

SIMULATION OF GAS-SOLID FLOW AND DESIGN MODIFICATIONS OF CEMENT PLANT CYCLONES

V. SINGH¹, S. SRIVASTAVA², R.CHAVAL², V. VITANKAR¹, B. BASU^{1*}, M.C.AGRAWAL²

¹Aditya Birla Management Corporation Ltd, Mumbai, India.

²Grasim Industries Ltd., Cement Business, Mumbai, India.

(Aditya Birla Management Corporation Ltd and Grasim Industries Ltd. are
member companies of Aditya Birla Group)

ABSTRACT

The paper presents Design development of cement industry cyclone based on CFD along with pilot plant trials.

Based on simulation of laboratory scale cyclone as expressed in literature and in-house pilot plant experiments, Reynolds stress model has been found to predict the turbulent flow field, pressure drop and particle collection efficiency closely as compared to other standard turbulence models.

Using the methodology developed, coal mill cyclone of one of the cement plant in India was simulated by CFD. The predicted pressure drop and separation efficiency were compared with measured data to show good agreement. Using the CFD model, the cyclone dimensions were optimized to improve the cyclone separation efficiency without any major change in pressure drop. The new design was successfully implemented in the plant.

NOMENCLATURE :

C_D	drag coefficient
d	particle diameter
U	Velocity
u_i	velocity component
u'	fluctuating velocity
v_p	particle velocity
k	kinetic energy
ϵ	kinetic energy dissipation
ρ	gas density
t	time
μ	viscosity of fluid

INTRODUCTION

In cement manufacturing industries, large-sized cyclone separators are used as main process equipments in significant numbers for handling high volumetric flow rates of dust-laden gases. Typically, the cyclones in a cement plant of capacity about 1 million TPY are 10m tall and 5m in diameter. Their performance is most commonly expressed as the collection efficiency or separation efficiency η , measured as the weight ratio of the dust collected to the dust entering in the cyclone, and the pressure drop. Their operating pressure drop ranges between 100–200 mm Wg. Cyclones efficiencies strongly depend on their design parameters and could vary from 80 % to over 95 % for high efficiency designs, depending upon particle cut-size. These two critical process

parameters are strongly linked to the geometry and the flow patterns in the cyclone.

In its simplest form, a cyclone consists of a cylindrical shell fitted with tangential inlet through which the dusty gas enters; an axial inlet pipe for discharging the cleaned gas, and a conical base with dust discharge. The main objective in the cyclone is to create a vortex which will centrifuge the dust particles to the walls; whence they can be transported into the dust collecting hopper out of the influence of the spinning gases through the boundary layer.

In order to understand the important features of a well designed cyclone, it is necessary first to study the flow pattern and particle separation efficiency. Prediction of gas flow and performance of cyclone by Computational Fluid Dynamics (CFD) have become increasingly popular. CFD helps to understand the complex flow patterns in cyclones and develop efficient cyclone with low pressure drops and high separation efficiency.

The conventional method of predicting the flow field and the collection efficiency of a cyclone is empirical (Stairmand (1951) Leigh and Licht (1972), Muschelknautz, and Greif, (1997)). Computation fluid dynamics (CFD) has great potential to predict the flow field characteristics and particle trajectories inside the cyclone as well as the pressure drop (Griffins and Boyson (1996)).

One of the first CFD simulations was done by Boysan et al. (1982). He found that the standard $k-\epsilon$ turbulence model is inadequate to simulate flows with swirl because it leads to excessive turbulence viscosities and unrealistic tangential velocities. The standard $k-\epsilon$, RNG $k-\epsilon$ and realizable $k-\epsilon$ model was not optimized for strongly swirling flows found in cyclones (Gimbin et.al (2005)). However, to reduce the computational effort, the RNG $k-\epsilon$ model can be used with about 12 % deviation on experimental data (Griffins and Boyson, 1996). The pressure-strain correlation in Reynolds stress model (RSM) with linear model by Laufer, Reece and Rodi (LRR model) (Laufer et. al, 1975) under-predicts the tangential velocity component close to the axis and with the quadratic model by Speziale, Sarkar and Gatski (RSM model) (Speziale et. al. 1991) is qualitatively correct (Montaven et. al., 2000)

If the inlet duct is ignored, the cyclone geometry is almost axi-symmetric and a number of past CFD models have utilized this feature to reduce the model to a two-dimensional case (Dugins and Frith, 1987, Hoffman et al., (1996), Meina and Mori (1999)). Although this greatly

* Corresponding author

reduces computational time, a two-dimensional model will be limited, as the inlet location will break flow pattern symmetry. Furthermore such geometric simplifications cannot be used to assess changes to inlet design nor offset vortex finders.

Particle turbulent dispersion due to interaction between particles and turbulent eddies of fluid is generally dealt with by two methods (Loth, 2000); mean diffusion which characterizes only the overall mean (time averaged) dispersion of particles caused by the mean statistical properties of the turbulence, and structural dispersion which includes the detail of the non-uniform particle concentration structures generated by local instantaneous features of the flow, primarily caused by the spatial-temporal turbulent eddies and their evolution. To predict the mean particle diffusion in turbulent flow, both Lagrangian and Eulerian techniques can be used. Since the early work of Yuu et al. (1978) and Gosman and Ioannides (1981), the stochastic Lagrangian model has shown significant success in describing the turbulent diffusion of particles.

