PAPER • OPEN ACCESS

Circular Free Jets: CFD Simulations with Various Turbulence Models and Their Comparison with Theoretical Solutions

View the article online for updates and enhancements.

Circular Free Jets: CFD Simulations with Various Turbulence Models and Their Comparison with Theoretical Solutions

Miroslava Kmecova [1], Ondrej Sikula [2], Michal Krajcik [1]

[1] Slovak University of Technology, Faculty of Civil Engineering, Radlinskeho 11,
   81005 Bratislava, Slovakia
[2] Brno University of Technology, Faculty of Civil Engineering, Veveri 95, 60200
   Brno, Czech Republic

   miroslava.kmecova@stuba.sk

Abstract. The theory of turbulent free jet is fundamental for the design of comfort ventilation, as free jets frequently occur in mixing and personalized ventilation systems and their characteristics strongly influence the air quality in the breathing zone of an occupant. The aim of this research is to provide recommendations that help researchers and practitioners improve the accuracy and reliability of their computational models of ventilation systems involving circular free jets. To accomplish this, a review of existing theoretical calculation models is performed, and these models are subsequently investigated by computational fluid dynamics. The theoretical solutions of free jets are compared with CFD simulations using various turbulent models such as the standard k-epsilon model, the k-epsilon realizable model, the standard k-omega model, the shear stress transport (SST) k-omega model, and the Reynolds stress model (RSM). The simulated models are represented by profiles of the centreline velocity for a free jet emanating from a round nozzle, because such presentation of the data proved to be particularly helpful for the comparison of the turbulence models. The k-omega SST turbulence closure scheme with standard coefficients produced results of the centreline velocity closest to the average of theoretical solutions investigated, whereas the discrepancy between the simulations and the theoretical models was about 60 % with the k-epsilon standard turbulence model.

1. Introduction

Circular jets are used in a variety of applications, some of which include drying processes, air curtains, and room space heating, air conditioning and ventilation. A free air jet is a term used to describe a flow of air issuing from an opening or a nozzle into a space where there are no solid boundaries to influence the flow [1]. The boundary layer at the exit of the supply develops as a free shear layer, mixing with the ambient fluid thereby entraining the ambient fluid in the jet stream. The mass flow at any cross section of the jet progressively increases, while the jet centreline velocity decreases with downstream distance [2]. A circular free jet can emanate from a pipe, from a nozzle with smoothly contracting shaping upstream of the nozzle exit plane or from a sharp-edged orifice. These three different exit conditions lead to different developments of the flow downstream of the exit plane [3]. Regardless of the exit shape used, the flow field of a circular jet can be divided into regions, related to the centreline velocity decay, as shown in figure 1.

The *core* of the jet is defined as the region of flow where the centreline velocity, U_m, is greater than or equal to 95 % of the supply velocity, U_0 [4]. The centreline velocity, U_m, is constant and equal to the supply velocity, U_0. It has a conical shape, and it typically extends up to $4d_0$ to $6d_0$ [5]. In the *transition* zone the centreline velocity starts to decay at a rate that can be approximated as proportional to x-0.5. This usually corresponds to a region from $6d_0$ to $20d_0$, and it is known as the interaction region where shear layers from both sides merge. Beyond the transition zone, the transverse velocity profiles are similar at different values of x and the centreline velocity decay is approximately proportional to x-1. This region of *profile similarity* is dominated by a highly turbulent flow generated by viscous shear at the edge of the shear layer. For three-dimensional jets this is usually referred to as the "fully developed flow region" [1, 2]. The zone of *termination* is a region of rapid diffusion, where the jet becomes indistinguishable from the surrounding air, and the centreline velocity decays with the square of the distance [1]. Because of the large velocity difference at the surface of discontinuity between the jet fluid and ambient, large eddies are formed, which cause intense lateral mixing. As a result of this mixing, fluid within the jet is decelerated and the fluid surrounding the jet is accelerated and entrained into the jet flow [2]. The different mechanisms of jet disintegration where the jet flow stops entraining room air and supply air starts to diffuse into the occupied zone are still not well understood and should be subject to further investigations.
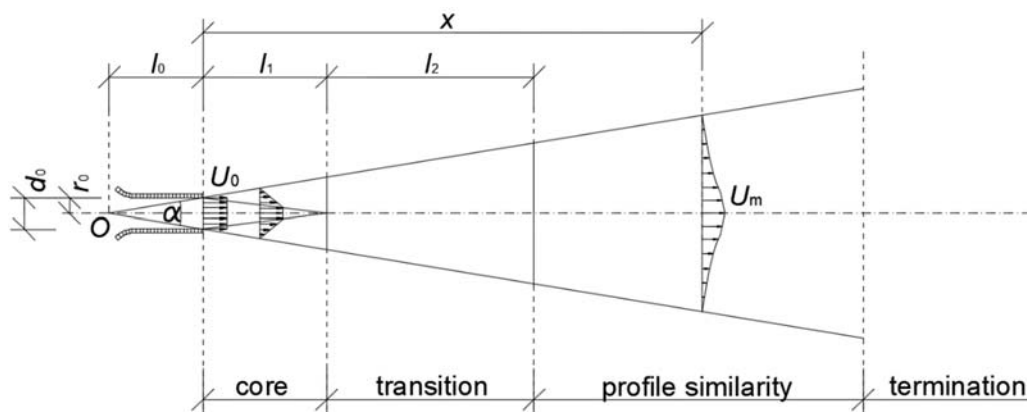
Figure 1. Development of a free jet (adapted from [5, 6])

