MICRO-MODULAR AIR DRIVEN COMBUSTION NOZZLE Experimental and Numerical Modelling Studies towards Optimal Geometric Design

by

Abay M. DOSTIYAROV and Samal B. SADYKOVA*

^a Almaty University of Power Engineering and Telecommunications named after Gumarbek Daukeev, Baytursynuly, Almaty, Kazakhstan

Original scientific paper https://doi.org/10.2298/TSCI210410257D

Experimental and numerical modelling studies on optimized design of micro-modular air driven combustion nozzle have been developed. Moreover, optimal angle of outlet swirler (vortex generator) connected in series with the nozzle has been analyzed in the context of minimization of NO_x in the flue gases. Numerical modelling by ANSYS verifying the experimental data has been carried out. It has been established that an angle of about 20° of the outlet vortex generator has been able to assure optimal flame length and combustion efficiency.

Key words: nozzle, combustion chamber, vortex generator, CFD modelling, swirler, swirling flow, blade angle

Introduction

Intensive development of energetics for electricity and heat supply in past century led to significant contaminations of air and soils by emitted combustion products [1]. The modern trend in energy transformation is the minimization of toxic combustion emissions [2] upon strong environmental control and legislative regulations [3], oriented mainly towards minimization of emissions of NO_x and CO down to 25 ppm. All this restrictions demand developments of new combustion technologies and use of the natural gas principle environmentally friendly fuel [4, 5].

It is well-known that one the most effective approach in the combustion proves performance is the premixed combustion [6] where lean premixed prevaporized and rich burn-quick quench-lean burn premixed stream allows reduction of NO_x emissions due to attained low flame temperature granted by almost perfect fuel-oxygen mixing and absence of local hot zones. However, the premixed combustion of fuel-air mixtures strongly depends on the flame stability and the mixing efficiency [7, 8]. Therefore, the development of new combustion devices requires optimal geometry assuring almost high efficiency of the process in them satisfying all imposed regulatory restrictions with regards to toxic components emissions [3].

For the sake of clarity of explanations and comprehensiveness of the preliminary analysis of existing situation a summary of literature data with relevant comments is listed in tab. 1.

^{*} Corresponding author, e-mail: kundyzdy@mail.ru

Dostiyarov, A. M., *et al.*: Micro-Modular Air Driven Combustion Nozzle Experimental and ... THERMAL SCIENCE: Year 2022, Vol. 26, No. 2B, pp. 1557-1566

Reference	Methods	Object	Problem studied	
Bulat <i>et al.</i> [9]	Numerical study	Premixed combustion of methane in a annular combustion chamber and channel	Emissions of CO and NO _x	
Jerzak and Kuznia [10]	Experimental study	Premixed combustion of natural gas in quartz tube chamber	Effect of the vortex number and, oxygen content in the fuel in reverse flows and flame instability	
Wu <i>et al.</i> [11]	Numerical study	Turbulent vortex flow is chambers related to gas turbines	The effect of the exist geometry on the structure of the vortex flow and vortex cores in the chamber	
Hossein et al. [12]	Numerical study	Premixed combustion of methane-air mixtures by a nozzle with a perforated exist	Effect of the geometry on the combustion efficiency in isothermal flow	
Runyon et al. [13]	Experimental study	Premixed combustion of lean methane in vortex chamber	Determination of the reasons leading to flame	
Lin <i>et al.</i> [14]	Experimental study	Estimation of the effect of fuel replacements on the premixed combustion efficiency	Analysis of Wobbe index, heat release ability and the theoretical demand of air supply	
Zhao <i>et al.</i> [15]	Numerical study	Effect of the mixing distance and the mixing efficiency with a double-cone burner	It was demonstrated that dimensionless distance affects the mixing process and the location of the high temperature zone as well as the emissions	
Wu <i>et al.</i> [16]	Numerical study	Structure and flow characteristics of propane-air mixture over a stabilized bluff-body	Combustion and limits of flame stability	
Chen <i>et al.</i> [17]	Numerical study	A model of combustion chamber with partial preliminary premixing of the methane	Effects of auto-oscillations on the fuel-air mixing	
Karyeyen and Ilbas [18]	Experimental study	A nozzle for combustion of low caloric fuel	Determination of the operating regime relevant to stable combustion process	
Enagi <i>et al.</i> [19]	Numerical study	Development of a combustion chamber for a micro-gas turbine	Numerical study towards optimization of combustion chamber geometry	
Yilmaz et al. [20]	Experimental study	Flame stability in chamber with a premixing	Effects of fuel composition, the equivalence coefficient, the vortex number on the flame behaviour (static and dynamic) in premixed flow	
Rowhani and Tabejamaat [21]	Experimental study	Combustion of biogas in swirling flow	Effect of swirling the flow on the stability of the combustion of biogas fuel was experimentally investigated	
Wang <i>et al.</i> [22]	Numerical study	Combustion of fuel in tangentially boiler	Mechanism of NO _x formation is being studied and the factors affecting their emission are studied too	

