CFD Analysis of a Cyclone Seperator

Sumalatha Tankari[1] Pradeep Halder[2] Edla Franklin[3] K.Vineetha Peter[4], Kalyan Aravalli[5]
[1,2,3,4]Department of Mechanical Engineering [5]Department of Electrical Engineering
[1,2,3,4]CBIT, Hyderabad, Telangana state, SRM University, Tamilnadu, India

Abstract— This work deals with the application of Computational Fluid Dynamics (CFD) for cyclone modeling on three-dimensional unstructured mesh using the Reynolds Stress turbulence model, a standard k-ε or a k-ω model and Large Eddy simulation. Large-eddy simulations (LES) is performed on the gas flow in a cyclone at R_e = 280,000. Numerical analysis of flow characteristics and separation efficiency in a high-efficiency cyclone is carried out. The model is only estimated the cyclone's performance under the limited environments; it is difficult to obtain a general model for all the types of cyclones. The purpose of this study is to find out the flow characteristics and separation efficiency numerically using ANSYS Fluent software. The Reynolds stress model (RSM), standard k-ε model and Large Eddy simulations are used in this work to know the flow separation characteristics. The models represent the 3-D, time-dependent flow analysis. CFD velocity profiles, and pressure drops for all the time-dependent flows are compared and discussed. Some details of the flow in the relatively small region in the vicinity of the inlet have strong influence on the separation process is checked with the simulations. The cyclone flow field pattern is simulated and analyzed with the aid of velocity components and static pressure contour plots.

Keywords: Cyclonic Separation, K-Epsilon, Large Eddy simulation, Hydrocyclone, Recirculation, RSM

I. INTRODUCTION

Cyclonic separation is a method of removing particulates from an air, gas or liquid stream, without the use of filters, through vortex separation. Rotational effects and gravity are used to separate mixtures of solids and fluids. The method can also be used to separate fine droplets of liquid from a gaseous stream.

A high speed rotating (air) flow is established within a cylindrical or conical container called a cyclone. Air flows in a helical pattern, beginning at the top (wide end) of the cyclone and ending at the bottom (narrow) end before exiting the cyclone in a straight stream through the center of the cyclone and out the top. Larger (denser) particles in the rotating stream have too much inertia to follow the tight curve of the stream, and strike the outside wall, then falling to the bottom of the cyclone where they can be removed. In a conical system, as the rotating flow moves towards the narrow end of the cyclone, the rotational radius of the stream is reduced, thus separating smaller and smaller particles[1]. The cyclone geometry, together with flow rate, defines the *cut point* of the cyclone. This is the size of particle that will be removed from the stream with 50% efficiency. Particles larger than the cut point will be removed with a greater efficiency, and smaller particles with a lower.

Large scale cyclones are used in sawmills to remove sawdust from extracted air. Cyclones are also used in oil refineries to separate oils and gases, and in the cement industry as components of kiln preheaters. Cyclones are increasingly used in the household, as the core technology in bagless types of portable vacuum cleaners and central vacuum cleaners[2]. Cyclones are also used in industrial and professional kitchen ventilation for separating the grease from the exhaust air in extraction hoods. Smaller cyclones are used to separate airborne particles for analysis. Some are small enough to be worn clipped to clothing, and are used to separate respirable particles for later analysis.

Analogous devices for separating particles or solids from liquids are called hydrocyclones or hydroclones. These may be used to separate solid waste from water in wastewater and sewage treatment.

A. Dust collector

A dust collector is a system used to enhance the quality of air released from industrial and commercial processes by collecting dust and other impurities from air or gas[3]. Designed to handle high-volume dust loads, a dust collector system consists of a blower, dust filter, a filter-cleaning system, and a dust receptacle or dust removal system. It is distinguished from air cleaners, which use disposable filters to remove dust.

