DESIGN ANDDEVELOPMENT OF AUTOMATIC PNEUMATIC BUMPER

Mr.Nitin S. Adwani[1], Dr.B.M.Shinde[2]

[1]Scholar, Dept of Mechanical Engg, Dhole Patil College of Engineering, Pune, Maharashtra, India
[2]Professor, Dept of Mechanical Engg, Dhole Patil College of Engineering, Pune, Maharashtra, India

---

Abstract -

The development of an automatic pneumatic bumper with FEA (Finite Element Analysis) simulation is the focus of this study. Bumpers play a critical role in absorbing and dissipating impact energy during collisions, thereby protecting the vehicle and its occupants. The objective of this research is to design and analyze a bumper system that can automatically adjust its stiffness and damping characteristics based on the impact force. The design process involves the selection of suitable materials, consideration of geometrical parameters, and integration of pneumatic components. The proposed bumper system utilizes pneumatic actuators to modulate the stiffness and damping properties in real-time. This allows for adaptive response to varying impact forces, enhancing the overall crashworthiness and safety of the vehicle.

Finite Element Analysis (FEA) simulation is employed to evaluate the structural performance of the bumper system. The simulation models are developed using appropriate material properties, boundary conditions, and impact scenarios. The FEA analysis provides valuable insights into the stress distribution, deformation, and energy absorption capabilities of the bumper design. The results of the FEA simulation are used to optimize the bumper system design, ensuring adequate strength and durability under different impact conditions. The simulation outcomes also aid in identifying potential areas of improvement, allowing for iterative refinement of the design.

***Keywords*—**Pneumatics Bumper,Fea, Structural performance, , Stress distribution, , Vehicle safety

1. INTRODUCTION

The concept of an automatic pneumatic bumper involves the integration of pneumatic actuators within the bumper system. These actuators allow for real-time adjustment of the bumper's stiffness and damping properties based on the magnitude of the impact force. By adapting to varying impact forces, the bumper system can effectively mitigate collision energy and enhance crashworthiness. Vehicle safety has always been a paramount concern in the automotive industry. Bumpers play a crucial role in protecting vehicles and their occupants during collisions by absorbing and dissipating impact energy. Traditional bumpers have fixed stiffness and damping characteristics, which may not provide optimal performance in various impact scenarios. To address this limitation, the development of an automatic pneumatic bumper with Finite Element Analysis (FEA) simulation has gained significant attention.

The design process for the automatic pneumatic bumper encompasses several key considerations. The selection of suitable materials with appropriate mechanical properties is crucial to ensure adequate strength and durability. Geometrical parameters, such as bumper shape and size, are also important factors that influence the overall performance of the system. Integration of pneumatic components and control systems enables the seamless adjustment of stiffness and damping characteristics.A.Pnuematic Bumper Requirements

2 PROBLEM DEFINITION

The design and development of an automatic pneumatic bumper system with Finite Element Analysis (FEA) simulation face several challenges and objectives to address. The primary problem can be defined as follows:

1. **Inadequate Adaptability**: Traditional bumper systems have fixed stiffness and damping characteristics, which may not provide optimal performance in varying impact scenarios. A lack of adaptability limits the ability of the bumper to effectively absorb and dissipate collision energy, potentially leading to increased damage to the vehicle and increased risk to occupants.

2. **Limited Energy Absorption**: Bumper systems need to efficiently absorb and dissipate impact energy to minimize damage to the vehicle and reduce the risk of injury to occupants. Traditional bumpers may have limited energy absorption capabilities, leading to greater structural damage and compromised safety.

3. **Suboptimal Design:** The design of bumper systems needs to consider various factors such as material selection, geometrical parameters, and integration of pneumatic components. A suboptimal design can result in inadequate strength, poor durability, and compromised crashworthiness.

4. **Lack of Real-time Adaptation:** Bumpers should be able to adapt their stiffness and damping properties in real-time based on the magnitude of the impact force. The lack of real-time adaptation limits the effectiveness of the bumper system in responding to dynamic impact conditions.

5. **Complex Structural Analysis**: Assessing the structural performance of the bumper system under various impact scenarios is essential for ensuring its effectiveness and safety. However, conducting physical testing can be costly and time-consuming. Therefore, there is a need for a reliable

and efficient method to evaluate the structural performance through simulation.

