Analysis of a cross-flow influence for a turbulence progress in an initial segment of axisymmetric duct

Wojciech Judt[1*], and *Jarosław* Bartoszewicz[1]

[1]Poznan University of Technology, Chair of Thermal Engineering, Piotrowo 3 Street, 60-965 Poznań, Poland

Abstract. Influence for a nondisturbed stream by cross-flow is present in many technical applications. In part of applications, this phenomenon is caused intentionally, but in others, it is a side-effect. Flow through the closed duct is one of the most frequently realized types of flow in thermal devices. The paper shows an influence of cross-flow for a laminar stream, which is flowing through closed, round duct. The considered case was calculated in numerical calculations and analyzed during experimental research. Cross-flow stream was generated by four circular nozzles located on the duct walls in some distance from the duct inlet, which was distributed along the circumference of analyzed construction in the right angle to the mainstream. Experimental research was realized by a constant temperature anemometry measurement method. A numerical model of a fluid flow was created in ANSYS Fluent software. Unsteady numerical calculations of a cross-flow influence for a mainstream was realized with the Detached Delayed Eddy Dissipation model of turbulence application. Realized research established how a cross stream can turbulent a laminar flow in a closed, circular duct. Authors showed velocity distribution in the main duct after cross-flow influence and demonstrated, that this method allows ascending turbulence of a flow in external parts of fluid flow.

Keywords: axisymmetrical duct, closed duct, cross-flow, turbulent intensity, Ansys Fluent

1 Introduction

Fluid flow through a closed duct is commonly used during work of different types of technical devices. They can be used for a fluid transportation. Also, this type of a flow is present in combustion engines, because of the wide application of closed circular ducts or circular chambers in this type of technical devices [1]. The paper presents results of a research connected with a fluid flow inside a closed circular duct. The main stream of a fluid was disturbed by a cross-flow. This specific type of a flow is present in different applications. Influence of a cross flow on the main stream is present for a delivering air into combustion

---

* Corresponding author: wojciech.judt@put.poznan.pl

process inside retort burners during solid fuels combustion [2] and for gas burners, where similarly a secondary air is delivered [3, 4].

During realized research, an additional cross-flow stream was provided into a duct by transversal nozzles. Authors of paper analyzed experimental data obtained from experimental research [5] and compared with numerical calculations, which were prepared by the main author. The purpose of the performed research was a detection of the velocity distribution of the main flow after the influence of additional, cross-stream. Authors analyzed what quantity of additional cross-flow can increase fluctuations inside the main flow and what is an area of influence.

2 Experimental analysis

Experimental research was prepared on a test stand, which is presented in Figure 1. The test stand is built from a circular pipe with a 50 mm diameter and a 2.5 meters length. The measurement part of test stand has 2-meter length. Measurements were realized in holes distributed along measurements part of the test stand at a distance of every 100 mm. The additional stream was supplied to the duct by four round nozzles with a diameter equal to 2 or 4 mm. These nozzles were connected to the wall of the main duct at the right angle to the axis of the duct. Nozzles were distributed along to circumference of the pipe. During research, an additional stream of fluid was injected into duct only by one type of applied nozzles.
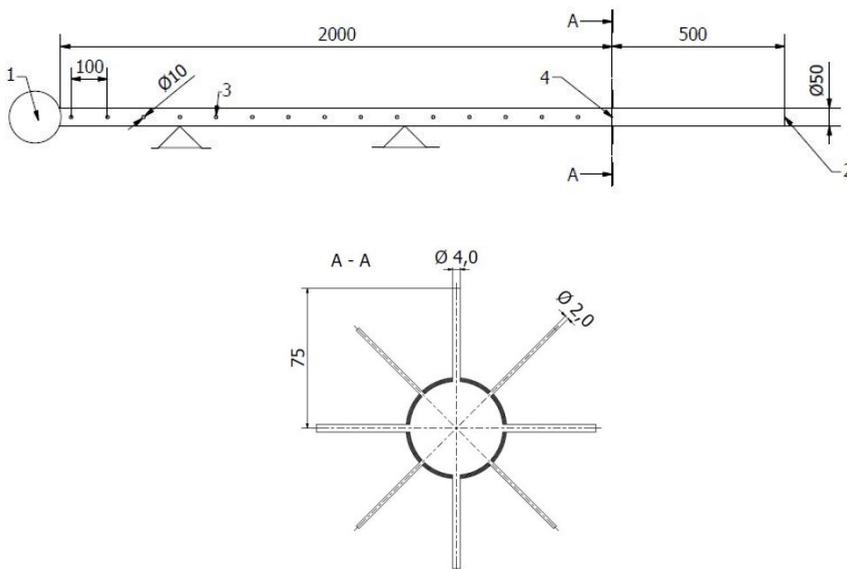

