**ISSN**: 1231-4005
**e-ISSN:** 2354-0133
**DOI:** 10.5604/01.3001.0012.2466

# ANALYSIS OF POSSIBILITIES OF INCREASING A TURBULENCE BY THROTTLED AXISYMMETRIC STREAM FOR A FLUID FLOW INSIDE CLOSED DUCT

**Wojciech Judt, Jarosław Bartoszewicz**

*Poznan University of Technology, Chair of Thermal Engineering*
*Piotrowo Street 3, 60-965 Poznan, Poland*
*tel.: +48 61 6652209, +48 61 6652215, fax: +48 61 6652281*
*e-mail: wojciech.judt@put.poznan.pl, jaroslaw.bartoszewicz@put.poznan.pl*

***Abstract***

*Fluid flow through the closed duct is a common phenomenon, which is used in many technical applications. A stream of a fluid is often disturbed by different shapes of elements, which have an effect for a flow and causes the growth of a turbulence. Turbulence rising causes an increase of heat transfer process and eliminates areas of flow stagnation in thermal devices. Article presents methodology and results of research about possibilities of turbulence increasing for a fluid flow inside a closed duct. Authors analysed capabilities of application of an internal stream of a fluid in the axis of the tested duct, in the inlet to analysed construction for mainstream turbulence expanding. An additional stream of a fluid was added into the model by internal, partially throttled pipe, which was clogged by special disc. Contact between the disc and internal pipe generated a small gap, where an additional stream of a fluid was directed. Numerical analysis of a level of turbulence for a fluid flow through analysed duct was realized in ANSYS Fluent environment, as an unsteady simulation with Delayed Detached Eddy Dissipation model of turbulence. Experimental research was realized with constant anemometry measurement method. Results of experimental and numerical analysis show, what part of a fluid inside duct disturbed mainstream of a flow.*

***Keywords:*** *closed duct, turbulence increasing, numerical calculations, fluent*

## 1. Introduction

Usually, designers of devices, which have a contact with fluids, strive to decrease a turbulence of a flow for a flow neighbouring to the device during work. The high amount of turbulence causes the occurrence of energy loss due to vortices formation inside the flow. It also causes an increase in aerodynamic drag. This phenomenon is not desirable for example in aircraft or automotive branch.

Although a high, value of turbulence in a flow has some positive effects. Increasing of turbulence for a flow is applied in situations when the analysed device takes part in a heat transfer process [3]. Turbulence increasing causes maximizing of a heat transfer process for devices, which has an association with fluids. Increasing of heat transfer coefficient allows for a reduction of the area of heat transfer and can improve the efficiency of the realized heat transfer process. Turbulent flow is a very important scientific topic, which was analysed by many authors [2].

Authors of article analysed the possibility of increasing a turbulence level in the closed circular pipe. Authors mounted inside duct interior pipe located in the axis of the analysed duct. This element is injecting an additional axial stream of fluid, which aim is to increase a level of turbulence for the main stream. The additional stream is throttled by a plug, which is creating a 1 mm chink between a plug and edge of the interior pipe. Scheme of proposed construction of turbulence mixer is shown on Fig. 1.

## 2. Experimental research methodology

Experimental analysis was realized by usage of a constant temperature anemometry method. This method depends on the intensity of heat transfer between measurement probes, which is flooded inside the heated fluid. Usually, a probe is made of a very thin wire from wolfram, which is heated by electric current. Preparation of an energy balance for a neighbouring fluid and probe allows measuring the intensity of turbulence of a flow and average velocity of a flow. Velocity in a flow is measured based on a loss of heat, which is occurring on the probe during a flow of fluid in the probe surroundings. Experimental research was realized by using of IFA-300 constant temperature anemometer.

