



รายงานวิจัยฉบับสมบูรณ์

การออกแบบอุปกรณ์ไฮโดรไซโคลนในกระบวนการแยก
สารโดยใช้กระบวนการทางคอมพิวเตอร์

โดย ดร.วรรณวิไล ไกรเพ็ชร และคณะ

มิถุนายน 2548

สัญญาเลขที่ MRG4680084

รายงานวิจัยฉบับสมบูรณ์

การออกแบบอุปกรณ์ไฮโดรไซโคลนในกระบวนการแยกสารโดยใช้
กระบวนการทางคอมพิวเตอร์

คณะผู้วิจัย สังกัด

1. ดร.วรรณวิไล ไกรเพ็ชร คณะวิศวกรรมศาสตร์ มหาวิทยาลัยศรีนครินทรวิโรฒ
2. Professor Tom Dyakowski Dept. of Chemical Engineering, University of Manchester, UK
3. Dr. Andrew Nowakowski Dept. of Mechanical Engineering, University of Sheffield, UK
4. พันโท ผศ.ดร.อโณทัย สุขแสงพนมรุ้ง กองวิชาวิศวกรรมเครื่องกล โรงเรียนนายร้อยพระจุลจอมเกล้า

สนับสนุนโดยทบวงมหาวิทยาลัย และสำนักงานกองทุนสนับสนุนการวิจัย

(ความเห็นในรายงานนี้เป็นของผู้วิจัย ทบวงฯ และสกว. ไม่จำเป็นต้องเห็นด้วยเสมอไป)

ABSTRACT

The computational fluid dynamics (CFD) code with the finite element method (FEM) and 3D unstructured grids was developed to simulate the flow within a solid-liquid hydrocyclone. An influence of the particle-particle and particle-fluid interactions on the flow was investigated by using the time-scale analysis method. It was found that the liquid-solid or Drag interaction is predominant in the most area of the hydrocyclone. Therefore, the particle trajectory method was used to modeling the flow path within the separator. The computational results demonstrate the double-vortex flow, which depict the two separation flows. Turbulence characters were modeled by using RANS method with the Smogorinsky model. This model cannot give a good prediction on this type of flow, because the flow is highly, not fully-developed, anisotropic turbulent flow. The simulation of the flow within the hydrocyclone operating with an air core by using the finite volume method (FVM) was performed. Reynolds stresses model was used to modeling the turbulence characteristic of the flow. The low pressure zone or the air-core in the center core was simulated. The redesign hydrocyclone for reducing the energy loss due to this low pressure core was done by insert the metal-rod in the middle of hydrocyclone. The effect of the inserted-rod on the velocity distributions and the separation performance was investigated. It was found that the separation performance can only be improved with the proper size of the inserted-rod.

Key words: Hydrocyclone, Finite Element Method, Finite Volume Method, Numerical Simulation, Computational Fluid Dynamics, Particle- Fluid Interaction and Liquid-solid separation

บทคัดย่อ

งานวิจัยนี้ได้พัฒนาการคำนวณพลศาสตร์ของไหลในภายในไฮโดรไซโคลน โดยใช้เทคนิคการคำนวณระเบียบวิธีเชิงตัวเลขแบบ Finite Element Method และ กริดสามมิติเรียงตัวแบบไม่เป็นระเบียบ การศึกษาอิทธิพลของปฏิสัมพันธ์ระหว่างของไหลและอนุภาคของแข็ง และระหว่างอนุภาคของแข็งด้วยกันที่มีต่อการไหลภายในไฮโดรไซโคลน โดยใช้การวิเคราะห์ Time-scale ของแรงกระทำระหว่างของไหลและอนุภาคของแข็ง และระหว่างอนุภาคของแข็งด้วยกัน พบว่าปฏิสัมพันธ์ระหว่างของไหลและอนุภาคของแข็ง มีอิทธิพลต่อการไหลมากกว่าปฏิสัมพันธ์ระหว่างอนุภาคของแข็งด้วยกันในพื้นที่การไหลส่วนใหญ่ ดังนั้นจึงสามารถใช้เทคนิค particle trajectory ในการจำลองการเคลื่อนที่ของอนุภาคของแข็ง ผลการจำลองลักษณะการไหลให้ผลสอดคล้องกับผลการทดลอง แสดงให้เห็นว่าเกิดกระแสไหลวน 2 กระแส ซึ่งจะไหลออกทางด้านบนและด้านล่าง จากการคำนวณการไหลแบบปั่นป่วนโดยใช้สมการ Reynolds Averaged Navier Stokes และแบบจำลอง Smogorinsky model พบว่าไม่สามารถอธิบายการไหลได้เนื่องจากการไหลเป็นการปั่นป่วนแบบ anisotropic และ เป็นการปั่นป่วนแบบไม่สมมาตร การจำลองการไหลในอุปกรณ์ไฮโดรไซโคลนแบบมาตรฐานและแบบที่มีการใส่แกนโลหะ โดยใช้เทคนิคการคำนวณระเบียบวิธีเชิงตัวเลขแบบ Finite Volume Method และใช้ Reynolds stresses model สามารถจำลองลักษณะของแกนอากาศความดันต่ำบริเวณตรงแกนกลางได้ดี และการใส่แกนโลหะเพื่อกำจัดแกนอากาศความดันต่ำมีผลให้การสูญเสียพลังงานลดลง และเมื่อทำการศึกษาสมรรถนะของไฮโดรไซโคลนทั้งแบบมาตรฐานและแบบใส่แกนโลหะ พบว่าการใส่แกนโลหะที่มีขนาดเหมาะสมสามารถเพิ่มสมรรถนะของไฮโดรไซโคลน

คำสำคัญ : ไฮโดรไซโคลน การคำนวณพลศาสตร์ของไหล การคำนวณระเบียบวิธีเชิงตัวเลข Finite Element Method Finite Volume Method ปฏิสัมพันธ์ระหว่างของไหลและอนุภาคของแข็ง กระบวนการแยกสาร (ระหว่างของไหลและของแข็ง)

Content

Abstract	page	i
Abstract (Thai version: บทคัดย่อ)		ii
Executive summary		1
Project description		7
I. An investigation of the effect of the particle-fluid and particle-particle interactions on the flow within a hydrocyclone		8
II. The simulation of the flow within a hydrocyclone operating with and without an air core.		27
III. The experimental study of the flow within a hydrocyclone operating with an air core		60
Exploitation (Output of the project)		74
Appendix A: An investigation of the effect of the particle-fluid and particle-particle interactions on the flow within a hydrocyclone <i>Publish in : Chemical Engineering Journal xxx (2005) xxx-xxx</i>		A
Appendix B: Numerical prediction of outlet velocity patterns in solid-liquid separators <i>Publish in: Chemical Engineering Journal xxx (2005) xxx-xxx</i>		B
Appendix C: The experimental study of the flow within a transparent hydrocyclone with a vortex finder attachment <i>Present in: American Filtration and Separations Society Annual Conference 2005 at Atlanta, GA, 10-13 April 2005.</i>		C

Executive summary

Project title: An intelligent computer-based procedure for design of hydrocyclones for separation process

1. Background and problem statement (ความสำคัญและที่มาของปัญหา)

A hydrocyclone is a piece of separation equipment for solid-liquid and liquid-liquid systems. It is used to separate dispersed particles from a continuous fluid as the effect of a swirl flow. Hydrocyclones have been used in many mineral processing and mining industries, for example, in the separation of clay in drilling mud, when sizing sand into fractions, and in the classification of iron ore into coarse and fine fractions. Recently hydrocyclones have been introduced in the food, oil and textile industries as a result of their simplicity of design and operation, high throughput, low maintenance, low operating cost and small physical size of the unit. As they have many advantages and have been used in many industries, this research programme seeks to develop a new approach and improvement in the design and operation of hydrocyclones. The development of hydrocyclone design should consider its simplicity and low operational cost. The solution of problems, such as the reduction of turbulence level and input flow shear, and the increase of the particle residence time, would open up new industrial applications.

Hydrocyclone geometry and operational parameters can be calculated by applying empirical models, which consist of a set of design equations. Most of the design equations are based on the correlations obtained experimentally, using dimensionless similarity numbers. These equations have their limitations due to the specific systems used for the model development. The application of the empirical model to different systems is still limited and unreliable. Therefore, in many cases, they cannot be used with confidence to predict the design of an individual separator, or the overall plant configuration that is required to meet different user requirements. This is a main problem, which has always been found in industry. Additionally, they do not offer an insight into the underlying physical mechanisms of fluid flow in the hydrocyclone.

Alternatively, an attempt to model the flow in a hydrocyclone has been made by using the theoretical models, which is based on solving a set of conservation equations, resulting from mass and momentum principles. This approach provides a physical

insight into the fundamental causes of the observed phenomena. The earlier theoretical models are steady state and 2D-axisymmetrical models, which are limited to dilute flow only. In addition, it is difficult to describe the behaviour of high turbulent swirling flow caused by the 3D-flow entry. Therefore, more advanced modelling is needed that allows, for example, the study of such phenomena as an adjustment of three-dimensional flow to axisymmetrical, particle-fluid, particle-particle and particle-wall interactions. Such models will allow the description of particle effects on suppressing or generating turbulence and non-Newtonian slurry flows. Additionally, in the context of modelling turbulence, a physical model is needed to show how a fluid turbulent deformation characterises swirl flows.

As discussed above, the development of hydrocyclone design should consider the complex physical phenomena affecting the performance of hydrocyclones. Therefore, advanced theoretical technique is required to obtain a better understanding of such phenomena as a 3D flow, particularly in the entry region of hydrocyclone, and the effect of the presence of particles on the flow. Therefore, the main aim of the work is to introduce Computational Fluid Dynamics (CFD) techniques to the simulation of 3D flow within a hydrocyclone by using the Finite Element Method (FEM). This technique is a better choice for a design tool since this method is unlimited, reliable and flexible in the change of hydrocyclone geometry. It can be apply to all hydrocyclone systems.

The deliverables from the research are expected to include; an advanced fluids dynamics modelling programme; this will be encapsulated in a software-package for design of particle separators for use by engineers in the industry and academic; improved fundamental knowledge of the mechanism of separation based on three-dimensional simulation.

2. Aims and objectives of the research programme (วัตถุประสงค์)

The ultimate goal of undertaken research is to take advantage of computational simulations more effectively in the overall design process and consequently optimise separation and classification process. The main objectives of the project are:

1. to introduce the powerful numerical method "Computational Fluid Dynamics (CFD) techniques", based on the finite element method to the simulation of multi-phase flow problem in chemical engineering field.

2. to simulate the hydrocyclone performance based on a particle trajectory method. The results will allow an assessment of the effect of flow behaviour on the efficiency of its performance.
3. to investigate an importance of the particle-fluid interaction mechanism of the flow within hydrocyclones by using the new approach of time-scale analysis.
4. to use the methodology to predict hydrocyclone design for a specified duty and to test the prediction by fabricating the novel hydrocyclone design and testing its performance. This refers to as an intelligent design procedure.

3. Methodology and task explanations (ระเบียบวิธีวิจัย)

Methodology

An intelligent procedure based on a computer model for hydrocyclone design was developed in this work, whereby the user specifies target objective functions to the designer. This procedure will enable a specific type of hydrocyclone to be designed to meet the requirements of various processes such as liquid classification, slurry thickening or solids washing. Hydrocyclone efficiency as a function of its geometrical parameters and solid concentration in a feed stream will be calculated. Firstly, the simulation of 3D flow within a hydrocyclone was carried out by using the Finite Element Method (FEM) with an unstructured grid. The turbulence flow character was modelled by using Smagorinsky turbulence model. The numerical results showed that this turbulence model could not give good prediction on this type of high-swirling flow. Then, the Reynolds stresses model (RSM) together with the finite volume method (FVM) was used to simulate the flow. A particle tracking method was applied to predict particle distribution. This method is based on the calculation of a particle trajectory in a Lagrangian co-ordinate system. The simulation results will be validated by the comparison of simulation data with the experimental data, in terms of the flow characteristic such as a pressure drop, split flow or low pressure zone.

A work schedule is presented in Table 1

Task 1

The arrangement and setting of the computational facilities and instruments used in numerical simulation, which was placed at Department of Chemical Engineering, Faculty of Engineering, SWU. The facilities and instruments include the PC computer work

station, C++ program compiler, RedHat Linux 7.1 operator, AVS/express Visualization edition.

Task 2

The new work proposed here has developed advanced modelling which based on the application of a 3D-particle trajectory (Lagrangian approach) for describing particle behaviour within a hydrocyclone. An importance of the particle-fluid interaction mechanism of the flow within hydrocyclones was analysed.

Task 3

The Reynolds Averaged Navier-Stokes equations (RANS) modelling approach enables the approximation of the effects of the turbulence on the mean flow. This method with the proper turbulence model was implemented to investigate the turbulent flow in this work .

Task 4

This task was concerned with solving a set of mass and momentum conservation equations. The partial differential equations were solved in 3D numerically by using the FEM and FVM methods, and for this purpose the use of a powerful computer facilities was needed.

Task 5

Since the obtained results from Task 4 were calculated from the unstructured grid system. The data were dimensionless. To visualise these data, the AVS/express visualisation edition was used as the post-processor of the obtained results. The flow simulation results were shown in forms of flow velocity (contour, velocity vectors, isoline and so on), flow pressure, and fluid and particle pathlines.

Task 6

Calculating the separation performance of hydrocyclones from the results of task 4 by using the concept of mass and momentum balance.

Task 7

This task was focus on validation of data obtained from numerical simulations. The experimental work on a transparent hydrocyclone was carried out. The flow characteristics obtained from the numerical simulations were compared with the data from literatures and the experimental work.

Task 8

The study of the effect of feed conditions and hydrocyclone geometries (inserted-rod) on the separation performance of hydrocyclone was carried out,

Table 1

Tasks	Six-month period			
	1th	2nd	3rd	4th
1. Arrangement and setting of the computational facilities and instruments.				
2. Constructing the 3D model based on particle trajectory approach.				
3. Constructing model for the approximation of the effects of the turbulence.				
4. Solving a set of partial differential equations numerically by using the FEM method.+FVM				
5. Visualisation the obtained data from Task 4.				
6. Calculating the separation performance of hydrocyclone.				
7. Validation of data obtained form numerical simulation by using the data from experiment and literature.				
8. Investigation the effect of feed conditions and hydrocyclone geometries (inserted-rod) on the separation performance and using the CFD code as a design tool.				

4. Project plan & Management (แผนการดำเนินงานวิจัยตลอดโครงการในแต่ละช่วง 6 เดือน)

The project will be managed following the project schedule (Table 1), which describes list of tasks and time scale for each six months. The detail explanations of each task is described in Section 3 (Methodology and task explanations). The tasks will be executed by the author at Srinakarinwirote University and supervised by the mentors from UMIST using common software and Internet communication. Six monthly summary research reports will be held to monitor progress against the planned programme in Table 1. Yearly review meeting between the researches may be held if necessary.

5. **Exploitation** (ผลงานหัวข้อเรื่องที่กำลังจะตีพิมพ์ในวารสารวิชาการระดับนานาชาติ)

The results of the project have been disseminated international publicity via refereed publications submitted to the established journals and a major conference.

In the first year,

Title: Influence of particle-fluid and particle-particle interactions on the performance of a hydrocyclone.

Journal: Chemical Engineering Journal (impact factor 0.677)

In the second year,

Title: Advanced modelling for hydrocyclone design

Journal: International Journal of Mineral Processing (impact factor 0.539)

6. **Research expenses** (งบประมาณโครงการ)

Table 2 งบประมาณรวมสำหรับ 2 ปี

รายการ	งวดที่ 1 (บาท)	งวดที่ 2 (บาท)	งวดที่ 3 (บาท)	รวม (บาท)
1. หมวดค่าตอบแทน				240,000
- ค่าตอบแทนหัวหน้าโครงการ	120,000	60,000	60,000	
2. หมวดค่าวัสดุ				46,000
- ค่าวัสดุคอมพิวเตอร์	10,000	10,000	-	
- ค่าวัสดุทำหุ่นจำลองพลาสติกของ Hydrocyclone	-	26,000	-	
3. หมวดค่าใช้สอย				85,000
- ค่าเอกสารประกอบการวิจัย	10,000	10,000	-	
- ค่าใช้จ่าย ค่าเดินทาง ค่าเบี้ยเลี้ยง ค่าที่พักในการไปประชุมวิชาการและการอบรมภายในประเทศ	-	15,000	-	
- ค่าทำรายงาน ค่าถ่ายเอกสาร ค่าสืบค้นข้อมูล	10,000	20,000	-	
- ค่าใช้จ่ายการตีพิมพ์วารสารทางวิชาการระดับนานาชาติ	-	20,000	-	
4. หมวดค่าครุภัณฑ์				109,000
- AVS/Express visualisation work station	90,000	-	-	
- CD RW	-	6,000	-	
- Harddisk 80 GB	-	8,000	-	
- DDRAM 1GB 512 MB	-	5,000	-	
รวมงบประมาณโครงการ	240,000	180,000	60,000	480,000

เนื้อหางานวิจัย (Project description)

Project title: An intelligent computer-based procedure for design of hydrocyclones for separation process

The research project is divided into 4 main topics as following:

- I. An investigation of the effect of the particle-fluid and particle-particle interactions on the flow within a hydrocyclone**
- II. The simulation of the flow within a hydrocyclone operating with and without an air core.**
- III. The experimental study of the flow within a hydrocyclone operating with an air core**

I. An investigation of the effect of the particle-fluid and particle-particle interactions on the flow within a hydrocyclone

1. Abstract

The effect of the particle-fluid and particle-particle interactions of the flow within a hydrocyclone is investigated. These were studied by applying the time scale analysis. It is shown that the particle-particle interactions, due to the lubrication and collision mechanisms, only play an important role in the vicinity of a hydrocyclone wall, and near the air core. In the remaining region, particle-fluid interactions are dominating. These play a vital role on the separation efficiency as illustrated by the significance of the wakes generated behind larger particles on dragging finer particles.

2. Introduction

A hydrocyclone is a type of separation equipment used for solid-liquid and liquid-liquid systems. It is used to separate dispersed particles from a continuous fluid as the effect of a swirl flow, and has been used in many mineral processing and mining industries.

In modelling the hydrocyclone performance, the influence of the particles on the flow is significant, particularly in the dense slurry flow, when the exchange of momentum from the particle-fluid, particle-particle and particle-wall interactions affect the velocity of the fluid. This may cause inefficiency in separation performance. The previous works presented by Bloor and Ingham (1973a, 1975a, 1987), Pericleous and Rhodes (1986), Pericleous (1987) and Hsieh and Rajamani (1991) discounted the effects of the presence of particles on a slurry velocity field by assuming that the flow is diluted (solids concentrate less than 5%), and the particle-fluid interaction for a single particle moving through a liquid without the presence of other particles was applied. However, when solids concentrate exceeds 5%, the presence of particles changes the velocity stresses and results in the generation of extra inertial stresses. The constitutive formulae, describing complex particle-fluid and particle-particle interactions are required. Generally, these formulae are very complicated as they take into account the spatial and temporal non-uniformities in the particle distributions as well as the acceleration of relative velocity.

There are two main approaches that can be applied to modelling multi-phase systems. They are a multi-fluid model and particle tracking method. The choice of one particular approach is determined by the character of the predominant interaction between phases. The purpose of this study is to investigate the method for determining the predominant particle interaction of the flow within a hydrocyclone. From this knowledge, the proper method for modelling particle flow in this separator can be chosen.

In this paper, the influence of neighbouring particles is described. The particle-fluid and particle-particle interactions are analysed using the concept of a time scale analysis introduced by Roco (1990). Here a case study is presented, based on the results published by Rajamani and Milin (1992) for a 75-mm hydrocyclone. On this basis, the zones of the predominant interaction mechanism can be defined and in turn, a proper choice of a specific approach for modelling the flow within hydrocyclone can be made. As a result, the complexity of the constitutive formulae describing these interactions might be significantly reduced.

3. Particle-Particle Interaction Related to Hydrocyclone Performance

In general, the hydrocyclone has been assumed to be operating at a steady state where the classical fluid dynamics can be applied. Stokes' law is generally assumed to be valid in order to develop models because it simplifies the mathematical formulation. Brownian movement, entrance effects on fluid, particle interactions, spinning of particles, curl of the fluid, wall interaction and the effect of turbulent fluctuations, are normally ignored or neglected (Roldan-Villasana *et al.*, 1993). The equations describing the separation function (see Flintoff *et al.* 1987), are based on an assumption that the particle settling velocity is not affected by the presence of other particles, and that it is a monotonic function of the particle size. In such a case it seems reasonable to assume that the separation process is linear and is a monotonic function of particle diameter. In fact, as the particles move to the wall their concentration increases and the interactions between them start to manifest through the interstitial fluid by secondary currents and pressure field changes. These particle interaction mechanisms cannot be neglected. Recently, Kumar *et al.* (2000) measured the settling velocities for particles in a poly-dispersed mixture. It is obvious that Stokes' law cannot be applied in a study where the particle interactions are considered.

Their results show that the motion of large particles is influenced only by the total volume fraction of particles within the system. Therefore, the settling velocity of large particles can be described by the Richardson-Zaki equation (Richardson and Zaki, 1954). On the other hand, the smaller particles move at almost identical velocities to the larger particles. These small particles appear to be dragged with the larger ones. The settling velocities of small particles are even larger than the corresponding Stokes velocities or those predicted by the existing theories (Batchelor, 1982) or correlations.

Taneda's (1956) experimental results showed that the permanent vortex-ring behind a sphere begins to form in the rear of a sphere when the critical particle Reynolds number, Re_p , is equal to 24. The size of the vortex-ring increases for higher Reynolds numbers and the wake behind a sphere begins to oscillate at the rear of the permanent vortex-ring when the Reynolds number is about 130. The latest experimental data of Yang *et al.* (2000) and numerical simulations of Tang *et al.* (1992) show how the dispersion of particles in a wake is organised for small Stokes number particles. Particles with Stokes number less than one are able to respond to the small-scale flow pattern and therefore become essentially flow-tracers. The trajectories of these particles distribute themselves throughout the large-scale vortex cores. Conversely particles with a Stokes number much greater than one are little affected by the fluid fluctuations at any scale and therefore simply move in the direction of their initial trajectories with only slight deviation.

The ratio between the centrifugal acceleration, w^2/r , and the gravitational acceleration, g , varies along the hydrocyclone radius. The tangential velocity reaches its peak near the hydrocyclone axis. According to Cilliers (2001), for a 10-mm diameter hydrocyclone, the ratio between the centrifugal and gravitational acceleration is around 60,000 and the particle residence time is in the order of milliseconds. These are extreme conditions in relation to the gravitational buoyancy-driven separation of the dispersed phase from the continuous phase.

Therefore, interactions similar to those predicted by Yang *et al.* (2000), and caused by the presence of the vortex ring behind particles, seems to be worthy of consideration in terms of hydrocyclone performance. These can provide an additional mechanism for finer particles

reporting to the underflow in the wake behind the larger particles (see Figure 1). Such a mechanism may explain the shape of the selectivity curve, and the fact that the bypass value is higher than the water recovery to the underflow. This selectivity curve does not have a sigmoidal shape, but exhibits a dip in regions of finer particle size. This dip is known as the fish-hook effect. The methods of modelling the fish-hook effect of the flow within hydrocyclones, based on this mechanism, were investigated by Kraipech *et al.* (2002).

The hydrodynamic behaviour of hydrocyclones treating concentrated slurries has yet to be fully understood and no accurate theory exists to simulate the phenomena occurring within hydrocyclones. Therefore, the other mechanisms explaining the fish-hook effect proposed by previous researchers such as Roldan-Villasana *et al.* (1993) and Frachon and Cilliers (1999), who introduced the idea of an influence from the turbulent dispersion on the motion of fine particles, could be possible and should not be dismissed. Figure 2 illustrates the possible mechanisms that describe the fish-hook phenomenon.

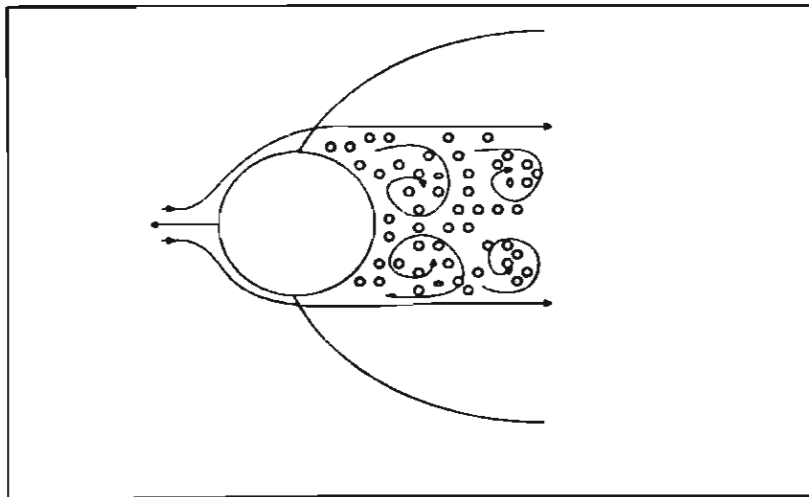


Figure 1. Dragged mechanism of fine particles by large particle.

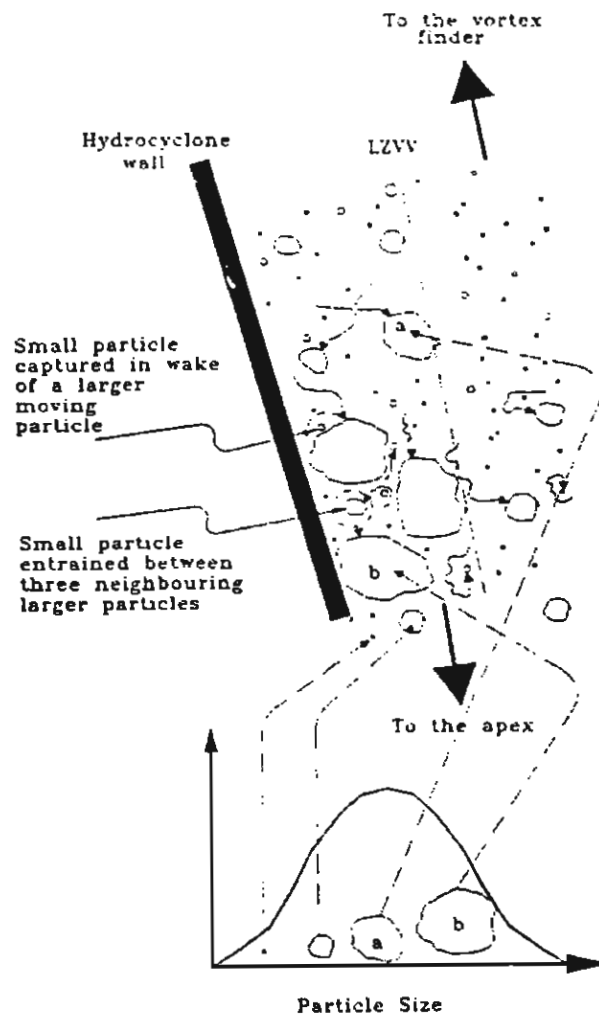


Figure 2. Mechanisms that describe the fish-hook phenomenon (Roldan-Villasana, 1992).

4. Time scale analysis

The frequency of an interaction mechanism's occurrence between particles, or between particles and fluid, is inversely proportional to the time required by the particle to respond to the interaction mechanism.

An interaction mechanism " n " that is characterised by the time scale t_n is more frequent and therefore has a larger distribution to the momentum transfer than another interaction

mechanism “ m ” characterised by the time scale t_m , if $t_n < t_m$. This relation is defined as the relative particle interaction number, $N_{n,m}$.

$$\left(\frac{1}{t_m} \right) / \left(\frac{1}{t_n} \right) = t_n / t_m = N_{n,m} \quad (1)$$

If $N_{n,m} < 1$, the n -th mechanism is prevalent comparative to the m -th mechanism.

The time scales responding to liquid-solid (drag), lubrication and collision interactions and the relative particle interaction numbers $N_{n,m}$, are shown in Tables 1 and 2, respectively. They were derived using a similar method as discussed by Roco (1986) (see Kraipech, 2002). On the contrary to the drag time scale, the lubrication and collision time scales are dependent on the velocity profile and the solids volume fraction.

Table 1. Time scales for particle interactions in the hydrodynamic range (Roco, 1990).

Type of particle interaction	Time scale, t_m	Comments
Liquid-Solid Interaction (drag)	$t_d = \frac{4}{3} \frac{sd}{C_D \mathbf{u} - \mathbf{u}_p } \quad (2)$	where $C_D = f(\text{Re}_p)$ Re_p is Particle Reynolds number.
Lubrication	$t_{\text{lub}} = \frac{18}{s} \frac{\lambda}{\dot{\gamma}_{ij}} \quad (3)$	$\lambda = \frac{1 - (\alpha^*)^{0.33}}{(\alpha^*)^{0.33}}$ where $\alpha^* = \alpha / \alpha_{\text{max}}$
Collision	$t_{\text{col}} = \frac{45}{s \dot{\gamma}_{ij}} + \frac{4.5}{s \dot{\gamma}_{ij}} \ln(\lambda) \quad (4)$	The first term on the right-hand side corresponds to the particle roughness.

Table 2. Relative particle interaction number, $N_{n,m}$ (Roco, 1990).

Relative interaction mechanisms	Significant Ratio	Relative particle interaction number
Particle-particle lubrication/ Particle-liquid drag	t_d/t_{lub}	$N_{d,lub} = \frac{s^2 d^2 \dot{\gamma}_y}{18^2 \nu \lambda} \quad (5)$
Particle-particle collisions/ Particle-liquid drag	t_d/t_{col}	$N_{d,col} = \frac{s^2 d^2 \dot{\gamma}_y}{18 \nu} \frac{1}{(45 + 4.5 \ln(\lambda))} \quad (6)$
Particle-particle collisions/ Particle-particle lubrication	t_{lub}/t_{col}	$N_{lub,col} = \frac{18 \lambda}{(45 + 4.5 \ln(\lambda))} \quad (7)$

Note: t_d is replaced by its expression for the Stokesian ($Re_p \leq 0.1$) in this Table.

5. A case study for calculating the time scales of the flow within a hydrocyclone

In this study, an application of a time scale analysis to identify the predominant interaction is presented for the 75-mm hydrocyclone, based on the experiment and prediction of Rajamani and Milin (1992). The time scales are calculated for seven flow zones within a hydrocyclone, instead of six flow zones, which was carried out in the author previous work, Kraipech (2002). The locations of each zone are described and shown in Table 3 and Figure 3.

Table 3. Location of zones.

Zone	Location
I	The upper cylindrical part of the hydrocyclone, from the top to the bottom of the vortex finder tube, outside the boundary layer on the outer wall of the vortex finder. Because of the high level of turbulence and secondary flows that originate due to the tangential entrance, the flow is well mixed containing a homogeneous suspension.
II	The middle part of the hydrocyclone from the bottom of the vortex finder tube to the middle of the conical section outside the boundary layer on the lateral walls and the boundary layer near the air core.
III	The area near the air core.
IV	The boundary layer on the lateral walls. The boundary layer starts at the feed and develops its maximum thickness when reaching the level of the bottom of the vortex finder tube.
V	The boundary layer at the outer wall of the vortex finder.
VI	The air core, which is assumed to have a cylindrical shape.
VII	The lower part of the hydrocyclone from the middle to the end of the conical section outside the boundary layer on the lateral walls and the boundary layer near the air core.

The feed slurry of the case study is 35% by wt. of limestone (16.6% by vol.). The liquid phase is water, which has a density of 1000 kg/m^3 and a kinematic viscosity of $10^{-6} \text{ m}^2/\text{s}$. The density of limestone is 2700 kg/m^3 and its volume fraction at maximum packing is 0.7. The particle size is in the range of 1 to 90 microns (Table 4). The volumetric concentration map and tangential velocity profile are shown in Figures 4 and 5, respectively.