Fig. 1. Test stand used during experimental analysis; 1 – fan, 2 – air inlet, 3 – location of measurement points, 4 – additional stream injection point

During the research a constant volume flow of air injected through nozzles was used. The volume flow of air delivering through a single nozzle was equal to $1.1 \cdot 10^{-4}$ m^3/s. Also, a volume flow of air, which was transported through the main duct was constant during research. Average velocity flow of gas at the inlet to the main duct was equal to 21.5 m/s. This value is a result of the power of applied fan for experimental research. Measurement of average gas flow in the duct and fluctuations of fluid flow was realized by using a constant temperature anemometer IFA-300. This measuring device allows measuring an average flow velocity and fluctuations of velocity in the obtained flow.

Figures 2-4 presents average velocity distribution and obtained fluctuations of a flow for during experimental research. Figure 2 presents collected data for confused flow by using a 2 mm nozzles. Figure 3 collects information for a case with 4 mm nozzles using. Figure 4 presents researched distributions for undisturbed flow, which can be a comparison with previous cases.
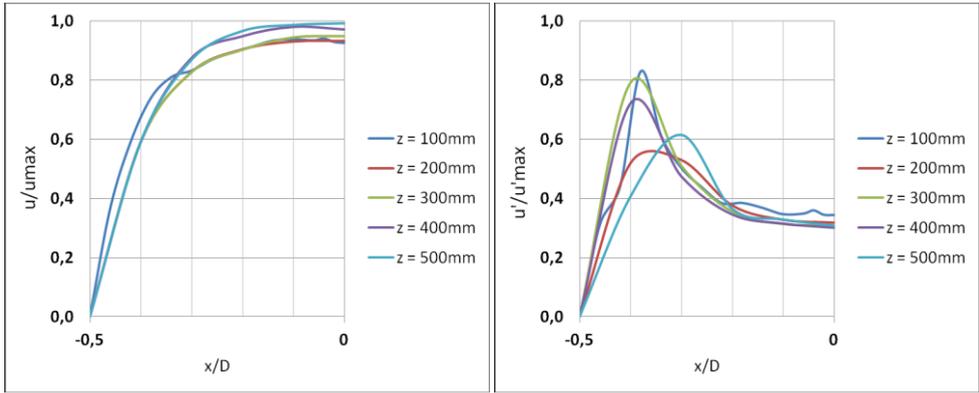

Fig. 2. Velocity magnitude and velocity fluctuations distribution for the case, where 2 mm nozzles were implemented (based on [5])
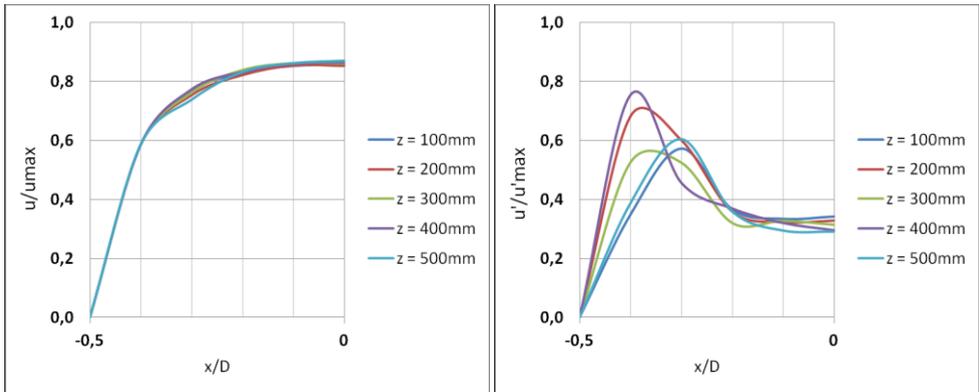

