

# Computational fluid dynamics analysis of a resistance muffler

DRAGOS TUTUNEA

Department of Vehicles, Transportation and Industrial Engineering  
University of Craiova  
Street Calea Bucuresti, number 107, Code 200512, Craiova, Dolj  
ROUMANIA  
[dragostutunea@yahoo.com](mailto:dragostutunea@yahoo.com)

MADALINA CALBUREANU

Department of Applied Mechanics and Civil Engineering  
Faculty of Mechanical Engineering- Industrial and Agricultural Civil Engineering, Craiova  
ROMANIA  
[madalina.calbureanu@gmail.com](mailto:madalina.calbureanu@gmail.com)

LUNGU MIHAI

University of Craiova, Faculty of Electrical Engineering  
Electric, Energetic and Aerospatiale Engineering Department  
[lma1312@yahoo.com](mailto:lma1312@yahoo.com)

**Abstract:** - Resistance muffler is the principal device to attenuate the exhausted noise of engines used in present. In this study, a resistance muffler was designed for automotive industry. The Computational Fluid Dynamics (CFD) method was used to analyse the effect which the internal flow field has on the performance of the muffler. With this method the pressure distribution in the muffler is simulated and the pressure loss was predicted. The integrated performance of design and construction of muffler has been advanced.

**Key-Words:** resistance muffler, CFD, flow field, pressure loss, automotive, design.

## 1 Introduction

The muffler is defined as a device for reducing the amount of noise emitted by a machine. To reduce the exhaust noise, the engine exhaust is connected via exhaust pipe to silencer called muffler. The various types of mufflers [1] used in automobiles are:

- Baffle type;
- Resonance type;
- Wave cancellation type;
- Combined resonance and absorber type;
- Absorber type mufflers.

Virtually all reciprocating internal combustion engines are fitted with mufflers. The muffler fitted to an engine is intended to reduce the pressure pulses associated with the exhaust gas leaving the cylinders of the engine. Generally mufflers fitted to such engines are essentially reactive devices as opposed to being dissipative devices. Reactive mufflers operate by the destructive interference of the acoustic waves propagating within them. Dissipative mufflers operate by the dissipation of

acoustic energy, usually within porous fibrous materials. Practical reactive mufflers also have some dissipative function. An ideal muffler for a reciprocating internal combustion engine should function as a low pass filter. The steady or mean flow should be allowed to pass unimpeded through the muffler while the fluctuating flow which is associated with the acoustic pressure fluctuation is impeded. If the steady flow is not significantly impeded the so-called 'back pressure' will be very low and the engine will function more efficiently. It is desirable to be able to predict the pressure drop associated with the steady flow through the muffler [2]. Perforated tubes are commonly used inside automotive mufflers. They can be found in the form of perforated tubes to confine the mean flow in order to reduce the back-pressure to the engine and the flow generated noise inside the muffler. They can also be used to provide resistive damping to enhance the acoustic performance. On the other hand, if the flow is forced through the perforations, this provides a significant back pressure on the

engine. Being able to theoretically model perforated mufflers enables car manufacturers to optimize their performance and increase their efficiency in attenuating engine noise, and at the same time to minimize the back pressure exerted on the engine [3]. Reactive and resonance silencers, which are commonly used in automotive applications, reflect the sound waves back towards the source and prevent sound from being transmitted along the pipe. Reactive silencer design is based either on the principle of a Helmholtz resonator or an expansion chamber, and requires the use of acoustic transmission line theory [4]. There an increasing trend to use computer simulations in the design of various products. This is in special due to the demands on shortened time to market, higher product performance and greater product complexity. To be useful in the design process it is important that the simulation models are kept as simple as possible while still being accurate enough for the characteristics they are supposed to describe. To reveal weaknesses in the simulation models experimental investigations is often necessary [5]. Computational Fluid Dynamics (CFD) has been around for quite some time for prediction can be made reliably. It is also desirable to be able to predict the acoustic performance of the muffler. The present paper is concerned with describing how CFD can be used for this purpose.

