Advances in Science and Technology
Research Journal
Volume 8, No. 23, Sept. 2014, pages 45–52
DOI: 10.12913/22998624.1120322

Research Article

CFD MODEL OF THE CNG DIRECT INJECTION ENGINE

Zbigniew Czyż[1], Konrad Pietrykowski[1]

[1] Department of Thermodynamics, Fluid Mechanics and Aviation Propulsion Systems, Faculty of Mechanical Engineering, Lublin University of Technology, 36 Nadbystrzycka Str., 20-618 Lublin, Poland, e-mail: z.czyz@pollub.pl

Received: 2014.07.21
Accepted: 2014.08.11
Published: 2014.09.09

ABSTRACT
The paper presents CFD analysis of fuel flow in the CNG injector. The issues such a pressure drop along an injector channel, mass flow through the key sections of the injector geometry, flow rates, the impact of the needle shape on the deflection of the sprayed gas cone and the impact of the wall head are analyzed in the article. The simulation was made in the transient states conditions for full injection process, including the opening and closing of the injector. An injection time of 6 ms, velocity of 0.33 mm/ms and a lift of 0.5 mm were selected for opening and closing of injector based on experimental test. The simulation shows that the volume inside the injector is a kind of fuel accumulator, and the opening process of the needle influence the flow parameters in an inlet cross-section after a certain time, depending on a channel cross section. The calculations allowed to select the ratio of an injector duct cross sectional area to the aperture area of the injection capable of the reducing pressure loss. The unusual location of the injector in the socket of a glow plug in the Andoria ADCR engine makes a stream be impaired by a part of the head. This research result would be useful in developing an injector construction which will be used for an investigation of CNG addition into diesel engine.

Keywords: simulation and modeling, gas injection, methane injection, direct injection.

INTRODUCTION

Performance of internal combustion engines is largely dependent on the characteristics of the fuel injection system. Transient nature of the flow through these systems and their small sizes makes it very difficult to analyze them experimentally. Simulation tests are often used to analyze the flow inside the injectors and provide the physical flow processes visibility that occurs in them and the entire fuel system [1, 2, 3, 4, 5]. Because of that, computational flow dynamics analysis (CFD) of flow through the injector was performed. In the past, most of the fuel injector CFD analysis was limited to the steady state time conditions. This approach is simplistic and does not fully reflect the phenomena occurring in the undetermined flow. In order to improve computing technologies, the moving mesh is used nowadays. Depending on the computing solver, there are many ways to simulate the movement of the elements. In work of Margot et al. and Payri et al. [6, 7], the moving mesh was done using STAR CD software in which the movement does not affect the change in the number of elements. Lee and Reitz [8] used a special algorithm of needle movement, which contained the structural grid consisting of hexahedral elements. However, such an approach increases the difficulty of mesh generation for complex shapes of injectors. It is possible to carry out the whole analysis including the geometry preparation and model discretization, not only in STAR CD software, but also in Ansys Fluent software. In the case of transient time analysis (taking into account the time-varying phenomena), there are two possibilities of moving mesh behavior. The first one does not make any changes to the finite element of mesh and is called Sliding Meshes. The second one modifies the elements (remeshing, layering, smoothing). In the simulation, the layering moving mesh was used. This method allows for dynamic layering of hexahedral or wedge mesh elements. This makes it possible to add or remove cells layer placed near the movement surface, based on the thickness of the layer adjacent to the moving surface [9]. Certainly, the development of model discretization methods and movement simulation increases design options.

Injector designing is still challenged by constant striving to improve economic and ecological aspects in combustion engines, alternative-fuel supply and varied injection pressure. This paper focuses on the development of the model of compressed natural gas injector, using Ansys Fluent. CFD tools are often applied to simulate parameters, difficult and sometimes impossible to measure experimentally, of the flow inside the injector and injection. The injector investigated in this paper was developed for dual-fuel supply in a compression ignition engine. Methane seems to have fairy high heating value, so as an additional fuel, it requires special injection conditions to power an engine, mainly for its relatively low density and large-volume flows. Therefore, developing a model that can help to identify and exclude any structural mistakes in flow channel geometry.

RESEARCH OBJECT

The assumptions of a geometric numerical model were based on real geometry and experimental studies. It will be used to supply the CNG fuel to Andoria ADCR diesel engine. The injector will be placed in the cylinder head in glow plug socket. Other cases of construction solutions for dual-fuel engine were presented in [10]. Figure 1 presents an injector used for experimental research along with its longitudinal cross section.

