DOI: https://doi.org/10.15407/pmach2024.03.016

UDC 621.165.62-192

CFD SIMULATION OF THE INFLUENCE OF THE TYPE OF GAS DISTRIBUTION IN THE BURNERS ON THERMAL AERODYNAMIC PROCESSES IN THE DKVR 10-13 BOILER

[1] **Olha Yu. Chernousenko**
chernousenko20a@gmail.com
ORCID: 0000-0002-1427-8068

[1] **Artur Yu. Rachynskyi**
arturrachinskiy@gmail.com
ORCID: 0000-0001-6622-1517

[1, 2] **Oleksandr V. Baraniuk**
olexandr.baranyuk@gmail.com
ORCID: 0000-0003-0610-1403

[1] **Oleksandr A. Siryi**
Seruy_Alex@i.ua
ORCID: 0000-0001-5811-9037

[1] National Technical University of Ukraine "Igor Sikorsky Kyiv Polytechnic Institute"
37, Beresteiskyi ave., Kyiv, 03056, Ukraine

[2] Thermal Energy Technology Institute of NAS of Ukraine
19, Andriivska str., Kyiv, 04070, Ukraine

Topics related to fuel combustion and its impact on the environment will never lose their relevance, as the issues of efficient combustion and emission reduction are key in power generation and environmental protection. The countries of the European Union are massively abandoning the use of natural gas as a fuel for thermal power stations. However, in Asian countries, the ease of using natural gas as the main fuel and its environmental friendliness compared to coal made it possible to widely use natural gas in industry and energy sector. Comparing natural gas with alternative combustible gases (generator, blast furnace, mine, biogas), the main conclusion that it has the most attractive characteristics for its use in industry, including energy facilities, can be drawn. Therefore, it is impossible to replace it with alternative fuels in the chemical, heavy industry and energy sector in the near future. The paper is devoted to CFD modeling of stabilized combustion without premixing in a burner with low swirl for two operating modes of the boiler unit - nominal one and at 60% capacity. The study was carried out using numerical methods with the ANSYS-Fluent application program package. The object of the study is a burner built according to the technology based on the use of jet-niche systems with gas distribution of fuel by circular jets fed perpendicularly into the flow of the oxidizer through a single-row system of holes. Hydrodynamics and heat exchange processes were chosen as the subject of research, based on the analysis of which a model of NO_x generation in jet-niche systems was obtained. The authors of the paper believe that replacing the regular burners of the DKVR-10-13 water heating boiler with a jet-niche ones can contribute to better mixing of fuel and air and ensure more complete combustion. In this paper, two types of burners are considered. In one of the burners, fuel is supplied through rectangular slits, in the other – through round holes arranged in a row. Air is supplied to both burners through rectangular slits. It was determined that gas distribution through round holes increases the spraying of the mixture and increases the area of combustion products spraying. Visualization of the distribution of pressure, temperature, kinetic energy profiles of turbulent pulsations and vorticity was carried out. The obtained results indicate that there are no changes in the flow regime, flame displacement or its instability. It was determined that both the axial velocity and the tangential velocity of the flow affect the distribution of combustion products and harmful impurities such as NO_x. Gas distribution in circular jets stabilizes combustion and reduces flame expansion.

Keywords: *gas distribution, jet-niche technology, ANSYS-Fluent, modeling, gaseous fuel, combustion, methane, boiler fuel.*

Introduction

Nowadays, there is a need to optimize the design and operating parameters of burners. For this, it is necessary to study the combustion processes. As is known, combustion between fuel and air involves the conversion of chemical energy into thermal energy. In view of this, in order to obtain maximum thermal efficiency, it is desirable to have complete combustion. Therefore, incomplete combustion, during which toxic unburned hydrocarbons are released into the environment along with CO_2, NO_x, etc. [1], which leads to environmental pollution and a decrease in fuel combustion efficiency, is not suitable to get the desired result.

Natural gas is the primary fuel for industrial gas turbines. Let's add that the composition of natural gas, despite the predominance of methane, is different. In addition, the size of the detailed chemical kinetic model is too large to be used in the CFD-Fluent code. Based on this, the goal of the study [2] is to find a reduction in the number of species and reactions in order to obtain a fast and effective calculation mechanism (Fig. 1) and implant it in Fluent. The study of the methane-air flame reaction was performed on a model without premixing using the k-ε turbulent model and an 8-stage reaction model. The authors confirmed the results of calculations with experimental results.

