MODELLING OF CYCLONE SEPARATORS USING CFD

A DISSERTATION
Submitted in partial fulfillment of the
requirements for the award of the degree
of
MASTER OF TECHNOLOGY
in
CHEMICAL ENGINEERING
(With Specialization in Computer Aided Process Plant Design)

By
Y. ALLWYN ABRAHAM

DEPARTMENT OF CHEMICAL ENGINEERING
INDIAN INSTITUTE OF TECHNOLOGY ROORKEE
ROORKEE - 247 667 (INDIA)
JUNE, 2008

INDIAN INSTITUTE OF TECHNOLOGY ROORKEE
ROORKEE

CANDIDATE'S DECLARATIONS

I hereby certify that the work which is being presented in the dissertation entitled **MODELLING OF CYCLONE SEPARATORS USING CFD** in partial fulfilment of the requirements for the award of the degree of Master of Technology, submitted in the Department of Chemical Engineering, Indian Institute of Technology Roorkee, is an authentic record of my own work carried out during a period from June 2007 to June 2008 under the supervision of Dr. Vijay Kumar Agarwal, Professor, Department of Chemical Engineering, Indian Institute of Technology Roorkee, Roorkee.

The matter embodied in this dissertation has not been submitted by me for the award of any other degree of this or any other Institute.

Y.Allwyn

(Y. Allwyn Abraham)

This is to certify that the above statement made by the candidate is correct to the best of my knowledge.

(Prof. Vijay K. Agarwal)
Supervisor

Date: 30 June, 2008

ABSTRACT

Cyclones are devices that employ the resulting force between the drag, gravitational, inertial and centrifugal force to separate particles from the carrier gas. Due to these they follow a high turbulence. The CFD (Computational Fluid Dynamics) appears to be a strong engine to predict engineering projects. They are capable of modelling the high turbulence in the cyclones by the use of the equations of continuity and momentum balance. These equations are applied to the problem of the cyclone flow and were solved using FLUENT 6.2.16. This study investigates the effect of cyclone outlet (vortex finder) length and the cylindrical portion length on cyclone pressure drop

A series of numerical experiments were performed by using the computational fluid dynamics technique (CFD) for the evaluation and validation of three-dimensional models for cyclones. The cases sought to reproduce the simulated geometry and operating conditions given by Lapple (1951)and that of Stairmand.

The simulations were out using the CFD software FLUENT, the discrete phase model was used along with the RSM model of turbulence (Reynolds Stress Model) and second order upwind for the variables descriptive.

The first part of the preliminary study involved simulations of a cyclone with geometric characteristics given by Stairmand using the RNG k-Є model and the RSM. As values calculated by the RSM model were much closer to the experimental values, the same was used for further studies.

The computed pressure drop values from six different geometries clearly indicates that at low cylinder heights, the pressure drop was high and at less vortex length the pressure drop were less. Hence for cyclones operating at high velocity, the pressure can be greatly reduced by increasing the cylinder height or by decreasing the vortex length. This may be due to the additional parts which exceed cyclone natural vortex length. Although there is no vortex they may serve as a chamber to release air pressure It was also found that, the vortex length increases the swirl flow inside the cyclone separators and thereby showing the greater chances for high separation efficiencies.

ACKNOWLEDGEMENT

I dedicate this work to my Lord and Saviour, for His glory and honour.

I am highly indebted to my father S. Yesu Joshwa, mother Jothy Joshua, and my brother Jeba Selwyn for their support and motivation. Words cannot express fully their love and affection.

I am grateful to my thesis Supervisor, Dr. Vijay Kumar Agarwal, Professor, Department of Chemical Engineering, IIT Roorkee for his keen interest in me and my work has made this thesis possible. His warm personal approaches and painstaking efforts in going through the manuscript are gratefully acknowledged.

I am highly obliged to the Head of the Department of Chemical Engineering, I express a sense of gratitude to all faculty members for their valuable suggestions and help. I wish to thank my fellow students, Mr. Sagar G. Wankhede, my lab friends kusuma, Mohamad Rashid and my dear brother and friend Mr. Edwin Raj. R for their help and encouragement in my work.

I thank Dr. B.S.S. Daniel-Supriya and family, Mr. Emmanuel-Phoebe & family, Mr. R. Edwin Raj & family, Pr. Rajan Thomas & family, Mr. Krishnamoorthy & family for the supporting and comforting fellowship to cherish and making my time in Roorkee as purposeful by their tremendous love and affection that created me the home feel in Roorkee.

I treasure the happy and sad moments, shared and enjoyed with my friends in IIT campus, especially with Mr. Ashok Kumar, Mr. Arun Kumar, G. Swetha, M.V. Niharika, Mr. Ananth Kumar, Mr. Sathish Kumar & family, Mr. Ganapathy,

At this moment, I am thankful to all of my friends back home especially, I.R. Jelin Edwin, J. Raman Kumar and C. Jayageeth. I am indebted to my mentors R. Edwin Raj, P. Emmanuel Shubakhar for their motivation and encouragement during tough times.

Y.Allwyn Abraham

TABLE OF CONTENTS

List of Tables

List of figures

Nomenclature

a	Cyclone inlet duct width
b	Cyclone inlet duct height
D	Cyclone diameter, dissipation length scale
D_e	Outlet pipe diameter
D/Dt	Total time derivative
d_P	Particle diameter
N_T	Number of trajectories
N_P	Local number of particles per package and/or cell
p	Static pressure
t	Time
v	Fluid velocity
v_r	Relative velocity between particle surface and wall
v_{rel}	Relative velocity between particle and fluid
VP	Particle volume
x, y, z	Cartesian coordinates
ϵ	Dissipation rate
ρ_F, ρ_P	Density of the fluid and particle

Chapter 1
Introduction

1.1 Separation Processes

In chemical engineering, a separation process is used to transform a mixture of substances into two or more compositionally-distinct products. Barring a few exceptions, almost every element or compound is found naturally in an impure state such as a mixture of two or more substances. Many times the need to separate it into its individual components arises. Separation applications in the field of chemical engineering are very important. Separation processes can essentially be termed as mass transfer processes. The classification can be based on the means of separation, mechanical or chemical. The choice of separation depends on the pros and cons of each. Mechanical separations are usually favoured if possible due to the lower cost of the operations as compared to chemical separations. Systems that can not be separated by purely mechanical means, chemical separation is the remaining solution.

Depending on the raw mixture, various processes can be employed to separate the mixtures. Many times two or more of these processes have to be used in combination to obtain the desired separation. In addition to chemical processes, mechanical processes can also be applied where possible.

1.1.1 Gas Solid separation processes

Year by year, requirements for controlling particulates in gases entering the atmosphere tend to become more stringent and hence gas – solid separation processes play a vital role in industries. Their separation becomes much more tedious as the size of the solid particle decreases. There is a wide variety of devices for separating particulates from gas streams before the gases are discharged into the atmosphere. The term "particulates" includes both solid particles and liquid droplets light enough to be swept along in a flowing gas stream. All such particles are small, but size difference is so significant that particle size is measured in microns. These many devices have evolved because industrial processes vary greatly in the nature of the particulate matter that they generate, as well as in the conditions under which particulate collection must be accomplished.

Particulate collectors may be classified as:

- Mechanical collectors.
- Wet scrubbers.
- Fabric filters.
- Electrostatic precipitators.

The many different dust collectors available today according to L. Theodre and A.J.Buonicore is summarized as shown in Table 1, which is a simplified review of the whole collector field, ranges and limits tabulated are typical values but naturally may vary widely for unusual applications. Suppliers can be found listed in the directory at the back of this issue and should be consulted especially for integrated systems, packaged units and air cleaners.

Table 1.1 Different dust collectors – range and limits

Types of dust collecting equipments	**Particle size, microns**	**Loadin grains per cu. Ft**	**Collection Efficiency Weight %**	**Pressure Loss**		**Utilities per 100cfm**	**Gas velocity, Fpm**	**Size range limits, 100Cfm**	**Space Required (Relative)**
				Gas, in WG	**Liquid, in Psi**				
Dry Inertial Collectors									
Settling chamber	>50	>5	< 50	< 0.2	-------	--------------	300 – 600	None	Large
Baffle chamber	>50	>5	< 50	0.1– 0.5	---------	--------------	1000-2000	None	Medium
Skimming chamber	>20	>1	< 70	< 1	---------	--------------	2000 – 4000	50	Small
Louver	>20	>1	< 80	0.5 – 2	---------	--------------	2000 – 4000	30	Medium
Cyclone	>10	>1	< 85	0.5 – 3	---------	--------------	2000 – 4000	50	Medium
Multiple cyclone	>5	>1	< 95	2 – 6	---------	--------------	2000 – 4000	200	Small
Impingent	>10	>1	< 90	1 – 2	---------	--------------	3000 – 6000	None	Small
Dynamic	>10	>1	< 90	Provides head	——	1 - 2 hp	------------	50	
Wet scrubbers									
Gravity Spray	> 10	>1	< 70	< 1	20 – 100	0.5-2 gpm	100-200	100	Medium
Centrifugal	>5	>1	< 90	2 – 6	20 – 100	1-10 gpm	2000-4000	100	Medium
Impingement	>5	>1	< 95	2 – 8	20 – 100	1-5 gpm	3000-6000	100	Medium
Packed bed	>5	>0.1	< 90	1 – 10	5 – 30	5-15 gpm	100-300	50	Medium
Dynamic	>1	>1	< 95	Provides head	5 - 30	1-5 gpm 3-20 hp	3000-4000	50	Small
Submerged nozzle	< 2	> 0:1	< 90	2 – 6	None	No pumping	3000	50	Medium
Jet	0.5 - 5	> 0.1	< 90	Provides head	50–100	50 – 100 gpm	2000-20000	100	Small
Venturi	> 0.5	> 0.1	< 99	10 – 30	5 – 30	3-10 gpm	12000-42000	100	Small
Fabric filters	> 0.2	> 0.1	< 99	2 – 6	---------	------------	1-20	200	Large
Electrostatic precipitators	< 2	> 0.1	< 99	0.2 - 1	---------	0.1–0.6 kw	100-600	10-2000	Large

1.2. Centrifugation

Centrifugation or Cyclonic separation is a method of removing particulates from an air (or gas) stream, without the use of filters, through vortex separation. Rotational effects and gravity are used to separate mixtures of solids and fluids. A high speed rotating air flow is established within a cylindrical or conical container called a cyclone. Air flows in a spiral pattern, beginning at the top (wide end) of the cyclone and ending at the bottom (narrow) end before exiting the cyclone in a straight stream through the center of the cyclone and out the top. Larger (denser) particles in the rotating air stream have too much inertia to follow the tight curve of the air stream and strike the outside wall, falling then to the bottom of the cyclone where they can be removed.