The actual geometry of the injector was used to prepare the geometric model shown in Figure 2.

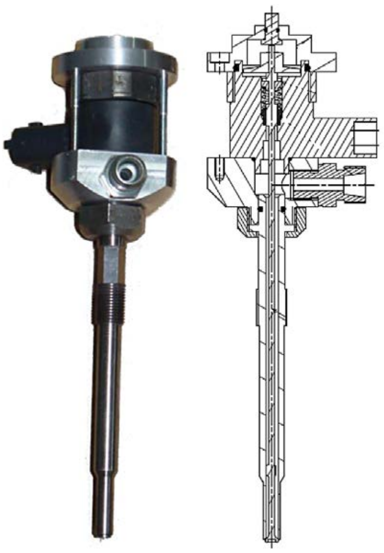

Fig. 1. The actual design of the injector (left) and its longitudinal cross section (right)

In addition, different geometries of injectors were used for numerical calculations (Figures 2 and 3). The second version is completely changed by the injector nozzle, but the third version has an expanded needle nozzle, in comparison to the first version. This is a consequence of previously performed tests on the first version of the injector and is intended to inject the gas correctly to the cylinder. In all of the cases, there is the same cross-section area (5.1 mm^2) of the outlet channel at the outlet of the injector. Simulation studies carried out on other versions of injector were made to provide an injector characterized by small pressure drop over the length of the flow channel with a relatively large mass flow rate and the correct shape of the gas spray injected into the cylinder.

The experiments on real injector determined the characteristics of the injector opening and closing which leads to a simplified model of the needle movement that had been assumed in CFD analysis. Similarly to Czarnigowski [11], table 1 and in Figure 4 present the time of injector open-

Fig. 2. Digital models of studied injectors – first from the left compatible with the real geometry

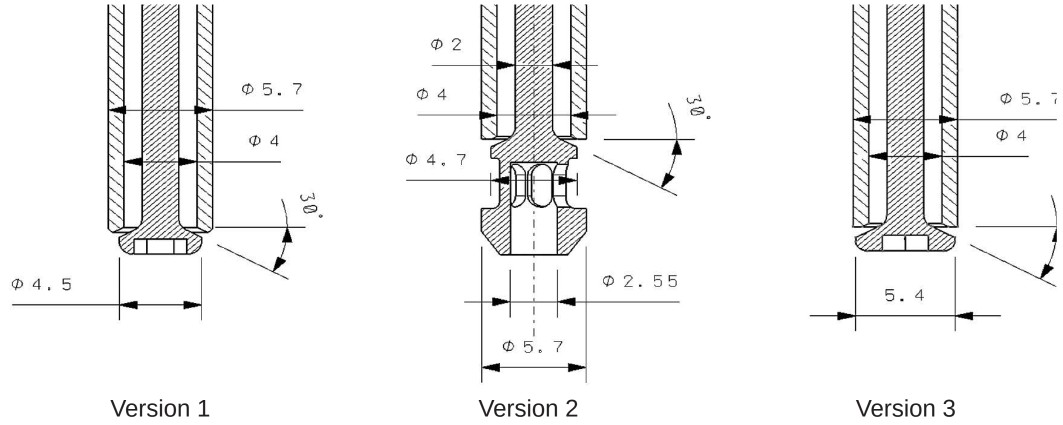

Fig. 3. Cross–sections of studied version of injectors

ing and closing, which was calculated on the basis of the opening and closing velocity that was chosen on the basis of the experiment equal to 0.33 mm/ms for the needle lift of 0.5 mm. Injection time 6 ms was assumed for the calculation.