Key: U_0 - air velocity at the outlet from the jet channel; U_m - centreline velocity in the assumed cross-section of the stream; O - pole; r_0 - radius of the outlet channel of the jet; d_0 - diameter of the outlet channel of the jet, equal to $2r_0$; l_0 - distance of the pole from the nozzle outlet; l_1 - core of the stream, where $U_m = U_0$; l_2 - transition interval of the stream with a decreasing velocity; x - distance from the supply; α - stream angle

CFD modelling of free jets has become an important tool to design and optimize ventilation systems. One of the crucial aspects in the computer simulations of fluid dynamics to obtain reliable results is the selection of a turbulence model that is most appropriate for the specific situation. This study investigates the performance of five turbulence models available in the software ANSYS Fluent v15 when applied in CFD simulations of a circular free jet. Moreover, some of the major theoretical solutions valid for free streams flowing out of a cylindrical jet are summarized and compared with the results obtained by the computer simulations.

2. Theoretical solutions of free jets

A theoretical model of the outflow of air from the turbulent free stream of a circular jet is shown in figure 1. The air flows out from the outlet channel at a mean velocity U_0. On the axis of the jet at the initial interval l_1, the centreline velocity, U_m, is very close to the supply velocity, U_0. At distances from

the channel outlet that are greater than l_1 the velocity decreases. Theoretically, the free stream has the profile of a cone with a stream angle α. The top of this cone is placed at point O (pole), also called the virtual origin [6]. For circular jets, the virtual origin was found to range from $0.6d_0$ to $2.2d_0$ behind the nozzle. Because of the uncertainty involved in predicting this distance, for practical purposes it is suggested that the virtual origin be located at the nozzle itself [7]. On the boundary of the stream and the surrounding, the axial velocity decreases to zero. Here a turbulent exchange of mass takes place that causes an increase in the mass of the free stream. The basic dimensions connected to the geometry of a free stream, as given by Abramovich, can be calculated for the pole distance by [6, 8]:

$$l_0 = \frac{0.29 * r_0}{a}$$

(1)

and for the initial interval by:

$$l_1 = \frac{0.67 * r_0}{a}$$

(2)

The larger the value of a, the faster is the decay of centreline velocity. For jets of a circular cross-section, Abramovich [6, 8] recommends values of a between 0.066 and 0.076, and a slightly higher value of a = 0.089 at a higher initial turbulence. For a nozzle and uniform velocity distribution, Rajaratnam [7] recommends to use a = 0.066. As an alternative, Cihelka et al. [9] give a relationship to calculate the constant a by:

$$a = \frac{\frac{tg\alpha}{2}}{3.4}$$

(3)