## 2 Problem Formulation

The exhaust system is defined as the hardware necessary to vent the exhaust from the vehicle beginning at the exhaust plane defined by the engine manufacturer and necessary to isolate the exhaust thermally from vehicle structures. The virtual design of the exhaust muffler, as a minimum, include an accurate estimate of space required for the exhaust, backpressure to the engine, system weight, gas species distributions, gas temperature distributions, the interaction of the plume with external surfaces both on the vehicle and the ground, and the thermal interaction of the exhaust system with external surfaces through internal convection, conduction, and radiation [6]. CFD analysis of exhaust mufflers can take weeks to converge and are very difficult to model since the flow regimes include high subsonic compressible internal flow, low velocity wake regions, and regions with high vortices. To be useful in the design process, the CFD analysis must be responsive to the design process. The exhaust muffler was designed in Solidworks Fig. 1 and has the following material characteristics (Table 1).

Table 1 Steel Stainless 302

Property	Units	Value
Density	kg/m <sup>3</sup>	7900
Specific heat	J/(kg*K)	500
Conductivity type	isotropic	isotropic
Thermal Conductivity	W/(m*K)	16,2999

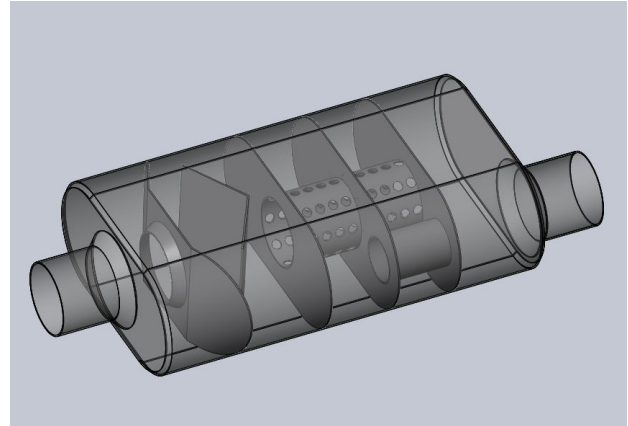


Fig. 1 Solidworks model of designed muffler

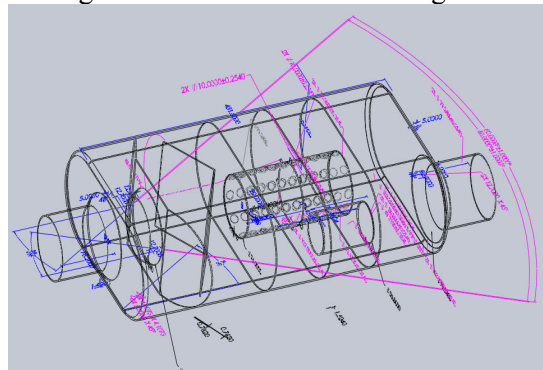


Fig. 2 Main dimension of the muffler

## 3 Problem Solution

The designed exhaust manifold is designed by reducing the angle of bend pipes throughout the exhaust gases path to reduce the air friction. After the construction of exhaust muffler model in Solidworks the model is analyzed with Cosmos Flow from Solidworks. As initial data we consider the flow of burning gases at a speed of 25 m/s and a temperature of 280°C. For the environment condition is selected an ambient temperature of 20°C and a normal atmospheric pressure. Due to the high subsonic compressible internal flow the flow simulation was computed on an I7 Intel Core System with 6 Gb. of RAM. The solution has converged after 2745 iteration.

