CFD Simulation Of The Flow Field In A Uniflow Cyclone Separator

I. Mokni, H. Dhaouadi, H. Mhiri and G. Lepalec

***Abstract*— A numerical analysis of the turbulent and strongly swirling flow field of a uniflow cyclone and its performances is described. The gas flow is obtained by the use of the Reynolds stress model while, the determination of particle flow is ensured by the use of a Stochastic Lagrangian model. The flow features are examined in terms of tangential, axial and radial velocities, pressure drop and separation efficiency. The validity of the proposed approach is verified in terms of cyclone separator performances by the good agreement between the measured and the predicted results.**

***Index Term* — CFD, cyclone, uniflow, separation.**

I. Introduction

The separation processes are crucial to industry like petrochemical, chemical, mining, pharmaceutical and agro-alimentary processing. Generally, it is a physical separation of two phases (Gas/Liquid, Gas/Solid, Liquid/Solid), i.e. a continuous phase (carrying phase), and a dispersed phase (particles). Cyclones are the most widely used separator devices which are based on the particle centrifugal force created by vortex in the cyclone. A difference is generally made between two types of cyclone: uniflow and reverse flow. They differ essentially in the configuration of their outlet pipe with respect to the entrance. They can be also classified according to the fluid carrying the dispersed phase: cyclones gas-solid and the hydrocyclones (liquid-solids). In the past, extensive research has been focused on traditional reverse flow cyclones while little was on the uniflow cyclones. Experimental and theoretical researches were carried out to improve their separation performances. An experimental study was made by Gauthier et al. [1] on a uniflow cyclone. They noted that the collection efficiency depends on the length of the separation zone whose optimal value varies between 1 and 10.5 times the diameter of the cyclone. It increases with the inlet velocity of the cyclone. Recently, Zhongchao Tan [2] studied a uniflow cyclone with a tangential inlet and developed an analytical model to predict the fractional efficiency which is a function of particles Stokes number and the geometry of the cyclone body. This model was validated by six sets of experiments carried out under various conditions. Generally, improvement of cyclone performances requires complex structure and additional exploitation cost. That's why best comprehension of the flow field in the cyclone is necessary in order to make other design improvements. Computational fluid dynamics (CFD) can provide information about the flow field and particle trajectory in this type of separation apparatus. This paper reports our recent effort in modeling the anisotropic turbulence flow in a uniflow cyclone. We used RSM and Stochastic Lagrangian model available in Fluent to describe the fluid flow field in this type of gas-solid cyclone and also to study its performances. Pressure drops and separation efficiency are compared with experimental results available in the literature.

Manuscript received Mars 23, 2009
I. Mokni is with UTTPI, National Engineering School, Monastir, TUNISIA (phone: +21698606843; e-mail: mokni_ines@yahoo.fr)
H. Dhaouadi is with Faculté des Sciences, Département de Chimie, Monastir, TUNISIA (e-mail: hatem.dhaouadi@fsm.rnu.tn)
H. Mhiri is with the National Engineering School, Monastir, TUNISIA (e-mail: hatem.mhiri@enim.rnu.tn)
G. Lepalec is with IUSTI, UMR CNRS 6595, 5 rue Enrico Fermi, Technopole de Château-Gombert, 1303 Marseille, FRANCE (e-mail : georges.lepalec@univmed.fr)

II. Cyclone description

The studied device, invented by Gauthier [3], is a uniflow cyclone which is used for the ultra-rapid fluidized (URF) reactor. The cyclone is characterized by an asymmetric configuration because of an only one tangential inlet of fluid charged with particles. Fig. 1 shows a general view of the studied cyclone and its dimensions notations.

