CFD Letters www.cfdl.issres.net Vol. 4 (1) – March 2012

CFD-DP Modelling of Multiphase Flow in Dense Medium Cyclone

Okan Topcu[C]
Mechanical Engineering Department, Faculty of Engineering
TOBB Economy and Technology University, TURKEY

Received: 03/08/2011 – Revised 17/11/2011 – Accepted 18/11/2011

Abstract

A numerical study of the gas-liquid-solid multi-phase flow in a hydrocyclone is summarized in this paper. The turbulent flow of the gas and the liquid is modelled using the realizable k-epsilon turbulence model, the interface between the liquid and the air core is modelled using the Eulerian multi-phase model and the simulation of the particle flow described by the dense discrete phase model in which the data of the multi-phase flow are used. Separation efficiency, particle trajectories, split ratios, flow field and pressure drop are the examined flow features. The results show that the flow fields in the hydrocyclones are possible to simulate by realizable k-epsilon model which is a fast solver for turbulent flows. The cut size is achieved between 3 and 15 μm. The air-core development is observed to be a transport effect due to the velocity of surrounding fluid rather than a pressure effect. The approach offers a useful method to observe the flow of a hydrocyclone in relation to design of the system and operational conditions.

Keywords: multiphase flow; dense medium cyclone; realizable k-epsilon model.

1. Introduction

Hydrocyclone, which is a key unit operation in mineral process industry, in process design and in CFD techniques governing optimization simulations, usually has a single inlet that scatters the feed stream and separates the flow based on volumetric ratio. Hydrocyclones have been used in the coal cleaning, paper and pulp cleaning, sand removal, oil removal and starch washing applications and their working principle has been examined in detail in the literature [1, 2, 11]. Dense medium cyclone is a kind of hydrocyclone which is also known as heavy medium cyclone [3]. The success of the simulation methodology depends on mainly how good the outcomes are matching with the experimental data and the computational time it requires obtaining the desired solutions.

A numerical study of the gas-liquid-solid multi-phase flow in a hydrocyclone is summarized in this paper. The turbulent flow of the gas and the liquid is modelled using the realizable k-epsilon turbulence model, the interface between the liquid and the air core is modelled using the Eulerian multi-phase model and the simulation of the particle flow described by the dense discrete phase model in which the data of the multi-phase flow are used. Separation efficiency, particle trajectories, split ratios, flow field and pressure drop are the examined flow features. The results

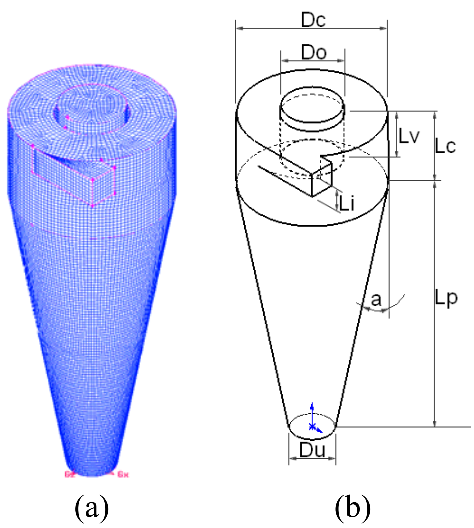
[C] Corresponding Author: O. Tobcu
Email: otopcu@etu.edu.tr Telephone: +90 312 292 6498
 PII: S2180-1363(12)4133-X

show that the flow fields in the hydrocyclones are possible to simulate by realizable k-epsilon model which is a fast solver for turbulent flows. In this study, the mesh played an important role in obtaining the right data throughout the flow field, likewise the volume ratios of the phases had been indispensable part in the simulation of the model.