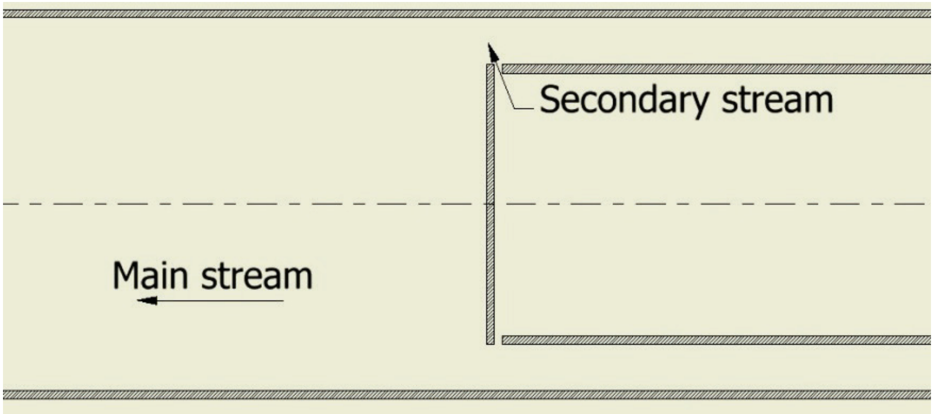


*Fig. 1. Scheme of turbulence mixer*

Measurements were realized in symmetry plane of the tested duct in distance equal to 100 mm between another measurement points by small, special gaps, which were drilled in a tested duct. The velocity of a flow was measured along the pipe diameter. Flow in main flow was generated by the fan, which delivers air through the duct. During research, two different velocities of a flow inside the duct were applied. Tests were realized with a velocity of mainstream equal to 1.25, 7.5 and 22 m/s. The secondary stream was developed to the test stand by an interior duct, which was supplied with a constant velocity equal to 1.5 m/s.

## 3. The methodology of numerical calculations

The numerical calculation was prepared in ANSYS Fluent environment. This software allows preparing numerical analysis for fluid flow in different applications. Obtained results for a fluid flow modelling can be divided into two parts. The first group of results is based on a Reynolds-averaged Navier-Stokes equation, which allows obtaining average results of analysed parameters for a flow in a steady state solution. This type of calculations is popularly named RANS, which is an acronym from equation used for calculations. This method is most widely used to calculate industrial application of numerical calculations. RANS calculations allow obtaining average results of calculations, which can be easily compared with experimental analysis.

The second group of numeric calculations is called as Large Eddy Simulations (LES). This group of modelling allows calculating unsteady parameters of the analysed model. Numerical calculations, in this case, are realized as an unsteady state. This type of numerical modelling includes a model of turbulence, which can modelling eddies generation inside of a flow. This type of calculation uses more computational resources and needs a higher resolution of a mesh for calculations. LES calculation refers to all turbulence models, which can realize analysis [1].

Numerical calculations for the analysed case were prepared in two main steps. Firstly, a numerical model of a fluid flow was prepared by a solution of RANS calculation. Obtained results from this step were compared with experimental results. In this step, three numerical models were prepared, each for a different velocity of the main stream according to the velocities applied in experimental research. This part of calculations was realized by implementation of Reynolds Stress model of turbulence. Secondly, results of RANS calculation were transferred into LES model, where a Delayed Detached Eddy Simulation (DDES) model of turbulence was applied. This step allows simulating how vortices are generated inside the flow. Calculations in unsteady calculations were prepared with time step equal to $1 \cdot 10^{-5}$ s and were realized to achieve expanded flow in the whole volume of the analysed duct.

A numerical model was prepared as three-dimensional although a geometry is axisymmetric. It is caused because LES models needed a three-dimensional case of geometry for obtaining of real distribution of eddies generated inside the flow. Numerical calculations were realized on a structural, hexagonal mesh.

## 4. Results

Figure 2 presents velocity distribution for experimental and numerical calculations obtained on a cross-section of the analysed duct, which was located 20 centimetres from secondary stream outlet from turbulence mixer.

