ISSN: 1231-4005
e-ISSN: 2354-0133
DOI: 10.5604/01.3001.0012.4790

NUMERICAL MODELLING OF COMBUSTION PROCESS WITH THE USE OF ANSYS FLUENT CODE

Mirosław Kowalski

Air Force Institute of Technology
Ksiecia Boleslawa 6, 01-494 Warsaw, Poland
tel.: +48 22 851301, fax: +48 22 851313
e-mail: miroslaw.kowalski@itwl.pl

Antoni Jankowski

Institute of Aviation
Krakowska Avenue 110/114, 02-256 Warsaw, Poland
tel.: +48 22 8460011, fax: +48 22 846432
e-mail: antoni.jankowski@ilot.edu.pl

Abstract

The article presents the modelling of the combustion process of liquid fuels using professional ANSYS FLUENT software. This program allows modelling the dynamics of compressible and incompressible, laminar and turbulent flows as well as heat exchange phenomena with occurrence and without chemical reactions. The model presented in the article takes into account the influence of the gas phase on the liquid phase during the fuel combustion process. The influence of velocity and pressure of the flowing gas and the type of flow has a significant impact on the combustion of liquid fuels. The developed model is fully reliable and the presented results are consistent with experimental research. The occurrence of a laminar sublayer in a turbulent flow was confirmed, and the thickness of this layer and the turbulent layer significantly influences the course of the combustion process. The use of the flat flow model reflects the basic phenomena occurring during the combustion of liquid fuels under turbulent conditions. The use of the program for flows with different flow velocity profiles is justified. It gives important information about the processes taking place during the combustion of liquid fuels. The results of numerical tests are presented graphically. The article presents graphs of velocity field, absolute pressure, power lines, temperature and density.

Keywords: *liquid fuels, combustion processes, turbulent flow, combustion process model, ANSYS FLUENT*

1 Introduction

In order to compare the experimental results obtained in tests on a special position in which the flow was realized with the addition of the weight, with the capabilities of a theoretical analysis of such phenomena occurring in the flow, calculations based on the modelling of the flow in the pipe were performed. In modelling, the ANSYS FLUENT program, which gives the possibility to model a wide range of issues related to computational flow dynamics (CFD), both in relation to the flow of compressible and incompressible fluids, laminar and turbulent flows, phenomena of transport, heat exchange, occurrence and without occurrence chemical reactions etc. was applied. The program allows analysing steady, transient and unsteady flows. Particular emphasis in this program is on the modelling of the phenomena occurring during combustion, which includes models of energy dissipation and models of the probability density function. If there is heat exchange or compressibility in the flows, the energy conservation equation is also solved. With regard to flows, in which substance mixing occurs, chemical reactions occur, particle conservation equation, or in the case of the non-premixed combustion model, the equations of the conservation of the mixture fractions and their variances are solved. Turbulent flows are characterized by fluctuations in velocity fields. These fluctuations cause the mixing of the transport size, such as moment, energy, particle concentrations, and are the reason for the fluctuation of transported quantities. Because these fluctuations can occur on a small scale, but with high frequency, they are time-consuming in engineering practice for direct calculations. Therefore, instead of these equations, which relate to a given moment, the equations averaged over a period may be given, or they may be modified in such a way as to remove solutions relating to a small scale, which greatly simplifies the solution of equations by computer methods. However, the modified equations may contain additional unknown variables, and therefore it was necessary to introduce turbulence models allowing determining these variables in relation to known models. There are many models of turbulence, the use of which depends on the course of the phenomenon being studied.

2 Results of numerical calculations

The calculation domain scheme

The calculations concerned turbulent flow, in which the mass of the factor was added at the bottom of the domain (Figure 1). The upper and sidewalls were set as isothermal with a temperature of 300 K. Additionally, the slip option was chosen for them. For the V_d 0.05 and 0.1 m s^{-1} velocities, the laminar flow model was used, and for 1 m s^{-1} velocity − the SST turbulent flow model, which was developed for solutions of similar cases.

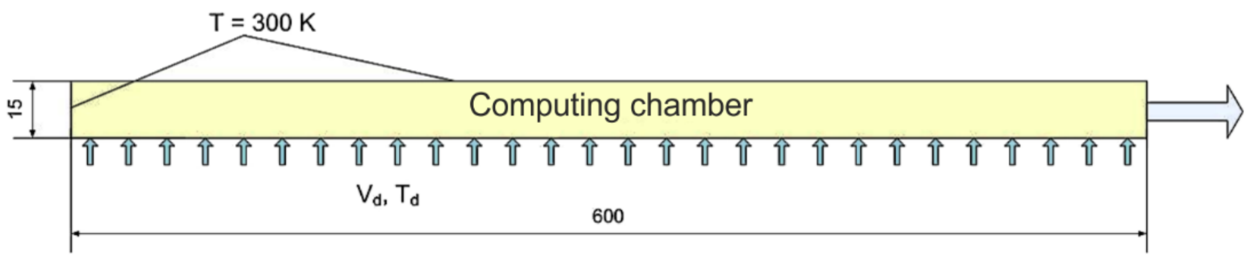

Fig. 1. Diagram of a two-dimensional computational domain with the addition of mass at the bottom of the channel for T = 300 K

The parameters of the analysed cases are presented in Tab. 1. The calculations for 9 variants were made. Velocity V_d, and T_d temperature side stream added.

Tab. 1. 9 variants of calculations

Item	V_d	T_d
1	0.05 m/s	300 K
2		500 K
3		1 000 K
4	0.1 m/s	300 K
5		500 K
6		1 000 K
7	1.0 m/s	300 K
8		500 K
9		1 000 K

Variant 1

Fig. from 1.1 to Fig. 1.6 show diagrams of fields: velocity V, velocity component Vx, velocity composition V_y, absolute pressure p and vector field, vector field of velocity and stream lines.

