Modeling of Multiphase Flow in Hydrocyclone and Validation

by

Muhamad Khairuddin bin Saad

Dissertation submitted in partial fulfillment of
the requirements for the
Bachelor of Engineering (Hons)
(Chemical Engineering)

SEPTEMBER 2011

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Modeling of Multiphase Flow in Hydrocyclone and Validation

by

Muhamad Khairuddin bin Saad

A project dissertation submitted to the
Chemical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfillment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(CHEMICAL ENGINEERING)

Approved by,

____

(Assoc. Prof. Dr. Nurul Hasan)

UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK
SEPTEMBER 2011

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

MUHAMAD KHAIRUDDIN BIN SAAD

ABSTRACT

Cyclone separators have exist since the 1800's and are still widely used in many industries. Although hydrocyclone are geometrically simple, the physics describing the flow and separation processes which occur in them is complex. Over the decades many researchers have studied these devises and have developed a number of theories and empirical models for design purposes. In practice, most cyclones are design using some type of empirical information. Physical prototypes are then built, tested and tuned until an acceptable level of performance is obtained. Recent advancement in numerical methods and in the performance capabilities of moderately priced computers have opened the possibility of developing computer-based methods, which can be effectively used for hydrocyclone design study. This is where this project play part, a study of multiphase flow in hydrocyclone with different configuration of parameters are manipulate using computer model will be proposed. In this model, the mixture of multiphase flow model is used to simulate the internal three-dimensional flow field of the hydrocyclone using computational fluid dynamics (CFD) method. AutoCAD and NUMECA FINE/Open are used as a medium to design and simulate the CFD model of hydrocyclone in this project to obtain optimum design configuration. The outcome of research is very helpful to explain the separation process and to optimize the hydrocyclone design. This study provides the potential to produce hydrocyclone designs with the required performance characteristics more quickly and more economically than older methods which use experimental design approach exclusively.

ACKNOWLEDGEMENTS

I would like to thank everyone who had contributed to the successful completion of this project. I would like to express my gratitude to my supervisor, Assoc. Prof. Dr. Nurul Hasan for his valuable advice, guidance and her enormous patience throughout the development of this study. All the knowledge that I learned from her will be come in handy in the future.

In addition, I would also like to express my gratitude to my loving family and friends who had helped and give me encouragement. I am very fortune to be blessed by our course mate and seniors whose help, stimulating suggestions and encouragement. Without the mentioned parties, it is impossible for me to complete this final year project.

Finally, to all that involved in contributed to this thesis. Thank you to all of them that help me either direct or indirectly.

TABLE OF CONTENTS

Contents

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1
INTRODUCTION

1.1. Background of Study

Hydrocyclone have been fully utilized for many years in various industries. Common hydrocyclone application include classification of solids or removal of particles from primary fluid, a liquid or a gas stream. The use of the solid-liquid hydrocyclone has emerged as a alternative to separation systems, which are bulky, required backwashing, frequent replacement of filters, chemical additives and have greater pressure drop, resulting in higher operating costs.

It's simple design and easy to configure make it more preferred unit in many industrial applications (Elsayed & Lacor, 2009). Accurate prediction of hydrocyclone flow field has become essential to determine the separation efficiency of the hydrocyclone device. Furthermore, hydrocyclone performance basically is affected by several parameters such as geometry, dimension, diameter, cylinder length and inlet velocity, etc (Richard, 1982).

An implementation of CFD using engineering software has become upper hand in helping to determine optimum configuration for simulation of flow field of mixtures model inside the hydrocyclone. Although the geometry structure of hydrocyclone is simple, the distribution of the internal flow field is extremely complex and many approaches and study have been carried out with excellent results. Using CFD method, it give great advantages in elevating the design level of cyclone, shortening design cycles, reducing developments costs and improving operational efficiency (Zhang, You, & Niu, 2011).

1.2. Objectives

The main objective of this study is to develop a CFD model of multiphase flow in hydrocyclone to have better understanding of fluid flow behavior. The model will enable the prediction of the velocity profile, the pressure drop and particle distribution, which are used to determine particle trajectories in analyzing the separation efficiency. Experimental data from the journals are used to validate and refine the proposed model.

Others objective in this study are:

a) To design a CFD model of hydrocyclone using AutoCAD (CAD software)
b) To solve the simulation using NUMECA FINE/Open (Engineering software)
c) To identify the optimum configuration of hydrocyclone that give higher separation efficiency.

1.3. Scope Of Study

Rising needs for efficient and reliable solids removal systems in various industries such as the mineral and energy industries, the hydrocyclone has emerged as a proven technological alternatives. Proper hydrocyclone design is therefore crucial for achieving maximum performance and ensuring the highest and most reliable separation efficiency. However, the complexity in understanding of the dynamic fluids flow behavior and separation mechanism that occur in the hydrocyclone, thus more research is needed in order to achieve these goals. With a rapidly increasing in technology development especially in computational fluid dynamics (CFD) gives extra hands for people to study hydrocyclone. Furthermore, CFD simulations give better understanding in study of flow fields within the hydrocyclone to bring it optimum design. Even though, CFD models require a large amount of computing power for start-up operation. However, the simulation still have advantages than experimental that time-consuming and costly.

In this paper, the study mainly focused at developing a CFD simulation of multiphase flow model capable of predicting the fluids flow behavior and separation efficiency of the hydrocyclone over a different configuration. Analysis and validation of the simulation results is against the experimental data obtained from journals. A non-commercial engineering software, NUMECA FINE/Open is used to simulate the model.

1.4. Thesis Structure

Current Chapter is a brief preface to the study. It begins with the background of the study, objectives and scope of study. The Chapter 2 follows with literature review of hydrocyclone. At early chapter presents an overview of hydrocyclone technology and typical hydrocyclone geometry function. Next, this chapter cover hydrocyclone characteristics, flow behavior and basic definitions in hydrocyclone study. Last part of chapter describe some of the theoretical of equation models involve in order to study fluids behavior in hydrocyclones.

Chapter 3 introduce the methodology and steps taken throughout the study from early stage of the literature review until the validation of simulation. In chapter 4, the results obtained from proposed model is validate against the experimental data and refined accordingly. Detailed discussions and comparisons of simulation-experimental results are presented in this chapter. Finally conclusions and recommendations for future work are described in chapter 5.

