3D CFD MODELLING OF AEROCYCLONES OPERATING AT HIGH TEMPERATURE

Jolius Gimbun*[1], T. G. Chuah[2], Thomas S. Y. Choong[2], A. Fakhru'l-Razi[2]

[1] Faculty of Chemical and Natural Resources Engineering, University Collage of Engineering and Technology Malaysia, KUKTEM, Locked Bag 12, 25000 Kuantan, Pahang Darul Makmur, Malaysia.
[2] Department of Chemical and Environmental Engineering, Faculty of Engineering, Universiti Putra Malaysia 43400 UPM Serdang, Selangor D. E., Malaysia.
*Corresponding author. Phone: +609-549 2413, Fax: +609-549 2399
Email: jolius@kuktem.edu.my

ABSTRACT

The design of aerocyclones operating at high temperatures is still a difficult problem still practical experience shows that the pressure drop and collection efficiency really obtained is mostly different than that calculated. This work presents a Computational Fluid Dynamics calculation to predict and to evaluate the flow field, pressure drop and collection efficiency of aerocyclones under extreme operating temperature. The numerical solutions commercial CFD code FLUENT 6.1. All the predictions proved to be satisfactory when compared with the presented experimental data. The CFD simulations predict excellently the cyclone cut-off size and pressure drop for all operating conditions with a deviation about 5% from the experimental data respectively. Results obtained from the computer modelling exercise have demonstrated that CFD and is a reliable method of modelling the aerocyclones performance. It has also been shown to be useful for examining the effects of a number of design changes have on their performance. This method of analysis is almost certainly less expensive than experiment, and represents a cost-effective route for design optimisation.

Keywords: Cyclone; Temperature; CFD; Efficiency; Pressure drop; Solid-gas separation

1 INTRODUCTION

Cyclone separator is one of the most widely used devices to separate and recover the industrial dusts from air or process gases. Their simple design, low capital cost and nearly maintenance-free operation make them ideal for use as pre-cleaners for more expensive final control devices such as baghouses or electrostatic precipitators. Aerocyclones are particularly well suited for high temperature and pressure conditions because of their rugged design and flexible components materials. Application in extreme condition includes the removing of dust in power generation plant, incineration plant, and the use as a spray dryer or gasification reactor.

Engineers are generally interested in two parameters in order to carry out an assessment of the design and performance of a cyclone. These parameters are the collection efficiency of particle and pressure drop through the cyclone. An accurate prediction of cyclone pressure drop is very important because it relates directly to operating costs. In other hand, the prediction of cyclone efficiency is important because an inaccuracy in cyclone efficiency prediction may result in the design of an inefficient cyclone. CFD has a great potential to predict the flow field characteristics and particle trajectories inside the cyclone as well as the pressure drop (Griffiths and Boysan, 1996; Gimbun et al., 2004; 2005). The complicated swirling turbulent flow in a cyclone places great demands on the numerical techniques and the turbulence models employed in the CFD codes when modelling the cyclone pressure drop.

This article presents an application of computational fluid dynamics, FLUENT 6.1 CFD, in the prediction of aerocyclones flow field and performance under extreme operating condition. The CFD simulations results are then are compared with experimental data found in the literature for different inlet flow rates and temperatures. In this study, the CFD calculations are carried out using commercial finite volume code FLUENT 6.1.

Many different types of cyclones have been built but the reverse flow cyclone with tangential inlet in Figure 1 is most often used for industrial gas cleaning (Li and Wang, 1989; Altmeyer *et al.*, 2004). In this study, the numerical simulation deals with the reverse flow aerocyclone with a tangential rectangular inlet. The key geometrical features of this type of aerocyclone are identified in the figure.

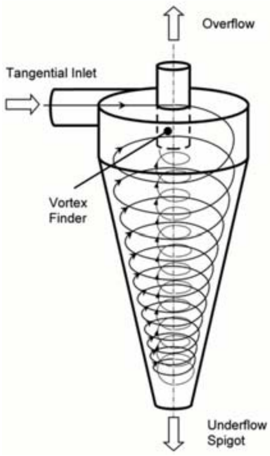

Figure 1: A conventional aerocyclone (Svarkovsky, 1984)

The inflowing fluid rotates within the main body of the chamber and is constrained to follow a spiral flow path. Any particles suspended within the fluid are subjected to an enhanced radial acceleration. Cyclones are commonly used when the density of the inflowing fluid (the carrier phase) is less than the particle phase. In cyclones of this type the larger particles migrate outwards to the cone wall where they travel in a downward spiral to the base of the chamber and exit at the underflow. The smaller particles migrate more slowly and therefore their distribution across the flow changes little. Those in the centre are captured in the upward flow and spiral upward and out through the vortex finder, see Figure 1. The remainder is discharged with the coarse fraction at the underflow or spigot.

