LARGE-EDDY SIMULATION OF SINGLE PHASE TURBULENT FLOWS IN CYCLONES

Ricardo de Vasconcelos Salvo, rvsalvo@hotmail.com
Faculdade de Engenharia Mecânica – FEMEC, Universidade Federal de Uberlândia – UFU, Av. João Naves de Ávila, 2121, Santa Mônica, Uberlândia, Brazil.

Francisco José de Souza, chicao99br@gmail.com
Sygma Motors – Parque Tecnológico de São José dos Campos –Rodovia Presidente Dutra, km 113 São José dos Campos, Brazil

Elie Luis Martinez Padilla, epadilla@mecanica.ufu.br
Faculdade de Engenharia Mecânica – FEMEC, Universidade Federal de Uberlândia – UFU, Av. João Naves de Ávila, 2121, Santa Mônica, Uberlândia, Brazil.

Aristeu da Silveira Neto, aristeus@mecanica.ufu.br
Faculdade de Engenharia Mecânica – FEMEC, Universidade Federal de Uberlândia – UFU, Av. João Naves de Ávila, 2121, Santa Mônica, Uberlândia, Brazil.

Abstract. *A Large-Eddy Simulation (LES) of a single-phase turbulent flow in model cyclone geometry was performed. The computational code utilized is a dedicated type of code which incorporates the finite volume method using SIMPLE algorithm on unstructured three-dimensional computational grid. The standard Smagorinsky sub-grid scale model, including Van Driest wall damping function, and Yakhot's RNG sub-grid model were applied. The simulation was performed at a moderated Reynolds number, and the results for average axial and tangential velocities as well as RMS velocities in these directions show consistent agreement when compared with experimental ones. The large-scale structures were visualized, and revealed important features of this high unstable and anisotropic flow. This LES simulation was run on a PC on a reasonable time frame, suggesting that the application of this accurate methodology is affordable in an industry environment for designing and optimizing cyclones.*

Keywords*: Cyclone separator, LES, Smagorinsky, Sub-grid model, CFD.*

1. INTRODUCTION

Cyclones are widely used in many industrial process where the separation of one phase is necessary. The application of those equipments covers from process inside the food industry to mineration process. The first patent of a cyclone dates from 1891 (Svarovsky, 1984), in other words, these equipments have been used for more than a hundred years. It Stands out among other separators, because its high efficiency allied with small size, a low maintenance necessity, a geometrical simplicity (without any moving parts), and a relative low energy consumption. Over the years of using these equipments were aim of several researches, and it was revealed that although the geometry of cyclone separators is simple, the flow field in their interior is extremely complex. Initially the experimental studies used intrusive probes for measuring the local flow velocity but later breakthrough studies have used exclusively optical methods, nowadays most of the experimental studies only still cover the influence of operating parameters or geometry changing on the separation result (Bergström and Vomhoff, 2007).

As a consequence, of the lack of data about the internal flow field and the complexity of it, the design of cyclone separators has relied on empirical models. These empirical models are based on experimental data which include the effects of geometrical and operational parameters. However, different sets of experimental data lead to different equations to the same basic parameters, in other words, these models suffer from an inherent shortcoming of any empirical model. These models only can be used within the limits of the experimental data from which the models parameters were determined, Souza (2003); Narasimha et al. (2005); Narasimha et al. (2006) and several other authors.

An alternative to the use of empirical models on the design of cyclones separators is the use of computational fluid dynamics – CFD – models, which are based on the fundamental transport equations, the Navier-Stokes equations. They have shown more generic and, in theory, also more precise approach (Hoekstra et al., 1999). As mentioned, the flow field in cyclones is complex and, as a part of that complexity, it presents strong anisotropy, three dimensional boundary layers (with strong stream line curvature) and high turbulence levels normally occurring at high Reynolds numbers. Nowadays it is still impossible to apply DNS (Direct Numerical Simulation, where all the scales of turbulence are solved) for this type of flow. It means that obtaining good results with CFD models in this type of flow is directly linked with the correct modelling of turbulence. In this context, the recent literature shows that there are numerous turbulence models divided between URANS/RANS (Unsteady/Reynolds Averaged Navier-Stokes) and LES (Large Eddy Simulation) models, and that most of the URANS/RANS models do not apply properly to cyclone simulations because they are not able to compute the anisotropy.

In this paper we present the results of a Large Eddy Simulation of flat bottom cyclone in an unstructured three dimensional grid. Our results are compared with experimental data and shown consistent agreement.

2. MATEMATICAL FORMULATION

The mass and momentum conservation equations can be written, adopting the Einstein convention, for an incompressible Newtonian fluid, respectively as:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j}\left(u_i u_j\right) = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\nu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] \tag{2}$$

The variables on the above equations can be separated in a large scales part and a sub-grid part:

$$f(\vec{x},t) = \bar{f}(\vec{x},t) + f'(\vec{x},t) \tag{3}$$

Where the filtered part is given by:

$$\bar{f}(\vec{x},t) = \int_D f(\vec{x}',t) G(\vec{x}-\vec{x}') d\vec{x}' \tag{4}$$

And the filtering function can be defined as:

$$G(\vec{x}) = \frac{1}{\Delta_x \Delta_y \Delta_z}, \text{ if } |\vec{x}| \le \Delta/2 \tag{5}$$

or zero otherwise.

