Swirling Stability of Fluid at High Velocity Flow in Pipe

Nor Amirah bt Che Mohamad Nor and Hanafiah bin Zainal Abidin,

Faculty of Chemical Engineering, Universiti Teknologi Mara

***Abstract* – This paper investigate the performance and flow of fluid inside a cyclone. A cyclone present a complex flow structure which gave different prediction depends on geometry and flow velocity. A numerical simulation is performed to simulate the flow of fluid. A three-dimension simulation of single phase fluid within cyclone has been carried out using Computational Fluid Dynamics (CFD) with FLUENT software. The solver predictions using k-ε model to predict the turbulent flow inside the cyclone. Patterns of flow in every parts of cyclone are observed in order to identify the performance of separation process. Comparisons between sweep surfaces are made based on vectors and streamlines. Velocity profile and swirling stability at the vortex core is observed. High velocity fluid as an inlet stream which is 10 m/s caused the forced vortex at the center of cyclone bigger in shape. It can be shown that high velocity region mostly occurred at the wall of cyclone and cone chamber experienced low velocity region.**

Keywords – ***cyclone, Computational Fluid Dynamics (CFD), turbulent flow, vortex core and velocity profile.***

I. INTRODUCTION

In chemical process industries, CFD often used to simulate flow inside cyclone or hydrocyclone. Moreover, research had studied the single phase hydrocyclone flow field using CFD simulation (Dlamini et al, 2005). The studied is about the flow structure and flow pattern for single phase in hydrocyclone. Plus, CFD models have been used to study the structure of swirling flow in cyclone. The limitation in analytical methods had been the problem. But flow engineers pursued CFD models based on Navier-Stokes equations. There are also research for simulation and validation of turbulent gas flow in cyclone by Stephens et al (2015). It is to investigate the effect of turbulence model in cyclone using CFD simulation. The research is about the investigation of swirling flow structure in hydrocyclones by Nowakowski, A.F. and Dyakowski (2003). Three different swirling angles were investigated in this study using simulation. The research on CFD simulation is widely expands in various industries.

Mostly research findings were to manipulate the characteristics of the vortex flow in cyclone in terms of the fluctuating velocity components for different flow-rates and at different sections of the model (Papoulias, 2015). In most of industries, the problem arise in various vortex apparatuses is crucial. This can be observed in equipment used for chemical reactions, heat transfer intensification and separators. It is important to get a better understanding about the idea to overcome the limitation of the analytical methods.

The main objective in this study was to investigate the swirling stability of fluids at high velocity flow. Fluids may come in different forms. Different forms of fluids will give different flow characteristic. High velocity flow usually exists in turbulent condition. Turbulent flow is difficult to handle compare to laminar flow. Next, the second objective was to study the velocity profile in cyclone using Computational Fluid Dynamics (CFD). Recently, computational fluid dynamic is popular in designing a system or equipment that involves flow. Last but not least, the other objective was to simulate the fluid flow at vortex core using k-ε model.

In this paper, it was focus on turbulence effects in fluid flow. The Reynolds number for turbulent flow is between 2300 and 4000. This is because the inertial forces tend to produce eddies in flow stabilities. The modelling for both flows is obviously different. Particles that enter the cyclone may come with different density. The lighter particles in gas stream have less inertia that can easily influenced by the circular vortex and travel up. These particles are then leaving at the top of the cyclone. As for larger particles, they are not easily influenced by the vortex because of their high density. They have more inertia compare to lighter particles.

Therefore, larger particles have some difficulty following the high-speed circular motion of the air in the cyclone. As the air continues to spin, these particles begin to separate by hitting the inner side of walls of the cyclone and sliding towards the bottom of the container into a collection hopper. But, due to high velocity flow inside the cyclone, it produced high swirl and very large curvature streamlines within the flow. It is a challenge to researcher to module the flow stability in cyclone.

CFD is a method that replacing PDE systems by a set of algebraic equations where it can be solved using digital computers. Computational fluid dynamics (CFD) had reached an extent where it can visualize the flows of a gas and liquid and how flows affect the system. In order to simulate the flows of fluid, it needs high-speed computers. Computer will perform all the calculations required based on the types of flow. Recently, research found that the CFD software had improves the accuracy of the simulation such as transonic or turbulent flows.

