17th International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2013, Istanbul, Turkey, 10-11 September 2013

AN ANALYSIS OF GEOMETRIC PARAMETERS' EFFECTS ON FLOW CHARACTERISTIC OF A REACTIVE MUFFLER

Erdem Özdemir, Rifat Yılmaz, Zeynep Parlar, Şengül Arı
İstanbul Technical University, Faculty of Mechanical Engineering
İnönü Street No: 65 Gümüşsuyu 34437 İSTANBUL, Turkey

ABSTRACT

Internal combustion engines are typically equipped with an exhaust muffler to suppress the acoustic pulse generated by the combustion process. A high intensity pressure wave generated by combustion in the engine cylinder propagates along the exhaust pipe and radiates from the exhaust pipe termination. Exhaust mufflers are designed to reduce sound levels at certain frequencies. New regulations and standards for noise emission increasingly compel the automotive firms to make some improvements about decreasing the engine noise. On the other hand, developments on automobile technology and increasing competition between manufacturers necessitates having being reduced weight, having capability of higher sound absorption and lower back pressure mufflers. Lightness could be possible if the thickness is decreased or the volume is reduced. However, this causes high back pressure. Therefore, the optimum design requires. Recently finite element methods are used to obtain flow characteristics and back pressure values of mufflers. Having used of this method, effect of different parameters can be examined without prototyping and best suitable muffler can be determined in the design process. Furthermore time and money can be saved. In this study, a reactive, perforated and cross-flow muffler which damped the noise of a spark ignition engine is investigated. Some geometrical dimensions of the muffler are decreased and new models are redesigned. Models are analyzed to obtained flow characteristic by using computational fluid dynamics. .

Keywords: exhaust muffler, CFD analysis, back pressure

1. INTRODUCTION

End of the combustion process high temperature gas have to be released outside. To overcome the sound which is occurred by high velocity and pressure of gas mufflers are being used. Developments on automobile technology and increasing competition between manufacturers necessitates having being designed lighter, having capability of higher sound absorption and lower back pressure mufflers. Lightness could be possible if the thickness is decreased or the volume is reduced. However, this causes high back pressure. Therefore, the optimum design requires. Recently finite element methods are used to obtain flow characteristics and back pressure values of mufflers. Having used of this method, effect of different parameters can be examined without prototyping and best suitable muffler can be determined in the design process. Furthermore time and money can be saved.

It is seen that there is a limited research about studying the flow characteristics in a muffler. In 2011 J. Chen, X. Shi made a research on Fluent, which is a computational fluid dynamics program, on a very basic model which has a perforated inlet pipe and has 4 pipes between expansion chambers [1]. In 2010 X. Hou modeled an exhaust system which consists of pre and after muffler and he analyzed it by using Fluent program. It was shown that the main pressure drop happens in the after muffler and declares that back pressure is 20,8 kPa. He emphasized that when he looked at velocity range in the after muffler according to sudden direction changes of velocity vectors the back pressure increases. It is understood that back pressure can be optimized by size of baffle holes [2]. In 2009 J. Fang studied flow characteristics of exhaust muffler of a steam shovel by finite element methods. Also he verified analysis results with the experiment. Analysis