Table 1. Time of opening and closing of injector form the experiment

Time [ms]	Stroke [mm]
0	0
0.18	0
1.68	0.5
5.18	0.5
6.68	0
7.18	0

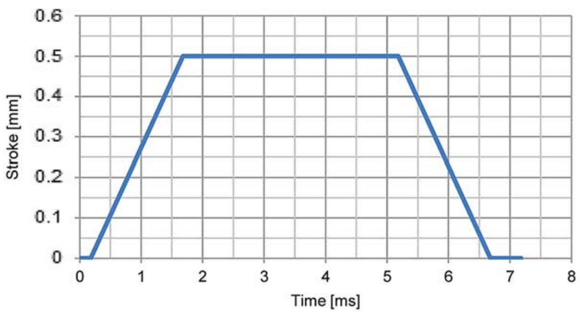

Fig. 4. Opening and closing characteristics of the injector

NUMERICAL METHOD AND BOUNDARY CONDITIONS

In order to reduce the number of elements and the calculation time it was decided to perform the calculation of the symmetric half geometrical (Fig. 5). In order to make transient type analysis with moving parts it was necessary to divide of the geometry that will be useful for the development of a moving mesh (Fig. 6). The usage of layering method requires to have a structural mesh in the moving surfaces. Structural mesh enables the separation of rows and columns of elements belonging to a given volume. This kind of the geometry results in the distribution of regular and uniform formation of new layers. As it is illustrated in figure 7 in the area of injector needle seat, moving elements consist of a hexahedral type, and at different locations they are tetrahedral with the inflation boundary layer on the walls of the flow channel (Fig. 8).

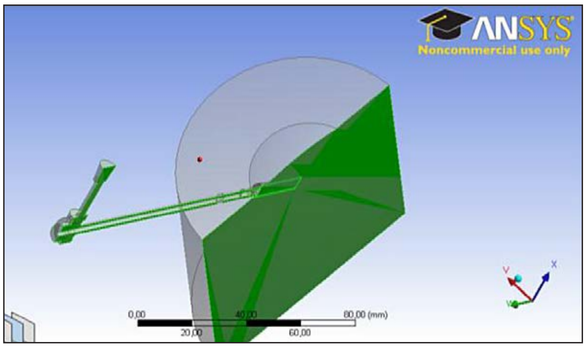

Fig. 5. Geometrical model of the injector with the cylinder

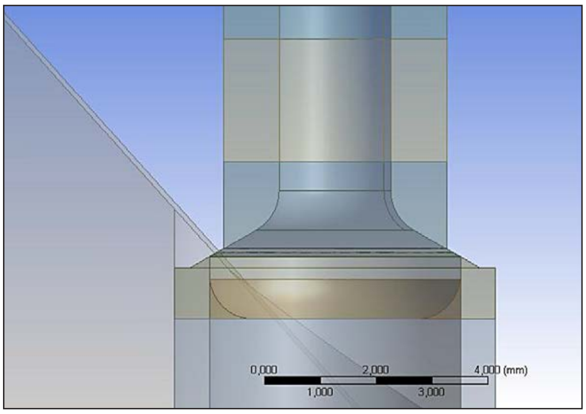

Fig. 6. Division of injector geometry

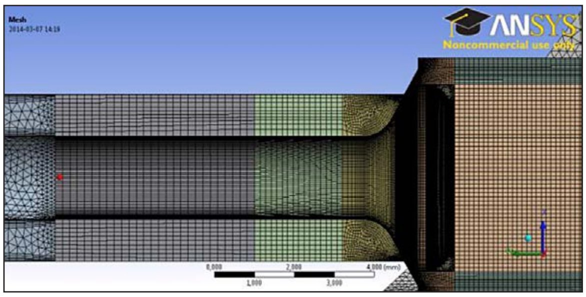

Fig. 7. Computational mesh in the area of injector needle seat

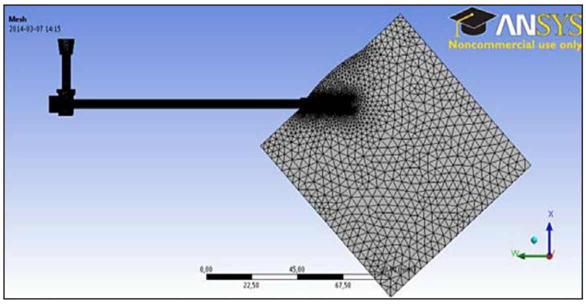

Fig. 8. Computational mesh

Using computational solver Fluent, flow phenomena simulation with transient time conditions was prepared. Due to the nature of flow, to solve the phenomena of turbulence, equation of energy and turbulence were assumed. In all analyzes turbulence model Realizable k-ε (RKE) was used. The flowing gas was, in the first step, the ambient air while in the second step it was methane, for which the ideal gas model was adopted. As a reference pressure, a normal pressure with a value of 101325 Pa was assumed.

In the simulations, the following boundary conditions were assumed:

- for inlets: pressure-inlet, air pressure at the inlet 10 bar,
- the air outlets: pressure-outlet, the air pressure at the outlet was equal to ambient pressure 101325 Pa.

