NUMERICAL SIMULATION OF TURBULENT GAS-SOLID FLOWS IN CYCLONE SEPARATORS

Jean-Pierre Minier[+], Jean-Daniel Mattei
Laboratoire National d'Hydraulique, Electricité de France
6 quai Watier, 78400 Chatou, France
[+]e-mail: Jean-Pierre.Minier@der.edf.fr

Mehdi Ouraou
Simulog, 1 rue James Joule, 78286 Guyancourt, France

ABSTRACT

This paper discusses theoretical and numerical issues that are relevant to computational calculations of cyclone separators. The importance of satisfactory predictions of gas velocities, of an adequate particle stochastic model and of a consistent numerical scheme is stressed. Numerical exemples are provided for two Stairmand-type cyclones and two different kind of particles to illustrate these various points.

INTRODUCTION

Reverse-flow cyclone separators are devices used to separate particles (solid particles or droplets) from gas streams. They are used for environmental purposes, like gas cleaning, or constitute key elements of a number of industrial processes. This is the case of the Circulating Fluidized Bed process in power plants where cyclones ensure solid particle (usually coal particles) recycling to the furnace. For these reasons, there is growing interest in realiable ways to evaluate cyclone performances. Current designs are often based on past experience and engineering know-how. While useful for some purposes, these design rules were developed mostly at a time when requirements were not as stringent as today and it remains unclear how non-standard design performances can be assessed. It is thus of key importance to check current designs with respect to new requirements, to provide help for improvement and hopefully to come up with new design rules.

Numerical simulations are helpful for troubleshooting and for analysing new design ideas. However, numerical computations of turbulent gas-solid two-phase flows is not an easy or standard case for available codes. Indeed, one can say that simulations of gas-solid flows in cyclone separators is at the crossroads between theoretical modelling, engineering calculations and practical needs. Consequently, care must be taken when carrying out the calculations and analysing computational outcomes.

The purpose of the present paper is not to claim that satisfactory predictions are readily obtained with a given code. It is rather to report some experience with numerical computations of cyclone separators. The purpose is actually three-fold. First, to present a stochastic model. Second, to outline numerical issues and related requirements. Finally, to present numerical results that are helpful to assess current state and also to point out the main areas in which improvements are needed.

EQUATIONS OF THE MODEL

Most of the flows considered are high-Reynolds-number flows, and a statistical approach as well as a turbulence model are needed. In the present work we concentrate on two-phase flows with low loading-ratios (dilute flows). The simulation is performed in two stages: the gas flow is first computed and, then, particle properties are calculated using the gas flow predictions. Since the main result is cyclone efficiency over a range of particle diameters, the particle phase is simulated with a Lagrangian (or particle-tracking) approach. This approach treats convection as well as any variation in particle properties, however complicated, without approximation and is therefore well suited for poly-dispersed two-phase flows. The general approach is a coupled Eulerian/Lagrangian approach. It is also a stochastic approach in which a large number of particles are followed through the flow and from which mean quantities are obtained by ensemble averaging. The stochastic models are introduced since each particle trajectory must be calculated using the instantaneous fluid velocities encountered as the particle moves across the flow. However, gas flows are computed using classical models which provide only mean fields, such as the mean gas velocity field, the mean kinetic energy, ... Consequently, insufficient information is available and the instantaneous fluid velocities sampled by the particles have to be reconstituted by the stochastic model.

The present model represents these fluid velocities by a stochastic diffusion process (Arnold, 1974) and is based on Langevin equations already used in numerous fields from Brownian motion to biological studies. The governing particle stochastic equations have the following form (Pozorski and Minier, 1998; Pozorski and Minier, 1999; Minier, 1998)

$$dx_{p,i} = U_{p,i}\, dt, \tag{1}$$

$$dU_{p,i} = \frac{U_{fs,i} - U_{p,i}}{\tau_p}\, dt, \tag{2}$$

$$dU_{fs,i} = -\frac{1}{\rho}\frac{\partial \langle P \rangle}{\partial x_i}\, dt - \frac{U_{fs,i} - \langle U_{fs,i} \rangle}{T^*}\, dt + \sqrt{C_0^* \langle \epsilon \rangle} dW_i, \tag{3}$$

where τ_p is the particle relaxation time, T^* the timescale of the fluid velocity sampled, C_0^* a function which defines the diffusion coefficient of the stochastic process and $\mathbf{W}$ a vector of independent Wiener stochastic processes. Details on general expressions of the drift and diffusion coefficients are developed in Minier (1999). In the present paper, we limit ourselves to the simplified form above and further take C_0^* as a constant.

