IOP Conference Series: Earth and Environmental Science

PAPER • **OPEN ACCESS**

Study of the aerodynamics of air flow in burners using the Comsol Multiphysics software

To cite this article: A K Makzumova *et al* 2023 *IOP Conf. Ser.: Earth Environ. Sci.* **1234** 012014

Study of the aerodynamics of air flow in burners using the Comsol Multiphysics software

A K Makzumova[1*], N N Sarakeshova[2], A M Dostiyarov[2] and I K Iliev[3]

[1] L.N.Gumilyov Eurasian National University, Astana, 2 Satpayev street, 010008, Kazakhstan
[2] S.Seifullin Kazakh Agrotechnical Research University, Astana, 62 Jenis avenue, 010000, Kazakhstan
[3] Angel Kanchev University of Ruse, Ruse, 8 Studentska street, POV 7017, Bulgaria

***Email: a.makzumova@gmail.com**

Abstract. The article presents the processes of modeling the aerodynamic flow of air in two burner microflame devices: 1. The burner device of the combustion chamber of a gas turbine plant; 2. The micromodular gas burner. The aerodynamic flow is modeled in the Comsol Multiphysics software package, which is reasonable in terms of availability and quality. The performed analysis of microflame fuel-burning devices revealed that the dynamic and geometric characteristics affect the aerodynamics and velocity profiles. Different airflow rates significantly affected the intensity of the formation of recirculation zones, which also led to an increase in pressure in the circuit. Accordingly, the flow at different heights of the cylindrical nozzle showed that the dimensions will affect the combustion processes and, as a result, the stabilization and formation of nitrogen oxides.

1. Introduction

Numerical modeling is the process of creating computer models of real systems or phenomena that can be used to study and predict their behavior. It is a key technology in science, engineering and other fields where it is necessary to study complex systems that cannot be analytically solved.
For the purpose of improving burner devices applicable in the energy sector, the Comsol Multiphysics numerical simulation program was used in this article [1].
Burners are used for the production of electricity, heat and steam, as well as for the production of various types of fuel, such as gas or liquid fuel. In fuel-burning plants, fuel is burned to produce thermal energy, which is then used to drive turbines, generate electricity, or to produce heat and steam. Burner devices can use various types of fuel, including natural gas, coal, oil and other sources, depending on the availability and cost of fuel in the region. However, the use of fuel can be associated with a negative impact on the environment, since various harmful substances and emissions are released during combustion, such as carbon dioxide, sulfur and nitrogen oxides, as well as solid particles. Methods of reducing harmful emissions are discussed in the article [2,3].
In general, burners continue to play an important role in supplying energy to many regions of the world, but at the same time efforts are required to reduce their negative impact on the environment. In this context, it is important to develop new technologies and solutions for more efficient use of fuel and reduction of emissions of harmful substances.
The aerodynamics of the air flow in fuel-burning installations (FBI) can have a significant impact on harmful emissions, especially on emissions of nitrogen oxides (NOx), which are among the main harmful emissions emitted by these installations.
In FBI, air is fed into the inlet compression chamber, where it is compressed and heated, then mixed with fuel and fed into the combustion chamber, where the mixture burns and releases thermal energy.

During the combustion process, nitrogen oxides are released, which are formed from nitrogen in the air at high temperatures and pressures [4].
The aerodynamics of the air flow affects the combustion processes, thereby affecting the amount and composition of emissions emitted. For example, poor mixing of air and fuel can lead to incomplete combustion and the formation of large amounts of NOx. In addition, mismatch of air flow parameters, such as speed and temperature, can lead to the formation of zones of high temperature and increased NOx concentration.
To reduce harmful emissions it is important to ensure proper mixing of air and fuel, optimize air flow parameters and control the temperature in the combustion chamber. Various methods can be used for this, including modeling of air flow, optimization of the installation design and the use of special control and regulation systems.
Microflame combustion technology (MFCT) [5-7] is an innovative technology that is used to reduce emissions of nitrogen oxides (NOx) in fuel-burning plants.
It consists in using special burners that create microflame - small flames on the surface of the burner, which ensures uniform combustion of fuel. Thus, MFC allows to achieve high combustion efficiency at low temperatures, which reduces the formation of NOx.
The use of MFCT in gas turbine installations allows to reduce nitrogen oxide emissions by up to 25-30%, compared with traditional combustion methods. In addition, it allows you to achieve a higher degree of fuel combustion, which increases the efficiency of installations and reduces operating costs.
MFCT also has other advantages such as lower noise and vibration levels, a wider range of operating conditions and a smaller burner size. In general, MFCT is one of the promising areas for the development of gas turbine installations, which can significantly reduce harmful emissions and improve their economic performance [8,9].

