NUMERICAL MODEL AND EXPERIMENTAL RESEARCH OF A NEW TWO-TIER MICRO-FLAME COMBUSTOR FOR GAS TURBINES

by

Abay M. DOSTIYAROV^a, Iliya K. ILIEV^b, and Ainura K. MAKZUMOVA^{c*}

^a Gumarbek Daukeyev Almaty University of Power Engineering and Telecommunications, Almaty, Kazakhstan ^b Angel Kanchev University of Ruse, Ruse, Bulgaria

°L. N. Gumilyov Eurasian National University, Astana, Kazakhstan

Original scientific paper https://doi.org/10.2298/TSCI240318141D

The study of a novel two-tier micro-flame combustor for gas turbines involved comprehensive modelling using SolidWorks and ANSYS FLUENT software packages, alongside physical experiments. Analysis of air-flow velocities, pressures, temperatures within the combustion chamber, and NO_x emissions led to several key findings. The SolidWorks modelling confirmed stabilized air-flow and uniform pressure distribution, enhancing fuel combustion. Optimal air-flow velocities (15 m/s and 30 m/s) for different operating modes were determined, highlighting the importance of air-flow regulation. The ANSYS FLUENT analysis revealed efficient mixing of the fuel-air mixture through the device's two tiers. The analysis of NO_x emissions confirmed compliance with regulatory standards, up to 15 ppm. These results underscore the efficiency and stability of the combustor, making a significant contribution the advancement of gas turbines while meeting requirements for efficiency and environmental safety.

Key words: micro-flame combustion, combustor, combustion chamber, a gas turbine, experimental stand, numerical modelling

Introduction

In recent decades, the energy industry has been experiencing a period of intense development and transformation in response to the challenges of climate change, sustainable development, and ensuring energy security. In this context, gas turbine installations (GTI) are attracting increasing attention as an efficient and environmentally friendly source of energy.

Worldwide, the development of GTI is closely linked to the constant pursuit of increasing energy efficiency, reducing emissions, and minimizing environmental impact. Countries are actively investing in research and development aimed at creating more advanced and economically viable technologies in the field of GTI.

In Kazakhstan, as in many other countries, the energy sector plays an important role in ensuring sustainable economic growth and social development. With extensive reserves of natural resources, including gas and oil [1], Kazakhstan has the potential for efficient use of gas turbine technologies in electricity production.

^{*}Corresponding author, e-mail: a.makzumova@gmail.com

At the same time, countries with sharply continental climates, such as Kazakhstan, face challenges related to energy supply instability and seasonal fluctuations in energy consumption. The RES, including solar and wind energy, cannot always provide reliable energy supply under such conditions. This emphasizes the need for the development of alternative energy sources capable of ensuring stable and efficient electricity production in extreme climatic conditions.

According to [2, 3], GTI, among traditional energy sources, represent one of the most efficient and environmentally friendly options for providing base and backup power in energy-intensive industries. Their high efficiency, operability, and relatively low emission levels make them an attractive choice for many countries seeking to modernize their energy infrastructure and reduce environmental impact.

Purpose and objectives of the research

The purpose of this article is to investigate the new two-stage micro-flame combustor (MFC) for the combustion chamber of a GTI using computer modelling in Solid Works and ANSYS Fluent programs, as well as conducting experimental research using a high precision gas analyzer TESTO 350.

Taking into account the aforementioned factors, the development of GTI appears to be an important direction for ensuring energy stability and economic growth. This approach is actively supported by scientific and engineering research aimed at improving design, materials, management, and operation technologies of GTI. In this study, the authors focused on analyzing a new MFC of a GTI to identify its potential for efficient energy production in sharply continental climates.

Expected results

We anticipate that the research findings will allow us to evaluate the effectiveness of the new micro-flame burner device, determine its aerodynamic characteristics, temperature distribution, and combustion product concentrations in the combustion chamber of the GTI. These data will be valuable for further optimization and development of GTI, taking into account efficiency and environmental safety requirements.