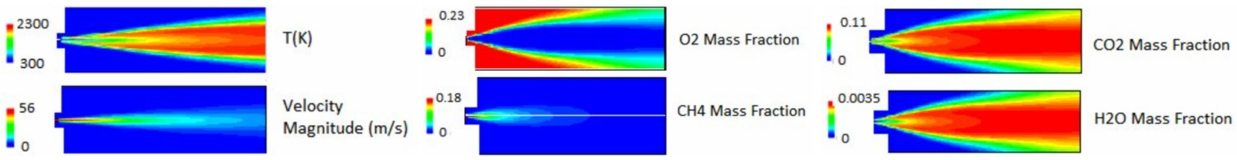


Fig. 1. Results of a free flame simulation without CH_4-air premixing using a one-step scheme [2]

According to the analysis of the available literature, nowadays it is necessary to increase the efficiency of combustion and minimize emissions. Based on this, researchers have developed various designs of burners. For example, the authors of [3] determined the flame temperature and CO_2 temperature under different conditions of the methane-air mixture flow using CFD modeling. To simulate the combustion process, they used a combustion model without premixing. In [4], the physical mechanism of flame ignition and stability limits during combustion without premixing were studied using theoretical and numerical simulations. A one-dimensional flame calculation was used to construct a stability diagram, and three separate stability limits were used to study the flame ignition mechanism. The authors of [5] focused on the combustion behavior of a premixed propane-air mixture using the k-ε model and the model of reduced chemicals. As the results showed, the highest temperature is observed in regions with strong vorticity.

The authors of [6] used CFD analysis to study the combustion of a partially premixed methane-air mixture in a 2D combustion chamber. Their results showed that the static temperature is high in the combustion zone, and it decreases towards the outlet from the burner.

However, as of now, not only well-known experimental and numerical methods, but also the latest technologies in the form of laser diagnostics are used to study combustion processes. Thus, in [7], the characteristics of the reaction zones of the preliminary mixture turbulent combustion were studied. They showed that the thickness of the preheating and reaction zone increases many times compared to the laminar counterpart. The authors concluded that energy vortices can potentially penetrate into the core of the flame and thus expand its thickness.

From the analysis of the available literature, it follows that in order to reduce the emission of harmful substances, in addition to improving the design of the burner, the possibility of replacing methane with other types of fuel should also be taken into account. Thus, the authors of [8] studied the behavior of propane and methane combustion using a CFD-based combustion model without premixing. Based on the obtained results, they proposed propane instead of methane as an effective fuel for the combustion chamber they developed.

The method of CFD modeling of methane-air fuel for steady-state conditions using computational fluid dynamics with the k-ε turbulence model is presented in [9]. Thus, the analysis of the abovementioned papers shows that in the calculation practice, general features of computer models of combustion processes and gas transport have already been formed. Those features should be held to in order to achieve a correct, from a physical point of view, result, namely: the combustion process should be modeled using the species transport model and the k-ε turbulence model with standard or scaled wall functions.

The mentioned technique can be applied in the analysis of the flow and heat exchange in the fuel (gas space) of the boiler unit by changing the design and mode parameters of the burners installed in the boiler unit.

The aim and problems of the research

The aim of the paper was to determine the methods of reducing the emissions of harmful substances from the DKVR-10-13 boiler into environment.

To achieve the set goal, the following problems were solved:

– to develop models of the DKVR-10-13 fuel boiler equipped with burners made using jet-niche technology;

– to study the processes of hydrodynamics and mixture formation in the fuel (gas space) of the boiler, depending on the type of gas supply and regime parameters of the flow in the jet-niche flame stabilizer;

– to provide recommendations regarding the modernization of the installation in the most economical method.

Research objects and features of geometric models

It is known that the main principles of modern fuel combustion technology are the rational initial distribution of fuel in the oxidizer flow, stable and controlled aerodynamic structure of the flow of fuel, oxidizer, and combustion products, as well as self-regulation of the fuel mixture composition in the flame stabilization zone. These requirements are met by the jet-niche technology of natural gas burning [10], which is currently implemented on many types of equipment. Thus, the list of modernized equipment includes: water-heating and steam boilers, in particular power boilers, with a capacity of up to 200 MW, open-hearth furnaces, drying furnaces, etc.

Boilers of the DKVR type are very common in the small energy sector. The author's research [10] shows that almost all modernized boiler units, due to the reduction of the excess air ratio to 1.04–1.07 and a significant reduction in the temperature of the gases flowing in a wide load range, ensured a high brute efficiency (about 95%) compared to the nominal value. On one of the DKVR-6.5 boilers operating in water heating mode, comparative tests of SND-43 burners with a system of two niches (jet and end) and standard HMG-4 burners were conducted. The results of measuring the parameters of boiler operation before and after modernization are shown in Fig. 2, from which it can be seen that the boiler, which was in operation for about 40 years and somewhat reduced its economic characteristics, significantly exceeded the nominal values of efficiency after modernization.