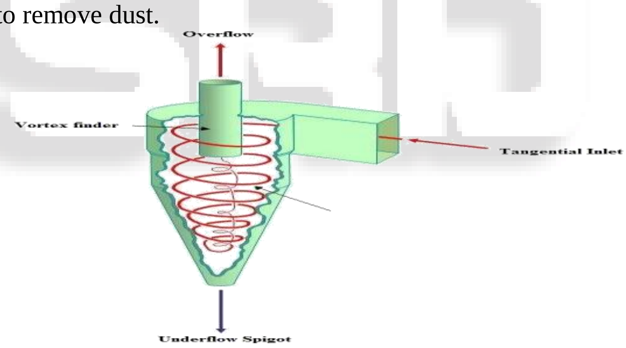

Fig. 1: Cyclone separator

Airflow diagram for Aerodyne cyclone in standard vertical position. Secondary air flow is injected to reduce wall abrasion. Airflow diagram for Aerodyne cyclone in horizontal position, an alternate design. Secondary air flow is injected to reduce wall abrasion, and to help move collected particulates to hopper for extraction.

An alternative cyclone design uses a secondary air flow within the cyclone to keep the collected particles from striking the walls, to protect them from abrasion. The primary air flow containing the particulates enters from the bottom of the cyclone and is forced into spiral rotation by stationary spinner vanes[4]. The secondary air flow enters from the top of the cyclone and moves downward toward the bottom, intercepting the particulate from the primary air. The secondary air flow also allows the collector to optionally be mounted horizontally, because it pushes the particulate toward the collection area, and does not rely solely on gravity to perform this function.

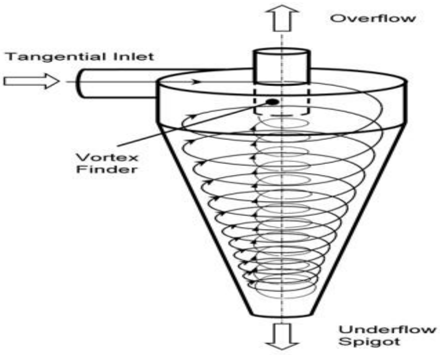

Fig. 2 Schematic view of cyclone separator

Reverse flow cyclones are probably the most widely used separator devices in industrial environments, being applied in many industrial branches, ranging from food and pharmaceutical industries to mining and petrochemical industries. Their popularity is based on their relative geometrical simplicity, low manufacturing, operational and maintenance costs. Despite their deceitful simplicity, cyclones are complicated to design and hardly optimized, since the flow field within them is extremely complex [5]. The RSM model is naturally suitable for calculating the average properties of swirling flows as it is capable of handling anisotropic effects. This has been extensively demonstrated [6-7].

II. RESULTS AND DISCUSSIONS

A. Discrete Phase Model

The Discrete Phase Model tracks the motion of individual (discrete) particles. Note that the same principles apply whether the object is a solid p, q p article, or as in this case, a liquid droplet. The trajectory of each particle (droplet) is computed over a large number of steps as it passes through the flow domain. Since we know the mass and the surface area of each particle at each step the solver can compute and the surface area of each particle, at each step the solver can compute the balance of forces acting on it. Integrated over the flow domain, the overall trajectory can be determined[8]. Unlike the continuous phase (air) where material flows through the grid cells (known as an 'Eulerian' reference frame), the DPM moves particles were each particle has its own x,y,z co-ordinate (known as a 'Lagrangian' reference frame). In this case the particles (droplets) are inert, however the DPM does support much more complex cases where the particles may evaporate or combust.

B. Setup Single Phase

– General - Enable gravity: -9.81 m/s² in y-direction to include the effect of gravitational force on the droplets.
– Models – Turbulent Flow (Re~400,000*)
– Enable the Realizable k-epsilon Model With Enhanced Wall Treatment
– Materials – Process Gas assumed to be Air
– Process Gas and Liquid Droplets come into a Reactor
– We will use the Discrete Phase Model (DPM) to track the Liquid Droplets
– For the process, it is important that only small droplets come into the Reaction Zone, to avoid blockage in the catalyst.
– If droplets hit the Reactor Wall, they run down – So, it is not necessary to treat them as droplets after hitting the wall – Larger droplets will be removed with this logic and only smaller droplets will stay in the domain and pass through to the outlet which is the inlet to the reaction zone.
– The problem is treated as isothermal

