NOVATEUR PUBLICATIONS
International Journal Of Research Publications In Engineering And Technology [IJRPET]
ISSN: 2454-7875
VOLUME 2, ISSUE 9, Sep. -2016

FLOW ANALYSIS OF REACTIVE MUFFLER USING CFD

SUYOG S. MANE,
PROF. S.Y. BHOSALE,
PROF. H.N. DESHPANDE,
Modern College of Engineering, Pune.
Email: suyogsmane@gmail.com

ABSTRACT:

Muffler design is traditionally a trial and error process. This paper describes the flow analysis of a reactive muffler using CFD simulation in order to improve its performance by reducing the back pressure created on the engine. The back pressure of the muffler is computed from CFD simulation. The CFD analysis is done to avoid the tedious experimentation. The flow simulation is carried out using k-ε turbulent model as it is most suitable for turbulent flows having less converging time. Total four cases were analyzed including the base model muffler. Thus three modifications were done in muffler geometry. The modification with reduced baffle spacing produced least back pressure with reduction in back pressure by 8.59%.

INTRODUCTION:

Mufflers are used to dampen the high intensity pressure pulse generated by the combustion process from an internal combustion engine. Generally speaking, there is no technical distinction between a silencer and muffler and the terms are frequently used interchangeably. A silencer has been the traditional name for noise attenuation devices, while a muffler is smaller, mass-produced device designed to reduce engine exhaust noise. The main function of a muffler is to reduce the noise emitted by the engine.

1.1 MUFFLER:

A muffler is a device for decreasing the amount of noise emitted by the exhaust of an internal combustion engine. Mufflers are installed within the exhaust system of most internal combustion engines, although the muffler is not designed to serve any primary exhaust function.

1.1.1 ABSORPTIVE:

This type of muffler uses only absorption to reduce the sound. These mufflers produce much less restriction, but don't reduce the sound level as much as conventional mufflers. An absorptive muffler is shown in fig. 1.1

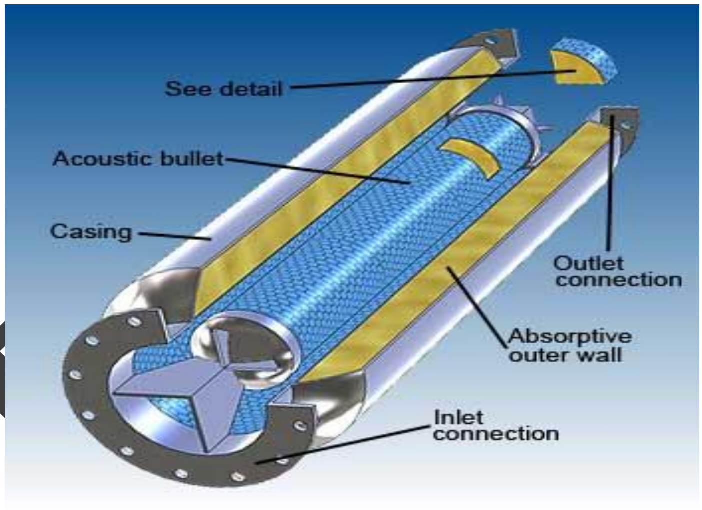


Fig 1.1 An Absorptive Muffler

1.1.2 REACTIVE MUFFLER:

In this type of muffler Inlet and outlet tube are extended in chambers as shown in fig 1.2. These mufflers generally consist of several pipe segments that interconnect with a number of larger chambers. These are most widely used to attenuate the exhaust noise of internal combustion engines.

Fig. 1.2 A Reactive Muffler

1.2 BACK PRESSURE:

Back pressure is defined as the extra static pressure exerted by muffler on engine through restriction in flow of exhaust gases. Higher Back pressure can cause decrease in engine efficiency or increase in fuel consumption, overheating, and may result in a complete shutdown of engine.

LITERATURE REVIEW:

Many researchers have worked in the field of acoustics and back pressure in order to increase the transmission loss of the muffler as their prime objective and to reduce the back pressure as the secondary. Some of them have also tried to develop an empirical relation between the transmission loss and the muffler dimensions. Some of the reputed journal papers describing the findings of the authors are discussed below:

P. S. Yadav et al.[1] 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 back pressure constraints and finds the performance parameters like TL and backpressure, which form the basis for the design of silencer.

Jianhua Fang et al. [2] studied the Pressure Loss for a Muffler Based on CFD and experimented on it. They simulated the flow field of a muffler based on CFD technique at a certain engine velocity and analyzed 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.

Shital Shah et al. [3]developed a practical approach to design, develop and test muffler particularly reactive muffler for exhaust system, which gave advantages over the conventional method with shortened 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.