Existing theoretical knowledge of hydrocyclones is still ineffective in explaining the governing flow characteristics. In addition applied and demanded experiments with complex and expensive models are not feasible in industrial studies. Wang et al. [8] reported a model which is designed for the examination of the different body dimensions and their effects on the performance of the hydrocyclone also shows good agreement with experimental data. Another reported model, which has examined the different configurations of the vortex finder, from Wang et al. [9] indicates that changing the length of the vortex finder affects the separation ratio of the particle sizes that change from finer to coarser or vice versa. The paper, which especially governs experimental methods and experimental verification, represented by Narashima et al. [10] has mentioned the effect of the shape and size of a hydrocyclone, as well. The paper presented by Nowakowski et al. [11] have focused on reviews and how algorithms could be extended for the CFD models of the hydrocyclones and also has addressed the problem of avoiding the air core in the calculations. The CFD methods differ based on equations which are used in the model. Some of these methods require a lot of computational time and power (like LES (Large Eddy Simulations)). Complex and expensive methods always do not mean that they will give the right solutions. When obtaining the solution of a hydrocyclone flow by means of CFD methods, it is mandatory to choose the right set of equations. The paper represented by Bhaskar et al. [12] has compared the different turbulence models and in their study they have shown that RSM (Reynolds Stress Method) has given better results than k-epsilon and k-epsilon RNG models. Another study by Krebs Engineers [13] has reported that higher order Reynolds Stress Turbulence model has provided the best agreement with velocity profiles. Using higher order equations or more complex models, sometimes provides the best solution but these methods also come with a big obstacle that is experienced operator and high computational needs. Slack et al [14] has described how automated tools could solve these obstacles. In their study Narashima et al [15] has developed a LES turbulence model for CFD calculations which is capable of predicting the flow patterns inside the hydrocyclone including accurate prediction of the size of the air core as well as the flow split.

The purpose of this paper is to show that k-epsilon realizable method is useful in obtaining the preliminary data of a hydrocyclone simulation without using the other complex and time taking CFD methods before deepening the analysis.

2. Geometry and Flow Description



(a) (b)

Figure 1. Mesh representation (a) and geometry (b) of the simulated DMC

The computational domain considered in the present study is the same as that presented by Chu et al. [3]. Figure 1 (a) shows the original model's computational domain. Figure 1 (b) shows the hydrocyclone design with its main geometrical parameters; the diameter of the inlet (Di), the diameter of the vortex finder (Do), the diameter of the apex (Du), the diameter of the cylindrical body (Dc), length of the cylindrical part (Lc), length of the vortex finder (Lv) and included angle (a). Table 1 shows these parameters and their values in SI units.

TABLE 1: PARAMETERS USED IN THE MODEL

Parameter	Symbol	Value [mm]
Diameter of the body	Dc	350
Side length of inlet	Li	65
Diameter of vortex finder	Do	145
Diameter of spigot	Du	105
Length of cylindrical part	Lc	200
Length of vortex finder	Lv	127
Length of conical part	Lp	695
Cone angel	a	19.5º

The inlet velocities of the water and the steel micro particles are both 3.7275 m/s. The pressure at the outlets is 101.325 kPa. Steel micro particles with a density of 8030 kg/m3 were injected at the inlet. The rest of the parameters used in the CFD simulation are given in Table 2. Note that the current study examines the possibility of the usage of the realizable k-epsilon model as a model for the hydrocyclone simulations, and hence was conducted under basic assumptions of a turbulent flow.

TABLE 2: GEOMETRY AND MESH PARAMETERS OF THE DMC

Phase	Parameter	Symbol	Units	Value
Solid	Density	ρ	kg/m^3	8030
	Particle diameter	d_i	μm	3, 15, 27, 39, 51 and 63 (mono sized particles of same densities)
	Particle velocity at inlet	u_p	m/s	3.7275
Gas	Density	ρ	kg/m^3	1.225
	Viscosity	μ	$kg/m/s$	1.8x10e-5
Liquid	Density	ρ	kg/m^3	998.2
	Viscosity	μ	$kg/m/s$	0.001
	Velocity at inlet	u	m/s	3.7275
Pressure at vortex finder outlet		-	Pa	101325
Pressure at spigot outlet		-	Pa	101325

3. Mathematical modelling and simulations

Momentum equations for three-dimensional turbulent flow were numerically solved based on the assumption that the flow is steady in mean and incompressible and the governing time averaged continuity. The Realizable k-ε turbulence model was used to treat turbulence and the corresponding transport equations.