In the industry, the major challenge is to increase production and reduce manufacturing cost. In order to achieve these objectives, cyclones are operated well beyond the design conditions resulting in the increased pressure drop and reduced separation efficiency. In the present work, a systematic pilot-scale experimental studies and CFD simulation of cyclone were undertaken to understand the flow pattern and the complex interaction between gas-solid flow to improve the cyclone performance, i.e., higher solid separation and reduce pressure drop.

The objective of this work is to develop a CFD model to understand the flow patterns, predict cyclone performance for laboratory scale, pilot scale and industrial scale cyclone and its application to carry out design modification of industrial cyclones. In order to select appropriate turbulence models to predict swirling turbulent flow in the cyclone, experimental data of laboratory scale cyclone was used and pilot plant experiments were carried out. A CFD simulation methodology was developed for design of industrial cyclones. A number of cyclone designs have been successfully modified and implemented using the above concept.

MODEL DESCRIPTION

To predict the gas flow pattern and particle collection efficiency, in cyclones the governing equation consists of the continuity and momentum balance equation for the gas phase and the particles are tracked in the Lagrangian manner.

Gas Phase

Continuity equation for the gas phase is written as:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = 0 \quad (1)$$

The momentum balance equation for the continuous phase, gases flowing through the cyclone, is written as:

$$\frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho U \otimes U) = -\nabla p + \nabla \cdot \mu \nabla U - \nabla \cdot (\overline{\rho u_i u_j}) \quad (2)$$

The last term in the above equation requires closure in terms of turbulence models. A very commonly used turbulence model is standard k-ε model. However, due to its isotropic nature, it does not capture the turbulent swirling flow. In such case, the full Reynolds stress model is recommended. The standard Reynolds stress model in ANSYS CFX is based on the ε- equation.

$$\begin{aligned} \frac{\partial}{\partial t} (\overline{\rho u_i u_j}) + \frac{\partial}{\partial x_k} (U_k \overline{\rho u_i u_j}) &= P_{ij} + \phi_{ij} + \\ \frac{\partial}{\partial x_k} \left[\left(\mu \delta_{kl} + C_s \rho \frac{k}{\varepsilon} \overline{u_k u_l} \right) \frac{\partial u_i u_j}{\partial x_l} \right] &- \frac{2}{3} \delta_{ij} \rho \varepsilon \end{aligned} \quad (3)$$

$$P = -\rho (\overline{u \otimes u} (\nabla U)^T + (\nabla U) \overline{u \otimes u})$$

The equation for kinetic energy dissipation is given by:

$$\begin{aligned} \frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_k} (\rho U_k \varepsilon) &= \frac{\varepsilon}{k} (C_{\varepsilon 1} P - C_{\varepsilon 2} \rho \varepsilon) \\ + \frac{\partial}{\partial x_k} \left[\left(\mu \delta_{kl} + C_s \rho \frac{k}{\varepsilon} \overline{u_k u_l} \right) \frac{\partial \varepsilon}{\partial x_l} \right] \end{aligned} \quad (4)$$

where,

$$\phi_{ij} = \phi_{ij1} + \phi_{ij2}$$

$$\phi_{ij1} = -\rho \varepsilon \left(C_{s1} a + C_{s2} \left(a a - \frac{1}{3} a \bullet a \delta \right) \right)$$

$$\phi_{ij2} = -C_{r1} P a + C_{r2} \rho k S - C_{r3} \rho k S \sqrt{a \bullet a}$$

$$+ C_{r4} \rho k \left(a S^T + S a^T - \frac{2}{3} a \bullet S \delta \right)$$

$$+ C_{r5} \rho k (a W^T + W a^T)$$

$$a = \frac{\overline{u \otimes u}}{k} - \frac{2}{3} \delta$$

$$S = \frac{1}{2} (\nabla U + (\nabla U)^T)$$

$$W = \frac{1}{2} (\nabla U - (\nabla U)^T)$$

where a is the anisotropy tensor, S is the strain rate, and W is the vorticity

Particle transport

To predict the particle capture efficiency, it is necessary to solve for the particle phase. In this work, it is assumed that the particle transport is affected by the gas flow however, the gas phase remains unaffected by particle phase momentum. The particle transport is given by the following equation which considers the drag and buoyancy forces:

$$m_p \frac{dv_p}{dt} = \frac{1}{8} \pi \rho_f d^2 C_D |v_f - v_p| (v_f - v_p) + \frac{1}{6} \pi d^3 (\rho_p - \rho_f) g \quad (5)$$

The application of Lagrangian tracking in ANSYS CFX involves the integration of particle paths through the discrete domain. In the present case, the parameters for particle tracking and integration were carefully selected.

SIMULATIONS OF A LABORATORY SCALE CYCLONE SEPARATOR

Ayers et. al. (1983) have measured axial and tangential velocities in a laboratory scale cyclone using LDA. The cyclone is 0.205 m in diameter and 0.82 m tall.

