Experimental testing and CFD simulation to enhance the performance of existing 3-wheeler muffler

Chetana Vitthal Ahire
Heat Power Engineering
Matoshri College of Engineering & Research Centre
Maharashtra,India

Dr. D. D. Palande
Heat Power Engineering
Matoshri College of Engineering & Research Centre
Maharashtra,India

Prof. Chetana Nitish Purkar
Heat Power Engineering
Guru Gobind Singh Polytechnic,Nashik
Maharashtra,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 drops 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 using the concept of perforation in CFD simulation in ANSYS software. In present project existing 3-wheeler vehicle muffler and modified is experimentally tested to determine the mass flow rate, velocity and temperature and also simulation is performed in CFD to validate the experimental results.

Keyword: *ANSYS, CATIA, CFD, Muffler, Perforation*

---

1. INTRODUCTION

In internal combustion engines as a necessary component for exhaust section mufflers are installed. It is an acoustic device to reduce the loudness of the sound pressure created by burning-hot exhaust gas exiting the engine at high speed and also by a series of passages and chambers lined and resonating chambers harmonically tuned to cause destructive interference, where in opposite sound waves cancel each other out to produce loud noise. Due to this phenomenon back, pressure is created in muffler which is an unavoidable side effect of noise reduction which directly declines the engine efficiency. Today, finite element and other numerical models are used for predicting the streamline flow, pressure contour, kinetic energy of exhaust particles in CFD simulation of components related fluid namely air and gas. Convenient but unrealistic values are often chosen because many properties of the physical structure are very difficult to measure.

Figure1. Existing 3-wheeler muffler

Due to use of CFD software it becomes easy to visualize the results along the interior design with modified design systematically. Damping factors or loss factors are required in FE modelling to predict deflections and stresses caused by dynamic loads. Finite element modellers often have to pick damping factors at random because damping is impossible to obtain accurately from material properties and geometry alone. To reduce the vibration and temperature of hot gases coming out of engine muffler are used as a very important part of exhaust system. So, it becomes a necessary parameter to discharge this exhaust gases with minimum restriction to environment. While, discharging back pressure is created so due to which it consumes more fuel and decline in efficiency of engine is observed.

2. LITERATURE REVIEW

C.P. OM ARIARA GUHAN et.al [1], In this journal it presented the weight reduction of existing exhaust system by optimizing muffler volume with the help of 3D design tool CATIA V5 and CFD tool ANSYS CFX software. For any design verification in muffler following are the parameters to be considered namely noise level, back pressure, exhaust gas temperature sound quality etc. It is concluded that existing muffler volume has been reduced by 15 % and weight reduced by 2 %. In optimized design of muffler pressure drop and noise level parameter is observed. Finally, it is observed that noise output from the engine through muffler, quickly and efficiently is obtained. Final design concept of muffler with increased no of holes is mentioned as down sized muffler in upcoming discussion are showing the 3D geometry and cross-sectional view of existing muffler.

Ashok Patidar et.al [2], in present paper it predicted weight optimization parameter in exhaust system where muffler volume is optimized using Computational Fluid Dynamics (CFD). It is concluded that change in number of holes in inlet pipe and baffle position leads to change in backpressure value. In present research while performing CFD simulation distance between baffle were increased from 220 to 275mm and also number of holes were increased from 80 to 120. There is 6.4% less pressure drops in downsized muffler with respect to existing design. So, in quality of vehicle noise and vibration no deterioration is observed. The overall volume reduction in muffler is 14.1 % leading to reduction in muffler weight by 2% and yielding cost benefits.

Dr. Anselm Hopf et.al [3] In this paper, several Ford applications have been presented showing the optimization of three-dimensional flow problems of engine parts. The proposed solution is a combined optimization strategy using CFD topology optimization with Tosca Fluid and a following shape optimization with the Adjoint Solver. Initially, CFD engineer generates an initial design of a new CFD optimized part and give it to the designer. So, to perform a proper simulation an optimized design solution is formulated by a designer. To extend the parameter set of parametric optimizations new surfaces of the parts serves as an idea generator of new flow shapes. It is concluded that this new optimization workflow is highly efficient, reduces development time, improves result quality, and may reduce the number of expensive and time-consuming test-rig measurements.