CHAPTER 2
LITERATURE REVIEW

2.1. Introduction

Initially, the first U.S. Patent on a hydrocyclone design was granted to Bretney in 1891. However, it was until after World War II when the hydrocyclone technology gained popularity in different industrial applications. Recently, the hydrocyclone plays an important role in different industrial application such as oil and chemical industries, fluid clarification, classification, solid removal, liquid-liquid separation, solid-liquid separation, and particle size distribution measurement.

2.2. Solid-Liquid Hydrocyclone

The solid-liquid hydrocyclone is type of separator that facilitates the centrifugal separation of solid particles from a primary fluid (liquid stream). As stated in background of study, . Main reasons this device widely used are because it economy, simplicity in construction and ability to operate at various temperatures and pressure (Elsayed & Lacor, 2009).

The principle of hydrocyclone separation is simple, different from the slow gravity vessel separator, the hydrocyclone utilize the energy obtained from fluid pressure where the mixture enters tangentially from the inlet section to create rotational fluid motion called centrifugal force. Then, it will flow into the cylindrical body where induces a spinning forces to the mixture. Centrifugal forces that develop due to the spinning forces inside the hydrocyclone will separate the mixture based on their density and particle size characteristic. Large particles are centrifuge outwards to the hydrocyclone wall and leave through the underflow orifice and fine particles dragged in by the fluid flow are removed through the overflow at top hydrocyclone (Guofeng, Jong-Leng, & Andrew, 2010).

2.3. Solid-Liquid Hydrocyclone Geometry

Hydrocyclones are simple, compact and highly efficient separators when properly designed and configured. Figure 1 represent a typical hydrocyclone separator used in various industries. The hydrocyclone generally consist of a vertical cylinder with a conical section attached into it. The cylindrical part is closed at the top by a cover where the vortex finder extends to a certain length into the body of the cyclone. Near at the top cover is the feed inlet orifice, either circular or rectangular shape, through which the fluid mixture enter tangentially into the cylindrical part. Spigot diameter or underflow outlet, serves as the exit of the separated phase stream. This underflow stream consists of a mixture of some liquid and solid particles coarses than the cut size (d_{50}). Most of the liquid stream along with some particles finer than cut size exit through the overflow outlet via the vortex finder. The hydrocyclone utilize the centrifugal forces promoted by the tangential entry to separate the solid particles from the liquid mixture.

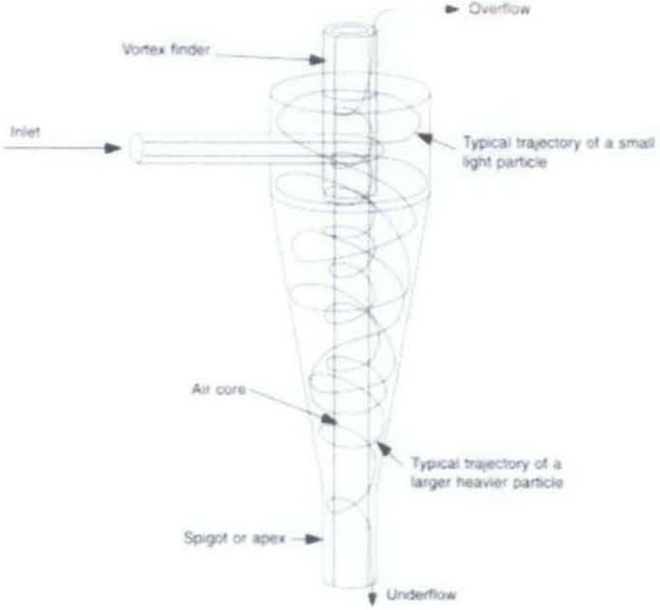


Figure 1: Typical Design of a **hydrocyclone** Separator

2.3.1. Feed Inlet

The Inlet orifice has the important role of providing a smooth flow patter at the point of entry into the cyclone. The main goal is to inject the feed in a way so as to achieve the highest tangential acceleration possible, reducing turbulence effect, pressure drop and shear stress to an acceptable level. Rectangular or circular shaped, single or twin inlets have been most frequently used by different researchers. Two commonly used are feed inlet configurations are the tangential and the involuted entry. The involuted feed entry aims at maximizing the efficiency conversion of kinetic energy to centrifugal force, while minimizing turbulence effect that could be detrimental to fine particle separation and causing excessive wear. This is achieved by minimizing the intersecting angle between the incoming feed and the already rotating fluid inside the hydrocyclone (Svarovsky, 1984). On the other hand, the twin inlets have been considered to maintain better symmetry, resulting in a more stable reverse core (Thew, Wright, & Colman, 1984).

2.3.2. Overflow Outlet

This is small diameter orifice that plays a major role in the split ratio, defined as the relationship between the overflow rate to the inlet flow rate. Most commercial hydrocyclones allow for changing the diameter of this orifice to suit a wide range of operating conditions.

2.3.3. Vortex Finder

The vortex finder is the overflow pipe located at the center top of the cylindrical section extending some length into the cyclone body. It is necessary that length of the vortex finder extends below the feed entry in order to increase separation efficiency by avoiding short-circuiting, that is the early exit of the feed stream to the overflow. the diameter of the vortex finder is generally that of the overflow orifice. Similarly, some manufactures provide interchangeable vortex finders for increased efficiency and a more flexible operation over a wide range of feed conditions.

2.3.4. Underflow Outlet

Also called spigot, the underflow is a small diameter orifice located at the apex of the cone. The spigot plays an important role in the control of the volumetric flow split and underflow density, as it has a direct effect on the underflow to throughput ratio, the underflow concentration and the cut size (Svarovsky, 1984). Most commercial unit are also supplied with a variable, changing, or adjustable orifice size to accommodate for a wide range of operating conditions and optimize the separation process. Maximum particle size at the feed entry should be considered in order to avoid spigot clogging or malfunctioning and thus, operation interruption.

2.4. Hydrocyclone Operating Principle

The hydrocyclone utilize the principle of centrifugal sedimentation to separate particulate matters based on size, shape and density. A liquid flow containing a concentration of fine particles is fed tangentially into the body of the hydrocyclone. The tangential inlet flow induces centrifugal forces causing solids coarser than the cut point size to be pushed radially toward the wall, move downward, and exit from the underflow via the spigot, along with some liquid. Most of the liquid flow with some solids finer than the cut point size move upward and exit through the overflow via the vortex finder. The term 'd_{50} cut point' stands for the particle size at which the cyclone is 50% efficient.