As a solver pressure-based solver was selected. In the solution settings as momentum equation algorithm and default SIMPLE algorithm was selected. For the equations of momentum, energy, dissipation energy of turbulence, kinetic energy of turbulence, the interpolation schemes of the second row were chosen. The convergence calculation solution for the above mentioned equations, as well as the pressure and the velocity on the plane of the injector symmetry were monitored during the simulation. To prepare the movement, it was necessary to develop the profile file as a text file describing the position of the moving components of the geometry depending on the time. Below a sample file is presented while table 2 contains its description:

((movement_of_valve 6 point)
(time 0 0.00018 0.00168 0.00518 0.00668 0.00718)
(y 0.0 0.0 -0.0005 -0.0005 0.0 0.0))

Table 2. Description of the motion profile

Designation		Description
Movement of valve		Name of this profile
6 point		Number of points of the curve shown in figure 5
Time [s]	0	Moments of time the individual moves in seconds
	0.00018	
	0.00168	
	0.00518	
	0.00668	
	0.00718	
y [m]	0.0	Changing the position of the Y axis in meters for given points in time
	0.0	
	-0.0005	
	-0.0005	
	0.0	
	0.0	

Sliding mesh layers on each other and the exchange of information between them without having to connect the nodes of each cell is guaranteed by the use of interfaces. This situation required to choose the interfaces cooperating with each other, so it was necessary to create a pair of interfaces on the contact place, one for each of the adjoining. The next step was to set move components and dynamically changing mesh by layering options. The appropriate properties of rigid body or stationary were given to the surfaces or volumes. The first properties correspond to moving elements and for them, previously developed profile: movement_of_valve was assigned. In the case of moving mesh it was necessary to determine the side from which layers will be built or deleted, dealing with the size of newly formed layers.

RESULTS AND DISCUSSIONS

The following figures illustrate exemplary results of the simulation in a form of contours of velocity, pressure and volume rendering for the

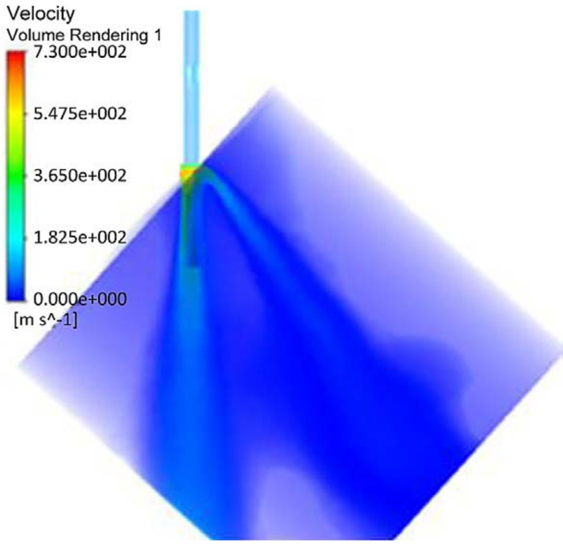

Fig. 9. The shape and the gas velocity profile exiting the first injector version

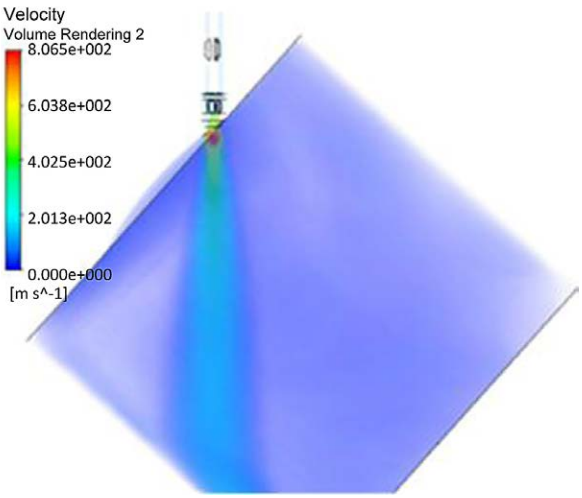

Fig. 10. The shape and the gas velocity profile exiting the second injector version

