CFD ANALYSIS OF EFFICIENCY AND PRESSURE DROP IN A GAS-SOLID SQUARE CYCLONES SEPARATOR

NOVI SYLVIA[1]*, YUNARDI[2], ELWINA[3], WUSNAH[1], YAZID BINDAR[4]

[1]*Chemical Engineering Dept., University of Malikussaleh, Lhokseumawe, Indonesia*

*Email: nxsylvia@gmail.com

[2]*Chemical Engineering Dept., Syiah Kuala University, Banda Aceh, Indonesia*

[3]*Chemical EngineeringDept., State Polytechnics of Lhokseumawe, Lhokseumawe, Indonesia*

[4]*Energy and Processing System of Chemical Engineering Dept., Faculty of Industrial Technology, Bandung Institute of Technology, Bandung, Indonesia.*

Abstract.

In this paper, two small cyclones with the same hydraulic diameter and volume, which one is square and the other one is round (Lapple cyclone), are numerically compared. A pre-processor software GAMBIT was employed to set up the configuration, discretisation, and boundary conditions of the cyclone. The characteristics of the cyclone being studied was 0.2 m in diameter, receiving a gas flow rate of 0.1 m3/s with a particle mass loading of 0.01 kg/s. A commercial CFD code FLUENT 6.2.16 was employed to simulate the flow field and particle dynamics in the cyclone. The Reynolds averaged Navier–Stokes equations with Reynolds Stress Turbulence Model (RSTM) are solved by use of the finite volume method based on the SIMPLE pressure correction algorithm in the computational domain. The Eulerian–Lagrangian computational procedure is used to predict particles tracking in the cyclones. The velocity fluctuations are simulated using the Discrete Random Walk (DRW).The results show that collection efficiency of square cyclone is the better with increasing flow rate than round cyclone. The pressure drop in square cyclone is higher than the pressure drop in small round one.

Keywords: square cyclone; round cyclone; CFD; efficiency and pressure drop

Introduction

Cyclones are one of the most common equipment used for controlling dust emissions of gaseous flow in industrial processes. Although current engineering developments have enabled to employ cyclone for example as dryers and reactors, their main application remains in the area of air pollution control where high efficiencies are required to meet the stringent regulations. In comparison with other equipment used for air pollution control, cyclones are more preferable due to their simplicity in the design, inexpensiveness to manufacture, low maintenance costs, and adaptability to a wide range of operating conditions such as high temperature and pressure. Despite they are frequently used as final collectors where large particles to be removed, it has been also a common practice to employ cyclones as pre-cleaners for a more efficient collector such as an electrostatic precipitator, scrubber or fabric filter (Swamee, 2009).

There are four major parts to a cyclone, the inlet, the cyclone body, the dust discharge system, and the outlet all affect the overall efficiency of the cyclone. The principle of cyclone separation is simple: the flow of gas–solid mixture is directed into the cyclone through the inlet on the top section. Then, the cylindrical body induces a spinning, forcing particulate matter to the wall of cylinder. The gas continues down the cyclone body to the cone, which gives the gas enough rotational velocity to keep the particulates against the wall. At the bottom of the cone, the gas changes direction from downward to upward. The ascending vortex enters a tube extension that is sometimes called a vortex finder and exits the cyclone. Meanwhile, the collected particulate matter drops into a hopper, where it is periodically or continuously removed.

The main performance of a cyclone is primarily judged from its collection efficiency and pressure drop. In spite the fact that its design and operation are simple, the flow behavior and particle dynamics inside the cyclone are complicated, requiring efficient mathematical models to provide accurate predictions of the efficiency and pressure drop for the purpose of design and operation of a cyclone. Numerous semi-empirical models leading to the predictions of collection efficiency and pressure drop have been developed by many investigators. Leith

(1990) summarized a number of these models, including those developed by Stairmand (1951), Barth (1956), Shepherd and Lapple (1939), Lapple (1951), Leith and Licht (1972). Although the majority of the empirical models were developed based on the experimental data of particular cyclone geometry, a substantial error between the prediction and measured data in the cyclone efficiency is evidence due the use of different assumptions and geometry conditions. As a consequence, the use of semi-empirical models has limitation in the prediction of cyclone performance.