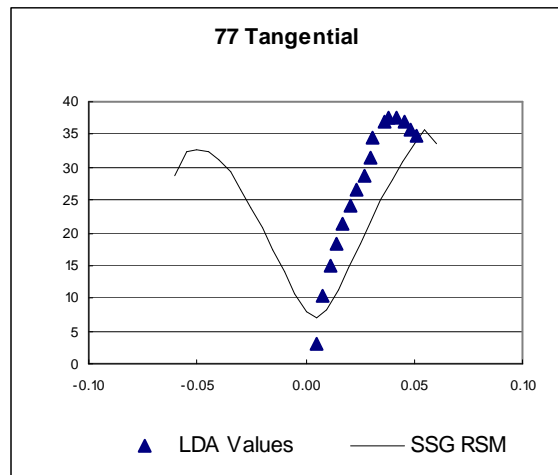
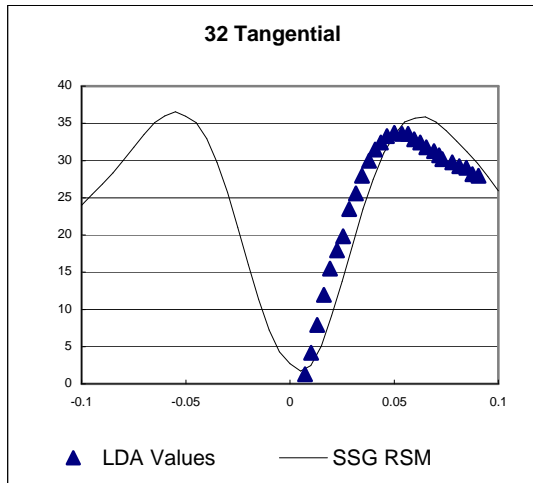
In this work, the simulation of gas flow in the cyclones was carried out using two different turbulent models the k-ε model and the Reynolds stress model (Launder et.al., 1975). 66,000 grid nodes were used along with tetrahedral meshes. The predicted axial and tangential velocity components have been compared with the experimental

data of Ayers et al. (1983). Due to anisotropic nature of the turbulent flow in cyclone, the simulation with k-ε model failed to predict the measured profile of axial and tangential velocity profiles (these are not presented here). The results from the Reynolds stress model are presented in Figures 1 and 2. It can be seen that predicted tangential and axial velocity are in close agreement with the measured values.

It can be concluded that Reynolds stress model is able to capture key features of turbulent swirling flows in the cyclone separator. Harwood & Slack (2003), Witt et. al. (1999) have also recommended Reynolds stress model for the flows in cyclone.

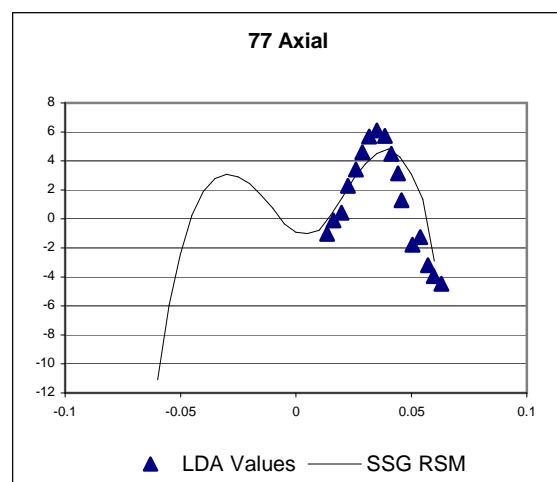
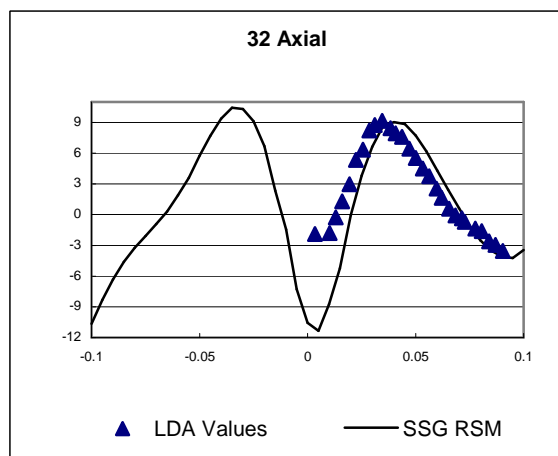
PILOT SCALE CYCLONE: EXPERIMENT AND SIMULATION

In order to further validate the CFD model at larger scale, a pilot scale cyclone was developed, and the pressure drop and particle separation efficiency were studied and compared with the predicted values. The pilot cyclone was fabricated and commissioned in-house. The geometry of pilot plant is shown in Figure 3.



i) Location 32 cm from the cyclone top ii) Location 77 cm from the cyclone top

Figure 1. Comparison of predicted and measured (Ayers et al., 1983) tangential velocity component.



i) Location 32 cm from the cyclone top ii) Location 77 cm from the cyclone top

Figure 2. Comparison of predicted and measured (Ayers et al., 1983) axial velocity component