Article structure

In the following sections of this article, we will present more detailed research results, including a description of the MFC geometry, results of computer modelling, findings from experimental research, data analysis, and conclusions.

Investigated burner device

The micro-flame combustion technology involves organizing fuel combustion by creating a whole system of numerous small flames in the combustion chamber and is a type of zone combustion. The two-tier micro-flame burner device [4] for burning gaseous fuel consists of a distribution cone, at the outlet of which stabilizers are installed in two tiers: six stabilizers in the inner tier and twelve in the outer tier. Fuel is supplied through two circular copper tubes along the outer and inner radii, with holes leading to the inner part of the stabilizer.

The burner device operates air, supplied by a fan, enters through the stabilizing tube into the expanding part of the burner - the cone. At the same time, gaseous fuel (in this case, propane) from the gas cylinder enters the fuel tube into the circular fuel collectors of the outer and inner radii, depending on the mode. From the circular fuel tubes, fuel is supplied through nozzle holes to each of the stabilizers. Then, the air-flow, upon entering the expanding cone, reduces its velocity and mixes in the throat of the burner with the fuel coming through the tubes. At the burner outlet, the fuel-air mixture burns due to mixing with additional external air.

The angular stabilizers at the burner outlet provide micro-flame combustion [5-7], which results in low emission combustion due to the absence of localized high temperature zones. The angular stabilizers create numerous flame backflows that ignite fresh portions of the fuel-air mixture, thereby increasing combustion stability.

Thus, the invention can provide low emission and stable combustion of gaseous fuel in gas turbine combustion chambers.



The design of the investigated device is illustrated in figs. 1 and 2.

Figure 1. The 3-D model of a two-tier MFC for natural gas combustion with dimensions

Figure 2. Front view of the burner with two tiers of stabilizers

Numerical modelling in the SolidWorks program

The SolidWorks Flow Simulation is software specifically designed for modelling and analyzing hydrodynamic, thermal, and other physical processes related to the flow of liquids and gases in engineering systems and devices.

The results of the studies [8, 9] demonstrate high efficiency when using this program.

The model is based on the relationship between the velocity and pressure values of the air and the initial parameters of the air entering from the stabilizing tube into the cone-shaped burner device, as shown in fig. 3.

The purpose of the modelling was to develop a computer model and numerically simulate the aerodynamic air-flow in the investigated burner device to determine the zones of re-circulation and turbulence in the air-flow.

For the investigation, the k- ε turbulence model was chosen, and an adaptive mesh was constructed, as shown in fig. 4 (the total number of elements exceeding 50000).

The initial air parameters selected were velocities: 5 m/s, 15 m/s, and 30 m/s, pressure: 101325 Pa, ambient temperature 300 K. The section of the pipe-line located at the outlet of the burner was created to visually represent the air-flow behind the stabilizers.



Figure 3. The 3-D model of the MFC in the SolidWorks flow simulation program

Figure 4. Adaptive computational mesh of the modeled domain in SolidWorks

Velocity contours

Table 1 presents velocity contour plots at different initial air velocities: 5 m/s, 15 m/s, and 30 m/s.

Velocity contours allow us to see how the velocity is distributed inside the simulated system.

Based on the images, it can be noted that the air-flow entering the simulated area stabilizes in the pipe and begins to expand and slow down at the entrance to the cone. Upon reaching the corner stabilizers at the exit of the cone, the air-flow encounters an obstacle, resulting in the formation of a zone of reverse flows behind the stabilizers. This process helps to maintain the flame and prevent its rapid entrainment.

The low velocity re-circulation zone (denoted by blue color) serves to detain the fuel-air mixture until complete combustion of the fuel and contributes to high flame stabilization and reduced fuel incompleteness.

