POLITYKA ENERGETYCZNA – ENERGY POLICY JOURNAL

2022 ✦ Volume 25 ✦ Issue 3 ✦ 21–34

DOI: 10.33223/epj/152805

Konstantin Vasilev Kostov[1], Ivan Nikolaev Denev[2], Neven Yordanov Krystev[3]

Research of the combustion process in the initial mixing section of the injection gas burner

Abstract: The economical combustion of gas fuel implies that it takes place with a minimum coefficient of excess air and minimal losses. Constructive, aerodynamic and physical factors have a determining influence on the completeness of combustion and the conditions of ignition. Using the ANSYS software program, the main characteristics of the combustion process in the cylindrical mixing section of a flat flame injection burner are investigated through computer simulation. A geometric model was created on which it is possible to study both straight and rotating jets. The possibility of numerically investigating the combustion of gaseous fuel (C_3H_8) in a confined air flow produced by injection is considered. A *k*-ε model of turbulence was used, which is based on the equation for turbulent kinetic energy and its dissipation rate. The purpose of the work is to study and analyze the changes and distribution of temperature and speed as well as the concentration of nitrogen oxides and carbon monoxide along the axis of the combustion chamber. The results are presented for the angles of inclination of the nozzles of 45° and 0°. Based on these, an analysis was made, where it

✉ Corresponding Author: Konstantin Vasilev Kostov; e-mail: kostov_77@abv.bg

[1] Department of Mechanical Engineering, Manufacturing and Thermal Engineering, Technical University of Sofia, Faculty of Engineering and Pedagogy of Sliven, Bulgaria; ORCID iD: 0000-0001-6134-6783; e-mail: kostov_77@abv.bg

[2] Department of Mechanical Engineering, Manufacturing Engineering and Thermal Engineering, Technical University of Sofia, Bulgaria; ORCID iD: 0000-0001-6086-0764; e-mail: ivan_denev.eng@abv.bg

[3] Department of Mechanical Engineering, Manufacturing and Thermal Engineering, Faculty of Engineering and Pedagogy of Sliven, Technical University of Sofia, Bulgaria; ORCID iD: 0000-0003-3061-0726; e-mail: nkrystev@yahoo.com

was found that with the increase in the degree of rotation, the absolute values of the temperature increase and the change in the mass concentration of the fuel along the length of the mixing section can be used to regulate the combustion process. The created numerical model can be successfully used to determine the main parameters of the burner under the same initial conditions, changing the angle of inclination of the nozzles. The obtained results can be considered as a basis for further research related to increasing the efficiency of the combustion process and lowering the harmful emissions produced by it.

Keywords: numerical research, combustion, gas fuel, regime and design parameters, harmful emissions

Introduction

Gaseous fuel is an energy source of growing importance for the public sector, energy and industry worldwide. The use of natural gas in industrial furnaces and boilers with the rational organization of its combustion is accompanied by a significant increase in the efficiency of the unit as a whole, improving the quality of heated products, reducing fuel consumption and reducing emissions. In order to increase the efficiency of the use of gaseous fuel, new operational solutions are being sought in the field of fuel technology and control systems for combustion devices. Numerical modelling and the simulation of combustion processes provides an effective tool for obtaining quantitative results of basic torch parameters with reliable accuracy (Xue et al. 2020; Wang et al. 2022; Yuanshu et al. 2022; Khabbazian et al. 2022; Ongar et al. 2022). There are enough examples in the literature for it to be expected that based on mathematical modelling, reliable data on the structure of the torch can be obtained (Ramos 1981; Kaufmann et al. 2002; Bernardi et al. 2003; Guo et al.2019). A prerequisite for numerical research is the difficulty that accompanies this type of research in real conditions – the presence of accurate recording elements, the interference they bring in the course, the long period of time and the high costs of the implementation of the full-scale model. Although diffusion combustion is the most common organization of the combustion process, injectable combustion devices are used in some industries (Markovich et al. 2014; Sharaborin et al. 2016; Lu and Chen 2018; Chanphavong and Zainal 2019; Mafalda et al. 2022; Preethi et al. 2022).

The present paper presents the results of a computer simulation of the combustion process in the mixer of an injection flat flame burner when the fuel leaks under high pressure and draws air from the surrounding space. The possibility for determining the main parameters of the combustion process and its management is shown.

The ANSYS/CFX program was used to study the combustion process. The software product ANSYS offers a number of possibilities for the numerical simulation of heat and mass transfer processes, combustion processes and flows of multiphase flows (Penkova et al. 2020; Krumov et al. 2021; Gaurav et al. 2021).

1. Problem formulation

The aim of the present work is to investigate the main characteristics of the combustion process in the cylindrical mixing section of an injection flat flame burner using computer simulation.

Such information would be very useful in terms of managing and regulating the combustion process in real conditions.

A simplified geometric 3D model of the coupled injection rotary apparatus and the mixing tract of an injection flat flame burner in SolidWorks environment has been compiled. The model was imported into the design modeler of ANSYS CFX.

With the geometric model thus created, it is possible to study both direct and rotating jets at 0°; 30°; 45°; 60° (Krystev 2021).

The combustion of propane gas (C_3H_8), which flows through a system of nozzles directed tangentially to the inner surface of the mixer, is simulated. The environment is air at atmospheric pressure. The walls of the mixer are adiabatic. The length of the mixer is L = 200 mm. Its inner diameter is Ø = 69 mm.

