MICRO-MODULAR AIR DRIVEN COMBUSTION NOZZLE: EXPERIMENTAL AND NUMERICAL MODELLING STUDIES TOWARDS OPTIMAL GEOMETRIC DESIGN

***Abay M. DOSTIYAROV*[1], *Samal B. SADYKOVA*[*2]**

[1] Gumarbek Daukeev Almaty University of Power Engineering and Telecommunications, 050013
Baytursynuly, 126/1, Almaty, Kazakhstan, https://orcid.org/0000-0002-7289-8986

[2] Gumarbek Daukeev Almaty University of Power Engineering and Telecommunications, 050013
Baytursynuly, 126/1, Almaty, Kazakhstan, https://orcid.org/0000-0001-8250-8614.

[*] Corresponding author; E-mail: kundyzdy@mail.ru

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 NOx 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 degree of the outlet vortex generator has been able to assure optimal flame length and combustion efficiency.

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

1. 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 NOx 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 (LPP) and Rich Burn - Quick Quench - Lean Burn (RQL) premixed stream allows reduction of NOx 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.

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.

Table 1. Studies on premixed combustion of gaseous fuels

Reference	Methods	Object	Problem studied
Bulat G. et al [9]	Numerical study	Premixed combustion of methane in a annular Combustion chamber and channel	Emissions of CO and NOx
Jerzak W. [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 Y et al. [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 M. 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 J. et al. [13]	Experimental study	Premixed combustion of lean methane in vortex chamber	Determination of the reasons leading to flame
Lin X. et al. [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 et al. [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 B. et al. [16]	Numerical study	Structure and flow characteristics of propane-air mixture over a stabilized bluff-body	Combustion and limits of flame stability
Chen Zh.X. et al. [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 S. et al. [18]	Experimental study	A nozzle for combustion of low caloric fuel	Determination of the operating regime relevant to stable combustion process
Enagi I.I. et al. [19]	Numerical study	Development of a combustion chamber for a micro-gas turbine	Numerical study towards optimization of combustion chamber geometry
Yilmaz H. 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 A. et al.[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 W.-S. et al.[22]	Numerical study	Combustion of fuel in tangentially boiler	Mechanism of NOx formation is being studied and the factors affecting their emission are studied too.

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 NOx. The study presents two main directions in pursuing this task: 1) Combustion experiment studies on a physical model using results obtained by isothermal modelling (see reports in [24]), and 2) Numerical modelling by CFD of the same physical model oriented to geometry optimization and rational fuel supply.

2. Study approaches

2.1. Experimental

The micromodular 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 outlet. The swirler parameters are: 8 blades of 1.5 mm in length and 12 mm width; the angle of the inlet vortex blades is β_1=40°, while for the outlet swirler it is β_2=30°. 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°.

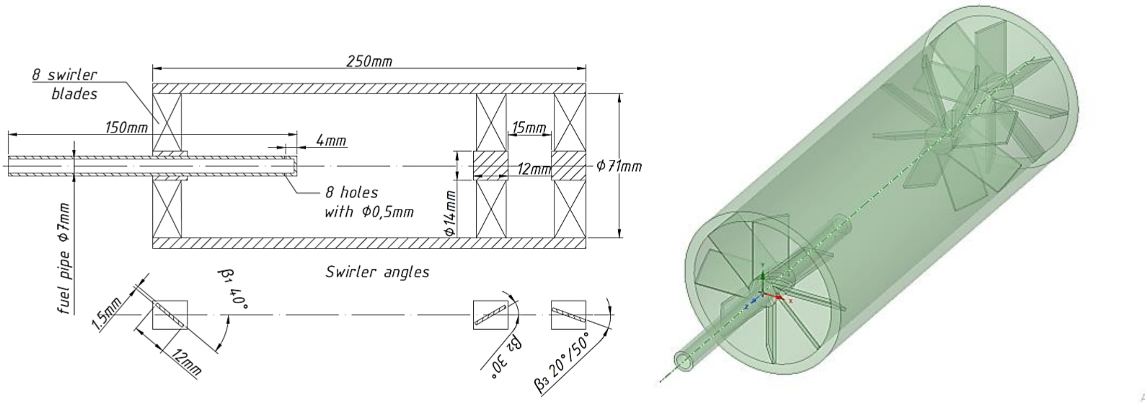


Figure. 1. Schematic presentation of the micromodular air driven combustion nozzle

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 there high turbulence intensity exists [24]. One caliper is a length that is equal to the inner diameter of the micro-modular nozzle, where the gas-air mixture flows.