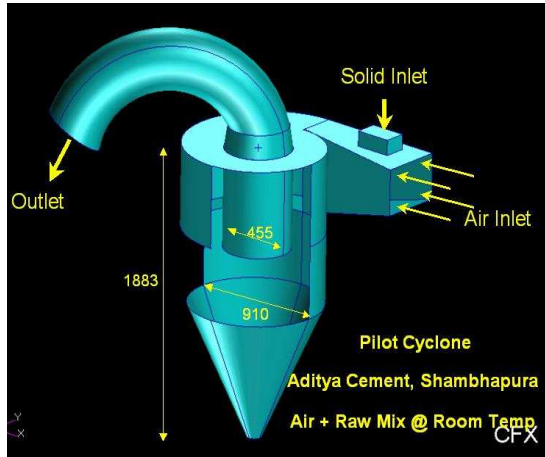


Figure 3 (i). The geometry of the pilot cyclone

The cyclone is about 1.88 m in height and 0.91 m in diameter. As seen in the Figure 3 (ii), an ID fan is used to suck air in the cyclone separator from the inlet, which is open to the atmosphere. Solid particles are fed through the feed silo with a capacity 1 MT of raw mix solid powder. The rotary air lock valve was used to control the solid feed rate.

The coarse solid particles were collected at the bottom and the fine particles which did not get arrested, left the cyclone through the vortex tube. The solid was fed from the feed silo to the cyclone through a rotary air lock valve. The rotary air lock valve speed was controlled by a variable frequency drive, which in turn controls the solid particle feed rate.

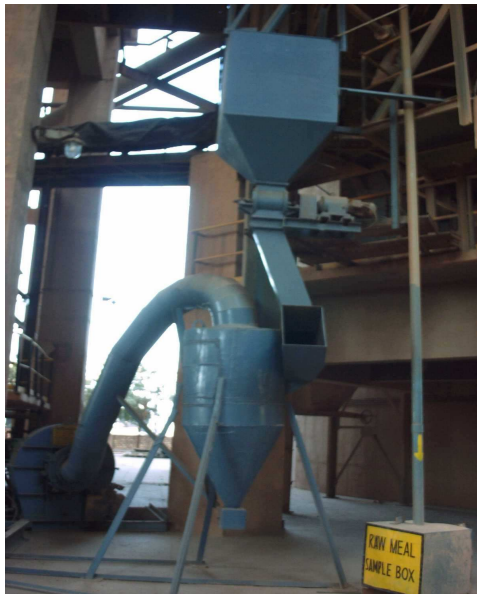


Figure 3 (ii) Pictorial view of pilot cyclone

The pressure drop across the cyclone separator was measured by pressure transmitters. The airflow rates were calculated based on the velocity measured at the inlet of the cyclone separator. The velocities were measured at different points in a plane near the inlet using the vane meter.

Experiments were carried out at 1.8kg/s of air flow with cement raw mix particles. The particle size distribution analysis of the raw mix powder was carried out using silas

particle size analyzer. Following Rosin Rammler parameters as obtained are:

$$\text{Rosin Rammler power} = 0.83,$$

$$\text{Rosin Rammler size} = 1.44 \mu$$

For different solid feed rates, collection efficiency and the pressure drop of the cyclone were measured experimentally. The experimental observations have been tabulated in Table 1. For these case, the pressure drop evaluated falls in the range 125-135 mmWC.

Trial No	Feed Rate, Kg/s	Collection Efficiency %
1	0.197	89.0 %
2	0.208	87.8 %
3	0.190	88.1 %
Average. Efficiency		88.3 %

Table 1 Experimental results on cyclone efficiency

SIMULATIONS OF THE FLOW IN THE PILOT CYCLONE

Using experimental data with one feed rate, the particle-wall interaction parameters have been obtained in such a way that the predicted particle efficiencies match with the measured one. For simulation and validation of the CFD predictions using average feed rate, these particle-wall interaction parameters were used

Sensitivity of meshes:

Simulations were carried out with two meshes to understand mesh sensitivity on pressure drop and particle efficiency. The results have been summarized in Table 2.

As can be observed, the predicted pressure drop and particle collection efficiency remained close when the number of nodes were increased from 50,000 to 84,000.

Effect of turbulence models:

The CFD simulation was carried out with different turbulence, k-ε, shear stress transport (SST) and the Reynolds stress model (RSM). In the earlier section while comparing with the measured gas flow in the cyclone, it was established that the Reynolds stress model (RSM) was suitable to predict swirling flow in the cyclone. In the simulation of pilot plant experiments, the effect of different turbulence models on particle collection efficiency has also been studied.

Details	Mesh I	Mesh II
Mesh details: Nodes	50,000	
Elements	1.8 Lacs	84,000 3.2 Lacs
Air flow rate, kg/s	2.6	2.6
Pressure Drop, mm Wg	259	261
Particle efficiency, wt%	96.1%	95.9%

Table 2. Mesh sensitivity studies

Figures 4 and 5 show the CFD predictions of the tangential and the axial velocities by different turbulent models just below the vortex tube.