The choice of network type as well as its density is determined in the ANSYS Workbench Mesh. An automatic crosslinking method was used in these studies. The constructed 3D model of the coupled rotating nozzles, the mixing section and the crosslinking of the model are presented in Figure 1.

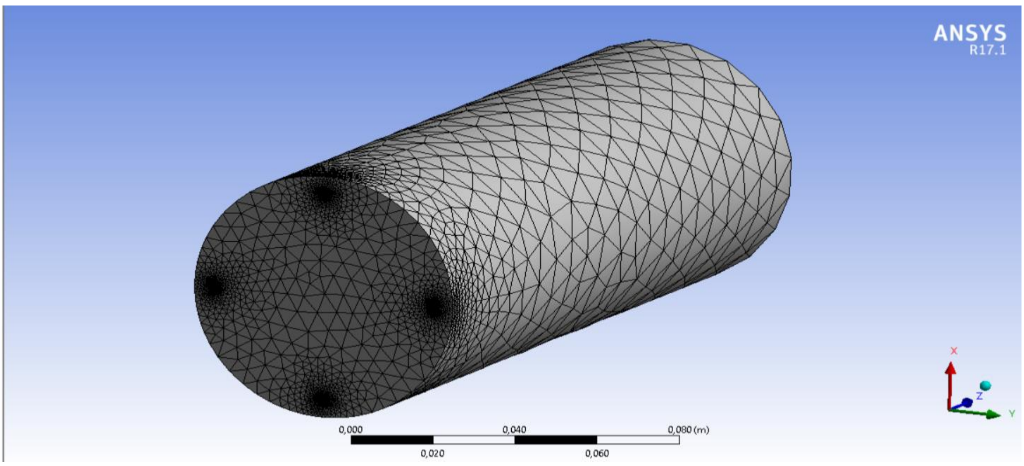

Fig. 1. 3D model of the coupled rotating nozzles and mixing section with a network of finite elements

Rys. 1. Model 3D sprzężonych dysz obrotowych i sekcji mieszania w sieci elementów skończonych

The purpose of ANSYS Workbench networking is to provide stable and easy-to-use networking tools that simplify network generation. Furthermore, the user is given the opportunity to control the network.

The mixing chamber with the injection rotary apparatus is part of the injection burner. The conditions of mixture formation are formed in the mixer and the combustion process takes place.

The following fuel models are available in ANSYS CFX: Eddy Dissipation, Finite Rate Chemistry, Combined EDM/FRC, Laminar Flamelet, Burning Velocity.

Analyzing and considering the individual models, the Finite Rate Chemistry (FRC) model taking into account the consequences of the final rates of chemical reactions is not appropriate in this case. This is due to the fact that the fuel (C_3H_8) and the oxidizer (air) are subjected to “fast” combustion (combustion rate dominates over the mixing speed of the reactants) (Fooladgar et al. 2021; Chung et al. 2021).

The Eddy Dissipation model and the Burning Velocity model are suitable for modelling “fast” combustion.

A standard *k*-ε model of turbulence was chosen. The standard *k*-ε turbulence model is a semi -empirical model based on the transport equations for turbulent kinetic energy (*k*) and dissipation rate (ε). According to this model, the flow is assumed to be fully turbulent and the effects of molecular viscosity are negligible. For this reason, it applies to fully developed turbulent currents. The equation for turbulent kinetic energy and the rate of its dissipation are:

$$\frac{\partial \rho k}{\partial \tau}+\frac{\partial(\rho V_x k)}{\partial x}+\frac{\partial(\rho V_y k)}{\partial y}+\frac{\partial(\rho V_z k)}{\partial z}=\frac{\partial}{\partial x}\left(\frac{\mu_t}{\sigma_k}.\frac{\partial k}{\partial x}\right)+\frac{\partial}{\partial y}\left(\frac{\mu_t}{\sigma_k}.\frac{\partial k}{\partial y}\right)+\frac{\partial}{\partial z}\left(\frac{\mu_t}{\sigma_k}.\frac{\partial k}{\partial z}\right)$$
$$+\mu_t\Phi-\rho.\varepsilon+\frac{C_4.\beta.\mu_t}{\sigma_t}.\left(g_x\frac{\partial T}{\partial x}+g_y\frac{\partial T}{\partial y}+g_z\frac{\partial T}{\partial z}\right) \tag{1}$$

$$\frac{\partial \rho \varepsilon}{\partial \tau}+\frac{\partial(\rho V_x \varepsilon)}{\partial x}+\frac{\partial(\rho V_y \varepsilon)}{\partial y}+\frac{\partial(\rho V_z \varepsilon)}{\partial z}=\frac{\partial}{\partial x}\left(\frac{\mu_t}{\sigma_\varepsilon}.\frac{\partial \varepsilon}{\partial x}\right)+\frac{\partial}{\partial y}\left(\frac{\mu_t}{\sigma_\varepsilon}.\frac{\partial \varepsilon}{\partial y}\right)+\frac{\partial}{\partial z}\left(\frac{\mu_t}{\sigma_\varepsilon}.\frac{\partial \varepsilon}{\partial z}\right)$$
$$+C_{1\varepsilon}\mu_t\frac{\varepsilon}{k}\varphi-C_2\rho.\frac{\varepsilon^2}{k}+\frac{C_\mu.(1-C_3).\beta.\rho}{\sigma_t}.\left(g_x\frac{\partial T}{\partial x}+g_y\frac{\partial T}{\partial y}+g_z\frac{\partial T}{\partial z}\right) \tag{2}$$

where:

- C_μ – the turbulence constant, *k*- turbulent kinetic energy,
- ε – the dissipation rate of the turbulent kinetic energy,
- σ_k, σ_ε – Schmid’s criteria for turbulent kinetic energy and the rate of its dissipation, respectively,
- σ_t – turbulent Prandtl (Schmid) criteria.