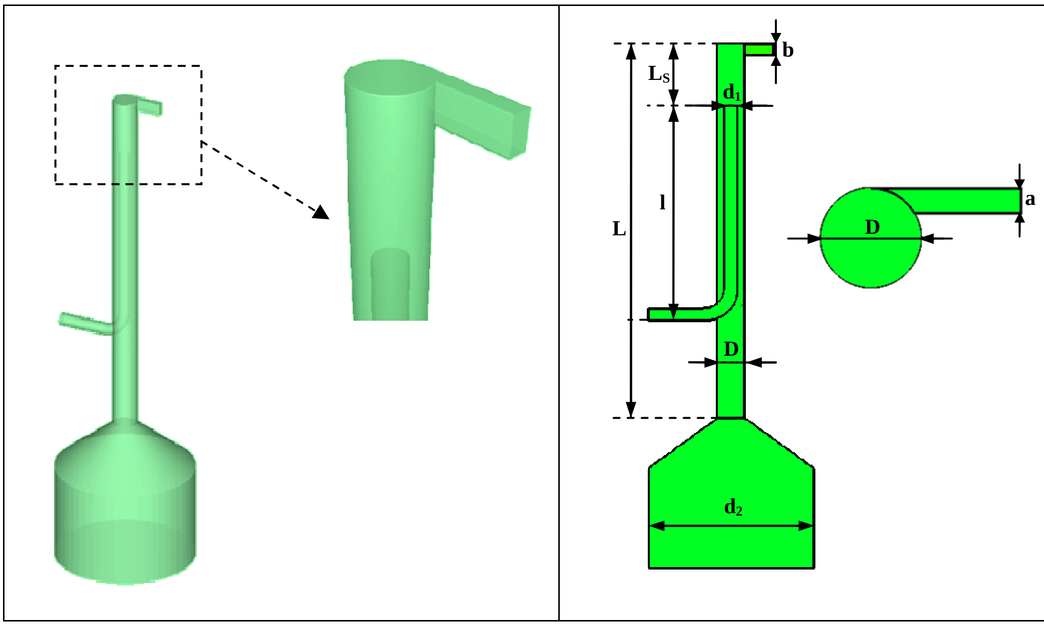
The particles granulometric distribution can be suitably described by the Rosin-Rammler equation:

$$R(d) = \mathbf{exp}\left[-\left(d/\bar{d}\right)^n\right] \tag{1}$$

R(d) is the mass fraction of particles having a diameter greater than d. n and $\bar{d}$ are respectively the spread parameter and the mean diameter of the distribution. To determine theses parameters, it is necessary to fit the particle size data to the Rosin-Rammler exponential equation. In our case, the mean diameter $\bar{d}$ is equal to 43µm and the distribution parameter n is equal to 2.57. The particle density is 2500 kg/m3.

III. Mesh and boundary conditions

The computational domain, constituted of 219443 CFD cells, was carried out by using the preprocessor "Gambit". Everywhere in the computational domain and when it is possible, the hexahedron cells were more preferable than tetrahedral cells, to have a grid which adapts well to the geometry considered. Fig. 2 shows an overall view of the grid generated and zooms on inlet and outlets of the fluid and particles. It is delicate to choose a grid which gives results that can predict the studied flow field.



D=5	
d_1	2,5
d_2	30
a	1,3
b	2,5
L	75
L_S	12,5
l	42,5

All dimensions are given in cm

Fig. 1: Dimensions of the studied uniflow cyclone.

Thus, sufficiently fine grids give good precision but excessively fine grids can lead to high computational times. Grid refinement tests are conducted in order to make sure that the solution is not grid dependent. Those tests allow also concluding that relatively fine grids with an adaptation on walls can suitably describe the flow inside the cyclone. A “velocity inlet” boundary condition was used at the cyclone inlet, and the inlet velocity was 25m/s. The pressure at the gas exit is 1atm, thus the boundary condition used was the “pressure outlet” condition. No slip boundary condition was used in wall boundary, and near-wall treatment was standard wall function. Particles are injected from the inlet surface.

IV. MODEL DESCRIPTION

For the rotating turbulent flow in cyclone, the key to the success of CFD lies with the accurate description of the turbulent behavior of the flow. A number of turbulence models are available. The commonly used in cyclone simulation are: k-ε model, RSM. The present work is based on the RSM which can describe anisotropic turbulence.

