

Experimental Testing and CFD Simulation of Existing Automotive Muffler

¹Nilesh Kurangal, ²Dr. Sangram Patil

¹PG Scholar, ²Assistant Professor

^{1,2}Department of Mechanical Engineering

^{1,2}Sinhgad Academy of Engineering, Pune, India

Abstract— A muffler is a device used to reduce the noise and vibration of gas emitted by internal combustion engine. So, it becomes necessary equipment in automobile to have a proper emission of gases to surrounding. Due to improper design some muffler decreases the mass flow rate due to which there is increase in fuel consumption, large pressure drop across its cross section and higher knock sensitivity. Thus, recent research is being performed to increase the mass flow rate and decrease the pressure drop. In present project existing 4-wheeler vehicle muffler is experimentally tested to determine the mass flow rate, velocity and temperature and also simulation is performed in CFD to validate the experimental results.

Index Terms— Muffler, CFD Simulation, Mesh, Ansys, Deformation, Modal Analysis, Perforation

1. INTRODUCTION

Mufflers are installed within the exhaust system of most internal combustion engines. The muffler is engineered as an acoustic device to reduce the loudness of the sound pressure created by the engine by acoustic quieting. The noise of the burning-hot exhaust gas exiting the engine at high speed is abated by a series of passages and chambers lined with roving fiberglass insulation and resonating chambers harmonically tuned to cause destructive interference, wherein opposite sound waves cancel each other out. The function of a Muffler or a Silencer in Vehicle is to cool the exhaust gases by expansion through it and to reduce the noise of outgoing gases. The exhaust gases must be discharged into the atmosphere with minimum restriction. The restriction in flow of exhaust gases causes back pressure. [5][6][7]

2. PROBLEM STATEMENT

Comparison of experimentally performed and CFD simulation of the existing 4-wheeler muffler component to increase the mass flow rate as well as decrease the pressure drop across the cross-sectional area.

3. OBJECTIVE

1. Experimentally determining the mass flow rate, velocity and temperature of existing 4-wheeler vehicle muffler.
2. To increase the mass flow rate for low pressure drop across its cross-sectional area.
3. Validation of experimental results with CFD simulation parameters (velocity, temperature, mass flow rate).

4. SCOPE

1. CFD simulation for velocity, mass flow rate and temperature profile.
2. The similar study can be performed for different mufflers to validate experimental results.

5. METHODOLOGY

Step 1: - started the work of this project with literature survey. Gathered many research papers which are relevant to this topic. After going through these papers, we learnt about muffler.

Step 2: - After that the selection of muffler which are required for our project are decided.

Step 3: - After deciding the components, the 3D Model and drafting will be done with the help of CATIA software.

Step 4: - Computational Fluid Dynamics (CFD) simulations of muffler will be done with the help of ANSYS Fluent software.

Step 5: - The manufacturing of optimized model will be done, after that experimental reading are not down.

Step 6: - Comparative analysis between the experimental & CFD result & then the result & conclusion will be drawn.

5.1 Design of the System

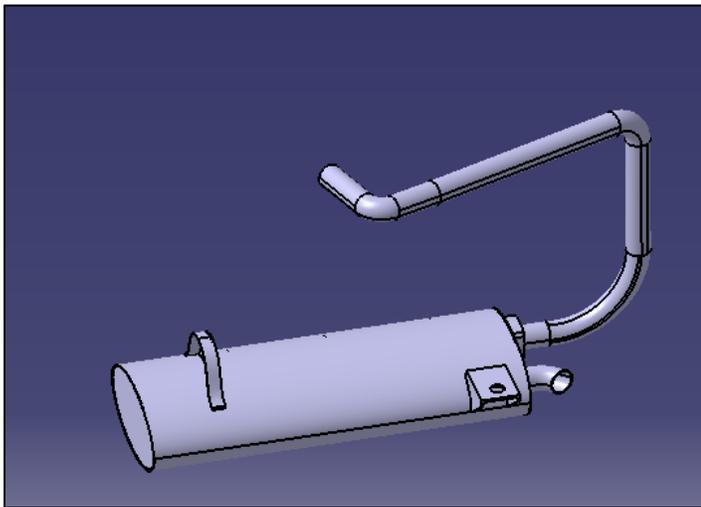


Figure 1: Catia Model of the Muffler

Computer-aided design (CAD) is the use of computers to aid in the creation, modification, analysis, or optimization of a design. CAD software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing. CAD output is often in the form of electronic files for print, machining, or other manufacturing operations.

