

## Simulation of Turbulent Axisymmetric Waterjet Using Computational Fluid Dynamics (CFD)

PhD. Eng. Nicolae MEDAN<sup>1</sup>

<sup>1</sup>Technical University Cluj-Napoca, North University Center Baia Mare, Nicolae.Medan@cunbm.utcluj.ro

**Abstract:** This paper presents numerical simulation of axisymmetric turbulent free waterjet developed in water using RANS method with  $k$ - $\epsilon$  model. GAMBIT and FLUENT software has been used. There are presented graphs and contours for axial and radial velocities of waterjet, contours for the turbulence kinetic energy (TKE) and graphs representing the turbulence intensity (IT). There is briefly presented also the concept of computational fluid dynamics and methods developed for numerical simulation of fluid flow.

**Keywords:** turbulent axisymmetric jet, CFD, FLUENT

### 1. Introduction

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to analyze work processes and phenomena involving fluid flow [1], [9]. Computational fluid dynamics is an interdisciplinary and practical approach linking disciplines: fluid dynamics, mathematics and computing. In terms of fluid dynamics, computational fluid dynamics (CFD) is dedicated to studying fluid motion and fluid flow affects how processes are involved. In terms of mathematical equations are required to describe the state of the process fluid to be investigated. In order to solve these mathematical equations that describe fluid are converted by computer programs software packages. In the last years, the usage of CFD in the studies of fluids has increased, due to the advantages that it presents [6], [8].

#### 1.1 Developed methods for numerical simulation of fluid flow

Using the Reynolds decomposition representation of turbulent motion, the movement is divided into an average component and a pulsating and using mathematics to describe the movement system Navier-Stokes equations, are reported in the literature several methods developed for numerical simulation: **DNS** (Direct Numerical Simulation), **LES** (Large Eddy Simulation), **DES** (Detached Eddy Simulations), **RANS** (Reynolds Averaged Navier-Stokes equation).

**RANS** method is the oldest and also the most used method. It consists in mediating temporal Navier-Stokes equations based on the representation Reynolds turbulent motion. Following this operation, there are a number of additional terms due to turbulent pulsations temporal mediation with tensions apparent significance (Reynolds stresses). The method uses different models which explains these tensions and can close the system of equations. Turbulence models used in RANS method can be grouped into two categories:

1) Models using Boussinesq's hypothesis. In this group are included: models based on mixing length theory (Prandtl), models with "zero equation" and models with "two equations",  $k$ - $\omega$  or  $k$ - $\epsilon$ . The standard  $k$ - $\epsilon$  model proposed by Spalding is the most used.

2) Models for Reynolds stresses. These models introduce one transport equation for each Reynolds stress resulting from mediation Navier-Stokes equations. Number of unknowns increases considerably and therefore the calculation effort is high. These models are seldom used.

RANS method with  $k$ - $\epsilon$  model is the professional software package CFD (Computational Fluid Dynamics) for simulating turbulent flows.

#### 1.2 Mathematical description of RANS method for turbulent axisymmetric fluid flows

The mathematical model used to describe the movement of axisymmetric turbulent jets feature, use the pressure and velocity coupling Cartesian coordinate system. To solve the system of equations is using standard  $k$ - $\epsilon$  turbulence model [5], [7]. Forming equations describing turbulent flow system can be written in the following general form [5]:

$$\underbrace{\frac{\partial}{\partial x_j} (\rho u_j \phi)}_{\text{convection}} = \underbrace{\frac{\partial}{\partial x_j} \left( \Gamma_\phi \frac{\partial \bar{\phi}}{\partial x_j} \right)}_{\text{diffusion}} + \underbrace{S\phi}_{\text{source}} \quad (1)$$

## 2. Numerical Simulation of Axisymmetric Turbulent Jet Using CFD

CFD software package contains three modules, namely [4]:

- 1) Pre-processor,
- 2) Solver (governing equations solve on a mesh),
- 3) Post-processor.