Wang Jie et al. [4] created a three-dimensional solid model of muffler by ProE and analyzed by ANSYS. The method of data transformation between ProE and ANSYS was prepared. Muffler natural frequencies and modal shapes were calculated by the FEM analysis software named ANSYS, by which modal shapes are displayed in the form of animation. The natural frequencies and mode shapes were considered during the design of the muffler; so as to avoid the resonance occurred in exhaust system.

N. V. Pujari et al. [5] developed an integrated methodology using CFD technique to predict the acoustical and engine performance of the muffler.This research work resulted in the best optimized design achieved using integrated methodology, which meets the acoustic as well as backpressure target requirements. This study presents a benchmark 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 Acoustic performance (db) and backpressure, which form the basis for the design of silencer. The methodology developed is helpful for the manufacturers as well OEMs to reduce the design cycle time for silencer.

2.1 SUMMARY OF LITERATURE REVIEW:

From the literature review and all the other aspects, some highlights of finding are below:

1. If the diameter of the perforation is increased the backpressure decreases sharply. The change in diameter of perforations has remarkable effect.
2. Design methodology emphasis on modern CAE tools for optimization of overall system design to choose the best concept.
3. The Backpressure reduced almost by 75% if the porosity is doubled. Also, if the diameter of the perforation increases the backpressure decreases sharply. The change in diameter of perforations has remarkable effect on back pressure.
4. The CFD simulation software can be used in designing and simulations. The simulations give valuable information regarding the velocity field, pressure field, density field and temperature field of the exhaust muffler. This is important because save time and many in the production process through the identification of eventual problems before the exhaust muffler is build.

2.2 GAP ANALYSIS:

1. The effect of baffle spacing on back pressure needs to be studied in detail.

2. Most of the research papers focus on transmission loss characteristics of muffler and less importance is given to the effect of back pressure.
3. Since the back pressure has significant effect on the performance and life of engine the study of back pressure and efforts to minimize it are necessary.

   So in this work, the effect of baffle spacing on back pressure is studied along with effect of change in perforation size on the value of back pressure.

2.3 OBJECTIVES:

1. To analyse numerically the flow parameters of exhaust gases like velocity field, pressure field by using CFD package.
2. To study effect of variations of the number of perforations, diameter of perforations and the distance between baffle plates on back pressure of an automotive silencer.
3. To select the optimum configuration based on CFD package result which produces the least back pressure.

CFD SIMULATION:

The simulation of the CAD models is carried out in software package STAR CCM+. Muffler has a complex geometry structure. The Fig.3.1 shows the insight of muffler structure. In the given muffler model, two baffle plates, one inlet and one outlet pipe are used. The inlet pipe is bent at 90° on which the perforations are made. The outlet pipe is straight one having two baffle plates. One of the baffle plate on the outlet pipe is having perforations of 3 mm diameter. Other baffle is without any perforations. The geometry is modelled in PRO ENGINEER 5.0 software.

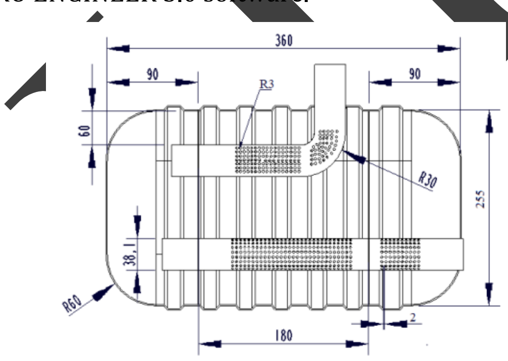


Fig.3.1 2D Geometry of Muffler

3.1 COMPUTATIONAL MESH:

The CAD Model in Para solid format was imported in STAR CCM+. Then geometry clean-up was done and the surface mesh was created using triangular elements. For the gas volume, unstructured mesh has to be created. The Fig. 3.2 shows the meshed model

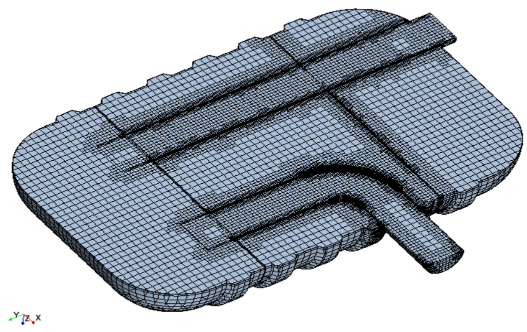


Fig. 3.2 Meshed Model

3.2 BOUNDARY CONDITIONS:

The Table 3.1 shows boundary conditions used for simulations. For simulation, velocity inlet condition and pressure outlet condition were used for inlet and outlet respectively.