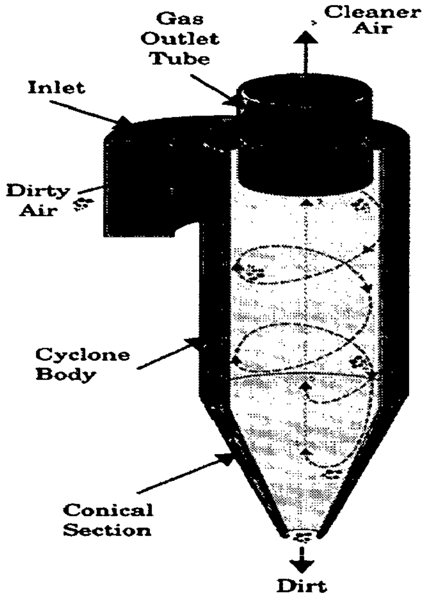
1.2.1 Cyclone Separator - operation and classification

Centrifugal separators commonly referred to as cyclone separator, are used in industries for the removal of solid and liquid matter from gas streams. Particles suspended in a moving gas stream possess inertia and momentum and are acted upon by gravity. If the gas stream be forced to change direction, these properties can be utilized to promote centrifugal forces to act on the particles. In the conventional units the entire mass of the gas stream with the entrained particles tangentially enters the unit and forced into a constrained vortex in the cylindrical portion of the cyclone. Upon entering the unit, a particle develops an angular velocity. Due to its inertia, it tends to move outward across the gas stream lines in a tangential rather than a rotary direction; thus attaining a net outward radial velocity. By virtue of its rotation with the carrier gas around the axis of the tube and its higher density with respect to the gas, the entrained particle is forced toward the wall of the unit. Eventually the particle may reach the outer wall where it is carried by gravity, and assisted by the downward movement of the outer vortex and/or secondary eddies, towards the dust outlet at the bottom of the unit.

Cyclones may be generally classified into four categories, depending on the entry of the gas stream and on exit of the collected particles.

1. Tangential inlet and axial dust discharge.
    a. Conventional (large diameter).
    b. High efficiency (small diameter, less than 10 in.)
2. Tangential inlet and peripheral dust discharge.
3. Axial inlet and axial dust discharge.
4. Axial inlet and peripheral dust discharge.

Fig. 1.1 cross section of a cyclone separator



Types 1 and 3 are the most widely used. Large-diameter cyclones, with body diameters three to five times the diameter of the inlet duct, are useful where large gas handling capacity and moderate particle collection efficiency are required. The ratio of gas volume to capital investment dollar is greater than that for most cleaning devices. As can be deduced from the definition of "high-efficiency" cyclones, decreasing the body diameter will increase the efficiency. This is due to increased separation forces caused by the smaller vortex radius. Individual high-efficiency, small-diameter cyclones have a small capacity, and they must be operated in parallel to handle typical gas volumes. They generally have a common gas inlet, dust hopper, and gas outlet and can be arranged in banks of up to several hundred cyclones each.

The cone portion of the cyclone is not necessary to convert the downward vortex to an upward one, although its presence does reduce the length of cyclone needed to effect this reversal. As the inlet gas stream enters the annular region at the top of the cyclone, it is squeezed by the existing gas to about half of its inlet width. This causes a significant pressure loss, which can be reduced by the addition of vanes to the annular area. The presence of the vanes, however, reduces the efficiency, apparently due to the prevention of vortex formation in the annulus. Helical and involute inlets are attempts to reduce interference between the incoming gas and the vortex already present in the annulus. Axial inlets are free from most of these problems. However, they introduce new problems; the inlet vanes must be designed so that they impart adequate rotation to the gas and yet resist erosion and plugging.

1.3 Literature gap

Since the first application of aerocyclones in 1886, several models have been developed for the calculation of the design parameters, pressure drop and efficiency. All these approaches can be divided into six groups.

Due to the physically simplified assumptions and the high calculation expenditure, the use of the first four models is rather limited and only the residence time models and force balance models are used widely.

A full understanding of how the cyclone separator works and how individual particles behave within it is not yet available. Little information has been gathered until the invention of the measuring equipment necessary to measure fluid velocities within cyclones (laser Doppler anemometry - LDA). Ultimately, it was only due to the development of computational fluid dynamics (CFD) codes that was able to accurately model the swirling flow within cyclones. CFD study revealed the drawbacks and the inaccuracy in the classical models.

Eventhough the cyclone separators were in use for a longer period of time, only a less investigations has been carried on the effect of the geometrical parameters like the vortex length. The computational fluid dynamics, study of these parameters were still much lower.

Furthermore, many practical issues such as outlet and inlet configurations have not been investigated properly or at all. A relatively much lesser work only has been carried on the cyclone geometries.

1.4 Objective of the study

- The investigations were aimed at studying and controlling the instability of vortex flow, as well as examining the effect of outlet piping (vortex finder) on cyclone performance.
- This work aims to examine the effect of the axial dimensions.
- The effect of the vortex length and the height of the cylindrical section over pressure drop is studied
- The Re-normalization group theory and the RSM models are to be tested using the stairmand cyclone and the suitable model among the two were to be used for further simulations.

Chapter 2
Literature Review

A detailed analysis and discussion on modelling the gas and particle flow inside cyclone separators was carried out by Cristo bal Corte's and Antonia Gil recently in 2007. They have reviewed the models developed for the flow field inside inverse-flow cyclone separators. In a first part, traditional algebraic models and their foundations are summarized in a unified manner, including the formulae for tangential velocity and pressure drop. The immediate application to the prediction of collection efficiency is also reviewed. The approach is the classical, treating first the dilute limit (clean-gas correlations), and afterwards correcting for ''mass loading'' effects. They found that, although all the classical methods have had a remarkable success, more advanced ideas are needed to model cyclones. This is put forward by exploring the work done on the so-called ''natural'' length of the cyclone, that has led to the discovery of instability and secondary flows. The resort to computational fluid dynamics (CFD) in this case is difficult, however, due to the very nature of the flow structure. A closing section on the subject reviews past and recent CFD simulations of cyclones, the single-phase and two-phase, steady and unsteady, aiming at delineating the state-of-the-art, present limitations and perspectives of this field of research.

A. Avci and I. Karagoz, Department of Mechanical Engineering, University of Uludag have developed a mathematical model of two phase flow in tangential cyclone separators, by definitions of new parameters including the effects of cyclone geometry, surface roughness and concentration of particles. The critical diameters, fractional efficiencies and pressure losses are calculated under the assumptions that each phase has the same velocity and the same acceleration in the spiral motion of the flow, a relative velocity occurs in the radial direction, and drag coefficient remains constant under certain conditions. The results obtained under these assumptions are compared with their experimental and theoretical counterparts in literature, and very good agreement was observed with the experimental.

It was found that, One of the important parameters is the cyclone height. Experiments prove that efficiency increase with increase of height. But it should be a limit for the height due to fi'iction and separation from the surfaces and this should be analyzed experimentally. The fractional efficiencies obtained in this study were also satisfactory in despite of, especially, estimation of the shape factor. Although fractional efficiency curves are similar to each other some discrepancies occur, especially, at the below and upper limits of particle diameter. However, drag coefficients and surface roughness should be analyzed in detail with the support of experiments to get more reliable and practical equations which serve to design optimum cyclone separators.

Mathematical models have been developed by G. Ramachandran, P.C. Raynor and D. Leith, for predicting the pressure drop and grade collection efficiency for a rotary flow cyclone. The models are based on simplifying assumptions about the nature of gas flow through the cyclone, as well as the mixing of particles in the gas stream. Despite these simplifications, the predictions of the models are close to experimental measurements of pressure drop and grade collection efficiency, made on an industrial rotary-flow cyclone which is used as a precleaner for an oil mist collection system. Using these models, optimization curves were developed which predicted the dimensions (radius and pitch) of an optimized rotary-flow cyclone that would yield the minimum cut diameter dSO for a given pressure drop. Optimisations were performed for one flow rate. Scaling equations were developed so that the results of the study could be used to design rotary-flow cyclones handling other flows.

The different models of cyclone prediction performance for various operating conditions were studied by S. Altmeyer et al., in 2002 and a comparison has been made between the various classic models using a new software which allows to calculate cyclone efficiency for a given geometry or to determine a geometry for a desired efficiency. It has been established for cyclones with relatively low solids loading (<10 g/m3) and it applies for pressure drop between 10 and 10 000 Pa, for cut diameter between 0.2 and 20 _m, for volumetric flow rate from 10–4 to 1000 m3/s and for clone diameter from 0.01 to 3 m. The calculations are realised with four models presented in the literature. Comparison between model predictions and published measurements shows that models used in the software predict pretty well the experimental results, obtained in a large range of operating conditions. Moreover, a comparison of the results obtained with these four models permits to select the model the most adapted, depending on inlet flow rate, temperature and pressure used.

Bingtao Zhao et al., developed a symmetrical spiral inlet to improve the performance of cyclone separator. Three cyclone separators with different inlet geometry were designed, which included a conventional tangential single inlet CTSI), a direct symmetrical spiral inlet (DSSI), and a converging symmetrical spiral inlet (CSSI). The effects of inlet type on cyclone performance characteristics, including the collection efficiency and pressure drop, were investigated and comparison was made as a function of particle size and flow rate. Experimental result indicated that the symmetrical spiral inlet (SSI), especially CSSI inlet geometry, has effect on significantly increasing collection efficiency with insignificantly increasing pressure drop. In addition, they made a comparison of the results of collection efficiency and pressure drop between the experimental data and the theoretical model.

The Effect of the bottom-contracted and edge-sloped vent-pipe on the cyclone separator performance was studied by Jihui Chen et al., in 2006. The influence of the bottom-contracted and edge-sloped vent-pipe on the separation efficiency and pressure drop of a cyclone separator under different vent-pipe insert depth and different orientation of the sloped edge were studied at a visual cold circulating fluidized bed (CFB) experimental setup according to a commercial 100MW CFB boiler with a scale of 20:1. And the correlative results were also compared with the traditional linear-pipe-shaped cyclone separator. Results indicate that the cyclone inlet stream velocity has a strong influence on the separation efficiency and pressure drop, and the results are similar to that of the conventional cyclone. Namely, both separation efficiency and pressure drop increase with increasing cyclone inlet stream velocity. The separation efficiency of the modified cyclone separator increases firstly and then decreases with the increasing of the vent-pipe insert depth. However, there is not a very clear rule of the effect of vent-pipe insert depth on the pressure drop for modified cyclone separator. Both the separating efficiency and the pressure drop change with the orientation of the sloped edge, and they have the same rule of change, the maximum at 90° and the minimum at 270°. Due to the configuration of the bottom-contracted and edge-sloped vent-pipe is suitable with the flow field inside cyclone separator, the separation efficiencies of the modified cyclone separator are usually observed to be higher than those of traditional cyclone separators.

The nature of the vortex end in centrifugal separators was studied by A.C. Hoffmann, in 2005. The nature of the vortex end (or the "tail end" or "tip") in reverse-flow centrifugal gas cleaning equipment, cyclones and swirl tubes, has been observed by visualization using a stroboscope and high-time-resolution pressure measurements. The core of the vortex seems to bend to the wall of the separator, and rotate around the wall, forming the ring-shaped pattern normally observed at the vortex end. The end of the vortex was found to occur higher in the swirl tube ("less stable vortex") when the solid loading is increased, and when the volumetric flow rate to the tube is decreased. The frequency with which the vortex core rotates varies with the gas flow rate and was found to be about the same as the frequency with which the gas rotates higher in the separator. The time-averaged wall pressure suddenly decreases in the separator body at the point where the vortex end attaches to the wall.