GAMBIT 2.4 was used for the creation of geometry and meshing as the pre-processing tool. Cooper Hex/Wedge grids are used to divide the entire computational domain. The commercial CFD software ANSYS FLUENT 12.1 was used to solve numerically the governing equations based on the finite volume method along with the boundary conditions. The first order upwind scheme was used to discretize the governing equations. The Phase Coupled solver was employed to obtain the numerical solutions of the governing equations for the conservation of the mass, momentum, and energy and other scalar variables, such as, turbulence. The SIMPLE algorithm was used to relate velocity and pressure corrections to enforce mass conservation and to obtain the pressure field. Turbulence multiphase model is selected as dispersed. A “Velocity Inlet” boundary condition is used at the cyclone inlet.

The separation efficiency, the split ratio and the pressure drop are the characteristic parameters [4-7] of a hydrocyclone. In order to compare the predicted results with the other reported elsewhere [8, 9, 10, 13], the base hydrocyclone of a previous work [3] have been used.

The equation of motion of a particle in the fluid flow given by a force balance (in direction i) is:

$$\frac{du_p}{dt} = F_D\left(u - u_p\right) + \frac{g_i\left(\rho_p - \rho\right)}{\rho_p} \tag{1}$$

Where the gravitational acceleration is g_i in direction *i*, the drag force per unit particle mass is:

$$F_D = \frac{18\mu}{\rho_p d_p^2}\frac{C_D \mathrm{Re}}{24} \tag{2}$$

where u is the fluid phase velocity, the particle velocity is u_p, the dynamic viscosity of fluid is μ, ρ is the fluid density, ρ_p is the density of the particle, and the particle diameter is d_p. The relative Reynolds number is Re and defined as:

$$\mathrm{Re} \equiv \frac{\rho . d_p\left|u_p - u\right|}{\mu} \tag{3}$$

The value of the drag coefficient C_D was based on the study by Morsi and Alexander [16]

$$C_D = \alpha_1 + \frac{\alpha_2}{\mathrm{Re}} + \frac{\alpha_3}{\mathrm{Re}^2} \tag{4}$$

where α_1 , α_2 , and α_3 are constants that apply to smooth spherical particles.

Model validation has been made with the existing experimental data [8, 9, 10, 13] which matches with the obtained results. The method of modelling of water-gas flow in a DMC is very similar with the flow in a hydrocyclone. Therefore, the results obtained for a hydrocyclone match with the results of a DMC as well.

4. Results and discussion

Figure 2 shows the distributions of the flow which are actually snapshots of the transient simulation when the flow field is less sensitive to time except the small fluctuations caused by the air core. It can be seen from the figures that the regions corresponding to the air core have a peculiar property compared to the liquid phase. Representative snapshots of the simulated flow are given inside the Figure 2 as radial velocity, tangential velocity, static pressure and volume fraction. It can be seen from Figure 2(a) that the radial velocity gets negative values near the wall where fluid flow is dominant and it gets positive values inside the air core where the flow is upwards. Figure 2(b) shows that the tangential velocity increases at the inlet and diminishes at the centre of the DMC. The pressure gets negative values at the centre of the medium as seen in the Figure 2(c) which also shows that the highest pressure values are at the boundary of the system where the fluid flow is strong and forced to change course. As shown in Figure 2(d) the volume fraction of the gas phase occupies the centre and it gets its shape from the field has left from the liquid flow. Therefore the shape of the air core appears to be a helix which gets wider but at the same time gets much longer.