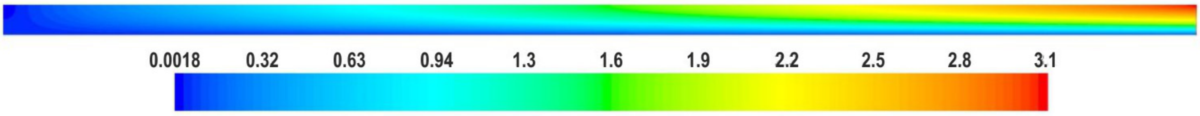

Fig. 1.1. Velocity

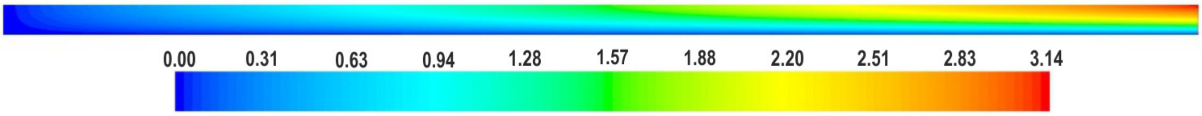

Fig. 1.2. Velocity component V_x

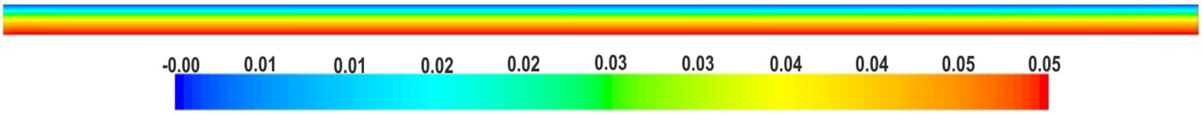

Fig. 1.3. Velocity component V_y

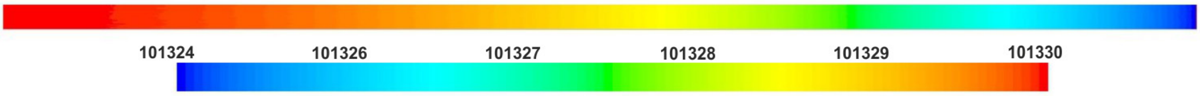

Fig. 1.4. Absolute pressure

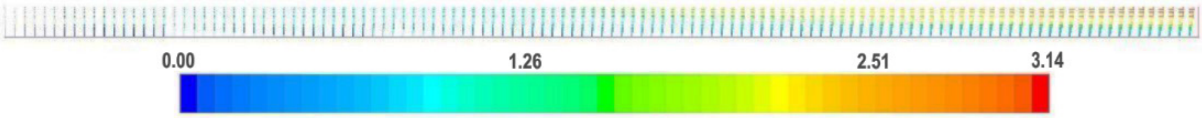

Fig. 1.5. Vector field of velocity

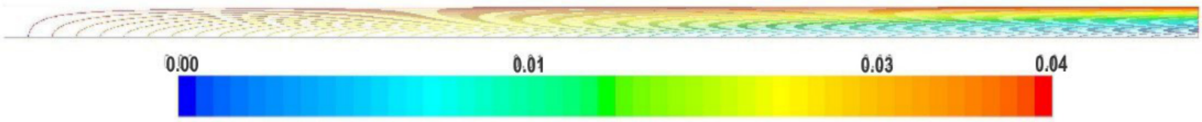

Fig. 1. 6. Stream lines

Variant 2

Fig. from 2.1 to Fig. 2.8 show diagrams of fields: velocity V, velocity component V_x, velocity composition V_y, absolute pressure p, vector field of velocity and stream lines, Temperature T and density ρ.

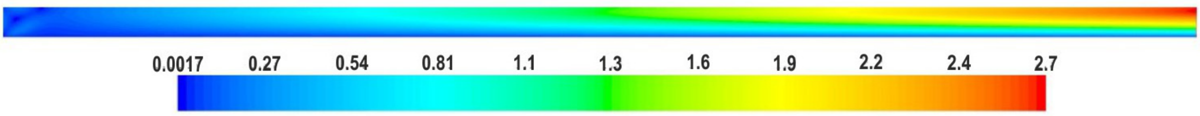

Fig. 2.1. Velocity

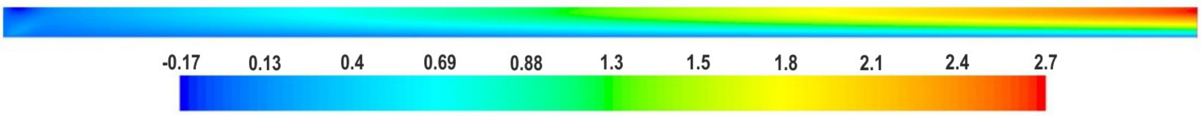

Fig. 2.2. Velocity component V_x

Fig. 2.3. Velocity component V_y

Fig. 2.4. Absolute pressure

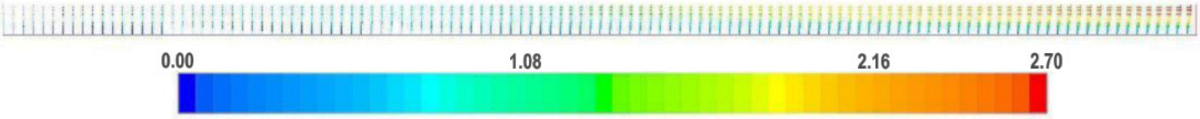

Fig. 2.5. Vector field of velocity

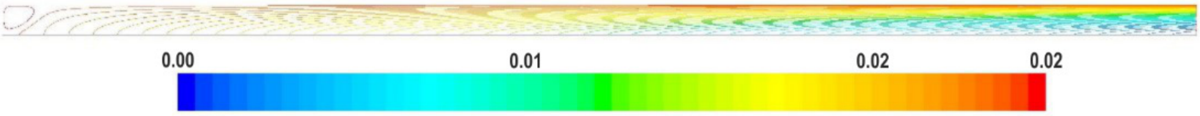

Fig. 2.6. Stream lines

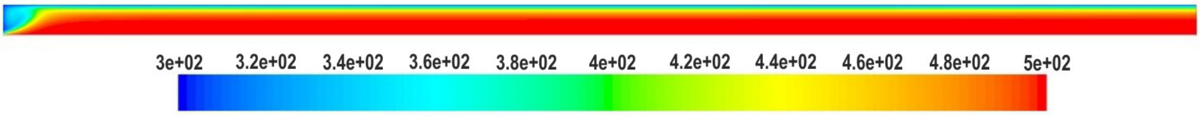

Fig. 2.7. Temperature field

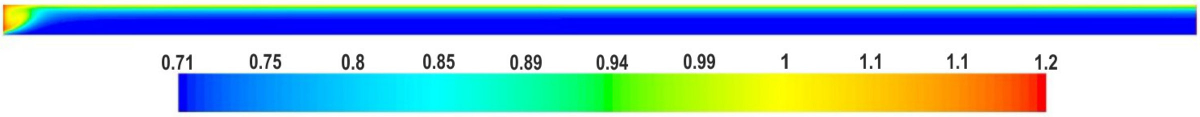

Fig. 2.8. Density field

Variant 3

In Fig. from 3.1 to 3.8 show diagrams of fields: velocity *V*, velocity component *Vx*, velocity composition *Vy*, absolute pressure p, vector field of velocity and stream lines, Temperature *T* and density ρ.