Studies were made on experimental flow patterns, determined by Laser Doppler Anemometry (LDA) for two types of reverse-flow centrifugal separators, by P.J.A.J. Boot et al., The flow patterns in a conventional cylinder-on-cone cyclone with tangential inlet and a swirl tube with vane-generated swirl and a cylindrical body are compared. The experimental data are also used to test the validity of the flow assumptions of some widely used cyclone separation models for each of the two devices. The measured flow patterns are also compared with the results of computational fluid dynamics (CFD) simulations wherever this was helpful for the interpretation. Although the data globally support the standard flow assumptions, some features were quite surprising; for instance, it appears that the surface delimiting the central region of upward flow is largely determined by the diameter of the body and not that of the vortex finder. The similarities and differences between the flow patterns in cyclones and swirl tubes are discussed, as is the suitability of using cyclone separation models for the prediction of swirl tube performance.

A detailed flow analysis inside cyclone diplegs equipped with bottom gas extraction was presented by A. Gil, L.M. Romeoet al., in 2002. Tests varying inlet solid loading and gas extraction flow rate were carried out in a one-fifth cold-flow model of a PFBC cyclone. Velocity and pressure measurements along cyclone dipleg have been carried out. Results from experimental measurements reveal that the swirling flow penetrates inside the dipleg, deeper than can be anticipated by literature predictions on conventional cyclone designs. This effect is caused by the high inlet velocities of this kind of cyclones and the gas suction at dipleg bottom. Dipleg pressure coefficient is shown to be a function of the inlet solid loading and the tangential to axial velocity ratio. Although for standard cyclone designs it has been claimed that underneath the vortex end there is a stagnant region where solid recirculation and re-entrainment can occur, it has been experimentally verified that even with a small percent of gas extracted at the bottom a substantial tangential velocity component is induced at dipleg bottom. This velocity assures solids conveyance to the extraction device. As solid loading is increased, an upward movement of the vortex end is detected from wall pressure measurements. The solid loading causes an important decrease of the vortex energy and, consequently, a weakening of the dipleg tangential velocity. A new method for measuring the vortex penetration in dipleg was presented.

A Numerical simulation of complex particle–fluid flows was carried out by K.W. Chu, A.B. Yu. They presented a numerical study of particle–fluid flow in complex three-dimensional (3D) systems by means of Combined Continuum and Discrete Method (CCDM). In the CCDM, the motion of discrete particles phase is obtained by Discrete Element Method (DEM) which applies Newton's laws of motion to every particle and the flow of continuum fluid is described by the local averaged Navier–Stokes equations that can be solved by the traditional Computational Fluid Dynamics (CFD). This method has been increasingly used worldwide, but so far its application is limited to relatively simple flow systems. The simulation is achieved by incorporating a DEM code into the commercial CFD software package Fluent that can be readily used for complex CFD problems. The applicability of this development is demonstrated in the study of the particle–fluid flow in various 3D systems including pneumatic conveying bend, cyclone separator and circulating fluidized bed. It is shown that the numerical results are, either qualitatively or quantitatively depending on the availability of experimental data for comparison, in good agreement with those measured, and can generate information leading to better understanding of the internal flow structure of these systems.

The performance of different analytical methods in evaluating the grade efficiency of centrifugal separators was done by Madhumita B. Ray er al., The results were presented from four different particle size analysis techniques used to characterize the performance of a novel centrifugal gas}solid separator. The Post Cyclone (PoC), is a new secondary collector situated at the top of a conventional reverse flow cyclone which utilizes the residual swirl available at the outlet (vortex finder) to capture some of the escaped dust. A prototype PoC was tested for a Stairmand high-efficiency cyclone (diameter"0.4 m) under a range of operating conditions. It is inherent to the process that the size distributions of the captured and escaping dust from the PoC are very close. Moreover, only little dust bigger than 5 km escaped from the cyclone to the PoC, making grade effciency curves unreliable above this particle size. This means that demands on the quality of the particle sizing for this application are especially strenuous. Therefore, four different types of analytical equipment, namely a disc centrifuge, a cascade impactor, a cyclone train and a laser scattering sizer were used to collect information about the size distribution of the particles. This work highlights the apparent merits and shortcomings of each of these methods and compares the experimental grade efficiencies obtained by the different methods.

Simulation of mass-loading effects in gas–solid cyclone separators was studied by J.J. Derksen et al., and the three-dimensional, time-dependent Eulerian–Lagrangian simulations of the turbulent gas–solid flow in a cyclone separator have been performed. The Eulerian description of the gas flow is based on lattice-Boltzmann discretization of the filtered Navier–Stokes equations, where the Smagorinsky subgrid-scale model has been used to represent the effect of the filtered scales. Through this large-eddy representation of the gas flow, solid particles with different sizes are tracked. By viewing the individual particles (of which there are some 107 inside the cyclone at any moment in time) as clusters of particles (parcels), we study the effect of particle-to-gas coupling on the gas flow and particle behavior at appreciable mass-loading (0.05 and 0.1). The presence of solid particles causes the cyclone to lose some swirl intensity. Furthermore, the turbulence of the gas flow gets strongly damped. These two effects have significant consequences for the way the particles of different sizes get dispersed in the gas flow. It is anticipated that the collection efficiency gets affected in opposite senses: negatively by the loss of-swirl, positively by the reduced turbulence.

A study of the effect of high inlet solids loading on a cyclone separator pressure drop and collection efficiency was done by Fassani and Leonardo. The particles used were FCC catalyst. An extended range of concentrations, up to 20 kg of solidsrkg of gas was used, well beyond the loading range reported before. The average entrance velocities were 7, 18 and 27 mrs. The experiments showed that, in the range of concentrations tested, the cyclone pressure drop for the solids laden air flow was about 47% of that for clean air. A trend of increasing separation efficiency with concentration was observed, up to 12 kg of solids/kg of gas, above which, the efficiency decreased. At test conditions, the collection efficiency was higher for the entrance velocity of 18 m/s than for 27 m/s.

The operation and mechanistic modelling of a cyclone gas–liquid separator was performed by E.S. Rosa et al., They focused on testing scaled-down models and prototypes and developing mechanistic modelling for the phase separation and flow hydrodynamic processes. Description on the operational principles of the cyclone separator, discloses laboratory and field data and presents the modelling foundations. The laboratory tests were performed in downsized models operating with mixtures of air and water or water-based viscous liquids. The analysis of experimental data, extensive flow visualization and the identification of the operational constraints set the basis for the mechanistic modelling. The capability of the model to represent the separation processes was checked against field tests conducted with actual fluids in full dimension prototypes. Based on these results, prospective field applications are also presented. B. Wang et al., presents a numerical study of the gas–powder flow in a typical Lapple cyclone. The turbulence of gas flow is obtained by the use of the Reynolds stress model. The resulting pressure and flow fields were verified by comparing with those measured and then used in the determination of powder flow that is simulated by the use of a stochastic Lagrangian model. The separation efficiency and trajectory of particles from simulation are shown to be comparable to those observed experimentally. The effects of particle size and gas velocity on separation efficiency are quantified and the results found to agree well with experiments. Some factors which affect the performance of cyclone were identified. It is shown that the collision between gas streams after running about a circle and that just entering occurred around the junction of the inlet duct and the cylinder of the cyclone, resulting in a short-circuiting flow. The combination of flow source and sink was distributed near the axis of cyclone, forming a flow dipole at axial section. Particles entering at different positions gave different separation efficiency. A particle with size exceeding a critical diameter, which was condition-dependant, would stagnate on the wall of cyclone cone. This was regarded as one of the main reasons for the deposition on the inner conical surface in such cyclones used in the cement industry.

The hydrodynamic flow behavior in a Gas-Liquid Cylindrical Cyclone (GLCC) compact separator is studied experimentally and theoretically by S. Movafaghian et al., New experimental data were acquired utilizing a 7.62 cm I.D, 2.18 m high, GLCC separator for a wide range of operating conditions. Investigated parameters included three different inlet geometries (5.08 cm I.D single, 7.62 cm I.D single and 7.62 cm I.D dual inlets), four different liquid viscosities (1, 2.5, 5 and 10 cps), three system pressures (101.3, 273.6 and 487.2 kPa), and the effect of surfactant. The measured data comprise of equilibrium liquid level, zero-net liquid flow holdup and the operational envelope for liquid carry-over. The data was utilized to verify and refine an existing GLCC mechanistic model. Comparison between the modified model predictions and the experimental data showed a very good agreement.

Theoretical analysis of pressure losses in cyclone separators under the consideration of geometrical and flow parameters including inlet geometry, surface roughness, velocity and particles concentration, has been performed by Karagoz and a new equation has been developed. The results obtained in this study are compared with experimental values for different type of cyclones. It was found that the proposed equation could be used to predict the pressure losses easily and it is worthy especially industrial applications.

A simple model has been predicted for solid collection efficiency of a gas–solid separator by R. Zhang and P. Basu. The semiempirical model developed for the prediction of the grade efficiencies of a gas–solid cyclone for circulating fluidized bed (CFB) boilers is based on the equilibrium orbit theory. Experiments were conducted in a large cyclone to gather data on grade efficiencies for model verification and the model was found to give a good agreement between predicted and experimental values of grade efficiencies.

A universal model is developed by Jianyi Chen and Mingxian Shi to calculate cyclone pressure drop. The definition and composition of the pressure drop over a tangential inlet, reverse flow cyclone was analyzed. It is assumed that two factors mainly contribute to the pressure drop, i.e., the local loss and the loss along the distance. The former includes the expansion loss at the cyclone inlet and the contraction loss at the entrance of the outlet tube (or vortex finder). The latter consists of the swirling loss resulting from friction at the cyclone walls and the dissipation of gas dynamic energy in the outlet tube. By use of the measured results of the flow field in cyclones, the calculation methods for each loss have been developed. And a universal model to predict the cyclone pressure drop is thus obtained simply by summing each loss. A detailed comparison between the calculated and experimental results shows that this accurate model is suitable either for pure or for dust laden gases at normal or high temperatures and can meet the requirement of most cyclone designs.

The cut size was predicted by M. Narasimha et al., by modelling hydrocyclone using CFD.. The shape and size of a hydrocyclone separator has a direct influence on the internal flow structure of the continuous phase and, thereby, the separation of the particulate phase. Hydrocylcones usually have a single inlet that distributes the feed stream near the end wall between the vortex finder and the sidewall. Effect of spigot diameter, i.e., 10 and 20 mm and inlet water velocities (5.91–12.35 m/s) on the water splits and particle classification in the hydrocyclone have been studied. The cut size of the hydrocyclone, operated at very low pulp density, has been predicted using discrete phase modelling technique. The studies revealed that with an increase in feed flow rate and decrease in spigot diameter the cyclone sharpness of separation improves. These predictions were found to be similar in line with the experimental observations.

The effects of flow and geometrical parameters on the collection efficiency in cyclone separators was studied by Atakan Avci, Irfan Karagoz, through the development of a mathematical model for calculation of cut-off size and fractional efficiencies in cyclone separators, by taking into account the effects of flow, particle and geometrical parameters, and acceleration assuming that the mixture of fluid and particles is homogenous, and acceleration diminishes depending on the friction and geometry. Collection efficiency curves and cut-of size values predicted by the proposed model showed a good agreement with experiments over a wide range of inlet velocities for different types of cyclones. Comparison of the obtained results with semi empirical models available in literature also indicated that the present model may be used successfully for determination of the performance of a tangential inlet cyclone. Analyses of the effects of various parameters reveals that, in addition to flow and geometrical parameters, surface friction, vortex length and ow regimes play an important role on cyclone performance especially in small cyclones.