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 NOx in the flue gases was determined with gas analyzer Testo 350.

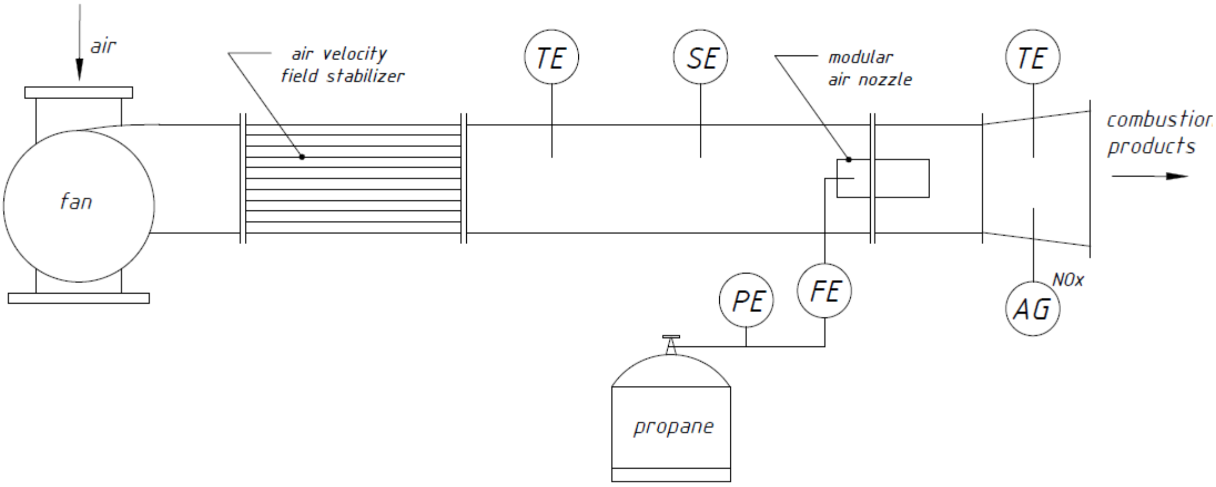


Figure. 2. Experimental set up

TE -thermocouple chromel-alumel type k complete with device OVEN TRM-138, PE – manometer complete with BPO-5-3 reducer, SE – anemometer Testo 416, FE – fuel flow meter BK-G4, AG $^{(NOx)}$ – gas analyzer Testo 350

