STUDY OF DIFFERENT APPROACHES FOR MODELING CYCLONES USING CFD

Daniel de Brito Dias, School of Chemical Engineering, University of Campinas, Campinas, SP, Brazil
Milton Mori, School of Chemical Engineering, University of Campinas, Campinas, SP, Brazil
Waldir Pedro Martignoni, Cenpes/Petrobras, Rio de Janeiro, RJ, Brazil

Abstract

Due to their relative robustness and low cost, cyclones are widely used in industries as gas-solid and liquid-solid separators. Their performance is usually measured by two parameters: pressure drop and collection efficiency. While this equipment is simple in concept, studies have shown that the confined vorticial flow inherent to cyclones is in fact quite complex. Because of this, computational fluid dynamics (CFD) has been employed to model flows and solve them numerically, and has shown relative success in dealing with phenomena such as vortex breakdown, reversal flow, and high turbulence intensity.

Cyclone simulations are usually expensive in terms of computer processing, mainly due to the fact that an anisotropic turbulence model must be used. Several studies have addressed that issue, and Reynolds Stress Model (RSM) has been considered by many researchers as being the most suitable model. In order to reduce computational times while still acquiring reliable results, other aspects of the simulation must be studied, such as the modeling approach to the particulate phase, which is the subject of this work.

While the Eulerian approach is generally applied for the gas, different schemes can be considered for the particulate phase. In this work, two different methods are explored: the classic Eulerian-Eulerian (E-E), where a mean diameter is considered to calculate the entirety of the flow field of the particulate phase; and Eulerian-(Eulerian)$_n$ (E-E$_n$), where particles with different diameters are included in the same simulation, each as a distinct phase. The fractional collection efficiency curve is obtained for both methods. An analysis is made regarding accuracy and computational costs.

The simulations make use of ANSYS CFX, a commercial software package, running in parallel mode in a high performance computing cluster. The fractional and overall collection efficiencies comparison with the experimental data showed that E-E$_n$ achieved more accurate results.

KEYWORDS: CFD, cyclone, gas-solid

Introduction

Cyclones are the most commonly used gas-solid separation equipments in industry, mainly due to their relatively low manufacturing and operational costs. Although the operation principles involved are simple, the fluid dynamics are not: the dust-laden gas enters a cylindrical chamber tangentially and spirals downwards. The particles, being denser, concentrate on the wall due to their inertia and are collected at the bottom. The clean gas reverses its flow and exits through a central opening known as a vortex finder.

The result is a complex confined double vortex which is part of the reason why cyclone design remained, for many years, restricted to empirical or semi-empirical methods which are still in use. Aiming to maximize collection efficiency while minimizing pressure drop, design models in which the geometrical parameters of the equipment are defined as a function of its diameter were created. Also, several algebraic models for estimating velocities, pressure drop and collection efficiency are available and have been compiled from literature in reviews [1, 2].

Recent advances in computer hardware, numerical codes and parallel processing have made possible the use of computational fluid dynamics (CFD), a technique that proposes the solution of the equations of conservation of mass, momentum and energy via numerical methods such as finite volumes. In the field of cyclones it has proven the complexity of the phenomena involved with the need for unsteady simulations, higher order discretization schemes and anisotropic turbulence models such as the Reynolds Stress Model (RSM) [3,4]. Even more difficulties arise when taking multiphase flow into account. This can be done initially considering two different approaches: Eulerian-Eulerian, in which the phases are considered continuous and interpenetrating and Eulerian-Lagrangian, where the solid phase is considered discrete and each individual particle is tracked along the continuous phase. Each method has its particular pros and cons, and both have been applied to cyclone modeling with relative success, which can be exemplified by references [5] and [6].