The time scales and their relative particle interaction numbers are the functions of flow data such as the velocity field, particle sizes and the solid concentration, as shown in Table 1 and 2. Therefore, it is possible to calculate the time scales and their relative particle interaction numbers for a given set of data describing the flow.

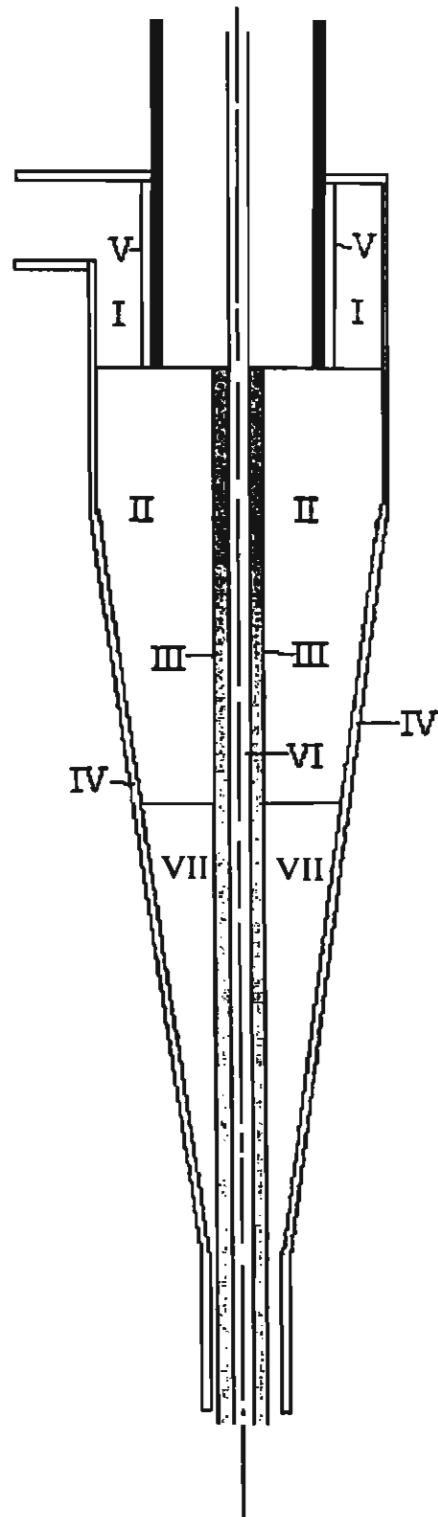


Figure 3. Zones in the hydrocyclone.

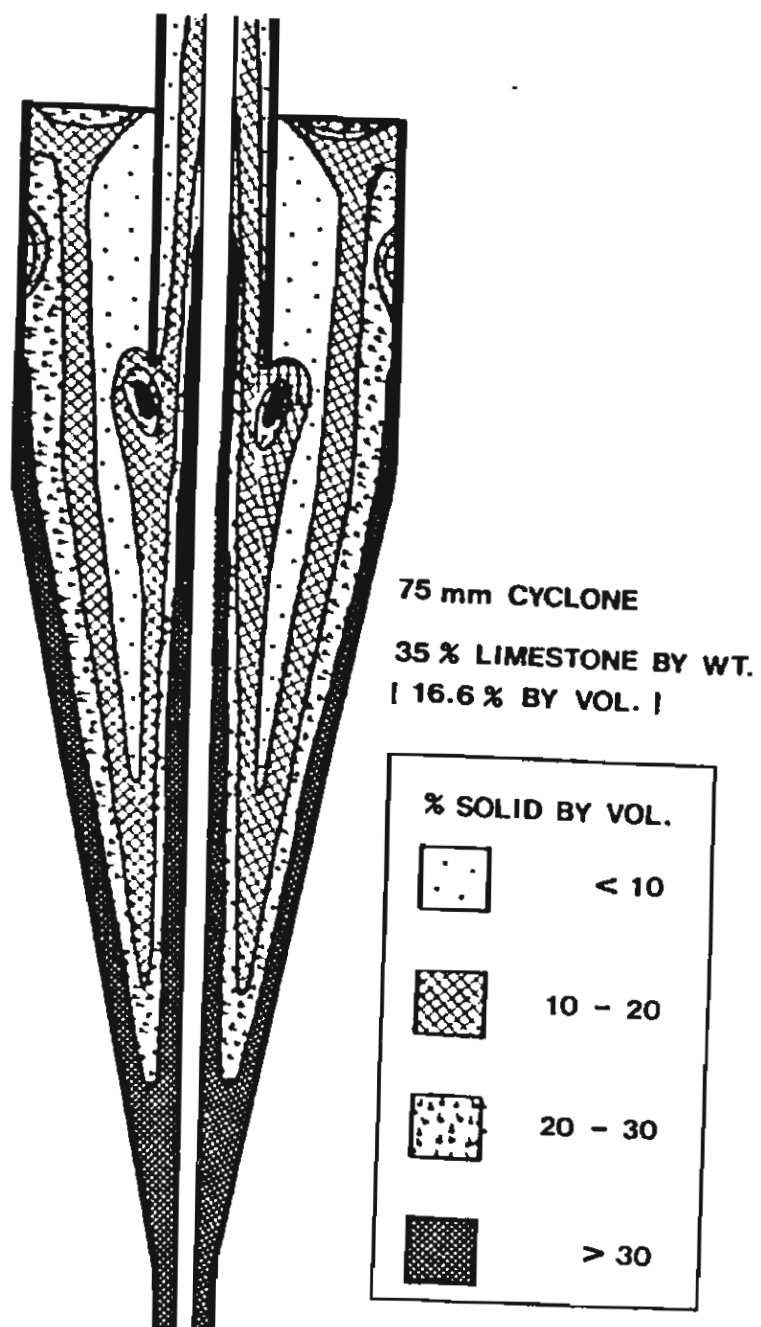


Figure 4. Predicted volumetric concentration map for 35% limestone by wt. (16.6% by vol.) in the feed (Rajamani and Milin, 1992).

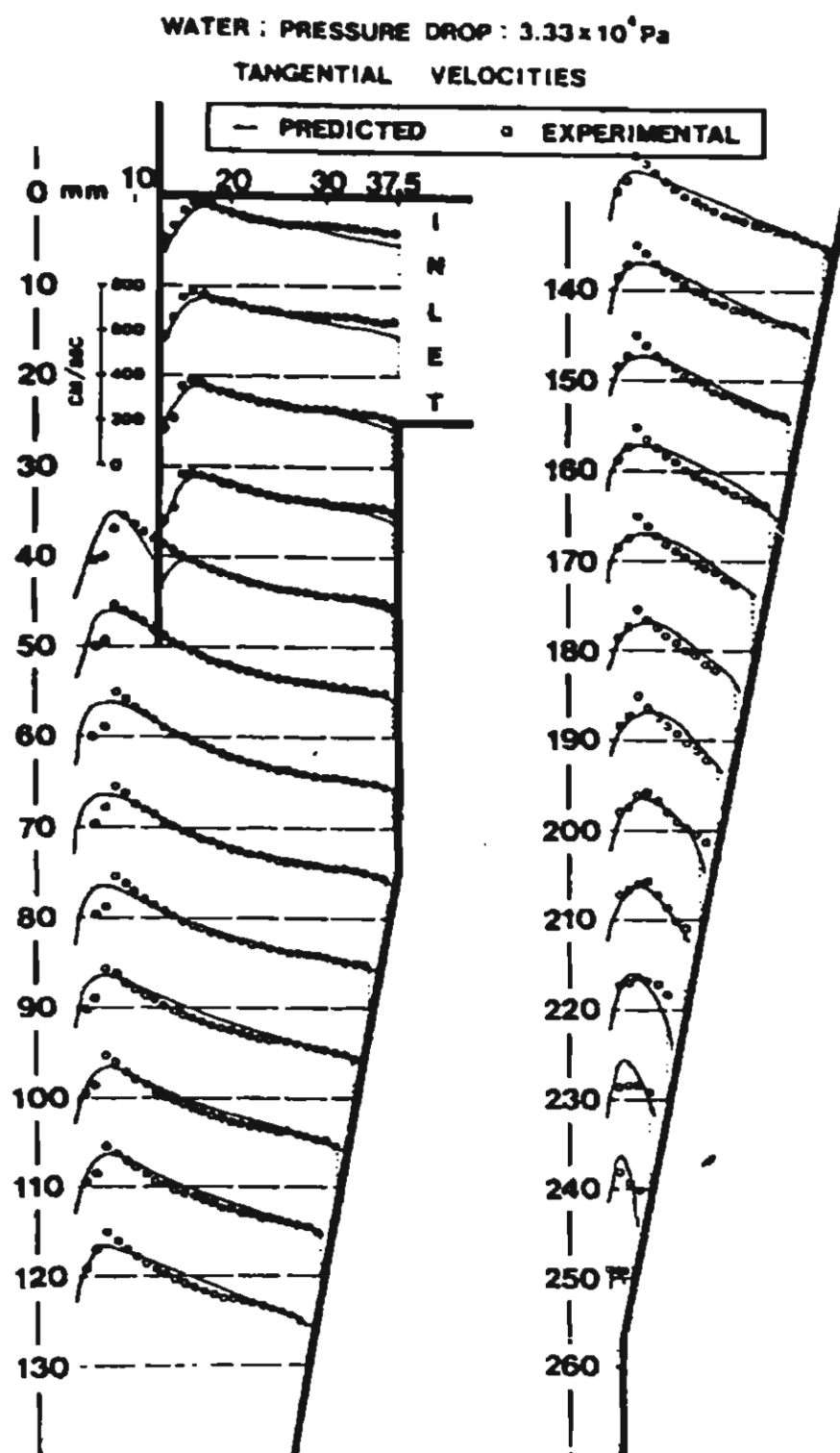


Figure 5. Measured and predicted tangential velocities in a 75-mm hydrocyclone (Rajamani and Milin, 1992).

Table 4. The experimental selectivity values (Rajamani and Milin, 1992).

Particle size (microns)	Percent of solid recovery to underflow (%)
90	100
65	92
45	80
33	52
23	30
16	20
11	16
8	11
4	10
3	9
1	7

Equations 2 to 4 are used to calculate the drag, lubrication and collision time scale. The flow around the particle is assumed to be in the Stokes' law region. Therefore, $C_D = 24/Re_p$. The aqueous suspension of limestone is assumed to be a Newtonian fluid for all range of weigh fraction in order to simplify the calculation, even though the suspension shows a non-Newtonian behaviour when its weight fraction is high. The rate of strain tensor can be calculated from the tangential velocity profile, which is presented in Figure 5. The

component $\dot{\gamma}_{r\theta} = \frac{r}{2} \frac{\partial}{\partial r} \left(\frac{w}{r} \right) + \frac{1}{2r} \frac{\partial v}{\partial \theta}$ of the rate of strain tensor is taken into this analysis,

instead of $\dot{\gamma}_{r\theta} = \frac{\partial v}{\partial r}$, which is used in Nowakowski *et al.* (2000). Assuming that the flow is

axisymmetrical, the term $\frac{1}{2r} \frac{\partial v}{\partial \theta}$ is neglected. The absolute value of the rate of strain

tensor can be calculated as $|\dot{\gamma}_{r\theta}| = \frac{r}{2} \left| \frac{\partial}{\partial r} \left(\frac{w}{r} \right) \right|$. The ratio between the inter-particle distance

and the particle diameter, λ , is equal to $\frac{1 - (\alpha^*)^{0.33}}{(\alpha^*)^{0.33}}$, where $\alpha^* = \frac{\alpha}{\alpha_{\max}}$. $\alpha_{\max}$ is the solids

volume fraction at maximum packing and α is the solids volume fraction obtained from the volumetric concentration as shown in Figure 4. In this figure, the solids concentration

distribution is presented in three ranges of volumetric concentration. According to the experimental results, it is assumed that there are no particles equal to or larger than 90 microns in diameter in Zone V (see Figure 3).

The flow characteristics are summarised in Table 5. The drag time scale as a function of the particle diameter is shown in Figure 6. The time scales for lubrication and collision interactions are presented in Table 6.

Table 5. The flow characteristics in each zone in the hydrocyclone.

Zone	Volume fraction, α	Particle size (microns)	The mean rate of strain tensor, $ \dot{\gamma}_{r\theta} = \frac{r}{2} \left \frac{\partial}{\partial r} \left(\frac{w}{r} \right) \right , (\text{s}^{-1})$
I	$\alpha < 0.1$	1 to 90	107
	$0.1 < \alpha < 0.2$		
	$0.2 < \alpha \leq 0.3$		
II	$\alpha < 0.1$	1 to 90	110
	$0.1 < \alpha < 0.2$		
	$0.2 < \alpha \leq 0.3$		
III	$\alpha > 0.3$	1 to 90	570
IV	$\alpha > 0.3$	1 to 90	179
V	$\alpha < 0.1$	1 to 65	19
VI	-	-	-
VII	$\alpha < 0.1$	1 to 90	187
	$0.1 < \alpha < 0.2$		
	$0.2 < \alpha \leq 0.3$		

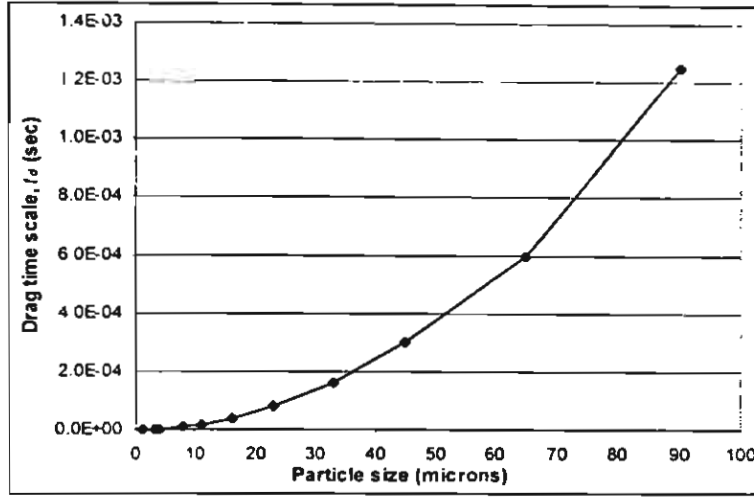


Figure 6. Drag time scale, t_d , of the limestone particle flow in water (kinematic viscosity of $10^{-6} \text{ m}^2/\text{s}$) obtained from the experimental results of Rajamani and Milin (1992).

Table 6. Lubrication time scale, t_{lub} , and collision time scales, t_{col}

Zone	Solid concentration (Volume fraction)	$t_{lub} = \frac{18}{s} \frac{\lambda}{\dot{\gamma}_{ij}}$ (s)	$t_{col} = \frac{45}{s \dot{\gamma}_{ij}} + \frac{4.5}{s \dot{\gamma}_{ij}} \ln(\lambda)$ (s)
I	$\alpha < 0.1$	$t_{lub} > 0.0561$	$t_{col} > 0.0952$
	$0.1 < \alpha < 0.2$	$0.0561 > t_{lub} > 0.0319$	$0.0952 > t_{col} > 0.0864$
	$0.2 < \alpha \leq 0.3$	$0.0319 > t_{lub} \geq 0.0201$	$0.0864 > t_{col} \geq 0.0792$
II	$\alpha < 0.1$	$t_{lub} > 0.0546$	$t_{col} > 0.0926$
	$0.1 < \alpha < 0.2$	$0.0546 > t_{lub} > 0.0319$	$0.0926 > t_{col} > 0.0840$
	$0.2 < \alpha \leq 0.3$	$0.0319 > t_{lub} \geq 0.0201$	$0.0840 > t_{col} \geq 0.0770$
III	$\alpha > 0.3$	$t_{lub} < 0.0037$	$t_{col} < 0.0149$
IV	$\alpha > 0.3$	$t_{lub} < 0.0120$	$t_{col} < 0.0473$
V	$\alpha < 0.1$	$t_{lub} > 0.0316$	$t_{col} > 0.0536$
VI	-	-	-
VII	$\alpha < 0.1$	$t_{lub} > 0.0321$	$t_{col} > 0.0545$
	$0.1 < \alpha < 0.2$	$0.0321 > t_{lub} > 0.0183$	$0.0545 > t_{col} > 0.0494$
	$0.2 < \alpha \leq 0.3$	$0.0183 > t_{lub} \geq 0.0115$	$0.0494 > t_{col} \geq 0.0453$

Figure 6 shows that an increase in the particle size leads to an increase in the drag time scale, since a larger particle needs more time to respond to the change in fluid velocity than a smaller one. The lubrication and collision time scales decrease with an increase in the solids volume fraction and the mean rate of strain tensor. This indicates that these two mechanisms are not only dependent on the flow concentration but also depend on the flow velocity field. The prevalent interaction mechanisms in each flow zone are determined by using the data shown in Figure 6 and Table 6, and are presented in Table 7 and Figure 7.

From this analysis, it is found that for the majority of the area within the hydrocyclone, where the solids concentration is less than 30% by vol., the main particle interaction mechanism is the liquid-solid interaction (drag). In the high solids concentration areas, the lateral boundary layer (zone IV) and the area near the air core (zone III), the lubrication and collision interaction mechanisms are prevalent compared with the drag interaction mechanisms. This is in agreement with the work of Nowakowski *et al.* (2000).

Table 7. Prevalent interaction mechanism for each flow zone in the hydrocyclone.

Zone	Solid concentration condition	Relative particle interaction numbers			Prevalent particle interaction mechanism
		$N_{d,lub}$	$N_{d,col}$	$N_{lub,col}$	
I	$\alpha \leq 0.3$	$N_{d,lub} < 1$	$N_{d,col} < 1$	$N_{lub,col} < 1$	Liquid-solid interaction (Drag)
II	$\alpha \leq 0.3$	$N_{d,lub} < 1$	$N_{d,col} < 1$	$N_{lub,col} < 1$	Liquid-solid interaction (Drag)
III	$0.3 < \alpha < 0.52$	$N_{d,lub} < 1$	$N_{d,col} < 1$	$N_{lub,col} < 1$	Liquid-solid interaction (Drag)
	$\alpha \geq 0.52$	$N_{d,lub} > 1$	$N_{d,col} < 1$	$N_{lub,col} < 1$	Lubrication interaction
IV	$0.3 < \alpha < 0.64$	$N_{d,lub} < 1$	$N_{d,col} < 1$	$N_{lub,col} < 1$	Liquid-solid interaction (Drag)
	$\alpha \geq 0.64$	$N_{d,lub} > 1$	$N_{d,col} < 1$	$N_{lub,col} < 1$	Lubrication interaction
V	$\alpha < 0.1$	$N_{d,lub} < 1$	$N_{d,col} < 1$	$N_{lub,col} < 1$	Liquid-solid interaction (Drag)
VI	-	-	-	-	-
VII	$\alpha \leq 0.3$	$N_{d,lub} < 1$	$N_{d,col} < 1$	$N_{lub,col} < 1$	Liquid-solid interaction (Drag)

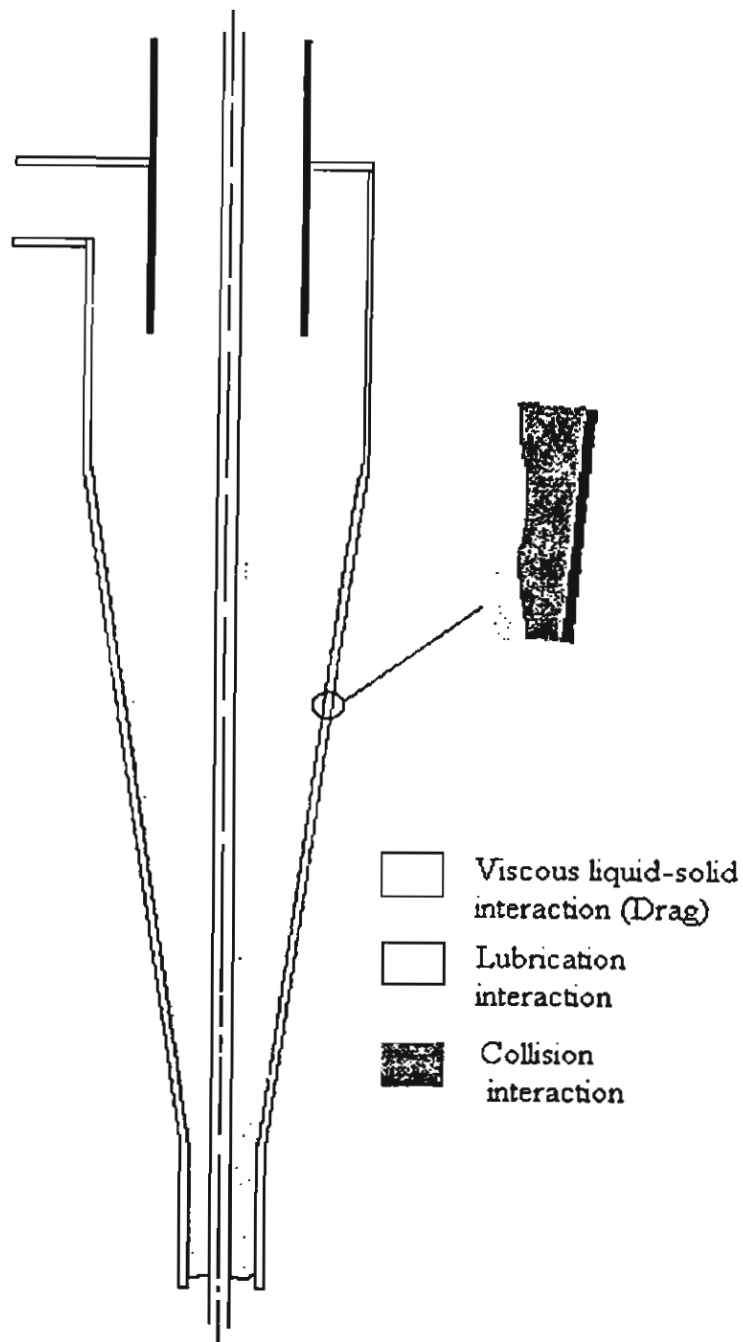


Figure 7. Main particle interaction mechanisms in the 75-mm hydrocyclone.

6. Conclusion

An application of a qualitative analysis based on a "time scale" concept was presented and discussed to assess the predominant interaction mechanisms within a hydrocyclone. It was found that the liquid-particle interaction (drag) plays an important role in the main body of a

hydrocyclone. However, within the regions close to the walls and the air core, both lubrication and collision mechanisms are predominant. This means that the flow pattern within a hydrocyclone should not only be described by interactions between particles and carrying fluid but that the results of solid mechanics should also be included in modelling particle-particle collisions in the vicinity of the hydrocyclone walls. The future work should lead to deriving a simplified model for the particle transport taking into account only the dominating relevant forces in each region. The extension of the analysis to the non-Newtonian behaviour of the high weight fraction of limestone suspensions is also planned. .

7. References

- Batchelor, G. K. (1982)** Sedimentation in a dilute polydisperse system of interacting sphere: Part 1. General theory, *Journal of Fluid Mechanics*, Vol.119, pp. 372-408.
- Bloor, M. I. And Ingham, D. B. (1973a)** Theoretical investigation of the flow in a conical, *Transactions of the Institution of Chemical Engineers*, Vol. 51, pp. 36-41.
- Bloor, M. I. And Ingham, D. B. (1975a)** Turbulent spin in a hydrocyclone, *Transactions of the Institution of Chemical Engineers*, Vol. 53, pp. 1-6.
- Bloor, M. I. And Ingham, D. B. (1987)** The flow in industrial cyclones, *J. Fluid Mech*, Vol. 178, pp. 507-519.
- Cilliers, J. J. (2001)** Private Communication.
- Flintoff, B. C., Plitt, L. R., Turak, A. A. (1987)** Cyclone modelling a review of present technologies, *CIM Bulletin*, Vol. 80, pp. 39-50.
- Frachon, M. and Cilliers, J. J. (1999)** A general model for hydrocyclone partition curves, *Chemical Engineering Journal*, Vol. 73, pp. 53-59.
- Hsieh, K. T. and Rajamani, R. K. (1991)** Mathematical model of hydrocyclone based on physics of fluid flow, *AIChE Journal*, Vol. 37. No. 5, pp. 735-745.
- Kraipech, W. (2002)** Studying the performance of Mineral Hydrocyclones, Ph.D. Thesis, University of Manchester, Institute of Science and Technology.
- Kraipech, W., Chen, W., Parma, F. and Dyakowski, T. (2002)** Modelling the fish-hook effect of the flow within hydrocyclones, *International Journal of Mineral Processing*, Vol.66, pp.49-65.

- Kumar, S., Pirog, T. W. and Ramkrishna, D. (2000)** A new method for estimating hindered creaming/settling velocity of particles in polydisperse systems. *Chemical Engineering Science*, Vol. 55, pp.1893-1904.
- Nowakowski, A., Kraipech, W., Dyakowski, T. and Williams, R. (2000)** The hydrodynamics of a hydrocyclone based on a three-dimensional multi-continuum model, *Chemical Engineering Journal*, Vol. 80, pp. 275-282.
- Pericleous (1987)** Mathematical simulation of hydrocyclones, *Applied Mathematical Modelling*, Vol. 11, pp. 242-255.
- Pericleous, K. A. and Rhodes, N. (1986)** The hydrocyclone classifier: A numerical approach, *International Journal of Mineral Processing*, Vol. 17, pp. 23-43.
- Rajamani, R. K. and Milin, L. (1992)** Fluid-flow model of the hydrocyclone for concentrated slurry classification, in Savarovsky, L and Thew, M. T. (Eds), *Hydrocyclone Analysis and Applications*, London: Kluwer Academic Publishers, pp. 59-108.
- Richardson, J. F. and Zaki, W. N. (1954)** Sedimentation and fluidisation: Part 1., *Transactions of the Institution of Chemical Engineers*, Vol. 32, pp. 35-53.
- Roco, M. C. (1986)** Dense Slurry Flow, *Proceedings of Advancement in Aerodynamics, Fluid Mechanics And Hydraulics*, ASCE, EM & HY Div., pp. 365-376.
- Roco, M. C. (1990)** Turbulent flow of incompressible mixtures, *Encyclopedia of Fluid Mechanics*, Houston, Texas: Gulf Publisher Company, Vol. 10, pp. 1-68.
- Roldan-Villasana, E. J., Williams, R. A. and Dyakowski, T. (1993)** The origin of the fish-hook effect in hydrocyclone separators, *Powder technology*, Vol. 77, pp. 245-250.
- Taneda, S., (1956)** Studies on wake vortices (III): Experimental investigation of the wake behind a sphere at low Reynolds numbers. *Reports of Research Institute for Applied Mechanics IV*, Vol.16, pp. 99-105.
- Tang, L., Wen, F., Yang, Y., Crowe, C. T., Chung, J. N. and Troutt, T. R. (1992)** Self-organizing particle dispersion mechanism in a plane wake. *Physics of Fluids A4*, Vol. 10, pp. 2244-2251.
- Yang, Y., Crowe, C. T., Chung, J. N. and Troutt, T. R. (2000)** Experiments on particle dispersion in a plane wake. *International Journal of Multiphase Flow*, Vol. 26, pp. 1583-1607.

II. The simulation of the flow within a hydrocyclone operating with and without an air-core.

1. Abstract

The computational fluid dynamics (CFD) code with the finite element method (FEM) and 3D unstructured grids was developed to simulate the flow within a solid-liquid hydrocyclone. The approach offers significant advantages in the solution process of convection dominated internal flows having one inlet and more than one outlet. It also deals with the complex geometry of the head entry part of hydrocyclone. The boundary conditions represent forces and are efficiently incorporated into the numerical formulation. Such formulation is very useful since it allows modelling the characteristic velocity profile in the outlet. From previous work of the time-scale analysis, it was found that the liquid-solid or Drag interaction is predominant in the most area of the hydrocyclone. Therefore, the particle trajectory method was used to modelling the flow path within the separator. The computational results demonstrate the double-vortex flow, which depict the two separation flows. Turbulence characters were modelled by using RANS method with the Smogorinsky model. This model cannot give a good prediction on this type of flow, because the flow is highly, not fully-developed, anisotropic turbulent flow. The interaction between the swirling flow and velocity profile at the outlet was investigated. The studies are carried out for fluids with different properties and can be extended to hydrocyclones with different geometrical configurations.

The simulation of the flow within the hydrocyclone operating with an air core by using the finite volume method (FVM) was performed. Reynolds stresses model was used to modelling the turbulence characteristic of the flow. The low pressure zone or the air-core in the centre core was simulated. The redesign hydrocyclone for reducing the energy loss due to this low pressure core was done by insert the metal-rod in the middle of hydrocyclone. The effect of the inserted-rod on the velocity distributions and the separation performance was investigated. It was found that the separation performance can only be improved with the proper size of the inserted-rod.

2. Introduction

A hydrocyclone is a piece of separation equipment for solid-liquid and liquid-liquid systems. Recently hydrocyclones have been introduced in the food, oil and textile industries as a result of their simplicity of design and operation, high throughput, low maintenance, low operating cost and small physical size of the unit. As they have many advantages and have been used in many industries, this research programme seeks to develop a new approach and improvement in the design and operation of hydrocyclones.

The development of hydrocyclone design should consider its simplicity and low operational cost. The solution of problems, such as the reduction of turbulence level and input flow shear, and the increase of the particle residence time, would open up new industrial applications. Hydrocyclone geometry and operational parameters can be calculated by applying empirical models, which consist of a set of design equations. Most of the design equations are based on the correlations obtained experimentally, using dimensionless similarity numbers. These equations have their limitations due to the specific systems used for the model development. The application of the empirical model to different systems is still limited and unreliable. Therefore, in many cases, they cannot be used with confidence to predict the design of an individual separator, or the overall plant configuration that is required to meet different user requirements. This is a main problem, which has always been found in industry. Additionally, they do not offer an insight into the underlying physical mechanisms of fluid flow in the hydrocyclone.

Alternatively, an attempt to model the flow in a hydrocyclone has been made by using the theoretical models, which is based on solving a set of conservation equations, resulting from mass and momentum principles. This approach provides a physical insight into the fundamental causes of the observed phenomena. The earlier theoretical models are steady state and 2D-axisymmetrical models, which are limited to dilute flow only. In addition, it is difficult to describe the behaviour of high turbulent swirling flow caused by the 3D-flow entry. Therefore, more advanced modelling is needed that allows, for example, the study of such phenomena as an adjustment of three-dimensional flow to axisymmetrical, particle-fluid, particle-particle and particle-wall interactions. Such models will allow the description of particle effects on suppressing or generating turbulence and non-Newtonian slurry flows.

Additionally, in the context of modelling turbulence, a physical model is needed to show how a fluid turbulent deformation characterises swirl flows.

In a hydrocyclone operational control, the primary operating parameter that can be controlled without changing fluid properties or measuring the solids in the under/overflow is the discharge angle from the apex. Assuming that two distinct operational states of hydrocyclone exist, which are the following: roping and spray. The states differ by the discharge profile from the apex. Roping tends to form a rope-like discharge, while spray resembles an umbrella. Concha et al. (1996) and Neesse et al. (2003) point out that the best separation occurs near the formation of rope discharge. Similarly, the experimental work in this study (described in the next section) has found that the hydrocyclone achieves better separation when there is less spraying but more roping in the underflow discharge profile. With the calculation of the angle, determination of the operating performance of the hydrocyclone can be predicted. The approach of this simulation work can be applied as a tool to determine the effectiveness of an existing solid-liquid separator by calculating the discharge angle.