The present model does not claim to be the definitive answer to the modelling problem. However, it is ensured that the particle tracer limit is correctly obtained. Indeed, when the particle characteristic time τ_p goes to zero, as is the case for very small particles, the model reverts to a stochastic model well known in single-phase PDF modelling (Pope, 1994). A formulation of the model in terms of the instantaneous fluid velocities and a proper account of the mean pressure-gradient avoids the appearance of so-called spurious drifts. The development of a two-phase flow model free of any spurious drift is a key issue for theoretical consistency (the model for fluid particle must simply be consistent with the mean Navier-Stokes equation) and for present applications, since cyclone efficiency for small particles is precisely what is sought.

NUMERICAL ISSUES

Numerical calculation of cyclone separators represents a challenging test case and computation of both phases raises difficulties that must be properly addressed. First of all, we are dealing with a three-dimensional flow in a complex geometry. Furthermore, this is a swirling flow with very high rotational velocities in confined geometries. Developments and tests of suitable turbulence models for present configurations remains a subject of research, not to mention accurate and efficient numerical algorithms to solve the coupled mean gas equations. It has been found that second-order turbulence models, or Reynolds stress models, bring valuable improvements compared to calculations using $k - \epsilon$. Yet, the turbulence model is not the only key point and numerical resolution may play an even more important role. As is explained in the next section, numerical treatment of the convective operator (first treated using the characteristic method and then with a completely conservative finite-volume formulation) has a marked influence on numerical outcomes, particularly on gas tangential velocities.

This is also a challenging test case for the computation of particle trajectories. One has to integrate a set of stochastic differential equations which are far more difficult and trickier to solve than ordinary differential ones. When dealing with stochastic equations, it is crucial to pay special attention to the discretisation of the stochastic term so as to respect its defining properties. Failure to do so amounts to introducing numerical spurious drifts (Kloeden and Platen, 1992). Then, we are also dealing with a range of particle diameters and we are interested in the limit of small particles when $d \to 0$ which implies $\tau_p \to 0$. This means that the set of equations becomes a set of stiff stochastic differential equations. It is important to avoid using explicit scheme having time step constraints, since this would result in far too small time steps. Lastly, present computations require calculating complete particle trajectories until each particle reaches either the vortex finder or is entrained through the gas outlet. Since particle typically describes a long and spiralling trajectory, this can be a time-consuming task. At this stage, it is often assumed in the literature that particles touching a wall boundary in the conical section, is collected. This obviously relieves the numerical burden, but is not physically justified and may cut off any possible re-entrainment. In order to use higher time steps (for a given precision), a second-order scheme appears preferable. Computations reported in this paper have been obtained with an explicit, yet unconditionally stable, second-order scheme (in the weak sense) based on (proper) prediction-correction ideas (Minier and Talay, 1999).

NUMERICAL APPLICATIONS

The gas flow inside a cyclone has quite complicated patterns. This is a reverse swirling flow having quite high rotational velocities. The swirl is created by the tangential inlet, but experimental measurements indicate that the gas flow has the structure of a double-helix, spiralling downwards with the same intensity to the vortex finder where it reverses and then spirals upwards in a cylindrical volume having roughly the diameter of the gas exit. In a cyclone separator, particles are not separated by gravity but precisely by the effect of this very double gas swirl. The separation mechanism is based on the centrifugal forces developed by the spinning gas flow. Particles are entrained towards the outer walls where the downward spiralling gas motion brings them to the particle exit while the flow reverses. From this description, it appears that the two key features of the gas flow are: first, a correct prediction of the rotational intensity, which creates the centrifugal force, and, second, a correct reproduction of the negative axial velocities near the wall boundaries in the cylindrical and conical sections.

Two calculations are reported in the present paper. Both of them concern high-efficiency Stairmand-type cyclones. This is a classical design where the cyclone outer diameter is the only free parameter. The first case is a Stairmand design with a diameter of $20cm$. Detailed profiles of the gas tangential and axial mean velocities are available at various sections inside the cyclone which span nearly its total height (Boysan et al., 1983). However, the efficiency curve was not measured in the same experiment and available data for the same case is still made up by Stairmand original values of 1951 (reported in Boysan et al, 1986). For that case, the particles are solid particles with a density of $2500kg/m3$. The second comparison is made against the experimental measurements of Dirgo and Leith (1985) in a $30cm$ Stairmand cyclone and using liquid droplets having a density of $800kg/m3$. For this second case, the particle efficiency curve was measured but no data are available for the gas mean velocities inside the cyclone. It is regrettable that (to the authors' knowledge), no recent complete data set, including both gas velocity distributions and particle efficiency curve, is available. Indeed, both results are needed, for the same case, to pinpoint the sources of discrepancies that may be observed on the efficiency curve.