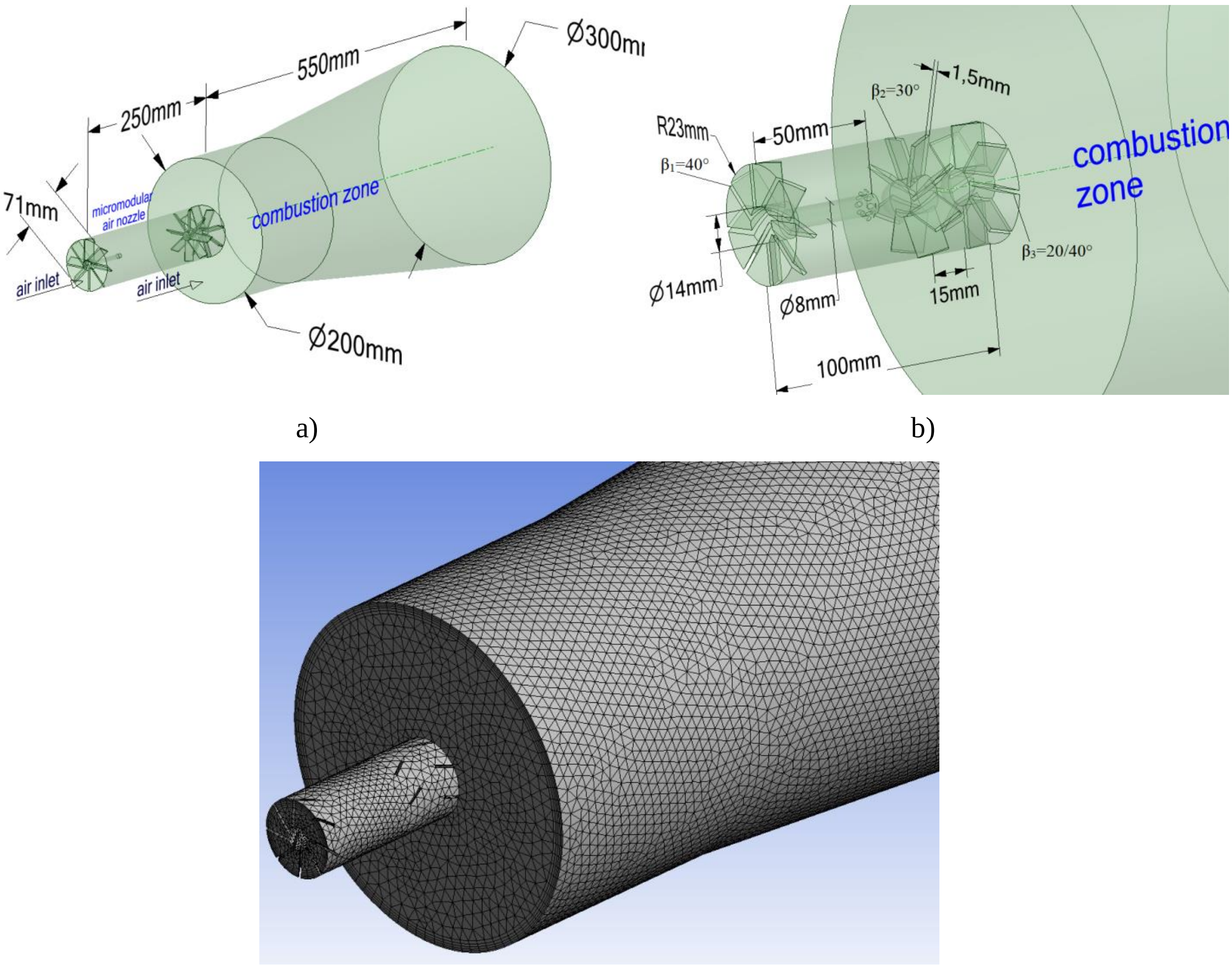
The following values were measured during the experiment:

- 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;
- 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 ϕ=0,185. Combustion temperatures and NOx emissions are given below in the section 3 Results.

2.2. Numerical modelling

The numerical modelling and verifications were carried out with CFD utilizing ANSYS 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.3a correspond to the physical model studied (see 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.



a) b)

c)

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

The first step in numerical modeling is mesh creation. The study modul area was divided into a set of unstructured tetrahedral small volumes (Fig. 3c) in the Meshing preprocessor of the Ansys software package. Mesh characteristic: ~ 555101 elements, ~ 155839 nodes, orthogonal quality - 0.8, skewness - 0.2, aspect ratio - 2.9.

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

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

momentum-conservation equation

$$\frac{\partial}{dt}(\rho\vec{v}) + \nabla \cdot (\rho\vec{v}\vec{v}) = -\nabla p + \nabla \cdot (\bar{\bar{\tau}}) + \rho\vec{g} \tag{2}$$

where ρ - density, $\vec{v}$ - velocity vector, p- static pressure, $\bar{\bar{\tau}}$ - stress tensor, $\rho\vec{g}$- gravitational body force.

The stress tensor $\bar{\bar{\tau}}$ is

$$\bar{\bar{\tau}} = \mu\left[(\nabla\vec{v} + \nabla\vec{v}^T) - \frac{2}{3}\nabla\cdot\vec{v}I\right] \tag{3}$$

where μ - molecular viscosity, I- 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.:

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

where ϕ is a scalar quantity, Γ is 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 equation (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 (PDF) 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, m/s	11	Mass flow rate, kg/s	2.16	Backflow temperature, K	288
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 NOx emissions. Partial-equilibrium approach was chosen for the formation of O and OH radicals.