According to Abramovich [10], a increases linearly with the ratio of the average to the maximum velocity at the nozzle, and the value of a also appears to increase with the turbulence level of the jet. If the velocity distribution is non-uniform, a appears to take larger values [7, 11]. The value of a also depends on the type of supply opening. According to Baturin [12], it can range from 0.066 for a convergent nozzle, up to 0.27 for a swirl diffuser with eight vanes at 45° to jet centreline. The decay of the centreline velocity can be expressed as [1]:

$$\frac{U_m}{U_0} = \frac{K_v}{\frac{x}{d_0}}$$

(4)

where K_v is a constant, usually referred to as the throw constant, and d_o is the effective diameter of the supply opening, equal to $2r_o$. The value of K_v can vary from 5.75 up to 7.32, depending on the author. The centreline velocity decay has been studied by a number of authors. Using extensive experimental data for different free axisymmetric jets, Baturin [12] obtained the velocity decay equation:

$$\frac{U_m}{U_0} = \frac{0.48}{\frac{a*x}{d_0} + 0.145}$$

(5)

The solution to produce centreline velocity decay by Tollmien [7, 13]:

$$\frac{U_m}{U_0} = \frac{0.965}{\frac{a*x}{r_0}}$$

(6)

The solution to produce centreline velocity decay by Hinze and Zijnen [7, 14]:

$$\frac{U_m}{U_0} = \frac{6.39}{\frac{x}{d_0}+0.6} \tag{7}$$

The solution to produce centreline velocity decay by Albertson et al. [7,15]:

$$\frac{U_m}{U_0} = \frac{6.2}{\frac{x}{d_0}} \tag{8}$$

For practical purposes, the value of K_v equal to 6.3, lying between the extreme variations, is suggested for the velocity scale by Rajaratnam [7]:

$$\frac{U_m}{U_0} = \frac{6.3}{\frac{x}{d_0}} \tag{9}$$

The solution to produce centreline velocity decay equations by Aziz [16]:

$$\frac{U_m}{U_0} = \frac{A_4}{\frac{x}{d_0}+\alpha_2} \tag{10}$$

where A_4 is equal to 6.3. The value of α_2 represents a correction for the virtual origin. Comparison of the various theoretical solutions, expressed as centreline velocity profiles in the fully developed flow region, are shown in figure 2. The supply air velocity, U_0, and the constant a, were determined based on a set of preliminary experimental measurements by the particle image velocimetry (PIV) method. The inputs to calculate the profiles were as follows: Baturin $U_0 = 4.2$ m/s, $a = 0.051139$; Tollmien $U_0 = 4.2$ m/s, $a = 0.066$; Hinze and Zijnen $U_0 = 4.85$ m/s; Albertson $U_0 = 4.85$ m/s; Rajaratnam $U_0 = 4.85$ m/s; Aziz $U_0 = 4.85$ m/s, $\alpha_2 = 0.6d_0$.
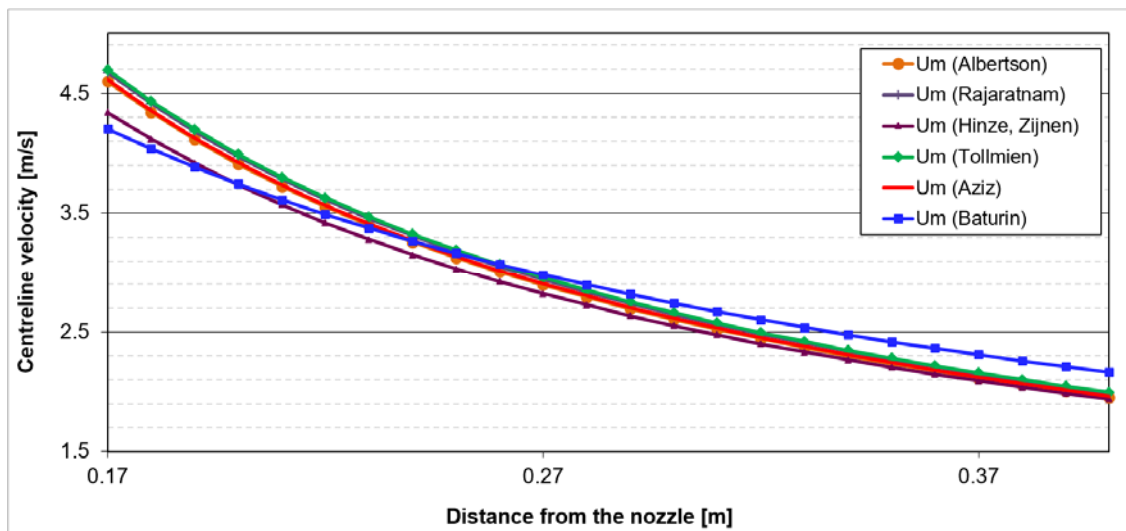

