ISSN: 1231-4005
e-ISSN: 2354-0133
ICID: 1130419
DOI: 10.5604/12314005.1130419

NUMERIC MODELLING OF FUMES FLOW IN EXHAUST MANIFOLD OF THE COMBUSTION ENGINE

Monika Andrych, Lech J. Sitnik, Marcin Tkaczyk, Wojciech Walkowiak

Wroclaw University of Technology
Department of Vehicle Engineering
Wybrzeże Wyspiańskiego Street 27, 50-370 Wroclaw, Poland
tel.: +48 71 347 7918, fax: +48 71 347 7918
e-mail: monika.andrych@pwr.edu.pl, lech.sitnik@pwr.edu.pl
marcin.tkaczyk@pwr.edu.pl, wojciech.walkowiak @pwr.edu.pl

Abstract

In the paper there is presented the methodology of researches as well as the numeric researches results, which aim is the analysis of fumes flow parameters in the exhaust manifold in the diesel engine, particularly the velocity, the temperatures and the turbulence of gas. It has been presented that the research methodology depending on three dimensional model of air flow in the outlet systems with the application of standard numeric methods based on the principles of conservation of: mass, energy and momentum can be the way of researches. The model may be to be the basis for performing changes in geometry of outlet system concerning minimizing the flow losses and shaping the field of velocity. The model enables of calculations of outlet system of internal combustion engines on the stage of its construction. Results show that the numerical simulations methods are one of the possible to use for achievements of investigations low costs, especially that the hardware engine parts must be not prepared. Inter alia, complete model of exhaust manifold, model of fluid volume fitting in the manifold with the numbers of ducting, the grid with applied edge conditions, linear outline of static pressure on the walls of exhaust manifold, total pressure, tension, distribution of total temperature, distribution of fluid velocity on the walls, turbulence intensity are presented in the paper.

***Keywords*:** *combustion engine, exhaust manifold, fumes flow, numeric modelling*

1. Introduction

The exhaust manifold is the essential element of an engine when the exchange of load in a cylinder is concerned. Manifold construction is impeded because of the necessity of counting the presence of different engine elements around it. The manifold has an impact on fumes flow dynamics as well as thermal tension and deformation. Making use of the numeric method of modelling is therefore the helpful for projecting and optimization of shapes and sizes of the manifold. For modelling, the calculative program FLUENT has been used [1].

The package of FLUENT software is for numeric modelling of flow (mixing and transporting chemical individuals) with the possibility of concerning the accompanying chemical reactions. The software enables the modelling of multiple chemical reactions including the influence (effects) of the walls limiting the flow and their porosity [2, 4].

The main phases of solving the issue of transport and chemical reactions require:

- the formulation of description (choice) of transport model and reactions connected with the volume change of considered individuals and giving the mixture ingredients,
- if necessary, counting the influence of model walls on the reaction process,
- defining of mixture quality: mixture ingredients, the formula of accompanying chemical reaction and the other physical qualities (e.g. tenacity, specific heat),
- defining of physical qualities of every individual in the mixture,
- determination of edge condition of mixture flow.

2. Object and research methodology

The object of numeric researches has been the exhaust manifold of modern four-stroke diesel engine with cubic capacity of 1998 cm3, of power 104 kW that cooperates with ECU of code EDC16U34 [4-6].

The realized researches have concerned the flow resistances in exhaust manifold in internal combustion engine. The presented results are limited to outflow of fumes from the fourth cylinder, on the other hand, the researchers methodology is analogical when is analysed the outflow of fumes out the other cylinders.

In the early phase of studies has been modelled the manifold construction though digital mapped in the modelling program 3D SolidWorks and then the volume of fluid filling the Manifold ductings has been created [7-9].

The researches results have been presented in Fig. 1 and 2.

Fig. 1. Complete model of exhaust manifold

Fig. 2. Model of fluid volume fitting in the Manifold with the numbers of ducting

On the resulting channel model applied the grid in the GAMBIT- Fig. 3.

