STUDY OF GAS–SOLID FLOW IN A MULTICHANNEL CYCLONE

Pranas Baltrėnas¹, Petras Vaitiekūnas², Inga Jakštonienė³, Simona Konoverskaitė⁴

Department of Environmental Protection, Vilnius Gediminas Technical University,
Saulėtekio al. 11, LT-10223 Vilnius, Lithuania
E-mail: ¹pbalt@vgtu.lt; ²vaitiek@vgtu.lt; ³ingajakstoniene@gmail.com (corresponding author); ⁴saimutmis@gmail.com

Submitted 28 Jan. 2011; accepted 29 Nov. 2011

Abstract. This paper aims to analyse the problem of the gas–solid particle (SP) flow in the multichannel cyclone (three rings) with tangential inlet (KDГ – equipment for separation of solid particles from gaseous fluid flow). It provides a review of experimental and theoretical papers that describe cyclones with a very complex swirling flow. The paper describes the experimental study and numerical modelling of the flow in the multichannel cyclone, the height of which is 0.72 m and the diameter – 0.50 m; with the height of the cylindrical part amounting to 0.29 m, the height of the conical part – 0.43 m, and the inlet area – 0.29·0.034 m². The multi-functional measuring instrument Testo 400, intended for measuring the flow velocity in the inlet and outlet of the multichannel cyclone was used in experimental studies of the cyclone. Three-dimensional transport differential equations (Reynolds) for incompressible turbulent flow inside a cyclone are solved numerically using finite volume-based numerical method and turbulence models, namely the Standard k-ε model, the RNG k-ε model. According to results obtained during the experiments with quartz sand and quartz sand dust pollutants, the highest SP treatment efficiency as regards these pollutants, reaching 85.8–90.4%, was obtained. Modelling results obtained from the numerical tests with the inlet velocity of 6.27–10.78 m/s and, the flow rate of 0.111–0.190 m³/s have demonstrated a reasonable agreement with experimental and theoretical results. The average relative error was ± 4.3%.

Keywords: multichannel cyclone, solid particles, numerical modelling, turbulence, airflow, one-phase, two-phase flow.

1. Introduction

Cyclones are widely used in oil and recycling industries to separate particles from fluids. This can be explained by the fact that cyclones are easy to use, and they do not require heavy construction, exploitation, maintenance and energy consumption expenses. The use of appropriate materials and construction methods enables scientists to exploit cyclones in high temperature and high pressure when in energy and recycling industries, the use of very efficient equipment can influence such processes as pressurized fluidised-bed combustion (PFBC), integrated gasification combined cycle (IGCC) and fluidised catalytic cracking (FCC). In these processes, cyclones are now almost the only fully-commercial method to separate particles from high-temperature gases (Zhou, Sao 1990; Hu et al. 2005; Gujun et al. 2008).

Cyclones are defined as funnel-shaped industrial inertial equipment. They are very popular because of simplicity; besides, they are compact and cheap to produce; they have no moving parts and do not require much maintenance. (Boysan et al. 1982; Bernardo et al. 2006; Jakštonienė, Vaitiekūnas 2009).

Tangential inlet allows separating particles from gases (Altmeyer et al. 2004, Kaya, Karagöz 2008). They are called inverse conical flow (ICF) cyclones (Derkessen 2003). A spiral in the tangential inlet cyclones was modified, which is very effective in separating solid particles from the contaminated gas stream (Hashremi 2006), in which analysis (experiment and modelling) of two types of cyclones was made: ICF cyclone and a modified spiral in the ICF cyclone.

Computational fluid dynamics (CFD) has a great potential to predict the sediments in water (Vaikasas 2010), the flow-field characteristics (Petraitis, Vasarevičius 2001; Braduliene, Vasarevičius 2010), particle trajectories (Baltrenas et al. 2008) and the pressure drop in cyclones (Gimbutėnė et al. 2005).

