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

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

[c] L.N.Gumilyov Eurasian National University, Astana, Kazakhstan

[*] Corresponding author; E-mail: a.makzumova@gmail.com

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

1. 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 (GTIs) are attracting increasing attention as an efficient and environmentally friendly source of energy.

Worldwide, the development of GTIs 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 gas turbine installations.

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.

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. Renewable energy sources, 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], GTIs, 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.

1.1. 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 modeling 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 factors mentioned above, the development of GTIs 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 GTIs. 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.

1.2. 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 gas turbine installations, taking into account efficiency and environmental safety requirements.

1.3. 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 modeling, findings from experimental research, data analysis, and conclusions.

2. 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 as follows: 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 Figure 1 and Figure 2.

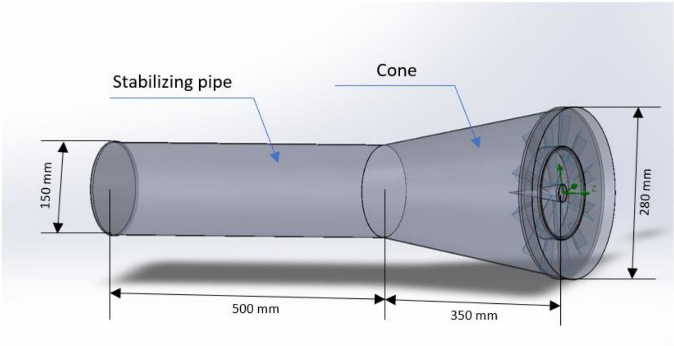


Figure 1. 3D model of a two-tier MFC for natural gas combustion with dimensions

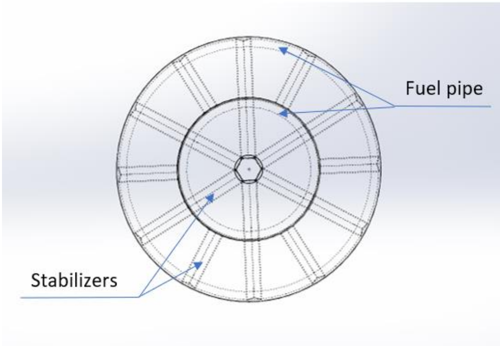


Figure 2. Front view of the burner with two tiers of stabilizers

3. Numerical modeling in the SolidWorks program

SolidWorks Flow Simulation is software specifically designed for modeling 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 Figure 3.

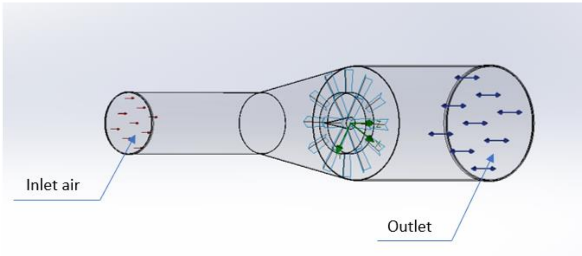


Figure 3. 3D model of the MFC in the SolidWorks Flow Simulation program

The purpose of the modeling was to develop a computer model and numerically simulate the aerodynamic airflow in the investigated burner device to determine the zones of recirculation and turbulence in the airflow.

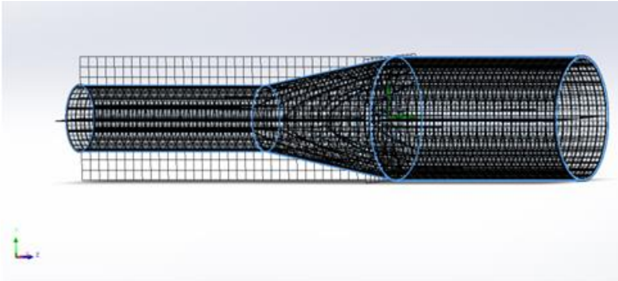


Figure 4. Adaptive computational mesh of the modeled domain in SolidWorks

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, 15, 30 m/s, pressure: 101325 Pa, ambient temperature 300 K. The section of the pipeline located at the outlet of the burner was created to visually represent the airflow behind the stabilizers.