The profiles of tangential and axial velocities were compared with the typical velocity profiles as observed in the cyclone. Figures 1 and 2 present such velocity profiles. Following inferences can be made from the above comparative study with different turbulence models:

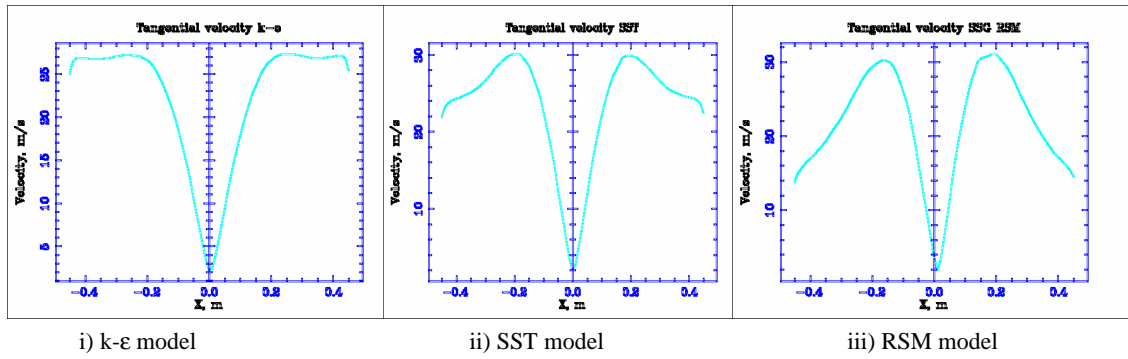


Figure 4. Predictions of tangential velocities using different turbulence model

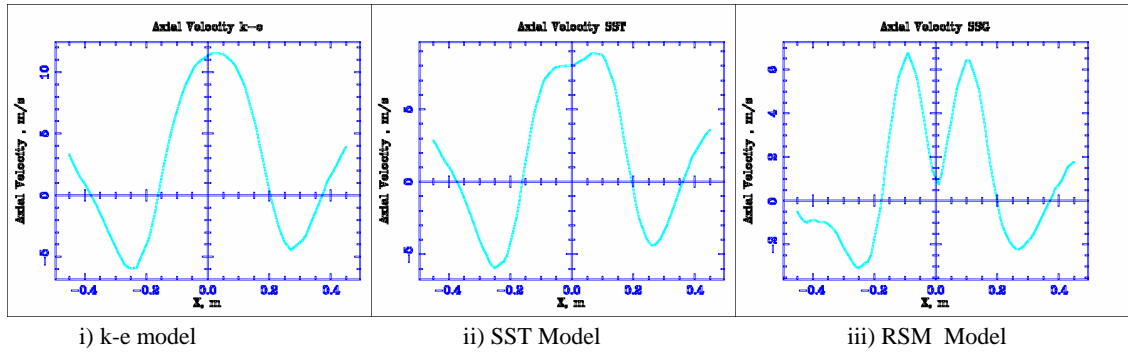


Figure 5 Prediction of axial velocity component using different turbulence model

- The tangential and the axial velocity profile predicted by the RSM model showed gradual decrease towards the wall as observed in the cyclones (Ayers et al. 1983). k-ε and SST models predicted a larger spread of tangential velocities in the radial direction. The predicted tangential velocity with SST and RSM were close (~ 30 m/s) while k-ε showed marginally higher value.
- Due to the isotropic nature, k-ε and SST models did not predict the dip at the center of the cyclone for axial velocity distribution as observed experimentally (Ayers et.al, 1983). RSM model effectively captured this characteristic closely.

The pressure drop and particle collection results of the simulations for k-ε, SST and RSM models are given in the Table 3 for flow rates of 1.8 kg/s.

The measured pressure drop in the cyclone is in the range of 125 – 135 mm Wg. The CFD predicted values of RSM model were 141 mm Wg. The observation from this simulation study and comparisons are summarized below:

	Pressure Drop across the cyclone mm Wg	Particle collection efficiency (%)
Experiment	125-135	88.3
k-ε	179.7	94.8
SST	182.4	92.7
RSM	141.0	91.6

Table 3. Comparisons of Pressure Drop and particle efficiency predictions of various turbulent models

- Isotropic models such as k-ε, SST, over predicted the pressure drops in the cyclone separator.
- The pressure drop prediction of RSM model, though slightly higher, was closest to the measured values.
- The particle efficiency predicted by RSM and SST were closest to the measured values whereas those by k-ε model were over-predicted
- The RSM model predicts both the pressure drop and the collection efficiency close to measured value.

RE-DESIGN OF COAL MILL CYCLONE

Having benchmarked with laboratory and pilot scale cyclone separator, the developed CFD model has been used for performance evaluation and design development of a coal mill cyclone. The cyclone was used as a pre - collector in coal mill circuit before the coal mill bag filter. The following problems have been continuously persisting since last few years

- High Pressure drop across the cyclone
- Poor cyclone collection efficiency
- High pressure drop across the coal mill bag filter
- Low life of bag filter bags due to high inlet dust loading

To address the above problem, a new cyclone has been designed for flow of 72000 m³/hr of air. In order to develop a compact cyclone, a vortex breaker is used in the present design which enforces flow reversal and also creates low velocity zone at the bottom where materials are collected. As per the in-house cyclone design experience, the dimensions of various parts in cyclones were selected with the following design options: Figure 6 shows the sketch of the cyclone with vortex breaker.