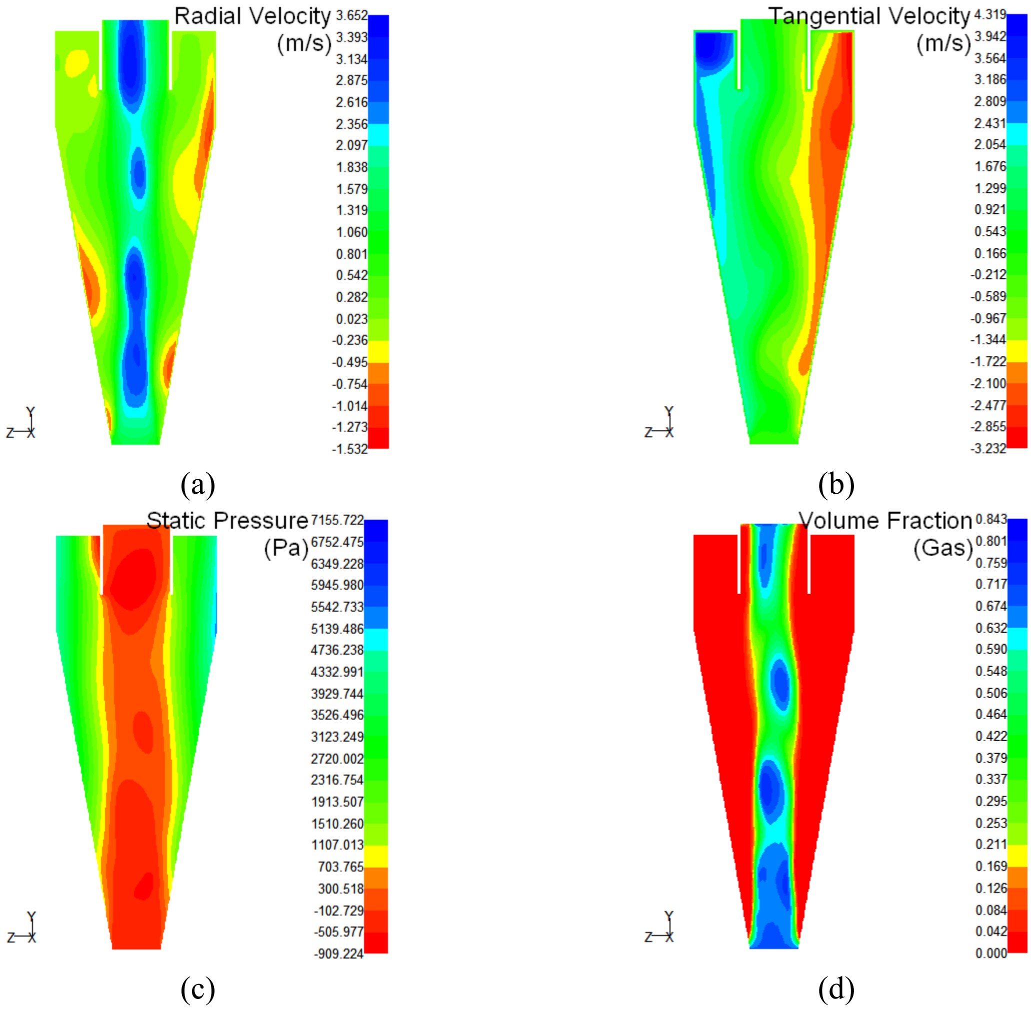


Figure 2. Distributions of radial velocity (a), tangential velocity (b), static pressure (c) and volume fraction (d) in the DMC

Figure 3 shows the liquid and gas phases' velocity vectors which are taken from the Y-Z plane at the centre of the computational domain. It can be seen from the figures that the region corresponding to the liquid phase has shown fast flow at the boundary of the DMC compared to the rest of the computational domain. In both figures, the mixture entered the DMC lost its velocity throughout the flow on the contrary air core which starts with small values of velocity has left the DMC with higher values. Figure 3(a) shows the liquid phase's main motion pattern which enters from the inlet, has followed a path that gradually gets closer to the spigot and, but most of it left the hydrocyclone from the vortex finder. This pattern is a well-known characteristic of hydrocyclones. Because air core is not a straight structure, velocity vectors of the air phase expand upwards following the main pattern of the core. Air core gets different shapes due to the design of the hydrocyclone. The air phase, which has changed the direction of its flow after interacting with the fluid phase at the boundary of the air core, has its vectors aiming upwards as the flow can be seen from Figure 3(b).

The flow field which decides the flow of solid particles is also important for the verification of the model. In this study particles of six different diameters were chosen to detect their distributions and trajectories in the hydrocyclone. Figure 4 shows the y axis position values of the solid particles inside the flow field. The residence time and trajectories of individual particles with different particle diameters are examined. This was done by simulating the set of different sized particles' flows. Figure 4 shows the obtained trajectories of the single particles as a representative of their sets.

