The computational fluid dynamics (CFD) study of fluid dynamics performances of a resistance muffler

D. Tutunea, M.X. Calbureanu and M. Lungu

***Abstract*—** The exhaust pollution has become one of the important problems of environment pollution with applications in automobile industry, and the exhausted muffler has been paid attention to improve the performance of engines. Computational Fluid Dynamics (CFD) method was used to explore the aerodynamic performance of the muffler. The dimensions of muffler design should be limited in the demand of assembly with simple structure. Resistance muffler research relates with the fields of acoustics, fluid dynamics, heat transfer and mechanism design. The author simulated the field by numerical method with Cosmos Flow and analyzed the effect which the internal flow field has on the performance of the muffler, which may be a credible guidance of the muffler structural design. With this method the pressure distribution in the muffler is simulated and the pressure loss is predicted for the structure modification. The experiment results verify that the assembly performance of the muffler modified is better than the original muffler.

***Keywords*—** CFD, design, flow field, pressure loss.

I. Introduction

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 is 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. Also the simulation models can then be updated to better correlate with experimental results [1,2,3,4].

CAD systems and their geometric representations have been around for quite some time. Almost all CAD systems have involved into similar representations for their models. One of the most important aspects to mesh Generation is accessing CAD geometry. CAD systems often use relatively large tolerances on basis to provide model operations. This approach is referred to as variable tolerances and modeling by different CAD systems. The use of these large variable tolerances produces gaps and overlaps in the geometry and topology of the CAD system. The major issue with CAD geometry access for mesh generation is the need to understand the analysis requirements. An appropriate mesh and geometry is to be used for meshing. Meshing is a function of the analysis to be performed and the desired accuracy. There does not is an optimal mesh independent of the analysis to be performed. The element shape quality test for good mesh independent of the analysis is to be performed for the accuracy desired. The appropriate mesh is one that produces the desired accuracy for the problem to be solved [5,6,7].

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 [8] 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

D. Tutunea is with the University of Craiova, Faculty of Mechanics, Department of Vehicles, Transportation and Industrial Engineering, Calea Bucuresti, 107, Zipe code 200512, Craiova, Dolj (corresponding author to provide phone: 0745523262; e-mail: dragostutunea@yahoo.com).

M.X. Calbureanu, is with the University of Craiova, Faculty of Mechanics, Department of Applied Mechanic and Civil Construction, Calea Bucuresti, 107, Zipe code 200512, Craiova, Dolj (e-mail: madalina.calbureanu@gmail.com).

M. Lungu is with the University of Craiova, Faculty of Electrical Engineering Electric, Department of Energetic and Aerospatiale Engineering (e-mail: mlungu@elth.ucv.ro).

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 [9]. 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 [10]. 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 [11]. 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 is 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 [1]. 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. Automotive mufflers come in all different shapes, styles and sizes depending on the desired application. Generally automotive mufflers consist of an inlet and outlet tube separated by a larger chamber that is oval or round in geometry. The inside detail of this larger chamber may be one of numerous constructions. The end user of the muffler usually does not care what is inside the chamber so long as the muffler produces the desired sound and is aesthetically pleasing. It is therefore the task of the muffler designer to ensure that the muffler is functional as well as marketable [12]. In this study the authors proposed a method to design a complex internal muffler for automotive industry with focus on the main steps of CAD construction.

Fig. 1 Typical muffler design [13]

II. 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 [14]. 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. 2,3 and has the fallowing material characteristics (Table 1).

