International Journal of I.C. Engines and Gas Turbines
Vol. 3: Issue 1
www.journalspub.com

CFD Analysis and Modification of Muffler of 1067 cc4 Cylinder Inline Engine for Better Performance

*Tridib Kumar Mahata**
Department of Mechanical and Manufacturing Engineering, MIT-MU, Manipal, Karnataka, India

ABSTRACT
Back pressure is one of the important parameter which determine the performance of the muffler. Pressure drop of the exhaust system includes losses due to piping, silencer, and termination. The most critical mechanical factor regarding backpressure of any commercial modified and available muffler is cross flow perforated tube in which the diameter of the perforated tube hole and porosity of the perforations are most critical. In this paper, the objective is to study and to reduce the backpressure on a 4-cylinder 1067 cc engine by making design changes in its muffler followed by computation fluid dynamics (CFD) analysis for its better performance.

Corresponding Author
E-mail: tridibkumarmahata@gmail.com

INTRODUCTION

The muffler (or silencer) is an integral component of the exhaust system of an automobile. The purpose of the muffler is to provide obstruction to the flow of gases (up to a suitable degree) so as to reduce unwanted noise, while ensuring smooth flow and assist the engine in scavenging. First patented in 1897, this component has been the subject of much research since then, both from an acoustic point of view (for manufacturers looking to achieve a certain sound) as well as performance point of view (reducing the backpressure on the engine, and thus allowing it to aspirate easier, with less pumping losses). The following is a study focused on increasing the performance of a standard commercial vehicle muffler [1–3].

The muffler is the last component in the exhaust system, starting from the exhaust manifold of the engine. It is placed after the catalytic convertor in most modern cars. There are many design requirements to be fulfilled when designing a muffler for a particular automobile. These include: allowable backpressure limits, noise emission restrictions, durability, vibration characteristics, cost, as well as space constraints on the volume of the muffler. Most passenger cars in use today employ a reactive muffler, which is simple in construction and helps to reduce the high frequency noise. But the principal drawback of such mufflers is the restriction placed on the airflow, which is the area of interest in this study. The strategy employed here is to study the flow of exhaust gases through the muffler by performing computation fluid dynamics (CFD) analysis on the existing muffler, and consequently making the appropriate design changes in the geometry of the muffler to alleviate pressure buildup and recirculation zones inside the muffler. The process of CFD allows us to test the prototypes and obtain the comparative performance of the models without investing in manufacturing and personnel costs.

Current work on CFD analysis of muffler has focused on the steady-state analysis, i.e. the gases entering the muffler are assumed to flow continuously and steadily through the muffler. However, a much more meaningful and accurate solution is obtained when a transient analysis is performed, since this enables the designer to get a time-based view of the pressure buildup zones, eddy formation zones, as well as frame-by-frame velocity streamlines through the muffler. The transient analysis performed in this study assumes a three-cylinder engine feeding the exhaust gases into the muffler, wherein the exhaust duration and amount is accurately simulated; it is thus a superior method of producing prototypes of the automobile muffler without investing in tooling, material, time, money and personnel. This type of analysis enables the designer to predict the advantages obtained by modifying a certain geometry feature of the muffler.

The main objective of this work is to identify which geometry features of the muffler can be modified towards achieving lower backpressure on the engine. Three different geometries are compared, and the one with the best performance characteristics is chosen [4–7].

METHODOLOGY

1. The muffler of 1067 cc 4 cylinder inline engine was obtained, and its cut section was taken to obtain its internal geometry.
2. CATIA was used for modelling the muffler.
3. Transient (varying mass flow at each inlet), pressure-based, fluid flow analysis was performed in ANSYS-Fluent, for two different rpm.
4. Plot backpressure changes, pressure contour and velocity profile was obtained for the different models, with respect to time-steps.
5. Modifications are made in the base model, based on the inferences of the CFD results.
6. The Fluid flow analysis is performed again, on the modified model.

Boundary Conditions

In the next step, the meshed model was imported to the FLUENT solver, with a density-based, transient solution being chosen. The SIMPLEC solving scheme was used, as it provides the most stable solution results. The working fluid was specified as air at a temperature of 673 K which behaves as an ideal gas.

The mass flow rate through the exhaust port was assumed to be constant during the exhaust stroke for each cylinder, and drops to zero during the suction, compression and exhaust strokes. This condition was satisfied by using a User Defined Function for specifying the inlet rate and timing of flow at each of the inlet faces. The function, written in the C programming language varies the mass flow with time for each inlet, taking time-step as input parameter and mass flow rate as output parameter. The firing order followed was 1-3-4-2, with each exhaust gas pulse coming at 180° crankshaft rotation apart. A total of 720° of crankshaft rotation was run for each model.

