Analysis of turbulence models performance for the predictions of flow yield, efficiency, and pressure drop of a gas-solid cyclone separator

[1]Novi Sylvia, [2]Yunardi, [3]Ilham Maulana, [4]Elwina, [1]Wusnah and [5]Yazid Bindar

[1] Chemical Engineering Dept., University of Malikussaleh, Lhokseumawe, Indonesia; [2]Chemical Engineering Dept., Syiah Kuala University, Banda Aceh, Indonesia; [3] Mechanical Engineering Dept., Syiah Kuala University, Banda Aceh, Indonesia; [4] Chemical Engineering Dept., State Polytechnics of Lhokseumawe, Lhokseumawe, Indonesia; [5] Energy and Processing System of Chemical Engineering Dept., Faculty of Industrial Technology, Bandung Institute of Technology, Bandung, Indonesia. Corresponding Author: yunardi@unsyiah.ac.id

Abstract. This paper presents the results obtained from the application of computational fluid dynamics (CFD) to modelling the flow field of a Lapple cyclone and to optimizing the cyclone based upon its geometrical parameters. 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/m3. A commercial CFD code FLUENT 6.2.16 was employed to simulate the flow field and particle dynamics in the cyclone. The objective of this research was to investigate the performance of a number of turbulence models on the prediction of the flow field, collection efficiency and pressure drop in the Lapple cyclone. A number of five turbulence models under Reynolds Averaged Navier Stokes (RANS) category, including Spallart-Allmaras, standard k-ε model, RNG k-ε model, standard k-ω model, and Reynolds Stress Model (RSM) were examined in the simulation of the flow field and particle dynamics inside the cyclone. A validation of all calculation was performed by comparing the predicted results in terms of axial and tangential velocities, efficiency and pressure drop against experimental data of a Lapple cyclone taken from literature. The results of the investigation show that out of five turbulence models being tested, the RSM presented the best predicted results. The predictions of axial and tangential velocities as well as cyclone efficiency by this model are in excellent agreement with the experimental data. Although the pressure drop in the cyclone is under-predicted, the RSM predictions are far better than those of other model. Other turbulence models are over-predicted and under-predicted the axial and tangential velocity, respectively. With respect to efficiency and pressure drop of the cyclone, other models are capable of following the trend of the experimental data but they failed to agree with the experimental values. These results suggest that the RSM is the most suitable turbulence model to represent the flow field and particle dynamics inside a cyclone gas-solid separator.

Key words: cyclone, computational fluid dynamics, turbulence model, efficiency, 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 al.l 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. Zhou and Soo (1990) applied two-equation turbulence model, k-ε standard for the modeling of gas solid flow and collection of solid in a cyclone separator. They found out that the k-ε standard is capable of providing good predictions with respect to axial velocity; however, it fails to simulate strongly swirling flow near the axis. Hoekstra et al. (1990) evaluated the performance of the k-ε model, the RNG-k-ε model, and the Reynolds stress transport model (RSTM) in predicting the gas flow field in a cyclone separator. They found out that the RSTM demonstrated reasonable predictions in terms both axial and tangential velocities inside the cyclone. On the contrary, the other two models were considered not suitable for the predictions of cyclonic flow. However, it is important to note here that the RNG-k-ε model could predict well the pressure drop in three type of cyclone sampler (Griffiths and Boysan, 1996). This finding suggests that any simpler turbulence model does not always results in poor prediction, as also demonstrated by Suyitno (2005) when one-equation turbulence model, Spallart-Allmaras, was found to give better predictions than that of the RNG-k-ε model in a study of cyclone performance. On this basis, it is necessary to investigate the current commonly used turbulence models in an attempt to have a better understanding on the performance of each model in the simulation of flow and pressure fields as well as efficiency of a cyclone.

The objective of this paper is to present the results of evaluation of the performance of five turbulence models under Reynolds Averaged Navier Stokes (RANS) category, namely Spallart-Allmaras, standard k-ε model, RNG k-ε model, standard k-ω model, and Reynolds Stress Model (RSM) in predicting flow and pressure fields as well as efficiency in a Lapple cyclone separator. Validations of predictions are made by comparing the predictions resulted by each model with experimental data taken from Wang et al. (2005).

