

NUMERICAL STUDY OF LIQUID-SOLID SEPARATION PROCESS INSIDE THE HYDROCYCLONES WITH DOUBLE CONE SECTIONS

George IPATE¹, Tudor CĂȘĂNDROIU²

Obiectivul major al acestui studiu a fost ca prin utilizarea metodelor numerice moderne, să se analizeze mișcarea particulelor solide într-un hidrociclon cu două secțiuni conice folosit la epurarea apelor uzate. Aceasta cercetare cuprinde calculul curenților de fluid în hidrociclon, incluzând traiectoria particulelor, caderea de presiune și eficiența separării. Hidrociclonul a fost proiectat ținând cont de relațiile geometrice dintre diametrul ciclonului, aria secțiunii conductei de alimentare, conducta de suprascurgere, orificiul de evacuare, precum și de timpul necesar separării particulelor. Rezultatele obținute prin calcul numeric sunt verificate destul de bine prin compararea cu datele din literatura de specialitate. Predictia vitezei particulelor sau recuperării particulelor solide pe fracții de dimensiuni în hidrociclon în funcție de proprietățile fizice ale fluidului, de încărcarea cu solide sau debitul de fluid are o precizie ridicată

The major objective of this study was, using the modern numerical techniques, to investigate particle transport processes within a hydrocyclone with double cone sections, where the wastewater is depurated. This investigation consists of calculations of the fluid flow inside the hydrocyclone, including particle trajectory, pressure losses and separation efficiencies. The hydrocyclone has been modeled with the proper geometrical relationship between the cyclone diameter, inlet area, vortex finder, apex orifice, and sufficient length providing retention time to properly separate particles. Obtained results of calculations were numerically verified as well as compared with results published in the subject literature. The model will predict the velocity of particles and fractional recovery of solid particles given the dimensions of the cyclone, the physical properties of the fluid, and the volumetric flow rate.

Keywords: hydrocyclones; model; mixture; performance; geometrical proportions; efficiency

1. Introduction

Hydrocyclones are widely used in the treatment of waste water streams from poultry processing to remove feathers, sand and grit, fatty solids, and other wastes. They are essentially a passive device with a short residence time,

¹ Assist., Depart. of Biotechnical Systems, University "Politehnica" of Bucharest, ROMANIA

² Prof., Depart. of Biotechnical Systems, University "Politehnica" of Bucharest, ROMANIA

which makes them easy to run. A review of earlier simplified models for the so-called dilute flow separation in the hydrocyclone, i.e. for relatively small solid concentrations, can be found in a book by Svarovsky (1981) [1]. Mathematical models based on fluid mechanics involving simplifying assumptions have clarified some aspects of the hydrocyclone vortex-flow problem was developed by Monredon et al (1992) [2]. Numerical calculations of the separation of suspensions with different particle size distribution in the hydrocyclone computing by Dueck (1998) show that feed solid concentration affects the separation parameters of the hydrocyclone [3]. However the fact that they treat particle-laden flows means that wear and its minimization is a major problem.

The main goal of the paper was to create a computer model of a cyclone separator unit operation. This model allows the user to either design a new cyclone or rate the performance of an existing cyclone. There are many calculation options available to the user. Additional options, such as series cyclones and dip leg sizing, can be incorporated into the model to increase the usefulness of the simulation. Another major goal of the project is to evaluate the performance of the computer model. This was done using literature examples and industrial cyclone data [10,11,15,16]. The literature examples were used to produce performance curves on graphs.

2. Geometrical model

In this study a commercial CFD (Computational Fluid Dynamics) package called Ansys is applied to build a computational model and calculate results. Computational Fluid Dynamics is the technique which solves problems involving fluid flow by means of computer-based simulation. The technique spans also a wide range of industrial and non-industrial application areas. The coding of the program is in FORTRAN 77.

In the first stage of this work a parametric three-dimensional geometrical model of the hydrocyclone with multiple cone sections, was designed. For this purpose a CAD-type software (called Solid Works), capable of designing even very complex geometrical objects, was applied (figure 1). Geometry transferred from Solid Works to CFD package preprocessor is much more flexible and accurate than that created with preprocessor itself.