Table 1 Steel Stainless 302

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

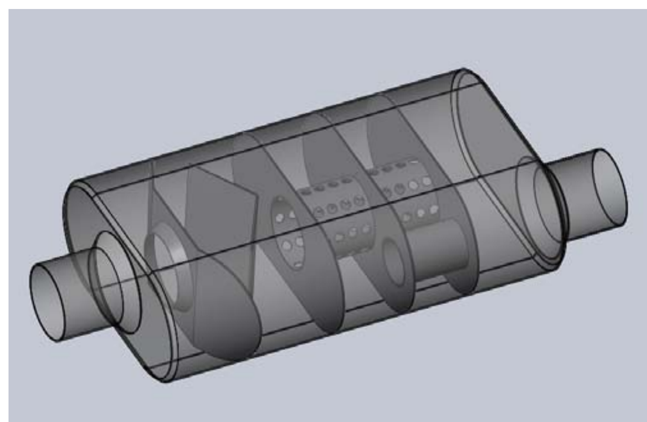

Fig. 2 Solidworks model of designed muffler

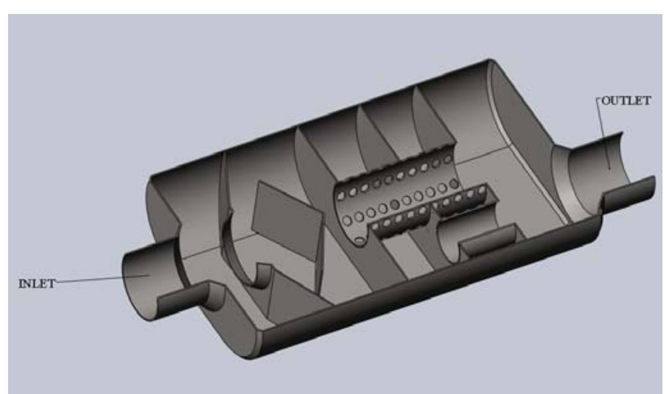

Fig. 3 Muffler section view

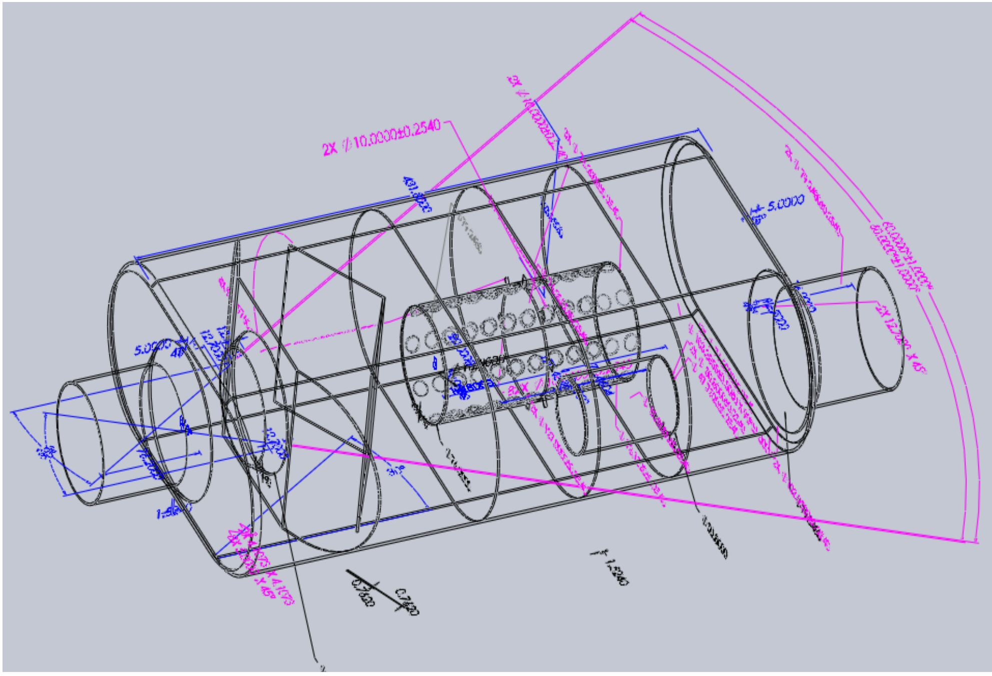

Fig. 4 Main dimension of the muffler

III. 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.

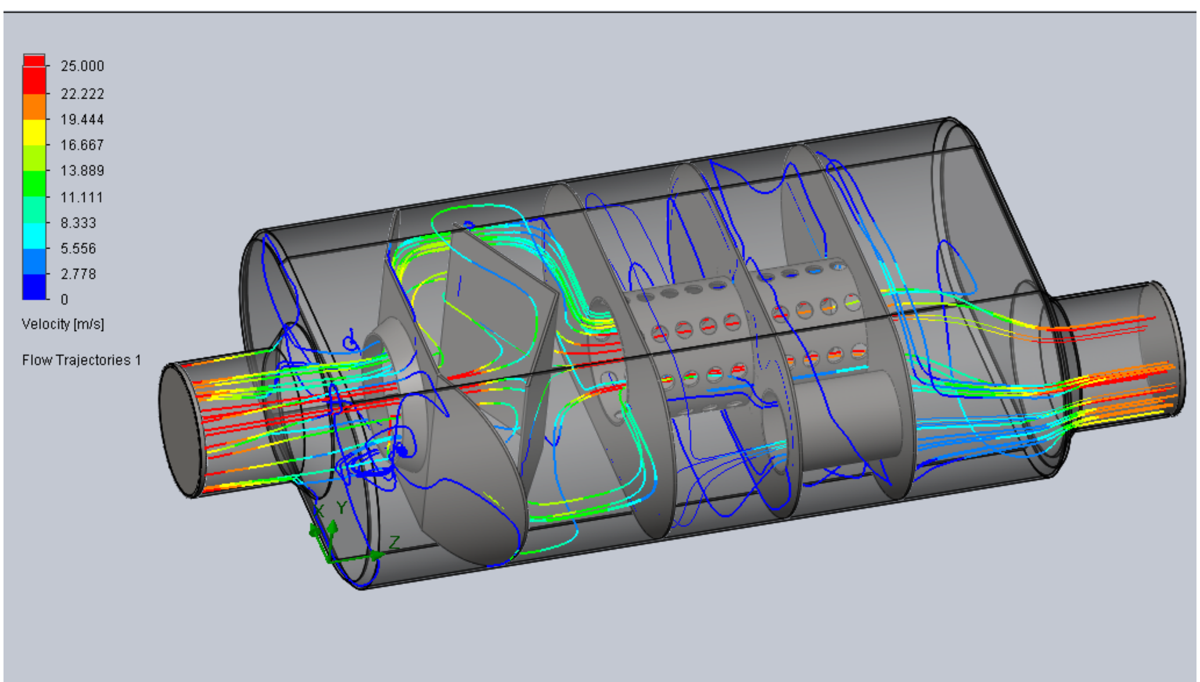

Fig. 5 Muffler velocity

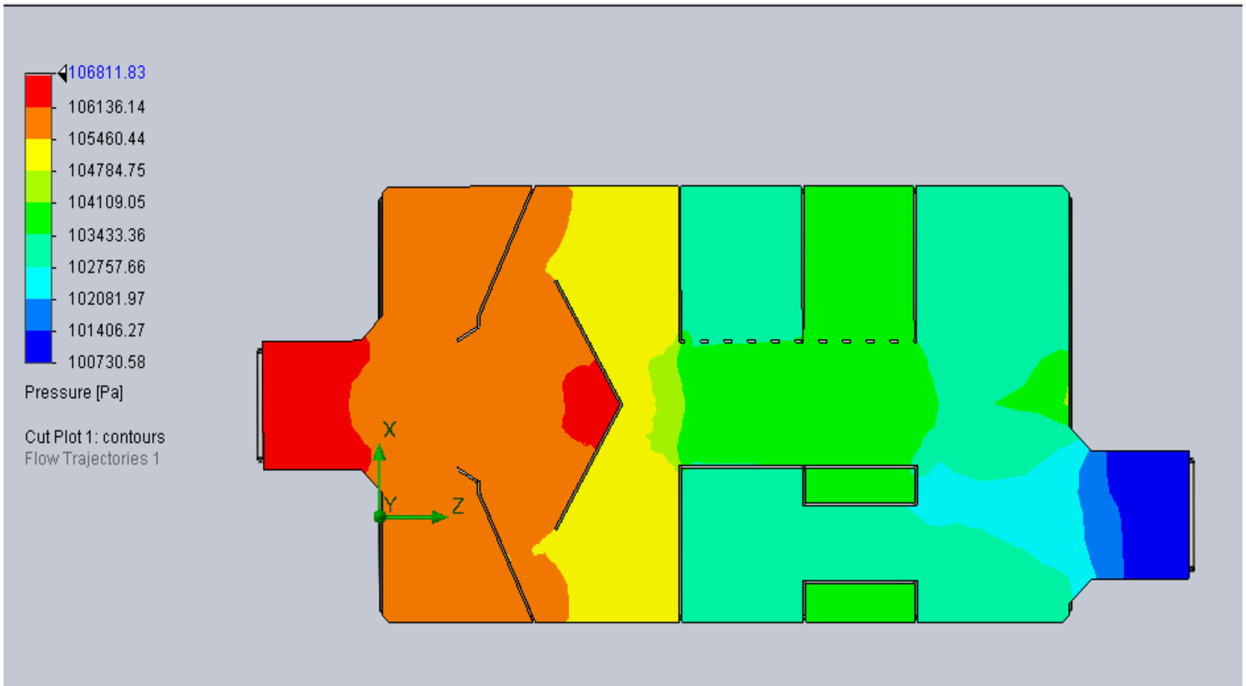

Fig. 6 Cut plot of pressure field