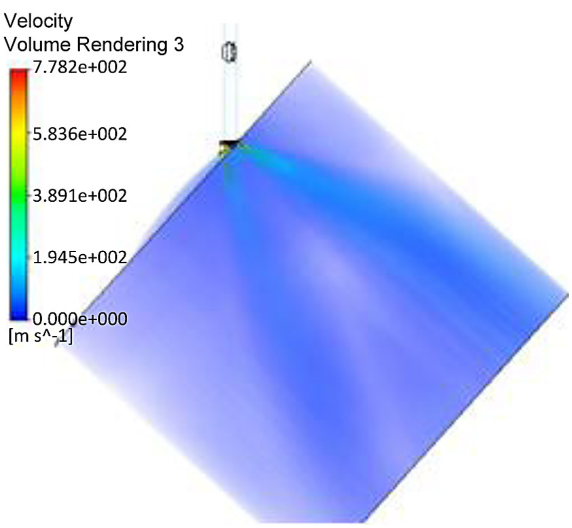

Fig. 11. The shape and the gas velocity profile exiting the third injector version

full opening of the injector. Exemplary velocity contours in the same cross section where methane was injected are presented in Figures 9, 10, 11. The figures show that the maximum flow velocity for methane reaches the value of 730 m/s to 805.1 m/s. The first version is characterized by the smallest value of the flow velocity. The velocities also translate into mass flow rate. This is due to the occurrence of flow accumulations. However, these are only local increases in speed and they are not indicative of the size of the mass flow rate. They depend on the average flow speed. As it can be seen from Figure 15, version 1 is characterized by the largest mass flow rate of both air and methane. By comparing different versions of injector supply by the air and methane, it was found that for the first version the mass flow rate difference between air and methane supply is the smallest 4% in air is preferrable. In the latter case, this is much higher, up to 27%, while in the third it is 19% (relative to air). Thus, the mass flow rate of the injector version 1 is about 40.5% higher than version 2 and 24.9% from version 3**.**

Differences air mass flow and methane are mainly due to the physical and chemical properties of these gases. Density of methane amounting to 0.717 kg/m^3 and air equals 1.205 at 20 °C significantly affect the mass flow rate. During the incompressible flow through the duct the injector the required volume of methane is 68% greater while the mass flow rate remained unchanged. As the critical velocity of these gases are varied but it is not a proportional correlation so the same value of mass flow rate cannot be obtained. As is apparent from Figure 12 and 13 the maximum speed of the flow of methane from the injector is from 30% to 40% higher with respect to the air. This implies that the speed ratio does not compensate for the difference in density. In addition, the volume of elasticity ratio which amount to 1.42 × 105 Pa, and 1.013 × 105 must be considered respectively for air and methane. As shown, methane is more compressible and translates this unfavorably to the mass flow, especially in terms of gas injection at high positive pressure and flow at high speeds.

The first stage of the research allowed to choose the best shape of the injector nozzle of the three tested versions. On this basis, an analysis of the complete cycle of the injection for the first version was made. This version mass flow characteristics result from calculations and are shown in Figure 15. Red line “Inlet” corre-

