XX Polish Fluid Mechanics Conference,
Gliwice, 17-20 September 2012

NUMERICAL ESTIMATION OF THE PARTICLE COLLECTION EFFICIENCY OF A CYCLONE SEPARATOR

Paweł KOZOŁUB, Wojciech ADAMCZYK, Ryszard BIAŁECKI, Gabriel WĘCEL
Institute of Thermal Technology, Silesian University of Technology, Gliwice, Poland
E-mail: pawel.kozolub@polsl.pl

Abstract

The article presents numerical simulation of the flow inside a cyclone separator. Two cases have been analyzed: single gas flow and gas-particle flow. The numerical calculations have been carried out using commercial CFD code ANSYS-FLUENT. The Reynolds Stress Model (RSM) and k-epsilon RNG model have been used for the turbulence modeling. The results of axial and tangential velocity fields obtained with the RSM show good agreement with the experimental data taken from literature. The k-epsilon model is insufficient to model the complex flow inside the cyclone. The two-phase flow (gas and solid particles) has been modeled within the Lagrangian frame of reference in order to determine the solid particle collection efficiency. Obtained value of the total separation efficiency of the investigated cyclone is in agreement with the measurements.

Key words: computational fluid dynamics, cyclone, separation efficiency

INTRODUCTION

Cyclone separators are widely used in various branches of industry for the separation of solid particles from air or process gases. An important application of these type of devices are in Circulating Fluidized Bed (CFB) installations, where they are used for the separation of recirculating material. Cyclones utilize the centrifugal force caused by the swirling motion of fluid as a separation mechanism. These type of particle separators gained their popularity due to simple design, low manufacturing and maintenance costs. Moreover, they have a relative high efficiency and ability of operating at wide ranges of temperature and pressures which make them accessible for adaptation to the specific installations.

The pressure drop through the cyclone and particle collection efficiency are the key parameters that characterize the cyclone performance. They depend on the cyclone geometry and operating conditions (e.g. gas inlet velocity, particle mass loading). There are many semi-empirical correlations that can estimate cyclone flow characteristics, which result from experimental work done on various cyclone installations. These type of relations could be very useful for the cyclone design and optimization. However the flow pattern inside these devices is very complex and many physical aspects are not taken into account by this methodology. Recent advances in experimental and computational methods give a possibility to understand the complex nature of the flow and simulate the flow behavior in more detailed manner (Cortes and Gil, 2007).

The work presents a numerical simulation of the granular flow inside a cyclone separator carried out by the commercial Computational Fluid Dynamics (CFD) code ANSYS FLUENT. Based on the flow field solutions the cyclone pressure drop and separation efficiency has been obtained. Obtained results have been compared to experimental data described in literature.

CYCLONE GEOMETRY AND FLOW CONDITIONS

The cyclone geometry used in the numerical analysis has been taken from the experiment described by Solero and Coghe (2002). Fig. 1 presents the geometrical features of the separator along with its dimensions. Four basic parts of the cyclone can be distinguished: tangential inlet duct, main cylindrical body, conical body and exhaust duct (also called as vortex finder).

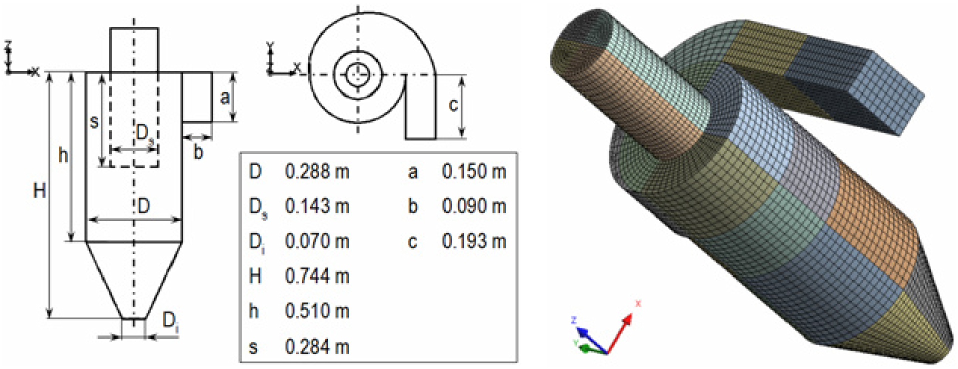

Fig. 1. Cyclone geometry with dimensions and numerical mesh used in computations

In the present work two cases have been considered: pure gas flow and gas with solid particles flow. Operating conditions of the pure gas flow are reported in the work of Solero and Coghe (2002). Conditions of the experiment regarding the case of gas and particle flow are described by Cristea et al. (1996), where the presence of solid particles in the experiment was simulated using an advanced ceramic powder (ACP). All details about the flow conditions for both cases are listed in Table 1.