One particular disadvantage of the traditional Eulerian-Eulerian approach is that the solid phase is given an average diameter, so that any information about particle size distribution is lost. Such information is very important in cyclone analysis because it provides design engineers with more complete information on collection efficiency in the form of the fractional collection efficiency curve. This work analyzes and compares two different methods for obtaining the mentioned curve while still retaining the E-E scheme. The first method (E-E) consists on performing a fixed number of simulations, with one particulate phase per simulation, varying particle diameter. Each time, it is considered that the single particulate phase represents the entire solid load fed to the cyclone, obtaining one point in the fractional collection efficiency curve per simulation. In the second method (referred to as $E\text{-}E_n$) one single simulation considering multiple solid phases of different diameters weighted according to the particle distribution is performed, thus directly obtaining an approximation of the fractional collection efficiency curve. A comparison on the accuracy and applicability of each method is made, and the results are validated by experimental data by Zhao [7].

The authors hope that this discussion adds to the cyclone CFD literature, and helps establish it as a tool not only for analysis but proposal of new designs, as it has been in recent years [8, 9]. The simulations make use of ANSYS CFX 11, a commercial software package, running in parallel mode in a high performance computing cluster.

Mathematical Modeling

The conservation equations of mass and momentum solved in this work for phase α (out of n total phases) are described by Equations 1 and 2, respectively.

$$\frac{\partial \rho_\alpha}{\partial t} + \nabla(\xi_\alpha \rho_\alpha v_\alpha) = 0 \tag{1}$$

$$\frac{\partial(\xi_\alpha \rho_\alpha v_\alpha)}{\partial t} + \nabla(\xi_\alpha \rho_\alpha v_\alpha v_\alpha) = \nabla\{\xi_\alpha \mu_\alpha [\nabla v_\alpha + (\nabla v_\alpha)^T]\} - \xi_\alpha \nabla p + \xi_\alpha \rho_\alpha g + \sum_{i\neq\alpha}^{n} \beta(v_i - v_\alpha) \tag{2}$$

Where ξ is the volumetric fraction, ρ is the density, v the velocity, p the pressure, g the acceleration due to gravity, μ the viscosity and β is the inter-phase momentum transfer coefficient.

As for closure equations, the continuity of volumetric fractions was guaranteed by Equation 3, and the inter-phase drag was calculated by Equations 4 and 5. Due to dilute flow conditions, solid pressure force and particle-particle drag were neglected. Solid phases were considered to be inviscid.

$$\sum_{i}^{n} \xi_i = 1 \tag{3}$$

$$\beta = \frac{3}{4} Cd \frac{|v_g - v_p| \xi_p \rho_\alpha}{d_p} \tag{4}$$

$$Cd = max\left[\frac{24}{Re}(1 + 0.15Re^{0.687}), 0.44\right] \tag{5}$$

Turbulence Model

For the complex turbulent flow inherent to cyclones, a robust model such as RSM must be considered. First, the Reynolds average, which splits a variable into a time-average and a fluctuating part ($i.e.\ v = \overline{v} + v'$) is applied to the conservation equations. Transport equations for the individual Reynolds stress ($\rho\overline{v'v'}$) components obtained (Equations 6 and 7) and for the turbulence dissipation rate ε (Equation 8) are then solved, rather than adhering to the eddy viscosity hypothesis.

$$\frac{\partial\left(\rho\overline{v_i'v_i'}\right)}{\partial t} + \frac{\partial\left(U_k\rho\overline{v_i'v_i'}\right)}{\partial x_k} = P_{ij} + \phi_{ij} + \frac{\partial}{\partial x_k}\left[\left(\mu + \frac{2}{3}c_s\rho\frac{k^2}{\varepsilon}\right)\frac{\partial\left(\overline{v_i'v_i'}\right)}{\partial x_k}\right] - \frac{2}{3}\delta_{ij}\rho\varepsilon \tag{6}$$

$$P = -\rho\left[\overline{v'v'}(\nabla v)^T + (\nabla v)\overline{v'v'}\right] \tag{7}$$

Where P is the exact production term and ϕ is the pressure-strain correlation (Equations 8 through 12).