2 CFD APPROACH

The Fluent software version 6.1.22 was used for the purpose of the CFD simulations. The cyclone chamber was represented using a relatively fine structured hexahedral cooper mesh that consisted of 20846 nodes (Fig. 2).

A velocity inlet boundary was used to specify an air inlet velocity ranged from 4.62 to 14.16 m/s. The simulations were carried out with a zero underflow component. The underflow was therefore represented using a wall boundary. An outflow boundary condition was used to represent the cyclone overflow. The air properties such as viscosity and density at predefined temperature are manually inserted into a Fluent solver.

The finite volume methods have been used to discretised the partial differential equations of the model using the SIMPLE method for pressure-velocity coupling and the second order scheme to interpolate the variables on the surface of the control volume. The segregated solution algorithm was selected. The Reynolds stress (RSM) turbulence model was used in this model due to the anisotropic nature of the turbulence in cyclones. Standard Fluent wall functions, after Launder and Spalding (1974), were applied and high order discretisation schemes were also used.

Having obtained the solution for the fluid flow, the particulate flow simulation has been performed by numerically tracing the motion of the particles in a Largrange frame of reference. The CFD simulation was performed with a Pentium IV 2.8 GHz HP workstation XW8000 with 512 cache-memory, 1 GB RAM-memory, and 110 GB hard-disc memory. Computational results are then compared with the experimental data obtained from the literature.

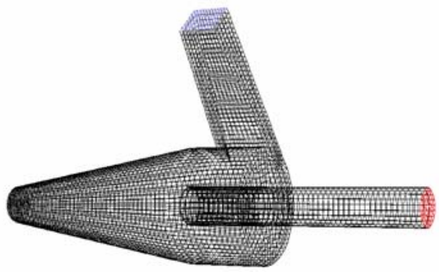

Fig. 2: The CFD computational mesh of the cyclone

3 RESULTS COMPARISON

The results of the CFD analysis are presented in direct comparison with the experimental data in Figures 3 and 5. It is shown that CFD with RSM turbulence model predicts the cyclone performance pretty well within the domain of condition studied. The direct plot of static pressure contour inside the cyclone is presented in Figure 4. The low-pressure centre in Figure 5 can be responsible for the flow reversion and deviation of the axial velocity peak to the wall of the vortex finder pipe (Gimbun, 2005). The particle tracking has been carried out via discrete phase modeling (DPM) available FLUENT code. The trajectories of 1.5 μm particle are shown in Figure 6.

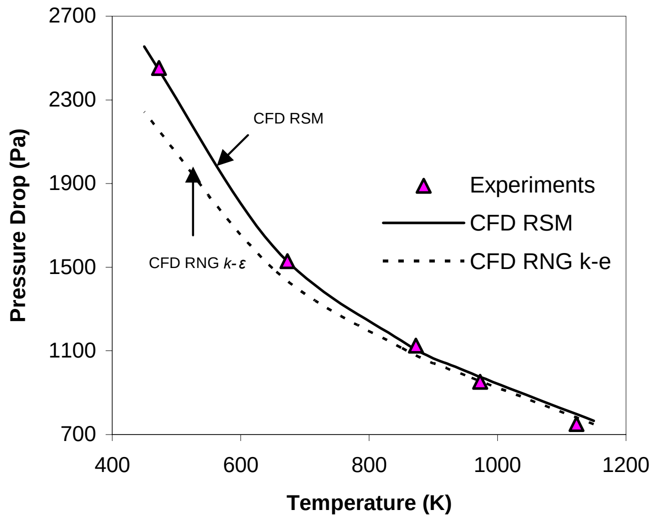

Fig. 3: Evaluation of cyclone pressure drop with temperature. Comparison between data presented by Bohnet (1995) and the CFD predictions (v = 20.08 m/s).