In the present study is performed 2D axisymmetric simulation of turbulent jets using the Fluent 6.3 [2]. For definition and scope of work meshing program Gambit [3] was used.

### 2.1 Field work

Consider a cylindrical nozzle through which fluid effluent, which in this case is water. Jet is formed in the ambient fluid that is all water.

In figure 1 is presented the schematic diagram of the computational domain with dimensions.

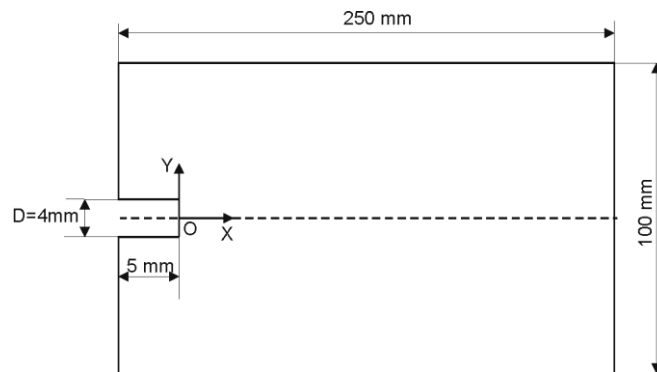


Fig. 1. Schematic diagram of the computational domain

Is studied the evolution of axisymmetric turbulent jets for different speeds effluent fluid out of the nozzle. We deal with free jets drowned (ambient fluid is at rest). It is known that the main property of turbulence is given by the Reynolds number. In most cases if the Reynolds number exceeds 2300, flow becomes turbulent. In this case we want to study the axisymmetric turbulent jets and therefore the lowest value of Reynolds number which will be modelled is  $Re = 3,636$ .

Table 1 presents the input parameters for conducting experiments.

TABLE 1: Input parameters for conducting experiments.

Speed Parameters	v=1 m/s	v=3 m/s	v=5 m/s	v=7.76 m/s
Reynolds Re	3,636	10,908	18,180	28,200
Turbulence Intensity T.I.	0.44	0.51	0.54	0.57

### 2.2 Initial conditions (pre-processor)

To define the initial conditions using Gambit software [3] and includes the following steps:

- 1) Creating geometry, based on field work schedule (figure 1).
- 2) Meshing method. There are used rectangular cells with dimensions 0.33mmx0.5mm in area next to the entrance and rectangular cells with dimensions 0.5mmx0.5mm in rest.
- 3) Boundary conditions. How is define the types of boundaries is shown in figure 2.

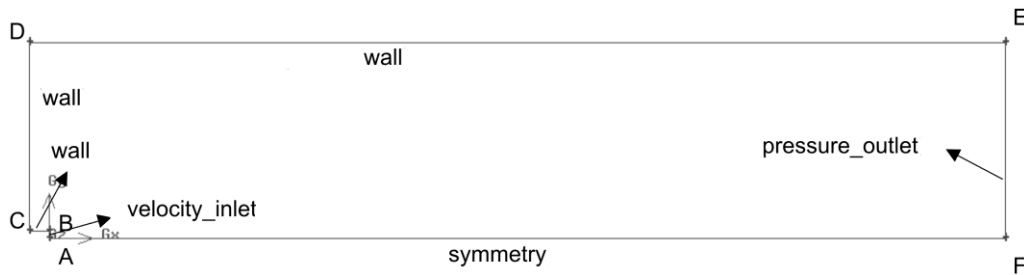


Fig. 2. Computational domain and boundary conditions

**2.3 Perform simulation (solver)**

After defining the domain and meshing, the simulation was performed using FLUENT 6.3 software. It defines the turbulent motion model solved. To define the turbulent flow model that solves movement RANS method was chosen to model 2 equation k-ε standard. Table 2 presents discretization scheme used for turbulent jet flow equation. It used the pressure-velocity coupling (SIMPLE method).