II. METHODOLOGY

A. Geometry

To perform the CFD simulation, there are three basic procedures that need to follow. First and foremost is pre-processing stage. In this stage, the geometry of desired domain is selected. In addition, mathematical model is required to determine the effect of the fluid flow. Suitable flow model is defined such as a viewpoint or reference frame. Then the selected geometry is divided or discretized. Discretization is a method of approximating the differential equations by a system of algebraic equations for the variables at some set of discrete locations in space and time (Bakker, A et al, 2001). Discretization methods that mainly used in simulation are finite volume (FV), finite element (FE) and finite differential (FD). In this research, finite volume is used because the flow involved high velocity. Finite volume method can handles high Reynolds number as turbulent flows. This method typically used Navier-Stokes partial differential equations as information to solve the simulation. The equation describes the properties of the moving fluid.

B. Meshing

The geometry of interest is further divided into a number of computational cells also known as meshing. Cells can be variety in shapes. It depends on the dimension used. For two-dimensional (2-D) problems, triangular and quadrilateral cells are usually used. For three-dimensional (3-D) problems, hexahedral, tetrahedral, pyramidal and prismatic cells are used. But the most applicable shape is triangular shape as it is more accurate. After the meshing is complete, boundary condition is chosen. Boundary condition is a property that needs to simulate in the system. Through this research, velocity is specified at inlets and outlets of the equipment. It is because the objective is to study the high velocity fluid in cyclone.

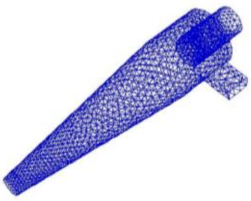


Figure 1 : Meshing sketch of cyclone model

C. Solver

The solving process can be proceeds after grid and boundary condition is created. The process is done by computer. There are many types of CFD software are introduced nowadays. ANYSY FLUENT is used in this paper. The solver will iterate the equations until convergence is achieved. The turbulence kinetic energy, *k* and its rate of dissipation, ε are obtained from the following transport equations,

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon-Y_M+S_k \qquad \text{(Eq 1)}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon)+\frac{\partial}{\partial x_i}(\rho \varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon \qquad \text{(Eq 2)}$$

D. Analyzed

Last stage is called post-processing stage. In this stage, the result is analyzed with different methods. It can be shown in contour plot, vector plot and also in data curve which is in graph shape.

III. RESULTS AND DISCUSSION

Case study	High velocity fluid
No. of inlet	1
No. of outlet	2
Working fluid	Engine oil
Velocity	10 m/s (normal to boundary)

Table 1 : Data for experimental case

The study of this paper was about swirling velocity of fluid at high velocity in industrial equipmnent. Cyclone was taken as the equipment for this project. Cyclone has unique principle that enable particles to be separated according to their size and mass. Usually, the separation involves solid-solid materials, solid-gas materials and liquid-solid materials. As the particles become smaller in their size, it is difficult to control and monitor the separation inside the cyclone. The invention of the simulation system called Computational Fluid Dynamics (CFD) helps in improving the process of separation where users can manipulate and identify the flow of substances at certain time and point.

A. The effects of flow in geometry

The design of cyclone contribute to the simulation result at the end of the experiment. The data for this experiment is provided at Table 1. Refer to Table 1, the case involves turbulent flow where Reynolds number is $1.06\text{x}10^5$ which is more than 4000. The velocity of engine oil is set to 10 m/s as an inlet velocity. This value can be clarified as high velocity fluid flow inside the cyclone. The material for the inlet substance is engine oil.

This experiment is carried out for single phase flow instead of multiphase to ease the simulation process. As stated above, geometry of cyclone will affect the efficiceny of separation process. As for the inlet design, rectangular shape is used where the dimension for width and height are 0.2m and 0.4m respectively. The standard design for cyclone contains 2 outlets. The diameter for top outlet is 0.5m while outlet at the bottom of the cyclone is 0.3m.

In the case of efficiency, the design of cone in cyclone affect the separation process as the heavy particles will move downward to the bottom of cyclone. The increase in number of height of cone will give higher efficiency of separation. Because the time travel by the particles increase gives longer separation process occurs in cyclone. Other than that, diameter of cyclone also affects the efficiency of separation process. Diameter of cylindrical shape represents the diameter of cyclone. The sizing of diameter needs to be small to increase the separation process. This is because, centrifugal force react better at smaller density followed by increasing in velocity inlet. The balance between diameter cyclone and velocity inlet have to be correct and suite the process involve.