Figure 2. Centreline velocity in fully developed flow region

3. Model description

The computer program for CFD simulations ANSYS Fluent v15 was employed to calculate the turbulent air flow characteristics with various turbulence models. The simulations are based on Reynolds-

averaged Navier–Stokes equations (RANS) and five different turbulence models: the standard k-epsilon model, the k-epsilon realizable model, the standard k-omega model, the shear stress transport (SST) k-omega model, and the Reynolds stress model (RSM). The k-epsilon model has been popular for industrial applications due to its good convergence rate and relatively low memory requirements. It solves for two variables: k, the turbulence kinetic energy; and ε (epsilon), the rate of dissipation of turbulence kinetic energy. The k-omega model is similar to the k-epsilon model, but it solves for omega, the specific rate of dissipation of kinetic energy. The k-omega model is useful in many cases where the k-epsilon model is not accurate, such as internal flows, flows that exhibit strong curvature, separated flows, and jets. The SST model is a combination of the k-epsilon model in the free stream and the k-omega model near the walls. It has similar resolution requirements to the k-omega model and the low Reynolds number k-epsilon model, but its formulation eliminates some weaknesses displayed by k-omega and k-epsilon models [17].

**Figure 3.** Geometry of the model

The geometry of the simulated CFD model, representing a 2D symmetrical rotating body from which the supply air is discharged, consists of a rectangle with a width of 0.5 m and a length of 1 m, as shown in figure 3. Model with such dimensions is sufficiently large so that its walls do not influence the free stream discharged from the air supply nozzle. The supply air originates from a bottle-shaped outlet channel with a starting diameter of 0.08 m, which is gradually narrowed down to 0.026 m at the air supply nozzle where the air is discharged into the space. The air supply nozzle is located in the centreline, and its offset from the start of the rectangle is 0.29 m. The detail of the geometry and mesh of the outlet channel is shown in figure 4. The minimum orthogonal quality is $2.795*10^{-1}$. The orthogonal mesh has the maximum aspect ratio of $1.021*10^{1}$, and the face area between $8.481*10^{-5}$ and $2.542*10^{-2}$ m^2. The mesh was created in ANSYS Workbench 15.0 and subsequently adapted in Fluent, and it contains 5 781 cells. For all turbulence models, the turbulence intensity was set to 10 %. The temperature of the supply air is 25 °C, equal to the room temperature. The air velocity at the beginning of the outlet channel of the jet, thus 0.29 m behind the air supply nozzle, is 0.45 m/s. The values of air velocity at the outlet from the jet channel, U_0, for the five turbulence models were as follows: k-omega SST 4,655 m/s; k-omega standard 4,664 m/s; RSM 4,780 m/s; k-epsilon standard 4,794 m/s; k-epsilon realizable 4,779 m/s.
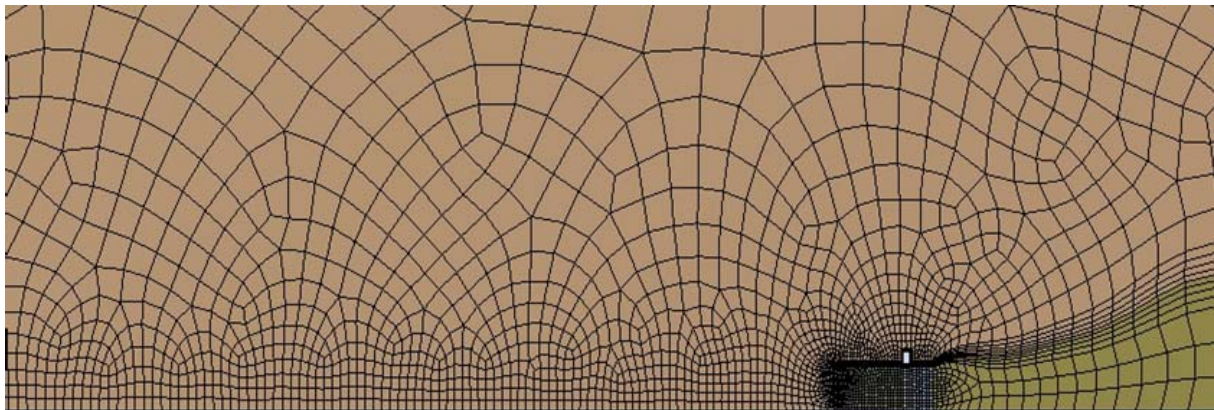
Figure 4. Detail of the geometry and mesh of the outlet channel

