RESEARCH ARTICLE OPEN ACCESS

Numerical Study Of Flue Gas Flow In A Multi Cyclone Separator

Ganga Reddy C* Umesh Kuppuraj**
*(Mechanical Centre of Excellence, HCL Technologies, Bangalore-562106)
** (Mechanical Centre of Excellence, HCL Technologies, Bangalore-562106)

ABSTRACT
The removal of harmful particulate matter from power plant flue gas is of critical importance to the environment and its inhabitants. The present work illustrates the use of multi-cyclone separators to remove the particulate matter from the bulk of the gas exhausted to the atmosphere. The method has potential to replace conventional systems like electrostatic precipitator due to inherent low power requirement and low maintenance. A parametric model may be employed to design the system based on the requirement of the power station. The present work describes the simulation of flue gas flow through a cyclonic separator. A Finite volume approach has been used and the pressure-velocity coupling is resolved using the SIMPLE algorithm. Discrete phase model is used to inject solid particles from inlet. In this numerical analysis a cluster of four cyclonic separators are considered. Comparisons are made between the available experimental results and the computational work for validation of the numerical models and schemes employed in the work. The separation efficiency and particle trajectories are shown and found comparable to similar cases from literature. The experimental results correlate well for the model under consideration.

Keywords – Flue gases, discrete phase model, computational fluid dynamics, Separation efficiency, cyclone separator

I. INTRODUCTION

Particulates also referred to as particulate matter (PM) or suspended particulate matters (SPM) are tiny subdivisions of solid material suspended in a fluid. Particulates are naturally in nature like those in volcanoes, dust storms and sea spray; while others are created by human activities like combustion of fossil fuels in power generation, propulsion and various industrial processes.

Many different processes are employed to separate particulate matter in power plants such as:

- Bag Filters
- Gravitational Separator
- Electrostatic precipitators
- Cyclone Separators

Of all the devices used to separate dust from gas flows Cyclone Separators finds wide application in the industry today due to its inherent advantage like geometric simplicity, low power consumption and flexibility with regard to operating conditions. The illustration and model is shown in fig.1 and fig. 2 respectively [3].

Fig.1 General depiction of Cyclone Separator (Source:imimg.com)

Fig.2 General Depiction of Cyclone Separator (Source: ssenvirotech.com)

The typical geometry of a gas cyclone is used to separate particles from a gaseous stream. The cyclone utilizes the energy obtained from the fluid pressure gradient to create rotational fluid motion. This rotational motion causes the dispersed phase to separate relatively fast due to the strong acting forces. In widely used reverse flow cyclones of the cylinder, gases spiral down from a tangential inlet towards the apex of a conical section, where the flow is reversed and the particles are collected in a dust collector. The continuous phase then proceeds upward in an inner core flow towards the gas exit passing through the vortex finder. Swirl and turbulence are the two challenging actions responsible for separation process; swirl induces a centrifugal force on the solids phase which is the driving force behind the separation. Prediction of the separation process therefore requires an adequate representation of the gas flow field (including its turbulence characteristics) in the presence of a particulate phase.

The Reynolds Stress Model (RSM) was selected to simulate the strongly swirling turbulent flow in the annular space of a cyclone separator by Xue et al. [5]. They concluded that the predictions with the Reynolds Stress Model (RSM) are in reasonable agreement with experimental results, and it reveals that the RSM is suitable for investigating the flow of cyclone separators.

II. COMPUTATIONAL MODEL

The illustration below is the graphical representation of the geometric model being considered for the CFD analysis. The model constitutes of four individual cyclone separators connected around the circumference of a common inlet.

The flow is being fed through the inlet (shown in red) against gravity into the cyclones via a tangential inlet. The flow progresses into the cyclones and then through the dust outlet at the bottom. The gas outlet is positioned at the top and the four outlets are connected to a solitary chamber which exhausts downstream and/ or into the atmosphere.

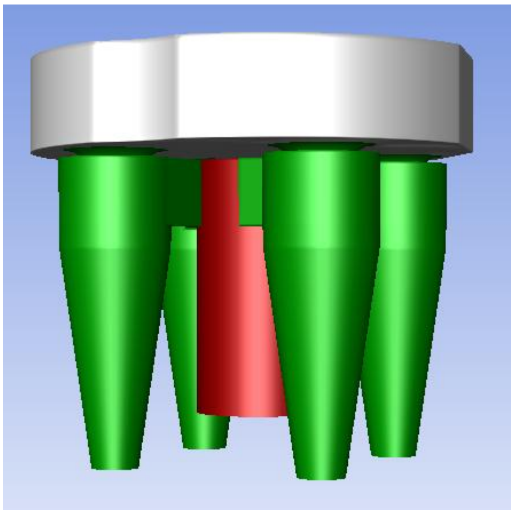


Fig.3 Computational model illustrating the common inlet (Red), the cluster of cyclones (Green) and the outlet (Grey)