In order for the simulation to converge, CFD software required iteration based on each of meshing part. Time taken for the iteration to converge depends on the geometry of the cyclone. It can be in short time or long time. For this experiment, the iteration is up until 283 which taking only 10 minutes. The pattern of graph is shown in Figure 2. At the beginning of the graph, the pattern seems to be unstable because fluid starts to flow at the inlet section throughout the body parts. Then, after a few minutes, the fluid starts to stabilize during 20th iteration. The iteration remains stable until iteration of 283.

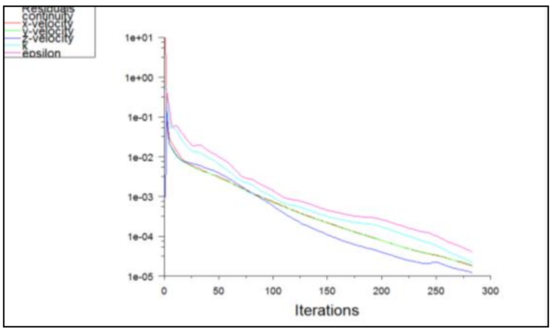


Figure 2 : Graph for iterations pattern

Dynamic response of fluid inside the cyclone is observed. Through this paper, the observation is focused on velocity profile of the engine oil flowing inside the cyclone. Flow in cyclone can be presented in three different axis which are axial, tangential and radial. In CFD software, the simulations can be used to predict gas hold-up, transfer rate, mass-transfer and process performance of fluid inside the selected equipment. The results gave very clear trends of fluid flow.

B. Velocity profiles

Second objective stated the studies on velocity profile using CFD. Typical velocity profile prediction sampled at the middle of the cyclone where 0-mm from the center. The results from the sweep surface can be presented in variety types. For example are based on contour, vector and pathline or streamline. RANS results presented in this paper have been obtained with the high-Reynolds realizable k-epsilon model.

The results from simulation predicted quite accurately the axial and tangential velocity components. In case of radial velocities, the CFD software should also predicted accurately since the three values are linked through the continuity equation. The flow of fluid entering the cyclone is depicted at Figure 5. Figure 5 shows the differentiation of streamline based on the number of points or layer. It can be observed that the inlet starts to enter and filled the cyclone.

The fluid flows at high velocity towards the inlet chamber which is about 11 m/s. It has been specified according to the fluid flowrate through the cyclone. The swirling patterns were clearly illustrated according to the velocity of fluid. Apparently, the core of the forming vortex inside the cyclone created most of the swirling of fluid.

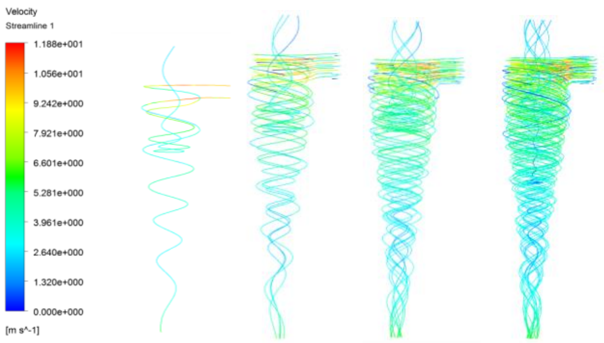
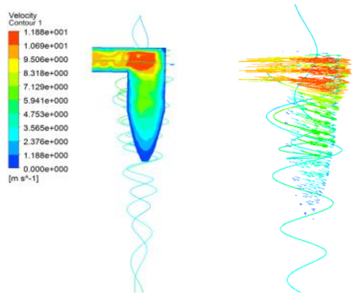


Figure 5 : Streamline inside the cyclone

The separation dynamics of fluid particles occurred inside vertical inlet flow. The velocity of fluid decreasing as they approach the bottom of the cyclone. The velocity profiles can be shown based on the legend provided. The movement of fluid can be categorized as low velocity at the conical section of the cyclone. The fast movement at the inlet affect the separation process to be more efficient.

The radial velocity magnitude is minimal when the vortex center is located in the measurement plane and increase when the distance of the vortex center to the measurement plane increase. High flowrate results in strong rotating centrifugal force thereby making the effective particles arrangements more efficient. Other than that, the turbulent kinetic energy is sharply increased which also influence the stability of the inner flow field of the cyclone (M.F. Dlamini, 2005).