2. Method

Comsol Multiphysics [1] is a software for numerical simulation of physical processes that allows engineers and scientists to analyze and optimize various engineering systems and processes using an integrated approach that combines several physical disciplines.
The software is commonly used to solve various engineering problems, such as electronics design, structural strength and stability analysis, optimization of heating and cooling systems, analysis of the flow of liquids and gases, as well as for modeling other physical processes in various industries, including the automotive industry, energy, medicine and science.
This article discusses the aerodynamics of air flow in 3D models of fuel-burning devices used in the energy industry, namely: 1. The burner device of a gas turbine installation [10]; 2. The micromodule air nozzle used in hot water boilers [11]. The initial air parameters are indicated in Table 1.

Table 1. Initial parameters

№	Initial speed	Pressure
1	5-15 m/s	101 325 Pa
2	3-10 m/s	

The purpose of the simulation is to develop physical models and numerical simulation of the aerodynamic flow of air (gaseous fuel) in the above combustion devices to determine the effectiveness of the design by studying aerodynamic processes, the choice of initial air velocities for efficient combustion of fuel assemblies.

2.1 Description of the burner device the combustion chamber of a gas turbine plant

The existing simple flare burner devices of the combustion chamber of the GTP have a large emission of nitrogen oxides. In this regard, a patent was developed [10], the design of which is illustrated in Figure 1 and Figure 2.

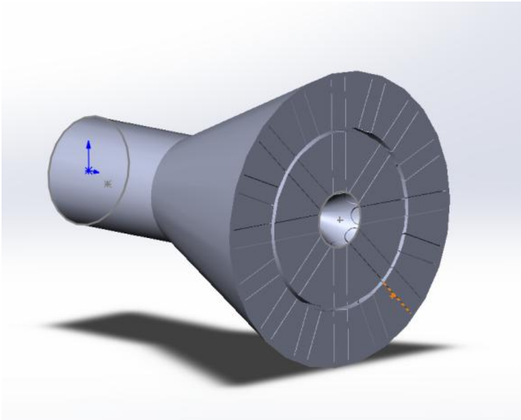


Figure 1. Three-dimensional model of a two-tier microflame burner for natural gas combustion

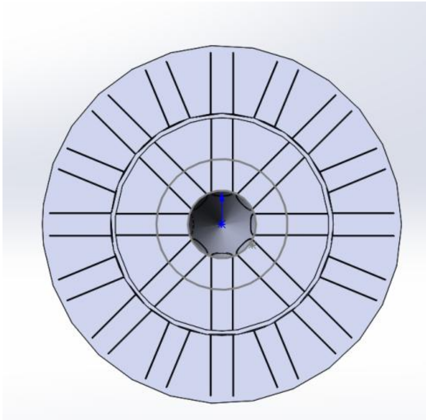


Figure 2. The front of the burner with two tiers of stabilizers

A two–tier microflame burner for burning natural gas has a distribution cone at the outlet along the burner axis, connected to a housing with stabilizers installed in two tiers: eight stabilizers in the inner tier, sixteen in the outer tier. Fuel is supplied through an annular copper tube with twenty-four holes, which is located along the axis between the outer and inner tiers of the stabilizers. 16 holes are drilled up and 8 holes down so that the injection of gaseous fuel falls into the concave (inner) part of the stabilizers.