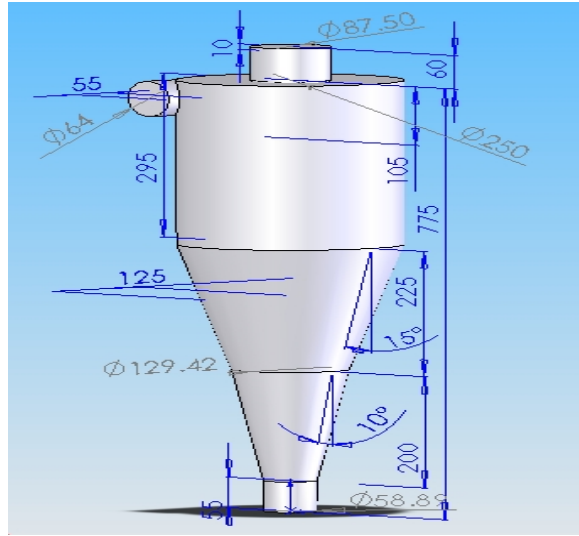


Fig. 1 Hydrocyclone dimensions and geometry

The main parameter is the hydrocyclone diameter $D_c=250$ mm. This is the inside diameter of the cylindrical feed chamber. The basic area of the inlet nozzle at the point of entry into the feed chamber approximates $0.05D_c^2$. The size of the vortex finder equals $0.35D_c$. The next section is the double conical sections, typically referred to as the cone section. The included angle of the first, respectively, second cone section is normally 15° , respectively 10° , and, similar to the cylinder section, provides retention time. The termination of the cone section is the apex orifice and the critical dimension is the inside diameter at the discharge point. The size of this orifice is determined by the application involved and must be large enough to permit the solids that have been separated to underflow to exit the cyclone without plugging. The normal minimum orifice size would be $0.1D_c$ and can be as large as $0.35D_c$. A mixture of fluid and particles is fed tangentially into the upper or larger diameter part of the hydrocyclone whit double cone sections. The resulting spinning effect forces solids to the wall of the device and they exit from the bottom or apex of the cone, while the cleaned liquid and fine particles exits at the top.

3. Numerical model.

Mathematical model of the coupled fluid flow in the hydrocyclones is based on the classical continuity, momentum and turbulent kinetic energy equations[4].

Lagrangian Tracking Implementation. Particle transport modeling is a type of multiphase model, where particulates are tracked through the flow in a Lagrangian way, rather than being modeled as an extra Eulerian phase. The full

particulate phase is modeled by just a sample of individual particles. The tracking is carried out by forming a set of ordinary differential equations in time for each particle, consisting of equations for position, velocity and masses of species. These equations are then integrated using a simple integration method to calculate the behavior of the particles as they traverse the flow domain. The following section describes the methodology used to track the particles.

Integration. The particle displacement is calculated using forward Euler integration of the particle velocity over time step, δt .

$$x_i^n = x_i^o + v_{pi}^o \cdot \delta t \quad (1)$$

Where the superscripts o and n refer to old and new values respectively and v is the particle velocity. In forward integration, the particle velocity calculated at the start of the time step is assumed to prevail over the entire step. At the end of the time step, the new particle velocity is calculated using the analytical solution to (Eqn. 3):

$$v_p = v_f + (v_p^o - v_f) \exp\left(-\frac{\delta t}{\tau}\right) + \tau \cdot F_{all} (1 - \exp\left(-\frac{\delta t}{\tau}\right)) \quad (2)$$

The fluid properties are taken from the start of the time step. For the particle momentum, v_p^o would correspond to the particle velocity at the start of the time step. In the calculation of all the forces, many fluid variables, such as density, viscosity and velocity are needed at the position of the particle. These variables are always obtained accurately by calculating the element in which the particle is traveling, calculating the computational position within the element, and using the underlying shape functions of the discretisation algorithm to interpolate from the vertices to the particle position.

