Back pressure analysis of an engine muffler using cfd and experimental validation

[#1]Mr. S.S. Mane,[#2]S.Y.Bhosale

[#1]Mechanical Engineering, PES's Modern College of engineering, Pune, INDIA
[#2]Mechanical Engineering, PES's Modern College of engineering, Pune, INDIA

ABSTRACT— In this paper, a research on back pressure of a muffler is done using CFD. At different engine velocities, the flow field of a given geometry of muffler is simulated and the total back pressure inside the muffler is calculated. The muffler is then simulated for different configurations in an attempt to minimize the back pressure. It is resulted out that the perforated tube is the most critical parameter from back pressure point of view. The change in diameter of the perforated tube and that of the holes on the tube changes the back pressure significantly. The least value of back pressure from CFD simulation is 356.55Pa. The experimental and CFD values for base model are observed in good agreement with each other.

Keywords— Back Pressure,Muffler

I. INTRODUCTION

Internal combustion engines are responsible for a great amount of environmental noise. This noise is controlled through the use of mufflers. A silenceris a noise attenuation device, whereas a muffler is a device specially designed to controltheexhaust noise from an engine. Silencers are constantly worked on to improve the performance of the exhaust system. Exhaust mufflers are responsible for reducing the noise of an engine and the noise of other possible sources in vehicles. To maintain the required lower noise the muffler needs to be analyzed.

In this work, a single cylinder diesel engine experimental set up is selected, having an in built reactive muffler. The aim is to minimize the back pressure of the muffler and hence to improve the thermal efficiency of the engine. The flow field of the muffler is simulated using CFD and computed.

1. Basic theory of fluid dynamics for reactive muffler

For the muffler, the mass flow rate of the exhaust gas at inlet is equal to that from the outlet. The flow volume of the muffler must satisfy the law of conservation of mass, the law of conservation of momentum, and the law of conservation of energy.

Assumptions made for analysis:

1. The physical parameters of the solid and fluid sections for the reactivemuffler are constant;
2. The flow is considered turbulent steady flow;
3. The effect of the gravity is not considered;
4. The velocity at the inlet of muffler is homogeneous.

Through the muffler, the velocity of gas flow is higher. Considering ideal turbulence and selecting k –ε turbulent modal, the general control equation is given by as:

$$\frac{\partial(\rho\phi)}{\partial t} + div(\rho v\phi) = div(\Gamma grad\phi) + S$$

2. Computation of back pressure

According to the measuring norm of pressure loss for muffler, equal-located points are selected in the inlet/outlet sections, and the average value of total pressure was taken at the one of the section. Fig.1 is the sketch map for measuring the pressure in the inlet/outlet sections. The difference of total pressures of the two sections will be considered as the pressure loss of the muffler. The computation formula for pressure loss is

$$\Delta p = pt1 - pt2$$

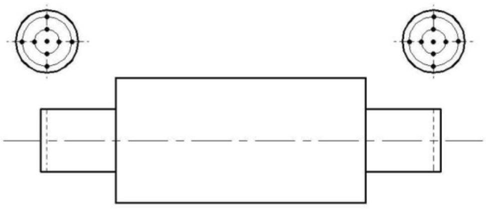


Fig.1.Measuring norms of pressure loss for muffler

II. LITERATURE REVIEW

In 2007, P. S. Yadav, A. Muthukumar, V. V. PhaniKiran, V. Tandon and S. Raju, studied the optimization of silencer as an integrated approach i.e., for transmission loss and backpressure. They developed an integrated methodology to predict the performance of the silencer at the design stage resulting in an optimized time and cost effective design. The acoustical and engine performance of silencer was predicted using FEM/BEM and CFD techniques. Parametric studies were carried out to find the effect of geometry on the transmission loss and backpressure. Transmission loss of various configurations was evaluated using a 1D simulation code GT-Power. A steady state analysis was carried out using 3D CFD code FLUENT to predict the backpressure for various silencer configurations.They achieved the optimized design using integrated methodology, which met the acoustic as well as backpressure target requirements. They benchmarked a methodology for the optimized design of silencer, which takes into account the noise as well as the backpressure constraints and finds the performance parameters like TL and backpressure, which form the basis for the design of silencer [1].