Fig. 3. Velocity magnitude and velocity fluctuations distribution for the case, where 4 mm nozzles were implemented (based on [5])
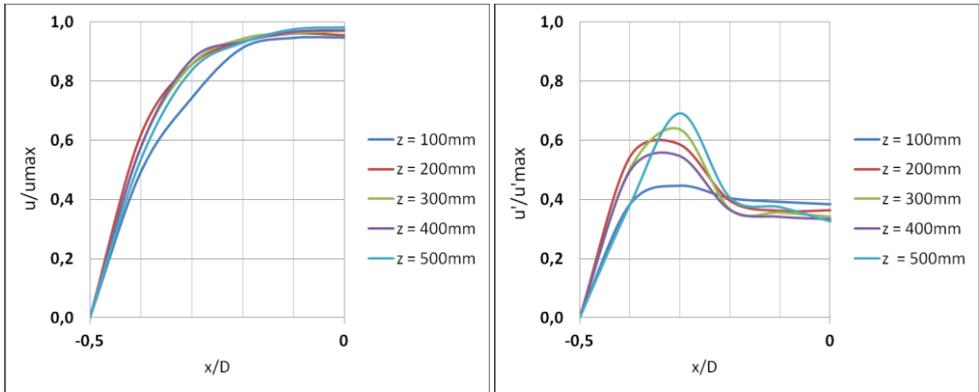

Fig. 4. Velocity magnitude and velocity fluctuations distribution for undisturbed flow (based on [5])

Results from the experimental research were presented in relation with maximum obtained velocity magnitude and velocity fluctuation for analyzed cases. Location of measurement points is related to the diameter of the duct. Relation $x/D = 0$ mean location of the axis of the analyzed duct. Position $x/D = $ -0.5 is located on the external wall of the duct. Dimension Z describes the distance between the plane of injection additional stream into duct and planes where measurements were realized. Measurements were realized perpendicularly to the axis of the duct along to his diameter.

Obtained results related to velocity magnitude shows, that additional cross-flow application to the duct has no crucial consequence for a velocity magnitude in the core of a flow. Profile of velocity distribution obtained for prepared research is similar for each of analyzed case. Obtained differences are better visible during analysis of fluctuation distributions in the analyzed flow. Application of additional nozzles with smaller dimension causes, that injected additional stream has higher velocity. It causes, that influence for a flow by the additional stream for velocity profile is more visible, than in cases where injection of the additional stream was realized by bigger nozzles. Values of fluctuations are higher for cases, where the additional stream was implemented.

3 Numerical calculations

Numerical analysis was created by using the ANSYS Fluent software. Numerical calculations were prepared for the same flow parameters and geometrical dimensions of the duct as in experimental research. Numerical analysis can be divided into two steps. First of all steady-state calculations were prepared by using k-Omega SST model of turbulence. This step allows obtaining averaged results from Reynolds-averaged Navier-Stokes (RANS) equation solving. Owing to the fact was possible to obtain results of average velocity magnitude and energy of turbulence k, which can be compared with experimental results. Value k, which is calculated during a model of turbulence equations solving is described as:

$$k = \tfrac{1}{2}\left(\overline{(u')^2} + \overline{(v')^2} + \overline{(w')^2}\right) \tag{1}$$

This parameter can be quantified by the mean of the turbulence normal stresses in Cartesian coordinates. One of the elements owing to the fact in this equation *(u')* was measured during experimental research, so this parameter describes the wider intensity of turbulence, that value which was measured during experimental research. Results obtained from RANS calculations are showed on Figures 5-7.
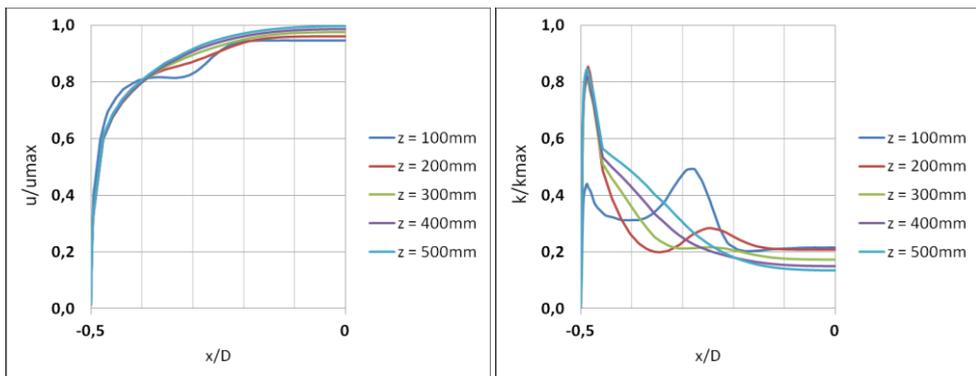