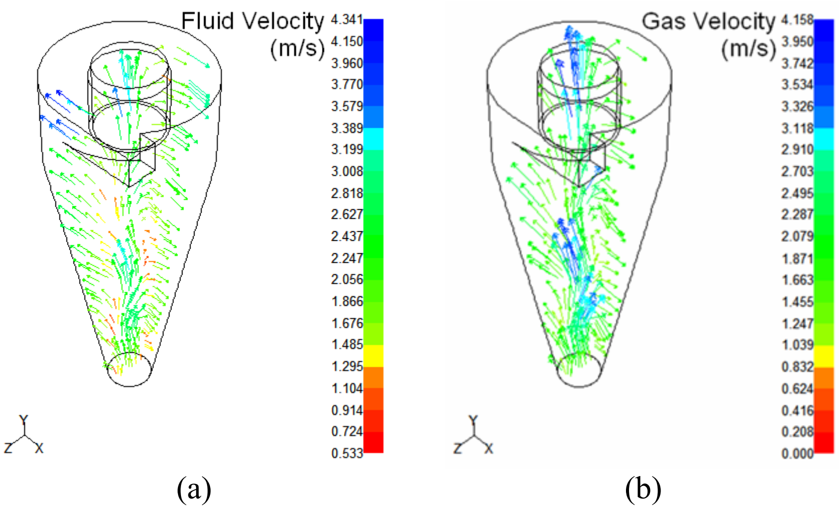


Figure 3. Distributions of fluid (a) and gas (b) velocity in the flow field

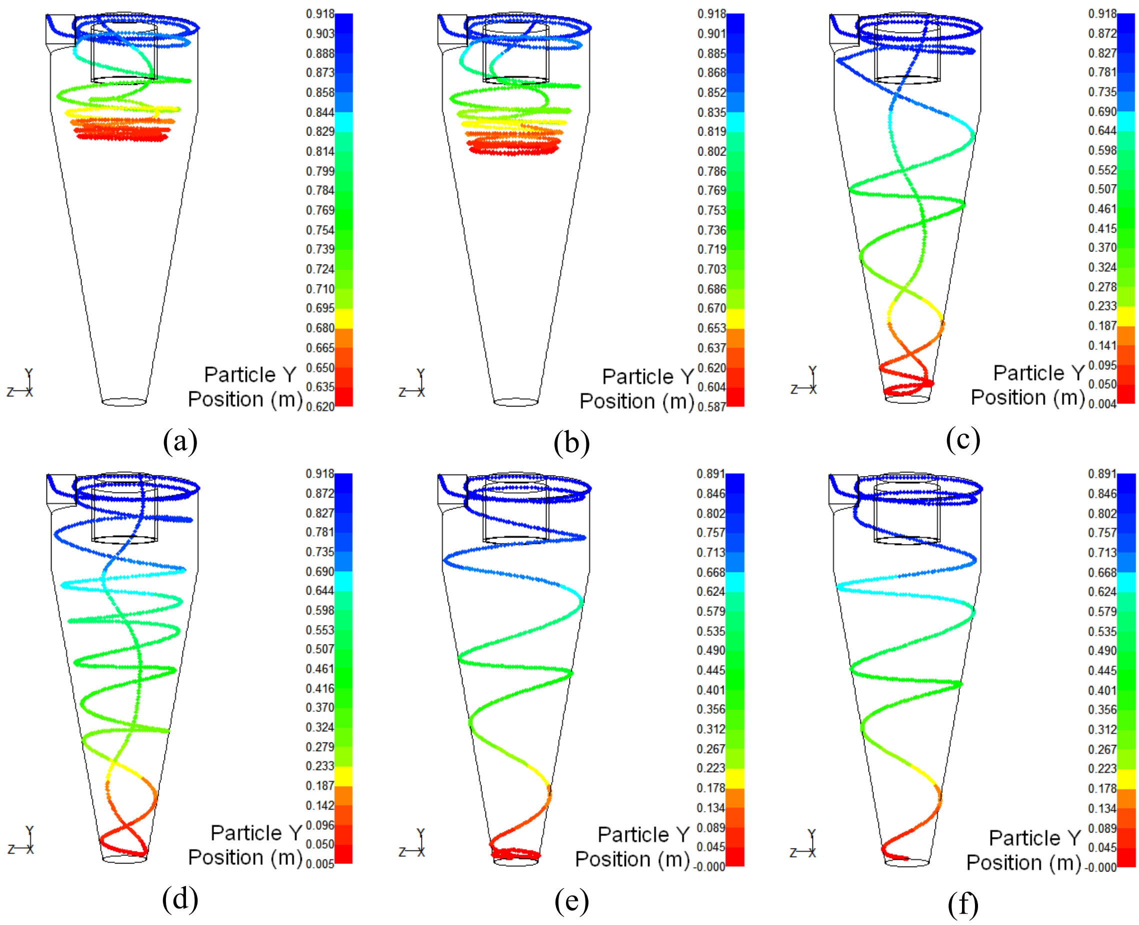


Figure 4. Y axis position values of particles 3 µm (a), 15 µm (b), 27 µm (c), 39 µm (d), 51 µm (e) and 63 µm (f) inside the flow field