The cut-size was predicted by M.S. Brennan through the Multiphase modelling of hydrocyclones. A comprehensive multiphase model of cyclone separators using Computational Fluid Dynamics is under development. The model is capable of predicting velocity profiles, flow splits, air core position and efficiency curves in classifying hydrocyclones. The model approach uses the Mixture model with the granular options and large eddy simulation (LES) to resolve the turbulent mixing of the particles. Multiphase simulations of Hsieh's [Hsieh, K.T., 1988. A phenomenological model of the hydrocyclone, Ph.D. thesis, University of Utah] data show a very good prediction of the cyclone efficiency curve. Whilst further model development is needed, the approach is showing promise as a cyclone design tool.

The turbulent characteristics of the gas–solid suspension in a square cyclone separator were observed by Yaxin Su. Three-dimensional particle dynamics analyzer was employed to study the gas–solid flow in a square-shaped cyclone separator which was designed for large CFB application. Distribution of flow vector, fluctuating velocity, turbulent kinetic energy, turbulent intensity and particle concentration were discussed. The swirling flow inside the cyclone showed the Rankine vortex characteristics, i.e., strong swirling vortex at the central region of the cross-section and weak swirling quasi-free vortex near the wall. The quasi-laminar motion of particles enhanced the turbulent movement at the corners due to particle–particle/wall collision, which led to the local peak value of the turbulent kinetic energy and turbulent intensity. The corner is one of the major region to cause pressure drop because the suspension at the corners consumed more energy of the flow. The corners were found to be beneficial to particle separation mainly because the strong fluctuating flow consumed much of the kinetic energy of both the particle and the gas.

The different works in the classical models was summarised briefly by Cristobal Corttes as described in table 2.1

Table 2.1 – Widely used classical models

Model	Equation		Remarks
Shepherd and Lapple [42]	$\xi_g = \frac{16ab}{D_e^2}$	(35)	Tangential inlet; ambient air conditions
Alexander [11]	$\xi_g = 4.62\left(\frac{ab}{D_c D_e}\right)\left[\left(\left(\frac{D_c}{D_e}\right)^{2n} - 1\right)\left(\frac{1-n}{n}\right) + f_g\left(\frac{D_c}{D_e}\right)^{2n}\right]$	(36)	Experiments with scroll and tangential inlets
	$f_g = 0.8\left[\frac{1}{n(1-n)}\left(\frac{4-2^{2n}}{3}\right) - \left(\frac{1-n}{n}\right)\right] + 0.2\left[(2^{2n}-1)\left(\frac{1-n}{n}\right) + 1.5(2^{2n})\right]$	(37)	Air and combustion gases, up to 1100 °C
	$n = 1 - (0.67 D_c^{0.14})\left(\frac{T}{283}\right)^{0.8}$	(4)	
Barth [13]	$\xi(\lambda = \lambda_g) = \left(\frac{ab}{\pi D_e^2/4}\right)^2 (\xi_b + \xi_e)$	(38)	
	Loss in the cyclone body $\xi_b = \frac{D_e}{D_c}\left(\frac{1}{(v_{ze}/v_{te} - ((H-S)/(0.5D_e))\lambda)^2} - \left(\frac{v_{te}}{v_{ze}}\right)^2\right)$	(39)	
	Loss in the vortex finder $\xi_e = K\left(\frac{v_{te}}{v_{ze}}\right)^{4/3} + \left(\frac{v_{te}}{v_{ze}}\right)^2$	(40)	$3.41 < K < 4.4$
Muschelknautz and Kambrock [43]	$\xi(\lambda = \lambda_g) = \left(\frac{ab}{\pi D_e^2/4}(\xi_b + \xi_e)\right)$	(41)	Tangential and scroll inlets
	$\xi_b = \lambda \frac{A_S}{0.9V}\frac{\rho_g}{2}(v_{tw} v_{te})^{1.5}$	(42)	Flow field based on Barth's model [13]
	$\xi_b = 2 + 3\left(\frac{v_{te}}{v_{ze}}\right)^{4/3} + \left(\frac{v_{te}}{v_{ze}}\right)^2$	(43)	Ambient P,T conditions $\lambda = \lambda_g \approx 0.006$ A_S is the total inner area of cyclone contributing to friction
Casal et al. [44]	$\xi_g = 11.3\left(\frac{ab}{D_e^2}\right)^2 + 2.33$	(44)	Comparative study of six correlations

CHAPTER 3
MODELLING OF CYCLONE SEPARATORS

3.1 MODELLING OF CYCLONES USING CFD

A full understanding of how the cyclone separator works and how individual particles behave within it is not yet available. Little information has been gathered until the invention of the measuring equipment necessary to measure fluid velocities within cyclones (laser Doppler anemometry - LDA). Ultimately, it was only due to the development of computational fluid dynamics (CFD) codes that was able to accurately model the swirling flow within cyclones.

3.1.1 Ideal Vortex Laws from the Navier-Stokes Equations

The Navier stokes equation forms the basis for any CFD modelling and calculations. Hence the essential equations for swirling flow were derived from the basic equations of fluid mechanics: the Navier-Stokes equations. For an incompressible fluid the equation of continuity and momentum are,

$$\nabla \cdot \mathbf{v} = 0 \qquad \text{------------- } 3.1$$

$$\rho \frac{D\mathbf{v}}{Dt} = -\nabla p - \nabla \cdot \tau + \rho \mathbf{g} \qquad \text{------------- } 3.2$$

The terms in the above equation from left to right are,

- The mass times acceleration per unit volume. This is the density multiplied by the absolute (or 'material') derivative of the velocity. The material derivative *D/Dt* gives the acceleration of a fluid element in a Eulerian3 frame of reference.
- The net force due to normal stresses per unit volume.
- The net force due to shear stresses per unit volume. τ is the 'deviatoric' stress tensor, meaning that the pressure has been subtracted from the total stress tensor, so that the sum of the three diagonal elements is zero. This essentially leaves us with the shear stresses.
- The gravitational force per unit volume.

The momentum equation in terms of co-ordinates can be written as,

$$\rho\left(\frac{\partial v_\theta}{\partial t}+v_r\frac{\partial v_\theta}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_\theta}{\partial \theta}+\frac{v_r v_\theta}{r}+v_z\frac{\partial v_\theta}{\partial z}\right)= \quad -\frac{1}{r}\frac{\partial p}{\partial \theta}-\left(\frac{1}{r^2}\frac{\partial}{\partial r}\left(r^2\tau_{r\theta}\right)+\frac{1}{r}\frac{\partial \tau_{\theta\theta}}{\partial \theta}+\frac{\partial \tau_{\theta z}}{\partial z}\right)+\rho g_z.$$

In the shear stress components τ, the first index indicates the plane on which the stress acts, and the second its direction. This complicated looking equation can be simplified to give useful \ information about swirling flow. In steady, axisymmetrical vortex flow with negligible velocity in the *r* and the *z*-directions, the terms listed in Table can be eliminated.

Table 3.1 – Suitable condition to be applied to simplify Navier Stokes Equation

Term eliminated	Reason
I	steady flow, no change with time
II and IV	no radial velocity
III and VI and VIII	no gradients in the θ-direction (axisymmetric flow)
V and IX	no gradients in the z-direction

This leaves with the only term

$$\frac{\partial}{\partial r}\left(r^2\tau_{r\theta}\right)=0 \tag{3.4}$$

By integrating the above expression and relating the shear stress wit the velocity,

$$\tau_{r\theta}=-\mu\left[r\frac{\partial}{\partial r}\left(\frac{v_\theta}{r}\right)+\frac{1}{r}\frac{\partial v_r}{\partial \theta}\right] \tag{3.5}$$

Similarly, the Navier stokes equation for the θ and z components can be solved to get,

$$\frac{\partial p}{\partial r}=\rho\frac{v_\theta^2}{r} \tag{3.6}$$

$$\frac{\partial p}{\partial z}=\rho g_z \tag{3.7}$$

3.2 Turbulence Models

In cyclone separators as the direct turbulence modelling is not yet possible, it is therefore necessary to mimic the effect of the turbulence on the mean flow pattern by means of some model. When the velocity components in the Navier-Stokes equations are split into two parts:

- a fluctuating part due to the turbulence with a mean of zero and
- a mean part,

and the equations are then time-averaged, the effect of the turbulence appearsas extra stresses augmenting those caused by the molecular viscosity. These extra stresses are called the 'Reynolds stresses'. There are 9 Reynolds stresses, 3 in each of the three coordinate planes. It can be shown that the Reynolds stress tensor is symmetrical, so that only 6 of the stresses are independent. Reynolds stresses again give rise to the notion of a 'turbulent viscosity' augmenting the molecular one. In most practical situations, the turbulent viscosity turns out to be much higher than the molecular one. Contrary to the molecular viscosity, the turbulent viscosity needs neither to be homogeneous, *i.e.* the same at all points in the flow field, nor isotropic, *i.e.* the same in all directions.

The most used turbulence models are briefly described below. In the 'k-ε turbulence model', balance equations are solved for the turbulence kinetic energy per unit mass, k, and the dissipation rate of the turbulence per unit mass, ε, and a turbulent viscosity is calculated from these two parameters. The turbulent viscosity found is necessarily isotropic. In the 'Reynolds stress model', or 'RSM' for short, transport equations are solved for all 6 independent Reynolds stresses. These transport equations can be formulated by manipulating the time-averaged Navier- Stokes equations, but it is very time consuming to solve all these coupled equations.

The ASM gives conceptual problems in swirling flows, and for this reason Boysan et al. (1986) formulated a hybrid between ASM and RSM in which transport equations for some stress components are solved, while, for the rest, the algebraic expressions from the ASM were used. The newest technique is 'Large Eddy Simulation' or LES, where the larger eddies, which are mostly responsible for anisotropy in the turbulence, are simulated directly, while the effect of the smaller eddies is accounted for in a simple turbulence model. Thus this is an intermediate step toward direct turbulence modelling.

3.2.1 Reynolds Stress Model

Eddy viscosity models have significant deficiencies. In three-dimensional flows, the Reynolds stress and the strain rate may not be related in such a simple way. This means that the eddy viscosity may no longer be a scalar.

The most complex models in common use are Reynolds stress models which are based on dynamic equations for the Reynolds stress tensor itself. The exact transport equation for the Reynolds-stress tensor uiuj is obtained from the momentum equation 2.3.6 by multiplying the instantaneous component ui equation by the fluctuation velocity u'.

adding the two and then time-averaging the result. The result can be written for constant-density flows neglecting body forces after some rearrangement as

$$\underbrace{\frac{\partial \overline{u_i' u_i'}}{\partial t}}_{C_{ij}} = \underbrace{-\left(\overline{u_i' u_k'}\frac{\partial u_j}{\partial x_k} + \overline{u_j' u_k'}\frac{\partial u_i}{\partial x_k}\right)}_{P_{ij}}$$
$$\underbrace{-\frac{\partial}{\partial x_i}\left[\overline{u_i' u_j' u_k'} + \frac{1}{\rho}\left(\overline{p' u_i'}\,\delta_{ij} + \overline{p' u_j'}\,\delta_{ik}\right) - \nu\frac{\partial \overline{u_i' u_j'}}{\partial x_k}\right]}_{D_{ij}}$$
$$\underbrace{+\frac{p'}{\rho}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)}_{\phi_{ij}} \underbrace{-2\nu\left(\frac{\partial u_i}{\partial x_k}\frac{\partial u_j}{\partial x_k}\right)}_{\varepsilon_{ij}}$$

The pressure-strain term is important term since its contribution is significant.