There were some researchers had studied the performance of the hydrocyclone operating without an air core. Luo et al. (1989) studied the velocity profiles of hydrocyclones by using the LDA technique. They removed the air core by sealing the apex with water and compared the velocity profiles of the ordinary and water-sealed hydrocyclones. They concluded that the water sealed hydrocyclone was superior, but that the capacity was reduced and the seal had to be carefully maintained. Xu et al. (1990a) and (1990b) also measured the velocity components of the hydrocyclone without an air core by using LDA. They replaced the air core with a solid rod and claimed that the air core contributes to the energy loss, and the separation efficiency should increase by removing it. However, they did not validate their statement. In order to test this postulation, Lee and Williams (1993) carried out an extensive series of experiments whereby they measured the classification and separation efficiency of the conventional hydrocyclones compared with modified hydrocyclones, in which the air core was replaced by a steel rod insert. They reported that the insertion of a rod into the hydrocyclones does not appear to improve the separation efficiency. In the later, Chu et al. (2004) carried out the experiment on the hydrocyclone

with an inserted-rod, and they found that the inserted-rod could improve the separation performance of hydrocyclone. They reported that negative effect on the separation found in the experimental study of Lee and Williams (1993) might be mainly due to their body supports designed for fixing the solid rod as the main flow field inside the hydrocyclone might be disturbed by the body supports, and the negative effect of this on separation performance might be more remarkable than the positive effect of eliminating the air core. As the result, the hydrocyclone separation performance was not improved, but deteriorated.

The main aim of the work is to introduce Computational Fluid Dynamics (CFD) techniques to the simulation of 3D flow within a hydrocyclone. This procedure is employed to predict velocity fields in the hydrocyclone with different geometries operating under a wide range of conditions. A method for predicting particle trajectories in hydrocyclone and its separation efficiency is demonstrated. The numerical results are compared with available experimental data showing quiet good agreement. The flow behaviour and the separation performance of a hydrocyclone operating with an inserted-rod are also studied.

3. The simulation of the flow within a hydrocyclone operating without an air core using the finite element method (FEM)

3.1. Governing equations and variational formulation

In the study, the necessary numerical data are obtained using finite-element approximation of incompressible viscous flow. The "stress-divergence" form of the continuity equation and the Navier-Stokes equations (Gresho, 1991) are solved. The form is commonly used in finite element methods and rarely in finite difference or volume methods. The advantage of the "stress-divergence" form is that it permits formulation of a physically meaningful Neumann boundary conditions via proper accounting of viscous forces.

The variational formulation of the Navier-Stokes equations and the continuity equations are:

$$\int_{\Omega} \left\{ \phi \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) + \nu \nabla \phi \cdot (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \nabla \phi \cdot \mathbf{p} \right\} d\Omega$$

$$= \int_{\Gamma} \phi \left(-\mathbf{n} p + \nu \mathbf{n} \cdot (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) \right) d\Gamma = \rho \int_{\Gamma} \phi \mathbf{n} \cdot \boldsymbol{\sigma} d\Gamma \quad (1)$$

$$\int_{\Omega} \psi \nabla \cdot \mathbf{u} d\Omega = 0 \quad (2)$$

In this work, the finite element solution of the discrete problem is defined using the variational problem 1 and 2 on ten-nodes tetrahedral elements (Figure 1). In the finite element approach, the solution is expanded in a set of basis functions, and the discrete equations are obtained by requiring the residuals to be orthogonal to the set of test functions. The approach, which will follow here, is the Galerkin weighted residual statement, where (ϕ, ψ) are test functions, which are equated to the interpolation functions used for $\mathbf{u}$ and p , respectively.

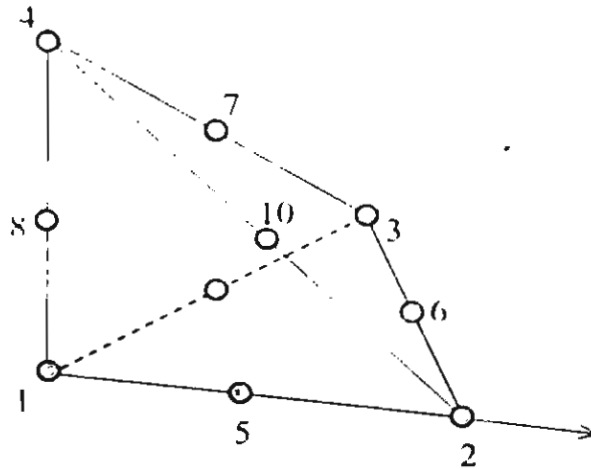


Figure 1. Ten-nodes tetrahedral element.

The element interpolation functions for dependent variables are approximated by expansions of the form:

$$u_i = \sum_{m=1}^M \phi_m(x) u_i^m = \Phi^T \mathbf{u}_i \quad (3)$$

$$p = \sum_{l=1}^L \psi_l(x) p^l = \Psi^T \mathbf{p} \quad (4)$$

where Φ and Ψ are column vectors of test (interpolation) functions, and $\mathbf{u}$, and $\mathbf{p}$ are vectors of nodal values of velocity components and pressure. The solution expansion is obtained by multiplying these test functions by vertex-based solution variables and summing over the entire grid. The structure of the global equation system can be written in the following way.

$$\begin{bmatrix} \mathbf{M} & \mathbf{0} & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \mathbf{M} & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{M} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{0} & \mathbf{0} \end{bmatrix} \begin{Bmatrix} \dot{\mathbf{u}} \\ \dot{\mathbf{v}} \\ \dot{\mathbf{w}} \\ \dot{\mathbf{p}} \end{Bmatrix} + \begin{bmatrix} \mathbf{N}(\mathbf{u}) & \mathbf{0} & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \mathbf{N}(\mathbf{u}) & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{N}(\mathbf{u}) & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{0} & \mathbf{0} \end{bmatrix} \begin{Bmatrix} \mathbf{u} \\ \mathbf{v} \\ \mathbf{w} \\ \mathbf{p} \end{Bmatrix} + \begin{bmatrix} 2\mathbf{K}_{11} + \mathbf{K}_{22} + \mathbf{K}_{33} & \mathbf{K}_{21} & \mathbf{K}_{31} & -\mathbf{C}_1 \\ \mathbf{K}_{12} & \mathbf{K}_{11} + 2\mathbf{K}_{22} + \mathbf{K}_{33} & \mathbf{K}_{32} & -\mathbf{C}_2 \\ \mathbf{K}_{13} & \mathbf{K}_{23} & \mathbf{K}_{11} + \mathbf{K}_{22} + 2\mathbf{K}_{33} & -\mathbf{C}_3 \\ -\mathbf{C}_1^T & -\mathbf{C}_2^T & -\mathbf{C}_3^T & \mathbf{0} \end{bmatrix} \begin{Bmatrix} \mathbf{u} \\ \mathbf{v} \\ \mathbf{w} \\ \mathbf{p} \end{Bmatrix} = \begin{Bmatrix} \mathbf{F}_1 \\ \mathbf{F}_2 \\ \mathbf{F}_3 \\ \mathbf{0} \end{Bmatrix} \quad (5)$$

The vectors $\mathbf{u}$, $\mathbf{v}$, $\mathbf{w}$, $\mathbf{p}$ contain the nodal unknown velocities and pressure. The coefficient matrices shown in equations 5 are defined by

$$\mathbf{M} = \int_{\Omega} \Phi \Phi^T d\Omega \quad (6)$$

$$\mathbf{N}(\mathbf{u}) = \int_{\Omega} \Phi (\Phi^T \mathbf{u}_j) \frac{\partial \Phi^T}{\partial x_j} d\Omega \quad (7)$$

$$\mathbf{K}_{ij} = \int_{\Omega} \nu \frac{\partial \Phi}{\partial x_i} \frac{\partial \Phi^T}{\partial x_j} d\Omega \quad (8)$$

$$\mathbf{C}_i = \int_{\Omega} \frac{\partial \Phi}{\partial x_i} \Psi^T d\Omega \quad (9)$$

$$\mathbf{F}_i = \int_{\Gamma} \Phi f_i d\Gamma \quad (10)$$

Index notation (indices i, j) is used for $\mathbf{u} = \{u, v, w\}$.

In the global coefficient matrix, the diagonal $\mathbf{M}$ and $\mathbf{N}(\mathbf{u})$ sub-matrices represent the effects of transient and advection terms, respectively. The sub-matrices $\mathbf{K}$ represent the effect of diffusion in conservation equations (1). The $\mathbf{C}$ matrices are the pressure gradient operators and their transposes, and the $\mathbf{C}^T$ matrices appearing in the continuity equation (2), are the velocity divergence operators. In the right-hand-side in equation 5, vectors $\mathbf{F}$ contain surface-flux type contributions from the natural boundary conditions.

3.2. Boundary conditions

The vertical cross-section of a hydrocyclone with the different parts of boundary conditions indicated are introduced in Figure 2. For fluid adjacent to a solid wall, Γ_{D_w} and at the inlet of the hydrocyclone, Γ_{D_o} the Dirichlet boundary conditions specifying the velocity vector are imposed. At outflow sections, Γ_N Neumann boundary conditions are needed to truncate the computational domain. In the present application, they represent forces and are expressed by:

$$\mathbf{f} = \boldsymbol{\tau} * \mathbf{n} = -p\mathbf{n} + 2 * \nu * \mathbf{D}(\mathbf{u}) * \mathbf{n} = \hat{\mathbf{f}} \quad (11)$$

where $\boldsymbol{\tau}$ represents total stress tensor and $\mathbf{D}(\mathbf{u})$ deformation tensor equal to the following:

$$\boldsymbol{\tau} = -p * \mathbf{I} + 2 * \nu * \mathbf{D}(\mathbf{u}) \quad \mathbf{D}(\mathbf{u}) = 0.5 * [\nabla \mathbf{u} + (\nabla \mathbf{u})^T] \quad (12)$$

Here, $\mathbf{n}$ is unit outer vector normal to the boundary, $\hat{\mathbf{f}}$ is the prescribed body force on the boundary, ν denotes the kinematic viscosity of the fluid, $\mathbf{u}$ is the velocity vector and p is the scalar pressure. In the performed numerical simulations the "no-stress" boundary conditions were prescribed at the outlets. This is the equivalent of setting $\mathbf{f} = 0$ at the spigot and vortex finder outlet. Such assumption is physically correct, it does not predefine the parameters of operations, although it may be considered idealistic. The assumption of zero-valued components of forces is natural and advantageous compared to the imposition of specific velocity profile as boundary conditions.

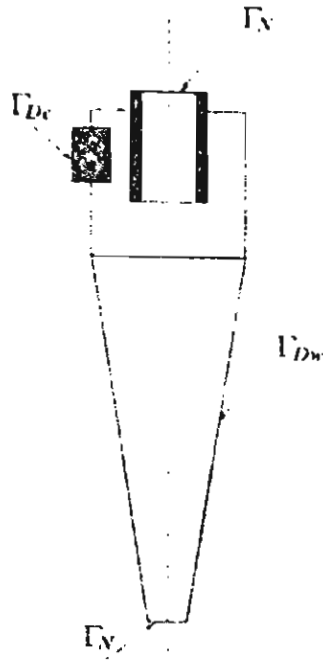


Figure 2. Boundary Conditions of Hydrocyclone.

3.3. Solution algorithm

The solution method used in this study is the pressure projection algorithm. The general concept of this algorithm was introduced by Haroutunian *et al.* (1993). The algorithm comprises the main steps as shown in the diagram comprising Figure 3. These steps were realised after discretising the Navier-Stokes equations spatially. Thus, the system (equation 5) formed the basis for the construction of the decoupled algorithm. The approach was derived using algebraic manipulation of the momentum and continuity equations. As a consequence, the boundary conditions implicit in equation 5 are consistently incorporated in the algorithm.

THE PRESSURE PROJECTION ALGORITHM

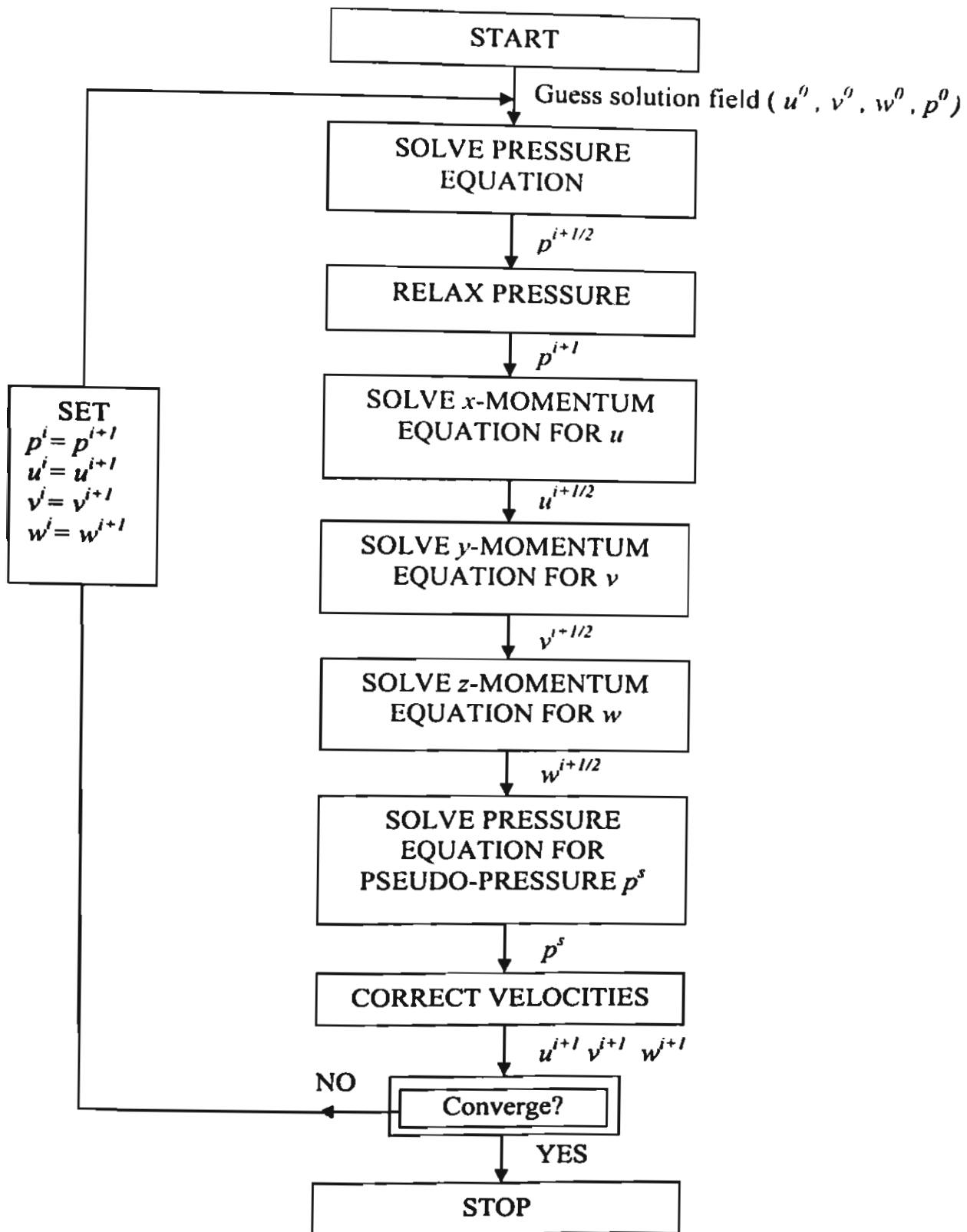


Figure 3. Diagram of the pressure projection algorithm.

3.4. Numerical simulation

The numerical code has been applied to investigate the flow in a 50-mm hydrocyclone. The fluid density and viscosity are 998.2 kg/m^3 and $6.0 \times 10^{-2} \text{ kg/m-s}$, respectively. The linear systems associated with the solution of the pressure and momentum equations have been solved by the conjugated gradient method with a tolerance on the residual to 10^{-10} . The iterative procedure is declared converged when all residuals have been reduced below 10^{-6} . The computations were carried out on a PC computer work station, C++ program compiler, RedHat Linux 7.1 and SuSe Linux operators. The CPU time was 4 days on average for the case with 37,882 nodes depending on the exact geometry of inlet and viscosity (turbulence model). It was necessary to perform from 5,000 up to 20,000 iterations.

The initial phase of the numerical solution began with the generation of 3D mesh. Owing to the complex geometry the unstructured mesh generation technique is applied instead of popular structure or block-structured generators. The generated unstructured grid consisting of tetrahedral elements is shown in Figure 4.

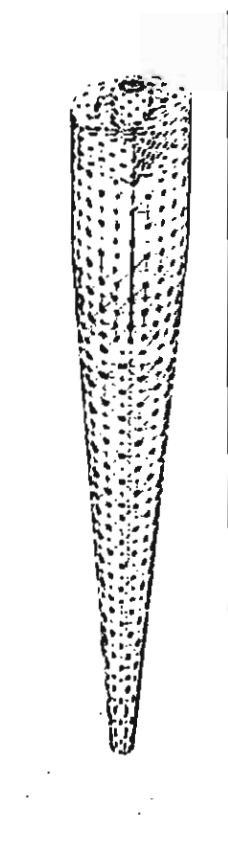


Figure 4. Generated unstructured grid consisting of tetrahedral elements within a hydrocyclone.

3.5. Numerical prediction of flow characteristics

In this work, the fluid particle pathlines were calculated by solving the following equation:

$$\frac{dx_i}{dt} = u_i \quad (13)$$

with the given initial location of point, $P(x_i^0)$ and the known velocity field, u_i .

Figure 5 shows the spiral flow in the hydrocyclone for different numbers of pathlines. Two separation flows is demonstrated. The double helical flow is observed as it can be seen that some of the incoming fluid particles move in an outer helical flow into the outer portion of the inverted cone where it begins to feed across toward the centre. Some of the downward flow leaves through the underflow orifice in the apex of the cone while the rest reverses its vertical direction and goes up via the inner helical flow and out through the vortex finder.



Figure 5. Fluid-particle pathlines.

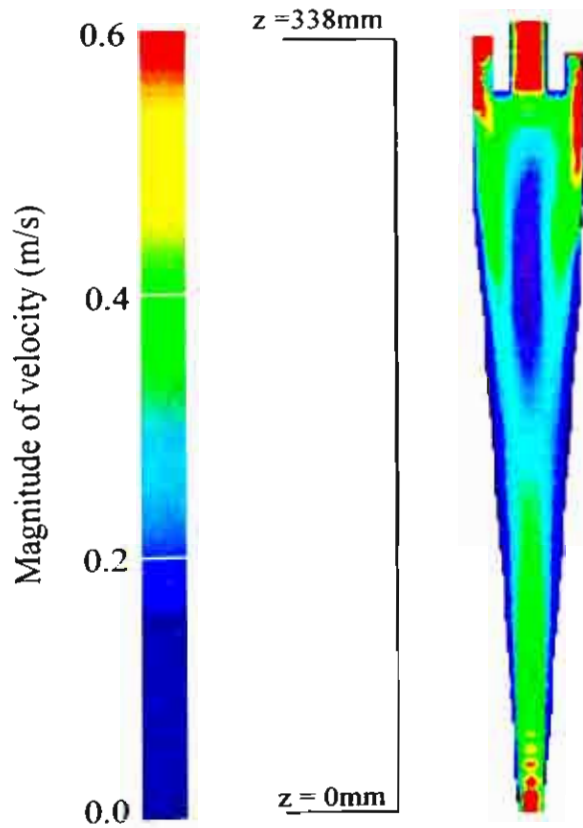


Figure 6. magnitude of fluid velocity (m/s).

The magnitude of fluid velocity (m/s) projected on a vertical plane is shown in Figure 6. The high velocity at inlet and outlet flows can be calculated. The re-circulation and separation zones were also observed as shown in Figure 7. It can be seen that the flow within hydrocyclone is asymmetric.

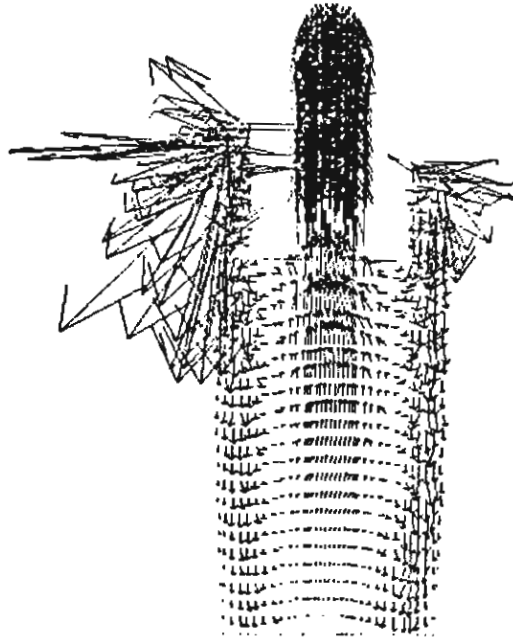
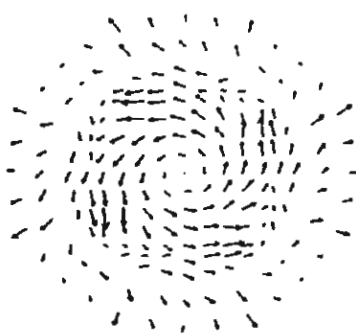
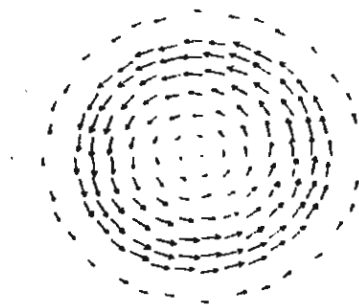


Figure 7. Obtained 3D fluid velocity field projected on a vertical plane.

One of the benefits of the FEM is the use of no stress boundary conditions at the outlets of a hydrocyclone. These boundary conditions enable the avoidance any "a priori assumptions" concerning the velocity field, and in particular do not define the mass split-ratio. Therefore, the flows at the overflow and underflow outlets have been successfully simulated as shown in Figure 8 and 9. The plane $z = 0$ mm is the plane at the apex.

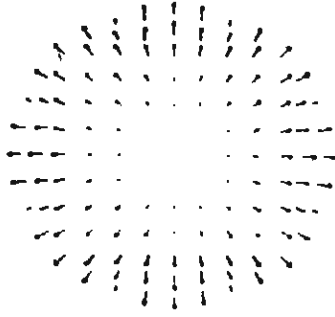


a) at Overflow outlet $z = 340$ mm

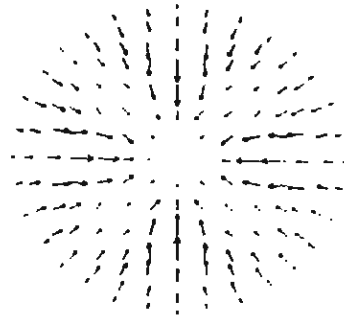


b) at Vortex finder tip $z = 313$ mm

Figure 8. Obtained 3D fluid velocity field in the vortex finder tube projected on a horizontal plane at: a) overflow outlet and b) vortex finder tip.



a) at Underflow outlet $z = 0$ mm



b) at Conical part at $z = 13$ mm

Figure 9. Obtained 3D fluid velocity field in the underflow area projected on a horizontal plane at: a) underflow outlet and b) conical part.

3.6. Numerical prediction of outlet velocity patterns

The results of the computational fluid dynamics simulation of the hydrocyclone provide the data for calculating the discharge angle. Neesse et al. (2003) showed that the form of the underflow discharge can be used as an indication of the operating state of hydrocyclones. The angle is determined from the velocity components at the spigot and fluid properties. Lacking the information from the 3-D flow field, Neesse et al. (2003) derived the following equation:

$$\alpha = \arctan\left(\frac{v}{u}\right) \approx \arctan \frac{\rho_m \frac{D_u}{z} w^2}{\mu_m} \quad (14)$$

here u , v , and w are the velocity components in the cylindrical coordinate system of the suspension in the axial(u), radial(v), and tangential direction(w), z is two times the velocity in the axial direction, ρ_m is the density of the mixture, μ_m is the viscosity of the mixture, and D_u is the apex diameter. The assumption of symmetrical flow is used, but it is not an accurate representation of the flow in the hydrocyclone. The presented approach, which will be referred throughout this report as the AFN method, is not limited to axisymmetrical flow. Thus, the AFN method having generated a 3D velocity field the angle is calculated directly. An unstructured grid that forms to the shape of a hydrocyclone using tetrahedral elements is generated. Using an unstructured grid helps not only to eliminate the occurrence of singularities but provides full geometrical flexibility.

In order to compare the results of the two methods, several runs were done at different viscosities while keeping the density constant. The range of viscosities that were chosen kept the flow in the laminar regime. Due to the laminar flow properties, the comparison would not introduce any questionable results from using a specific turbulence model. Laminar flow would be equivalent to feeding a viscous slurry in the hydrocyclone. For both methods, the angle calculations were performed for the same numerically obtained velocity data. In order for equal comparison, the viscosity in the computational domain was assumed to be constant for the Neesse method. The viscosities values decreased after the initial run to compare the change in the form of discharge. Due to the properties of the fluid, the hydrocyclone was operating in the roping region, as shown in Figure 10. As can be seen from Figure 10, the angle of the exiting fluid is predominately in the downward direction. The roping region can be distinguished from a spray discharge by observing the angle at which the slurry exits along the outer rim of the apex. The roping state tends to show that at the outer edge of the apex the discharge angle is closer to 90 than in spray discharge. However, operating in the experimented region, spray discharge does not occur.

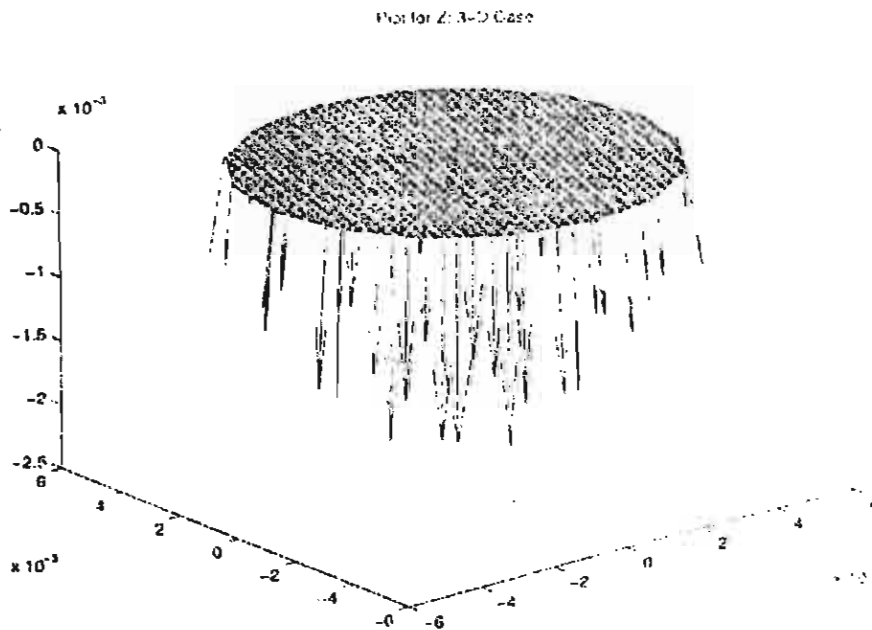


Figure 10. 3D quiver cross-section of the velocity profile exiting the apex.

Though in the model we did not take into account the air core, our model reveals the likely mechanism of air core creation. Atmospheric pressure at the centre of the apex is the reference point for the pressure field in the hydrocyclone. As noticed in Figure 11, a

pressure reduction occurs toward the centre of the hydrocyclone in each of the three different cross sections. The lowest cross section indicates that sediment has started to hinder the formation of the low pressure near the centre. As Figure 11 shows, the bottom of the hydrocyclone has already been semi-plugged with sediment, thus not allowing the low pressure field to develop at the tip. However, the examination of Figure 12 reveals that the flow is moving away from the centre at the apex.

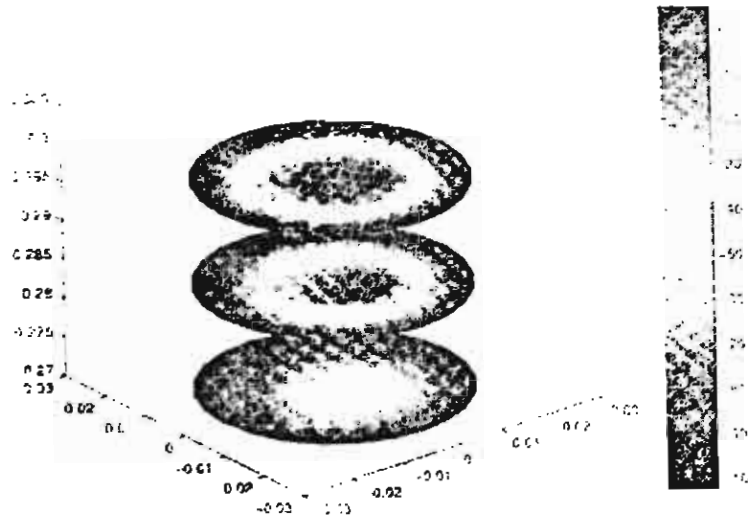


Figure 11. Cross-section of the pressure profile at the height of 50, 60, and 70 mm.

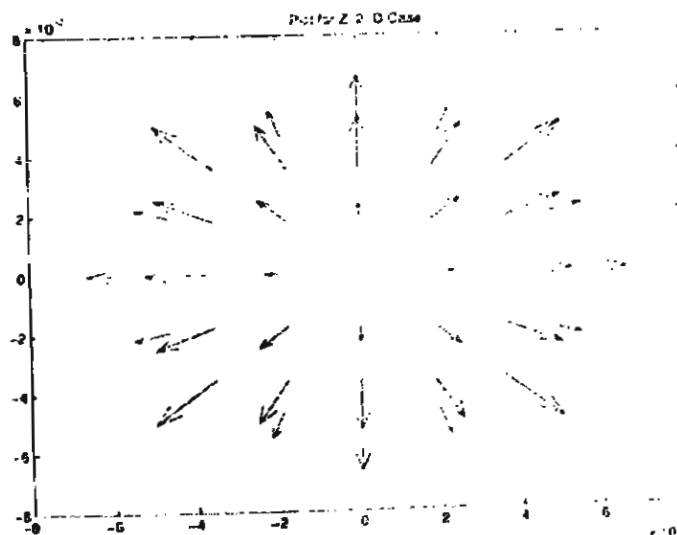


Figure 12. The 2-D velocity profile projected onto the x-y plane cross-section at the apex.

Processing the data consists of using basic trigonometric functions and using equation 14 to calculate the discharge angle. The results of the angle calculated at different viscosities

were compared between the AFN and Neesse methods (see Table 1). The comparison of the results obtained from both methods presents an interesting occurrence. The presented AFN method shows that the angle actually slightly decreases with increasing viscosity. The Neesse method predicts less of a change in the angle than the AFN method at corresponding outlet points. Since the operational state of the hydrocyclone is in roping, the expected angle exiting the spigot should be approximately 90 degrees, which is predicted by both methods at constant viscosity. The possible cause of the inversion trend of the discharge angle at the apex in both methods could be due to the high pressure that occurs at the apex.

Table 1. Comparison of AFN to the Neesse method at the apex.

Comparison of angles from the two methods at the apex					
$\mu = 0.09 \text{ Pas*sec}$		$\mu = 0.15 \text{ Pas*sec}$		$\mu = 0.2 \text{ Pas*sec}$	
AFN	Neesse	AFN	Neesse	AFN	Neesse
-87.8287	-89.9999	-86.4223	-89.9999	-85.5935	-89.8999
-85.5519	-89.9986	-83.7236	-89.9985	-82.7164	-88.9985
-80.1776	-89.9999	-77.7033	-89.9985	-76.4056	-89.9999
-78.4363	-89.9973	-75.8757	-89.9974	-74.5503	-89.9974

3.7. Conclusion

In conclusion, the Computational Fluid Dynamics (CFD) techniques to the simulation of 3D flow within a hydrocyclone operating without an air-core by using the Finite Element Method (FEM) has been introduced. This technique is a better choice for a design tool since this method is unlimited, reliable and flexible in the change of hydrocyclone geometry. It can be apply to all hydrocyclone systems. The study done on the prediction of the angle showed that both approaches tended to show similar angle profiles. Even with the simplification of the equations from 3-D to 2-D, the Neesse method shows a comparable trend to the AFN method. The difference between the two methods is the small variations in the angle calculated with increasing viscosity, which may be due to the increased pressure at the high viscosity. Additionally, experimental verification will need to be done in order to confirm whether or not one method is more accurate than the other, though the Neesse method has already been shown to work for controlling a operational cyclone. However, the trend of

both methods at the wall boundary condition coincides with experimental results from Fisher and Flack (2002) makes the results look very promising for both methods. The advantage of using the proposed approach is that the velocity profile and split ratio is not explicitly specified initially, thereby allowing the flow field to develop naturally. In the future, the plan is to look at assuming the presence of a non-Newtonian fluid, which allows the viscosity to change through the system. The ultimate goal of the study is to use the analysis of the discharge angle as a tool to assist in hydro cyclone design.