The combustion process model used corresponds to Arrhenius’ law (Yeh et al. 2013), which gives the relationship between the rate of a chemical reaction and the change in temperature depending on:

$$K = A_k T^{\beta_k} exp\left(\frac{E_k}{RT}\right) \tag{3}$$

where:

A_k – a pre-exponential factor,
β_k – a dimensionless temperature component,
E_k – the energy of activation,
T – the absolute temperature in K.

2. Results

The results obtained at the angles of inclination of the nozzles of 45° and 0° are presented.

On the basis of these results, an analysis was made of the possibilities for regulation and management of the combustion process, while increasing its efficiency and maintaining or reducing harmful emissions from it. Results of changes in temperature, the concentration of nitrogen oxides and carbon monoxide along the axis of the combustion chamber and the distribution of temperature and velocity in the intersecting plane centrally along the axis of the mixer are presented sequentially.

Figure 2 shows the change in temperature and Figure 3 presents the change in NOx concentration along the axis of the mixer at a nozzle angle of 45°. The temperature increases gradually

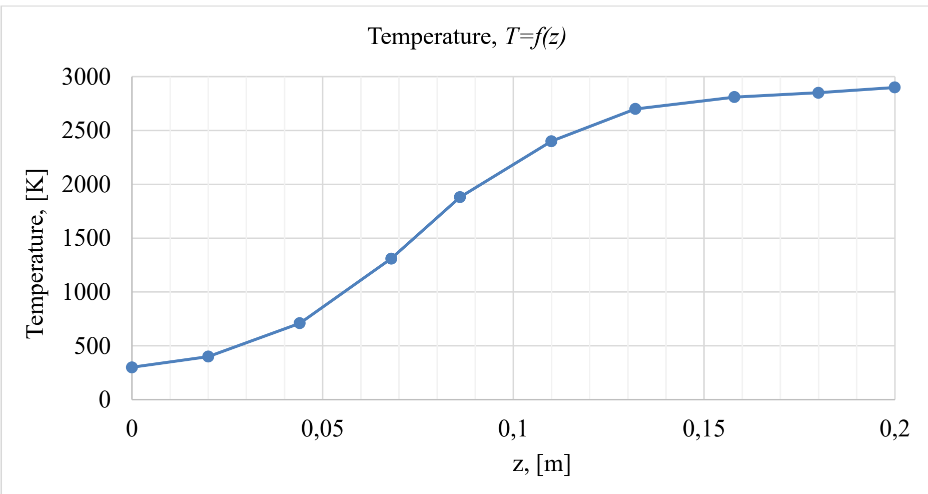

Fig. 2. Temperature change along the axis of the mixer at an angle of inclination of the nozzles of 45°

Rys. 2. Zmiana temperatury wzdłuż osi mieszadła przy kącie nachylenia dysz 45°

until the NOx concentration reaches its maximum at z = 0.16 m and then decreases. Under the same conditions (angle of inclination of the nozzles of 45°) a numerical study was conducted on the change in the concentration of CO, which is presented in Figure 4.

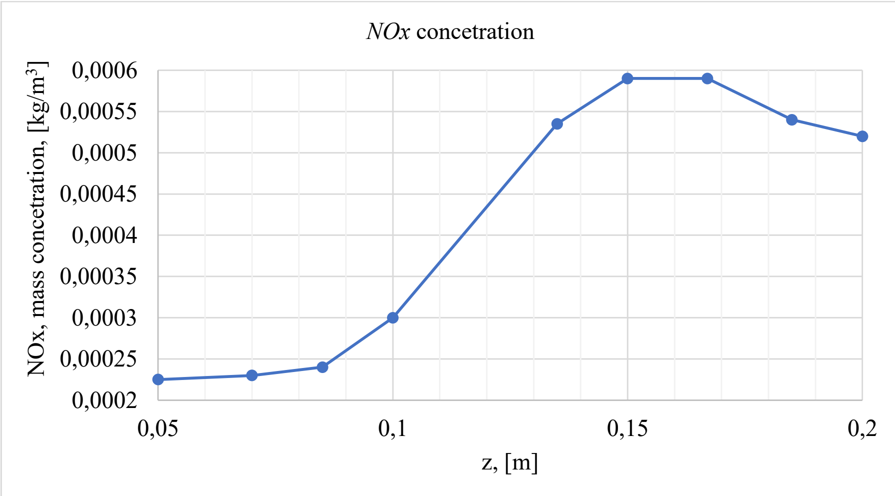

Fig. 3. Change in NOx concentration along the axis of the mixer at a nozzle angle of 45°