The way to implement effective technology is the location of the jet-niche system on an autonomous pylon-collector. Such a burner module covers all stages of the work process: fuel distribution in the oxidizer flow, mixture formation to the required concentration level, fuel mixture ignition, flame stabilization and formation of concentration, velocity, and temperature fields of combustion products, is actively cooled by oxidizer and fuel flows, does not require additional control automation. Everything mentioned is achieved due to the successful constructive placement of the system on the autonomous collector, from which the burner consists of (Fig. 2, b).

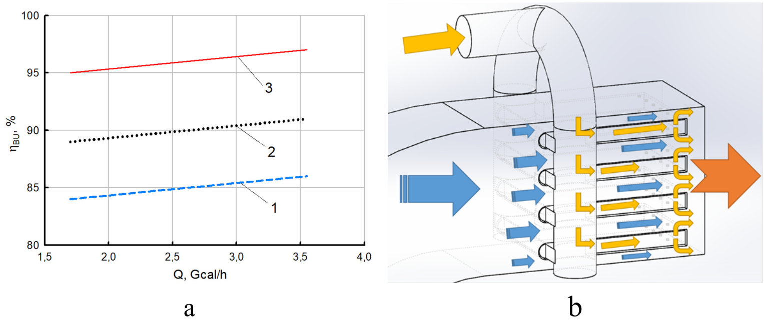


Fig. 2. The experience of modernization of the DKVR type boilers by the SNT company [10] (a) and the appearance of the burner (b):
1 – boiler unit efficiency before modernization (the specific fuel consumption in the maximum mode was 166 kg cond.fuel/Gcal);
2 – boiler unit efficiency is nominal (the specific fuel consumption in the maximum mode was 160 kg cond.fuel/Gcal);
3 – boiler unit efficiency after modernization (the specific fuel consumption in the maximum mode was 147 kg cond.fuel/Gcal)

This design makes it possible to combine all stages of the working process of the burners, such as: distribution of fuel in the flow, formation of a mixture with the required levels of fuel concentrations, ignition of the combustible mixture, stabilization of the torch and formation of combustion products of the required quality and without harmful emissions, thermal preparation of fuel and self-cooling of thermally loaded burner elements.

It is known that the result of modeling depends on many parameters, but one of the most important ones is the density of the calculation grid, the bevel and aspect ratio of the finite elements, as well as their type. In ANSYS, it is possible to build a calculation grid based on finite elements in the form of a tetrahedron, a parallelepiped, and their combination, the so-called hybrid finite-element grid.

Fig. 3 shows a CFD model of the fuel with an installed jet-niche flame stabilizer, which is used as a burner. Unstructured (finite element type – tetrahedron), structured (finite element type – parallelepiped) and hybrid finite element grids were used to approximate the computational domain. The "quality" values of the finite-element grid – parameters of aspect ratio (AspectRatio) of the finite element, orthogonal quality (Orthogonal Quality) and skewness (Orthogonal Skew) – are in the middle of the reference interval issued by the program for verification. The density of the calculation grid is selected as maximum, within the limits allowed by the ANSYS-Student version.

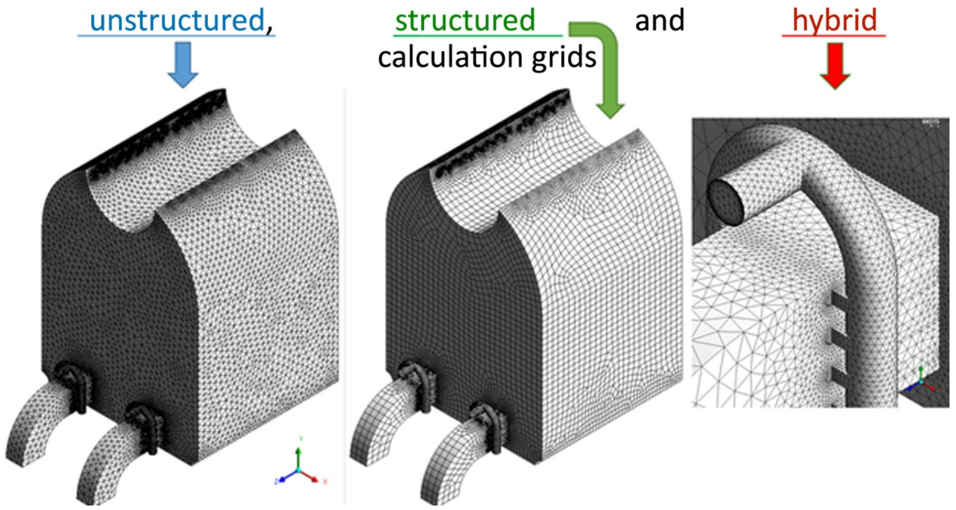


Fig. 3. Finite element grid of the CFD model of a boiler unit with a jet-niche flame stabilizer