2.5. Hydrodynamic Flow Behavior

The centrifugal force is produced by the tangential injection of the pressurized fluid mixture into the hydrocyclone. The flow pattern consist of a spiral within another spiral moving in the same circular direction (Seyda & Petty, 1991). These are the most conspicuous flow in the hydrocyclone and are sometimes called primary and secondary vortices. The primary or outer vortex moves downward carrying suspended particles along the axis of the cyclone to underflow outlet. The secondary or inner (forced) vortex is located inside the primary vortex in the region close to the cyclone core moving upward carrying mainly a clean liquid stream to the overflow outlet (Rushton, Ward, & Holdich, 2000).

2.6. Pressure Drop and Flow Rate

The pressure drop is the differential pressure between the locations right before the feed entry and right after the overflow outlet. The hydrocyclone develops its swirling motion utilizing the fluid pressure energy. A hydrocyclone of fixed dimensions, operating with a given flow mixture and flow conditions, gives a fixed relationship between the volumetric throughput and the pressure drop. The two variables are therefore interdependent where increasing flow rate results in increasing pressure drop.

2.6.1. Flow Reversal

With a high swirl at the inlet region, the pressure is high near the wall region and very low toward the centerline, in the core region. As a result of the pressure gradient profile across the cyclone diameter, which decreases with downstream position, the pressure at the downstream end of the core is greater than at the upstream, causing flow reversal (Hargreaves, 1990) in the region along the cyclone axis.

2.7. Definition of Separation Efficiency

The main application of the hydrocyclone subject to this study is to analyze the separation efficiency. Separation efficiency is a measure of the hydrocyclone ability to recover solids through the underflow outlet, while allowing most of the clean liquid to continue through the underflow outlet, and thus, through the rest of the process. Following are the definitions of some important parameters used to define hydrocyclone separation efficiency.

2.7.1. Water Split ratio

The water split ratio is the ratio of the overflow rate to the inlet flow rate, as given by the following expression:

$$F = \frac{q_o}{q_1} \times 100\%$$

where F is the water split ratio, q_0 is the total flow rate at the overflow of the hydrocyclone, and q_1 is the total inlet flow rate.

2.7.2. Cut Size

A common approach to define hydrocyclone efficiency is based on the cut size, d_{50}, the size at which particle separation or classification is 50% efficient. That is, the size having probability of going to the overflow or the underflow (Rushton, Ward, & Holdich, 2000). Figure 2 represents an idealized size distribution or feed split into overflow and underflow.

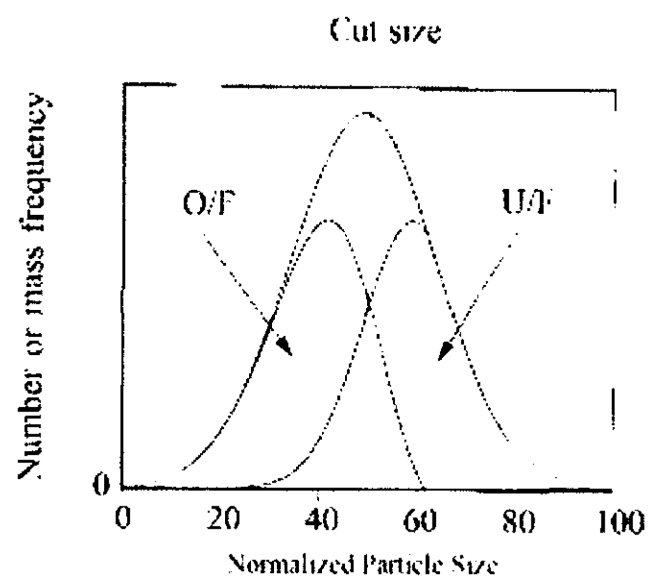


Figure 2:Idealized Particle Size Distribution Curves

2.7.3. Separation Efficiency Based on Particle Tracking

In this study, the hydrocyclone separation efficiency is predicted based on particle trajectory analysis. Particle trajectories are traced in the fluids flow using a Lagrangian approach. This is accomplished by performing a force balance on each characteristic particle size present in the feed in order to predict its velocity. Thus, it is possible to predict it characteristic particle is either able to reach the underflow outlet and be separated, or if it reaches the reverse flow region, dragged by the primary flow and carried to the overflow.

2.8. CFD and Numerical Studies

In Recent years, the advancement of computer technology has promoted the use of Computational Fluid dynamics (CFD) to study complex fluid flow systems, such as the hydrocyclone. Fundamental equation, as well as, turbulence closure models are solve numerically over a grid system domain. The exact geometry and flow conditions can be reproduced, reducing the need for complex and costly experiments.

Rigorous phenomenological models based on fluid dynamics have three main components; the mass balance described by the continuity equation, the momentum balance described by the Navier-Stokes equation and the turbulence effect closure model. Solving the continuity and the Navier-Stokes equation for non-turbulent flow can be achieved with the computational resources available today for simple or complex geometries. However, at large Reynolds number current resources struggle to attain the instantaneous velocity and the pressure fields, even for simple geometries (Hubred, Mason, Parks, & Petty, 2000). (Slack, Cokljat, & Vasquez, 2003) proposed an automated CFD modeling interface for hydrocyclone design, providing the non-CFD analyst or design engineer with a flexible hydrocyclone simulation tool.

CFD has been used in the past to numerically solve the governing equations and to study hydrocyclone turbulent flow phenomena. However, choosing an appropriate turbulence model and the numerical solution scheme is required for achieving good results. Some recent studies have attempted to compare the different turbulence closure model and their variations into separation efficiency (Matvienko, 2004).

2.9. Factor Affecting Solid-Liquid Separation in Hydrocyclone

2.9.1. Effect of Geometry

The geometrical configuration and the different dimensions of each component of the hydrocyclone, such as the cone angle, length and diameter of the cylindrical chamber, length and diameter of the vortex finder, outlet orifice diameter and feed pipe diameter, have been found by several researchers to have significant influence on particle separation performance. Hence, the understanding of the impact of each component size and geometry on performance could lead to significant improvements to hydrocyclone design.