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho U_k\varepsilon)}{\partial x_k} = \frac{\varepsilon}{k}(c_{\varepsilon1}P - c_{\varepsilon2}\rho\varepsilon) + \frac{\partial}{\partial x_k}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_k}\right] \tag{8}$$

$$\phi_{ij} + \phi_{ij1} + \phi_{ij2} \tag{9}$$

$$\phi_{ij1} = -\rho\varepsilon\left[C_{s1}a + C_{s2}\left(aa - \frac{1}{3}a{\cdot}a\delta\right)\right] \tag{10}$$

$$\phi_{ij2} = -C_{r1}Pa + C_{r2}\rho kS - C_{r3}\rho kS\sqrt{a{\cdot}a} + C_{r4}\rho k(aS^T + Sa^T - \frac{2}{3}a{\cdot}S\delta) + C_{r5}\rho k(aW^T + Wa^T) \tag{11}$$

$$\begin{cases} a = \frac{\overline{v\prime v\prime}}{k} - \frac{2}{3}\delta \\ S = \frac{1}{2}[\nabla v + (\nabla v^T)] \\ W = \frac{1}{2}[\nabla v - (\nabla v^T)] \end{cases} \tag{12}$$

Where a is the anisotropy tensor, S is the strain rate and W is the vorticity. From the many varieties of RSM available, Speziale-Sarkar-Gatski (SSG) was chosen, as its quadratic relation for the pressure-strain correlation is recommended for swirling flows [10]. The model constants are as follows:

$$c_s = 0.22; c_{\varepsilon 1} = 1.45; c_{\varepsilon 2} = 1.83 \tag{13}$$
$$C_{s1} = 1.7; C_{s2} = -1.05; C_{r1} = 0.9; C_{r2} = 0.8; C_{r3} = 0.65; C_{r4} = 0.625; C_{r5} = 0.2$$

The RSM approach makes the overall model far more computationally expensive but also theoretically more suited to predict complex anisotropic flows.

Numerical Methods

The aforementioned equations were solved using commercial CFD code ANSYS CFX 11, by means of the finite volume method. A numerical algorithm based on Rhie-Chow was applied to the pressure-velocity coupling and the higher upwind high order interpolation scheme was considered. A very detailed description of these methods can be found in [11]. A time step of 0.0001 seconds was used to simulate, for each case, five seconds of real time. The error criterion was 0.0001 for the RMS residue for all equations.

Case Description

The simulation attempted to numerically reproduce the experimental work of Zhao [7] (specifically the tangential inlet cyclone data), which has been previously studied [12]. Equipment dimensions are summarized in Table 1. The 3-D geometry and its corresponding numerical grid (consisting of approximately 350,000 hexahedrical elements) were built using meshing code ICEM CFD, and are displayed in Figure 1.

The fluid considered in the simulations was air at 25°C. The particulate phase was a talcum powder of density 2700 kg/m^3 obeying a log-normal size distribution with a mass-mean diameter of 5.97μm and geometric deviation of 2.08. The concentration of the solids was of 5.0 g/m^3. Four different inlet velocities were simulated: 12, 16, 20 and 24 m/s. The air exit was set to an atmospheric pressure opening. Wall conditions were set to no slip for the gas and free slip for the talcum, and a dust hopper was added at the bottom, with the same wall boundary conditions applied to it.

Three diameters were taken into account, so that three repetitions for the E-E simulation and one E-E_3 simulation were performed. Considered particle sizes were the mean diameter (5.97 μm), the mean diameter divided by the geometric deviation (corresponding to 2.87 μm) and the mean diameter multiplied by the geometric deviation (corresponding to 12.42 μm). In the E-E_3 simulation they correspond to 68, 16 and 16 percent in volume of the solid load respectively.