Insufficient understanding of the process of two-phase flow in a cyclone does not allow improving its exploitation. Such inadequate understanding arises due to the fact that despite the supposed simplicity, dynamics of flows in cyclones is very complex and includes features such as swirling movement and in certain cases – several reverse-flow circular zones. Theories of closed swirling flow have not yet succeeded in distinguishing many characteristics of the analysed flow. The problem of mathematical modelling of the detailed flow structure includes the solution of closely related non-linear partial differential equations of mass and impulse conservation and has no analytical solution. Besides, discontinuation of turbulence based on the assumption of isotropy (e.g. the k-ε model) cannot be used for strongly swirling flows (Boysan et al. 1982, Bernardo et al. 2006). Hoffmann and Stein (2002) claim that the standard k-ε model has its disadvantages when used for strongly swirling flows.
Meier and Mori (1999) provided timely-averaged Navier-Stokes equations for the gaseous phase and related them to the anisotropic turbulence model in combination with the $k$–ε model and algebraic stress equations. After this innovative work, several studies aimed to model turbulence in order to better foresee velocity and pressure. All these studies accepted axial symmetry that allows using two-dimensional model where the solid phase almost does not contact the gaseous field.

Bernardo et al. (2006) used a specific turbulence model, known as the Reynolds stress model (RSM), to find certain values for Reynolds stress terms. This model is based on transport equations for all Reynolds stress tensor components and dissipation velocity. RSM provides anisotropic turbulence to flows; when the hypothesis of turbulent viscosity is used, model gives isotropic turbulence. In the first case, Reynolds stress transfer equations are being solved for separate stress components.

Wang et al. (2006) applied Reynolds stress model in order to model gas flow in the Lapple cyclone. RSM very precisely predicts swirling flow features, axial velocity, tangential velocity and pressure loss in the model of a cyclone (Sommerfeld 2003; Gujun et al. 2008). The following mathematical model can be drawn for this case:

$$\frac{Du}{Dt} = \rho F_{ij} - \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_i}{\partial x_j} - \rho \mu_{ij} \right), \quad (1)$$

$i, j = 1, 2, 3$. Terms in the (1)-type equations $\mu_{ij}$ are called turbulent or Reynolds stresses that require additional differential equations. For this purpose, Reynolds stress equations are used. These equations for the two-dimensional problem are described in the Kavaliauskas and Vaitiekūnas (2001) study. RSM (Reynolds stress model) can reasonably predict swirling flows (Gujun et al. 2008). The probabilistic Lagrange model is used in order to model the flow of particles.

Recently pervasive are the cross-type cyclones – centrifugal filters – (Balan et al. 2009; Serebrianskij, Novakovskij 2009), the efficiency of which depends on the number of channels.

The aim of this work is to investigate the experiment and numerical modelling of the multichannel cyclone. For this purpose, three-dimensional transport equations with the standard $k$–ε and RNG $k$–ε turbulence model were used.

2. Analysis methods

2.1. Experimental analysis method

The experimental research using the multichannel cyclone was carried out in the Scientific Environmental Technology Laboratory of Vilnius Gediminas Technical University.

The schematic diagram of experimental set-up is presented in Fig. 1. The cyclone was filled with experimental dust using a dust nozzle and dispersed with the help of compressed air pressure (8 bars). Air was blown to the cyclone system using a channel ventilator (1). Testo 400 (2) was used to measure an airflow velocity at the inlet and outlet of the multichannel cyclone. In order to minimise the sampling error occurring due to the formation of a swirling flow in cyclone’s outlet, which may first throw particles to external walls, a lattice is fitted inside a pipe in the cyclone’s outlet (3). Measurements were carried out using the optical dust concentration monitor Microdust pro (Casela) (4).