For the original Stairmand design, the inlet velocity is about $25m/s$ and the gas rotates at around $40m/s$. Satisfactory tangential velocities, which are key results for the separation mechanism, have been obtained with a Reynolds-Stress Model (RSM) on a fine enough grid (about 400000 nodes) and for a complete 3D computation (see Fig. 1). In the same figure, a first numerical outcome is also plotted (referred to as first simulation in Fig. 1). The two simulations differ only in the numerical treatment of convective terms and both used the same turbulence model. It is seen that, for the first simulation, gas rotational velocities are clearly underpredicted which will have marked effects on predicted efficiencies. Although not displayed, satisfactory agreement has also been obtained for mean axial gas velocities at the same sections.

As already mentioned, cyclone performance is characterized by the efficiency curve which represents the proportion of particles which are collected as a function of the particle diameter. This is calculated by a Monte Carlo approach since the present model is a stochastic model: for each class of diameter, a large number of particles are followed and the fraction of those being collected represents the numerical prediction of the cyclone efficiency for the same diameter. For the original Stairmand design, computed efficiency curves are plotted in Fig. 2, where standard boundary conditions, namely elastic bouncing, have been used for particles hitting solid walls. It is seen that satisfactory results are obtained, in particular for the prediction of the diameter d_{50} which is the diameter corresponding to an efficiency of 50%. The predicted efficiency has a steeper slope compared to the (old) measurements, a feature nearly always observed in similar cyclones. The computed efficiency goes quickly to its limit value of total collection while measured values seem to level off at about 0.9. Apart from experimental uncertainties, this could indicate that other physical phenomena, such as possible particle re-entrainment due to turbulent bursts, may be present. In the same Figure, the efficiency curve computed with mean gas velocities from the first simulation is also plotted. It is seen that lower tangential velocities result in a shift of the efficiency curve towards higher particle diameters and lead here to an overestimation of the d_{50} diameter. These results illustrate that, in order to obtain reasonable estimation of the cyclone performance, the two key features of the gas flow (correct rotational intensity and existence of negative axial velocities in the vicinity of the outwer wall boundaries) must be first correctly reproduced. However, this is not the end of the numerical story. As underlined in the previous section, the particle trajectories must be accurately simulated. In particular, both computed efficiency curves corectly goes to zero as the particle diameter becomes very small, indicating that the stiff model equations are properly handled. The same time step was used for the whole range of particle diameter, and the numerical scheme remains of second-order even for fluid particles ($d \to 0$).

The difference between the slopes of the experimental and predicted numerical curves led to us to consider another case, investigated by Dirgo and Leith (1985). The cyclone has a higher outer diameter and a lower inlet gas velocities of around $25m/s$, resulting in more moderate gas rotational intensities inside the cyclone. Particles have a smaller density compared to the first case and are of a different nature. Since droplets can interact with solid walls in more ways than solid particles, the correct boundary conditions to use for droplet-wall interactions (elastic bouncing, non-elastic bouncing, sticking conditions or possibly droplet breakup) are not obvious. It is likely that these different conditions may all apply depending upon the way a particular droplet hits a wall (angle of impact, droplet velocities, ...). Numerical calculations are shown in Fig. 3. The two curves corresponding to standard conditions (elastic bouncing) seem to have the correct shape and slope compared to the experimental values but are shifted and, therefore, underestimate the cyclone efficiency. For this case, gas mean velocities did not look as satisfactory as in the first case with respect to the two key criteria. Thus, it is possible that the undestimation of the efficiency is due to an underestimation of gas rotational intensities. However, in the absence of experimental measurements on the gas flow, this cannot be clarified or proved. This illustrates that *detailed and complete* data set are needed if one is to assess rigorously numerical outcomes. Further information can be obtained from Fig. 3. Two efficiency curves correspond to standard conditions, but to different numerical parameters. Compared to the first curve, the second one is obtained with a time step ten times smaller and with nearly four times more particles in each particle class. Both curves are nearly identical, showing the robustness of the particle scheme and that the 'limit results' are reached even for a reasonable time step and not too many particles per class. Such considerations are important to reduce computational requirements, without stopping particle trajectory calculations simply for lack of cpu time. Then, another efficiency curve has been simulated in which droplets are assumed to be collected whenever they touch a solid wall in the conical section. This results in higher numerical efficiencies for droplets up to 4μ without modifying efficiencies for bigger ones. By changing boundary conditions, applying the sticking conditions throughout the cyclone for example, the predicted curve can be dricen towards the experimental one. Yet, it remains unclear if this is a justified way to carry out the simulations and if the predicted efficiency has necessarily a more reliable value. Additional information is definitively required to pinpoint where the actual shortcomings.