The RSM has been proven to be an appropriate turbulence model for cyclone flow, although it is computationally more expensive. The governing equations for an incompressible fluid can thus be written as:

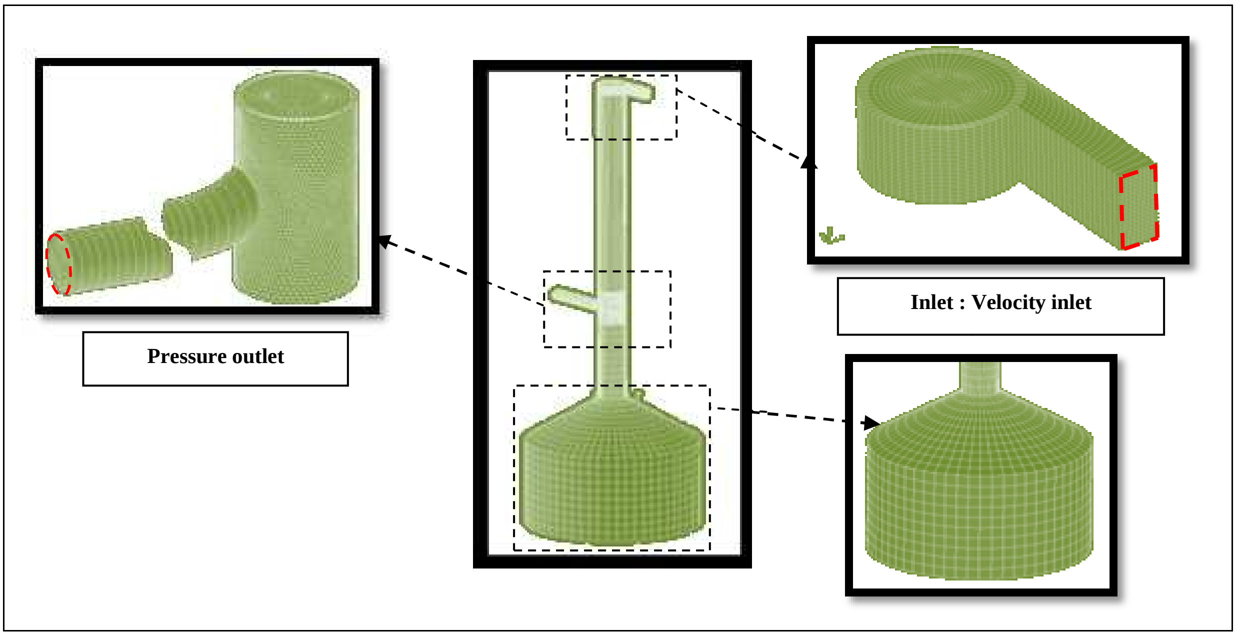


Fig. 2: CFD grid of the studied cyclone and boundary conditions.

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_i}(\rho u_i)=0 \qquad (1)$$

$$\frac{\partial}{\partial t}(\rho u_i)+\frac{\partial}{\partial x_j}(\rho u_i u_j)=-\frac{\partial P}{\partial x_i}(\rho u_i)+ \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right]+\frac{\partial}{\partial x_j}\left(-\rho\overline{u_i' u_j'}\right) \qquad (2)$$

Where the velocity components are decomposed into the mean $\overline{u_i}$ and fluctuating u_i' velocities which are related as given by:

$$u_i=\overline{u_i}+u_i' \qquad (3)$$

The Reynolds stress term $R_{ij}=\rho\overline{u_i' u_j'}$ includes the turbulence closure, which must be modeled to close (2).

A. Turbulence model RSM

Transport equations of the Reynolds stresses terms ($R_{ij}=\rho\overline{u_i' u_j'}$), are written as [4]:

$$\frac{\partial R_{ij}}{\partial t}+C_{ij}=P_{ij}+D_{ij}-\varepsilon_{ij}+\phi_{ij}+G_{ij} \qquad (4)$$

Where $C_{ij}, P_{ij}, D_{ij}, \varepsilon_{ij}, \phi_{ij} et G_{ij}$ are respectively: the convective transport term, the stress production term, the diffusion term, the dissipation term, the pressure strain term and the buoyancy production term [4]. The RSM model requires the following empirical constants: C_μ= 0.09, $C_{\varepsilon 1}$ =1.44, $C_{\varepsilon 2}$ =1.92, σ_ε =1, σ_k =0.82.