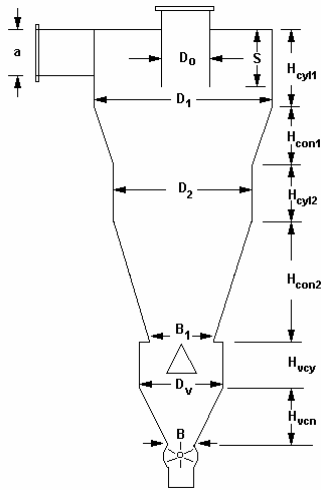


Figure 6. Schematic configuration of a coal mill cyclone.

Option I: Cyclone was designed for above flow rate maintaining the inlet velocity of **14 m/s**.

Option II: Keeping other dimensions same as option-I, the cylindrical height is increased up to dip tube height, to reduce the short-circuiting. In option-I dip tube was extended into the conical section

Option III: Cyclone designed for above flow rate maintaining the inlet velocity of **9 m/s**. With same design criterion of option-I, to reduce pressure drop across cyclone, the diameter of the cylindrical portion was increased and hence all the subsequent dimensions were changed

The performance of these new cyclone designs was predicted through CFD simulation to select the best option before implementation in the plant. Studies were carried out for coal and petcoke as the trend in cement industry is to replace coal with petcoke. Table 4 summarises the particle size distribution for coal and pet coke.

The results obtained were analysed to understand the flow patterns in different cyclone configurations considered.

Particulars	South African coal	Pet coke
Particle size distribution		
1-8 μm	8.60	28.87
1-45 μm	55.48	84.30
1-90 μm	85.20	97.40
1-212 μm	99.70	99.98
Particle flow rate	6.0	4.20

Table 4: Particle details

Figure 7 shows the velocity vector in the vertical plane close to the dip tube. It has been observed that minor short-circuiting is taking place around the vortex finder for option-I. In this case, the vortex finder is extended beyond the cylindrical section and within the conical section. This results in accelerating swirling flow at the entrance of vortex finder causing short-circuiting as can be seen from the Figure 7. Based on this finding, the cylindrical section of the cyclone was increased to match the height of the dip

tube in the option II and diameter was increased in the option III.

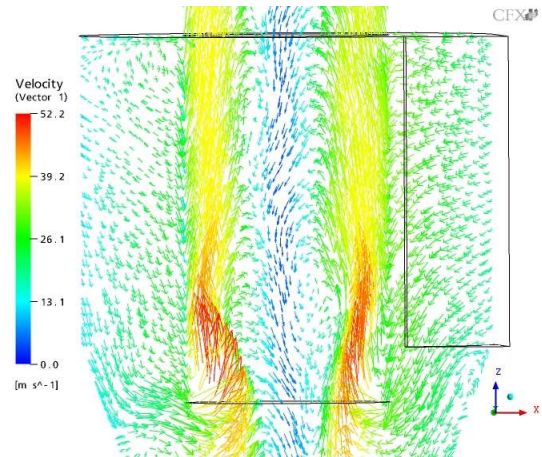


Figure 7. Velocity vector in the base case cyclone close to the vortex finder

Figure 8 shows the velocity contour in the vertical plane. The central blue colored region is a reverse vortex and the yellow colored region is the outer vortex. The velocity gradient between the outer vortex and the inner vortex is smaller in option I where as it is higher in option II where the cylindrical section height has been increased. The red colored region close to the vortex finder indicates short-circuiting. Re-circulation zones can be seen inside the cyclone in all the three options. It can be observed that short circuiting is quite high in option II as compared to option I and minimum in option III.

Figure 9 presents streamline and velocity distribution of gas flow close to the vortex breaker with the option I. The streamline plot shows that a swirl below the vortex breaker has considerably lower strength as can be seen from the velocity magnitudes. Figure 9 (ii) presents the velocity contours at the two locations above and below the vortex breaker to show that it reduces the swirl significantly.

Maximum cut size of particle

Based on the spinning velocity of the fluid, there is a critical size of the particle, of the given density on which the centrifugal and the inward viscous forces just balance so that the particle moves neither outwards to the walls nor inwards to the cyclone axis. All particles greater than this critical size would get collected and all smaller ones would escape. This critical size of the particle is called cut size of the particle. The cut size of the particles was evaluated in all the simulations and has been tabulated in Table 5. It will be observed that the cut size for option I and II is the same, 65 micron whereas for the larger cyclone, option III, the cut size predicted is marginally higher, 69 microns. The cyclone performance parameters were evaluated for all the cases studied. Table 5 summaries the findings for the case with imported coal and petcoke. It can be seen that the option I design predicted higher efficiency (80%) as compared to the other two. As expected, the pressure drop is the lowest for option III as it is the cyclone with smaller inlet velocity and larger size. Similarly for petcoke, option I gives better separation efficiency but has