The burner works as follows: air for the preparation of a depleted fuel-air mixture enters the swirling flow into the expanding part of the burner through the inlet swirler. Further, the hydraulic resistance of the outlet section of the burner is reduced in the distribution cone, and as a result, the flame slip is eliminated. Fuel is supplied from the fuel tube to the annular fuel tube from which fuel is supplied to each of the stabilizers by branches of copper tubes. At the outlet of the burner, the fuel assembly burns due to mixing with additional air from the outside.

Corner stabilizers at the burner outlet provide microflame combustion, which gives low-emission combustion due to the absence of local high-temperature zones. Corner stabilizers create a lot of reverse flame flows that ignite fresh portions of fuel assemblies. Thus, they increase the stability of combustion.

The modeling process is represented by constructing a model in the 3-dimensional space of a realizable k-ε turbulence model. Figure 3 shows the modeling area.

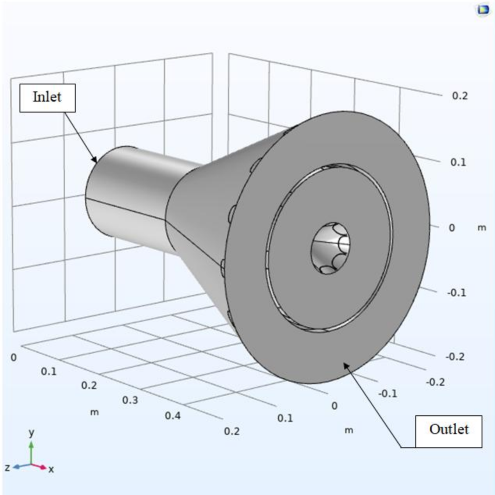


Figure 3. 3D modeling area

Combustion is based on the dependence of the values of the velocity and pressure of the air in the combustion zone on the initial parameter of the air coming from the lower part of the combustion zone and coming out of the upper part.

Figures 4 and 5 illustrate the tetrahedral adaptive computational grid of the simulated area. The number of elements in the area in Fig. 4 is 8838. The grid consists of a group of tetrahedral, which makes it possible to efficiently calculate various variations of the stress field and obtain a highly accurate result.

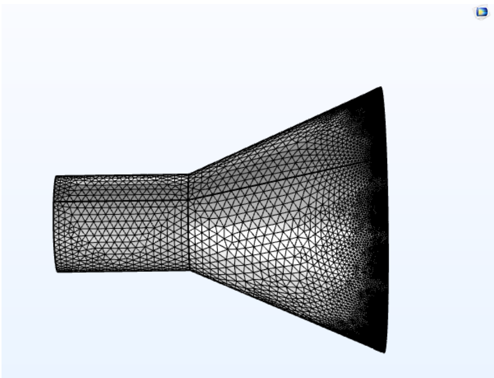


Figure 4. Adaptive computational grid of the simulated area in COMSOL Multiphysics