B. Discrete Phase model DPM

The motion of a particle is described by the stochastic Lagrangian multiphase flow model. Its trajectory is obtained by integrating the force balance on particle. There are many forces that act on a particle in cyclone as centrifugal force, drag force and gravitational force. Thus, the particle motion equation can be written in the following form [4]:

$$\frac{d\vec{u}_p}{dt}=F_D\left(\vec{u}_f-\vec{u}_p\right)+\vec{g}\left(\frac{\rho_P-\rho_f}{\rho_P}\right)+\vec{F} \qquad (5)$$

F is a source term which expresses the presence of an additional force. $F_D(\vec{u}_f-\vec{u}_p)$ is the drag force per unit particle mass and F_D is given by [4]:

$$F_D=\frac{18\mu_f}{\rho_P {d_P}^2}\frac{C_D \operatorname{Re}_P}{24} \qquad (6)$$

Where: $$\operatorname{Re}_p=\rho_f(u_f-u_p)d_p/\mu_f \qquad (7)$$

C_D, ρ_f, ρ_p, μ_f, dp are respectively the drag coefficient, the density of the fluid, the density of the particle, the molecular viscosity of the fluid and the particle diameter.

V. Results and Discussions

In the following, the fluid flow field in a uniflow cyclone and the behavior of the dispersed phase (solid particles) are treated in order to examine its performances and validate the proposed model by comparing results given by numerical simulation with experimental results available in the literature [1]. The analyzed results are velocities profiles, the pressure drop and the separation efficiency.

A. Fluid flow field

To explore the inner flow in the cyclone, tangential, axial and radial velocities are represented in different sections. Fig.3 shows the distribution of the calculated tangential velocity, which is the most dominant component of the flow in the cyclone. The figure describes well the combined vortex (Rankine vortex) composed of a free vortex (outer vortex) and a forced vortex (inner vortex). Indeed, the radial evolution of the tangential velocity (Fig.4) has a shape of “W” as well in the separation chamber of the cyclone as in the evacuation pipe and the annular zone (the part limited by walls of the cyclone body and walls of the evacuation pipe).

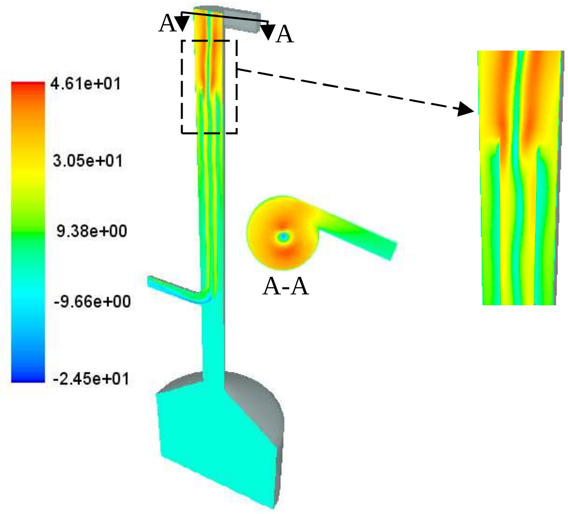


Fig. 3: Contour of tangential velocity distribution.

