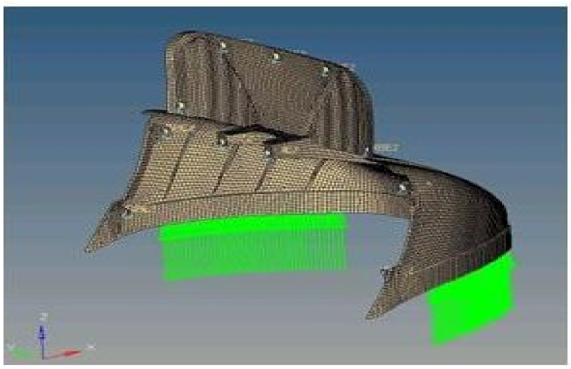


Figure 5: Applied loads & boundary condition on model Critical load condition

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.

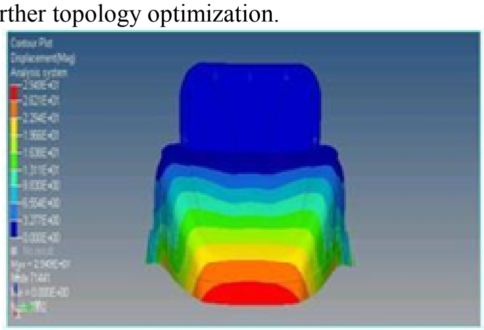


Figure 6 (i) : Displacement & stress plots of the existing model.

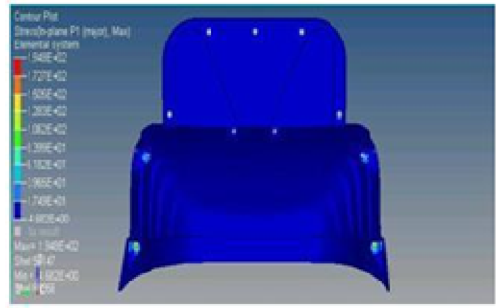


Figure 6 (ii) : Displacement & stress plots of the existing model.

III. 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.

IV. VALIDATION OF OPTIMIZED DESIGN

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

V. 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

5.1 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 were obtained for all load cases upto100kg 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.

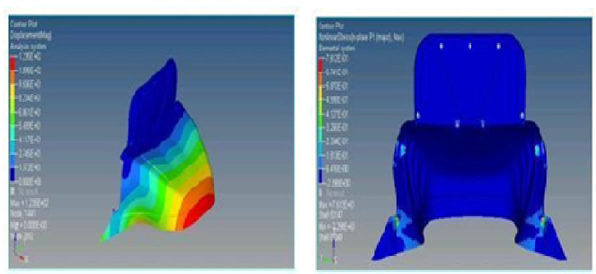


Figure 7 : Stress & displacement plots of the existing model

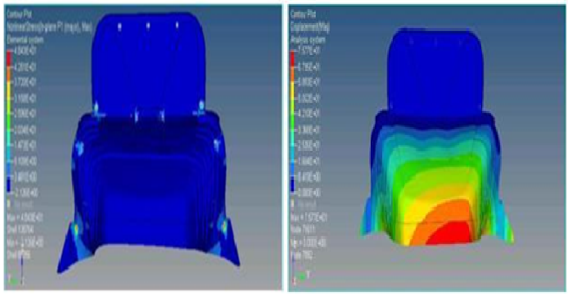


Figure 8 : Stress & displacement plots of the optimized topology model

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.

VI. CRASH SIMULATION IN LS-DYNA SOLVER:

After the mesh of the surfaces is generated, the model was prepared for simulation as explained below:

6.1.Material Model Selection & Assignment Of Material Properties:

Front mudguard materials is mainly for polypropylene (PP). LS-DYNA is having a wide range of material and equation of state model each with a unique of history variables from the material library material model named MAT_PIECEWISE_ LINEAR_PLASTICITY is used for the MUDGUARD, the material number is MAT24.

Since the RIGID PLATE has been considered rigid in their design, and no deflection has been observed .so the material of the Rigid Plate is modeled by MAT_RIGID (020).

Material properties for Rigid Plate	**Symbol (Unit)**	**Value**
Young's modulus	E (MPa)	210000
Poisson's ratio	ν	0.3
Density	ρ (ton/mm³)	7.89 −9
Velocity	V (mm/s)	2222.220

Table 2: Material Properties of Rigid plate

6.2 Initial And Boundary Conditions:

To fix the Mudguard in space, three SPC-Node sets were considered to model the fixed points of the mudguard in the areas where it is assembled on the vehicle. Initially, a set of nodes were selected for applying the boundary condition by means of SPC-Node set,

representing the area of connection. All 6 DOFs of the selected nodes in the FE model were constrained in the for mentioned SPC-Node set. For Flat Rigid Plate, according to IIHS and CMVSS standard for flat barrier impact using low-speed collision test standard, the standard of collision speed is 8 km/hr., the velocity of the rigid plate was defined as 8 km/hr. along the normal vector of contact surface's elements of the flat rigid plate via INIVEL card and the velocity was applied to the entire nodes of the rigid plate using velocity load collector.