Table 1. Geometrical Parameters of the Studied Cyclone

	D	**De**	**h**	**H**	**B**	**S**	**a**	**B**
Size [mm]	300	150	450	1200	112.5	150	150	60

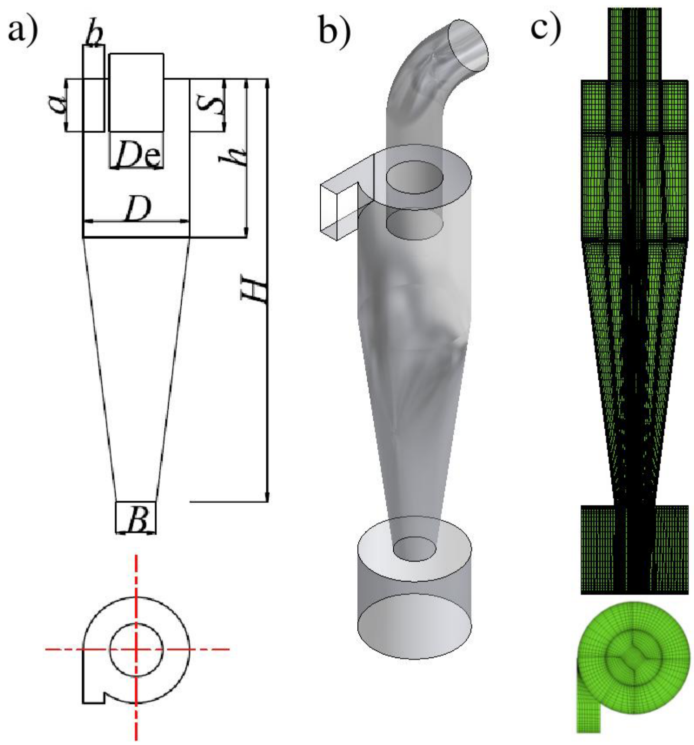

Figure 1. Studied cyclone and its a) Geometrical parameters b) CAD Geometry c) Hexa mesh

Results and Discussion

The pressure and velocity profiles for all simulations, exemplified in axial cut planes in Figure 2, came to good agreement with what is expected of cyclone flow.

Along the axis, in the central zone of the equipment, the formation of an ascending low pressure zone can be observed, contrasting with the downward high pressure zone closer to the walls. The negative and positive tangential velocities, peaking at almost two times the inlet velocity, form a quasi-symmetrical plane along the central axis and indicate the swirling motion of the double vortex: the left (negative) section represents fluid *entering* the plane and the right (positive) section *leaving* the plane.

This adequate description of the fluid dynamics is validated by the good agreement with experimental data, showed in Table 2 and Figures 3 and 4. Overall and fractional efficiencies (η) are calculated as functions of the mass flows in the two boundaries of the system and analyzed.