Methods

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

Figure 1 on the left illustrates a Lapple cyclone, having a diameter of 0.2 m employed in this study with the ratio of geometric parameters is shown in Table 1. On the right of Figure 1, the cyclone geometry drawn using GAMBIT code was set up with boundary conditions. Initially, the geometry of the cyclones has a number of 57,000 cells. However, during computation in FLUENT, a mesh adaptation was performed allowing the increase of the number of cells of around 30% of the initial state. 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 by 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

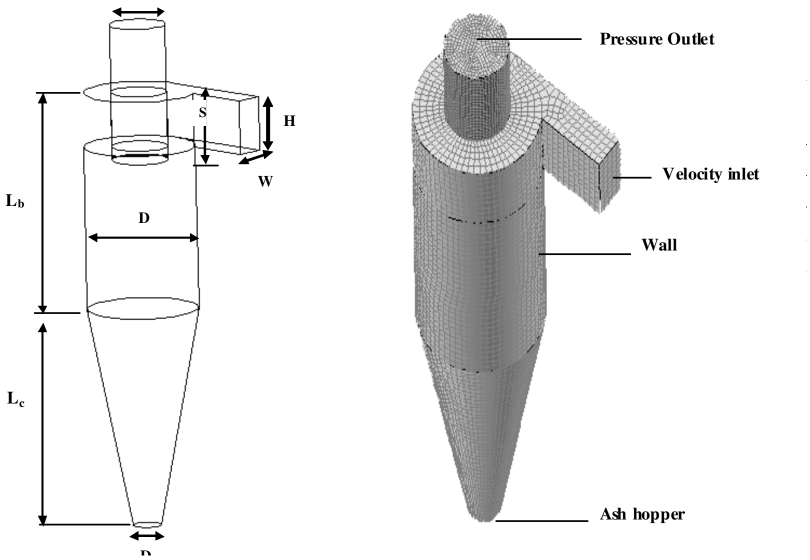

Figure 1. Cyclone geometry used in this study

Flow Field Calculation and Validation

Flow field computation was carried out using FLUENT 6.3 with varying turbulence model as listed in Table 3. All constant used in each model were those of default without any adjustment. Computation were performed using a laptop of 2.00 GB RAM having a 2.13 GHz of speed. The computation time for each run highly depended on the turbulence model used. However, in general each run required a computation time from 4 to 24 hours.

Table 3. Turbulence model were tested in this study

Classification of model	Derivative Model	Tested model
RANS	One equation model	Spalart Allmaras
	Two equation model	1. k-ε standar 2. RNG k-ε 3. k-omega
	Reynold Stress Model	RSM

The validation of the predictions was performed by comparing with the experimental data reported by Wang et al. (2005). In this experiment, Wang et al. (2005) used a Lapple cyclone model. Experimental measurements were presented in the forms of flow tangential and axial velocities, pressure drop and efficiency.

Results and Discussion

Prediction and Validation of Flow Field in Cyclones

Figure 2 shows comparison of predicted axial velocity by various turbulence models and experimental data. Both experimental data and predictions show that the axial velocity reached the peak in the center of the cyclone having the lowest values at the positions near the wall of the cyclone. All turbulence models qualitatively produce the similar trend, following the evolution of experimental data. However, among five turbulence models tested in this study, Reynolds Stress turbulence model (RSTM) produced more satisfactory prediction. The RSTM Predicted axial velocity both in the área near the wall and in the core of the cyclone was in close agreement with experimental data. Both qualitatively and quantitatively, the performance shown by the RSTM in the prediction of axial velocity in cyclone is more superior tan those of other turbulence models tested in this study. Although the predictions of other models follow the trend of experimental data, turbulence models such as the k-ε standard, RNG k-ε, k-ω and Spalart Allmaras produced a substantial deviation in peak area. The main reason for the accuracy of the RSTM predictions is most likely due to its performance to predict a complex flow involving swirling and vortex as in the case of cyclone.