CONCLUSION

The considerations developed in the present paper and the reported applications have tried to emphasize that numerical simulations of cyclone separator performances require some care and a thorough analysis of the results. The idea that performances can be simply assessed, and that new designs can be devised, through blind calculations or by applying ready-to-use code is misleading. Actually, more validation appears necessary and this case should receive more than the limited attention it usually gets. Computing accurately gas flow inside a cyclone is a difficult but interesting problem for numerical turbulence modelling. This is also a challenging problem for two-phase flow modelling and numerical simulations of particle properties. Calculating cyclone efficiency is a practical problem which requires theoretical developements, such as high-order numerical schemes for stiff stochastic differential equations and efficient Monte-Carlo methods.

REFERENCES

L. Arnold, *Stochastic Differential Equations: Theory and Applications*, (New-York, Wiley, 1974).

F. Boysan, J. Swithenbank and W.H. Ayers, "Mathematical Modelling of Gas-Particle Flows in Cyclone Separators," Chap 42 in Encyclop. of Fluid Mech Vol 4, N.P. Cheremisinoff editor (Gulf Publishing Company, 1986).

F. Boysan, B.C.R. Ewan, J. Swithenbank and W.H. Ayers, "Experimental and theoretical studies of cyclone separators aerodynamics," I. Chem. E., POWTECH Conference, 1983.

J. Dirgo and D. Leith, "Cyclone collection efficiency: comparison of experimental data with theoretical predictions," Aerosol Sci. Technol., **4**, 401-415, 1985.

W.D. Griffiths and F. Boysan, "Computational fluid dynamics (CFD) and empirical modelling of the performance of a number of cyclone samplers," J. Aerosol Sci. **27**, 281-304, 1996.

P. E. Kloeden and E. Platen, *Numerical Solution of Stochastic Differential Equations*, (Berlin, Sprinler Verlag, 1992).

J.P. Minier "Closure proposals for the Langevin Equation Model in Lagrangian two-phase flow modelling", to be presented at the ASME Gas-Solid Flow Conference, San Francisco, July 18-22, 1999.

J.P. Minier, "Lagrangian Stochastic Modelling of Turbulent Flows", Lectures Notes of the Von Karman Institute, Session on *Advances in Turbulence Modelling*, 23-27 March 1998.

J.P. Minier and D. Talay, "Weak second-order schemes for Stochastic Differential Equations used in PDF Modelling and in two-phase flow simulations", in preparation, (1999).

S. B. Pope, "Lagrangian PDF methods for turbulent flows," Annu. Rev. Fluid Mech., 26:23–63, 1994.

Pozorski J. and Minier J.P., "On the Lagrangian Turbulent Dispersion Models Based on the Langevin Equation", Int. J. Multiphase Flow 24 (1998), 913-945.

Pozorski J. and Minier J.P., "Probability Density Function Modelling of Dispersed two-phase Turbulent Flows", Phys. Rev. E, 24(1), 855-863 (1999).