6.3 Contact Conditions:

In this case contact between MUDGUARD and a RIGID plate are defined. Contact type used in LS-DYNA for impact is Surface to Surface (STS).The surface which is more coarsely zoned should be chosen as the master surface. When using the one-way slide surface with rigid materials, the rigid material should be chosen as the master surface. So for impact simulation mudguard surface is slave surface and flat rigid plate is master surface.

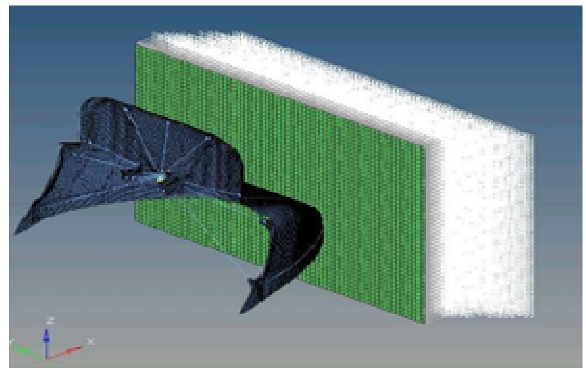


Figure 9: Crash Simulation of fender at Fixed Position with Rigid Plate Having Initial Velocity.

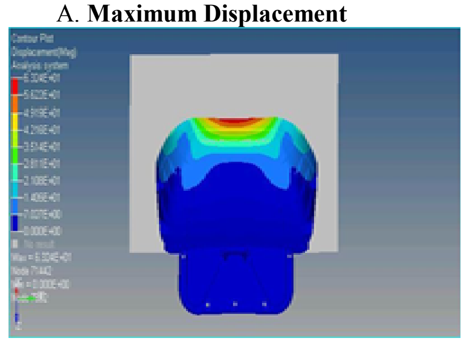
6.4 Comparison Of Simulation Results And Disscusion:

After performing low speed impact analysis of a both existing and optimized mudguard model using LS-DYNA as a solver. Results of Post-processing are obtained is reviewed using Hyper View as post processor. For stress counter plot, If the ''Von Mises stress'' exceeds the yield stress, then the material is considered to be at the plastic deformation condition. The Von Mises stress distribution may be viewed from Hyperveiw post processor, and the values can be determined along mudguard at any time of the collision. Now the impact analysis result achieved in hyperveiw & graphs in hyper graph are shown as follows,

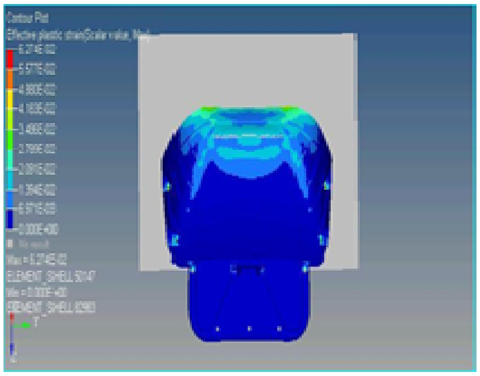
NOTE: For Good Viewing Purpose All Counter Plot are plotted in Rear View.

1) Counter Plots Existing Model :