$$\eta = \frac{\dot{m}_{in} - \dot{m}_{out}}{\dot{m}_{in}} \quad (14)$$

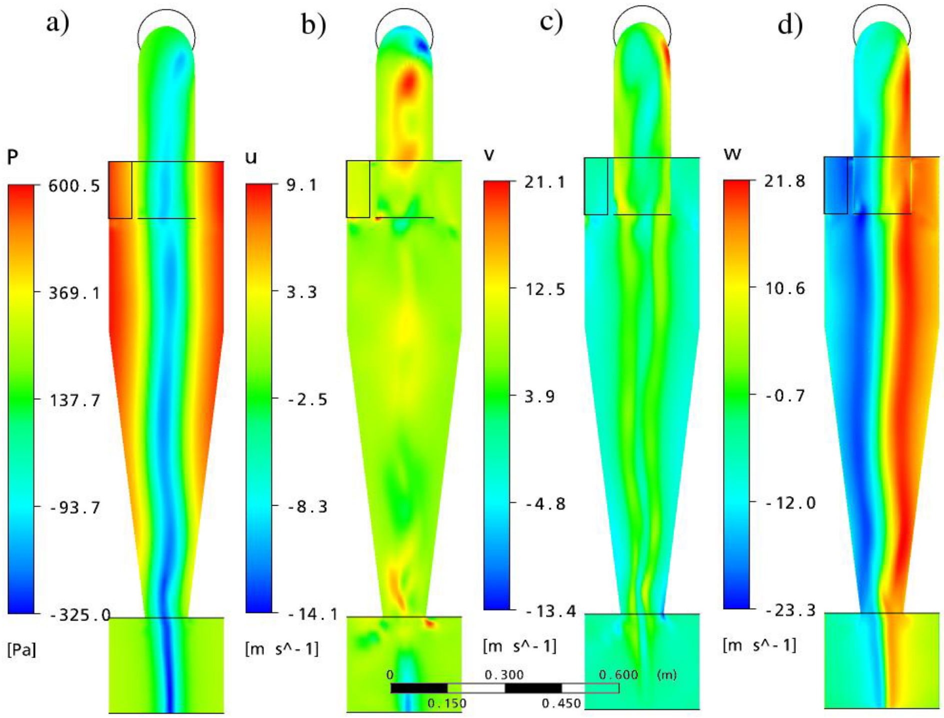

Figure 2. Profiles for a) Pressure b) Radial Velocity c) Axial Velocity and d) Tangential Velocity for the E-E_3 case with an inlet velocity of 12 m/s.

Table 2. Comparison of fractional and overall collection efficiencies

v_{in} = 12 m/s			
	Zhao [7] [%]	**E-E [%]**	**E-E_3 [%]**
2.87 µm	70.2	57.6	63.4
5.97 µm	98.0	99.3	98.8
12.42 µm	99.7	99.8	99.1
Overall	91.0	92.8	93.3

v_{in} = 16 m/s			
	Zhao [7] [%]	**E-E [%]**	**E-E_3 [%]**
2.87 µm	82.8	81.9	80.4
5.97 µm	98.8	99.5	99.2
12.42 µm	99.8	99.9	99.3
Overall	94.7	96.8	96.2

v_{in} = 20 m/s			
	Zhao [7] [%]	**E-E [%]**	**E-E_3 [%]**
2.87 µm	88.1	92.4	87.1
5.97 µm	99.0	99.6	99.4
12.42 µm	99.9	99.9	99.5
Overall	95.8	98.5	97.4

v_{in} = 24 m/s			
	Zhao [7] [%]	**E-E [%]**	**E-E_3 [%]**
2.87 µm	92.7	96.3	90.1
5.97 µm	99.1	99.7	99.5
12.42 µm	99.9	99.9	99.6
Overall	95.7	99.2	98.1

At a first glance, it can be seen that both methods predict the separation reasonably well. Experimentally the increase of velocity also sees an increase in collection efficiency, and this holds true for the CFD runs. Nonetheless, the E-E_3 approach proved to have a more acceptable accuracy in the numerical results. This is most likely due to the fact that the gas-solid momentum transfer in this case is closer to experimental data. The particles with different diameters are included in the inter-phase drag equation and consequently three number ranges are taken into account.

The overall collection efficiency, showed in Figure 4, agrees as well as the numerical results of fractional collection efficiency, and gives a better visualization on how the E-E_3 method was superior in representing the separation phenomena.