The inlet is modeled with a diameter of 1m. The total inlet area is 0.793 m2. The cyclone has a total height of 2.45 m while the diameter is 0.889m. The cyclone bottom or the dust outlet is 0.439 m2 while the outlet discharging into the atmosphere is 0.748m2

III. DOMAIN DISCRETIZATION

A single cyclone model was discretized using hexahedral cells owing to their superior characteristics. The single cyclone was composed of 93096 elements and these were arrayed around the axis of the tangential inlet’s circumference to evolve a multi-cyclone model comprising of 4 individual cyclone separators connected via a common inlet. The total cell count of the computational domain excluding the domain extension numbered at 835944.

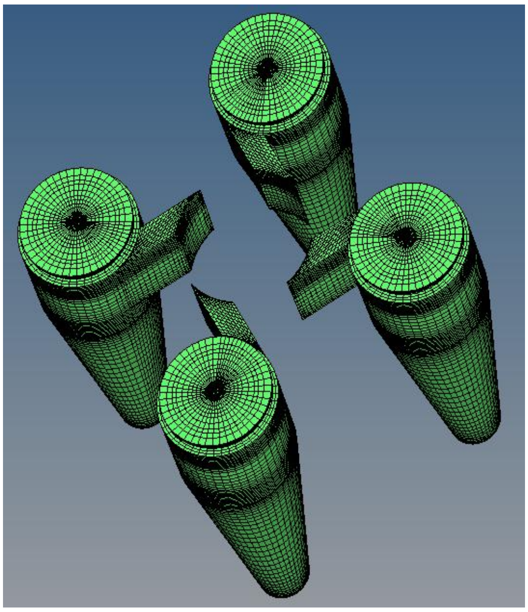


Fig.4 Meshed model of the cyclones

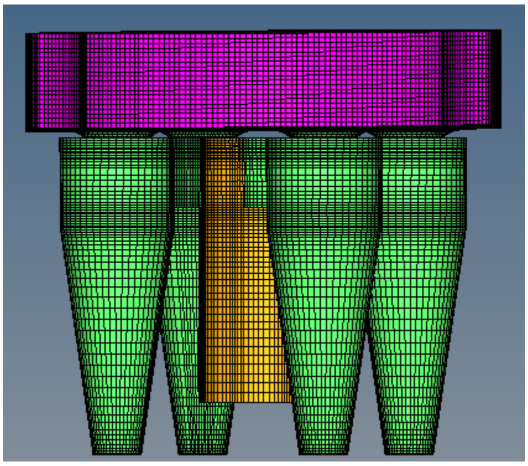
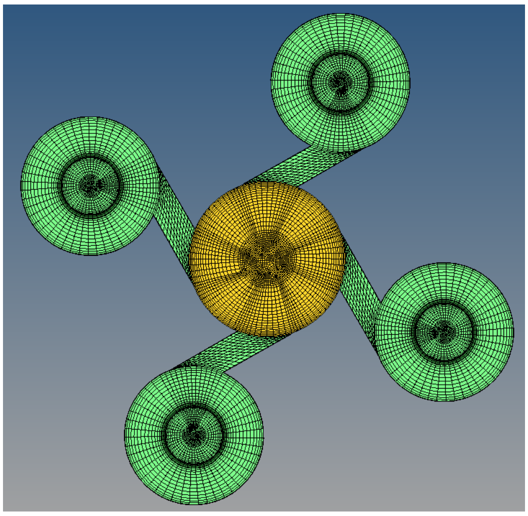


Fig 5. Mesh model of computational domain



1. *Fig. 6 Sectional view of computational domain*

IV. NUMERICAL SETTINGS

FLUENT is a commercially offered CFD code which employs the finite volume formulation to accomplish coupled or segregated calculations (with reference to the conservation of mass, momentum and energy equations). For turbulent flow in cyclones, success of CFD is consummate with the accurate depiction of the turbulent behavior of the flow.

To model the swirling turbulent flow in a cyclone separator, there are a number of turbulence models available in FLUENT. These range from the standard K-ε model to the more complicated Reynolds stress model (RSM) and Large Eddy Simulation (LES). The Reynolds Stress model is employed for the numerical computation for the case under consideration. The Reynolds stress turbulence model (RSM) necessitates the solution of transport equations for each of the Reynolds stress components. The Reynolds stress turbulence model yields a precise prediction of the swirl flow pattern, axial velocity; tangential velocity and pressure drop in cyclone simulations.