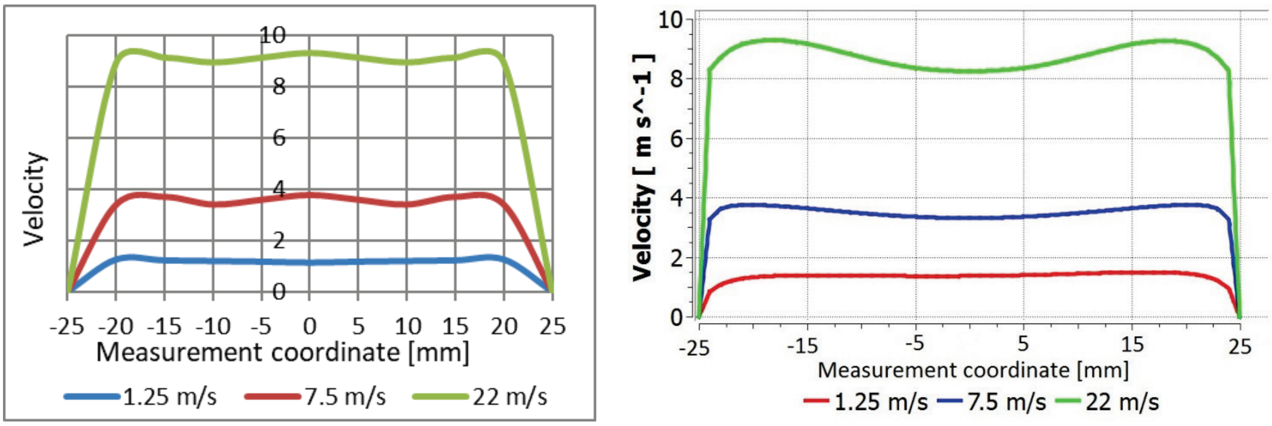


*Fig. 2. Velocity distribution measured in cross-section of duct located 20 centimetres from secondary stream inlet a) experimental research b) numerical calculations*

Obtained velocity distribution in this plane shows, that velocity is increasing in exterior parts of main flow along to the radius of the duct. It shows that internal pipe implementation into the main duct causes modification of velocity profile in this part of a flow. It is related to increasing of turbulence level in this place. Obtained velocity distribution for each of analysed flow during experimental research is close to numerical calculations. Velocity profile obtained during experimental research has a partially different course in the axis of the duct, where velocity grew slightly.

Figure 3 presents results of experimental and numerical research obtained for cross-section located 30 centimetres from the outlet of turbulence mixer.

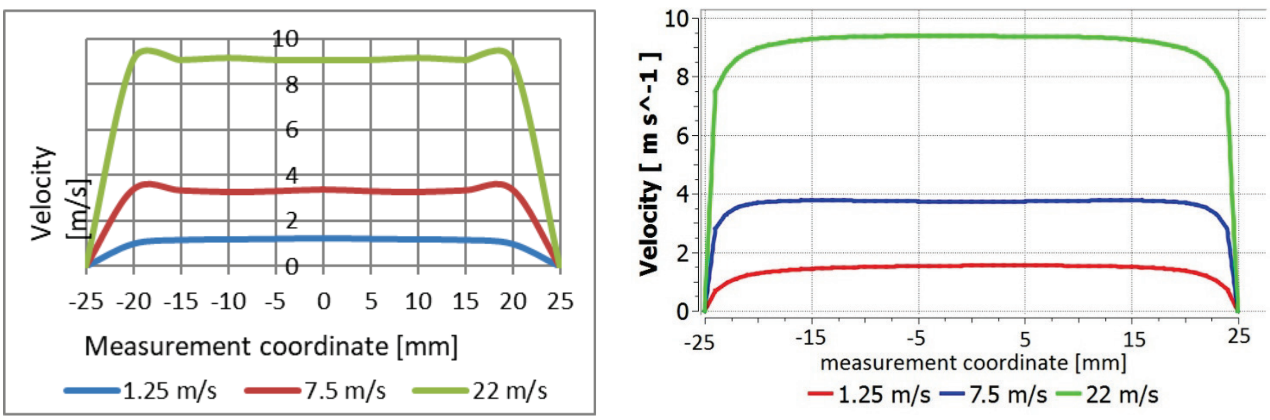


*Fig. 3. Velocity distribution measured in cross-section of duct located 30 centimetres from secondary stream inlet a) experimental research, b) numerical calculations*

Velocity distribution for the analysed place is more uniform than distribution showed for cross-section showed on Fig. 2. It means that influence of turbulence mixer on a flow inside the main duct has no effect on flow in this cross-section of a flow or impact is insignificant. Also, obtained results during experimental research showing, that distribution of velocity is more homogenous than distribution obtained in cross-section located 10 centimetres closer to turbulence mixer. Results of experimental research did not show an increase of velocity in the axis of the duct, which has occurred for the previously analysed cross-section.

Figures 4-6 present a velocity distribution for three analysed cases obtained from unsteady calculations with Delayed Detached Eddy Simulation model of turbulence. This model allowed to show a real distribution of velocity with a showing of places of eddies occurrence.