TABLE 2: Discretization schemes for jet flow

Variable	Discretization scheme
Pressure	Presto
Momentum	Second Order Upwind
Turbulent kinetic energy	Second Order Upwind
Turbulent dissipation rate	Second Order Upwind

**3. Results**

Still present some results obtained from simulations with FLUENT software [2].

**3.1 Graphs representing the heart jet axial velocity and radial velocity**

For graphical representation of flow velocity, I present results for all four cases considered velocity. In figure 3 is represented the numerical simulation of variation of axial velocity in the centre jet speed for all cases considered and for a distance of 100 mm from the jet origin.

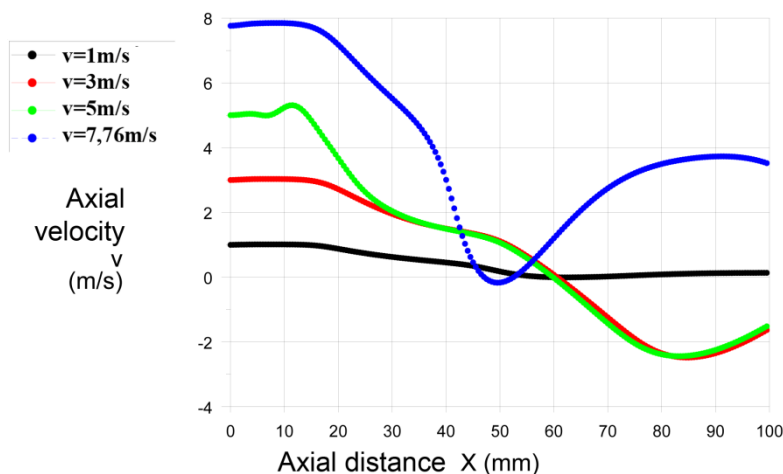


Fig. 3. Variation of axial velocity in the centre of jet for x=0÷100 mm

In figure 4 is represented the numerical simulation of radial flow velocity variation for different distances from the jet origin x = 1 mm, x = 2mm and x =4mm and fluid velocity at the nozzle exit is for all cases considered.