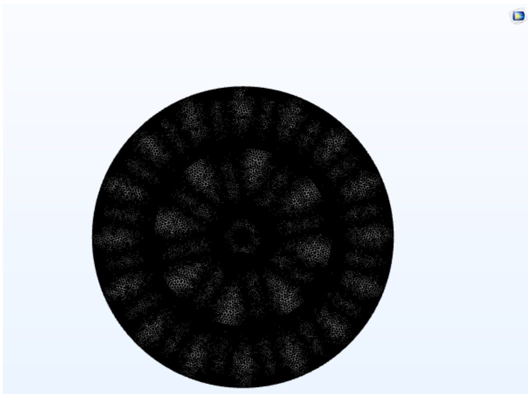


Figure 5. Adaptive calculation grid of the front side of the burner

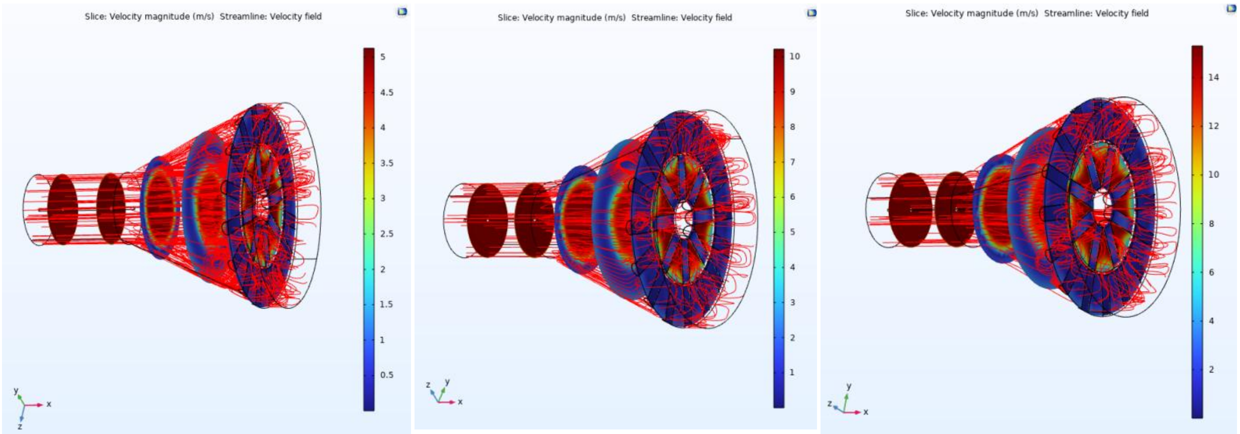
2.1.1 Results

High-speed contours and recirculation zones. Figures 6a, 6b, 6b show velocity contours and recirculation zones of reverse currents at different initial air velocities (5, 10, 15 m/s).

As can be seen from the figures, the air flow, entering the modeling area, meets an obstacle in the form of a corner stabilizer, rounding it, loses speed, a zone of reverse currents forms behind the stabilizer, thereby preventing rapid flame entrainment.

It can be seen that at low initial air velocities, namely 5 m/s, an uneven recirculation zone with low velocity values is formed. A more stable recirculation zone is observed at initial velocity values equal to 10, 15 m/s.

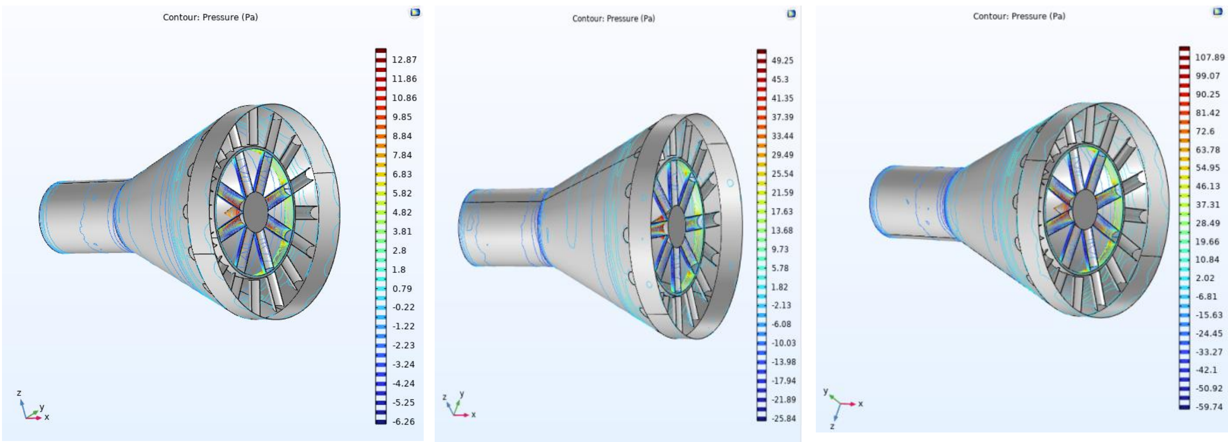
High air velocities are observed at the end of the expanding part of the burner device, the recirculation zone with low speeds serves to detain fuel assemblies until complete combustion of fuel and leads to high flame stabilization and reduction of under burning of fuel.