Fig. 1. The schematic diagram of experimental set-up: 1 – channel ventilator, 2 – multifunctional meter Testo 400, 3 – lattice, 4 – dust concentration monitor Microdust pro (Casela)

It is designed for assessing the concentration of solid particles. Its operation is based on the measurement of the angle of scatter of infrared light of 880 nm wavelength spreading in the air polluted with particulates. The monitor consists of: a sampling probe fitted with an infrared light, measurement slot, a light collector, an infrared ray detector; an indicator with analytical signal intensification, processing and representation system, a control button and a liquid crystal display. The main functions of the dust concentration monitor: identification, calculation and indication of solid particles concentration; verification of the technical condition and indication of a liquid crystal display; verification of the condition of a power battery (indication in segments); graphic representation of dust concentration measurement; sensitivity coordination with known concentrations in optical filters from an instrument kit; identification of initial measurement zero (0.0 mg/m$^3$) concentration point.

Materials used for the preparation of a pollution source: quartz sand (physical density 3200 kg/m$^3$, particle size 50–400 µm), quartz sand dust (crystalline silicon dioxide) (physical density 2450 kg/m$^3$, particle diameter 10–100 µm), calcium carbonate (CaCO$_3$) (physical density 2 700 kg/m$^3$, particle diameter 5–100 µm), which must be dry during measurement. All materials were dried at a temperature of 80 °C, and afterwards stored at a temperature of 24–26 °C for 24 hours. Humidity of the particles may not exceed 5%. During the experiment the ambient
temperature varied from 24.4 °C to 25.9 °C, and relative humidity of air reached 58%. Control samples for inlet dust concentrations were supplied to the multichannel cyclone at the fixed time.

Upon completion of each dust measurement and prior to measuring particles of another material, hoses of the aspiration system are cleaned by blowing a strong airflow through them. Each material for the dusty airflow was drawn at a rate of 20 l/min. The optical dust concentration monitor Microdust pro (Casela) must be calibrated anew by setting it in a conditional zero position (Baltrėnas, Kvasauskas 2005).

During the experiments air enters the cyclone system via a tangential inlet and airflow velocity is measured in the inlet and outlet with the instrument Testo 400.

A place for flow velocity measurement is selected in the linear section of the air pipe with the settled gas flow in the cyclone. The air pipe section must be free of any obstacles to the flow (valves, bends, ventilators) in 5–6 D (air pipe diameter) up to the measurement point and in 3–4 D behind the measurement point.

To make measurements in the air pipe, holes are bored in the selected place and metal connecting pipes, 40–50 mm long, with screw caps are welded, or openings, 20–30 mm in diameter, are made.

The multifunctional meter Testo 400 is used to determine dimension readings, i.e. m/s, showing an airflow velocity. When parameters of the multifunctional measuring instrument Testo 400 are set, the Pitot tube is placed into the air pipe in which the end of the tube is directed against a gas flow and the tube is slowly pulled from the inner side of the tube towards the middle and backwards. Measurements are started and the average of readings is calculated. Analysis in each point was made three times and the average airflow velocity was calculated.

Principle scheme of the multichannel cyclone. A cyclone consists of two main parts (Fig. 2): cylindrical part (diameter (D₁) – 0.5 m, height (H₁) – 0.29 m), height of the conical part (H₂) – 0.43 m, base diameter (D₂) – 0.16 m, downward (for solid particles, diameter (D₃) – 0.10 m, general cyclone height (H) – 0.72 m. In the cylindrical part fitted there are three different centrifugal radius semicircle 0.29 m in width, which consists of four channels (Fig. 3).

When compared to usual centrifugal air-cleaning equipment, the multichannel cyclone cleans fine solid particles with centrifugal force and filtration, performed in curvilinear channels (7) that have spaces between the rings of channels (3). These spaces divide flow into two parts: larger particles enter the channel of larger radius (6) and fall into the hopper, finer particles are filtered through the air-flow and enter the channel of smaller radius. Cleaned air outflows through the outlet.