4. The simulation of the flow within a hydrocyclone operating with an air core and with an inserted metal-rod using the finite volume method (FVM)

4.1. Governing equations and discretised form

In this part, the numerical data are obtained using finite-volume approximation of incompressible viscous flow. The continuity and the Navier-Stokes equations are discretised by using the QUICK differencing scheme to create the discretised form of the governing equations for each cell. The resulting discretised equations for the variable are given by:

$$a_p \phi_p = \sum a_{nb} \phi_{nb} + S_p \quad (15)$$

where S_p is source term.

4.2. Modelling turbulence and particle motion

The Reynolds stresses turbulence model was used to calculate the turbulent character of the flow. It is based on the transport equations of the Reynolds stresses and its dissipation rate. The equation describing the components of the Reynolds stress tensor, in a tensor notation, has the form:

$$\frac{D\langle u'_i u'_j \rangle}{Dt} = P_{ij} - \varepsilon_{ij} + \varphi_{ij} + \phi_{ij} \quad (16)$$

where:

$$P_{ij} = - \left(\overline{u'_j u'_m} \frac{\partial \bar{u}_i}{\partial x_m} + \overline{u'_i u'_m} \frac{\partial \bar{u}_j}{\partial x_m} \right) \quad (17)$$

Production term,

Pressure strain term,
$$\varphi_{ij} = \frac{p'}{\rho} \overline{\left(\frac{\partial u'_i}{\partial x_j} + \frac{\partial u'_j}{\partial x_i} \right)}$$
 (18)

Dissipation term,
$$\varepsilon_{ij} = 2\nu \overline{\left(\frac{\partial u'_i}{\partial x_m} \frac{\partial u'_j}{\partial x_m} \right)}$$
 (19)

Diffusion term,
$$\phi_{ij} = \frac{\partial}{\partial x_m} \left[\nu \frac{\partial \overline{u'_i u'_j}}{\partial x_m} - \overline{u'_i u'_j u'_m} - \frac{p'}{\rho} \overline{(u'_i \delta_{jm} + u'_j \delta_{im})} \right]$$
 (20)

The motion of particles due to turbulence in the fluid phase was predicted by using the particle trajectory method. In this method, the trajectory of the discrete phase particle is obtained by integrating the force balance on the particle. The reduced momentum equations describing a balance between the drag and centrifugal or gravity forces are:

for the radial direction:

$$(\rho_p - \rho_m) \frac{w^2}{r} \pi \frac{d^3}{6} = \frac{1}{2} \rho |\mathbf{V}_{rel}| (v_p - v) \frac{\pi d^2}{4} C_D \quad (21)$$

for the axial direction:

$$(\rho_p - \rho_m) g \pi \frac{d^3}{6} = \frac{1}{2} \rho |\mathbf{V}_{rel}| (u_p - u) \frac{\pi d^2}{4} C_D \quad (22)$$

where $|\mathbf{V}_{rel}|$ is the absolute relative velocity between fluid and particle, equal to

$$\sqrt{(u_p - u)^2 + (v_p - v)^2 + (w_p - w)^2}$$

4.3. Boundary conditions and numerical technique

Since partial differential equations are incorporated in the model, it is necessary to define boundary conditions for all boundaries of the flow domain. Uniform velocity boundary condition was applied at the inlet. Pressure boundary condition, which outlet gauge pressure equal to zero, was applied to the two outlets. This kind of outlet boundary

condition was used to simulate the occurrence of the region of negative gauge pressure in the centre of hydrocyclone, where we considered as the air-core zone. No-slip boundary condition is assumed at the solid wall. As a result, all velocity components are zero at the wall. The solution method used in this study is the SIMPLEC algorithm developed by Patankar (1980).

4.4. Numerical simulation

The investigation the water flow within a 50-mm hydrocyclone was carried out. The water density and viscosity are 998.2 kg/m^3 and $1.003 \times 10^{-3} \text{ kg/m-s}$, respectively. The linear systems associated with the solution of the pressure and momentum equations have been solved with a tolerance on the residual to 10^{-6} . The iterative procedure is declared converged when all residuals have been reduced below 10^{-6} . The computations were carried out on a PC computer work station, with FLUENT 6.0 Code and the CPU time was 2 days on average for the case with 241,551 computational grids. The generated unstructured grid consisting of tetrahedral elements is shown in Figure 13.

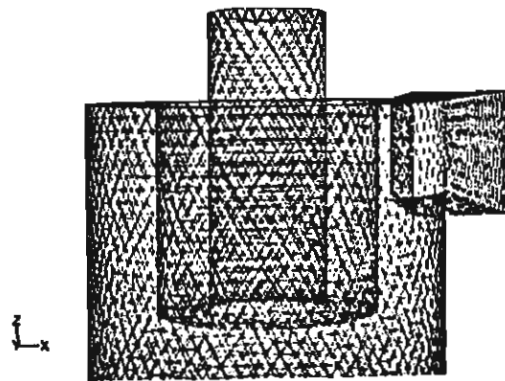


Figure 13. Unstructured grid consisting of tetrahedral elements within a hydrocyclone for the finite-volume approximation.

4.5. Numerical results

4.5.1. Flow characteristics

The flow characteristics of the flow within the hydrocyclone are obtained from the post-processing process of the predicted data of the flow pressure and velocity components. The schematic diagram shows the plane positions of 50-mm hydrocyclone in Figure 14.

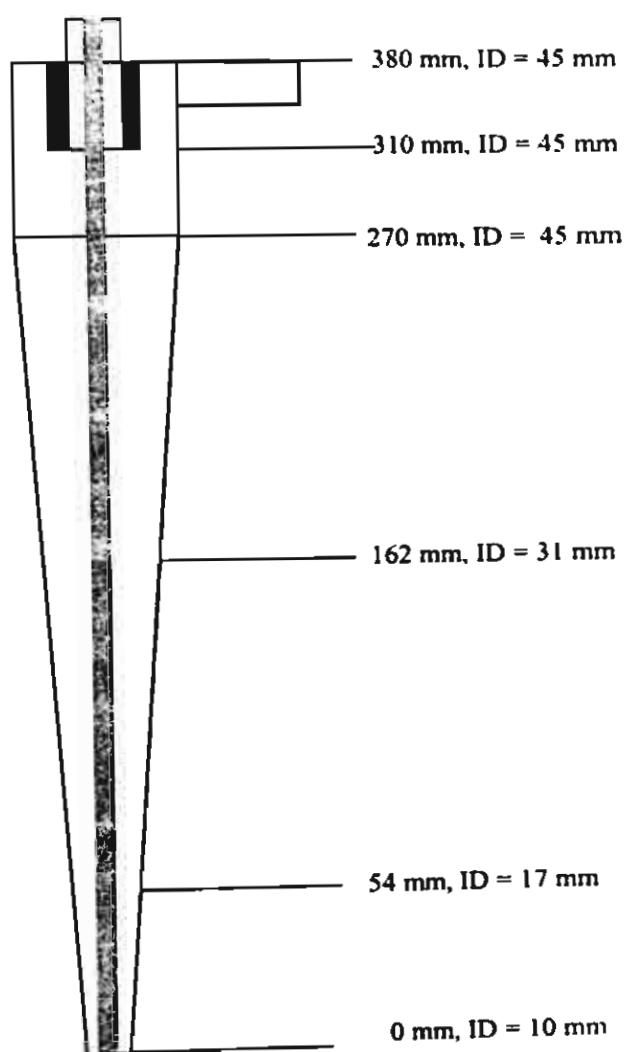


Figure 14. Schematic diagram shows the plane positions of 50-mm hydrocyclone with inserted-rod.

The obtained pressure of fluid projected on a vertical plane of the hydrocyclone operating with an air core and a 4-mm inserted-rod are shown in Figure 15. The numerical results

can depict the air core region, where the pressure is negative, in the centre of the hydrocyclone. The air-core characteristics obtained from this numerical simulation is similar to the experimental observation as can be seen in Figure 15. The air-core was found to be instable and its size, shape and position are unfixed because of the instability of the gas-liquid interface. It is also found that the region of the low pressure does not exist in the flow within the hydrocyclone operating with 4-mm inserted-rod. This is due to the inserting of the rod eliminating the occurrence of the air core.

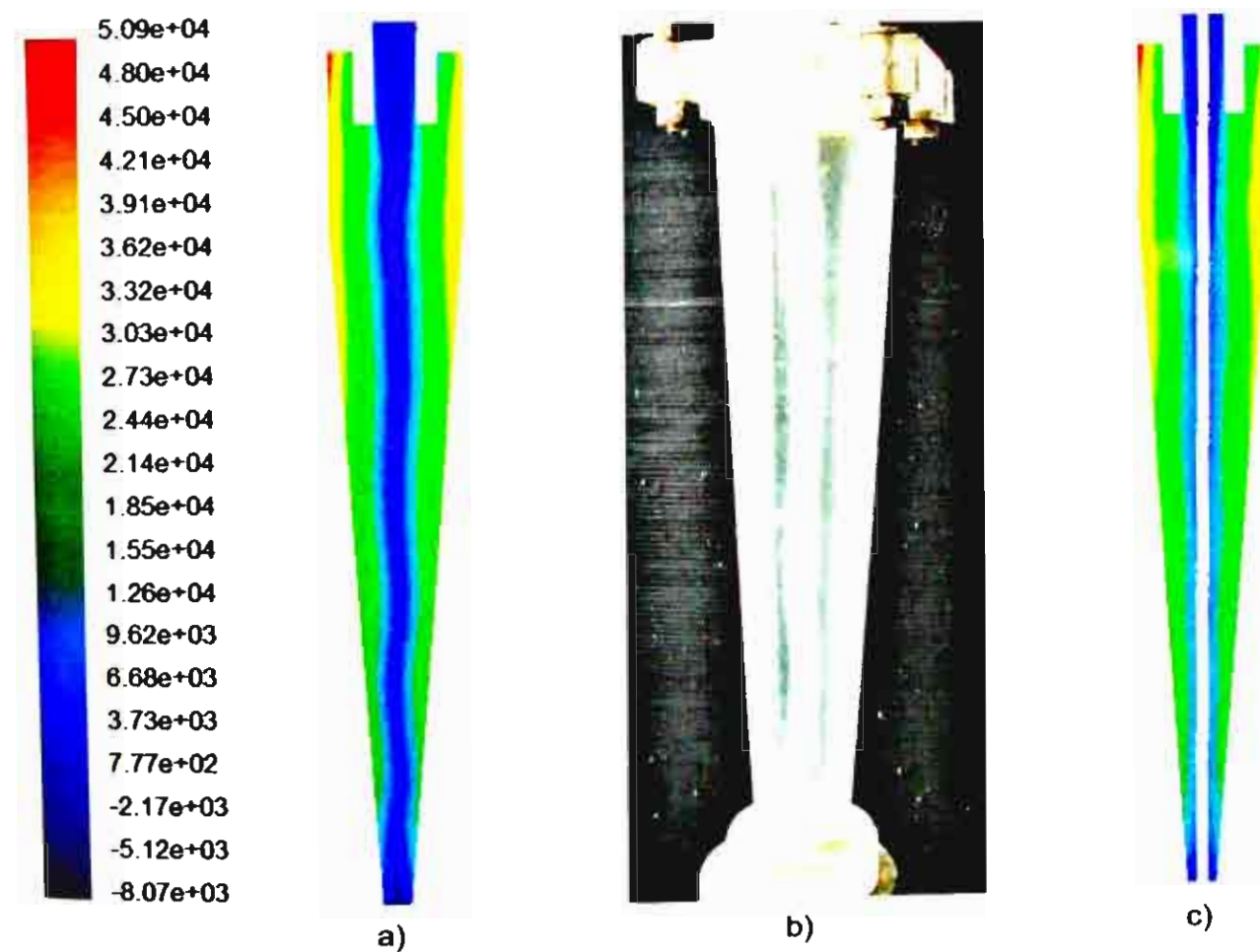


Figure 15. The obtained pressure (Pascal) of fluid projected on a vertical plane of the hydrocyclone: a) operating with an air core, b) experimental observation and c) with 4-mm inserted-rod.

The obtained pressure of fluid projected on a horizontal plane of the hydrocyclone operating with an air core, a 4 and 6-mm inserted-rod, are also shown in Figure 16.

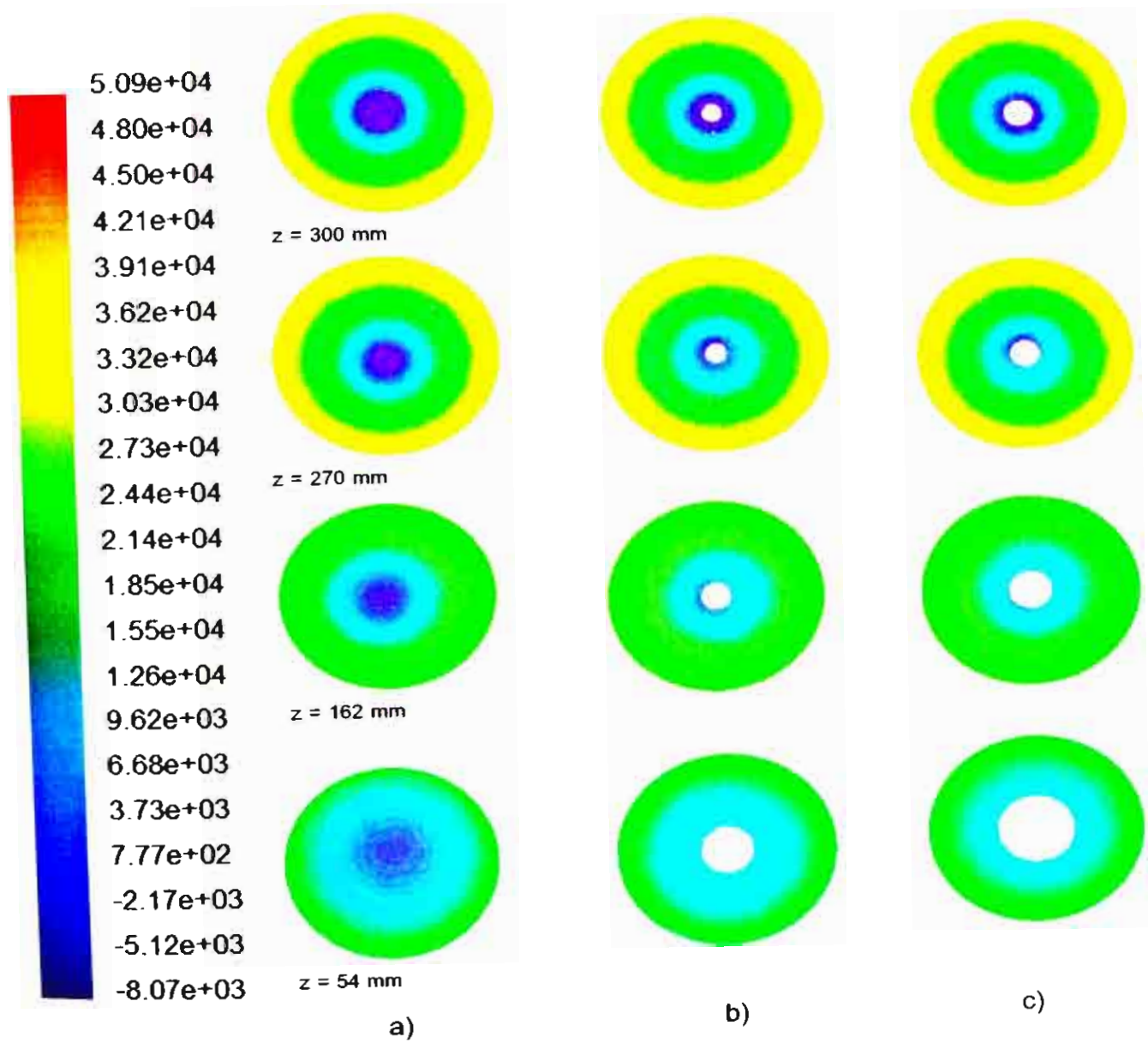


Figure 16. The obtained pressure (Pascal) of fluid projected on a horizontal plane of the hydrocyclone: a) operating with an air core, b with 4-mm inserted-rod and c) with 6-mm inserted-rod.

The tangential velocity is an important velocity component as it creates the high swirling field in hydrocyclone as can be seen in Figure 17. The fluid enters through the tangential inlet with high inlet velocity and its initially linear motion is converted to angular motion by the hydrocyclone. The numerical result demonstrates a double vortex pattern or two swirling flows, which are the outer downward and the inner upward flows. It can be seen that the flow within the hydrocyclone is not symmetric.

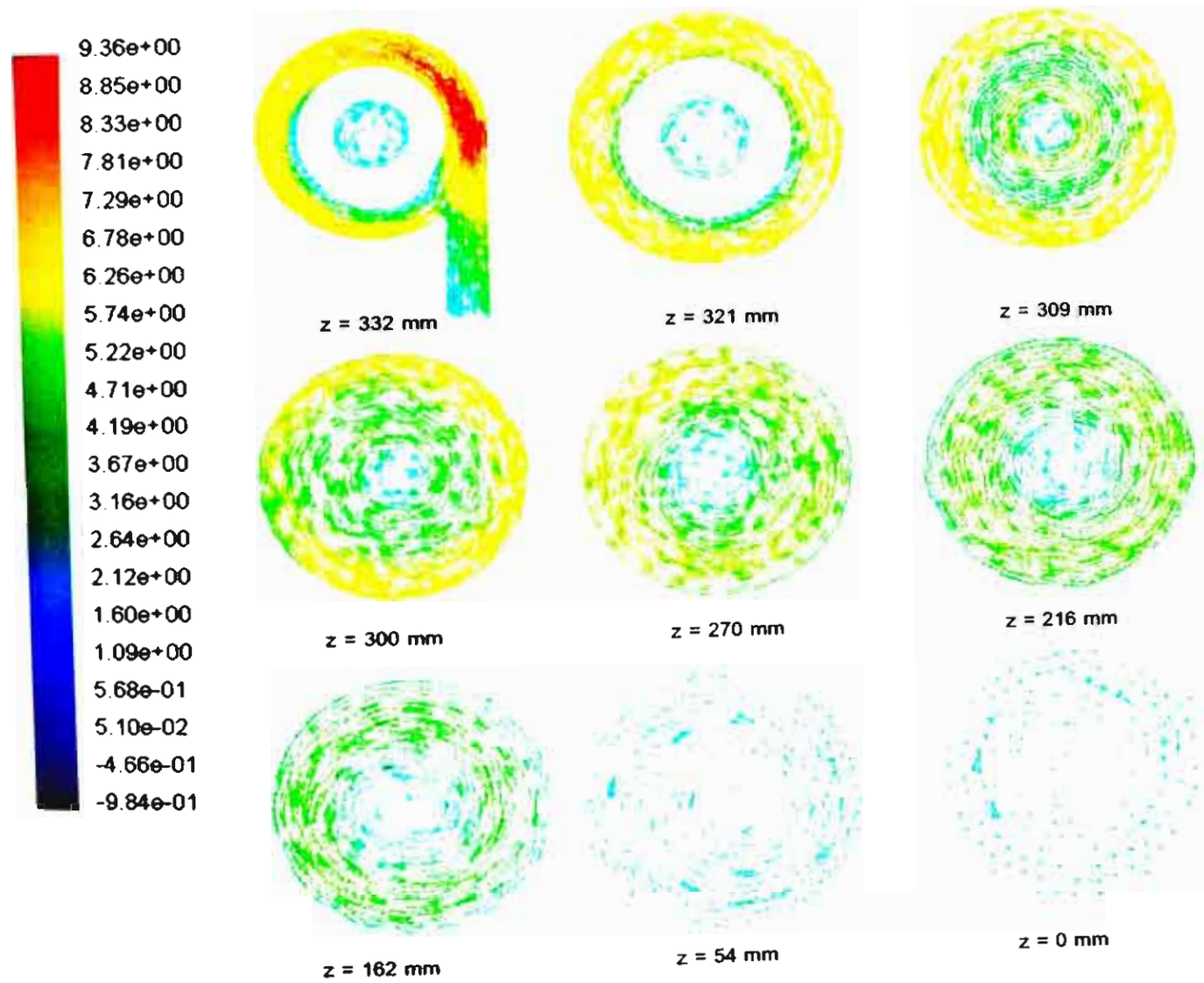


Figure 17. The obtained tangential velocity vector (m/s) of fluid projected on a horizontal plane for the inlet velocity of 7 m/s.

The separation flow is successfully simulated and can be demonstrated by the axial velocity component as shown in Figure 18 for both hydrocyclone operating with and without the inserted-rod. The positive and negative values of the axial velocity indicate the upward flow and downward flow, respectively. The inserted-rod causes a lower axial velocity at the underflow outlet area of the separator, because it reduces the underflow outlet area and this causes the reversed upward flow to increase. The typical high axial velocity at the overflow area is observed.

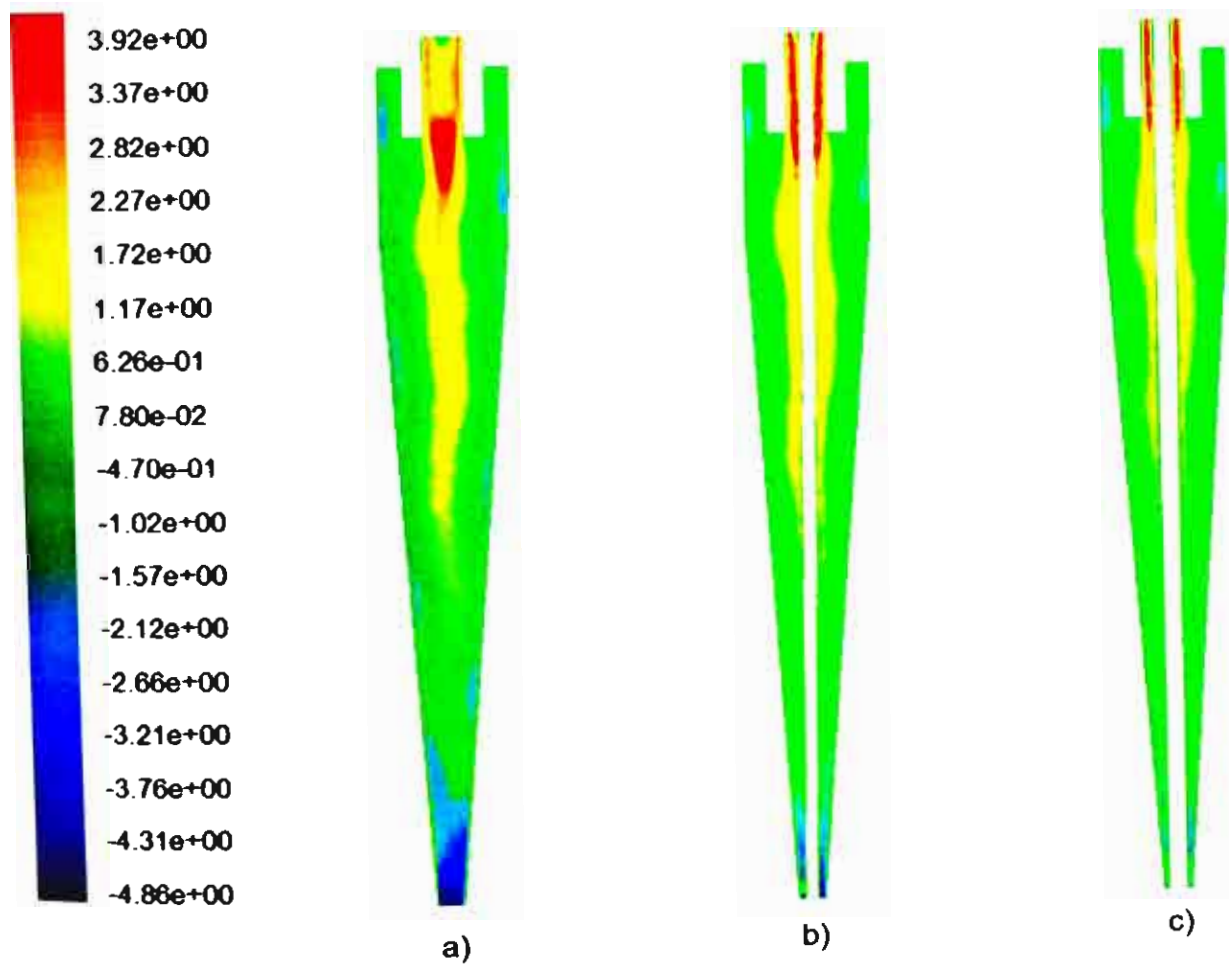


Figure 18. The obtained axial velocity (m/s) of fluid projected on a vertical plane of the hydrocyclone: a) operating with an air core, b) with 4-mm inserted-rod and c) with 6-mm inserted-rod.

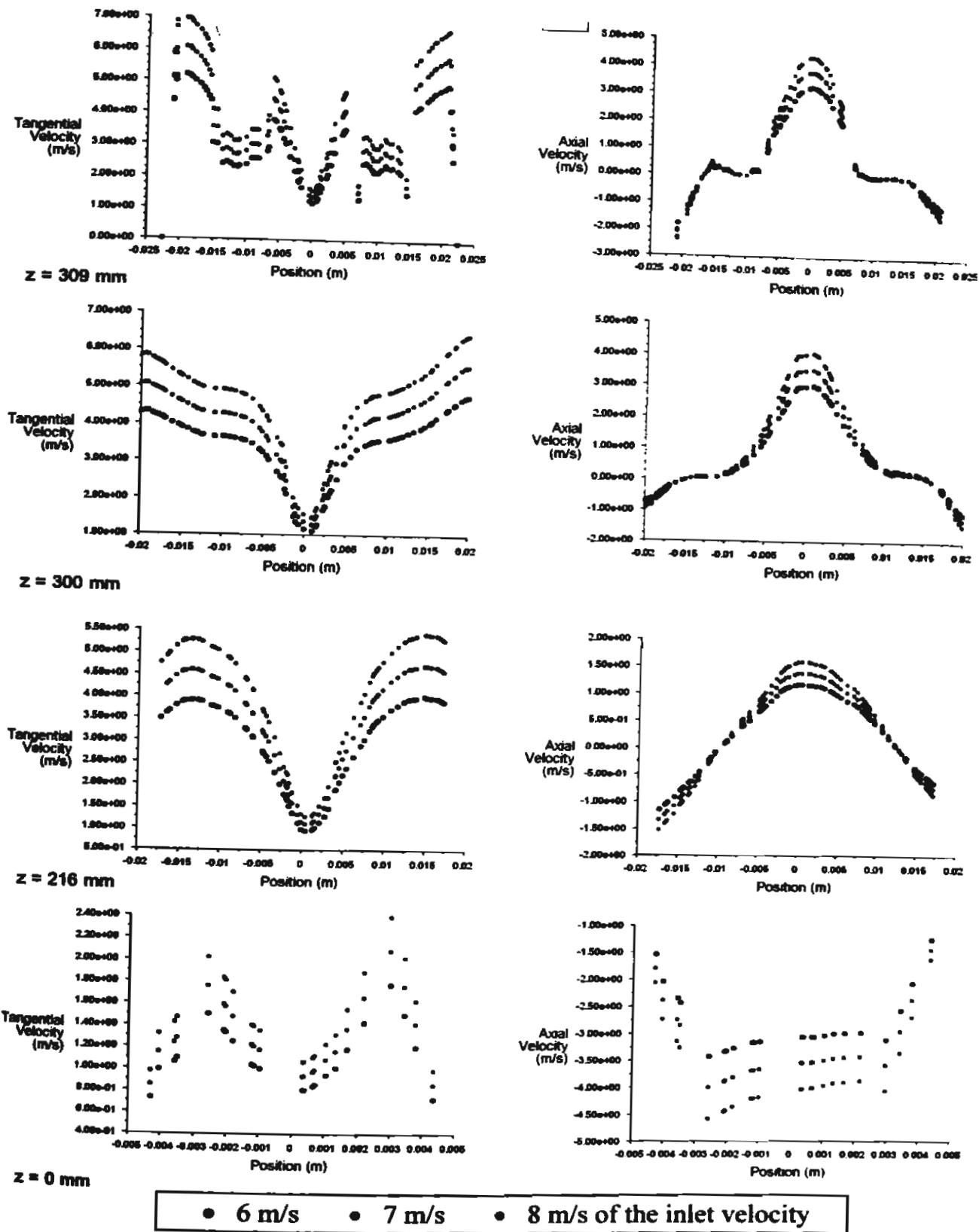


Figure 19. The axial and tangential velocity profiles (m/s) of fluid on a horizontal plane of the hydrocyclone, operating with an air core, for the inlet velocity of 6, 7 and 8 m/s.

From Figure 19, it can be seen that the profiles of both axial and tangential velocity components of the higher inlet velocity case follow the same pattern as that of the lower inlet velocity case. However, the magnitude of both velocity components increases with an increase in the inlet velocity. The axial velocity profiles prove the existence of the LZVV (the location where the zero axial velocity exists), where the separated flow occurs in hydrocyclone. At the plane $z = 309$, which located in the bottom of the vortex finder tip, the tangential velocity profile is not found to decrease towards the centre like the profile at the plane $z = 300$. This is due to the effect of the upward flow entering the vortex finder tube. The tangential velocity, in the conical section at the plane $z = 216$, tends to increase with decreasing radius until it reaches a maximum at the some point. At radial distances less than this point, the velocity decreases proportionally with the radius. This demonstrates two different zones of forced vortex flow. The tangential velocity profile at the underflow outlet shows the two zones of the swirling flow, while the axial velocity profile demonstrates the reduction of the velocity in the centre caused by the external pressure force.

