American Journal of Engineering Research (AJER)
e-ISSN: 2320-0847 p-ISSN : 2320-0936
Volume-6, Issue-2, pp-20-27
www.ajer.org
Research Paper Open Access

Numerical Analysis of the Thermal and Aerodynamic Influence in an Automotive Exhaust System Using Computational Fluid Dynamics

G. de P. Rúbio[(1)],R. L. P. Teixeira[(1)], R. A. M. Carrillo [(1)], V. T. Signoretti[(1)], J.C. de Lacerda [(1)*], R. F. Brito [(1)]
[(1)]*Universidade Federal de Itajubá – Campus Itabira, 35903-087, Itabira MG, Brasil*

ABSTRACT: *This study aims to analyze the thermal and fluid dynamics profile of an automotive engine's exhaust system in order to identify points to be improved in the project. The exhaust system is one of the automotive components working under the most severe conditions of thermal transfer, as the passage of combustion gases from the engine operation takes place in this system. The exhaust system must be designed for better performance from the knowledge of the heat transfer inside it and with a view to identify critical points in terms of corrosion, thermal stress, fatigue, among others. Therefore, this work has carried out a computer simulation by the finite volume method within the system of heat exchange and mass flow from the fuel gas in an automotive exhaust system of a 2004 Volkswagen Gol GT vehicle, with the engine speed at 1000rpm.*

Keywords: *Numerical Analysis, Computational Fluid Dynamics, Exhaust System.*

I. INTRODUCTION

The exhaust system, or tail pipe system, is an automotive component that works under the most severe conditions of thermal transfer, as the passage of combustion gases from the engine operation occurs in this system. This system must be carefully designed to obtain better performance. It is essential to know how the heat transfer happens inside the system to foresee critical points in terms of corrosion, thermal stresses, and fatigue, among others. For the use of computational fluid dynamic (CFD) techniques, the knowledge of the type of flow is necessary, as well as the types of heat transfer, so that we can choose the most compatible physical simulation with more reliable results.

The exhaust system is one of the components present in an automotive that works in an extreme situation of thermal transfer. This situation may lead to faults such as deformations and cracks [1]. Automotive engineering continues to aim toward producing exhaust systems with a longer useful life and with lower pollutant emissions, as can be seen in current vehicles [2]. Even with new technologies, about 30 to 40% of the energy coming from the burning of fuel is lost in the exhaust systems [3]. Therefore, to know how the heat exchange happens is of vital importance so that we can understand how and why deformations and cracks happen in the component, besides increasing the useful life of a component and improving its performance.

The exhaust system is made of the combination of the expansion chamber (baffle and muffler), conductive tubes, and perforated pipes and sheets [4]. Therefore, a system like this needs a reliable project and analysis, due to high thermal and mechanical requirements they have to bear, besides accomplishing requirements of weight, cost, and useful life. This is a favorable condition for a thermal and mechanical analysis using the finite element method [5]. Commercially available computer-assisted engineering (CAE) programs may perform this analysis with a high degree of reliability, assuring an efficient analysis of the several thermal and mechanical profiles where the solution of mass conservation equations, quantity of movement, and energy in the permanent regime use the Calculus of Fluid Dynamics by the Finite Volume Methods (FVM) with the Euleriano scheme for the discretization of the physical domain, using a finite number of control volumes [6-7]. A higher degree of details on the concepts involved in the FVM, may be found in the works of Barth and Ohlberger [8], which explore discretization techniques, integral approach techniques, convergence criteria, and calculus stability. As illustrated by the studies of Kandylas and Stamatelos [9], computational methods allied with lab data have enabled the evaluation of several thermal fluid dynamic profiles associated with the pipe walls of an automotive exhaust system.

As proposed, a numerical analysis of an automotive exhaust system was performed in order to obtain the thermal aerodynamic field of the exhaust gases, aimed at determining the highest and the lowest temperature gradients. Such data may be used, for instance, to know the points where corrosion may occur in the system. Chemical corrosion occurs in high temperatures, in the absence of water, whereas, the process of electrochemical corrosion, more frequent in nature, necessarily involves the presence of water and the transfer of electrons. This spontaneous process occurs due to the chemical potential difference between metal and the ambient, involving the reaction of these materials present in the ambient with aggressive substances for metal (O_2, H_2S and CO_2) [10]. Moreover, the exhaust system, due to being a complex part where combustion gases in high temperature are conducted, has areas subject to several corrosive influences, such as the outside ambient, high temperatures, output residues and many other interferences that may lead to erosive chemical actions of the metallic material. All this, besides contributing to corrosion, may cause the functional and structural loss of the exhaust system [11].

