Numerical simulation of the dynamics of particle motion with different sizes

Alibek Issakhov\textsuperscript{a,b}, Ruslan Bulgakov\textsuperscript{a} and Yeldos Zhandaulet\textsuperscript{a}

\textsuperscript{a}Department of Mathematical and Computer Modelling, Al-Farabi Kazakh National University, Almaty, Republic of Kazakhstan; \textsuperscript{b}Center of Mathematics and Cybernetics, Kazakh-British Technical University, Almaty, Republic of Kazakhstan

**ABSTRACT**

In this paper, was presented the numerical simulation of air pollution by particles with different sizes from the Ekibastuz State district power plant-1 (Ekibastuz SDPP-1), which is located on the northern side of Zhngyldy Lakeshore, the Republic of Kazakhstan in real scales using the three-dimensional mathematical model. It was found that the deterioration of the environment is due to the release of a large amount of SO\textsubscript{x}, NO\textsubscript{x} and the volatile particles of Suspended Particulate Matter and Respirable Suspended Particles matter (SPM and RSPM), which cause human and animal diseases. For the adequacy of the mathematical model, the test problem was solved. The experimental data were used to evaluate the applicability of the mathematical model and the numerical algorithm for the test problem. The obtained numerical simulation results are in good agreement with the numerical results of other authors and the experimental data. In addition, to select the optimal turbulent model, the obtained simulation results for different turbulent models were compared with experimental data. Moreover also the boundary conditions for turbulent models ($k - \varepsilon$, $k - \omega$), boundary conditions for turbulent kinetic energy were chosen to match the experimental data.

**ARTICLE HISTORY**

Received 25 July 2018
Accepted 4 November 2018

**KEYWORDS**

Large eddy simulation (LES); Reynolds averaged Navier–Stokes (RANS); detached eddy simulation (DES); jet in crossflow; particle dispersion; thermal power plant

**Introduction**

The current state of the atmosphere and the biosphere is critical. The reasons for the approach of mankind to an ecological catastrophe are many factors directly affecting the stable prosperity of the planet. To date, one of the main problems is polluted air and the globe atmosphere. To determine the most optimal solution to these problems, it is necessary to consider pollution sources their location and physical characteristics. One such is thermal power plants (TPP), vehicles, factories, enterprises. These pollution sources throw dust, soot, ash and other waste into the atmosphere, which pollutes not only air but also soil, by settling on the ground. All waste materials are the smallest solid particles invisible to the naked eye.

In 2013, specific pollutants such as lead and its compounds amounting to 572.4 tons entered into the air basin of the Republic of Kazakhstan, 186.2 tons of manganese and its compounds, 336.7 tons of copper oxide, 346.7 tons of sulfuric acid, arsenic – 121.8 tons, chlorine – 53.3 tons, mercury – 189 kg (SAMRYK ENERGY, 2013). The actual release of these substances also did not exceed the volume of the established MPE (maximum permissible emission) norms. From the total amount of pollutants discharged into the air, 85.2% were gaseous and liquid substances, 14.8% were solid (SAMRYK ENERGY, 2013). The data show that the quality of atmospheric air in large cities often exceeds the norms established in the health field, and leads to significant expenditures in the public health field in the country (WHO, 2002). Pollution by particulate matter is considered one of the most serious air pollutants in health effects terms, and even small steps to reduce its level can lead to significant benefits. It can be concluded that reducing the particulate matter level by at least 60 $\mu$g/m\textsuperscript{3} will lead to an annual savings of $57 million in health care through reducing premature mortality and increasing labor productivity (Environmental, Health, and Safety Guidelines for Thermal Power Plants, 2017; Wilson & Spengler, 1996).

The problems associated with the movement dynamics of the smallest solid and liquid particles in the air are widespread in science (aerosols, dust propagation in the engine, ash dynamics during a volcanic eruption, etc.). For an example of the need to study the distribution of harmful microparticles in the air, it is worth mentioning that, from 1990 to 2013, the pollutant emissions bulk into the air in the Republic were accounted for by sulfur dioxide and solid particles. The greatest volume of dust, sulfur dioxide and nitrogen oxide emissions falls on three main
sectors of Kazakhstan’s economy: electric power, mining and transport (SAMRYK ENERGY, 2013) (Figure 1).

Particulate matter is an airborne contaminant that contains both solid microparticles and tiny liquids droplets. And these solid microparticles and tiny liquids droplets in size from about 10 nm to 2.5 µm. Other designations and names for PM2.5 particles: fine particulate matter (Flermoneca, 2013; Wilson & Spengler, 1996). All these particles and droplets smaller than 2.5 µm are suspended in the air. They are in the forest and the sea, but those that are in the city that they represent the greatest danger. Firstly, there are usually more of them in the city, and secondly, the chemical composition of the fine aerosol in the city is more dangerous than in nature. It should be noted that in different cities, the composition of the PM2.5 aerosol and the parameters of individual particles may differ greatly (Wilson & Spengler, 1996).

Note that in the world there is no single standard of maximum permissible values of PM10, for each country this rate is different. In Kazakhstan, the maximum permissible daily average concentration of PM10 is 60 µg/m³, with this value a person can safely breathe this air for years. The maximum permissible single concentration of PM10 is 300 µg/m³, this concentration by inhalation for 20–30 min should not cause reflex reactions in the body (WHO, 2002).

Priority pollutants include PM10 and PM2.5 suspended solids, NOx, SO2 and ozone (O3) (Wilson & Spengler, 1996). Nitrogen dioxide and sulfur dioxide, as well as suspended, substances are included in the national air quality monitoring system from the presented list of priority pollutants. Nitrogen dioxide, sulfur dioxide and PM2.5 were identified as priority pollutants. The exception of PM10 is explained in the papers (Akbarian et al., 2018; Ardabili et al., 2018; Wilson & Spengler, 1996), in which the authors indicate the limited effect of these particles on the human body. Consequently, suspended particles of a given size do not represent a hygienic value, such as PM2.5, which in turn penetrate the upper and lower respiratory tract. If the dust particle size is greater than 10 µm, then it practically does not penetrate into the respiratory system. Particles size from 10 to 7 µm accumulate in the nasal cavity, into the throat penetrate in size from 7 to 4.7 µm, in the trachea and primary bronchi – 4.7–3.3 µm, in the secondary bronchi – 3.2–2.1 µm. In the deep part of the bronchi, dust accumulates if the particle size is 2.1–1.1 µm, it enters the bronchioles if they are 1.1–0.65 µm, and the particles in the lung alveoli penetrate 0.65–0.43 µm (Flermoneca, 2013). A similar kind of problem was solved in Vasistha (2014). In this study, the pollutants impact on the environment, the sources of which are thermal power plants, was shown. Pollutants, especially fly ash, have several uses to reduce environmental pollution and economic development through social needs such as the production of bricks, cement, ceramics, paving, sidewalks, embankments, etc. Fly ash can be used as a fertilizer, polymer products to make the system economical and minimize pollution of the environment, soil and water.

In this paper, it was studied the problem related to pollution from the activities of the coal-based Ekbastuz State district power plant-1 (Ekbastuz SDPP-1), which is located on the Zhyngyldy Lake, at 16 km from the Ekbastuz city, Pavlodar region, the Republic of Kazakhstan. Since these types of pollutants are widely used in thermal power plants in Kazakhstan, this problem is acute. To solve these problems, some solutions are given.