a) 5 m/s b) 10 m/s c) 15 m/s

Figure 6. Velocity contours and recirculation zones of reverse currents

Pressure contours. Figure 7 shows the pressure contours at different initial air velocities (5, 10, 15 m/s).
As can be seen from the figures, the corner stabilizers take the greatest pressure from the air, this is confirmed by the zone with high pressures indicated in red in the figure.
It can be noticed that at the end of the expanding part of the stabilizer wall, randomly located pressure isobars are observed, this is explained by the turbulent air flow.
The air, encountering an obstacle in the form of a stabilizer, loses speed and reduces pressure, thereby slowing down the flow and increasing the time for more complete mixing of fuel assemblies, resulting in efficient combustion.



a) 5 m/s b) 10 m/s c) 15 m/s

Figure 7. Pressure contours

2.2 Description of the micromodule gas burner

Numerical simulation was performed on a micromodule gas burner [11], with a Venturi pipe and a cylindrical nozzle. Since the nozzle is of large diameter, a sudden expansion is formed at the junction

of the nozzle with the Venturi pipe, which is a good stabilizer. In addition, secondary air enters through the slots to the nozzle, which ensures high combustion, combined fuel-air mixture with low NOx output. The result is achieved by burning natural gas in a micromodule gas burner consisting of a Venturi tube, a fuel tube and a sprayer, characterized in that a large diameter cylindrical nozzle with outlet slots is installed to the diffuser part.
A 3D-simulation of this burner is shown in Figure 8.

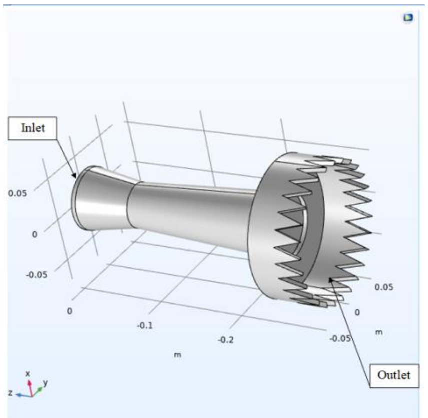


Figure 8. 3D model of the burner

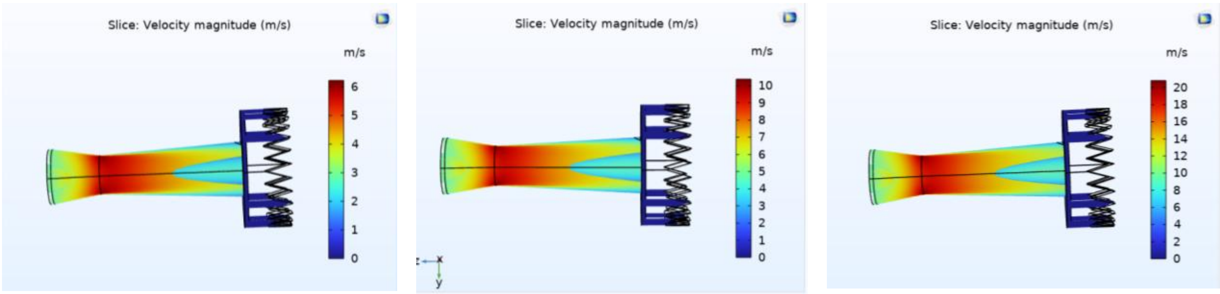
The modeling object consists of a Venturi pipe and a cylindrical nozzle with a larger diameter, input and output areas.
In view of the fact that the article considered only the effect of fuel supply, the value of fuel consumption was not taken into account when modeling.
When solving flow turbulence, different turbulence models are used. In this case, the 3D model k-ε realizable (realizable) was applied.
Figure 9 shows a tetrahedral adaptive computational grid of the simulated area. The grid consists of a group of tetrahedra, with more than 5000 elements in it, which makes it possible to efficiently calculate various variations of the stress field and obtain a highly accurate result.