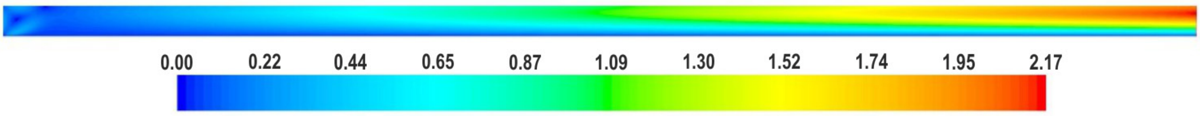

Fig. 3.1. Velocity

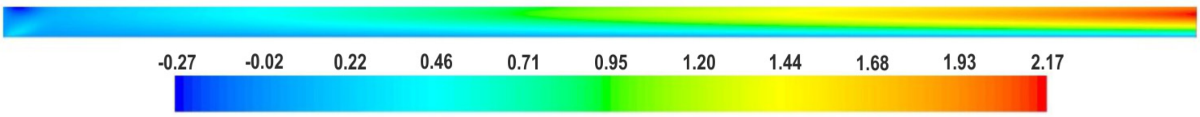

Fig. 3.2. Velocity component V_x

Fig. 3.3. Velocity component V_y

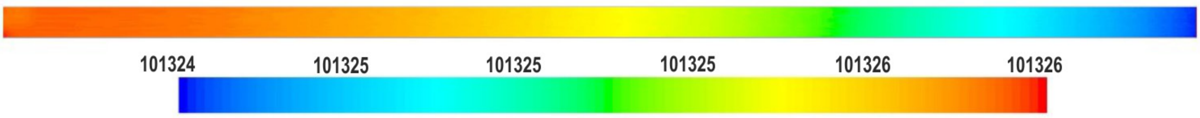

Fig. 3.4. Absolute pressure

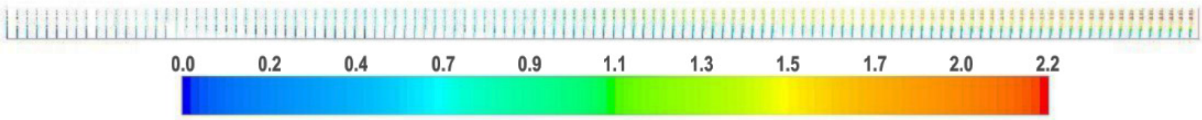

Fig. 3.5. Vector field of velocity

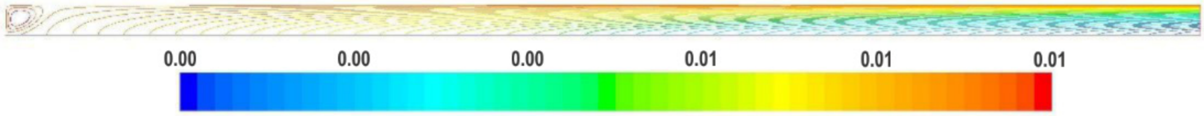

Fig. 3.6 Stream lines

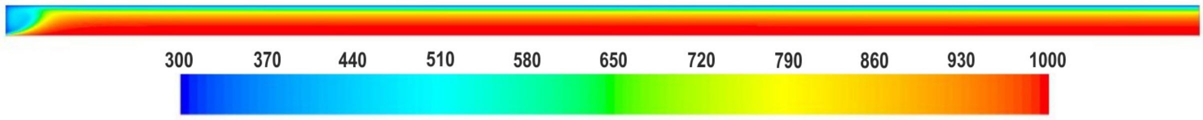

Fig. 3.7. Temperature field

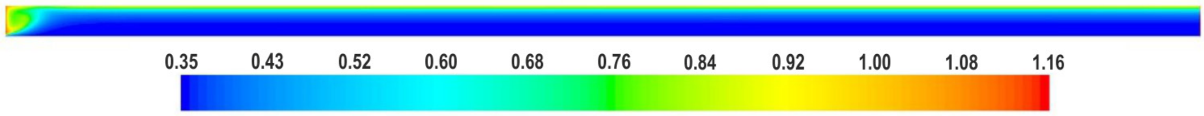

Fig. 3.8. Density field

Variant 4

In Fig. from 4.1 to 4.6 show diagrams of fields: velocity *V*, velocity component *Vx*, velocity composition *Vy*, absolute pressure p, vector field of velocity and stream lines.