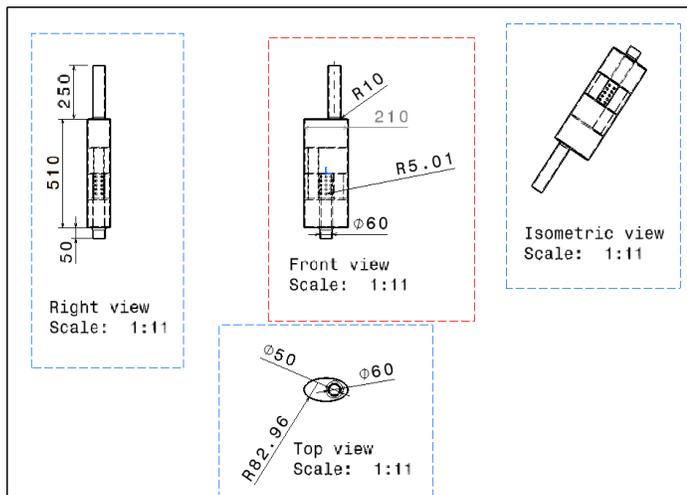


Figure 2: Drafting of the Muffler

5.2 Analysis

The finite element method (FEM) is a numerical method for solving problems of engineering and mathematical physics. The method yields approximate values of the unknowns at discrete number of points over the domain. To solve the problem, it subdivides a large problem into smaller,

simpler parts that are called finite elements. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem. FEM then uses variation from the calculus of variations to approximate a solution by minimizing an associated error function. Studying or analyzing a phenomenon with FEM is often referred to as finite element analysis (FEA).

5.3 Computational Flow Dynamics

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows.

CFD is now recognized to be a part of the computer-aided engineering (CAE) spectrum of tools used extensively today in all industries, and its approach to modelling fluid flow phenomena allows equipment designers and technical analysts to have the power of a virtual wind tunnel on their desktop computer.

5.4 Geometry

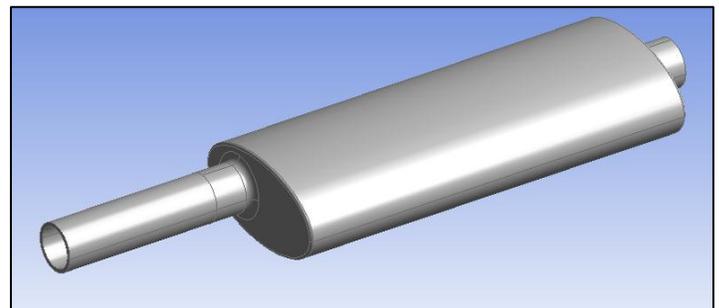


Figure 3: Geometry imported in ANSYS

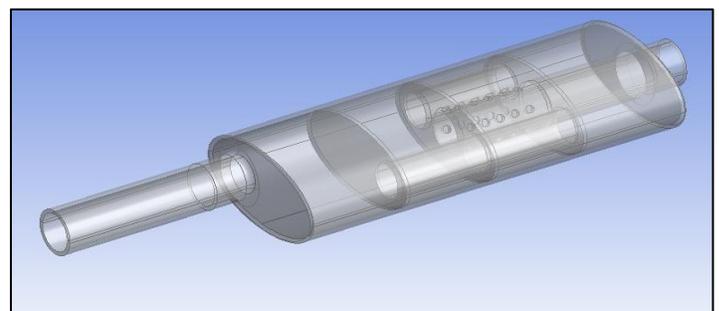


Figure 4: Volume Extracted in ANSYS

5.5 Mesh

ANSYS Meshing is a general-purpose, intelligent, automated high-performance product. It produces the most appropriate mesh for accurate, efficient Multiphysics solutions. A mesh well suited for a specific analysis can be generated with a single mouse click for all parts in a model. Full controls over the options used to generate the mesh are available for

the expert user who wants to fine-tune it. The power of parallel processing is automatically used to reduce the time you have to wait for mesh generation.