It is well known, for example in swirling flows, that the k-ε model predicts a too rapid decay of the swirl because it assumes isotropic viscosity, when in fact the Reynolds stresses are highly anisotropic (Petty and Parks, 2001). With respect to Figure 6, the contour plots showed high velocity region at the inlet stream. The image was zoom in by using vector shown at Figure 6b. The movement of fluid at the inlet chamber can be observed clearly. Moreover, high velocity of fluid is only experienced at the beginning of the flow. The region at the inlet chamber experienced most of the turbulence flow.



(a) (b)

Figure 6 : Contour for velocity magnitude at the inlet chamber

Surface area at the wall was small compared to the other regions. This is the reason of the high velocity fluid showed at the wall of the cylone. Surface area of the regions can affect the movement of the fluid. The small area inhibit the free movement of fluid and result in high kinetic energy. Compare to the region at the center of the cyclone shown in Figure 7a, enough space is provided for the fluid to move freely in circulate movement. Thus, the regions experienced low velocity of fluid where kinetic energy is not stronger as at the wall.

The overall movement of flow inside the cyclone is served in Figure 6. In these contour planes, the predicted relevant positions and particles are shown at different coordinates of y-axis. The diagram is drew on X-Z plane. The samples are taken with 5 different spots. According to the experiment, the main chamber which is the cylindrical shape chamber having a diamater of 1 m while the inner chamber is 0.5 m. As the results, the planes are served with own patterns and profiles on velocity magnitude. Comparison at the same vertical cross-section revealed the difference between the velocity fields. The discusion started at the plane located at -0.3-m from y-axis. This vertical plane was the nearest plane at the inlet section. The velocity at the circled point can still be categorized as high velocity.

The similarity that can be observed from each contour planes are it shows that small eddies is presented at the wall of the cyclone. The fluid with high velocity traveled through the inlet chamber towards the vortex finder at the center of the main chamber. The droplets move in circular direction and hit the wall resulting high velocity region occurred at the wall of the chamber. The inner cylinder function as a gas outlet chamber.

At the center of the cyclone where plane is located exactly at 0-m from y-axis, the fluid experienced most largely swirling activities. Based on this plane, it can be seen that the low velocity region occurs at the center of the cyclone. This is because, the movement of fluid starts to slow down and settled downwards. The conical section in cyclone acts as a hopper where it promote particles with high density to moves downwards towards the bottom of the cyclone. A forced vortex labelled at Figure 7a is quite big, results in unstable flow field.

The vortex finder will stimulate the movement of the fluid for further separation. High velocity fluid will cause the heavy particles to settle at the bottom and lighter particles will move upwards towards the vortex core and exit the top of the inner cyclindrical chamber as overflow.

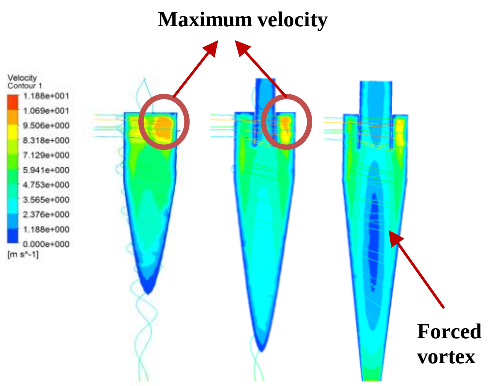


Figure 7a : Contour at -0.3-m, -0.2-m and 0-m from y-axis

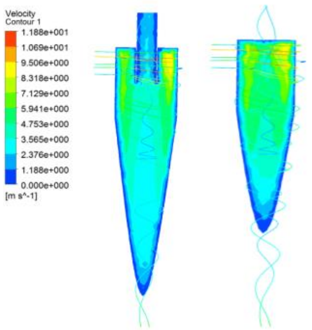


Figure 7b : Contour at 0.2-m and 0.3-m from y-axis

Refering to Figure 7a, the blue region at the center of the plane represented low velocity field about 1.2 m/s. This means that a swirling low decays just below the vortex finder. Particles with light density accumulated and captured at the center of conical section. The swirling fluid promote by the vortex finder will cause this particles to move upwards. Nevertheless, at the bottom of the 0-m plane, the velocity magnitude is increased. The forced exerted at small surface area increased the velocity of the particles to exit at this region.

The movement of fluid in cyclone can be further illustrated on Figure 8. These figures help to interpret the pattern of fluid occurs in the whole chamber. Based on this Figure 8a at 0-m from center, it can be describes that the droplets inside the cyclone body separate effectively. Most of the heavy droplets go to outer vortex and discharge at the bottom of the conical section. Plus, the fluid does not accumulate at the vortex core that can prevent short-circuit during separation process.