In 2009, Jianhua Fang1, Yiqi Zhou2, Peigang Jiao, Studied the Pressure Loss for a Muffler Based on CFD and Experimentedit. They simulated the flow field of a muffler based on CFD technique atacertainengine velocity and analysed the pressure distribution internal the muffler. They found that with increased engine velocity, the pressure drop across the muffler became higher. For the same engine velocity with lower punching rate, the pressure loss increased [2].

In 2010, Shital Shah, Prof. D. G. Thombare, developed a practical approach to design, develop and test muffler particularly reactive muffler for exhaust system, which gave advantages over the conventional method with shorten product development cycle time and validation. They emphasized the importance of the design methodology –a practical approach from the concept design to proto manufacturing and validation of exhaust muffler [3].

In 2013, N. V. Pujari, Mahajan S.R., Mohite Y.B., developed an integrated methodology using CFD technique to predict the acoustical and engine performance of the muffler [4].

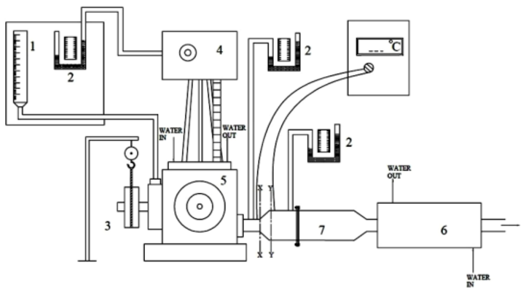
In 2013, S.S. Pangavane, AmolUbale, made CFD analysis and experimentation on perforated inner tube of muffler in an attempt to minimize its backpressure on the engine. They also varied the porosity of the perforated tube to check the effect on backpressure. They tested three different models of muffler with varying porosity. It was concluded that the backpressure was reduced by 75% if the porosity was doubled. Also increase in hole diameter had a remarkable effect on backpressure by about 40% [5].

Remarks:

1. Back pressure of the muffler has a considerable effect on engine performance.
2. Design of muffler involves two conflicting requirements i.e. TL and back pressure.
3. For a given muffler, its back pressure can be minimized by modifying its geometry.
4. CFD can be effectively used as a tool to evaluate the back pressure of muffler.

3. Experimental set up:

Fig. 2.Shows the schematic view of the experimental set up for the measurement of back pressure of the engine. The exhaust gas velocity is maintained at 20 m/s, 40 m/s, 60 m/s and 80 m/s at different times during the experiment. The exhaust gas velocity is measured by Pitot tube. The back pressure is measured as the difference between the pressures at the inlet and exit of engine muffler.



1 Fuel Flow Measurement 5 C.I. Engine
2 U- Tube Manometers
Fig 2. Schematic view of experimental set up

4.Physical model

Figure 3 shows the internal geometry of the muffler. The muffler is composed of four chambers. The perforated tube is the critical component which responsible for producing reduced noise of higher frequency and insert pipe can reduce noise of lower frequency.

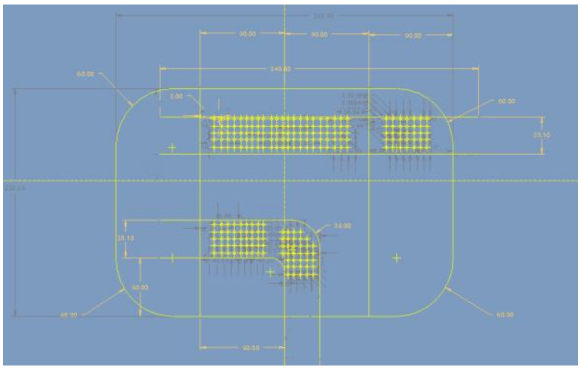


Fig. 3.Internal Geometry of muffler

5.Boundary Conditions

The exhaust from the engine consists of gases CO, HC, NO_X, SO_2. The amount of these gases less than 0.6%.Thus the molecular weight of exhaust gas can be considered to be approximately equal to that of air. The exhaust gas from engine is considered as air having density 1.2 kg/$m^3$and viscosity 1.7894e-05pa.s. Before the actual process of computation, a Segregated/Implicit solution method is selected. The flow is taken as steady flow, k −ε standard turbulent modal is considered, energy equation is started considering air as flow fluid.