2.2. Governing equations and solution method

Turbulent flows are very complex. This is clearly seen in the increased complexity of turbulent velocity equations (such as 1) where additional terms (Reynolds stresses) are used. When modelling these terms, we try to provide simple connections as the finite equation form that is solved numerically (simplification of the full equations). This means that the accuracy of the mathematical model that describes flow can be reduced (Vaitiekūnas 1998). The use of the hypothesis of turbulent viscosity allows forming the following differential transport equations (for air flow only):

$$\frac{\partial}{\partial t} [\rho \Phi] + \text{div} (\rho \mathbf{v} \Phi - \Gamma_\Phi \text{grad} \Phi) = S_\Phi, \quad (2)$$

where: $t$ – time; $\rho$ – density; $\Phi$ – dependent variable, as a moment to the unit of mass, turbulence energy, its dissipation rate; when $\Phi = 1$ – continuity equation; $\mathbf{v}$ – velocity vector; $\Gamma$ – exchange coefficient of the variable $\Phi$;
$S_\Phi$ – flow (source) term to variable $\Phi$. Exchange coefficient for the turbulent flow can be written as:

$$\Gamma_\phi = \rho (v_t + v_i),$$

(3)

where $v_i$ – molecular coefficient of kinematic viscosity; $v_t$ – coefficient of turbulent viscosity. The turbulent viscosity $\mu_t$ or $v_t$ can be computed by combining the turbulent kinetic energy $k$ and its dissipation rate $\varepsilon$ as follows:

$$\mu_T = C_{\mu} \rho \frac{k^2}{\varepsilon}. \quad (4)$$

Transport equations for variables $k$ and $\varepsilon$ in the RNG $k$-\varepsilon model, which is derived from Navier-Stokes equations using the renormalisation group theory (Yakhot, Orszag 1986) can be written as:

$$\rho \frac{D k}{D t} = \frac{\partial}{\partial x_i} \left( \rho \mu_{eff} \frac{\partial k}{\partial x_i} \right) + G_k - \rho \varepsilon, \quad (5)$$

$$\rho \frac{D \varepsilon}{D t} = \frac{\partial}{\partial x_i} \left( \rho \mu_{eff} \frac{\partial \varepsilon}{\partial x_i} \right) + C_1 \mu \frac{G_k}{k} - C_2 \rho \frac{\varepsilon^2}{k} - R. \quad (6)$$

Unlike the standard $k$-\varepsilon model, this model includes analytical expressions in addition to having an extra term $R$ in the second equation. The model constants are assumed to have the following values: $C_1 = 1.42$, $C_2 = 1.68$ and $C_\mu = 0.0845$, $Pr_k = Pr_\varepsilon = 0.7194$.

These governing equations are solved numerically using the finite volume-based method (Spalding 2002; Patankar 1980). According to the basic idea of this method, the computational domain is divided into a number of cells, and differential equations are integrated over each cell using the theorem of divergence (the Gauss–Ostrogradski theorem) to obtain algebraic equations. These algebraic equations are solved iteratively to obtain the field distribution of dependent variables.

Numerical modelling was performed with a numerical calculation grid where the cyclonic area is a three-dimensional space in the cylindrical coordinate system that is divided into cells $x$, $y$, $z$ directions, $x$ is angle in radians (Bernardo et al. 2005).

Governing equations were solved numerically using the finite volume-based method. The use of CFD when modelling flow in cyclones is the best decision as this method is more universal. The use of the optimum modelling scheme CFD allows recording complex flow–particle interaction with a great accuracy (Youngmin et al. 1999).

In general, the cellular region of a cyclone is formed of $x \times y \times z = 40 \times 36 \times 45 = 64800$ (Fig. 4) volume cells. These are control velocity cells where velocity components, pressure and turbulent characteristics in radial, tangential and axial directions are calculated.

Airflow velocity at the cyclone inlet is assumed to be uniformed and reaches 6.27–10.78 m/s. The outflow boundary condition was used at the exit. At the walls, the law of flow-solid wall adhesion was applied for velocity, and near-wall treatment was achieved using the standard and non-equilibrium wall functions.

3. Results and analysis

3.1. Experimental results

Experimental results were obtained when inlet air–solid flow rate is: 0.111; 0.143; 0.166 and 0.190 m$^3$/s.

Solid particle (SP) concentrations in incoming and outgoing air–dust flow from the KDG cyclone were determined by the spectrometric technique. This technique was employed to measure the concentrations of quartz sand and quartz sand dust in the flows incoming to and outgoing from the KDG cyclone. Data received from spectrometric measurements are presented in Figs. 5 and 6.

