

# Hydrodynamic Simulation of Cyclone Separators

Utikar<sup>1</sup>, R., Darmawan<sup>1</sup>, N., Tade<sup>1</sup>, M., Li<sup>1</sup>, Q, Evans<sup>2</sup>, G.,  
Glenny<sup>3</sup>, M. and Pareek<sup>1</sup>, V.

<sup>1</sup>*Department of Chemical Engineering, Curtin University of Technology, Perth, WA 6845,*

<sup>2</sup>*Centre for Advanced Particle Processing, University of Newcastle, Callaghan, NSW 2308,*

<sup>3</sup>*BP Kwinana Refinery Pty Ltd, Mason Road, Kwinana, WA 6167,  
Australia*

## 1. Introduction

Cyclone separators are commonly used for separating dispersed solid particles from gas phase. These devices have simple construction; are relatively inexpensive to fabricate and operate with moderate pressure losses. Therefore, they are widely used in many engineering processes such as dryers, reactors, advanced coal utilization such as pressurized and circulating fluidized bed combustion and particularly for removal of catalyst from gases in petroleum refinery such as in fluid catalytic cracker (FCC). Despite its simple operation, the fluid dynamics and flow structures in a cyclone separator are very complex. The driving force for particle separation in a cyclone separator is the strong swirling turbulent flow. The gas and the solid particles enter through a tangential inlet at the upper part of the cyclone. The tangential inlet produces a swirling motion of gas, which pushes the particles to the cyclone wall and then both phases swirl down over the cyclone wall. The solid particles leave the cyclone through a duct at the base of the apex of the inverted cone while the gas swirls upward in the middle of the cone and leaves the cyclone from the vortex finder. The swirling motion provides a centrifugal force to the particles while turbulence disperses the particles in the gas phase which increases the possibility of the particle entrainment. Therefore, the performance of a cyclone separator is determined by the turbulence characteristics and particle-particle interaction.

Experimental and numerical studies have been carried out in the last few decades to develop a better understanding of the flow field inside the cyclone separators. In the early years, empirical models were built (e.g. Shepherd & Lapple, 1939; Lapple, 1951; Barth, 1956; Tengbergen, 1965; Sproul, 1970; Leith & Licht, 1972; Blachman & Lippmann, 1974; Dietz, 1981 and Saltzmann, 1984) to predict the performance of industrial cyclones. However, these models were built based on the data from much smaller sampling cyclones therefore; they could not achieve desired efficiency on industrial scales as the industrial cyclone operates in the turbulent regime while sampling cyclones operate under the transitional conditions. One of the major drawbacks of these empirical models is the fact that they ignore two critical factors that determine the performance of a cyclone namely the unsteadiness and asymmetry. Many flow phenomena such as high turbulence, flow reversal, high

Source: Computational Fluid Dynamics, Book edited by: Hyoung Woo OH,  
ISBN 978-953-7619-59-6, pp. 420, January 2010, INTECH, Croatia, downloaded from SCIYO.COM

vorticity, circulating zones and downflow also occur. The empirical models do not include these phenomena in their analysis and hence are limited in their application. Computational fluid dynamics (CFD) models on the other hand can accurately capture these aspects and thus can take a significant role in analyzing the hydrodynamics of cyclone separators. A validated CFD model can be a valuable tool in developing optimal design for a given set of operating conditions. However, cyclone separators pose a peculiar fluid flow problem. The flow in cyclone separators is characterized by an inherently unsteady, highly anisotropic turbulent field in a confined, strongly swirling flow. A successful simulation requires proper resolution of these flow features. Time dependent turbulence approaches such as large eddy simulation (LES) or direct numerical simulation (DNS) should be used for such flows. However, these techniques are computationally intensive and although possible, are not practical for many industrial applications. Several attempts have been made to overcome this drawback. Turbulence models based on higher-order closure, like the Reynolds Stress Model, RSM, along with unsteady Reynolds averaged Navier - Stokes (RANS) formulation have shown reasonable prediction capabilities (Jakirlic & Hanjalic, 2002). The presence of solids poses additional complexity and multiphase models need to be used to resolve the flow of both the phases.

In this chapter we review the CFD simulations for cyclone separators. Important cyclone characteristics such as the collection efficiency, pressure and velocity fields have been discussed and compared with the experimental data. Several significant parameters such as the effect of geometrical designs, inlet velocity, particle diameter and particle loading, high temperature and pressure have also been analysed. The chapter discusses peculiar features of the cyclone separator and analyses relative performance of various models. Finally an example of how CFD can be used to investigate the erosion in a cyclone separator is presented before outlining general recommendations and future developments in cyclone design.

## 2. Basic design of cyclone separators

A cyclone separator uses inertial and gravitational forces to separate particulate matter from gas. Accordingly various designs have been proposed in literature (Dirgo & Leith, 1986). Figure 1 shows a schematic of widely used inverse flow cyclone and depicts main parts and dimensions. The particle laden gas enters the cyclone separator with a high rotational velocity. Different inlet configurations like tangential, scroll, helicoidal and axial exist to provide high rotational velocity. Of these, the tangential and scroll configurations are most frequent. The rotational flow then descends near the wall through the cyclone body and conical part until a reversal in the axial velocity making the gas flow in the upwards direction. Where this occurs is called as the vortex end position. The upward rotating flow continues along the cyclone axis forming a double vortex structure. The inner vortex finally leads the flow to exit through a central duct, called the vortex finder. The vortex finder protrudes within the cyclone body. It serves both in shielding the inner vortex from the high inlet velocity and stabilizing its swirling motion. The solids are separated due to the centrifugal force and descend helicoidally along the cyclone walls and leave the equipment through the exit duct.

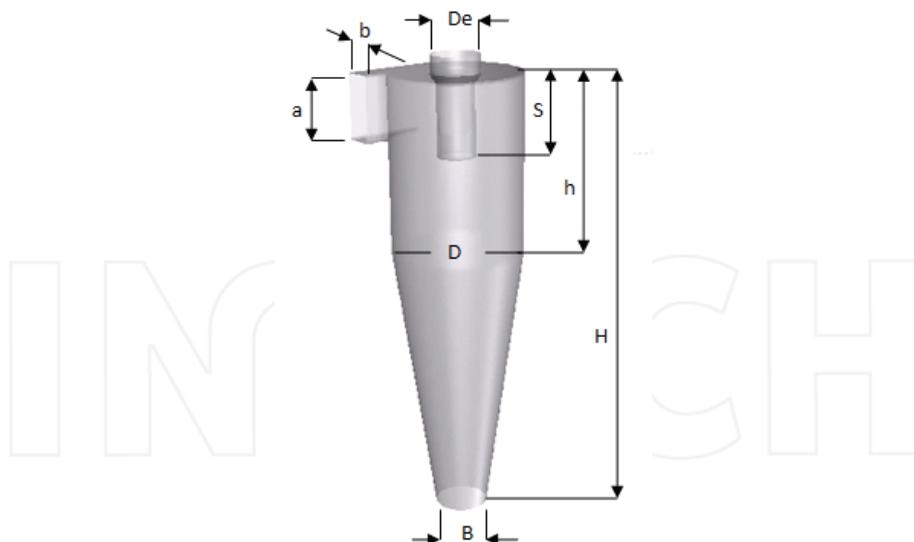


Fig. 1. Typical design of cyclone separator

Source	Stairmand (1951)	Stairmand (1951)	Lapple (1951)	Swift (1969)	Swift (1969)	Swift (1969)
Duty	High efficiency	High throughput	General purpose	High efficiency	General purpose	High throughput
D	1	1	1	1	1	1
a/D	0.5	0.75	0.5	0.44	0.5	0.8
b/D	0.2	0.375	0.25	0.21	0.25	0.35
De/D	0.5	0.75	0.5	0.4	0.5	0.75
S/D	0.5	0.875	0.625	0.5	0.6	0.85
h/D	1.5	1.5	1	1.4	1.75	1.7
H/D	4	4	4	3.9	3.75	3.7
B/D	0.375	0.375	0.25	0.4	0.4	0.4

Table 1. Standard Geometrical Design of Industrial Cyclone Separator

For convenience, the dimensions of various cyclone parts are usually stated in dimensionless form as a ratio to the cyclone diameter,  $D$ . This method allows a comparison between the cyclone designs, without using the actual size of each individual part. Table 1 lists a few examples of industrial cyclone types (Leith and Licht, 1972). A more comprehensive range of designs can be found in Cortes and Gil (2007). The performance of a cyclone separator is measured in terms of the collection efficiency defined as the fraction of solids separated and the pressure drop. By nature, the flow in a cyclone separator is multiphase (gas–solid) and shows strong gas–solid–solid interactions. The gas–solid interactions can only be neglected at very low solid loadings. Early CFD models focused on single phase flow and turbulence interactions inside the cyclone. Multiphase CFD simulations that account for the gas–solid and gas–solid–solid interactions and its immediate results concerning cut sizes and grade-efficiency are relatively scarce in the

literature. The subsequent sections discuss available CFD models and their predictive capabilities with respect to the flow field, pressure drop and collection efficiency.

### 3. Computational fluid dynamics models for cyclone separators

The flow inside a cyclone separator is inherently complex and poses many practical difficulties for numerical simulations. The primary difficulty arises from the fact that the turbulence observed in cyclones is highly anisotropic. This renders most of the first order turbulence closures, like the popular  $k-\epsilon$  model, unusable for reliable prediction of the flow characteristics. Several attempts were made to overcome this limitation. Boysan et al. (1982, 1983) were one of the first to report CFD studies of cyclone flows. These early studies realized that the standard  $k-\epsilon$  turbulence model is not able to accurately simulate this kind of flow and that at least a second-order closure, e.g., RSM is needed to capture the anisotropy and achieve realistic simulations of cyclone flows. The authors found reasonable agreement between the experimental data and simulations using a mixed algebraic-differential, stationary RSM. Many studies have since been performed to capture the turbulence characteristics accurately. The next section will review these in detail.

Selection of numerical parameters, especially the discretization of the advection terms, poses an additional difficulty and plays an important role on the accuracy of simulations. First order discretization is prone to numerical diffusion and often produces misleading results in cyclone separator simulations. The use of hexahedral grids for the main flow region (Harasek et al., 2004) and a second order accurate advection scheme (Bunyawanchakul et al. 2006) has shown a significant improvement in CFD predictions. The flow in a cyclone separator is characterized by unsteady structures like secondary eddies and the precessing vortex core (PVC). An adequate resolution in space and time is necessary to capture these dynamic features. Early CFD studies focused on the steady state solution of the flow (Boysan et al. 1982) due to limited availability of computational power and low spatial resolution that resulted into artificial dampening of instabilities. With increasing computational power, unsteady state simulations with a sufficiently resolved mesh have become standard (Derksen et al. 2006).

