Numerical Simulation of the Combustion Characteristics in a Flue Gas Internal Recirculation Burner

Lianjie Zhang, Chenghao Wu, Jianghui Zhang, Bin Zhang, and Chunjie Sui*

ABSTRACT: The combustion characteristics and NOx emissions of a newly designed flue gas internal recirculation low-NOx burner (FIR) were studied. In the study, experimental and numerical simulations of the FIR low-NOx burner were conducted under natural inlet air conditions at three different powers. Results show that the fuel inlet strongly influences the jet effect, thus influencing the flue gas recirculation rate, flame stability, and NOx emissions. With a medium power of 20 kW, the NOx emission of a FIR low-NOx burner is lower than 30 mg/N m$^3$. Higher or lower power will increase the NOx emissions or induce combustion instability. The swirling flow and bluff body structure can effectively improve the combustion stability. Three dominant frequencies of 54, 264, and 448 Hz can be observed from the power spectral densities of axial velocity, corresponding to the shedding vortex in the shear layer, the swirl frequency of processing vortex core (PVC), and the vortex induced by the PVC structure, respectively. The influences of vortex shedding and PVC structure are weak and inadequate to affect the overall flame stability.

1. INTRODUCTION

With the constant attention given to the environmental pollution problem, the emission standards for pollutants from gas boilers are becoming increasingly strict. In China, the NOx emission limit for new gas boilers has been reduced to below 50 mg/m$^3$, which poses an intensified challenge for research on technologies related to NOx reduction. To reduce NOx emissions from combustion equipment effectively, combustion product denitrification or low-nitrogen combustion technology is often used. Combustion product denitrification technology needs high first input and use cost, and the waste cannot be effectively recycled, which is prone to make secondary pollution. On the contrary, low-nitrogen burners do not produce additional waste in achieving low-NOx combustion and effectively reduce use costs. Hence, designing high-performance low-NOx burners and studying the combustion characteristics are significant.

Current low-NOx combustion technologies include staged combustion technology, flue gas recirculation, lean premixed combustion, and porous media combustion. Among them, the flue gas recirculation technology employs the burner structure for high-temperature flue gas dilution to reduce oxygen concentration and combustion temperature, therefore lessening NOx generation. Song et al. applied flue gas recirculation technology in coal swirling combustion and found that the recirculation zone induced by the swirling flow enhanced the effect of flue gas recirculation and achieved stable low-NOx combustion. Using numerical simulations, researchers have conducted numerous studies on flue gas recirculation low-NOx combustion technology. Torkzadeh et al. successfully applied the $k$-omega SST model and the steady-state flamelet model in the numerical simulation of flue gas recirculation combustion and obtained consistent results with the experimental results. Through experimental and numerical simulations, Zhu et al. investigated the effect of flue gas recirculation rate on the temperature distribution and NOx emission in an industrial furnace. Zhou et al. used numerical simulation to optimize the flue gas recirculation structure, aiming to reduce NOx generation by changing the flow field distribution and forming a larger reducing atmosphere zone. The flue internal recirculation (FIR) technology achieves flue gas recirculation by specific structural design. By numerical simulation and experiments, Nhan et al. verified that FIR can effectively change the flow characteristics and temperature distribution in a furnace and significantly reduce NOx emissions.

To achieve decreased NOx emissions and stable combustion, flue gas recirculation is often combined with swirling flow.

Received: August 8, 2022
Accepted: November 1, 2022
Published: November 14, 2022
The swirling flow strengthens the turbulence in a combustion chamber. It can enhance the mixing of fuel with air and recirculated flue gas, which can reduce the generation of nitrogen oxides. Nevertheless, it may cause flame surface folds, resulting in heat release pulsations, which increase NOx emissions and combustion instability. Lückoff et al. combined OH-LIF and particle image velocimetry techniques to investigate the effect of processing vortex core (PVC), which is generated by swirling flow and bluff body. Gatti et al. investigated the kinetic characteristics and flame response processes in the internal recirculation region of a combustion chamber with swirling flow and bluff body. Stöhr et al. examined the combustion instability in partially premixed swirling flames and found that the presence of swirling flow and bluff body may lead to a local quench at the root of the flame. Zhang et al. also determined combustion instability at the root of the swirling flame. Meanwhile, the flame folds produced by the swirling flow are beneficial to the stabilization of the downstream flame. Some researchers have found that nitrogen oxide emissions decrease with the increase in flue gas recirculation rate in experiment and simulation studies, while further combustion instability would be caused. In the FIR technology, the flue gas recirculation rate is strongly power dependent. Therefore, to achieve efficient and stable combustion, keeping appropriate power in FIR burners is crucial.