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

3. 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. 3b. 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.

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 β_3=20°. Moreover, when β_3=50°, in the same cross-section, temperature profile is almost flat (and symmetrically distributed) in a wide range located lower than in the case with β_3=20°. This can be attributed to the fact that when β_3=50° 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.

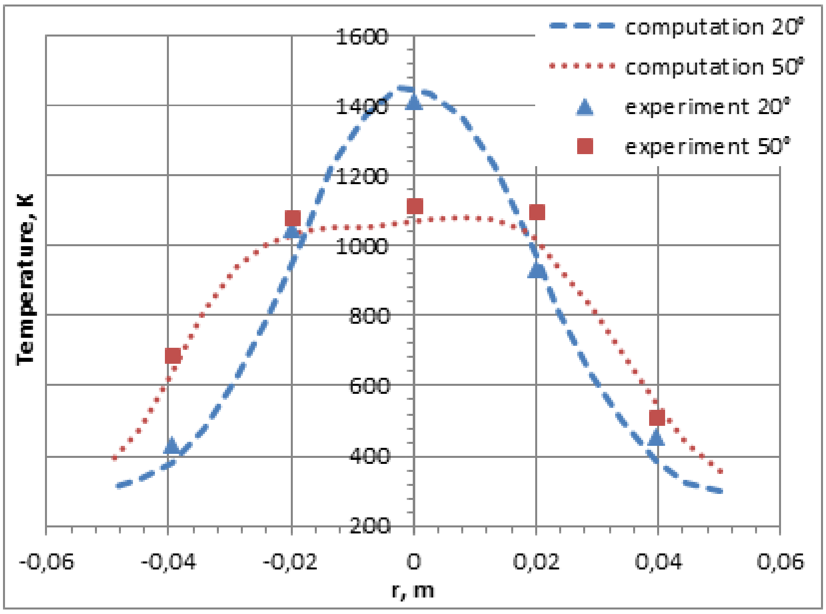


Figure. 4. Temperature profiles: Experimental and simulated

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.