4. Results and discussion

In figure 5 the centreline velocities calculated by the simulation software using the five turbulence models are compared with each other and with the theoretical solutions in the fully developed flow region. The theoretical solutions are here represented by the average of the centreline velocities produced by the six theoretical solutions (5 to 10), indicated by the red curve in figure 5. The results of the simulation of the circular air jet suggest that in particular the k−omega SST formulation can be used to effectively predict the air flow characteristics. The centreline velocity profile obtained with the Reynolds stress model (RSM) is also in good agreement with the average of theoretical solutions.
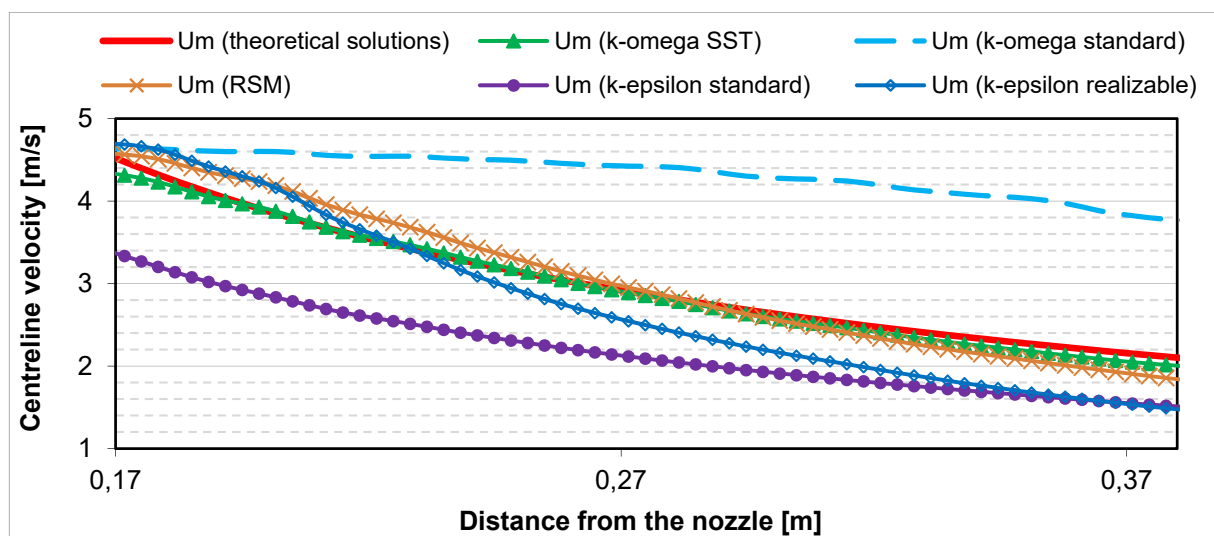
Figure 5. Centreline velocity produced by various turbulence models and its comparison with the average of theoretical solutions

Figure 6 shows discrepancies between the simulation results obtained by the five turbulent models and the average of theoretical solutions in the fully developed flow region. A discrepancy is defined as the absolute value of the difference between the average of theoretical solutions and the result obtained by the turbulent model, averaged over the distance from the nozzle of 0.17 to 0.4 m. The lowest average discrepancy of 4,06 % is represented by the k-omega SST turbulence model, followed by the Reynolds stress model (RSM), the k-epsilon realizable model, and the k-epsilon standard model, with the discrepancies of 10.23 %, 20.88 %, and 27.62 %, respectively. The least accurate approximation to the average of theoretical solutions is obtained for the k-omega standard turbulence model, with the