V. DPM SETTINGS

The Lagrangian discrete phase models in FLUENT follows the Euler-Lagrange approach. The fluid phase is treated as a continuum by solving the time-averaged Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of particles through the calculated flow field. The dispersed phase can exchange momentum, mass, and energy with the fluid phase.

A fundamental assumption made in this model is that the dispersed second phase occupies a moderate volume fraction (usually less than 30 %, where the volume fraction is the ratio between the total volume of particles and the volume of fluid domain); even though high mass loading is acceptable. The particle trajectories are computed individually at specified intervals during the fluid phase calculation. This makes the model appropriate for the modeling of particle-laden flows. The particle loading in a cyclone separator is reasonable and therefore, it can be safely assumed that the presence of the particles does not affect the flow field (one-way coupling).

In FLUENT, the drag coefficient for spherical particles is calculated by using the correlations developed by Morsi and Alexander. The equation of motion for particles was integrated along the trajectory of an individual particle. Collection efficiency statistics were obtained by releasing a specified number of mono-dispersed particles at the inlet of the cyclone and by monitoring the number escaping through the outlet. Collisions between particles and the walls of the cyclone were assumed to be perfectly elastic (coefficient of restitution is equal to 1).

VI. RESULTS AND DISCUSSIONS

1. Validation of Results

In order to validate the results obtained during the course of this analysis; a single cyclone from the cluster was simulated against a case presented by Hoekstra using a Laser Doppler Anemometry (LDA). Comparisons were made for axial and tangential velocity profiles. The Reynolds Stress Model (RSM) simulation predicts a similar trend as observed experimentally though deviations were noticed at the peaks for the tangential velocity. On the whole considering the complexity of the swirl and turbulence in the cyclone results are in good agreement.

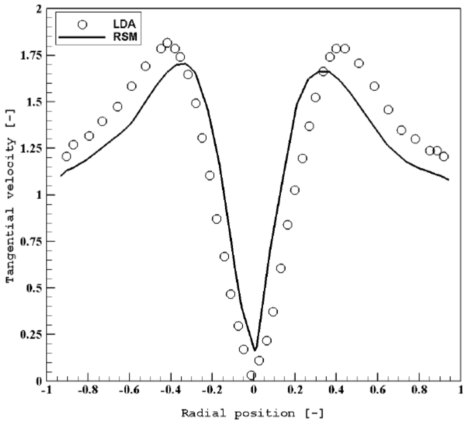


Fig.7 Comparison between RSM simulation and LDA tangential velocity

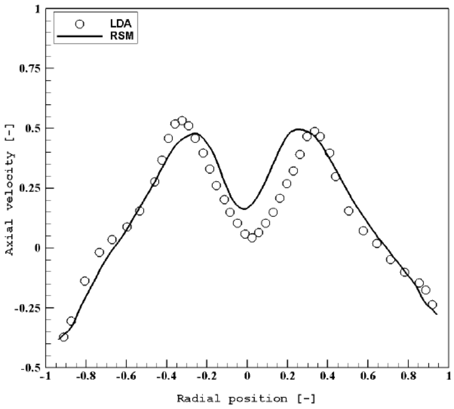


Fig.8 Comparison between RSM simulation and LDA radial velocity

VII. Flow field Patterns

The Pressure Field

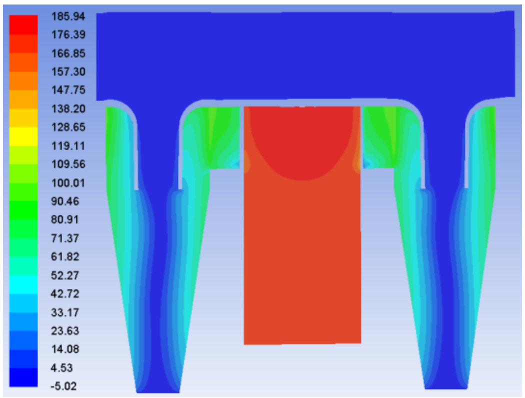


Fig.9 Static pressure plot in Pa

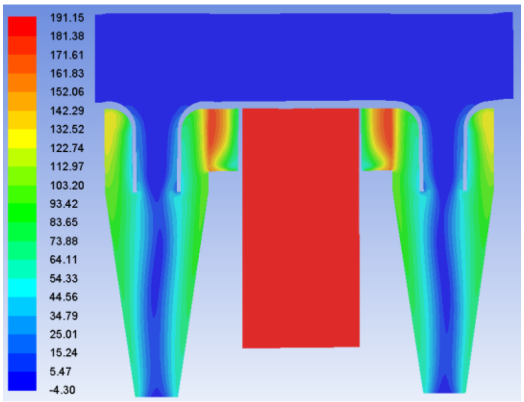


Fig.10 Total pressure plot in Pa

A negative pressure appears in the vortex region owing to high swirl velocity whilst the pressure reduces from the wall to the center. The flow is not symmetric about the cyclone axis as is clear from the low pressure zone. The flow with the regard to the axis of the inlet is symmetrical as can be inferred from figure 9.