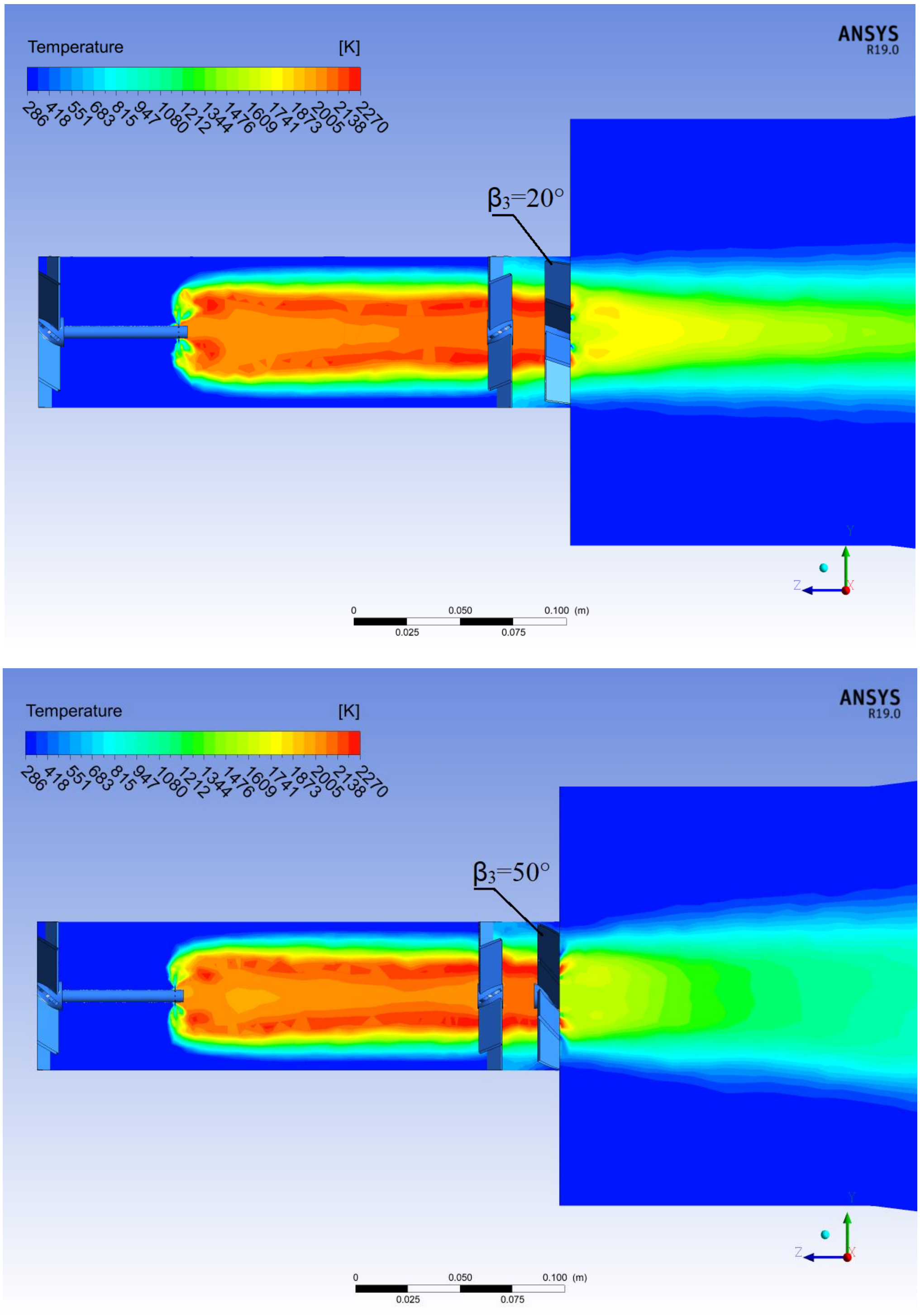


Figure. 5. Temperature profiles as a result of the numerical modelling of the 1st model

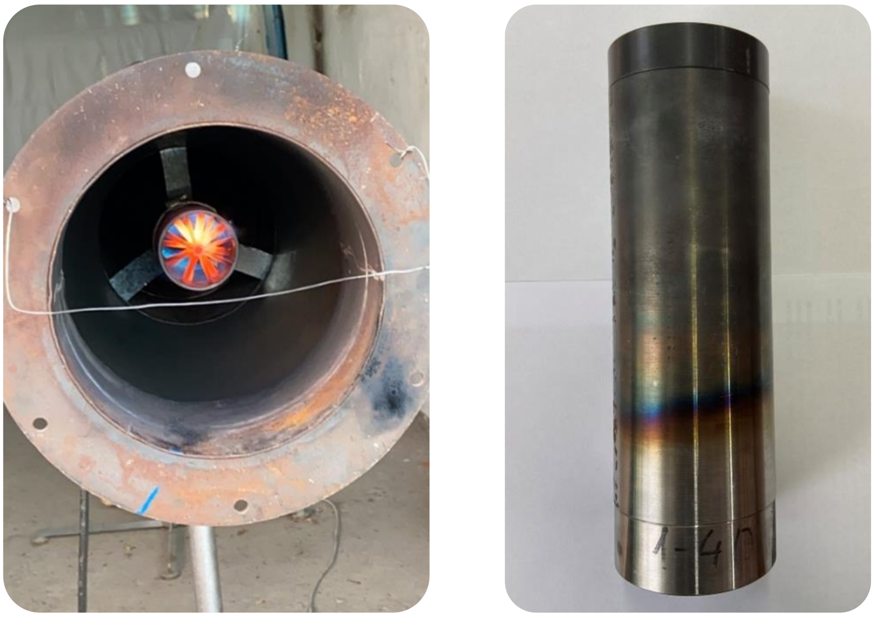


Figure 6. Photos of the studied micro-modular nozzle used in the experiments. Left: Front view a with vortex generator at the central axis. Right: Vertical view

The comparative analysis of the experimental results and the outcomes of the numerical modelling with respect to the NOx 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 β_3=20° the NOx concentration is greater than when β_3=50°. The main reason explaining this can be attributed to the fact that at β_3=50° the common temperature at the nozzle exist is lower than when β_3=20° (see Fig.5).