Table 1. Operating conditions of the simulated flow

	Case A	Case B
Gas medium	SF_6 59% and air 41%	SF_6 59% and air 41%
Volumetric flow rate, m^3/s	0.063	0.063
Inlet mean velocity, m/s	4.64	4.64
Gas density, kg/m^3	3.41	3.41
Dynamic viscosity, Pa·s	$0.19 \cdot 10^{-4}$	$0.19 \cdot 10^{-4}$
Solid particulate	-	ACP
Particle density, kg/m^3	-	3700
Dust-to-gas mass ratio, kg_p/kg_g	-	0.851

NUMERICAL SETUP

Accurate numerical simulation of the flow inside cyclone requires usage of a turbulence model which is able to take into account the complex flow behavior. The swirling fluid motion has high turbulence level and strong anisotropy. Standard turbulence models based on eddy viscosity (k-epsilon) are insufficient to model this type of flows. Thus, the accurate CFD solution requires more advanced turbulence models such as Reynolds Stress Model or Large Eddy Simulation (Slack et al., 2000, Shalaby et al., 2005). In the present work two turbulence closure models for Reynolds Averaged Navier-Stokes equations have been compared, namely k-epsilon RNG and Reynolds Stress Model (RSM). The first is based on turbulent viscosity concept which assumes isotropic flow behavior. The latter is more suitable for the swirling flow but it is computationally more expensive. It solves additional transport equations for all

Reynolds stresses in order to predict an anisotropic behavior of swirled flow inside the cyclone.

Numerical mesh used for computations consisted of 216 670 elements. The majority of grid elements have been of a hexahedron type, however some wedge elements in the scroll inlet zone have been used. The view of the numerical mesh is presented in Fig. 1. The finite volume methods have been used to discretize partial differential equations of the model. The SIMPLE method has been used for the pressure-velocity coupling and the second order QUICK scheme has been applied to interpolate variables on the surface of the control volume. Standard wall functions implemented in ANSYS FLUENT have been used to model the boundary layer region.

The presence of solid particles has been modeled using Euler-Lagrange approach implemented in ANSYS FLUENT as the Discrete Phase Model (DPM) which has common roots with Discrete Element Method (DEM). The DPM model in contrary to the DEM approach does not take into account mutual interactions betweeen particles due to collisions. In this methodology the fluid phase is treated as a continuum by solving Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of particles through the calculated flow field. The trajectories of individual groups of particles are predicted by integrating the force balance of forces acting on a single group of particles. The present study assumes that gravity and drag forces influence the particulate flow behavior, but the presence of particles do not affect the fluid flow pattern (one way coupling). These assumptions are valid when diluted two-phase flow is simulated (Wang et al., 2006).

SIMULATION RESULTS

Velocity field

This section presents results for the case of gaseous medium flow. Results obtained with the use of RSM and k-epsilon RNG turbulence models are compared. Fig. 2 presents radial profiles of axial and tangential velocities located at different heights of the cyclone. The experimental data used for the comparison has been taken from the work of Solero and Coghe (2002). Measured tangential velocity profiles are similar at different heights of the cyclone. They indicate the presence of the Rankine type vortex which consists of an outer free vortex and a solid body rotation (forced vortex) at the core region. The highest swirl velocity is located at the interface of free and forced vortices. The profiles of axial velocity component taken from the experiment show that the flow is directed downward (towards conical part of the cyclone) in an annular section close to cyclone walls. The inner region of the flow is directed upward towards the exit of the vortex finder, where the axial velocity reaches its maximum. A zone in which there is a dip in the axial velocity or even the flow is reversed downwards exists inside the inner region.

It can be seen that the RSM is more accurate in predicting velocity fields than the k-epsilon RNG model, especially for the axial component of velocity. The solution obtained with the RSM is much closer to experimental data due to its ability of taking into account anisotropic flow behavior. The k-epsilon model cannot predict flow phenomena inside the cyclone due to strong curvature of the streamlines of the moving fluid (Shalby et al., 2005). Thus the performance of the RSM is superior compared to the k-epsilon RNG model, even though it requires more computational time and resources.