2.9.2. Effect of Particle Properties

The effect of the properties of the solid-phase on hydrocyclone performance was examined by (Salcudean, Gartshore, & Statie, 2003). They observed a decreased in particle carry-over as particle density increased. They also found out that the magnitude of this effect was affected by the particle diameter and length. The carry-over sharply decreased with larger particle diameters, as would have been expected. This is agreement with (Dwari, Biswas, & Meikap, 2004) observation where they reported that larger particles are remove easily with an increase in particle size at a particular inlet pressure, separation efficiency increases.

2.9.3. Effect of Temperature and Pressure

A CFD model was used by (Shi, Bayless, Kremer, & Stuart, 2006) to predict the pressure drop and velocity profiles in cyclones at high temperatures and high pressures. The results showed that density had a considerable effect on the pressure distribution, while the effect of viscosity was insignificant. Temperature increase led to decrease in tangential velocity, while the reverse flow in the center of the cyclone became weaker, resulting in a decrease in the cyclone efficiency. On the other hand, an increase in pressure led to an increase in the collection efficiency for the same inlet velocity. Fluid density increase with pressure, and this in turn increases the tangential velocity in the outer vortex region.

(Su & Mao, 2006) studied experimentally the effect of cyclone wall temperature on the flow field. They observed that the flow field became more uniform with increased suspension temperature. They also noticed that local vortices at the corners were weakened and the swirling intensity lowered, which led to decreased total mean separation efficiency from 81% to 76%. These results are in agreement with observation by (Shi, Bayless, Kremer, & Stuart, 2006).

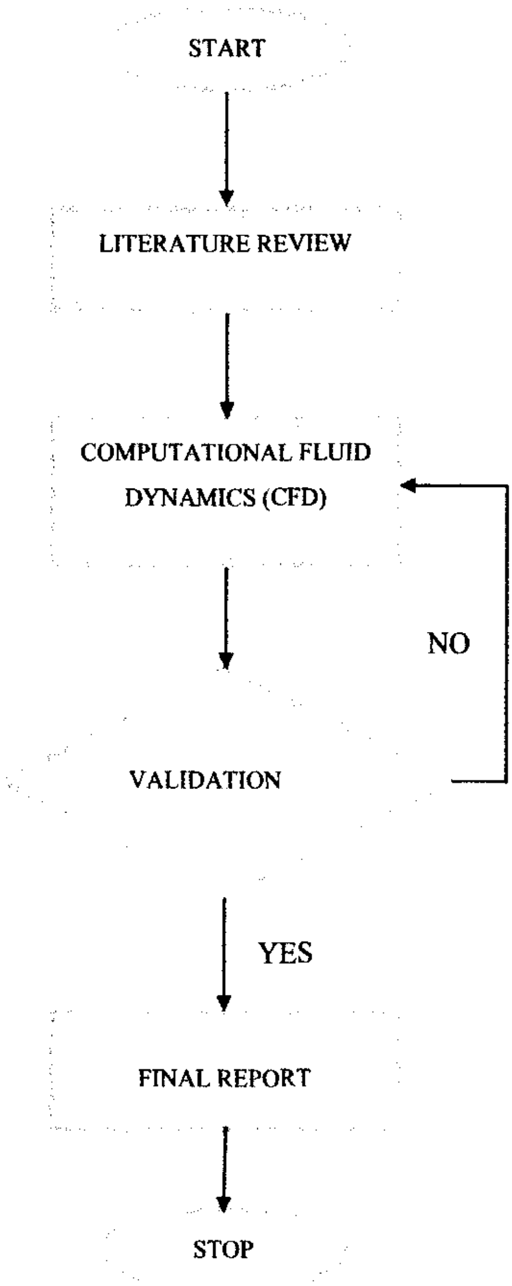
2.9.4. Effect of the Air Core

The air-core formed in the cyclone is a very important internal structure of the cyclone. Stability of the flow field is necessary for effective performance. The effect of the air core on the main flow field was investigated by (Luo & Xu, 1992) concluding that it is detrimental to particle separation. They observed that the air core enhanced instability and asymmetry of the flow filed and disturbed the regular distribution of classified particles. Many researchers have neglected the effect of the air core in their modeling and simulation for simplicity. However, this simplification can lead to inaccuracies in the prediction of the flow field and overall hydrocyclone separation efficiency.

CHAPTER 3
METHODOLOGY

3.1. Introduction

In this chapter, the methodology of the whole study as stated in Figure 3 will be organize by a systematic method to analyze the separation efficiency of the hydrocyclone. The project will begin with literature review on fundamental studies about multiphase flow in hydrocyclone. The literature review is firstly done by studying and understanding the concept of how hydrocyclone operate in various conditions. After the study, a complete hydrocyclone model will be developed in engineering software, (AutoCAD and NUMECA FINE/Open) then are simulated using several sets of mathematical method in solving CFD model to get satisfied outcomes. This stage is done simultaneous with modeling stage as it depends on each others. After computational solution is successful, all of the results will be documented and be compiled in final report.



- Literature Review.
- Introduction on multiphase flow in hydrocyclone.
- Fundamental studies from references and journals.

- Develop sets of CFD analysis method for the project.
- Identify suitable solving method for the project.
- Develop mathematical model for this project.
- Formulation of governing equations.
- Mathematical solution of the governing equations.

- Simulation analysis using engineering software (e.g. AutoCAD and NUMECA FINE/Open).
- Interpretation of the results.

- Recommendation on process application.
- Complete final report for submission.

Figure 3: Methodology Charts

3.2. CFD Simulation method

In order to optimize and fully utilize the time use for the project study, a test cases table has been produce to track the work progress. As shown in the Table 1 below, it is separate into 3 tests with different value for each criterion. Firstly, mesh geometry will be test with different mesh densities which are 100,000, 200,000, 300,000 and 400,000 of grid cells. Discretization test also will be conducted with 1st order and 2nd order discretization. While for turbulence model, a little bit of modification is made in this simulation by simulate the CFD model using K-omega (**K-Ω**). These steps again will be repeated for pressure parameter as stated in objective to find the optimum configuration.