Finally, the complexity arises from the presence of solids and their interaction with the gas phase flow. Two approaches, namely the Eulerian-Eulerian approach and the Eulerian-Lagrangian approach have been adopted in the literature to predict the multiphase flow. In the Eulerian-Eulerian approach both the solid particles and the fluid are treated as the interpenetrating continua. The governing equations are then formulated and solved for each phase. This approach can account for the complex phenomena such as the agglomeration and break-up by using a population balance model. The Eulerian-Eulerian approach requires that the interactions between the phases are modelled and are accounted for. These interactions are not yet well understood. The Eulerian-Eulerian approach also requires a specification of the boundary conditions for the particulate phase mutual interaction between particles, and interactions with the wall. In many situations, this information is not readily available. Due to these inherent drawbacks this approach has found limited application in cyclone separator simulation (See for example, Meier et al. 1998 and Qian et al. 2006).

In the Eulerian-Lagrangian approach, particle trajectories are obtained by integrating the equation of motion for individual particles, whereas, the gas flow is modelled using the Navier-Stokes equation. The flow structures in dispersed two-phase flows are a direct result

of the interactions between the two phases. Accordingly, a classification based on the importance of the interaction mechanisms has been proposed (Elghobashi, 1994). Depending on the existence of mutual, significant interaction between particles, two different regimes namely dilute and dense two-phase flow can be distinguished (See figure 2). For  $\alpha_p < 10^{-6}$  and  $L/d_p > 80$ , the influence of particles on the gas can be neglected. This is known as “one-way coupling”. The influence of the particle phase is pronounced at higher volume fractions and has to be accounted for. This is known as “two-way coupling”. For larger particles at higher volume fraction ( $\alpha_p > 10^{-3}$ ,  $L/d_p < 8$ ), the interparticle interactions become important, both through the physical collisions and indirect influence on the nearby flow field. The collisions can lead to coalescence and break-up, which must then be considered. This regime is frequently called the “four-way coupling” regime. The Eulerian-Lagrangian approach is more suited to dilute flows with one- or two-way coupling. The approach is free of numerical diffusion, is less influenced by other errors and is more stable for the flows with large gradients in particle velocity. The treatment of realistic poly-dispersed particle systems is also straightforward. These attributes make Eulerian-Lagrangian approach more suitable for the simulation of gas - particle in cyclone separators. The Eulerian-Lagrangian approach is discussed in section 1.3.2.

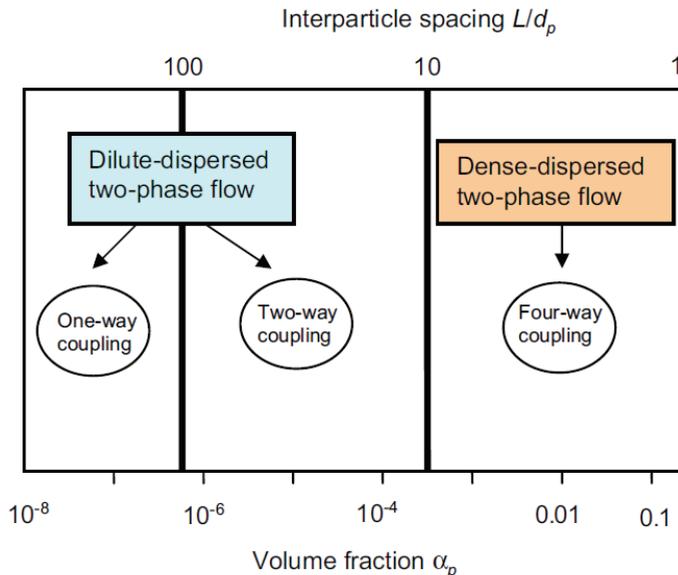


Fig. 2. Regimes of dispersed two-phase flow as a function of the particle volume fraction/interparticle spacing. Adapted from Elghobashi, 1994.

### 3.1 Choice of turbulence model

The preceding discussion makes it clear that the choice of the turbulence model is the most critical aspect of CFD simulation of cyclone separators. An appropriate turbulence model should be selected to resolve these flow features. As mentioned previously, the models based on first order turbulence closure have a limited ability for capturing the real flow. Generally it is thought that at least a second-order closure is needed to capture the

anisotropy and achieve realistic simulations (Hoekstra et al., 1999). While stressing the need for a higher order turbulence model, one needs to keep in mind that as we resolve larger ranges of time and length scales, the computational requirements escalate tremendously. A trade-off between the accuracy and speed of computation is therefore needed for practical simulations.

Of the three available approaches to capture the turbulent characteristics, namely RANS, LES and DNS, RANS approach are the oldest approach to turbulence modeling. In the unsteady RANS, an ensemble averaged version of the governing equations that also includes transient terms is solved. Turbulence closure can be accomplished either by applying the Boussinesq hypothesis, i.e. using an algebraic equation for the Reynolds stresses or by using the Reynolds stress model (RSM), i.e. by solving the transport equations for the Reynolds stresses. In the LES approach, the smaller eddies are filtered and are modeled using a sub-grid scale model, while the larger energy carrying eddies are simulated. The DNS solves fully-resolved Navier - Stokes equations. All of the relevant scales of turbulent motion are captured in direct numerical simulation. This approach is extremely expensive even for simple problems on modern computing machines. Until sufficient computational power is available, the DNS will be feasible only for model problems; leaving the simulation of industrial problems to LES and RANS approaches. Although LES of full-size equipment is possible, it is still costly partly due to the escalating computational cost near the wall region. The unsteady RANS approaches are comparatively far less expensive.

Within the RANS approach, comparative studies have been performed for different turbulence models. Hoekstra et al. (1999) compared the relative performance of the  $k-\epsilon$  model, RNG  $k-\epsilon$  model (a variation of the  $k-\epsilon$  model based on renormalization group theory) and Launder, Reece, Rodi and Gibson (LRRG) models (a differential RSM model). The simulations were compared with Laser Doppler Anemometry (LDA) velocity measurements. Tests were performed with three different vortex finder diameters, which produced three different swirl numbers. The results for the tangential velocity are shown in Figure 3. For all runs, the  $k-\epsilon$  model predicted only the inner vertex structure clearly contradicting the experimental observations showed two distinguishing vortices. The RNG  $k-\epsilon$  model showed significant improvement, while the RSM exhibited the best behavior. Pant et al. (2002) and Sommerfeld and Ho (2003) have also reported similar observations. Gimbin et al. (2005) studied the effect of temperature and inlet velocity on the cyclone pressure drop. They compared four different empirical models, the  $k-\epsilon$  model, and the RSM with the experimental data. Their study of the effect of the inlet velocity on the pressure drop found that the RSM gave the closest agreement with the experimental results. The superiority of the RSM over other models has been established by Meier et al. (1999), Xiang et al. (2005), Qian et al. (2006), Wan et al. (2008) and Kaya et al. (2009). These investigations of various characteristics of cyclone separator flow field, such as velocity profiles, pressure drop, effect of particle size, mass loading, separation efficiency, effect of pressure and temperature, have reemphasized the ability of the RSM for realistic prediction of the flow field inside cyclone separators.

Although, the superiority of the RSM over the other models has been established, it is still not clear which is the most suitable form of the RSM for cyclone separator simulations as both algebraic and differential RSMs have been employed. Between these two, the differential form of the RSM is more accurate and should be preferred over the algebraic

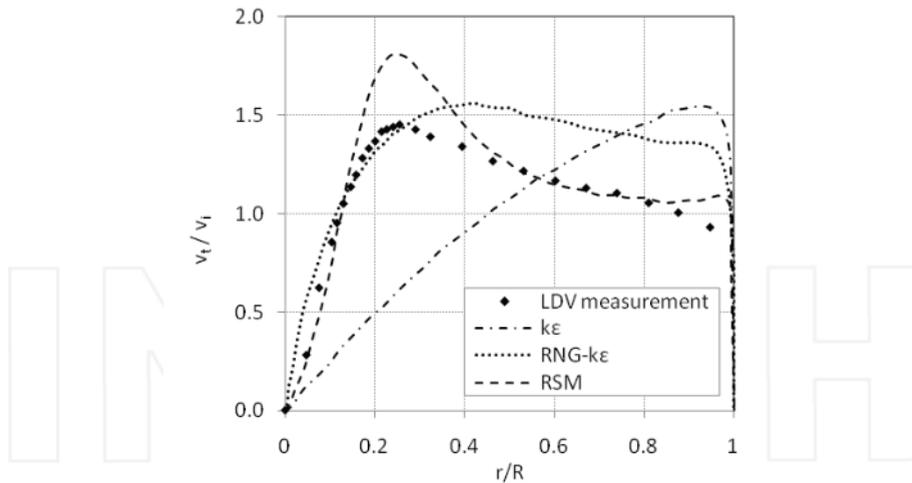


Fig. 3. Comparison of tangential velocity profiles (Adapted from Hoekstra et al., 1999)

form when the extra cost of the calculation is affordable (Hogg & Leschziner, 1989). Within the differential RSMs, the difference between a basic and an advanced differential RSM is also of relevance. For example, Grotjans et al. (1999) compared the predictions of various turbulence models with LDA measurements for the tangential velocity profile in an industrial hydrocyclone. Turbulence models including two differential RSM implementations, the basic Launder, Reece, Rodi (LRR) implementation and the advanced Speziale, Sarkar and Gatski (SSG) implementation along with the standard  $k-\epsilon$  and a  $k-\epsilon$  model modified to account for the streamline curvature (the  $k-\epsilon$  cc model) were tested. They found the flow field to be highly sensitive to the model choice, whereas the pressure distribution predictions were relatively robust. The typical Rankine profile was obtained only by means of the RSMs. The SSG model produced more acceptable results compared to the LRR model in the lower part of the cyclone. The LRR model also underpredicted tangential velocity near the cyclone center.

Despite a number of advances, the ability of unsteady RANS simulations with advanced RSM to accurately predict complex flow structures has not been fully established. Only relatively stable and ordered flows have been simulated. In order to fully establish their viability for cyclone separator simulations, these models should be tested for conditions of highly incoherent and variable PVC. Meanwhile, LES simulation of swirling and cyclone flows is presently becoming a new standard (Derksen, 2008). Derksen and van den Akker (2000) were among the first to simulate the PVC phenomenon by means of LES simulations. The capabilities of LES to simulate the turbulent flow in a cyclone separator have been reported by Shalaby et al. (2005), Derksen (2003), Derksen et al. (2006) and Shalaby et al. (2008). Early simulations (Derksen & van den Akker, 2000) were limited to small scale cyclones at a moderate inlet Reynolds number. With increasing computational power, simulation of industrial scale equipment (with  $Re = 280000$ ) have been reported (Derksen et al. 2006). The LES approach seems to offer a very realistic simulation. However due to the scale and complexity of today's industrial cyclone separator simulations, the unsteady RANS approach with higher order turbulence closures is the only practical approach that

offers affordable realistic predictions of flow inside cyclone. It is only a matter of time that resolved simulations using LES will become the preferred alternative. The behavior of particles and their interaction with continuous phase is paramount in cyclone separators and should be accounted for regardless of the turbulence models.