Here $\Delta_{(i)}$ is the characteristic size of the filter in all three spatial directions, which determines the cut-off frequency of filtering process. Applying this process on Eq. (1) and Eq. (2) and introducing the Boussinesq hypothesis. We obtain:

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j}\left(\bar{u}_i \bar{u}_j\right) = -\frac{1}{\rho}\frac{\partial \bar{p}^*}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\left(\nu + \nu_t\right)\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)\right] \tag{6}$$

Where:

$$\bar{p}^* = \bar{p} + \frac{2}{3}\rho K \quad , \qquad k = \frac{1}{2}\left(\overline{u_i' u_i'}\right) \tag{7}$$

and in Eq. (6), the turbulent viscosity has been modeled.

3. TURBULENCE MODELS

3.1. Smagorinsky turbulence model

The earliest, the simplest and the most used sub-grid turbulence model is the Smagorinsky model (Smagorinsky, 1963), and it consists on the basis of many other more “sophisticated” models (Pope, 2003). In this model is assumed that the production, Eq. (8), and the dissipation, Eq. (9), of sub-grid turbulent stress are equal to each other (Germano et al., 1990; Silveira-Neto, 2002).

$$\wp = 2\nu_t S_{ij} S_{ij} \tag{8}$$

$$\varepsilon = -c_1 \frac{\overline{u_i' u_j'}^{3/2}}{l} \tag{9}$$

Then the turbulent viscosity is represented, in analogy to the mixture length hypothesis, by:

$$\mu_t = \rho (Cs\Delta)^2 \overline{S} \tag{10}$$

Where:

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \quad , \quad \overline{S} = \sqrt{S_{ij} S_{ij}} \quad , \tag{11}$$

and the total viscosity is given by the sum of the turbulent and molecular viscosities. Here, the term, Cs is a parameter to be determined and it varies according to the type of flow, the Reynolds number, the grid resolution, and several other parameters (Germano et al, 1990; Ferziger and Peric, 2002). Another difficulty associated with the utilization of this model is the fact that the turbulent viscosity does not reduce itself in near wall flows as it should. A common practice is to use a damping function with this model, like the Van Driest wall damping function, Eq. (12) (Ferziger and Peric, 2002).

$$Cs = Cs_0 \left(1 - e^{-n^+/A^+}\right)^2 \tag{12}$$

3.2. Sub-grid turbulence model from "Yakhot et al. (1986)"

The main difference in the model proposed by Yakhot et al. (1986), when compared to the standard Smagorinsky model, is the way how the total viscosity is calculated. In this model the viscosity is given by:

$$\mu_{tot} = \mu \left[1 + H\left(\frac{\mu_{SGS}^2 \mu_{tot}}{\mu^3} - C\right)\right]^{1/3} \tag{13}$$

In the equation above μ_{SGS} is calculated in the same way that in Eq. (10), however, the difference that in this model Cs is a theoretical constant (Cs = 0.157), C is another constant (C = 100) and H is the Heaviside Ramp function. Besides, this model has the main advantage of yielding correctly the zero value for SGS viscosity in low Reynolds numbers without any *ad hoc* modifications (Slack et al., 2000).

4. NUMERICAL METHOD

For our proposal, we performed the simulations by using the computational code UNSCYFL3D, this code has been developed as a dedicated tool for simulation of highly rotational flows, aiming at cyclones/hydrocyclones separators and swirl tubes. It incorporates the finite volume method using SIMPLE algorithm for the pressure correction equation on unstructured three-dimensional computational grid. The Finite Volume method requires two approximation levels, approximation of the integrals and approximation of the variable values at locations other them control volume center (Ferziger and Peric, 2002). The UNSCYFL3D code utilizes for these approximations, respectively, the midpoint rule and linear interpolation, which are second order approximations (spatial interpolations are made with second order central difference scheme-CDS, but if more stability is needed it is possible to use a blending with the first order upwind interpolation scheme-UDS). A fully implicit second order accuracy temporal scheme, three-time level is also used. The non-smoothness of the grid and non-orthogonality effects are also taken into account (for more details, see section 8.8 of the book by Ferziger and Peric, 2002 - third, rev. edition).

The cyclone geometry simulated, as shown on figure 1-a, is a flat bottom gas cyclone without the underflow (only the gas phase was simulated). This geometry is the same utilized by Hoekstra et al. (1998), who kindly supplied the experimental data for comparison.