Table 1: Test Cases Table

	1st test	2nd test	3rd test	3rd test
Mesh geometry	**M1**= 100000	**M2**= 200000	**M3**= 300000	**M3**= 400000
Discretization	**D1**= 1st order, **D2**= 2nd order			
Pressure	**P1**= 83kPa			
Model	Turbulence model: **K-Ω**			
	Multiphase model: Discrete phase model			

3.3. Model Description

3.3.1. Hydrocyclone Geometry

In this project, the focus is in geometry effects in hydrocyclone, the geometry model plays important parts in this simulation as different configuration shows different result of fluid flow inside the hydrocyclone. CFD simulation study are carried out on 76 mm diameter hydrocyclone. The geometrical description is describe in details in Table 2 below:

Table 2:Design Details of 76mm Hydrocyclone

Dimensions (mm)	Hydrocyclone
Cyclone diameter (CD)	76
Cylindrical length (CyL)	85
Vortex finder diameter (VFD)	25
Vortex finder length (VFL)	90
Feed inlet (FI)	20 x 10
Cone angle (CA)	10^o
Spigot diameter (SPD)	10

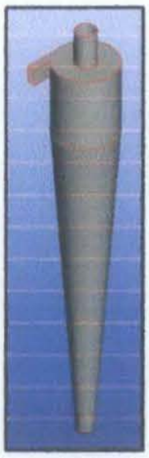


Figure 4:Complete Hydrocyclone Geometry Produce Using AutoCAD

3.3.2. Meshing Scheme

There are 3 type of mesh classification which are structured meshes, unstructured meshes, and hybrid meshes. Depends on the analysis and solver used, type of mesh will be choose based on its application. For hydrocyclone, it is cannot be modeled based on 2D plane due to geometry, non-axisymmetric while 3D model give better match with the experimental data [www.psl.bc.ca/downloads /presentations/cyclone/cyclone.html]. In this study, unstructured hexahedral mesh is chosen as it give better prediction in analyzing the fluid flow inside hydrocyclone.

3.3.3. Grid Independence Test

Grid independence test is used to describe the improvement of simulation results by using smaller grid cell sizes for the calculation. The study is start with a coarse grid cell sizes and gradually decrease it cell sizes until the changes in the results are smaller and can be neglected. Hence, the optimize grid cell size is chosen as it given the better prediction and reasonable computational time for simulations.

The present computational model is based on 3D geometry. Hexahedral mesh was chosen in this study which is known to be less diffusive compared to other types of meshed like tetrahedral. Grid independence test is carried out using FINE/Hexpress with mesh sizes created of 100,000, 200,000, 300,000 and 400,000 cells however due to software adaptation and optimization of mesh, the actual cell size form is differ from the setting. The actual sizes of grid cells form in hydrocyclone geometry is as at Table 3.

3.3.4. Initial and boundary conditions

A mass flow condition is used to prescribe pressure inlet through the hydrocyclone feed inlet with pressure of 83kPa. The outlet, overflow and underflow outlets were set as pressure outlets. The primary phase is set to water with density = 1001 kg/m^3 while for particle distribution were carried out using inert solid spherical particles with density 2300 kg/m^3. For above condition, the fluid flow was simulated using NUMECA FINE/Open and FLUENT, with 2nd order discretization and a steady state inside the hydrocyclone. This FINE/Open software have user-friendly interface that help to simulate the fluid flow inside hydrocyclone. Turbulence flow inside a hydrocyclone is anisotropic in nature which is important in order to determine the flow characteristic, thus K-omega (**K-Ω**) model is chosen.

3.3.5. Experimental

Based on main journal, the experimental setup consisted a slurry tank of 200 L capacity mounted on a stable platform. A Centrifugal pump with 3-phase, 5.5 kW motor was connected to the slurry tank at the bottom. Feed slurry consisting of flyash material at different solid consistency was pumped into the cyclone body through the pipeline connected to the pump. The other body end of the pipeline was connected to the inlet opening of hydrocyclone in study. The pressure drop inside the cyclone was maintained at required level with the help of a diaphragm type pressure gauge fitted near the feed inlet. The hydrocyclone was positioned upright above the slurry tank.

The experimental program was designed to achieve a wide range of water splits into the overflow and underflow product suitably selecting the spigot opening and feed inlet pressures. Hydrocyclone main body was fixed to the test-rig. Initially, distribution studies were carried out by pumping water into the cyclone at different spigot openings and feed pressures. Required level of solid consistency was maintained in the slurry tank by mixing measured amount of flyash and water. Timed sample were collected simultaneously in suitable containers. The underflow and overflow products collected were filtered, dried and weighed. Particle size distribution of representative samples of the dried products was analyzed using in Malvern laser particle size analyzer. Distribution points based on report of each size fraction in the feed to the underflow product were generated.

3.4. Software Required

There are various commercialize CFD software applicable to use in order to study the fluid behavior inside the model. In this study, it can be separated into 3 section where each section will used different approach of software to analyze the fluid behavior inside the hydrocyclone.

3.4.1. First Section

3.4.2. Geometry creation - AutoCAD (CAD software)

2.1.1. Second Section

b) Meshing scheme - NUMECA FINE/Hexpress (CFD software)

c) Solver - NUMECA FINE/Open / FLUENT (CFD software)

3.1.2. Third Section

1.1.1.1. Post-processing - NUMECA CFView

CHAPTER 4
RESULTS AND DISCUSSION

4.1. Introduction

This chapter will discussed all the test results have been conducted and validation against experimental data.

4.2. Grid Independence Test

In chapter 3 have discussed about grid independence test. As mention earlier in previous chapter, there are 4 test conducted using FINE/Hexpress with mesh densities created are 100,000, 200,000, 300,000 and 400,000 grid cells however due to software adaptation and optimization of mesh, the actual cell size form is differ from the setting. The actual sizes of grid cells form in hydrocyclone geometry is as Table 3 below.

Table 3:Grid Independence Test Variation

Test Variation	M1	M2	M3	M4
Number of mesh create	100,000	200,000	300,000	400,000
Actual number of mesh form	51191	98899	139085	180795

Based on the result in Table 3, M4 shows the higher mesh cell sizes compare to others with 180795 cells. Water distribution studies have indicated that the higher mesh densities (cell sizes) inside the model give better prediction of fluids flow. However as the mesh cell sizes keep increasing, the computational time needed to solve the calculation also keep increasing. Thus, an optimum value of cell sizes is required in order to have good prediction at the same time have a reasonable computational time for simulation.