Pokale (2012) was shown that thermal power stations have a very strong influence on the ecological part of the surrounding region. Environmental degradation occurs due to the constant release of large amounts of SOx, NOx, PM (PM2.5, PM10) and RSPM, which are distributed within a radius of 25 km and cause respiratory diseases in humans and animals. This also affects the photosynthesis process, the minerals balance, micro and basic nutrients in plants, soil layers, structures and buildings are exposed to corrosive reactions.

From the fundamental and application point of view, the study of turbulent jets in a crossflow is an important problem due to the diverse phenomena of fluid dynamics, as evidenced in Mahesh (2013). The reason is primarily in countless applications in which this flow pattern is detected, including pollutants emissions, dilution streams in combustion chambers and film cooling. The latter, perhaps, is the most effective and most widely used cooling technology in the thermal design of gas turbines.

The turbulent jet in a transverse flow is basically a three-dimensional flow field. The flow becomes complex due to the presence of large coherent structures and the strong pressure gradient along the jet and sidewall effects (Acharya, Tyagi, & Hoda, 2001). Flow behavior and heat

![Figure 1. The schedule of emissions of the main pollutants into the atmosphere for 1990–2013. (1000 tons/year)](image-url)
transfer analysis of the turbulent jet in a transverse flow can be found in the papers (Carter, 1969; Hwang & Chiang, 1995; Sarkar & Bose, 1995; Shi, Ray, & Mujumdar, 2003). In the papers (Flacks, Dullenkopf, & Scherer, 1994; Haniu & Ramaprian, 1989; Hoda & Acharya, 2000; Kalita, Dewan, & Dass, 2002; Karagozian, 2014; Keimasi & Taebi-Rahni, 2001; McGuirk & Rodi, 1978; Mou, He, Zhao, & Chau, 2017; Ramaprian & Haniu, 1983), only the behavior of a plane jet in a transverse flow is investigated. In most papers modeling of the turbulent jets in a transverse flow velocity and scalar fields, mainly concern canonical and simplified cases. In the paper (Muppidi & Mahesh, 2008), a normal jet was extracted from a long tube into a laminar boundary layer. They obtained an important understanding of scalar transport, especially from the turbulent flows point of view. However, the jet dynamics in the turbulent transverse flow is very different.

Despite the fact that there are many studies on single-phase jets in a crossflow, there are fewer studies on gas particles jets in the crossflow. One of the first to study this problem was Edelman, Economos, and Boccio (1971), this paper was studied a jet in a crossflow with gas particles. Graphite was used as the basis of the particles. Salzman and Schwartz (1978) and Han and Chung (1992) measured 15 μm of silicate particles in an air jet that was introduced into the crossflow. Yi and Plesniak (2002) experimentally investigated the jet in a crossflow with spherical and non-spherical gas particles with different diameters. Campolo, Salvetti, and Soldati (2005) numerically investigated the mechanisms of scattering of inertial particles in a jet with a crossflow. Ahmed et al. (2007), Achim, Naser, Morsi, and Pascoe (2009) and Tian, Witt, Schwarz, and Yang (2010) numerically modeled coal combustion in boilers with tangential firing with standard $k-e$ turbulent models.

Works devoted particle distribution analysis using two-dimensional and three-dimensional numerical modeling can be found in the papers (Balachandar & Eaton, 2010; Monchaux, Bourgoin, & Cartellier, 2012). In most of these studies considered the particles propagation in turbulent flows (Ahmed & Elghobashi, 2001; Goto & Vasilicos, 2008), channels (Fessler, Kulick, & Eaton, 1994; Liu, Ji, Xu, Xu, & Williams, 2017; Marchioli et al., 2008; Rouson & Eaton, 2001) and tubes (Marchioli, Giusti, Salvetti, & Soldati, 2003; Vreman, 2007). With a few exceptions gravity is taken into account (Armenio & Fiorotto, 2001; Marchioli, Picciotto, & Soldati, 2007), and gravitational forces are the only forces considered in the balance of the particles forces. In the paper (Chung & Troutt, 1988) for numerical study the particles dispersion in an axisymmetric jet was used the large eddy method (LES). In this work was proposed that the particle dynamics is controlled by large-scale structures in the near flow field of a circular turbulent jet. These particles dispersions strongly depend on the dimensionless Stokes number $(St)$, which is defined as

$$St = \frac{t_0 \mu_0}{l_0},$$

where $t_0$ is the particle relaxation time, $\mu_0$ is the fluid velocity of the flow well away from the obstacle and $l_0$ is the obstacle characteristic dimension.

The particles with large dimensionless Stokes numbers are scattered less and can be thrown away of the mixing range of a turbulent jet stream. These results are close to the experimental data (Longmire & Eaton, 1992). Also, the authors have suggested that the existence of certain intervals of the intermediate state, wherein the optimal particles dispersion in a turbulent mixing layer free jet can be achieved.

Balachandar and Eaton (2010) provide an overview point particles approaches to a multiphase turbulent flow. The proposed point-particle method for the Stokes number is greater than 1 in comparison with various methods for turbulent multiphase flows is suitable. Also in this paper, an estimate was made of the Stokes number for determining the application of the point-particle method and it was found that the method of point particles can be used more effectively for very heavy particles matter (PM10).

The aim of this work is to simulate the motion dynamics of solid particles with different sizes (from $dp = 0.1 \mu m$ to $dp = 10 \mu m$), as well as the analysis of different turbulent models for the optimal combination of accuracy and speed of the problem with the ANSYS Fluent software for calculating the motion equations for the viscous incompressible medium and performing efficiency analysis of different turbulent methods.

### Mathematical models

In the paper (Margason, 1993) a detailed description of the flow field from a jet in a crossflow can be found. In the papers (Fric & Roshko, 1994; Kelso, Lim, & Perry, 1996) the velocity field was numerically simulated, and the field of the passive scalar mass fraction was numerically simulated in the papers (Shan & Dimotakis, 2006; Su & Mungal, 2004). Numerical methods were used to solve all these problems. So for simulation Hasselbrink and Mungal (2001), Muppidi and Mahesh (2005) and Muppidi and Mahesh (2007) were used the Reynolds averaged Navier–Stokes equations (RANS) and the obtained results were compared with the experimental data. In the papers (Acharya et al., 2001; Chai, Iyer, & Mahesh, 2015; Chochua et al., 2000; Schluter &
Schonfeld, 2000; Yuan, Street, & Ferziger, 1999) for modeling this problem, direct numerical simulation (DNS) was used and obtained numerical results were also compared with the experimental data. However, it should be noted that this method requires large computational resources, which is not entirely acceptable when solving applied problems on a real scale. As a result, the $k - \varepsilon$ turbulence model was used in this paper. To describe this process, the Navier–Stokes equations are used, which consist of the continuity equation and the momentum equation.

$$\frac{\partial u_i}{\partial x_j} = 0,$$

(2)

$$\frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} = f_i - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu_{\text{eff}} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right),$$

(3)

where $\mu_{\text{eff}}$ is the effective viscosity, $p$ is the pressure, $\mu_{\text{eff}} = \mu + \mu_t$, where $\mu_t$ is the turbulence viscosity. The external force of the body considered is gravity, so that $f = \rho g$, where $g$ is the acceleration due to gravity, and $\rho$ is the density.