Before starting the computational modeling process, it is important to have a good understanding of what occurs with the system physics. It is known that the exhaust system is subject to two main ways of heat transfer: conduction and convection. The conduction may be seen as an energy transfer from the most energetic to the least energetic particles of a similar substance, due to the interaction among their molecules. Heat transfer by convection (natural or forced) happens through the diffusion from the random molecular movement or the global movement of its molecules. This type of molecular movement is commonly observed when the heat transfer happens from a surface due to the flow of a specific fluid [12].

Another important aspect to be observed in the study of the exhaust system is in terms of the laminar or turbulent flow regimes. Identification of the flow regime may be made through the number of Reynolds in the system: 2000 (laminar), 2000 and 2400 (laminar-turbulent transition), and higher than 2400 (turbulent) [13]. Turbulent flows are not of easy resolution; therefore, an adequate computational modeling to each physical problem is needed [14].

This paper presents a simulation performed with CFD software for the 1000 revolutions per minute of an engine of the proposed exhaust system, using a geometry of reference and a chosen mesh. The analysis of the data obtained and the discussion about what can be improved in the system are provided below.

II. PROPERTIES OF THE COMBUSTION GAS

Through Equation 1, using REFPROP® software, thermal physical properties foreseen in the gas in the exhaust system were obtained, as can be seen in Table 1. These properties were used in the ANSYS CFX® software taking into account a boundary condition with a pressure of 0.098MPa.

Table 1 - Properties of the combustion gas in the exhaust system.

RPM	T [ºC]	P [MPa]	ρ[kg/m³]	h [kJ/kg]	s [kJ/kg.K]	c_p [kJ/kg.K]	c_p/c_v	K [mW/m.K]	μ[μPa.s]	ν [cm²/s]
1000	67.4	0.098	0.99854	367	6.7578	1.0247	1.393	28.664	19.924	0.19953

III. GEOMETRY AND MESH USED

Using the Solidworks® software, the computational domain was designed representing only the volume of gas inside the system. For the design, we considered the exhaust system of a Volkswagen® Gol GT® vehicle, as shown in Figure 1(a).

To obtain the mesh, the ANSYS CFX Mesh® software was used. An automatic mesh with the maximum number of possible elements was used (1,722,071 tetrahedral) with 326174 knots, as can be seen in Figure 1(b). A study of the influence of the computational mesh refining was performed through the results obtained. The mesh with 1,722,071 tetrahedral obtained the lowest variance of results when compared with the mesh with the least elements.

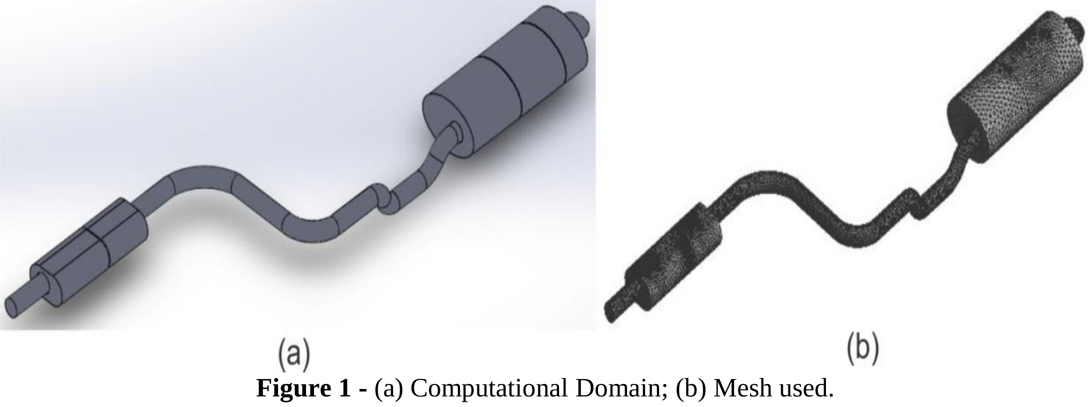

Figure 1 - (a) Computational Domain; (b) Mesh used.

IV. BOUNDARY CONDITIONS

At point 1, represented in Figure 2, the Opening condition was adopted, as this point represents the exit of the exhaust gases. Such condition was used so that it was possible to compare the velocity obtained in the numerical analysis with those obtained through experiments. In the system input, an Inlet condition with a mass flow system established as 6.7×10^{-3} kg/s was adopted. The mass flow was determined by multiplying the volumetric flow obtained in Table 2 by the gas specific mass determined in Table 1 (temperature of reference 67.4°C). The input temperature was 352°C, obtained through experiments. The other surfaces were considered as walls with the heat exchange by natural convection with the external environment being the heat transfer coefficient by convection of h =20 W/m^{2}K (air ambient in temperature of 25°C). The turbulence model used was k-Epsilon, which was adopted as we could not know precisely if the flow was totally turbulent. The k-Epsilon turbulence model can be applied with precision for the laminar and the turbulent flow and with the coexistence of both, as it is the most used model currently in CFD simulators and does not demand homogeneous boundaries [15].

