ISSN: 1231-4005
e-ISSN: 2354-0133
DOI: 10.5604/01.3001.0012.4339

NUMERICAL ANALYSIS OF A VELOCITY DISTRIBUTION OF A FLUID FLOW INSIDE A REVERSING CHAMBER

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

Axisymmetric stream, which is directed into an obstacle, is an important type of a fluid flow for technical applications in a large amount of thermal and flow devices. The article presents a case wherein stream of a fluid is directed into a flat surface and changes the direction of a flow by an angle equal to 90°. After that, the free stream is changing a character of a flow into impinging stream. The article presents a methodology of numerical calculations preparation in ANSYS Fluent environment for a velocity distribution of an airflow inside a reversing chamber. Numerical calculations were prepared for a three-dimensional model as an unsteady simulation with Delayed Detached Eddy Simulation model of turbulence. A stream of an air, which was analysed inside a reversing chamber, was not initially swirled. Obtained results of realized calculations were compared with experimental analysis and numerical calculations, which was realized in a different environment by co-author. Model of reversing chamber, which was implemented into numerical analysis has the same dimensions as used in experimental research. Obtained results show areas of intense flow turbulence inside reversing chamber. Prepared numerical calculations agreed with experimental results of research and allowed to designate areas of stream core and impinging stream inside modelled chamber.

Keywords: *numerical analysis, reversing flow, fluent, axisymmetric flow*

1. Introduction

Axisymmetric streams have many different possibilities of application in technical devices, so this kind of flows was widely studied by many authors [2]. A special example of axisymmetric flow, which is widely applied are restricted flows. Restricted flow is obtained for a stream, where the flow has a contact with the surface of the solid body. This kind of a flow is commonly used for technical application in thermal machines. Often encountered example of this type of a flow is a striking stream. Scheme of an analysed case of a flow is shown in figure 1. This type of a flow can be divided into 3 parts. Firstly, a stream, which is injected by the nozzle, is treated as a free jet. When a flow is directing into the area of solid wall stream has contact with a wall and it comes to create an area of stagnation of a flow near to wall area. When will come to contact of a stream with the wall then comes to changing the direction of a stream. Jet is directing radially along to the wall, which causes generation of impinging stream.

Experimental research of velocity distribution for a free and restricted stream analysis can be realizable by many different measurement methods. One of the most popular approaches is the application of constant thermal anemometry method. This method was used by co-author in his experimental work about the analysed phenomenon. To the measured quantities by this method belong average velocity of a flow and value of flow fluctuation. Obtaining of the real velocity distribution of a flow during experimental research more complicated. Implementation of a probe into a flow allows obtaining the average velocity of a flow. Realization of analysis of deviation from average velocity allows designating fluctuation for a flow. Real distribution of velocity inside of a flow can be obtained from numerical calculations based on application into calculations appropriate model of turbulence.

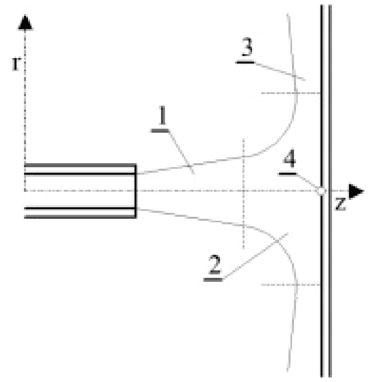

Fig. 1. Scheme of striking stream: 1 – free jet region, 2 – region of stagnation of a flow, 3 – wall jet region, 4 – stagnation point [3]

2. Description of numerical calculations methodology

The numerical calculation was realized for a geometry, which was used by co-author during experimental research. Scheme of modelled reversing chamber is shown in Fig. 2.