Rys. 3. Zmiana stężenia NOx wzdłuż osi mieszadła przy kącie dyszy 45°

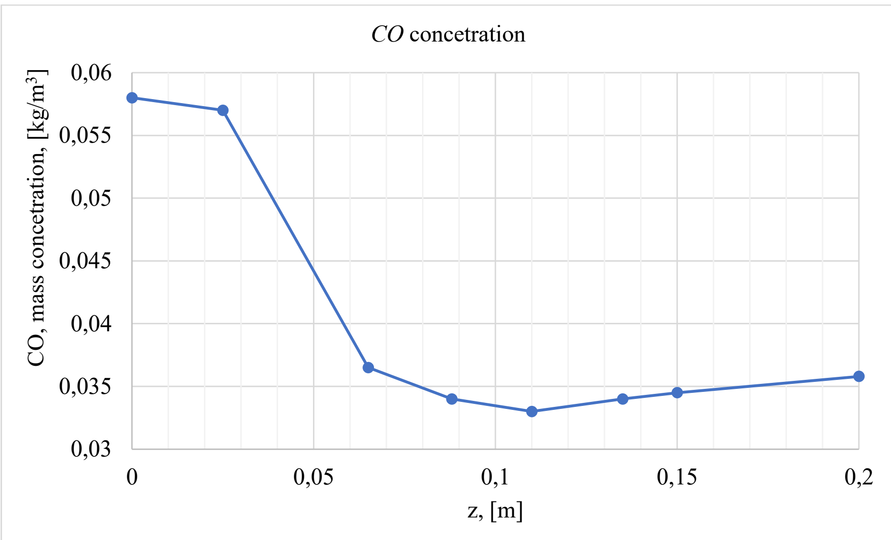

Fig. 4. Change in CO concentration along the axis of the mixer at a nozzle angle of 45°

Rys. 4. Zmiana stężenia CO wzdłuż osi mieszadła przy kącie dyszy 45°

Figures 5 and 6 show, respectively, the temperature and velocity fields in the intersecting plane through the axis of the mixer at an angle of inclination of the nozzles of 45°.

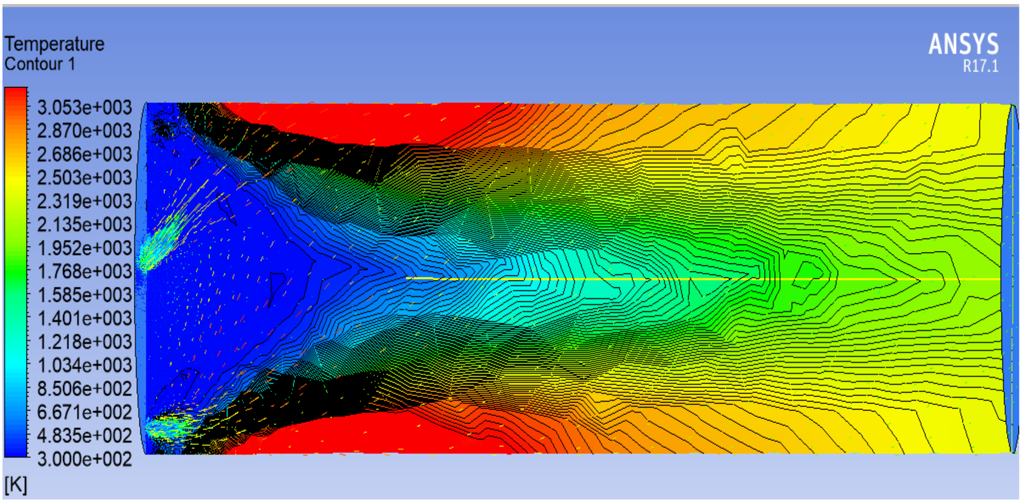

Fig. 5. Temperature fields in the intersecting plane through the axis of the mixer at an angle of inclination of the nozzles of 45°

Rys. 5. Pola temperatur w płaszczyźnie przecięcia przez oś mieszadła przy kącie nachylenia dysz 45°

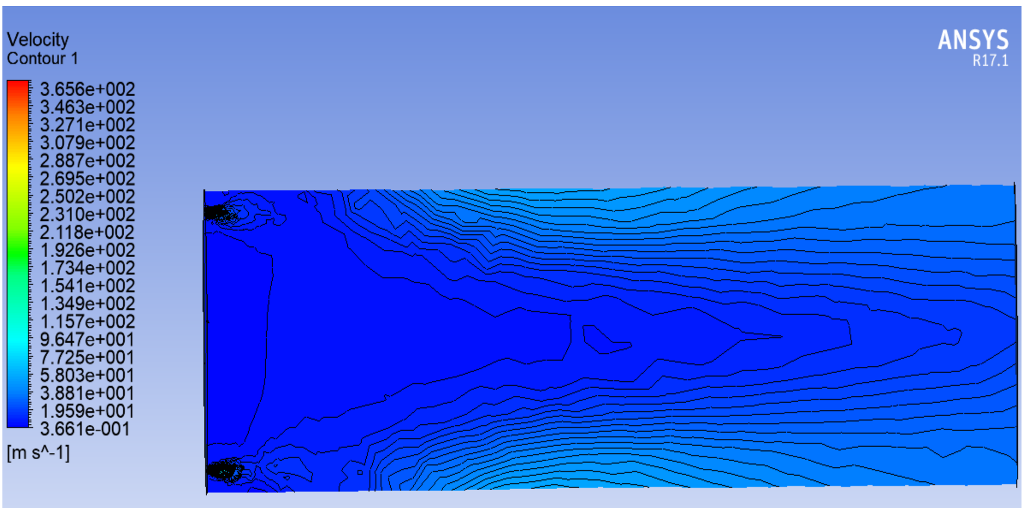

Fig. 6. Speed fields in the intersecting plane through the axis of the mixer at an angle of inclination of the nozzles of 45°