In this study, a FIR low-NOx burner under natural inlet air conditions is proposed on the basis of an oilfield heater burner. The FIR low-NOx burner adopts jet induction to produce flue gas recirculation, reducing the NOx emissions. Numerical simulations and experiments are conducted to explore the effects of power and flue gas recirculation rate on flame kinetic characteristics and NOx emissions.

2. FIR BURNER AND EXPERIMENT

The FIR low-NOx burner adopts powerless FIR technology, and the structure of the combustion nozzle is shown in Figure 1. The burner is 110 mm long and 60 mm in diameter. The diameter of the fuel pipe is 6 mm. The length ratio of the convergent segment of the Venturi tube and the divergent segment is about 1:3. The fuel gas is composed of CH₄ and air, both with a volume fraction of 50%. It is injected from the fuel jet with high speed and flows into the Venturi tube, where the fuel gas expands adiabatically and mixes with the recirculation flue gas. Under the acceleration and deceleration effects of venturi tubes, the mixing is more complete. Then, the mixtures rush out of the Venturi for combustion. Flue recirculation is produced by the pressure difference between the fuel jet and the recirculation cavity, which is induced by the fuel’s high velocity, corresponding to the Bernoulli equation. The recirculation flue will dilute the combustion area, reducing the mole fraction of oxygen and the temperature inside the flame and thus lessen the generation of thermal NOx and fast NOx.

A FIR low-NOx burner is installed and tested in a heating furnace of Hekou Oil Production Plant, Shengli Oilfield. The experimental system is shown in Figure 2. It is mainly composed of a heating furnace (combustion system), a gas supply system (continuous delivery of fuel, 300 K), and a data sampling and analysis system (with a Testo 350XL multifunctional flue gas analyzer) able to obtain the NOx emissions under different operating conditions. The analysis system is calibrated by the manufacturer, and the error is within 0.5%.

3. NUMERICAL METHODS

In this study, a FIR low-NOx burner and combustion chamber are simulated in three dimensions on the basis of Ansys Fluent. The effect of different powers on the combustion characteristics of a FIR low-NOx burner is studied.

In the simulation, a scale-adaptive simulation (SAS) turbulence model is adopted to solve the strong swirling and turbulent flow in the combustion chamber. The SAS model combines the Reynolds equation (RANS) and large eddy simulation (LES). It can automatically determine whether the flow has entered a nonconstant flow state on the basis of the local flow and then achieve conversion in the RANS and LES models by local turbulence scales. The SAS model can accurately capture the nonconstant vortex structure in the flow field with acceptable time spent, which is more suitable for engineering problems. SAS models at different levels are calibrated and validated with numerous practical engineering applications.

The eddy-dissipation-concept (EDC) model is adopted to solve the interaction between the chemical reaction and turbulence in this study. This model is applicable to complex chemistry-based turbulent combustion problems. Coupled with LES, it can better capture the detailed aspects of the combustion process. The EDC model assumes that the reaction occurs in a fine-scale turbulent structure. The length fraction of the fine scale is defined as eq 1, where $\xi^{*}$ is the fine-scale length fraction; $C_{l}$ is a volume fraction constant, taking 2.1377; and $\nu$ is the kinematic viscosity. The characteristic timescale of chemical reactions occurring in the fine turbulent structure is defined as eq 2, where $r^{*}$ is the characteristic timescale; and $C_{r}$ is a time scale constant, taking 0.4082.

$$\xi^{*} = C_{l} \left( \frac{\nu}{k} \right)^{1/4}$$

$$r^{*} = C_{r} \left( \frac{k}{\nu} \right)$$
\[ \tau^* = C \left( \frac{U}{F} \right)^{1/2} \]

The simulations employ a two-step methane mechanism and consider thermal, prompt, and intermediate NOx reaction mechanisms.\(^7\) The finite volume method is used to solve the basic equations,\(^2\) and the SIMPLE solver is used for pressure–velocity coupling. In spatial discretization, the convection terms of all transport equations are in a second-order upwind format.