![Fig. 4. The multichannel cyclone outline of a calculation grid](image-url)

![Fig. 5. Dependence of quartz sand SP concentration treatment on air-solid flow velocity in the KDG cyclone with the following SP concentrations for incoming flow (6.27; 8.09; 9.41; 10.78 m/s): 1) 136.35; 2) 115.06; 3) 60.9; 4) 60.8 mg/m$^3$](image-url)

![Fig. 6. Dependence of quartz sand dust SP concentration treatment on air-solid flow velocity in the KDG cyclone with the following SP concentrations for incoming flow (6.27; 8.09; 9.41; 10.78 m/s): 1) 154.20; 2) 96.45; 3) 76.30; 4) 75.20 mg/m$^3$](image-url)
Graphic representation shows the solid particle concentrations determined by the spectrometric technique in the airflow outgoing from the KDG cyclone when air was cleaned from quartz sand and quartz sand dust by changing the operation parameters of the device, i.e. supplied airflow velocity and time of inlet dust concentration. 4 positions of airflow velocity values were set: 6.27; 8.09; 9.41; 10.78 m/s. Time of inlet dust concentration: 50 s for quartz sand, 25 s for quartz sand dust (Figs. 5 and 6).

Figs. 5 and 6 show the dependences of the treatment of SP concentrations determined by the spectrometric technique on airflow velocity (on inflow into the KDG cyclone) when air is cleaned from quartz sand and quartz sand dust with the following initial concentrations: 136.35; 115.06; 60.9; 60.8 mg/m$^3$ for quartz sand, and 154.20; 96.45; 76.30, 75.20 mg/m$^3$ for quartz sand dust.

As Figs. 5 and 6 show, with a gradual increase of velocity of the airflow passed through the KDG cyclone (from 6.27 to 10.78 m/s), the treatment efficiency of solid particle concentrations gradually increases: initially (at a velocity of 6.27 m/s) air is cleaned from 136.35 mg/m$^3$ quartz sand and 154.20 mg/m$^3$ quartz sand dust up to the concentration of 16.63 and 22.67 mg/m$^3$, respectively; but when an airflow velocity reaches 6.27 m/s, the concentration of solid particles decreases by around 2.8 and 3.1 times, respectively. Such a decrease in SP concentration testifies to a high treatment efficiency of the air treatment device – the KDG cyclone.

Below are presented the dependences of the treatment of SP concentrations determined during experiments on airflow velocity (on inflow into the KDG cyclone) when air is cleaned from quartz sand and quartz sand dust. The efficiency of cyclone was analysed at the four inflow velocities (see Figs. 5 and 6).

According to results obtained during the experiments with quartz sand and quartz sand dust pollutants, the highest SP treatment efficiency as regards these pollutants, reaching 85.8–90.4%, was obtained.

### 3.2. Modelling results and analysis

After taking account of the dimensions of the multichannel cyclone structure (see Figs. 2 and 3), aero-dynamic processes – transport of the inlet airflow velocity in the gas flow at the inlet velocity of 6.27–10.78 m/s within an air pipe – were modelled.

Fig. 8 shows the inflow of the airflow supplied through the multichannel cyclone’s inlet, which is within the entire height of a cylindrical part of the cyclone, to the circular part of the cyclone (between cyclone’s body and ring R2, Fig. 3). An airflow is supplied to the KDG cyclone via the channel (1) and (2), afterwards the enters channel (3) and finally enters the channel (4), from which the flow splits into two parts: part of the flow enters the channel (3), another – ascends in the channel (4) via the cylindrical part of the cyclone’s body to the top towards the airflow outlet.

Having passed through openings intended for solid particle falling into a hopper in the conical part of the cyclone, located by the cyclone body, the flow causes a complicated movement (Fig. 8). Velocity vectors by cyclone walls, as presented in Fig. 8, show movement of the turbulent airflow in the KDG cyclone.