3.1. Velocity contours

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

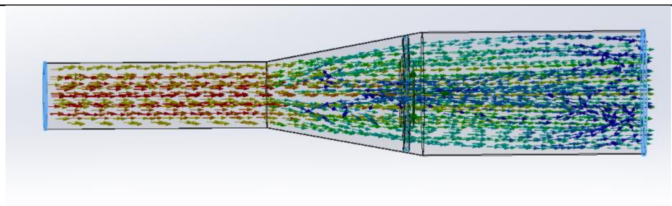
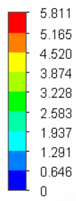
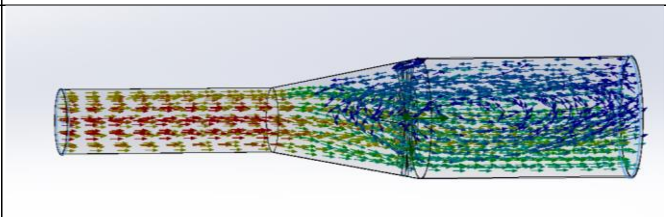
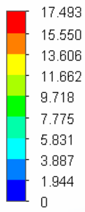
Velocity contours allow us to see how the velocity is distributed inside the simulated system.

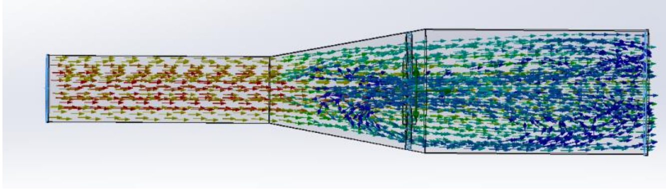
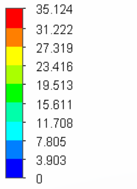
Based on the images, it can be noted that the airflow entering the simulated area stabilizes in the pipe and begins to expand and slow down at the entrance to the cone. Upon reaching the corner stabilizers at the exit of the cone, the airflow encounters an obstacle, resulting in the formation of a zone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

The low-velocity recirculation 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 recirculation zone is observed even at low initial air velocities of 5 m/s, while a more extensive recirculation zone is noticeable at velocities of 15 and 30 m/s.

Table 1. Velocity contours

Initial parameters	Velocity contours	Velocity, m/s
5 m/s		
15 m/s		

<table>
<tr><td>30 m/s</td><td></td><td>
</td></tr>
</table>

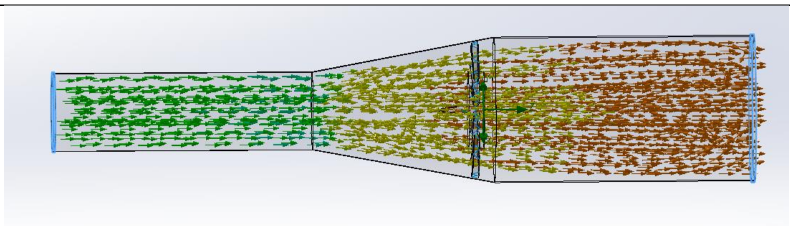
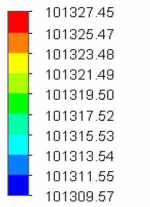
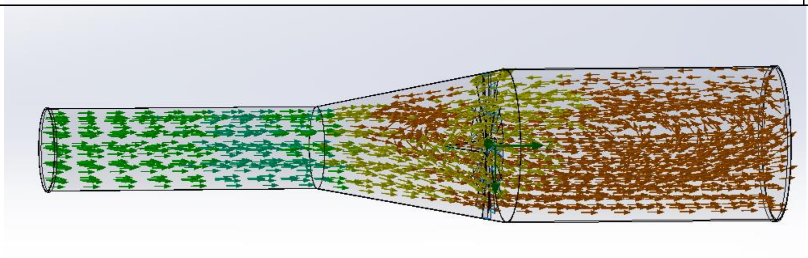
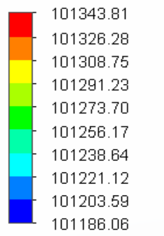
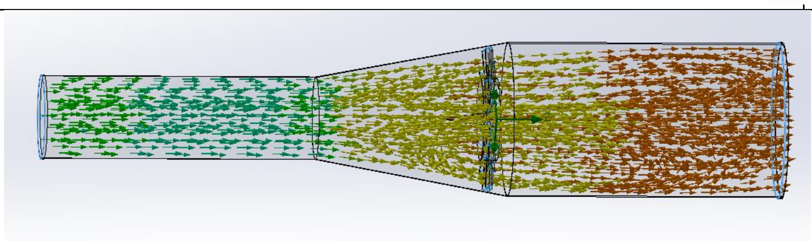
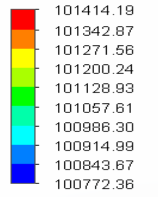
3.2. Pressure contours

Table 2 shows the pressure contour plots for different initial air velocities: 5, 15, 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.