The characteristic of fluid flow inside the hydrocyclone need to be observed in order to determine the optimum mesh density. Static pressure and velocity profile inside the hydrocyclone were analyze and shows that M4 with mesh density of 180795 cell sizes is the optimum. It also take around one hour 7 minutes to complete the simulation where is good computational time. For static pressure and velocity profile results can be obtained at section 4.3.2

4.3. Discretization Test

Table 4: Discretization Test

Test	D1	D2
Discretization Scheme	Central	Central
Scheme accuracy	1st order	2nd order

For discretization test, as shown at Table 4 D2 is chosen with central scheme and 2nd order accuracy as it give better prediction than 1st order accuracy. Analysis of D2 test given the simulation result can be converged while D1 test shows error due to simulation cannot be converged. Another test is conducted to analyze the result between 1st order and 2nd order accuracy and it show the higher order give better prediction. It is in agreement with (Juan & Chi-Wang) where for simple geometry and smooth mesh, then the high order finite scheme would be the top choice because of its simplicity and efficiency for multi-dimensional calculations.

4.4. Throughput and Water Split

In order to have better understanding in fluid flow inside hydrocyclone, simulation results were compared with experimental data obtained from main journal. The inlet flow rates (water entering the hydrocyclone) and water-split (%) report of total water into the overflow) into overflow product of 76 mm diameter hydrocyclone are presented in Table 5.

Table 5:Experimental and Simulated Values of Water Throughput

Spigot opening	M1		M2		M3		M4	
	Exp	Sim	Exp	Sim	Exp	Sim	Exp	Sim
Throughput (kg/s)	1.16	1.16	1.16	1.16	1.16	1.16	1.16	1.16
Water split (%)	94.8	97.7	94.8	96.80	94.8	95.1	94.8	95.0
Err. split (%)	3.06		2.12		0.344		0.241	

It can be observed from the Table 5 that the actual experimental and simulation results matching over each other. While water splits for simulated is higher than the experiment results with percentage error varies from 0.241 to 3.06. Having a close matching of the results from both experiment and simulation, it is a first assumption the fluid flow pattern in hydrocyclone are similar to each other. This result also support that M4 with the higher mesh density give the better prediction to simulation.

4.5. Static Pressure

Fluid flow characteristic is indirectly measured by considering the static pressure distribution inside the hydrocyclone. An increase in the inlet pressure has also increased the static pressure differential along the radius within the hydrocyclone and as a result more water split into overflow. To understand further the static pressure characteristic in the hydrocyclone, the maximum and minimum values obtained in a particular radial line are plotted against axial height in Figure 5.

From Figure 5, at each axial height, the static pressure is maximum at the wall and decreases toward the center core of hydrocyclone. Furthermore, the static pressure also decreasing from the top to bottom (spigot) of hydrocyclone.

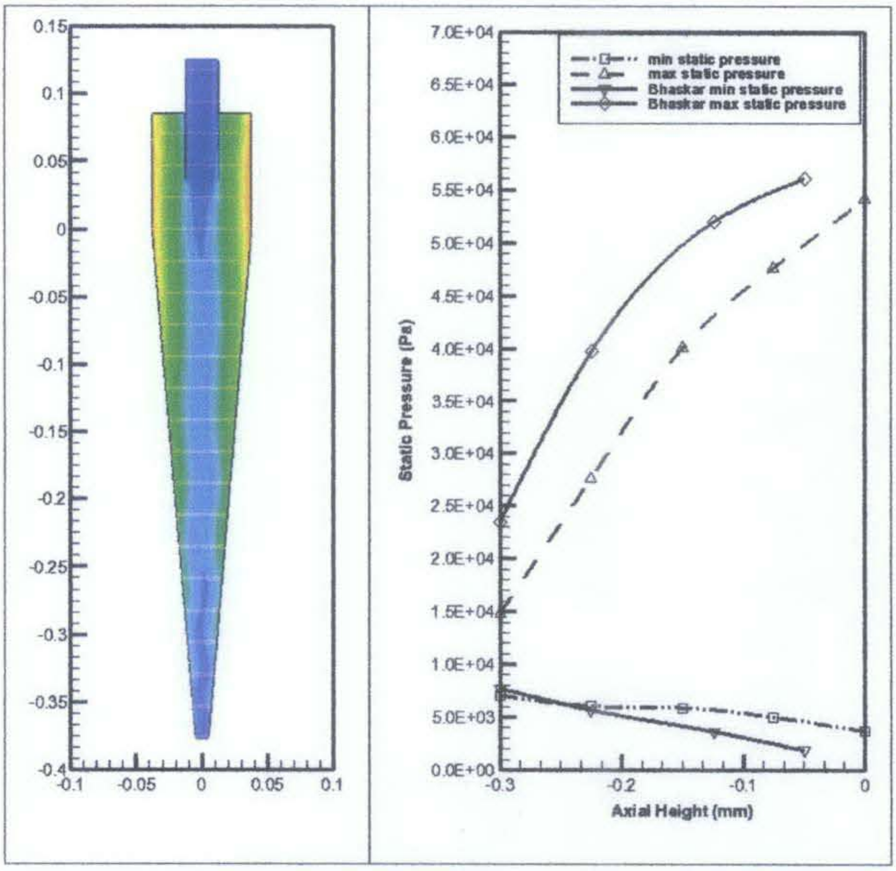


Figure 5:Minimum and Maximum Values of Static Pressure at Different Axial Heights.

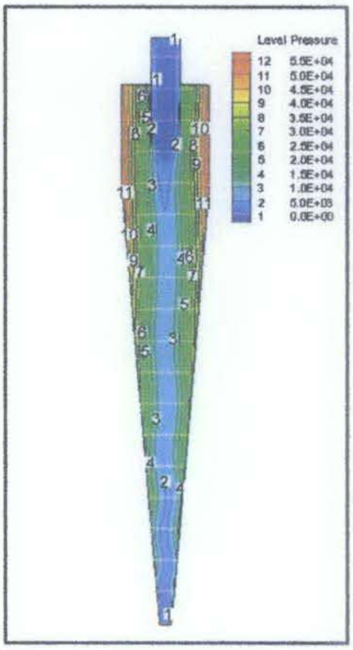
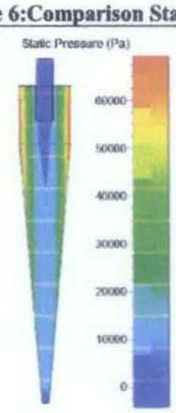
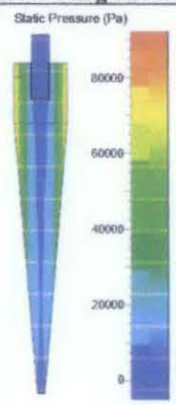
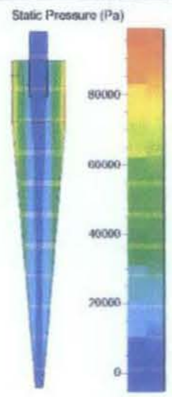
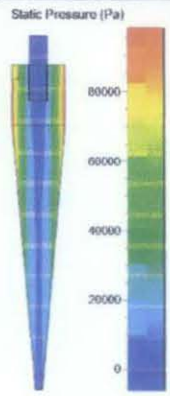