Rys. 6. Pola prędkości w płaszczyźnie przecięcia przez oś mieszadła przy kącie nachylenia dysz 45°

Figures 7, 8 and 9 present the results of numerical experiments along the axis of the mixer at an angle of inclination of the nozzles of 0°for the change in temperature, NOx concentration and CO concentration, respectively.

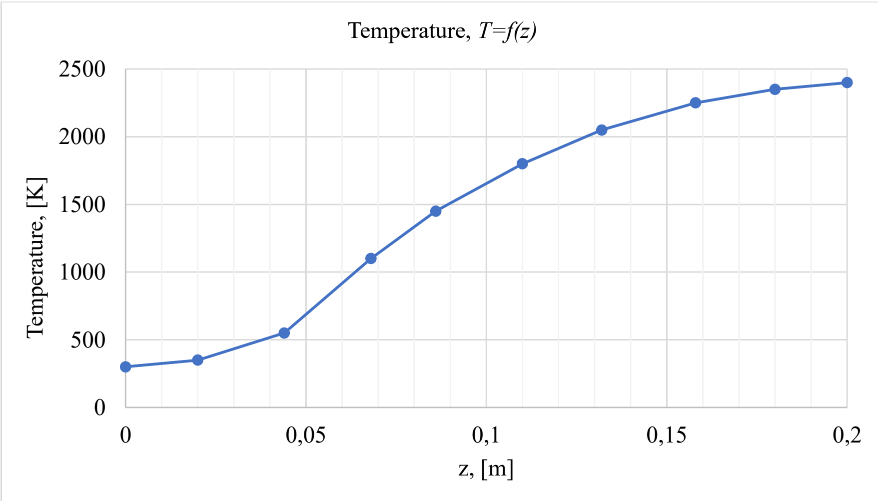

Fig. 7. Change in temperature along the axis of the mixer at an angle of inclination of the nozzles of 0°

Rys. 7. Zmiana temperatury wzdłuż osi mieszadła przy kącie nachylenia dysz 0°

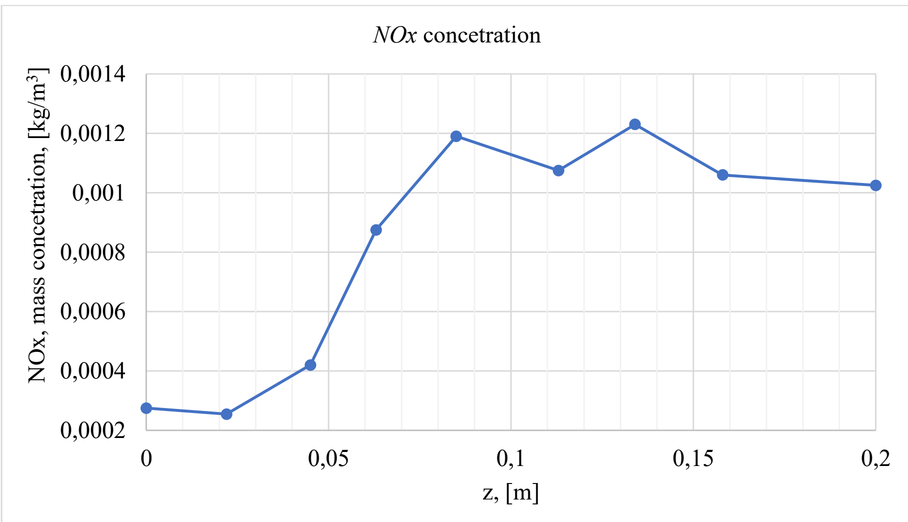

Fig. 8. Change in NOx concentration along the axis of the mixer at a nozzle angle of 0°

Rys. 8. Zmiana stężenia NOx wzdłuż osi mieszadła przy kącie dyszy 0°

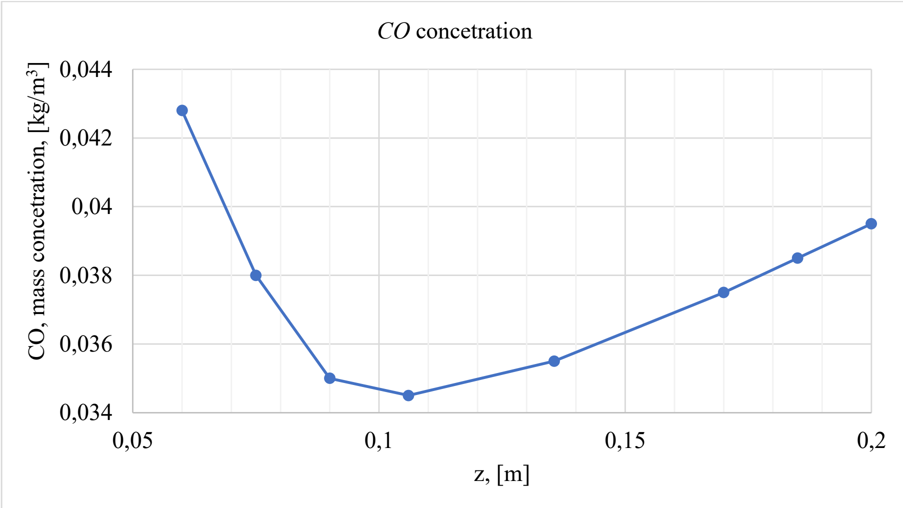

Fig. 9. Change in CO concentration along the axis of the mixer at a nozzle angle of 0°