### 3.2 Eulerian–Lagrangian approach for multiphase flow

The Eulerian-Lagrangian approach is generally more suitable for cyclone separator simulation over the Eulerian-Eulerian approach. In the Eulerian-Lagrangian approach, the gas is treated as a continuum while the solid phase is resolved by tracking individual solid particles (Lagrangian tracking) through the flow field. Lagrangian tracking essentially applied the Newton's second law of motion to a particle to determine its position. The forces generally considered are the drag force, gravity buoyancy, virtual mass and Basset forces. For cyclone separators only the drag and gravity forces are of significance due to the large gas-particle density ratio. Of these, drag force, due to the relative slip between the particle and gas, is the dominating force and is typically modeled using an empirical correlation like the Morsi Alexander model (Morsi & Alexander, 1972).

Depending on the volume fraction, either one-way or two-way coupling is applied to account for the interactions between the two phases. For dilute flows, the gas phase flow is not influenced by the solid flow and Lagrangian tracking decoupled from the gas flow calculation is sufficient. The advantage of this is that the Lagrangian tracking can be performed as a post-processing step as a calculation using the converged and time averaged single phase simulation. To achieve a statically meaningful solution in the simulation, a large number of tracked particles (at least  $3 \times 10^5$  particles) are required (Sturgess et al., 1985). Furthermore, the time averaged gas flow field smooth out all the turbulent fluctuations. Only particles of large size will behave as exclusively influenced by the time-averaged gas flow. While very small particles will tend to fluctuate following turbulent fluctuations (known as turbulent diffusion) of the gas velocity there will be a complete range of intermediate behaviors between these two extremes. Turbulence fluctuations are random functions of space and time and several ways are available to accommodate them, mostly by including additional terms in the time averaged equation. Amongst these, the stochastic discrete random walk model is the most popular. In this model a prefixed probability distribution of velocity is assumed. The equation of motion for the discrete velocities and particle sizes is solved and the average of the forces is obtained (Gosman & Ioannides, 1981). The Stochastic Lagrangian model has been used successfully by many researchers including Yuu et al. (1978), Boysan et al. (1982), Hoekstra et al. (1999), Sommerfeld and Ho (2003) and Wang et al. (2006). The one-way coupling approach assumes negligible effect of particles on the gas flow. As a result of the collection process, high local solid concentration is observed in the near wall regions. These regions are not effectively modelled using one-way coupling. Hence in most of the Lagrangian tracking results the computational simulations show larger cut-sizes than those observed experimentally.

The effectiveness of the LES to accurately predict the gas flow field in cyclone separators has been established (Derksen, 2003). Subsequently, the Lagrangian tracking has been applied to calculate the particle flow in the LES simulations (Narsimha et al., 2006). The dynamic nature and enormous quantity of time dependent data generated by the LES prohibits post-priori calculation of the Lagrangian tracking and demands instantaneous tracking of a large number of particles. Moreover, since all the turbulence length scales are not fully resolved, the stochastic model for turbulent particle diffusion still needs to be applied. This leads to an

extremely intensive method for predicting the cyclone performance. Several alternatives have been proposed, based on average, frozen and periodic LES-velocity fields. Amongst these methods, the periodic approximation produces closest match to experimental data, however, it is the most costly method in terms of computational requirements. Using this approximation, the simulation results are much closer to the experimental data than the classical Lagrangian tracking (Derksen, 2003).

Depending on the particle size distribution, agglomeration may also become an important factor in predicting the cyclone efficiency. Particle sizes ranging from 1 to 10  $\mu\text{m}$  tend to agglomerate due to the turbulent flow. For this range of particle size the turbulence induced motion is more dominant compared to that of both Brownian motion and gravitational motions. Van der Waal forces are considerably strong enough between the particles to result in particle agglomeration and bigger size particles. Sommerfeld and Ho (2003) observed that the separation efficiency increased considerably for smaller particles in an agglomerating regime. Although, the predictions were not in a perfect agreement with the measurements regarding the grade efficiency curve, they revealed the importance of particle agglomeration on the total separation efficiency.

At higher solid concentrations, the interactions between the two phases become significant and a two-way coupling for the momentum between the particulate and fluid phases needs to be considered. Traditionally, the particle-source-in cell (PSIC) model (Crowe et al. 1977) is used for this purpose. In this model, the flow field is calculated first without the particle-phase source terms until a converged solution is achieved. A large number of "parcels" (i.e. discrete particles representing large groups with the same properties) are then tracked through the flow field. The source terms are then obtained from these tracks for a second Eulerian calculation of the gas flow. The procedure is repeated iteratively until convergence is achieved. The accuracy of this method depends on the number of parcels. Typically a minimum of 10000 to 20000 parcels are used. Computational effort also escalates as the number of particles needed to represent the dispersed phase increases. For this reason, two-way coupling therefore is still uncommon. Derksen et al. (2008) studied the effect of mass loading on the gas flow and solid particle motion in a Stairmand high efficiency cyclone separator using a two-way coupled Eulerian-Lagrangian simulation. They observed that compared to one-way coupling the two-way coupled simulation yield higher overall efficiencies. They found that the dependence of the separation efficiency on the inlet solid loading is the result of two competitive two effects namely, the attenuation of the swirl, which lowers the efficiency due to a lowered centrifugal force, and the attenuation of turbulence, which augments the efficiency through a decreased turbulent diffusion of particles.

The standard Lagrangian approach neglects the particle-particle interactions. However at higher solid concentration, these interactions must be included. The discrete element method (DEM) solves the force balance on individual particles and takes into consideration both the particle-particle and particle-gas interactions and has been used to simulate the motion of particles for highly dense flows (Zhu et al. 2007). This approach gives information about the position and velocities of individual particles. Conventional DEM approaches assume a simplified flow field and are not suitable for simulating the particle flow in cyclone separators. Recent advances in DEM and its coupling with the CFD codes has allowed simulation of particle flow within complex flow fields (e.g. Chu et al. 2009), but at this stage the method remains very costly and is limited by the number and size of particles.

## 4. Cyclone flow and pressure fields

The collection efficiency and pressure drop performance of the cyclone separator are a direct result of the flow patterns of gas and solid and pressure field inside the cyclone. In a time-averaged basis, the dominant flow feature in a cyclone separator is a vortical flow that can be described as the Rankine vortex, which is a combination of a free outer vortex and a forced inner vortex. Apart from the inlet gas velocity and geometrical parameters, the wall friction and solid loading also influence the strength of the vortex. The empirical models often neglect the later two aspects and hence are limited in their application. Computational modelling is needed to resolve the velocity and pressure fields (Kim et al., 1990, Hoekstra et al., 1999, Ma et al., 2000, Slack et al. 2000 and Solero et al 2002).

### 4.1 Axial velocity

The axial velocity of the gas phase is a major influence in the transportation of particles to the collection device. Empirical models based on the double vortex structure postulate radially constant values for the downward flow in the outer vortex and upward flow in the inner vortex. Both these values are zero at the axial position where the vortex ends. In reality, the profiles are not flat but exhibit maxima and minima. Typically the downward flow shows a maximum near the walls, while the upward flow shows either a maximum or a minimum at the symmetry axis. The diameter of the swirl of gas entering the vortex finder is larger than the vortex finder diameter itself. Consequently, the gas velocity expected to increase and peak at the vortex finder either on the centre or at the sides. This results in an inverted V or an inverted W shaped profile as seen in figures 4a and 4b for the inner vortex. The V pattern forms an axial velocity maximum at the vortex core of the cyclone while the W pattern forms an axial velocity maximum at the vortex finder radius with a minimum at the vortex core.

Figures 4a and 4b show the axial velocity profile in a cyclone separator at a horizontal position of 0.125 m below the vortex finder for two  $D_e/D$  ratios of 0.3 and 0.375, respectively. It can be seen that with relatively small difference in the dimensionless length parameters, the axial velocity in the inner vortex region shift from a V pattern to a W pattern. Harasek et al. (2008) studied this phenomenon by investigating the effect of the vortex finder diameter on the axial velocity profile. Their simulation findings could not determine the conditions when there is a transition in the velocity profile. For smaller  $D_e/D$  ratio ( $< 0.45$ ) the V shaped axial velocity is more stable was dominant. They also observed temporary W patterns due to turbulent fluctuations. At higher  $D_e/D$  ratio ( $> 0.53$ ), the possibility of back flow occurring increases and the air from outside is more likely to be drawn into the core due to the low pressure at the centre of the vortex. Thus, a W pattern is more stable.

The  $D_e/D$  ratio is not the only factor that affects the axial velocity profile. The downstream exit conditions, at both gas and particle outlets, have been shown to significantly affect the reversed flow to the vortex finder and the whole internal flow. For example, Hoffmann et al. (1996) showed that for cyclone separators with diplegs, the upward flow has a V-shape profile. This was confirmed by the numerical study by Velilla (2005). The V-shape profile is expected to have greater separation efficiency due to a narrower ascending flow region with higher swirl. Some observation by Wang et al. (2006) and Liu et al. (2006) indicate that the centre of the upward flow does not always coincide with the centre of the geometrical cyclone. This is attributed to the chaotic flow within the cyclone. In such cases, an eccentric vortex finder can be designed to reduce the pressure drop and weaken the chaotic flow.

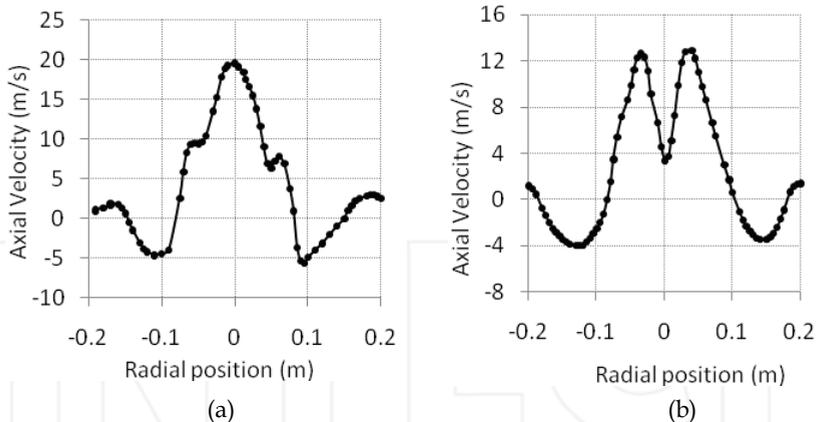


Fig. 4. Typical axial velocity profile (a) V pattern and (b) W pattern (Adapted from Harasek et al. 2008)

#### 4.2 Radial velocity

The radial velocity affects the particle bypass and is an important factor in analyzing the particle collection and losses of efficiency. Frequently the radial velocity is assumed to be of much lesser magnitude than the other components. However, this is valid only in the outer vortex, and especially near the vortex finder, the radial velocity increases rapidly towards the vortex core (Muschelknautz, 1972).