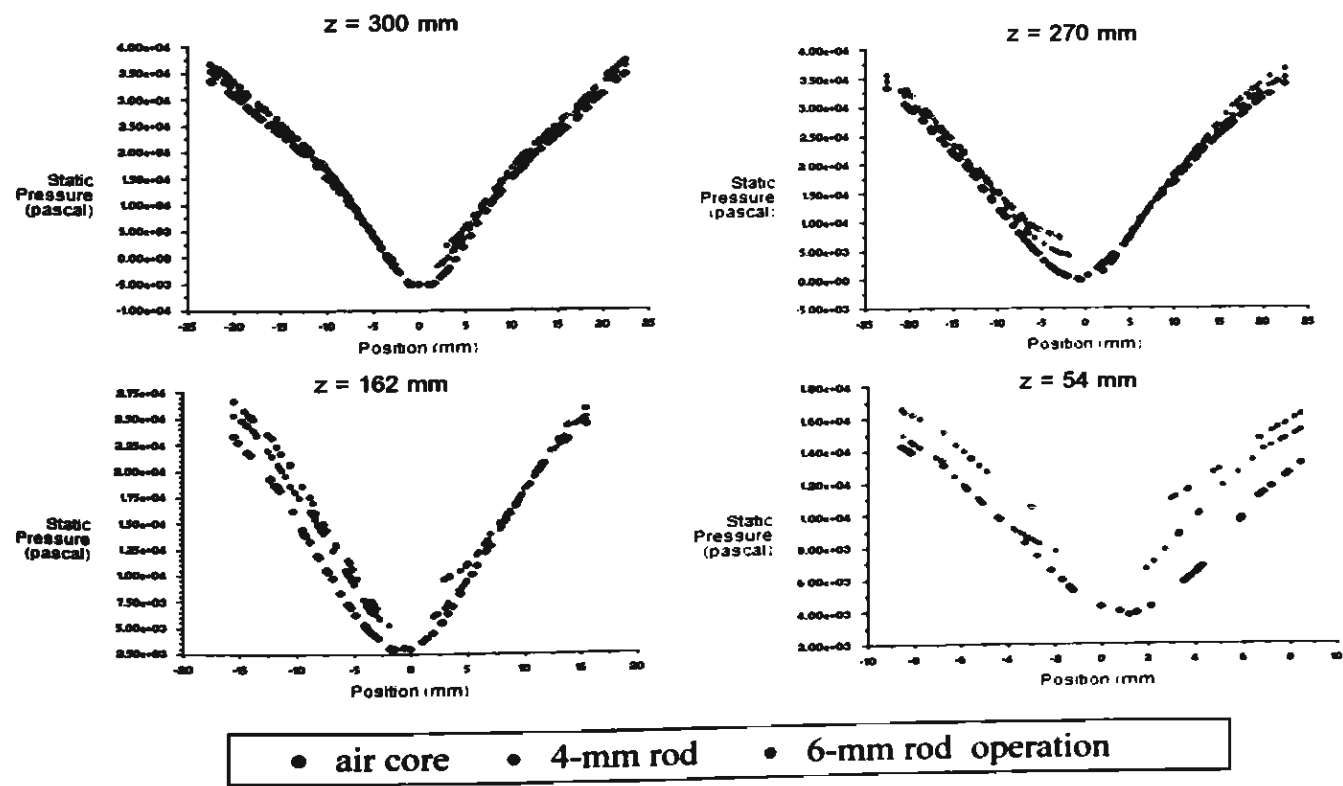


Figure 20. The profile of static pressure (Pascal) of fluid on a horizontal plane of the hydrocyclone operating with: an air-core, 4-mm rod and 6-mm rod.

The pressure is an important parameter of hydrocyclone operation. High pressure drop in operation means high energy loss of the flow and high pumping cost. As the result of this numerical simulation, the reduction of pressure loss was found to be reduced by an inserted-rod as shown in Figure 20. The hydrocyclone operating without the inserted-rod has lower pressure field that that of the operating with inserted-rod. The 6-mm rod can give the lowest pressure field.

The axial velocity profile projected on a horizontal plane of the hydrocyclone operating with an air core, a 4 and 6-mm inserted-rod, comparing with the experimental data are shown in Figure 21.

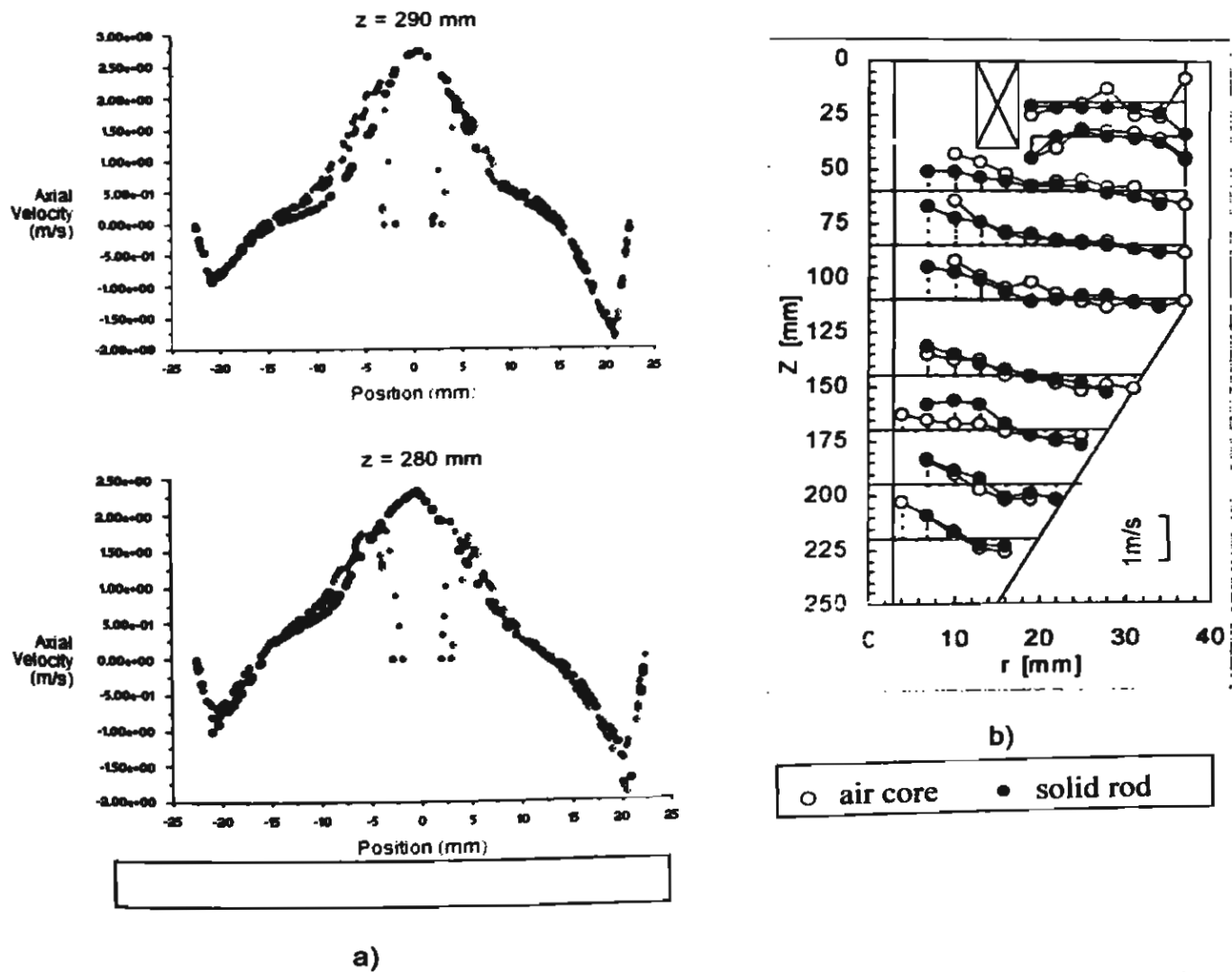


Figure 21. The axial velocity profiles (m/s) of fluid on a horizontal plane of the hydrocyclone: a) operating with: an air-core, 4-mm rod and 6-mm rod and b) experimental data obtained by Chu et al. (2004).

From the numerical result and the experimental data of Chu et al. (2004), the upward axial velocity of the fluid in the central area decreased when the air core was replaced by inserted-rod. The decrease of the axial velocity of fluid nearby the entrance of the vortex finder is beneficial to reducing the mixing of coarse particles in overflow product. In Figure 22, it can be seen that the radial velocity distribution of the flow operating with and without the inserted-rod are similar but the magnitude of the radial velocity of the inserted-rod cases is obviously smaller. The reduction in radial velocity is advantageous for separating fine particles because the drag forces, which acting towards the center, are reduced. The smallest magnitude of the radial velocity was obtained from the flow with the 4-mm rod, which the best separation performance was found as discussed in the next section.

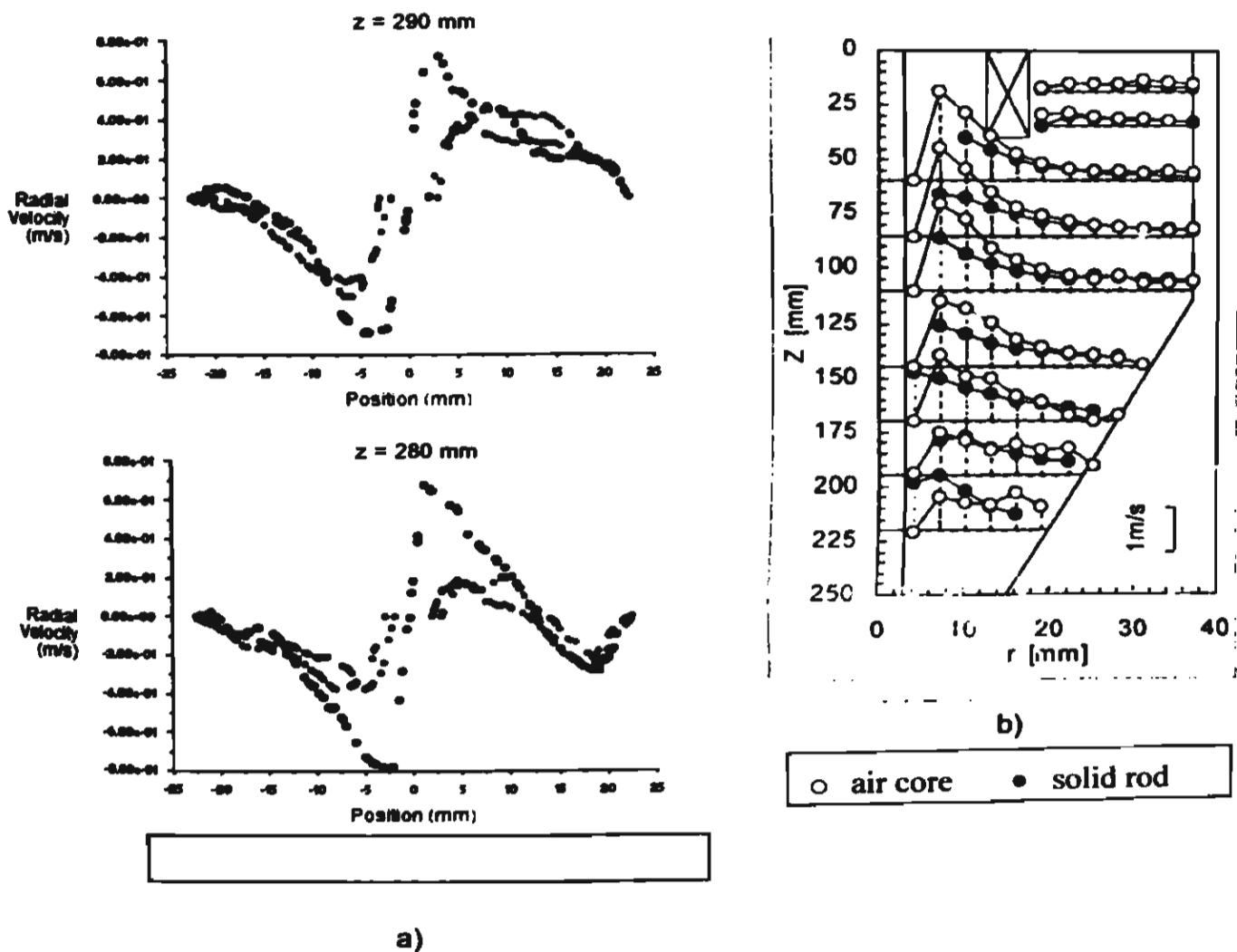


Figure 22. The radial velocity profiles (m/s) of fluid on a horizontal plane of the hydrocyclone: a) operating with: an air-core, 4-mm rod and 6-mm rod and b) experimental data obtained by Chu et al. (2004).

and the decrease in cut size tend to decline above a certain pressure drop. This is thought to be due to resistance effects within the hydrocyclone.

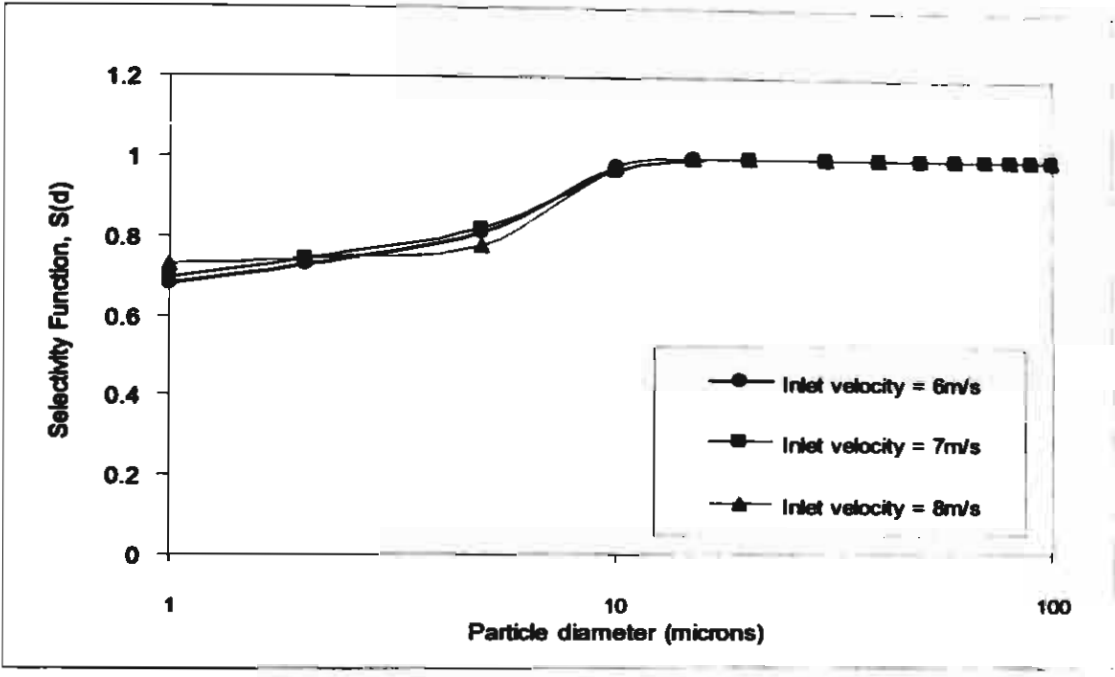


Figure 25. The selectivity curves, presenting the predicted separation performance, of the flow with an air core for different inlet velocities.

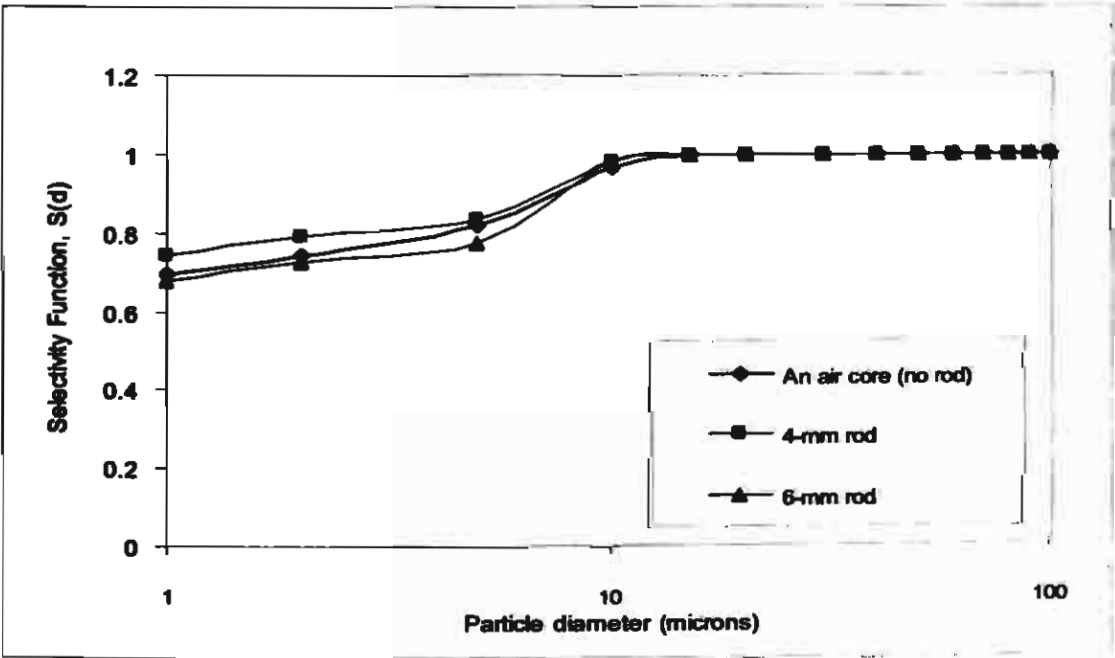


Figure 26. The selectivity curves, presenting the predicted separation performance, of the flow with an air core (no rod) and 4 and 6-mm inserted rods.

The selectivity curves for the hydrocyclone operating with 4 and 6-mm inserted-rods comparing with that of operating without inserted-rod are presented in Figure 26. It can be seen that the hydrocyclone operating with the 4-mm inserted- rod can gives the best separation performance, while the operating with the 6-mm inserted-rod gives the worst separation performance. It can be concluded that the hydrocyclone separation performance can be improved by replacing the air core with the proper size of the inserted-rod. The negative effect on the separation found in the experimental study of Lee and Williams (1993) might be due to both the design of the inserted-rod body supports and the size of inserted-rod.

4.6. Conclusion

In conclusion, the Computational Fluid Dynamics (CFD) techniques to the simulation of 3D flow within a hydrocyclone operating with an air-core and with inserted-rod by using the Finite Volume Method (FVM) has been introduced. The particle motion was successfully predicted by using the particle trajectory method. The separation performance of the separator was determined by the relationship between percentages of the each particle size of feed reporting to the underflow discharge. The numerical results from this study were in good agreement with the experimental data. It was found that the radial and axial velocity components in the area that just below the vortex finder were reduced, when replacing the air core by inserting a metal rod. This causes the flow field inside hydrocyclone become more beneficial for the separation process. However, the separation performance can only be improved with the proper size of the inserted-rod. The reduction of pressure loss in the hydrocyclone was also found to be reduced by replacing the air core by inserting a metal rod.

5. References

- Chu, L., Yu, W., Wang, G., Zhou, X., Chen, W. and Dai, G. (2004) Enhancement of hydrocyclone separation performance by eliminating the air core, *Chemical Engineering and Processing*, Vol. 43, Issue 12, pp. 1441-1448.
- Concha, F., Barrientos, A., Montero, J., Sampaio, R. (1996) Air core and roping in hydro cyclones. *International Journal of Mineral Processing*, 44-45, 743-749.

- Fisher, M., Flack, R. (2002)** Velocity distributions in a hydrocyclone separator. *Experiments in Fluids*, 32, 302-312.
- Gresho, P. (1991)** Incompressible fluid dynamics: Some fundamental issues. *Annual Review of Fluid Mechanics*, 23, 413-453.
- Haroutunian, V., Engelman, M. S. and Hasbani, I. (1993)** Segregated finite element algorithms for the numerical solution of large-scale incompressible flow problems, *International Journal for Numerical Methods in Fluids*, Vol. 17, pp. 323-348.
- Lee, M. S. and Williams, R. A. (1993)** Performance characteristics within a modified hydrocyclones, *Minerals Engineering*, Vol. 6, No. 7, pp. 743-751.
- Luo, Q., Deng, C., Xu, J. R., Yu, L. and Xiong, G. (1989)** Comparison of the performance of the water sealed and commercial hydrocyclones, *International Journal of Mineral Processing*, Vol. 25, pp. 297-310.
- Neesse, T., Schneider, M., Dueck, J., Golyk, V., Buntenbach, S., Tiefel, H. (2003)** Hydrocyclone operation at the transition point rope/spray discharge. In: *Hydrocyclones 2004*, in South Africa.
- Patankar, S. V. (1980)** Numerical heat transfer and fluid flow, New York: Hemisphere Publishing Corporation, Taylor and Francis Group.
- Xu, J., Luo, Q. and Qiu, J. (1990a)** Studying the flow field in a hydrocyclone with no forced vortex, Part I - average velocity, *Filtration and Separation*, Vol. 27, pp. 181-182.
- Xu, J., Luo, Q. and Qiu, J. (1990b)** Studying the flow field in a hydrocyclone with no forced vortex, Part II- average velocity, *Filtration and Separation*, Vol. 27, pp. 356-359.

III. The experimental study of the flow within a hydrocyclone operating with an air core

1. Abstract

The experimental studies of the flow behaviour within 50-mm hydrocyclones with and without a vortex finder attachment (Fin) have been performed. The flow characteristics within both hydrocyclones were observed visually and recorded both photographically and on video as the hydrocyclones are made of a transparent material. The development of the double-helical flow was observed; also the shape and diameter of the air-core have been examined for different values of operating pressure drop. It was found that the air-core is unstable and its size, shape and position are unfixed because of the instability of the gas-liquid interface. The results also showed that the air core is related to the shape of the underflow discharge and is a function of the pressure drop. A regular spray discharge from the spigot was observed. The roping discharge was observed when an extension pipe was connected to the underflow outlet. An increase in pressure drop leads to an increase in air-core diameter and the angle of the underflow profile. However, the relationships between these three parameters are non-linear. The knowledge of the size and location of the air-core relative to the operating pressure drop can assist in the validation of computational fluid dynamics (CFD) simulation work.

The separation performance of these two transparent hydrocyclones for a CaCO_3 in water system has been investigated. In this study, the results showed that a fin-type vortex finder attachment could not increase the sharpness of separation but it could help in the thickening process. It can be seen that the separation performance does not depend only on the geometry and operating conditions, but also on slurry concentration and particle size distribution. The concentration of the feed slurry and the operating pressure drop strongly affect the sharpness of separation.

2. Introduction

Hydrocyclones are widely used in the mineral, chemical, coal, powder-processing industries for de-watering, or classifying according to a required particle size or density separation.

The reasons for this popularity lie in their simplicity of design and operation, high throughput, low maintenance, low operating cost and small physical size of the unit. Hydrocyclones are physically simple and robust separation devices with no moveable parts. A typical hydrocyclone consists of three sections; a cylindrical section, conical section and a tangential inlet section. The tangential inlet section is attached to the cylindrical section. The vortex finder is located centrally through the lid and at the top of the cylindrical section. Figure 1a shows a conventional hydrocyclone, which is inherently simple in construction. The design variables which influence the performance of the unit operation are the inlet and outlet diameters. The separation process is achieved as a result of the centrifugal force acting on the slurry and its components. However, there is a secondary flow down the outside wall of the vortex finder to the overflow, which is called a short-circuit flow. This flow was reported to cause low particle separation efficiency (Kelsall, 1952). Svarovsky (1994) suggested that a redesigned-vortex finder with a skirt can eliminate this inefficiency problem.

Despite its apparent simplicity, the elucidation of the mechanism of separation and detailed models of fluid flow with hydrocyclone separators has yet to be achieved. The main reasons for these short comings are: the complexity of solving Navier-Stokes equations for non-Newtonian anisotropic fluids that contain high concentrations of interacting particles, The complex high swirling and varying turbulence conditions prevailing in the separator, the presence of an interface (the air-core) inside the separator and the lack of on-line measurement methods to provide reliable experimental data for model development under credible process conditions (Bond et al., 1999).

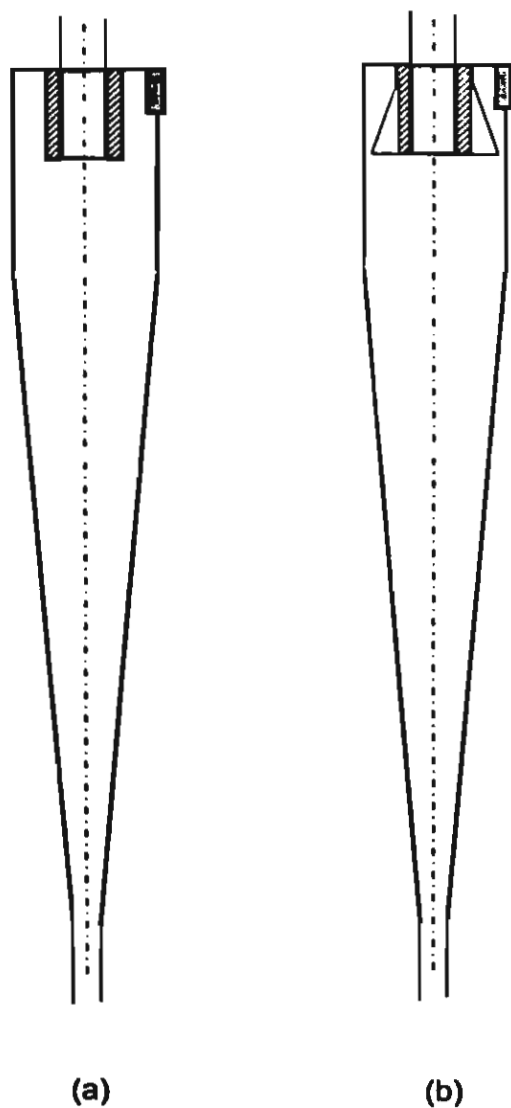
An attempt to model the flow in a hydrocyclone has been made by using a theoretical model, which is based on solving the Navier-Stokes equations. This approach provides a physical insight into the fundamental causes of the observed phenomena. The earlier theoretical models were presented by Bloor and Ingham (1973), (1975) and (1987), Pericleous and Rhodes (1986), and Hsieh and Rajamani (1991). These steady state and 2D-axisymmetrical models have been considered, but the results are limited to dilute flow only. In addition, it is difficult to describe the behaviour of high turbulent swirling flow caused by the 3D flow entry. Therefore, more advanced modelling is needed that allows,

for example, the study of such phenomena as an adjustment of three-dimensional flow to axisymmetrical, particle-fluid, particle-particle and particle-wall interactions. Such models will allow the description of particle effects on suppressing or generating turbulence and non-Newtonian slurry flows. Additionally, in the context of modelling turbulence, a physical model is needed to show how a fluid turbulent deformation characterises swirl flows and the deformation of the air-core inside the separator. Advanced theoretical and experimental techniques are needed to obtain a better understanding of the complex physical phenomena affecting the performance of hydrocyclones.

The purpose of this work is to study the flow behaviour, focusing on the air-core and underflow discharge characteristics, by using 50-mm transparent hydrocyclones with and without a vortex finder attachment (Fin). The experiments of these hydrocyclones treating CaCO_3 in water system are carried out. The knowledge of the underflow discharge characteristics relative to the operating pressure drop and feed property affecting the separation performance is investigated.

3. Experimental systems

Figure 1 shows the schematic diagram of both 50-mm diameter acrylic hydrocyclones with and without a vortex finder attachment. They were equipped and set up with a feed pump and pressure gauge to measure the feed inlet pressure (see Figure 2). The vortex finder and apex diameters were kept constant at 14mm and 10 mm. Hydrocyclone overflows and underflows were directed back to the sump for recirculation. To study the flow behaviour and the characteristics of the air core, water was used. The operating pressure drop was varied from 10 psi to 35 psi by varying the inlet water flow rate. The density and the viscosity of water are 1031 kg/m^3 and 1.330 cp, respectively. The size and the location of the air core were observed relative to the pressure drop. From each experiment, samples were collected from the feed, overflow and underflow streams. The collected samples were weighed and mass flow rates determined.



Geometry	Value (mm)
underflow internal diameter	10
vortex finder internal diameter	14
vortex finder outer diameter overflow	28
cylindrical section diameter	45
conical section length	270
cylindrical section length	40
width of inlet orifice	22
length of inlet orifice	40
height of inlet orifice	16
length of fin	6
height of fin	20

Figure 1. Schematic diagram of 50-mm diameter acrylic hydrocyclones (a) conventional hydrocyclone and (b) with a vortex finder attachment (Fin).

The separation performance was examined by using the solution of CaCO_3 in water as feed slurry. The density of CaCO_3 is 2700 kg/m^3 . The flow rate and concentration of the feed was varied. In this case, the solid phase in each stream was also weighed to determine the concentration. The particle size distribution of each stream was measured by using the laser-diffraction size-analysis technique.

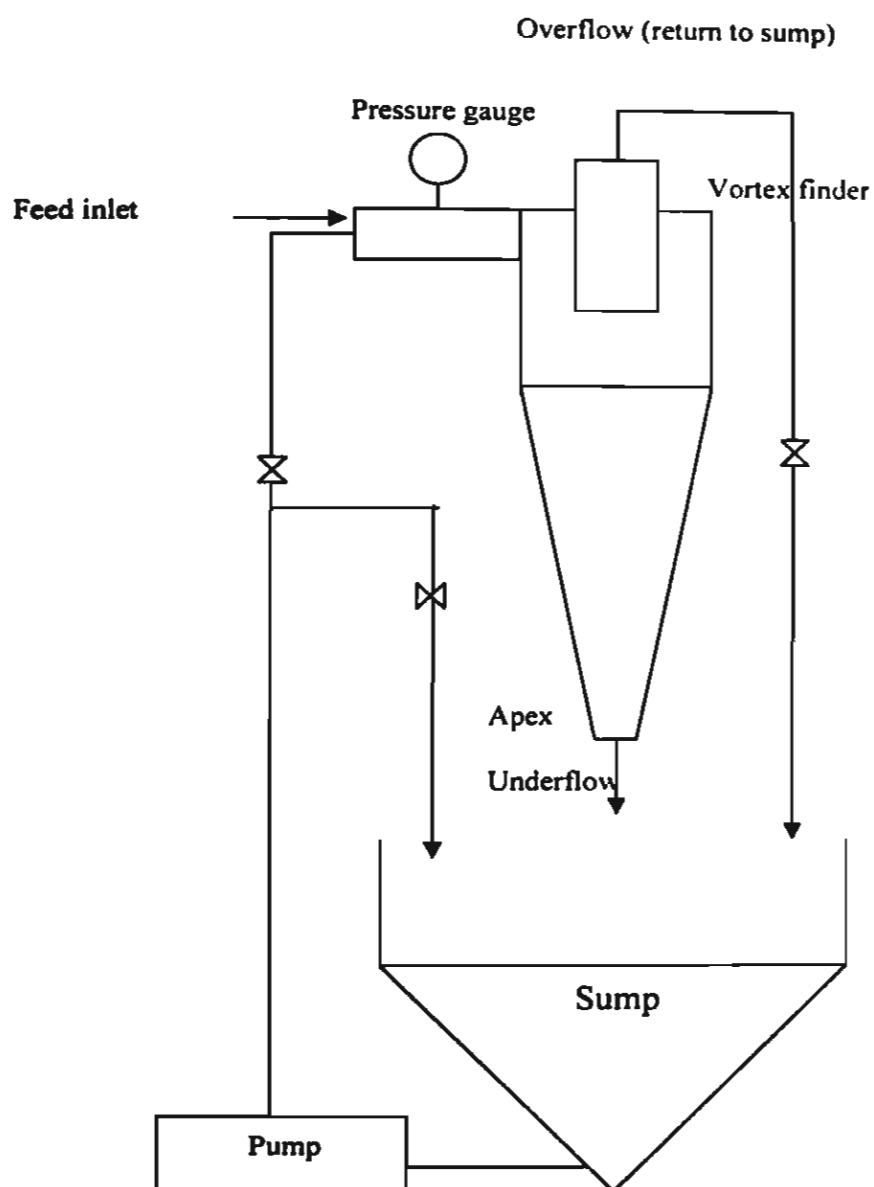
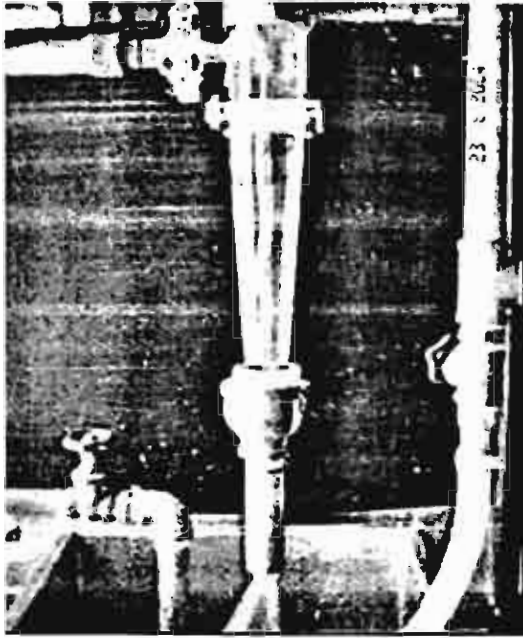


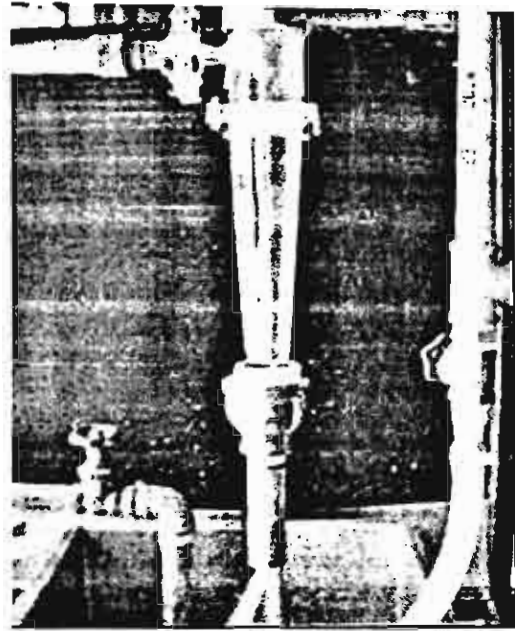
Figure 2. Hydrocyclone apparatus.