The kinematic relationship between the particles position and the particle velocity is

$$\frac{dx_p}{dt} = u_p,$$

(4)

$$m_p \frac{du_p}{dt} = F_D + F_G,$$

(5)

where $x_p$ is the particles location, $u_p$ is the velocity of particles, $u_f$ is the velocity of fluids, $m_p$ is the mass of particles, $F_G$ is the gravity force, $F_D$ is the drag force, and $F_D$ is calculated as follows:

$$F_D = \frac{1}{2} \rho_f \frac{\pi d_p^2}{4} C_D (u_f - u_p) |u_f - u_p|,$$

(6)

where the resistance coefficient

$$C_D = \begin{cases} \frac{24}{Re} & (Re < 1), \\ \frac{24}{Re} (1 + 0.15 Re^{0.687}) & (1 \leq Re \leq 1000). \end{cases}$$

(7)

For testing, four turbulent models were identified:

(1) The turbulent model $k - \varepsilon$ is a model with two equations and with a vortex viscosity. Two equations for the kinetic turbulent energy and the kinetic turbulent energy dissipation are added to the basic group of Reynolds equations (Jones & Launder, 1972). The system of equations is as follows:

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \frac{\partial k}{\partial x_j} \right] + \rho g + P_{kb},$$

(8)

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{\varepsilon}{k} (C_{1k} P_k - C_{2k} \rho g + C_{1k} P_{gb}),$$

(9)

where $\mu_t = \frac{k}{\omega}$,

The constants have the following meanings (Launder & Spalding, 1974):

$$C_{1k} = 1.44, C_{2k} = 1.9, \sigma_\varepsilon = 1.2, \sigma_k = 1, C_3 = 1,$$

where $v'$ is the velocity component along the $Y$-axis parallel to gravity, is the velocity component along the $X$-axis perpendicular to gravity.

(2) The turbulent model SST $k - \omega$ (Menter, 1994) is a model with two equations and with a vortex viscosity. The shear stresses transfer (SST) connected the best from all sides. The use of this turbulent model in the inner parts of the boundary layer allows it to be directly relevant to the wall itself through a viscous layer, for this reason, the SST $k - \omega$ model can be used as a ‘turbulent Low-Re model’ without additions or additional damping functions:

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \frac{\partial k}{\partial x_j} \right] + P_k - \beta^* \rho k \omega + P_{kb},$$

(10)

$$\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho \omega u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \frac{\partial \omega}{\partial x_j} \right] + \frac{\omega}{k} P_k - \beta \rho k \omega^2 + P_{\omega b},$$

(11)

where
\[ P_k = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \left( 3\mu_t \frac{\partial u_k}{\partial x_k} + \rho k \right), \]

\[ P_{ob} = \frac{\omega}{k} ((\alpha + 1)C_3 \max(P_{kh}, 0) - P_{kh}). \]

The constants of the model are defined as follows (Wilcox, 1988):

\[ \sigma_k = 2, \sigma_{\omega} = 2, \beta = 0.075, \beta^* = 0.09, \alpha = 5/9. \]

(3) The idea of the DES model (Menter & Kuntz, 2003; Spalart, Jou, Stretlets, & Allmaras, 1997) is to combine the best properties of the RANS and LES methods into one method. This method attempts to simulate the near-wall areas using the RANS method and simulate the rest of the flow using the LES method:

\[ \varepsilon = \beta \ast k\omega = k^{3/2}/L_t \rightarrow k^{3/2}/(C_{DES}\Delta) \]

for \((C_{DES}\Delta < L_t)\),

\[ \Delta = \max(\Delta_i); L_t = \left( \sqrt{k} \right) / \beta \ast \omega. \]

The main reason for setting the maximum edge length in the turbulent DES model is that the model must return to the RANS model in fixed boundary layers. The DES-modification can be formulated as a factor to the destruction term in the \( k \)-equation (Issakhov, 2014; Issakhov, Zhandaulet, & Nogaeva, 2018; Menter & Kuntz, 2003; Spalart et al., 1997):

\[ \varepsilon = \beta \ast k\omega \rightarrow \beta \ast k\omega \cdot F_{DES}, \]

where

\[ F_{DES} = \max\left( \frac{L_t}{C_{DES}\Delta}, 1 \right) \quad \text{and} \quad C_{DES} = 0.61. \]

Since the limit should be only in the model area. The numerical scheme at the same time changes between the forward-shifted and central difference scheme in RANS and DES, respectively.

(4) Turbulent flows contain a wide range of lengths and timescales, with large-scale movements usually more vigorous than small-scale ones. Forecasting of industrially important fluctuating flow problems can be performed using the LES technique. LES is an approach that solves for large-scale fluctuating motions and uses turbulence models of subnet scale for small-scale motion (Issakhov, 2015, 2016a, 2016b, 2017a, 2017b).

### The numerical algorithm

Numerical algorithm SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) is used for the numerical solution of the system of equations (2)–(3). This algorithm can be represented in the following sequence of steps (Patankar, 1980):

1. Representations of the initial pressure \( P^0 \) and give fields \( P^* = P^0, t = 0 \).
2. Definitions of the initial velocity field \( u^0, v^0 \).
3. Solving the momentum equations for obtaining \( u^*, v^* \).
4. Solving the equation for \( P^* \) and calculating \( P \) by adding \( P^* \) to \( P^0 \).
5. Finding \( u, v \) by means of formulas for velocity correction.
6. In the case \(|P^*|\) that there is little in all nodes of the computational grid, then we assume \( P^0 = P, u^0 = u, v^0 = v, t = t + \Delta t \). Otherwise, it uses what was found \( P \) like \( P^* \) and go to step 3.
7. If, \( t < T_{max} \) then we have to return the step 3.

The SIMPLE procedure was successfully used to solve a number of problems in the incompressible fluid flows calculation.

For Equations (4)–(5) and additional equations for the turbulent model ((8)–(9) or (10)–(11)) were solved by Euler implicit discretization and explicit time discretization, respectively. For all additional equations in ANSYS FLUENT, explicit methods are used in the finite difference discretization, which are computed in the previous time step.

All simulations were done in the PC (Intel Core i7, 2.80 GHz, 3 GB RAM) and the cluster system with 312 cores (Intel(R) Xeon(R) CPU E5645, 2.40 GHz, 624 GB RAM) is used to decrease CPU time.

### Implementation of the test problem

To test 3D the mathematical model of the reactor was constructed based on the data provided in the paper (Keimasi & Taeibi-Rahni, 2001).

The reactor consisted of two channels (Figure 2):

- Horizontal channel has a rectangular shape, with a width equal to 3D (where \( D \) is the vertical channel diameter, which is equal to 12.7 mm), a reactor length 45D and a reactor height 20D.
- Vertical channel has a square shape, with a diameter of \( D \) and a height of 5D.
The vertical channel is located exactly in the center of the Z-axis and is displaced by a distance equal to 5D from the beginning of the horizontal channel.