Pressure field

Fig. 3 presents contours of static pressure obtained for k-epsilon RNG and RSM turbulence models. The difference between two models can be noticed in the core vortex. In the case of RSM model the shape of core region is more distinct due to lower value of pressure in this region. The overall pressure drop in the cyclone is estimated by the difference between static pressure average values at the inlet and vortex finder exit. Table 2 contains evaluated values of pressure drop obtained from results of numerical computations and pressure measurements reported by Cristea et al. (1996). The CFD simulation is capable of reproducing pressure drop with satisfactory accuracy for only for the RSM model due to its better performance in prediction of the velocity field.

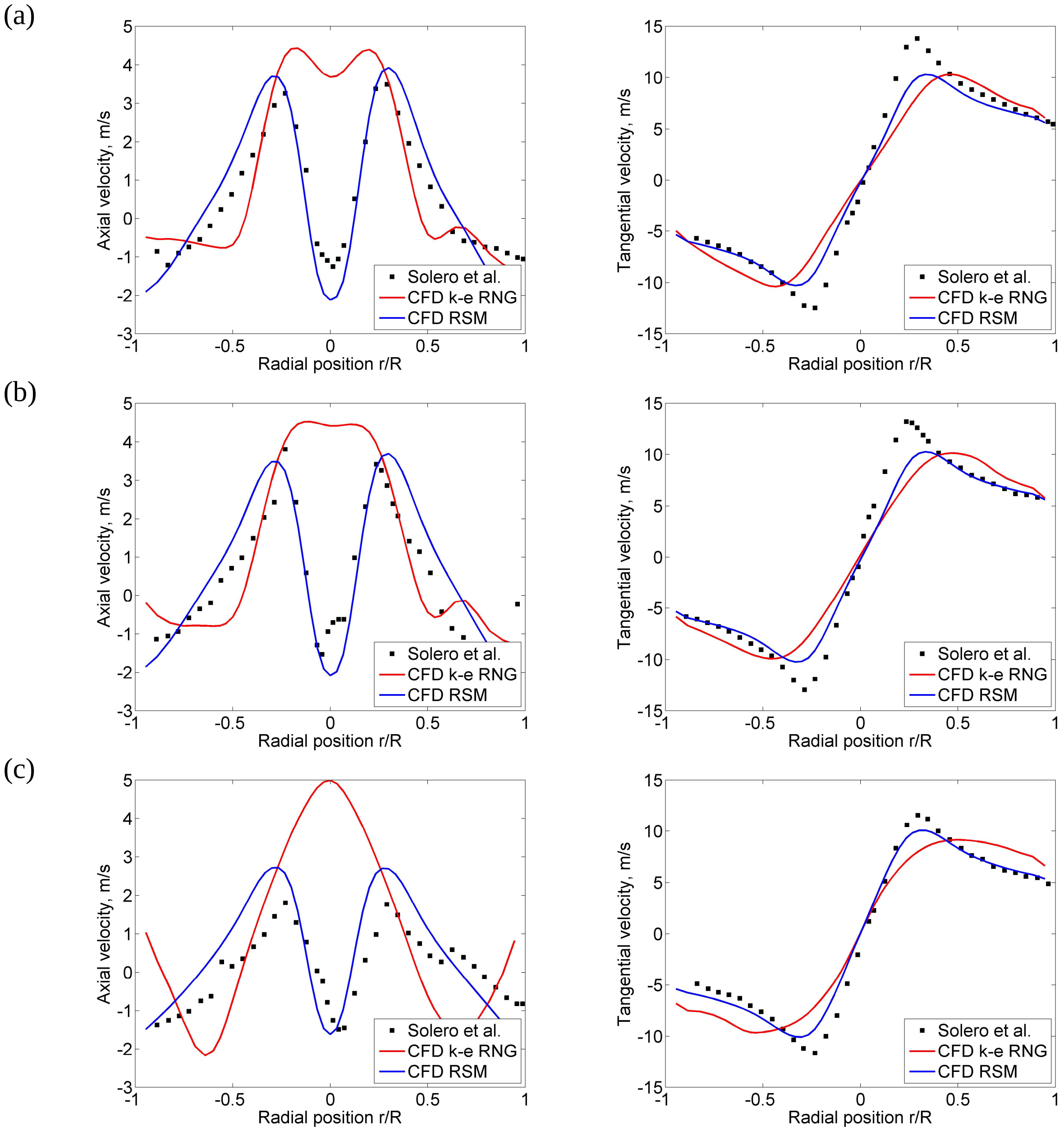

Fig. 2. Radial profiles of axial and tangential velocities at different cyclone heights: (a) 317 mm, (b) 330 mm, (c) 400 mm

Table 2. Values of the cyclone pressure drop obtained from measurements and numerical simulation