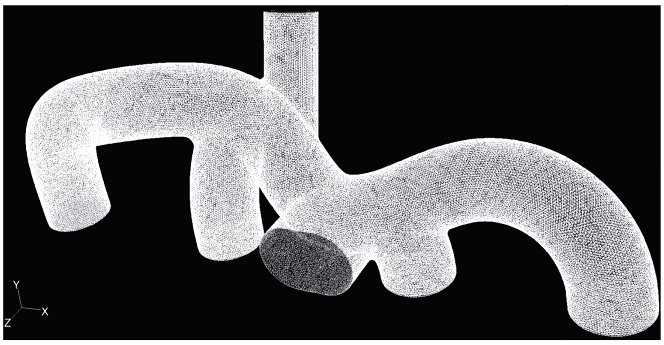

Fig. 3. Grid visualization of space limiting fluid volume in ANSYS FLUENT

For modelling process, the following edge conditions have been assumed:

- Intake of fumes in the 4th ducting, v= 24.61m/s,
- Turbulence =4 [%], 1000 mm length scale,
- Outlet EGR – 0.2 of volume,
- Outlet in turbine – 0.8 of volume,
- Fumes temperature (intake air) =823 K,
- Temperature of body walls =673 K,
- Body walls material: cast iron,
- Gauge of body walls: 4-5 mm,
- Energy equation- on.

The grid with applied edge conditions is presented in the Fig. 4.

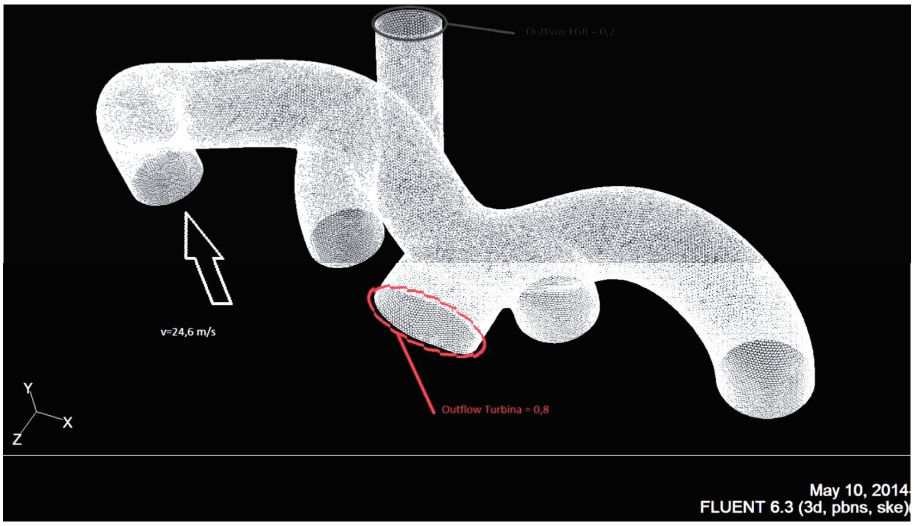

Fig. 4. The grid with applied edge conditions

3. Research results

The realized numerical analysis shows that from the 700th iteration the courses become practically rectilinear which allows to assume that the above-mentioned number of iterations is sufficient for obtaining as adequate model.

In the Fig. 5 has been presented the course of static pressure on manifold walls, and Fig. 6 shows the course of total pressure.