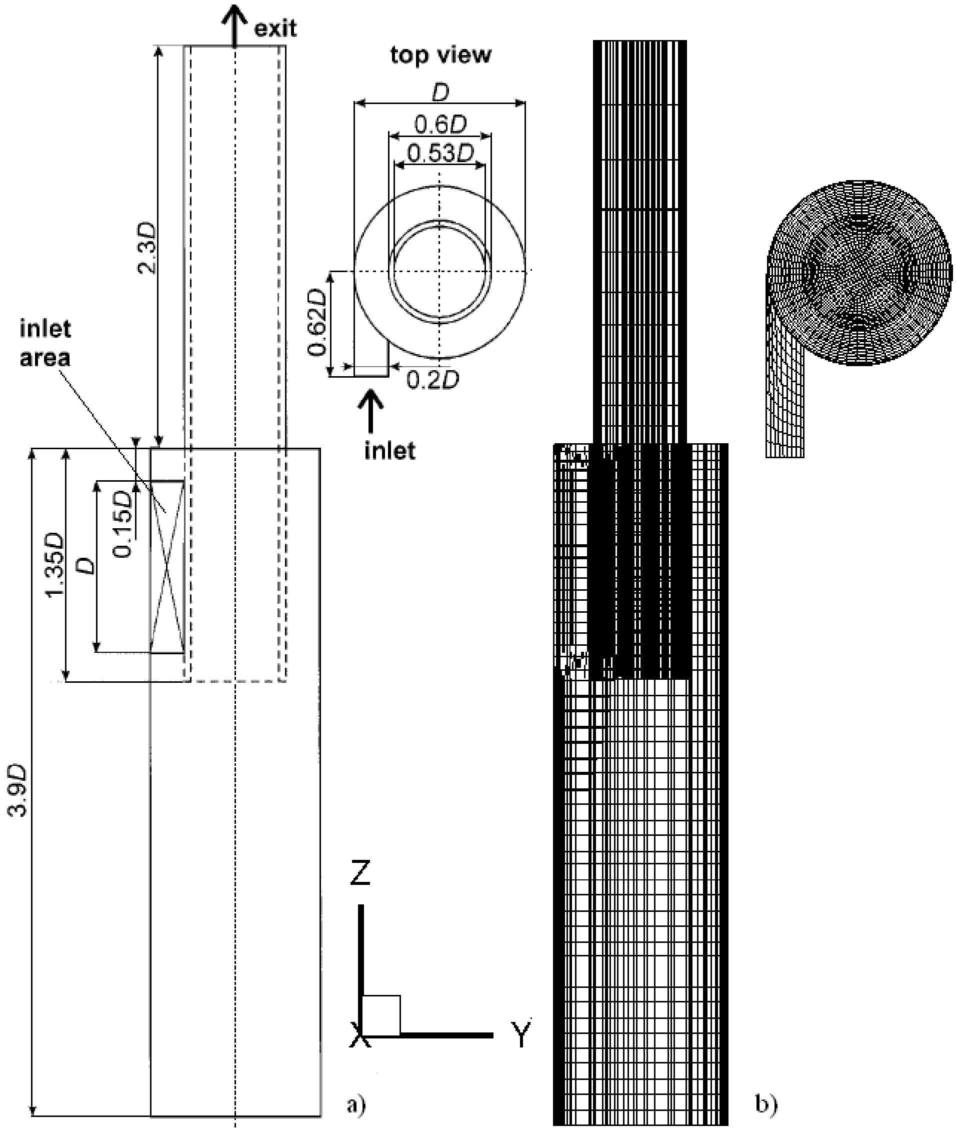


Figure 1: a - Flow geometry (adapted from Derksen and Van den Akker, 2000). b- Computational grid.

In all simulations performed the cyclone diameter was 0.1 m, the inlet velocity was 2.26 m/s, the density was 1.2 Kg/m³ and the viscosity was 1.808E-05 m²/s, resulting in a Reynolds number of 15.000. The tridimensional grid utilized had a little more than 100000 hexahedral elements, as shown in Figure 1-b. For each simulation, a zero initial flow field was utilized, in a steady state, to obtain a developed initial flow field. This condition was utilized as an initial field in the unsteady simulation. The averages were made by using a time period of 80D/Uin, where D is the diameter of the cyclone and Uin is the inlet velocity.

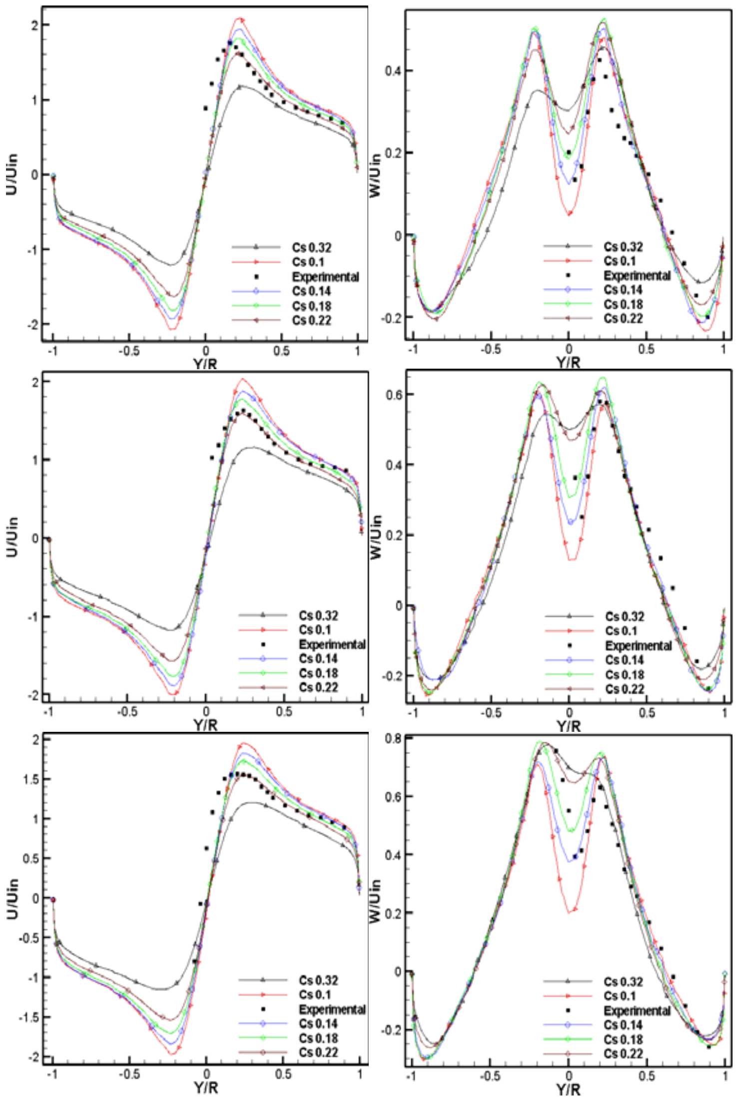
5. RESULTS AND DISCUSSION

The tangential, axial, RMS tangential and RMS axial velocities profiles were analyzed in four different axial positions (Z/D = 0.89, 1.39, 1.89 e 2.39), and have been compared with LDA experimental data (provided by Hoekstra et al., 1998). The simulation conditions were as close as possible from those utilized in the experiments, and they differed mostly in the inlet and outlet regions (the differing regions and the reasons that they occurred were the same presented by Derksen and Van Den Akker, 2000).