The necessary boundary conditions are prescribed by taking in to account the correlative inlet/outlet data at a certain speed for similar type of engine. Specific choices are as follows: velocity-inlet type, at 20m/s of inlet speed, 80 m/s of inlet speed, and pressure outlet type. In order to simplify the problem, the density of gases in the muffler is seen as a stable value of 1.255kg/m^3 at 813K. The walls are considered adiabatic, stationary no slip walls. The initial condition is 1 atm pressure. After finishing computation, the convergent results were saved.

III.RESULTS AND DISCUSSION

Fig.5(a)shows the velocity distribution and 5(b) shows the pressure flow distribution across the muffler for an inlet velocity of 20 m/s and perforation of 3 mm .It is observed that the back pressure varies in every chamber.

The back pressure at the inlet of muffler is observed to be 540 Pa. The first chamber experiences maximum back pressure of all the three chambers. The pressure then drops in the second chamber and becomes about 350 Pa. Finally, in the last chamber the pressure slightly decreases further and its value becomes 290 Pa. Now the back pressure inside the muffler is the difference between the inlet pressure and the exit pressure and it is 540Pa for the velocity of 20 m/s

Fig. 6.1 shows the results of simulation carried out for an increased hole diameter of 4 mm. The value of back pressure for this configuration is 444Pa. Fig. 6.2 shows the results of simulation for the same 4 mm and 80 m/s velocity of flow. The observed back pressure value is 4343 Pa. Fig. 7.1 shows the simulation results for another modification in geometry with changed baffle spacing to 165 mm. The observed value of back pressure for this configuration is 356 Pa. Thus it is observed that the minimum value of back pressure is achieved for the changed baffle spacing configuration. Similar simulations are carried out at different velocities. The corresponding back pressures are tabulated below in table 2. The comparison between the CFD and experimental values is represented on graphs of velocity vs back pressure for base model.