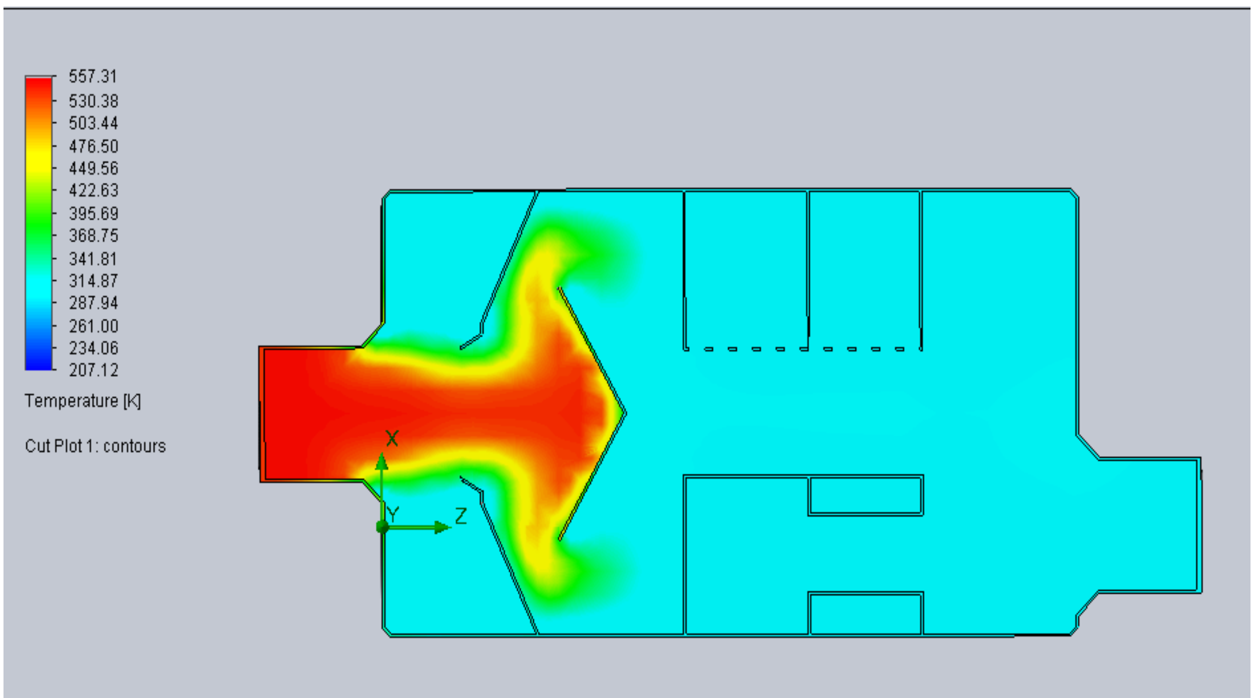

Fig. 7 Cut plot of temperature field

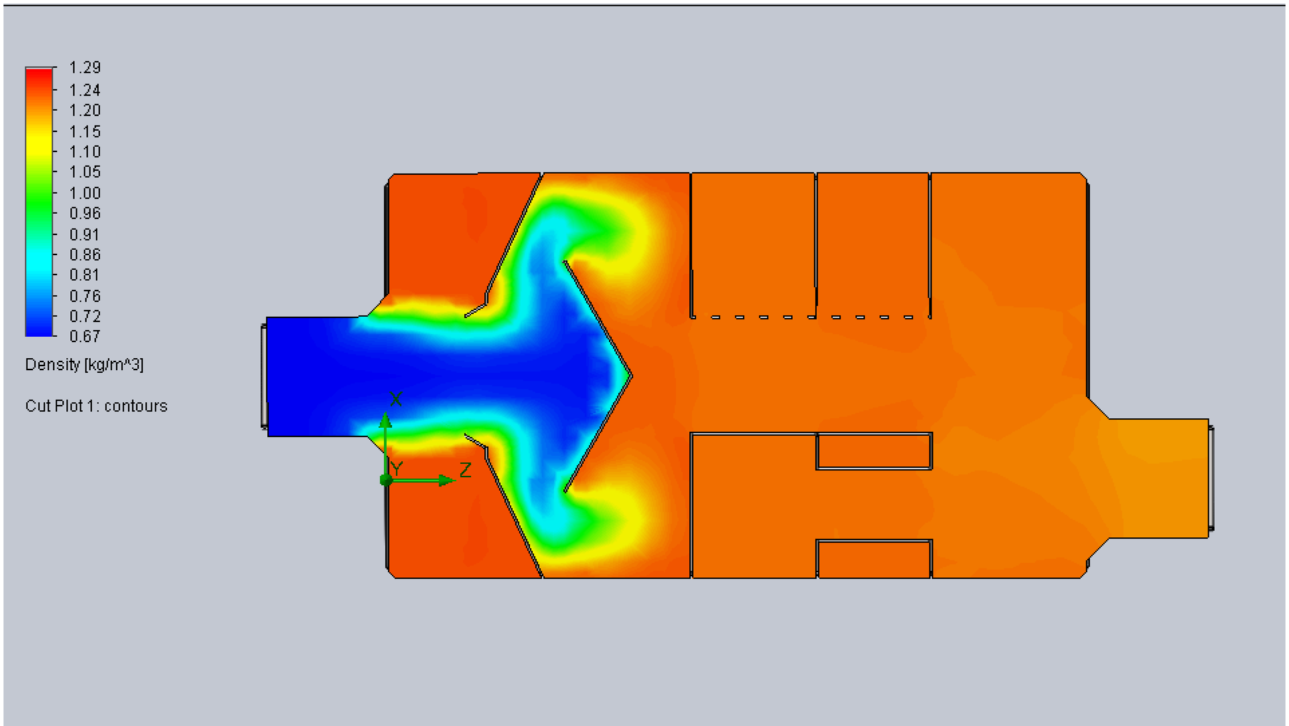

Fig. 8 Cut plot of density field

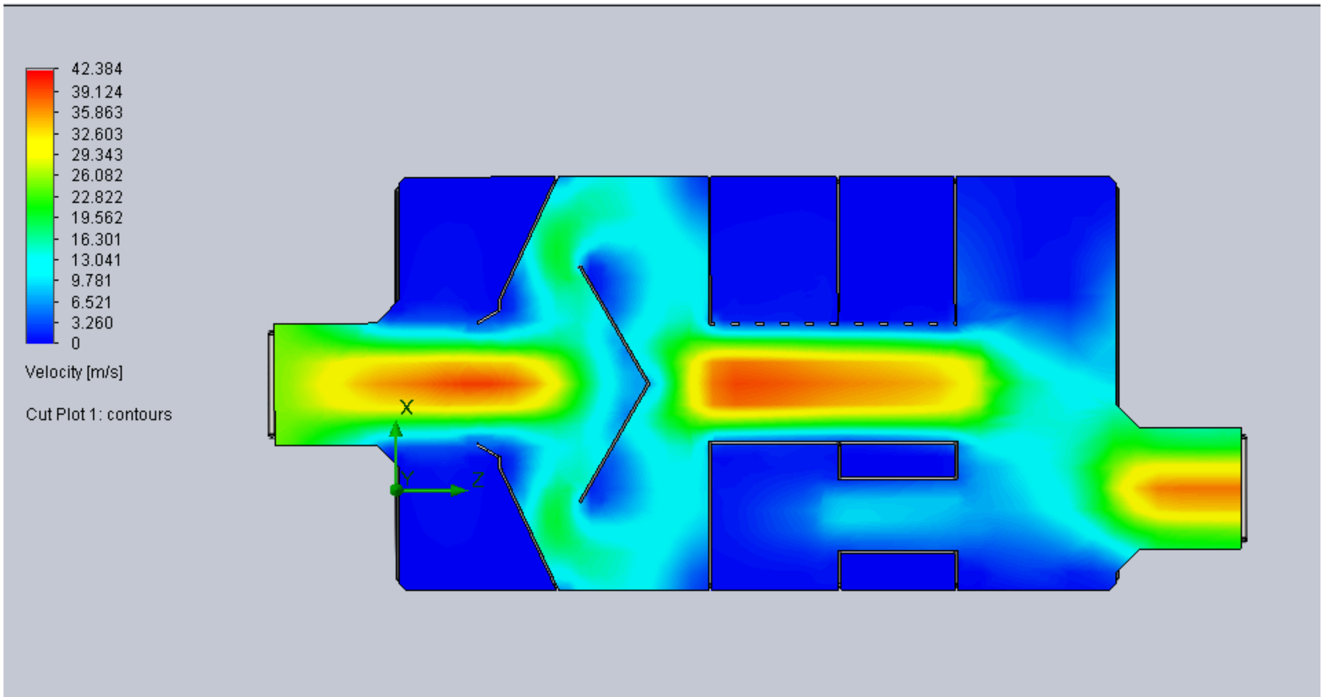

Fig. 9 Cut plot of velocity field

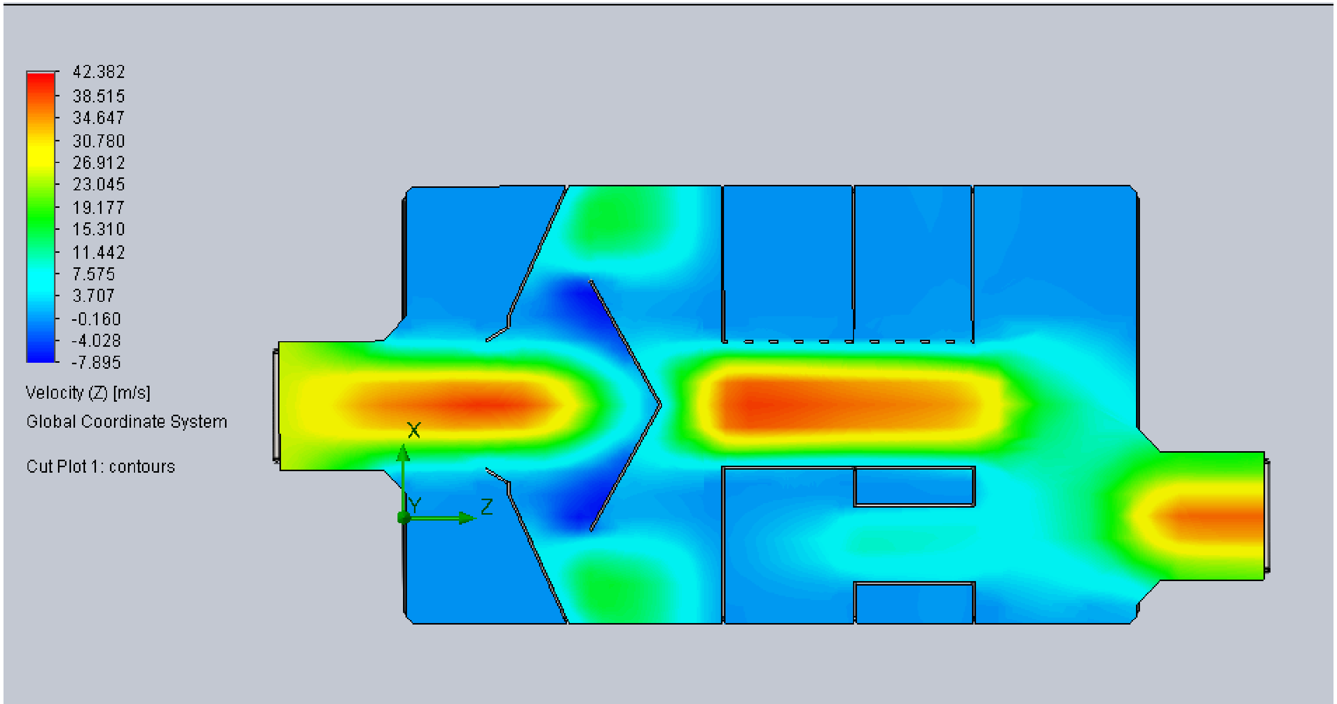

Fig. 10 Cut plot of z velocity field

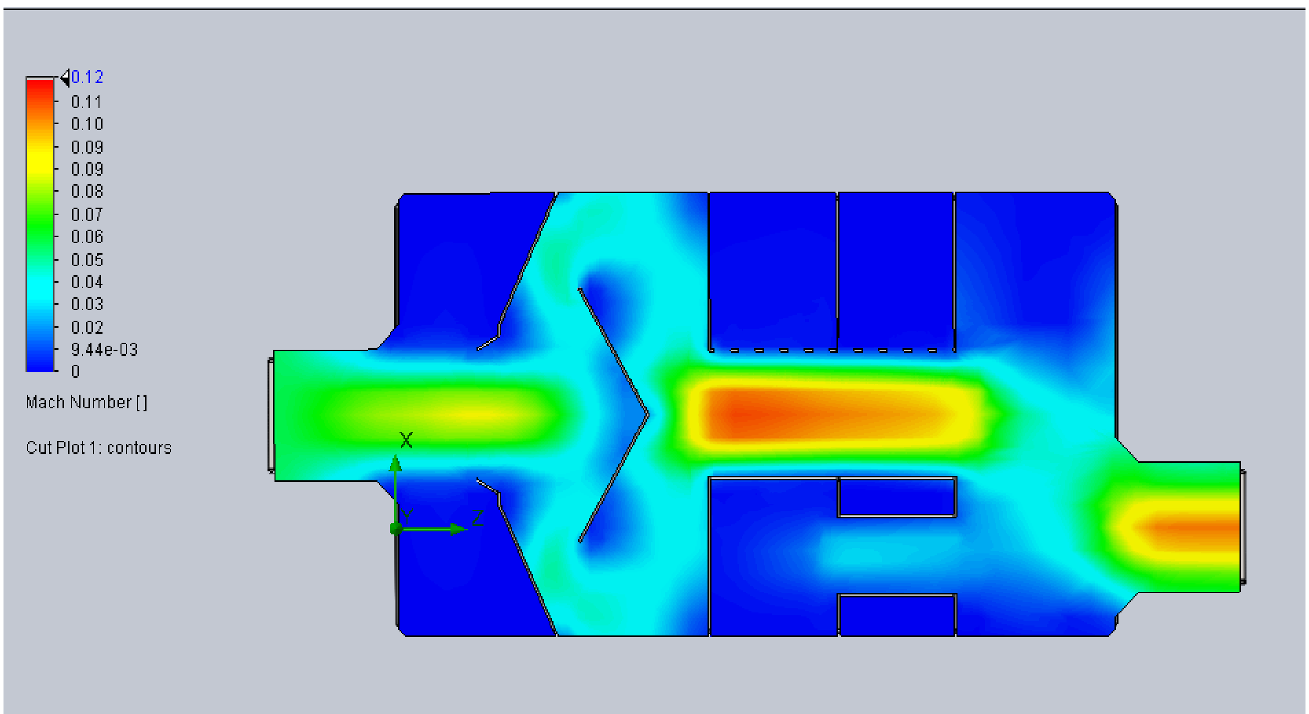

Fig. 11 Cut plot of Mach Number field

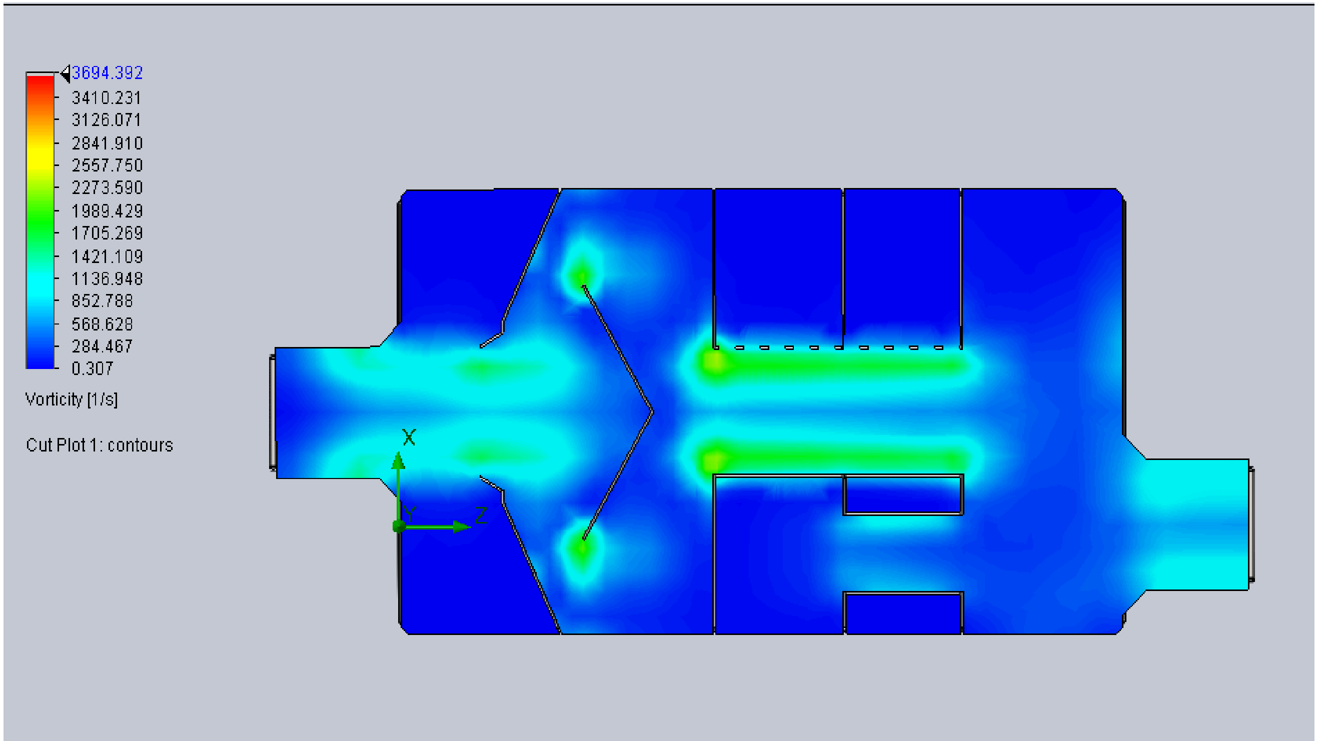

Fig. 12 Cut plot of vorticity field