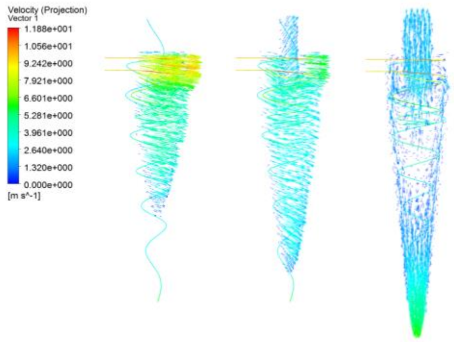


Figure 8a : Vectors at -0.3-m, -0.2-m and 0-m from y-axis

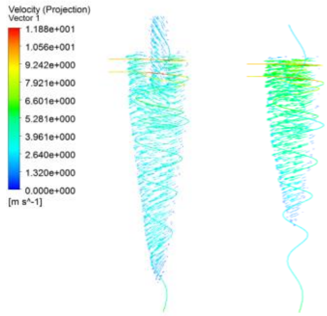


Figure 8b : Vectors at 0.2-m and 0.3-m from y-axis

C. *Vortex Core*

The third objective dictated about the studies of fluid flow at vortex core using RANS model. Separation of fluid mainly occurred at vortex core section. Vortex finder acts as the medium in separation process which will generate the circular motion in cyclone. Fluid that travels through this section produce high kinetic energy and move faster in circular direction. The high velocity particles inside the fluid hit the wall of the cyclone and settle downwards if the density is high enough. The performance of turbulence model relies ultimately on the experimental parameters.

In fact, the largest strength for the particles to experience is occurred at the vortex finder. The velocity field exhibits a more complex pattern around the vortex finder (A.F. Nowakowski & T. Dyakowski, 2003). The streamline of flow at the vortex core is presented at Figure 9a. The behavior of the vortex flow inside the cyclone is realized by looking at the velocity field from the simulation results. The strength of the circulating regions related to the geometric configuration and rate of the fluid flow through the feed.

By referring to this Figure 9, the result shown that fluid around vortex finder encountered high velocity flow. But the region inside the vortex finder experienced low velocity flow because the swirling of fluid at the vortex core forced the low density particles to move inside the spiral section and exit at the top of the chamber. The red region shows a high velocity inlet. The fluid continues to undergo high velocity at the vortex core. At the wall of the main chamber, the speed of the fluid slightly decreases as the fluid flows downwards.

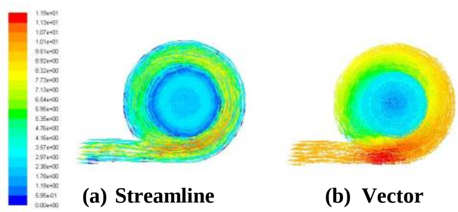


Figure 9 : Streamline and vector of fuid flow at vortex core

The turbulent kinetic energy contours with horizontal cross-section through the cyclone is served at Figure 10. The velocity profiles are depicted for selected depth along the cyclone. The flow for each selected depth are analyzed for the effectiveness of separation process. At the vortex core, turbulence would bring intense disturbance and pulsation to cyclone inner flow field. Thus, high turbulent kinetic energy may depleted the stability of flow field and separation efficiency. This also results in decreases in pressure drop. Figure 10 indicated the decay of the vortex flow as it evolves futher downstream. The tangential velocity starts to increase with decreasing radius as shown in Figure 10h.