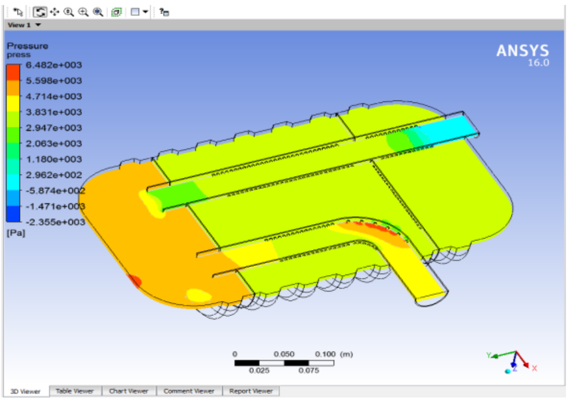


Fig .4. Pressure Flow Contour for Base Model

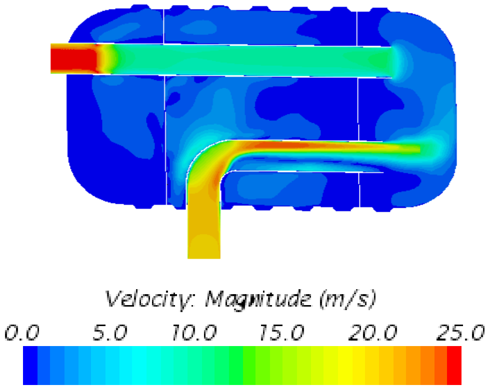


Fig. 5(a) Perforation 3 mm, Plate Distance 208 mm, Flow Velocity = 20 m/s

Base Model Simulation

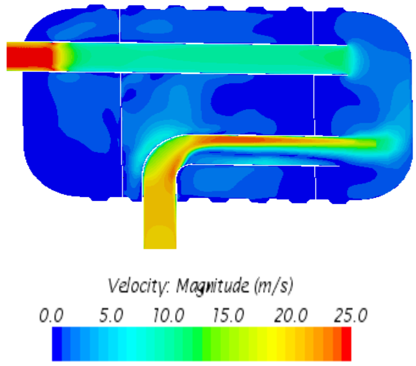


Fig. 6.1 (a) Perforation 4 mm, Plate Distance 208 mm, Flow Velocity = 20 m/s

Modification 1

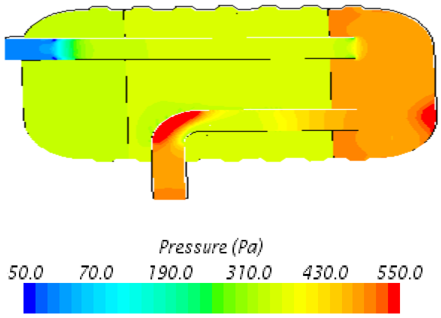


Fig. 5(b) Back pressure at muffler inlet 540.5 Pa

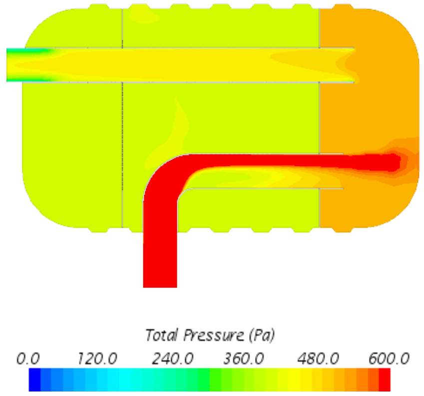


Fig. 6.1(b) Back pressure at muffler inlet 444.6Pa

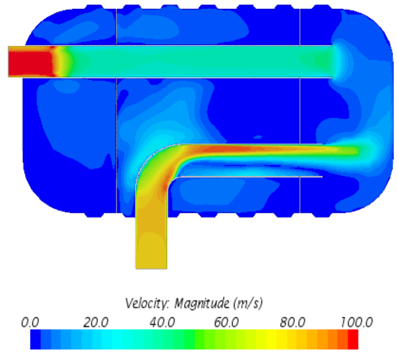


Fig 6.2(a) Perforation 4 mm, Plate Distance 208 mm, Flow Velocity= 80 m/s

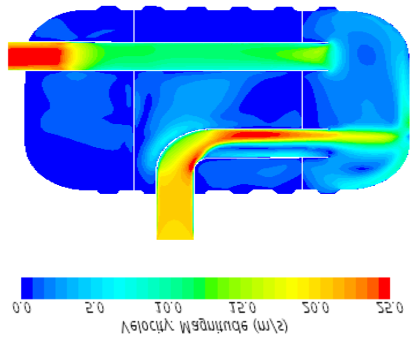


Fig. 7.1(a) Perforation 3 mm, Plate Distance 165 mm, Flow Velocity = 20 m/s

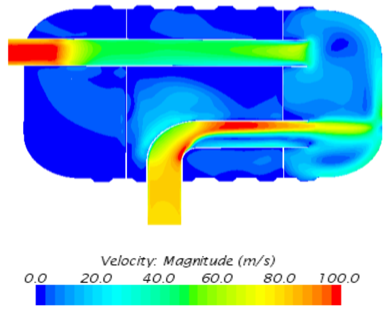


Fig. 7.2(a) Perforation 3 mm, Plate Distance 165 mm, Flow Velocity = 80 m/s

Modification 2

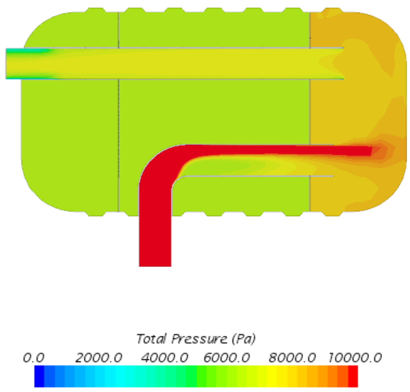


Fig 6.2(b) Back pressure at muffler inlet 4343Pa

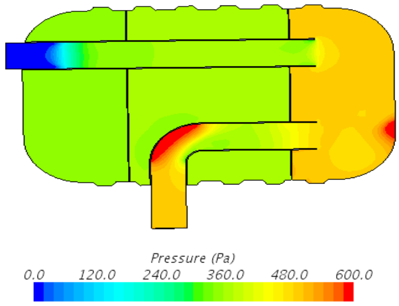


Fig. 7.1(b) Back pressure at muffler inlet

356.55 Pa

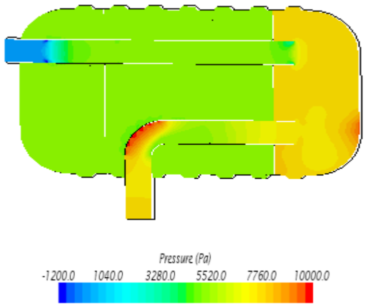


Fig. 7.2(b) Backpressure at muffler inlet 4033 Pa

Table 1.Back pressure for CFD simulation

	Perforation & Baffle Spacing(mm)	**Back Pressure, Pa (CFD)**		**Back Pressure, Pa (Experimental)**	
		At 20 m/s	**At 80 m/s**	**At 20 m/s**	**At 80 m/s**
Case 1 (Original)	3 & 208	540.5	4377	512	4015
Case 2	3 & 165	356.55	4033	-	-
Case 3	4 & 208	444.6	4343	-	-

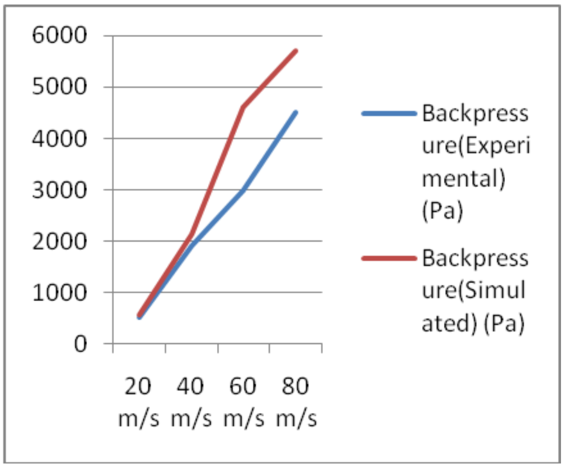


Fig No.8. Backpressure Vs Flow Velocity

IV.CONCLUSIONS

1. The configuration with hole diameter 3 mm and baffle spacing 165 mm creates the least back pressure.
2. This configuration is selected for modification in the muffler.