4. Flow behaviours

The diameter of the air-core of a conventional hydrocyclone for water-only flow was measured and found to be increased due to an increase in the pressure drop. It is also related to the angle of the underflow profile.

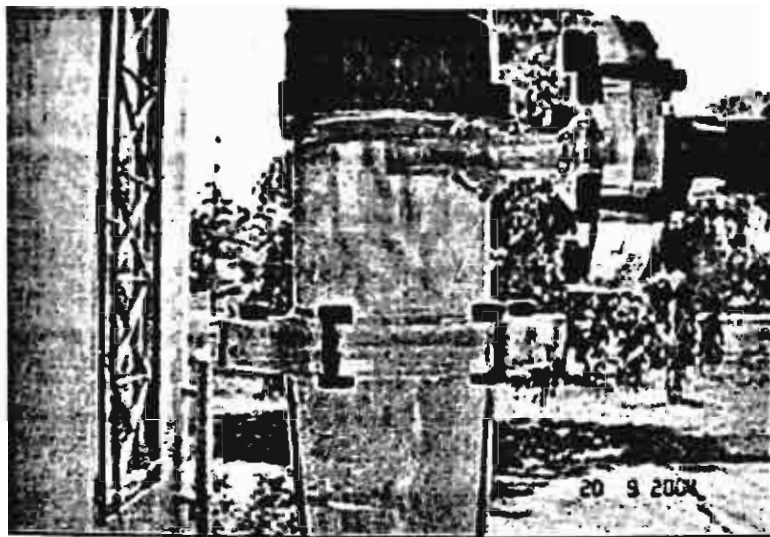


(a)

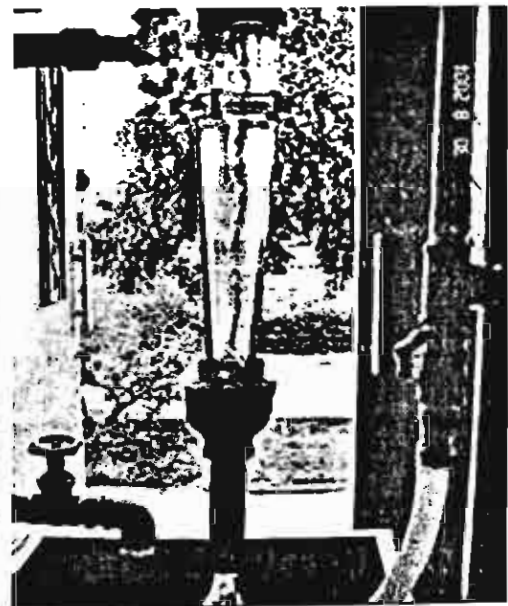


(b)

Figure 3. The flow pattern within the conventional hydrocyclone operating at: a) 10 psi b) 30 psi.



(a)



(b)

Figure 4. The flow pattern within the hydrocyclone with a vortex finder attachment operating at 10 psi.

In addition, the angle of the underflow discharge of a conventional hydrocyclone treating CaCO_3 slurries was observed for 1%, 2% and 5% by volume of solid in feed as shown in Figure 5.

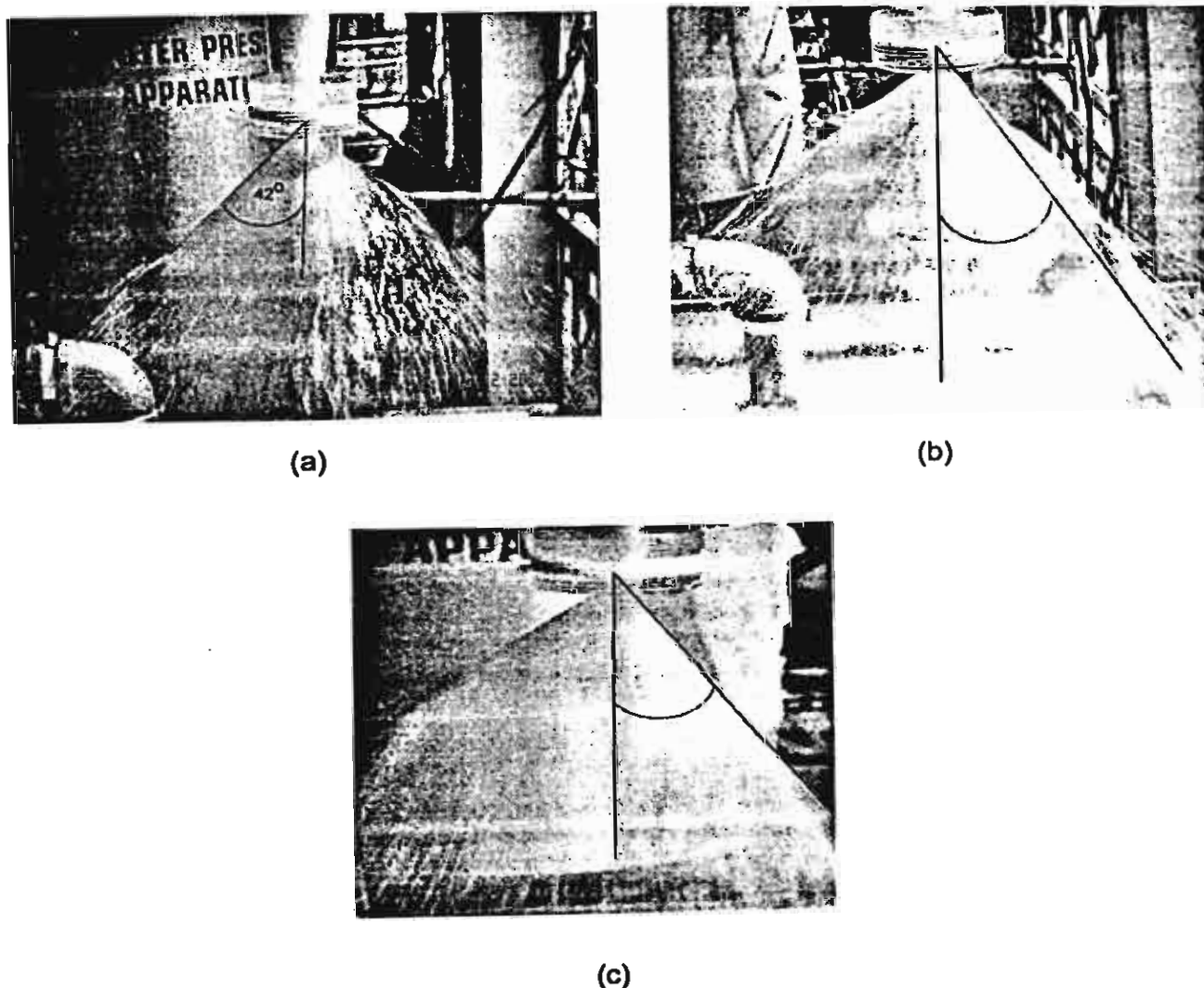


Figure 5. The underflow discharge of the hydrocyclone operating at 30 psi for feed solid concentrations of: a) 1% b) 2% and c) 5% by volume.

A regular spray discharge from the spigot was observed in every experiment. The shape of the underflow discharge was found to be altered according to the change of feed-solid concentration and operating pressure drop as shown in Figure 6. Figure 6 also depicts the relationships between the air-core diameter, the angle of the underflow discharge and the pressure drop. It was found that an increase in pressure drop leads to an increase in the air-core diameter. However, in low-solid concentration cases, which are 1 and 2% by

volume of solid in feed, it was found that an increase in pressure drop leads to an increase in the angle of the underflow profile. For higher solid concentration, which is 5% by volume, it was found that above a certain operating pressure (25 psi), the discharge angle decreases due to the increase in pressure drop. This is due to the sedimentation of solids in the underflow section caused by high operating pressure in dense-flow separation.

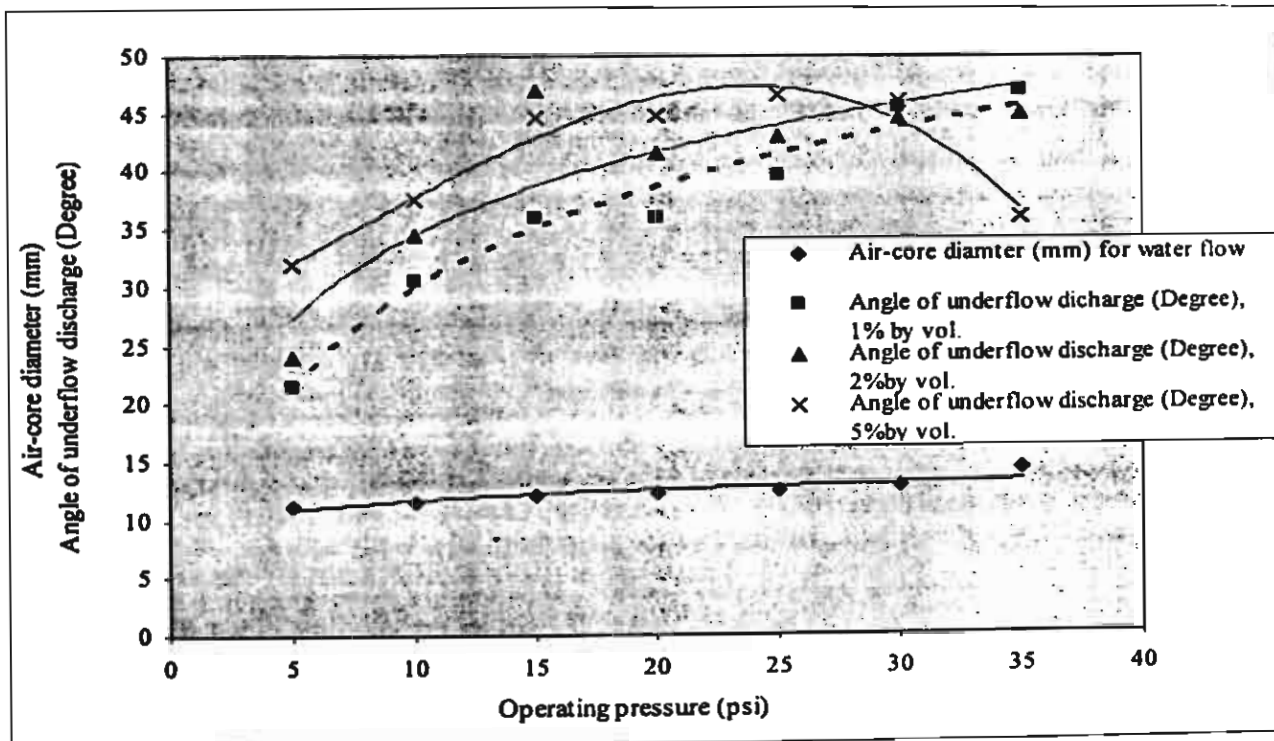


Figure 6. The relationship between the pressure drop, the air-core diameter and the angle of underflow discharge.

The solid concentration has a very strong effect on the angle of underflow discharge. The relationships between these three parameters are non-linear.

5. Separation performance

The separation performance of this hydrocyclone with the extension pipe, treating CaCO_3 slurry, was examined. The operational data (pressure drop), the concentration and flow rate of each stream, the cut size and throughput ratio (R_t) of each test are shown in Table 1.

The throughput ratio, R_r , is the ratio between the volumetric flow rate of the underflow and that of the feed. The particle size distribution of the feed is shown in Figure 7. The selectivity curves are presented in Figures 8-11.

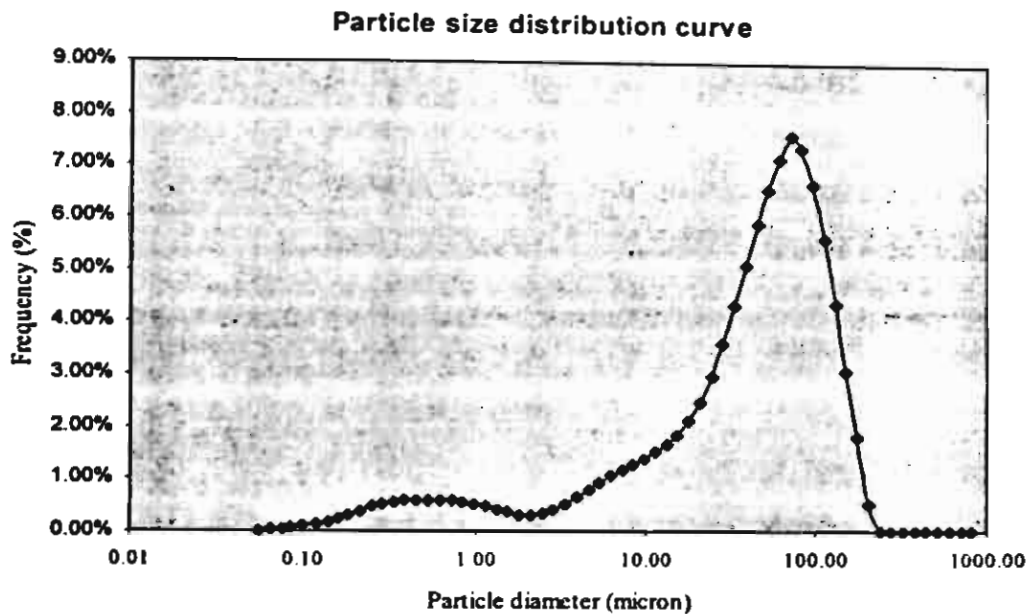


Figure 7. Particle size distribution curves.

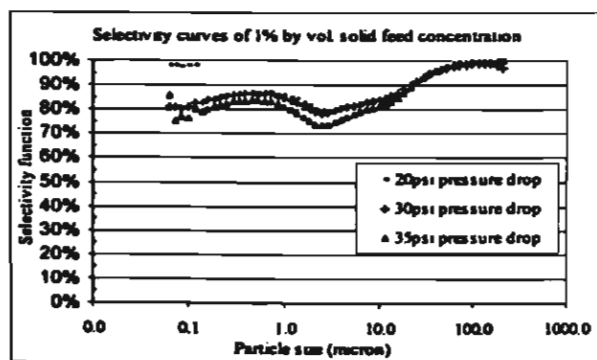
In this study, the operating pressure is limited. The highest pressure drop is 35 psi. In this range, it was found that the performance of the 50-mm hydrocyclone unit treating CaCO_3 slurry at 1 and 2% by volume was poor. The cut size cannot be obtained as can be seen in Figure 8a and 8b. However, the solid concentration in the underflow was three times higher than the concentration in the feed. The separation performance of the hydrocyclone treating the feed in this concentration range could be improved for a higher pressure drop.

The selectivity curves of the operation at the feed concentration of 5% by volume are shown in Figure 9. The separation performance of the hydrocyclone on this feed solid concentration is good. The results showed that the operating pressure of 35psi gave the smallest cut size. It is shown that the change in pressure drop obviously affects the hydrocyclone separation performance since an increase in pressure drop increases all velocities throughout the hydrocyclone. Therefore, an increase in pressure drop causes a decrease in the cut size.

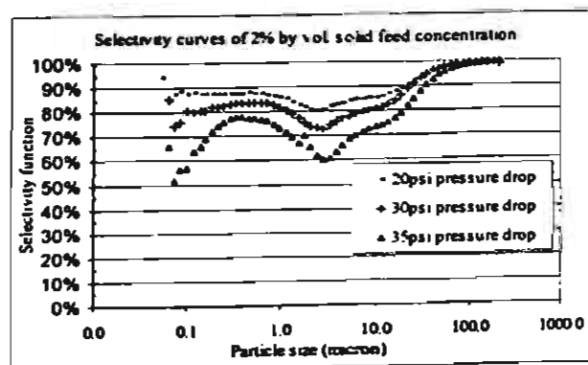
Table 1. Experimental results.

Test No.	Pressure Drop Δp (psi)	Feed		Over flow		Under flow		d_{50} (μm)	R_f
		% by vol.	(gal/min)	% by vol.	(gal/min)	% by vol.	(gal/min)		
I	20	1	8.98	0.096	5.38	2.548	3.60	-	0.40
II	30	1	10.89	0.133	6.68	3.559	4.21	-	0.38
III	35	1	11.74	0.107	7.35	3.362	4.39	-	0.37
IV	20	2	8.64	0.207	5.08	5.088	3.56	-	0.41
V	30	2	10.53	0.218	6.54	6.321	3.99	-	0.38
VII	35	2	11.81	0.216	7.42	3.632	4.39	-	0.37
VII	20	5	8.47	3.030	4.91	11.340	3.56	40	0.42
VIII	30	5	11.28	3.415	6.93	14.163	4.35	11	0.38
IX	35	5	11.56	0.978	7.21	8.200	4.35	2	0.37
X*	30	0.20	9.62	3.74	3.15	3.95	6.47	-	0.67
XI*	30	0.37	9.32	0.78	1.45	3.92	7.87	-	0.84
XII*	35	1	10.01	0.22	3.62	0.87	6.39	-	0.64
XIII*	35	2	10.61	0.38	3.95	1.43	6.66	-	0.63

* Hydrocyclone with vortex finder attachment (Fin).



(a)



(b)

Figure 8. Selectivity curves of the feed solid concentration: a) 1% and b) 2% by volume for 20, 30 and 35psi.

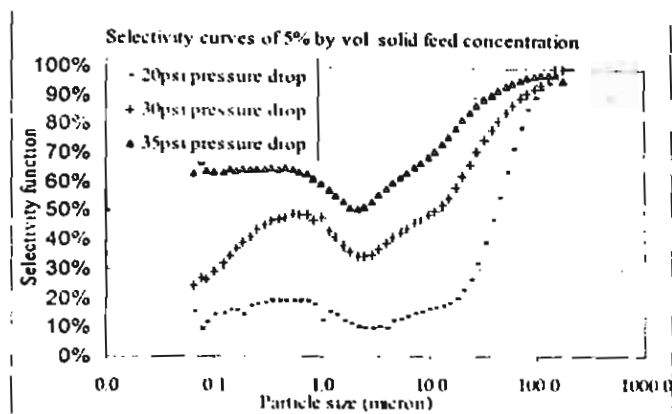


Figure 9. Selectivity curves of the feed solid concentration of 5% by volume for 20, 30 and 35psi.

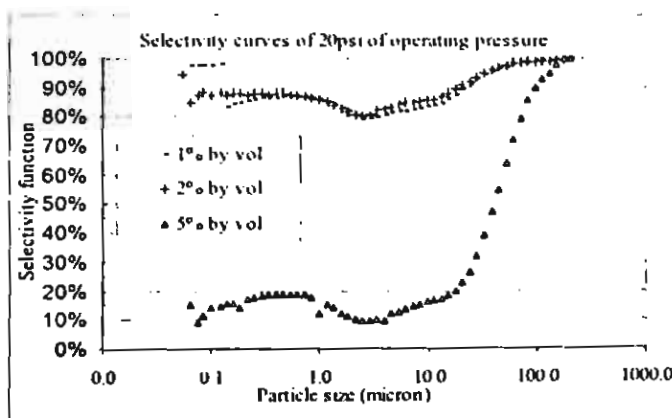
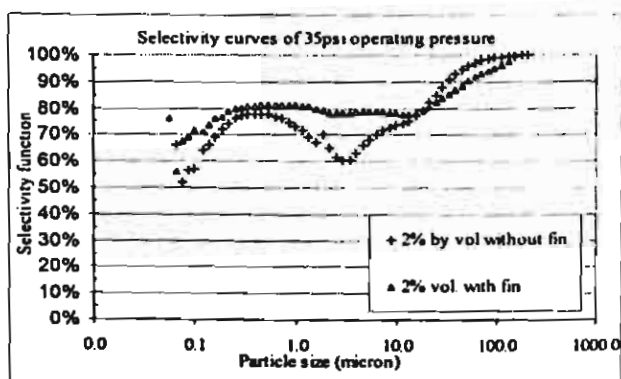
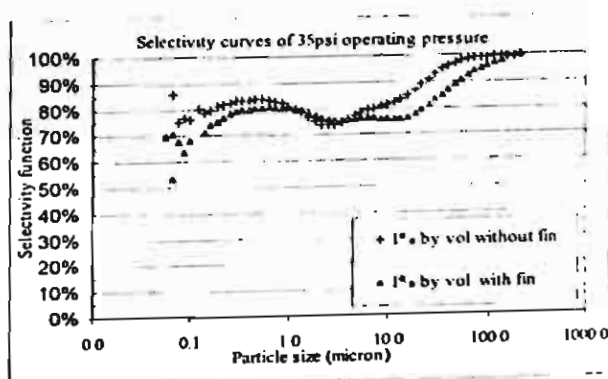


Figure 10. Selectivity curves of the operating pressure of 20 psi for 1, 2 and 5% by volume of solid in feed.



(a)



(b)

Figure 11. Selectivity curves of the operating pressure of 35 psi for: a) 1% and b) 2% by volume of solid in feed for the hydrocyclone operated with and without fin.

The pressure drop used in practice usually depends on economic considerations. Operating at high-pressure means less units are required to treat a given flow. Lower capital costs, finer cut sizes and sharper separations can be obtained. These benefits must be offset against drawbacks including higher pumping cost and increased abrasion. Furthermore, the increase in feed flow-rate and the decrease in cut size tend to decline above a certain pressure drop. This is thought to be due to resistance effects within the hydrocyclone (Bradley, 1965). More details about the effects of the feed properties and the operational conditions on the separation performance can be found in Kraipech (2002).

The cut size of the 50-mm hydrocyclone unit treating CaCO_3 slurry also strongly depends on the feed concentration. It also appeared that the angle of the underflow discharge at an operating pressure of 35 psi was the smallest angle compared to those at 20 and 30 psi. The discharge of the separator contains the information which could possibly be used for better operational control, assuming that two distinct operational states of the hydrocyclone exist, which are the following: spray and roping. The states differ by the discharge profile from the apex. Roping tends to form a rope-like discharge, while spray resembles an umbrella. Concha et al. (1996) points out that the best separation occurs near the formation of rope discharge. Similarly, Neesse et al. (2003) states on a broader note that a hydrocyclone achieves the best separation at a transitional discharge phase between roping and spraying. Therefore, with the calculation of the angle, determination of the operating performance of the hydrocyclone can be predicted. It is of interest to establish a procedure for improving the operation of hydrocyclones based on computational simulations.

A characteristic dip or fish-hook effect was observed in all experiments. It started to occur at particle sizes of approximately 2 microns. The analysis of this phenomenon is described in Kraipech (2002).

The separation performance of the hydrocyclone with the fin attachment was investigated for the feed concentration of 0.20, 0.37, 1 and 2% by volume. The results of the dilute feed (0.20 and 0.37% by volume) showed that the concentration of the underflow stream was ten to twenty times higher than that of the feed, but the selectivity curve showed low sharpness of separation and the cut size could not be obtained. The fin attachment causes an

increase in the throughput ratio. The results of the 1 and 2% by volume of the feed concentration showed very poor results as shown in Figure 11. The operation with 5% by volume of the feed slurry was also examined, and it was found that there was no overflow stream. This is due to the effect of the fin attachment forcing the flow swirling down to the apex tip and the high viscosity of the high concentration feed causing higher drag force, which reduces the effect of the centrifugal force. It can be summarised that the fin-type vortex finder attachment could not increase the sharpness of separation but it could help in the thickening process for a dilute feed system.

6. Conclusion

The results demonstrate that the air core is related to the type of underflow discharge and is a function of the pressure drop, which has an effect on the separation performance of the hydrocyclone. It can be seen that the separation performance does not depend only on the geometry and operating conditions, but also on slurry concentration. The results of this study are useful since they offer an ideal for improving hydrocyclone design and will assist in the validation of computational fluid dynamics (CFD) simulations.

7. References

- Bloor, M. I. And Ingham, D. B.** "Theoretical investigation of the flow in a conical", Transactions of the Institution of Chemical Engineers, Vol. 51, pp. 36-41, 1973.
- Bloor, M. I. And Ingham, D. B.** "Turbulent spin in a hydrocyclone", Transactions of the Institution of Chemical Engineers, Vol. 53, pp. 1-6, 1975.
- Bloor, M. I. And Ingham, D. B.** "The flow in industrial cyclones", J. Fluid Mech, Vol. 178, pp. 507-519, 1987.
- Bond, J., Cullivan, J. C., Climpson, N., Dyakowski, T., Faulks, I., Jia, X., Kostuch, J. A., Payton, D., Wang, M., Wang, S. J., West, R. M. and Williams, R. M.** "Industrial monitoring of hydrocyclone operation using Electrical Resistance Tomography", 1st World Congress on Industrial Process Tomography Buxton, Greater Manchester, April 14-17, 1999.
- Bradley, D.** "The Hydrocyclone", Oxford: Pergamon Press, 1965.
- Concha, F., Barrientos, A., Munoz, L., Bustamante, O. and Castro, O.** "A phenomenological model of a hydrocyclone", in Claxton, D., Svarovsky, L. and Thew, M.

(Eds), *Hydrocyclones '96*, London: Mechanical Engineering Publications Limited, pp. 63-82,1996.

Hsieh, K. T. and Rajamani, R. K. "Mathematical model of hydrocyclone based on physics of fluid flow", *AIChE Journal*, Vol. 37. No. 5, pp. 735-745,1991.

Kelsall, D. F. "A study of the motion of solid particles in a hydraulic cyclone", *Transactions of the Institution of Chemical Engineers*, Vol. 30, pp. 87-108, 1952.

Kraipech, W. "Studying performance of industrial hydrocyclones", Ph.D. Thesis, UMIST, UK, 2002.

Neesse, T., Schneider, M., Dueck, J., Golyk, V., Buntenbach, S., Tiefel, H. "Hydrocyclone operation at the transition point rope/spray discharge". In: *Hydrocyclones 2004 in South Africa*, 2003.

Pericleous, K. A. and Rhodes, N. "The hydrocyclone classifier: A numerical approach", *International Journal of Mineral Processing*, Vol. 17, pp. 23-43, 1986.

Svarovsky, L. "A short course in cyclones", Manual to Course at the Dow Chemical Co. in Freeport/Texas, January 10-11, 1994.

Exploitation (or Output) of the project

1. Two Publications in International Journals

- 1.1. Title: An investigation of the effect of the particle-fluid and particle-particle interactions on the flow within a hydrocyclone
Authors: W. Kraipech, A. Nowakowski, T. Dyakowski and A. Suksangpanomrung
Journal: Chemical Engineering Journal xxx (2005) xxx-xxx
Impact No.: 0.677
Paper: See Appendix A
- 1.2. Title: Numerical prediction of outlet velocity patterns in solid-liquid separators
Authors: Michael J. Doby, Wanwilai Kraipech and Andrzej F. Nowakowski
Journal: Chemical Engineering Journal xxx (2005) xxx-xxx
Impact No.: 0.677
Paper: See Appendix B

2. Three Presentations in International Conferences

- 2.1. Title: An investigation of the effect of the particle-fluid and particle-particle interactions on the flow within a hydrocyclone
Authors: W. Kraipech, A. Nowakowski, T. Dyakowski and A. Suksangpanomrung
Conference: Solid-Liquid Separation Systems IV Conference organized by American Filtration Society, British Filtration Society and American Institute of Chemical Engineers (AIChE) at Pucon, Chile, 14-19 December 2003.
Paper: See Appendix A
- 2.2. Title: Numerical prediction of outlet velocity patterns in solid-liquid separators
Authors: Michael J. Doby, Wanwilai Kraipech and Andrzej F. Nowakowski
Conference: Solid-Liquid Separation Systems IV Conference organized by American Filtration Society, British Filtration Society and American

Institute of Chemical Engineers (AIChE) at Pucon, Chile, 14-19 December 2003.

Paper: See Appendix B

2.3. Title: The experimental study of the flow within a transparent hydrocyclone with a vortex finder attachment

Authors: Wanwilai Kraipech and Andrew F. Nowakowski

Conference: American Filtration and Separations Society Annual Conference 2005 at Atlanta, GA, 10-13 April 2005.

Paper: See Appendix C

3. One Preparing Manuscript for Future publication for International Journals

3.1. Title The simulation of the flow within a hydrocyclone operating with an air core and with an inserted metal-rod.

Authors: W. Kraipech, A. Nowakowski, T. Dyakowski and
A. Suksangpanomrung

Journal: International Journal of Mineral Processing

Impact No.: 0.539

Appendix A

Title: An investigation of the effect of the particle-fluid and particle-particle interactions on the flow within a hydrocyclone

Authors: W. Kraipech, A. Nowakowski, T. Dyakowski and A. Suksangpanomrung

Journal: Chemical Engineering Journal xxx (2005) xxx-xxx

An investigation of the effect of the particle–fluid and particle–particle interactions on the flow within a hydrocyclone

W. Kraipech^{a,*}, A. Nowakowski^b, T. Dyakowski^c, A. Suksangpanomrung^d

^a Department of Chemical Engineering, Srinakharinwirot University, Ongkharuk, Nakorn-Nayok 26120, Thailand

^b Department of Mechanical Engineering, University of Sheffield, Mappin Street, Sheffield S13JD, UK

^c Department of Chemical Engineering, UMIST, PO Box 88, Manchester M601QD, UK

^d Department of Mechanical Engineering, Academic Division, Chulachomklao Royal Military Academy, Nakorn-Nayok 26001, Thailand

Abstract

The effect of the particle–fluid and particle–particle interactions of the flow within a hydrocyclone is investigated. These were studied by applying the time scale analysis. It is shown that the particle–particle interactions, due to the lubrication and collision mechanisms, only play an important role in the vicinity of a hydrocyclone wall, and near the air core. In the remaining region, particle–fluid interactions are dominating. These play a vital role on the separation efficiency as illustrated by the significance of the wakes generated behind larger particles on dragging finer particles.

© 2005 Elsevier B.V. All rights reserved.

Keywords: Particle–fluid interaction; Hydrocyclone; Time-scale analysis

1. Introduction

A hydrocyclone is a type of separation equipment used for solid–liquid and liquid–liquid systems. It is used to separate dispersed particles from a continuous fluid as the effect of a swirl flow, and has been used in many mineral processing and mining industries.

In modelling the hydrocyclone performance, the influence of the particles on the flow is significant, particularly in the dense slurry flow, when the exchange of momentum from the particle–fluid, particle–particle and particle–wall interactions affect the velocity of the fluid. This may cause inefficiency in separation performance. The previous works presented by Bloor and Ingham [3–5], Pericleous and Rhodes [16], Pericleous [15] and Hsieh and Rajamani [10] discounted the effects of the presence of particles on a slurry velocity field by assuming that the flow is diluted (solids concentrate less than 5%), and the particle–fluid interaction for a single particle moving through a liquid without the presence of other particles was applied. However, when solids con-

centrate exceeds 5%, the presence of particles changes the velocity stresses and results in the generation of extra inertial stresses. The constitutive formulae, describing complex particle–fluid and particle–particle interactions are required. Generally, these formulae are very complicated as they take into account the spatial and temporal non-uniformities in the particle distributions as well as the acceleration of relative velocity.

There are two main approaches that can be applied to modelling multi-phase systems. They are a multi-fluid model and particle tracking method. The choice of one particular approach is determined by the character of the predominant interaction between the phases. The purpose of this study is to investigate the method for determining the predominant particle interaction of the flow within a hydrocyclone. From this knowledge, the proper method for modelling particle flow in this separator can be chosen.