The mean flow performance of the muffler considered in the acoustic analysis has been 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 shows decreases of speed of burning gases in the exhaust muffler with the increase of the speed on the superior part of outlet pipe. In the simulation are 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.

IV. 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] T. Englund, J. Wall, K. Ahlin, G. Broman, Signifiance of nonlinearity and component-internal vibration in an exhaust system, Proceedings of the 2nd WSEAS International Conference on Simulation, Modelling and Optimization, Skiathos Island, Greece, 2002.

[2] V. Tut, A. Tulcan, C.Cosma, and I. Serban, Application of CAD/CAM/FEA, reverse engineering and rapid prototyping in manufacturing industry, International Journal of Mechanics, Issue 4, Volume 4, 2010, pp. 69-86

[3] Robert Rajca, Lukasz Matysiak, Michal Banas, Robert Sekula, A Novel Simulation Approach for Analyzing Reactive Molding Process, International Journal of Mathematics and Computers In Simulation, Issue 4, Volume 4, 2010, pp.99-106

[4] Baron C., Esteve D., Rochet S., How evolutionary computation can be introduced to select and op-timize scenarii along a product design process, WSEAS Transactions on Systems, Issue 2, Volume 3, April 2004

[5] Tsamasphyros G., Vrettos C., Numerical simulation of flow through porous media. A control volume approach in combination with higher order finite elements, Proceedings of the 4th IASME/WSEAS International Conference on Continuum Mechanics (CM'09).

[6] Iliescu M., Comănescu B., Nuțu E., "Rapid Prototyping Use in Manufacturing components of a Medical Laser device", 2nd WSEAS International Conference on Additive Technologies ICAT 2008, DAAAM Specialized Conference, ISBN 3-901509–72-0, Ptuj, Slovenia, september, 2008

[7] Dr. S. Rajadurai, Suresh Natarajan, N. Manikandan, Muffler Pre-Processing Methodology and Comparative Study Using HyperMesh, Simulation Driven Innovation, HTC 2012.

[8] 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.

[9] 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.

[10] T. Elnady, S. Elsaadany and M. Åbom, Investigation into modeling of multi-perforated mufflers, 16th International Congress of Sound and Vibration, Krakow (2009).

[11] Bankim Bihari Ghosh, Paritosh Bhattacharya, Rajsekhar Panua and Prabir Kumar Bose, Prediction of Noise Level by Mathematical

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.

[12] Potente Daniel, General Design Principles for an Automotive Muffler, Proceedings of Acoustics 2005, 9-11 November 2005, Busselton, Western Australia.

[13] http://auto.howstuffworks.com/muffler3.htm, Accessed: 2013-01-22.

[14] 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.