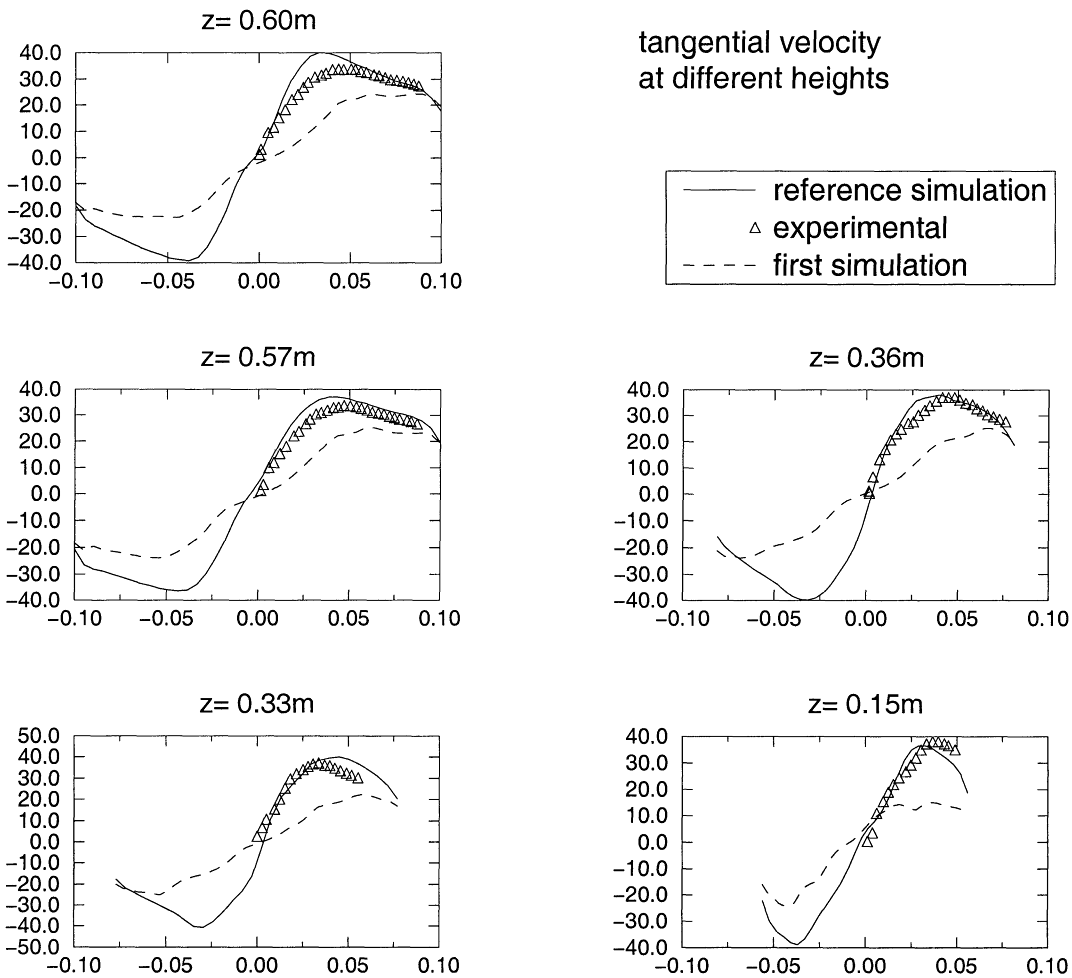

Figure 1: Comparison between computed gas tangential velocities and experimental measurements. Two computational results are plotted, corresponding to two different numerical treatments of the convective part of the gas equations.

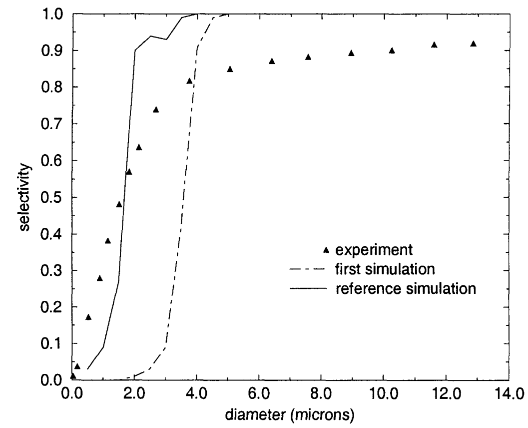

Figure 2: Comparison between computed efficiencies and experimental measurements for the $20cm$ diameter Stairmand cyclone. The two curves have been obtained using mean gas velocities shown in Fig. 1.

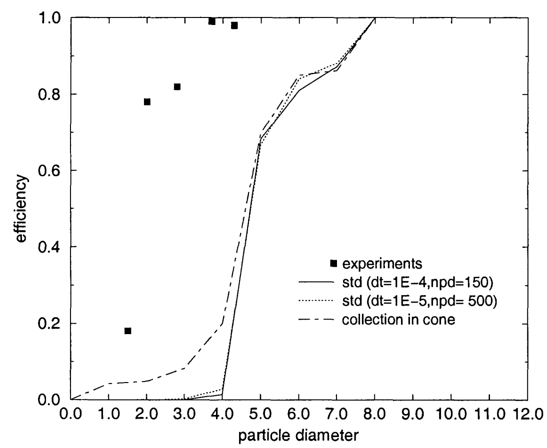

Figure 3: Comparison between computed efficiencies and experimental measurements of Dirgo and Leith. Three curves have been computed for different numerical parameter (time step dt and number of particles per class npd) and different boundary conditions.