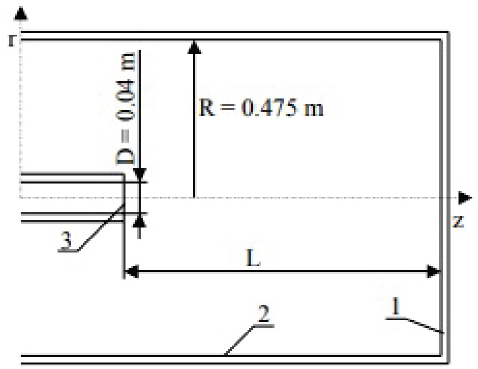

Fig. 2. Scheme of reversing chamber: 1 – impinging wall, 2 – sidewall, 3 – the outlet of nozzle [1]

A numerical model was prepared in ANSYS Fluent software. Numerical calculations were divided into two parts. Firstly, numerical calculations were realized by a solution of a Reynolds-averaged Navier-Stokes equation (RANS), which allows obtaining average results of analysed parameters for a flow in a steady state solution. RANS calculations allow obtaining average results of calculations, which can be easily compared with experimental analysis. Numerical calculations were realized for three different velocities of the stream. Calculations were prepared for airflow inside reversing chamber. The velocity of fluid was equal to 10, 30 and 50 m/s at the inlet to the nozzle.

The second part of numeric calculations was relied on solving of model based on Large Eddy Simulations (LES). This group of modelling allows calculating unsteady parameters of the analysed model. Numerical calculations, in this case, were realized as an unsteady state. This type of numerical modelling includes a model of turbulence, which can modelling generation of eddies inside of a flow. This part of calculations was realized by implementation of Delayed Detached Eddy Simulation (DDES) model of turbulence, which is a part of LES models. This step allows simulating how vortices are generated inside the flow. Calculations during 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.

3. Results of RANS profile distributions

Figure 3 presents velocity distribution along to the axis of the reversing chamber obtained during numerical calculations in steady state conditions. In addition, results of experimental research obtained by co-author are shown, which was used for results comparison.

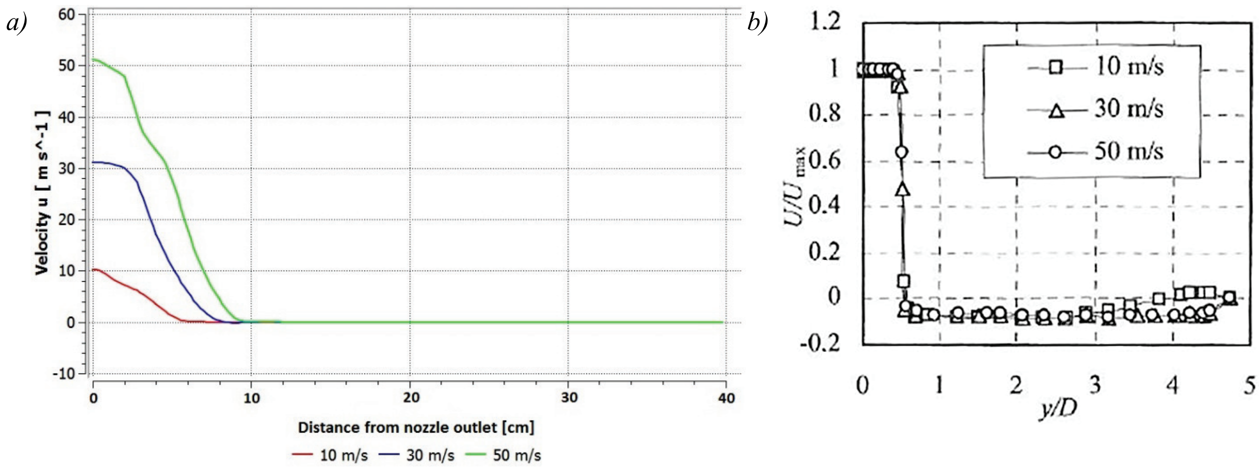

Fig. 3. Axial velocity distribution measured in the axis of the reverse chamber a) RANS numerical calculation, b) experimental research [1]

Calculations of RANS model allowed obtaining average velocity distribution in axis for analysed flow. The course of velocity distribution obtained during numerical calculations is similar to the experimental research prepared by co-author. A stream of a fluid, which is injected into the chamber with initial velocity, was slowing after injection to velocity equals to a stagnation point. This part of a stream is treated as a free stream. Subsequently came to decelerating of the velocity of a stream by the influence of a fluid, which is filling volume of the researched chamber. This process occurs on the distance equal about 6-10 cm from the outlet of a nozzle, depending on the initial velocity of a fluid. This distance determines on 15-25% length of the chamber. During experimental research, decreasing of velocity was occurring on 12-15% length of the chamber. When the stream is crossing area where comes to intense degreasing of velocity obtained average velocity in the axis of a flow was close to stagnation flow. Analysed experimental results also showed that average velocity was close to stagnation of a flow, but in part of a flow, a very low amount of reversed flow was noticed.