Output of the Program:

The program has four output variables and one input variable. The input variable is time and output variable is mass flow rate of all the inlets. The mass flow pattern at each inlets are shown graphically which is the output of the program. 4 exhaust cycles are shown in graph which takes 8 revolutions of crank shaft to complete. Inlet for first, second, third and fourth cylinders are specified as inlet 1, inlet 2, inlet 3, inlet 4.

Modifications Made to Achieve Reduced Pressure Within the Third Chamber

- Modification 1: Model with 5 holes in the second baffle plate
- The second baffle plate is located in between the second and third chambers. In order to relieve the pressure quicker from the third chamber, 5 holes each of 8 mm diameter were made in the plate.

- Modification 2: Model with 20% increase in length of third chamber
- The advantage of increasing the length of the third chamber lies in the fact that as the volume increases, pressure decreases. As the size of the third chamber increases, more volume of air is able to be cleared out within the same time as compared to a smaller chamber, leading to reduced backpressure [8–10].

5 Holes Were Made on the Second Baffle Plate

Five holes were made on the last baffle i.e. between second and third chamber of diameter 8 mm one at placed at the center of the baffle and other four holes at 90° apart as shown in Figure 1 on a green plane.

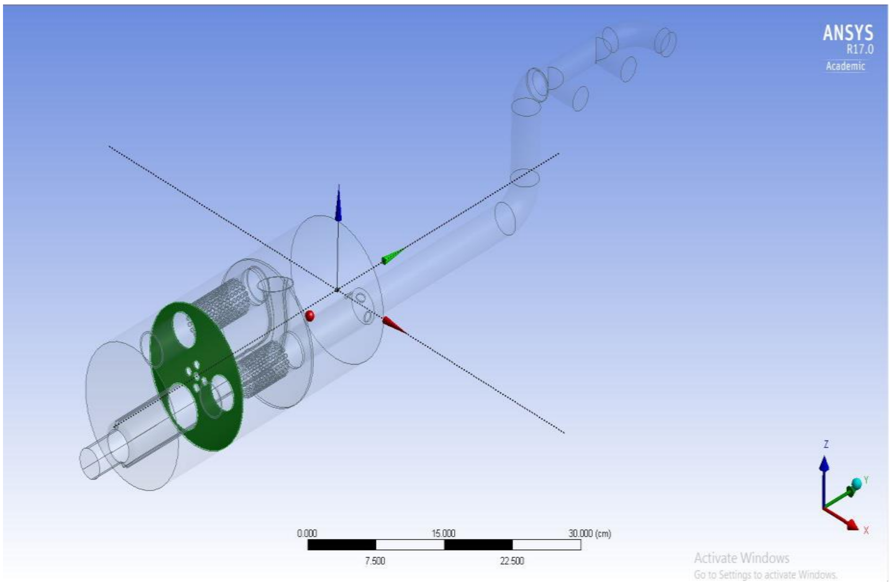

Fig. 1. *Holes were made on the last baffle CAD model.*

Twenty Percent Increased Volume of Third Chamber

In order to satisfy this condition, the he cross sectional area of the chamber was kept constant and the length of the third chamber was increased by 20%. In the base model, each chamber had a length of 100 mm, in this modification the third chamber length was taken as 120 mm as shown in Figure 2.

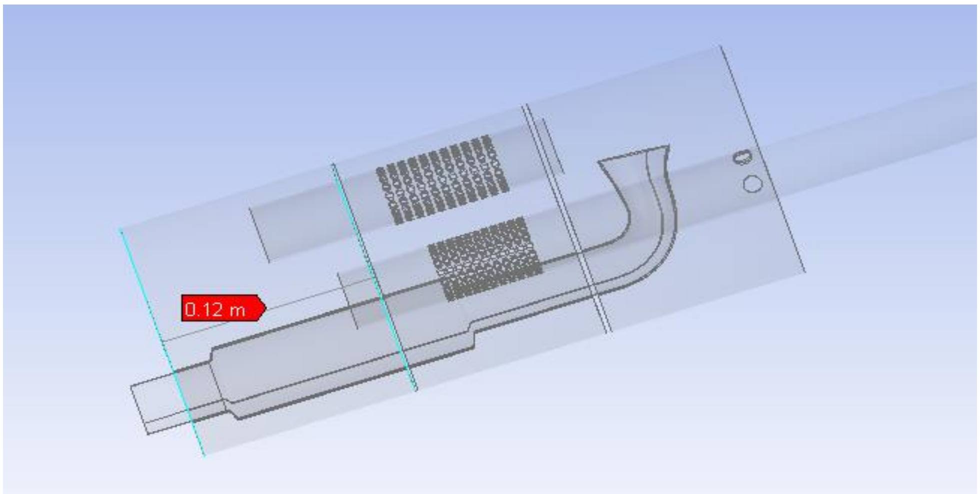

Fig. 2. *Percent increased volume of third chamber.*

All the models are tested with same condition and solved at following speeds.
(1) 5000 rpm
(2) 2500 rpm