2.1 Objectives:

- To improve the adaptability of the bumper system, ensuring optimal energy absorption and dissipation during collisions.
- Design a bumper system with enhanced energy absorption capabilities to minimize vehicle damage and reduce the risk of injury to occupants
- To achieve a balanced design that ensures sufficient strength, durability, and crashworthiness.
- To evaluate the structural behavior, stress distribution, deformation patterns, and energy absorption capabilities of the bumper design.
- Validate the performance of the automatic pneumatic bumper system through experimental testing and comparison with simulation results.
- The ultimate objective is to enhance vehicle safety by developing an automatic pneumatic bumper system that effectively absorbs impact energy, minimizes damage, and protects vehicle occupants during collisions.

2.2 Methodology:

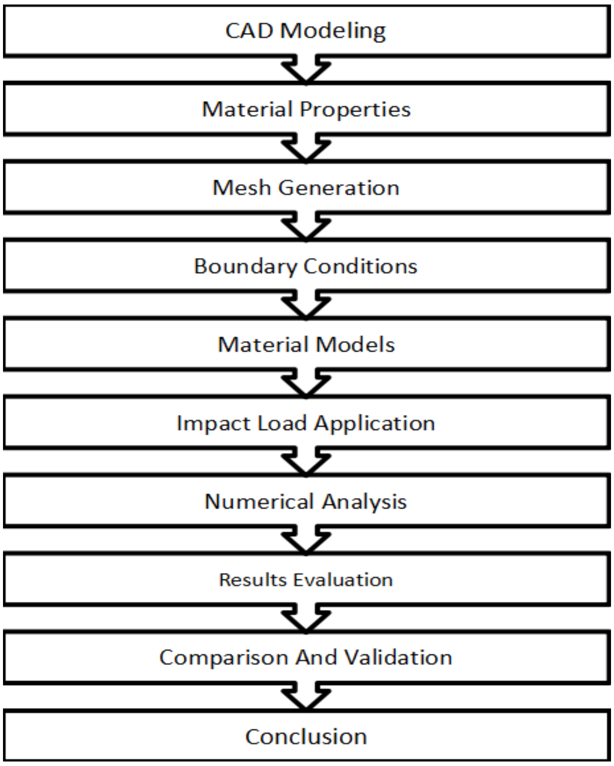

Fig 2.1: Methodology

3. FINITE ELEMENT ANALYSIS

The Finite Element Method is a numerical technique employed by engineers to accurately address the mechanics of solid materials. It is a mathematical modeling approach that involves dividing a continuous domain into smaller elements, known as finite elements. These elements are interconnected at nodes to simulate the behavior of the system under both two-dimensional and three-dimensional movements.