Momentum Transfer. The forces acting on the particle which affect the particle acceleration are due to the difference in velocity between the particle and fluid and due to the displacement of the fluid by the particle. The equation of motion for such a particle was derived by Basset, Boussinesq and Oseen for a rotating reference frame:

$$\frac{\pi d^3 \rho_p}{6} \frac{dv_p}{dt} = \frac{1}{8} \pi \rho_f d^2 C_D |v_f - v_p| (v_f - v_p) + F_b - \frac{\pi d^3}{6} (\rho_p - \rho_f) \omega \times (\omega \times R) - \frac{\pi d^3 \rho_p}{6} \omega \times v_p \quad (3)$$

where d is the particle diameter, v is velocity, ρ is density, C_D is the drag coefficient, F_b is the buoyancy force due to gravity, ω is the rotational velocity, is a vector directed from the axis of rotation, subscript f refers to the fluid and the subscript p refers to the particle. The term on the left-hand side is a summation of all of the forces acting on the particle expressed in terms of the particle acceleration. In this form, the equation of motion has particle acceleration terms on both sides of the equation and would require solution by an iterative method.

- *Term I* is the drag force acting on the particle:

$$F_D = \frac{1}{8} \pi \rho_f d^2 C_D |v_f - v_p| (v_f - v_p) \quad (4)$$

• *Term II* is the buoyancy force due to gravity, which for a spherical particle is given by:

$$F_b = \frac{\pi d^3}{6} (\rho_p - \rho_f) g \quad (5)$$

Where g is the gravitational acceleration.

• *Term III* is the centripetal force, present only in a rotating frame of reference:

$$F_{centripetal} = -\frac{\pi d^3}{6} (\rho_p - \rho_f) \omega \times (\omega \times R) \quad (6)$$

• *Term IV* is the Coriolis forces, present only in a rotating frame of reference:

$$F_{coriolis} = -\frac{\pi d^3 \rho_p}{3} \omega \times v_p \quad (7)$$

Where v_p is the particle velocity, ω the angular velocity of the rotating frame and r is the vector from the axis of rotation to the current particle position.

Turbulence in Particle Tracking. In turbulent tracking, the instantaneous fluid velocity is decomposed into mean, $\overline{v_f}$, and fluctuating, v'_f , components. Now particle trajectories are not deterministic and two identical particles, injected from a single point, at different times, may follow separate trajectories due to the random nature of the instantaneous fluid velocity. It is the fluctuating component of the fluid velocity which causes the dispersion of particles in a turbulent flow. The model of turbulent dispersion of particles that is used assumes that a particle is always within a single turbulent eddy. Each eddy has a characteristic fluctuating velocity, v'_f , lifetime, τ_e , and length, l_e . The turbulent velocity, eddy and length and lifetime are calculated based on the local turbulence properties of the flow:

$$v'_f = \Gamma (2k/3)^{0.5} \quad (8)$$

$$l_e = \frac{C_\mu^{3/4} k^{3/2}}{\varepsilon} \quad (9)$$

$$\tau_e = l_e / (2k/3)^{1/2} \quad (10)$$

Where k and ε are the local turbulent kinetic energy and dissipation, respectively, and C_μ is turbulence constant. The variable Γ is a normally distributed random number which accounts for the randomness of turbulence about a mean value. Because of this randomness, each component of the fluctuating velocity may have a different value in each eddy.

4. CFD Experiments

In this part of our work CFD package Ansys was used to study only the hydrodynamic behavior of a liquid-solid flow in a hydrocyclone. Main region of interest was the particles solids, where radial particle velocity profiles were computed as a function of system parameters, e.g. particle size and density or inlet velocity. Concerning low volume fractions of a solid phase the Eulerian-Eulerian multiphase model and the standard k- ϵ turbulence model were used. The steady-state problem formulation was used to simulate the start-up of the apparatus. One type of computational grid was used. It was unstructured triangular grid (Fig. 2) with 1693 nodes, 7594 tetrahedron-elements and 1376 faces. The workstation used for all simulation was Notebook Dell Inspiron-1501, 799 MHz, 500 MB RAM. The average CPU time consumed for each iteration was 4.9 s. Convergence was assumed to be reached when no further changes in the interesting happened, and never before the residuals decreased to 10^{-3} .