As governing equations used in the mathematical model of the process, the continuity equation, the Navier-Stokes equation for the flow of a compressible viscous liquid with variable properties, the energy equation and the equation of conservation of mixture components [11], which are solved by numerical methods in Ansys-Fluent environment, are adopted. The modeling process also used the standard Ansys-Fluent technique for predicting NO_x generation, namely calculating the generation of both thermal and fast NO_x. The standard Ansys-Fluent technique takes into account the turbulent-chemical interaction, which allows the calculation of NO_x formation taking into account the influence of turbulent pulsations on the time-averaged reaction rates. A partial equilibrium model is used to predict the concentration of the O radical, which is necessary for predicting thermal NO_x.

An implicit installation algorithm (Pressure Based Implicit) was chosen as the solution algorithm. The calculation was made in a stationary setting (Steady).

As a calculation one, the authors used the Realizable k-ε turbulence model in the Realizable modification because it is designed for flows containing jets (injection of a fuel-air mixture).

As boundary conditions on impermeable walls, the condition of equality of all components of the velocity vector to zero and uneven temperature distribution on the surface of the walls were set, for the determination of which the temperature pressure between the lower distribution collector and the drum was set, which was selected from the nominal characteristics of the boiler.

At the inlet to the calculation area, mass flow (Massflow inlet) and temperature are set, at the outlet – static pressure (Pressure outlet). The values of the parameters of the reacting flows are shown in Table 1.

Table 1. Numerical values of boundary conditions set in work and control parameters

Parameter	Value	Dimensions	Load, %	
			100	63
Heat load	Q_k	Gkal	9.39	5.86
Gas consumption	G_{gas}	m³/h	1215	760
Gas temperature	t_{gas}	°C	18	18
Behind the economizer	NO_x	ppm	86	77
Excess air coefficient	α	–	1.27	1.33
Air consumption	G_{air}	m³/sec	4.11	2.69
Air temperature	T_{air}	°C	21	21

The result of CFD modeling of the flow and heat exchange in the fuel of the DKVR-10-13 boiler

Heat exchange in the boiler fuel is primarily determined by the flow regime organized by the burners. As expected, the jet flow from the burner is directed to the "outlet" of the fuel, where the pressure is the lowest. A stagnant zone with a circulating current is formed above the jet. The movement inside the zone is supported by moving masses of gas due to the effect of the temperature difference between the red-hot flame torch (approximately 1600 °C) and the surface of the drum (the average temperature of the wall of which is 120 °C).

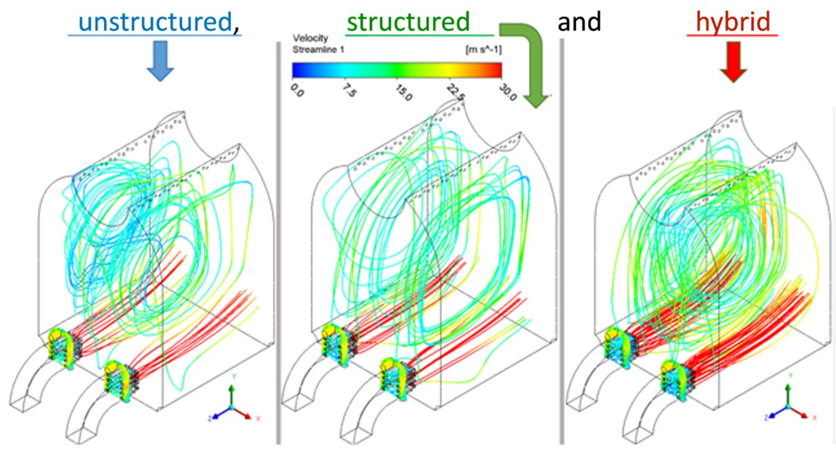


Fig. 4. Trajectories of flow particles in the volume of fuel depending on the type of calculation grid

Fig. 5 shows the temperature distribution of the flow in the cross-section by the plane, which is perpendicular to the spread of the jet of incandescent gas and is located at a distance of 100 mm from the "outlet" of the flow from the fuel. Data analysis of Fig. 5 shows that with the same color filling, the isotherms in the case of using a structured and hybrid grid are smooth, in contrast to the model built using an unstructured grid. Therefore, it is suggested to choose a hybrid grid for further modeling.