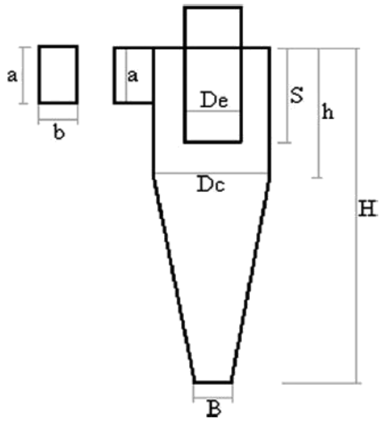

Fig. 3: Schematic view of cyclone separator

The geometric model of cyclone separator is as shown in the below Fig. 3. Modeling of cyclone separator is carried out on gambit and element mesh of tetrahedron with 1.2 million elements as shown in the Fig. 5.

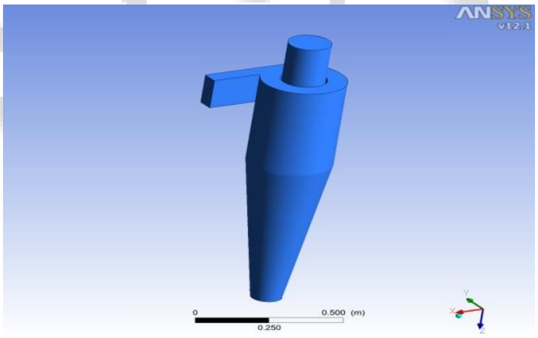

Fig. 4: Cyclone separator 3D model

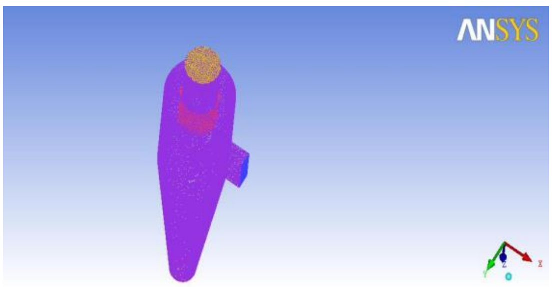

Fig. 5: Tetrahedron meshing for cyclone reactor (1.2 million elements)

The converged mesh of cyclone separator shown in Fig.5 is made of tetrahedron elements, and mesh gradation is followed based on the geometry and other associated conditions with the model and boundary conditions

C. K-epsilon model

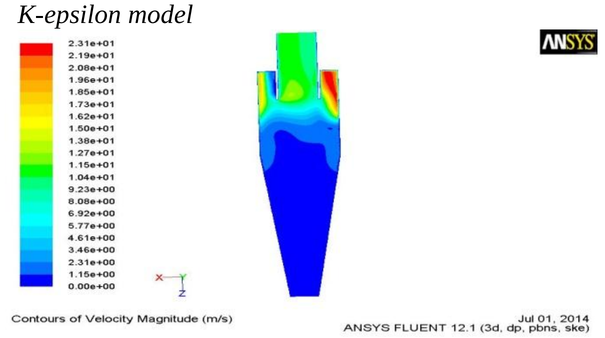

Fig. 6: Variation of velocity magnitude.

The above contour shows that velocity variation of air in cyclone separator. From the above Fig 6 it is observed that maximum is 23.1 m/s at top side and it is due to the inlet particles.