results showed that inlet total pressure is 256,566 Pa. It is seen that back pressure value reduces when the inlet velocity is decreased. Afterwards Fang gets real pressure loss between muffler inlet and outlet at 20m/s and he compared these values with analysis results. In 2010, Avcu and etc. analyzed the flow characteristics of a dry and wet type exhaust muffler of a diesel-driven ship with Ansys CFX. Analysis results of the dry type muffler shows that the back pressure is approximately 63,7 mbar. With some geometrical changes, the back pressure of the system was decreased to 23 mbar. The same analyses were repeated for wet type muffler and back pressure of it was obtained 22mbar[4]. In 2008, Lee and Ih analyzed flow and acoustic behavior of the different type mufflers which has the same number of perforated hole; but different hole distribution. It was emphasized that the predicted value of the back pressure is less than the measured one and this will not change the performance of the engine so much. They asserted that the error likely arise from the faults during measurement of the velocities.[5] In 2010, Saf investigated the acoustic and flow characteristics of perforated reactive muffler with finite element methods in his master thesis. Pressure loss in expansion chamber is 1330 Pa, in contraction chamber is 1150 Pa. It was argued that the reason of obtained low pressure loss is velocity vectors cannot distribute homogenously. It was indicated that as the velocity increases, pressure loss increases with the square of velocity. As temperature increases, pressure loss decreases some [6]. In 2010, Zheng et.al. analyzed the flow characteristic of four different mufflers with complex geometry. In the study, pressure and velocity distributions of these four mufflers were compared.
In this study, exhaust back muffler of a gasoline engine is analyzed to attain the flow characteristic with the help of computational fluid dynamics (CFD) and new alternative designs are obtained by changing the geometric dimensions of the base muffler. These new alternative models are also analyzed and compared with base muffler. Finally, the various parameters of the muffler were investigated in terms of pressure and velocity distributions.

2. MODEL BUILDING

CAD model of the present muffler which will be examined in this paper is shown in figure 1. As it is seen from figure 1, the present muffler consists of perforated inlet and outlet pipes and two perforated baffles. Furthermore, the muffler is divided into three main parts; front, middle and expansion chamber. Perforated parts of inlet and outlet pipes create a cross flow inside the muffler. Front and back baffles are also perforated.

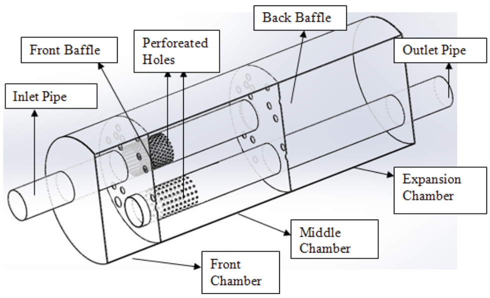


Figure 1.CAD model of base muffler

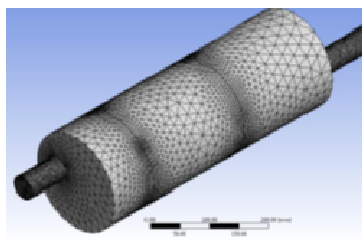


Figure 3. Mesh Generation of Base Muffler

The design changes for the purpose of reducing the total volume of the muffler is made with the help of the relevant literature and taking into consideration various studies. Expansion chamber has an important role for the muffler. In this chamber, jet flow or cross flow occur and acoustic attenuation is obtained by reflecting sound wave to source. In this view, front chamber in the present muffler can be thought as expansion chamber. According to related researches increasing the number of expansion chamber in the mufflers which have more than one expansion chambers enhance the transmission loss of the mufflers; however, the influences on back pressure changes depending on the rate of perforation. Increasing the number of expansion chamber in the same length provides noise attenuation in high frequencies [8]. For this purpose, in this study, three alternative models are attained by decreasing front chamber length, middle chamber length and expansion chamber length of the base muffler with certain percentage. Obtained alternative muffler model can be seen in Figure 2.

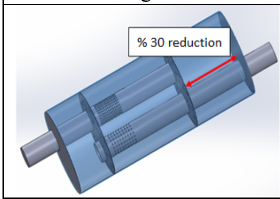
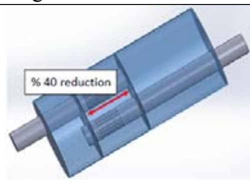
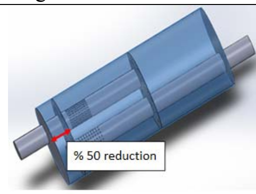
Model I: The main expansion chamber's length is reduced 30%	Model II: The Middle chamber's length is reduced 40%	Model III: The Front chamber's length is reduced 10%
		

