Konstantin Vasilev Kostov¹, Ivan Nikolaev Denev², Neven Yordanov Krystev³

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

Abstract: The economical combustion of gas fuel implies that it takes place with a minimum coefficient of excess air and minimal losses. Constructive, aerodynamic and physical factors have a determining influence on the completeness of combustion and the conditions of ignition. Using the ANSYS software program, the main characteristics of the combustion process in the cylindrical mixing section of a flat flame injection burner are investigated through computer simulation. A geometric model was created on which it is possible to study both straight and rotating jets. The possibility of numerically investigating the combustion of gaseous fuel (C₃H₈) in a confined air flow produced by injection is considered. A k-ε model of turbulence was used, which is based on the equation for turbulent kinetic energy and its dissipation rate. The purpose of the work is to study and analyze the changes and distribution of temperature and speed as well as the concentration of nitrogen oxides and carbon monoxide along the axis of the combustion chamber. The results are presented for the angles of inclination of the nozzles of 45° and 0°. Based on these, an analysis was made, where it
was found that with the increase in the degree of rotation, the absolute values of the temperature increase and the change in the mass concentration of the fuel along the length of the mixing section can be used to regulate the combustion process. The created numerical model can be successfully used to determine the main parameters of the burner under the same initial conditions, changing the angle of inclination of the nozzles. The obtained results can be considered as a basis for further research related to increasing the efficiency of the combustion process and lowering the harmful emissions produced by it.

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

---

### Introduction

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

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

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

The aim of the present work is to investigate the main characteristics of the combustion process in the cylindrical mixing section of an injection flat flame burner using computer simulation. Such information would be very useful in terms of managing and regulating the combustion process in real conditions.

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

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

The combustion of propane gas (C\textsubscript{3}H\textsubscript{8}), which flows through a system of nozzles directed tangentially to the inner surface of the mixer, is simulated. The environment is air at atmospheric pressure. The walls of the mixer are adiabatic. The length of the mixer is L = 200 mm. Its inner diameter is Ø = 69 mm.

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

![Fig. 1. 3D model of the coupled rotating nozzles and mixing section with a network of finite elements](Rys. 1. Model 3D sprzężonych dysz obrotowych i sekcji mieszanina w sieci elementów skończonych)

The purpose of ANSYS Workbench networking is to provide stable and easy-to-use networking tools that simplify network generation. Furthermore, the user is given the opportunity to control the network.
The mixing chamber with the injection rotary apparatus is part of the injection burner. The conditions of mixture formation are formed in the mixer and the combustion process takes place.

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

Analyzing and considering the individual models, the Finite Rate Chemistry (FRC) model taking into account the consequences of the final rates of chemical reactions is not appropriate in this case. This is due to the fact that the fuel (C$_3$H$_8$) and the oxidizer (air) are subjected to “fast” combustion (combustion rate dominates over the mixing speed of the reactants) (Fooladgar et al. 2021; Chung et al. 2021).

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

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

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

(1)

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

(2)

where:

- $C_\mu$ – the turbulence constant, $k$- turbulent kinetic energy,
- $\varepsilon$ – the dissipation rate of the turbulent kinetic energy,
- $\sigma_k, \sigma_\varepsilon$ – Schmid’s criteria for turbulent kinetic energy and the rate of its dissipation, respectively,
- $\sigma_t$ – turbulent Prandtl (Schmid) criteria.

The combustion process model used corresponds to Arrhenius’ law (Yeh et al. 2013), which gives the relationship between the rate of a chemical reaction and the change in temperature depending on:
\[ K = A_k T^{\beta_k} \exp \left( \frac{E_k}{RT} \right) \]  

(3)

where:
- \(A_k\) – a pre-exponential factor,
- \(\beta_k\) – a dimensionless temperature component,
- \(E_k\) – the energy of activation,
- \(T\) – the absolute temperature in K.

2. Results

The results obtained at the angles of inclination of the nozzles of 45° and 0° are presented. On the basis of these results, an analysis was made of the possibilities for regulation and management of the combustion process, while increasing its efficiency and maintaining or reducing harmful emissions from it. Results of changes in temperature, the concentration of nitrogen oxides and carbon monoxide along the axis of the combustion chamber and the distribution of temperature and velocity in the intersecting plane centrally along the axis of the mixer are presented sequentially.

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

![Temperature change along the axis of the mixer at an angle of inclination of the nozzles of 45°](image)

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

**Rys. 2. Zmiana temperatury wzdłuż osi mieszadła przy kącie nachylenia dysz 45°**
until the NOx concentration reaches its maximum at $z = 0.16$ m and then decreases. Under the same conditions (angle of inclination of the nozzles of 45°) a numerical study was conducted on the change in the concentration of CO, which is presented in Figure 4.

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

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

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

Rys. 4. Zmiana stężenia CO wzdłuż osi mieszadła przy kącie dyszy 45°
Figures 5 and 6 show, respectively, the temperature and velocity fields in the intersecting plane through the axis of the mixer at an angle of inclination of the nozzles of 45°.

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

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

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

Rys. 6. Pola prędkości w płaszczyźnie przecięcia przez oś mieszadła przy kącie nachylenia dysz 45°
Figures 7, 8 and 9 present the results of numerical experiments along the axis of the mixer at an angle of inclination of the nozzles of 0° for the change in temperature, NOx concentration and CO concentration, respectively.

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

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

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

**Rys. 8.** Zmiana stężenia NOx wzdłuż osi mieszadła przy kącie dyszy 0°
Figures 10 and 11 show the obtained data for the temperature and velocity fields in the intersecting plane through the axis of the mixer at an angle of inclination of the nozzles of 0°.

![Figure 9](image1.png)

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

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

![Figure 10](image2.png)

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

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

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

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

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

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

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

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

References


Konstantin Vasilev Kostov, Ivan Nikolaev Denev, Neven Yordanov Krystev

Badania procesu spalania w sekcji wstępnego mieszania wtrysku palnika gazowego

Streszczenie

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

SŁOWA KULCZOWE: badania numeryczne, spalanie, paliwo gazowe, reżim i parametry projektowe, emisy szkodliwe