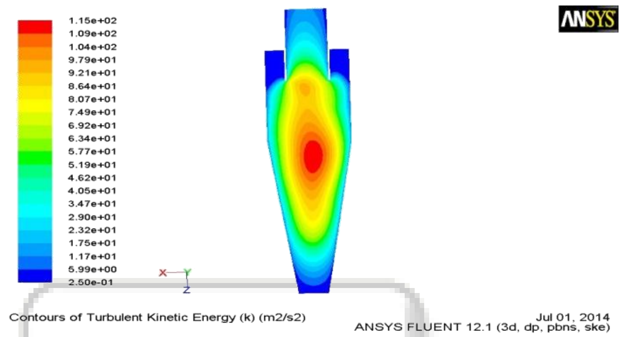

Fig. 7: Variation of turbulent kinetic energy in cyclone separator

The turbulent kinetic energy variation in the separator given in Fig. 7. It shows the maximum turbulent kinetic energy at the middle because in standard K-Epsilon model the particle flows at the middle section of the cyclone separator. The concentration turbulence of kinetic energy increasing gradually to maximum of 1.15e3 K towards the center.

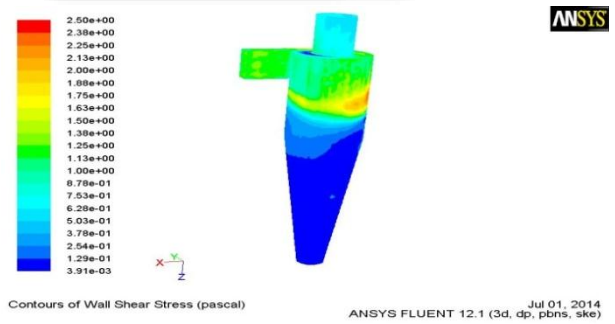

Fig. 8: Variation of wall shear stress

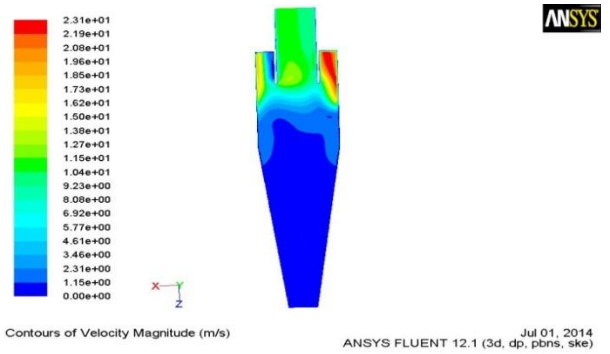

Fig. 9: Variation of velocity magnitude of cyclone separator.

The Fig 9 shows the variation of the velocity plot inside the cyclone separator. From the Figure it is observed that the velocity is increasing at the top side of cyclone separator. The maximum velocity inside the scramjet is 2.31e3 m/s.

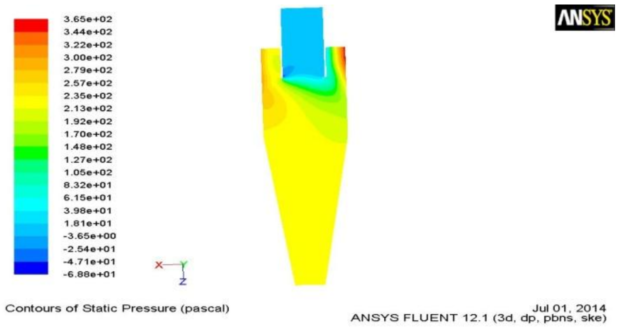

Fig. 10: Variation of static pressure.

From the above Fig. 10 it is clear wall static pressure is maximum of 3.65 e2 pascal at the top inlet wall and it is minimum at the outlet.

D. Large Eddy Simulation

The Fig 11 static pressure variation in the cyclone separator ,From the Figure it is clear that at the entrance up to some distance the maximum of static pressure 1.29 e3 bar is seen, it is decreased to 3.43e0 bar at the outlet of the cyclone separator. The maximum pressure at the inlet is due to the particles striking the walls. The Fig 12 shows the Variation of static pressure in cyclone separator the maximum shear stress of 1.29 e3 is at the outlet of the cyclone separator the minimum