In hydrocyclone applications, there is a possibility that a certain particle diameter might accumulate in the hydrocyclone and may also cause a blocking for the rest of the particles and the flow [3]. It can be seen from the Figure 5 that the y position value of the particle shown in Figure 4 and the time it spent in the flow. It can be seen from the Figure 4 and the Figure 5 that the particle which has a diameter value of 39 µm had a slow movement and slightly descending orbit, analogously the particle that has a diameter of 27 µm also had a slow movement around the core. The particles that had trapped inside are generally the ones which could not contact with the air core during the flow.

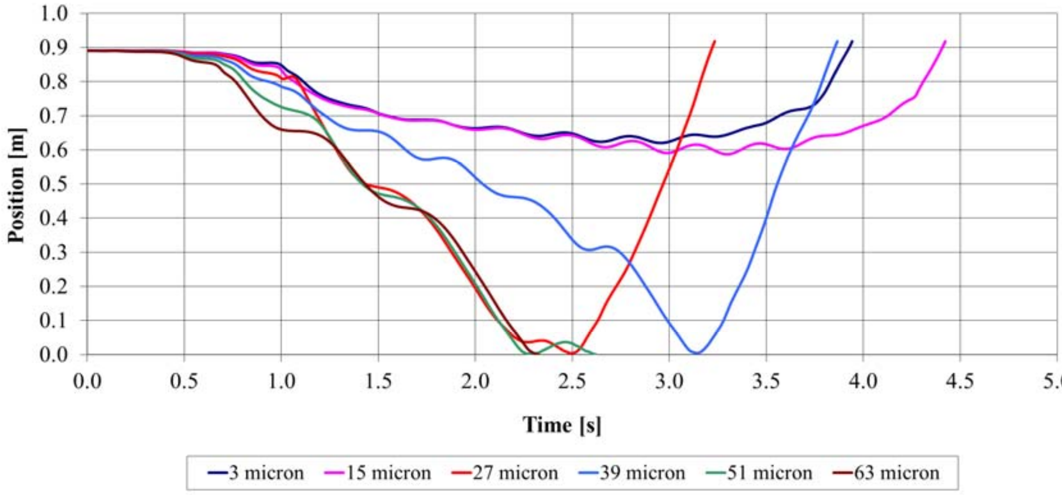


Figure 5. Y axis position values of particles with diameters 3 µm (a), 15 µm (b), 27 µm (c), 39 µm (d), 51 µm (e) and 63 µm (f) versus time

Figure 6 shows the velocity versus distance travelled data of the particles. In the hydrocyclone examined, the smallest particles such as 3 µm and 15 µm cumulated around the air core and then left from the vortex finder along with the upward flow. The bigger particles 51 µm and 63 µm which left early compared to the rest of the particles are concentrated on the boundary and collected from the spigot. In Figure 5 bigger particles had left the DMC first but in Figure 6 they have not shown very high velocity values because of the liquid phase had a spiral movement characteristic which has very little y velocity vector. On the contrary the air core which had high velocity values along the y axis accelerated the medium sized particles which entered the air core near the spigot. This circumstance can be seen from the Figure 6.