V. EXPERIMENTAL PROCEDURES

Temperatures and average velocities of the gas exits and flows were measured at the points presented in Figure 1, for1000rpm. For the experimental measures, the following instruments were used: Digital infrared thermometer (Instrutherm brand, model TI-920, resolution 0.1ºC, scale -50ºC to 1600ºC and precision of ±1.5%) and Portable Digital Anemometer Term (Instrutherm brand, TAD-800 model, resolution 0.01 on a scale of 0.60 m/s to 30.00 m/s and error of ± 3%).

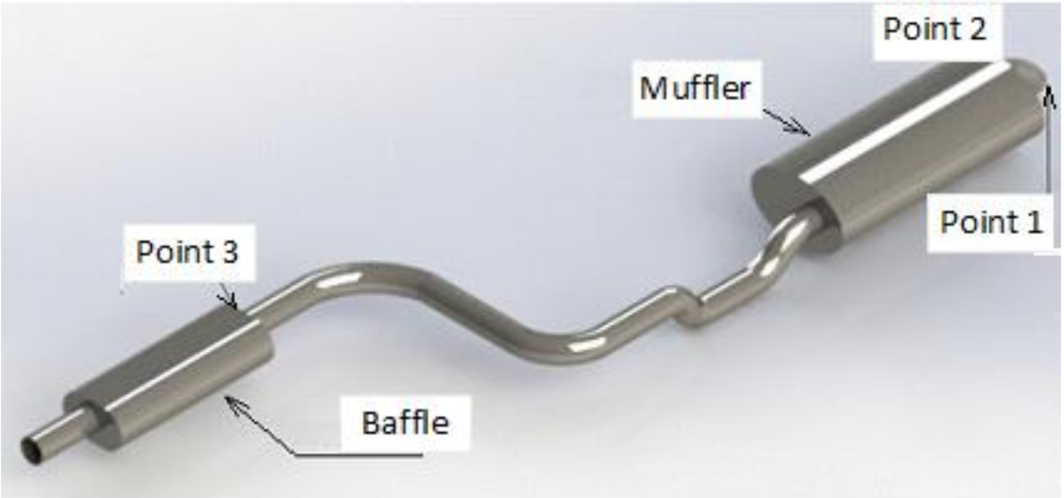

Figure 2 – Measuring points in the exhaust system.

The experimental data obtained are presented in Table 2. The flows were determined from the diameter of the flow pipe (46.68mm).

Table 2 - Experimental temperatures, velocities, and volumetric flows.

POINT	AVERAGE TEMPERATURE (ºC)	VELOCITY (m/s)	VOLUMETRIC FLOW (m³/s)
1	67.4	3.92	6.71×10^{-3}
2	77.3	-	-
3	87.6	-	-

According to Leal [16], using gasoline with 25% of ethanol, the combustion of the gases may be analyzed by Equation 1.

$$0.7499992(C_8H_{18}) + 0.25(C_2H_5OH) + 0.0000007805(S_2) + 8.4999919(O_2+3.76N_2) \rightleftarrows 3.249997(CO) + 0.0000007805(SO_2) + 3.249997(CO_2) + 7.499928(H_2O) + 31.9599695(N_2) \quad (1)$$

VI. RESULTS AND DISCUSSION

As can be seen in Figures 3 and 4, the maximum temperature observed in the exhaust gas was 352.1°C (exhaust system input), which was kept in the baffle (Figure 4a).

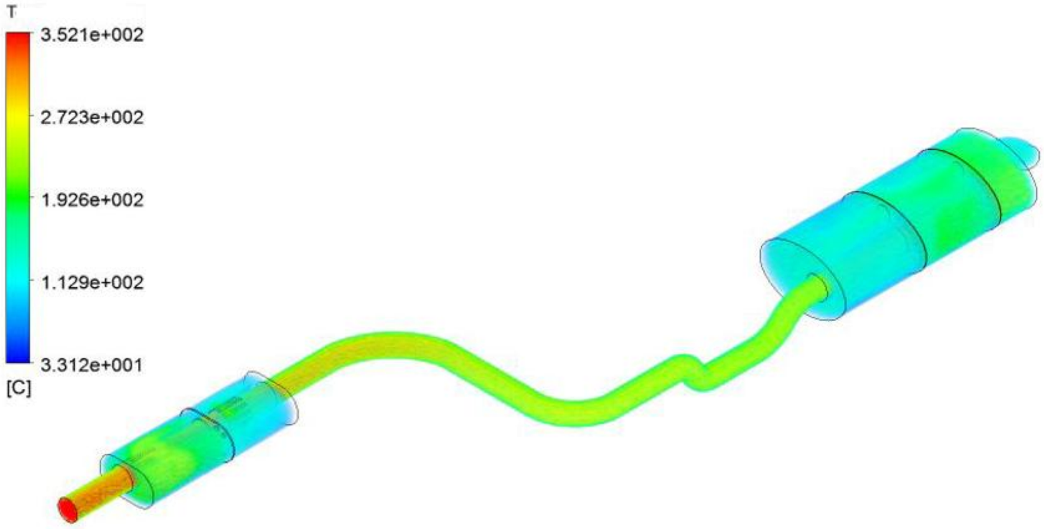

Figure 3 - Gas temperature inside the exhaust system.