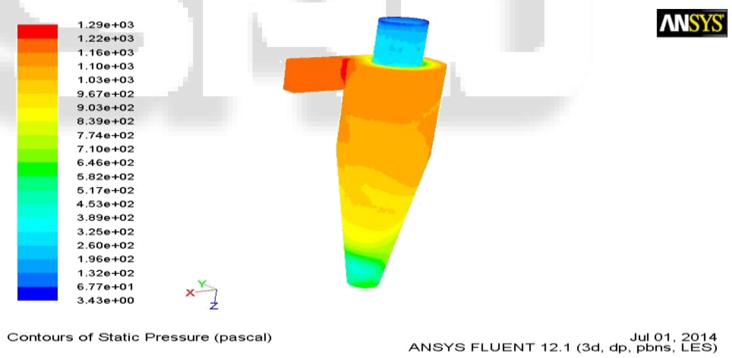

Fig. 11: Variation of static pressure in cyclone separator

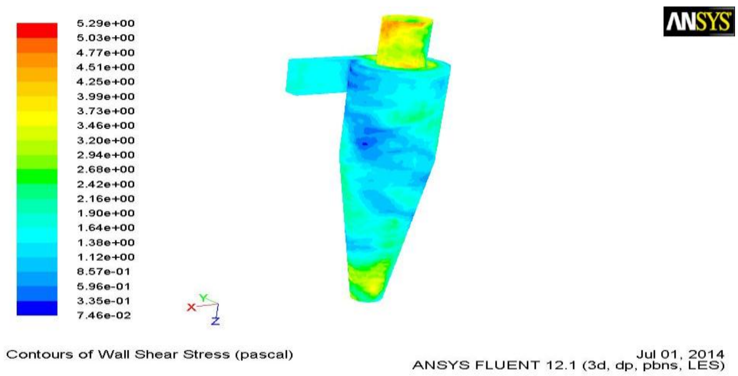

Fig. 12: Variation of wall shear stress in cyclone separator

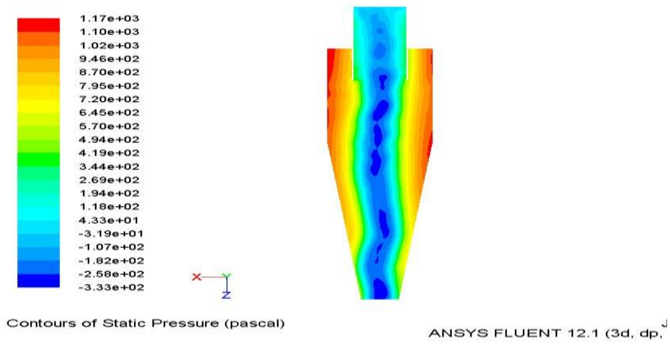

Fig. 13: Variation of static pressure in cyclone separator.

Fig 13 shows the variation in velocity magnitude the maximum value of 35.4 m/s inside the cyclone separator

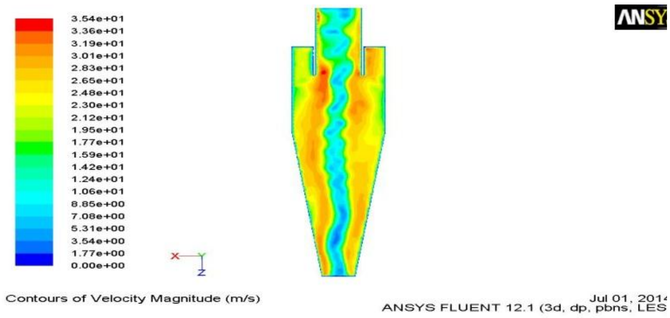

Fig. 14: Variation of velocity magnitude in cyclone separator of LES.