Table 3.1 Boundary Conditions

Parameters	**Value**
Inlet Condition	Velocity Inlet
Inlet Velocity	20, 40, 60, 80 m/s
Outlet Condition	Pressure Outlet
Turbulent Model	Standard k-ε turbulent model with standard wall functions
Wall Surface Specification	Smooth, No-slip wall

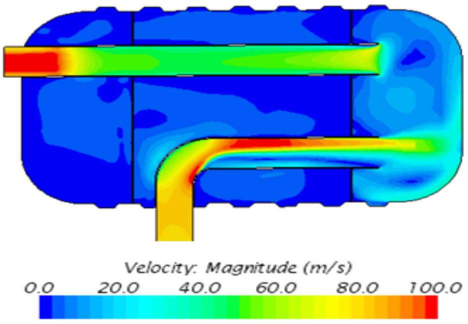
SIMULATION RESULTS AND DISCUSSION:

The results of simulation of different muffler configurations are discussed below.

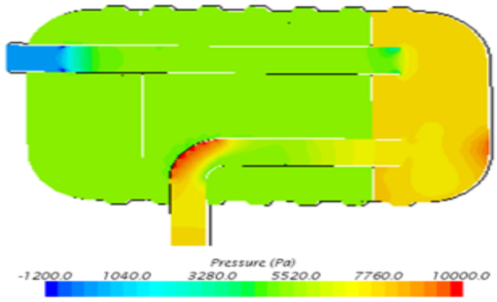
4.1 BASE MODEL SIMULATION (CASE-1):

The simulation results for base model muffler are shown in fig. 4.1.

The velocity near inlet pipe walls (after bent section) was higher. The velocity near outlet was found higher. It was observed that the pressure changes among the expansion chambers. The pressure in inlet pipe at bent section was observed higher. The back pressure observed was around 4196 Pa.



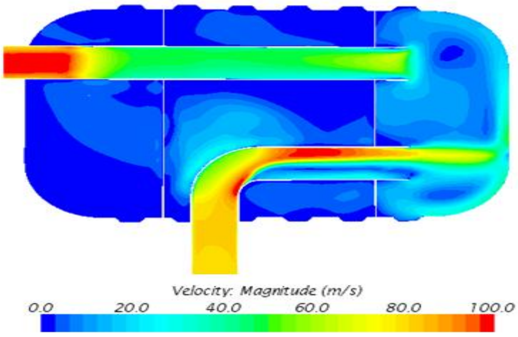
a) Velocity Distribution



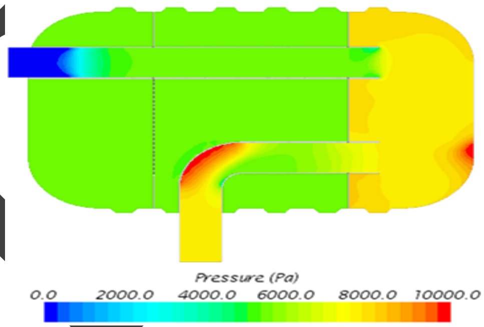
b) Pressure Distribution

Fig 4.1 Velocity and Pressure Distribution for Case-1

4.2 SIMULATION OF MODIFICATION 1 (CASE-2):



a) Velocity Distribution

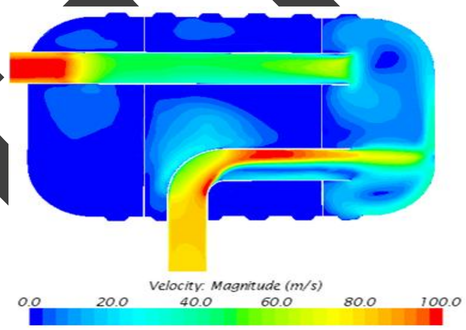


b) Pressure Distribution

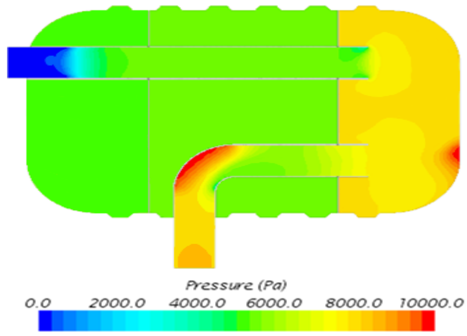
Fig 4.2 Velocity and Pressure Distribution for Case-2

The Fig. 4.2 (a and b) shows the velocity and pressure distribution for modified perforation diameter of 4 mm. The simulation back pressure observed was around 4343 Pa. Thus the back pressure increases in this modification.