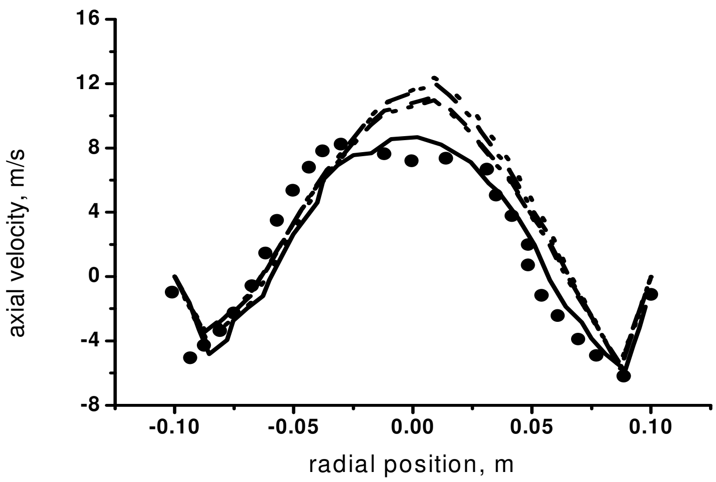

Figure 2. Comparison between axial velocity predictions with experimental data (symbols = experimental data; Line = Prediction results; ; – RSM; – – k-ε; ···· k-ε RNG; – · – k-ω standar; – ·· – Spalart Allmaras)

Figure 3 shows a comparison of predicted tangential velocity by various turbulence models and experimental data. Tangential velocity profile along the radial position inside the cyclone is different from the axial velocity profile. The axial velocity forms a single peak at the core of the cyclone, while the tangential velocity profile forms two peaks in the left and right of the centerline, with the minimum peak occurs in the core of the cyclone. With regard to the prediction of tangential velocity by various turbulence models, all turbulence models quantitatively were unable to capture the evolution of the experimental data. Although qualitatively, predictions by RSTM is superior than those of other turbulence models, it has not been able to capture the tendency of the tangential velocity data as its ability to capture the trend of the axial velocity data. It should be noted here that all models were run on the basis of the default of Fluent code, without adjusting any constant in the model. Therefore, the improvement of the prediction, particularly by the RSTM could be obtained by adjusting the empirical constants associated with spreading rate.

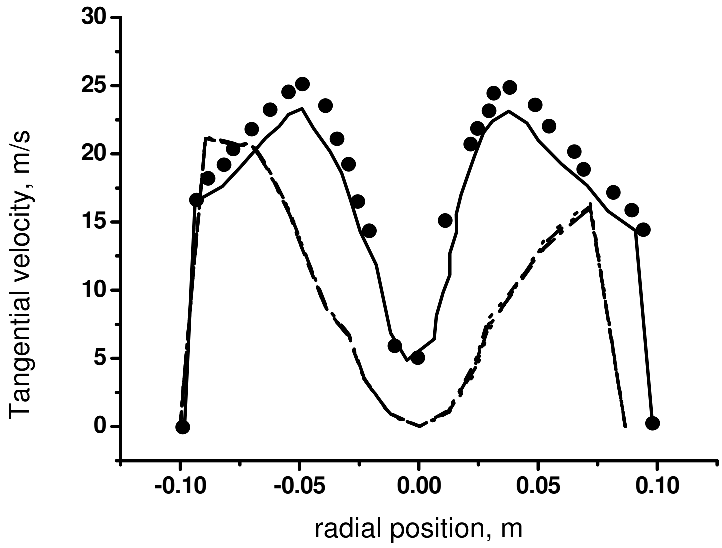

Figure 3. Comparison between the predictions of tangential velocity with experimental data (symbols = experimental data; Line = Prediction results; – RSM; – – k-ε; ···· k-ε RNG; – · – k-ω standar; – ·· – Spalart Allmaras)