It can be seen that the re-circulation zone is observed even at low initial air velocities of 5 m/s, while a more extensive re-circulation zone is noticeable at velocities of 15 m/s and 30 m/s.



Table 1. Velocity contours

Pressure contours

Table 2 shows the pressure contour plots for different initial air velocities: 5 m/s, 15 m/s, and 30 m/s.

Pressure contours illustrate how pressure changes within the burner device model.

From the plots, it can be observed that pressures are evenly distributed, with no areas of critically low or high pressure.

Moderate pressure in the fuel injection zone indicates favorable conditions for thorough mixing of the fuel-air mixture. This suggests that there is sufficient time in the system for thorough fuel-air mixing. Such effective mixing creates favorable conditions for efficient fuel combustion and ensures optimal conditions for the combustion process.



Conclusion on solid works modelling

Based on the analysis of velocity and pressure contours in the combustion chamber at various initial air velocities (5 m/s, 15 m/s, and 30 m/s), the following conclusions can be drawn:

- Flow stabilization: The air-flow entering the combustion chamber stabilizes in the tube and slows down upon entering the cone. This indicates the presence of a certain flow control system, which is important for ensuring uniform and stable fuel combustion.
- Formation of re-circulation zones: Corner stabilizers at the exit of the cone create zones of
 reverse flow in the air-flow. This process contributes to retaining the flame in the combustion zone and prevents its rapid entrainment.
- Uniform pressure distribution: Pressure contours show a uniform distribution of pressures inside the combustion chamber. The absence of areas with critically low or high pressure indicates the stability of the system.
- *Effective fuel-air mixing*: Moderate pressure in the fuel injection zone indicates favorable conditions for complete mixing of the fuel-air mixture. This creates optimal conditions for efficient fuel combustion and ensures the stability of the combustion process.

Thus, the observed flow and pressure characteristics in the combustion chamber indicate the stability and efficiency of the system, which is an important factor in ensuring efficient fuel combustion.

Numerical modelling in ANSYS FLUENT

The ANSYS FLUENT is a leading software for numerical modelling and analysis of fluid-flow, heat transfer, and other physical phenomena in engineering systems. The program offers a wide range of capabilities, including turbulence modelling, chemical reactions, multi-component flows, and thermal phenomena. The ANSYS FLUENT provides powerful tools for building complex models as well as for visualizing and analyzing results, as demonstrated in [10-14].



Figure 5. The 3-D model of the combustor in ANSYS FLUENT

Mesh independence test

Considering the characteristics of the MFC (3-D model in fig. 5) and the defined tasks, the turbulent combustion model with chemical reactions k- ε releasable was selected, with the combustion model being Non-premixed combustion.

The goal of using visualizationols in ANSYS FLUENT was to display the results regarding temperature distribution. The section of the pipe-line located at the outlet of the burner was created for a clear representation of temperature contours behind the stabilizers.

The mesh independence test is a process used to find the optimal grid condition that has the smallest number of grids without generating a difference in the numerical results based on the evaluation of various grid conditions [15].



In fig. 6, Points 1 and 2 are shown, where temperature values were measured.

Figure 6. Mesh independence test points

When changing the mesh size, both the total number of mesh elements and the average temperature values at each point changed. These values are presented in tab. 3.

	Total mesh elements	Size [m]	<i>T</i> ₁ [K]	<i>T</i> ₂ [K]	d <i>T</i> [K]
1	1121658	0.035	1848.17	1556.92	1702.55
2	1184842	0.030	1979.13	1524.08	1751.61
3	1277731	0.025	2128.36	1625.99	1877.18
4	1328770	0.023	2096.40	1555.37	1825.89
5	1390000	0.021	1779.12	1574.05	1676.59
6	1470900	0.019	1694.77	1564.27	1629.52
7	1584224	0.017	1740.11	1600.26	1670.19

Table 3. The values for the mesh independence test

The best way to check the independence of the solution from the mesh is to plot a graph, fig. 7, of the resulting value of the monitor point against the number of elements in the simulation. This is illustrated below, where we have results from our monitor points for the average temperature at the outlet.