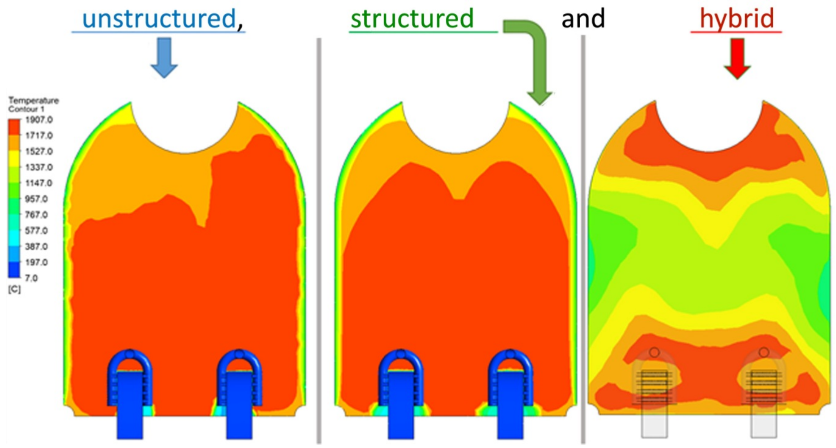
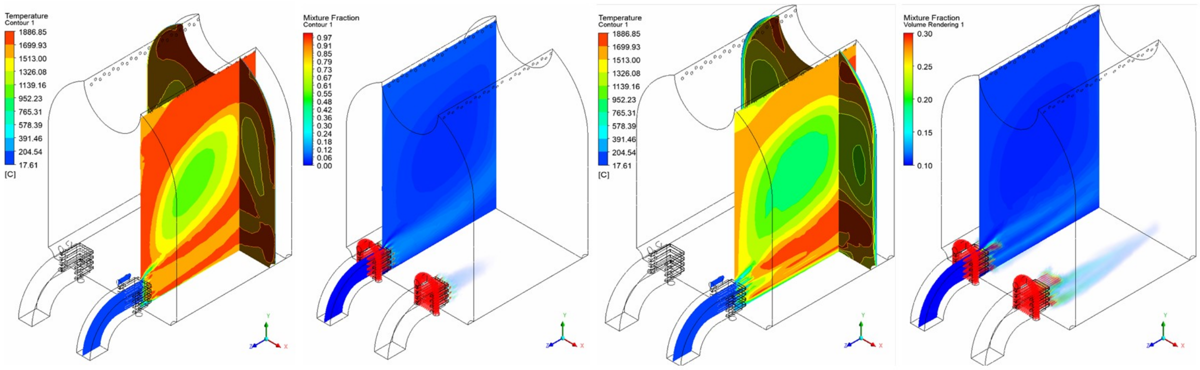


Fig. 5. Flow temperature at the fuel outlet depending on the type of calculation grid

The influence of the type of gas distribution (gas supply either through rectangular slits or through round holes located in a row) is characterized by the temperature distribution in the volume of the fuel (Fig. 6). Analysis of Fig. 6 shows that in the center of the fuel there is a zone of reduced pressure, relative to which a circulation current is observed. On the periphery of this current, zones, the temperature of which corresponds to the temperature of the flame torch, are observed. This phenomenon is explained by the free-convective movement of gas masses due to changes in density. The length of the flame torch can be determined using the Mixture Fraction parameter, which reflects the intensity of the mixing process and most fully characterizes the degree of mixing of fuel and oxidizer (for example, in a reacted flame). A value of zero corresponds to 100% oxidizer, and 1 to 100% fuel. The zone of gas mixed with air (Fig. 6) indirectly depicts the shape of the torch of incandescent gases, since it is in this area that the reaction of the chemical interaction of methane with oxygen actually takes place.

As shown in Fig. 6, a, the length of the flame torch does not exceed 1 m, which corresponds to the nominal characteristics for the regular burner of the DKVR boiler. In the case of gas supply through rectangular slits (Fig. 6, b), the torch "crosses" the entire section of the fuel. In the practice of operating boiler units, they try to avoid such cases, since there is a risk of burning the pipes of the back screen of the boiler.



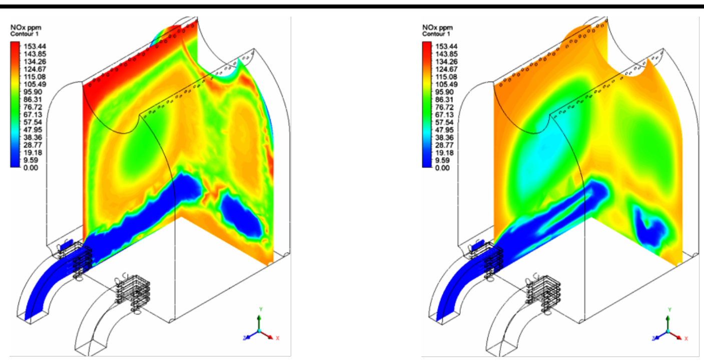
a b

Fig. 6. The temperature of the flow in the fuel volume and calculation of the torch length for the case of gas distribution through round holes located in a row (a) and through rectangular slits (b)

The distribution of nitrogen oxides NO_x in two mutually perpendicular cross-sections - through the middle of the burner and the cross-section of the "outlet" flow from the fuel shown in Fig. 7 reveals that the maximum concentration of pollutants is observed in zones with high temperatures. These zones are formed according to the hydrodynamic structure of the flow in the fuel and the generation of polluting particles in them, which directly depends on the time the air particle stays in the zone of high temperatures.