Rys. 9. Zmiana stężenia CO wzdłuż osi mieszadła przy kącie dyszy 0°

Figures 10 and 11 show the obtained data for the temperature and velocity fields in the intersecting plane through the axis of the mixer at an angle of inclination of the nozzles of 0°.

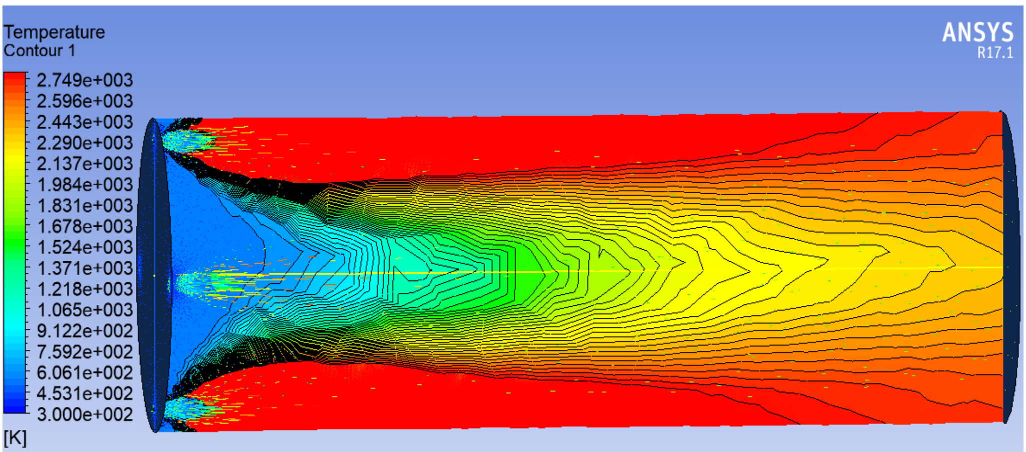

Fig. 10. Temperature fields in the intersecting plane through the axis of the mixer at an angle of inclination of the nozzles of 0°

Rys. 10. Pola temperatur w płaszczyźnie przecięcia przez oś mieszadła przy kącie nachylenia dysz 0°

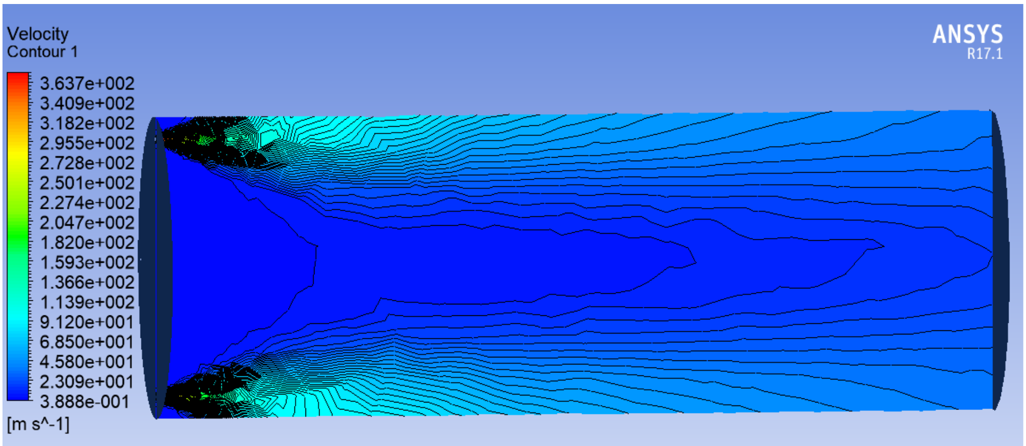

Fig. 11. Speed fields in the intersecting plane through the axis of the mixer at an angle of inclination of the nozzles of 0°

Rys. 11. Pola prędkości w płaszczyźnie przecięcia przez oś mieszadła przy kącie nachylenia dysz 0°

3. Discussion

Comparing the results obtained at nozzle angles of 45° and 0°, respectively, Figures 4 and 9 show that the use of a rotating jet leads to better mixing and as a final result, the intensification of the combustion process. The maximum temperature at the outlet cross section of the mixing path at a nozzle rotation angle of 45° is approximately 2900 K, while in a direct flow, under the same initial conditions it is approximately 2400 K.

When comparing the change in the concentration of nitrogen oxides as harmful emissions formed under high temperature conditions, it is seen that the maximum levels are twice as low in a rotating jet compared to a direct jet. As an explanation for the results obtained in a rotating jet are the presence of circulating zones (closed contours), both axial and wall, which cool the local high-temperature zone of their formation. With a direct current, such zones are missing. With regard to the change in the concentration of carbon monoxide as a factor for chemical incompleteness of combustion, it can be seen that when the jet is rotated, the levels are also lower compared to the direct jet.