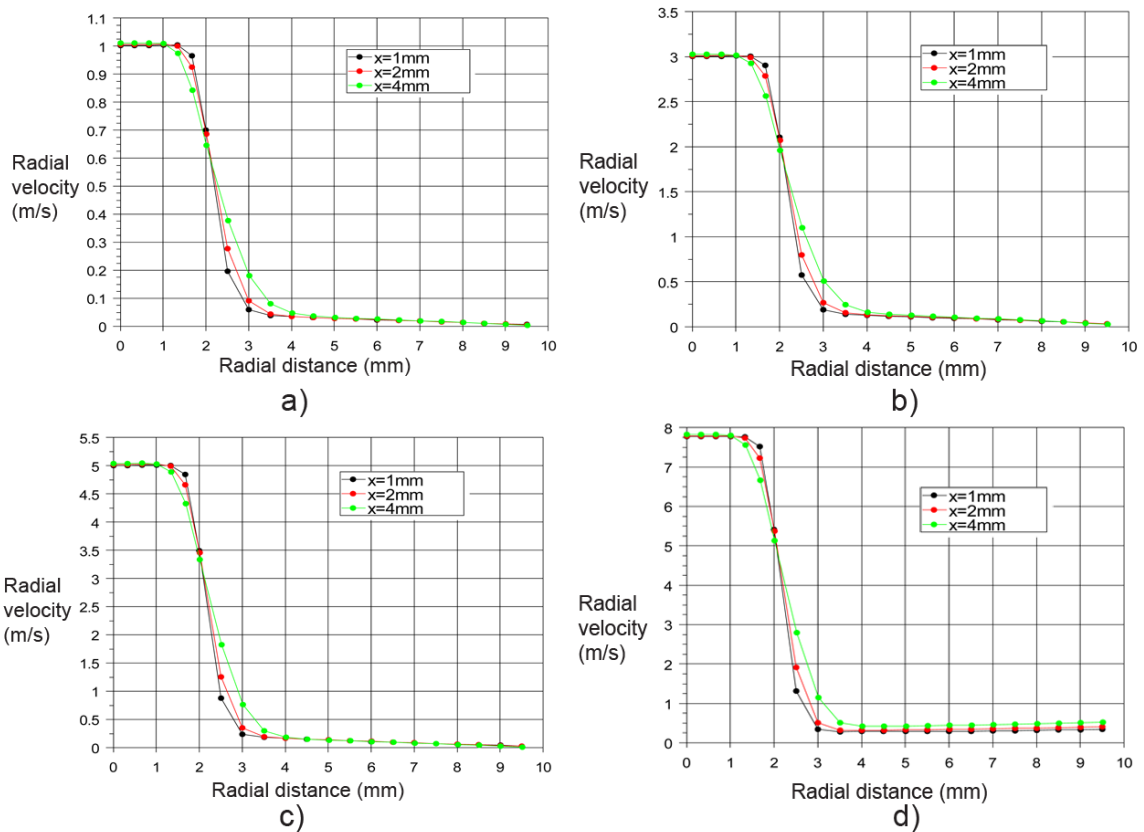


Fig. 4. Variation of radial velocity for a) v=1 m/s, b) v=3 m/s, c) v=5 m/s, d) v=7.76 m/s

### 3.2 Contours represent the axial and radial velocities of jet

In figure 5 are the axial flow velocity contours for all 4 values of the speed, for the entire area define.

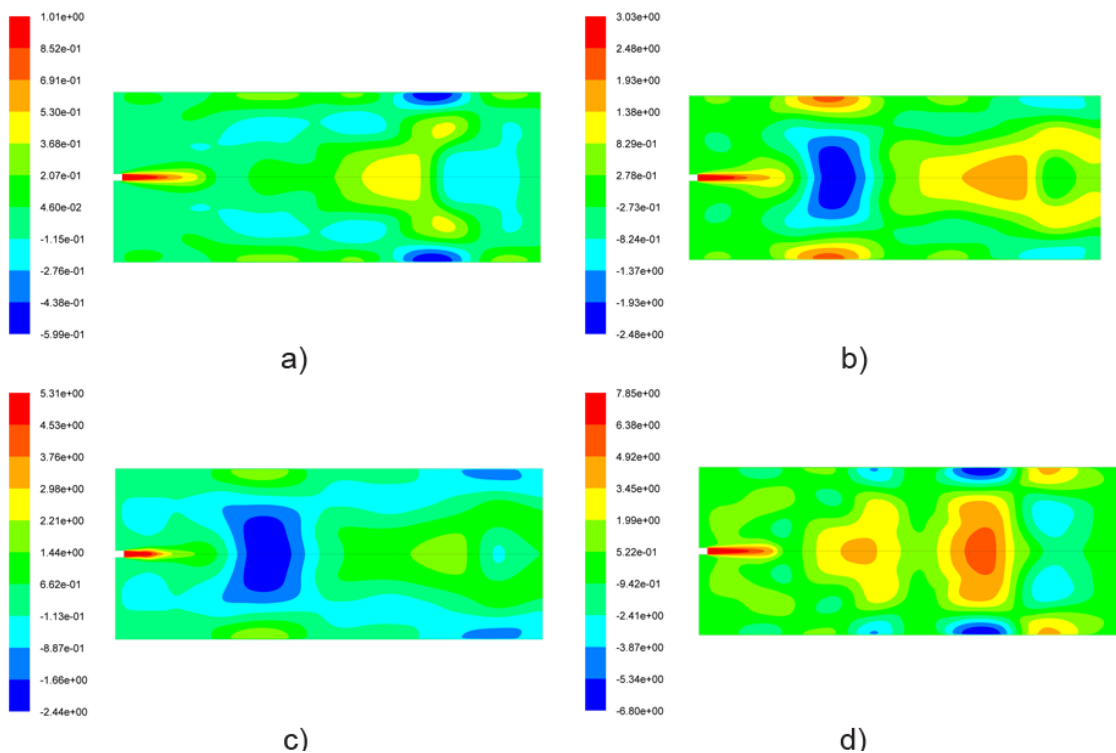


Fig. 5. Contours of axial velocity for a) v=1 m/s, b) v=3 m/s, c) v=5 m/s, d) v=7.76 m/s

In figure 6 are the radial flow velocity contours for all 4 values of the speed.