To obtain the most accurate numerical result, the structured computational grid was constructed with the greatest accuracy as in Figure 3 with dimensions (Table 1).

The boundary conditions were taken absolutely identical, as in Keimasi and Taebi-Rahni (2001), while the velocity was set at the entrance to each of the channels (Figure 4).

Horizontal channel: on the boundary layer, with a height equal to $h = 2D$, the velocity along the X-axis was given by the $1/7$ power law. At a height above
Table 1. Structured computational grid dimensions.

|                         | Number of points along the X-axis | Number of points along the Y-axis | Number of points along the Z-axis | Number of elements |
|-------------------------|-----------------------------------|-----------------------------------|-----------------------------------|--------------------|
| Horizontal channel      | 315                               | 140                               | 21                                | 872,920            |
| Vertical channel        | 7                                 | 35                                | 7                                 | 1224               |
| Overall number of elements |                                  |                                    |                                    | 874,144            |

2D, the velocity along the X-axis was set at a constant of 11 m/s.

Vertical channel: a constant velocity along the Y-axis was set equal to 5.5 m/s. Such velocity values were set to repeat the experiment carried out in the paper (Ajersch, Zhou, Ketler, Salcudean, & Gartshore, 1995).

Also, conditions on the \((k - \varepsilon)/(k - \omega)\) turbulent models components were given at the horizontal flow boundary inlet.

For the \(k - \varepsilon\) turbulent model was used following boundary conditions for kinetic turbulent energy:

\[
k = \begin{cases} 
0.3 + 0.3 \left( \frac{0.008 - y}{0.0254} \right)^{1/7} V_{\text{hor}} & y < 0.008, \\
0.3 \cdot V_{\text{hor}} & y = 0.008, \\
0.3 - 0.3 \cdot \left( \frac{y - 0.008}{0.0254} \right)^{1/7} V_{\text{hor}} & 0.008 < y < 0.0254, \\
0.0012 \cdot V_{\text{hor}} & y > 0.0254.
\end{cases}
\]

In this work, for a turbulent kinetic energy, a profile was derived. It depends on the initial velocity to obtain the maximum correspondence with the experimental data,
which was obtained by the paper (Ajersch, Zhou, Ketler, Salcudean, & Gartshore, 1995).

The boundary condition for the turbulent energy dissipation was taken like that (Versteeg & Malalasekera, 1996)

$$
\varepsilon_{in} = C_\mu^{3/4} \left( \frac{k^{3/2}}{L_m} \right),
$$

where $C_\mu = 0.09$ is the experimental constant.

For boundary conditions acting as a ‘wall’, the standard conditions proposed by the ANSYS Fluent software were chosen. On the $Z$-axis, periodic boundary conditions were established for the horizontal channel.

During the calculations with the $k - \omega$ turbulent model, the following boundary conditions were set. For the turbulent energy dissipation rate, the formula (Menter, 1994)

$$
\omega = (1 \rightarrow 10) \left( \frac{V_{hor}}{L} \right),
$$

where $L$ is the reactor length.

Also, special attention was paid to the flow boundary. At the outlet, the horizontal acceleration was set to 0.

On Figures 5–8 comparisons of the numerical results of turbulent kinetic energy and profiles for the horizontal, vertical and transverse velocity components with the computational data from paper (Keimasi & Taeibi-Rahni, 2001) and the experimental data (Ajersch et al., 1995).

**Figure 6.** Comparison of the horizontal velocity component profiles at the point $x/D = 3$ and $z/D = 0$ with the computational data (Keimasi & Taeibi-Rahni, 2001) and the experimental data (Ajersch et al., 1995).

**Figure 7.** Comparison of profiles for the vertical velocity component at the point $x/D = 3$ and $z/D = 0$. 
Figure 8. Comparison for the transverse velocity component profiles at the point $x/D = 3$ and $z/D = 0$ with the computational data (Keimasi & Taeibi-Rahni, 2001) and the experimental data (Ajersch et al., 1995).

Figure 5 shows the comparison of turbulent kinetic energy profiles of different turbulent models with the numerical results and experimental data of other authors (Ajersch et al., 1995; Keimasi & Taeibi-Rahni, 2001). The conclusion can be drawn from the numerical results shown in Figure 5. Comparing the turbulent kinetic energy profiles with the numerical results and experimental data of other authors (Ajersch et al., 1995; Keimasi & Taeibi-Rahni, 2001), one can see a difference in the numerical results even when using the $k-\varepsilon$ and $k-\omega$ SST turbulent models. This difference can be explained by the use of a 500,000 computing grid in the paper (Keimasi & Taeibi-Rahni, 2001), whereas in this paper a grid element more than 870,000 is used. The numerical results obtained with $k-\varepsilon$ and $k-\omega$ SST turbulence models are in well coincide with the experimental results (Ajersch et al., 1995), than the numerical results obtained from the paper (Keimasi & Taeibi-Rahni, 2001). Figure 6 shows that the horizontal velocity profiles almost for all turbulent models are in well coincide with the experimental results (Ajersch et al., 1995).

From Figure 7 can be seen that the vertical velocity component profiles for different turbulent models describe behavior well, but apart from the DES and LES models, deviations from experimental data appear in all turbulent models (Ajersch et al., 1995). Among the

Figure 9. Comparison of kinetic energy profiles for different points $x/D = 0$ and $z/D = 0$, $x/D = 3$ and $z/D = 0$ with numerical results without particles.
Figure 10. Comparison of velocity profiles for different cross sections with numerical results without particles.
turbulent models (Figure 7), it is necessary to distinguish the turbulent models DES and LES, the numerical results of these models are very close to the experimental data (Ajersch et al., 1995), both for kinetic energies and for all velocity components. It can be noted from Figure 8 that the numerical results of the DES and LES models of the transverse velocity component are in well coincide with the experimental results (Ajersch et al., 1995). During the turbulent models testing (Figures 5–8), the most accurate and optimal model was the turbulent models DES and LES. However, it is worthwhile to check the effect of the particles introduced into the reactor and to see the turbulent models effectiveness.

Analysis of numerical simulation results with particles

After approbation of the mathematical model and numerical algorithm, the problem of the particles propagation with different sizes (from $d_p = 4.3\mu m$ to $d_p = 10\mu m$) is considered. For this problem, it is assumed that particles with different dimensions (from $d_p = 4.3\mu m$ to $d_p = 10\mu m$) are ejected from the vertical channel in just the same way as the turbulent flow in the transverse direction. These particle sizes were taken, since these are the most dangerous dimensions for human health (WHO, 2002).

Let us first consider the numerical results for the turbulent model $k - \varepsilon$:

It can be seen from Figure 9 that the particles appear in the vertical channel affected the kinetic energy profile. This phenomenon is due to the fact that the used area is very small and the small particles are quite significant under the given conditions. Figure 10 shows the comparisons of horizontal, vertical and transverse velocity profiles for different cross sections with and without particles. In both cases (with and without particles), velocity components profiles are practically the same. However, for some cross sections, changes in the horizontal, vertical, and transverse velocity components profiles are sufficiently noticeable, this phenomenon is due to the fact that particles with different sizes affect the overall flow.