In three dimensions, this model requires the solution of seven partial differential equations in addition to the equations for the mean flow. These equations are solved in a manner similar to that for the k −□ equations and even more care is required in their solution. There is no doubt that the Reynolds stress model has greater potential to represent turbulent flow phenomena more correctly than the k−□ model. Excellent results have been obtained for some flows in which the k − □ types of models are known to perform badly; however, in some flows their performance is not better at all (e.g., swirling flows, flows with strong curvature and with separation from curved surfaces, etc.).

3.3 Modelling of Pressure Drop

Flow in the inner space of a cyclone separator possesses several characteristics that makes its numerical simulation difficult. Obviously, intense swirl and shear and confined and unstable flow structures will not be easy to duplicate. But, to be more concrete, the main issue here is rather the necessity of a model of turbulence that simaltaneously accounts for,

1. High curvature of the average streamlines.
2. High swirl intensity and radial shear.
3. Adverse pressure gradients and recirculation zones

The two important parameters that is of greater interest while modelling the cyclone separator are the collection efficiency and the pressure drop. This dissertation work focuses on the latter parameter, the pressure drop of the cyclone.

The normal procedure for measuring a pressure drop in the process industry is to measure the static pressure at the wall in the upstream and downstream piping or ducting. This is complicated in cyclones by the swirl in the exiting gas. In the first place, the swirl causes the static pressure at the wall to be higher than the cross sectional average, and in the second place, there is the issue of what to do with the dynamic pressure stored in the swirling motion.

The pressure drop over a cyclone is normally subdivided in three contributions:

1. losses in the entry,
2. losses in the separation space (the main cyclone body), and
3. losses in the vortex finder.

1. The losses in the entry are often negligible compared to the other contributions, at least in tangential entry cyclones. For swirl tubes with inlet vanes little information is available, but if the vanes are properly contoured aerodynamically, the losses are generally small.

2. The losses in the cyclone body are relatively higher, but, their main significance is in limiting the intensity of the swirl in the separation space: more frictional losses at the walls lead to a less intensive vortex. Such *wall losses* do not dominate the overall pressure drop. The losses in the vortex finder are the largest, in both through-flow and reverse-flow tangential-inlet cyclones.

3. Vortex finder losses may be an order of magnitude larger than the two other contributions. The one notable exception is highly (solids) loaded primary or 'rough-cut' cyclones wherein wall losses associated with frictional drag at the walls can become a significant contribution to the overall pressure loss—at the expense of losses in the vortex core, and the vortex finder. The pressure drop over a cyclone, Δp, is proportional to—or very close to being proportional to—the square of the volumetric flow rate, similar to that of all processing equipment with turbulent flow.

3.4 WORK DONE

This dissertation work, focuses on the study of the pressure drop in cyclone separators and hence they are studied on six different geometries by varying the cylinder and vortex height. The basic geometry considered for this study is the stairmand high efficiency cyclone. Slight variations to the stairmand high efficiency cyclone in the vertical direction. The geometrical parameters that were varied are the cylindrical height and the height of the vortex cylinder or the gas outlet pipe. Also the pressure drop in a stairmand cyclone was studied by both the Reynolds stress model and the RNG k- ε models available in the fluent package.

The design of the control volumes was important for several reasons. First of all the control volumes had to be small enough to be able to resolve the significant length scale of the flow. In general this means that a more complex flow with large gradients needs more cells, i.e. a higher cell density in order to capture the correct flow field. More cells will on the other hand lead to a computationally larger problem and require more CPU time. A compromise is to generate a denser mesh where large gradients are expected, by decreasing the mesh size, and to decrease the total number of cells in areas where the flow calculation is less sensitive to the cell size.

The computational grid geometries were built using Gambit 2.2.30 by hybrid meshing and the simulation works were carried out on a commercial fluent package Fluent 6.2.16. The calculation time varied from about 4 hours to around 10 hours. The solution was found to converge with iterations varying from 600 to 1200. The calculatios were carried out on a pc with intel Pentium core 2 duo processor and 2 giaga bytes of RAM.

The whole computational domain is divided by structured hexahedron grids. At the zone near wall and vortex finder the grids are dense, while at the zone away from wall the grids are refined. The discrete phase model was used as the concentration of the solids were less than 10%. The density of the solid was taken to be 1550 Kg/m^3. Two way coupling was used and the particles were tracked down by stochastic tracking. The solids were considered to have a diameter varying from 5μm to 15μm.

CHAPTER 4

RESULTS AND DISCUSSION

In the present study, experimental and prediction results are presented for pressure drop for different inlet velocities and two different sizes of vortex finder. The different inlet velocities considered ranges from 5 to 25 m/s

Figure 4.1 shows the general dimensions of the cyclone. Table 4.1 gives the list of the different geometries considered for this study. Figures 4.2, 1.3, 4.4 shows the computational grids that were developed in this study. Fig 4.6 compares the model predicted using CFD to that of the experimental, for the stairmand high efficiency cyclone. The CFD models used for the stairmand cyclone were the RSM and the RNG k-□. This was done so as to choose the relatively better model.

Figure 4.6 shows the contour of velocity over a plane created on the vertical mid section of the cyclone. The latter figures compare the pressure drop variations with respect to the geometry of the system that were considered as described in the table 4.1.

Validation examines whether the physical models used in computer simulations agree with real world observations. It is a process that addresses the question 'Have we accurately formulated and solved the equations?'. Validation is one of the two fundamental columns upon which the credibility of numerical simulations is built. The basic validation strategy is to identify and quantify both error and uncertainty through comparison of simulation results with experimental data.

Fig. 4.1: Dimensions of cyclone separator

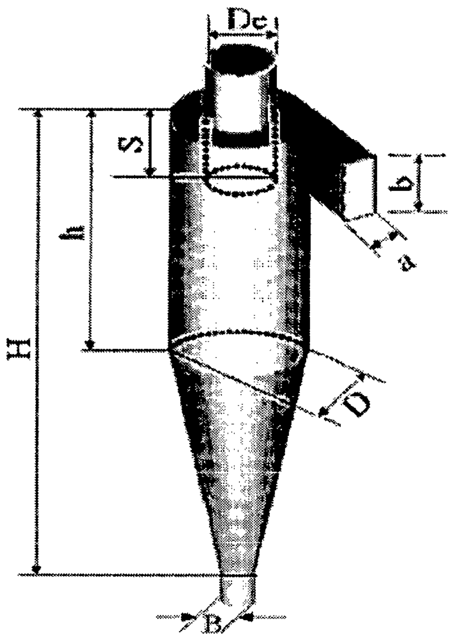
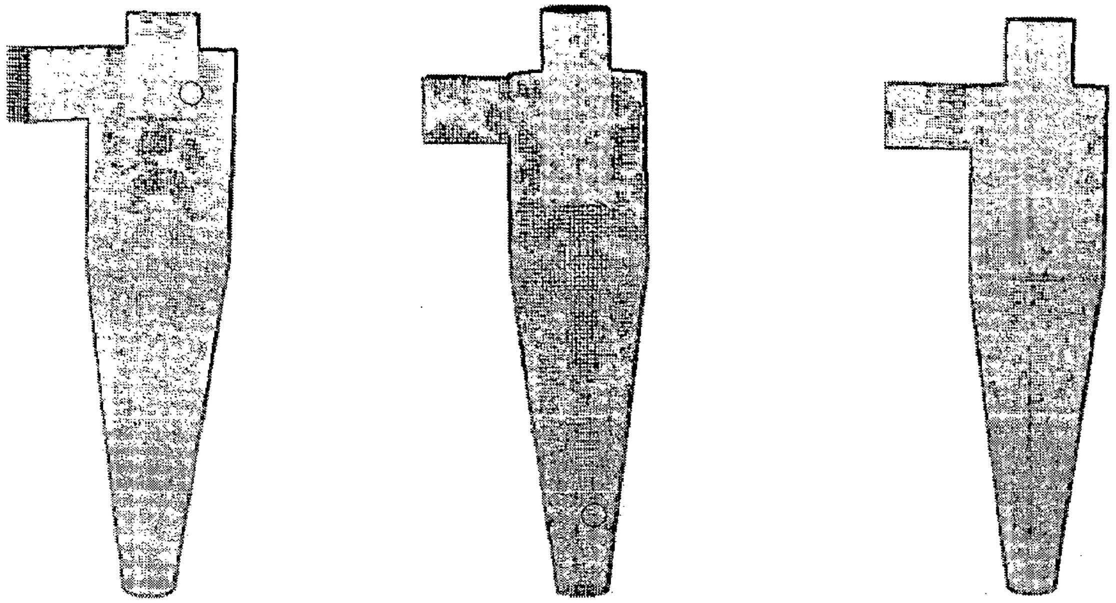


Table 4.1 Dimensions of Geometries under study

Geometry no	Cylinder height, cm	Cyclone height cm	Vortex height, cm	Common dimensions, cm
Geometry 1	45.7	122	15.25	Cyclone dia = 30.5 Cone height = 76.3 Inlet width =6.10 Inlet height =15.25 Solid out dia =11.4 Vortex dia =15.25
Geometry 2	45.7	122	30.5	
Geometry 3	45.7	122	45.75	
Geometry 4	22.9	99.2	30.5	
Geometry 5	68.6	144.9	30.5	
Geometry 6	91.5	167.8	30.5	
Lapple Cyclone	D=20cm, H=80cm, h=40cm, De=10cm, b=10cm, a=5cm, S=12.5cm, B=5cm			