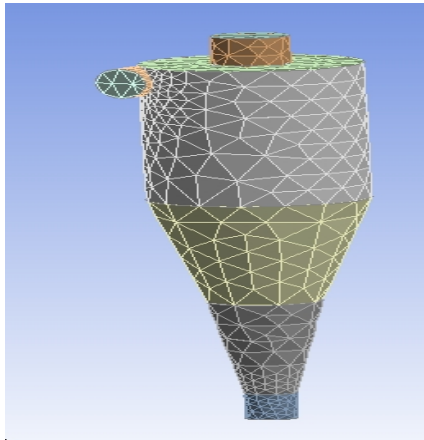


Fig. 2 Triangular structure grid of hydrocyclone

CFD simulation of given multiphase system were computed for different mixture volumetric concentration in range between 1.5-3.5 %. Also different particle sizes were concerned. Chosen results were compared with literature experimental velocity profiles. Water was used as a continuous primary phase. Two different materials were used as a solid phase. It was all rubber scrubs, sands with densities of 1 100 and 2 650 kg/m³ respectively. Particles were small spheres with uniform distribution diameter by diameter in range of 5 to 400 μm (figure 3).

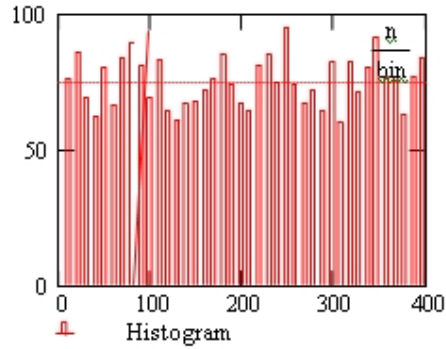


Fig. 3 Uniform distribution of solid particles

5. Simulation results and discussion

The flow chart shown below illustrates the general solution procedure. The solution of each set of equations shown in the flow chart consists of two numerically intensive operations. For each time step: The non-linear equations are linearised (coefficient iteration) and assembled into the solution matrix. The linear equations are solved (equation solution iteration) using an LES method. The timestep iteration is controlled by the physical timestep (global) or local timestep factor (local) setting to advance the solution in time for a steady state simulation. In this case, there is only one linearization (coefficient) iteration per timestep.

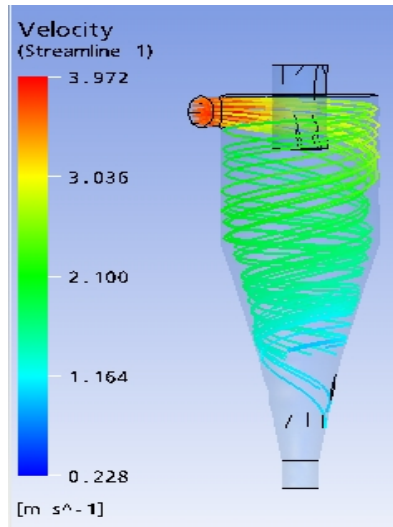


Fig. 4 The velocity profiles in hydrocyclones

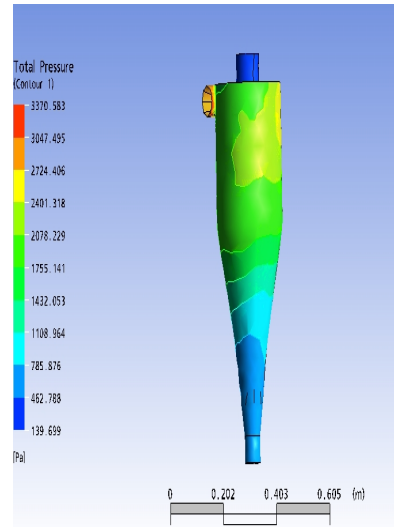


Fig. 5 Distribution of total pressure

In Fig. 4 are shown general velocity profiles for different outlet size to particle diameter ratio. This geometrical parameter is very important for proper apparatus operation. If this value is smaller than $5 \mu\text{m}$ there is high possibility that the doming can occur and the particle flow in the cell can be blocked. In agreement with published data the descending particle velocity is increasing with growing outlet size to particle diameter ratio (d_{50}/d) [5,6,7]. In other words smaller particles move faster. In comparison with experimental data the computed velocities are approximately at the same level.