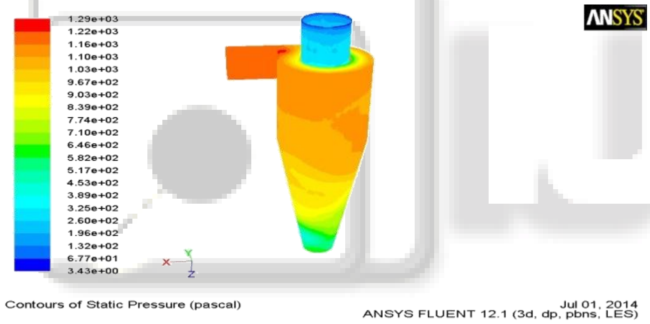

Fig. 15: Variation of maximum static pressure in cyclone separator of LES

The Fig 14 Variation of velocity magnitude in the cyclone separator, from the figure it is clear that at the walls the maximum static pressure of 1.17e3 bar in the cyclone separator. But in the Fig 15 the maximum pressure of 1.29 e3 bar is at the inlet

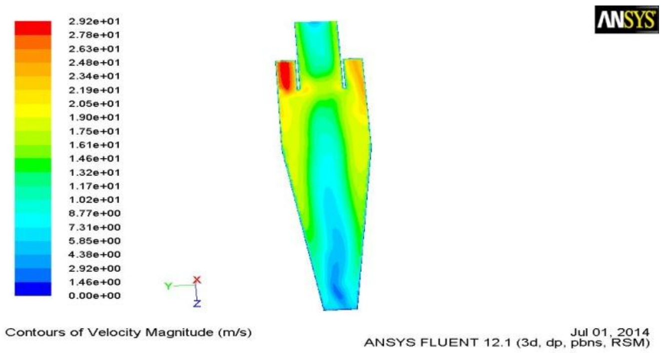

Fig. 16: Static pressure variations in cyclone separator using RSM model.

The Fig 16 static pressure variation in the cyclone separator, from the figure it is clear that at the walls the maximum static pressure is 5.44e2 bar, it is decreased. The Fig 17 shows the variation of the velocity plot inside the cyclone separator. From the Figure it is observed that the velocity is increasing at the top side of cyclone separator. The maximum velocity inside the cyclone separator is 29.2 m/s. The above Fig 18 shows the particles pathlines in Reynolds stress model of a cyclone separator0

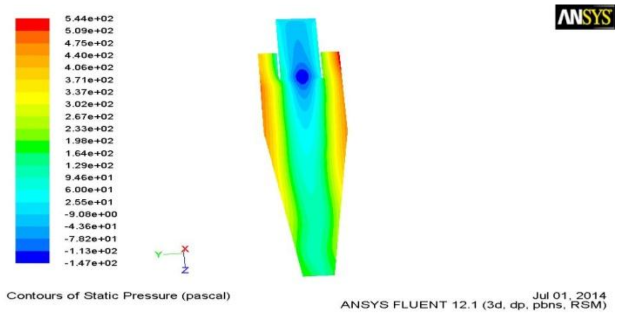

Fig. 17: Variation of velocity magnitude of cyclone separator using RSM model

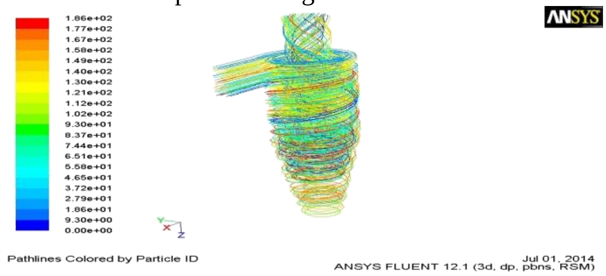

Fig. 18: Variation of particle pathlines of cyclone separator with RSM model

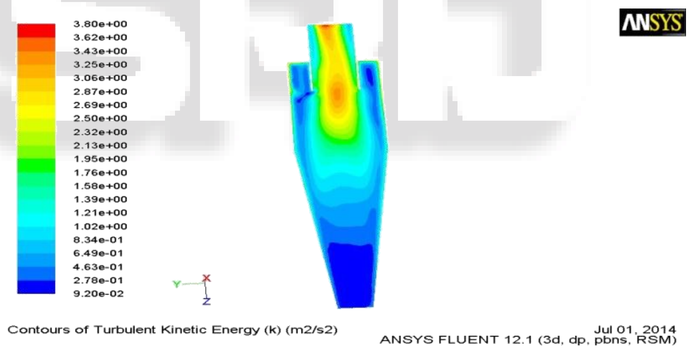

Fig. 19: Variation of turbulent kinetic energy in cyclone separator

Fig 19 shows the Variation of turbulent kinetic energy in cyclone separator the maximum turbulence kinetic energy is at the outlet of 3.80e0 m^2/s^2.