Comparing All the Models Based on the Results

The results were compared based on the pressure drop across the muffler. A sectional plane was selected shown in fig located 40 mm away from the muffler. The outlet pressure was set as 0 Pa. which refers to atmospheric pressure so the difference between the plane and outlet surface gives the amount of pressure drop across the muffler. Area average of pressure was monitored for all the models with respect to time on the specified plane as shown in the Figure 3.

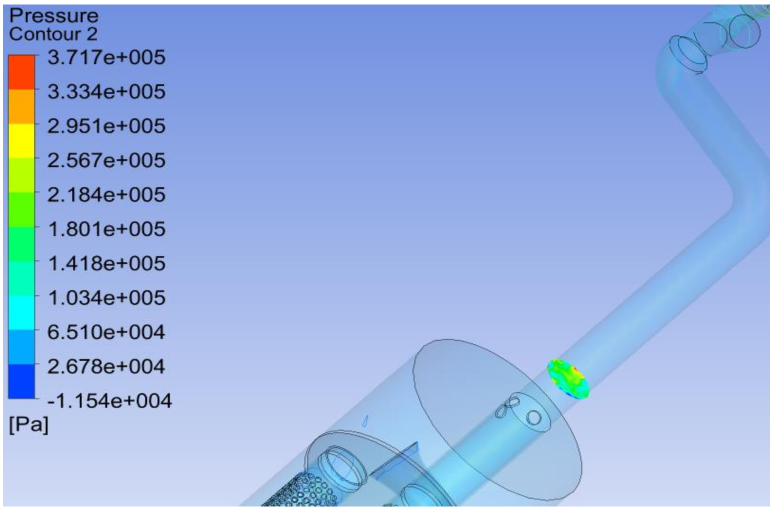

Fig. 3. *Plane location.*

Result Analysis

The results after eight revolutions of crank shaft were taken for the comparison. Area average of pressure at the specified plane was collected for 720° of revolutions of crank shaft and plotted on a graph to observe the behavior of each model at a specific rpm. Time taken for 2 full revolution of crank shaft for 5000 and 2500 rpm are 0.048 and 0.096 sec, respectively. For 5000 rpm the results were plotted with a time step of 0.0005 second and for 2500 rpm 0.001 sec with total 24 data points which refers to last two revolutions of crank shaft, respectively (Figure 4) [11, 12].

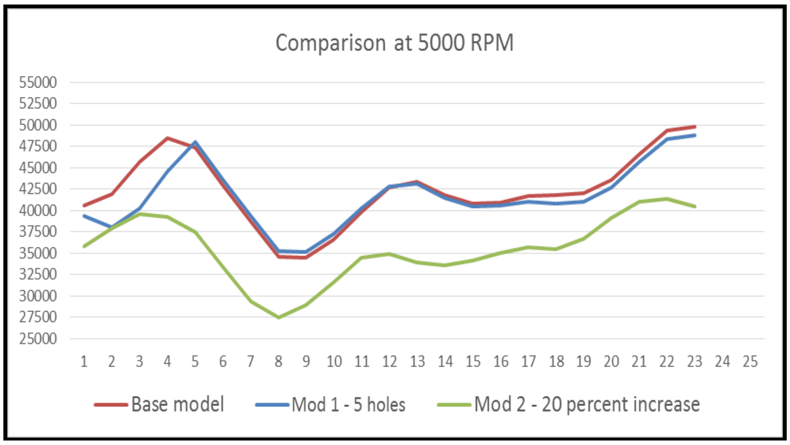

Fig. 4. *Pressure comparison graph at 5000 rpm.*

At 2500 rpm, pressure is seen to be in the range of 3500–16000 Pa. Modified model 1 has same performance as the base model, while modified model 2 produces a reduction of 12.7% over the range of time-steps taken (Figures 5–7).

Each unit on the X-axis represents 2 time-steps, while the Y-axis shows backpressure in Pascal.