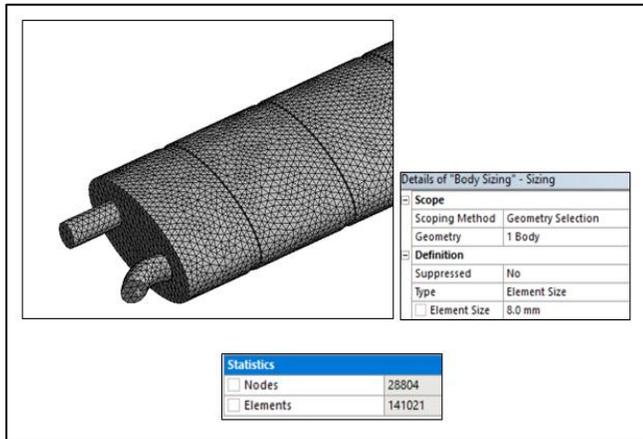
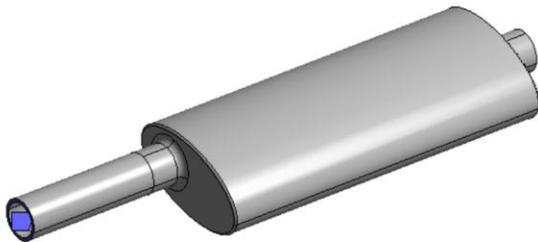