Table 1. Studies on premixed combustion of gaseous fuels

1558

All these works summarized in tab. 1 are oriented towards either pure experimental studies or to numerical simulations by CFD of premixed combustion with the main task to determine the optimal nozzle design.

This work addresses results from a complex series of studies on a micro-modular air driven nozzle (MMADN) claimed by a patent [23]. The main task is determination of optimal geometric parameters of the nozzle and effects of the blade angle of the outlet swirler (vortex generator) on the gross combustion process performance and related emissions of NO_x . The study presents two main directions in pursuing this task: combustion experiment studies on a physical model using results obtained by isothermal modelling, see reports in [24], and numerical modelling by CFD of the same physical model oriented to geometry optimization and rational fuel supply.

Study approaches

Experimental

The micro-modular air driven combustion nozzle is shown schematically in fig. 1. The device consists of cylinder with inner diameter 71 mm with vortex generators: one at the inlet and two outlets. The swirler parameters are: 8 blades of 1.5 mm in length and 12 mm width, the angle of the inlet vortex blades is $\beta_1 = 40^\circ$, while for the outlet swirler it is $\beta_2 = 30^\circ$. It is important to mention that angles of blade orientations of both vortex generators are in opposite directions. The blade angle β_3 of the third swirler was possible to vary 20° to 50°.



Figure 1. Schematic presentation of the micro-modular air driven combustion nozzle

The fuel inlet was chosen to be at a distance of one caliper (one caliper is a length that is equal to the inner diameter of the micro-modular nozzle, where the gas-air mixture flows) from the inlet frontal plane of the micro-modular nozzle. This point was selected after the isothermal aerodynamic experiments since the thermo-anemometric measurements revealed that high turbulence intensity exists [24].

The experiments were carried out with the experimental installation shown in fig. 2. It consists of fan for air supply, flow stabilizators, gas bottle with propane fuel, micro-modular nozzle as well as measuring and controlling devices.

The air supplied by the fan passes through a tube bundle for in order almost homogeneous flow velocity profile to be attained. The fuel is supplied from the gas bottle. The emissions of NO_x in the flue gases was determined with gas analyzer Testo 350.



Figure 2. Experimental set-up; TE – the k-type (Chromel/Alumel) thermocouple sensor complete with device OVEN TRM-138, PE – manometer complete with BPO-5-3 reducer, SE – anemometer Testo 416, FE – fuel-flow meter BK-G4, and AG (^{NOx)} – gas analyzer Testo 350

The following values were measured during the experiment:

- temperature, pressure and consumption of the fuel,
- air temperature and velocity at the entrance of the micro-modular nozzle,
- barometric pressure of the environment,
- temperature of gases at the exit of the micro-modular nozzle, and
- composition of combustion products with the gas analyzer.

In all experimental runs the air flow, with temperature of 15 °C, supplied to the nozzle was established at 11 m/s. The fuel supply was 2.16 kg/h with a pressure of 30 kPa. The equivalence coefficient inside the micro-modular nozzle $\phi = 0,185$. Combustion temperatures and NO_x emissions are given below in section *Results*.

Numerical modelling

The numerical modelling and verifications were carried out with CFD utilizing AN-SYS FLUENT package. The geometry of the model studied is shown in fig. 3. Two models of the micro-modular nozzle were studied numerically, with two different angles of the outlet vortex blades (20° and 50°). The geometric dimensions of the model shown in fig. 3(a) correspond to the physical model studied, fig. 1. The second model has the following parameters: 46 mm diameter, 100 length, and fuel supply point located at 50 mm from the inlet frontal plane of micro-modular nozzle.

The first step in numerical modeling is mesh creation. The study module area was divided into a set of unstructured tetrahedral small volumes, fig. 3(c) in the meshing preprocessor of the ANSYS software package. Mesh characteristic: \sim 555101 elements, \sim 155839 nodes, orthogonal quality – 0.8, skewness – 0.2, aspect ratio – 2.9.