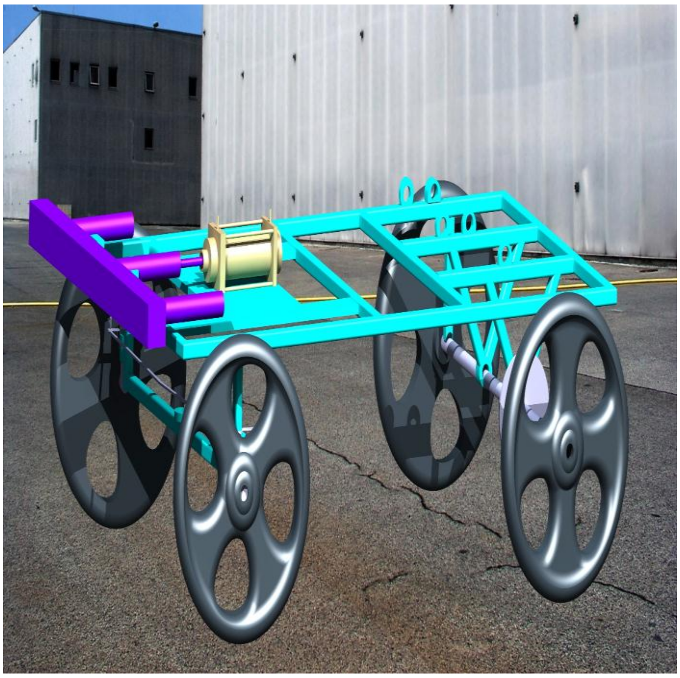

Fig 3.1: Automatic Pneumatic Bumper 3D model

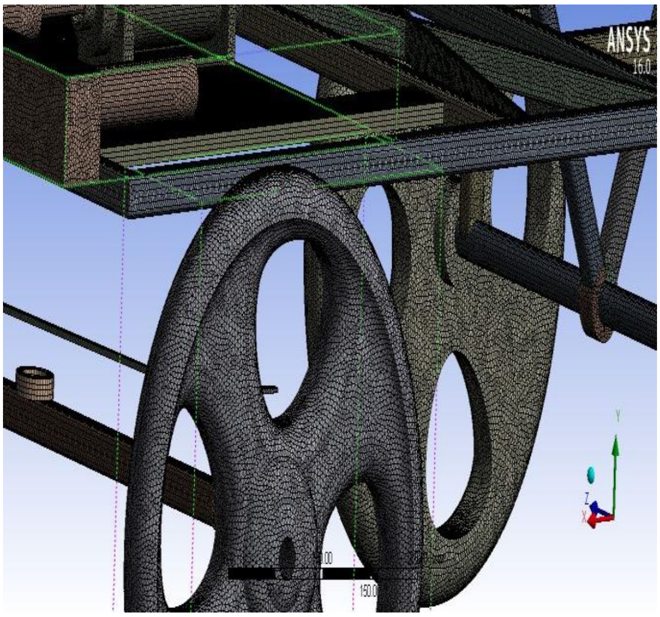

Fig 3.2: Automatic Pneumatic Bumper Meshing Models

3.1 Boundary Conditions Force

Impact Force calculated is 151 N

We will take 1000N.

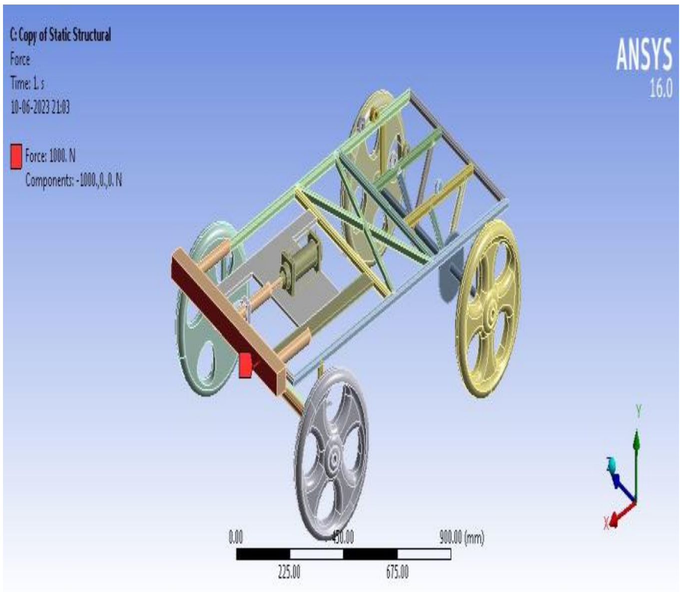

Fig 3.3: Automatic Pneumatic Bumper Boundary Conditions

3.2. Post-processing of Structural Analysis

The total deformation contour plot is as shown in below Figure.

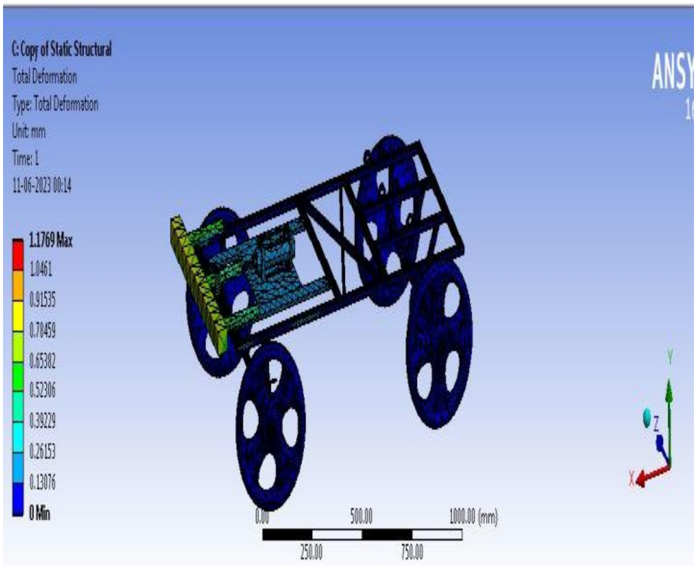

Fig 3.4: Total deformation contour plot for Pnuematic Bumpert Model

The maximum deformation shown by the Automatic Pnuematic Bumper model is 1.17 mm.