Fig 4.2: Constructed computational grids by varying the vortex length

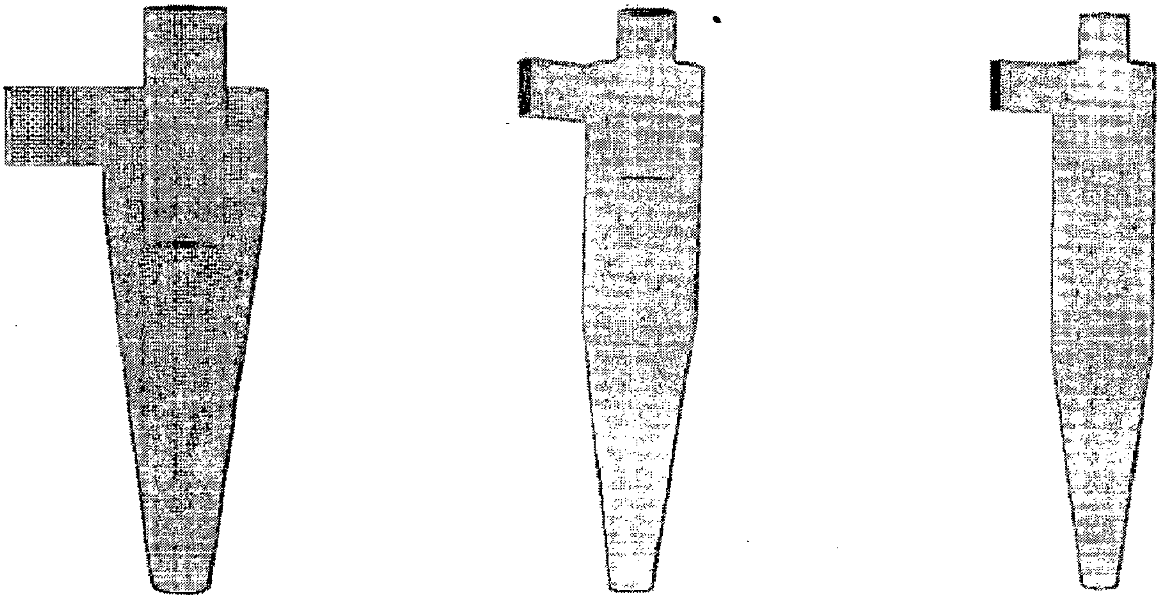


Geometry 1
no. cells = 218513

Geometry 2
no. cells = 124965

Geometry 3
no. cells = 264933

Fig 4.3: Constructed computational grids by varying the cylinder length

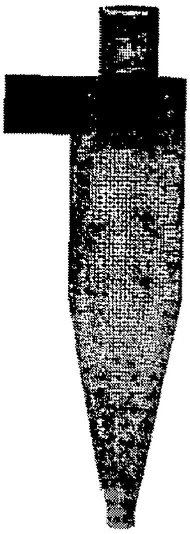


Geometry 4
no. cells = 194974

Geometry 5
no. cells = 262603

Lapple Cyclone
no. cells = 355730

Fig. 4.4: Computational grid of Lapple cyclone separator



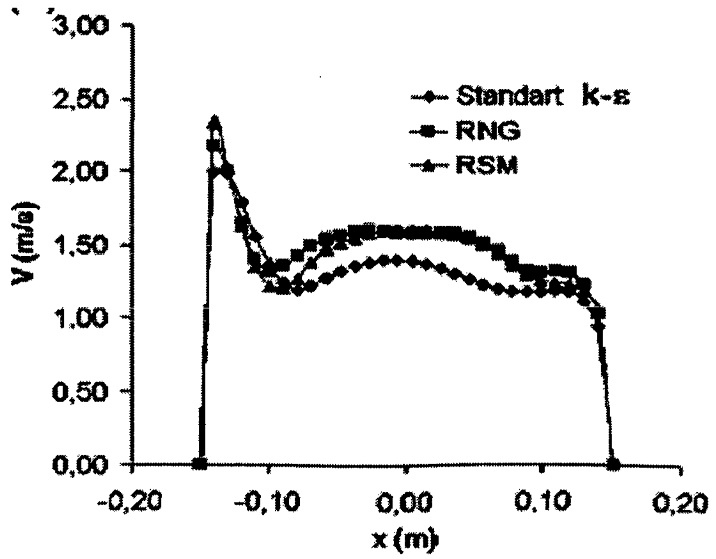
no. cells = 205850

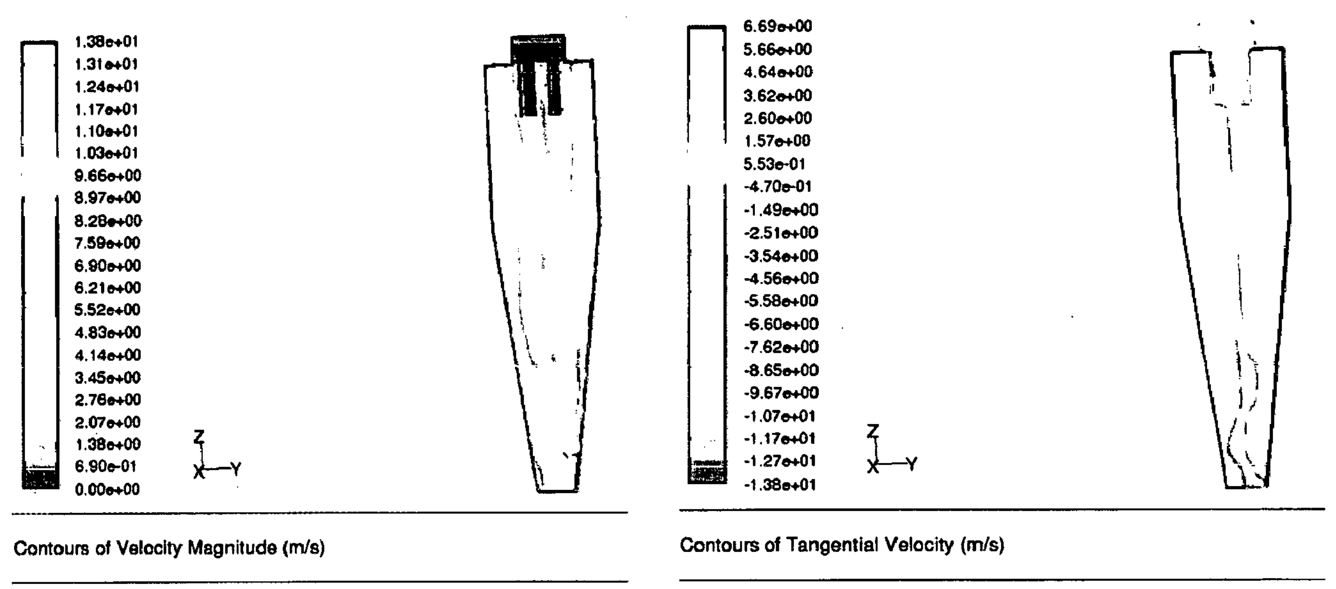
4.1 Validation Process:

Validation examines whether the physical models used in computer simulations agree with real world observations. It is a process that addresses the question 'Have we accurately formulated and solved the equations?'. Validation is one of the two fundamental columns upon which the credibility of numerical simulations is built. The basic validation strategy is to identify and quantify both error and uncertainty through comparison of simulation results with experimental data.

From literature it was found that, among the three models the RSM and the RNG k- Є models are found to give reasonable similar results and hence these two models are studied over a stairmand cyclone for which the experimental data was taken from literature.

Fig 4.5: Comparison of the three models





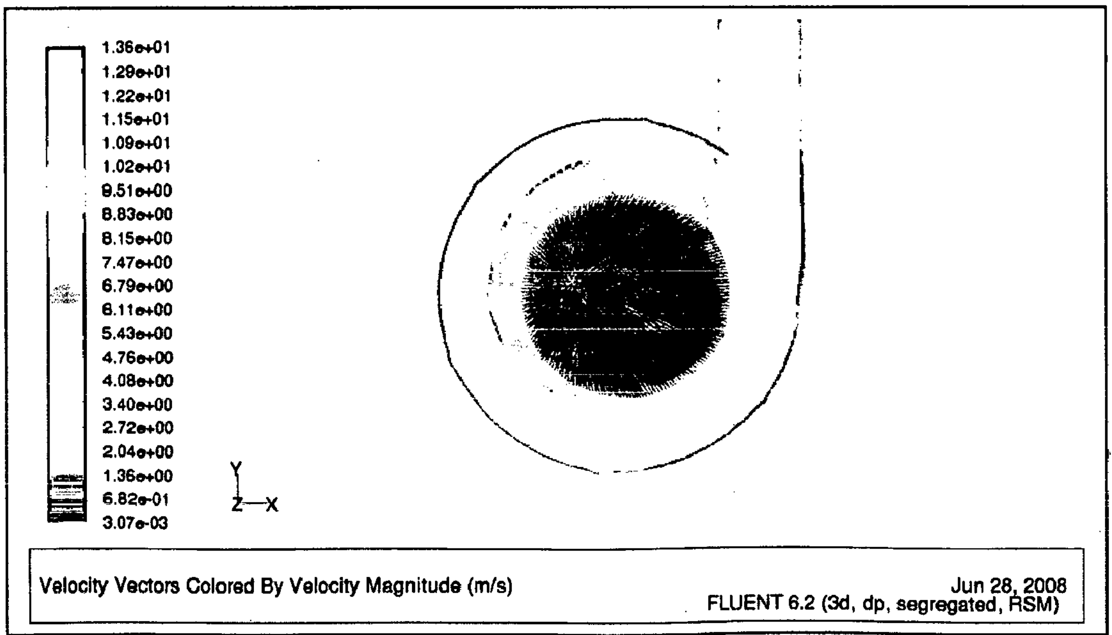
Contours of Velocity Magnitude (m/s)

Contours of Tangential Velocity (m/s)

Fig 4.6: Contours of tangential and total velocity at vertical mid-section

From the above diagrams it can be clearly understood that at the center, ie. At the vortex region the axial velocity is very high clearly indicating the vertical reverse flow motion. But due the high swirl in the cyclone separators, the overall velocity magnitude on the walls is found to be very high. This may also be more likely due to the high swirl of the particles. The above diagrams do match well with reality and to that values observed in literature.

Fig. 4.7 : Elevation of the magnitude of the Velocity vectors



Velocity Vectors Colored By Velocity Magnitude (m/s)

Jun 28, 2008
FLUENT 6.2 (3d, dp, segregated, RSM)

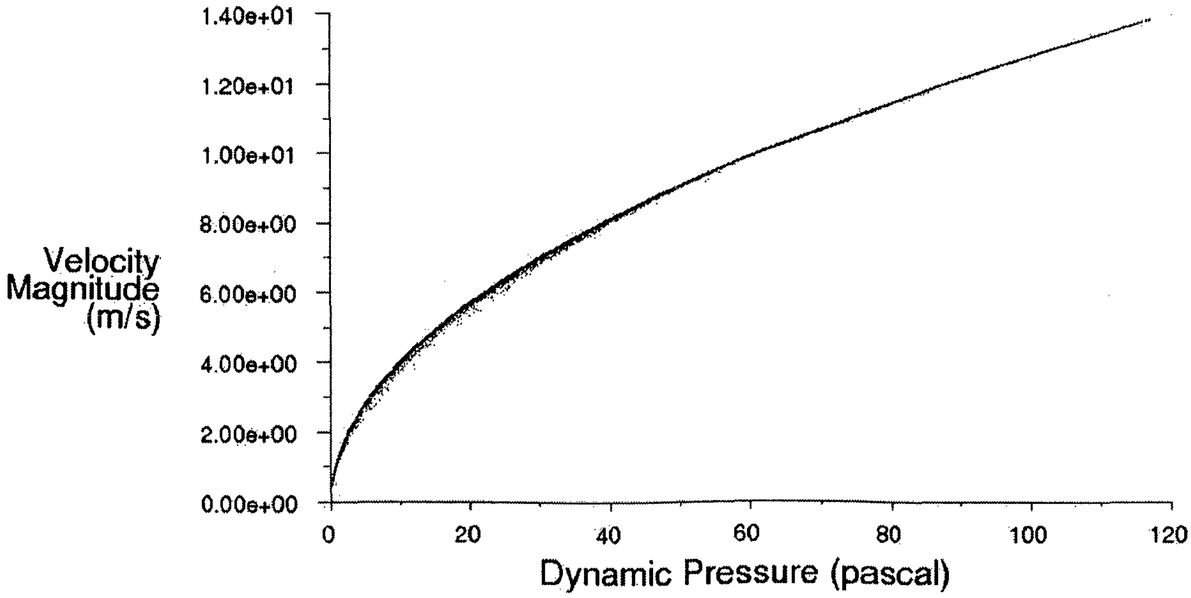
4.2 Dynamic Pressure drop:

A large amount of static pressure is transformed into dynamic pressure. If the inlet velocity is 15 m/s, the tangential velocity in the center could be about 45 m/s. Swirl dynamic pressure of the order of ($1/2\rho 45^2$) Pa is then dissipated in the vortex finder and downstream piping per unit volume of gas. In case b), on the other hand, with a lot of wall friction robbing the inlet jet of all its dynamic pressure (visualize this by incredibly rough walls reducing the spin to almost zero in the cyclone body), we are dissipating of the order of ($0.5\rho 15^2$) Pa per unit volume of gas, which is an order of magnitude less. In this second case, the air will then enter the vortex finder without any swirling motion, and at a much higher static pressure. The dissipation in the vortex finder will then be very small. The result is that the dissipation, and therefore the pressure drop.