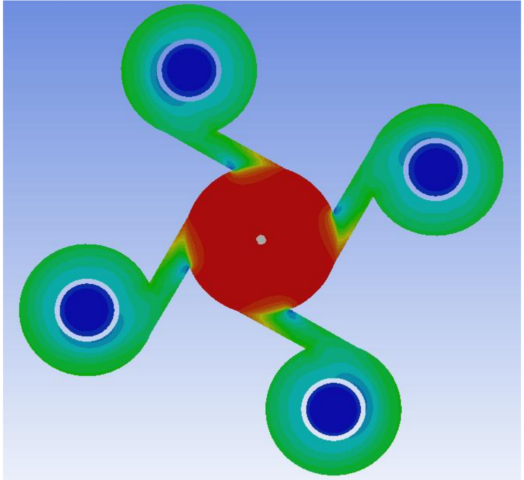


Fig.11 Static pressure plot in Pa

The gradient of pressure in the axial direction is narrow while is highest in the radial direction.

The Velocity Field

The contour plots for the absolute, tangential and radial velocities are analyzed to ascertain the characteristics of the flow field.

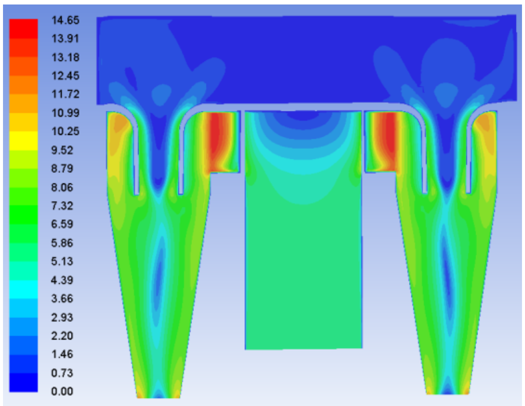


Fig. 12 Velocity Magnitude plot in m/s

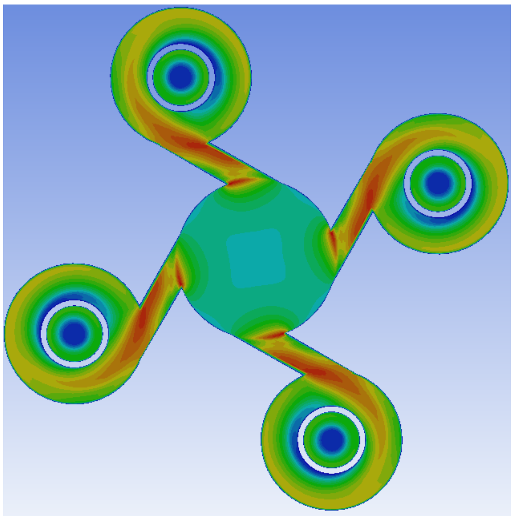


Fig.13 Velocity Magnitude plot in m/s

The absolute velocity is elevated at the tangential inlet to the individual cyclone and is seen to diminish progressively.