A FIR low-NOx burner is mounted on a cylindrical combustor, as shown in Figure 3a. A polyhedral meshing approach is used in this study, and the grid is refined near the burner and the combustion region. Three sets of mesh with grid numbers of 0.64, 1.31, and 2.63 million are used to verify the grid irrelevance. The mesh with 0.64 million grids obtains an enormous difference from the two other ones. As shown in Figure 3b, the meshes with 1.31 and 2.63 million grids obtain results with less difference. Thus, the simulations will adopt the mesh with 1.31 million grids to reduce computation time cost. The mesh is shown in Figure 3c, with boundary conditions indicated.

The fuel inlet is set with flow velocities of 10, 20, and 30 m/s, corresponding to the thermal powers of 10 kW (case A), 20 kW (case B), and 30 kW (case C), respectively. The temperatures of fuel and inlet air are 300 K. The air inlet and outlet are defined as pressure boundaries, and the pressure difference is \(-5\) Pa according to real working conditions. The walls are defined as adiabatic and no-slip walls.

The time step is \(2 \times 10^{-5}\) s in all cases, keeping the max value of Courant number less than 0.5 to satisfy the calculation stability and convergence. The calculation is done after the full development of combustion statistics.

4. RESULTS AND DISCUSSION

4.1. Effects of Thermal Powers. 4.1.1. Flow Field. The mean axial velocity and temperature distributions of the \(y = 0\) cross section in three different cases are depicted in Figure 4. The power has a large influence on the velocity at the burner exit. As the heat power increases, the flame gradually becomes longer, and the flame center moves downstream. The high-speed zone locates in the Venturi throat. When the fuel jets out from the burner, the axial velocity increases rapidly due to the sudden drop in pressure and exothermic combustion. The reflux areas locate in the recirculation cavity and in the central recirculation zone (CRZ) behind the bluff body. The reflux flow in the recirculation cavity is driven by the low-pressure area induced by the fuel jet. The recirculation flue mixes with air, reducing the flame combustion temperature and oxygen partial pressure in the combustion zone. Thus, the NOx emission because of local high temperatures and uneven combustion is reduced. As power increases, the reflux velocity in the recirculation cavity increases, and the reflux flue gas temperature gradually decreases.

The difference of power has a great influence on the position of the inner shear layer (ISL) and out shear layer (OSL) and the shape of the high-speed ring zone (HSRZ, the region between the ISL and OSL). As shown in Figure 4a, the velocity...
It makes a larger fuel jet angle and a wider CRZ. As the power increases, the axial flow rate on both sides of the bluff body is increased, the angle between the OSL and the horizontal direction gradually decreases, and the width of HSRZ is gradually narrowed. Under a high-power condition, the shape of the CRZ is compressed. The reflux velocity in the reflux chamber is larger, as shown in Figure 4c. Overall, the local high temperature zone of the combustor gradually moves to the downstream, the position of the shear layer and the thickness of the HSRZ are changed by the thermal expansion. The shear layer extends to the outside, and the HSRZ is thinned by extrusion.

The mean axial and tangential velocity distributions at $x = 0.111, x = 0.126, x = 0.156,$ and $x = 0.201$ in different cases are given in Figure 5. The width and location of the CRZ can be obtained from the negative axial velocity zone induced by the swirling flow. As shown in Figure 5, the power has a strong influence on the axial and tangential velocities in the flow field of the combustion zone, and the axial velocity at the nozzle exit is determined by the power. The negative axial velocity upstream of the flame front end corresponds to the CRZ. In case A, the low velocity cannot establish an effective swirling flow, as shown in Figure 5b. In the other cases, typical characteristics of a strong swirling flow can be observed as a large tangential velocity. In addition, it is found from Figure 5 that the reflux and swirl are stronger as the power increases. Therefore, when the power is 30 kW, the residence time in the high-temperature zone is longer.

4.1.2. NOx Emissions. Different flow fields due to diverse powers indirectly affect NOx emissions. This study tests the FIR low-NOx burner in real production environments through experiments, corresponding to the simulation cases. In the experiment, the fuel passes through the pipe and is ignited in accordance with the standard working pressure and power. When a burner is operating steadily, the NOx concentration in
the flue gas is recorded using a gas analyzer at the end of the combustion chamber.