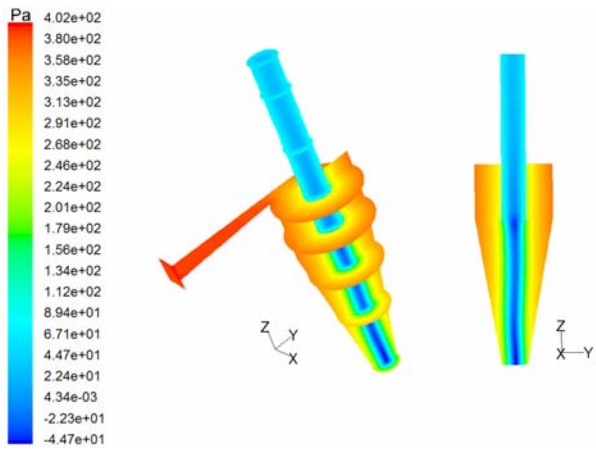

Fig. 4: 3D Map of static pressure of Bohnet (1995) cyclone at temperature 850 K.

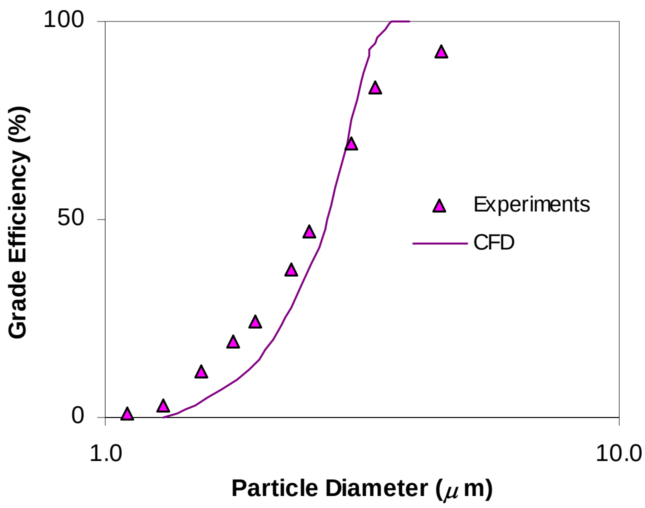

Fig. 5: Separation efficiency of Bohnet (1995) cyclone at high temperature (P = 1 Bar, T = 873 K, v_i = 8.61 m/s). Data point from Bohnet (1995).

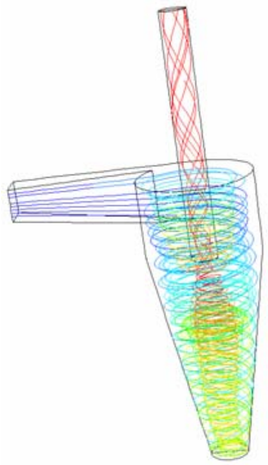

Fig. 6: Particle trajectories of Bohnet cyclone at 873 K (d_p = 1.5 μm)

4 CONCLUSIONS

A comparison of the experimental data with the results of the CFD analysis has shown good agreement. In each case, the shape of the aerocyclone collection efficiency and pressure drop was accurately predicted with an average deviation of 5%, which probably in the same magnitude of the experimental error. It was demonstrated that only a marginal improvement in the accuracy of the predicted collection efficiency and pressure drop was achieved when the same cyclone was modelled using an RNG k-ε turbulence model on a same computational mesh. It is concluded therefore that the collection efficiency and pressure drop within a cyclone chamber at extremely high temperature can be accurately modelled using CFD.

REFERENCES

S. Altmeyer, V. Mathieu, S. Jullemier, P. Contal, N. Midoux, S. Rode, J.-P. Lecrerc, (2004). Comparison of different models of cyclone prediction performance for various operating conditions using a general software, *Chem. Eng. Process*. **43**: 511-522

J. Gimbun, Thomas S. Y. Choong, T. G. Chuah, A. Fakhru'l-Razi. (2004) A CFD study on the prediction of cyclone collection efficiency, *Int. Journal of Computational Engineering Science* (in press).

J. Gimbun, T. G. Chuah, A. Fakhru'l-Razi and Thomas S. Y. Choong. (2005) The influence of temperature and inlet velocity on cyclone pressure drop: a CFD study, *Chemical Engineering & Processing* 44(1): 7-12.

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

B. E. Launder and D. B. Spalding, (1974). The Numerical Computation of Turbulent Flows. *Computer Methods in Applied Mechanics and Engineering*, **3**:269-289

Li Enliang and Wang Yingmin, (1989). A new collection theory of cyclone separators, *AIChE Journal* **35**: 666-669

L. Svarkovsky, (1984). Hydrocyclones, Holt, Rinehart and Winston, London.