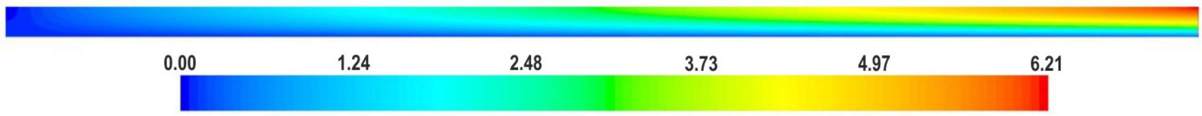

Fig. 4.1. Velocity

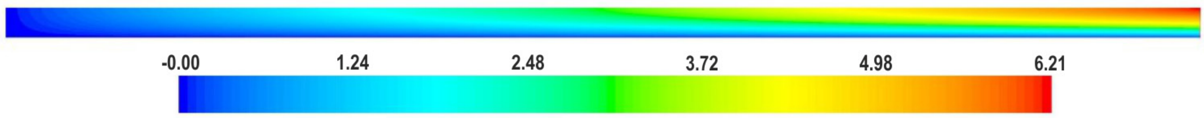

Fig. 4.2. Velocity component V_x

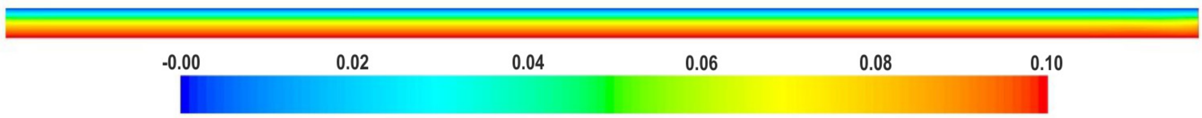

Fig. 4.2. Velocity component V_y

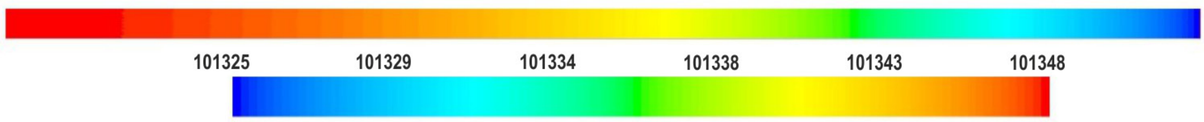

Fig. 4.4. Absolute pressure

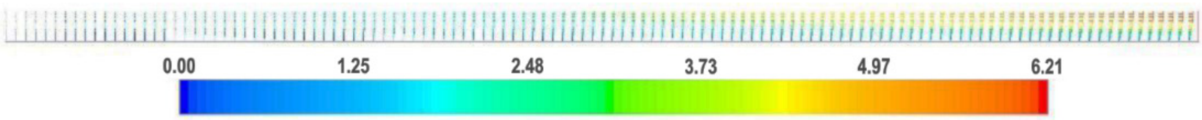

Fig. 4.5. Vector field of velocity

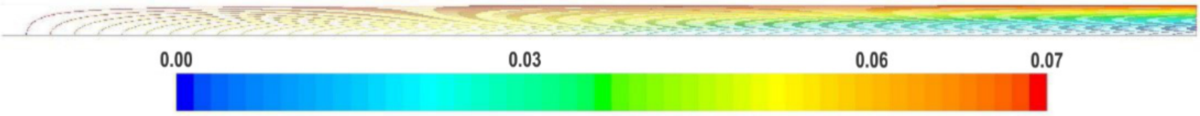

Fig. 4.6 Stream lines

Variant 5

In Fig. from 5.1 to 5.8 show diagrams of fields: velocity *V*, velocity component *Vx*, velocity composition *Vy*, absolute pressure p, vector field of velocity and stream lines, Temperature *T* and density *ρ*.

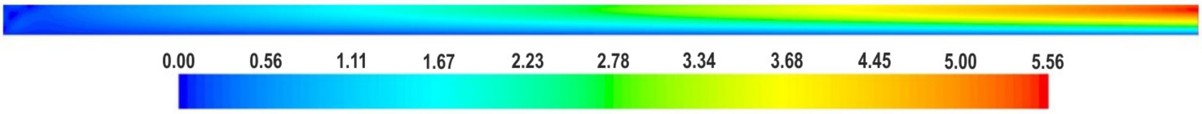

Fig. 5.1. Velocity

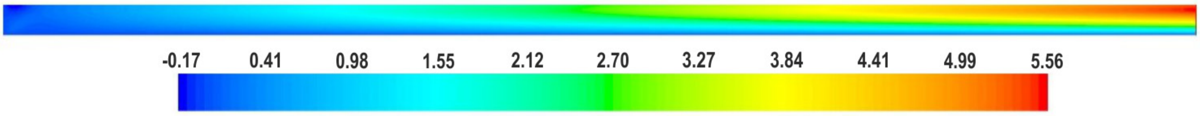

Fig. 5.2. Velocity component V_x

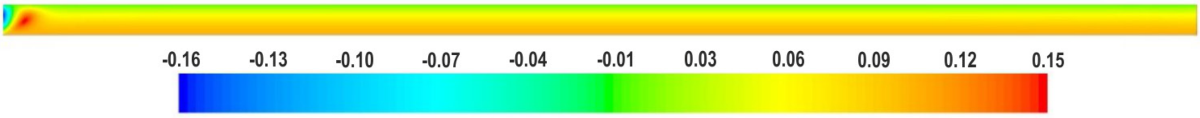

Fig. 5.2. Velocity component V_y

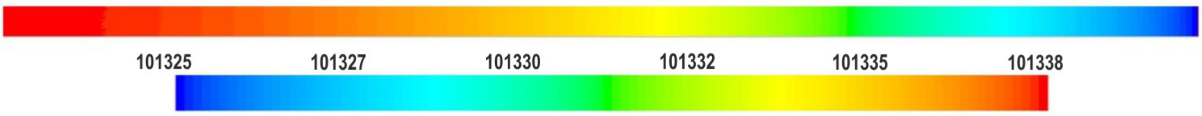

Fig. 5.4. Absolute pressure

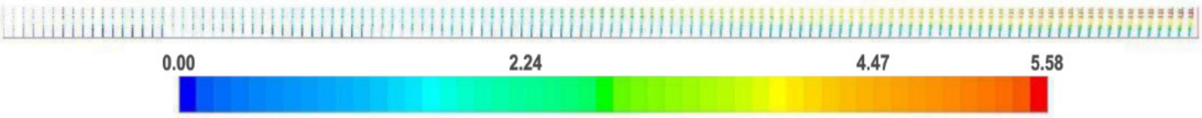

Fig. 5.5. Vector field of velocity

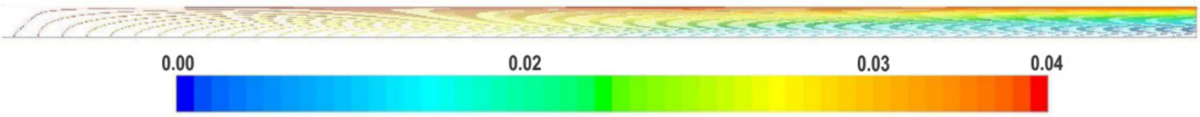

Fig. 5.6 Stream lines

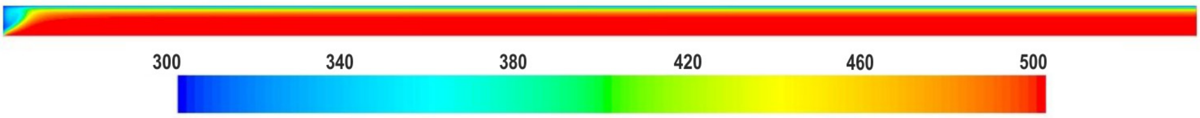

Fig. 5.7. Temperature field

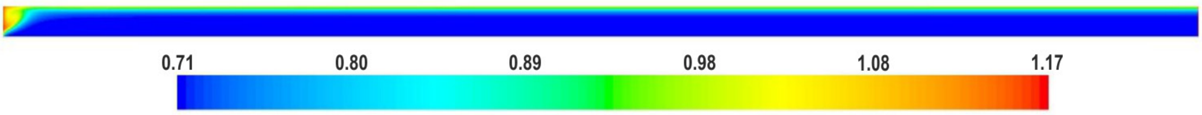

Fig. 5.8. Density field

Variant 6

In Fig. from 6.1 to 6.8 show diagrams of fields: velocity *V*, velocity component *Vx*, velocity composition *Vy*, absolute pressure p, vector field of velocity and stream lines, Temperature *T* and density *ρ*.