discrepancy of 60.62 %. The discrepancy between the simulations and the theoretical solutions partially originates from the outlet channel and nozzle geometry, characteristics of the discharged and ambient fluid, and the physical environment into which the jet is discharged, which were different in this study than in the experimental studies from which the theoretical solutions were derived.
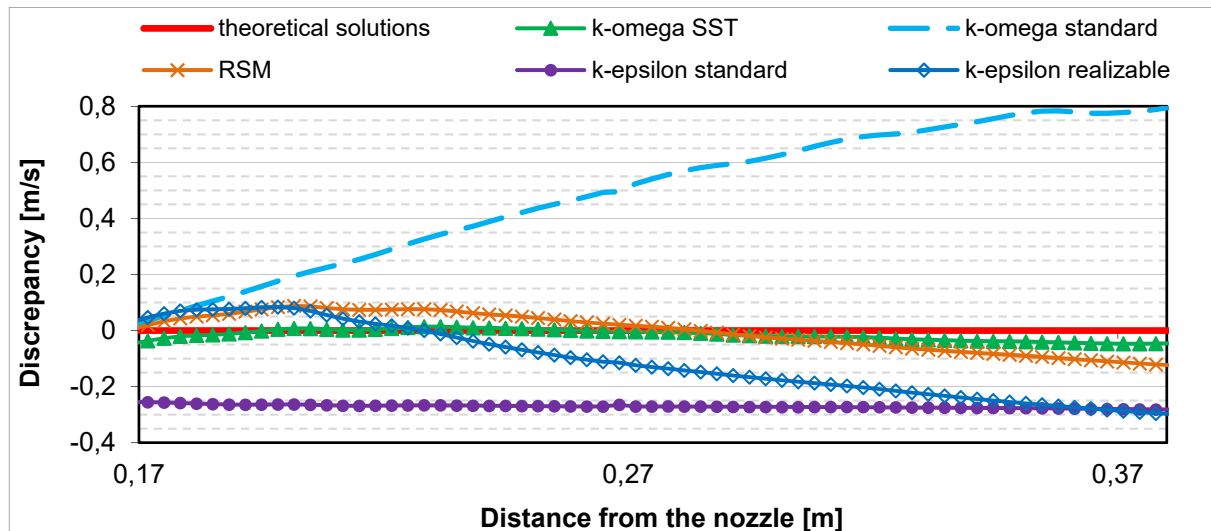
Figure 6. Discrepancies in centreline velocity obtained by CFD simulations and by theoretical solutions

The air velocity contours and the mesh for simulations with the k-omega SST turbulence model, which produced values of centreline velocity closest to the average of theoretical solutions, are shown in figure 7.
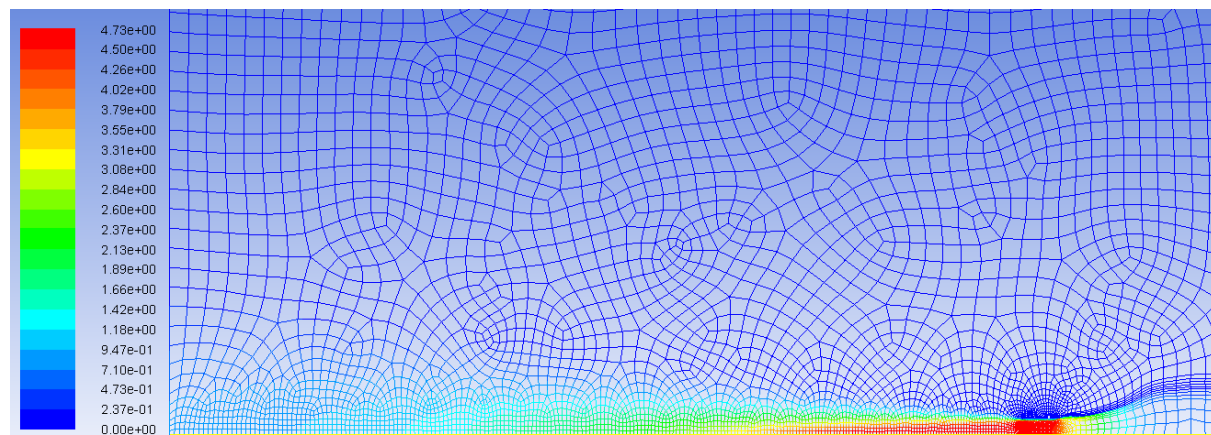
Figure 7. Air velocity contours and mesh for 2D simulations with k-omega SST turbulence model