Preparation of RANS calculation allowed only for obtaining an average profile of a flow. This model did not allow showing the real distribution of velocity inside the stream. It is certain, that velocity of a flow is variable in conjunction with turbulence flow occurring. RANS model did not show areas of velocity changing and treats a flow as steady state.

4. Results of numerical calculations

Figure 4 presents results of numerical calculations obtained for calculation of LES model. The first figure presents velocity distribution for initial velocity equal to 10 m/s. On the second figure is presented a velocity distribution for initial velocity equal to 30 m/s. The third figure presents velocity distribution for the case, where velocity in the nozzle was equal to 50 m/s.

The prepared numerical model shows, that instantaneous velocity distribution of a flow is much different as obtained during steady-state calculations. Velocity distribution is changing in the time during the influence of turbulence occurring inside of a flow. Changing of a velocity profile in the time causes the occurrence of distribution, which was showed during experimental research and steady-state calculation.

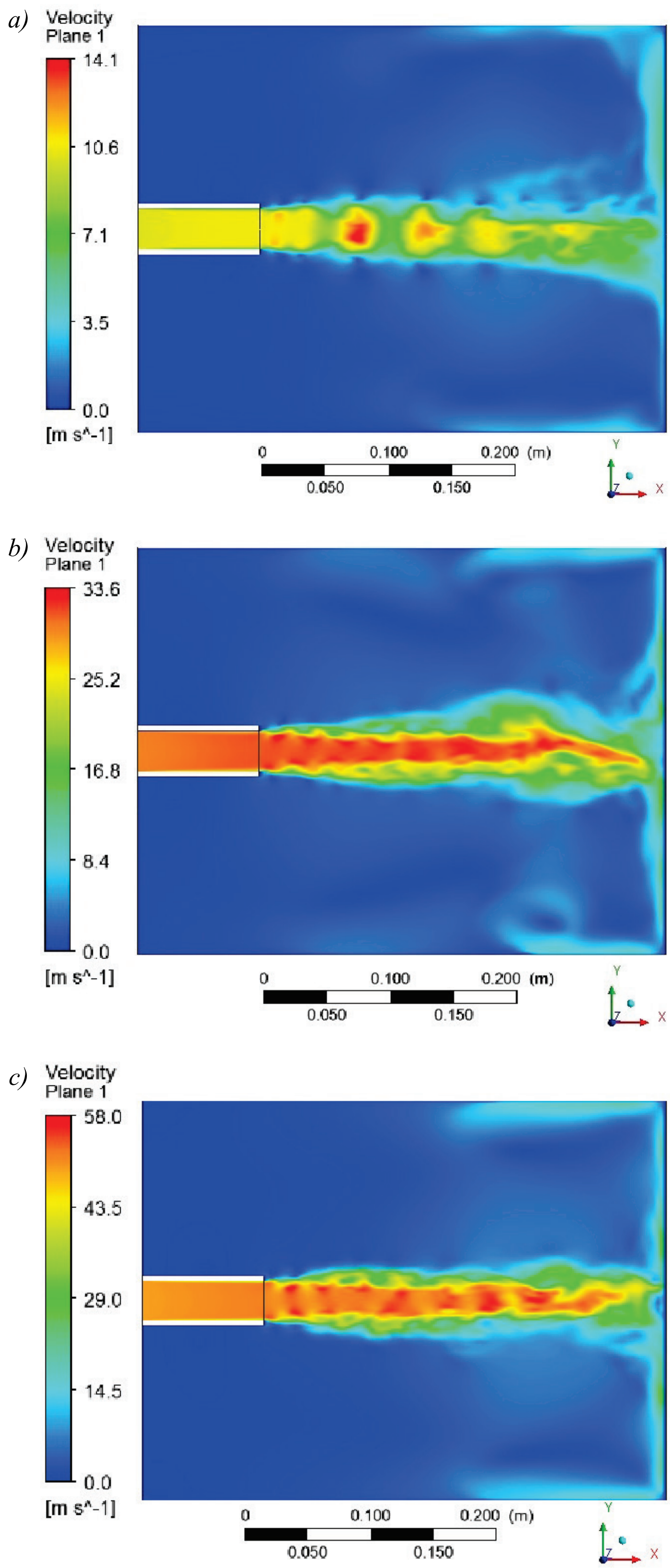

Fig. 4. Results of numerical calculations obtained for calculation of LES model for initial velocity equal to: 10 m/s (a), 30 m/s (b), 50 m/s (c)

Analysis of velocity profile obtained during unsteady calculations allowed to show, that flow is turbulent. It is confirmed by Reynolds Number, which was equal to 26 thousand for flow with a velocity equal 10 m/s, $7.8 \cdot 10^4$ when flow was realized with a velocity equal to 30 m/s and

$1.3 \cdot 10^5$ for a flow realized with an initial velocity equal to 50 m/s. Velocity profile, which was obtained for a case, where a flow was realized with an initial velocity equal to 10 m/s showed, that reached flow has not created a core inside of a flow. In this case, a stream located in the axis of the chamber is composed of parts of fluid in a ball shape, which decreasing his size during flowing into impinging wall place. The core of a flow is present inside a stream for cases where initial velocity was equal to 30 m/s and 50 m/s. In the first phase of flow, obtained core has stable character. It was shown also in steady state calculations and experimental research. After crossing of a place by the stream, where intense degreasing of velocity occurred core stream has unstable character. The direction of a flow for this part of a flow is changing fluently. It is visible on steady-state calculations as a stagnation flow. After that, flow is directing along to impinging wall in the radial direction. Subsequently, flow is directing into the sidewall and is flowing to the outlet according to the sidewall.