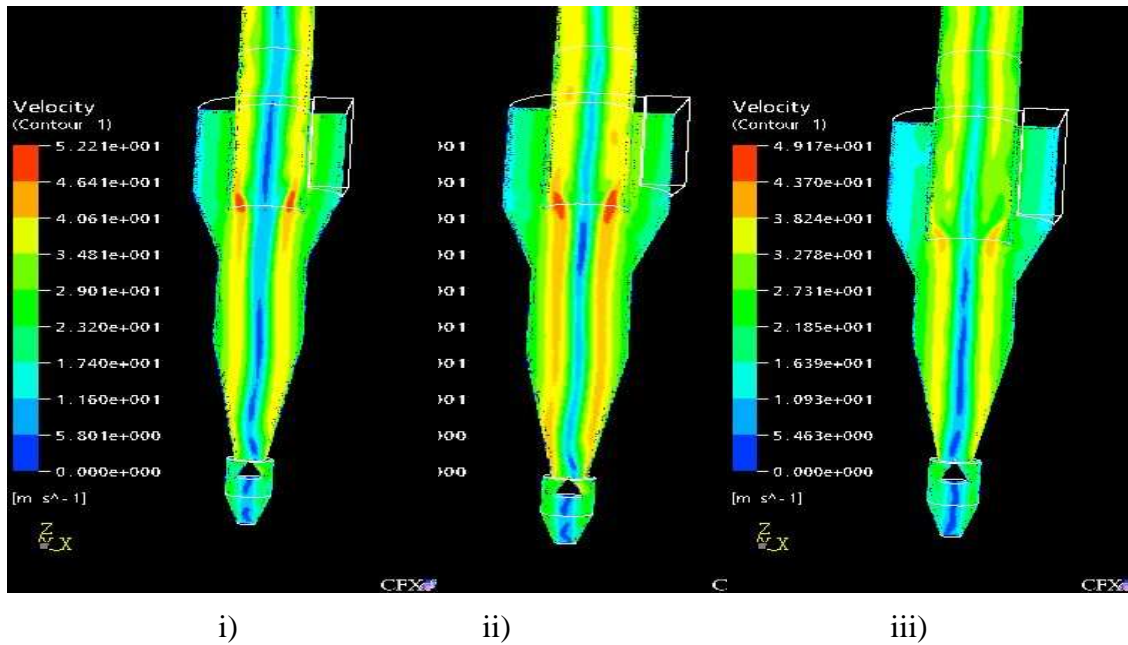


Figure 8. The velocity distribution in different design of cyclones, i) Option I, ii) Option II and iii) Option III.

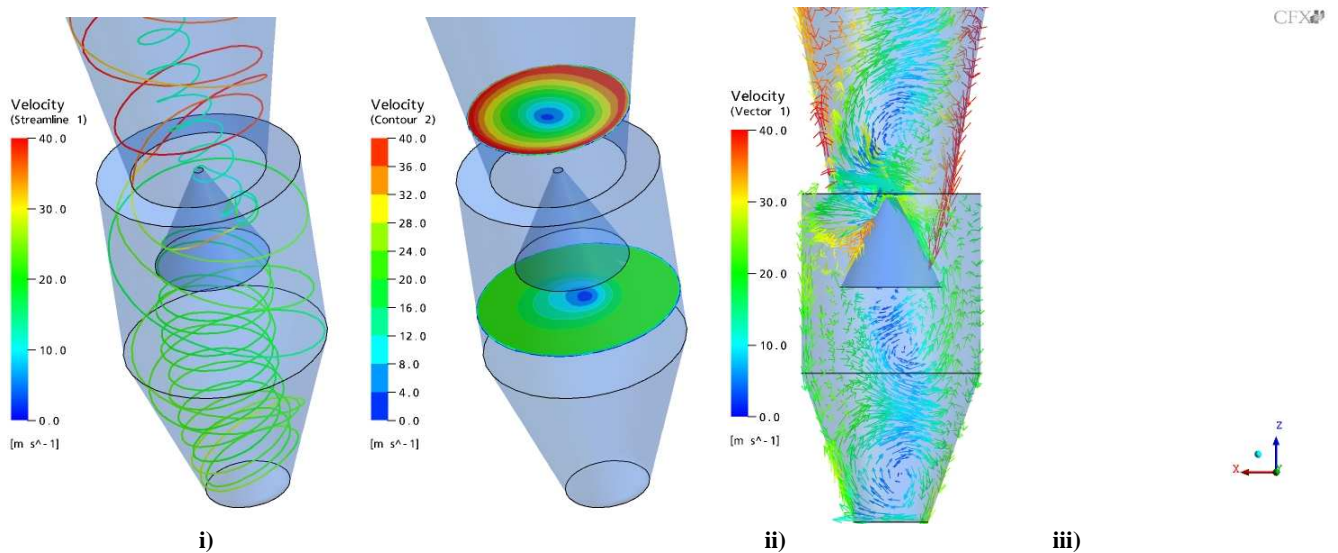


Figure 9. Flow field near the vortex breaker with i) streamline ii) velocity contour and iii) velocity vector

	Option I	Option II	Option III
Pressure drop (mm Wg)			
Coal	85	90	70
Pet coke	85	90	70
Particle collection efficiency (%)			
Coal	80	78	72
Pet coke	60	58	55
Particle Cut Size (μ)			
Coal	65	65	69
Pet coke	65	65	69

Table 5. Predicted Cyclone performance for South African coal and Petcoke

slightly higher pressure drop. From the point of particle efficiency, option I was recommended for implementation. Upon implementation, pressure drop was observed to be 90 mm Wg and the particle collection efficiency was 81% both of which were close to CFD predictions.