Dostiyarov, A. M., *et al.*: Micro-Modular Air Driven Combustion Nozzle Experimental and ... THERMAL SCIENCE: Year 2022, Vol. 26, No. 2B, pp. 1557-1566



Figure 3. Geometries of micro-modular nozzle models studied; (a) 1st model, (b) 2nd model, and (c) mesh

Governing equations were solved by the finite volume method [25] in CFD simulations. Governing equations are based on the Navier-Stokes equation which describes the movement of a turbulent flow, which includes:

– continuity equation

$$\frac{\partial \rho}{dt} + \nabla(\rho \vec{\mathbf{v}}) = 0 \tag{1}$$

momentum-conservation equation

$$\frac{\partial}{\partial t}(\rho \vec{\mathbf{v}}) + \nabla(\rho \vec{\mathbf{v}} \vec{\mathbf{v}}) = -\nabla p + \nabla(\overline{\overline{t}}) + \rho \vec{\mathbf{g}}$$
(2)

where ρ is the density, \vec{v} – the velocity vector, p – the static pressure, $\overline{\vec{\tau}}$ – the stress tensor, and $\rho \vec{g}$ – the gravitational body force.

The stress tensor $\overline{\overline{\tau}}$ is:

$$\overline{\overline{\tau}} = \mu \left[(\nabla \overline{\mathbf{v}} + \nabla \overline{\mathbf{v}}^T) - \frac{2}{3} \nabla \overline{\mathbf{v}} I \right]$$
(3)

where μ is the molecular viscosity, I – the unit tensor, and the second term on the right hand side is the effect of volume dilation.

The generalized transfer equation was used to determine the field of scalar quantities such as enthalpy, temperature, concentration of harmful substances, turbulence parameters k- ε , etc.:

$$\frac{\partial}{\partial t}(\rho\phi) + \nabla(\rho v\phi) = \nabla(\rho \Gamma \nabla\phi) \tag{4}$$

where ϕ is a scalar quantity and Γ – the diffusion coefficient.

The flow is assumed to be stationary and incompressible. The numerical modelling of the combustion process in the micro-modular device utilized the non-premixed combustion model, based on the assumption of instantaneous chemical reactions.

The investigated swirling flow is characterized by high turbulence of flow parameters (for example, speed, pressure, density, *etc.*). They have a random nature in the Navier-Stokes equation, so it is impossible to solve them directly. Therefore, the Navier-Stokes eqs. (1)-(4) was averaged by the Reynolds method in numerical modeling. A semi-empirical k- ε turbulence model (modification *realizable*) was used to close the Reynolds-averaged navier stokes (RANS) approach. The realizable k- ε model describes well the flow pattern in both the module volume and the boundary layer [26]. The probability density function was calculated with respect to propane. The model did not include radiation heat transfer since as suggested by some authors [27] this heat transfer process does not affect significantly the flame structure. The problem solved was considered as steady-state one boundary conditions summarized in tab. 2.

Table 2. Boundary conditions

Inlet air		Fuel inlet		Outlet	
Velocity, [ms ⁻¹]	11	Mass flow rate, [kgs ⁻¹]	2.16	Backflow temperature, [K]	
Temperature, [K]	288	Temperature, [K]	288	Gauge pressure, [Pa]	0
Gauge pressure, [Pa]	0	Gauge pressure, [kPa]	30	Turbulence intensity, [%]	2

A thermal and prompt mechanism was selected to calculate NO_x emissions. Partialequilibrium approach was chosen for the formation of O and OH radicals.

The applied sampling methods and the algorithm for the numerical solution of SIM-PLE are detailed in [25].

Results

Experimental studies with propane combustion have shown that the length of the micro-modular nozzle about three calipers is too long and therefore combustion begins inside the nozzle. It can be seen in the photo in fig. 6. Combustion begins approximately at a distance of two calipers from the frontal plane, which is also confirmed by the numerical studies, fig. 5. As a result, the MMADN length has been reduced to 2 calipers, as shown in fig. 3(b). That is, the task was that the fuel, leaving the nozzle, almost immediately goes through the outlet swirlers. The primary objectives of a pair of outlet swirler includes preventing the flame from flowing back into the micro-modular nozzle.