Table 2. Pressure Contours

<table>
<tr><th>Initial parameters</th><th>Pressure Contours</th><th>Pressure, Pa</th></tr>
<tr><td>5 m/s</td><td></td><td>
</td></tr>
<tr><td>15 m/s</td><td></td><td>
</td></tr>
<tr><td>30 m/s</td><td></td><td>
</td></tr>
</table>

3.3. Conclusion on Solid Works Modeling

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

1. Flow Stabilization: The airflow 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.

2. Formation of Recirculation Zones: Corner stabilizers at the exit of the cone create zones of reverse flow in the airflow. This process contributes to retaining the flame in the combustion zone and prevents its rapid entrainment.

3. 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.

4. 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.

4. Numerical Modeling in Ansys Fluent

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

Considering the characteristics of the MFC (3D model in Figure 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.

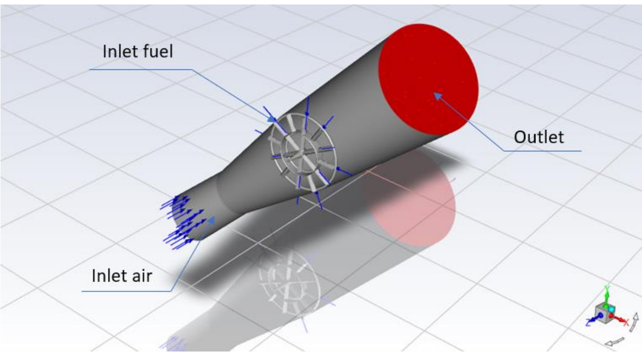


Figure 5. 3D Model of the Combustor in Ansys Fluent

The goal of using visualization tools in Ansys Fluent was to display the results regarding temperature distribution. The section of the pipeline located at the outlet of the burner was created for a clear representation of temperature contours behind the stabilizers.

4.1. Mesh independence test

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 Figure 6, points 1 and 2 are shown, where temperature values were measured.

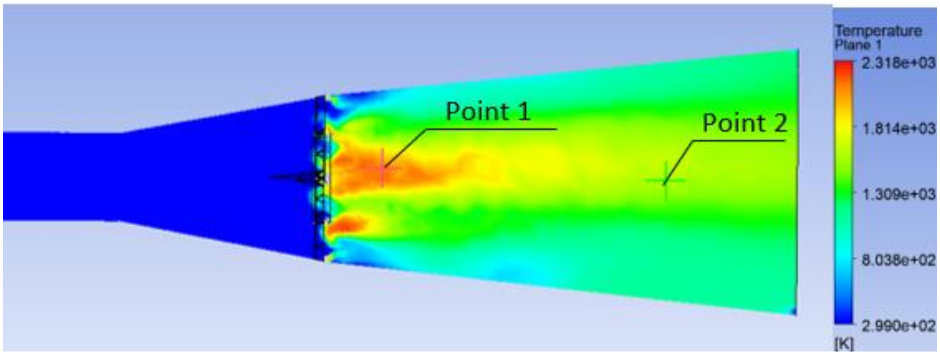


Figure 6. Mesh independence test points

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 Table 3.

Table 3. The values for the mesh independence test

	Total Mesh Elements	Size, m	T1, K	T2, K	dT, 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

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 to 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 to 0.021, we observe a balanced average temperature within acceptable deviations (+/-3 percent).