Analysis of Fig. 7 shows that in the case of gas distribution through a rectangular gap, the overall level of generation of nitrogen oxides NO_x is lower than in the case of gas distribution by jets. According to Figs. 4–6 along the periphery, the fuel flow moves with sufficiently low (4–7 m/s) local velocities. A zone with a high temperature is also observed here (1800 °C for the case of gas distribution by jets and 1500 °C for gas distribution by slits). These are ideal conditions for the generation of nitrogen oxides NO_x, which do not contradict the existing ideas about the generation of nitrogen oxides in boiler fuels and, if cleaning measures are not applied (for example, the oxidation of nitrogen oxides to carbon dioxide), those oxides can enter the atmosphere.

According to the Table 1, the value of nitrogen oxides averaged over the area of the "gas window" is set for the boiler economizer. The authors chose these values for CFD model verification. Thus, by means of CFD modeling at the nominal load at the "outlet" from the fuel in the case of gas distribution with round jets (proper jet-niche burner), the level of NO_x generation was determined to be 94 ppm. According to experimental data, this value is 86 ppm. An error of 8.6% in this case is acceptable and indicates the verification of the calculation model when the values of nitrogen oxides during fuel combustion are numerically studied.



a b

Fig. 7. Distribution of NO_x in the typical cross-sections of the fuel for the case of gas distribution through round holes located in a row (a) and through rectangular slits (b) at the nominal load of the boiler unit

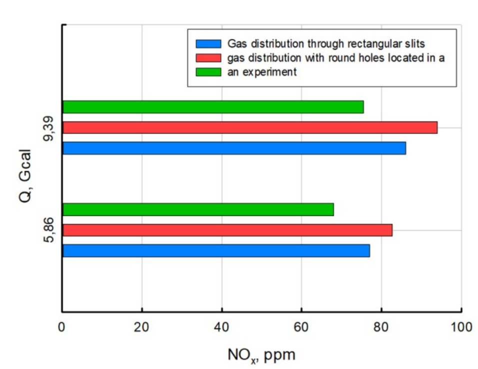


Fig. 8. Plane-averaged NO_x values at the "outlet" from the fuel

Such a comparison is allowed because the nitrogen oxides created in the zone of high temperatures are not destroyed, they enter the convective shaft of the boiler unit from the fuel and, together with the flow of flue gases, circumflex the boiler economizer, where it is possible to conduct experimental measurements.

It should be noted that in the case of gas supply through rectangular slits, the level of NO_x generation is 79.5 ppm, which is associated with a lower level of fuel temperatures.

The calculation of NO_x generation at 60% heat load was also carried out and its results are shown in Fig. 8.

Analysis of the data shown in Fig. 8 reveals that gas supply to rectangular slits reduces NO_x generation by 12%, compared to the experiment on a jet-niche burner. This is due to the fact that the average fuel temperature is much lower. The fact that the temperature field of the drum installed in the upper part of the fuel is uneven, which will negatively affect its strength, is also of high concern.

Conclusions

1. Verification using NO_x values averaged over the area of the "gas window" shows that the deviation of calculated data from experimental data does not exceed 8.6%.

2. It was determined that a lower level (≈12%) of NO_x generation is observed in the fuel of the DKVR-10-13 boiler if the gas distribution in the jet-niche burner is organized through rectangular slits, and not through round holes located in a row.

3. It was established that the temperature in the fuel at the periphery of the fuel for gas distribution through slits is 1500°C, and in the case of gas distribution by jets the temperature is 1800 °C, which, in turn, leads to a lower level of NO_x generation.

4. For a jet-niche burner, in which gas distribution is organized through round holes located in a row, the length of the flame torch does not exceed 1 m, which corresponds to the nominal characteristics for a regular boiler burner of the DKVR type. In the case of gas supply through rectangular slits, the torch "crosses" the entire section of the fuel. This situation is unacceptable, as there is a risk of burning the pipes of the back screen of the boiler.

5. It was determined that in the center of the fuel there is a zone of reduced pressure, relative to which a circulation current is observed. On the periphery of this current, zones, the temperature of which corresponds to the temperature of the flame torch, are observed. This phenomenon is explained by the free-convective movement of gas masses due to changes in density.

6. The temperature field of the drum installed in the upper part of the fuel with a burner, in which the gas supply is organized through rectangular slots, is uneven, which negatively affects its strength.