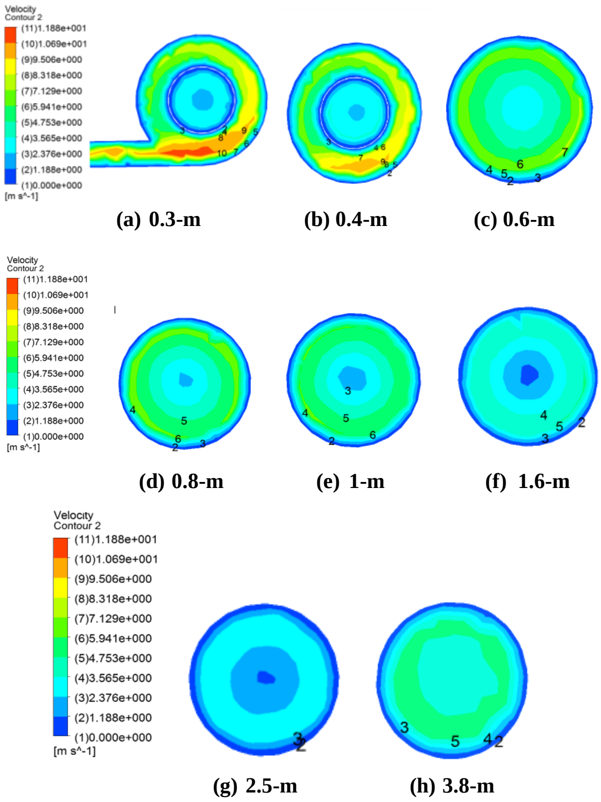


Figure 10 : Contours of velocity profiles from top of cyclone

IV. Conclusion

The study for this report is to analyze the fluid flow inside the cyclone including at the wall and the vortex core. In this paper, it can be concluded that the results obtained had meet the objectives of the experiment. The single fluid was analyzed instead of multiphase because the multiphase flow in a cyclone is quite complicated and might be a problem throughout the experiment. The overall separation for high velocity fluid is successful achieved. High velocity regions are transferred to the region near the wall thus concentrating the centrifugal field that is responsible for the increase in collection efficiency. Centrifugal force promoted the development of spiral movement at vortex finder.

The swirling inside the cyclone does not create any disturbance along the separation. But high velocity flow caused the large forced vortex at locus of zero vertical velocity. High pressure drop could also present at high velocity flow which can reduce the efficiency of the cyclone. The limitation arises where pressure inside the cyclone cannot be shown because of lack of variables involved in this paper. A way that can overcome this problem is by perform future research on implementation a mechanical device to the cyclone. In fact, the prediction for the multiphase flow pattern can also be a new upcoming research as it is rarely done by researchers.

Acknowledgment

A special thank you to my supervisor for all the guidance and time spent from the beginning of the project until completion. Because of his expertise in this scope of research, I able to gain as much knowledge as I can. In fact, it gave me plenty of ideas to write this research paper.

A huge gratitude goes to Universiti Teknologi Mara for giving me the opportunity to involve and establish this research paper successfully.

References

[1] A. Davailes, E. C. (2012). Analysis of swirling flow in hydrocyclones operating under dense regime. *Minerals Engineering*.

[2] Bakker, A. H. (2001). Realize Greater Benefits from CFD. *Chemical Engineering Progress*, 45-53.

[3] C. Esionwu-k, A. M. (2014). Further Aerodynamics and Propulsion and Computational Techniques CFD Solution Methodology.

[4] Dyakowski, A. N. (2003). Simulation and validation of turbulent gas flow in a cyclone using caelus. *Trans IChemE, Vol 81*.

[5] L. Ma, P. F. (2015). Cfd simulation study on particle arrangements at the entrance to a swirling flow field for improving the separation efficiency of cyclones. *Aerosol and Air Quality Research*, 2456-2465.

[6] Lo, D. P. (2015). Advances in CFD Modelling of Multiphase Flows in Cyclone Separators. *Chemical Engineering Transactions*.

[7] M. Dlamini, M. P. (2005). A CFD simulation of a single phase hydrocyclone flow field. *The Journal of The South African Instittute of Mining and Metallurgy*, 711-718.

[8] R. Hreiz, C. G. (2011). Chemical Engineering Research and Design Numerical investigation of swirling flow in cylindrical cyclones. *Chemical Engineering Research and Design*, 2521-2539.

[9] S. Aktershev, P. K. (2013). Stability of Swirl Axisymmetric Incompressible Flow. *Procedia IUTAM*, 13-21.

[10] Zuo, W. (n.d.). Introduction of Computational Fluid Dynamics. *FAU Erlangen-Nurnberg*.

[11] Bimal P. Singh, Sarama Bhattacharjee, Jouhari, A.K. & Vibhuti N. Misra. Quantitative Apporoaches In Mineral Processing. (2003). Allied Publishers PVT. Limited. pg 2.

[12] Wen, C. Y. Handbook of Fluidization and Fluid-Particle System. (2003). Marcel Dekker, Inc. USA. pg 846.

[13] Mohd Faizal Mohideen Batcha, Mohd Al-Hafiz Mohd Nawi, Shaharin Anwar Sulaiman and Vijay R. Raghavan. January 7, 2013. Numerical Investigation of Airflow in a Swirling Fluidized Bed. *Asian Journal of Scientific Research, 6: 157-166.*