Figure 9. Adaptive computational grid of the simulated area in COMSOL Multiphysics

2.2.1 Results

Speed contours. Figures 10a, 10b, 10b show velocity contours at different initial air velocities at the gas burner inlet. It can be seen from the figures that acceleration in the narrowing channel of the Venturi tube is characteristic for all the initial velocities of the air entering the burner, after which,

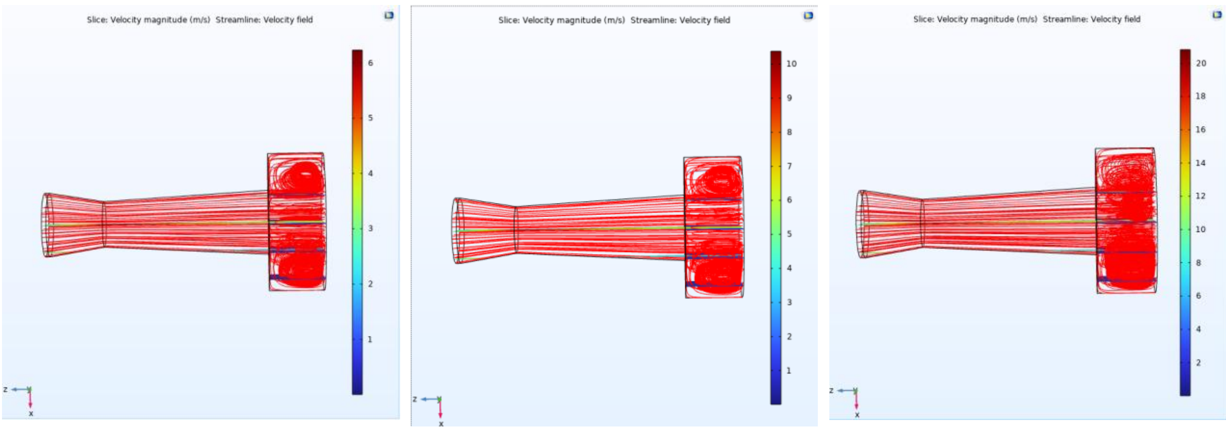
falling into the expanding part of the tube, the flow slows down, which gives time for effective mixing of gaseous fuel with air.
As a result of efficient mixing of the air-fuel mixture and with an increase in the mixing rate in the backflow zone, the effect on the combustion process is very effective, as shown in the figures.



a) 3 m/s b) 5 m/s c) 10 m/s

Figure 10. Speed contours

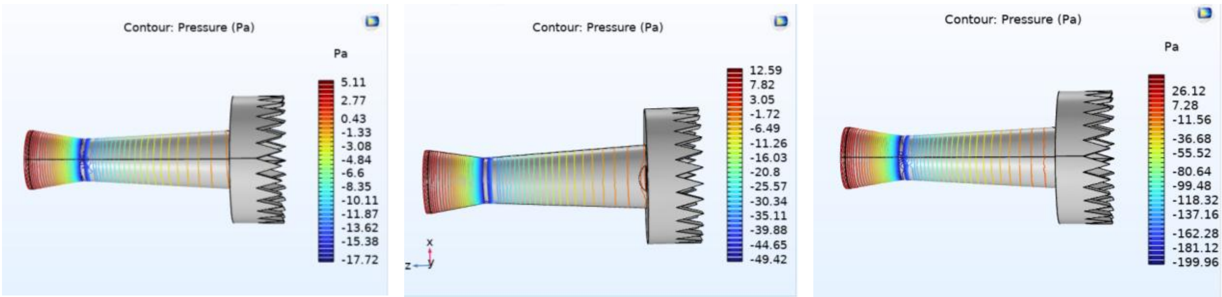
Recirculation zone. Figures 11a, 11b, 11b show recirculation zones at different speeds. Vortices are visible in the large diameter nozzle, which clearly demonstrates the formation of an extensive recirculation zone of the fuel-air mixture. At the junction of the nozzle with the Venturi pipe, a sudden expansion is formed, which leads to a high stabilization of the flame and a decrease in under burning of fuel.
That is, with an increase in the input speeds of the air flow in a wide part of the burner, we can notice an increase in the frequency of recirculation zones, in which vortex air flows are created, which are delayed for active mixing and complete combustion of gaseous fuel.