Table 3. NOx emission in case of 1st model

NOx ppm (O_2 15%)	β_3=20°	β_3=50°
Experiments	20.5	10.9
Numerical modeling	5.8	5.5

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 β_3=50° and β_3=20°) 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].

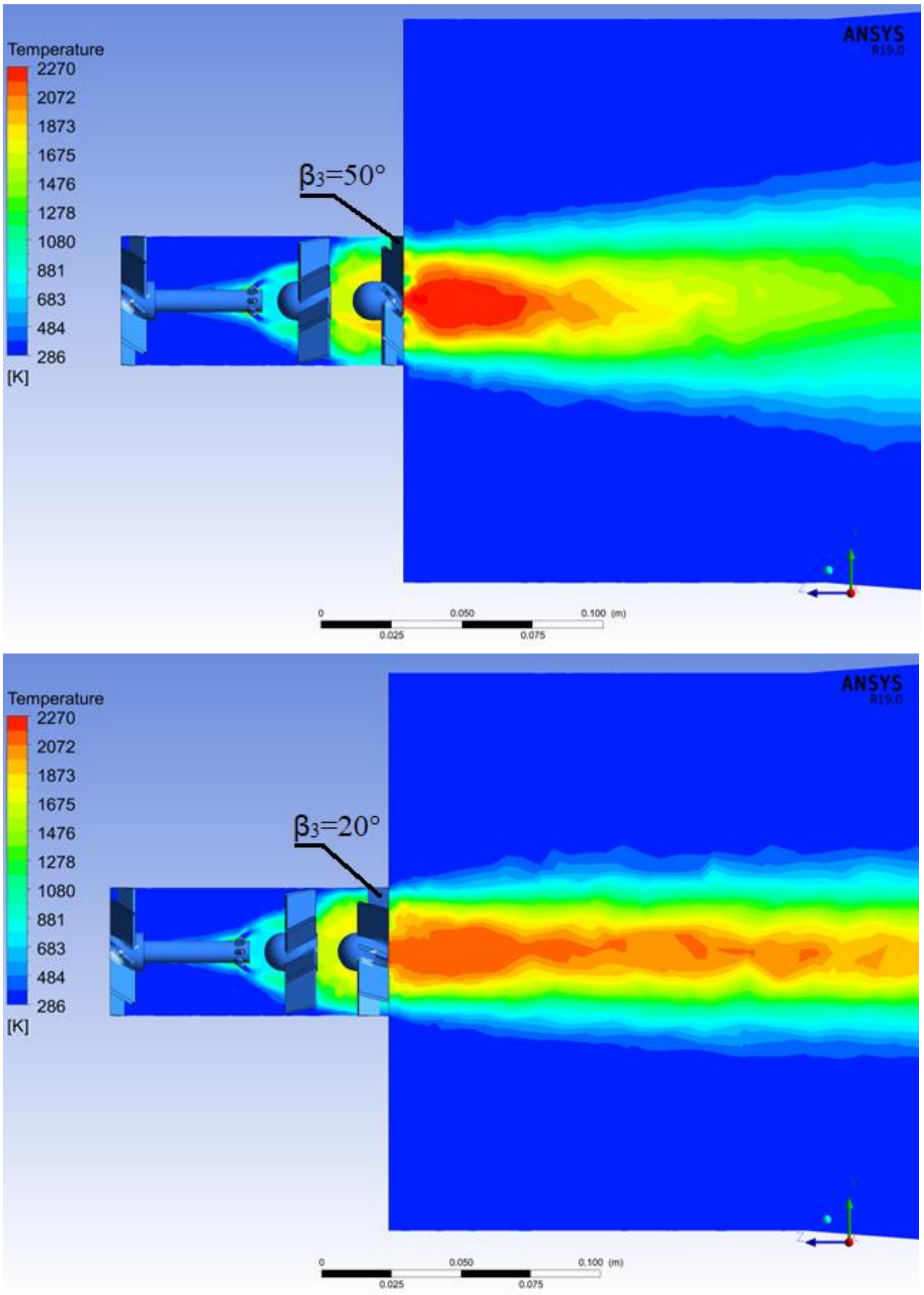


Figure. 7. Temperature profiles as outcomes of the numerical modelling with 2nd model