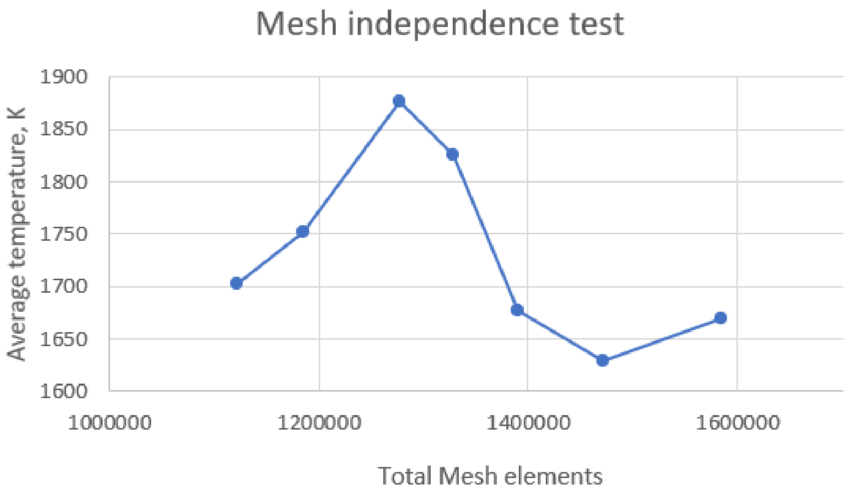


Figure 7. Mesh independence test

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.

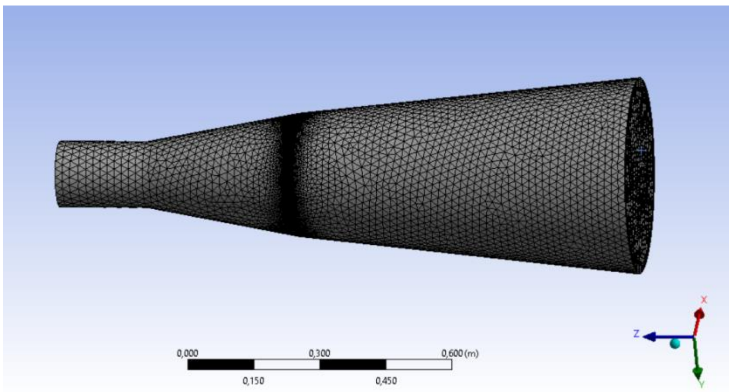


Figure 8. Adaptive computational mesh of the modeled domain in Ansys Fluent

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

Table 4. Specified Initial Parameters

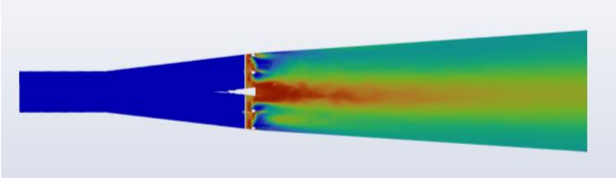
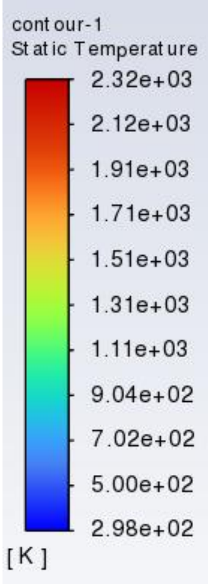
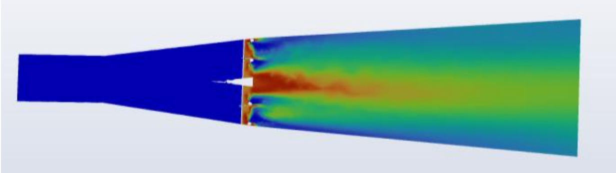
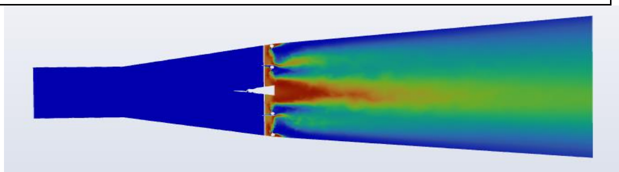
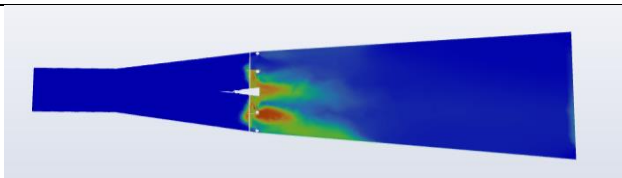
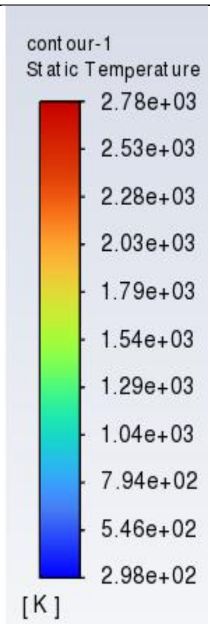
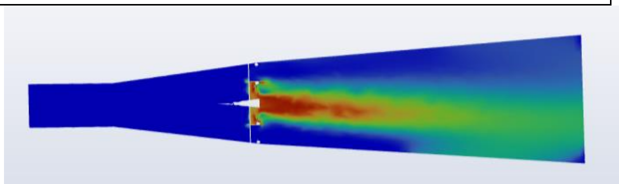
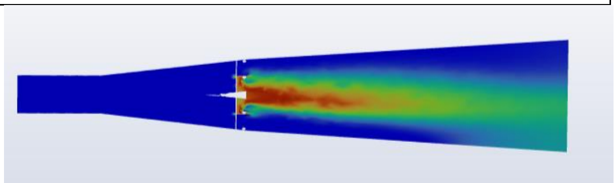
Initial Air Velocity, m/s	Pressure, Pa	Gas Flow Rate, kg/s	Ambient Temperature, K	Fuel
5, 15, 30	101325	0.006	300	Propane gas