Figure 4 illustrated axial velocity and tangential velocity contours in the left and right, respectively in the cyclone. Inspection the left figure, it is clear that low axial velocity dominated in the region closer to the wall. As the axial velocities have minus value, meaning the direction of the flow is downward. This means that the majority of the particles are moving downward, although small percentage of particles are moving upward as indicated by positive values of axial velocity in the core of the cyclone. Turning to the right figure, the tangential velocities of between 13 and 17 m/s dominated the flow field in the cyclone. It is this velocity that forced particles towards the wall went spinning downward. In contrast to the left figure, the tangential velocity at the center-line has very low values, indicating that the tangential velocity dominates the flow field at outside of the core regions of the cyclone, while the axial velocity dominates the flow field in the core of the cyclone.

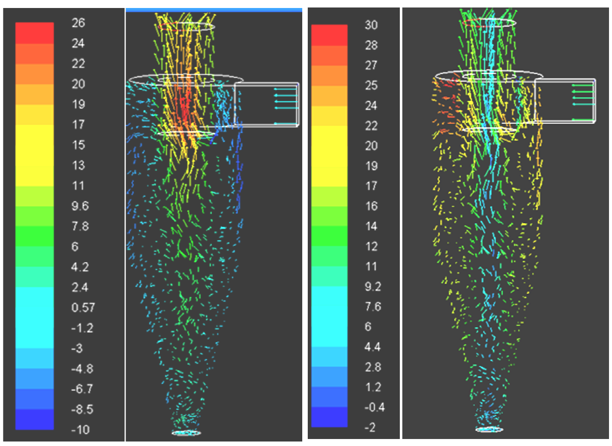

Figure 4. Axial velocity (left) and tangential velocity contours (right) in the cyclone

Prediction and Validation of the Efficiency and Pressure Drop

Figure 5 shows a comparison of efficiency prediction by various turbulence models against experimental data on the basis of gas entrance velocity into the cyclone. Predictions generated by RSTM is much better than those of other predictions, as they are closer with the experimental data. Efficiency predictions by other turbulence models are mostly below the experimental data, with the predictions generated by standar k-ω represents the most unsatisfactory. This is understandable as k-ω model is not prepared to predict swirling flow as occurs in the cyclone.

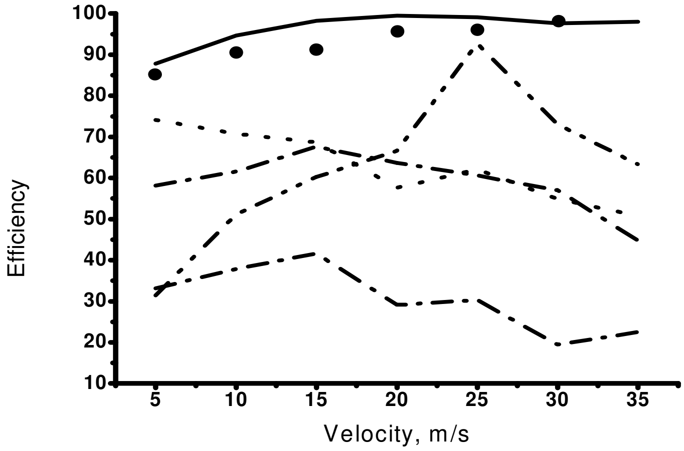

Figure 5. Comparison between efficiency predictions and experimental data (symbols = experimental data; Line = Results predictions– RSM; – – k-ε; ···· k-ε RNG; – · – k-ω standar; – ·· – Spalart Allmaras)

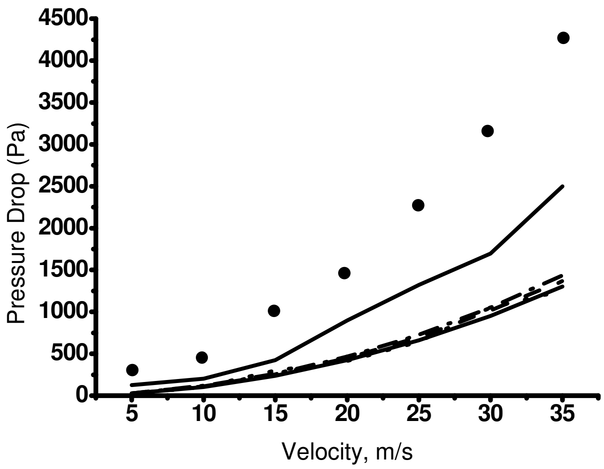

Figure 6. Comparison between pressure drop predictions and experimental data (symbols = experimental data; Line = Results predictions; – RSM; – – k-ε; ···· k-ε RNG; – · – k-ω standar; – ·· – Spalart Allmaras)