Figure 5: Details of the Meshing

6. MODAL ANALYSIS

F: Modal
Modal
Frequency: N/A
Fixed Support



F: Modal
Modal
Frequency: N/A
Fixed Support



Figure 6: Fixed Support

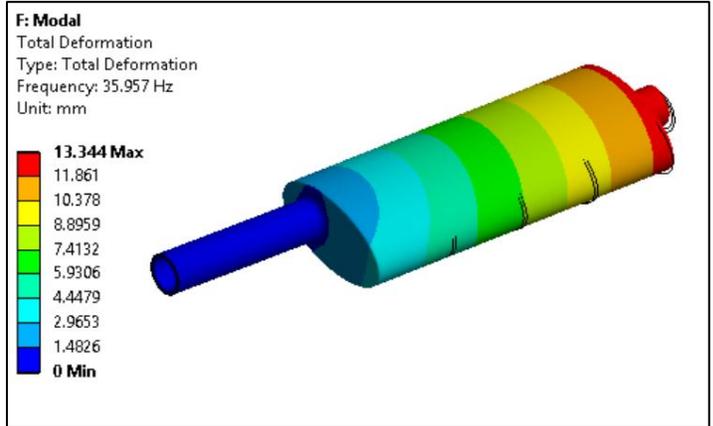


Figure 7: Natural frequency of mode 1 – 35.957 Hz

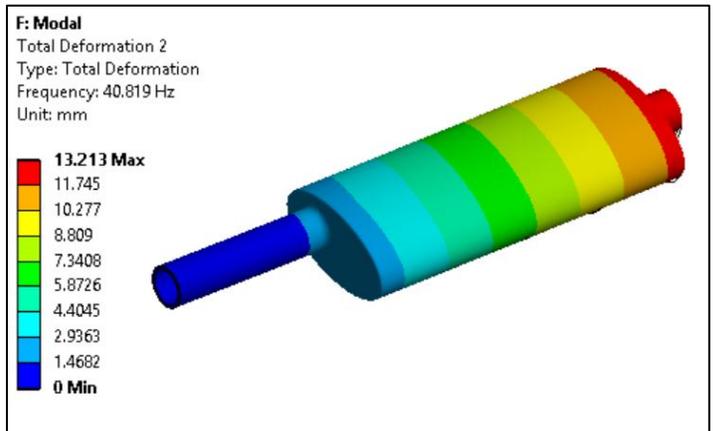


Figure 8: Natural Frequency of mode 2 – 40.819 Hz

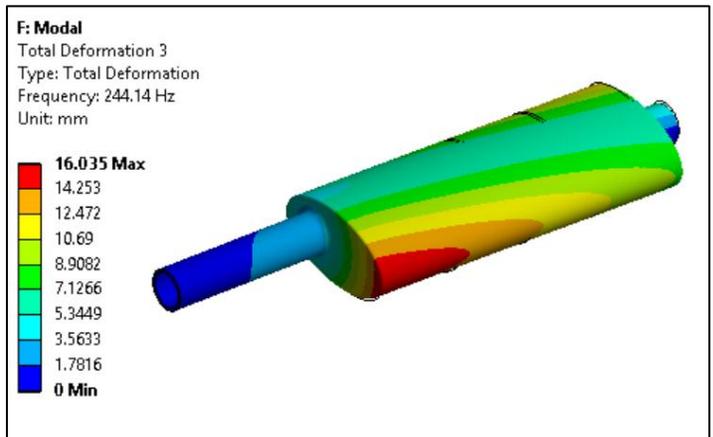


Figure 9: Natural Frequency of mode 3 – 244.14 Hz

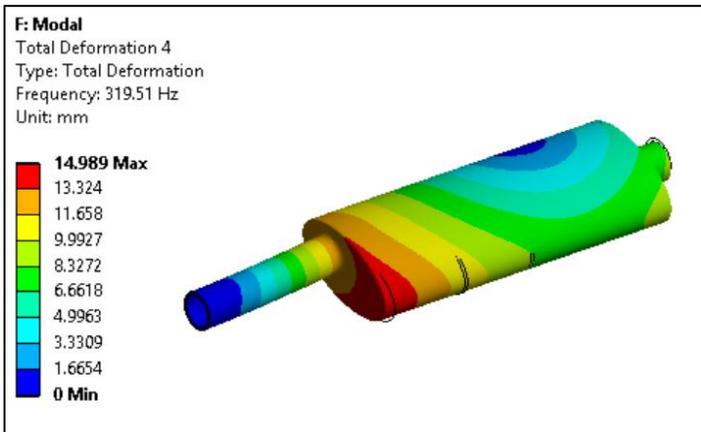


Figure 10: Natural Frequency of mode 4 – 319.51 Hz

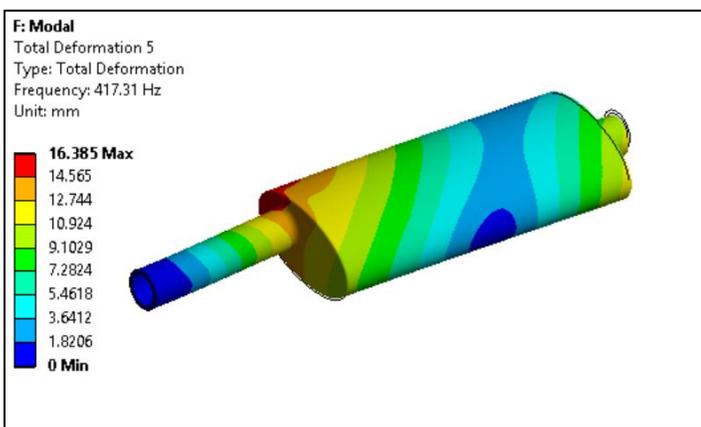


Figure 11: Natural Frequency of mode 5 – 417.31 Hz

Tabular Data		
	Mode	Frequency [Hz]
1	1.	35.957
2	2.	40.819
3	3.	244.14
4	4.	319.51
5	5.	417.31
6	6.	614.46

Table 1: Tabular representation of natural frequency w.r.t. mode shape

REFERENCES

- [1] CFD Topology and Shape Optimization of Ford Applications using Tosca Fluid by Dr. Anselm Hopf, Ford Motor Company, Aachen, Germany.
- [2] Integration of Exhaust Manifold with Engine Cylinder Head towards Size and Weight Reduction by M.A. Neshan, A. Keshavarz, K. Khosravi.
- [3] Exhaust System Muffler Volume Optimization of Light Commercial Vehicle Using CFD Simulation by C.P. Om Ariara Guhana, G. Arthanareeswaran, K.N. Varadarajan, S. Krishnan.
- [4] Methods for Evaluating In-duct Noise Attenuation Performance in a Muffler Design Problem by Jong Kyeom Lee, Kee Seung Oh, and Jin Woo Lee.
- [5] A Nonlinear Quasi-3D Approach for the Modeling of Mufflers with Perforated Elements and Sound-Absorbing Material by G. Montenegro, A. Della Torre, A. Onorati, and R. Fairbrother.
- [6] Modification of Muffler Design to Increase Exit Velocity by Nagisetty Lokhesh Kumar* and K. Veladri.
- [7] Enhancement of Engine Performance through Analyzing Internal Part Design on Modenas CT115s Exhaust Using CFD Simulation by Efi N E, Shahriman A B, Rojan M A, I Zunaidi1, Z M Razlan, W K Wan, M S M Hashim, N S Kamarrudin, A Harun, I Ibrahim and Azizul Aziz I.
- [8] Commercial Vehicles Muffler Volume Optimization Using CFD Simulation by Ashok Patidar, Shivdayal Prasad, Umashanker Gupta, and Mohan Subbarao.

7. CONCLUSIONS

- In present investigation existing muffler CFD simulation have been performed to determine exiting pressure drop, temperature, wall heat transfer coefficient.
- It is observed from contour that modifying the existing muffler with perforation of diameter 4 mm pressure drop is observed around 177 mpa compared to existing 350 Mpa.