The obtained field of velocity vectors in the KDG cyclone is presented in Fig. 9, where the centre axe of the plane is the top of the cyclone’s cylindrical part, i.e. the outlet for the cyclone (exit air pipe).

---

**Fig. 7.** Dependence of SP concentration treatment efficiency on air-solid flow velocity when air is cleaned from quartz sand and quartz sand dust in the KDG cyclone

As Fig. 7 shows, with a gradual increase of velocity of the airflow passed through KDG cyclone (from 6.27 to 10.78 m/s) treatment efficiency increases; however, at a velocity of 9.41–10.78 m/s the efficiency remains stable reaching around 90.3±0.13%.

**Fig. 8.** Field of air flow velocity vectors in the vertical plane of a multichannel cyclone. Scale of velocity vectors is 23.0 m/s

**Fig. 9.** Field of air flow velocity vectors in the centre axle (outlet) of a multichannel cyclone, scale of velocity vectors is 20 m/s (outlet for cyclone, $D_2 = 0.16$ m)
Fig. 10 shows the projections of the tangential velocity component to z coordinate in the vertical plane of the KDG cyclone. In Fig. 10 and Fig. 11 – velocity vectors with inlet velocity 6.27 and 10.78 m/s.

Comparison between numerical and experimental data results where measurements were made at two points of the KDG cyclone: point No. 1 – at the first ring \( R_1 \); point No. 2 – at the second ring \( R_2 \) of tangential velocities profiles 0.14 m from No. 2: \( x = 270^\circ, y = 12 \text{ cm} \) in the KDG cyclone. The results obtained are presented in the Fig 12. It gives a comparison between numerical and experimental results of tangential velocities at the axial position of the top of the cyclone \( z = 0.14 \text{ m} \) at the first ring \( R_1 \) (point No. 1: \( x = 90^\circ, y = 9.3 \text{ cm} \)) and the second ring \( R_2 \) (point No. 2: \( x = 270^\circ, y = 12 \text{ cm} \)) in the KDG cyclone. The results obtained are presented in the Fig. 12. Comparison is undertaken of numerical and experimental results of tangential velocities at the axial position of \( z = 0.14 \text{ m} \) from the top of the KDG cyclone.

![Fig. 10. Field of air flow velocity vectors in the centre axle of a multichannel cyclone, scale of velocity vectors is \( 40 \text{ m/s} \), in plane \( z = \text{const} \, 0.15 \text{ m} \) from top of the cyclone \( U_{in} = 6.27 \text{ m/s} \), \( D_1 = 0.50 \text{ m} \)](image)

![Fig. 11. Field of air flow velocity vectors in the centre axle of a multichannel cyclone, scale of velocity vectors is \( 40 \text{ m/s} \), in plane \( z = \text{const} \, 0.15 \text{ m} \) from top of the cyclone \( U_{in} = 10.78 \text{ m/s} \), \( D_1 = 0.50 \text{ m} \)](image)

Fig. 12. Comparison between numerical tangential air flow velocity on horizontal plane 0.14 m from the top of the KDG cyclone and experimental data (inlet velocity 6.27, 8.09, 9.41 and 10.78 m/s)

A comparison of experimental and numerical data shows that the numerical results of tangential velocity at the first ring in cyclone \( z = 0.14 \text{ m}, x = 90^\circ, y = 9.3 \text{ cm} \), when the number of cyclone cells is 97200, nearly coincide with the experimental data; and the average relative error is 6.8%. The best agreement of the experimental and numerical data was obtained from the centre of cyclone towards the second ring \( x = 270^\circ, y = 12 \text{ cm} \), in which case the average relative error reached 1.8%.

These results were obtained by applying a one-phase RNG turbulence model, which showed a reasonable agreement between experimental and numerical results. The modelling results of aerodynamic processes allow a conclusion that the values measured during airflow velocity investigations within the error limits corresponded to the tangential velocities determined during modelling, the total relative error of which reached 4.3% as in (Vaitiekunas et al. 2010).