Figure 2. Alternative models

3. FINITE ELEMENT METHODS

For flow analysis of the muffler, base muffler is drawn with a CAD program. Ansys Workbench is used to get mesh the muffler models. Tetragonal elements are preferred because of the geometry of the base muffler is very complex. Due to the thin thickness, small dimensions and especially perforated holes, hexagonal mesh is not preferred while meshing the muffler. Face sizing was applied to perforated holes in baffles, pipes and inlet and outlet pipes of the muffler (Figure 3).The element number of the base muffler is 5498964 and node number is 1510889. In addition to this, inflation comment was used because of the boundary layer approach on the outside liner of the muffler [9].
Maximum skewness used for model is 0,88. It is assumed that the flow inside the muffler is turbulent, so k-ε turbulence model is preferred as the mathematical model. Energy equation is defined to the program since temperature will change the density. More exact solutions are tried to obtained with the solution of both equation. Inlet temperature is 473 K and air is used as incompressible ideal gas in analysis. Since the muffler is connected to atmosphere at outlet, the boundary conditions are selected as atmosphere pressure, 101325 Pa, temperature, 343 K and other are default settings. The simulations were performed for five different inlet velocities, namely, 23, 46, 70, 93, and 116 m/s.Exhaust gas is represented as in compressible ideal gas.

4. RESULTS AND DISCUSSION

4.1. Compartions for total pressure

Total pressure contours on the surface passing through the axis of both pipes are shown in Figure 4. Pressure difference contours are given at 70m/s inlet velocity to make a comparison between suggested models.
Pressure distribution of base muffler is presented at Figure 4a. Maximum pressure value observed at inlet. The pressure values at the inlet along the pipe is high and stationary, This results in an easy passing to the middle chamber. There is a same distribution of pressure in the middle and front chambers and due to rapid expansion of the gas, less pressure values occur compared to rear chamber. At the inlet of the outlet pipe, pressure drops slightly and gas leaves the muffler. As shown in Figure 4a, maximum total pressure is observed at the inlet due to inlet velocity and at the point where gas crashes to the wall in the rear chamber. At this situation, maximum total pressure is 10024 Pa.
In Figure 4b which represents pressure distribution of Model I, it is realized that maximum total pressure which is 6426 Pa is less than base model. This model has similarity with base model about pressure loss locations but total pressure lost is less than base model. Reducing the length of middle chamber at Model II decreases gas out from perforated inlet pipe and according to this, pressure increases in rear expansion chamber. Thus, the most pressure lost occurs at Model II.

4.2. Compartions for velocity

Velocity distribution is shown at 70m/s for each geometry in Figure 5. Gas flows to rear chamber losing its velocity. Velocity values are observed to be very close at the inlet of outlet pipe for each geometry. There seems vortexes in rear chamber at base model, Model I and Model II. Gas flow characteristic is expected in this way according to reactive muffler working principal. It is shown that there is a uniform velocity distribution at the inlet of the outlet pipe and velocity increases after perforated part. Moreover, it is expected that there occurs cross flow at perforated locations of inlet and outlet pipes and acoustic performance increases.

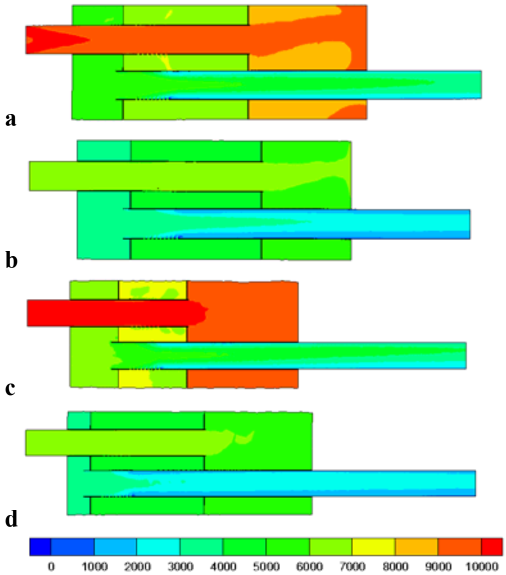


Figure 4. Pressure distribution inside the muffler a)Base Muffler, b)Model I, c) Model II and d) Model III