Numerical methods are therefore proposed to model the flow field and particle dynamics of these devices for the purpose of predicting the collection efficiency and pressure drop. After the first numerical simulation of cyclone using computational fluid dynamics (CFD) technique (Boysan et al, 1982), a number of CFD modeling works on the cyclone performances have been performed. Elsayed and Lacor (2010) optimized the cyclone geometry aiming at obtaining minimum pressure drop with the aid of response surface methodology. They used Reynolds Stress Model to represent the flow field inside the cyclone and the results showed that the most significant geometrical parameters are the vortex finder diameter, the inlet section width, the inlet section height and the cyclone total height. Shafikhani et al (2011) analyzes two small cyclones with the same hydraulic diameter, which one is square and the other one is round, are numerically compared. Obtaining results about pressure drop in small square cyclones is less than the pressure drop in small round ones and with increase in flow rate this difference is distinguished, but collection efficiency of small square cyclone is less than round one, but by increasing flow rate this difference decreases. In this paper, the characteristics of two small cyclones with the same hydraulic diameter and volume, which one is square and the other one is round (Lapple cyclone), are numerically compared. The objective of this paper is to present the results of evaluation of the performance cyclone about efficiency and pressure drop.

Turbulence Models Description

Turbulence theories, simulation and modelling have always been important subjects in fluid dynamics and engineering, descriptions of different turbulence approaches can be found in various computational fluid dynamics textbooks. Any modelling technique involves a number of descriptive equations whose solution needs to be obtained numerically. In general, with regards to turbulence prediction alone, three main classes of numerical simulations are currently being developed: (i) direct numerical simulation (DNS); (ii) large eddy simulation (LES); and (iii) Reynolds averaged Navier-Stokes (RANS) approaches.

The DNS of turbulent flows essentially involves a full numerical solution of the time dependent Navier-Stokes equations and accommodates all time and length scales of turbulence. From the conceptual point of view, it is fundamentally the simplest method to implement, since no turbulence modelling is needed. In DNS, all of the turbulent motions are resolved in the computational model from the largest scale to the smallest scale of turbulent eddy. As a consequence, the computational domain should be large enough to contain the largest eddies, and the grid spacing should be fine enough to resolve the smallest eddies. Therefore, it is extremely expensive to simulate even the simplest types of flow (e.g. homogeneous turbulence), primarily due to the refined grid required to resolve the small-scale turbulence structures, as well as the small time-steps required for the time-scales of the smallest eddies. In the Reynolds-averaged Navier- Stokes (RANS) approach, instead of directly solving for the turbulence field, solutions are obtained by solving time-averaged transport equations. The approach models all scales and solves the governing time-averaged equations which introduce unknown apparent stresses known as the Reynolds stresses. This adds a second-order tensor of unknowns for which various models can provide different levels of closure. Basically, two distinct types of RANS model have been developed: first-moment closure models and second- moment closure models. In the former, the unknown Reynolds stresses are reduced by correlation with the first-moment. The second moment closure models approximate the higher-order moments (i.e. the triple fluctuating velocity correlations) by second-moment terms, and solve transport equations for the Reynolds stresses directly. As a consequence of modeling the unknown terms, RANS turbulence models like Spalart-Allmaras, standard k-ε model, RNG k-ε model, standard k-ω model, and Reynolds Stress Model (RSM) are capable of producing much faster computation in comparison to those of LES and DNS. This is the reason to use such models in the present investigation.

Numerical Computation

All transport equations presented in the turbulence model description above are numerically solved using a commercial CFD code, Fluent 6.2.16 (Fluent Inc., 2005) . Control volume approach was used to discretize the transport equations. The SIMPLE algorithm was used to solve pressure-velocity coupling and first-order and second-order interpolation schemes for turbulent kinetic energy and momentum equations, respectively. Flows inside the cyclone was assumed to be in steady state. The numerical computation was carried out with an accuracy of 10-3 for the entire flow field parameters.