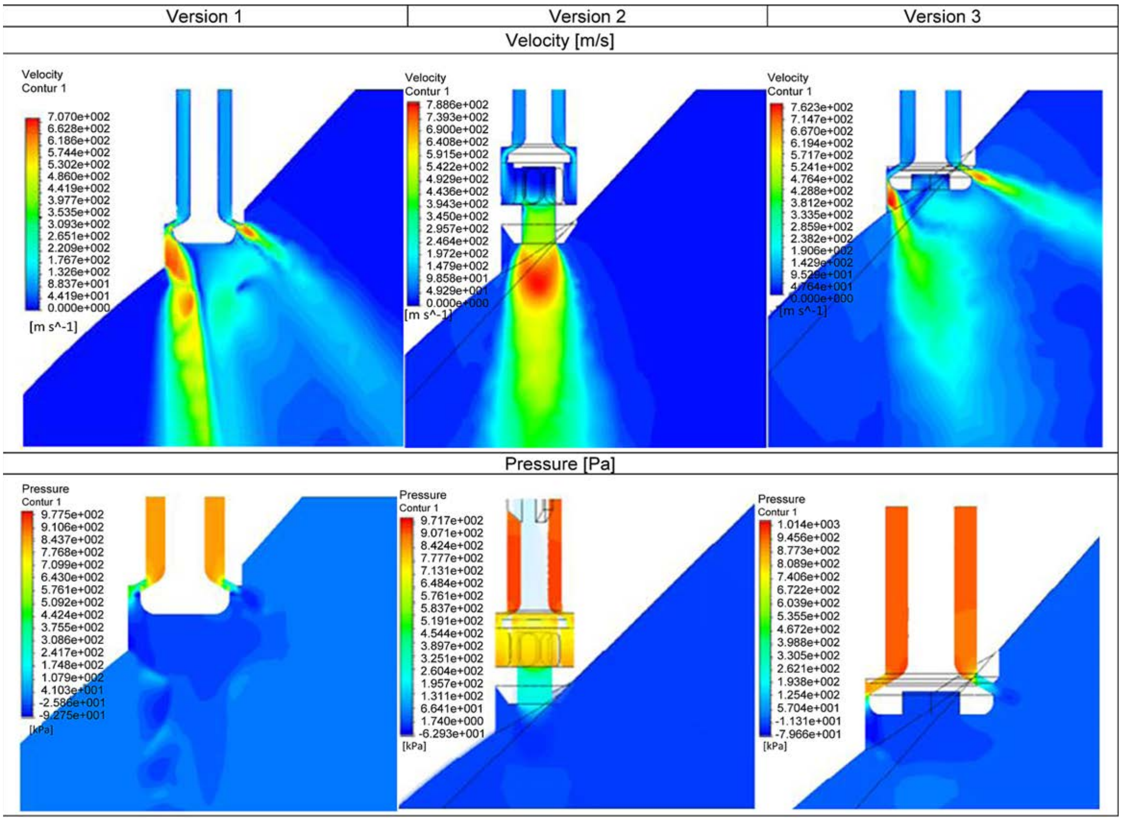

Fig. 12. The contours of velocity and pressure of methane around the needle seat of the injectors for the three versions of design solutions

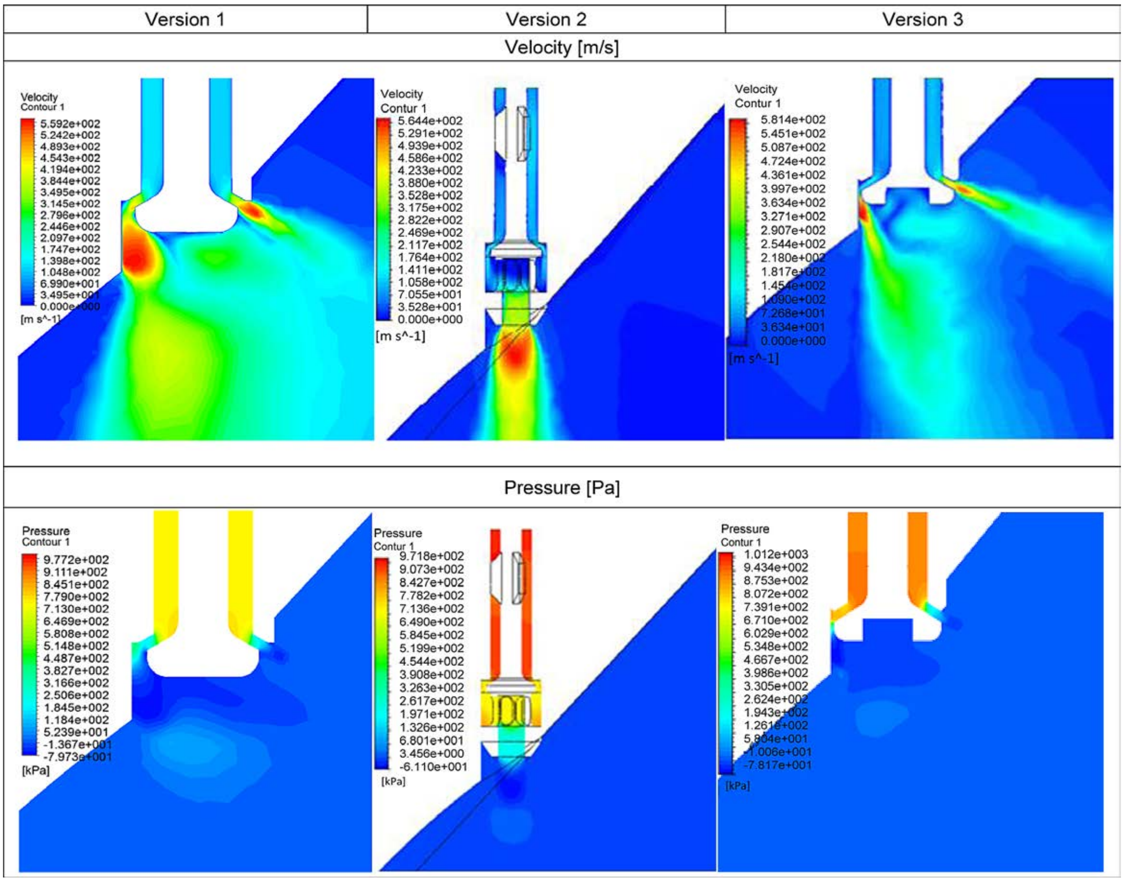

Fig. 13. The contours of velocity and pressure of air around the needle seat of the injectors for the three versions of design solutions