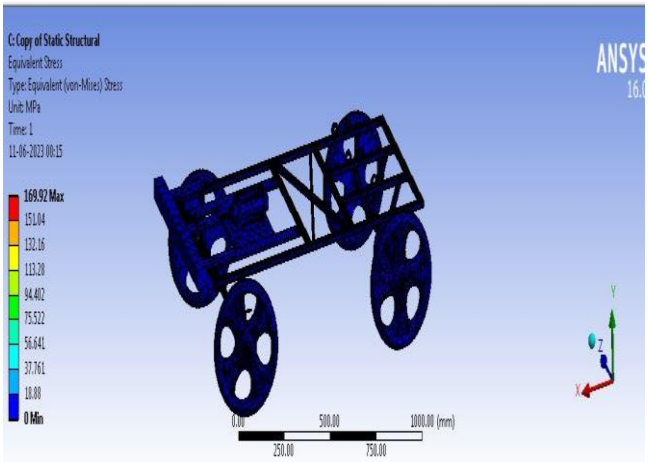

Fig 3.5: Von-Mises stress contour plot for Automatic Pnuematic Bumper Model

The Von-Mises stress observed in the model is 169 MPa.

3.3. Summary (From Finite Element Analysis Vs Experimental)

The analysis reveals that the Automatic Pneumatic Bumper experiences maximum stresses that align precisely with the failure area. The evaluation includes determining the overall deformation and equivalent stress values for both the initial baseline design and subsequent modified designs of the Pneumatic Bumper. The following table illustrates a comparison of different designs of the Pneumatic Bumper in Finite Element Analysis (FEA)..

Table 3.1 : FEA Vs Experimental Results

Test	FEA For Automatic Pnuematic Bumper	Experimental for Automatic Pnuematic Bumper	% Error
Deformation (mm)	1.17 mm	1.21 mm	2.23%
Stress acting on bumper	σ = 169 MPa	σ = 173 MPa	2.48%

4. CONCLUSION

A comparative study was conducted on different designs of the Electric Vehicle Automatic Pneumatic Bumper, leading to the following conclusions:

- The modified Finite Element Analysis (FEA) design of the Pneumatic Bumper exhibits a minimum deformation of 1.17 mm, which is in close agreement with the experimental value of 1.21 mm.

- The Von-Mises stress in the modified design of the Pneumatic Bumper is also minimized at 169 MPa, compared to the experimental value of 173 MPa.

- Therefore, based on design and manufacturing considerations, the modified design of the Automatic Pneumatic Bumper is deemed the most suitable and feasible choice for the current application.

REFERENCES

1. Pradeep Kumar Uddandapu, "Impact Analysis on Car Bumper by varying speeds using Materials ABS Plastic and Poly Ether Imide by Finite Element Analysis software Solid works," International Journal of Modern Engineering Research, Vol.3, Issue.1, Jan-Feb. 2013, pp-391-395.

2. Alen John, Nidhi M.B, "Modelling and Analysis of Automotive Bumper Used for a Low Passenger Vehicle," International Journal of Engineering Trends and Technology, Volume 15, Number 7, Sep 2014.

3. Yu Zhu et al., "Frontal structure improvement on car based on rear impact test," International Journal of Security and Its Applications, Vol.7, No.6, 2013, pp.147-154.

4. Qihui Li, Jikuang Yang, "Study of vehicle front structure crash worthiness on pole impact with different position," Fifth Conference on Measuring Technology and Mechatronics Automation, 2013.

5. Wenhao Mu, Jikuang Yang, "Optimization design of frontal bumper system for Pedestrian lower leg protection and low speed impact requirement."

6. Q.H.Ma et al., "Research on the Crash Safety of the Car Bumper based on different standards," International Journal of Security and Its Applications, Vol.7, No.6, 2013, pp.147-154.

7. Jeyanthi and J Janci Rani, "High velocity Impact analysis of the thermoplastic bumper in automobiles," International Journal of Security and Its Applications, Vol.7, No.6, 2013, pp.147-154.

8. Emil Evina, Miroslav, "Comparison of deformation properties of steel sheets for car body parts," ELSEVIER, Procedia Engineering 48, 2012, pp.115–122.

9. S.Popprath et al., "Experimental modal analysis of a scaled car body for metro vehicle," International Congress on Sound and Vibration, 2006.

10. Miss Priya Dongare, Prof. Dr. Suhas Deshmukh, "Static and Modal Analysis of composite Drive shaft," International Journal of Engineering Research & Technology, Vol. 1 Issue 10, December-2012.

11. Pawankumar.s, "Modal and static analysis of rear car guard for SUV," International Journal of Engineering Research & Technology, Vol. 3 Issue 7, July 2014.

12. Mr. Nithin S. Motgi, "Impact analysis of front bumper," International Journal of Engineering Trends and Technology, Volume 6 Number 5, Dec 2013.