Efi N E, Shahriman et.al [4], in this paper it predicts to analyze the internal part geometry of exhaust muffler in order to increase performance by using ANSYS software. Similarly, with the help of ANSYS software calculate the flow rate and the velocity of exhaust muffler. It is observed from analysis result that decreasing the size and number of holes has a major influence on performance of muffler exhaust. However huge variation in result is observed when there is increase in size of hole in muffler. For the mass flow rate result at improvement design B show that the mass flow is 0.15644 kg/s which is the highest compared to the others design. It is observed from analysis result that for existing design mass flow rate as per 0.0032 m/s is much lower than improved design B. Although, for design B magnitude of velocity is observed around $1.876e^2$ m/s than for existing design to $6.192e^2$ m/s. From the simulation result it shows that smaller the size of holes gives higher performance in term of mass flow rate and velocity. So, it is observed from simulation that the lower the value of diameter for perforated tube will contribute to higher performance of exhaust.

3. PROBLEM STATEMENT

In present study to improve the existing 3-wheeler muffler efficiency perforation technique is being used. In existing muffler due to continuous straight pipe, it led to accumulate more pressure of exhaust gas at last chamber due to which uneven noise and vibration lead to failure of internal structure. Due to this present study involves perforation of diameter 4 mm with 20 numbers of holes to evaluate the effect by CFD simulation and experimental results.

OBJECTIVES

1. Design of 3-wheeler muffler in CATIA software.
2. To study the effect of perforation on existing 3- wheeler muffler using CFD simulation in ANSYS software.
3. Experimentally determining the mass flow rate, velocity and temperature of existing 3-wheeler vehicle muffler.
4. To increase the mass flow rate for low pressure, drop across its cross-sectional area.
5. Validation of experimental results with CFD simulation parameters (velocity, temperature, mass flow rate).

4. METHODOLOGY

Step 1:- Initially research paper is studied to find out research gap for project then necessary parameters are studied in detail. After going through these papers, we learnt about muffler.
Step2:- Research gap is studied to understand new objectives for project.
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

CAD MODEL

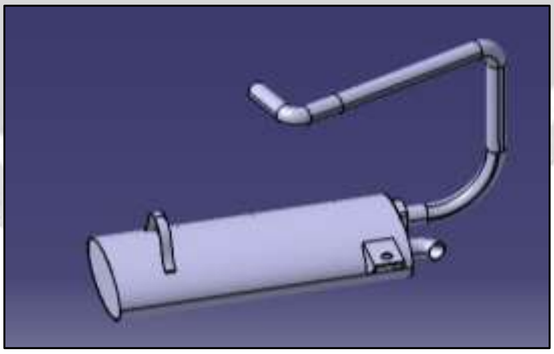


Figure 2. CATIA model of muffler

EXISTING 3-WHEELER MUFFLER

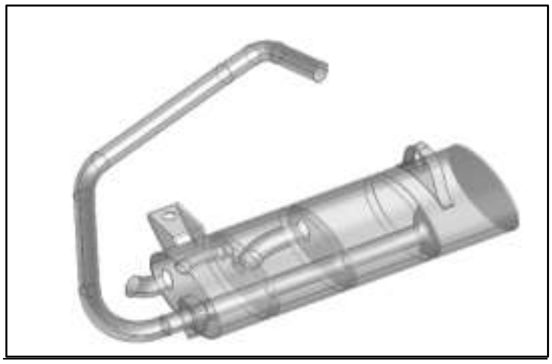


Figure 3. CAD model imported in ANSYS

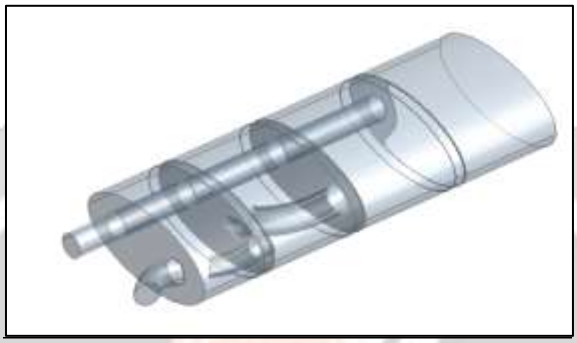
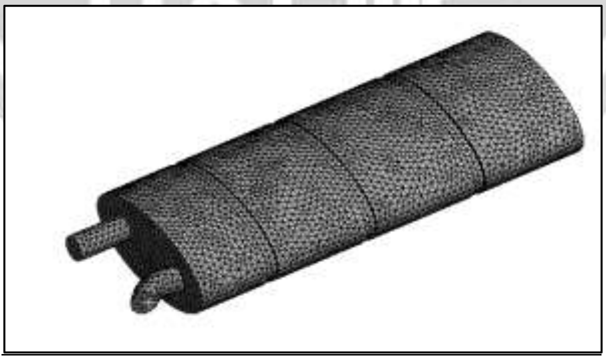