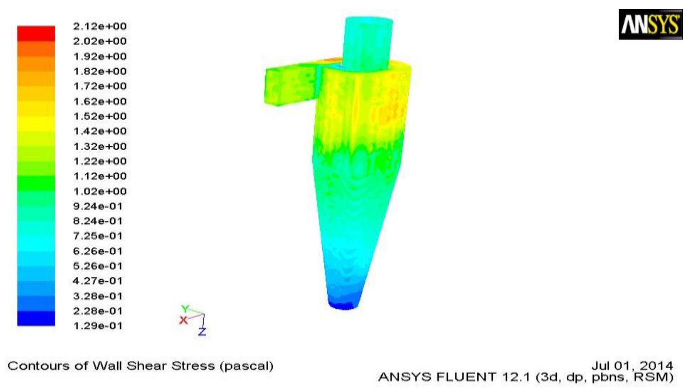

Fig. 20: Variation of wall shear stress in cyclone separator

The above Fig 20 shows the wall shear stress is maximum at the walls on the top side at inlet the maximum of 2.02 e0 pascals

Model type	**Velocity, m/s**	**Pressure, Pa**	**Turbulent kinetic energy, m^2/s^2**	**Shear stress, Pa**
K-epsilon	2.31	365	115	2.38
LES	35.4	1170	7.05 e-4	5.29
RSM	29.2	544	3.80	2.12

Table 1: Variation of velocity, pressure, kinetic energy and shear stress in different models

The fig 21 shows the variation of pressure in Large eddy simulation, Reynolds stress model and K-epsilon. It is clear that the variation of pressure inside the cyclone separator more in LES model on the either sides of axis, where as the two models has less variation in pressure drop towards positive axis than negative axis. In fig 22 the variation of velocities of different models can be observed, LES model has more velocity.