Table 1

Results from the simulation –rubber

Δp	Cv	Q_{mf}	Q_{mp}	Q_{ms}	v_s	ρ_s	Q_{mclar}	Q_{mrec}	Reynolds
[kPa]	[%]	[kg/s]	[kg/s]	[kg/s]	[m/s]	[kg/m ³]	[kg/s]	[kg/s]	[]
0.696	0.7	4.063	0.0316	4.095	1.28	997.721	2.39E+00	1.71E+00	7.07E+04
1.235	1.3	5.416	0.0787	5.494	1.71	998.339	3.23E+00	2.27E+00	6.08E+04
1.689	1.9	6.318	0.135	6.453	2.008	998.957	3.81E+00	2.6444	5.25E+04
2.785	2.4	8.072	0.219	8.291	2.579	999.472	4.9161	4.9161	5.48E+04
4.979	3.1	10.653	0.376	11.029	3.428	1000	6.5734	4.4553	7.28E+04

Table 2

Results from the simulation –sand

Δp	Cv	Q_{mf}	Q_{mp}	Q_{ms}	v_s	ρ_s	Q_{mclar}	Q_{mrec}	Reynolds
[kPa]	[%]	[kg/s]	[kg/s]	[kg/s]	[m/s]	[kg/m ³]	[kg/s]	[kg/s]	[]
1.612	0.7	6.191	0.116	6.307	1.944	1009.00	3.64E+00	2.67E+00	1.09E+05
1.819	1.3	6.627	0.232	6.859	2.093	1018.00	3.97E+00	2.89E+00	6.88E+04
2.241	1.9	7.265	0.374	7.639	2.309	1028.00	4.43E+00	3.2062	6.30E+04
3.753	2.4	9.241	0.604	9.845	2.952	1037.00	5.8006	4.0441	6.02E+04
5.337	3.1	10.772	0.916	11.688	3.466	1048.00	6.9490	4.7386	6.29E+04