Figure 6 presents a comparison of pressure drop predictions by various models of turbulence with experimental data. The results shown that there was no significant differences among predictions of one turbulence model to others, both qualitatively and quantitatively, with the exception to the predictions generated by RSTM. All models qualitatively produced a similar trend to the experimental data. However, quantitatively, all models could not provide a satisfactory performance when compared to the experimental data. From all turbulence models tested, only the RSTM that produced better predictions than those of others, although they under-predicted experimental data.

Conclusions

From the results and discussions, the following conclusions are drawn:

1. Predictions of flow fields are significantly influenced by the performance of a turbulence model to represent the vortex and swirling flows that occur in the cyclone. Among five turbulence models being tested in this study, only the RSM turbulence model is capable of producing reliable predictions of the flow field since the model was prepared to predict a complex flow.

2. Results of the simulation shows that the tangential velocity governs the flow field outside of the center line of the cyclone, forcing the particle towards to the wall and causing the particles to fall towards to the cones region for collection. On the other hand, the axial velocity dominates the flow field in the center line of cyclones, especially closer to the outlet pipe, even though the values are smaller than those of tangential velocity.
3. Better flow field predictions by RSM model provides positive impact on the predictions of collection efficiency and pressure drop that occurs in the cyclone. Calculated efficiency by the standard k-ε, RNG k-ε, standard k-ω and Spalart Allmaras models were under-predicted, but the predictions generated by the RSM model are in closer agreement to the experimental data.

References

Barth, W. 1956, Design and layout of the cyclone separator on the basis of new investigations. Brenn.Warme Kraft, 8 :1–9.

Boysan, F., Ayers, W.H., Swithenbank, J.A. 1982. Fundamental mathematical modeling approach to cyclone design. Trans IChemE, 60: 222-230.

Elsayed, K., Lacor, C. 2010. Optimization of the cyclone separator geometry for minimum pressure drop using mathematical models and CFD simulations. Chem Eng Sci*,* 65: 6048–6058.

Fluent 6.2.16. 2005. User's Guide. FLUENT Incoparated.

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

Hoekstra, A. J., Derksen, J. J., Van den Akker, H. E. A. 1999. An experimental and numerical study of turbulent swirling fow in gas cyclones.Chem Eng Sci, 54: 2055-2065.

Lapple, C.E. 1951. Process uses many collector types. Chem.Eng*,* 58:144-151.

Leith, D., Licht, W. 1972. The collection efficiency of cyclone type particle collectors. AIChE Symp*,* 68: 196–206.

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

Stairmand, C. J. 1951. The design and performance of cyclone separators. Trans IChemE, 29: 356-383.

Shepherd, C.B., Lapple, C.E. 1939. Flow pattern and pressure drop in cyclone dust collectors. Industrial and Engineering Chemistry. 31(8): 972-984

Swamee, P.K., Aggarwal, N., Bhobhiya, K. 2009. Optimum design of cyclone separator. AIChE, 55: 2279–2283.

Suyitno. 2005. CFD of analysis Performance cyclone with Spalart-Allmaras model and RNG k- ε. Media Mesin*,* 6:47-54.

Wang, B., Xu, D.L., Xiao, G. X., dan Yu, A.B. 2005. Numerical study of gas-solid flow in a Siklon**.** Appl Math Model*,* 30:1326-1342.

Zhou, L.X., Soo, S.L. 1990. Gas-solids flow and collection of solids in a cyclone separator. Powder Technol, 63: 45–53.