Figure 6:Contour of Static Pressure

In previous chapter 2 have been discussed that fluid flow characteristic can be determine with static pressure. In Table 6, can be observed with increasing mesh density, the static pressure inside hydrocyclone also change with increasing of pressure drop. Furthermore, at M1 with mesh density 100000 cell sizes, the flow inside hydrocyclone seems like not well developed thus the fluid flow inside hydrocyclone was not smooth enough and reduce it efficiency. The pressure drop for M1 also the lowest compared to others.

Slightly increasing pressure drop from M1 to M4 mesh density give the results of hydrocyclone efficiency increased. It can be seen that at core region of hydrocyclone, the minimum static pressure area keep increasing from M1 to M4 while maximum static pressure is more near the wall of hydrocyclone.

Table 6:Comparison Static Pressure at Different Mesh Cell Sizes Using FINE/Open

 **Static Pressure at 100,000 cell sizes**	 **Static Pressure at 200,000 cell sizes**
 **Static Pressure at 300,000 cell sizes**	 **Static Pressure at 400,000 cell sizes**

4.5. Velocity Profile

4.5.1. Axial Velocity

Velocity profile is another feature being observed in this study. In Figure 7, the cutting plane of hydrocyclone along y-axis for axial velocity, the negative downward axial velocity occurs near the hydrocyclone wall and cover the large part of the hydrocyclone while the upward axial velocity cover the upper core region. However, at heights approaching the bottom outlet, the contour show some kind on non-axisymmetric flow. A better flow visualization can be obtained from Figure 8 which show characteristic of axial velocity at different heights.

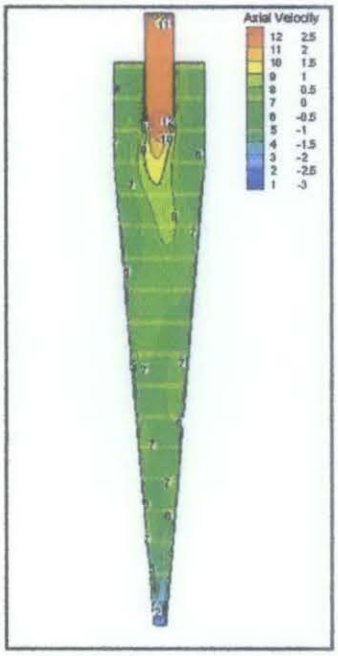


Figure 7:Axial velocity at Cutting Plane of y-axis

Observation from Figure 8 shows in respectively to z-axis along the hydrocyclone, the positive magnitude of velocity is acting upward direction at core of hydrocyclone. While when the velocity is near to the wall, the magnitude of velocity keep decreasing and change to negative value indicate the changes of direction toward bottom outlet. in addition, from the Figure 8 can be seen that the magnitude of velocity keep increasing in downward direction when reach spigot outlet as the diameter decreasing.

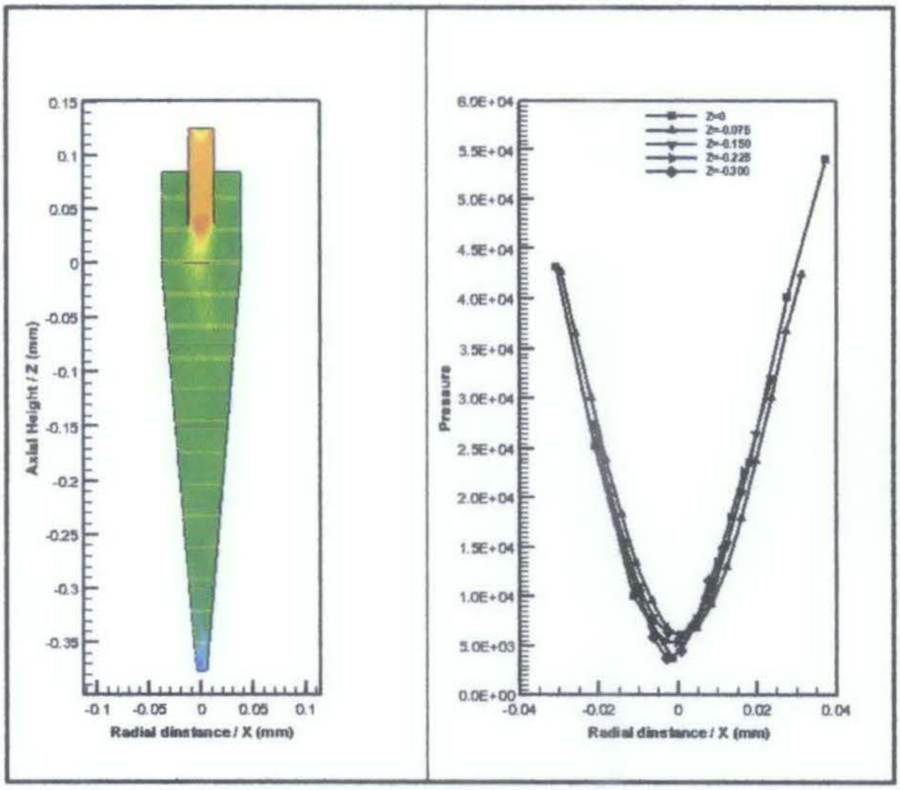
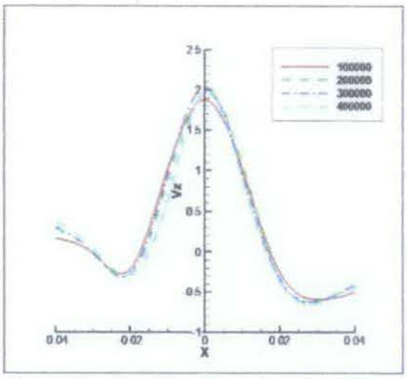
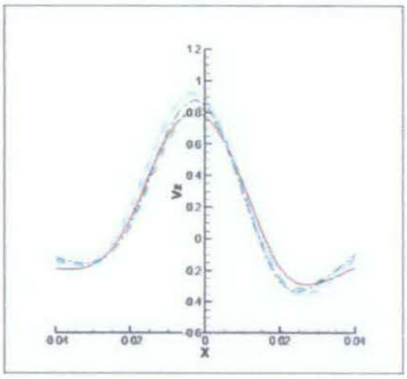