A typical example of radial velocity field is shown in figure 5. The plots are at the center of the cyclone (figure 5a) and at cut of sections B-B and C-C (figure 5b). The radial velocity profile at section D-D of the cyclone (figure 5c) has a helical shape. The axis of the vortex is slightly curved and not aligned with the geometrical axis of the cyclone. It can be seen from figure 5a that the distribution of the radial velocity is positive on one side and negative at the other side. This is due to the non-symmetrical shape of a conventional tangential inlet cyclone. It is also observed that the radial velocity increases sharply towards the vortex core. Alekseenko (1999) suggests that this phenomenon is the result of vortex rotation along the flow, in a helical shape, around the geometric axis of the cyclone. Points A and D in figure 5 show a short circuiting flow that can hinder the cyclone separator performance. Point A is called ‘lip leakage’. It is located below the rim of the vortex finder where a radial component of velocity flows inward directly to the vortex finder instead of flowing down the outside wall and returning in the central core. Point D is located near the inlet duct and the vortex finder. Just outside the vortex finder the radial velocity indicates an inward flow from the inlet (a negative value) but due to the effect of the centrifugal force around the vortex finder its value rapidly changes to zero and becomes positive. This results in an instability in the cyclone and it may affect the cyclone performance.

#### 4.3 Tangential velocity

The flow within a cyclone is dominated by the tangential velocity and strong shear in the radial direction which results in a centrifugal force that determines the particle separation. Subsequently, much discussion within cyclone separator studies is focused on the tangential velocity (Cortes & Gil, 2007).

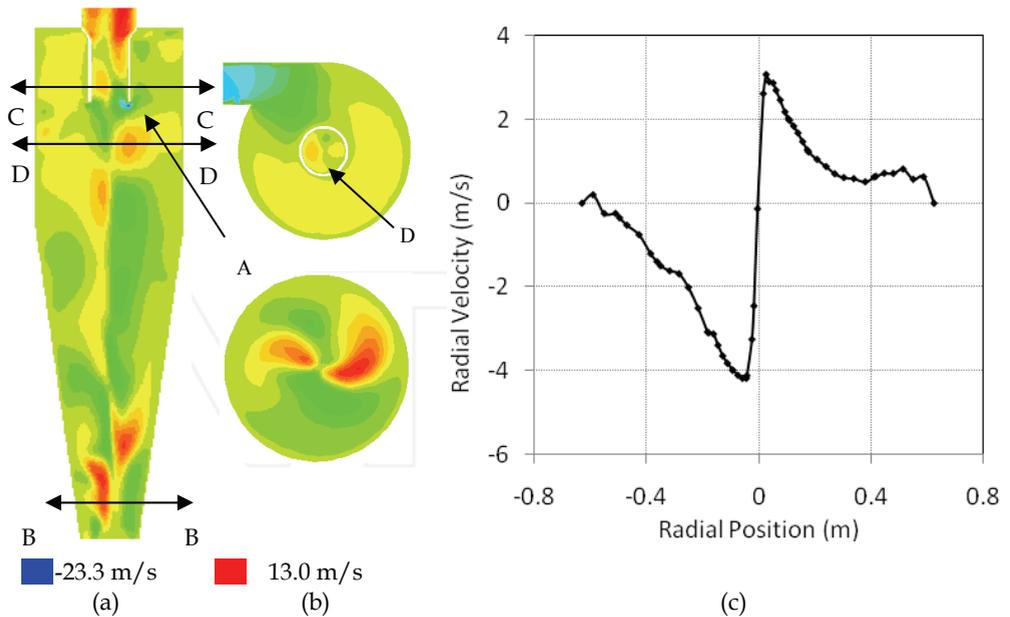


Fig. 5. (a) Contours of radial velocity at a vertical plane (b) Contours of radial velocity at horizontal cut off section B-B and C-C (c) radial velocity profile at section D-D

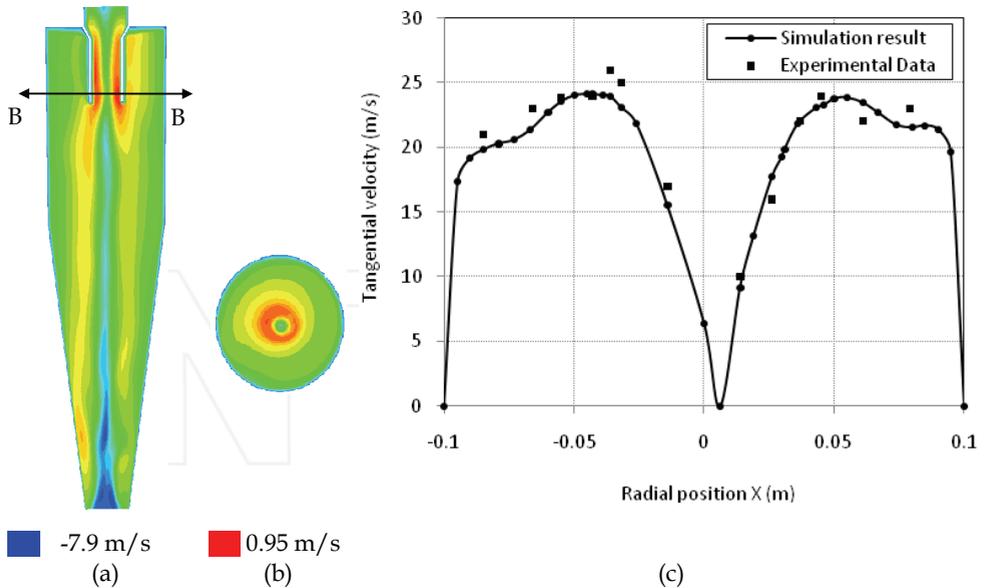


Fig. 6. (a) Contours of tangential velocity at a vertical plane (b) Horizontal cut off section at Line B-B (c) Example of comparison of numerical and experimental results for tangential velocities (Adapted from Wang et al., 2006)

Typical contour plots of the tangential velocity in both vertical and horizontal planes are shown in figures 6a and 6b, respectively. Figure 6c shows comparison of numerical and experimental results for tangential velocities from Wang et al. (2006). The cyclone has an asymmetrical shape and as can be seen from figure 6a, the axis of the cyclone does not exactly coincide with the axis of the vortex. The Rankine vortex can also be visualized. Figure 6b shows the plot of the tangential velocity across horizontal lines. It is observed that the inlet speed is accelerated up but then it decreases when the gas spins down along the cyclone wall. At a certain point flow reversal takes place and the gas flows in the reverse direction to the exit. Before entering the vortex finder, the gas collides with the follow-up flow and velocity decreases sharply. This causes energy loss and pressure drop. The tangential velocity is highly dependent on the geometrical design, wall friction and particle loading. Wang et al. (2006), Wan et al. (2008) and Raoufi et al. (2009) have demonstrated the use of CFD in reasonably predicting the tangential velocity under varied conditions. The temperature also has an effect on the tangential velocity (Shi et al., 2006). A minor decrease is noticed at the area of the inner vortex with increasing the temperature. The overall and maximum tangential velocity is also decreases on increasing the temperature. As the gas moving toward the vortex finder, the area of inner vortex become narrower and the outer vortex become wider. The main reasons for the changes are that on increasing the temperature the gas density decreases and viscosity increases. Furthermore, the centrifugal force is proportional to the square of the tangential velocity, therefore higher temperature causes the centrifugal force to decrease hence the lower separation efficiency.

#### 4.4 Pressure field

The pressure drop across the cyclone is a significant variable since it is directly related to the operating costs. The pressure drop is defined as the difference between the static pressure at the inlet and outlets. Conventional tangential inlet cyclone operations induce a spinning motion that creates radial pressure gradients, which provide a curvature for the gas flow. The particles usually follows these trajectories directed toward the cyclone wall. The Pressure drop within a cyclone is contributed to by both local losses and frictional losses. Local losses include the expansion loss at the inlet and the contraction loss at the outlet while the frictional losses include the swirling loss due to gas to wall friction and the dissipation loss of the dynamic energy of gas.

The total pressure drop, comprising both static and dynamic pressure, decreases on increasing the wall friction coefficient, particle concentration and cyclone length. The combination of the static pressure and the kinetic energy of the vortex is called the total pressure. The viscous dissipation of the kinetic energy in the vortex finder dominates the pressure loss within the cyclone (Dirgo, 1988 and Coker, 1993). Thus the pressure loss is directly proportional to the dynamic pressure. About 40% of the pressure drop is due to the swirl energy losses while the rest is from the sudden expansion at the inlet and the contraction at the outlet duct. Any influence that increases the vortex strength will increase the pressure loss. For example, an increase in the wall friction coefficient will increase the pressure loss as it will decrease the velocity magnitude and will lead to decreased loss in the vortex finder. Similarly, an increase in the wall friction will decrease pressure drop (Hoffman et al. 1992). At increased particle concentration, the tangential velocities will be lower and accordingly will yield a lower pressure drop.

Gimbun et al. (2005) studied the effect of the inlet velocity and particle loading on pressure drop. They compared experimental values by Bohnet (1995) with empirical models by

Shepperd and Lapple (1939), Casal and Martinez (1983), Dirgo (1988), Coker (1993), as well as with CFD predictions using the k- $\epsilon$  model and the RSM model. The result showed that the RSM model produced the closest pressure drop prediction. The k- $\epsilon$  results showed a reasonably good prediction at about 14% -18% deviation. The CFD studies on gas-solid flow in a cyclone separator by Wang et al. (2006) using the RSM model also showed an acceptable agreement with experimental data for Stairmand high efficiency cyclone Hoekstra et al. (1999).