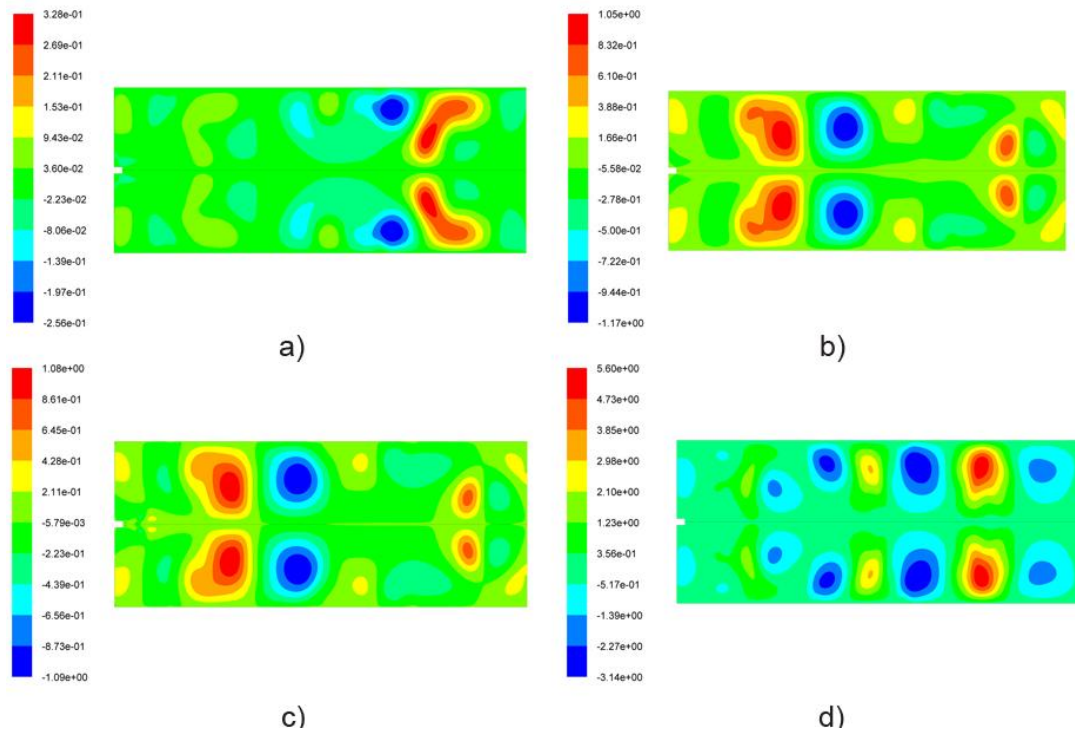


Fig. 6. Contours of radial velocity for a)  $v=1$  m/s, b)  $v=3$  m/s, c)  $v=5$  m/s, d)  $v=7.76$  m/s

### 3.3 Contours represent the turbulence kinetic energy (TKE)

In figure 7 are represented the turbulence kinetic energy (TKE) for all 4 values of the speed, for the entire area define. TKE is the kinetic energy per unit mass of the turbulent fluctuations in a turbulent flow.