Figure 4 . Volume extracted geometry for CFD simulation

4.1 Mesh

In ANSYS meshing is performed as similar to discretization process in FEA procedure in which it breaks whole components in small elements and nodes. So, in analysis boundary condition equation are solved at this elements and nodes. ANSYS Meshing may be a all-purpose, intelligent, automated high-performance product. It produces the foremost acceptable mesh for correct, economical metaphysics solutions. A mesh well matched for a selected analysis may be generated with one click for all elements in a very model. Full controls over the options accustomed generate the mesh are accessible for the skilled user who needs to fine-tune it. the ability of parallel processing is automatically accustomed reduce the time you have got to wait for mesh generation.



Statistics	
Nodes	29149
Elements	147199

Figure 5 Details of meshing of muffler

4.2 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.

4.3 Steps in CFD procedure

1. In general box gravity is defined in y direction as - 9.81 m/s^2 (negative sign for downward direction).
2. In model's energy is kept on with viscous model as k- epsilon, realizable, scalable wall function.
3. Inlet velocity of gas is defined as 220 m/s as mass flow rate is defined as 0.05 kg/s to convert mass flow rate into velocity following calculation are performed.
4. Mass flow rate = discharge x density of gas, discharge = area x velocity, area of discharge is known as 451 mm^2 (from geometry uploaded in ANSYS), velocity = 220 m/s with gas temperature as 400 degree Celsius (standard references from paper.
5. In solution method second order upwind are selected instead of first order upwind to improve the accuracy of result.
6. Hybrid initialization is selected in solution initialization.
7. Initial 1000 number of iterations is selected.

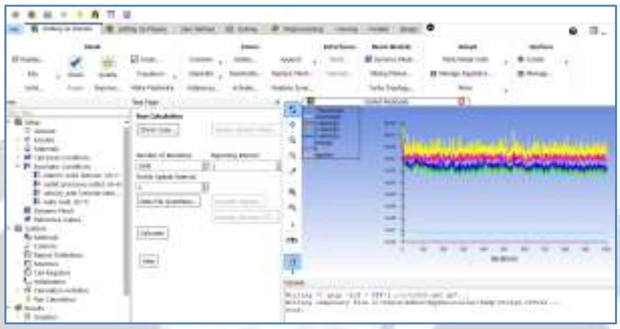


Figure 6. CFD simulation convergence graph

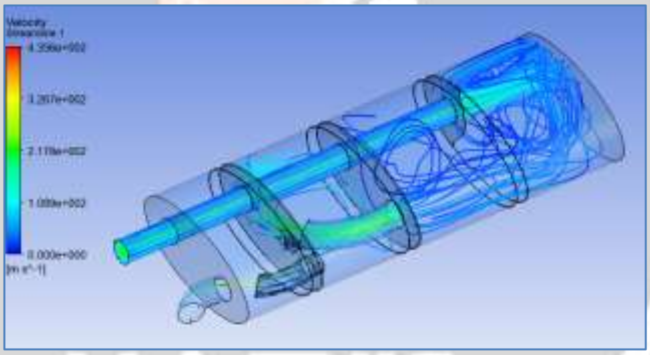


Figure 7. Stream line flow of exhaust gas in existing muffler

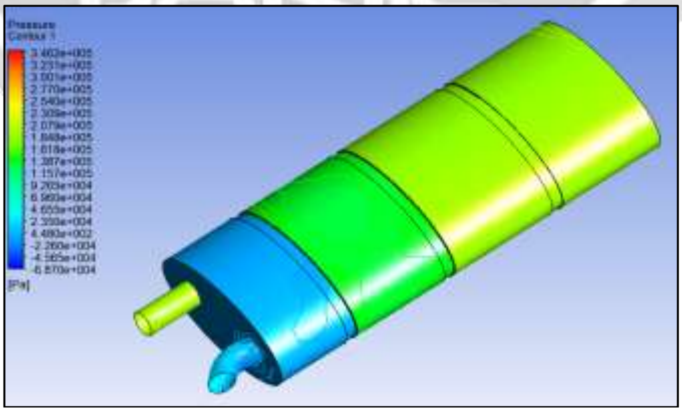


Figure 8 Pressure contour of existing muffler

It is observed that in existing 3-wheeler muffler pressure of maximum 346 MPa with contribution of more amounts of gases in last chamber to reduce or pressure drop perforation effect is being performed as shown in above figure 8.