a) 3 m/s b) 5 m/s c) 10 m/s

Figure 11. Contours of recirculation zones

Pressure contours. Figures 12a, 12b, and 12b show the pressure contours of a micromodule gas burner at different speeds. As can be seen from the figures, zones with negative pressures are formed in the cylindrical nozzle at minimum speeds, and at high speeds, the formation of zones with high pressures at the outlet of the cylindrical nozzle is observed, which characterizes the stabilization in the recirculation zones.
To stabilize the flame in the recirculation zone, it is very important to take into account the pressure in the cylindrical nozzle, since different pressures are formed at different speeds, as shown in the figures.



a) 3 m/s b) 5 m/s c) 10 m/s

Figure 12. Pressure contours

3. Conclusions

Optimal air velocities in fuel-burning plants for efficient mixing of air with fuel depend on several factors, such as the size of the chamber, the type of burner, air and fuel pressure, as well as the operating conditions of gas turbine units and hot water boilers, which is clearly shown in the figures presented in the article.
The vortex flows created behind the stabilizers in the air flow ensure better mixing of air with fuel in view of the active recirculation zone of reverse currents.
For the burners under consideration, the most effective initial velocities are 5 and 10 m/s for hot water boilers and much higher for gas turbine plants, respectively, this is due to the fact that rich combustion is possible at the outlet of the burner at low initial velocities due to insufficient air supply, and at high speeds, there is a risk of entrainment of the air-fuel mixture.
A preliminary study of the aerodynamics of burner devices will make it possible to correctly choose the ratio of geometric dimensions and the contours of the parameters of velocities and pressure in the sections.

References

[1] User manual Introduction to COMSOL Multiphysics www.comsol.ru pp 1-260

[2] Papapavlou S and Demis S 2020 The effect of fuel-bound nitrogen on NOx emissions during the combustion of biofuels *Applied Thermal Engineering* pp 115-194

[3] Zhang H Yao Z Zheng Q and Wang Z 2018 A review of low-temperature combustion and its effects on NOx emissions in compression-ignition engines *Energy 165 Part B* pp 1156-1170

[4] Lefevre A H 1986 Processes in the combustion chambers of the gas turbine engine *Mir* (Moscow) p 566

[5] Karim H and Dube P 2011 Micro-combustion in gas turbine systems *International Journal of Energy Research 35(4)* pp 267-286

[6] Dostiyarov A M, Umyshev D R, Katranova G S and Yamanbekova A K 2017 Combustion chambers and burners of gas turbine installations *Monograph* (Kazakhstan: Astana S. Seifullin KATU Press) p 205

[7] Gulde O L 2000 Fundamental aspects of micro-combustion *Proceedings of the Combustion Institute 28(1)* pp 2461-2474

[8] Ballal D R and Lefebvre A H 1979 Weak Extinction Limits of Turbulent Flowing Mixtures *Journal of Engineering for Power* (Vol. 101 №3) pp 343–348

[9] Ongar B Iliev I K Smagulova G K Nurova A I 2020 Numerical simulation of the formation of nitrogen oxides in pulverized furnaces *Journal of Engineering Science and Technology Review* 2 pp 171-175

[10] Makzumova, A K, Dostiyarov A M, Anuarbekov M A, Vernickas, P A, Sarakeshova N N and Biakhmetov B A 2022 Two-tier microflame burner device *Application for patent of the Republic of Kazakhstan (*No. 169582) (Kazakhstan: Astana)

[11] Dostiyarov A M, Sarakeshova N N 2022 Micromodule gas burner *Application for patent of the Republic of Kazakhstan* (No. 2022/0015.1) (Kazakhstan: Astana)