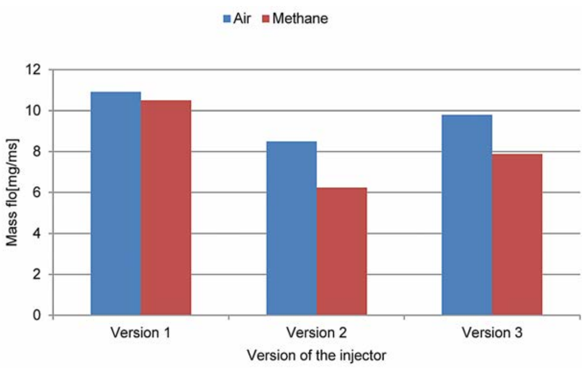

Fig. 14. Comparison of mass flow [mg/ms] examined three injector needle valves for air and methane

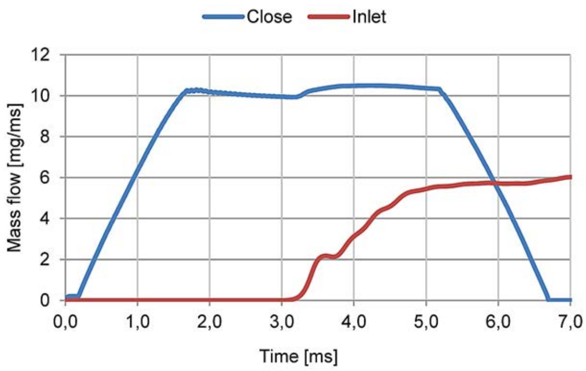

Fig. 15. Characteristics of mass flow of the version 1injector for methane

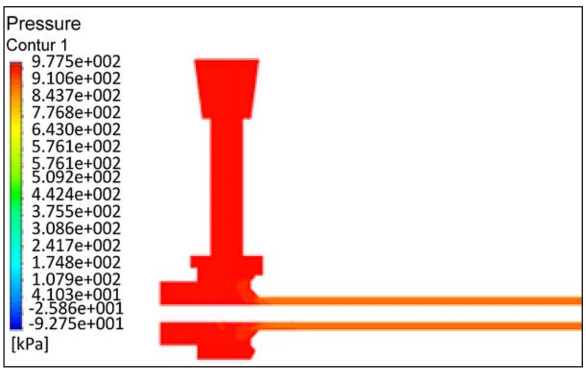

Fig. 16. The pressure distribution in the longitudinal cross-section of the injector close to the inlet

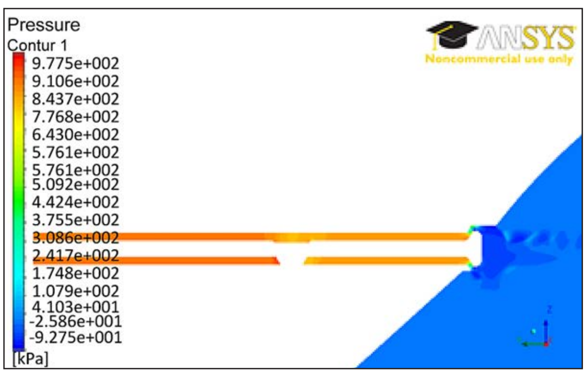

Fig. 17. The pressure distribution in the longitudinal cross-section of the injector close to injector nozzle

sponds to the mass flow in the inlet cross section of the injector, while the blue line "Close" corresponds to the mass flow in the cross-section in the valve seat, i.e. cross section of the flow channel at the outlet of the injector. The Close line values correspond to the injector outflow. Analyzing the mass flow at the time of injection it can be observed that in the cross-section corresponding to "close" line there is no constant flow, even if the valve is fully opened. In the first phase of the injection a decline occurs, accompanied by the pressure drop (Fig. 16 and 17). After some time period, in this case equal to approximately 1.5 ms, a slight increase occurs. The moment when the increase occurs coincides with the initiation of flow through the "Inlet" cross-section. From this moment, flow in this cross-section begins. Injection duration without affecting the "Inlet" cross-section depends on the flow channel volume. The value of mass flow through "Inlet" cross-section, for the tested injection time, has reached a value of about 58% of the maximum "close" cross-section mass flow.