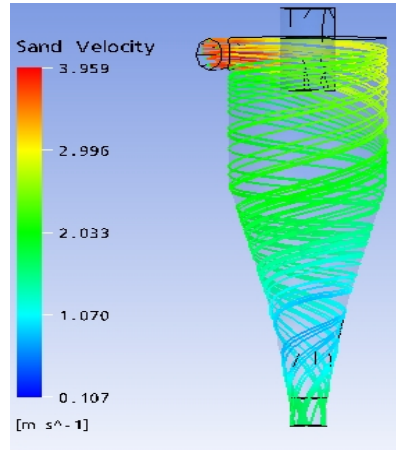


Fig. 6 Distribution velocity particles sand

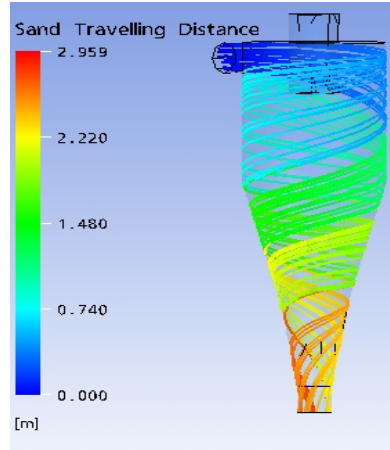


Fig. 7. Distribution of traveling distance

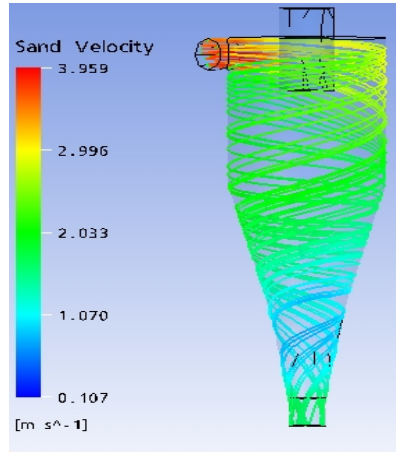


Fig. 6 Distribution velocity particles sand

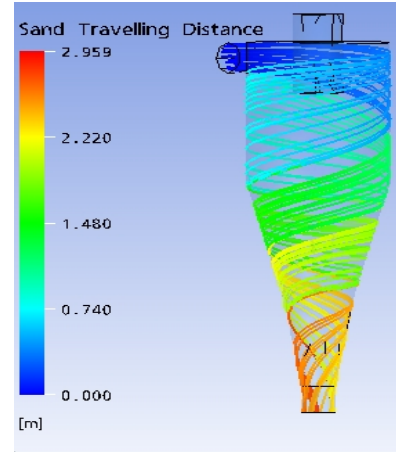


Fig. 7. Distribution of traveling distance

Fig. 5 documents effect of water inlet velocity on particle flow in the distribution of total pressure. In accordance with experiments the computed particle velocity increases with increasing inlet flow rate [8,9,10]. In this case the difference between simulation and experiment is slightly more triple. Experiments also show that for the higher particle density, the effect of inlet flow rate is smaller. The effect of polydispersion of particulate phase is just the same. The results from varying inlet velocity conditions are shown in Table 2, 3 and 4. Figs. 6, 7, 8 and 9 show examples of the results from the CFD analysis, all of them apply for the operating inlet velocity condition at 3.466 m/s.