In the inlet region the main difference was the lack of introduced turbulence on it, but it is worth commenting that Ma L., Ingham and Wen (1999) have varied the turbulent intensity in their cyclone inlet from 10 to 20 % and have not found any significant difference in the velocity profiles. Bisedes, Hovenden and Davidson (1997) have also analyzed the influence of different turbulent intensities on the entrance of a spray dryer. They did not find any significant changes in the velocity profiles, meaning that high swirling, recirculating flows, such as the flow in cyclone separators, do not suffer much influence from changes in the turbulence intensity at the entrance of the equipment. Hence, the difference that could affect the results probably is in the outlet region, once in the simulation a zero gradient condition was adopted, but in the experiments the outlet blew into free space.

Before we analyze the flow phenomena itself, some adjustments were made, and the influence of some parameters was checked, including the Smagorinsky constant, the solver tolerance and the time step.

First of all, since the Smagorinsky constant is not really constant, five simulations were made by using different Cs values (Cs = 0.22, 0.18, 0.14, 0.1), with a solver tolerance of 1.0E-03 and a time step of 1.0E-03. The resulting velocity profiles were compared at the four axial positions as can be seen in Figures 2 and 3. It is really clear that there is not a Cs value which provides good agreement with all experimental velocity profiles in all positions, but considering all data, the simulation where Cs 0.14 yield better results than the others values tested.



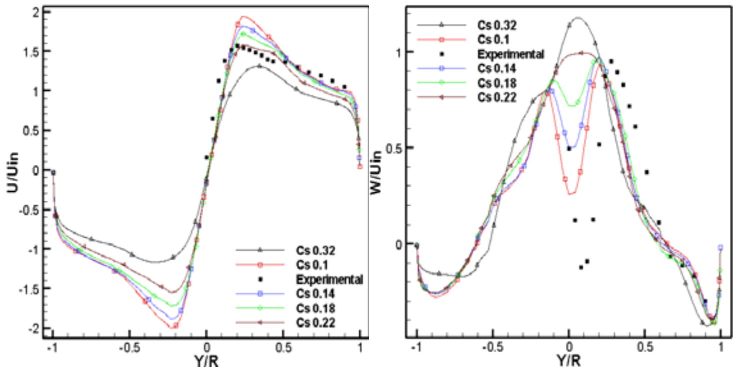
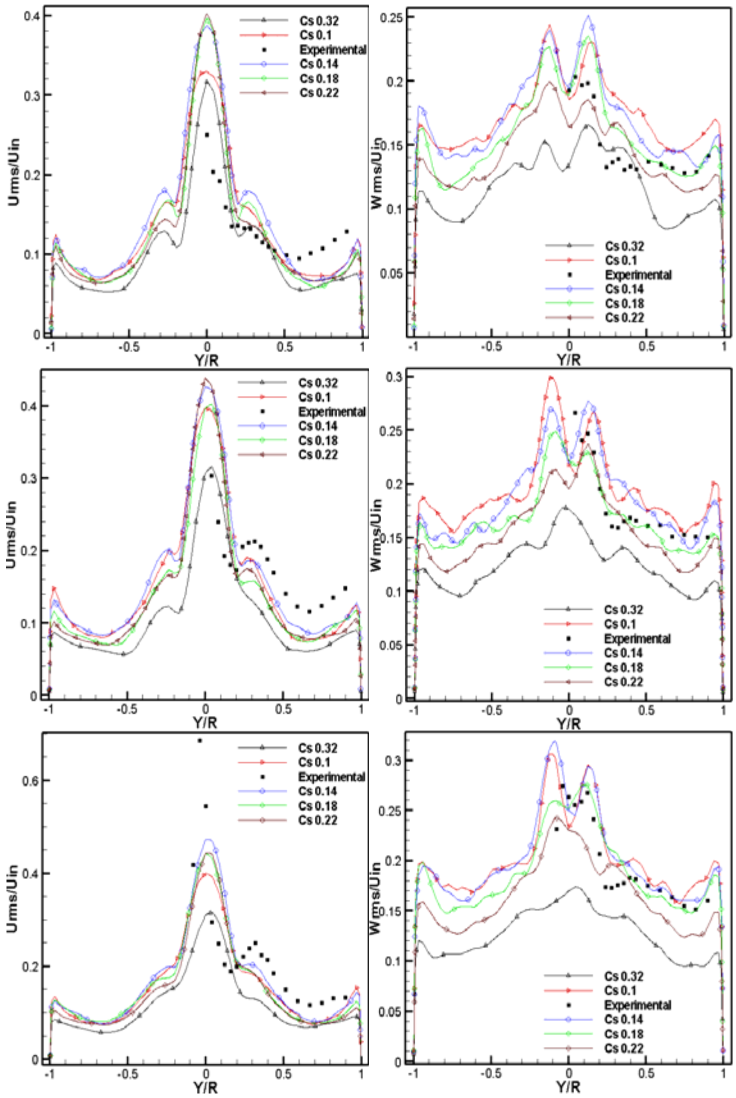


Figure 2: Radial profiles of the average velocities, tangential (left), axial (right), for different Smagorinsky constants, from top to bottom Z/D = 0.89, 1.39, 1.89 and 2.39.