Thus, the rougher the walls, the less the pressure loss becomes—quite contrary to what our intuition may lead us to think. This being the case, it should not be too difficult to understand that, due to their strong braking action, wall solids in heavily (solids)-loaded cyclones also reduce the overall pressure loss relative to the same cyclone operating with negligible solids loading.

Figure 4.8 gives the xy plot option available in the fluent domain between the velocity and the dynamic pressure. It can be seen that at high velocities the pressure drops were high and at low velocities the pressure drops were low. This was also observed in a contour plot of dynamic pressure which resembles to that of the velocity

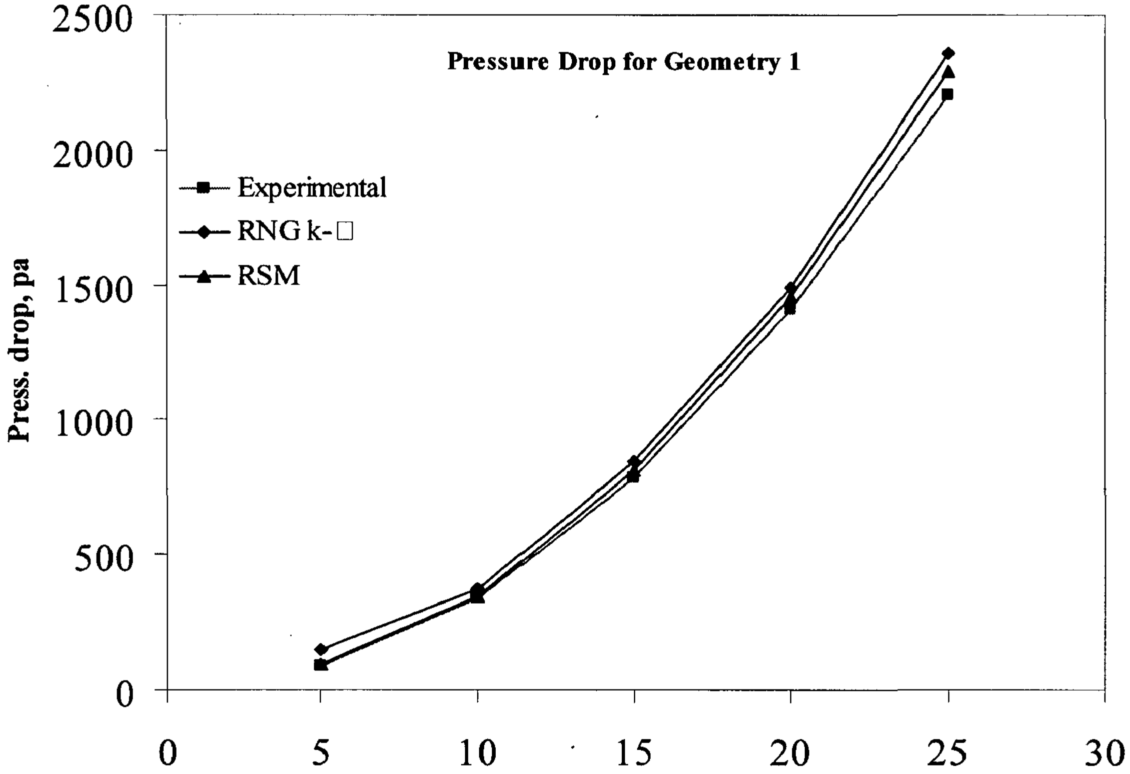
Fig. 4.8: Change in velocity with dynamic pressure



4.4 PRESSURE DROP RESULTS:

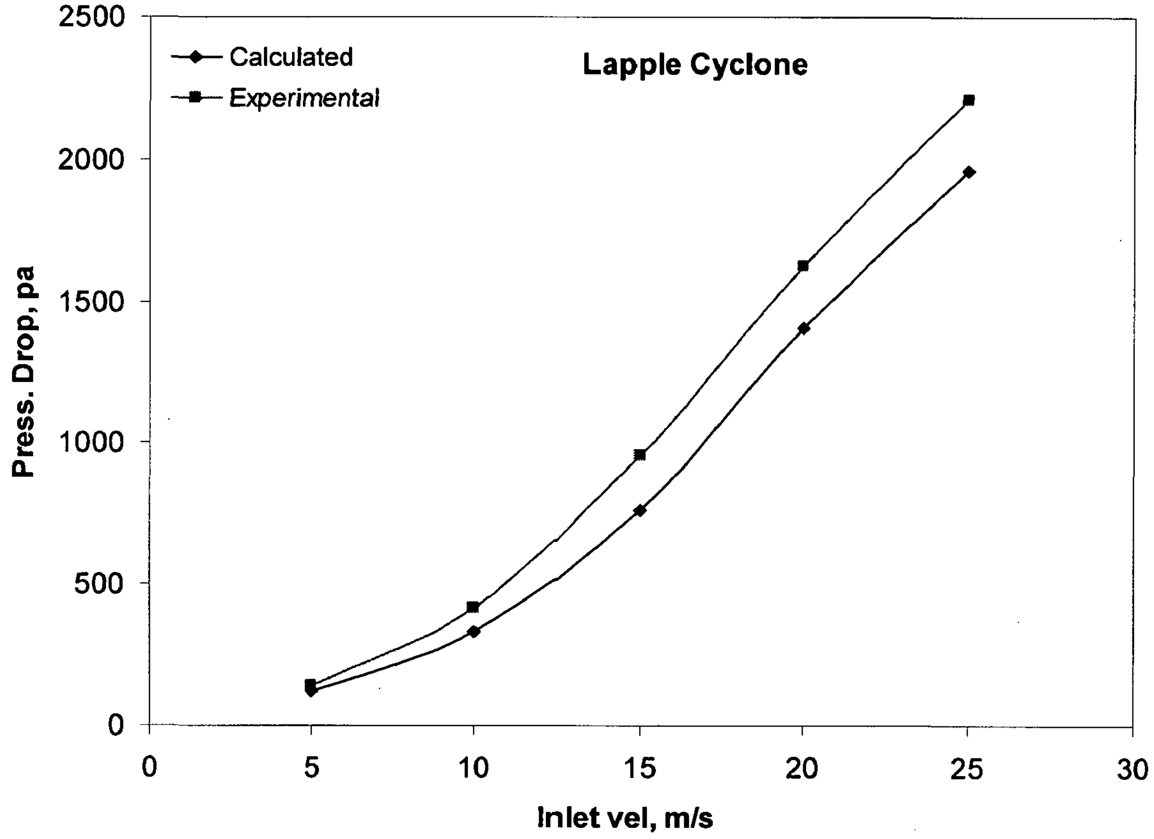
The suitable method for modelling the pressure drop was determined by using geometry 1. The RSM and the RNG k-Є models are compared with the experimental data by Dirgo and Leith. This comparison can be seen in figure 4.10. From the figure its clearly evident that the RSM model gives better closeness to the real experinmental values and hence the RSM model was applied to the geometries under study. Even from literatures and many authors suggest that the k- Є models would not be able to predict the values accurately due to the high vorticity and turbulence.

Fig. 4.10: Comparison of different models with experimental data for geometry 1



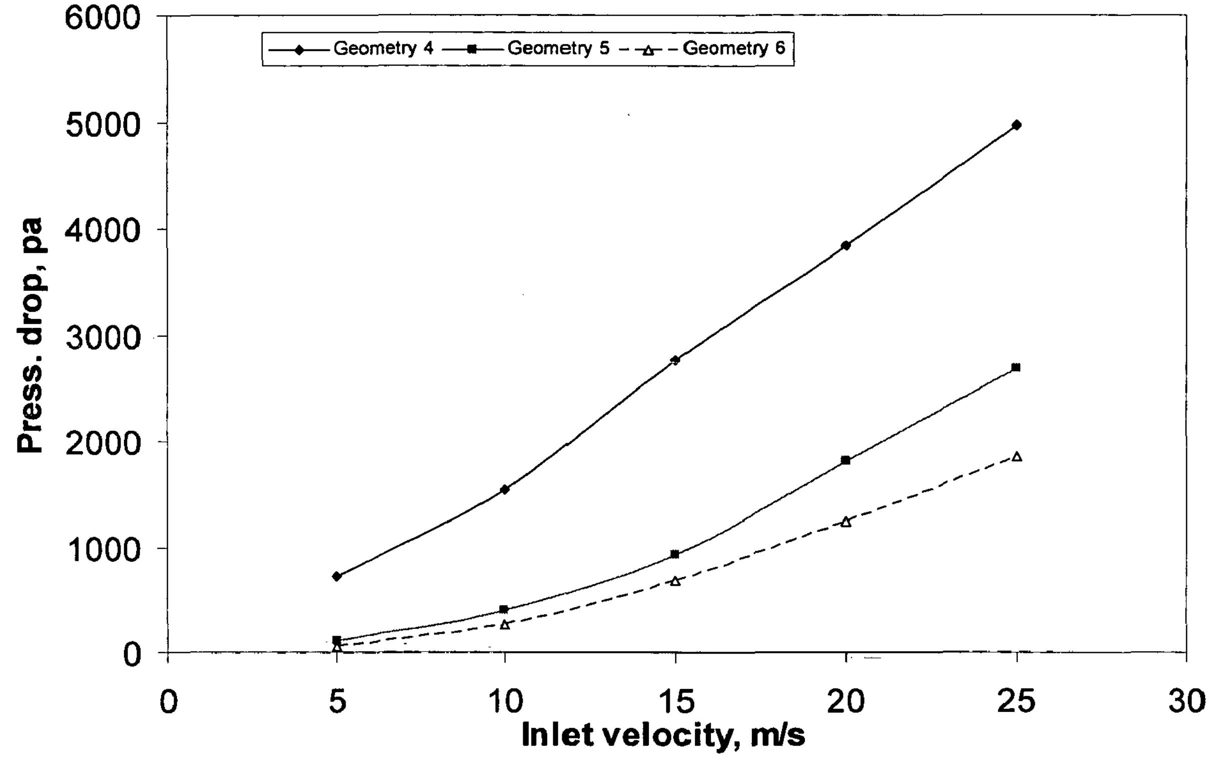
In case of RSM the maximum axial velocity tends to be close to the surface of the apex cone at the edge of the hopper with about 0.5 times the inlet velocity directed upwards and 0.36 directed downwards, respectively. In case of k − Є the maximum axial velocity tends to be close to the surface of the apex cone at the edge of the hopper with about 0.36 times the inlet velocity directed upwards and 0.18 directed downwards, respectively.