Fig. 5. Linear outline of static pressure on the walls of exhaust manifold

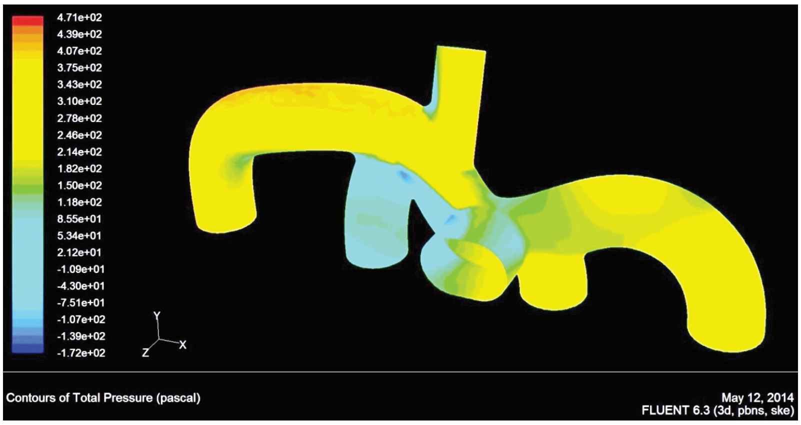

Fig. 6. Total pressure on the walls

It can be noticed that the highest total pressure is on the walls of the 4th inlet ducting (up to 470 Ps) and near the entrance for EGR outlet (up to 407 Pa). In the 3rd inlet ducting, the pressure is the lowest (about 53 Pa). In the 1st and 2nd inlet ducting, the values of pressure are moderate (182Pa-246Ps).

Tension on the walls of manifold is shown on the Fig. 7.

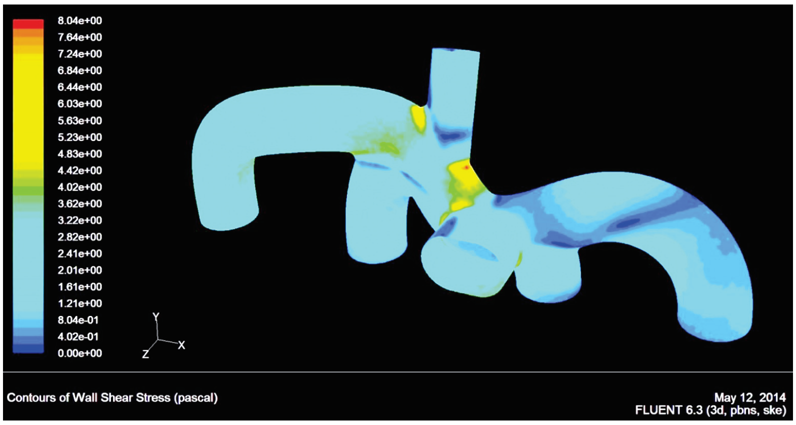

Fig. 7. Tension on the Manifold walls

According to Fig. 7, the highest tension is located in the places of rounding in EGR outlet and outlet of Turbine. These are very low values reaching up to 8 Pa.

From Fig. 8, there can be noticed that the temperature of fluid in inlet ducting 1 and 3 is very similar to the one of the body (673 K) and is oscillating from 678 K to 730 K. The fluid flowing in ducting 4, toward EGR outlets and the turbine is cooled from the temperature 823 K to about 787 K.