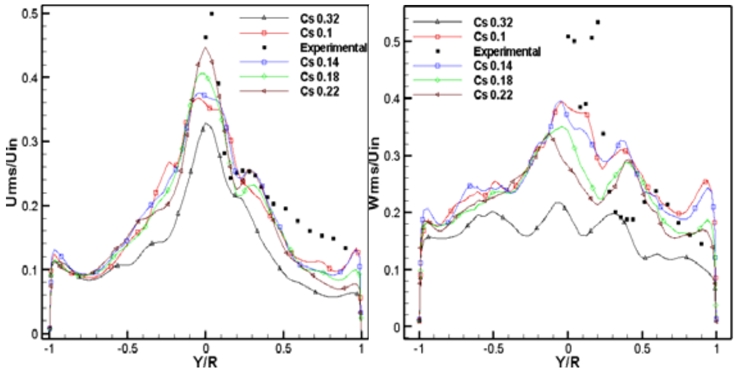
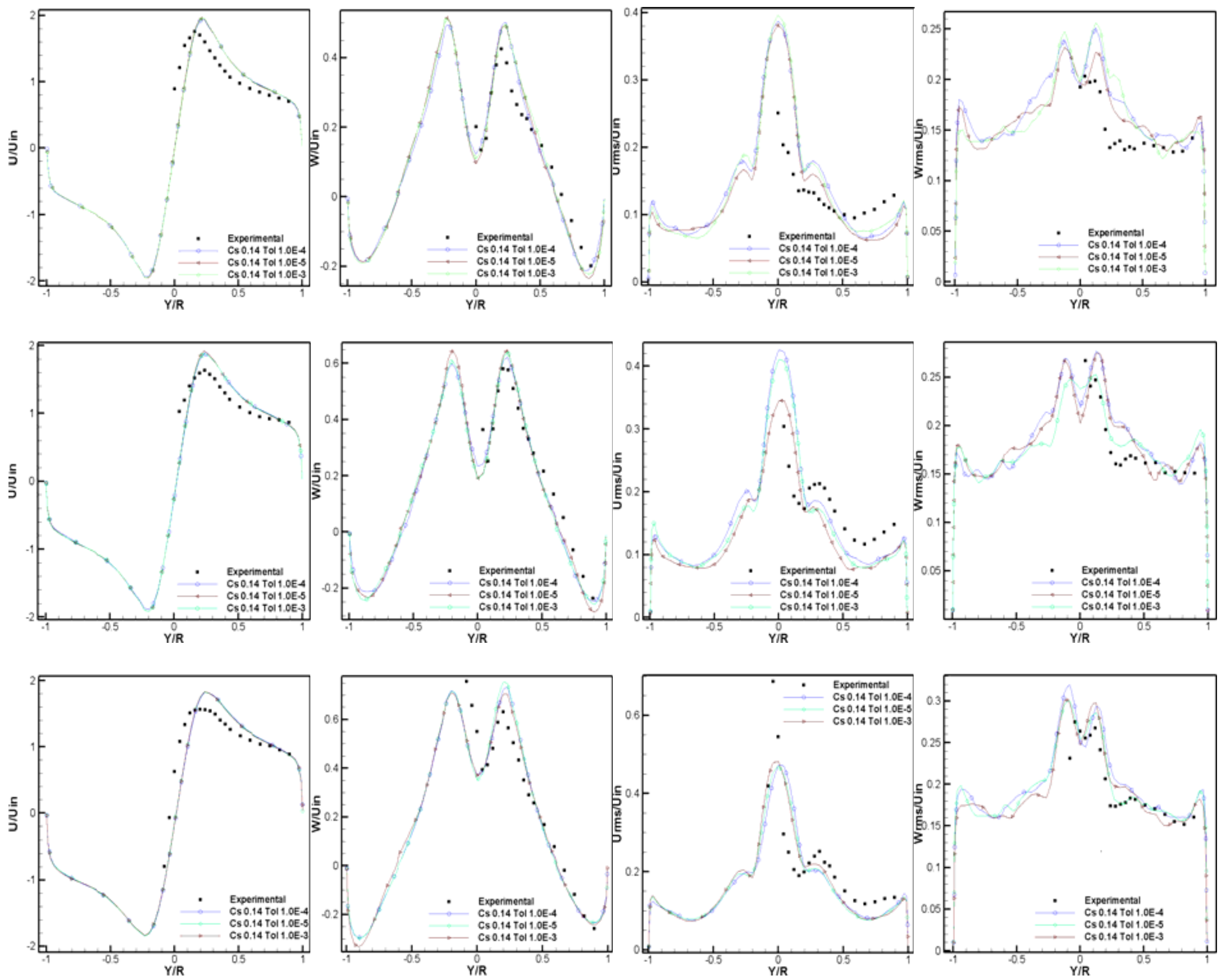


Figure 3: Radial profiles of the average RMS velocities, tangential (left), axial (right), for different Smagorinsky constants, from top to bottom Z/D = 0.89, 1.39, 1.89 and 2.39.