Figure 8:Axial velocity Distribution at Different Height

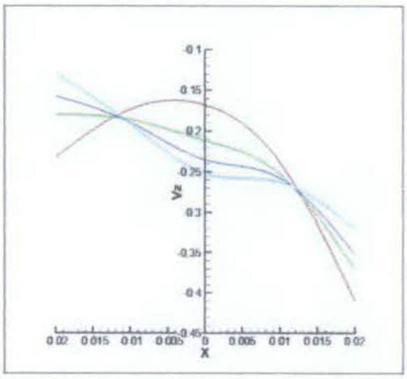
Figure 9 shows a comparison results of axial velocity at certain hydrocyclone heights for different mesh density. From observation, for part (a) and part (b) can be said that all test have the same curve pattern. However, at part (c) with height z= -0.3 mm from top hydrocyclone, the curves keep give better prediction as the mesh density keep increasing. However, it is not solid assumption to solely depend on axial velocity without supports from others data.



(a)



(b)



(c)

Figure 9:Axial Velocity at Height (a) z=0.03 mm, (b) z= 0 mm (c) z= -0.3 mm

CHAPTER 5
CONCLUSION AND RECOMMENDATION

5.1. Conclusion

In this study, computational fluid dynamics was performed to have better understanding and to predict the characteristic fluid flow behavior inside hydrocyclone in order to analyze the separation efficiency. In the first place, the solid CAD model of hydrocyclone have been manage to design in respect standard 76 mm diameter hydrocyclone. This study also, by using NUMECA FINE/Open manage to solve the simulation to analyze the flow behavior in term of static pressure, profile velocity and others more. From the results, it can be seen that the nature of fluid flow inside hydrocyclone is mainly due to centrifugal force from inlet flow. This force then will produce pressure drop in respects of flow velocity, an increasing the velocity magnitude will increase the pressure drop inside the hydrocyclone, thus make the flow separate into two section, where the lowest pressure will form a flow upward towards overflow exit while the higher pressure will keep near to the wall of hydrocyclone which form a flow in term of downward direction toward underflow exit.

However, due to limited time and limited resources, the study manage to complete the report in term of single-phase fluid flow behavior inside hydrocyclone. However, for future reference, it is recommend for next study to continue this project in term of studying the multiphase flow in hydrocyclone.

5.2. Recommendation

For future work research, this study should incorporate more important parameters such as temperature, geometries, and fluid flow, etc. More variation in term of test also need to consider to have a better agreement and consistent results. Thus, more conclusions can be made to improve this study.

REFERENCES

Dwari, R., Biswas, M., & Meikap, B. (2004). Performance Characteristics for Particles of Sand FCC and Fly Ash in a Novel Hydrocyclone. In *Chemical Engineering Science* (pp. 671-684).

Elsayed, K., & Lacor, C. (2009). Investigation of the Geometrical Parameters Effects on the Performance and the Flow Field of Cyclone Separators using Mathematical Models and Large Eddy Simulation. *ASAT-13* .

Guofeng, Z., Jong-Leng, L., & Andrew, N. (2010). Computational Study of Flow in a Micro-Sized Hydrocyclone. *17th Australasian Fluid Mechanics Conferences.* Auckland, New Zealand.

Hargreaves, J. (1990). *Computing and Measuring the Flow Field in a Deoiling Hydrocyclone.* 1990: University of Southampton, England.

Hubred, G., Mason, A., Parks, S., & Petty, C. (2000). Dispersed Phase Separations: Can CFD Help? *Proceeding of ETCE/OMAE Conference.* New Orleans, Louisiana.

J. C., & C.-W. S. *High Order Schemes For CFD: A Review.*

Luo, Q., & Xu, J. (1992). The Effect of the Air Core on the Flow Field within Hydrocyclones. *4th International Conference on Hydrocyclones,* (pp. 51-62). Southampton, England.

Matvienko, O. (2004). Analysis of Turbulence Models and Investigation of The Structure of The Flow in a Hydrocyclone. *Journal of Engineering Physics and Thermophysics* , 316-323.

Richard, A. A. (1982). The Sizing and Selection of Hydrocyclone. *Design and Installation of Comminution Circuits* .

Rushton, A., Ward, A., & Holdich, R. (2000). *Solid-liquid Fitration and Separation Technology.* Weinheim, Germany: Wiley-VCH.

Salcudean, M., Gartshore, I., & Statie, E. (2003). Test Hydrocyclones Before they are Built. *Chemical Engineering (www.che.com)* , 66-71.

Seyda, B., & Petty, C. (1991). *Separation of a Light Dispersion in a Cylindrical Vortex Chamber.* Michigan State University: Hydrocyclone Development Consortium.

Shi, L., Bayless, D., Kremer, G., & Stuart, B. (2006). CFD Simulation of the Influence of Temperature and Pressure on the Flow Pattern in Cyclones. In *Industrial and Engineering Chemistry Research* (pp. 7667-7672).

Slack, M., Cokljat, D., & Vasquez, S. (2003). Reynolds-Stress Model for Eulerian Multiphase. *Proc 4th Int. Symposium on Turbulence Heat and Mass Transfer* (pp. 1047-1054). Begell House Inc.

Su, Y., & Mao, Y. (2006). Experimental Study on the Gas-Solid Suspension Flow in a Square Cyclone Separator. *Chemical Engineering Journal vol. 121* , 51-58.

Svarovsky, L. (1984). *Hydrocyclones.* Holt, Rinehart & Winston.

Thew, M., Wright, C., & Colman, D. (1984). R.T.D. Characteristics of Hydrocyclones for the separation of Light Dispersions. *International Conference on Hydrocyclones*, (pp. 163-176).

Zhang, J., You, X.-Y., & Niu, Z.-G. (2011). Numerical Simulation of Solid-Liquid Flow in Hydrocyclone. *Chem. Biochem. Eng. Q.* , 37-41.