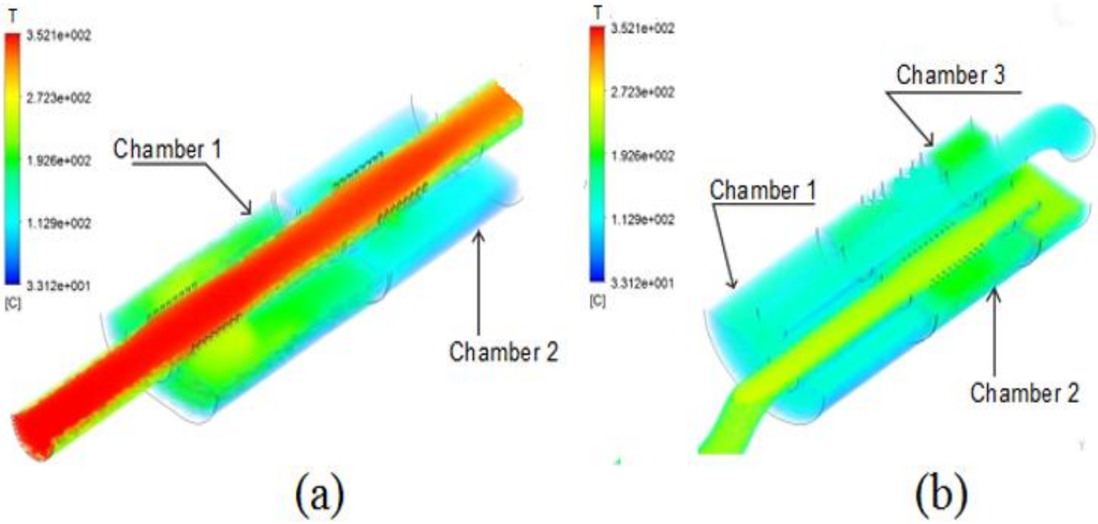

Figure 4 - Gas temperature inside the exhaust system: (a) Baffle (b) Muffler.

According to Figure 4(a), the temperature outside the main pipe within chamber 1 of the baffle is around 190°C. In chamber 2, the temperature outside the main pipe is around 100°C. The temperature of the gas inside the pipe in the muffler is around 170ºC. In the output pipe of the muffler, the gas temperature is around 130ºC, displaying, in this way, a drop in the gas temperature in the output around 40ºC in relation to the input. At the end of the duct output to the atmosphere, the temperature calculated is around 70°C, being consistent with the values measured through experiments.

In Figures 5 and 6, we can observe the following findings in relation to the temperature of the gas in contact with the walls of the exhaust system:

- Maximum temperature of the gas in contact with the exhaust system walls: 345.5°C (Figure 5);
- The temperature in chambers 1 and 2 of the baffle is around 50°C (Figure 6a), while in the main duct inside the baffle, the temperature is around 210°C. This large variation of the temperature gradient (around 160°C) may induce adverse conditions in this region of the system such as the following: undesirable condensations, thermal stress, thermal fatigue, corrosion under stress, and pitting corrosion, among others, depending on the material with which the exhaust system is manufactured.
- In the muffler (Figure 6b), different from what was observed in the baffle (Figure 6a), the temperature shows homogeneous values, that is, without significant variations.

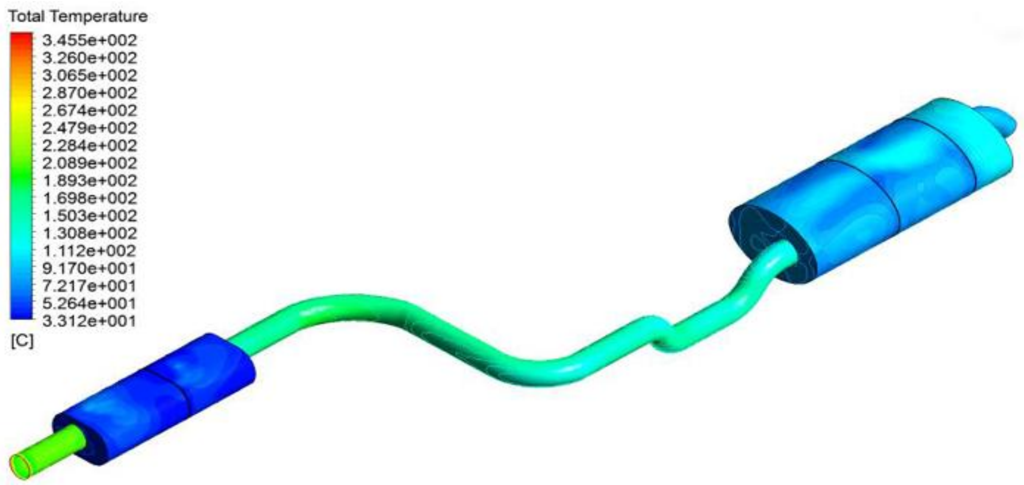

Figure 5 - Temperature of the gas in contact with the exhaust system wall.