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 β_3=50° there is high temperature elongated zone, while with β_3=20° the flame is longer but relatively colder: its temperature is about 300°C lower that at the same location but with β_3=50° and this can be explained by the fact that with β_3=50° the vortex flow structure at the outlet of the nozzle is stronger than when β_3=20° is used. The stronger vortex structure creates reverse flows towards the central combustion zone and they concentrate themselves at the nozzle exist. However, with β_3=20° 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 NOx emissions, on the other hand the extended flame zone also increase the NOx 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 NOx emissions. The numerical modelling reveals that the emissions of NOx (with 15% volumetric oxygen concentration) at the control position are 5.1 and 6.3 ppm for β_3 50° 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 β_3=50°, which actually means that the flame is inside the nozzle. With β_3=20° 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.

4. 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 NOx emissions. In all cases studied the blade angle 50° allows to reduce the NOx 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 β_3=20° 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 NOx emissions. The results of the research form a good scientific basis for future development of combustion chambers with micro-modular combustion nozzles.

References

[1] Health and environmental impacts of electricity generation systems: procedures for comparative assessment, International Atomic Energy Agency, Vienna, Austria, 1999, 204 p. (Technical reports series, ISSN 0074-1914)

[2] Bharvirkar, R, Dallas, B, & Alan, K., NO 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, 618 p.

[4] Lefebvre, A.H., Ballal, D.R., Gas Turbine Combustion: Alternative Fuels and Emissions, 3rd ed., CRC Press, 2010, 537 p.

[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), pp.90-96. https://doi.org/10.3103/S1068799817010135

[6] Benim, A.C., Syed, Kh. J., Concepts Related to Combustion and Flow in Premix Burners, in: *Flashback Mechanisms in Lean Premixed Gas Turbine Combustion*, Academic Press, 2015, 134 p, pp. 5-18. https://doi.org/10.1016/B978-0-12-800755-6.00002-7

[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, 2013, pp. 635-684 https://doi.org/10.1533/9780857096067.3.635

[8] Baubek, A., Atyaksheva, A., Zhumagulov, M., Kartjanov, N., Plotnikova, I., Chicherina, N., Complex Studies of the Innovative Vortex Burner Device with Optimization of Design., *Studies in Systems, Decision and Control, 351* (2021), pp. 139 – 153. https://doi.org/10.1007/978-3-030-68103-6_13

[9] Bulat, G., Jones, W., Marquis, A., 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. https://doi.org/10.1016/j.combustflame.2013.12.028

[10] Jerzak, W., Kuz´nia, 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), pp. 46-54. https://doi.org/10.1016/j.jngse.2015.12.054

[11] Wu, Y., Carlsson, C., Szasz, R., Peng, L., Fuchs, L., Bai, X.S., Effect of geometrical contraction on vortex breakdown of swirling turbulent flow in a model combustor, *Fuel, 170* (2016), pp. 210-225. https://doi.org/10.1016/j.fuel.2015.12.035

[12] Hossein, M., Moghaddam, S., Moghaddam, M.S., Khorramdel, M., Numerical study of geometric parameters effecting temperature and thermal efficiency in a premix multi-hole flat flame burner, *Energy, 125* (2017), pp. 654-662. https://doi.org/10.1016/j.energy.2017.02.116

[13] Runyon, J., Marsh, R., Bowen, Ph., Pugh, D., Giles, A., Morris, S., Lean methane flame stability in a premixed generic swirl burner: Isothermal flow and atmospheric combustion characterization, *Experimental Thermal and Fluid Science, 92* (2018), pp. 125-140. https://doi.org/10.1016/j.expthermflusci.2017.11.019