Compared to non-rotating torch rotations has a number of advantages – the presence of circulation zones that increase the stability of the flare increased injection capability and intense turbulent exchange.

Conclusion

The conducted study presents the importance and technical application of the rotated jets in the combustion practice. In this article, the rotation of the jet is presented as a means of controlling the main characteristics of the burning torch. The study of the features of the rotated jet and their consideration in the creation of new methods, and the improvement of existing methods of engineering design, is the basis for the creation of efficient combustion devices that do not endanger the environment at complete combustion of the fuel.

A numerical model was created for research of the combustion process in the initial mixing compartment of the gas injection burner. The combustion of propane gas is simulated, which flows through a system of nozzles directed tangentially to the inner surface of the mixer. Based on the performed numerical studies, the temperature distribution and the concentration of harmful emissions were obtained. The assembled model can be successfully used to determine the main parameters of this type of burner, given the initial conditions and changing the angle of inclination of the nozzles. The results for changes in temperature and the concentration of nitrogen oxides and carbon monoxide along the axis of the combustion chamber are successively presented. The obtained results clearly demonstrate the possibility of how, just by changing the angle of inclination of the nozzles, it is possible to model the main parameters of the flare. The influence of the change in the angle of inclination of the nozzles, both on temperature levels and on the concentrations of harmful emissions in the combustion process, was established. The possibility of controlling the combustion process in this type of combustion device in order to improve the combustion efficiency with reduced harmful emissions has been proven. The obtained logical and plausible results give the authors reason to continue their work in this direction.

The authors would like to thank the research and development sector at the Technical University of Sofia for their financial support.

References

Bernardi et al. 2003 – Bernardi, D., Colombo, V., Ghedini, E. et al. 2003. Three-dimensional modelling of inductively coupled plasma torches. *The European Physical Journal D – Atomic, Molecular, Optical and Plasma Physics* 22, pp. 119–125, DOI: 10.1140/epjd/e2002-00233-9.

Chanphavong, L. and Zainal, Z.A. 2019. Characterization and challenge of development of producer gas fuel combustor: A review. *Journal of the Energy Institute* 92(5), pp. 1577–1590, DOI: 10.1016/j.joei.2018.07.016.

Chung et al. 2021 – Chung, W.T., Mishra, A.A., Perakis, N. and Ihme, M. 2021. Data-assisted combustion simulations with dynamic submodel assignment using random forests. *Combustion and Flame* 227, pp. 172–185, DOI: 10.1016/j.combustflame.2020.12.041.

Dwivedi et al. 2021 – Dwivedi, G., Gohil, P.P. and Behura, A.K. 2021. Numerical investigation of thermodynamic parameters for performance evaluation of cooking gas stove burner by appending of flame shield. *Materials Today: Proceedings* 46(11), pp. 5696–5702, DOI: 10.1016/j.matpr.2020.09.836.

Fooladgar et al. 2021 – Fooladgar, E., Brackmann, C., Mannazhi, M., Ögren, Y., Bengtsson, P.-E., Wiinikka, H. and Tóth, P. 2021. CFD modeling of pyrolysis oil combustion using finite rate chemistry. *Fuel* 299, DOI: 10.1016/j.fuel.2021.120856.

Guo et al. 2019 – Guo, X., Alavi, S., Dalir, E., Dai, J. and Mostaghimi, J. 2019. Time-resolved particle image velocimetry and 3D simulations of single particles in the new conical ICP torch. *Journal of Analytical Atomic Spectrometry* 34(3), pp. 469–479, DOI: 10.1039/C8JA00407B.

Kalin et al. 2021 – Kalin, K., Penkova, M. and Zlateva, P. 2021. Cfd analysis of compressible flows through nozzles and diffusers. *UPB Scientific Bulletin, Series D: Mechanical Engineering* 83(2), pp. 149–158.

Kaufmann et al. 2002 – Kaufmann, A., Nicoud, F. and Poinsot, T. 2002. Flow forcing techniques for numerical simulation of combustion instabilities. *Combustion and Flame* 131, pp. 371–385.

Khabbazian et al. 2022 – Khabbazian, G., Aminian, J. and Khoshkhoo, R.H. 2022. Experimental and numerical investigation of MILD combustion in a pilot-scale water heater. *Energy* 239(A), DOI: 10.1016/j.energy.2021.121888.

Liu et al. 2022 – Liu, Y., He, X., Feng, L., Xue, R., Zhang, Y., Xu, C. and Du, J. 2022. Numerical simulation of the interaction between shock train and combustion in three-dimensional M12-02 scramjet model. *International Journal of Hydrogen Energy* 47(12), pp. 8026–8036, DOI: 10.1016/j.ijhydene.2021.12.126.

Lu, W. and Chen, H. 2018. Design of cylindrical mixing chamber ejector according to performance analyses. *Energy* 164, pp. 594–601, DOI: 10.1016/j.energy.2018.09.025.

Markovich et al. 2014 – Markovich D.M., Abdurakipov S.S., Chikishev L.M., Dulin V.M. and Hanjalić, K. 2014. Comparative analysis of lowand high-swirl confined flames and jets by proper orthogonal and dynamic mode decompositions. *Physics of Fluids* 26(6), DOI: 10.1063/1.4884915.