References

1. Hossain, M. A. (2022). Computational study of methane-air combustion using the species transport model. *AIAA Science and Technology Forum and Exposition*, January 3–7, 2022, San Diego, paper AIAA 2022-1102. https://doi.org/10.2514/6.2022-1102.
2. Guessab, A., Aris, A., Bounif, A., & Gökalp, I. (2014). Reduced chemical kinetic mechanisms: Simulation of turbulent non-premixed CH4-Air flame. *Jordan Journal of Mechanical and Industrial Engineering*, vol. 8, no. 2, pp. 66–74.
3. Eswara Kumar, A., Janjanam, N., Nagaraju, M., & Diwakar, V. (2015). CFD analysis of combustion parameters of methane-air mixture. *International Journal of Engineering Research & Technology (IJERT)*, vol. 4, iss. 05, pp. 92–95. https://doi.org/10.17577/IJERTV4IS050217.
4. Li, D. & Ihme, M. (2021). Stability diagram and blow-out mechanisms of turbulent non-premixed combustion. *Proceedings of the Combustion Institute*, vo1. 38, iss. 4, pp. 6337–6344. https://doi.org/10.1016/j.proci.2020.06.225.
5. Anetor, L., Osakue, E., & Odetunde, C. (2012). Reduced mechanism approach of modeling premixed propane-air mixture using ANSYS fluent. *Engineering Journal*, vol. 16, iss. 1, pp. 67–86. https://doi.org/10.4186/ej.2012.16.1.67.
6. Rajak, A., Datta, A., & Kureel, V. (2016). Analysis of combustion characteristics of methane/air mixture in coaxial combustor. *International Journal of Advance Research in Science and Engineering*, vol. 5, iss. 10, pp. 277–285.
7. Mohammadnejad, S., An, Q., Vena, P. C., Yun, S., & Kheirkhah, S. (2020). Thick reaction zones in non-flamelet turbulent premixed combustion. *Combustion and Flame*, vol. 222, pp. 285–304. https://doi.org/10.1016/j.combustflame.2020.08.047.
8. Nair, A. S., Mohan Krishna, B., & Ajith Kumar, S. (2021). Study of non-premixed combustion of propane and methane using CFD. *Materials Today: Proceedings*, vol. 46, part 10, pp. 4593–4597. https://doi.org/10.1016/j.matpr.2020.09.713.

9. Rajak, U. & Verma, T. N. (2017). Numerical investigation on cylindrical combustion chamber for methane-air fuel. *International Journal on Theoretical and Applied Research in Mechanical Engineering (IJTARME)*, vol. 6, iss. 1–2, pp. 148–150.
10. Abdulin, M. Z. (2019). *Rozroblennia teplofizychnykh zasad tekhnolohii spaliuvannia palyv z zastosuvanniam strumenevonishovykh system* [Development of thermophysical principles of fuel combustion technologies using jet niche systems]: D. Sc. (Engineering) dissertation. Institute of Engineering Thermophysics of NAS of Ukraine. Kyiv, 390 p. (in Ukrainian).
11. (2012). ANSYS FLUENT 14.5 Theory Guide. ANSYS Inc. ANSYS Help. https://ansyshelp.ansys.com.

Received 18 February 2024

CFD-моделювання впливу типу газороздачі в пальниках на теплоаеродинамічні процеси в котлі ДКВР 10-13

[1] О. Ю. Черноусенко, [1] А. Ю. Рачинський, [1, 2] О. В. Баранюк, [1] О. А. Сірий

[1] Національний технічний університет України «Київський політехнічний інститут імені Ігоря Сікорського»
03056, Україна, м. Київ, пр. Берестейський, 37

[2] Інститут теплоенергетичних технологій НАН України
04070, Україна, м. Київ, вул. Андріївська, 19