We can see that with element sizes ranging from 0.023-0.035, we have a result that shows significant fluctuations in values with an imbalance above 3%. By increasing the mesh resolution and reducing the element sizes to 0.017-0.021, we observe a balanced average temperature within acceptable deviations (+/-3%).

Figure 8 shows the adaptive computational mesh of the modeled domain in ANSYS FLUENT, with an element size of 0.019 m and a total number of elements of 1471476.



Figure 7. Mesh independence test

Encurs 9. A dontive computational mach

Figure 8. Adaptive computational mesh of the modeled domain in ANSYS FLUENT

The input parameters, as specified in tab. 4, were determined.

Table 4. Specified initial parameters

Initial air velocity [ms ⁻¹]	Pressure [Pa]	Gas-flow rate [kgs ⁻¹]	Ambient temperature [K]	Fuel
5, 15, 30	101325	0.006	300	Propane gas

Temperature contours

Three operating modes of the two-tier burner device were investigated, including operation with both tiers and each tier separately. As seen from the presented figures in tab. 5, in the operation mode of the two-tier and internal tier of the burner device, there is a concentration of maximum temperature in the central part of the burner device. This is explained by the centrifugal force acting on the directions of the openings where propane is supplied. The centrifugal force can contribute to more intensive mixing of fuel and air in the central zone, which may lead to temperature rise.

Operation of the external tier: In this operating mode, the external tier of the burner device demonstrated a more uniform temperature distribution. This occurs because the propane, supplied to the periphery of the burner device, is carefully mixed with air and carried into the combustion chamber. In this case, the centrifugal force has less influence on the temperature distribution, promoting uniform fuel combustion.

Conclusions from ANSYS FLUENT modelling

In the operation mode of the two-tiers and internal tier of the burner device, a concentration of maximum temperature is observed in the central part of the burner due to the centrifugal force acting on the directions of the openings from which propane enters. This contributes to more intense mixing of fuel and air in the central zone, resulting in a temperature rise.



 Table 5. Modelling results: temperature fields

The operation of the external tier of the burner device shows a more uniform temperature distribution. This is due to thorough mixing of propane with air at the periphery of the burner device, promoting uniform fuel combustion and reducing the influence of centrifugal force on temperature distribution.

Experiment

The experimental stand, presented in fig. 9, was created to analyze the temperature and gas characteristics at various combustion modes of the investigated device.

The principle of operation of the experimental stand is air enters through the fan -1, behind which, after the transition piece, the stabilizing tube -2 is located, filled with 30 mm nozzles to equalize velocity fields. Further along the air direction, there is a measuring section of air parameters -3, connected to the diffuser -7. This section also contains a collector of statistical and total pressure, chromel-copel thermocouples, and a flow metering device. Gaseous fuel enters from the gas pipeline -4, through the measuring section -5, via



Figure 9. Experimental stand: 1 - fan, 2 – stabilizing tube, 3 – air inlet measuring section, 4 – gas pipe-line, 5 – fuel inlet measuring section, 6 – fuel supply pipe, 7 – diffuser of the front device with a burner, 8 – multi-channel meter, 9 – measuring section behind the diffuser, and 10 - gas analyzer

the fuel-supplying nozzle -6, and directly on the tiers of the front device -7. Then, a measuring section is located behind the diffuser -9, with a multi-channel meter -8, and a gas analyzer probe -10.

The working experimental set-up and the front view of the burner device are presented in figs. 10 and 11.



Figure 10. Experimental set-up



Figure 11. Front view of the burner device

Results and discussions

Figures 12(a)-12(c) depict the dependence of fuel combustion efficiency on the initial air velocity at a constant gas pressure of 0.015 MPa and varying initial air velocity ranging from 5-50 m/s in all presented modes.