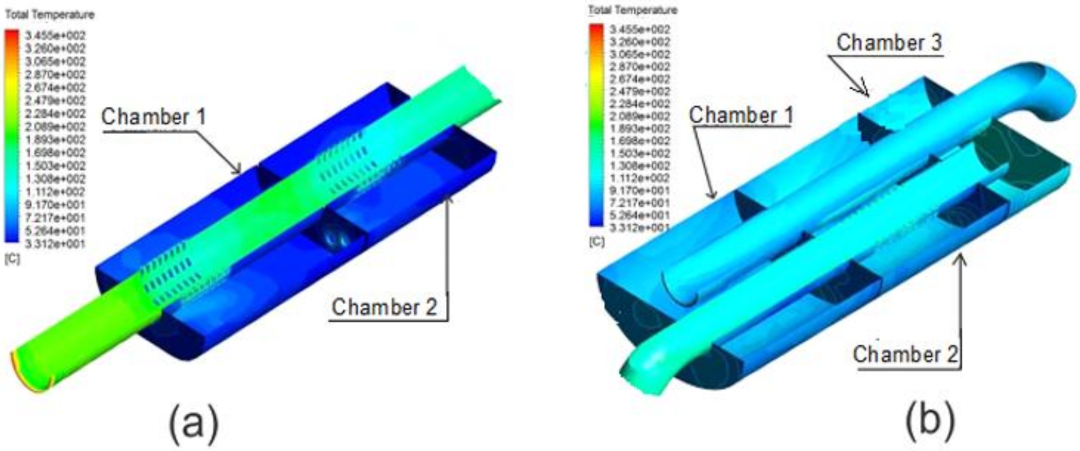

Figure 6 - Temperature of the gas in contact with the exhaust system wall in the (a) Baffle and (b) Muffler.

Variations of the gas pressure inside the exhaust system were analyzed in Figures 7 and 8. We can observe, generally, along the path of the gas inside the system, a gradual reduction of the gas pressure. In the baffle (Figure 8a), we can observe that the pressure was kept practically constant (± 60Pa) disclosing the low loss of load produced by this element of the system. In the muffler (Figure 8b), there is an evident reduction of the gas pressure when it goes through chamber 3 (± 50Pa), chamber 2 (± 40Pa), chamber 1 (± 20Pa), and finally in the output duct (± 2Pa). The system analyzed showed a total load loss of ± 58Pa.

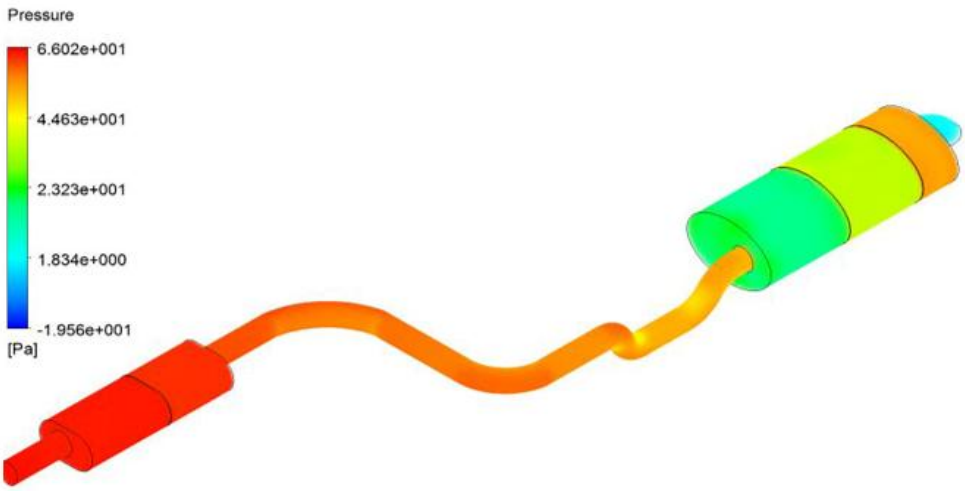

Figure 7 - Variation of the gas pressure in the exhaust system.