The second one is the numerical results for the turbulent model $k - \omega$:

Similarly, as in the $k - \varepsilon$ turbulent model case, in Figure 11, it is possible to observe deviations in the kinetic energy profiles with the injected tiny particles into the vertical flow. From the numerical results in Figure 12, it is possible to observe small changes in the horizontal, vertical, and transverse velocity components profiles with the smallest particles. These changes are influenced by the particles.

The third is the numerical results for the DES turbulent model:

In the case of simulating the particles motion dynamics with different sizes through the DES turbulent model, one can observe a much greater difference between the two cases (Figure 13), in the presence of particles and without them.

The differences in Figures 13 and 14 are due to the fact that DES model is a modernized SST model whose problems are modeling of turbulent flows, reducing the turbulence length effect by replacing it by the distance

**Figure 11.** Comparison of profiles for kinetic energy for different points $x/D = 0$ and $z/D = 0$, $x/D = 3$ and $z/D = 0$ with numerical results without particles.
Figure 12. Comparison of velocity profiles for different cross sections with numerical results without particles.
Figure 13. Comparison of profiles for kinetic energy for different points $x/D = 0$ and $z/D = 0$, $x/D = 3$ and $z/D = 0$ with numerical results without particles.

between grid cells in areas where it reaches maximum dimensions.

The fourth turbulent model is the numerical results for the LES model.

From the results of Figure 15, it could be seen that the velocity profiles with and without particles are approximately the same except for small deviations for some sections.

From the numerical results, it can be noted that the turbulent models DES and LES describe this problem more correctly than for the $k - \varepsilon$ and $k - \omega$ turbulent models. Since from the numerical results of the $k - \varepsilon$ and $k - \omega$ turbulent models for the velocity components, it can be seen that the velocities with the particles are a little larger than without particles, that the physically is not entirely correct. However, using the DES and LES turbulent models, one can observe the reverse picture that when a particle is introduced in a turbulent flow, the velocity components are lower than without them, which is physically more correct, since the particles create additional resistance and interfere with the flow. And also the simulation time for various turbulent models was estimated, since this indicator also plays a very important role. Table 2 clearly demonstrates the superiority over time of the turbulent model. However, one should take into account that the numerical results for the test problem from turbulent models DES and LES showed the numerical results close to the experimental data (Ajersch et al., 1995) in comparison with the other turbulent models ($k - \varepsilon$ and $k - \omega$). However, DES and LES turbulent models lose in the time spent in the modeling of that kind of problem.

**Modeling a problem with a modified geometry**

This section considers the geometry with a change in the layout of the vertical pipe (Figure 16). In this problem statement, the pipe was placed within the design area, and its vertex was determined as the vertical flow source. All dimensions have remained the same as the previous case, including the location of the vertical pipe relative to the horizontal and vertical boundaries of the calculated area. The computational grid constructions and discretization were also left as previously. The boundary conditions were not subject to absolutely any changes. Below are the simulation results using the DES turbulent model. Taking into account the numerical results obtained from the test problem, only the DES turbulent model was used in this simulation, since the DES and LES turbulent models showed the best numerical results in comparison with the $k - \varepsilon$ and $k - \omega$ turbulent models. And the difference between the numerical results obtained with the DES and LES turbulent models is minimal, but the time spent for modeling this process with the LES turbulent model is approximately almost 70%, 30% and 30% higher than for the $k - \varepsilon$, $k - \omega$ and DES turbulent models, respectively.

Figure 17 shows the horizontal velocity and turbulent kinetic energy profiles along the $Y$-axis at the point $x/D = 0$ and $z/D = 0$. Whereas Figure 18 shows the horizontal velocity and kinetic energy profiles along the $Y$-axis at the point $x/D = 3$ and $z/D = 0$. Furthermore, Figure 19 shows the vertical velocity and turbulent kinetic energy profiles along the $Y$-axis at the point $x/D = 0$ and $z/D = −1$. Figure 20 shows the vertical velocity and turbulent kinetic energy profiles.
Figure 14. Comparison of profiles for speeds for different cross sections with numerical results without particles.
Figure 15. Comparison of velocity profiles for different cross sections with numerical results without particles.
Table 2. Comparison of the time for solving by different turbulent models with the SIMPLE method.

| Turbulent model | Time without particles | Time with particles | Number of iterations |
|-----------------|------------------------|---------------------|----------------------|
| $k - \varepsilon$ | 3 min 27 s | 3 min 35 s | 108 |
| $k - \omega$     | 7 min 50 s | 8 min 00 s | 224 |
| DES             | 7 min 27 s | 8 min 11 s | 160 |
| LES             | 9 min 18 s | 10 min 27 s | 250 |

Figure 16. Geometry of the new calculation area.

The behavior of 3D turbulent flows is shown in Figures 23 and 24, where one can see the vortices dynamics and the particles settling. In this case (Figure 23) in the region located behind the vertical pipe, relative to the horizontal airflow, vortices are formed. These vortices are used to map non-standard trajectories of particle motion. These vortex motions are obtained due to the fact that the main flow surrounds the pipe and forms two large vortices at the behind. These vortices collide with each other, and after dissipate to more small eddies. With dissipation, many vortices begin to move irregularly, and after the flow, the particles also made an irregular motion. This phenomenon can be observed in Figure 24, behind the pipe of particles in the flow are moving irregularly. Also, the smaller particles were affected by the fact that the larger particles, having a larger resistance area, and the vertical airflow created a kind of barrier (Figure 24), which resulted in the formation of a velocity reduction area.

Moreover from Figure 24 one can notice the instantaneous nature of the swirl in the cross-section of the flow, together with the position of the particles. Here, for comparison, particles with different sizes (from $dp = 4.3\, \mu m$ to $dp = 10\, \mu m$) are used. In general, the dispersion of particles is controlled by three mechanisms: (1) inertial (2) vortex and (3) centrifugal (Uthuppan & Aggarwal, 1994). The vortex mechanism is responsible for the particle dispersion with very low inertia, such as the particles $dp = 4.3\, \mu m$, shown in Figure 24, that the particles exhibit dispersive behavior similar to the tracer particle behavior. It can be noted from Figures 23 and 24 that behind the pipe middle in the range from 4.3 to 10 $\mu m$, the particles are located to the near axis, because in this region it can be seen from the figures that the main flow has not large velocities.
Sbrizzai, Verzicco, Pidria, and Soldati (2004) also describe how particles are brought into the transition region with external flows by vortex rings, and after the particle follow the flow, so that they are almost particle-flow indicators. Along the flow direction, the most particles are collected predominantly in certain regions behind of the pipe. As can be noted from the numerical simulation results, inertial particles are mainly concentrated in areas with low vorticity (Squires & Eaton, 1990). It can be seen from Figure 24 that for particles with a larger inertia (size 10 μm), apparently, they do not affect local vortex structures. It can be noted that for the particles with the larger size, the inertia mechanism is still the main cause of particle dispersion. And the particles with intermediate sizes, shown in Figure 24, these dispersion data have the interesting phenomenon. The inertial mechanism in this kind of dispersion is still effective in the dispersing particles process of a given particle size. In Figure 24, the particles are located close to the axis when the vortex structures are not completely developed. In the transition region, during the flow development, more and more streamwise vortices of different sizes are formed. Some particles which have not very small and not very large size, in which it is assumed that the particle dispersion is controlled by a centrifugal mechanism, are brought into the external flow by flow vortices. It
can be noted from the Figures 23 and 24 that along the flow direction can be found the region where the particles are collated mainly in specific places. The particle dispersion for the medium size particles is controlled by the combined effect: the centrifugal and the inertial mechanisms.