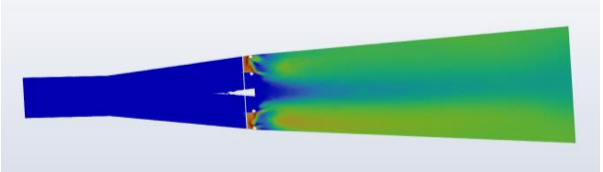
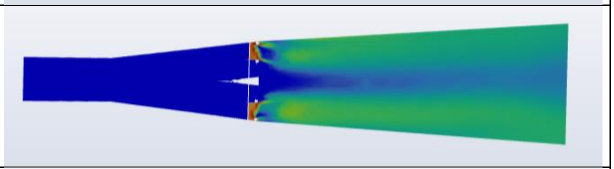
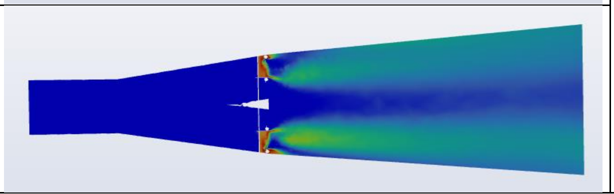
4.2. Temperature Contours

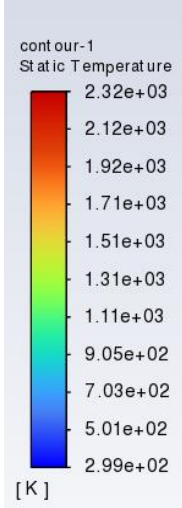
Three operating modes of the two-tier burner device were investigated, including operation with both tiers and each tier separately. As seen from the presented figures in Table 5, in the operation mode of the two-tier and internal tier of the burner device, there is a concentration of maximum temperature 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.

Table 5. Modeling Results: Temperature Fields

Two tiers		
5 m/s		
15 m/s		
30 m/s		
The internal tier		
5 m/s		
15 m/s		
30 m/s		
The external tier		

5 m/s	
15 m/s	
30 m/s	



4.3. Conclusions from Ansys Fluent Modeling

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.

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.

5. Experiment

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

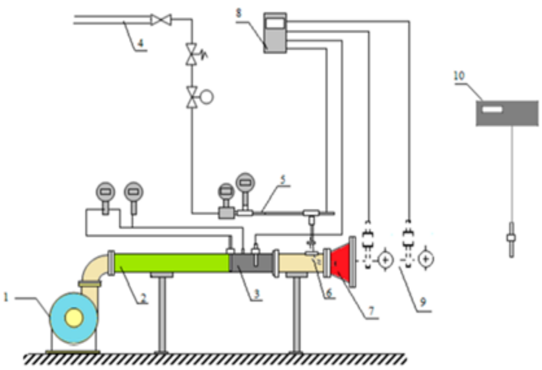


Figure 9. Experimental Stand: 1-fan; 2 – stabilizing tube; 3 – air inlet measuring section; 4 – gas pipeline; 5 – fuel inlet measuring section; 6 – fuel supply pipe; 7 – diffuser of the front device with a burner; 8 – multi-channel meter; 9 – measuring section behind the diffuser; 10 – gas analyzer.

The principle of operation of the experimental stand is as follows: 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 the fuel-supplying nozzle 6 directly onto 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 setup and the front view of the burner device are presented in Figures 10 and 11.