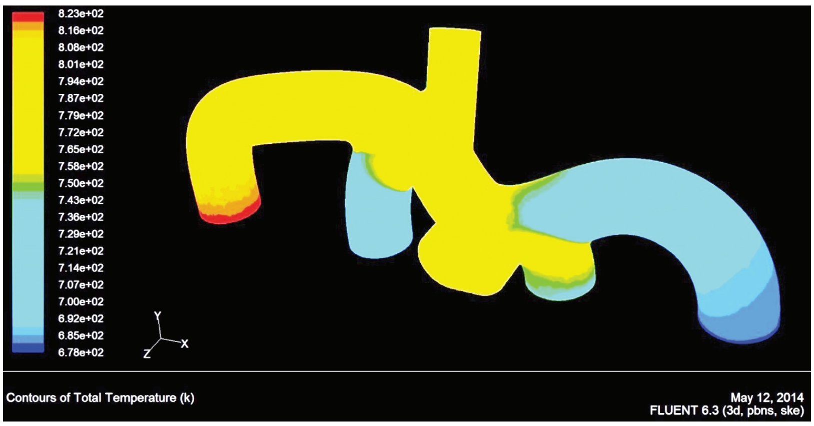

Fig. 8. The distribution of total temperature on the walls of exhaust manifold

Figure 9 presents that the fluid in the Manifold is characterized in the wide spectrum of velocity. The demanded velocity is 25 m/s. They can be observed at the beginning of the 4th inlet ducting. Later the fluid slows down on the curves and finally flows out with the velocity of about 21 m/s. It is worth noting, that by the rounding of outlets, the velocity raises up to about 41 m/s, which is presented in Fig. 10.

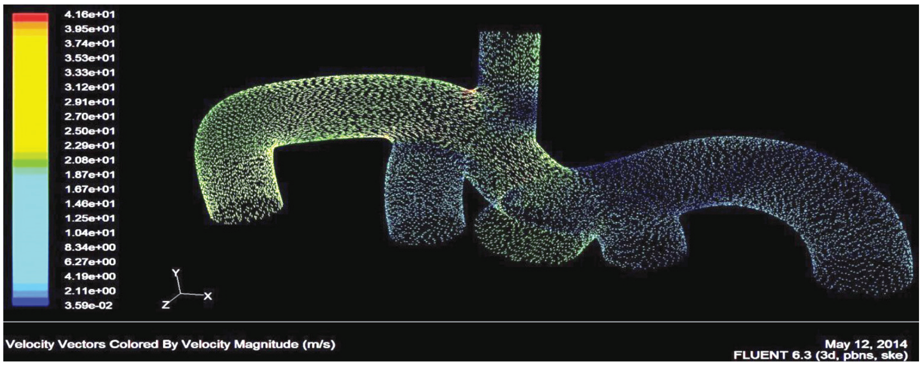

Fig. 9. Distribution of fluid velocity on the walls

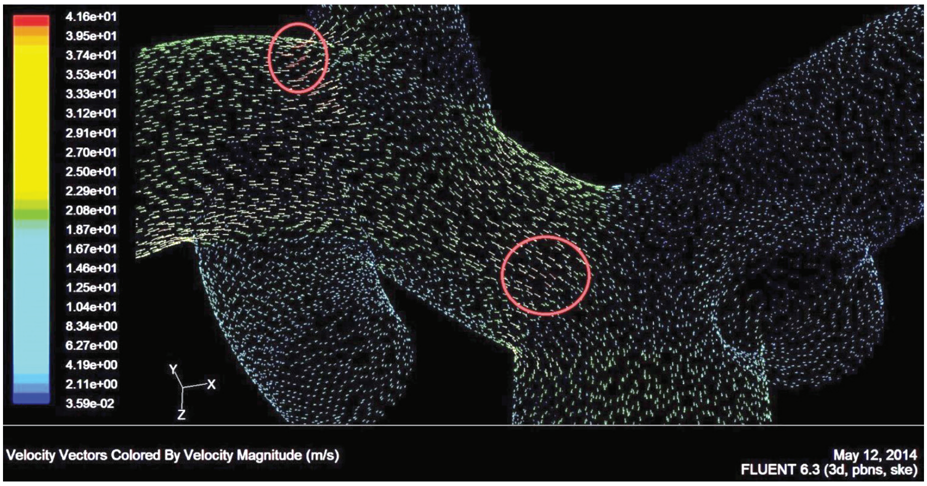

Fig. 10. Places where the velocity of fluid is the highest

According to Fig. 11, the most visible change of turbulence intensity is visible on the manifold creases (particularly EGR outlet and in the outlet of the turbine) values even up to 700%).