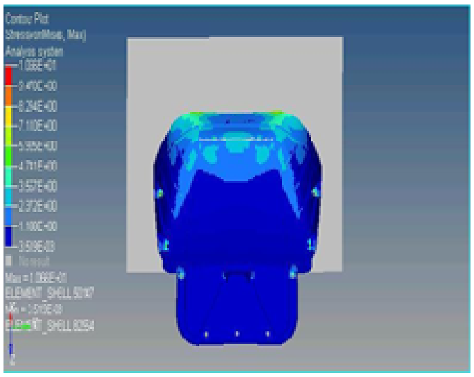
A. **Maximum Displacement**



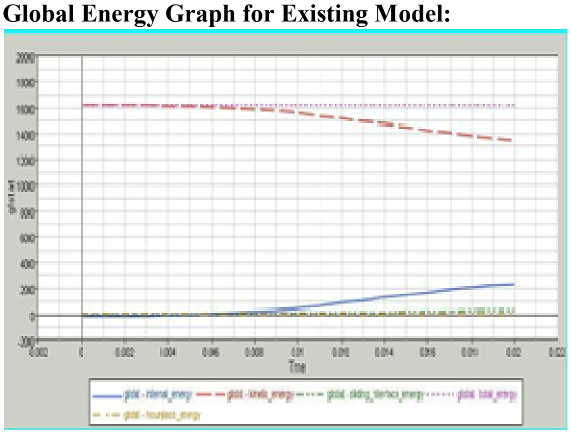
B. Maximum Effective Plastic Strain



C. Maximum Von Mises Stress



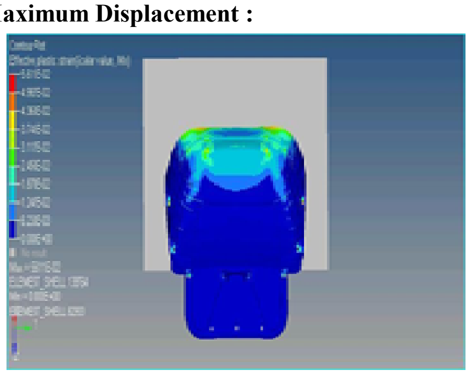
Global Energy Graph for Existing Model:



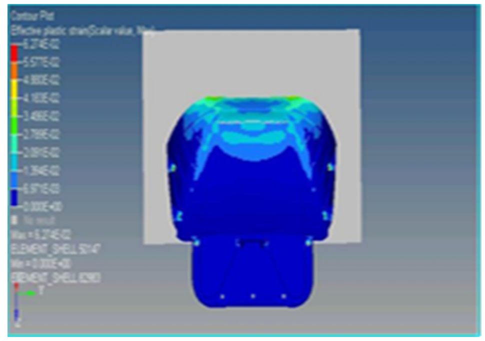
For Termination Time (20ms), Total internal energy = 2200 KJ

2) CountePlots For Optimized Model:

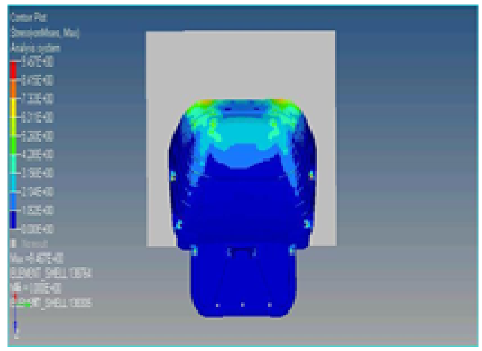
a) Maximum Displacement :



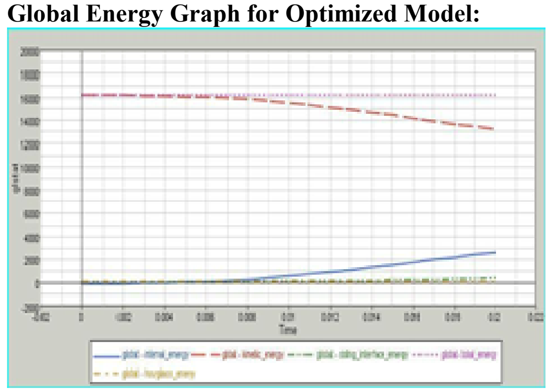
b) Maximum Effective Plastic Strain



c) Maximum Von Mises Stress



Global Energy Graph for Optimized Model:



For Termination Time (20ms), Total internal energy = 2800 KJ.

7. Result Tables

Table 1: Result Table for Optimization

Model	Weight (kg)	Displacement (mm)	Max Principal Stress (MPa)
Existing	1.48648	123.5	76.12
Optimized	1.4710	75.77	48.43

Table 2: Result Table for Low Velocity Impact Analysis

Model	Displacement (mm)	Max Effective Plastic Strain	Max Vonmises Stress (MPa)
Existing	63.24	0.06274	10.66
Optimized	67.88	0.05611	9.467

SUMMARY AND CONCLUSIONS

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

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.

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.

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.

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.

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.

REFERENCES

1. Rafat Ali, “Finite Element Study of a Composite Material Sump Pan of an I.C. Engine”, SAE Paper No.950942, 1995.
2. Muniyasamy K., Govindrajan R., Jayram N., Ravi kharul, “Vibration Fatigue Analysis of Motorcycle Front Fender” , SAE Paper No.2005-32-0030, 2005.
3. 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.
4. Mohammed M Ansari, “Validation of Finite Element (FE) Model for All Radiator End Tank”, SAE Paper No.2002-01-0951, 2002.
5. Bendsoe, M. P. (1995). Optimization of Structural Topology, Shape, and Material. Berlin: Springer. .
6. A.V. Kumar, D.C. Gossard, (1996) “Synthesis of optimal shape and topology of structures”, J. Mech Des, 118, 68-74.