The experimental data were obtained through direct measurements and observations in real operating conditions of the installation. The theoretical data were derived from the numerical model in ANSYS Fluent, equations, and calculations, based on the physical principles of fuel combustion in the GTI.

The fuel combustion efficiency coefficient was calculated according to the eq. (1) provided in [16]:

$$\eta_{g} = \frac{\left(1 + \alpha_{\Sigma}L_{0}\right)\left(c_{p,g}T_{g}^{*} - c_{p,g}T_{0}^{*}\right) - \alpha_{\Sigma}L_{0}\left(c_{p,a}T_{a}^{*} - c_{p,a}T_{0}^{*}\right) - \left(c_{p,f}T_{f}^{*} - c_{p,f}T_{0}^{*}\right)}{Q_{H}^{p}}$$
(1)

where T_g^* [K] is the temperature of gases at the outlet of the combustion chamber, T_0^* [K] – the standard temperature for determining the heat of combustion of fuel (calorimetry temperature), T_a^* [K] – the air temperature at the entrance to the combustion chamber, T_f^* [K] – the fuel temperature at the nozzle inlet, Q_H^p [kJkg⁻¹] – the lowest heat of combustion of the working fuel for propane, $c_{p,a}$ [kJkg⁻¹K⁻¹] – the average mass heat capacity of air at a temperature that is a multiplier of the named parameter, $c_{p,g}$ [kJkg⁻¹K⁻¹] – the average mass heat capacity of a gas at a temperature that is a multiplier of the named parameter, and $c_{p,f}$ [kJkg⁻¹K⁻¹] – the average mass heat capacity of fuel (propane) at a temperature that is a multiplier of the named parameter.

The dependency of the combustion efficiency coefficient on the air velocity allows for an assessment of fuel combustion efficiency depending on the operating conditions of the system.

Analyzing the graph in fig. 12(a), it can be concluded that during the simultaneous operation of two tiers, stable combustion is observed at air velocities ranging from 20-35 m/s, which is confirmed by both experimental and theoretical data ($\eta = 0.70$ -0.99). The wide operating range positively affects the maneuverability of the installation.

During the operation of the external and internal tiers, figs. 12(b) and 12(c), it is observed that, unlike when both tiers operate simultaneously, the air velocity values in these modes are limited to 25-30 m/s due to the peculiarities of the modes, namely, the supply of fuel through the fuel pipe located along the internal or external tiers. High combustion efficiency in these modes is observed at velocities of 10-15 m/s. At velocities exceeding 20 m/s, a lean mixture breakdown was observed during the experiment (more air than fuel).



Figure 12. Dependence of the combustion completeness coefficient on the initial air velocity; (a) when two tiers are operational, (b) when the external tier is operational, and (c) when the internal tier is operational

Figures 13(a)-13(c) depict the dependence of NO_x emissions on the temperature of exhaust gases, with constant gas pressure of 0.015 MPa and varying initial air velocity ranging from 5-30 m/s in all presented modes. During the experimental investigation, the TESTO 350-XL gas analyzer [17] was employed to measure the concentration of NO_x emissions in the exhaust gases.

A positive correlation between the temperature of the exhaust gases and the concentration of NO_x emissions is evident in figs. 13(a)-13(c). With an increase in the temperature of the exhaust gases, there is a rise in NO_x emissions, indicating more intense formation of nitrogen oxides as a result of fuel combustion. However, it is noteworthy that the observed NO_x emission

values are limited and do not exceed 15 ppm. This suggests that even at elevated temperatures of the exhaust gases, the level of NO_x emissions remains within regulatory requirements and permissible limits, which can be a significant factor in planning and optimizing fuel combustion processes to minimize environmental impact.