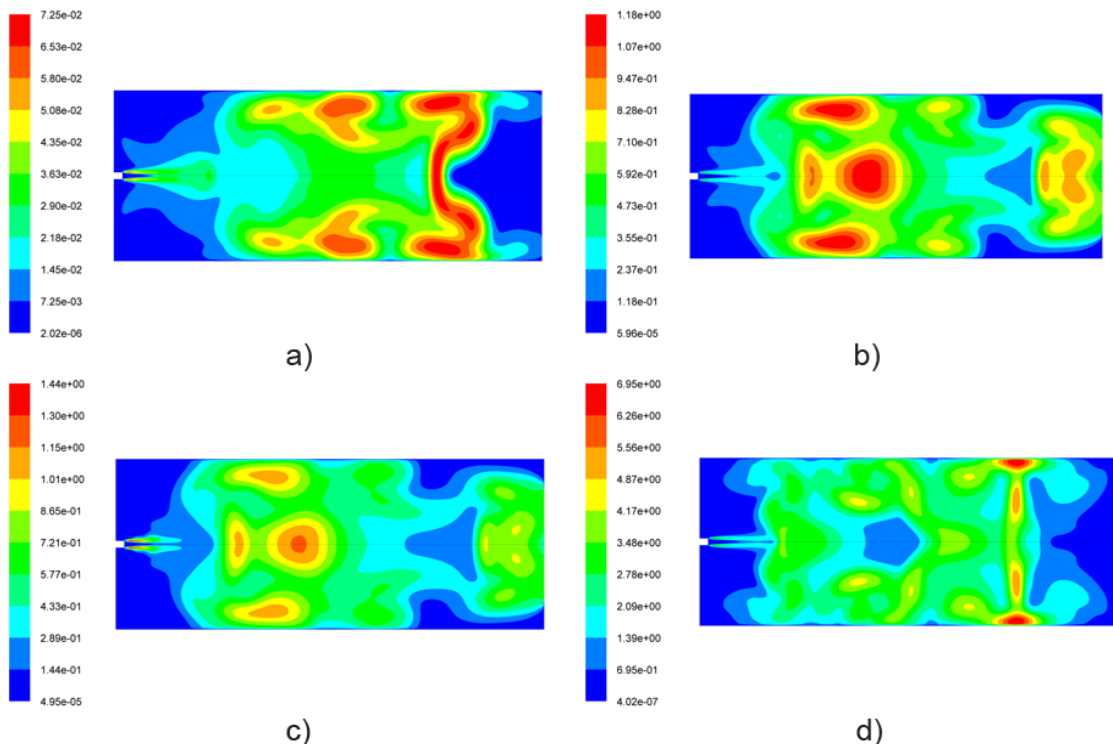


Fig. 7. Contours of turbulence kinetic energy (TKE) for a)  $v=1$  m/s, b)  $v=3$  m/s, c)  $v=5$  m/s, d)  $v=7.76$  m/s

### 3.4 Graphs representing the turbulence intensity (IT)

In figure 8 are represented the turbulence intensity (IT) for all 4 values of the speed, for the entire area define along the axial distance.

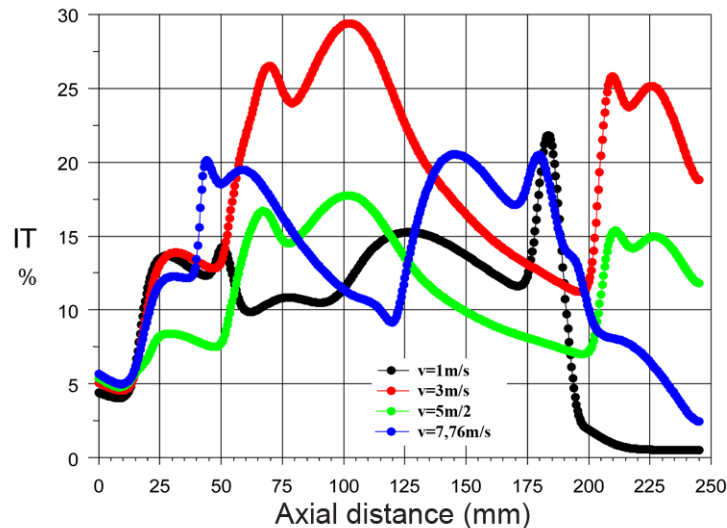


Fig. 8. Variations of turbulence intensity (IT) for  $v=1$  m/s,  $v=3$  m/s,  $v=5$  m/s,  $v=7.76$  m/s

## 4. Conclusions

In this study was performed CFD simulation of axisymmetric turbulent flow for waterjets using the RANS method with model 2 equations  $k-\epsilon$  standard. For this application was used Gambit and Fluent.