Figure 4. Temperature profiles: experimental and simulated

The temperature profiles shown in fig. 4 correspond to measurements in the nozzle cross-section at a distance 200 mm from the outlet place of the micro-modular nozzle related to the 1st (physical) model. Compared with the data generated by the CFD modelling we can see that there is a satisfactory agreement between them. In both type of experiments (physical and numerical) there are rapid reductions in temperatures when $\beta_3 = 20^\circ$. Moreover, when $\beta_3 = 50^\circ$, in the same cross-section, temperature profile is almost flat (and symmetrically distributed) in a wide range located lower than in the case with $\beta_3 = 20^\circ$. This can be attributed to the fact that when $\beta_3 = 50^\circ$ the vortexes are

stronger and more complete and occupies almost all section of the chamber, that is we may assume the flow field is almost homogeneous across the chosen cross-section. As a result the more effective mixing with the inlet air flow allows more effective control of both the temperature in the pressure drop.

1562

The axial temperature profile related to the 1st model shown in fig. 5 reveals that the high temperature zone is located inside the micro-modular nozzle and therefore the combustion process starts inside it even though it should be localized outside the nozzle. However, it was experimentally confirmed that the combustion process took place inside the micro-modular nozzle. In addition, the pictures in fig. 6 indicate that the first 60% of the module are covered by a dark shadow related to a high temperature field. This can be attributed to two main factors: the equivalence coefficient inside the module is less than 1 and the long distance traveled by the premixed fuel along the module axis. The flow structures inside the micro-modular nozzle (with both angles of blade orientations, *i.e.* different β_3) are almost equal since the angle of the blades of the vortex generator, β_1 , remained unchanged.



Figure 5. Temperature profiles as a result of the numerical modelling of the 1st model (*for color image see journal web site*)

The comparative analysis of the experimental results and the outcomes of the numerical modelling with respect to the NO_x emissions indicate significant differences (see the data summarized in tab. 3. The reason for the large discrepancy is possibly due to the fact that radiation heat transfer was not taken into account in the numerical modeling. Despite this the data have common trends, that is: in both cases $\beta_3 = 20^\circ$ the NO_x concentration is greater than when $\beta_3 = 50^\circ$. The main reason explaining this can be attributed to the fact that at $\beta_3 = 50^\circ$ the common temperature at the nozzle exist is lower than when $\beta_3 = 20^\circ$, see fig. 5.



Figure 6. Photos of the studied micro-modular nozzle used in the experiments; (a) front view with vortex generator at the central axis and (b) vertical view

When the task was the flame be outside the micro-modular nozzle then preliminarily numerical modelling was carried out varying the nozzle geometry. The outcomes of these numerical experiments are presented in fig. 7 and they confirm the fact that version presented at fig. 3b (2nd model) is the optimal one. The pictures presented in fig. 7 show the flow structures in the 2nd model when optimal configurations of blades, namely $\beta_3 = 50^\circ$ and $\beta_3 = 20^\circ$,

Dostiyarov, A. M., et al.: Micro-Modular Air Driven Combustion Nozzle Experimental and ... THERMAL SCIENCE: Year 2022, Vol. 26, No. 2B, pp. 1557-1566

NO _x ppm (O ₂ 15%)	$\beta_3 = 20^\circ$	$\beta_3 = 50^\circ$
Experiments	20.5	10.9
Numerical modeling	5.8	5.5

of the outlet swirler are used. The profiles in fig. 7 reveal that in both situations the flame is inside the micro-modular nozzle. Hence, due to reduction in the length and diameter of the module the flame can be outside the nozzle which actually reduces the residence time of the premixed fuel-air mixture. Moreover, additional bluff-bodies as hemispheres located at the nozzle axis just behind the outlet swirler allows the flame to be outside the nozzle [28].



Figure 7. Temperature profiles as outcomes of the numerical modelling with 2nd model (*for color image see journal web site*)

The profiles presented in fig. 7 indicate that the angle of the blades of the outlet swirler, β_3 , affects the flame structure at the exit of the nozzle. With $\beta_3 = 50^\circ$ there is high temperature elongated zone, while with $\beta_3 = 20^\circ$ the flame is longer but relatively colder: its temperature is about 300 °C lower than at the same location but with $\beta_3 = 50^\circ$ and this can be explained by the fact that with $\beta_3 = 50^\circ$ the vortex flow structure at the outlet of the nozzle is stronger than when $\beta_3 = 20^\circ$ is used. The stronger vortex structure creates reverse flows towards the central combustion zone and they concentrate themselves at the nozzle exist. However, with $\beta_3 = 20^\circ$ the vortices generated are no so strong and consequently the axial flow component due to the advection dominates and the combustion zone becomes larger. The high temperature results in increase in the NO_x emissions, on the other hand the extended flame zone also increase the NO_x emissions due to the longer residence time of the combustion products in the high temperature zone. Therefore, the angles of the blades of the swirler affects directly the level of the NO_x emissions. The numerical modelling reveals that the emissions of NO_x (with 15% volumetric oxygen concentration) at the control position are 5.1 and 6.3 ppm for $\beta_3 50^\circ$ and 20° accordingly. These results are similar the ones obtained with the 1st model, see tab. 3.