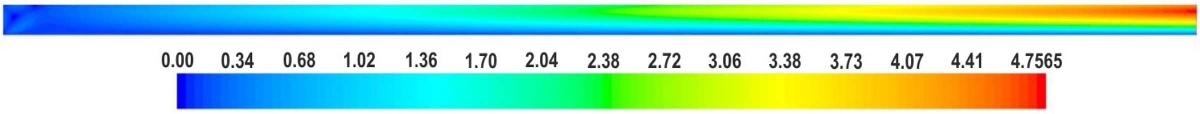

Fig. 6.1. Velocity

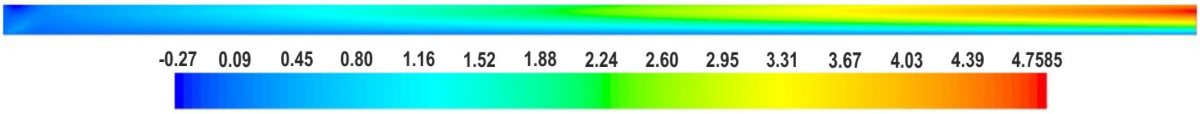

Fig. 6.2. Velocity component V_x

Fig. 6.2. Velocity component V_y

Fig. 6.4. Absolute pressure

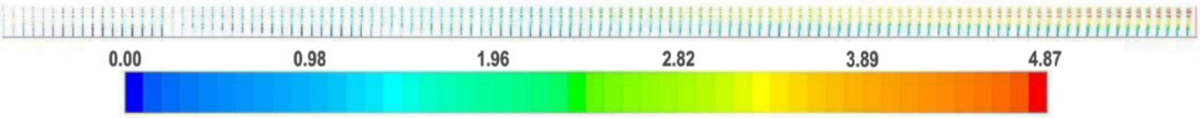

Fig. 6.5. Vector field of velocity

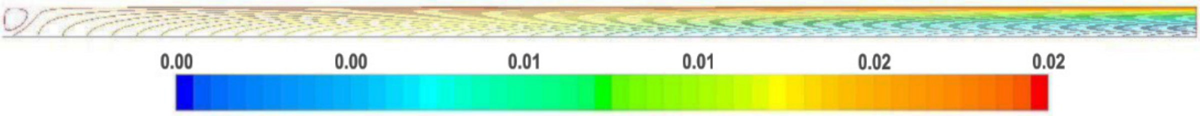

Fig. 6.6 Stream lines

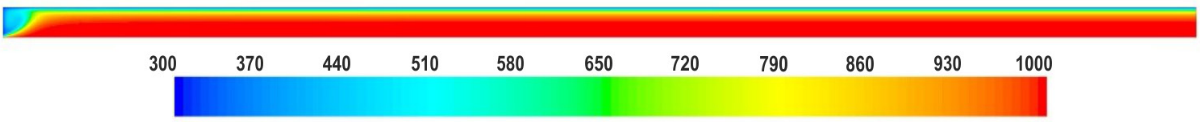

Fig. 6.7. Temperature field

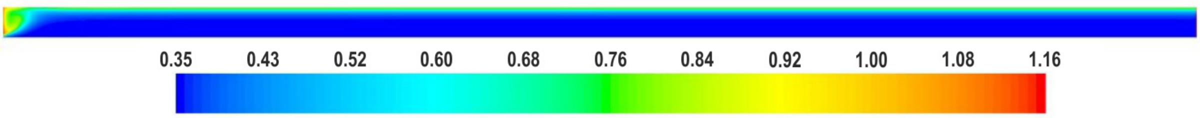

Fig. 6.8. Density field

Variant 7

In Fig. from 7.1 to 7.6 show diagrams of fields: velocity *V*, velocity component *Vx*, velocity composition *Vy*, absolute pressure p, vector field of velocity and stream lines, Temperature *T* and density *ρ*.