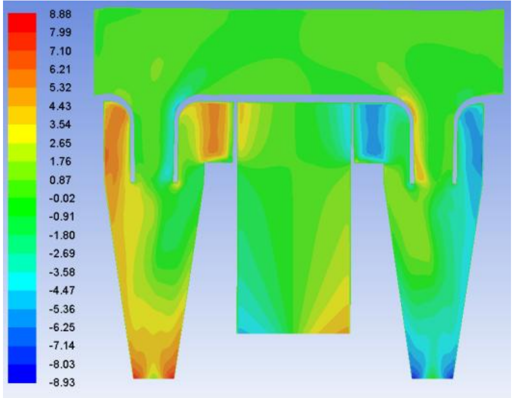


Fig. 14 Tangential Velocity plot in m/s

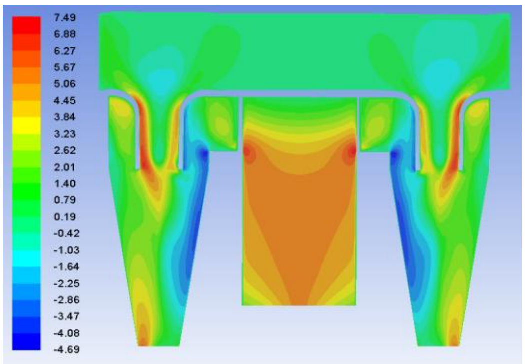


Fig.15 Radial Velocity plot in m/s

In order to calculate the separation efficiency of the multi-cyclone system, 13100 particles were injected from the inlet surface with velocity equal to that of the continuum phase at a mass flow rate of 1Kg/s to roughly constitute about 30% of the total inflow. These are typical for flue gases expelled from power generating stations.

The particles have been modeled applying the spherical drag law using the Rosin Raimler logarithmic function. The particle density is 600 kg/m^3 and the maximum number of time steps for each injection was 50000 steps.

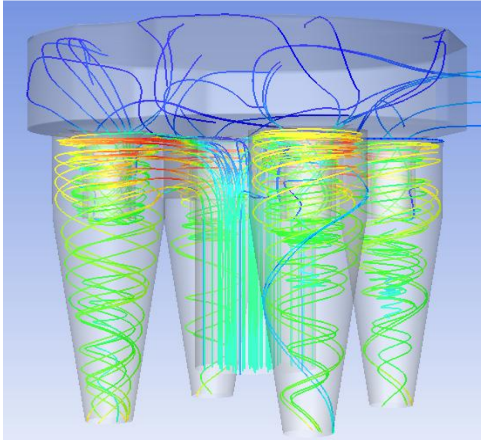


Fig.16 Fluid pathlines in the domain

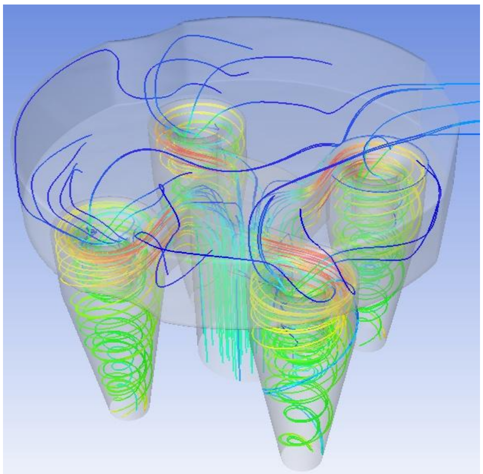


Fig.17 Swirl and turbulence

The DPM computation yields 694 particles escaping from the system outlet implying a separation efficiency of about 94.7 %

VIII. CONCLUSON

A mathematical modeling approach is applied to the simulation of multicyclone separators which could be successfully realized by the use of numerical models, to verify and optimize the geometrical and operating parameter of the collection process. The flow field is in concurrence for the validation case relative to experimental values presented in literature. The extension of the numerical parameters to the computational domain yields positive results in terms of flow field interpretation vis-à-vis velocity profile, pressure distribution along with the transition through the cyclones.

The separation system proposed here as an alternate to the presently used process offer advantage in terms of operating cost, higher efficiency, easy maintenance as well as scalability.

REFERENCES

[1] Khairy Elsayed, Chris Lacor, “*The effect of vortex finder diameter on cyclone separator performance and flow field*” Conputers and fluids, Vol 71 P224-239, Elsevier Publication, June 2010

[2] HoekstraA.J; Derksen J.J, Van Den Akker” *An experimental and numerical study of turbulent swirling flow in gas cyclones*” *Chemical Engineering Science 54 (1999) P2055-2065*

[3] Modern Power Station Practice, Volume B “*Boilers and Ancillary Plant*” *Third Edition British Electricity International London*, Pergamon Press.

[4] Shun.Rand Li Z.Q” *Simulation of strong swirling flow by use of different turbulence model*” *Power Engineering 2002*

[5] B.Wang, D.L.Xu, G.X. Xiao” *Numerical study of Gas-Solid Flow in Cyclone Separator*”, *Third International Conference on CFD*, December 2003.

[6] J.J Derksen, S.Sundaresan, Van Den Akker “*Simulation of mass-loading effects in gas-solid cyclone separators*”, *Powder Technology 163 (2006) P59–68*.

[7] Jolius Gimbun, Thomas Choong, T.G Chuah, A. Fakhru L Razi: *Prediction of the effect of dimension , particle density , temperature and inlet velocity on Cyclone collection efficiency* “*Jurnal Teknologi, 40, P37-50*, June 2004