In this paper, the equation of motion of a single particle in a fluid, neglecting the presence of other particles, is presented. The influence of neighbouring particles is described. The particle–fluid and particle–particle interactions are analysed using the concept of a time scale analysis introduced by Roco [20]. Here a case study is presented, based on the

* Corresponding author. Tel.: +66 6 999 8908; fax: +66 37 322 608.
E-mail address: wanwilai@swu.ac.th (W. Kraipech).

results published by Rajamani and Milin [17] for a 75 mm hydrocyclone. On this basis, the zones of the predominant interaction mechanism can be defined and in turn, a proper choice of a specific approach for modelling the flow within hydrocyclone can be made. As a result, the complexity of the constitutive formulae describing these interactions might be significantly reduced.

2. Particle–particle interaction related to hydrocyclone performance

In general, the hydrocyclone has been assumed to be operating at a steady state where the classical fluid dynamics can be applied. Stokes' law is generally assumed to be valid in order to develop models because it simplifies the mathematical formulation. Brownian movement, entrance effects on fluid, particle interactions, spinning of particles, curl of the fluid, wall interaction and the effect of turbulent fluctuations, are normally ignored or neglected [22]. The equations describing the separation function (see [8]), are based on an assumption that the particle settling velocity is not affected by the presence of other particles, and that it is a monotonic function of the particle size. In such a case it seems reasonable to assume that the separation process is linear and is a monotonic function of particle diameter. In fact, as the particles move to the wall their concentration increases and the interactions between them start to manifest through the interstitial fluid by secondary currents and pressure field changes. These particle interaction mechanisms cannot be neglected. Recently, Kumar et al. [13] measured the settling velocities for particles in a poly-dispersed mixture. It is obvious that Stokes' law cannot be applied in a study where the particle interactions are considered. Their results show that the motion of large particles is influenced only by the total volume fraction of particles within the system. Therefore, the settling velocity of large particles can be described by the Richardson–Zaki equation [18]. On the other hand, the smaller particles move at almost identical velocities to the larger particles. These small particles appear to be dragged with the larger ones. The settling velocities of small particles are even larger than the corresponding Stokes velocities or those predicted by the existing theories [2] or correlations.

Taneda's [24] experimental results showed that the permanent vortex-ring behind a sphere begins to form in the rear of a sphere when the critical particle Reynolds number, $Re_p = 24$. The size of the vortex-ring increases for higher Reynolds numbers and the wake behind a sphere begins to oscillate at the rear of the permanent vortex-ring when the Reynolds number is about 130. The latest experimental data of Yang et al. [27] and numerical simulations of Tang et al. [25] show how the dispersion of particles in a wake is organised for small Stokes number particles. Particles with Stokes number less than 1 are able to respond to the small-scale flow pattern and therefore become essentially flow-tracers. The trajectory

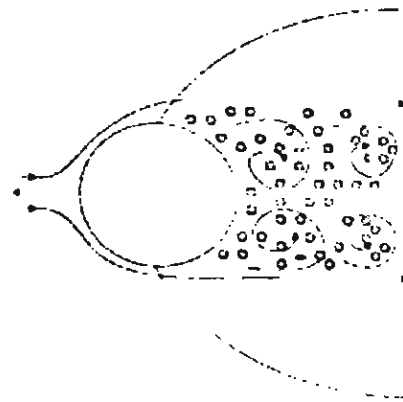


Fig. 1. Dragged mechanism of fine particles by large particle.

ries of these particles distribute themselves throughout the large-scale vortex cores. Conversely particles with a Stokes number much greater than 1 are little affected by the fluid fluctuations at any scale and therefore simply move in the direction of their initial trajectories with only slight deviation.

The ratio between the centrifugal acceleration, w^2/r , and the gravitational acceleration, g , varies along the hydrocyclone radius. The tangential velocity reaches its peak near the hydrocyclone axis. According to Cilliers [6], for a 10 mm

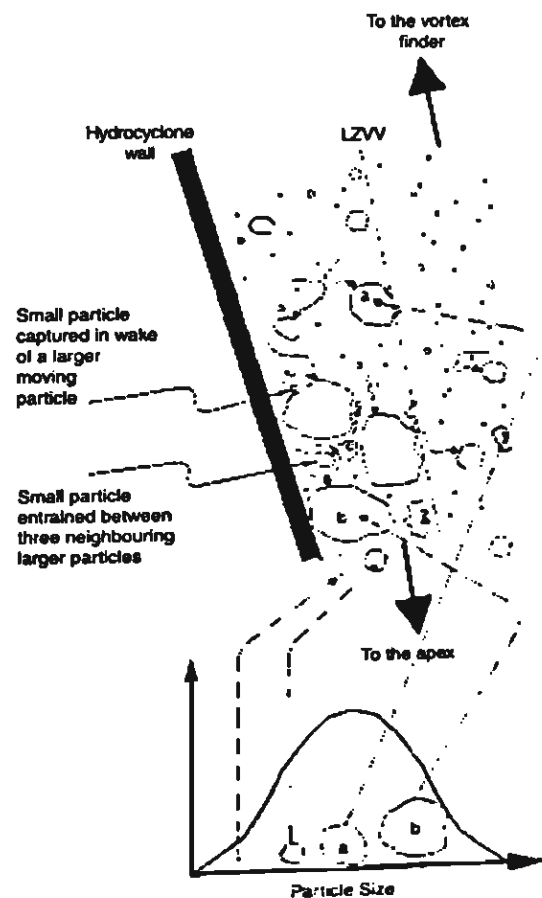


Fig. 2. Mechanisms that describe the fish-hook phenomenon (Roldan-Villasana [21]).

diameter hydrocyclone, the ratio between the centrifugal and gravitational acceleration is around 60,000 and the particle residence time is in the order of milliseconds. These are extreme conditions in relation to the gravitational buoyancy-driven separation of the dispersed phase from the continuous phase.

Therefore, interactions similar to those predicted by Yang et al. [27], and caused by the presence of the vortex-ring behind particles, seems to be worthy of consideration in terms of hydrocyclone performance. These can provide an additional mechanism for finer particles reporting to the underflow in the wake behind the larger particles (see Fig. 1). Such a mechanism may explain the shape of the selectivity curve, and the fact that the bypass value is higher than the water recovery to the underflow. This selectivity curve does not have a sigmoidal shape, but exhibits a dip in regions of finer particle size. This dip is known as the fish-hook effect. The methods of modelling the fish-hook effect of the flow within hydrocyclones, based on this mechanism, were investigated by Kraipech et al. [12].

The hydrodynamic behaviour of hydrocyclones treating concentrated slurries has yet to be fully understood and no accurate theory exists to simulate the phenomena occurring within hydrocyclones. Therefore, the other mechanisms explaining the fish-hook effect proposed by previous researchers such as Roldan-Villasana et al. [22] and Frachon and Cilliers [9], who introduced the idea of an influence from the turbulent dispersion on the motion of fine particles, could

be possible and should not be dismissed. Fig. 2 illustrates the possible mechanisms that describe the fish-hook phenomenon.

3. Equation of particle motion

The equation of motion of a spherical particle in a fluid, neglecting the interactions with other particles can be written as [1]:

$$m_p \frac{du_p}{dt} = m_p \left(1 - \frac{\rho}{\rho_p} \right) \mathbf{g} + \mathbf{F}_D + \mathbf{F}_{App} + \mathbf{F}_{Bas} + \mathbf{F}_{LS} + \mathbf{F}_{LM} + \mathbf{F}_{PG} \quad (1)$$

where m_p is the mass of the particle, u_p the instantaneous velocity of the particle and $\mathbf{g}$ the body acceleration. ρ and ρ_p are the densities of fluid and solid particles, respectively.

The term on the left-hand side of Eq. (1) describes the particle inertia, and the terms on the right-hand side are the forces caused by the particle–fluid interactions as explained in Table 1. When a particle's motion is affected by a neighbouring particle the other forces have to be altered as shown in Table 2.

There are two main causes for lateral lift force on a particle: one is due to the rotation of a particle moving in a fluid, and another is due to the shear of fluid itself, that is, the shear flow induces the lateral lift force even if the rotation of the

Table 1
Forces caused by particle–fluid interactions of a particle flow in a turbulent fluid

Forces	Sources of forces	Equations	
Steady-state drag force, F_D	The force acts on a particle in order to move the particle through a fluid with a uniform pressure and velocity field when there is no acceleration of the relative velocity between the particle and the conveying fluid	$F_D = \frac{1}{2} \rho C_D \frac{\pi d^2}{4} \mathbf{u} - \mathbf{u}_p (\mathbf{u} - \mathbf{u}_p)$	(4) d is the particle diameter, $(\mathbf{u} - \mathbf{u}_p)$ the relative velocity between fluid and particle and C_D the drag coefficient
Added mass force, F_{App}	The force of the particle on the fluid due to the acceleration of the relative velocity. When a particle is accelerated through the fluid, there is a corresponding acceleration of the fluid, which is at the expense of work done by the particle. This additional work causes the added mass force, which is required to accelerate the surrounding fluid	$F_{App} = \frac{\rho V_p}{2} \left(\frac{d\mathbf{u}}{dt} - \frac{d\mathbf{u}_p}{dt} \right)$	(5) V_p is the particle volume and $\left(\frac{d\mathbf{u}}{dt} - \frac{d\mathbf{u}_p}{dt} \right)$ is the relative acceleration of the fluid with respect to the particle acceleration
Basset force, F_{Bas}	The force due to the temporal delay in the boundary layer around the particle development as the relative velocity changes with time. This force takes into account the viscous effects due to the acceleration of the relative velocity	$F_{Bas} = \frac{3}{2} d^2 \sqrt{\pi \rho \mu} \int_0^t \frac{(d\mathbf{u}/dt) - (d\mathbf{u}_p/dt)}{\sqrt{t-t'}} dt'$	(6)
Saffman lift force, F_{LS}	The force produced by the pressure distribution developed on a particle due to the rotation induced by a fluid velocity gradient	$F_{LS} = 1.615 d^2 (\mu \rho \dot{\gamma})^{1/2} (\mathbf{u} - \mathbf{u}_p)$	(7) $\dot{\gamma}$ is the rate of fluid deformation
Magnus lift forces, F_{LM}	The force due to the rotation of the particle. This force is caused by a pressure difference between both sides of the particle resulting from the velocity difference due to the rotation	$F_{LM} = \frac{1}{2} \rho (\mathbf{u} - \mathbf{u}_p)^2 C_L \frac{\pi d^2}{4}$	(8) C_L is the lift force coefficient
Pressure gradient force, F_{PG}	The force due to the pressure gradient in the fluid surrounding the particle	$F_{PG} = -V_p \nabla p$	(9)

Table 2
The effect of neighbouring particles on the particle motion

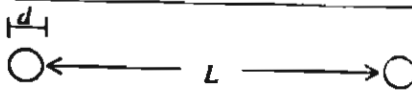
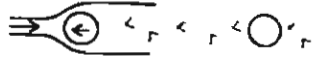
Distance between particles, L	Flow mechanism	Flow diagram
$\frac{L}{d} > 10$	No interaction between particles	
$\frac{L}{d} \approx 2$	Vortex shedding	
$\frac{L}{d} \approx 0.5$	Lubrication	
$\frac{L}{d} = 0$	Collision	

Table 3
Time scales for particle interactions in the hydrodynamic range [20]

Type of particle interaction	Time scale, t_m	Comments
Liquid–solid interaction (drag)	$t_d = \frac{4}{3} \frac{\pi d^2}{C_D u - u_p }$	(10) $C_D = f(Re_p)$, Re_p is the particle Reynolds number
Lubrication	$t_{lub} = \frac{18}{\pi} \frac{\lambda}{\dot{\gamma}_{ij}}$	(11) $\lambda = \frac{1 - (\alpha^*)^{0.33}}{(\alpha^*)^{0.33}}$, where $\alpha^* = \frac{a}{a_{max}}$
Collision	$t_{col} = \frac{45}{\pi \dot{\gamma}_{ij}} + \frac{4.5}{\pi \dot{\gamma}_{ij}} \ln(\lambda)$	(12) The first term on the right-hand side corresponds to the particle roughness

particle is absent. The former is called the Magnus effect and the latter is the Saffman effect, which are additionally described in Table 1. The Saffman lift force is predominantly induced when a solid particle moves in a region with a shear flow of a steep velocity gradient in the surrounding fluid. When the rotation of a particle is given from the beginning of the transport, the Magnus lift force is predominant.

Dense flow is characterised by high collision frequencies between particles, and hence their motion is dominantly influenced by particle–particle collisions. Interactions between the fluid and particles are of minor importance [23]. The models for predicting the collision forces are not discussed here but they can be found in the references of Crowe et al. [7], Tsirkunov and Panfilov [26] and Sommerfeld [23]. Generally speaking, the collision force depends on the properties of the particles such as density and surface roughness, as well as on the magnitude and the direction of the relative velocity.

The lubrication interaction is the particle–particle interaction due to the pressure in a fluid, which is generated by the particles approaching each other. This pressure can be calculated using the lubrication theory, which neglects all inertial forces in a fluid (creeping flow approach). By integrating this pressure distribution along the particle surface, an additional

force, the lubrication force, acting on the particle can be derived [7]. Assuming that the flow is symmetrical about the centre plane between two, this force can be described by the following equation:

$$F_{Lub} = -\frac{3\pi\mu d^2 \dot{h}}{8h_0} \quad (2)$$

where h_0 is the distance between the sphere and the symmetric plane, which is perpendicular to the plane, and $\dot{h}$ the rate at which the sphere is approaching the symmetric plane.

4. Time scale analysis

The frequency of an interaction mechanism's occurrence between particles, or between particles and fluid, is inversely proportional to the time required by the particle to respond to the interaction mechanism.

An interaction mechanism “ n ” that is characterised by the time scale t_n is more frequent and therefore has a larger distribution to the momentum transfer than another interaction mechanism “ m ” characterised by the time scale t_m , if $t_n < t_m$. This relation is defined as the relative particle interaction

Table 4
Relative particle interaction number, $N_{n,m}$ [20]

Relative interaction mechanisms	Significant ratio	Relative particle interaction number
Particle–particle lubrication/particle–liquid drag	$\frac{t_d}{t_{lub}}$	$N_{d,lub} = \frac{\pi^2 d^2 \dot{\gamma}_{ij}}{18^2 \nu \lambda}$ (13)
Particle–particle collisions/particle–liquid drag	$\frac{t_d}{t_{col}}$	$N_{d,col} = \frac{\pi^2 d^2 \dot{\gamma}_{ij}}{18 \nu} \frac{1}{45 - 4.5 \ln(\lambda)}$ (14)
Particle–particle collisions/particle–particle lubrication	$\frac{t_{lub}}{t_{col}}$	$N_{lub,col} = \frac{18 \lambda}{45 - 4.5 \ln(\lambda)}$ (15)

Note: t_d is replaced by its expression for the Stokesian ($Re_p \leq 0.1$) in this table.

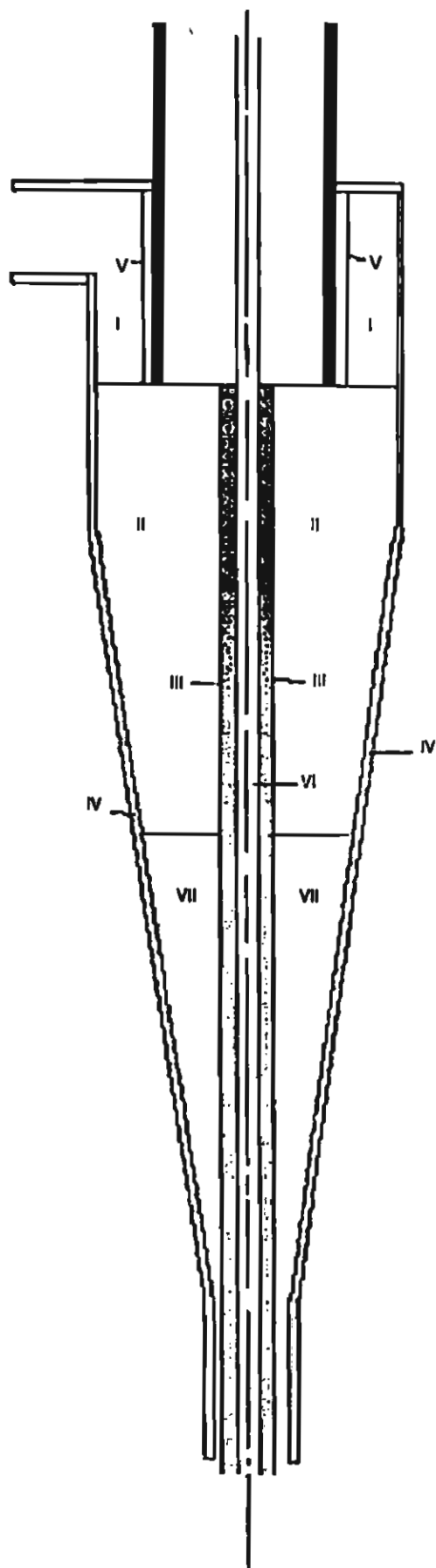


Fig. 3. Zones in the hydrocyclone.

number, $N_{n,m}$.

$$\frac{1/t_m}{1/t_n} = \frac{t_n}{t_m} = N_{n,m} \quad (3)$$

If $N_{n,m} < 1$, the n th mechanism is prevalent comparative to the m th mechanism.

The time scales responding to liquid–solid (drag), lubrication and collision interactions and the relative particle interaction numbers $N_{n,m}$, are shown in Tables 3 and 4, respectively. They were derived using a similar method as discussed by Roco [19] (see [11]). On the contrary to the drag time scale, the lubrication and collision time scales are dependent on the velocity profile and the solids volume fraction.

5. A case study for calculating the time scales of the flow within a hydrocyclone

In this study, an application of a time scale analysis to identify the predominant interaction is presented for the 75 mm hydrocyclone, based on the experiment and prediction of Rajamani and Milin [17]. The time scales are calculated for seven flow zones within a hydrocyclone, instead of six flow zones, which was carried out in the author's previous work,

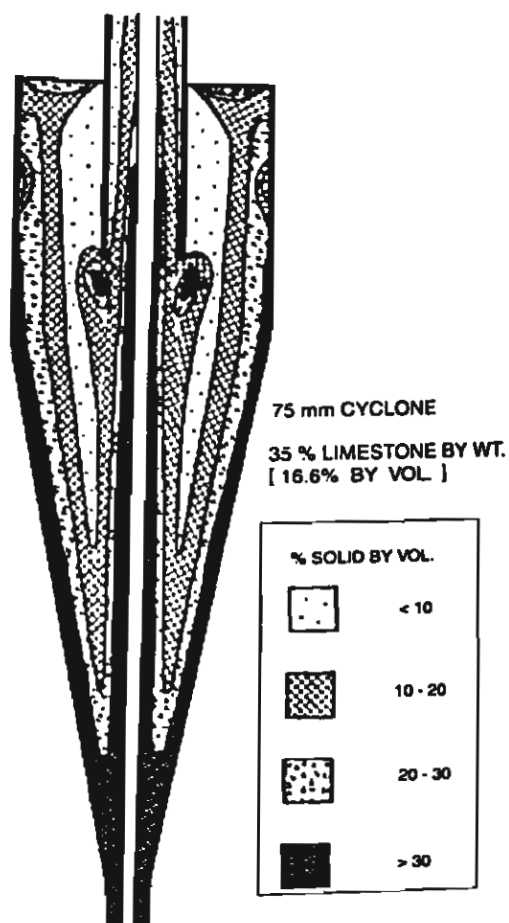


Fig. 4. Predicted volumetric concentration map for 35% limestone by weight (16.6% by volume) in the feed [17].

Table 5
Location of zones

Zone	Location
I	The upper cylindrical part of the hydrocyclone, from the top to the bottom of the vortex finder tube, outside the boundary layer on the outer wall of the vortex finder. Because of the high level of turbulence and secondary flows that originate due to the tangential entrance, the flow is well mixed containing a homogeneous suspension
II	The middle part of the hydrocyclone from the bottom of the vortex finder tube to the middle of the conical section outside the boundary layer on the lateral walls and the boundary layer near the air core
III	The area near the air core
IV	The boundary layer on the lateral walls. The boundary layer starts at the feed and develops its maximum thickness when reaching the level of the bottom of the vortex finder tube
V	The boundary layer at the outer wall of the vortex finder
VI	The air core, which is assumed to have a cylindrical shape
VII	The lower part of the hydrocyclone from the middle to the end of the conical section outside the boundary layer on the lateral walls and the boundary layer near the air core

Table 6
The experimental selectivity values [17]

Particle size (μm)	Percent of solid recovery to underflow
90	100
65	92
45	80
33	52
23	30
16	20
11	16
8	11
4	10
3	9
1	7

Kraipech [11]. The locations of each zone are described and shown in Table 5 and Fig. 3.

The feed slurry of the case study is 35% by weight of limestone (16.6% by volume). The liquid phase is water, which has a density of 1000 kg/m^3 and a kinematic viscosity of $10^{-6} \text{ m}^2/\text{s}$. The density of limestone is 2700 kg/m^3 and its volume fraction at maximum packing is 0.7. The particle size is in the range of 1–90 μm (Table 6). The volumetric concentration map and tangential velocity profile are shown in Figs. 4 and 5, respectively.

The time scales and their relative particle interaction numbers are the functions of flow data such as the velocity field, particle sizes and the solid concentration, as shown in Tables 3 and 4. Therefore, it is possible to calculate the time scales and their relative particle interaction numbers for a given set of data describing the flow.

Table 7
The flow characteristics in each zone in the hydrocyclone

Zone	Volume fraction, α	Particle size (μm)	The mean rate of strain tensor, $ \dot{\gamma} = \frac{r}{2} \left \frac{\partial}{\partial r} \left(\frac{u}{r} \right) \right (\text{s}^{-1})$
I	$\alpha < 0.1$; $0.1 < \alpha < 0.2$; $0.2 < \alpha \leq 0.3$	1–90	107
II	$\alpha < 0.1$; $0.1 < \alpha < 0.2$; $0.2 < \alpha \leq 0.3$	1–90	110
III	$\alpha > 0.3$	1–90	570
IV	$\alpha > 0.3$	1–90	179
V	$\alpha < 0.1$	1–65	19
VI	–	–	–
VII	$\alpha < 0.1$; $0.1 < \alpha < 0.2$; $0.2 < \alpha \leq 0.3$	1–90	187

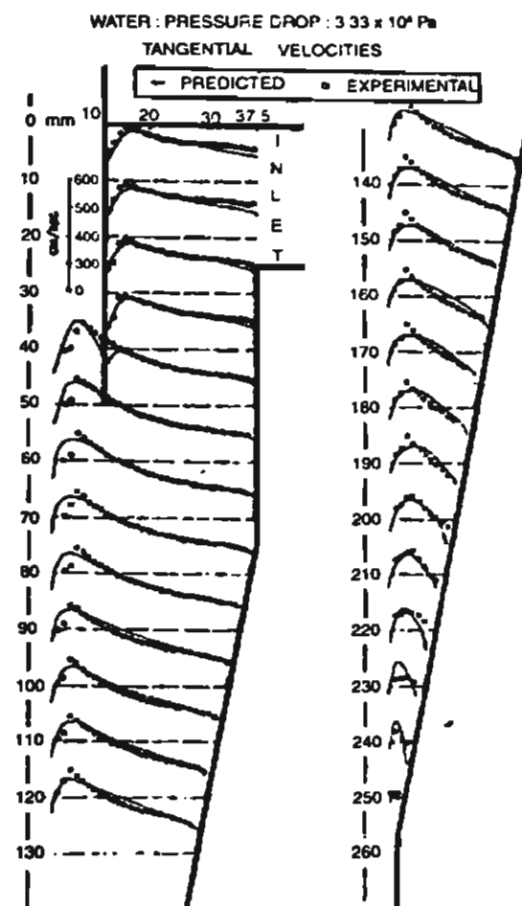


Fig. 5. Measured and predicted tangential velocities in a 75 mm hydrocyclone [17].

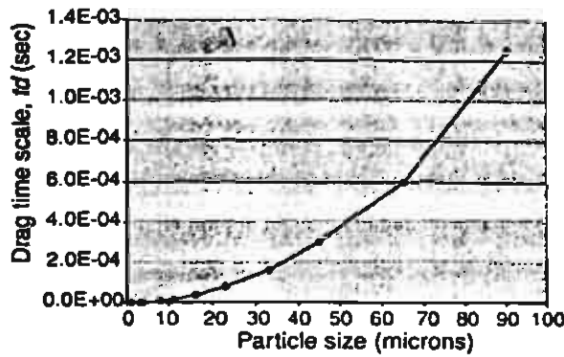


Fig. 6. Drag time scale, t_d , of the limestone particle flow in water (kinematic viscosity of $10^{-6} \text{ m}^2/\text{s}$) obtained from the experimental results of Rajamani and Milin [17].

Eqs. (10)–(12) are used to calculate the drag, lubrication and collision time scale. The flow around the particle is assumed to be in the Stokes' law region. Therefore, $C_D = \frac{24}{Re_p}$. The aqueous suspension of limestone is assumed to be a Newtonian fluid for all range of weigh fraction in order to simplify the calculation, even though the suspension shows a non-Newtonian behaviour when its weight fraction is high. The rate of strain tensor can be calculated from the tangential velocity profile, which is presented in Fig. 5. The component $\dot{\gamma}_{r\theta} = \frac{r}{2} \frac{\partial}{\partial r} \left(\frac{w}{r} \right) + \frac{1}{2r} \frac{\partial v}{\partial \theta}$ of the rate of strain tensor is taken into this analysis, instead of $\dot{\gamma}_{r\theta} = \frac{\partial w}{\partial r}$ which is used in Nowakowski et al. [14]. Assuming that the flow is axisymmetrical, the term $\frac{1}{2r} \frac{\partial v}{\partial \theta}$ is neglected. The absolute value of the rate of strain tensor can be calculated as $|\dot{\gamma}_{r\theta}| = \frac{r}{2} \left| \frac{\partial}{\partial r} \left(\frac{w}{r} \right) \right|$. The ratio between the inter-particle distance and the particle diameter, λ , is equal to $\frac{1-(\alpha^*)^{0.33}}{(\alpha^*)^{0.33}}$, where $\alpha^* = \frac{\alpha}{\alpha_{\max}}$. $\alpha_{\max}$ is the solids volume fraction at maximum packing and α the solids volume fraction obtained from the volumetric concentration as shown in Fig. 4. In this figure, the solids concentration distribution is presented in three ranges of volumetric concentration. According to the experimental results, it is assumed that there are no particles equal to or larger than $90 \mu\text{m}$ in diameter in Zone V (see Fig. 3). The flow characteristics are summarised in Table 7. The drag time scale as a function of the particle diameter is shown in Fig. 6. The time scales

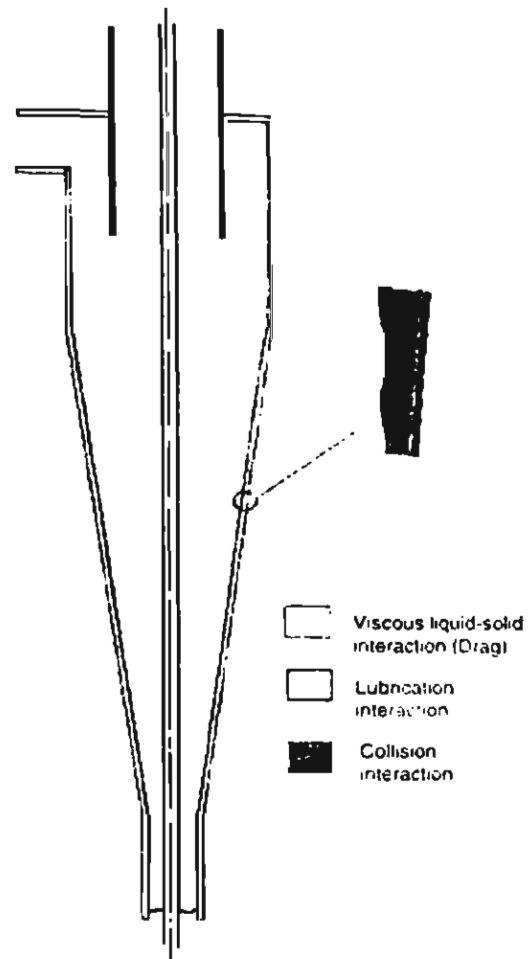


Fig. 7. Main particle interaction mechanisms in the 75 mm hydrocyclone.

for lubrication and collision interactions are presented in Table 8.

Fig. 6 shows that an increase in the particle size leads to an increase in the drag time scale, since a larger particle needs more time to respond to the change in fluid velocity than a smaller one. The lubrication and collision time scales decrease with an increase in the solids volume fraction and the mean rate of strain tensor. This indicates that these two

Table 8
Lubrication time scale, t_{lub} , and collision time scales, t_{col}

Zone	Solid concentration (volume fraction)	$t_{\text{lub}} = \frac{18}{s} \frac{\lambda}{\dot{\gamma}_{r\theta}} \text{ (s)}$	$t_{\text{col}} = \frac{4s}{\dot{\gamma}_{r\theta}} + \frac{4s}{\dot{\gamma}_{r\theta}} \ln(\lambda) \text{ (s)}$
I	$\alpha < 0.1$; $0.1 < \alpha < 0.2$; $0.2 < \alpha \leq 0.3$	$t_{\text{lub}} > 0.0561$; $0.0561 > t_{\text{lub}} > 0.0319$; $0.0319 > t_{\text{lub}} \geq 0.0201$	$t_{\text{col}} > 0.0952$; $0.0952 > t_{\text{col}} > 0.0864$; $0.0864 > t_{\text{col}} \geq 0.0792$
II	$\alpha < 0.1$; $0.1 < \alpha < 0.2$; $0.2 < \alpha \leq 0.3$	$t_{\text{lub}} > 0.0546$; $0.0546 > t_{\text{lub}} > 0.0319$; $0.0319 > t_{\text{lub}} \geq 0.0201$	$t_{\text{col}} > 0.0926$; $0.0926 > t_{\text{col}} > 0.084$; $0.084 > t_{\text{col}} \geq 0.0770$
III	$\alpha > 0.3$	$t_{\text{lub}} < 0.0037$	$t_{\text{col}} < 0.0149$
IV	$\alpha > 0.3$	$t_{\text{lub}} < 0.0120$	$t_{\text{col}} < 0.0473$
V	$\alpha < 0.1$	$t_{\text{lub}} > 0.0316$	$t_{\text{col}} > 0.0536$
VI	–	–	–
VII	$\alpha < 0.1$; $0.1 < \alpha < 0.2$; $0.2 < \alpha \leq 0.3$	$t_{\text{lub}} > 0.0321$; $0.0321 > t_{\text{lub}} > 0.0183$; $0.0183 > t_{\text{lub}} \geq 0.0115$	$t_{\text{col}} > 0.0545$; $0.0545 > t_{\text{col}} > 0.0494$; $0.0494 > t_{\text{col}} \geq 0.0453$

Table 9
Prevalent interaction mechanism for each flow zone in the hydrocyclone

Zone	Solid concentration condition	Relative particle interaction numbers			Prevalent particle interaction mechanism
		$N_{d,lub}$	$N_{d,col}$	$N_{lub,col}$	
I	$\alpha \leq 0.3$	$N_{d,lub} < 1$	$N_{d,col} < 1$	$N_{lub,col} < 1$	Liquid–solid interaction (drag)
II	$\alpha \leq 0.3$	$N_{d,lub} < 1$	$N_{d,col} < 1$	$N_{lub,col} < 1$	Liquid–solid interaction (drag)
III	$0.3 < \alpha < 0.52$	$N_{d,lub} < 1$	$N_{d,col} < 1$	$N_{lub,col} < 1$	Liquid–solid interaction (drag)
	$\alpha \geq 0.52$	$N_{d,lub} > 1$	$N_{d,col} < 1$	$N_{lub,col} < 1$	Lubrication interaction
IV	$0.3 < \alpha < 0.64$	$N_{d,lub} < 1$	$N_{d,col} < 1$	$N_{lub,col} < 1$	Liquid–solid interaction (drag)
	$\alpha \geq 0.64$	$N_{d,lub} > 1$	$N_{d,col} < 1$	$N_{lub,col} < 1$	Lubrication interaction
V	$\alpha < 0.1$	$N_{d,lub} < 1$	$N_{d,col} < 1$	$N_{lub,col} < 1$	Liquid–solid interaction (drag)
VI	–	–	–	–	–
VII	$\alpha \leq 0.3$	$N_{d,lub} < 1$	$N_{d,col} < 1$	$N_{lub,col} < 1$	Liquid–solid interaction (drag)

mechanisms are not only dependent on the flow concentration but also on the flow velocity field. The prevalent interaction mechanisms in each flow zone are determined by using the data shown in Fig. 6 and Table 8, and are presented in Table 9 and Fig. 7.