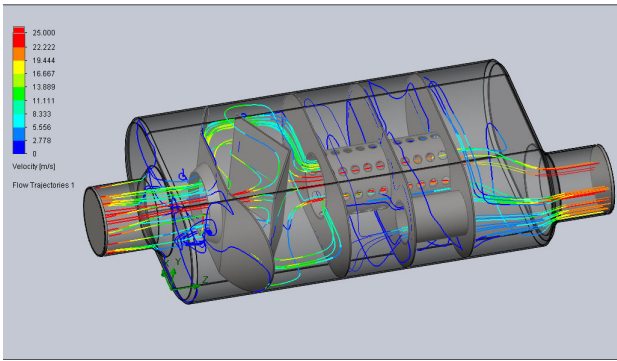


Fig. 3 Muffler velocity

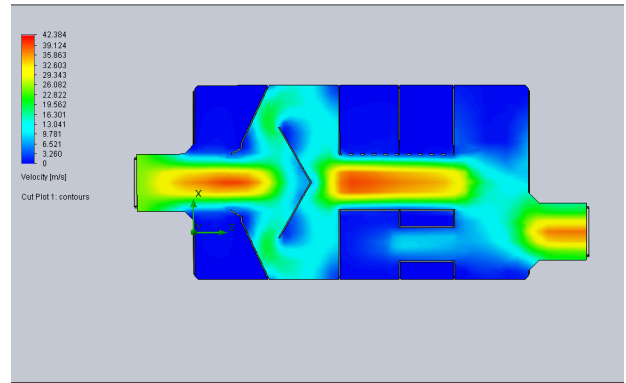


Fig. 7 Cut plot of velocity field

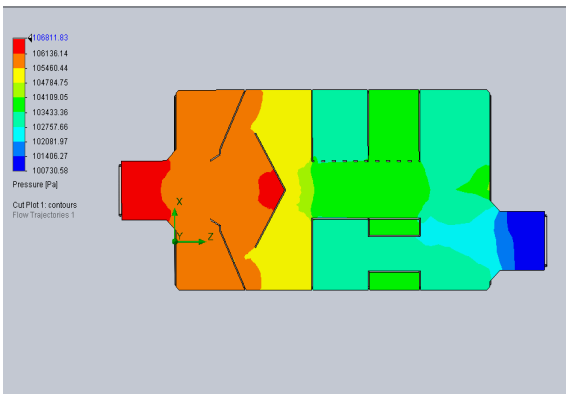


Fig. 4 Cut plot of pressure field

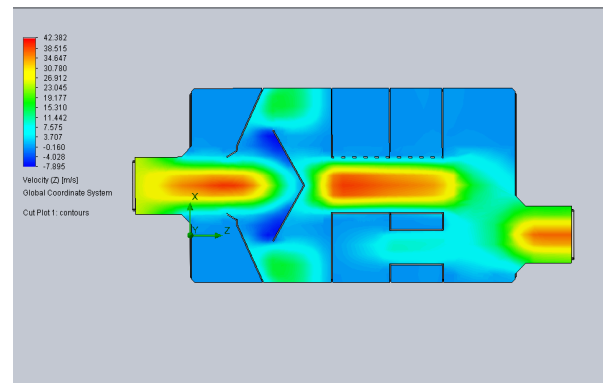


Fig. 8 Cut plot of z velocity field

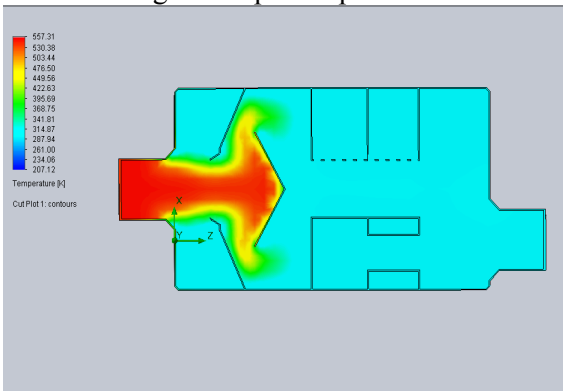


Fig. 5 Cut plot of temperature field

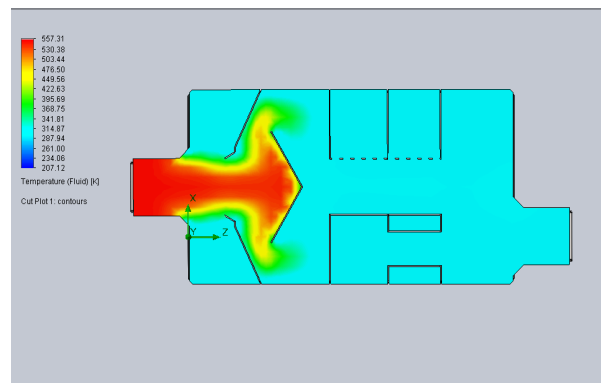


Fig. 9 Cut plot of fluid temperature field

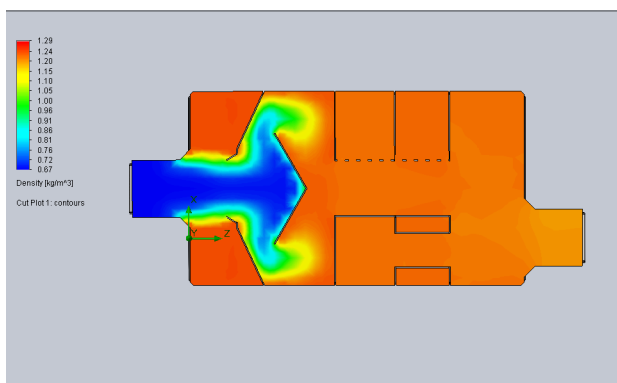


Fig. 6 Cut plot of density field

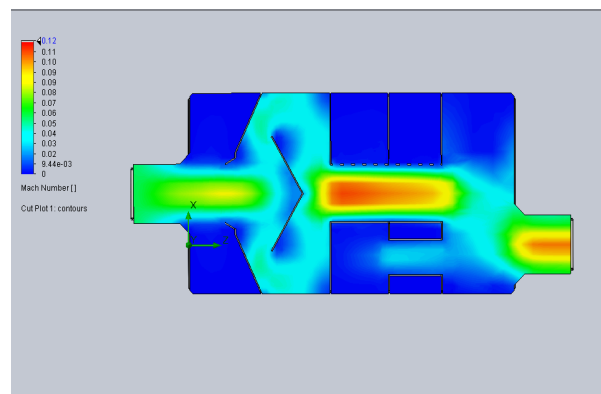


Fig. 10 Cut plot of Mach Number field

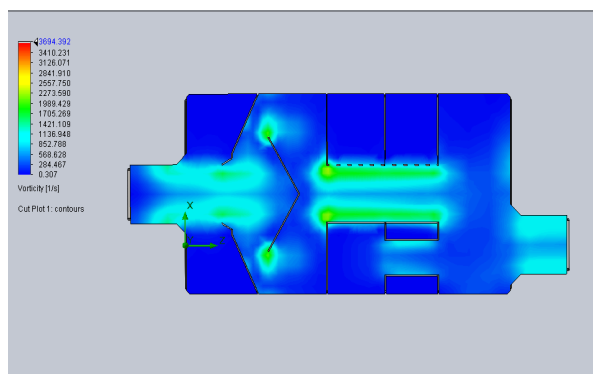


Fig. 11 Cut plot of vorticity field

The mean flow performance of the muffler considered in the acoustic analysis has been analyzed the local and integral parameters of different surfaces of the exhaust muffler. The results of the simulated muffler obtained with the use of CFD modeling are very promising.

#### 4 Conclusion

The CFD simulation software can be used in designing and simulations of the automobile exhaust system. Apart from that, it can also be used to design any other parts in automobile applications to simulate the flow in real condition. 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.

#### References

- [1] M.Rajasekhar Reddy, Dr K.Madhava Reddy, *Design And Optimization Of Exhaust Muffler In Automobiles*, International Journal Of Engineering Research and Applications (IJERA) ISSN: 2248-9622, Vol. 2, Issue 5, September- October 2012, pp.395-398.
- [2] Middelberg, J.M., Barber, T.J., Leong, S. S., Byrne, K.P and Leonardi, E., *Computational fluid dynamics analysis of the acoustic performance of various simple expansion chamber mufflers*, Proceedings of Acoustics, 3-5 November 2004, Gold Coast, Australia.