CONCLUSION

In this paper, CFD simulation of different scales of cyclone separators are presented. Using experimental data from laboratory and pilot scale cyclones, the predictions of velocity field, pressure drop and particle separation efficiency from the CFD model were critically compared with different turbulence models, namely, k-e, SST and the Reynolds stress models. It was found that the Reynolds stress model predicts the cyclone flow close to the experimental observation. Using pilot plant experimental data, a methodology of predicting particle separation efficiency in large industrial scale cyclones was

established. Using the CFD model as developed, coal mill cyclone of one of the cement plants has been designed to reduce pressure drop without affecting the separating efficiency. A number of design alternatives (with different length, diameter and location of vortex finders) were tested based on simulation and the optimal design was selected. The cyclone as per the recommended design was fabricated and installed in the plant. The performance of the cyclone (i.e., pressure drop and efficiency) was found to be close to the predicted values by CFD.

A total of 10 cyclones have so far been re-designed using CFD and successfully implemented in the plant to show performance improvement.

ACKNOWLEDGEMENT

The authors would like to acknowledge support and encouragement from Mr. S. K. Maheshwari, Group Executive President, Grasim Industries Ltd, Cement Business, to carry out pilot scale trials and plant scale implementation of this work. The authors also acknowledge encouragement and technical inputs from Dr. Hameed Bhombal, Senior President, Aditya Birla Management Corporation Ltd during different stages of the project.

REFERENCES

- AYERS, W. H., BOYSAN, F., SWITHEBANK, J AND EWAN, B.C.R (1983). Theoretical Modelling of Cyclone Performance. Filtech Conference
- BOYSAN, F. AYER, W. H., SWITHEBANK, J. A, "Fundamental mathematical modelling approach to cyclone design", Trans. Inst. Chem. Eng. 60 (4) (1982) 222-230.
- DUGGINS R. K. AND FRITH P. C. W., "Turbulence Anisotropy in cyclone", Filtration and Separation, 24, (1987), 394-397.
- GIMBIN, J., CHUAH, T. G., FAKHRU'L-RAZI, A., THOMAS, S. Y., CHOONG, "The influence of temperature and inlet velocity on cyclone pressure drop: a CFD study", Chemical Engineering and processing 44 (2005), 7-12.
- GOSMAN, A.D., IOANNIDES, E, Aspects of computer simulation of liquid-fuelled combustors. AIAA 81-0323, 1981.

GRIFFINS, W. D. AND. BOYSON, F "Computational fluid dynamics and empirical modeling of the performance of a number of cyclone samplers", J. Aerosol Sci. 27, (1996), 281-304.

HARWOOD, R. SLACK, M., CFD analysis of a cyclone, QNET-CFD Network Newsletter, Volume 1, No. 4 – November 2002

HOFFMANN A. C., DE GROOT, M. AND HOSPERS, A., "The Effect of Dust Collection System on the Flow pattern and Separation Efficiency of a Gas Cyclone", Can. J. Chem Eng. 74, (1996), 464-470.

LAUDER, B. E., REECE, G. J., RODI, W., "Progress in the development of Reynolds stress turbulence closure", J. Fluid Mech., 68 (1975), 537-566.

LEIGH, D. AND LICHT, W., "The collection efficiency of cyclone type particle separators- A new theoretical approach", AIChE Symp. Series 68, (1972), 196-206.

LOTH, E. Progr. Energy Combust. Sci. 26 (2000) 161-223.

MEINER, H. F. AND MORI, M., "Anisotropic behavior of the Reynolds stress in gas and gas-solid flows in cyclones", Powder Tech., 101, (1999), 108-119.

MONTAVON, C. A., GROTTJANS, H , HAMILL, I S, PHILLIPS, H W AND JONES, I P. "Mathematical modeling and experimental validation of flow in cyclones", BHR Conference on cyclone technologies, Warwick, 2000.

MUSCHELKNAUTZ, E., AND GREIF, V., "Cyclones and other gas-solids separators", Circulating Fluidised Beds, J. R., Grace, A. A. Avadin, and T. Knowlton, Eds. 1997, Chapman-Hall: London, 181-213.

SPEZIALE, C. G., SARKAR, S., GATSKI, T. B., "Modelling the pressure strain correlation of turbulence: an invariant dynamical system approach", J. Fluid Mech., 227 (1991), 245-272.

STAIRMAND, C. J., "The Design and performance of cyclone separators", Tran. Instn Chem. Egrs. 29, (1951), 356-373.

WITT, P. J , MITTONI, L. J , WU, J., SHEPHERD, I. C, Validation of a CFD Model for predicting gas flow in a cyclone. CHEMECA99, 26-29 September 1999, Newcastle, Australia

YU, S., HIROSAWA, N. Y, AIChE J. 24 (1978) 509-519