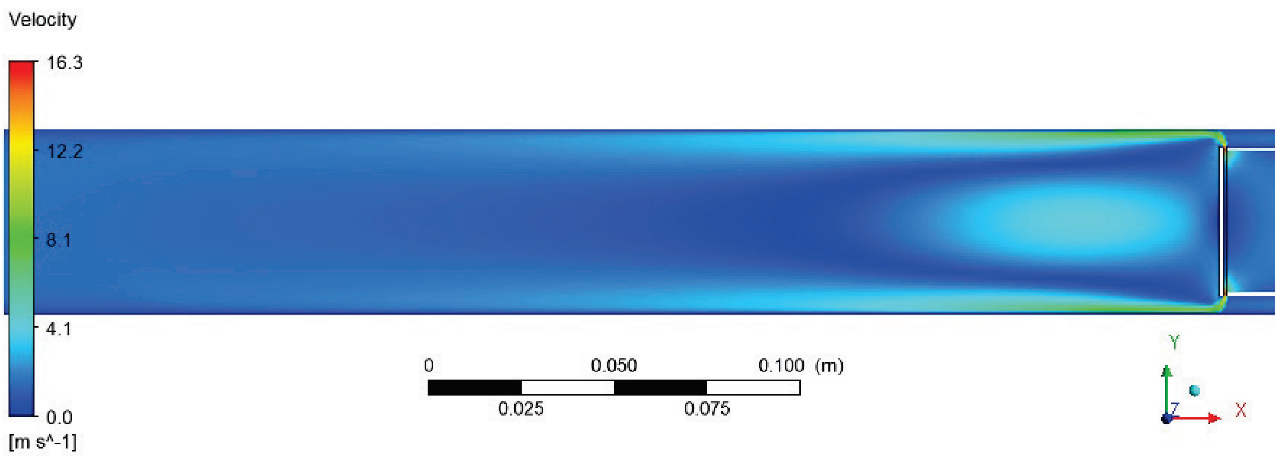


*Fig. 4. Velocity distribution for LES calculation for the velocity of a flow at the inlet to the duct equal to 1.25 m/s*

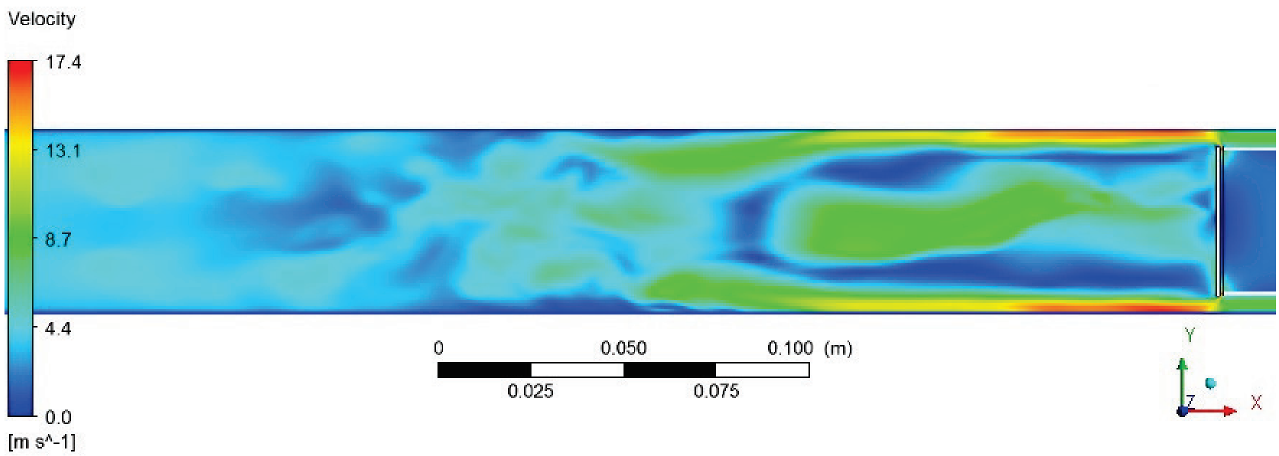


*Fig. 5. Velocity distribution for LES calculation for the velocity of a flow at the inlet to the duct equal to 7.5 m/s*

Velocity distribution obtained for the slowest analysed velocity of mainstream at the inlet to the duct showed that turbulent flow was not obtained. Results obtained during RANS calculations cannot be developed into LES model, so turbulent flow, in this case, does not exist. For cases, where flow was realized with higher velocity at the inlet to the geometry comes to the turbulence of a flow. In both of analysed cases, turbulent flow is present in some distance from turbulent mixer location. This distance is equal about 25 centimetres from the location of additional stream injection into the duct. Increasing of velocity at the inlet to the duct causes enhancement of turbulence level in a flow. It is strongly visible during a comparison of velocity profile presented in Fig. 5 and 6.