[14] Lin, X., Ma, H., Liu, Ch., Zhang, J., Zhang, Y., Miao, Zh., Experimental research on gas interchangeability indices for domestic fully premixed burners, *Fuel, 233* (2018), pp. 695-703. https://doi.org/10.1016/j.fuel.2018.06.097

[15] Zhao, Tiezheng, Liu, Xiao, Yang, Jialong, Zheng, Hongtao, Investigation on effect of mixing distance on mixing process and combustion characteristics of double-cone burner, *Fuel, 253* (2019), pp. 540-551. https://doi.org/10.1016/j.fuel.2019.05.043

[16] Wu, B., Zhao, X., Chowdhury, B.R., Cetegen, B.M., Xu, Ch., Lu, T., A numerical investigation of the flame structure and blowoff characteristics of a bluff-body stabilized turbulent premixed flame, *Combustion and Flame, 202* (2019), pp. 376-393. https://doi.org/10.1016/j.combustflame.2019.01.026

[17] Chen, Zh.X., Swaminathan, N., Stöhr, M., Meier, W., 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. https://doi.org/10.1016/j.proci.2018.08.042

[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), pp. 586-598. https://doi.org/10.1016/j.fuel.2018.02.052

[19] Enagi, I.I., Al-attab, K.A., Zainal, Z.A., Combustion chamber design and performance for micro gas turbine application, *Fuel Processing Technology*, *166* (2017), pp. 258-268. https://doi.org/10.1016/j.fuproc.2017.05.037

[20] Yilmaz, H., Cam, O., Yilmaz, I., Experimental investigation of flame instability in a premixed combustor, *Fuel*, *262* (2020), 116594. https://doi.org/10.1016/j.fuel.2019.116594

[21] Rowhani, A., Tabejamaat, S., Experimental study of the effects of swirl and air dilution on biogas non-premixed flame stability, *Thermal Science*, *19(6)* (2015), pp. 2161 – 2169. https://doi.org/10.2298/TSCI130112157R

[22] Wang, W.S., Huang, Z.H., Tian, M., Wang, J.H., Shangguan, S.S., Yao, M.Y., Numerical simulation of NOx emission characteristics during combustion in 350 MW supercritical cogeneration tangentially boiler, *Thermal Science*, *24* (2020), pp. 2717 – 2728. https://doi.org/10.2298/TSCI191010006W

[23] Dostiyarov, A.M., Sadykova, S.B., Kartjanov, N.R., inventors; G. Daukeyev AUPET university, assignee. Micro modular air nozzle. KZ patent 34790. 2019 Nov 14.

[24] Sadykova, S.B., Dostiyarov, A.M., Zhumagulov, M.G., Kartjanov, N.R., The influence of turbulence on the efficiency and reliability of combustion chamber of the gas turbine, *Thermal Science Journal*, (2021). https://doi.org/10.2298/TSCI200831064S

[25] ANSYS Fluent Theory Guide, ANSYS, Inc., 275 Technology Drive Canonsburg, PA 15317, November 2013.

[26] Elattar, H.F., Stanev, R., Specht, E., Fouda A., CFD simulation of confined non-premixed jet flames in rotary kilns for gaseous fuels, *Computers & Fluids*, *102* (2014), pp. 62–73. https://doi.org/10.1016/j.compfluid.2014.05.033

[27] Wenming, Y., Dongyue, J., Kenny, C.K.Y., Dan, Z., Jianfeng, P., Combustion process and entropy generation in a novel microcombustor with a block insert, *Chemical Engineering Journal*, *274* (2015), pp. 231-237. https://doi.org/10.1016/j.cej.2015.04.034

[28] Yiheng Tong, Xiao Liu, Zhenkan Wang, Mattias Richter, Jens Klingmann., Experimental and numerical study on bluff-body and swirl stabilized diffusion flames, *Fuel*, 217(2018), pp. 352-364. https://doi.org/10.1016/j.fuel.2017.12.061

Submitted: 10.04.2021.

Revised: 28.07.2021.

Accepted: 29.07.2021.