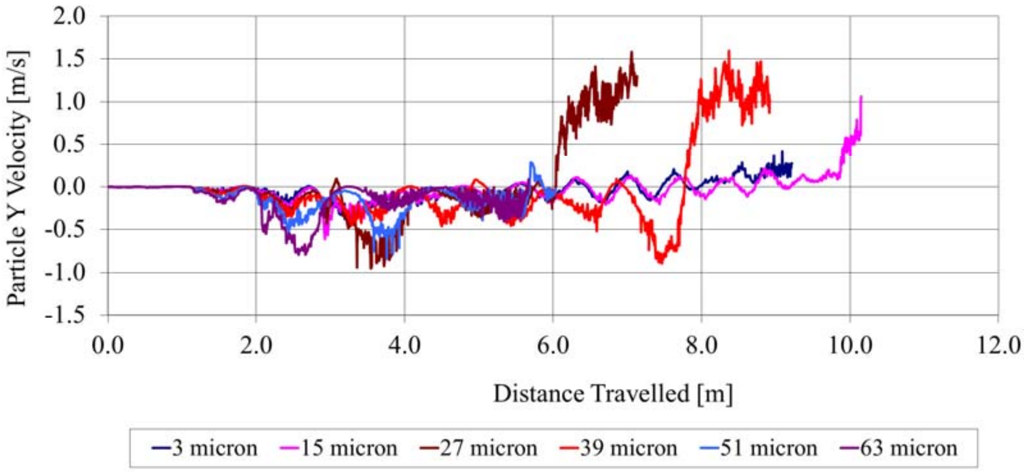


Figure 6. Y axis position values of single particles with diameters 3 µm (a), 15 µm (b), 27 µm (c), 39 µm (d), 51 µm (e) and 63 µm (f) versus distance travelled

Figure 7 shows the separation efficiency obtained in this study which matches with the documented best practices [8, 9, 10, 13]. These outcomes easily could be clarified from the diverse particle diameter values. As the body diameter increases, the orbit that particles travel increases. Consequently, it is expected for the bigger particles to lean against the boundary of the hydrocyclone and then exit from the bottom outlet. But the obtained graph of separation efficiency only shows the three main characteristics of the separation, owing to the fact that very little number of particles used as samples for this study. The lowest particle diameter that has left all from the spigot was between 51 µm and 63 µm which is a common value for hydrocyclones [8, 9, 10, 13]. The highest particle diameter which has obtained all from the vortex finder is between 3 µm and 15 µm which is also a common value for hydrocyclones [8, 9, 10, 13]. The slope of the graph happened to be very high on account of the a little number of particles used as samples and the other factors. These other factors which affects the separation efficiency are dimensions of the hydrocyclone which changes with design, particle characteristics which is a process constant and parameters of the model (like inlet speed and pressure values of the inlet and exit).

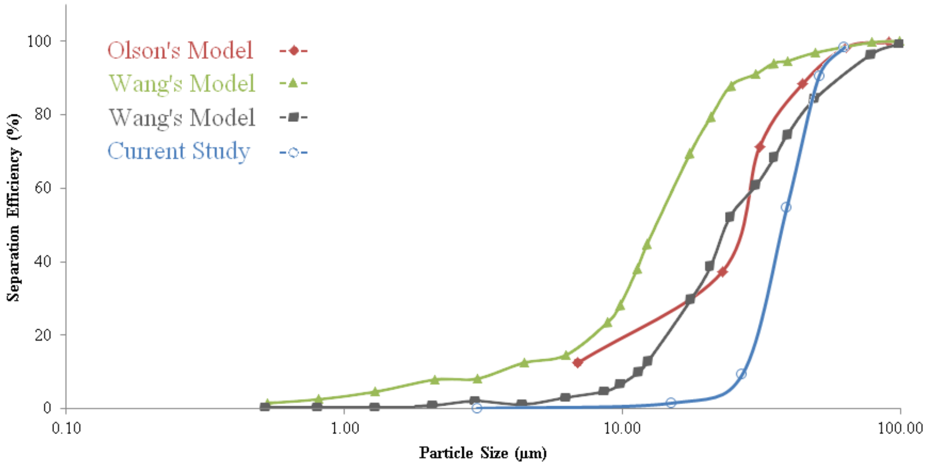


Figure 7. Separation efficiency

5. Conclusion

CFD techniques have shown appreciated potential in reproducing the fluid flow behaviour in a hydrocyclone. Flow of water through a hydrocyclone has been studied using ANSYS FLUENT Package. In order to understand the capability of the realizable k-epsilon turbulence model, the CFD model proposed by the previous work [3] has been used to identify the flow and the particle fields in the hydrocyclone. The realizable k-epsilon turbulence model has been used to simulate the anisotropic turbulent flow in a hydrocyclone, and the interface between the air core and the liquid is modelled using the Eulerian multi-phase model. Dense discrete phase model has been used to track the flow patterns of the micro particles in the multi-phase cyclone flow. Their applicability has been verified by the good agreement between the computationally calculated and experimentally measured flow fields. The achieved cut size has been between 3 and 15 µm and the air-core development has been shown to be a transport effect rather than a pressure effect.