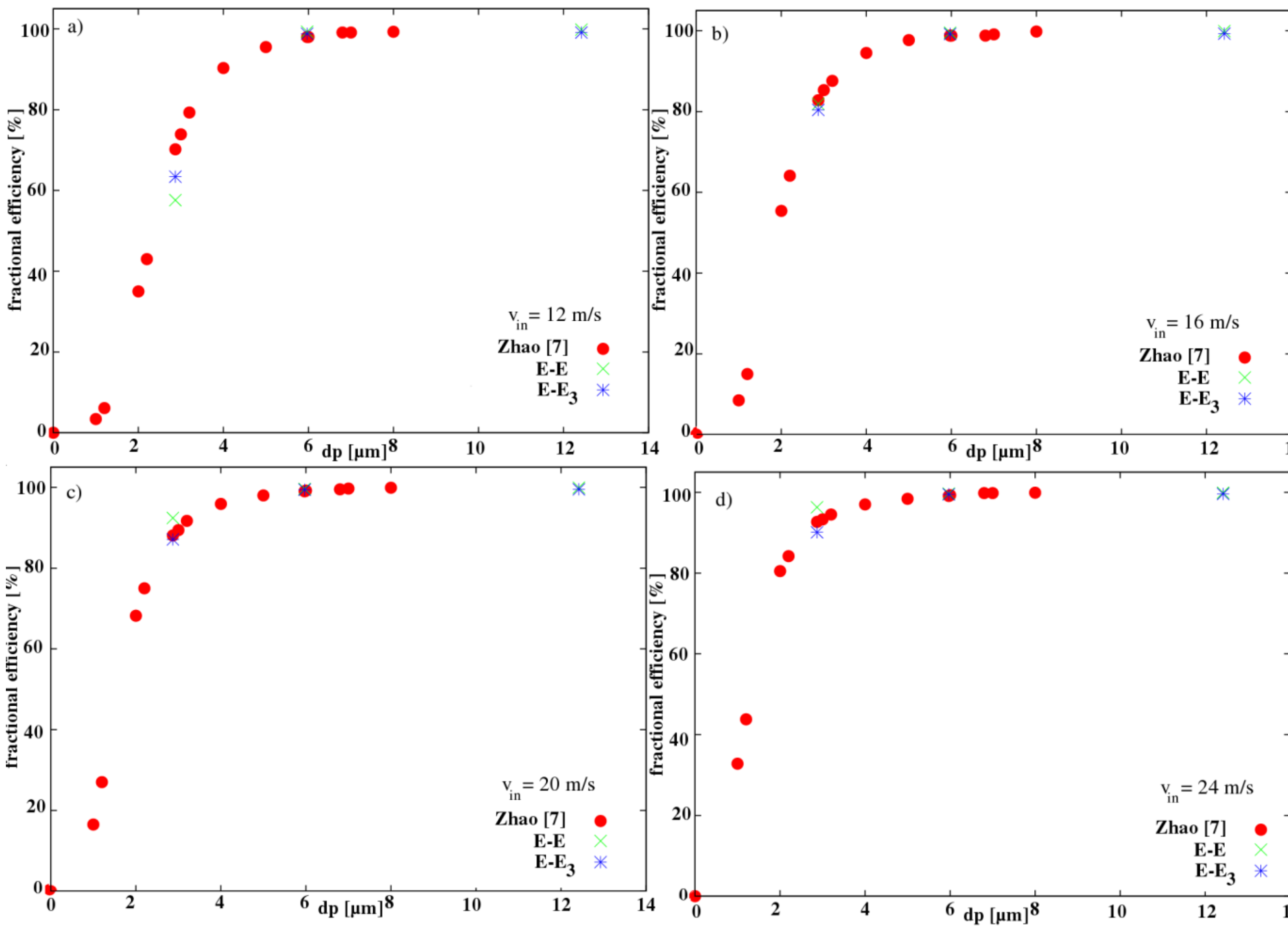

Figure 3. Fractional collection efficiency curves compared for both methods with inlet velocities of a) 12 m/s; b) 16 m/s; c) 20 m/s; d) 24 m/s.