	Pressure drop, Pa	Relative error, %
Measurements	250.0	-
CFD k-epsilon RNG	323.6	29.4
CFD RSM	279.0	11.6

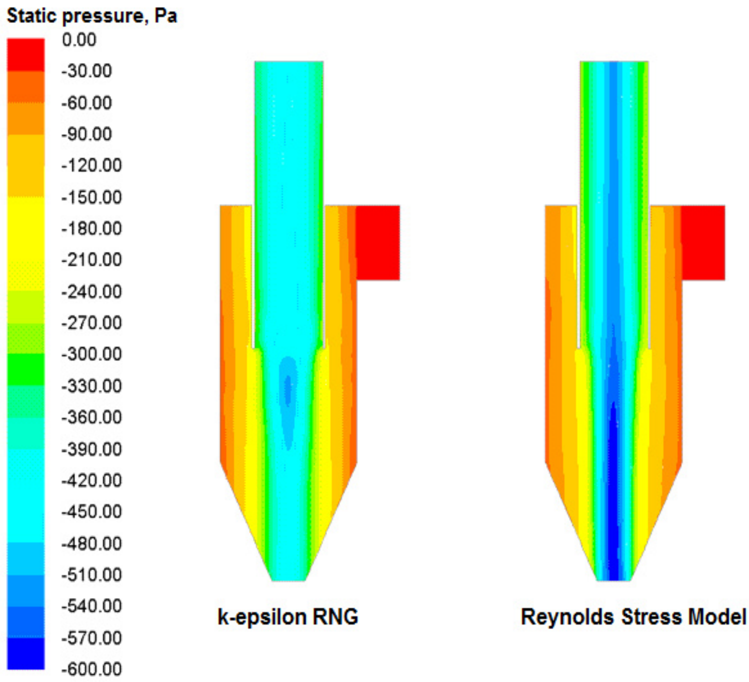

Fig. 3. Contours of static pressure obtained from CFD simulation

Separation efficiency

This section presents results of numerical simulation performed for the case in which two phase flow occurs. It has to be noted that all the computations for this case are carried out using RSM for turbulence modeling. Solid particles have been injected at the velocity of the gas through the surface of the cyclone inlet. The particle diameter distribution corresponds to the experiment described by Cristea et al. (1996). The minimum and maximum particle diameter is 1 and 24 μm respectively. The particle diameter distribution is described in terms of Rosin-Rammler distribution defined by 12.93 μm mean diameter and 1.056 spread parameter. The separation efficiency, which is defined as the mass of particles that is collected within the cyclone, is calculated using stochastic particle tracking method implemented in DPM. Fig. 4 presents the prediction of the cyclone grade efficiency curve obtained by the CFD simulation. The computed value of the total cyclone separation efficiency is around 96% and is very close to the experimental value.

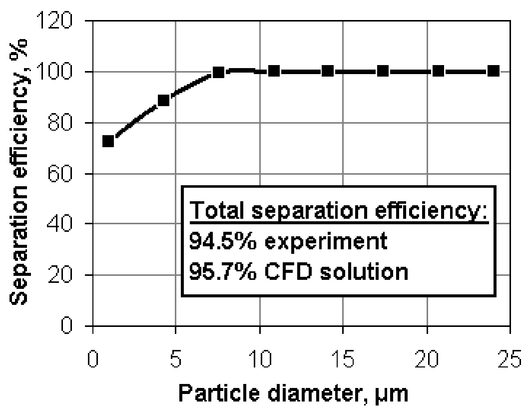

Fig. 4. Computed separation efficiencies of particles with different diameter sizes

In the simulation all the particle paths are tracked starting from the inlet surface. The particle is followed until it reaches the vortex finder exit (where it escapes) or the cone tip surface (where it is been collected). Fig. 5 presents the trajectories of particles with different diameter sizes. It can be noticed that particles with high diameters form a distinct strand and are mostly collected in the cyclone. On the other hand, particles with small diameter size are more likely to escape through the vortex finder and their movement path is more dispersed.