Calculations of unsteady-state model show places of eddies generation by axisymmetric stream influence. This kind of a flow has few places, where are occurring intense conditions of eddies generation. Especially visible can be eddy, which was generated for a flow where velocity at the inlet was equal 30 m/s. This model allowed showing an eddy generated on an area of the sidewall, which was trailing area for obtained flow. Different place, where eddies are generated is the area of stagnation point, where a change of flow direction is occurring.

5. Conclusions

Preparation of numerical calculation can help in research connected with fluid mechanics and flow in different technical applications. Conducting a research of fluid flow in basic and complex geometries allows analysing parameters for flows and allows designing better constructions of thermal and flowing devices. Analysis, which are belonging to the LES models can visualize areas of turbulent flow and obtain information about the character of a flow [4].

Analysis of reversing flow for analysed model agreed with experimental results in steady state step of calculations. Preparation of unsteady model allowed to expand obtained results to more information associated with eddies generation, places of unsteady velocity distribution and areas of direction changing in a flow.

References

[1] Bartoszewicz, J., *Analiza przepływu płynu przez komorę nawrotną*, Wydawnictwo Politechniki Poznańskiej, Poznan 2012.

[2] Ih, J.-G., Lee, B.-H., *Theoretical prediction of the transmission loss of circular reversing chamber mufflers*, Journal of Sound and Vibration, Vol. 112, Iss. 2, pp. 261-272, 1987.

[3] Kłosowiak, R., Bartoszewicz, J., Urbaniak, R., *Energy transformation in turbulent flow inside reversing chamber*, Engineering Mechanics 2017, 23rd International Conference, pp. 466-469, Svratka 2017.

[4] Popiolek, T. L., Awruch, A. M., Teixeira, P. R. F., *Finite element analysis of laminar and turbulent flows using LES and subgrid-scale models*, Applied Mathematical Modelling, Vol. 30, pp. 177-199, 2006.

Manuscript received 01 June 2018; approved for printing 05 September 2018