**Simulations results for real sizes**

After checking the mathematical model, numerical algorithm and testing of different turbulent models, this mathematical model and numerical algorithm are used in simulation the particle propagation from the pipe of the thermal power plant of the Ekibastuz SDPP-1 (Ekibastuz State district power plant-1) was carried out. Thermal power plant consists of several buildings, the length of the main building is 500 m, width is 132 m, and height is 64 m. Also, thermal power plant has two chimneys, the first chimney has a height of 300 m (built in 1980) and second one 330 m (built in 1982). The total operating capacity of the Thermal Power Plant is about 3500 MW and this TPP is the largest power plant in the Republic of Kazakhstan. The geometry of this area was presented in Figure 25. For numerical simulation, more than 12.8 million computing nodes were used for this problem. For calculation, grid clustered to the pipe, the minimum grid...
size was 1.623 m, and the maximum size was 5 m. In this problem statement, the pipe was also placed within the design area, and its vertex was determined as the vertical flow source. Boundary conditions were set as in the previous sections.

Figure 26 shows the particles movements of different sizes (from $dp = 0.1 \mu m$ to $dp = 10 \mu m$) in the computational domain for a real 330 m high pipe. From the numerical results, it can be noted the same pattern as in the previous section, that vortex motions are obtained due to the fact that the main flow around the pipe forms behind two large vortices, and after that the vortices collide and begin to move randomly and dissipate into smaller eddies. And the particles are also brought into the external flow by vortex rings, and after the particle follow the flow, so that they are almost particle-flow indicators.

Figure 27 shows the horizontal velocity component profiles at distances of 50, 100, 300, 500, 1000 and 2000 m from the pipe. It can be seen from this figure that when the position is closer to the pipe, the velocity profile has multiple fluctuations in horizontal velocity profiles. However, the velocities profiles when moving away from the pipe became more stabilized and assume a constant character. As shown in Figure 25, behind the pipe, where there are multiple fluctuations in the horizontal velocity profiles, the fine particles are well captured by the
vortex structures. It can also be noted that particles of a given size behave as fluid flow indicators. In addition to explaining, it should also be noted that circular dispersion also plays an important role in the particles dispersion in the certain flow region.

Figure 28 shows the mean concentration of the particle profiles at distances of 1, 10, 50, 100, 300, 500, 1000 and 2000 m from the pipe. It can be seen from this figure that the main changes in concentration occur up to 50 m from the thermal power plant’s pipe, and furthermore, starting from 100 m in the concentration of the particles profiles, no practical change is observed. However, it should be noted that the concentration of the particles still exists.

In Figures 24 and 26 illustrate that some particles move along a circular trajectory as flow with an almost equal radius. This phenomenon is called circular particles dispersion. It can be noted that these particles are not ejected from the core of the region, but also retain an almost equal radius in the considered region. This phenomenon depends on the vortex structures surrounding the particles. This instantaneous appearance and disappearance of the vortex structures cause particle dispersion along the circumferential direction. It can also be noted that for particles of the medium size, the particles dispersion depends to a large extent on the centrifugal mechanism, which is explained in detail above. In addition, it can be noted that in some regions the circular dispersion is more effective than the centrifugal mechanism. In these regions can be noted a rapid appearance and disappearance of the vortex structures. In the vortex structures region, the particles move around a circular trajectory and form a circular trajectory.

Conclusion
In this work, the accuracy of four turbulent models ($k - \varepsilon$, $k - \omega$, DES and LES) was tested by the ANSYS Fluent on the test problem of intersecting flows with numerical results of other authors and experimental results. The analysis of obtained numerical data with simulation results of other authors and the experimental data was also carried out. Moreover also the boundary conditions for the turbulent kinetic energy for the turbulent models ($k - \varepsilon$, $k - \omega$) were chosen to match the experimental data. The coincidences of numerical results of the investigated problem, comparison with the experimental results
(Ajersch et al., 1995) after study were the good agreement in comparison with the test problem (Keimasi & Taeibi-Rahni, 2001). After analyzing the accuracy, efficiency, and determining the time consumption of solving turbulent models, among all turbulent models, the DES turbulent model was chosen as the most optimal turbulent model for the transition to real sizes. In this paper for modeling, the problem was solved with changing geometry to the corresponding large-scale problem, without making corrections to the geometry dimensions,

**Figure 26.** Trajectories of particle motion.
boundary and initial conditions of the test problem. Numerical predictions have shown the particles motion trajectory under the real sizes conditions of the calculated region. In general, the numerical results are satisfactorily compared with the experimental data and give greater confidence in the effectiveness of the simulation. As a result of the numerical study, the obtained data can be used for further studies of the problems associated with the smallest particles spread in the air, as well as such particles spread in the human body. And also it is possible to note the enormous influence of the turbulent model on the obtained numerical results, the efficiency, accuracy and time consumption of the research.

In Section ‘Simulations results for real sizes’ the real physical model of the particles propagation from the Ekibastuz SDPP-1 was considered. The remarkable feature of this thermal power plant is that the particles distribution chimney of height 300 m. From the numerical results, it is obvious that the height of the chimneys significantly influences the particles distribution. The building of higher chimneys for the thermal power plant is more appropriate for the ecology safety.

Future research should continue to examine knowledge in describing the particles propagation with different sizes in the air from the activities of the Ekibastuz SDPP-1. This will help determine in advance the optimal location of thermal power plants in relation to human settlements. This will help minimize the damage caused by emissions to people, flora and fauna.

It is should be noted that many limitations exist in this work. The main limitation is the computational grid size and in the number of particles with different sizes: the computer resources restricted in the computational grid size, while very large grid and a lot of particles are needed to simulate accurately, but now it is not achievable, because it was limited by computer resource. Moreover, future increases in computer resources will be accompanied by demands of scientists for increased mesh resolution and the inclusion of additional physical parameters that will approach real problems. The second limitation of this work is the complexity of implementing

Figure 27. Velocity profiles at different distances from the pipe.

Figure 28. The profile of mean concentration of the particle at different distances from the pipe.
and analyzing experimental studies at particle distribution in the atmosphere from the activities of thermal power plants (environmental factors, shape, sizes, material of particles). The third limitation is that the shapes of the particles were taken as a sphere, but it should be noted that in reality the shape of the particle is not the correct shape (uniform shape of particles). These studies are useful for those who are interested in particles distribution in the atmosphere from the activities of thermal power plants.

**Disclosure statement**

No potential conflict of interest was reported by the authors.

**Funding**

This work is supported by the grant from the Ministry of Education and Science of the Republic of Kazakhstan.

**References**

Achim, D., Naser, J., Morsi, Y. S., & Pascoe, S. (2009). Numerical investigation of full scale coal combustion model of tangentially fired boiler with the effect of mill ducting. *Heat and Mass Transfer, 46*, 1–13.

Ahmed, A. M., & Elghobashi, S. (2001). Direct numerical simulation of particle dispersion in homogeneous turbulent shear flows. *Physics of Fluids, 13*(11), 3346–3364.