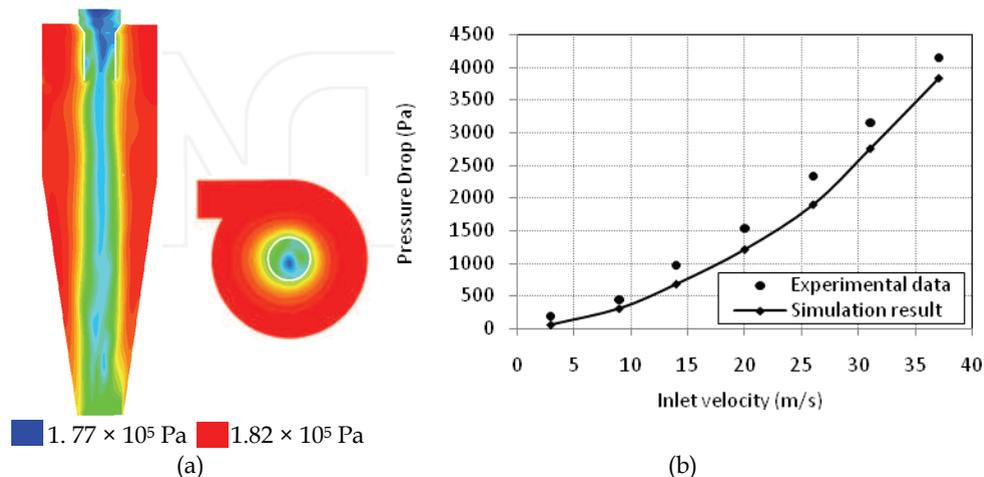


Fig. 7. (a) Contours of total pressure (b) Pressure drop comparison of CFD RSM measurement and experimental data (Stairmand cyclone) for various inlet gas velocities (Adapted from Wang et al., 2009)

Figures 7a and 7b show the contours of the total pressure and the pressure drop comparison between RSM model predictions and experimental data for the Stairmand cyclone at various inlet gas velocities, respectively. The total pressure increases in the radial direction from the the centre to the wall of the cyclone. Flow reversal in a cyclone is due to the low pressure centre. It can be seen from Figure 7b that the CFD simulations underpredict the pressure drop, across the cyclone only the static pressure is considered with the dynamic pressure being neglected. In reality swirl dissipation continues further down the cyclone outlet so that dynamic pressure will be lost without any chance to be recovered. Consequently, the actual pressure drop will be higher.

Any factors that may cause change in the absolute magnitude of the velocity, which in turn changes the strength of the vortex, will affect the pressure drop in the cyclone. Generally, the pressure drop will increase with increasing vortex strength. The pressure drop will decrease with an increase in the wall friction coefficient, particle loading or cyclone length (Yu et al., 1978, Parida & Chand, 1980, Hoffman et al., 1992). When the wall friction coefficient is increased, the swirl in the separation space decreases and causes an increase in the pressure loss. However, it also decreases the absolute velocity magnitude which results in a decreased pressure loss at the vortex finder. The latter effect is always much higher than the first effect and any increase in the wall friction decreases the pressure drop.

Some important processing industries such as pressurized fluidized bed combustion (PFBC), Integrated gasification and combined cycle (IGCC) and Fluid Catalytic Cracking

(FCC) operate at high temperatures and pressures. The operating temperature and pressure will influence the gas density and viscosity and their effect on the drag force. Therefore, for these industries, the operating temperatures and pressures are the important parameters that determine the pressure drop in the cyclone. Shin et al. (2005) (See figure 8) conducted numerical and experimental study on the effect of temperature and pressure on a high efficiency cyclone separator. They found that the pressure drop decreases at a higher pressure and lower temperature. Higher pressures and lower temperatures increase the gas density which in turn creates a higher dynamic pressure hence the higher pressure drop. This trend is confirmed by similar experimental and numerical studies by Gimbut et al. (2005) and Shi et al. (2006)

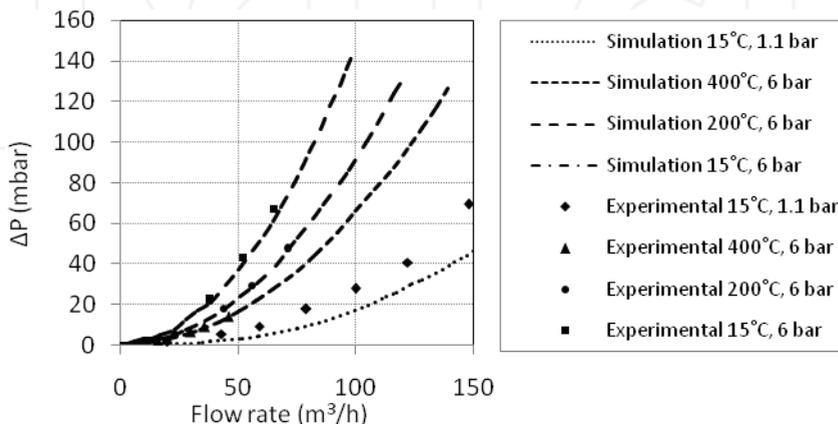


Fig. 8. Comparison of experimental and numerical result for pressure drop at a given flow rate in a elevated pressure and temperature ( Adapted from Shin et al., 2005)

## 5. Collection efficiency

The fraction of solids separated at the outlet is defined as the collection efficiency. Since cyclone separators usually handle various sizes of particles, the efficiencies are defined according to a continuous narrow interval of particular group size particles. The swirling motion within the cyclone separator causes large particles to travel swiftly to the cyclone walls and roll down to the outlet. On the other hand, the smaller particles are often drifted in upward spiral flow due to the slower speed and escape through the gas outlet. This typically yields an S shaped curve for the collection efficiency. Particle collection is the net effect of various forces acting on the particles. It is well known that the collection efficiency is governed by the centrifugal, gravitational and drag forces (Blachmann & Lipmann, 1974). Factors such as the particle-particle and particle-wall interaction also influence the cyclone efficiency. Their effect is not yet fully understood and hence often neglected in empirical modelling. Further, the empirical models are based on the lab scale data. Depending on the operating conditions, the flow inside the cyclone can be laminar, transitional or turbulent for the lab scale equipment. The actual industrial cyclones operate in the turbulent regime where the friction and its corresponding outcomes are significant. Therefore the particle collection efficiency models based on the lab scale data may not accurately predict the collection efficiency for industrial cyclones. At lower mass loading (<5-10 g/m<sup>3</sup>) the

empirical models perform reasonably well (Cortes and Gil, 2007). Many cyclone separator systems of industrial interest such as the FCC, PFBC and CFBC are well known for handling high particle loadings, where, the interphase and interparticle processes become important and the predictive ability of the conventional models is poor. Numerical studies then become necessary to achieve a better understanding of the cyclone collection efficiency.

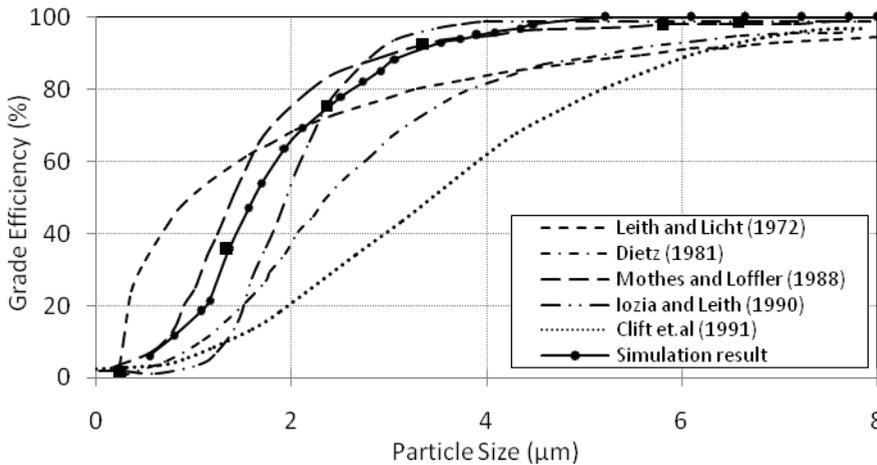


Fig. 9. Collection efficiency model comparison: theoretical, numerical and experimental (Adapted from Zhao et al., 2006)

A typical comparison of relative performance of computational models and conventional models in predicting the cyclone efficiency is shown in figure 9 (Zhao et al. 2006). The collection efficiency in a cyclone with conventional inlet (CI) and a spiral double inlet (DI) configuration is evaluated using the unsteady RANS model with the RSM turbulence model. These predictions are compared with the experimental data and empirical models. Both these profiles follow the S shape with lower collection efficiencies for the smaller particles and almost total capture of larger particles. The comparison also clearly demonstrates the superiority of CFD model over the empirical model in calculating the collection efficiencies. The collection efficiency is a primary measure of the performance of a cyclone separator and depends on the operating conditions and the geometrical characteristics. In the following subsections we look at how these affect the collection efficiencies.

### 5.1 Effect of mass loading, particle diameter and inlet velocity on cyclone efficiency

Qian et al. (2006) investigated the effect of mass loading on the collection efficiency. The results of their simulation are shown in figure 10. The collection efficiency is defined as the ratio of mass flow rate at the inlet and outlet for a converged steady condition. It is clearly evident that the collection efficiency increase on increasing the particle loading. The result is consistent with most of the previous studies (like Stern et al. 1955). Different mass loading for various particle group-sizes affect the grade efficiency differently. Smaller particle group sizes show a higher efficiency increase compared to the larger particle size groups. These findings are also confirmed by the simulation and experimental study by Luo et al. (1999)

and Ji et al. (2009). The increase in cyclone efficiency with solid loading is more pronounced at lower gas velocities (Hoffmann et al. 1991, 1992).

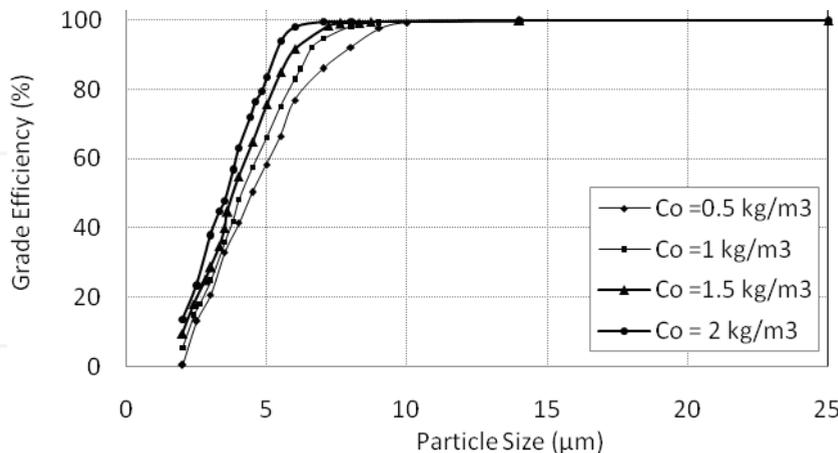


Fig. 10. Separation efficiency simulation result for various inlet particle concentration with constant inlet velocity (Adapted from Qian et al., 2006)

Mass loading effect is usually coupled with the particle diameter. At lower mass loadings, the smaller particles ( $< 10\mu\text{m}$ ) tend to be dispersed and hauled by the gas flow and escape from the vortex finder at the top of the cyclone separator (Derksen 2003 and Wan et al. 2008). But on increasing the particle mass loading, a sweeping effect of the coarser particle that sweeps away the smaller particles to the cyclone wall is observed. The swept particles then roll down and are collected at the bottom of the cyclone. This effect is also responsible for the formation of agglomerates. Agglomeration causes increased centrifugal force on the smaller particles improving their collection efficiency. Wan et al. (2008) also note that on increasing the particle loading, both the downward flow and the axial velocity at the centre (in upward direction) increase. This aids in higher collection efficiency in the cyclone separator.