Fig. 5. Velocity magnitude and velocity fluctuations distribution for the case, where 2 mm nozzles were implemented
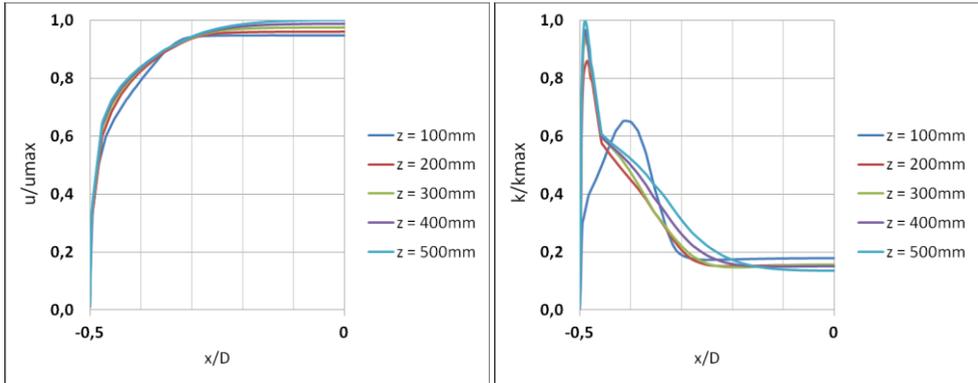
Fig. 6. Velocity magnitude and velocity fluctuations distribution for the case, where 4 mm nozzles were implemented
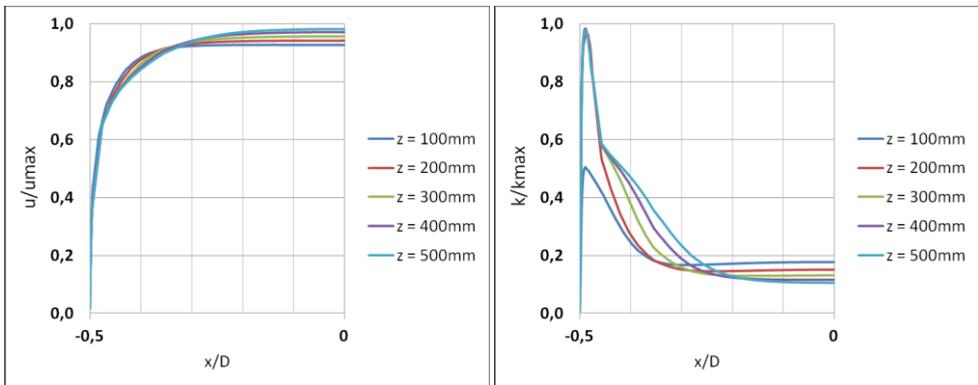

Fig. 7. Velocity magnitude and velocity fluctuations distribution for undisturbed flow

Velocity profile, which was obtained during numerical calculations for undisturbed flow is commonly occurring for flow through the closed duct. On distributions obtained for flow, were an additional stream was occurring, differences of velocity distribution are visible only for measurement realized on planes located near to the additional stream injection point. Analysis of fluctuations of velocity obtained for that step of research shows, that additional stream has influence for turbulence intensity in the analyzed flow. It is mostly visible for measurement planes located 100 and 200 millimeters from injection place. After that distance effect of the additional stream for the main flow is not noticeable. Influence of additional stream is more visible for the case, where smaller nozzles were used. Area of additional stream influence on fluctuation profile for first two measurement planes for smaller nozzles is present mostly at *x/D* equal to -0.25. It means that the biggest increase in fluctuations is occurring in the middle distance between the axis of the duct and his wall. For case, where an additional stream was injected by bigger nozzles influence for velocity distribution and for fluctuations of velocity are smaller than for earlier discussed case. Influence on a velocity profile is a much less visible for that case. Increasing of fluctuation of a velocity are occurring at *x/D* equal to -0.4 and is only visible for the first measurement plane.