Ahmed, S., Hart, J., Nikolov, J., Solnordal, C., Yang, W., & Naser, J. (2007). The effect of jet velocity ratio on aerodynamics of a rectangular slot-burner in the present of cross-flow. *Experimental Thermal and Fluid Science, 32*, 362–374.

Ajersch, P., Zhou, J. M., Kettler, S., Salcudean, M., & Garthshore, I. S. (1995, June). Multiple jets in a cross-flow: Detailed measurements and numerical simulations. *International Gas Turbine and Aeroengine Congress and Exposition* (pp. 1–16), ASME Paper 95-GT-9, Houston, TX.

Akbarian, E., Najafi, B., Jafari, M., Ardabili, S. F., Shamshirband, S., & Chau, K.-W. (2018). Experimental and CFD-based numerical simulation of using natural gas in a dual-fuelled diesel engine. *Engineering Applications of Computational Fluid Mechanics, 12*(1), 517–534.

Ardabili, S. F., Najafi, B., Shamshirband, S., Bidgoli, B. M., Deo, R. C., & Chau, K.-W. (2018). Computational intelligence approach for modeling hydrogen production: A review. *Engineering Applications of Computational Fluid Mechanics, 12*(1), 438–458.

Armenio, V., & Fiorotto, V. (2001). The importance of the forces acting on particles in turbulent flows. *Physics of Fluids, 13*(8), 2437–2440.

Balachandar, S., & Eaton, J. K. (2010). Turbulent dispersed multiphase flow. *Annual Review of Fluid Mechanics, 42*, 111–133.

Campolo, M., Salvetti, M. V., & Soldati, A. (2005). Mechanisms for microparticle dispersion in a jet in crossflow. *AICHE Journal, 51*, 28–43.

Carter, H. H. (1969). A preliminary report on the characteristics of a heated jet discharged horizontally into a transverse current, part 1 – constant depth. Technical Report No. 61, Chesapeake Bay Inst., Johns Hopkins University, Baltimore, MD.

Chai, X., Iyer, P. S., & Mahesh, K. (2015). Numerical study of high speed jets in crossflow. *Journal of Fluid Mechanics, 785*, 152–188.

Chochua, G., Shyy, W., Thakur, S., Brankovic, A., Lienau, K., Porter, L., & Lischinsky, D. (2000). A computational and experimental investigation of turbulent jet and crossflow interaction. *Numerical Heat Transfer A, 38*, 557–572.

Chung, J. N., & Troull, T. R. (1988). Simulation of particle dispersion in an axisymmetric jet. *Journal of Fluid Mechanics, 186*, 199–222.

Edelman, R. B., Economos, C., & Boccio, J. (1971). Analytic and experimental study of some problems in two-phase flows involving mixing and combustion with application to the B–O–H–N system. *AIAA Journal, 9*, 1935–1940.

Environmental, Health, and Safety Guidelines for Thermal Power Plants. (2017, June). International Finance Corporation.

Fessler, J. R., Kulick, J. D., & Eaton, J. K. (1994). Preferential concentration of heavy particles in a turbulent channel flow. *Physics of Fluids, 6*(11), 3742–3749.

Flacks, R., Dullenkopf, K., & Scherer, V. (1994). Constituency measurements in the mixing region of a cross flow jet using a laser velocimeter. *Experiments in Fluids, 17*, 198–204.

FLERMONECA, GIZ. (2013). *Report of quality of air* (p. 15).

Fric, T. F., & Roshko, A. (1994). Vortical structure in the wake of a transverse jet. *Journal of Fluid Mechanics, 279*, 1–47.

Goto, S., & Vassilicos, J. C. (2008). Sweep-stick mechanism of heavy particle clustering in fluid turbulence. *Physical Review Letters, 100*(5), 054503.

Han, K. S., & Chung, M. K. (1992). Numerical simulation of two-phase gas-particle jet in a crossflow. *Aerosol Science and Technology, 16*(2), 126–139.

Haniu, H., & Ramaprian, B. R. (1989). Studies on twodimensional curved nonbuoyant jets in cross flow. *ASME Journal of Fluids Engineering, 111*, 78–86.

Hasselbrink, E. F., & Mungal, M. G. (2001). Transverse jets and jet flames. Part 1. Scaling laws for strong transverse jets. *Journal of Fluid Mechanics, 443*, 1–25.

Hoda, A., & Acharya, S. (2000). Predictions of a film coolant jet in crossflow with different turbulence models. *ASME Journal of Turbomachinery, 122*, 558–569.

Hwang, R. R., & Chiang, T. P. (1995). Numerical simulation vertical forced plume in a crossflow of stably stratified fluid. *ASME Journal of Fluids Engineering, 117*, 696–705.

Issakov, A. (2014). Modeling of synthetic turbulence generation in boundary layer by using zonal RANS/LES method. *International Journal of Nonlinear Sciences and Numerical Simulation, 15*(2), 115–120.

Issakov, A. (2015). Mathematical modeling of the discharged heat water effect on the aquatic environment from thermal power plant. *International Journal of Nonlinear Science and Numerical Simulation, 16*(5), 229–238.

Issakov, A. (2016a). Mathematical modeling of the discharged heat water effect on the aquatic environment from thermal power plant under various operational capacities. *Applied Mathematical Modelling, 40*(2), 1082–1096.
Issakhov, A. (2016b). Numerical modelling of distribution the discharged heat water from thermal power plant on the aquatic environment. *AIP Conference Proceedings*, 1738, 480025.

Issakhov, A. (2017a). Numerical study of the discharged heat water effect on the aquatic environment from thermal power plant by using two water discharged pipes. *International Journal of Nonlinear Sciences and Numerical Simulation*, 18(6), 469–483.

Issakhov, A. (2017b). Numerical modelling of the thermal effects on the aquatic environment from the thermal power plant by using two water discharge pipes. *AIP Conference Proceedings*, 1863, 560050.

Issakhov, A., Zhandaulet, Y., & Nogaeva, A. (in press). Numerical simulation of dam break flow for various forms of the obstacle by VOF method. *International Journal of Multiphase Flow*, https://doi.org/10.1016/j.ijmultiphaseflow.2018.08.003.

Jones, W. P., & Launder, B. E. (1972). The prediction of laminarization with a Two-equation model of turbulence. *International Journal of Heat and Mass Transfer*, 15, 301–314.

Kalita, K., Dewan, A., & Dass, A. K. (2002). Prediction of turbulent plane jet in crossflow. *Numerical Heat Transfer, Part A*, 41, 1–12.

Karagozian, A. R. (2014). The jet in crossflow. *Physics of Fluids*, 26(10), 101303.

Keimasi, R. M., & Taeibi-Rahmi, M. (2001). Numerical simulation of jets in a crossflow using different turbulence models. *AIAA Journal*, 12(39), 2268–2277.

Kelso, R. M., Lim, T. T., & Perry, A. E. (1996). An experimental study of round jets in cross-flow. *Journal of Fluid Mechanics*, 306, 111–144.

Launder, B. E., & Spalding, D. B. (1974). The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering*, 3(2), 269–289.