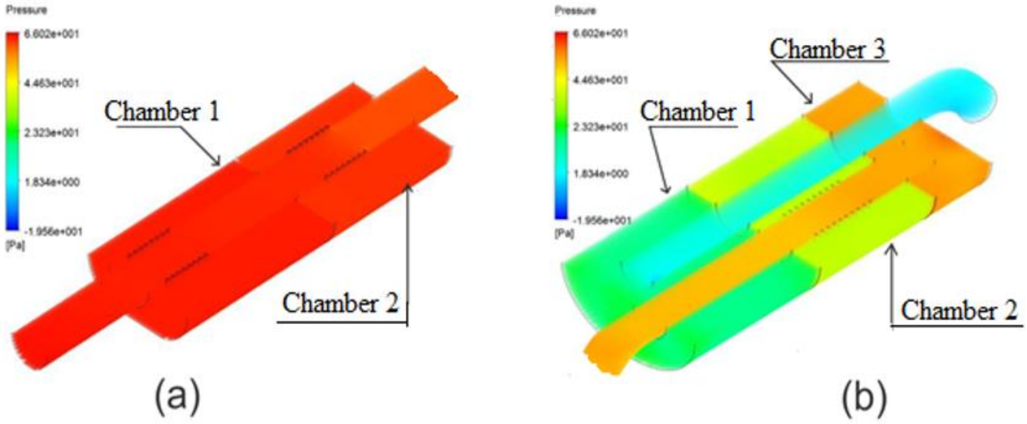

Figure 8 - Variation of the gas pressure inside the exhaust system: (a) Baffle, (b) Muffler.

Figures 9, 10, 11, and 12 show the profiles of the gas velocity inside the exhaust system. The velocities obtained are practically the same as those found through experiments [16]. The velocity of the gas inside the ducts (inclusive inside the baffle and the muffler) is practically constant (± 4m/s). In the baffle (Figures 10a and 12a) outside the main duct, the gas is kept static. In the muffler (Figures 10b and 12b), outside the main ducts (input and output), different from what was observed in the baffle, there is an intense gas deceleration when exiting the input duct (from ± 4m/s to ± 2m/s). The holes in the main ducts for chamber 2 and the holes in the separation drivers of the chambers lead to an intense acceleration of the gas, thus providing turbulence in the regime of the gas flow in this region.

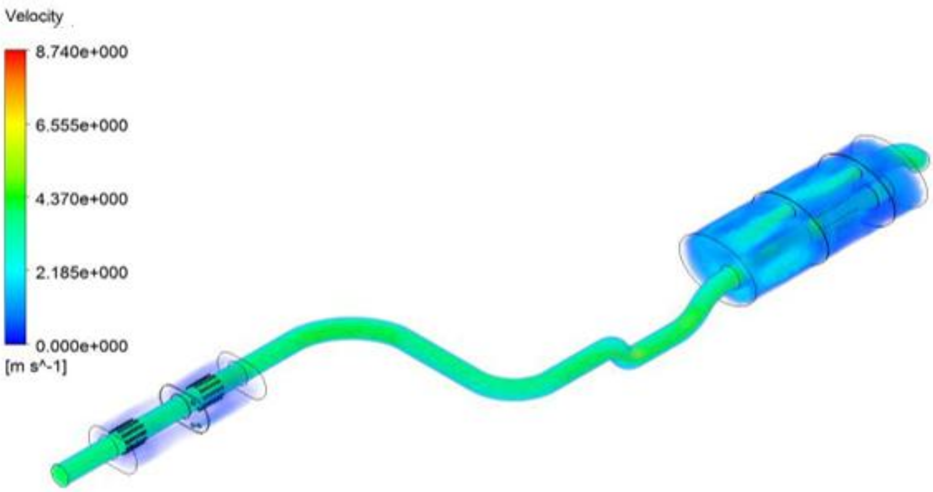

Figure 9 - Velocity of the gas inside the exhaust system.

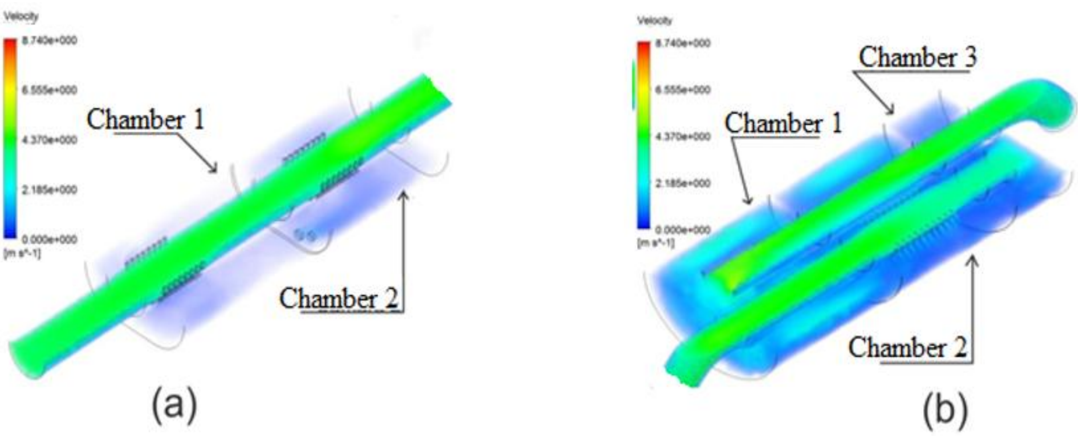

Figure 10 - Velocity of the gas inside the exhaust system: (a) Baffle, (b) Muffler.