Turbulent flow inside a tangential inlet cyclone was solved numerically using the CFD method with three different turbulence models, namely the standard \( k-\varepsilon \), the RNG \( k-\varepsilon \) turbulence model with non-equilibrium wall function and the RSM model. The received numerical results were compared with theoretical (Kaya, Karagoz 2008) and (Bernardo et al. 2006), as well as experimental (Patterson, Munz 1996, Cristea et al. 1996) data.

The complicated swirling turbulent flow in a cyclone places great demands on the numerical methods and turbulence models employed in the CFD codes when modelling the cyclone pressure drop, as well as axial and tangential velocities (Hoekstra et al. 1999; Ingham, Ma 2002).

Although literature widely analyses and describes turbulent transport models of various complexity used in numerical researches, it is not yet clear which of them are the most appropriate for mixed convection.

Researchers Kaya and Karagoz (2008) analysed the performance of various numerical methods and interpolation schemes when studying strongly swirling flows inside a tangential inlet cyclone and compared predictable results with experimental data and numerical values by Gong and Wang (2004).

Comparison of axial and tangential velocity profiles computed using three turbulence models with experimental (Gong, Wang 2004) data is shown that the RSM
turbulence model gives more coincident results than other turbulence models when comparing experimental data.

Although the tendency and behaviour of the theoretical velocity profiles are consistent with experimental data, there are some discrepancies, especially in the core part, when comparing the measured velocities from the literature and the RSM predictions. When this swirling flow is strongly affected by the flow and geometric conditions, and it is difficult to measure velocities precisely in such a complex flow, the conclusion can be made that these discrepancies are due to not only turbulence models and numerical methods, but also experimental and measurement errors. The highly rotating fluid flow generates strong anisotropy in the turbulent structure. This causes the standard $k$-$\varepsilon$ and the RNG $k$-$\varepsilon$ turbulence models to provide inaccurate predictions for the fluid flow. Although the RNG $k$-$\varepsilon$ model gives slightly better results when compared to the standard $k$-$\varepsilon$ model, it fails to provide Rankin-type velocity distributions due to its swirl. Besides, the standard $k$-$\varepsilon$ and the RNG $k$-$\varepsilon$ turbulence models predict the pressure drop. However, the best prediction of the pressure drop is given by the RSM model. This is also confirmed by the modelling results of the work (Kaya, Karagoz 2008).

4. Conclusions

1. Efficiency of the multichannel cyclone (KDG) at the investigated gas-solid flow velocities of 6.27–10.78 m/s was increasing in an asymptotic manner; however, a still higher increase in velocity could produce an inverse effect – either decreased or stable treatment efficiency.

2. The optimum inflow velocity (gas–solid flow) of the KDG cyclone is 9.510.0 m/s, at the presence of which the cyclone achieves the best efficiency, i.e. 90.3±0.15%.

3. The comparison of modelling results (tangential air flow velocities) and experimental data has shown the average relative error of ±4.3%.

This scientific research is supported by the COST action No MP0806 “Particles in Turbulence” and by the Research Council of Lithuania.

4. Conclusions

1. Efficiency of the multichannel cyclone (KDG) at the investigated gas-solid flow velocities of 6.27–10.78 m/s was increasing in an asymptotic manner; however, a still higher increase in velocity could produce an inverse effect – either decreased or stable treatment efficiency.

2. The optimum inflow velocity (gas–solid flow) of the KDG cyclone is 9.510.0 m/s, at the presence of which the cyclone achieves the best efficiency, i.e. 90.3±0.15%.

3. The comparison of modelling results (tangential air flow velocities) and experimental data has shown the average relative error of ±4.3%.

This scientific research is supported by the COST action No MP0806 “Particles in Turbulence” and by the Research Council of Lithuania.

References


Simona KONOVERSKYTĖ. Master at the Department of Environmental Protection, Vilnius Gediminas Technical University (VGTU). Bachelor of Environmental Protection and Ecology. Šiauliai University 2006. Publications: author of 1 scientific publication. Research interests: protection of the atmosphere.