5. Conclusions

In this study, (1) the overview of existing theoretical solutions to free air jets was presented, (2) a circular free jet was modelled in a CFD software using various turbulence models, and (3) the results obtained by the CFD simulations were compared with average of the theoretical solutions. The overall agreement between the CFD solvers and the theoretical solutions depends on the turbulence model used. The CFD simulations with various turbulence models yield results of the centreline velocity that can be very different from the results obtained by various theoretical solutions. For example, the k-omega SST turbulence model produced results very close to the average of theoretical solutions, whereas the results

obtained with the k-omega standard turbulence model differ from the average theoretical solutions by as much as 60 %. These results underline the fact that there is no universal turbulence model available, and the most suitable model for the problem at hand needs to be chosen. The next step of the research should be a thorough experimental measurement of the air flow profiles by the particle image velocimetry (PIV) method to validate the theoretical results.

Acknowledgment

References

[1]   H. Awbi, "Ventilation of Buildings", 2nd ed. London: *Spon Press*, ISBN 0-415-27056-1, 2003.

[2]   Theory of Jets [2017-10-05]. Available: http: //shodhganga.inflibnet.ac.in/bit stream/10603/6989/8/08_chapter%202.pdf

[3]   W. R. Quinn, "Upstream nozzle shaping effects on near field flow in round turbulent free jets", *European Journal of Mechanics - B/Fluids*, vol.25, pp. 279-301, ISSN 0997-7546, 2006.

[4]   F. Gori et al., "Flow evolution of a turbulent submerged two-dimensional rectangular free jet of air. Average Particle Image Velocimetry (PIV) visualizations and measurements", *International Journal of Heat and Fluid Flow*, vol. 44, pp. 764-775, ISSN 0142-727X, 2013.

[5]   Z. Yue, "Air jet in ventilation applications", Stockholm, ISSN 0284-141X, 2001.

[6]   L. Zawadzki, "Determination of the Air Velocity in the Free Stream Flowing out of a Cylindrical and Two-Gap Skewed Jet (Dual Slot Die)", *FIBRES & TEXTILES in Eastern Europe*, vol. 18, pp. 39-43, ISSN 1230-3666, 2010.

[7]   N. Rajaratnam, "Turbulent jets", 2nd ed. Amsterdam: *Elsevier Scientific Publishing Company*, pp. 27-49, ISBN 0-444-41372-3, 1976.

[8]   G. N. Abramovich, "Applied gas dynamics", *Nauka*, Moscow, 1976.

[9]   J. Cihelka et al. "Heating and Ventilation", 1st ed. Prague: *SNTL*, ISBN 04-216-75, 1969.

[10]   G. N. Abramovich, "Theory of Turbulent Jets", *MIT Press*, 1963.

[11]   S. Aloysius et al., "Comparison of flow and dispersion properties of free and wall turbulent jets for source dynamics characterisation", *Environmental Modelling & Software*, vol. 24, pp. 926-937, ISSN 1364-8152, 2009.

[12]   V. Baturin, "Fundamentals of Industrial Ventilation", *Pergamon*, Oxford, 1972.

[13]   W. Tollmien 1926 "Calculation of turbulent propagation processes", *Z.A.M.M.*, vol. 6, pp. 468-478, 1945.

[14]   J.O. Hinze, BGVDH Zijnen, "Transfer of heat and matter in the turbulent mixing zone of an axially symmetrical jet", *Appl. Sci. Res.*, vol. 1, 1949.

[15]   M.L. Albertson, Y.B. Dai, R.A. Jensen and H. Rouse, "Diffusion of submerged jets", *Transactions of the American Society of Civil Engineers*, vol. 115, pp. 639-664, 1950.

[16]   T. N. Aziz, "Numerical Simulation of Turbulent Jets", *Engineering Applications of Computational Fluid Mechanics*., vol. 2, pp. 234-243, ISSN 1997-003X, 2008

[17]   [2018-03-15]   Available: https://www.comsol.com/blogs/which-turbulence-model-should-choose-cfd-application/