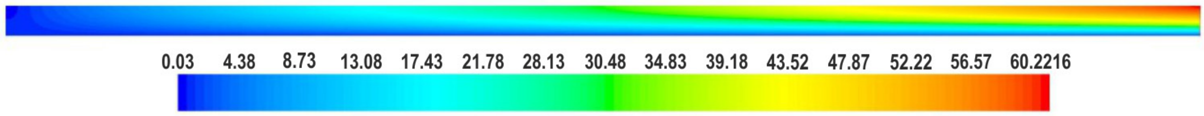

Fig. 7.1. Velocity

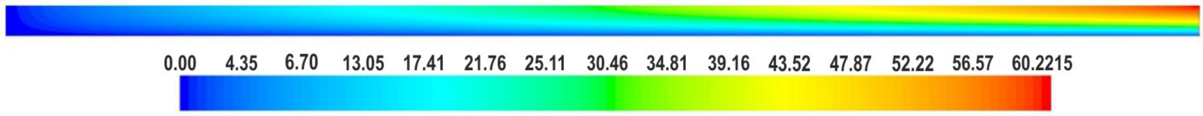

Fig. 7.2. Velocity component V_x

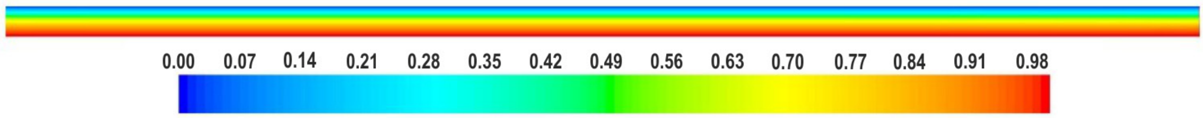

Fig. 7.3. Velocity component V_y

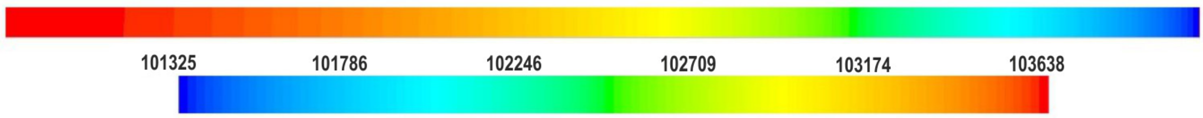

Fig. 7.4. Absolute pressure

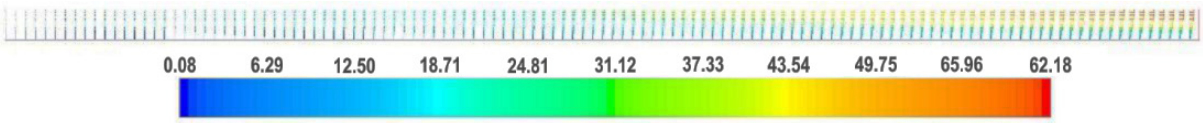

Fig. 7.5. Vector field of velocity

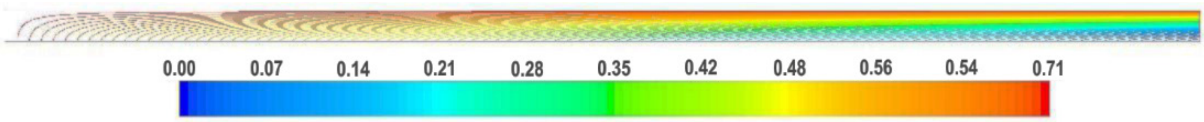

Fig. 7.6. Stream lines

Variant 8

In Fig. from 8.1 to 8.8 show diagrams of fields: velocity *V*, velocity component *Vx*, velocity composition *Vy*, absolute pressure p, vector field of velocity and stream lines, Temperature *T* and density *ρ*.

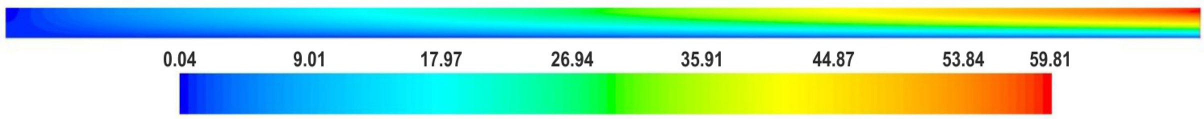

Fig. 8.1. Velocity

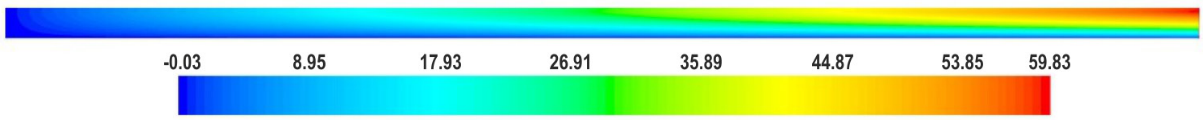

Fig. 8.2. Velocity component V_x

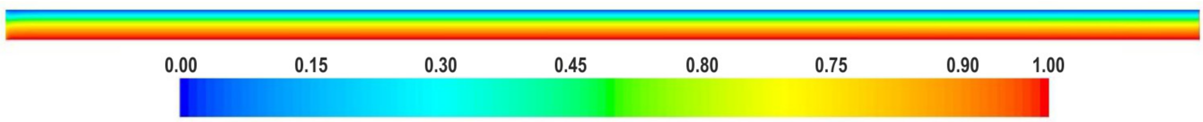

Fig. 8.2. Velocity component V_y

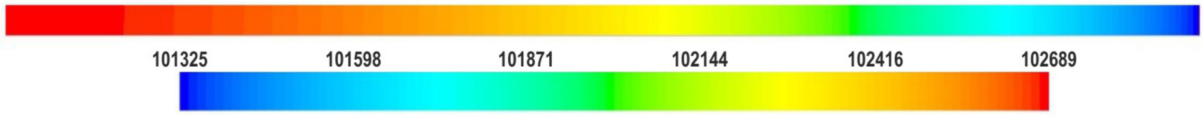

Fig. 8.4. Absolute pressure

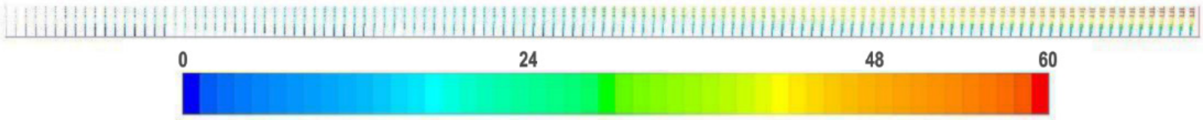

Fig. 8.5. Vector field of velocity

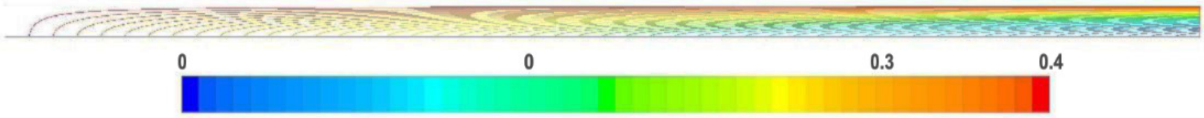

Fig. 8.6. Stream lines

Fig. 8.7. Temperature field

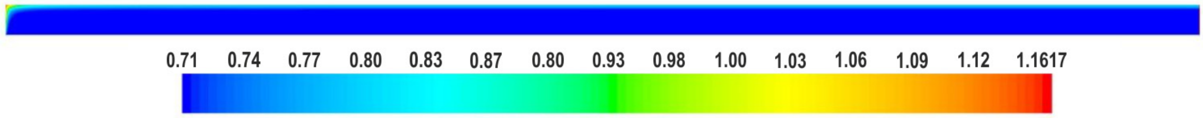

Fig. 8.8. Density field

Variant 9

In Fig. from 9.1 to 9.8 show diagrams of fields: velocity *V*, velocity component *Vx*, velocity composition *Vy*, absolute pressure p, vector field of velocity and stream lines, Temperature *T* and density *ρ*.