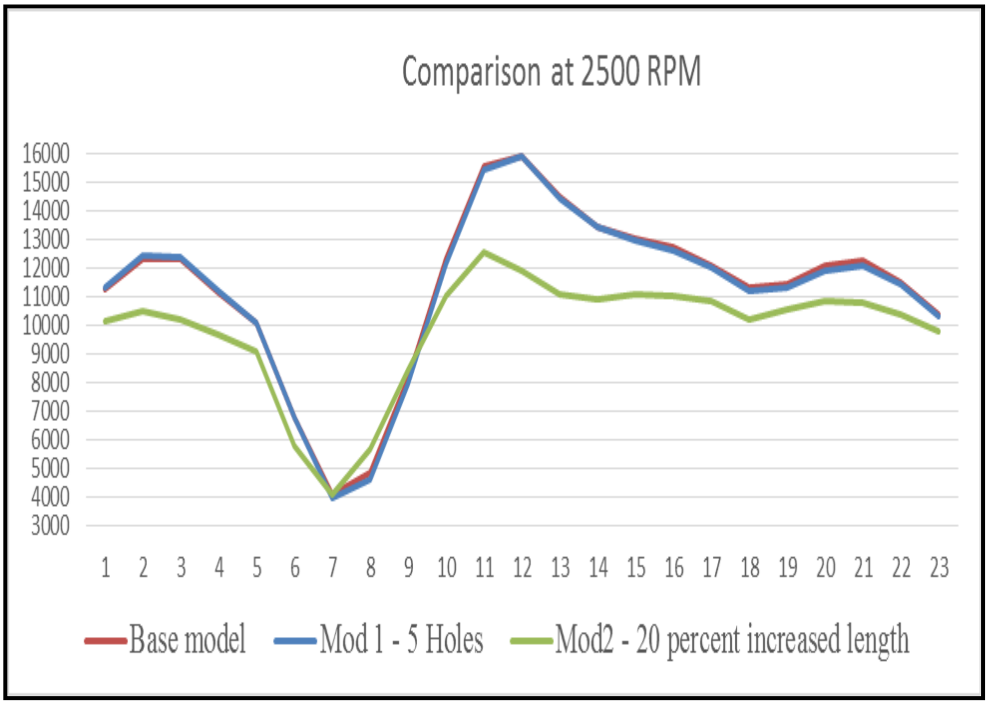

Fig. 5. *Pressure comparison graph at 2500 rpm.*

Pressure drop comparison:

At 5000 rpm:

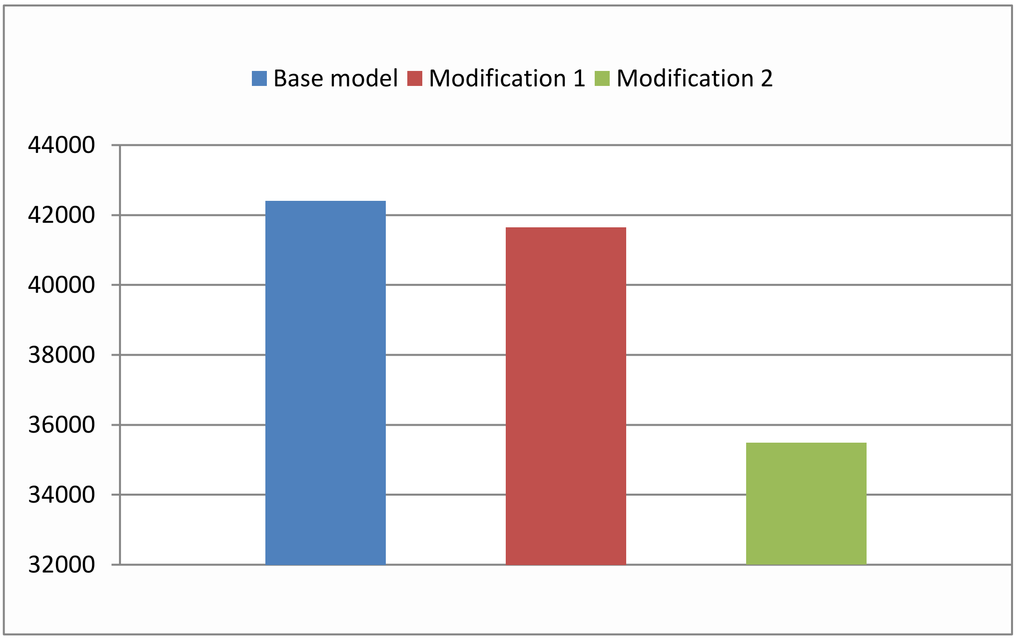

Fig. 6. *Bar chart of pressure comparison at 5000 rpm.*

At 2500 rpm:

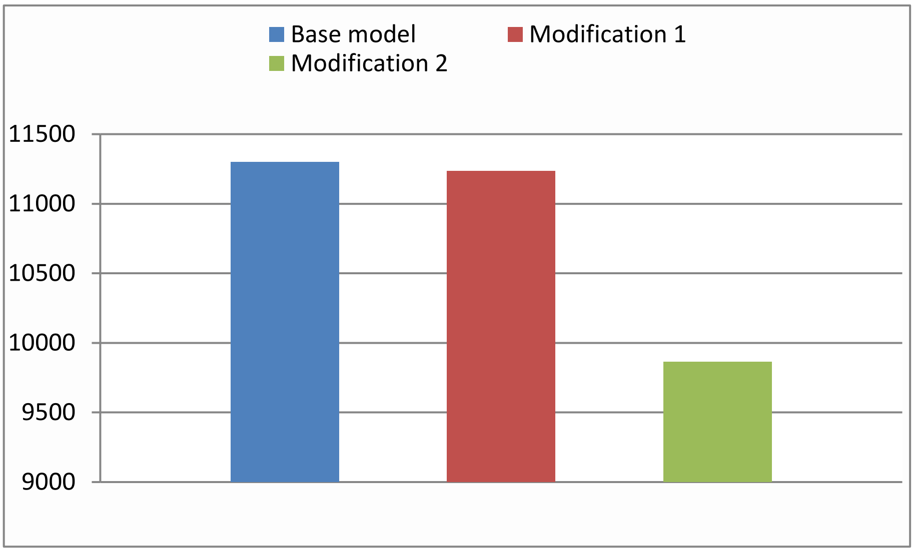

Fig. 7. *Bar chart of pressure comparison at 2500 rpm.*

CONCLUSION

The objective of this study was to reduce the backpressure on a 4-cylinder 1067 cc engine by making design changes in its muffler. This has been found to be achieved by the modification 2 which has given reduction of 12.7% and 16.3% in backpressure at 2500 and 5000 rpm, respectively. The design changes were focused on reducing the pressure in the third chamber of the muffler. Towards this aim, two modifications were made, one of which was to produce perforations in the baffle dividing the second and third chambers, and the second was to increase the length of the third chamber by 20%. The second modification is shown to have good performance, especially at high load condition.

REFERENCES

1. D. Thombare. A practical approach towards muffler design, development and prototype validation, *SAE Technical Paper*. 2010; 2010-32-0021, 2010, doi:10.4271/2010-32-0021.
2. M. Rehman. Design and construction of a muffler for engine exhaust noise reduction, In: *Proceedings of the International Conference on Mechanical Engineering 2005 (ICME2005)*. 28–30 December 2005, Dhaka, 2005.
3. Analysis of Six Stroke in an Internal Combustion Engine, Tridib kumar mahata, https://scholar.google.co.in/citations?view_op=view_citation&hl=en&user=Q0iWb74AAAAJ&citation_for_view=Q0iWb74AAAAJ:Y0pCki6q_DkC.
4. A Theoretical Investigation of Characteristics of Diesel Engine with Coil Cooler System, https://scholar.google.co.in/citations?view_op=view_citation&hl=en&user=Q0iWb74AAAAJ&citation_for_view=Q0iWb74AAAAJ:dhFuZR0502QC.
5. A Theoretical Investigation of Characteristics of Diesel Engine with Coil Cooler System, https://scholar.google.co.in/citations?view_op=view_citation&hl=en&user=Q0iWb74AAAAJ&citation_for_view=Q0iWb74AAAAJ:7PzlFSSx8tAC.
6. A Computational Study of Solar Air Heater for Its Performance Enhancement by Numerical Approach Using Standard CFD Tools, https://scholar.google.co.in/citations?view_op=view_citation&hl=en&user=Q0iWb74AAAAJ&citation_for_view=Q0iWb74AAAAJ:Wp0gIr-vW9MC.

7. M.P. Tambe. Analysis of exhaust system – ‘semi active muffler’, *IJIRSET*. 2016; 5.
8. U. Kalita. Absorption materials used in muffler. *Int J Mech Ind Technol*. 2015; 2: 31–7p.
9. J. Xu. Analysis of flow field for automotive exhaust system based on CFD, *Researchgate*. 2015.
10. V.D. Prajapati, A.J. Desai. Design and analysis of automotive muffler, *Int J Eng Res Technol*. 2016; 5.
11. R. Veloso. A 3D linear acoustic network representation of mufflers including higher order modes, *Researchgate*. 2016.
12. O.R.D. Pangavhane. Experimental and CFD analysis of a perforated inner pipe muffler for the prediction of backpressure, *CiteSeer*