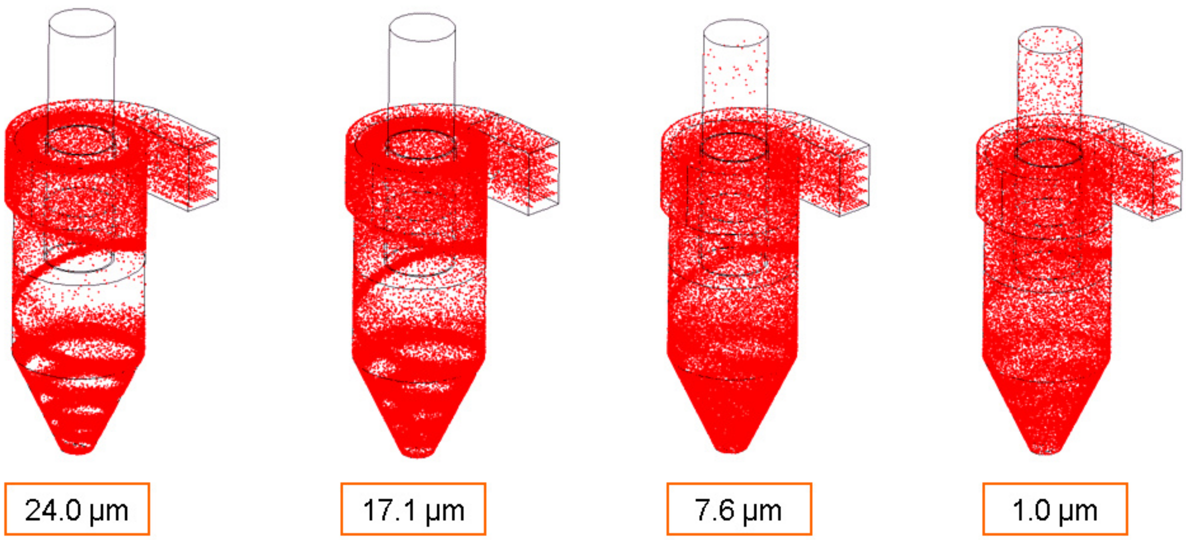

Fig. 5. Trajectories of particles with different diameter sizes

CONCLUSIONS

The flow pattern inside a cyclone is very complex. An accurate CFD simulation requires turbulence model which can handle swirling motion of highly turbulent fluid flow. The k-epsilon turbulence model which assumes eddy viscosity concept and isotropic behavior of the fluid is insufficient for this type of flows. The RSM which takes into account anisotropy by solving additional equations for Reynolds stresses gives flow field predictions in good agreement with the experimental data. However it requires more computational effort in comparison with k-epsilon models but it is not as numerically intensive as Large Eddy Simulation.

The pressure drop between the inlet and outlet duct is an important operating parameter of a cyclone separator. Numerical simulations performed with the use of RSM predict very well the pressure drop when compared to measurements.

The Euler-Lagrange approach (DPM) can be used to evaluate the cyclone separation efficiency. By tracking of a number of particle size classes the grade efficiency curve can be determined. The cyclone separation efficiency evaluated with results of simulation presented in this article matches the value obtained in the experiment. The two-phase model applied in the CFD simulation performed well, even though one way coupling between phases and no particle-particle interactions were assumed.

ACKNOWLEDGEMENTS

The work of WA, PK, RB, GW has been financed by the National Centre for Research and Development (NCBiR) under Grant No. SP/E/2/66420/10: Advanced technologies for energy generation, Activity No 2: Oxy-combustion technologies for pulverized fuel and fluidized bed boilers integrated with CO2 sequestration. The work of RB and GW has been also partially supported by RECENT project (European Commission, 7FP), Grant No. 245819. This financial support is gratefully acknowledged herewith.

REFERENCES

Cortes C., Gil A., (2007): *Modeling the gas and particle flow inside cyclone separators*, Progress in Energy and Combustion Science, Vol. 33, pp. 409-452

Cristea E.D., Malfa E., Coghe A., (1996): *3-D Numerical simulation and measurement of strongly swirling heavy dust-laden flow inside a cyclone separator*, 3rd International Symposium on Engineering Turbulence Modelling and Measurements, May 27-29, Crete

Shalaby H., Pachler K., Wozniak K., Wozniak G., (2005): *Comparative study of the continuous phase flow in a cyclone separator using different turbulence models*, International Journal for Numerical Methods in Fluids, Vol. 48, pp.1175-1197

Slack M.D., Prasad R.O., Bakker A., Boysan F., (2000): *Advances in cyclone modelling using unstructured grids*, Transactions of the Institution of Chemical Engineers, Vol. 78, Part A, pp. 1098-1104

Solero G., Coghe A., (2002): *Experimental fluid dynamic characterization of a cyclone chamber*, Experimental Thermal and Fluid Science, Vol. 27, pp. 87-96

Wang B., Xu D.L., Chu K.W., Yu A.B., (2006): *Numerical study of gas-solid flow in a cyclone separator*, Applied Mathematical Modelling, Vol. 30, pp. 1326-1342

ANSYS-FLUENT documentation, ANSYS Inc.