The second step of realized calculations relied on obtaining places, where the generation of eddies is occurring. This information can get from the implementation of hybrid calculations, where a part of calculations is realized by a solution of Large Eddy Simulation (LES) model. It is realized for a part of a domain, where solver detects the possibility of eddies generation in a flow. This type of calculations is based on calculating the unsteady parameters of the analyzed model. This model of numerical calculations allows extending

obtained information from RANS solution into additional data, which was averaged during Reynolds-averaged Navier-Stokes calculations. This stage of calculation was realized with a Delayed Detached Eddy Simulation (DDES) model of turbulence. This approach to the issue allows designating areas of increased turbulence inside of a flow.

Figure 8 presents a distribution of a velocity magnitude at the initial segment for the analyzed channel in the area of additional stream injection. This figure presents obtained results of unsteady calculations between injection point and first measurement plane location, which was located on distance equal to 100mm from nozzles location. The first picture presents a velocity distribution in the duct for the undisturbed flow. The second picture presents obtained solution for the case, where the additional stream was injected into the duct by nozzles with a diameter equal to 2 millimetres.
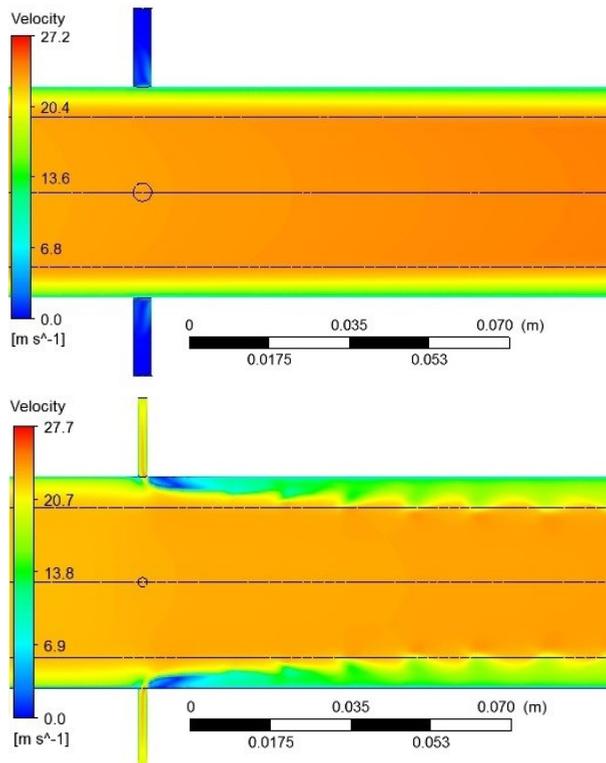

Fig. 8. Velocity magnitude and velocity fluctuations distribution for undisturbed flow

When the additional stream is injected into the duct, it moved to increase fluctuations of a flow, which was obtained in a volume of fluid located close to the wall of the duct. Eddies are expanding gradually along to the in axis direction but the most amount of eddies was generated in distance equal to about 0.8 radius from the channel axis. This phenomenon is present when a velocity of cross-flow is near to the velocity of the main flow in the duct. This situation is occurring only for a smaller nozzles application into the analyzed channel. Injection of the additional stream by bigger nozzles does not cause the creation of eddies inside of a flow. When the fluid is moving to the further sections of the duct, a turbulence of flow decreases and decisively disappear. It is caused because of duct wall has to influence on the stabilization of the fluid flow, and character of a flow returns to the laminar in a whole volume of the duct.