Velocity distribution obtained for the initial velocity of a flow equal to 7.5 m/s show, that the most turbulent flow is occurring in part of duct located between 10 and 20 centimetres. In the distance below 10 centimetres from turbulence, mixer obtained turbulent flow is expanding. In the axis of the duct, an area of fluctuate flow is created, which is responsible for transmission of turbulence to further parts of a flow. Turbulent flow disappears after traveling a distance of 25 centimetres from the location of an additional stream of a fluid injection. It was related with the influence of external walls of the duct, which has influence for a flow and was stabilizing a flow.

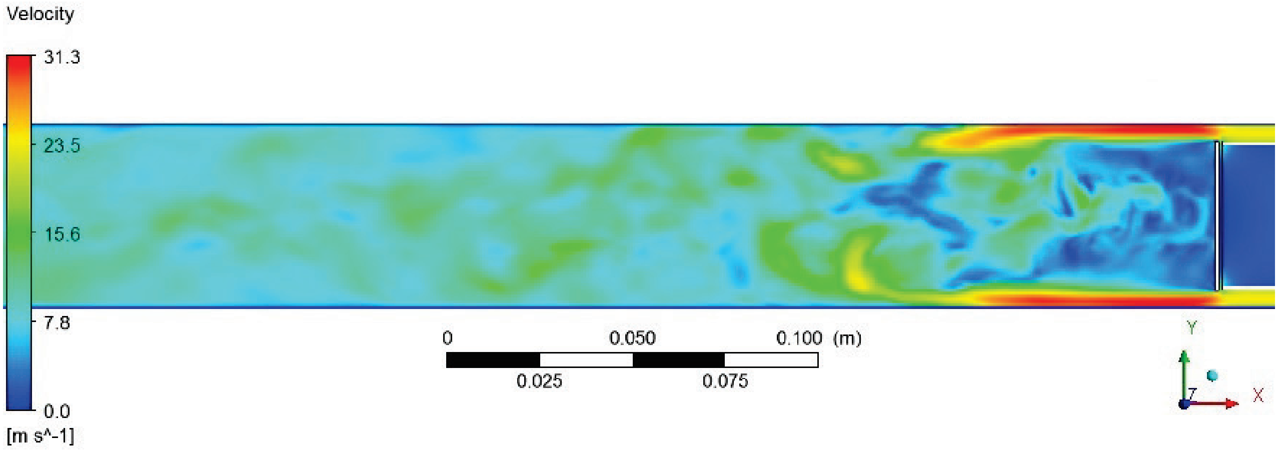


*Fig. 6. Velocity distribution for LES calculation for the velocity of a flow at the inlet to the duct equal to 22 m/s*

For a case, where an initial velocity of main flow was equal to 22.5 m/s an initial segment of a flow was not noticed. Turbulent flow is observable for a section of the main duct from a place of injection of an additional stream of fluid to the cross-section located about 25 centimetres. After that, flow was stabilized by external walls of a duct with the same effect than in case, which was calculated for initial velocity equal to 7.5 m/s.

## 5. Conclusions

Prepared analysis showed, that implementation of the internal stream of a fluid can increase a level of turbulence for flow realized in closed, circular duct. Obtained results show, that injecting an additional stream of fluid into the duct through narrow gap causes superposition of mainstream with the additional stream. Turbulent flow can be obtained only for flows, where the energy of a stream is sufficient for eddies generation. This solution can be applied in closed ducts in cases, where flow can be divided into two parts and is possible to mount a proposed construction of turbulence mixer inside the duct. Influence of proposed construction on a stream of a fluid is noticeable in distance depending on the velocity of a stream so in accordance with a theory about Reynolds number, which is commonly used in realized calculations of this phenomenon.

Application of numerical model into the calculation of turbulence increasing for technical usage allow obtaining more interesting results of research. Numerical calculations, which are realized as LES model, allow predicting places of turbulence increasing generation. This thing allows preparing more precious experimental research with the termination of places, where an exact measurement is needed.

## References

[1] Boudreau, M., Dumas, G., Veilleux, J.-C., *Assessing of ability of the DDES turbulence modelling approach to simulate the wake of a bluff body*, Aerospace, Vol. 4, Is. 41, 2017.

[2] Elsner, J., *Turbulencja przepływów*, Państwowe Wydawnictwa Naukowe, Warszawa 1987.
[3] Kunladub, S., Chuwattanakul, V., Kongkaitpaiboon, V., Promthaisong, P., Eiamsa-ard, S., *Heat transfer in turbulent tube flow inserted with lose-fit multi-channel* twisted tapes as swirl generators, Theoretical & Applied Mechanics Letters, Vol. 7, pp. 372-378, 2017.

*Manuscript received 19 December 2017; approved for printing 20 March 2018*