Figure 13. Dependency of NO_x emissions on the temperature of exhaust gases; (a) when two tiers are operational, (b) when the external tier is operational, and (c) when the internal tier is operational

Figures 14(a)-14(c) depict the dependence of carbon monoxide emissions on the air excess ratio at a constant gas pressure of 0.015 MPa and varying initial air velocity ranging from 5-30 m/s in all presented modes. During the experimental investigation, the TESTO 350-XL gas analyzer [17] was employed to measure the concentration of carbon monoxide emissions in the exhaust gases.



Figure 14. Dependency of CO emissions on the air excess ratio; (a) when two tiers are operational, (b) when the external tier is operational, and (c) when the internal tier is operational

The main reason for the formation of CO is incomplete combustion. During the operation of a GTI across a wide range of power levels, the problem of CO formation worsens. When transitioning to low power modes, the temperature in the combustion zone drops, leading to an increase in carbon monoxide emissions.

The experiment was conducted in an open-type combustion chamber. For this type of chamber, it is characteristic to have a shorter residence time for the fuel-air mixture in the combustion zone. Consequently, CO does not have enough time to fully oxidize into CO_2 .

Figures 14(a)-14(c) show a trend of decreasing carbon monoxide emissions with an increase in the air excess ratio, which occurs due to the decrease in temperature and incomplete combustion of the fuel.

The maximum value of carbon monoxide emissions in three modes under the specified initial parameters reached 140 ppm.

Conclusions

The investigation of the two-tier MFC of a gas turbine included modelling in Solid-Works, modelling in ANSYS FLUENT, and physical experiments. The results of studying velocities, pressures, and temperatures in the combustion chamber, as well as the analysis of NO_x emissions, the conclusions are as follows.

• *Flow stability and uniform combustion*: Modelling in SolidWorks demonstrated air-flow stabilization and uniform pressure distribution, contributing to efficient fuel combustion.

- *Influence of air velocity*: Optimal initial air velocities (20-35 m/s for two tiers, 10-15 m/s for external and internal tiers) for different operating modes were determined, highlighting the importance of regulating air velocity for efficient fuel combustion.
- *Effective fuel-air mixture mixing*: ANSYS FLUENT research showed that the operation of two tiers of the burner promotes intense mixing of fuel and air, improving the combustion process.
- *The NO_x emissions*: Analysis of NO_x emissions showed that even at elevated temperatures of exhaust gases, the level of emissions remains within regulatory limits, up to 15 ppm, which is important for environmental safety.
- *The CO emissions*: The maximum value of carbon monoxide emissions in three modes under the specified initial parameters reached 140 ppm.

Thus, the research results confirm the effectiveness and stability of the proposed burner device, which is a significant contribution the development of gas turbine technologies in line with efficiency and environmental safety requirements.

Acknowledgment

This research was supported by grant AR 14872041 on the topic *Development and* study of new combustion chamber front devices for improving the environmental safety and efficiency of gas turbine installations in Kazakhstan, from the State Institution Committee of Science of the Ministry of Science and Higher Education of the Republic of Kazakhstan.

Future developments

Within the scope of the grant, further research is planned to explore the potential of new combustion chamber front devices to enhance the longevity, efficiency, and environmental safety of gas turbine installations, as well as to adapt these technologies for use in other types of gas turbines.