Figure 10. Experimental setup

Figure 11. Front view of the burner device

5.1. Results and Discussions

Figures 12a, 12b, 12c 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 to 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 formula (1) provided in [16]:

$$\eta_g = \frac{(1+\alpha_\Sigma L_0)\cdot(c_{\mathrm{pg}}\mathrm{T}_g^* - c_{\mathrm{pg}}\mathrm{T}_0^*) - \alpha_\Sigma L_0(c_{\mathrm{pa}}\mathrm{T}_a^* - c_{\mathrm{pa}}\mathrm{T}_0^*) - (c_{\mathrm{pf}}\mathrm{T}_f^* - c_{\mathrm{p}f}\mathrm{T}_0^*)}{Q_{\mathrm{н}}^{\mathrm{p}}} \quad (1)$$

where T_g^* - the temperature of gases at the outlet of the combustion chamber, K;

T_0^* – standard temperature for determining the heat of combustion of fuel (calorimetry temperature), K;

T_a^* - the air temperature at the entrance to the combustion chamber, K;

T_f^* - fuel temperature at the nozzle inlet, K;

$Q_{\mathrm{н}}^{\mathrm{p}}$ – the lowest heat of combustion of the working fuel for propane, $\mathrm{kJkg^{-1}}$;

$c_{\mathrm{p}a}$, – the average mass heat capacity of air at a temperature that is a multiplier of the named parameter, $\mathrm{kJkg^{-1}K^{-1}}$;

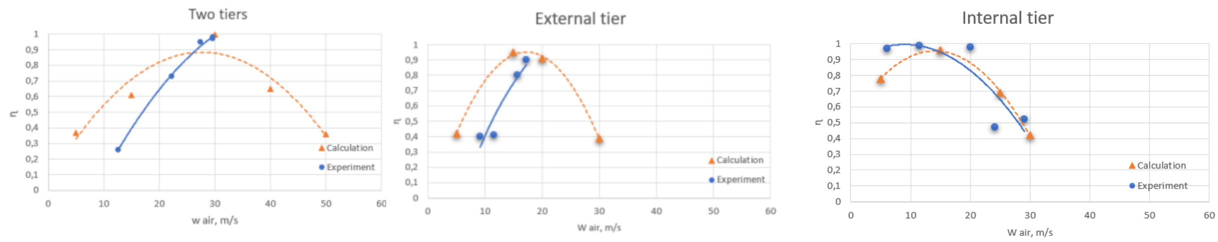
$c_{\mathrm{p}g}$ – the average mass heat capacity of a gas at a temperature that is a multiplier of the named parameter, $\mathrm{kJkg^{-1}K^{-1}}$;

$c_{\mathrm{p}f}$ – the average mass heat capacity of fuel (propane) at a temperature that is a multiplier of the named parameter, $\mathrm{kJkg^{-1}K^{-1}}$.

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

During the operation of the external and internal tiers (Fig. 12b, 12c), 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).

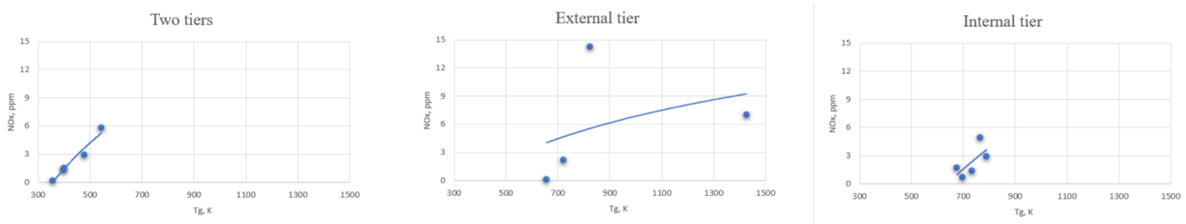


a b c

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; c) when the internal tier is operational

Figures 13a, 13b, 13c depict the dependence of NOx emissions on the temperature of exhaust gases, with constant gas pressure of 0.015 MPa and varying initial air velocity ranging from 5 to 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 NOx emissions in the exhaust gases.