Теми, пов’язані зі згорянням палива та його впливом на навколишнє середовище, ніколи не втратять своєї актуальності, оскільки питання ефективного згоряння й зменшення викидів є ключовими у виробництві електроенергії й охороні навколишнього середовища. Країни Європейського Союзу масово відмовляються від використання природного газу як палива для ТЕС. Проте в країнах Азії простота використання в промисловості природного газу як основного палива, його екологічність порівняно із вугіллям дали змогу ширше застосовувати природний газ у промисловості й енергетиці. Порівнюючи природний газ з альтернативними горючими газами (генераторним, доменним, шахтним, біогазом), можна зробити головний висновок про те, що він має найпривабливіші характеристики для використання в промисловості, зокрема і в енергетиці. Отже, у найближчий час заміна його на альтернативні палива в хімічній, важкій промисловості й енергетиці неможлива. Представлена робота присвячена CFD-моделюванню стабілізованого горіння без попереднього змішування в пальнику з низьким завихренням для двох режимів роботи котельного агрегату – номінального і на 60% потужності. Дослідження виконувалося за допомогою чисельних методів при використанні пакета прикладних програм Ansys-Fluent. Об’єкт дослідження – пальник, побудований за технологією, основаною на використанні струменево-нішевих систем із газорозподілом палива круглими струменями, що подаються перпендикулярно в потік окислювача через однорядну систему отворів. Предметом дослідження обрано процеси гідродинаміки і теплообміну, на підставі результатів аналізу яких отримана модель генерації NO_x в струменево-нішевих системах. Автори роботи вважають, що заміна штатних пальників водогрійного котла типу ДКВР-10-13 на струменево-нішеві може сприяти кращому змішуванню палива й повітря, а також забезпечити більш повне згоряння. У даній роботі розглянуто два типи пальників. В одному з пальників паливо подається крізь прямокутні щілини, в іншому – через розташовані в ряд круглі отвори. Повітря в обидва пальники подається через прямокутні щілини. Визначено, що газороздача крізь круглі отвори посилює розпилення суміші і збільшує зону розпилення продуктів згоряння. Проведена візуалізація розподілу середньої швидкості, температури, шкідливих домішок типу NO_x і компонентів реакції. Отримані результати свідчать, що зміна режиму течії, зміщення полум’я або його нестабільності відсутні. Встановлено, що на поширення продуктів згоряння і шкідливих домішок типу NO_x впливають як осьова, так і тангенціальна швидкість потоку. Газороздача круглими струменями стабілізує горіння і зменшує розширення полум’я.

Ключові слова: *газороздача, струменево-нішева технологія, ANSYS-Fluent, моделювання, газоподібне паливо, горіння, метан, паливня котла.*

Література

1. Hossain M. A. Computational study of methane-air combustion using the species transport model. *AIAA Science and Technology Forum and Exposition*, January 3–7, 2022, San Diego. Paper AIAA 2022-1102. https://doi.org/10.2514/6.2022-1102.
2. Guessab A., Aris A., Bounif A., Gökalp I. Reduced chemical kinetic mechanisms: Simulation of turbulent non-premixed CH4-Air flame. *Jordan Journal of Mechanical and Industrial Engineering*. 2014. Vol. 8. No. 2. P. 66–74.
3. Eswara Kumar A., Janjanam N., Nagaraju M., Diwakar V. CFD analysis of combustion parameters of methane-air mixture. *International Journal of Engineering Research & Technology (IJERT)*. 2015. Vol. 4. Iss. 05. P. 92–95. https://doi.org/10.17577/IJERTV4IS050217.
4. Li D., Ihme M. Stability diagram and blow-out mechanisms of turbulent non-premixed combustion. *Proceedings of the Combustion Institute*. 2021. Vol. 38. Iss. 4. P. 6337–6344. https://doi.org/10.1016/j.proci.2020.06.225.
5. Anetor L., Osakue E., Odetunde C. Reduced mechanism approach of modeling premixed propane-air mixture using ANSYS fluent. *Engineering Journal*. 2012. Vol. 16. Iss. 1. P. 67–86. https://doi.org/10.4186/ej.2012.16.1.67.
6. Rajak A., Datta A., Kureel V. Analysis of combustion characteristics of methane/air mixture in coaxial combustor. *International Journal of Advance Research in Science and Engineering*. 2016. Vol. 5. Iss. 10. P. 277–285.
7. Mohammadnejad S., An Q., Vena P. C., Yun S., Kheirkhah S. Thick reaction zones in non-flamelet turbulent premixed combustion. *Combustion and Flame*. 2020. Vol. 222. P. 285–304. https://doi.org/10.1016/j.combustflame.2020.08.047.
8. Nair A. S., Mohan Krishna B., Ajith Kumar S. Study of non-premixed combustion of propane and methane using CFD. *Materials Today: Proceedings*. 2021. Vol. 46. Part 10. P. 4593–4597. https://doi.org/10.1016/j.matpr.2020.09.713.
9. Rajak U., Verma T. N. Numerical investigation on cylindrical combustion chamber for methane-air fuel. *International Journal on Theoretical and Applied Research in Mechanical Engineering (IJTARME)*. 2017. Vol. 6. Iss. 1–2. P. 148–150.
10. Абдулін М. З. Розроблення теплофізичних засад технологій спалювання палив з застосуванням струменевонішових систем. дис. ... док. техн. наук: 05.14.06 «Технічна теплофізика і промислова теплоенергетика» / Інститут технічної теплофізики НАН України. Київ, 2019. 390 с.
11. ANSYS FLUENT 14.5 Theory Guide. ANSYS Inc. ANSYS Help, 2012. https://ansyshelp.ansys.com.