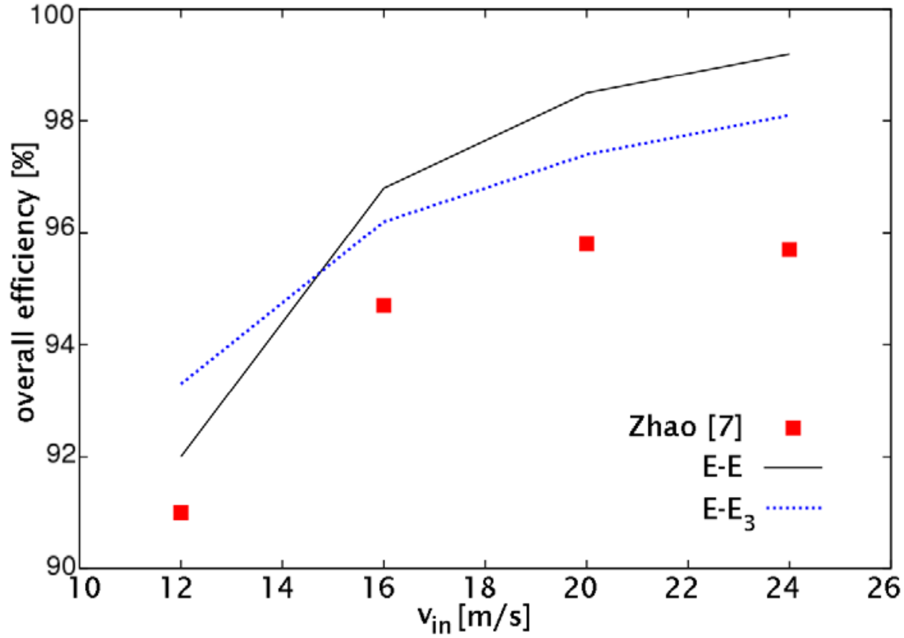

Figure 4. Overall collection efficiency comparison.

As a final analysis, the processing time and scalability of each method was tested. The average time taken per iteration was measured varying the number of parallel processes used for the run. Since three E-E simulations gave us the equivalent amount of data of one E-E_3 simulation, its processing time was multiplied by three. The simultaneous multiple solid phase approach performed far better in this analysis, with not only smaller simulation times but also better scalability. This happens because of two facts. First, it is only solving the gas phase once as opposed to three times and, second, more sets of differential equations are being solved simultaneously and consequently the solver can make better use of the additional parallel processes.

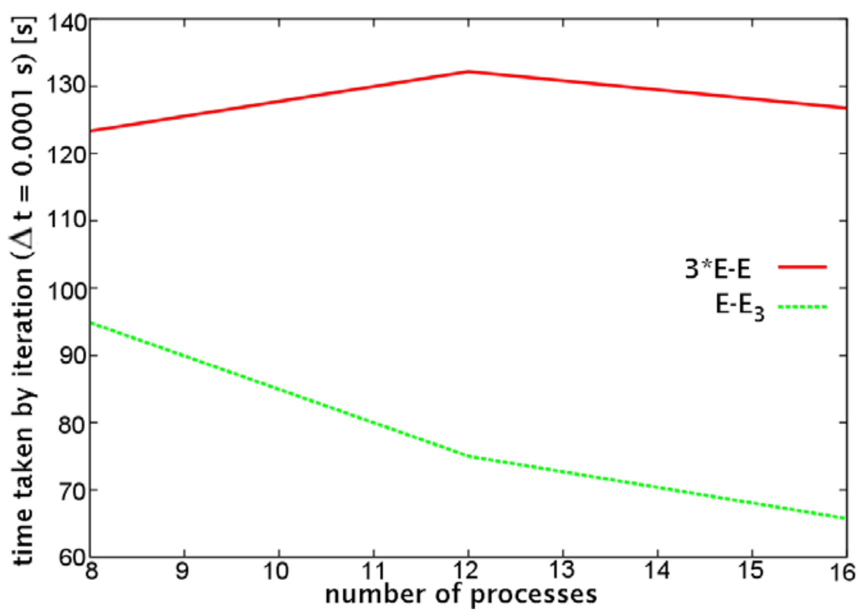

Figure 5. Processing time per iteration taken by each method *versus* number of processes used

Conclusions

The collection efficiency and flow characteristics of a gas-solid cyclone were obtained by numerical analysis. Two different approaches of modeling the solids have been studied in this work, aiming to obtain the fractional collection efficiency curve. These methods were referred to as E-E and $E\text{-}E_n$. The first consists in simulating several times a regular Eulerian-Eulerian flow with a different particle diameter each time. The second is achieved by performing one single simulation simultaneously involving multiple solid phases, with different diameters.

While both schemes correctly reproduced the complex fluid dynamics inside cyclones (which was confirmed by validation with experimental data [7]), $E\text{-}E_n$ proved itself not only more accurate, but also faster and more scalable, acting as a better alternative for obtaining the fractional collection efficiency curve for the case studied in this work.

A suggestion for future studies is the application of these methods to a case with a higher concentration of solids.