CONCLUSIONS

The main purpose of this simulation was to determine the mass flow through the needle of the injector at a define overpressure, depending on the injection time. The scope of work included the analysis of the flow through the injector with three versions of injector nozzles. On the basis of the research it can be decided which version of the injector is characterized by simplest geometry and, at the same time, preserves good flow properties. The first one has the highest mass flow rate value, low pressure drop along the channel and good shape of gas stream coming for the injector nozzle. The changes in the injector nozzles geometry, while maintaining the same cross-sectional area of injector outlet, cause the big changes in the characteristics of the gas flow. It turned out, that the change in shape, and thereby change of the local loss flow coefficients for the same pressure supply, leads to the changes in the mass flow rate even by 40%. With regard to the study of gases with different physical and chemical properties, such as air and methane, it is possible to obtain approximately the same values of the mass flow by adjusting the shape of the flow channel. In each of the tested versions, the methane mass flow value is smaller, but for the first version, this difference is very small, just over 4%. In relation to the second and third testes version, where the difference between the mass flow of air and methane is in the range 19–27%, the value for the first version is optimistic, so the next step is to verify the results obtained by simulation with experimental studies.

The study also shows the impact of the injection time on the value of the mass flow in the outlet section of the injector. The gas accumulated in the injector channel, in the moment of valve opening flows into the combustion chamber, thereby generating a pressure drop in the area of the outlet nozzle. The pressure drop is noticeable during the injection process and goes farther and farther from the nozzle inside into the injector. Consequently, the mass flow in the outlet cross-section reduces its value in the initial phase of the injection (when the injector valve is fully open). This effect is caused by a feedback pressure wave. Only after a certain time, which for the tested geometry was at about 1.5 ms, changes of the pressure wave direction occurs and the pressure increases moving in the direction of the outlet. The consequence of this is phenomenon is a small mass flow increase at the injector outlet what is presented in figure 15. The pressure drop depends on the volume of the injector channel, and its cross-section area. As shown in figure 17 and 18, the pressure decrease does not reach the inlet cross-section. To reduce its range, the cross-section of the channel must be increased. However, in this case it is limited due to the mounting of the injector onto a glow plug socket.

Acknowledgments

This work has been financed by the Polish National Centre for Research and Development, under Grant Agreement No. PBS1/A6/4/2012.

REFERENCES

1. Mitcham II C.E., at al., Simulations and Analysis of Fuel Flow in an Injector Including Transient Needle Effects. ILASS-Americas 24th Annual Conference on Liquid Atomization and Spray Systems, San Antonio, TX, May 2012.
2. Salvador F.J., Hoyas S., Novella R., Martinnez-Lopez J., Proceedings of the Institution of Mechanical Engineers. Part D: Journal of Automobile Engineering, 225, 2011, 545–556.
3. Tonini, S., Gavaises, M., Theodorakakos, A., Cossali, G.E., Numerical investigation of a multiple injection strategy on the development to high-pressure diesel sprays. Proc. IMechE, Part D: J. Automobile Engineering, 224 (1), 2010, 125–141.
4. Som S., Aggarwal S.K., El-Hannouny E.M., Longman, D.E., Investigation of Nozzle Flow and Cavitation Characteristics in a Diesel Injector. J. Eng. Gas Turbines Power, 132(4), 2010, (12 pages).
5. Schmidt D.P., Corradini M.L., The internal flow of diesel fuel injector nozzles: a review. Int. J. Engine Res., 2(1), 2001, 1–22.
6. Margot X., Hoyas S., Fajardo P., Patouna S., Mathematical and Computer Modelling. 52, 2010, 1143–1150.
7. Payri F., Margot X., Patouna S., Ravet F. et al., A CFD Study of the Effect of the Needle Movement on the Cavitation Pattern of Diesel Injectors. SAE Technical Paper 2009-24-0025, 2009.
8. Lee W.G., Reitz R.D., A Numerical Investigation of Transient Flow and Cavitation Within Minisac and Valve-Covered Orifice Diesel Injector Nozzles. Transactions – ASM Journal of Engineering for Gas Turbines and Power; 132, 5, 052802.
9. HELP program AVL Boost.
10. Pietrykowski K., Grabowski Ł., Sochaczewski R., Wendeker M., The CFD model of the mixture formation in the Diesel dual-fuel engine. Combustion Engines, 154(3), 2013, 476–482.
11. Czarnigowski J., Effect of calibration method on gas flow through pulse gas injector: Simulation tests. Combustion Engines, 154(3), 2013, 383–392.