References

- [1] Bealessio, B. A., *et al.*, A Review of Enhanced Oil Recovery (EOR) Methods Applied in Kazakhstan, *Petroleum*, 7 (2020), 3, pp. 1-28
- [2] Shamoushaki, M., Aliehyaei, M., Exergy, Economic, and Environmental (3E) Analysis of a Gas Turbine Power Plant and Optimization by Mopso Algorithm, *Thermal Science*, 22 (2018), 6A, pp. 2641-2651
- [3] Li, S., *et al.*, Research on the Performance Improvement of a Two-Shaft Gas Turbine with a Variable Area Nozzle Power Turbine, *Thermal Science*, 24 (2020), 6A, pp. 3721-3733
- [4] Makzumova, A., et al., Patent KZ No. 36479 Dvukhyarusnoye mikrofakelnoye ustroystvo, 2023 (in Russian), Makzumova, A., et al., Patent KZ No. 36479 Two-Tier Micro-Flame Burner Device, 2023
- [5] Chen, K., et al., Investigation on Micro-Flame Structure of Ammonium, Thermal Science, 27 (2023), 5B, pp. 4117-4134
- [6] Sosso Mayi, O., et al., Numerical Simulation of Premixed Methane/Air Micro Flame: Effects of Simplified one Step Chemical Kinetic Mechanisms on the Flame Stability, *Applied Thermal Engineering*, 73 (2014), 1, pp. 567-576
- [7] Jiang, F., et al., Flame Characteristics Influenced by the Angle of Burners for Non-Premixed C₃H₈-Air, *Thermal Science*, 26 (2022), 6B, pp. 5147-5156
- [8] Olaru, I., A CFD analysis in Solidworks Flow Simulation for Two Mixing Fluids with Different Temperatures in Nozzles, *Journal of Engineering Studies and Research*, 26 (2020), 1, pp. 41-46
- [9] Khairulmaini, M., et al., The Influence of Diameter and Winding on Heat Transfer Efficiency in Spiral Tube Heat Exchangers: A CAD-Integrated CFD Study Using Solidworks Flow Simulation, Journal of Physics: Conference Series 7th International Conference on Mechanical Engineering Research, 2688 (2023), 012002
- [10] Jovčevski, M., et al., Mathematical and CFD Methods for Prediction of Thermal Pollution Caused by Thermal Power Plant, *Thermal Science*, 27 (2023), 6A, pp. 4485-4496

Dostiyarov, A. M., *et al.*: Numerical Model and Experimental Research ... THERMAL SCIENCE: Year 2024, Vol. 28, No. 4A, pp. 3107-3119

- [11] Dostiyarov, A., et al., Numerical Investigation of Combustion Process Behind Bluff Bodies during Separation, Bulgarian Chemical Communications, 52 (2020), Spec., pp. 12-19
- [12] Fuzhang, W., et al., Numerical Analysis for the Effects of Heat Transfer in Modified Square Duct with Heated Obstacle Inside It, International Communications in Heat and Mass Transfer, 129 (2021), 105666
- [13] You, Y., et al., A CFD Model of Frost Formation Based on Dynamic Meshes Technique Via Secondary Development of ANSYS Fluent, International Journal of Heat and Fluid-Flow, 89 (2021), 108807
- [14] Dostiyarov, A, et al., Numerical Modelling Biogas Combustion in the Novel Burner, Proceedings, 8th International Conference on Energy Efficiency and Agricultural Engineering (EE&AE), Ruse, Bulgaria, 2022, pp. 1-4
- [15] Minhyung, L., et al., Improvement of Grid Independence Test for Computational Fluid Dynamics Model of Building Based on Grid Resolution in: Advances in Civil Engineering, (ed. Vipulanandan, C.), University of Houston, Houston, Tex., USA, 2020, pp. 1-11
- [16] Pchelkin, P., et al., Kharakteristiki raboty kamery sgoraniya GTD, Vyssheye tekhnicheskoye uchilishche imeni N. E. Baymana, Moskva, SSSR, 1978 (Characteristics of the Combustion Chambers of the Gas Turbine Engine, Higher Technical School named after N. E. Bayman - in Russian), Moscow, USSR, 1978
- [17] ***, TESTO, TESTO 350-XL Instruction manual, Testo AG Germany

Paper submitted: March 18, 2024 Paper revised: April 28, 2024 Paper accepted: May 3, 2024 © 2024 Society of Thermal Engineers of Serbia Published by the Vinča Institute of Nuclear Sciences, Belgrade, Serbia. This is an open access article distributed under the CC BY-NC-ND 4.0 terms and conditions