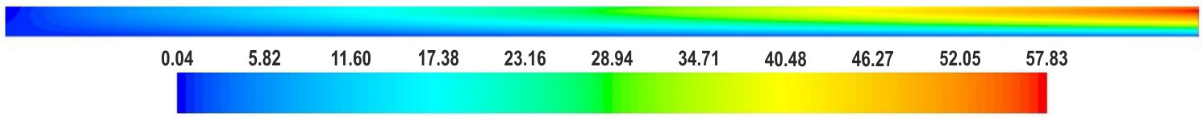

Fig. 9.1. Velocity

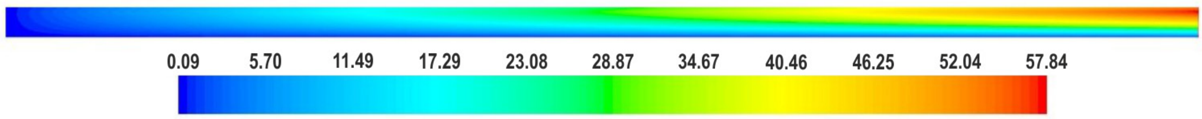

Fig. 9.2. Velocity component V_x

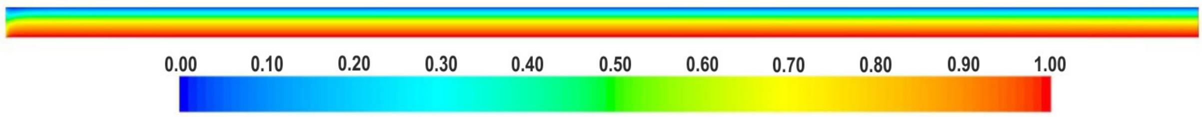

Fig. 9.3. Velocity component V_y

Fig. 9.4. Absolute pressure

Fig. 9.5. Vector field of velocity

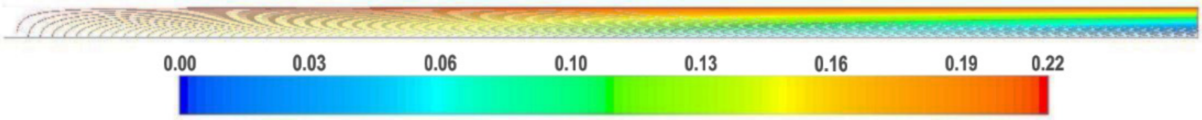

Fig. 9.6. Stream lines

Fig. 9.7. Temperature field

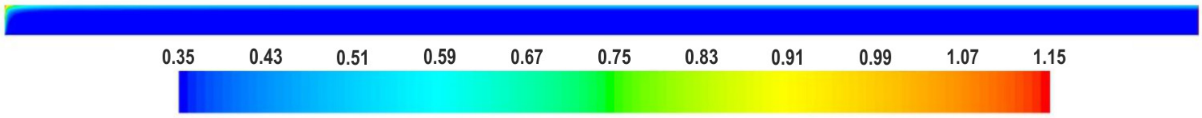

Fig. 9.8. Density field

3. Determination of the average speed

The distance from the beginning of the channel to the place, where the end of the air inlet is located was divided into six parts, every 100 m, and then in each of the cross-sections, the average values of horizontal velocity components V_x were determined. In the following diagrams (Fig. 10-12) they are presented for different velocities V_d and temperature T_d.

Using the simulation results shown in Fig. 1-9, the relationship between the velocity component V_x and the distance from the beginning of the channel for different values of temperature T_d was determined. Fig. 10 shows the graph for the speed V_d = 0.05 m / s, and in Fig. 69 for the speed V_d= 0.10 m / s. The velocity V_d models the velocity of adding a mass, which is gas with a temperature T_d.

In Fig. 12, the graph for velocity V_d = 1.0 m/s with reference to different values of the temperature of the added factor T_d is presented. The graphs in Fig. 68-70 show that the velocity of V_x increased as a function of the length of the channel and the higher the speed of adding mass V_d, the smallest differences in the value of the velocity component V_x were observed at different temperature values. Fig. 12 shows a graph for the velocity V_d = 1.0 m/s with reference to different values of the temperature of the added factor T_d. The graphs in Fig. 68-70 show that the speed V_x increased as a function of the length of the channel and the higher the speed of adding mass V_d, the smallest differences in the value of the component velocity V_x were observed at different temperature values. The calculations presented model the flows in the test chamber on the research bench. The following distributions were obtained: velocity, pressure, vector field of the tide, current line, temperature field and density field in a test chamber having a rectangular cross-section. Then, the axial velocity waveforms along the length for different mass addition rates and different temperatures of the added gas medium were compared on the graphs.

4. Conclusions

The theoretical model of the combustion process relating to liquid fuels has been developed. It takes into account the influence of the gas phase, since combustion is always present in this phase, and interacts with the liquid phase. We found that the velocity gas stream and the type of flow (laminar, transitional, or turbulent) have a significant impact on the burning liquid fuels, and the impact of pressure always exists, which forces the gas stream.