The inlet gas velocity also has an effect on the collection efficiency. The effect is also tightly related to the particle mass loading. Figure 11 shows the effect of increasing the inlet gas velocity on the collection efficiency for a given solid loading (Ji et al. 2009). The collection efficiency increases with the inlet gas velocity. For smaller particle sizes ( $< 10\mu\text{m}$ ), the increase in the efficiency with respect to the gas velocity is more pronounced. As the particle size increases, the effect of the inlet velocity becomes insignificant. These observations are in line with the experimental work of Fassani et al. (2000) and Hoffmann et al. (1991). The higher inlet velocity, results in the higher tangential velocity, thus leading to a higher centrifugal force and collection efficiency. Patterson and Munz (1989) analyzed the effect of several parameters including the gas temperature (300K - 2000K), inlet gas velocities (3 m/s - 42 m/s) and particle loadings (up to 235.2 g/m<sup>3</sup>) on the cyclone efficiency. Their analysis showed that there is an increase in the cyclone efficiency especially under high temperature condition.

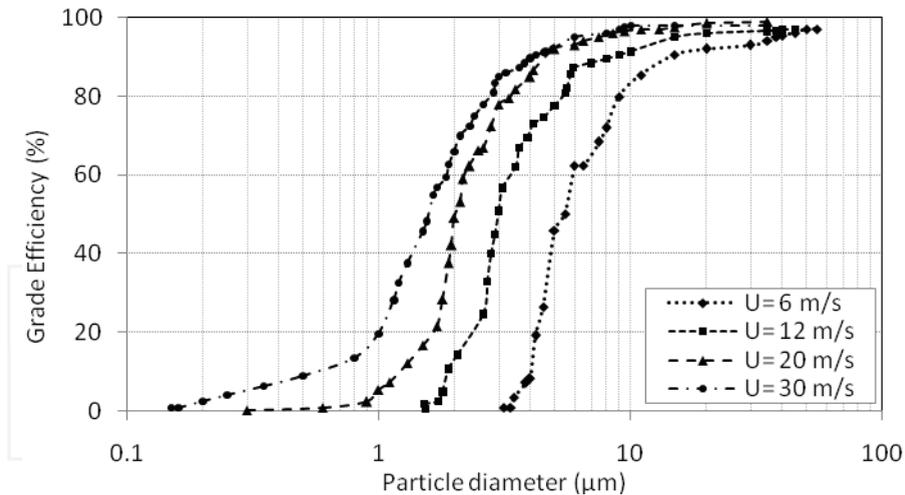


Fig. 11. Grade efficiency for different inlet velocity (Adapted from Ji et al., 2009)

## 5.2 Effect of geometrical configuration on cyclone efficiency

The geometrical configuration is probably the most crucial aspect affecting the performance of a cyclone separator. The cyclone separator performance is sensitive to the smallest of geometrical changes. Through meticulous experimentation, optimal designs of cyclone separators have been proposed. One of the most commonly used designs is the Stairmand high efficiency cyclone (Stairmand 1951). After almost 60 years, this design is still popular since it manages to maintain the velocity profile at every axial point well and achieves very high collection efficiency. The key for Stairmand high efficiency cyclone design lies on the optimal geometrical ratio of cone length to diameter ( $L/D=4$ ). Unfortunately, it is impossible to always accommodate Stairmand cyclone in the industrial processes, therefore an optimal design best suited for the need has to be explored. Cyclone peripherals like the dipleg and the vortex finder also affect the performance.

Several studies on the effect of dipleg on the cyclone performance have been reported (Hoffmann et al., 1996, Gil et al., 2002, Obermair et al., 2003 and Kaya et al., 2009). Hoffman et al. (1996) found that the  $L/D$  ratio of 5.65 is optimal, after this, the collection efficiency starts decreasing. Kaya et al. (2009) extended the effect of dipleg to investigate various inlet velocities and particle diameters for smaller cyclones. They showed that cyclone performance can be improved by adding a correct length of dipleg at the particle outlet. A prolonged cyclone improves the collection efficiency by providing additional space for the separation. According to their results, the optimum dipleg length would be approximately half the height of the cyclone. Placing the vortex end – the location at which the outer vortex changes its direction, exactly at the end of dipleg leads to high collection efficiency. Increasing the length further causes the vortex to fall short and leads to the re-entrainment of particle to the vortex finder thus decreasing the efficiency. Similar observations were made by Gil et al. (2002).

The vortex finder provides another avenue for optimizing the cyclone design. The size of the vortex finder is critical since it determines the inner and outer swirling flow pattern within a

cyclone separator. Concrete methodology is not yet available to optimize the vortex finder. Saltzmann et al. (1984) and Iozia et al. (1989) studied the effect of vortex diameter on the cyclone performance. Kim and Lee (1990) provide information on how the vortex finder and cyclone body diameter affects the cyclone performance. Lim et al. (2004) evaluated different geometries of the vortex finders to optimize the performance. Their results concluded that with smaller vortex finder diameters, the tangential velocity in the inner region of the cyclone increases resulting in a higher collection efficiency. These findings were supported by Raoufi et al. (2009) using CFD modelling.

CFD modelling has opened an avenue for the cost effective optimization of the cyclone geometry. Many geometrical designs have been proposed using CFD studies that can be used to improve the performance of cyclone separators. The list of these modifications include, (i) the use of different inlet types (scroll, helicoidal, axial spiral double inlet and square cyclone inlets) can be found in Cortes and Gil (2007), Wang et al. (1999) and Zhao et al. (2006), (ii) including a long coned (Lee et al., 2006), and (iii) the variation in body and cone height (Xiang et al., 2005).

## 6. Erosion in cyclone separator

The erosion often poses a serious problem in industrial cyclone separators. It is caused by the friction between the particles and the cyclone walls due to the continuous motion of the particles along the cyclone wall. The erosion often leads to a physical damage to the equipment (holes) and costly unscheduled shutdowns. Even pinholes in the wall can disrupt the pressure balance and can cause increased emissions. The eroded wall material may also contaminate the particles collected at the outlet. The erosion occurs due to directional or random impingement of the solid particles and due to friction of the sliding particles against the cyclone wall. Assuming a perfect elasticity, a particle colliding with the cyclone wall will rebound at the same angle (in opposite direction) as it hits the wall, the erosion is minimal at larger angles, however, at very acute angles of impingement, severe erosion results along the cyclone wall circumference. On a flat and ductile target the greatest erosion occurs at around  $15^\circ$  impingement angle (Noppenberger 2000). The erosion is proportional to the kinetic energy of the particles hitting the cyclone wall and the intrinsic strength of the particles. It also depends upon the angle of incidence on the cyclone wall.

At certain specific locations significantly higher erosion occurs compared to other locations, most possibly due to high local velocity which is a function of the particle size and particle density. The high erosion rates are also observed at the inlet. Severe erosion is also observed inside the cyclone, however, there is no consensus on the location of this. Jones et al. (1979) found that the area extending  $180^\circ$  from the inlet point has most severe erosion and the erosion maxima is observed at angular positions of  $105^\circ$ ,  $165^\circ$ ,  $205^\circ$  and  $245^\circ$  from the tangential inlet. Another study by Youngdahl (1984) found that the highest rate of erosion occurs at  $28^\circ$  and near the lower edge of the inlet stream. Because a detailed velocity and pressure fields and particle trajectories are available from an Eulerian-Lagrangian CFD simulation, it can be used to determine the otherwise impossible locations for likely erosion sites.

Figure 12 shows the erosion map for an industrial cyclone using (a) uniform particle size of  $80\mu\text{m}$  at the inlet and (b) Rosin – Ramler particle size distribution at the inlet with average particle size of  $80\mu\text{m}$ . The results show that the particles follow a distinct path rather than

cluttering on the cyclone walls while swirling down. For the uniform particle size, most severely eroded sites were the top part of the cyclone cylinder (near the inlet) and the cone of the cyclone. This observation is in accordance with the experimental findings of Jones et al. (1979) and Yongdahl et al. (1984). On using a particle size distribution, the most eroded part in the cyclone was at the intersection between the cyclone cylinder and cone. This demonstrates the sensitivity of the erosion to particle size distribution.

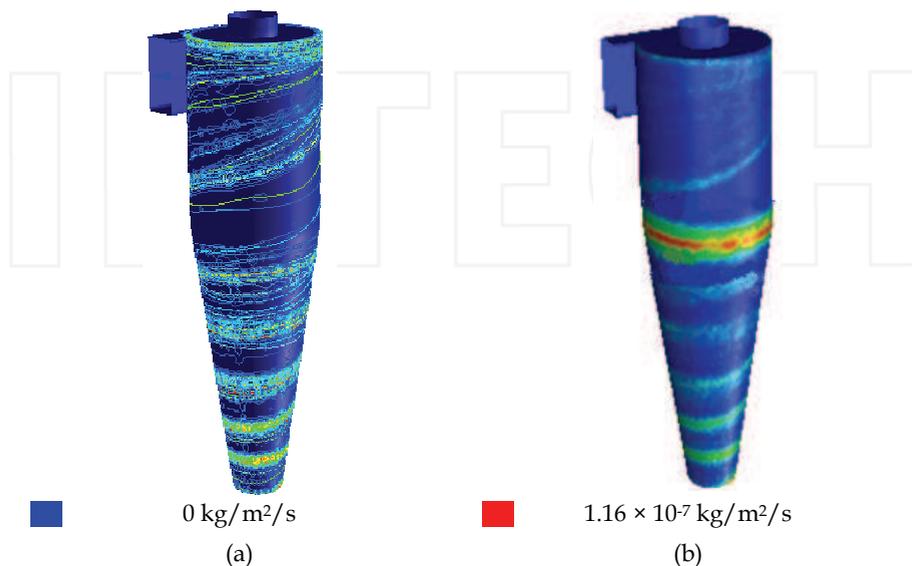


Fig. 12. (a) Contours of erosion (a) with uniform catalyst size (b) Rosin-Rammler PSD

### 6.1 Effect of particle size distribution on cyclone wall erosion

To investigate the effect of particle size distribution, a wider inlet particle size distribution ( $1 \mu\text{m}$  to  $160 \mu\text{m}$ ) was simulated using the Lagrangian tracking. The resulting particle trajectories are shown in Figure 13. It is clearly evident that the larger particles are collected at the walls while the smaller particles escape downwards in spiral manner. This is because the drag force on the smaller particles is larger than the centrifugal force preventing their transportation to the cyclone wall. The results show that the particle with sizes  $< 40 \mu\text{m}$  escape from the bottom of the cyclone after a certain residence time while the particle with sizes  $> 60 \mu\text{m}$  keep spinning around the mid level for significantly longer. Wang et al. (2006) have shown this phenomenon experimentally using ceramic balls. One possible explanation lies in the balance of the centrifugal force versus the gravitational force. As the larger particles roll down the conical part, the centrifugal force on the particle increases because the radius of the cyclone decreases while the tangential velocity of the particle remain almost the same. The larger particles will eventually be collected at the solid outlet due to the particle-particle interaction. However, some of the particles will stay at the cyclone wall. There also appears to be a critical value of the particle diameter below which the particle is not expected to be collected at the outlet. The critical value of particle diameter is related to the cyclone geometry, gas inlet velocity and particle properties.