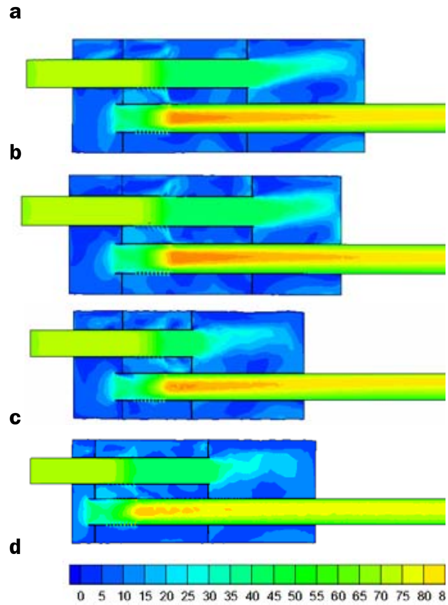


Figure 5. Velocity distribution inside the muffler a)Base Muffler, b)Model I, c) Model II and d) Model III

5. CONCLUSION

In this paper, length of each expansion chambers is studied to understand the effects to the flow characteristics of a cross-flowed perforated and 3-expansion-chambered reactive muffler. It is known that an increase in the total muffler axial length results in a better noise attenuation performance. However, due to the reason of need of a muffler with lower volume for cost reduction, the effect of a cut on length of the muffler is examined.It is observed that 30% reduction on length of rear chamber did not make any difference on acoustic chracteristics with base muffler model. To generate cross flow, inlet and outlet pipe's perforated part stand in the middle chamber. A decrease at the length of middle chamber prevents the cross flow. Thus, It can be concluded that a greater pressure loss occur at this model.

6. REFERENCES

[1] Chen, J. ve Shi, X.: CFD Numerical Simulation of Exhaust Muffler, Seventh International Conference on Computational Intelligence and Security,Wuhan, China.,2008.

[2] Hou, X., Guo, X., Liu, Z., Yan, F. vePeng, F.: Flow Field Analysis and Improvement of Automobile Exhaust System Cold End,Computational Intelligence and Software Engineering (CiSE), 2010 International Conference on, Wuhan, P.R. China,2010.

[3] Fang, J., Zhou, Y., Jiao, P., ve Ling, Z.: Study on Pressure Loss for a Muffler Based on CFD and Experiment, International Conference on Measuring Technology and Mechatronics Automation, China, 2009.

[4] Avcu, M., Kopuz, Ş., Liu, Z. veTeke, M.,: MTU 16V 4000 M90 Brand/Model Diesel Engine Exhaust System Design,Journal of Naval Science and Engineering, Vol. 6, pp. 39-58,201, 2010.

[5] Lee, S.-H. veIh, J.-G.,: Effect of non-uniform perforation in the long concentric resonator on transmission loss and back pressure, Journal of Sound and Vibration,311, 280–296, 2008.

[6] Saf, O.,: PerforasyonluSusturucularınAkışveAkustikEtkinliklerininİncelenmesi, Master thesis, İTÜ, İstanbul,2010.

[7] Zheng Zhong-Cai, Liu Na, Li Jian, GaoYan,ChenHai-Ou,(2010),:Numerical Analysis of Internal Flow in Mufflers with Complex Structures, Applied Mechanics and Materials, Vols. 29-32 (2010) pp 89-94,2010

[8] Munjal, M. L.,:Acoustic of Ducts and Mufflers,John Wiley & Sons, New York,1987.

[9] FLUENT Users' Manual, FLUENT Inc, 2003.