The angle of the outlet swirler's blades, β_3 , affects also the flow structure inside the micro-modular nozzle, precisely at the zone between the two swirlers at the exit located close to the nozzle exist. This zone is important because it allow controlling the flame escape. Comparing the temperature profiles in this zone we can see that there red spots $\beta_3 = 50^\circ$,

1564

which actually means that the flame is inside the nozzle. With $\beta_3 = 20^\circ$ due to the low hydrodynamic friction factor of the outlet swirler the temperature in this zone is lower because the flow rate is greater. Hence, low values of β_3 are preferable.

Conclusions

Optimal geometric parameters of a micro-modular air-driven nozzle for gaseous fuel combustion have been determined by experimental studies and numerical modelling (CFD modelling by ANSYS FLUENT). The main results can be outlined as.

- The length of the micro-modular nozzle of 3 calipers is too large and because of that the combustion zone is inside the nozzle. This was confirmed by the numerical modelling.
- The reduction in both the length and diameter of the module (down to 2 calipers) as well as the use of bluff bodies at the nozzle exist allow easily the flame to be in the outside zone and a stable flame to be attained.
- The angle of the outlet swirler blades β_3 affects the flame structure and the NO_x emissions. In all cases studied the blade angle 50° allows to reduce the NO_x emissions but the cost is the increased hydrodynamic resistance of the module. The increased hydrodynamic resistance of the module may allow flame to be partially in the inner nozzle zone.
- The studies released that the blades of the outlet swirler inclined at $\beta_3 = 20^\circ$ and the nozzle length of two calipers are optimal with respect to flame structure and flow pattern and may be recommended for micro-modular nozzle optimal design.
- The study reveals that the micro-modular air driven combustion nozzle design can provide efficient fuel combustion thus meeting both the contemporary and potential future requirements addressing NO_x emissions. The results of the research form a good scientific basis for future development of combustion chambers with micro-modular combustion nozzles.

References

- ***, Health and Environmental Impacts of Electricity Generation Systems: Procedures for Comparative Assessment, Technical reports series, ISSN 0074-1914, International Atomic Energy Agency, Vienna, Austria, 1999
- [2] Bharvirkar, R., *et al.*, NOx Emissions Trading and Episodic Air Quality Violations in Maryland, Final Report Prepared for Power Plant Research Program, 2003
- [3] ***, LCP BREF: Reference Document on Best Available Techniques for Large Combustion Plants, European Commission, 2006
- [4] Lefebvre, A. H., Ballal, D. R., Gas Turbine Combustion: Alternative Fuels and Emissions, 3rd ed., CRC Press, Boka Raton, Fla., USA, 2010
- [5] Baklanov, A. V., Neumoin, S. P., A Technique of Gaseous Fuel and Air Mixture Quality Identification Behind the Swirl Burner of Gas Turbine Engine Combustion Chamber, *Russian Aeronautics*, 60 (2017), 1, pp. 90-96
- [6] Benim, A. C., Syed, K. J., Concepts Related to Combustion and Flow in Premix Burners, in: *Flashback Mechanisms in Lean Premixed Gas Turbine Combustion*, Academic Press, New York, USA, 2015, 134 p, pp. 5-18
- [7] Huth, M., Heilos, A., Fuel Flexibility in Gas Turbine Systems: Impact on Burner Design and Performance, in: *Modern Gas Turbine Systems High Efficiency, Low Emission, Fuel Flexible Power Generation* (Ed. P. Jansohn), Woodhead Publishing, Sawston, UK, 2013, pp. 635-684
- [8] Baubek, A., et al., Complex Studies of the Innovative Vortex Burner Device with Optimization of Design., Studies in Systems, Decision and Control, 351 (2021), Apr., pp. 139-153
- [9] Bulat, G., *et al.*, NO and CO Formation in an Industrial Gas-Turbine Combustion Chamber Using LES with the Eulerian Sub-Grid PDF Method, *Combustion and Flame*, *161* (2014), 7, pp. 1804-1825