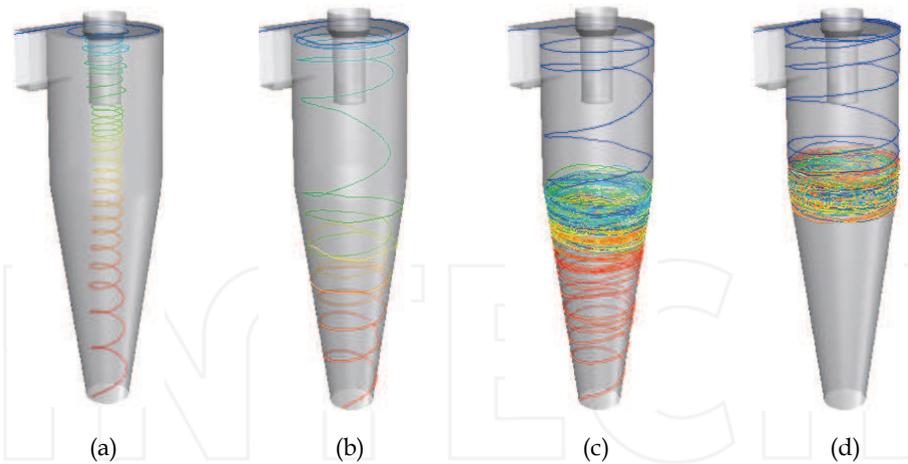


Fig. 13. Single particle trajectories of size (a) 5  $\mu\text{m}$  (b) 20  $\mu\text{m}$  (c) 40  $\mu\text{m}$  (d) >60  $\mu\text{m}$

**6.2 Effect of particle mass loading on cyclone wall erosion**

The effect of particle mass loading on the average erosion rate at different gas velocities is shown in figure 14. The average of erosion rate gives a global measure of this quantity within the entire cyclone. As expected, at a given gas velocity, the erosion rate decreases with the solid loading. For a given solid loading, however, the average erosion rate increases with gas velocity upto a certain velocity. Beyond this velocity, the erosion rate remains constant or decreases slightly.

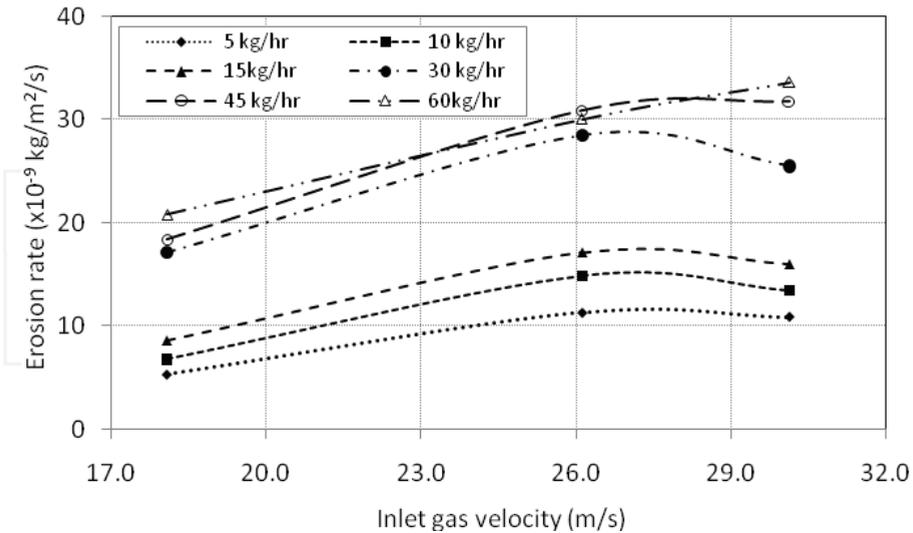


Fig. 14. Comparison of erosion rate with various gas and particle flow rate

At lower gas velocities, lower momentum is imparted by the gas on the particle, which sometimes prevents a second rebound to happen and the particle is forced to stay near the cyclone wall. Consequently, the rate of erosion is lesser at lower velocities. As the gas velocity is increased, the particle rebound is more likely to happen. Since the particle rebounds are not perfectly elastic, it reduces the impact angle gradually after the first rebound, thus increasing the average erosion rate. At even higher gas velocities, the centrifugal force on the particles increases. This makes some particles reach the cyclone wall faster. As a consequence, a layer of slow moving particles is formed that protects the walls from collision by other particles which then reduces the rate of erosion slightly.

## 7. Summary

Due to their simple and robust construction, cyclone separators are widely used in the chemical and process industries. In spite of their simple construction, flow patterns inside cyclones are highly complex and not fully understood. Understanding the flow is critical in accessing their performance and CFD is a useful tool in providing this information. However, due to the very nature of the flow, the application of CFD should be exercised with prudence. In order to accurately resolve the unsteady nature of the flow inside a cyclone, higher order numerical discretization along with unsteady simulation (unsteady RANS or LES) are needed. It also requires a higher order turbulence model (atleast second order like RSM) for the unsteady RANS simulations. For resolving the gas flow field, the LES provides superior results than the RANS approach. However, the cost of LES is prohibitive for the industrial-scale devices. Recent developments turbulence modelling such as the differential RSMs have shown a light of hope to achieve LES level of accuracies at RANS cost. However, the conditions under which the unsteady RANS solver can be used in place of the LES need to be explored.

To obtain the particle flow, the Eulerian-Lagrangian formulation with either one way or two way coupling should be employed. The necessity of capturing the unsteadiness of the gas flow in combination to the flow of poly-disperse particulates demands far superior computational power than what is currently available. Therefore, a considerable progress needs to be made in multiphase flow simulation of cyclones. Factors like interparticle phenomena and the boundary condition at the wall also need a careful attention. The advances in the DEM-CFD coupled simulations will bring new insight in the calculation of highly-loaded cyclones. Nevertheless, the two phase flow simulations have provided some useful insight into the cyclone operations and have provoked to question the existing theories on the particle flow and separation in cyclones.

The enormous computational requirements, even for the minimal modelling of cyclone phenomena, have limited our ability to go beyond a simple understanding of the flow structures, collection efficiency and global design issues. More systematic research for addressing the other important issues, such as reasonable estimates of the cyclone natural length and vortex finder dimensions, is needed. Furthermore, the loss of coherence in the vortex and the ensuing chaotic flow patterns and the effect of swirl-stabilization are some of the other topics that remain unanswered. With increasing computational power, it is envisaged that in the near future we will be able to perform fully resolved simulations on cyclones which will not only answer the above questions but will also advance our knowledge of cyclone operations and even optimize them for specific operational circumstances.

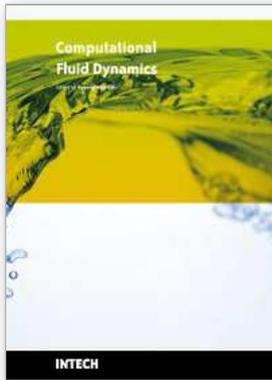
## 8. References

- Barth, W. (1956) Design and Layout of the cyclone separator on the basis of new investigation. *Brenn. Warne Kraft*, 8, 1-9.
- Blachmann, M. W., Lippmann, M. (1974) Performance characteristic of the multi cyclone aerosol sampler, *Ind Hyg. Ass. J.*, 35, 311-326.
- Bohnet, M. (1995) Influence of the Gas Temperature on the Separation Efficiency of Aerocyclones, *Chem. Eng. Process*, (34), 151.
- Boysan, F., Ayers, W.H., Swithenbank, J. (1982). A fundamental mathematical modeling approach to cyclone design, *Trans. Instn. Chem. Engrs*, 60, 222-230.
- Boysan, F., Ewan, B. C. R., Swithenbank, J., Ayers, W.H. (1983) Experimental and theoretical studies of cyclone separator aerodynamics, *Inst Chem Eng Symp Ser*, 69:305-19.
- Bunyanawichakul, P., Kirkpatrick, M. P., Sargison, J. E., Walker G. J. (2006) Numerical and Experimental Studies of the Flow Field in a Cyclone Dryer, *J Fluids Eng*, 128 (6), 1240
- Casal, J. M., Martinez, A. (1983) A better way to calculate cyclone pressure drop. *Chemical engineering*, 90, 99.
- Chu, K. W., Wang, B., Yu, A.B., Vince A. (2009) CFD-DEM modeling of multiphase flow in dense medium cyclone. *Powder Tech.*, 193(3), 235-247.
- Coker, A. K. (1993) Understand cyclone design. *Chemical Engineering Progress*, 28, 51-55.
- Cortes, C., Gil, A. (2007) Modeling the particle flow inside cyclone separators. *Progress in energy and combustion Science* (33).
- Derksen, J. J., van den Akker, HEA, (2000) Simulation of vortex core precession in a reverse flow cyclone. *AIChE J.*, 49, 1359-1371
- Derksen, J. J. (2003) Separation performance predictions of a Stairmand high-efficiency cyclone. *AIChE J* (49), 1359-1371.
- Derksen, J. J. (2005) Simulations of confined turbulent vortex flow. *Comp Fluids* (34), 301-318.
- Derksen, J. J., Sundaresan, S., van der Akker H.E.A. (2006) Simulation of mass loading effects in gas-solid cyclone separators. *Powder Tech.*, 163, 59-68.
- Dietz, P. W. (1981) Collection efficiency of cyclone separators. *A.I. Ch.E.J.*, 27, 888.
- Dirgo, J. (1988) Relationships between cyclone dimensions and performance. Havarad University, USA.
- Dirgo, J., Leith, D., (Ed.) (1986) Encyclopedia of fluid mechanics, (Vols. 4). Houston: Gulf Publishing Company.
- Elghobashi, S. E. (1994) On predicting particle-laden turbulent flows. *Appl Sci Res*; (52), 309-29.
- Gil, A., Cortes, C., Romeo, L. M., Velilla, J. (2002) Gas-particle flow inside cyclone diplegs with pneumatic extraction. *Powder Tech.* (128), 822-829.
- Gimbun, J., Chuah, T.G, Fakhru'l-Razi, A., Choong, T.S.Y. (2005) The influence of temperature and inlet velocity on cyclone pressure drop: A CFD study. *Chemical engineering and Processing*, 44, 7-12.
- Gosman, A. D., Ioannides, E. (1981) Aspect of computer simulation of liquid fuelled combusters. *AIAA paper*, 81-0323.
- Grotjans, H., Menter, F.R., Burr, R.C. and Gluck, M., 1999, "Higher Order Turbulence Modelling in Industrial Applications", 4<sup>th</sup> Int. Symp. on Engineering Turbulence Modeling and Measurement..