Nomenclature

Di	Diameter of the inlet, mm
Do	Diameter of the vortex finder, mm
Du	Diameter of the apex, mm
Dc	Diameter of the cylindrical body, mm
Lc	Length of the cylindrical part, mm
Lv	Length of the vortex finder, mm
a	Included angle, degree
d_i	Particle diameter, µm
g_i	Gravitational acceleration, ms^{-2}

F_D	Drag force per unit particle mass, Nkg^{-1}
u	Fluid phase velocity, ms^{-1}
u_p	Particle velocity, ms^{-1}
ρ_p	Density of the particle, kgm^{-3}
d_p	Particle diameter, μm
Re	Relative Reynolds number
C_D	Drag coefficient
α_1, α_2, α_3	Constants
Greek letters	
ρ	Density, kgm^{-3}
μ	Viscosity, $kg(ms)^{-1}$
Sub scripts	
D	Drag
i	Axis direction
p	Particle

References

[1] King, R.P., A.H. Juckes, *Cleaning of fine coals by dense-medium hydrocyclone*, Powder Technology, 1984. 40: p. 147-160.

[2] Bretney, E., *Water Purifier*, US Patent, 1891. 453105.

[3] Chu, K.W., B. Wang, A.B. Yu, A. Vince, *CFD-DEM modeling of multiphase flow in dense medium cyclones*, Powder Technology, 2009. 193: p. 235-247.

[4] Cierpisz, S., B.S. Gottfried, *Theoretical aspects of coal washer performance*, International Journal of Mineral Processing, 1977. 4: p. 261-278.

[5] Hu, S., B. Firth, A. Vince, G. Lees, *Prediction of dense medium cyclone performance from large size density tracer test*, Minerals Engineering, 2001. 14(7): p. 741-751.

[6] Yancey, H.F., M.R. Geer, *Efficiency and sharpness of separation in evaluating coal-washery performance*, Transactions AIME, Mining Engineering, 1951. p. 507-517.

[7] Gottfried, B.S., *A generalization of distribution data for characterizing the performance of float-sink coal cleaning devices*, International Journal of Mineral Processing, 1978. 5: p. 1-20.

[8] Wang, B., A.B. Yu, *Numerical study of particle–fluid flow in hydrocyclones with different body dimensions*, Minerals Engineering, 2006. 19: p. 1022-1033.

[9] Wang, B., A.B. Yu, *Numerical study of the gas–liquid–solid flow in hydrocyclones with different configuration of vortex finder*, Chemical Engineering Journal, 2008. 135: p. 33-42.

[10] Narasimha, M., R. Sripriya, P.K. Banerjee, *CFD modelling of hydrocyclone–prediction of cut size*, International Journal of Mineral Processing, 2005. 75: p. 53-68.

[11] Nowakowski, A.F., J.C. Cullivan, R.A. Williams, T. Dyakowski, *Application of CFD to modelling of the flow in hydrocyclones. Is this a realizable option or still a research challenge?*, Minerals Engineering, 2004. 17: p. 661-669.

[12] Bhaskar, K.U., Y.R. Murthy, M.R. Raju, S. Tiwari, J.K. Srivastava, N. Ramakrishnan, *CFD simulation and experimental validation studies on hydrocyclone*, Minerals Engineering, 2007. 20: p. 60-71.

[13] Olson, T.J., R. Van Ommen, *Optimizing hydrocyclone design using advanced CFD model*, Minerals Engineering, 2004. 17: p. 713-720.

[14] Slack, M.D., S. Del Porte, M.S. Engelman, *Designing automated computational fluid dynamics modeling tools for hydrocyclone design*, Minerals Engineering, 2004. 17: p. 705-711.

[15] Narasimha, M., Mathew Brennan, P.N. Holtham, *Large eddy simulation of hydrocyclone–prediction of air-core diameter and shape*, International Journal of Mineral Processing, 2006. 80: p. 1-14.

[16] Morsi, S.A. and A.J. Alexander, *An investigation of particle trajectories in two-phase flow systems*, Journal of Fluid Mechanics, 1972. 55: p. 193-208.