After these initial testes, a solver tolerance test was made. This test consists of three simulations all using Smagorinsky turbulence model, with Cs 0.14 and time step of 1.0E-03 s. The results for all axial positions are compared with the experimental data that can be seen in Figures 4-A and 4-B. By analyzing this data, it is possible to notice that even without any significant changes in tangential and axial velocity profiles for all the axial positions, the tolerance caused a considerable difference in all RMS profiles.



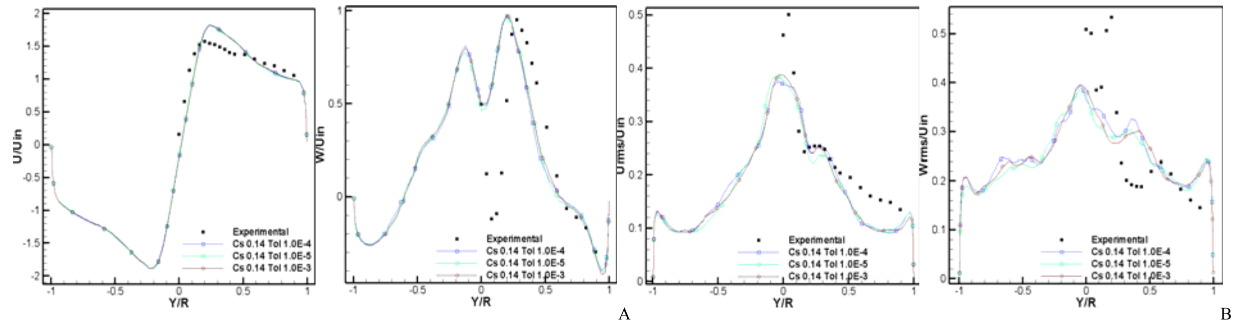
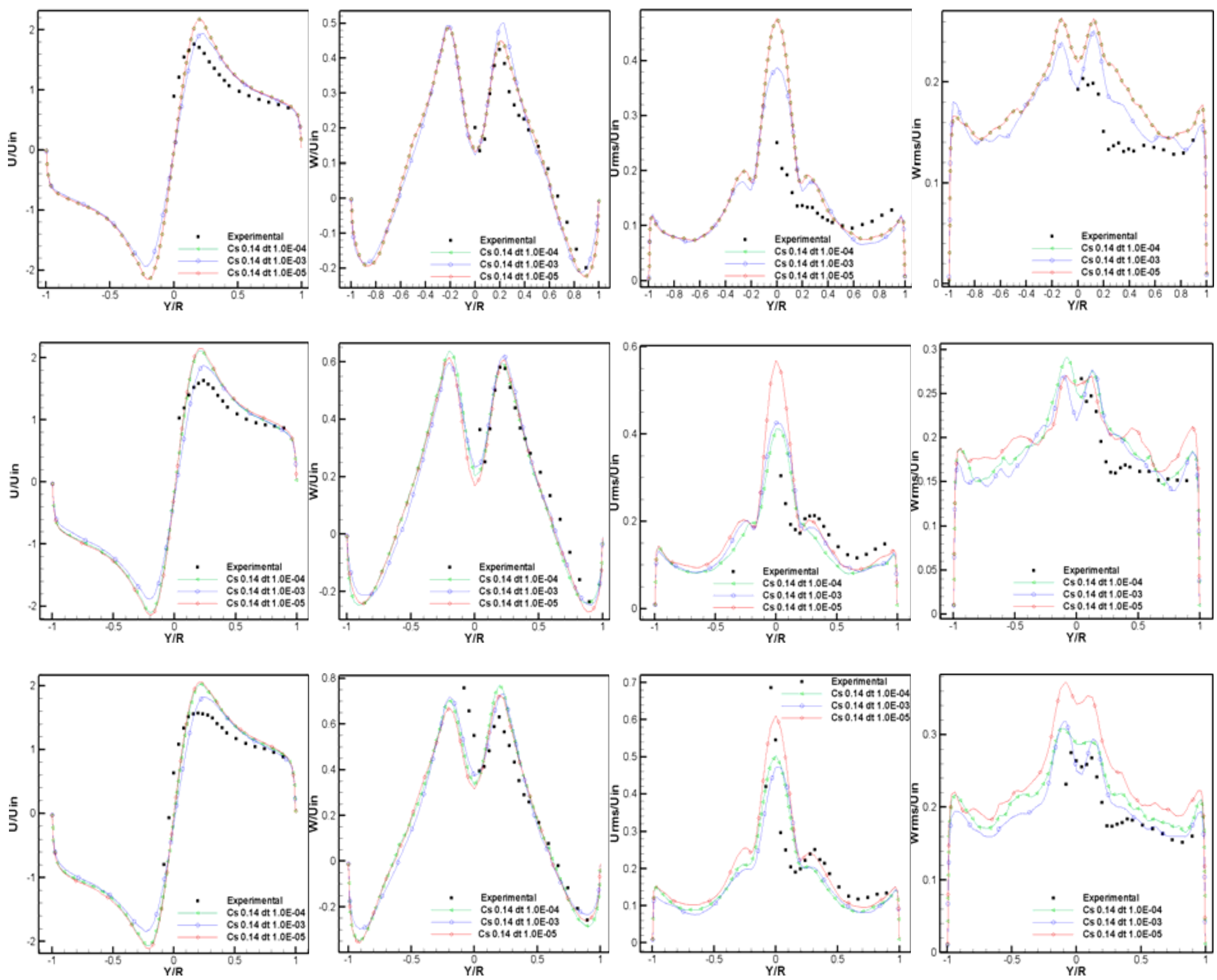


Figure 4: Radial profiles of the average velocities, tangential (left), axial (right), for different solver tolerance, from top to bottom Z/D = 0.89, 1.39, 1.89 and 2.39 (A). Radial profiles of the average RMS velocities, tangential (left), axial (right), for different solver tolerance, from top to bottom Z/D = 0.89, 1.39, 1.89 and 2.39 (B).