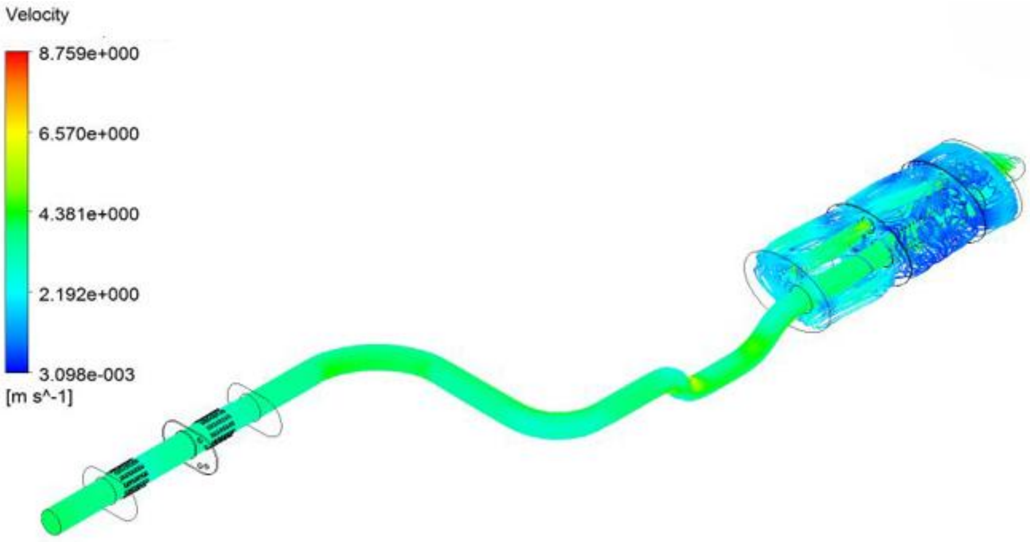

Figure 11 - Velocity lines of the gas inside the exhaust system.

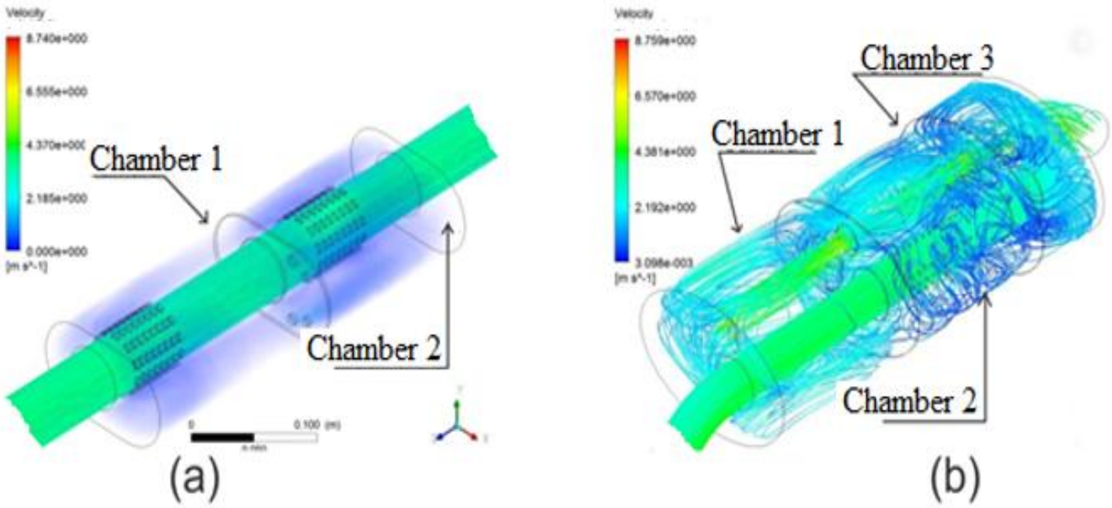

Figure 12 - Velocity lines of the gas inside the exhaust system: (a) Baffle, (b) Muffler.