Medeiros et al. 20122 – Medeiros, M.L.M.M., dos Santos, A. and Fernandes, E.C. 2022. Mathematical modelling and experimental study of an ejector burner. *Experimental Thermal and Fluid Science* 130, DOI: 10.1016/j.expthermflusci.2021.110482.

Neven, K. 2021. Multiplying the Effect of Nitrogen Oxides Reduction Under Vortex Burner Conditions at Gas Fuel Injection. *6th International Symposium on Environment-Friendly Energies and Applications (EFEA)*, pp. 1–4, DOI: 10.1109/EFEA49713.2021.9406255.

Ongar et al. 2022 – Ongar, B., Beloev, H., Iliev, I., Ibrasheva, A. and Yegzekova, A. 2022. Numerical simulation of nitrogen oxide formation in dust furnaces. *EUREKA: Physics and Engineering* 1, pp. 23–33, DOI: 10.21303/2461-4262.2022.002102.

Penkova et al. 2020 – Penkova, N., Krumov, K., Mladenov, B. and Stoyanov, Y. 2020. Modelling and numerical simulation of the heat transfer and natural ventilation in storage halls. *E3S Web of Conferences* 207, DOI: 10.1051/e3sconf/202020701005.

Ramos, J.I. 1981. A numerical study of turbulent swirling flows. Dept. of Mech. Eng., Carnegie-Mellon Univ., Pittsburgh, Pa.

Sharaborin et al. 2016 – Sharaborin, D.K., Dulin, V.M., Lobasov, A.S. and Markovich, D.M. 2016. Measurements of density field in a swirling flame by 2D spontaneous Raman scattering. *AIP Conference Proceedings* 1770, 030027, DOI: 10.1063/1.4963969.

Soundararajan et al. 2022 – Soundararajan, P.R., Durox, D., Renaud, A., Vignat, G. and Candel, S. 2022. Swirler effects on combustion instabilities analyzed with measured FDFs, injector impedances and damping rates. *Combustion and Flame* 238, DOI: 10.1016/j.combustflame.2021.111947.

Wang et al. 2022 – Wang, Y., Zhang, X. and Li, Y. 2022. Numerical simulation of methane-hydrogen-air premixed combustion in turbulence. *International Journal of Hydrogen Energy*, DOI: 10.1016/j.ijhydene.2022.05.167.

Xue et al. 2020 – Xue, R., Zheng, X., Yue, L., Zhang, S. and Weng, C. 2020. Study of shock train/flame interaction and skin-friction reduction by hydrogen combustion in compressible boundary layer. *International Journal of Hydrogen Energy* 45(31), pp. 15683–15696, DOI: 10.1016/j.ijhydene.2020.04.027.

Yeh et al. 2013 –Yeh, C.L., Hwang, P.W., Chen, W.K. and Li, J.Y. 2013. Modelling evaluation of Arrhenius factor and thermal conductivity for combustion synthesis of transition metal aluminides. *Intermetallic* 39, pp. 20–24, DOI: 10.1016/j.intermet.2013.03.021.

Konstantin Vasilev Kostov, Ivan Nikolaev Denev, Neven Yordanov Krystev

Badania procesu spalania w sekcji wstępnego mieszania wtrysku palnika gazowego

Steszczenie

Ekonomiczne spalanie paliwa gazowego oznacza, że odbywa się ono przy minimalnym współczynniku nadmiaru powietrza i minimalnych stratach. Czynniki konstrukcyjne, aerodynamiczne i fizyczne mają decydujący wpływ na kompletność spalania i warunki zapłonu. Za pomocą programu ANSYS, używając symulacji komputerowej, badano główne charakterystyki procesu spalania w cylindrycznej sekcji mieszania palnika wtryskowego z płaskim płomieniem. Powstał model geometryczny, na którym można badać zarówno strumienie proste, jak i wirujące. Rozważa się możliwość numerycznego badania spalania paliwa gazowego (C_3H_8) w zamkniętym strumieniu powietrza wytworzonym przez wtrysk. Zastosowano model turbulencji *k*-ε, który opiera się na równaniu energii kinetycznej turbulencji i szybkości jej rozpraszania. Celem pracy jest badanie i analiza zmian i rozkładu temperatury, a także prędkości oraz stężenia tlenków azotu i tlenku węgla wzdłuż osi komory spalania. Wyniki przedstawiono dla kątów nachylenia dysz 45° i 0°. Na ich podstawie przeprowadzono analizę, w której stwierdzono, że wraz ze wzrostem stopnia rotacji można wykorzystać wartości bezwzględne wzrostu temperatury i zmiany stężenia masowego paliwa na długości odcinka mieszania, do regulacji procesu spalania. Stworzony model numeryczny można z powodzeniem wykorzystać do wyznaczenia głównych parametrów palnika w tych samych warunkach początkowych, zmieniając kąt nachylenia dysz. Uzyskane wyniki można traktować jako podstawę do dalszych badań związanych ze zwiększeniem wydajności procesu spalania i obniżeniem wytwarzanych przez niego szkodliwych emisji.

Słowa kulczowe: badania numeryczne, spalanie, paliwo gazowe, reżim i parametry projektowe, emisje szkodliwe