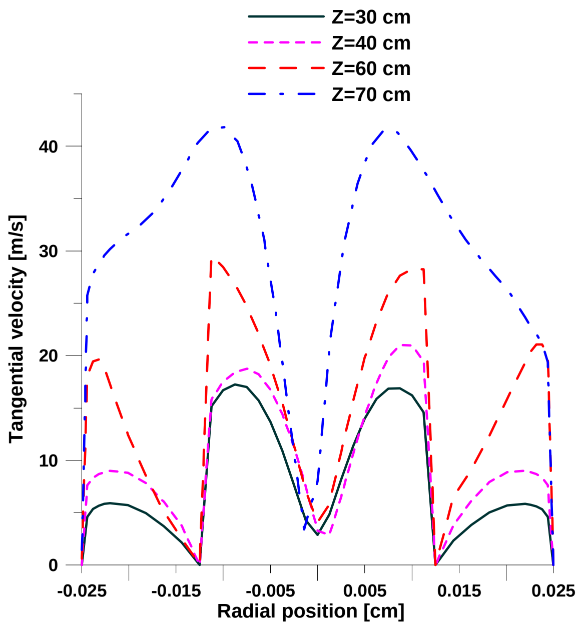


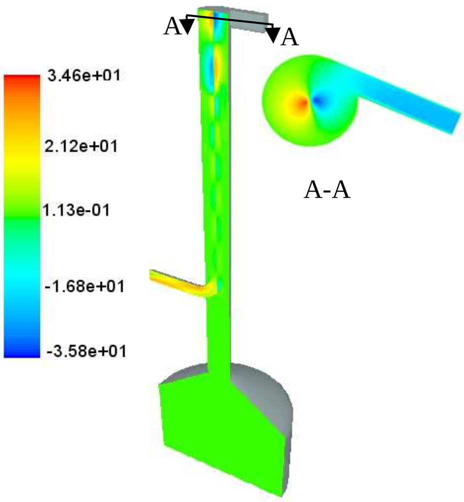
Fig. 4: Tangential velocity profiles.

This shape confirms the existence of a double vortex in these zones. Figure 4 also shows that the maximum of the tangential velocity is located at a ray almost equal to the ray of the outlet pipe and its value decreases throughout the flow.

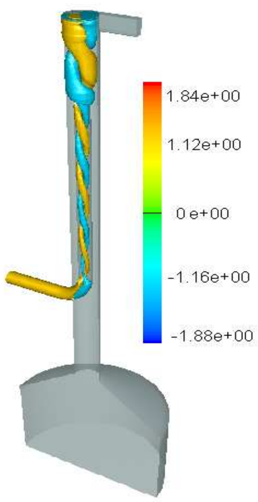
On Fig.5.a, radial velocity distribution is represented. This figure shows that the flow forms a dipole near the axis of the cyclone where radial velocity has the same absolute values but with opposite signs (The value of one side is positive and the other is negative). This explains the shape of the forced vortex which is in fact two helical cylinders (Fig.5.b). Fig.5.a (A-A) shows that the fluid, entering through the inlet pipe and moving towards the center of the cyclone, has a negative radial velocity that tends towards zero while approaching the center of the cyclone. Afterwards, it becomes positive due to the effect of the centrifugal force. The distribution of the axial velocity is represented on Fig.6 for various sections. Fig.6.a shows that there is, in the annular zone, a downward flow near cyclone walls compensated by an upward flow near the evacuation pipe from where it will be poured out. In fact the downward flow allows carrying away large particles to the low part of the cyclone (dustbin).

The locus of zero vertical velocity “LZVV” (locus of the dividing line between the upward flow and the downward flow), represented by a black line on the Fig.6.b, is asymmetrical and slightly larger than the evacuation pipe.

It is shown also on the same figure that the forced vortex has not the same contour at all vertical sections. This is explained by the fact that, in the central part of the cyclone, the vortex core is in precession.