Figure 9 shows a velocity distribution of fluid flow gained during numerical analysis in a cross-section of the duct, which was located 100 mm after the injection point of additional stream location. Presented results are also an effect of unsteady calculations. The cross-flow influence on the main stream in this cross-section is visible in two different ways. The first way is connected with the occurrence of classical velocity profile disorder for closed ducts. It is detectable in a radial velocity increasing. This phenomenon is responsible for a local increase of turbulence in a flow. The second effect of cross-flow influence is increasing the velocity of a flow around external walls of the main channel. It causes a change in a velocity profile of the main stream. Numerical analysis revealed no influence of stream turbulence in a whole cross-section of the analyzed duct.
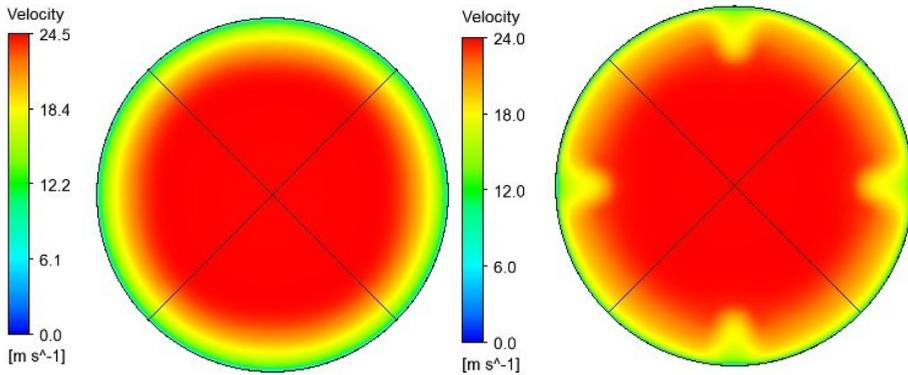

Fig. 9. Velocity magnitude and velocity fluctuations distribution for undisturbed flow

4 Conclusions

Realized experimental and numerical analysis confirmed the possibility of turbulence increasing in a flow by a cross-flow stream. Cross-flow implementation into a laminar stream can increase the intensity of turbulence in a flow, but only in external parts of the fluid, which are neighboring with walls of the closed channel. It is possible when a cross-flow velocity is close to the velocity of main stream. This method allows getting local increasing of turbulence in the flow on the circumference of the duct, with a maximum of turbulence in distance of $x/D = -0.3$ from the axis duct. Generating a turbulent flow in the whole cross-section of the duct requires to differentiate localization of nozzles on the duct wall. The second possibility of increasing a penetration by the cross-flow to the axis duct is application a different velocity flow from the succeeding nozzles. Comparison of results from experimental and numerical research confirms the propriety of realized analysis.

Injection of additional stream directed in a different direction than main flow can increase the energy of turbulence obtained inside of a flow. This parameter translates to increase a heat transfer coefficient for devices which are using a heat transfer phenomenon in his work. It can be used as an alternative to geometrical elements, which improving turbulence intensity inside a flow in places where implementation of that type of elements will be difficult [6]. This situation occurring for example in low power heating boilers for solid fuels [7]. Combustion of solid fuels generates a much amount of soot, which is clinging to boiler walls. Implementation of geometrical elements which are increasing turbulence intensity causes, that this elements easily will catch generated soot in the combustion process. Application of cross-flow in this type of heating devices will increase a turbulence intensity and help with cleaning internal walls of heating boilers from soot.

References

1. W. Bueschke M. Skowron, K. Wisłocki, (2017) *Investigations on gas-air mixture formation in the ignition chamber of two-stage combustion chamber using high-speed Schlieren imaging*, MATEC Web of Conf., vol. **118**, no. **12**, 1-6, [2017]

2. J. Bartoszewicz, A. Nygard, R. Urbaniak, *Characteristics of fans used in low-power boilers*, J. of Power Technologies, vol. 97, iss. 1, 69-74, [2017]

3. J. Jójka, R. Ślefarski, *Dimensionally reduced modeling of nitric oxide formation for premixed methane-air flames with ammonia content*. Fuel, vol. 217, 98-105, [2018]

4. D. Szewczyk, R. Slefarski, R. Jankowski, *Analysis of the combustion process of syngas fuels containing high hydrocarbons and nitrogen compounds in Zonal Volumetric Combustion technology*, Energy, vol. 121, 716-725, [2017]

5. M. Werner, *Badania systemu turbulizacji w kanale zamkniętym*, master thesis, Poznan University of Technology, [2012]

6. K.M. Lunsford, *Increasing Heat Exchanger Performance*, Hydrocarbon Eng., 1-13, [1998]

7. R. Urbaniak, J. Bartoszewicz, W. Judt, *Analysis of the possibilities of application of numerical methods in the improvement of the operating efficiency of low-power boilers*, Heat Trans. Res., vol. 49, no. 7, 675-683, [2018]