VII. CONCLUSIONS

- In the exhaust system studied, the gas temperature reduced from ± 350°C in the input to ± 70°C in the output;
- A large variation in temperature gradient was observed between the gas in the baffle and in the main duct inside it;
- The temperature variation of 160°C observed in the baffle may be the cause of eventual undesirable phenomena in this element of the system, such as: undesirable condensations, thermal stress, thermal fatigue, stress corrosion cracking, and pitting corrosion, among others, depending on the material with which the exhaust system is manufactured;
- There was a gradual reduction of the gas pressure inside the system, varying from ± 60Pa in the input to ± 62Pa in the output;
- The pressure in the baffle was kept practically constant, around ± 60Pa;
- In the muffler there was a considerable gas pressure reduction when passing from the input duct to the chambers and then to the output duct;
- In the duct, the gas velocity was kept constant, even inside the baffle and the muffler;
- The regime of the gas flow in the exhaust system was predominantly laminar, except in the muffler.

ACKNOWLEDGMENTS

The authors would like to acknowledge the UNIFEI (Universidade Federal de Itajubá, Campus Itabira – Brazil); FAPEMIG (Fundação de Amparo à Pesquisa do Estado de Minas Gerais – Brazil), and GPESE (Grupo de Pesquisa em Sistemas de Exaustão).

REFERENCES

[1]. SHIMIZU, T.; OHTANI, M. Numerical simulation of heat transfer and fluid flow applied to exhaust manifold. JSAE Review, v. 17, p. 17-23, 1996.

[2]. LIU, X.; DENG, Y.D.; CHEN, S.; WANG, W.S.; XU, Y.; SU, C.Q... A case study on compatibility of automotive exhaust thermoelectric generation system, catalytic converter and muffler. Case Studies in Thermal Engineering, v. 2, p.62-66, 2014.

[3]. HATAMI, M.; GANJI, D.d.; GORJI-BANDPY, M. Experimental and numerical analysis of the optimized finned-tube heat exchanger for OM314 diesel exhaust exergy recovery. Energy Conversion and Management. v. 97, p. 26-41, 2015.

[4]. SHAO, Ying-li. A Study on Exhaust Muffler Using a Mixture of Counter-phase Counteract and Split-gas Rushing. Procedia Engineering, v. 15, p.4409-4413, 2011.

[5]. HOSCHLER, K.; BISCHOF, J.; KOSCHEL, W. Thermo-mechanical analysis of an automotive diesel engine exhaust manifold. Temperature-Fatigue Interaction. v. 29, p.299-308, 2002.

[6]. VERSTEEG, H., and MALALASEKRA, W., An Introduction to Computational Fluid Dynamics: The Finite Volume Method, 2nd Edition, 520p., Prentice Hall. 2007.

[7]. LÖHNER, R. Applied Computational Fluid Dynamics Techniques: An Introduction Based on Finite Element Methods, 2nd Edition, ISBN-10: 047051907X, 544p., Wiley. 2008.

[8]. BARTH, T.; OHLBERGER, M. Finite Volume Methods: Foundation and Analysis, Encyclopedia of Computational Mechanics, John Wiley & Sons, Ltd. 2004.

[9]. KANDYLAS, I. P.; STAMATELOS, A.M. Engine exhaust system design based on heat transfer computation. Energy Conversion & Management, Thessaloniki, Greece, v. 40, p.1057-1072, 27 Mar. 1998.

[10]. AMBROZIN, A. R. P.; KURI, S. E.; MONTEIRO, M. R. Corrosão metálica associada ao uso de combustíveis minerais e biocombustíveis; Revista Quim Nova, v. 32, no. 7, p. 1910-1916, 2009.

[11]. GUIMARÃES, R.; FILHO, J. F. M. Sensor de corrosão à fibra óptica para poços de petróleo; XV Congresso de Iniciação Científica da UFPE, 2007.

[12]. INCROPERA, F.P. DEWITT, D. P.; BERGMAN, T. L.; LAVINE, A. S. Fundamentos de Transferência de Calor e de Massa. 6. ed. Rio de Janeiro: Ltc, 2008.

[13]. BRUNETTI, Franco. Mecânica dos Fluidos. 2. ed. São Paulo: Pearson Prentice Hall, 2008.

[14]. ÇENGEL, Yunus A. CIMBALA, John M. Fluid Mechanics: Fundamentals and applications. McGraw-hill Higher Education, 2006.

[15]. MIRANDA, W. R.; NASCIMENTO JR., E. D.; REZENDE, A. L. T. Simulação Numérica de uma Bolha de Separação em Bordo Arredondando Utilizando Equações Médias de Reynolds. In: XI Simpósio Brasileiro de Automação Inteligente. Fortaleza, 2013.

[16]. LEAL, M. B.; SOUZA, A. C.; CARRILO, R. M.; BRITO, R. F.; LACERDA, J. C.; GOUVEIA, L. L. A.; NEVES, L.; TEIXEIRA, R. L. P. Simulação termofluidodinâmica do escoamento de gases de combustão em escapamento de veículos automotivos. Revista vértice CREA-MINAS, p. 10-15, 2015.