Computational Domain and Boundary Conditions

Fig. 1 illustrates a square cyclone, having a diameter, D of 0.2 m employed in this study with the ratio of geometric parameters is shown in Table 1, where *W* inlet width, *H* inlet height, D_e outlet diameter, *S* outlet height, L_b cylinder height, *Lc* cone height and D_d dust outlet diameter. On Fig. 1, the cyclone geometry drawn using GAMBIT code was set up with boundary conditions. Information on material data for the cyclone computation is presented in Table 2. It should be noted here that these data are similar to the experimental data presented Wang et al (2005)

Table 1. Cyclone geometry used in this study (D=0.2 m)

a/D	b/D	De/D	S/D	h/D	H/D	B/D
0.25	0.5	0.5	0.625	2.0	4.0	0.25

Table 2 Material data used as input of the cyclone calculation

Temperature of air flow	25 ^{0}C
Min. diameter of particle	5 µm
Max. diameter of particle	200 µm
Mean diameter of particle	29.90 µm
Spread parameter	0.806
Ash density	3320 kg/m^3
Air density	1.225 kg/m^3

Flow field Calculation and Validation

Flow field computation was carried out using FLUENT 6.2.16 with Reynold Stress Turbulence Model (RSTM). Computations were performed using a laptop computer of 4.00 GB RAM having a 32 bit operating system. The validation of the predictions was performed by comparing with the experimental data reported by Wang et al (2005) who used a Lapple cyclone. Experimental measurements were presented in the forms of efficiency and pressure drop.

Prediction and Validation of efficiency and pressure drop in the Cyclone

Fig. 2 shows comparison of predicted efficiency and experimental data. Both experimental data and predictions show that are mostly below the experimental data. Efficiency cyclone as increase in rising velocity . The performance shown by the square cyclone in the prediction of efficiency is more superior to Lapple cyclone (Wang experiment) in this study. Although the predictions of efficiency in the round cyclone follow the trend of experimental data. Efficiency which has been proved in latter researches, is extremely depend on flow rate and at every flow rate the round cyclone has more efficiency than that of square one. This problem is less seen at high flow rates and according to less pressure drop of square cyclone, a square cyclone could be a better choice at high flow rates.

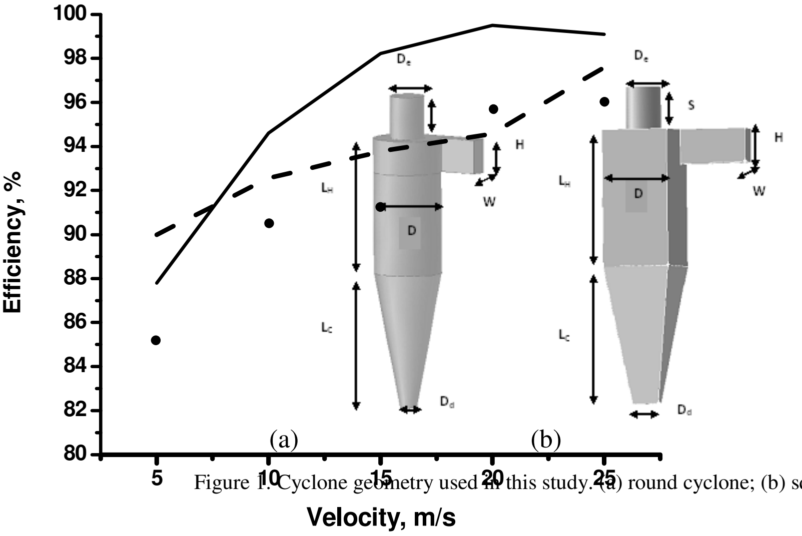

Figure 1. Cyclone geometry used in this study. (a) round cyclone; (b) square cyclone