A final test was made in order to verify the time step influence on the solution results. In this test, three simulations all using Smagorinsky subgrid model, with Cs 0.14, solver tolerance of 1.0E-04, and different time steps (1.0E-03, 1.0E-04 and 1.0E-05 s) were compared to the experimental data in all axial positions. The results can be seen in Figures 5-A and 5-B. The different time steps did not bring any significant changes in tangential and axial velocity profiles for all the axial positions, but, just as in the different tolerance case, a considerable difference in all the RMS profiles can be noticed.



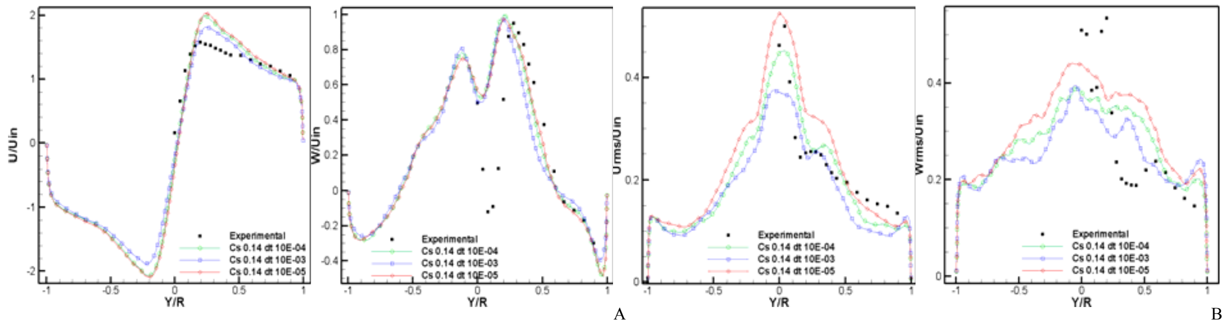


Figure 5: Radial profiles of the average velocities, tangential (left), axial (right), for different time steps, from top to bottom Z/D = 0.89, 1.39, 1.89 and 2.39 (A). Radial profiles of the average RMS velocities, tangential (left), axial (right), for different time steps, from top to bottom Z/D = 0.89, 1.39, 1.89 and 2.39 (B).

5.1 Flow field analysis

The stream traces in Figure 6-a show the complicated flow patterns inside the cyclone, with the outer spiral descending flow and the inner spiral upper flow. The instantaneous vortex core position can be seen in Figure 6-b. Some recirculation zones can be seen in Figures 6-c and 6-d.

The RMS velocity profiles show really complicated features, which according to Derksen and Van Den Akker (2000), as a consequence of the precessing vortex core (PVC). In that RMS values are much higher in the center of the cyclone when compared to the ones close to the walls, which would be the result of a combination between the core precession and the high tangential velocity gradients.

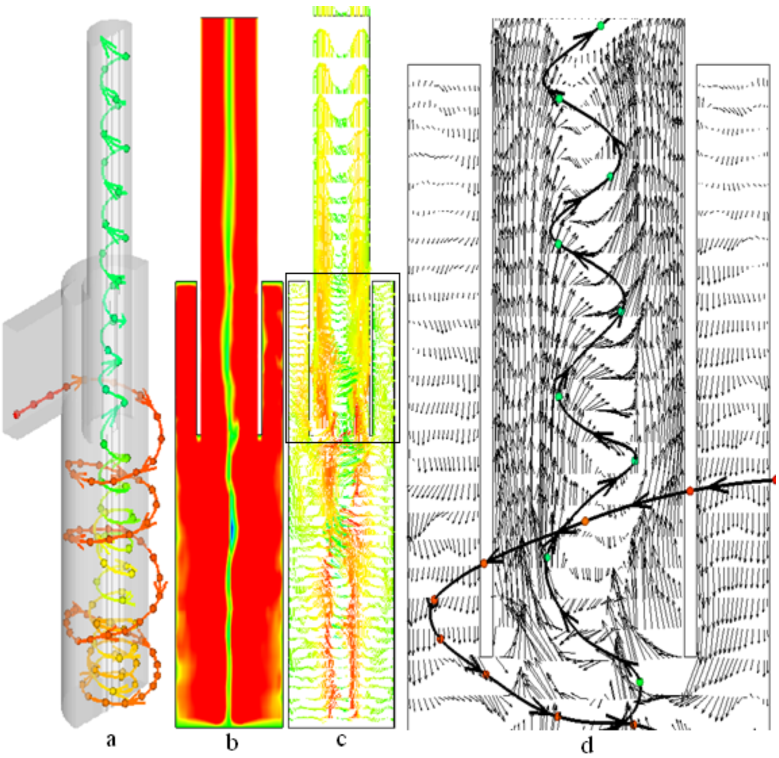


Figure 6: Details of the average (a) and instantaneous (b, c and d) flow field inside the cyclone.