- [3] T. Elnady, S. Elsaadany and M. Åbom, *Investigation into modeling of multi-perforated mufflers*, 16th International Congress of Sound and Vibration, Krakow (2009).
- [4] Bankim Bihari Ghosh, Paritosh Bhattacharya, Rajsekhar Panua and Prabir Kumar Bose, *Prediction of Noise Level by Mathematical*

assessed. The aerodynamic simulation was used to observe the streamlines and contours of velocity, vorticity, density, Mach number and temperature of the muffler. The burning gases enters with high temperature at inlet pipe, the heat is dissipate on the muffler structure and exit with an average temperature of 50 °C. At inlet pipe the burning gases have a high pressure which decreases with the recirculation of the gases through the perforated pipes of the exhaust muffler. The velocity field show a decreases of speed of burning gases in the exhaust muffler with yhe increase of the speed on the superior part of outlet pipe. In the simulation are

*Modeling in the Exhaust Muffler and Validation of these Analytical Results with the Experimental Results for 4-Stroke Diesel Engine*, Advances in Applied Mathematical Analysis, ISSN 0973-5313 Volume 2 Number 1 (2007), pp. 41–49.

- [5] Englund T., Wall J., Ahlin K., Broman, G., *Significance of nonlinearity and component-internal vibration in an exhaust system*, Proceedings of the 2<sup>nd</sup> WSEAS International Conference on Simulation, Modelling and Optimization, Skiathos Island, Greece, 2002.
- [6] James L. Means, Lance C. Grace “Suppression of Thermal Emission from Exhaust Components Using an Integrated Approach“ Proceedings of the Ground Target Modeling and Validation Conference [13th] Held in Houghton, MI on 5-8 August 2002.