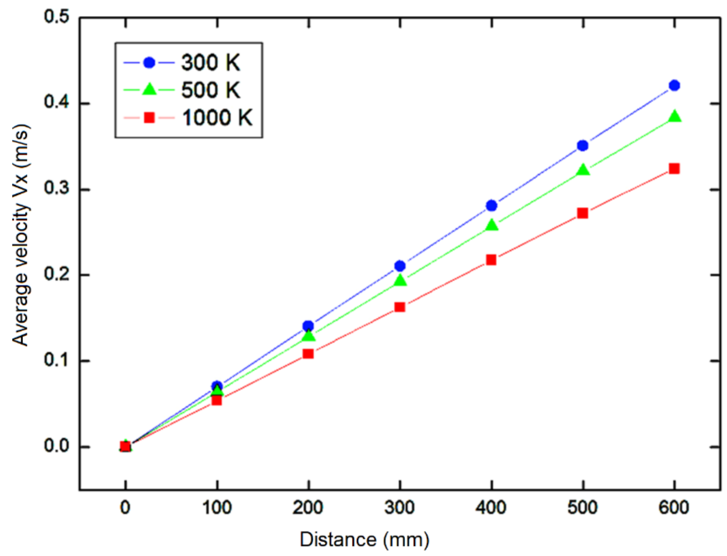

Fig. 10. Average speed Vx as a function of distance for Vd = 0.05 m/s

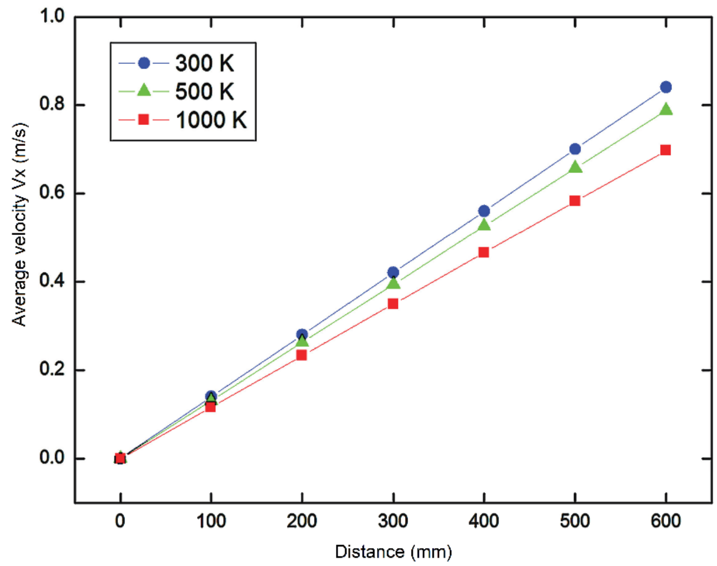

Fig. 11. Average speed Vx as a function of distance for Vd = 0.1 m/s

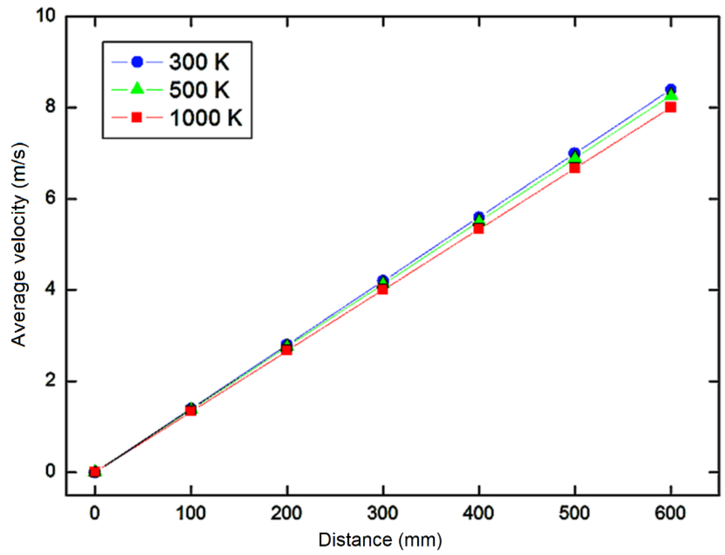

Fig. 12. Average speed V_x as a function of distance for V_d = 1.00 m/s

Developed a theoretical model burning has a good reference for an experiment conducted in conditions of model confirmed using numerical and experimental studies of the boundary layer. The results of numerical calculations confirmed the presence of the laminar sublayer in the turbulent flow. The thickness of the sublayer laminar and turbulent flow conditions laminar under laminar flow conditions has a significant impact on the course of the combustion process. The use of a flat flow model reflects the basic phenomena that occur during combustion of liquid fuels in turbulent conditions. In terms of turbulent flows basic relationship used in the modelling using ANSYS FLUENT were presented. Calculations for flows with different flow velocity profiles, modelling add mass and different values of module velocity were carried out. Velocity V_d modelling adding mass reflects the combustion process (adding mass to a stream of flowing gas). In place of breakdown the characteristics of velocity V_d on the inlet is an evident increase in the velocity component V_y with very small changes of this component in the rest of the channel. In the case of a fault this area increased significantly, and the breakdown velocity is greater, the greater the disturbance component of the velocity field.

References

[1]. *ANSYS FLUENT 12.0 Theory Guide*. 2009. [2] Gao J., Moon S., Zhang Y., Nishida K., Matsumoto Y., *Flame Structure of Wall Impinging Diesel Fuel Sprays Injected by Group-Hole Nozzles,* Combustion and Flame, Vol. 156, pp. 1263-1277, 2009.

[2]. Genzale, C. L., Reitz, R. D., Musculus, M. P. B., *Optical Diagnostics and Multi-Dimensional Modeling of Spray Targeting Effects in Late-Injection Low-Temperature Diesel Combustion,* SAE International Journal of Engines, Vol. 2, pp. 150-172, 2010.

[3]. Govardhan, J., Rao, G.V.S. In*fluence of Oscillating Combustion on Thermal Boundary Layer in a Diesel Fired Crucible Furnace*, International Journal of Computer Information Systems and Industrial Management Applications, Vol. 2, pp. 056-068, 2010.

[4]. Jankowski, A., Czerwinski, J., *Memorandum of Prof. A. K. Oppenheim and an example of application of the Oppenheim correlation (OPC)* for the heat losses during the combustion in IC-engine,* Journal of KONES, Vol. 17, No. 2, pp. 181-104, Warsaw 2010.

[5]. Jankowski, A., *Laser research of fuel atomization and combustion process in the aspect of exhaust gases emission*, Journal of KONES, Vol. 15, No. 1, pp. 119-126, 2008.

[6]. Jankowski, A., *Laser research of fuel atomization and combustion process in the aspect of exhaust gases emission*, Journal of KONES, Vol. 15, No. 1, pp. 119-126, Warsaw 2008.

[7]. Jankowski, A., *Laser Research of Fuel Atomization and Combustion Processes in the Aspect of Exhaust Gases Emission,* Journal of KONES, Vol. 15, No. 1 pp. 119-126, Warsaw 2008.

[8]. Jankowski, A., *Some Aspects of Heterogeneous Processes of the Combustion Including Two Phases*. Journal of KONES. Vol. 12, No. 1-2, pp. 121-134, Permanent Committee of KONES, Warsaw 2005.

[9]. 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.

[10].Jankowski, A., *Test Stand for Modelling of Combustion Processes of Liquid Fuels,* Journal of KONES, Vol. 21, No. 2, pp. 121-126, 2014.

[11].Kirchhartz, R. M., Mee, D. J., Stalker, R. J., *Effect of Boundary Layer Thickness and Entropy Layer on Boundary Layer Combustion,* 16th Australasian Fluid Mechanics Conference, Gold Coast, Australia, December 2007.

[12].Xia, J., Luo, K. H., *Conditional Statistics of Inert Droplet Effects on Turbulent Combustion in Reacting Mixing Layers,* Combustion Theory and Modelling, Vol. 13, pp. 901-920, 2009.

[13].Zurek, J., Jankowski, A., *Experimental and Numerical Modelling of Combustion Process of Liquid Fuels under Laminar Conditions,* Journal of KONES, Vol. 21. No. 3, pp. 30-316, Warsaw 2014.

Manuscript received 23 July 2018; approved for printing 24 September 2018