A positive correlation between the temperature of the exhaust gases and the concentration of NOx emissions is evident in Figures 13a, 13b, 13c. With an increase in the temperature of the exhaust gases, there is a rise in NOx emissions, indicating more intense formation of nitrogen oxides as a result of fuel combustion. However, it is noteworthy that the observed NOx emission values are limited and do not exceed 15 ppm. This suggests that even at elevated temperatures of the exhaust gases, the level of NOx 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.



a b c

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

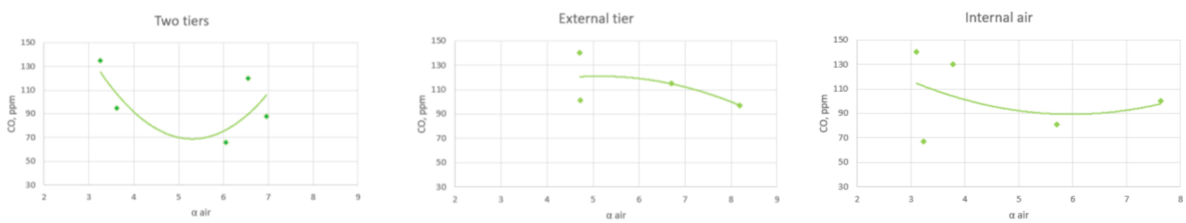
Figures 14a, 14b, 14c depict the dependence of carbon monoxide emissions on the air excess ratio at a constant gas pressure of 0.015 MPa and varying initial air velocity ranging from 5 to 30 m/s in all presented modes. During the experimental investigation, the TESTO 350-XL gas analyzer [17] was employed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

The main reason for the formation of carbon monoxide (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 CO2.

Figures 14a, 14b, 14c 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.



a b c

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

6. Conclusion

The investigation of the two-tier MFC of a gas turbine included modeling in SolidWorks, modeling in Ansys Fluent, and physical experiments. The results of studying velocities, pressures, and temperatures in the combustion chamber, as well as the analysis of NOx emissions, allowed for several conclusions to be drawn:

1. Flow stability and uniform combustion: Modeling in SolidWorks demonstrated airflow stabilization and uniform pressure distribution, contributing to efficient fuel combustion.

2. 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.

3. 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.

4. NOx emissions: Analysis of NOx 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.

5. 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 to 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(3)* (2020), pp.1-28

[2] Shamoushaki, M., Ehyaei, M., Exergy, economic, and environmental (3E) analysis of a gas turbine power plant and optimization by Mopso Algorithm, *Thermal Science, 22 (6A)* (2018), 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(6A) (2020), pp. 3721-3733

[4] Makzumova, A., *et al., Patent KZ №36479 Dvukhyarusnoye mikrofakelnoye ustroystvo, 2023* (in Russian language). (Makzumova, A., *et al., Patent KZ №36479 Two-tier Micro-flame Burner Device, 2023)*

[5] Chen, K., *et al*., Investigation on Micro-Flame Structure of Ammonium, *Thermal Science, 27(5B)* (2023), 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 (1)* (2014), pp. 567-576,

[7] Jiang, F., *et al.*, Flame characteristics influenced by the angle of burners for non-premixed C3H8-air, *Thermal Science, 26(6B)* (2022), 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(1)* (2020), 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*, (2023), pp. 1-8

[10] Jovčevski, M., *et al.*, Mathematical and CFD methods for prediction of thermal pollution caused by thermal power plant, *Thermal Science*, (2023), pp. 4485-4496

[11] Dostiyarov, A., *et al*., Numerical investigation of combustion process behind bluff bodies during separation, *Bulgarian Chemical Communications, 52*, (2020), 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), pp. 1-18

[14] Dostiyarov A, *et al*., Numerical modelling biogas combustion in the novel burner *8th International Conference on Energy Efficiency and Agricultural Engineering (EE&AE*), (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 *Advances in Civil Engineering* (2020) pp. 1-11

[16] Pchelkin, Yu.M, Lebedev, V.P., *Kharakteristiki raboty kamery sgoraniya GTD*, Vyssheye tekhnicheskoye uchilishche imeni N.E.Baymana, Moskva, SSSR, 1978 (in Russian language). (Pchelkin, Yu.M., Lebedev, V.P., *Characteristics of the combustion chambers of the gas turbine engine*, Higher Technical School named after N.E. Bayman, Moscow, USSR, 1978)

[17] TESTO, TESTO 350-XL Instruction manual, Testo AG Germany

Submitted: 18.3.2024.
Revised: 28.4.2024.
Accepted: 03.5.2024.