Fig. 4.11: Comparison of experimental and calculated values in Lapple cyclone



For a lapple cyclone the experimental data was found to vary with the calculated values by an error of 4%. Reynolds stress model has been used to simulate the anisotropic turbulent flow in a Lapple cyclone. Its applicability has been verified by the good agreement between the calculated and measured pressures and flow fields. But it can be observed that as the velocity increases the deviation from the experimental data becomes a little bit more.

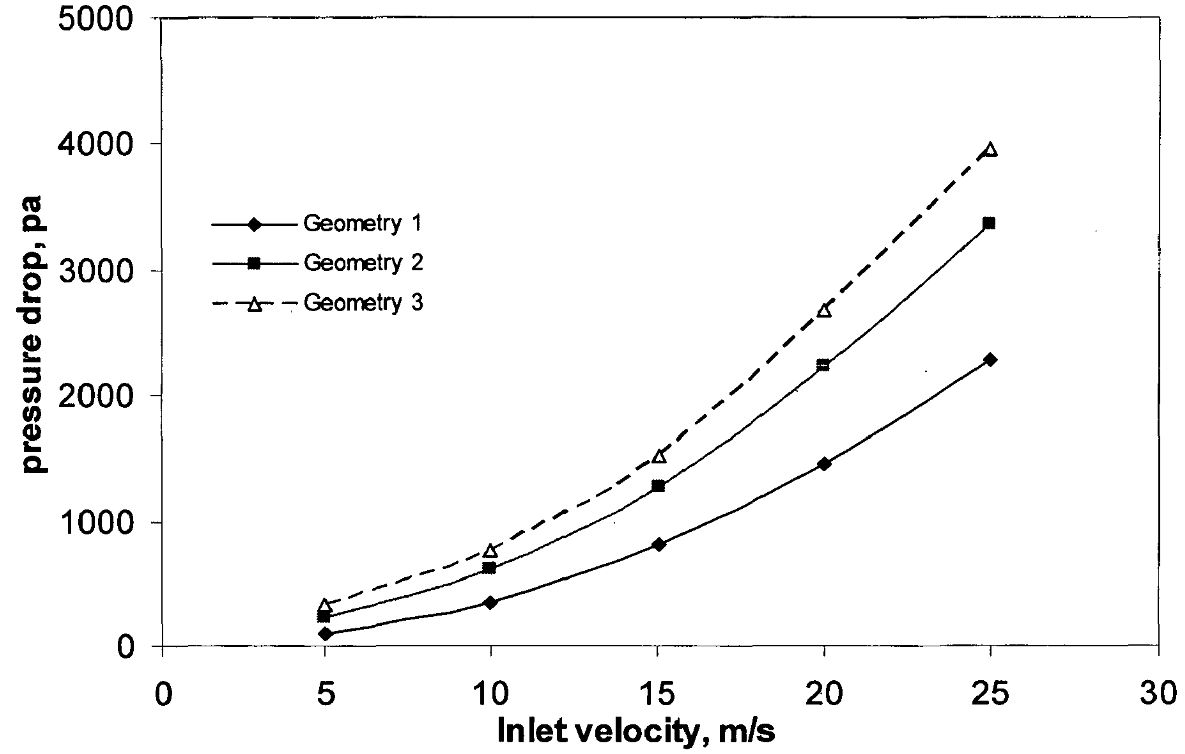
Figure 4.12: Effect of cylinder length on pressure drop at different velocities



Geometry 4 is the cyclone with the least cylinder size in which very high pressure drop was observed. On the contrary the highest pressure drop was observed in geometry 6 which has the largest cylinder among all the six geometries. Hence the pressure drop in a cyclone can be conveniently decreased by increasing the height of the cylinder. For cyclone operating at high inlet velocities this idea is very suitable.

It was observed that the static pressure decreases radially from wall to center and the minimum values appear in forced vortex (solid body rotation). The pressure gradient is the largest along radial direction since there exists a highly intensified forced vortex.

Figure 4.13: Effect of vortex length on pressure drop at different velocities



From the above figure it can be observed that the pressure increases considerably with the increase in vortex length. But on the contrary there are greater chances for high efficiencies also. Hence increase in vortex length enhances the power consumption. For lower pressure drops it is advisable to keep the vortex length equal to the height of the inlet pipe.

In case of RSM the maximum tangential velocity is about 0.26 times the inlet velocity directed upwards close to the wall of the apex cone. The minimum tangential velocity in case of RSM is about 0.02 times the inlet velocity directed downwards at the centre of the line. It can also be noted that there is a bigger difference between RSM and k – € results. However, the maximum tangential velocity always tends to be close to the walls.

CHAPTER 5

CONCLUSION AND FUTURE PERSPECTIVES

A series of numerical experiments were performed by using the computational fluid dynamics technique (CFD) for the evaluation and validation of a three-dimensional model for cyclones. The cases sought to reproduce the simulated geometry and operating conditions given by Lapple(1951)and that of Stairmand.

The conclusion drawn from the present study can be summarized as follows

- The RSM model was found to give much closer results than the RNG k-€ model. The deviations in the experimental data for the RSM was around 3 % and was about 5% in the case of RNG k-€
- The increase in the vortex finder height has a direct impact over the pressure drop. The pressure drop increases with increasing vortex length
- The increase in the length of the cylindrical section decreased the pressure drop.
- For cyclones operating at high velocity, the cylinder height can be increased to keep thee pressure drop at a low value
- The pressure drop can also be maintained less alternatively by reducing the length of the vortex. Reducing the vortex length on the other hand, decreases the turbulence and swirl flow in the cyclones.
- The investigations carried out in this study have proved that an increase in inlet flow rate rapidly increases the cyclone pressure drop.

FUTURE WORK

This work was focussed mainly on the vertical components of the cyclone separators. The effects of the geometrical parameters in the horizontal direction like the vortex diameter, cylinder diameter can also be studied over pressure drop.

Only the pressure drop of the cyclone separator was studied in depth in this work. The effects of the same geometrical parameters over the separation efficiency of a cyclone separator can be studied.

The modelling can be done by using the Large Eddy Simulation model (LES) instead of the RSM. The LES model was found to give better results by some authors like Fureby et al., Gunter et al.,

CHAPTER 6
REFERENCES

Atakan Avci, Irfan Karagoz , “Effects of good flow and geometrical parameters on the collection efficiency in cyclone separators”, Aerosol Science, Vol. 34, pp. 937-955, 2003.

A. Gil., “Gas-particle flow inside cyclone diplegs with pneumatic extraction”, Powder Technology Vol.128, pp. 78- 91, 2002.

A. Avci and I. Karagoz, “Theoretical investigation of pressure losses in cyclone separators”, Heat and mass Transfer, Vol. 28, No. 1, pp. 107-I 17, 2001.

Avci and I. Karagoz, “A Mathematical model for the determination of cyclone performance” Heat and Mass Transfer, Vol. 27, No. 2, pp. 263-272, 2000.

B. Wang a,b, D.L. Xu b, K.W. Chu a, and A.B. Yu a, “Numerical study of gas–solid flow in a cyclone separator” Applied Mathematical Modelling, Vol. 30, PP.1326–1342, 2006.

Bingtao Zhao et al., “Development of a symmetrical spiral inlet to improve cyclone separator performance”, Powder Technology, Vol.145, pp. 47– 50, 2004.

Cristo´ bal Corte´ s and Antonia Gil, “Modeling the gas and particle flow inside cyclone separators”, Progress in Energy and Combustion Science, vol.33, pp.409–452, 2007.

E.S. Rosa a, F.A. Franc.a a, and G.S. Ribeiro b, “The cyclone gas–liquid separator: operation and mechanistic modeling”, Petroleum Science and Engineering, Vol. 32, pp. 87– 101, 2001.

Fa´bio Lu´ıs Fassani and Leonardo Goldstein Jr, “A study of the effect of high inlet solids loading on a cyclone separator pressure drop and collection efficiency”, vol.107, pp.60-65, 2000.

Jianyi Chen and Mingxian Shi.B, “A universal model to calculate cyclone pressure drop” Powder Technology, vol. 17, PP. 184–191, 2007.

Jihui Chen et al., “Effect of the bottom-contracted and edge-sloped vent-pipe on the cyclone separator performance”, Chemical Engineering Journal, Vol.129, pp. 85–90, 2007.

J.J. Derkseni et al., “Simulation of mass-loading effects in gas–solid cyclone separators”, Powder Technology, Vol.163, pp. 59–68, 2006.

K.W. Chu and A.B. Yu, “Numerical simulation of complex particle–fluid flows”, Powder Technology, Article in Press.

Larri Ricci and Staff of Chemical engineering, “Separation Techniques 2: Gas/Liquid/Solid Systems”, McGraw Hill publications co., New York, First edition, pp. 201-208, 1980.

L. Theodre and A.J.Buonicore, “Industrial Air pollution control equipment for particulates”, CRC press inc., Second edition, pp.91-105, 1976.

M.S. Brennan et al., “Multiphase modeling of hydrocyclones – prediction of cut-size”, Minerals Engineering, Vol. 20, PP. 395–406, 2007.

M. Narasimha, R. Sripriya, and P.K. Banerjee, “CFD modeling of hydrocyclone—prediction of cut size”, Mineral Processing, Vol. 75, PP. 53– 68, 2005.

Madhumita B. Ray et al., “Performance of different analytical methods in evaluating grade efficiency of centrifugal separators, Aerosol Science, Vol. 31, No. 5, pp. 563-581, 2000.

R. Zhang, P. Basu, “A simple model for prediction of solid collection efficiency of a Gas-solid separator”, Powder Technology, Vol.147, PP. 86–93, 2004.

Robert E. Treybal, “Mass-Transfer Operations”, Third Edition, McGraw Hill.

S. Altmeyer et al., “Comparison of different models of cyclone prediction performance for various operating conditions using a general software”, Chemical Engineering and Processing, vol. 43, pp. 511–522, 2004.

S. Movafaghiana et al., “The effects of geometry, fluid properties and pressure on the hydrodynamics of gas-liquid cylindrical cyclone separators” International Journal of Multiphase Flow, vol. 26, pp. 999-1018, 2000.

W. Peng et al., “Experimental study of the vortex end in centrifugal separators: The nature of the vortex end”, Chemical Engineering Science Vol.60, pp.6919 – 6928, 2005.

W. Penga and A.C. Hoffmann, “Flow pattern in reverse-flow centrifugal separators Powder Technology” Vol.127, pp. 212– 222, 2002.

Yaxin Su, “The turbulent characteristics of the gas–solid suspension in a square cyclone separator”, Chemical Engineering Science, vol. 61, pp. 1395-1400, 2006.