The experimental results and simulation data are shown in Figure 6. All the NOx emission data are converted into 3.5% oxygen concentration. NOx gradually increases during 15–25 kW and tends to be consistent beyond 25 kW. When the power is 10 kW, the combustion is unstable, and the flame oscillates significantly; thus, no meaningful results are obtained in the experiment. The simulation results agree well with the trend of the experiment. Slightly higher NOx emissions can be observed in the experiment. They may be attributed to the fluctuation in the fuel inlet under experimental conditions. Overall, the NOx emissions can be accurately predicted by the simulations.

Figure 7 shows the averaged emission index of NO (EINO) and emission index of CO (EICO) generated per unit of fuel at the exit of the combustion chamber. The results are obtained from the simulation data after conversion in accordance with eq 3.

\[
E_l = \frac{1000 \cdot n_i x M_i}{x_{CO_2} M_{fuel}}
\]

where \(E_l\) is the emission index, \(n_i\) is the number of moles of carbon per unit mole of fuel, \(x_i\) is the measured concentration of pollutant \(i\), and \(M_i\) is the molecular weight of pollutant \(i\). \(x_{CO_2}\) and \(M_{fuel}\) are the measured \(CO_2\) concentration and molecular weight of fuel, respectively.

In eq 3, \(x_i\) and \(M_i\) are the measured concentration and molecular weight of pollutant \(i\), respectively. \(x_{CO_2}\) and \(M_{fuel}\) are the measured \(CO_2\) concentration and molecular weight of fuel, respectively.

From Figure 7, the nitrogen oxide generated per unit of fuel increases significantly with the increase in power, and the change in CO is not as significant as the change in NO. The amount of NOx is mainly by two factors: flame temperature and residence time. Figure 8 compares the temperature distribution and the thermal NO generation rate on the centerline of the chamber in three cases. As the fuel power increases, the maximum temperature gradually increases and moves downstream. The increase in the maximum temperature causes NO to show exponential growth and provides a positive environment for NO emissions. Combined axial and tangential velocities are shown in Figure 5, increased flame temperature and flame length lead to an increment in residence time of the high-temperature region, thus intensifying the generation rate of thermal NO.

The NO emission and flue gas recirculation situation under different operating conditions are shown in Table 1. The recirculation flue ratio is calculated as eq 4, where \(\chi\) is the ratio of recirculation flue to total air, \(Q_{fuel}\) is the fuel flow rate, \(Q_{flue}\) is the recirculation flue flow rate, and \(T_{re,flue}\) is the temperature of recirculation flue.

\[
\chi = \frac{300 \cdot Q_{flue}}{300 \cdot Q_{flue} + Q_{fuel} \cdot T_{re,flue}}
\]

Table 1 presents that the recirculation flue ratio in case A is higher than that in case B. It can effectively reduce the flame temperature and thus, the NOx emissions. However, in case A, the flame returns into the recirculation cavity easily, damaging the burner structure, and induces combustion instability. In case C, the temperature of recirculation flue is low, and the NOx emissions induced by the long residence time, as shown in Figure 8, cannot be ignored. The recirculated flue gas replaces part of the fresh air and has a "dilution effect" on the intake air. Flue gas contains more tri-atoms, such as carbon dioxide and water vapor, has higher specific heat capacity than air and has a "thermal effect" on the decrease in combustion chamber temperature. The combustion products of the flue gas re-participate in the combustion reaction and have a "chemical effect" on combustion. With regard to the NOx emissions and combustion instability, encouraging results are obtained in the case of 20 kW for the FIR low-NOx burner.

4.2. Flame Dynamic Characteristics in FIR Burner. To investigate the effects of flue gas recirculation and swirling flow on flame dynamic characteristics, two probe points are selected for power spectrum analysis of the axial velocity in this study, as shown in Figure 9. Point 1 is near the ISL, and point 2 locates in the fully developed combustion zone. Figure 9 demonstrates that the flame surface is wrinkled and in a state of strong turbulence, with an obvious vortex breakup phenomenon. At the root, the flame shows a dynamic axisymmetric distribution, swirling around the nozzle exit.
Figure 10 shows the frequency spectra at point 1. In this study, the velocity is measured using a sampling frequency of 50 kHz. A line of $k^{-5/3}$ is indicated in the figure to determine the range of energy decay corresponding to the inertial range. Most of the energy is contained in the low-frequency range. The simulation can obtain consistent results.