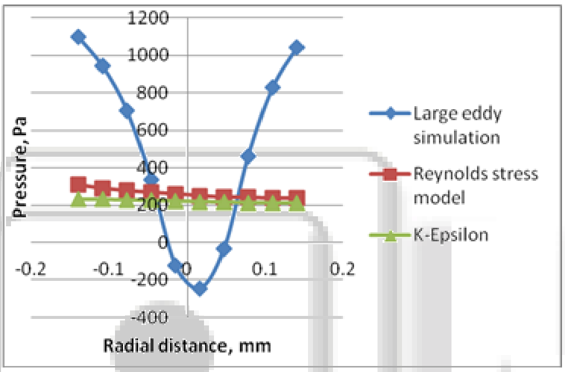

Fig. 21: Variation of pressure with the radial distance in various models

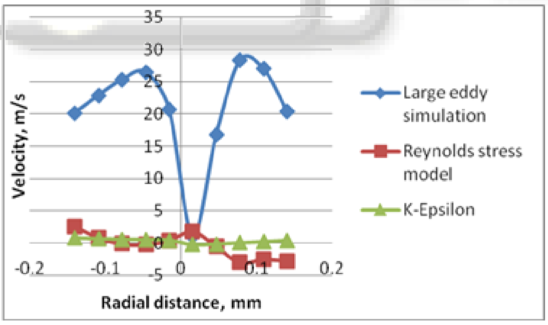

Fig. 22: Variation of velocity with the radial distance in various models

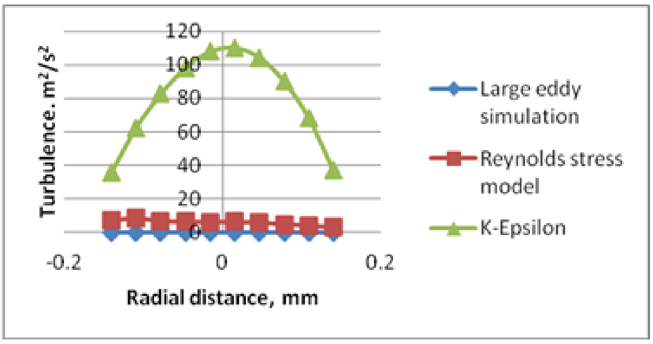

Fig. 23: Variation of turbulence with the radial distance in various models

The fig 23 shows the variation of turbulence in LES, RSM, K-epsilon models with respect to radial distance. Among the three models standard k-epsilon model has more turbulence as this model has more turbulence inside the cyclone separator as the particle flow is turbulent

III. CONCLUSIONS AND FUTURE SCOPE OF WORK

The following conclusions are drawn from the present work

A. Conclusions

- In this work, the effect of mass-loading on the gas flow and solid particle motion in a cyclone separator has been studied numerically. The simulations confirm that the separation process involves an interplay between centrifugal forces induced by swirl, and dispersion due to turbulence.
- Depending on the relative extents of turbulence attenuation and weakening of swirl intensity, the efficiency of the cyclone can either increase or decrease.
- The amplitude of the vortex core precession strongly depends on the axial position in the cyclone
- The complicated flow region in the top part of the cyclone (the region that contains the connection of the inlet channel to the cyclone body, as well as the vortex finder) is decisive for the separation behavior of particles slightly larger than the cut-size.
- The variation of pressure in Large eddy simulation is greater than the Reynolds stress model and K-epsilon model. It is clear that the variation of pressure inside the cyclone separator more in LES model on the either sides of axis, where as the two models has less variation in pressure drop towards positive axis than negative axis.
- The variation of velocities of different models can be observed, LES model has more velocity (35.4 m/s) where as the velocities in Reynolds stress model is 29.2m/s and and K-epsilon model is 2.31m/s

B. Future Scope Of Work

As a last remark, from both the experimental and simulation points of view, the grade efficiency is the final result of a combination of geometry, flow conditions and particle dynamics. It thus conveys all the errors possibly incurred in each measurement/calculation. Taking into account the complexity of the very intrinsic phenomena present in the gas–solid flow in cyclones, the results for the grade efficiency are need to be studied further.

REFERENCES

[1] T.G. Chuah, J. Gimbun, T.S.Y. Choong, A CFD study of the effect of cone dimensions on sampling aerocyclones performance and hydrodynamics, Powder Technology 162 (2006) 126–132.

[2] M. Azadi, M. Azadi, A. Mohebbi, A CFD study of the effect of cyclone size on its performance parameters, Journal of Hazardous Materials 182 (2010) 835–841.

[3] A. Raoufi, M. Shams, M. Farzaneh, R. Ebrahimi, Numerical simulation and optimization of fluid flow in a cyclone vortex finder, Chemical Engineering and Processing 47 (2008) 128–137.
[4] K.U. Bhaskar, Y.R. Murthy, M.R. Raju, S. Tiwari, J.K. Srivastava, N. Ramakrishnan, CFD simulation and experimental validation studies on hydrocyclone, Minerals Engineering 20 (2007) 60–71.
[5] A.C. Hoffman, L.E. Stein, Gas Cyclones and Swirl Tubes: Principles, Design and Operation, Springer, 2008.
[6] L. Shi, D. Bayless, Comparison of boundary conditions for predicting the collection efficiency of cyclones, Powder Technology 173 (2007) 29–37. A. Kepa, Divison of outlet flow in a cyclone vortex finder-the CFD calculations, Separation and Purification Technology 75 (2010) 127–131.
[7] J. Gimbun, T.G. Chuah, T.S.Y. Choong, A. Fakhru'l-Razi, Prediction of the effects of cone ip diameter on the cyclone performance, Aerosol Science 36 (2005) 1056–1065.
[8] S. Bernardo, M. Mori, A.P. Peres, R.P. Dionísio, 3-D computational fluid dynamics for gas and gas–particle flows in a cyclone with different inlet section angles, Powder Technology 162 (2006) 190–200.