- Harasek, M., Horvath, A., Jordan, C. (2004) *Investigation of dependence of gas flow on the geometry of cyclonic separators by CFD simulation*. Paper presented at the CHISA 2004 - 16th International Congress of Chemical and Process Engineering.
- Harasek, M., Horvath, A., Jordan, C. (2008) Influence of vortex finder diameter on axial gas flow in simple cyclone. *Chemical product and process modeling* 3(1), Article 5.
- Hoekstra, A. J., Derksen, J. J., Van Den Akker, H. E. A. (1999) An experimental and numerical study of turbulent swirling flow in gas cyclones. *Chem. Eng. Sci.*, 54, 2055-2056.
- Hoffmann AC, H Arends, H Sie, (1991), An experimental investigation elucidating the nature of the effect of solids loading on cyclone performance, *Filtration & Separation*, 28 (3), pp 188-193
- Hoffmann, A. C., van Santen A., Allen, R. W. K., Cliff, R. (1992) Effects of geometry and solid loadings on the performance of gas cyclones. *Powder Tech.*, 70, 83-91.
- Hoffmann, AC; deGroot, M; Hospers, A, (1996), The effect of the dust collection system on the flowpattern and separation efficiency of a gas cyclone, *Canadian journal of chemical engineering* , 74 (4), pp 464-470.
- Hogg, S., Leschziner, M.A. (1989). Computational of highly swirling confined flow with a Reynolds stress turbulence model, *AIAA J* ,1(27), 57-63.
- Iozia, D. L. a. L., D. (1989) Effect of cyclone dimensions on gas flow pattern and collection efficiency. *Aerosol Science tech.*, 10, 491-500.
- Jakirlic, S., Hanjalic, K. (2002). Modelling rotating and swirling turbulent flows: a perpetual challenge. *AIAA J*, 40:1984-96.
- Ji, Z., Xiong, Z., Chen, H., Wu, H. (2009) Experimental investigations on a cyclone separator performance at an extremely low particle concentration. *Powder Tech.* (191), 254-259.
- Jones, J. L., Arnold, J. M., Youngdahl, C. A. (1979) Erosion rates and patterns of the gas pilot plant's effluent cyclone.
- Kaya, F., Karagoz, I. (2009) Numerical Investigation of performance characteristics of a cyclone prolonged with a dipleg. *Che. eng. Journal*, 151, 39-45.
- Kim, J. (1990) Experimental study of particle collection by small cyclones, *Aerosol Science tech.*, 12, 1003-1015.
- Lapple, C. E. (1951). Processes use many collector types, *Chemical engineering*, 58, 144 - 151.
- Leith, D., Licht, W. (1972) Collection efficiency of cyclone type particle collector: A new theoretical approach, *A.I.Ch.E Symposium series (Air-1971)*, 68, 196-206.
- Ma, L., Ingham, D.B ., Wen, X. . (2000). Modelling of the fluid and particle penetration through small sampling cyclones. *J Aerosol Sci* (31), 1097-1119.
- Meier, H. F., Mori , M. (1998) Gas-solid flow in cyclones: the Eulerian-Eulerian approach. *Comput Chem Eng* , 22(Suppl 1):S641-4.
- Meier, H. F., Mori, M. (1999) Anisotropic behavior of the Reynolds stress in gas and gas-solid flows in cyclones. *Powder Tech.*(101), 108-119.
- Morsi, S. A., Alexander, A.J., . (1972). An investigation of particle trajectories in two phase flow systems. *Fluid Mech. J.*, 55(2), 193-208.
- Muschelknautz, E. (1972) Die Berechnung von Zyklonabscheidern für Gase. *Chem-Ing-Tech* (44), 63-71.
- Noppenberger, M. (2000) How to control erosion in FCC cyclone. *World refining*, 10(6), 36-38.
- Obermair, S. (2003). *Einfluss der Feststoffaustragsgeometrie auf die Abscheidung und den Druckverlust eines Gaszyklons*, Technical University Graz.

- Pant, K., Crowe, C. T., Irving, P. (2002) On the design of miniature cyclone for the collection of bioaerosols. *Powder Tech.*, 125, 260-265.
- Parida, A., Chand, P. (1980) Turbulent swirl flow with gas-solid flow in cyclone. *Che. Eng. Sci.*, 35(4), 949-954.
- Patterson, P. A. M., Munz R. J. (1989) Cyclone Collection Efficiencies at Very High Temperatures. *Can. J. Chem. Eng.*, 37.
- Qian, F., Huang, Z., Chen, G., Zhang, M. (2006) Numerical study of the separation characteristics in a cyclone of different inlet particle concentrations. *Computers and chemical engineering*, 31, 1111-1122.
- Raoufi, A., Shams, M., Kanani, H. (2009) CFD analysis of flow field in square cyclones. *Powder Tech.* (191), 349-357.
- Saltzman, B. (1984) Generalized performance characteristics of miniature cyclone for atmospheric particulate sampling. *Am. Ind. Hyg. Assoc. J.*, 45, 671-680.
- Shalaby, H., Pachler, K., Wozniak, K., Wozniak, G. (2005) Comparative study of the continuous phase flow in cyclone separator using different turbulence models. *International J. of Numerical methods in fluids*, 11(48), 1175-1197.
- Shalaby, H., Wozniak, K., Wozniak, G. (2008) Numerical calculation of particle -laden cyclone separator flow using LES. *Eng. app. of Comp. Fluid Mech.*, 2(4), 382-392.
- Shepherd, C. B., Lapple, C.E. (1939). Flow pattern and pressure drop in cyclone dust collectors. *Ind. Eng. Chem*, 31, 972-984.
- Shi, L., Bayless D. J., Kremer G., Stuart B. (2006) CFD Simulation of the Influence of Temperature and Pressure on the Flow Pattern in Cyclones. *Ind. Eng. Chem. Res.*(45), 7667-7672.
- Shin, M. S. K., Jang, D. S., Chung, J. D.; Bohnet, M. (2005) Numerical and Experimental Study on a High Efficiency Cyclone Dust Separator for High Temperature and Pressurized Environments, *Appl. Therm. Eng.*, (25), 1821.
- Slack, M. D., Prasad, R.O., Bakker, A., Boysan, F. (2000) Advances in cyclones modeling using unstructured grids, *Transactions of the Institution of Chemical engineers*, 78A, 1098.
- Sommerfeld, M., Ho, C.H. (2003) Numerical calculation of particle transport in turbulent wall bounded flows. *Powder Tech.*, 131, 1-6.
- Sproul (1970) Air pollution and its control, New York: Exposition Press.
- Stairmand (1951) The design and performance of cyclone separators. *Trans. Instn. Chem. Engrs*, 29, 356 - 383.
- Sturgess, G. J., Syed, S.A. (1985) Calculation of a hollow-cone liquid spray in uniform airstream, *J. Propul. Power* (1)
- Swift, P. (1969). Dust Control in Industry. *Steam Heat. Engr.*, 38, 453 - 456.
- Tengbergen, H. J. (1965) Comparative studies with cyclone, *Staub*, 25(11), 44-49.
- Velilla, J. (2005) Study of the flow at a PFBC cyclone dipleg. University of Zaragoza.
- Wan, G., Sun, G., Xue, X., Shi, M. (2008) Solids concentration simulation of different size particles in a cyclone separator, *Powder Tech.*, 183, 94-104.
- Wang, B., Xu, D.L., Chu, K.W., Yu, A.B. (2006) Numerical study of gas solid flow in a cyclone separator, *Applied Mathematical Modeling*, 30, 1326-1342.
- Wang, S., Fang, M., Luo, Z., Li, X., Ni, M., Chen K.. (1999). Instantaneous separation model of a square cyclone., *Powder Technology* (102), 65-70.

- Xiang, R. B., Lee, K.W. (2005) Numerical study of flow field in cyclones of different height. *Che. Eng. Sci.*, 44, 877-883.
- Youngdahl, C. A. (1984). Nondestructive monitoring of erosive wear in transfer lines and cyclones at synfuels pilot plants, Paper presented at the Corrosion/84, International Corrosion Forum Devoted Exclusively to the Protection and Performance of Materials
- Yuu, S., Jotaki T., Tomita, Y., Yoshida, K. (1978) the reduction of pressure drop due to dust loading in a conventional cyclone. *ChE. Eng. Sci.*, 33(12), 1573-1580.
- Zhao, B., Su, Y., Zhang, J. (2006). Simulation of gas flow pattern and separation efficiency in cyclone with conventional single and spiral double inlet configuration. *Chemical Engineering research and Design*, A12 (84), 1158-1165.



## **Computational Fluid Dynamics**

Edited by Hyoung Woo Oh

ISBN 978-953-7619-59-6

Hard cover, 420 pages

**Publisher** InTech

**Published online** 01, January, 2010

**Published in print edition** January, 2010

This book is intended to serve as a reference text for advanced scientists and research engineers to solve a variety of fluid flow problems using computational fluid dynamics (CFD). Each chapter arises from a collection of research papers and discussions contributed by the practiced experts in the field of fluid mechanics. This material has encompassed a wide range of CFD applications concerning computational scheme, turbulence modeling and its simulation, multiphase flow modeling, unsteady-flow computation, and industrial applications of CFD.

### **How to reference**

In order to correctly reference this scholarly work, feel free to copy and paste the following:

Utikar, R., Darmawan, N., Tade, M., Li, Q, Evans, G., Glenn, M. and Pareek, V. (2010). Hydrodynamic Simulation of Cyclone Separators, Computational Fluid Dynamics, Hyoung Woo Oh (Ed.), ISBN: 978-953-7619-59-6, InTech, Available from: <http://www.intechopen.com/books/computational-fluid-dynamics/hydrodynamic-simulation-of-cyclone-separators>

# **INTECH**

open science | open minds

### **InTech Europe**

University Campus STeP Ri  
Slavka Krautzeka 83/A  
51000 Rijeka, Croatia  
Phone: +385 (51) 770 447  
Fax: +385 (51) 686 166  
[www.intechopen.com](http://www.intechopen.com)

### **InTech China**

Unit 405, Office Block, Hotel Equatorial Shanghai  
No.65, Yan An Road (West), Shanghai, 200040, China  
中国上海市延安西路65号上海国际贵都大饭店办公楼405单元  
Phone: +86-21-62489820  
Fax: +86-21-62489821