Figure 11 shows the power spectral densities of axial velocity at points 1 and 2 in the case with a power of 20 kW, which is regarded as a reasonable operating case with stable combustion and low-NOx emission. Three dominant frequencies of 54, 264, and 448 Hz can be observed at point 1, corresponding to the vortex shedding frequency in the shear layer, the swirl frequency of PVC, and the vortex induced by the PVC structure, respectively. The PVC and the induced vortex are weaker than the shedding vortices, with lower amplitudes. In the power spectral densities of axial velocity at point 2, no obvious dominant frequency exists, indicating that the shedding vortex and PVC structure have minimal influences on the flame structure and are inadequate to affect the flame stability. Figure 12 demonstrates the development of the PVC structure for one cycle, corresponding to a cycle time of $3.8 \times 10^{-3}$ (1/264) s. When the PVC structure rotates around the central axis, sequential processing of the vortex is induced along with the axial and radial directions.
The authors declare no competing financial interest.

ACKNOWLEDGMENTS

This research was supported by the Natural Science Foundation of Shandong Province of China (no. ZR2021ME131).

REFERENCES

(1) Emission standard of air pollutants for boilers. DB44/765-2019.
(2) Xu, Q.; Shen, M.; Shi, K.; Liu, Z.; Akkurt, N.; Xiong, Y.; Liu, L.; Feng, J.; Wang, J. Effect of the Mixing Structure Parameters of a Self-reflux Burner on Combustion Characteristics and NOx Emissions. J. Therm. Sci. 2021, 30, 1224–1236.
(3) Boulahlib, M. S.; Medaerts, F.; Boukhalfa, M. A. Experimental study of a domestic boiler using hydrogen methane blend and fuel-rich staged combustion. Int. J. Hydrogen Energy 2021, 46, 37628–37640.
(4) (a) Ahn, S. Y.; Go, S. M.; Lee, K. Y.; Kim, T. H.; Seo, S. I.; Choi, G. M.; Kim, D. J. The characteristics of NO production mechanism on flue gas recirculation in oxy-firing condition. Appl. Therm. Eng. 2011, 31, 1163–1171. (b) Gamrat, S.; Poraj, J.; Bodys, J.; Smolka, J.; Adamczyk, W. Influence of external flue gas recirculation on gas combustion in a coke oven heating system. Fuel Process. Technol. 2016, 152, 430–437. (c) Baltasar, J.; Carvalho, M. G.; Coelho, P.; Costa, M. Flue gas recirculation in a gas-fired laboratory furnace: Measurements and modelling. Fuel 1997, 76, 919–929.
(5) (a) Lee, T.; Kim, K. T. Combustion dynamics of lean fully-premixed hydrogen-air flames in a mesoscale multinozzle array. Combust. Flame 2020, 218, 234–246. (b) Ti, S.; Chen, Z.; Kuang, M.; Li, Z.; Zhu, Q.; Zhang, H.; Wang, Z.; Xu, G. Numerical simulation of the combustion characteristics and NOx emissions of a swirl burner: Influence of the structure of the burner outlet. Appl. Therm. Eng. 2016, 104, 565–576.
(6) Kamal, M. M.; Mohamad, A. A. Combustion in Porous Media. Proc. Inst. Mech. Eng. J. Power Eng. 2006, 220, 487–508. (accessed 2022/06/02).
(7) (a) Zhou, Y.; Wang, C.; Chen, X. Combustion characteristic study with a flue gas internal and external double recirculation burner. Chem. Eng. Process. 2021, 162, 108345.
(8) Song, M.; Huang, Q.; Niu, F.; Li, S. Recirculating structures and combustion characteristics in a reverse-jet swirl pulverized coal burner. Fuel 2020, 270, 117456.
(9) Torkzadeh, M. M.; Bolourchifard, F.; Amani, E. An investigation of air-swirl design criteria for gas turbine combustors through a multi-objective CFD optimization. Fuel 2016, 186, 734–749.
(10) Zhou, H.; Yang, Y.; Liu, H.; Hang, Q. Numerical simulation of the combustion characteristics of a low NOx swirl burner: Influence of the primary air pipe. Fuel 2014, 130, 168–176.
(11) Nhan, H. K.; Kwon, M.; Kim, S.; Park, J. H. CFID investigation of NOx reduction with a flue-gas internal recirculation burner in a mid-sized boiler. J. Mech. Sci. Technol. 2019, 33, 2967–2978.
(12) (a) Terhaar, S.; Kröger, O.; Paschereit, C. O. Flow Field and Flame Dynamics of Swirling Methane and Hydrogen Flames at Dry and Steam Diluted Conditions. J. Eng. Gas Turbines Power 2014, 137, 041503. (accessed 6/2/2022) (b) Stöhr, M.; Arndt, C. M.; Meier, W. Transient effects of fuel–air mixing in a partially-premixed turbulent swirl flame. Proc. Combust. Inst. 2015, 35, 3327–3335.
(13) Fric, T. F. Effects of fuel-air unmixedness on NO(x) emissions. J. Propul. Power 1993, 9, 708–713. (accessed 2022/06/02).
(14) Ducruix, S.; Schuller, T.; Durox, D.; Candel, S. Combustion Dynamics and Instabilities: Elementary Coupling and Driving Mechanisms. J. Propul. Power 2003, 19, 722–734. (accessed 2022/06/02).
(15) Paschereit, C. O.; Gutmark, E. Combustion Instability and Emission Control by Pulsating Fuel Injection. J. Turbomach. 2008, 130, 011012. (accessed 6/2/2022)
(16) Lückoff, F.; Sieber, M.; Paschereit, C. O.; Oberleithner, K. Impact of the Precessing Vortex Core on NOx Emissions in Premixed Swirl-Stabilized Flames—An Experimental Study. J. Eng. Gas Turbines Power 2020, 142, 111010. (accessed 6/3/2022)