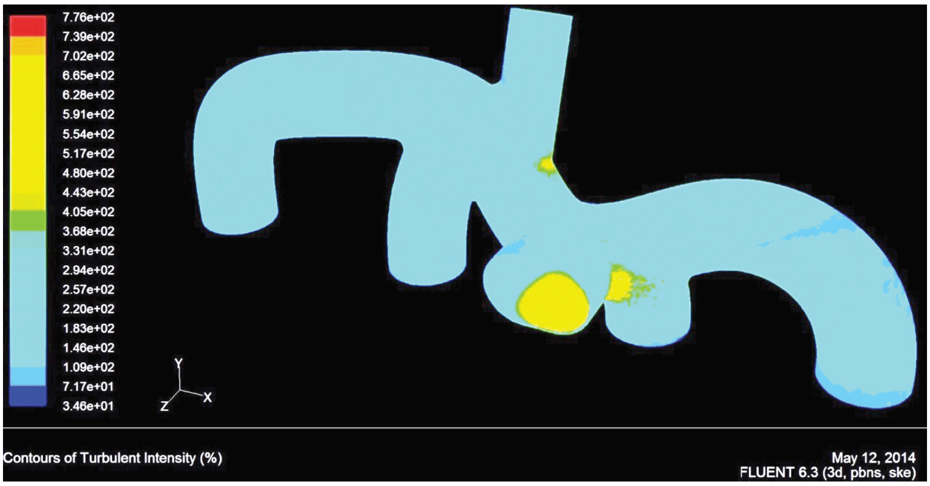

Fig. 11. Turbulence intensity

4. Conclusions

It has been presented that the research methodology depending on three-dimensional model of airflow in the outlet systems with the application of standard numeric methods based on the principles of conservation of: mass, energy and momentum can be the way of researches.

The model may be to be the basis for performing changes in geometry of outlet system concerning minimizing the flow losses and shaping the field of velocity. The model enables of calculations of outlet system of internal combustion engines on the stage of its construction.

The conclusions from the analysis of velocity and air distribution:

- the dominant resistance has been affirmed in the space of direction change where the highest pressure differences appear,
- the has been affirmed patchy distribution of pressure and velocity of fumes in the cross-section area of outlet ducting, which indicates the deficiencies in develop of manifold,
- the increase of flow velocity appears in the field of manifold, which is the result of decreasing the field of section space,
- the conducted considerations suggest the necessity of changes in geometry of manifold.

Bibliography

[1] Fluent Inc., Fluent 6.1 *User`s Guide,* 2003.
[2] Jankowski, A., *Some aspects of heterogeneous processes of the combustion including two phases,* Journal of KONES, Vol. 12, No. 1-2, pp. 121-134, 2005.
[3] Jankowski, A., *Chosen Problems of Combustion Processes of Advanced Combustion Engine,* Journal of KONES Powertrain and Transport, Vol. 20, No. 3 2013, pp. 203-208, 2013.
[4] Jankowski, A., *Study of the influence of pressure, speed and type of gas stream on the combustion process, S*cientific Papers of the Air Force Institute of Technology, Issue 28, (in Polish) Warsaw 2010.
[5] Jankowski, A., *Laser research of fuel atomization and combustion process in the aspect of exhaust gases emission*, Journal of KONES Powertrain and Transport, Vol. 15, No. 1, pp. 119-126, 2008.
[6] Jankowski, A., *Some Aspects of Heterogeneous Processes of the Combustion Including Two Phases*. Journal of KONES. Internal Combustion Engines. Vol. 12, No. 1-2, pp. 121-134, Permanent Committee of KONES, Warszawa 2005.
[7] Johnson, R., *Fluid Dynamics,* CRC Press LLC, Florida 1998.
[8] Wilcox, D., *Turbulence modeling for CFD,* KNI, Inc., Anaheim 2002.
[9] Sampath, M. K., Zheng, G., Zhang, Y., Kotrba, A., Golin, M., Lacin, F., *CFD Optimization of Exhaust Manifold for Large Diesel Engine Aftertreatment Systems,* SAE 2199 New York 2011.