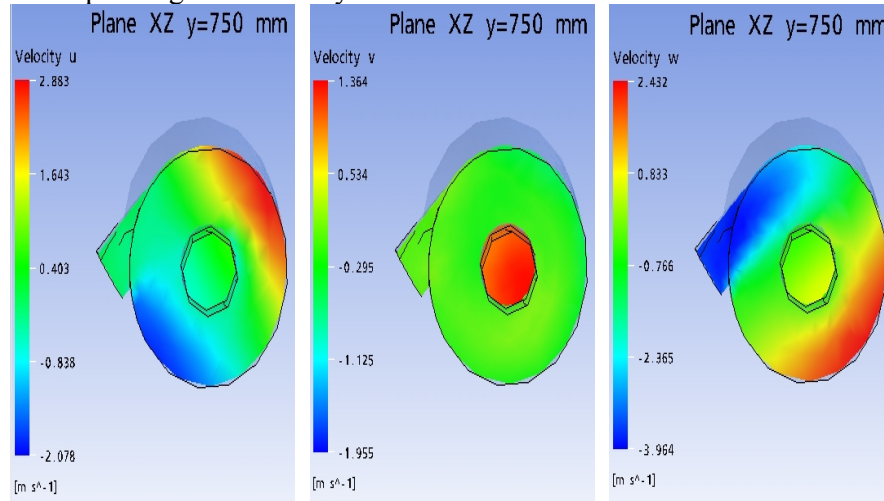


Fig. 8 Velocity u, v and w profiles in plane XZ at distance y=750mm

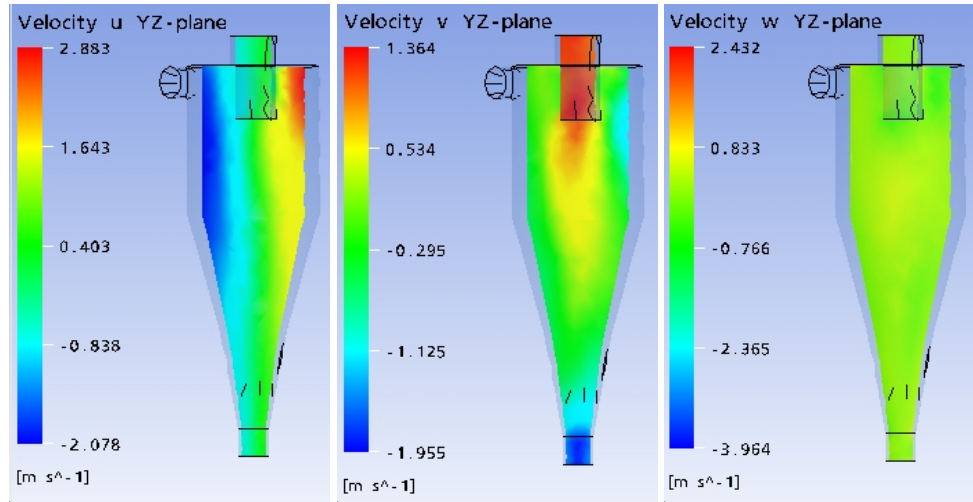


Fig. 9 Velocity u, v and w profiles in plane YZ at x=0mm

6. Separation efficiencies

The performance of hydrocyclone classifiers is determined using efficiency curves, which show the probability of a particle reporting to the hydrocyclone underflow as a function of its size [10]. The classification function can be expressed closely by equations such as (Plitt, 1976). Investigations have shown that this curve remains constant over a wide range of cyclone diameters and operating conditions when applied to a slurry containing solids of a single specific gravity and a typical or normal size distribution such as those encountered in most grinding circuits [10,11].

Table 4

Recovery efficiency of underflow			
Rubber		Sand	
Inlet Velocity (m/s)	Recovery at underflow (%)	Inlet Velocity (m/s)	Recovery at underflow (%)
1.28	99.33	1.944	99.13
1.71	99.22	2.093	99.01
2.008	98.59	2.309	98.67
2.579	98.08	2.952	98.37
3.428	97.71	3.466	97.81

Equation (11) gives a mathematical relationship which can be used to calculate the reduced recovery [10]. This recovery, along with the bypassed solids, is used to predict the complete size distribution for the underflow product.

$$R_r = \frac{(e^{4X} - 1)}{(e^{4X} + e^4 - 2)} \quad (11)$$

Where R_r is recovery to underflow on corrected basis, and X is ratio between particle diameter d and d_{50c} cut size particle diameter.

$$d_{50c} = \frac{50.5 \cdot D_c^{0.46} \cdot D_i^{0.6} \cdot D_o^{1.21} \cdot e^{(0.063 \cdot C_v)}}{D_u^{0.71} \cdot h^{0.38} \cdot Q^{0.45} \cdot (\rho_s - \rho)^{0.5}} \quad (12)$$

Figure 10 also shows that the actual recovery curve does not decrease below a certain level. This indicates that a certain amount of material is always recovered to the underflow and by passes classification in concordance whit Kawatra (2005).

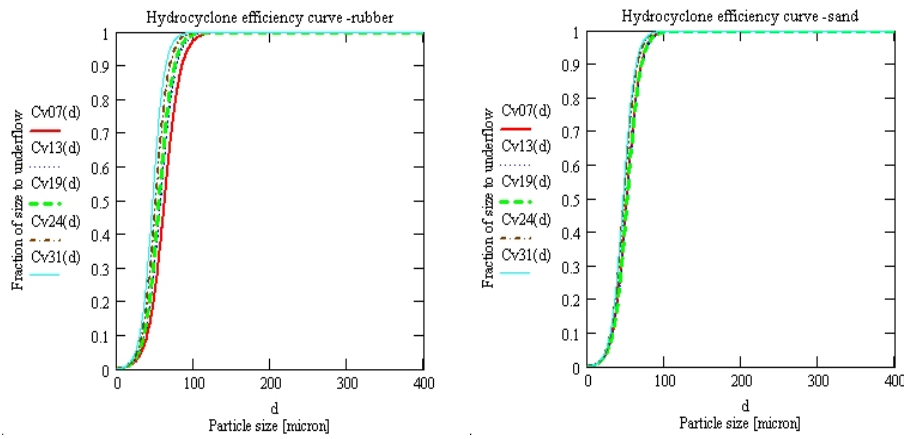


Fig. 10 Cumulative distribution curves

If a comparison is made between the minimum recovery levels of solids to the liquid that is recovered, they are found to be equal. Therefore it is assumed that a percent of all size fractions reports directly to the underflow as bypassed solids in equal proportion to the liquid split. As the d_{50c} point changes from one application to another, the recovery curves shift, along the horizontal axis.