5.2 Comparison between the two turbulence models

A simulation was performed with the Yakhot's turbulence model. In this simulation a time step of 1.0E-03 s and a solver tolerance of 1.0E-04 were used. The results were compared with the experimental data for all axial positions and the results of a simulation with the Smagorinsky model with Cs 0.14, time step of 1.0E-03 s and solver tolerance of 1.0E-04. The data can be seen in Figure 7.

The Yakhot's model was not able to predict the slope in the axial velocity profiles, it almost behaves like the Smagorinsky model with a larger Cs. Probably this happened due to the insufficient mesh resolution utilized (at least for an LES simulation).

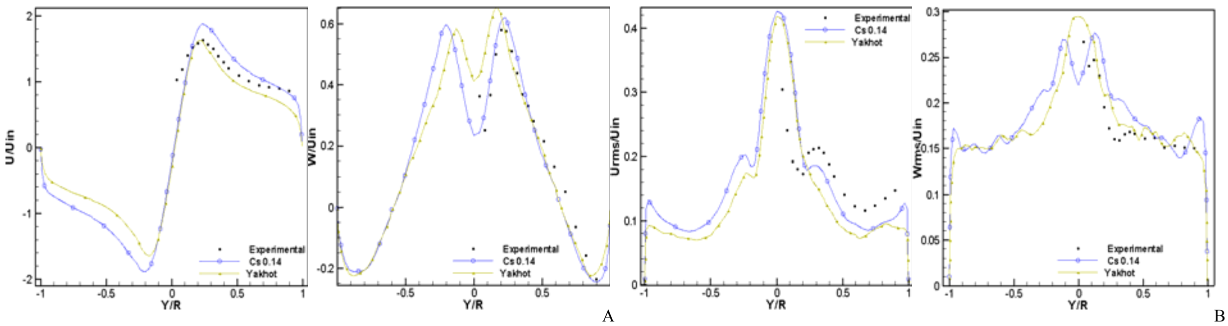


Figure 7: Radial profiles of the average velocities, tangential (left), axial (right), for different turbulence models, Z/D = 1.39 (A). Radial profiles of the average RMS velocities, tangential (left), axial (right), for different turbulence models, Z/D = 1.39 (B).

6. CONCLUSION

Several tests were run in order to analyze the influence of the time step, Smagorinsky constant and solver tolerance. On the results, unfortunately, due to computational limitations, only one grid was utilized, but with the data gathered in this work, it will be possible to setup a few more cases and analyze the influence of those parameters on a finer grid, and then make another Large Eddy Simulation with a computational grid even finer.

Considering the simulations with the Smagorinsky model, Cs 0.14, the results have shown good agreement with experimental data, diverging mostly for the axial RMS velocity profiles. This is probably due to the coarse grid utilized, since no grid tests were made. The flow field has also showed good agreement with the literature, enabling to the identification of flow phenomena, such as the PVC, recirculating zones.

7. ACKNOWLEDGEMENTS

The authors would like to thank A. J. Hoekstra for supplying the experimental data.

8. REFERENCES

Bergström, J. and Vomhoff, H., 2007, "Experimental hydrocyclone flow field studies", Separation and Purification Technology, Vol 53, pp. 8-20.

Derksen, J.J. and Van den Akker, H.E.A., 2000, "Simulation of vortex core precession in a reverse-flow cyclone", AIChE Journal, Vol 46, N° 7, pp. 1317-1331.

Ferziger, J.H. and Peric, M., 2002, "Computational methods for fluid dynamics", Springer.

Germano, M., Piomelli, U., Moin, P. and Cabot, W.H., 1990, "Adynamic subgrid-scale eddy viscosity model", American Institute of Physics, Vol 7, pp. 1760-1795.

Hoekstra, A.J., Derksen, J.J. and Van den Akker, H.E.A., "An experimental study of turbulent swirling flow in gas cyclones", Chemical Engineering Science, Vol 54, pp. 2055-2065.

Hovenden, R.J. and Davidson, M.R., 1997, "Turbulence modelling of single phase flow in a spray dryer", Inter Conf on CFD in Mineral & Metal Processing and Power Generation, CSIRO 1997, pp. 459-465.

Ma, L., Ingham, D.B. and Wen, X., 2000, "Numerical modelling of the fluid and particle penetration through small sampling cyclones", J. Aerosol Sci., Vol 31, N° 9, pp. 1097-1119.

Narasimha, M., Brennan, M. and Holtham, P.N., 2006, "Large eddy simulation of hydrocyclone – prediction of air-core diameter and shape", International Journal of Mineral Processing, Vol 80, pp.1-14.

Pope, S.B., 2003, "Turbulent Flows", Cambridge University Press, 2003.

Silveira-Neto, A., 2002, "Turbulência nos fluidos aplicada", Faculdade de Engenharia Mecânica, Universidade Federal deUberlândia – UFU, Uberlândia, Brasil.

Slack, M.D., Prasad, R.O., Bakker, A. and Boysan, F., 2000, "Advances in cyclone modelling using unstructured grids", Institution of Chemical Engineers, Vol 78, Part A, pp. 1098-1104.

Souza, F.J., 2003, "Simulação de grandes escalas de escoamentos emum hidrociclone", PHD teses, Faculdade de Engenharia Mecânica, Universidade Federal deUberlândia – UFU, Uberlândia, Brasil.

9. RESPONSIBILITY NOTICE

The four authors are the only responsible for the printed material included in this paper.