Liu, X., Ji, C., Xu, X., Xu, D., & Williams, J. J. (2017). Distributed characteristics of inertial sediment particles in the turbulent boundary layer of an open channel flow determined using Voronoi analysis. *International Journal of Sediment Research*, 32(3), 401–409.

Longmire, E. K., & Eaton, J. K. (1992). Structure of a particle-laden round jet. *Journal of Fluid Mechanics*, 236, 217–257.

Mahesh, K. (2013). The interaction of jets with crossflow. *Annual Review of Fluid Mechanics*, 45, 379–407.

Marchioli, C., Giusti, A., Salvetti, M. V., & Soldati, A. (2003). Direct numerical simulation of particle wall collision and deposition in upward turbulent pipe flow. *International Journal of Multiphase Flow*, 29(6), 1017–1038.

Marchioli, C., Picciotto, M., & Soldati, A. (2007). Influence of gravity and lift on particle velocity statistics and transfer rates in turbulent vertical channel flow. *International Journal of Multiphase Flow*, 33(3), 227–251.

Marchioli, C., Soldati, A., Kuerten, J. G. M., Arcen, B., Taniere, A., Goldensoph, G., ... Portela, L. M. (2008). Statistics of particle dispersion in direct numerical simulations of wall-bounded turbulence: Results of an international collaborative benchmark test. *International Journal of Multiphase Flow*, 34(9), 879–893.

Margason, R. J. (1993). Fifty years of jet in crossflow research. *AGARD Symposium on a Jet in Cross Flow*, Winchester, UK. AGARD CP 534.

McGurik, J. J., & Rodi, W. (1978). A depth-averaged mathematical model for the near field of side discharges into open channel flow. *Journal of Fluid Mechanics*, 86, 761–781.

Menter, F. R. (1994). Two-Equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32(2), 1598–1605.

Menter, F. R., & Kuntz, M. (2003). Development and application of a zonal DES turbulence model for CFX-5. CFX-validation report. CFX-VAL17/0503.

Monchaux, R., Bourgoin, M., & Cartellier, A. (2012). Analyzing preferential concentration and clustering of inertial particles in turbulence. *International Journal of Multiphase Flow*, 40, 1–18.

Mou, B., He, B.-J., Zhao, D.-X., & Chau, K.-W. (2017). Numerical simulation of the effects of building dimensional variation on wind pressure distribution. *Engineering Applications of Computational Fluid Mechanics*, 11(1), 293–309.

Muppidi, S., & Mahesh, K. (2005). Study of trajectories of jets in crossflow using direct numerical simulations. *Journal of Fluid Mechanics*, 530, 81–100.

Muppidi, S., & Mahesh, K. (2007). Direct numerical simulation of round turbulent jets in crossflow. *Journal of Fluid Mechanics*, 574, 59–84.

Muppidi, S., & Mahesh, K. (2008). Direct numerical simulation of passive scalar transport in transverse jet. *Journal of Fluid Mechanics*, 598, 335–360.

Patankar, S. V. (1980). *Numerical heat transfer and fluid flow*. New York: Taylor & Francis.

Pokale, W. K. (2012). Effects of thermal power plant on environment. *Scientific Reviews and Chemical Communications*, 2(3), 212–215.

Ramaprian, B. R., & Haniu, H. (1983). *Turbulence measurement in plane jets and plumes in cross flow*. Technical Report No. 266, IIHR, University of Iowa, Iowa City, IA.

Rouson, D. W., & Eaton, J. K. (2001). On the preferential concentration of solid particles in turbulent channel flow. *Journal of Fluid Mechanics*, 428, 149–169.

Salzman, R. N., & Schwartz, S. H. (1978). Experimental study of solid–gas jet issuing into a transverse stream. *Journal of Fluids Engineering – Transactions of the ASME*, 100, 333–339.

SAMRYK ENERGY. (2013). *Annual report*. Retrieved from https://www.samruk-energy.kz/en/shareholder/annual-reports.

Sarkar, S., & Bose, T. K. (1995). Comparison of different turbulence models for prediction of slot–film cooling: Flow and temperature field. *Numerical Heat Transfer Part B*, 28, 217–238.

Sbrizzai, F., Verzicco, R., Pidria, M. F., & Soldati, A. (2004). Mechanisms for selective radial dispersion of microparticles in the transitional region of a confined turbulent round jet. *International Journal of Multiphase Flow*, 30, 1389–1417.

Schluter, J. U., & Schonfeld, T. (2000). LES of jets in crossflow and its application to a gas turbine burner. *Flow Turbulence Combust*, 65, 177–203.

Shan, J. W., & Dimotakis, P. E. (2006). Reynolds-number effects and anisotropy in transverse-jet mixing. *Journal of Fluid Mechanics*, 566, 47–96.

Shi, Y., Ray, M. B., & Mujumdar, A. S. (2003). Numerical study on the effect of cross-flow on turbulent flow and heat transfer
characteristics under normal and oblique semi-confined impinging slots jets. Drying Technology, 21(10), 1923–1939.
Spalart, P. R., Jou, W.-H., Stretlets, M., & Allmaras, S. R. (1997). Comments on the feasibility of LES for wings and on the hybrid RANS/LES approach, advances in DNS/LES. Proceedings of the first AFOSR international conference on DNS/LES.
Squires, K. D., & Eaton, J. K. (1990). Preferential concentrations of particles by turbulence. Physics of Fluids A: Fluid Dynamics, 3, 1169–1178.
Su, L. K., & Mungal, M. G. (2004). Simultaneous measurement of scalar and velocity field evolution in turbulent crossflowing jets. Journal of Fluid Mechanics, 513, 1–45.
Tian, Z. F., Witt, P. J., Schwarz, M. P., & Yang, W. (2010). Numerical modelling of victorianbrowncoal combustion in a tangentially fired furnace. Energy and Fuels, 24, 4971–4979.
Uthuppan, J., & Aggarwal, S. K. (1994). Particle dispersion in a transitional axisymmetric jet: A numerical simulation. AIAA Journal, 32(10), 2004–2014.
Vasistha, V. (2014). Effects of pollutants produced by thermal power plant on environment: A review. International Journal of Mechanical Engineering and Robotics Research, 3(2), 202–207.
Versteeg, H. K., & Malalasekera, W. (1996). An introduction to computational fluid dynamics—the finite volume method (Chapters 5–7). Essex: Longman Malaysia.
Vreman, A. W. (2007). Turbulence characteristics of particle-laden pipe flow. Journal of Fluid Mechanics, 584, 235–279.
WHO. (2002). The World Health Report 2002—reducing risks, promoting healthy life. Retrieved from http://www.who.int/whr/2002/en/.
Wilcox, D. C. (1988). Re-assessment of the scale-determining equation for advanced turbulence models. AIAA Journal, 26(11), 1299–1310.
Wilson, R., & Spengler, J. (1996). Particles in our air. Cambridge, MA: Harvard University Press.
Yi, J., & Plesniak, M. W. (2002). Dispersion of a particle-laden air jet in a confined rectangular crossflow. Powder Technology, 125, 168–178.
Yuan, L. L., Street, R. L., & Ferziger, J. H. (1999). Large-eddy simulations of a round jet in crossflow. Journal of Fluid Mechanics, 379, 71–104.