4.4 MODIFIED MUFFLER WITH PERFORATION

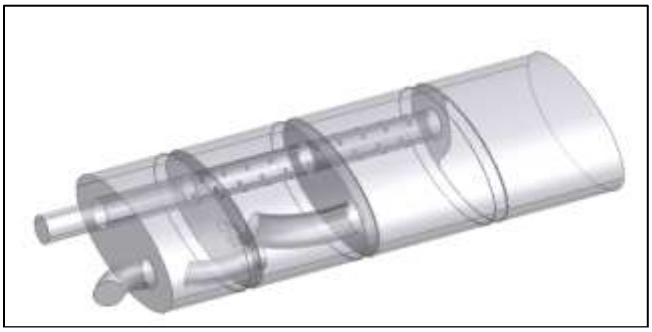


Figure 9. Modified muffler design

Modifying the existing muffler to drop the pressure by introducing perforation of diameter 4 mm alont inlet pipe with 20 number of holes.

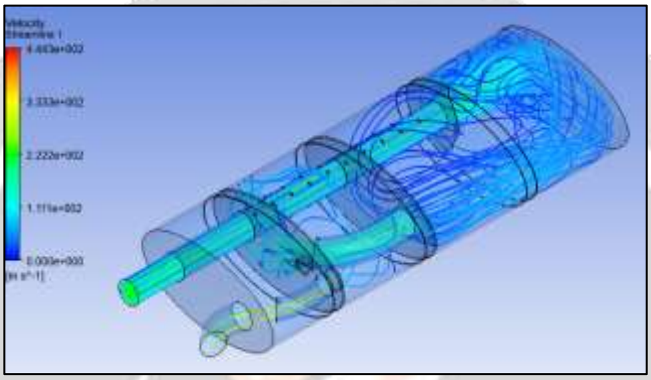


Figure 10. Stream line flow of exhaust gas in modified muffler

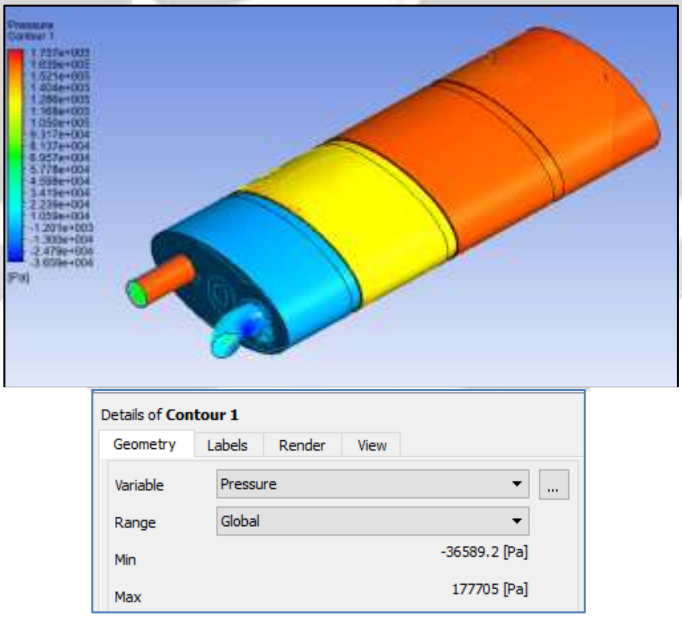


Figure 11 Pressure contour of modified muffler

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.

Figure12. Existing muffler

Figure13. Modified muffler

5. CONCLUSION

1. In present investigation existing muffler CFD simulation have been performed to determine exiting pressure drop, temperature, wall heat transfer coefficient.
2. 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.

6. REFERENCES

1. 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.
2. Commercial Vehicles Muffler Volume Optimization using CFD Simulation by Ashok Patidar, Shivdayal Prasad, Umashanker Gupta, and Mohan Subbarao.
3. CFD Topology and Shape Optimization of Ford Applications using Tosca Fluid by Dr. Anselm Hopf, Ford Motor Company, Aachen, Germany.
4. 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.
5. Modification of Muffler design to increase exit velocity by Nagisetty Lokhesh Kumar and K. Veladri .
6. Integration of exhaust manifold with engine cylinder head towards size and weight reduction by M.A. Neshan, A. Keshavarz, K. Khosravi.
7. M. Nazmul Haquea, Mohammad Alia , Ismat Araa, "Experimental investigation on the performance of NACA 4412 aerofoil with curved leading edge planform", Procedia Engineering 105 (2015) 232 – 240