Acknowledgments

The authors would like to thank PETROBRAS for their financial support.

Nomenclature

Cd	drag coefficient	-
$c_s, c_{\varepsilon 1}, c_{\varepsilon 2}, C_{s1}, C_{s2}, C_{r1}, C_{r2}, C_{r3}, C_{r4}, C_{r5}$	RSM constants	-
dp	particle diameter	m
g	gravitational acceleration	$m \cdot s^{-2}$
k	turbulence kinetic energy	$m^{-2}s^{-2}$
$\dot{m}$	mass flow	$kg \cdot s^{-1}$
P	exact production	$kg \cdot m^{-1} \cdot s^{-3}$
p	pressure	Pa
Re	Reynolds number	-
t	time	s
v	velocity	$m \cdot s^{-1}$
x	dimension	m

Greek Letters

β	inter-phase momentum transfer coefficient	$kg \cdot m^{-2} \cdot s^{-1}$
δ	Dirac delta	-
ε	turbulent dissipation rate	$m^{-2} \cdot s^{-3}$
η	collection efficiency	-
μ	viscosity	$kg \cdot m^{-1} \cdot s^{-1}$
ξ	volumetric fraction	-
ρ	density	$kg \cdot m^{-3}$
ϕ	pressure-strain	$kg \cdot m^{-1} \cdot s^{-3}$

Subscripts

i,j,k	indexes
in	inlet
n	total number of phases
out	outlet
t	turbulent
α	phase indicator

References

1. Hoffmann, A. C. and Stein, L. E. (2002), "Gas Cyclones and Swirl Tubes: Principles, Design and Operation", ed. Berlin, Springer.
2. Cortés, C. and Gil, A. (2007), "Modeling the Gas and Particle Flow Inside Cyclone Separators", *Progress in Energy and Combustion Science,* 33, pp. 409-452.
3. Boysan, F.; Ayers, W. H. and Swithenbank, J. A. (1982), "A Fundamental Mathematical Modeling Approach to Cyclone Design", *Institution of Chemical Engineers*, 60, pp. 222-230.
4. Meier, H. F. and Mori, M. (1999), "Anisotropic behavior of the Reynolds Stress in Gas and Gas-Solid Flows in Cyclones", *Powder Technology*, 101, pp. 108-119.
5. Meier, H. F. and Mori, M. (1998), "Gas-Solid Flow in Cyclones: the Eulerian-Eulerian Approach", *Computers and Chemical Engineering*, 22, pp. 641-644.
6. Derksen, J. J.; van den Akker, H. E. A. and Sundaresan, S. (2008) "Two-Way Coupled Large-Eddy Simulations of the Gas-Solid Flow in Cyclone Separators", *AIChE Journal*, 54, pp. 872-885.
7. Zhao, B.; Shen, H.; Kang, Y. (2004), "Development of a Symmetrical Spiral Inlet to Improve Cyclone Separator Performance", *Powder Technology*, 145, pp. 47-50.
8. Bernardo, S.; Mori, M.; Peres, A. P.; Dionísio, R. P., "3-D Computational Fluid Dynamics for Gas and Gas-Particle Flows in a Cyclone with Different Inlet Section Angles", *Powder Technology*, 162, pp. 190-200.
9. Noriler, D.; Vegini, A. A.; Soares, C.; Barros, A. A. C.; Meier, H. F.; Mori, M. (2004), "A New Role for Reduction in Pressure Drop in Cyclones Using Computational Fluid Dynamics Techniques", *Brazilian Journal of Chemical Engineering*, 21, pp. 93-101.
10. "ANSYS CFX Reference Guide", *ANSYS CFX Release 11.0*, December, 2006.
11. Patankar, S. V. (1980), "Numerical Heat Transfer and Fluid Flow", ed. New York, Hemisphere Pub Co.
12. Vegini, A. A. (2007), "Eulerian-Eulerian Approach for Numerical Simulation of Cyclones", *PhD Thesis*, University of Campinas, Campinas, 2007.