(17) Gatti, M.; Gaudron, R.; Mirat, C.; Zimmer, L.; Schuller, T. Impact of swirl and bluff-body on the transfer function of premixed flames. Proc. Combust. Inst. 2019, 37, 5197–5204.

(18) Stöhr, M.; Boxx, I.; Carter, C.; Meier, W. Dynamics of lean blowout of a swirl-stabilized flame in a gas turbine model combustor. Proc. Combust. Inst. 2011, 33, 2953–2960.

(19) Zhang, W.; Wang, J.; Lin, W.; Guo, S.; Zhang, M.; Li, G.; Ye, J.; Huang, Z. Measurements on flame structure of bluff body and swirl stabilized premixed flames close to blow-off. Exp. Therm. Fluid Sci. 2019, 104, 15–25.

(20) (a) Shi, B.; Hu, J.; Peng, H.; Ishizuka, S. Effects of internal flue gas recirculation rate on the NOx emission in a methane/air premixed flame. Combust. Flame 2018, 188, 199–211. (b) Shinomori, K.; Katou, K.; Shimokuri, D.; Ishizuka, S. NOx emission characteristics and aerodynamic structure of a self-recirculation type burner for small boilers. Proc. Combust. Inst. 2011, 33, 2735–2742.

(21) (a) Menter, F. R.; Egorov, Y. The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 1: Theory and Model Description. Flow, Turbul. Combust. 2010, 85, 113–138. (b) Egorov, Y.; Menter, F.; Lechner, R.; Cokljat, D. The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 2: Application to Complex Flows. Flow, Turbul. Combust. 2010, 85, 139–165.

(22) (a) Zhao, R.; Xu, J.-L.; Yan, C.; Yu, J. Scale-adaptive simulation of flow past wavy cylinders at a subcritical Reynolds number. Acta Mech. Sin. 2011, 27, 660. (b) Xu, C.-y.; Zhou, T.; Wang, C.-l.; Sun, J.-h. Applications of scale-adaptive simulation technique based on one-equation turbulence model. Appl. Math. Mech. 2015, 36, 121–130.

(23) (a) Magnussen, B. On the structure of turbulence and a generalized eddy dissipation concept for chemical reaction in turbulent flow. 19th Aerospace Sciences Meeting, Aerospace Sciences Meetings; American Institute of Aeronautics and Astronautics, 1981. (b) Gran, I. R.; Magnussen, B. F. A Numerical Study of a Bluff-Body Stabilized Diffusion Flame. Part 2. Influence of Combustion Modeling And Finite-Rate Chemistry. Combust. Sci. Technol. 1996, 119, 191–217.

(24) Yılmaz, İ. Effect of Swirl Number on Combustion Characteristics in a Natural Gas Diffusion Flame. J. Energy Resour. Technol. 2013, 135, 042204. (accessed 6/2/2022)