To define the initial conditions was used Gambit software and includes the following steps: creating geometry, based on field work schedule, defining the mesh and the boundary conditions.

The numerical simulation was performed for different values of axial velocity at the exit of jet:  $v = 1$  m/s,  $v = 3$  m/s,  $v = 5$  m/s and  $v = 7.76$  m/s. To generate the mesh area was used rectangular cells with dimensions  $0.33\text{mm} \times 0.5\text{mm}$  in area next to the entrance and rectangular cells with dimensions  $0.5\text{mm} \times 0.5\text{mm}$  in rest.

For axial velocities along the center of the jet can be seen that they decrease as moving away from the origin of the jet.

Representing the radial velocities of flow was made at distances equal to 1 mm, 2 mm and 4 mm from the jet origin for  $v = 7.76$  m/s. It can be seen that the radial velocity decreases with increasing radial distance from the origin.

Through the representation of the contours of the axial and radial speed of the waterjet can be noticed the way of develop the waterjet. With increasing velocity of a jet at the outlet saw an increase of the area in which develop the jet.

By viewing the contours of the turbulence kinetic energy it can view the way that develops the turbulence kinetic energy (TKE) in the field work. The turbulence kinetic energy (TKE) increases with increasing the speed at the outlet from the hole.

For the graphs representing the turbulence intensity (IT), it should be observed that with increasing velocity of the waterjet at the outlet, increases in direct proportion the turbulence intensity of the fluid in the jet. With increasing distance from the origin of the jet appear variations of the turbulence intensity due to the influence of the working area. Thus for  $v = 1$  m/s the turbulence intensity is almost 0 at a distance of 200 mm, which means that the jet is fully developed in the working area. In the other 3 cases considered appear variations of the turbulence intensity due to the limits of the working area.

Numerical simulation of fluid flow is an alternative that can replace the implementation of experiments, in the sense that it can reproduce the conditions of experiments and can be viewed

and further processed results. Numerical simulation can successfully replace experiments often involve high costs.

### References

- [1] T.J. Chung, “Computational Fluid Dynamics”, Cambridge University Press, ISBN: 978-0-521-76969-3, 2002;
- [2] FLUENT 6.3.26: User manual;
- [3] GAMBIT 2.3.16: User manual;
- [4] T. Jiyuan, H.Y. Guan, L. Chaogun, “Computational Fluid Dynamics A Practical Approach”, Elsevier Press, ISBN: 978-0-7506-8563-4, 2008;
- [5] B.E. Lauder, D.B. Spalding, “Lectures in Mathematical Modeling of Turbulence”, Academic Press, London, England, ISBN-10: 0124380506, 1972;
- [6] P. Oprețoiu, RANS Simulation of Combined Flow and Heat Transfer Through Open-cell Aluminium Foam Heat Sink, “Hidraulica” (No. 3/2013) Magazine of Hydraulics, Pneumatics, Tribology, Ecology, Sensorics, Mechatronics, ISSN 1453 – 7303, pp. 15-25;
- [7] D.B. Spalding, “Basic Equations of Fluid Mechanics and Heat and Mass Transfer and Procedures for Their Solutions”, Heat Transfer Section, Report HTS/77/9, Imperial College London, 1976;
- [8] K. Stračár, J. Krchnár, K. Prikkel, CFD Model of Flow in the Outlet Channel of Floating Chamber “Hidraulica” (No. 1/2014) Magazine of Hydraulics, Pneumatics, Tribology, Ecology, Sensorics, Mechatronics, ISSN 1453 – 7303, pp. 7-23;
- [9] D.C. Wilcox, “Turbulence Modeling for CFD”, DWC Industries Inc., La Canada California, ISBN: 0-9636051-0-0, 1994.