- [10] Jerzak, W., Kuznia, M., Experimental Study of Impact of Swirl Number as Well as Oxygen and Carbon Dioxide Content in Natural Gas Combustion Air on Flame Flashback and Blow-Off, *Journal of Natural* Gas Science and Engineering, 29 (2016), Feb., pp. 46-54
- [11] Wu, Y., et al., Effect of Geometrical Contraction on Vortex Breakdown of Swirling Turbulent Flow in a Model Combustor, Fuel, 170 (2016), Apr., pp. 210-225
- [12] Hossein, M., et al., Numerical Study of Geometric Parameters Effecting Temperature and Thermal Efficiency in a Premix Multi-Hole Flat Flame Burner, Energy, 125 (2017), Apr., pp. 654-662
- [13] Runyon, J., et al., Lean Methane Flame Stability in a Premixed Generic Swirl Burner: Isothermal Flow and Atmospheric Combustion Characterization, *Experimental Thermal and Fluid Science*, 92 (2018), Apr., pp. 125-140
- [14] Lin, X., et al., Experimental Research on Gas Interchangeability Indices for Domestic Fully Premixed Burners, Fuel, 233 (2018), Dec., pp. 695-703
- [15] Zhao, T., et al, Investigation on Effect of Mixing Distance on Mixing Process and Combustion Characteristics of Double-Cone Burner, Fuel, 253 (2019), Oct., pp. 540-551
- [16] Wu, B., et al., A Numerical Investigation of the Flame Structure and Blowoff Characteristics of a Bluff-Body Stabilized Turbulent Premixed Flame, Combustion and Flame, 202 (2019), Apr., pp. 376-393
- [17] Chen, Zh. X., et al., Interaction Between Self-Excited Oscillations and Fuel-Air Mixing in a Dual Swirl Combustor, Proceedings of the Combustion Institute, 37 (2019), 2, pp. 2325-2333
- [18] Karyeyen, S., Ilbas, M., Experimental and Numerical Analysis of Turbulent Premixed Combustion of Low Calorific Value Coal Gases in a Generated Premixed Burner, *Fuel*, 220 (2018), May, pp. 586-598
- [19] Enagi, I. I., et al., Combustion Chamber Design and Performance for Micro Gas Turbine Application, Fuel Processing Technology, 166 (2017), Nov., pp. 258-268
- [20] Yilmaz, H., *et al.*, Experimental Investigation of Flame Instability in a Premixed Combustor, *Fuel*, 262 (2020), Feb., 116594
- [21] Rowhani, A., Tabejamaat, S., Experimental Study of the Effects of Swirl and Air Dilution on Biogas Non-Premixed Flame Stability, *Thermal Science*, 19 (2015), 6, pp. 2161-2169
- [22] Wang, W. S., et al., Numerical Simulation of NOx Emission Characteristics During Combustion in 350 MW Supercritical Cogeneration Tangentially Boiler, *Thermal Science*, 24 (2020), 5, pp. 2717-2728
- [23] Dostiyarov, A. M., et al., G. Daukeyev AUPET University, Assignee, Micro Modular Air Nozzle, KZ patent 34790, 2019, 14
- [24] Sadykova, S. B., *et al.*, Influence of Turbulence on the Efficiency and Reliability of Combustion Chamber of the Gas Turbine, *Thermal Science*, 25 (2021), 6A, pp. 4321-4332
- [25] ***, ANSYS Fluent Theory Guide, ANSYS, Inc., 275 Technology Drive Canonsburg, PA 15317, November 2013
- [26] Elattar, H. F., et al., CFD Simulation of Confined Non-Premixed Jet Flames in Rotary Kilns for Gaseous Fuels, Computers & Fluids, 102 (2014), Oct., pp. 62-73
- [27] Wenming, Y., *et al.*, Combustion Process and Entropy Generation in a Novel Microcombustor with a Block Insert, *Chemical Engineering Journal*, 274 (2015), Aug., pp. 231-237
- [28] Yiheng, T., et al., Experimental and Numerical Study on Bluff-Body and Swirl Stabilized Diffusion Flames, Fuel, 217 (2018), Apr., pp. 352-364

Paper submitted: April 10, 2021 Paper revised: July 28, 2021 Paper accepted: July 29, 2021