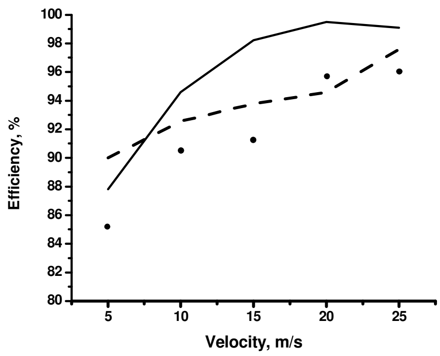

Fig. 2. Comparison between efficiency predictions and experimental data (symbols: experimental data; line: results predictions – round cyclone; – – square cyclone)

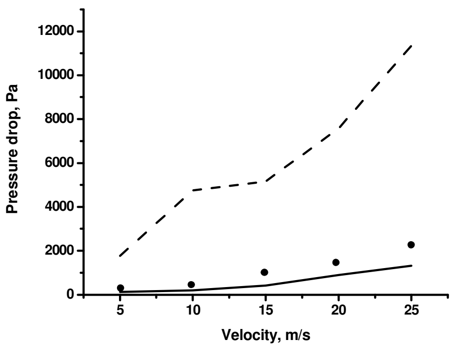

Fig. 3. Comparison between pressure drop predictions and experimental data (symbols: experimental data; line: results predictions – round cyclone; – – square cyclone)

Figure 3 presents a comparison of pressure drop predictions by round and square cyclones with experimental data. The results shown that there was no significant differences among predictions of round and square cyclones but round cyclone is the less pressure drop than square cyclone.

Conclusions

1. Obtained results show that CFD is a powerful tool for the study of the flow in cyclones
2. Round cyclone is the less pressure drop than square cyclone.
3. Efficiency which has been proved in latter researches, is extremely depend on flow rate and at every flow rate the round cyclone has more efficiency than that of square one.

4. Collection efficiency of square cyclone is the better with increasing flow rate than round cyclone
5. Instead of same volume, the other criterions like optimization square cyclone can be used for compare the characteristics of round and square cyclones for future studies.

References

1 W. Barth, "Design and layout of the cyclone separator on the basis of new investigations," Brenn.Warme Kraft, vol **8**, 1(1956).

2 F. Boysan, W.H. Ayers, and J.A. Swithenbank, "Fundamental mathematical modeling approach to cyclone design," *Trans IChemE*, vol **60**, 222 (1982).

3 K. Elsayed and C. Lacor, "Optimization of the cyclone separator geometry for minimum pressure drop using mathematical models and CFD simulations," Chem Eng Sci, vol **65**, 6048 (2010).

4 FLUENT 6.2.16, User's Guide. FLUENT Incoparated, (2005).

5 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*., vol **2**, 281(1996).

6 C.E. Lapple, "Process uses many collector types," *Chem.Eng*., vol **58**, 144 (1951).

8 D. Leith and W. Licht, "The collection efficiency of cyclone type particle collectors," *AIChE Symp,* vol **68**, 196 (1972).

9 D. Leith, Cyclone performance and optimization. Technical Report, Department of Environmental Sciences and Engineering, University of North Carolina.Chapel Hill, USA (1990).

10 C.J. Stairmand, "The design and performance of cyclone separators," *Trans IChemE*, vol **29**, 356 (1951).

11 C.B. Shepherd and C.E. Lapple, "Flow pattern and pressure drop in cyclone dust collectors". *Ind. Eng. Chem.*, vol **31**, 972 (1939).

12 P.K. Swamee, N. Aggarwal, and K. Bhobhiya, "Optimum design of cyclone separator". *AIChE*, vol **55**, 2279 (2009).

13. H. Shafikhani, M. Shams, and S. Dashti, "Numerical simulation of square cyclones in small sizes," *Advanced Powder Technology*, vol **22,** 359(2011).

14 B. Wang, D.L. Xu, G.X. Xiao, and A.B. Yu, "Numerical study of gas-solid flow in a Siklon," *Appl. Math. Model,* vol **30**, 1326 (2005).