4.3 SIMULATION OF MODIFICATION 2 (CASE-3):



a) Velocity Distribution



b) Pressure Distribution

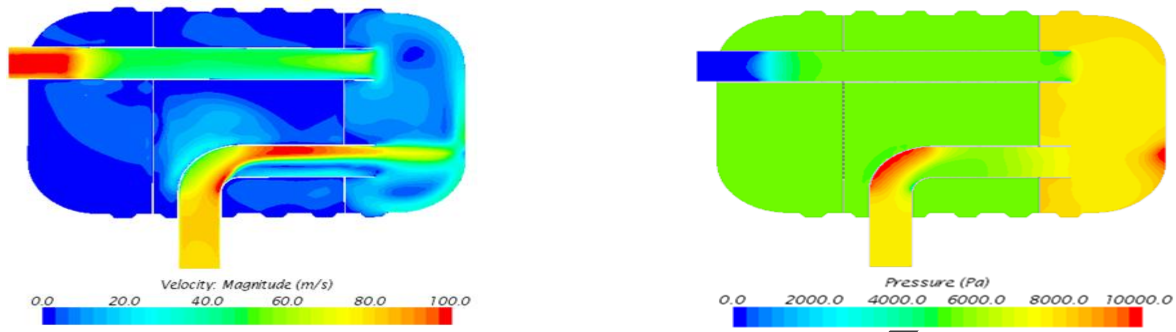
Fig. 4.3 Pressure and Velocity Distribution for case 3

The Fig. 4.3 above shows the velocity and pressure distribution for perforation size of 4 mm and baffle spacing 165 mm. The back pressure observed was around 4240 Pa. This is lower than the corresponding back pressure for the Case-2 muffler

.

NOVATEUR PUBLICATIONS
International Journal Of Research Publications In Engineering And Technology [IJRPET]
ISSN: 2454-7875
VOLUME 2, ISSUE 9, Sep. -2016

4.4 SIMULATION FOR MODIFICATION-3 (CASE-4):



a) Velocity Distribution b) Pressure Distribution

Fig 4.4 Velocity and Pressure Distribution for Case-4

Fig. 4.4 shows the pressure and velocity distribution for the muffler with perforation size of 3 mm and baffle spacing of 165 mm. The observed back pressure for this configuration was 3833 Pa. This is the least value of back pressure. From the results above, it was observed that the Case-4 muffler model is showing least back pressure than Case-1, Case-2 and Case-3 muffler model.

Table 4.1 Simulation Results of Back Pressure

Sr. No.	Perforation Size	Baffle Spacing	Back Pressure, Pa (CFD)
Case 1	3 mm	180 mm	4196
Case 2	4 mm	180 mm	4343
Case 3	4 mm	165 mm	4240
Case 4	3 mm	165 mm	3833

The fig.4.5 shows the graph of comparison of back pressure of all the four muffler cases against flow velocity. It is clearly seen that the modification with baffle spacing of 165 mm produces least back pressure.

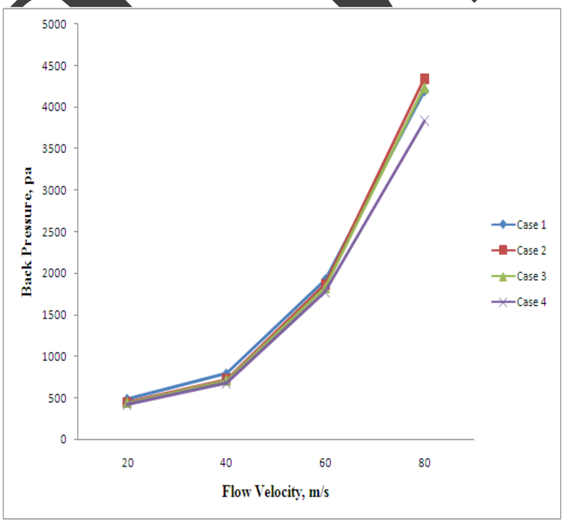


Fig. 4.5 CFD Simulation Results

CONCLUSIONS:

1. The configuration with perforation diameter 3 mm and baffle spacing 165 mm creates the least back pressure.
2. The back pressure for the case-4 muffler is 3833 Pa whereas the back pressure for the base model is 4196 Pa. Thus, there is a discrepancy of 3.25 % between CFD and experimental results.
3. The Case-3 muffler doesn't give satisfactory results compared to Case-4. This is due to the fact that though Case-3 combines Case-2 and Case-4, the effect of decrease in number of perforations is more predominant.

FUTURE SCOPE:

1. In addition to the above approach, the shape modification of the perforation can be considered for future work.
2. The variation in the diameter of the perforated tube may also be analyzed in the next stages.

REFERENCES:

1. P. S. Yadav, A. Muthukumar, V. V. Phani Kiran, 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 and Experiment*", 2009 International Conference on Measuring Technology and Mechatronics Automation.

3. Shital Shah, Prof. D.G.Thombare,2010, "*A Practical Approach towards Muffler Design, Development and Prototype Validation*", SAE Paper No. 2010-032-0021.
4. Wang Jie, "*The Modal Analysis of Automotive Exhaust Muffler Based on PROE and ANSYS*", 2010, 3rd International Conference on Advanced Computer Theory and Engineering (ICACTE)
5. 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.