3. Now the selected configuration is to be worked out and its experimental testing is to be done.
4. There is discrepancy of around 10 % between the CFD and experimental values for the base model. Hence, it can be said that the two are in good agreement with each other.
5. CFD can be conveniently used as a tool for back pressure analysis of muffler.

. **Symbol**	**Meaning**
ϕ	the general variable,
Γ	the generalised diffusion coefficient,
S	the generalised source term,
V	the velocity vector, m/s,
ρ	density of air, kg/m^3.
pt1	average pressure of the inlet
pt2	average pressure of the outlet

REFERNCES

1. P. S. Yadav, A. Muthukumar, V. V. PhaniKiran, V. Tandon and S. Raju,2007, “Optimized Design of Silencer -An Integrated Approach”, SAE Paper No. 2007-26-037
2. Jianhua Fang1, Yiqi Zhou2, Peigang Jiao, 2009, ‘Study on Pressure Loss for a Muffler Based on CFD andExperiment”, 2009 International Conference on Measuring Technology and Mechatronics Automation.
3. Shital Shah, Prof. D. G.Thombare,2010, “A Practical Approach towards Muffler Design, Development andPrototype Validation, SAE Paper No. 2010-032-0021.
4. N. V. Pujari, Mahajan S.R., Mohite Y.B., 2013, “Optimization of Silencer - An Integrated Approach of Acoustic Performances & Backpressure”,
International Journal of Emerging Science and Engineering (IJESE) ISSN: 2319–6378, Volume-2, Issue-2, December 2013.
5. SudarshanDilipPangavhane, AmolBhimraoUbale, Vikram A. Tandon, 2013, “Experimental and CFD Analysis of a Perforated Inner Pipe Muffler for the Prediction of Backpressure”, International Journal of Engineering and Technology (IJET), ISSN : 0975-4024 Vol 5 No 5.
6. Rahul D. Nazirkar, S.R. Meshram, Amol D. Namdas, “Design & Optimization of Exhaust Muffler & Design Validation”,10TH IRF international conference, ISBN: 978-93-84209-23-0.