7. Conclusions

As the aim of this phase of this work was to predict particle velocity profiles in hydrocyclones whit multiple cones by a CFD simulation and compare them with experimental profiles, the results are satisfactory. Simulation captured important trends in influence of system parameters (particle size and density, inlet velocity of carrier phase) on particle velocity. However quantitative agreement is not so good, simulation show faster moving particles then experiments. This trend occurs in all simulation results and probably it is due to neglecting the shear stress between front and rear walls and particles. Numerical results also show that type and shape of computational grid are not elementary parameters [13].

The experiments further show that techniques used for particle velocity profiles determination and experimental data evaluations are convenient. The experimental results for given apparatus show that for particles with higher density it is necessary to provide the higher inlet water velocities to ensure particle circulation, as expected. Moreover particle motion in hydrocyclones is strongly affected by cyclones geometry and entire apparatus construction [14,15,16]. The accurate representation of a computational domain allows researching into how changes in the shape of hydrocyclone will influence its operating performance. The ability of modern supercomputers allows the approximation of three-dimensional flow pattern in hydrocyclones to be investigated.

REFERENCES

- [1]. *Svarovsky, L.*, Solid-liquid Separation, London-UK:Butterworths, 1981.
- [2]. *Monredon T.C., Hsieh K.T., Rajamani R.K.*, Fluid flow of the hydrocyclones: an investigation of the devices dimensions, International Journal of Mineral Processing, 1992, **35**, pp 65-83.
- [3]. *Dueck J., Matvienko O., Neeße Th.*, Numerical calculations of the separation of dense suspensions with different particle size distribution in the hydrocyclone, In Proceedings of the 9th Workshop on Two-Phase-Flow Predictions, edited by M. Sommerfeld, Merseburg, , 1999, pp. 194-202.
- [4]. *** ANSYS – Finite Element System, User Guide, 1995.
- [5]. *Křišťál J, Jiříčný V. and Staněk V*, The CFD simulation and an experimental study of hydrodynamic behavior of liquid-solid flow, 2005.
- [6]. *Hsieh K.T. and Rajamani, R.K.*, Mathematical Model of the Hydrocyclone Based on Physics of Fluid Flow, *AIChE Journal*, 1991, **37**, (5), pp. 735-746.
- [7]. *Coelho, M. A. Z., and Medronho, R. A.*, A Model for Performance Prediction of Hydrocyclones, *Chemical Engineering Journal*, 2001, **vol. 84**, No. 1, pp. 7-14.
- [8]. *Del Villar, R., and Finch, J. A.*, “Modelling the Cyclone Performance with a Size Dependent Entrainment Factor”, *Minerals Engineering*, **vol. 5**, No. 6, 1992, pp. 661-669.
- [9]. *Castilho, L.R. and Medronho, R.A.*, A Simple Procedure for Design and Performance Prediction of Bradley and Rietema Hydrocyclones, *Minerals Engineering*, **13**, (2), 2000 pp. 183-191.
- [10]. *Kawatra S. K.*, Optimization of Comminution Circuit Throughput and Product Size Distribution by Simulation and Control Final Technical Report, 2005.
- [11]. *Tim Olson*, Custom Simulation Tool Helps Develop Cyclone with Sharper Recovery Profile, Journal articles by Fluent users JA-231,2006.
- [12]. *Medronho, J. Schuetze R. A. and Deckwer W.-D.*, Numerical Simulation Of Hydrocyclones For Cell Separation, *Latin American Applied Research* , 2005, **35** :1-8.
- [13]. *Neesse, Th., Dueck, J., and Minkov L.*, Separation of Finest Particles in Hydrocyclones, *Minerals Engineering*, **vol. 17**, 2004, pp. 689-696.
- [14]. *Plitt, L.R.*, A Mathematical Model of the Hydrocyclone Classifier, *CIM Bull.* 69, 1976, 114.
- [15]. *Peterson, R. D., and Herbst, J. A.*, Effects of Two-Stage Hydrocyclone Classification on Mineral Processing Plant Performance, *Canadian Metallurgical Quarterly*, **vol. 23**, No. 4, 1984, pp. 383-391.
- [16]. *Richard A. Arterburn*, The sizing and selection of hydrocyclones, , Krebs Engineers, Menlo Park, CA., 1976.