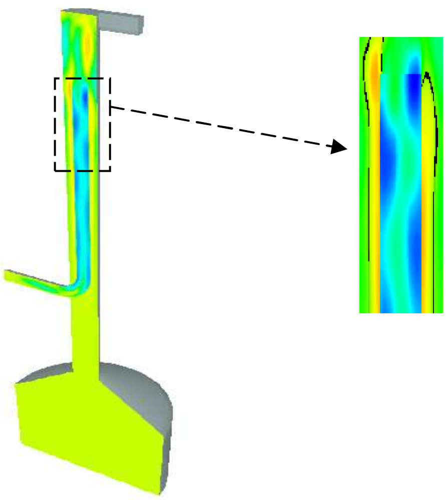


(a)
Contour of radial velocity

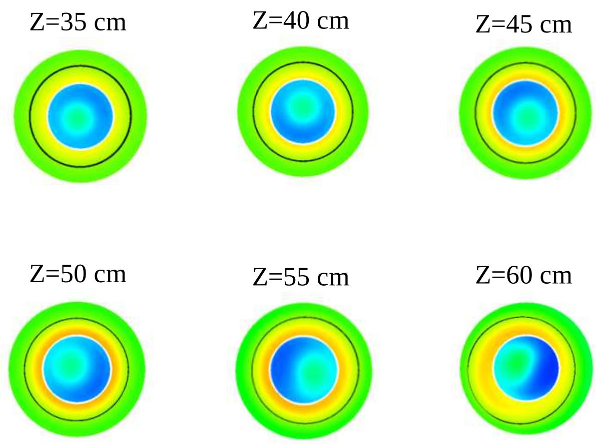


(b)
Forced vortex

Fig. 5: Radial velocity distribution



(a)



(b)

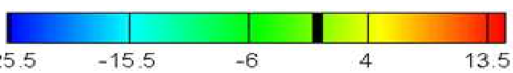


Fig. 6: Contour of axial velocity.

B. Cyclone performances

Pressure drop and separation efficiency are the principal parameters which characterize the performances of a cyclone separator. Fig.7 represents the distribution of the relative pressure. It is shown that the pressure is radially decreasing from wall to the center of the cyclone where it reaches a zone of low pressure that is already quoted in the previous paragraph and is called the precessing vortex core. It is spread on the entire length of the separation chamber and the evacuation pipe (zone in blue). The limits of the depression zone (lines in black) define the interface between the two free and forced vortexes. Integrating the pressure on the inlet and exit faces allows estimating the pressure drop in the considered cyclone.

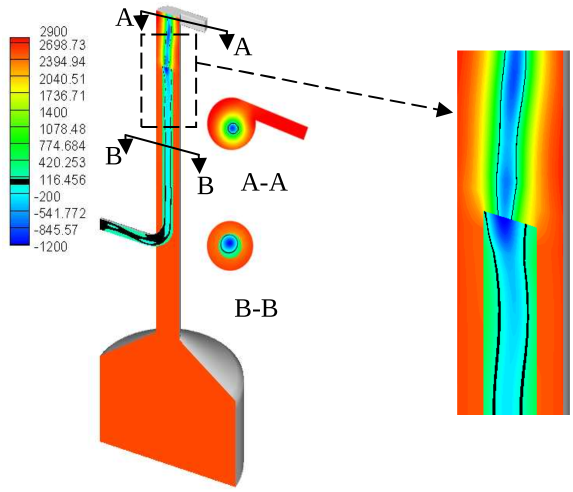


Fig. 7: Contour of relative velocity.

To calculate the separation efficiency, it's necessary to study motion of solid particles in the cyclone. On table I these parameters are listed and compared with the experimental results available in the literature [1]. The comparison shows that the developed model underestimates the separation efficiency. Thus, it is required to use other multiphase models such as the Euler-Euler model, which takes account of the interactions between particles when the mixture is highly loaded with particles. In general, numerical results obtained are in rather good agreement with the experimental results.

Table I: The experimental and numerical cyclone performances.

	Pressure drop [Pa]	Separation efficiency [%]
Exp. Gauthier	2800	99.98
CFD	2849	98.5

VI. Conclusion

A CFD simulation, based on the turbulence model "RSM" and the stochastic multiphase model "DPM", was elaborated in this paper. The CFD results, given by the developed model and compared with experimental available results in the literature, show that this model underestimates the separation efficiency of the studied cyclone although, it describes well the turbulent and strongly swirling flow of the continuous phase. However, it is essential to use another multiphase 74107model which could better estimate the separation efficiency of the studied uniflow cyclone which is very charged with particles (catalyst).

References

[1] T.A. Gauthier, C.L. Briens, M.A. Bergougnou, P. Galtier, , Powder Technology 62 (1990) 217–225.
[2] An analytical model for the fractional efficiency of a uniflow cyclone with a tangential inlet, Zhongchao Tan, Powder Technology 183 (2008) 147–151.
[3] "Séparateur extracteur à co-courant" Patent EP92403149, Gauthier Thierry (inventor).
[4] Fluent 6.2, "Documentation", User's Guide Fluent. Inc. (2001).