From this analysis, it is found that for the majority of the area within the hydrocyclone, where the solids concentration is less than 30% by volume, the main particle interaction mechanism is the liquid–solid interaction (drag). In the high solids concentration areas, the lateral boundary layer (zone IV) and the area near the air core (zone III), the lubrication and collision interaction mechanisms are prevalent compared with the drag interaction mechanisms. This is in agreement with the work of Nowakowski et al. [14].

6. Conclusion

An application of a qualitative analysis based on a “time scale” concept was presented and discussed to assess the predominant interaction mechanisms within a hydrocyclone. It was found that the liquid–particle interaction (drag) plays an important role in the main body of a hydrocyclone. However, within the regions close to the walls and the air core, both lubrication and collision mechanisms are predominant. This means that the flow pattern within a hydrocyclone should not only be described by interactions between particles and carrying fluid but that the results of solid mechanics should also be included in modelling particle–particle collisions in the vicinity of the hydrocyclone walls. The future work should lead to deriving a simplified model for the particle transport taking into account only the dominating relevant forces in each region. The extension of the analysis to the non-Newtonian behaviour of the high weight fraction of limestone suspensions is also planned.

References

- [1] M.K. Akbar, M.A.R. Sharif, R.C. Bradt, Effect of forces on a particle in a straight channel turbulent flow, in: Proceedings of the Fourth International Conference on Multiphase Flow, vol. 53, New Orleans, LA, USA, May 27–June 1, Institution of Chemical Engineers, 2001, pp. 1–6.

- [2] G.K. Batchelor, Sedimentation in a dilute polydisperse system of interacting sphere. Part 1. General theory, *J. Fluid Mech.* 119 (1982) 372–408.
- [3] M.I. Bloor, D.B. Ingham, Theoretical investigation of the flow in a conical, *Trans. Inst. Chem. Eng.* 51 (1973) 36–41.
- [4] M.I. Bloor, D.B. Ingham, Turbulent spin in a hydrocyclone, *Trans. Inst. Chem. Eng.* 53 (1975) 1–6.
- [5] M.I. Bloor, D.B. Ingham, The flow in industrial cyclones, *J. Fluid Mech.* 178 (1987) 507–519.
- [6] J.J. Cilliers, Private communication, 2001.
- [7] C.T. Crowe, M. Sommerfeld, Y. Tsuji, *Multiphase Flows with Droplets and Particles*, CRC Press, USA, 1998.
- [8] B.C. Flintoff, L.R. Plitt, A.A. Turak, Cyclone modelling a review of present technologies, *CIM Bull.* 80 (1987) 39–50.
- [9] M. Frachon, J.J. Cilliers, A general model for hydrocyclone partition curves, *Chem. Eng. J.* 73 (1999) 53–59.
- [10] K.T. Hsieh, R.K. Rajamani, Mathematical model of hydrocyclone based on physics of fluid flow, *AIChE J.* 37 (5) (1991) 735–745.
- [11] W. Kraipech, Studying the performance of mineral hydrocyclones, Ph.D. Thesis, University of Manchester, Institute of Science and Technology, 2002.
- [12] W. Kraipech, W. Chen, F. Parma, T. Dyakowski, Modelling the fish-hook effect of the flow within hydrocyclones, *Int. J. Miner. Process.* 66 (2002) 49–65.
- [13] S. Kumar, T.W. Pirog, D. Ramkrishna, A new method for estimating hindered creaming/settling velocity of particles in polydisperse systems, *Chem. Eng. Sci.* 55 (2000) 1893–1904.
- [14] A. Nowakowski, W. Kraipech, T. Dyakowski, R. Williams, The hydrodynamics of a hydrocyclone based on a three-dimensional multi-continuum model, *Chem. Eng. J.* 80 (2000) 275–282.
- [15] Pericleous, Mathematical simulation of hydrocyclones, *Appl. Math. Model.* 11 (1987) 242–255.
- [16] K.A. Pericleous, N. Rhodes, The hydrocyclone classifier: a numerical approach, *Int. J. Miner. Process.* 17 (1986) 23–43.
- [17] R.K. Rajamani, L. Milin, Fluid-flow model of the hydrocyclone for concentrated slurry classification, in: L. Savarovsky, M.T. Thew (Eds.), *Hydrocyclone Analysis and Applications*, Kluwer Academic Publishers, London, 1992, pp. 59–108.
- [18] J.F. Richardson, W.N. Zaki, Sedimentation and fluidisation, Part 1, *Trans. Inst. Chem. Eng.* 32 (1954) 35–53.
- [19] M.C. Roco, Dense slurry flow, in: *Proceedings of the Advancement in Aerodynamics, Fluid Mechanics and Hydraulics*, ASCE, EM & HY Div., 1986, pp. 365–376.
- [20] M.C. Roco, Turbulent flow of incompressible mixtures, in: *Encyclopedia of Fluid Mechanics*, vol. 10, Gulf Publisher Company, Houston, TX, 1990, pp. 1–68.
- [21] E.J. Roldan-Villasana, Modelling and Simulation of Hydrocyclone Networks for Fine Particle Processing, Ph.D. Thesis, UMIST, Manchester, UK.

- [22] E.J. Roldan-Villasana, R.A. Williams, T. Dyakowski, The origin of the fish-hook effect in hydrocyclone separators, *Powder Technol.* 77 (1993) 245–250.
- [23] M. Sommerfeld, Validation of a stochastic Lagrangian modelling approach for inter-particle collisions in homogeneous isotropic turbulence, *Int. J. Multiphase Flow* 27 (2001) 1829–1858.
- [24] S. Taneda, Studies on wake vortices (III): experimental investigation of the wake behind a sphere at low Reynolds numbers, *Rep. Res. Inst. Appl. Mech.* IV 16 (1956) 99–105.
- [25] L. Tang, F. Wen, Y. Yang, C.T. Crowe, J.N. Chung, T.R. Troutt, Self-organizing particle dispersion mechanism in a plane wake, *Phys. Fluids A* 10 (1992) 2244–2251.
- [26] Y.M. Tsirkunov, S.V. Panfilov, Modelling of particle-wall interaction in two-phases flows at moderate and high particle impact velocity, in: *Proceedings of the Third International Conference on Multiphase Flow, ICMF 98*, Lyon, France, June 8–12, 1998.
- [27] Y. Yang, C.T. Crowe, J.N. Chung, T.R. Troutt, Experiments on particle dispersion in a plane wake, *Int. J. Multiphase Flow* 26 (2000) 1583–1607.

Appendix B

Title: Numerical prediction of outlet velocity patterns in solid-liquid separators
Authors: Michael J. Doby, Wanwilai Kraipech and Andrzej F. Nowakowski
Journal: Chemical Engineering Journal xxx (2005) xxx-xxx

Numerical prediction of outlet velocity patterns in solid–liquid separators

Michael J. Doby^{a,*}, Wanwilai Kraipech^b, Andrzej F. Nowakowski^{a,c}

^a Chemical Engineering Department, UMIST, P.O. Box 88, Manchester M60 1QD, UK

^b Department of Chemical Engineering, Srinakharinwirot University, Rangsit-Nakhonnayok Rd, Klong 16, Ongcharak, Nakhonnayok 26120, Thailand

^c Department of Mechanical Engineering, University of Sheffield, Mappin Street, Sheffield S1 3JD, UK

Abstract

Three-dimensional simulations of incompressible fluid flow within hydrocyclone have been performed using the developed numerical technique. A discretization of the physical problem has been done by using a finite element method based on mixed approximation of the velocity and pressure space. The approach offers significant advantages in the solution process of convection dominated internal flows having one inlet and more than one outlet. It deals with the complex geometry of the head entry part of hydrocyclone. The boundary conditions represent forces and are efficiently incorporated into the numerical formulation. Such formulation is very useful since it allows modeling the characteristic velocity profile in the outlet. We investigate the interaction between the swirling flow and velocity profile at the outlet. The studies are carried out for fluids with different properties and can be extended to hydrocyclones with different geometrical configurations.

© 2005 Elsevier B.V. All rights reserved.

Keywords: Solid–liquid separators; Discharge angle; Hydrocyclone design; Numerical simulation

1. Introduction

The discharge of solid–liquid separators contains the information, which could possibly be used for better operational control. The work presented endeavors to establish a procedure for improving operation of hydrocyclones based on computational simulations. The complex nature of the flow in hydrocyclones drastically changes depending on the operating conditions. Since the hydrodynamics of a hydrocyclone are not clearly understood, the design and control of hydrocyclones are primarily based on empirical data. Due to the complicated nature of flow in a hydrocyclone, the models developed to predict the operation and control are still under development. Heiskanen [1] argued that hydrocyclone empirical models available for determining the operational state should be used carefully, because the models did not accurately control or predict the operation of a hydrocyclone. In spite of the shortcomings, the design process of hydrocyclones consists of using empirical models and classification

curves, which are based on specific fluid properties. Plitt [2] proposed one of the first models, which used operational parameters and calculated the mean particle size to determine the operational state. Other empirical relationships have been proposed by Lynch and Rao [3] and Nageswararao et al. [4], which compare well with specific experimental data. In spite of the inaccuracies and uncertainties of the empirical models, steps have been made in improving the design of hydrocyclones as demonstrated by Chu and Luo [5]. Due to lack of complex empirical databases for different mixtures and geometries, various methods of controlling the operation of a hydrocyclone are considered.

In order to develop a good control technique, the significant variables to the operation must be determined. The primary operating parameter that can be controlled without changing fluid properties or measuring the solids in the under/overflow is the discharge angle from the apex. Assuming that two distinct operational states of hydrocyclone exist, which are the following: roping and spray. The states differ by the discharge profile from the apex. Roping tends to form a rope-like discharge, while spray resembles an umbrella. Concha et al. [6] points out that the best separa-

* Corresponding author.

E-mail address: m.doby@postgrad.umist.ac.uk (M.J. Doby).

tion occurs near the formation of rope discharge. Similarly, Neesse et al. [7] states on a broader note that a hydrocyclone achieves the best separation at a transitional discharge phase between roping and spraying. Many researchers have tried to control the operation of a hydrocyclone by using varying non-obtrusive experimental techniques. Van Latum [8] suggested using X-ray imaging during operation to produce density profile cross-sections. Williams [9] used electrical impedance tomography, which produced a cross-section of an operating hydrocyclone. Petersen et al. [10] used image analysis as a controlling mechanism for the hydrocyclone, based on the discharge spray angle. Neesse et al. [11] developed a non-obtrusive manner of controlling the performance based on the angle of the spray discharge using a combination of techniques, which effectively controlled the operation of a hydrocyclone. Van Deventer et al. [12] demonstrated a method of calculating the angle of discharge with the inclusion of gravity. The method that was presented shows that the spray discharge proceeds through three distinct stages, which are the following: initial increasing of angle, flat and the gravitational driven regions [12]. With the calculation of the angle, determination of the operating performance of the hydrocyclone can be predicted as shown by Petersen et al. [10].

Deriving from the analysis of the discharge spray, we proposed a novel approach for controlling the operation of a hydrocyclone. The approach, which is an extension of the computational code of Nowakowski and Dyakowski [13] can be implemented in the design stage to effectively control the operation of the hydrocyclones, or can be applied as a tool to determine the effectiveness of an existing solid–liquid separator by calculating the discharge angle.

2. Problem formulation

In the study, the necessary numerical data are obtained using finite element approximation of incompressible viscous flow. The governing partial differential equations are the continuity equation and the Navier–Stokes equations. The former is a mathematical realization of the incompressibility of the flow, whereas the latter is momentum equation along with a linear constitutive law relating stresses to rates of strains. The primitive variable formulation is expressed in the most general and fundamental “stress-divergence” form [14]. The form is commonly used in finite element methods and rarely in finite difference or volume methods. The advantage of the “stress-divergence” form is that it permits formulation of physically meaningful Neumann boundary conditions via proper accounting of viscous forces.

2.1. Boundary conditions

Fig. 1 introduces the vertical cross-section of a hydrocyclone with the different parts of boundary conditions indicated.

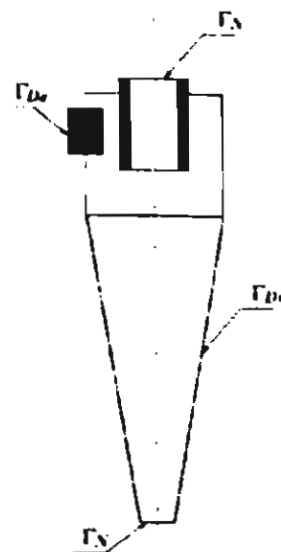


Fig. 1. Boundary conditions of hydrocyclone.

For fluid adjacent to a solid wall, Γ_{Dw} and at the inlet of the hydrocyclone, Γ_{De} the Dirichlet boundary conditions specifying the velocity vectors are imposed. At outflow sections, Γ_N Neumann boundary conditions are needed to truncate the computational domain. In the present application, they represent forces and are expressed by:

$$f = \tau n = -pn + 2\nu D(u)n = \hat{f} \quad (1)$$

where τ represents total stress tensor and $D(u)$ deformation tensor equal to the following:

$$\tau = -pI + 2\nu D(u) \quad D(u) = 0.5[\nabla u + (\nabla u)^T] \quad (2)$$

here, n is unit outer vector normal to the boundary, $\hat{f}$ the prescribed body force on the boundary, ν denotes the kinematic viscosity of the fluid, u is the velocity vector and p is the scalar pressure. In the performed numerical simulations the “no-stress” boundary conditions were prescribed at the outlets. This is the equivalent of setting $f=0$ at the spigot and vortex finder outlet. Such assumption is physically correct, it does not pre-define the parameters of operations, although it may be considered idealistic. Clearly, the stresses exist and can play an important role in the establishment of the velocity profile. The precise value for such stresses is difficult to determine experimentally. Independent of mathematical legitimacy of such boundary conditions, the physics of the matter provides only guidance at best. However, with the lack of the necessary information, the assumption of zero-valued components of forces is natural and advantageous compared to the imposition of specific velocity profile as boundary conditions. The work of Nowakowski and Dyakowski [13] shows that such description has less significant impact on the velocity field. Consequently, the method proposed enables simulations of the characteristic velocity profile at the outlet. In contrast, majority of existing finite volume computational codes for hydrocyclones usually

specify the split ratio or make some assumptions regarding the character of the velocity profile. Such assumptions limit the applicability of simulations performed and consequently the obtained data cannot be used to control the operation of hydrocyclones. Another important advantage of boundary condition, Eq. (1), can be of use when modeling the air core (interface between fluids can be considered as a free boundary).

2.2. Numerical technique

In the finite element method, the flow equations and the associated boundary conditions are solved using the weak form of the governing equations. Then, the continuum problem governed by partial differential equations is reduced by discretization to a system of algebraic equations. The finite element procedure consists of meshing the hydrocyclone geometry into a number of tetrahedral elements. Within each of the elements, the dependent variables (three components of velocity and pressure) are interpolated by suitable polynomials at a set of nodal points. In the present implementation, the adopted element involves a piecewise continuous quadratic approximation of velocities and piecewise constant approximation of pressure. An account of the suitability of different approximation functions and element numerical stability for hydrocyclone simulation is presented in [15]. The comprehensive documentation of the finite element method applied to incompressible fluid mechanics was presented by Gresho and Sani [16].

2.3. Solution of flow problem

As a solution method, the pressure projection algorithm is implemented by Nowakowski and Dyakowski [13], which is based on some ideas introduced by Haroutunian et al. [17]. The method solves a convection–diffusion equation for velocity, excluding pressure from the momentum equations and updates the pressure while imposing the incompressibility constraint. The velocity field obtained in the first step does not satisfy continuity equation in general. Thus, the velocity field has to be projected onto a divergence-free subspace of the approximation space while updating the pressure. The above described procedure is carried out iteratively using the discrete operators after spatially discretizing the weak form of the Navier–Stokes equations. As a consequence, the boundary conditions, Eq. (1), are consistently incorporated in the algorithm.

2.4. Calculation of exit profile

The results of the computational fluid dynamics simulation of the hydrocyclone provide the data for calculating the discharge angle. Neesse et al. [7] showed that the form of the underflow discharge can be used as an indication of the operating state of hydrocyclones. The angle is determined from the velocity components at the spigot

and fluid properties. Lacking the information from the three-dimensional flow field, Neesse et al. [7] derived the following equation:

$$\alpha = \arctan\left(\frac{v}{u}\right) \approx \arctan\frac{\rho_m(D_u/2u)w^2}{\mu_m} \quad (3)$$

here, u , v and w are the velocity components in the cylindrical coordinate system of the suspension in the axial (u), radial (v) and tangential direction (w), ρ_m the density of the mixture, μ_m the viscosity of the mixture and D_u is the apex diameter. Neesse et al. [7] model assumes boundary conditions that simplify the equations from a three- to two-dimensional problem. For simplification, the flow conditions at the inlet assume an axial symmetrical flow. The assumption of symmetrical flow is not an accurate representation of the flow in the hydrocyclone, as shown by He et al. [18]. The radial velocity was chosen by corresponding a flux to the given flow rate through the inlet. The model assumes that the viscosity of the fluid changes throughout the hydrocyclone, though in the present work the viscosity is set constant. The boundary conditions at the outlet assume that the effluent does not contact the air. Even with the simplifying assumptions for the method proposed by Neesse et al. [7], which will be referred to throughout the paper as the Dueck method, the Dueck method was able to be used to control the operation of the hydrocyclone.

The presented approach, which will be referred throughout the paper as the AFN method, is not limited to axisymmetrical flow. Thus, the AFN method having generated a three-dimensional velocity field the angle is calculated directly. Due to the three-dimensional nature of the problem, the complete set of Navier–Stokes equations was solved in a three-dimensional framework. The necessary velocity field profiles are obtained using the described finite element approach. An unstructured grid that forms to the shape of a hydrocyclone using tetrahedral elements is generated. One of the reasons that a structured grid was not used may be attributed to a singularity that occurs with the governing equations [19]. Using an unstructured grid helps not only to eliminate the occurrence of singularities but provides full geometrical flexibility. The computer output is in the form of velocity vectors in the u , v and w directions in a Cartesian coordinate system. Using basic trigonometric functions the calculation of the angle is obtained.

Table 1
Fluid properties at different runs with diameter at 22 mm

Number	Reynolds number (Re_{in})
Run 1	220
Run 2	259
Run 3	293
Run 4	338
Run 5	440
Run 6	488
Run 7	505

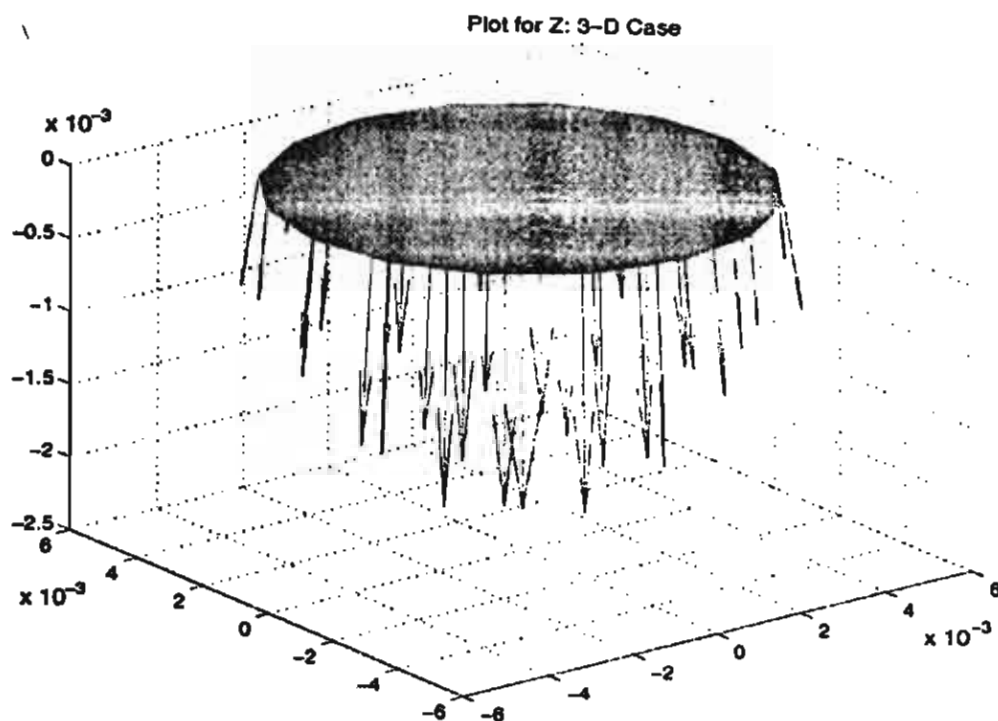


Fig. 2. Three-dimensional quiver cross-section of the velocity profile exiting the apex.

3. Results and discussion

In order to compare the results of the two methods, several formed at different Reynolds numbers at the inlet. The range of viscosities that were chosen kept the flow in the laminar regime. Due properties, the comparison would not introduce any questionable results from using a specific turbulence model. Laminar flow would be equivalent to feeding a viscous slurry in the hydrocyclone. For both methods, the

angle calculations were performed for the same numerically obtained velocity data. In order for equal comparison, the viscosity in the computational domain was assumed to be constant for the Dueck method. Though the neglect of turbulence hydrocyclones is significant even with low inlet Reynolds numbers, a general behavior of the discharge angle can be seen to develop. Table 1 presents the numerical experiment by showing the changes in the Reynolds number at the inlet duct.

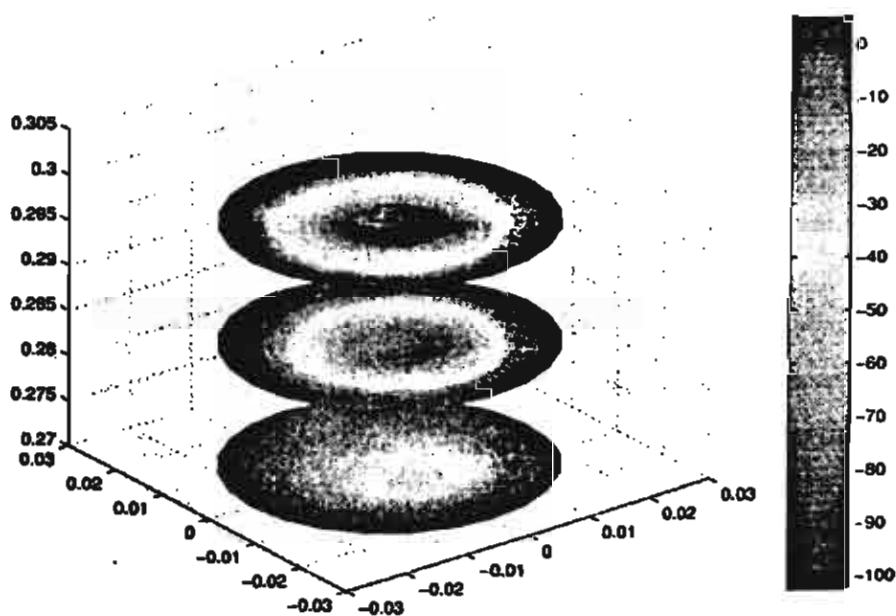


Fig. 3. Cross-section of the pressure profile at the height of 27, 28 and 29 cm.

The Reynolds number was calculated by $Re_{in} = \rho V D / \mu$, where ρ is the density, D the diameter and μ is the dynamic viscosity. The viscosities values decreased after the initial run to compare the change in the form of discharge. Due to the properties of the fluid, the hydrocyclone was operating in the roping region, as shown in Fig. 2.

For clarification, the solid-colored circular contour indicates the location of the exit. As can be seen from Fig. 2, the angle of the exiting fluid is predominately in the downward direction. The roping region can be distinguished from a spray discharge by observing the angle at which the slurry exits along the outer rim of the apex. The roping state tends to show that at the outer edge of the apex the discharge angle is closer to 90° than in spray discharge. However, operating in the experimented region, spray discharge does not occur.

Though in the model we did not take into account the air core, our model reveals the likely mechanism of air core creation. Atmospheric pressure at the center of the apex is the reference point for the pressure field in the hydrocyclone. As noticed in Fig. 3, a pressure reduction occurs toward the center of the hydrocyclone in each of the three different cross-sections. The lowest cross-section indicates that sediment has started to hinder the formation of the low pressure near the center.

As Fig. 3 shows, the bottom of the hydrocyclone has already been semi-plugged with sediment, thus not allowing the low-pressure field to develop at the tip. However, the examination of Fig. 4 reveals that the flow is moving away from the center at the apex.

Processing the data consists of using basic trigonometric functions and using Eq. (3) to calculate the discharge angle. The results of the angle calculated at different viscosities were compared between the AFN and Dueck methods and are listed in Table 2 below.

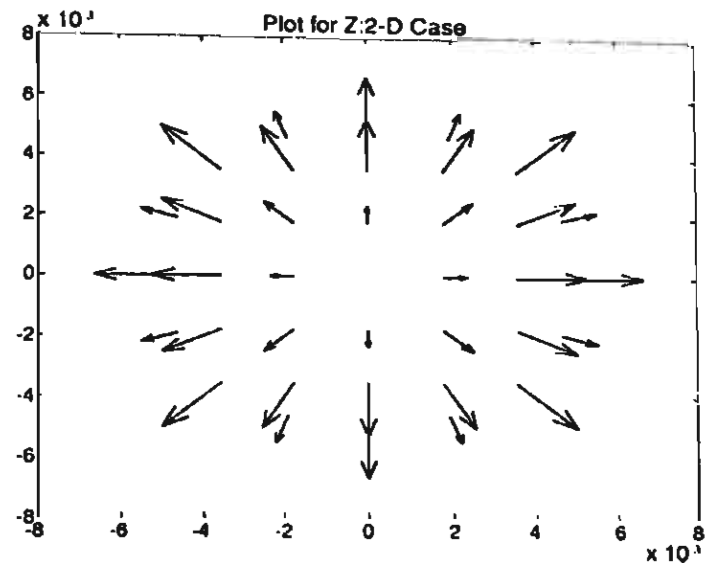


Fig. 4. The two-dimensional velocity profile projected onto the x - y plane cross-section at the apex.

The specific numerical data is not as important as the trend that is shown over the increasing viscosities. The comparison of the results obtained from both methods presents an interesting occurrence. The presented AFN method shows that the angle actually slightly decreases with increasing viscosity. The Dueck method predicts less of a change in the angle than the AFN method at corresponding outlet points. Since the operational state of the hydrocyclone is in roping, the expected angle exiting the spigot should be approximately 90°, which is predicted by both methods at constant viscosity. The possible cause of the inversion trend of the discharge angle at the apex in both methods could be due to the high pressure that occurs at the apex.

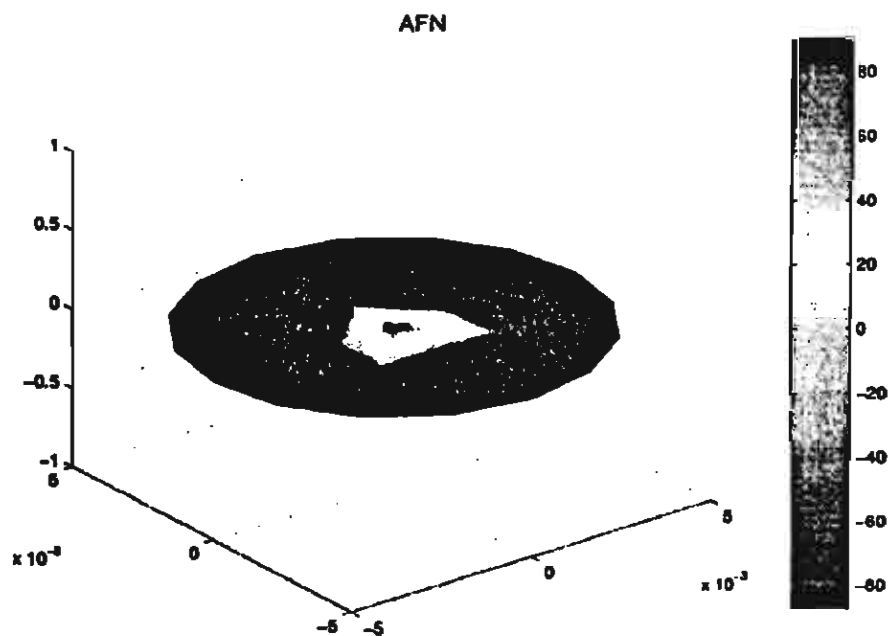


Fig. 5. Cross-section at the apex showing the value of the angle calculated using AFN method [13].

Table 2
Comparison of AFN to the Dueck method at the apex^{a,b}

$\mu = 0.09$		$\mu = 0.15$		$\mu = 0.2$	
AFN	Dueck	AFN	Dueck	AFN	Dueck
90	0	90	0	90	0
–87.8287	–89.9999	–86.4223	–89.9999	–85.5935	–89.8999
–85.5519	–89.9986	–83.7236	–89.9985	–82.7164	–88.9985
–80.1776	–89.9999	–77.7033	–89.9985	–76.4056	–89.9999
–78.4363	–89.9973	–75.8757	–89.9974	–74.5503	–89.9974

^a The other runs not presented did not differ excessively.

^b μ has units of Pa s and density is held constant.

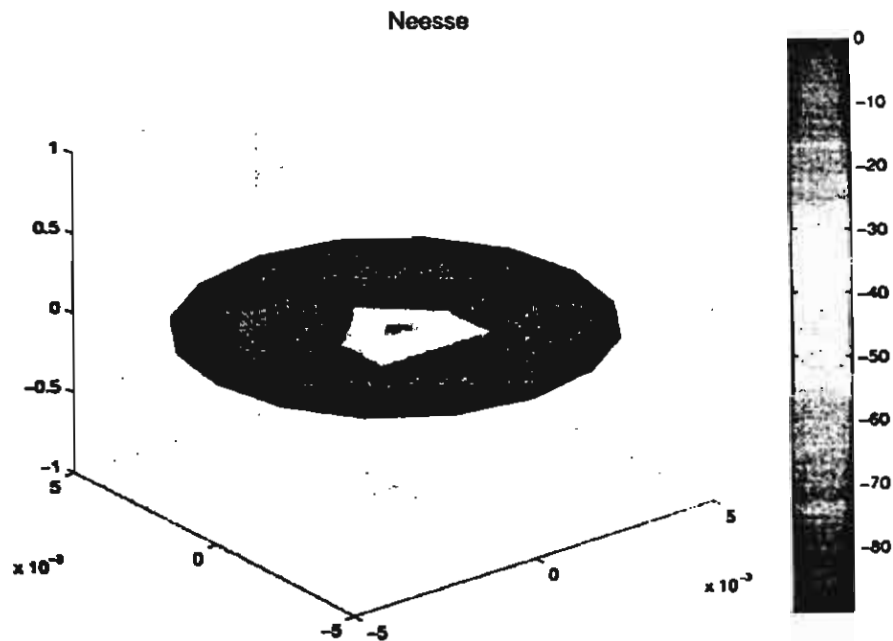


Fig. 6. Cross-section at the apex showing the value of the angle calculated using the Dueck method [7].

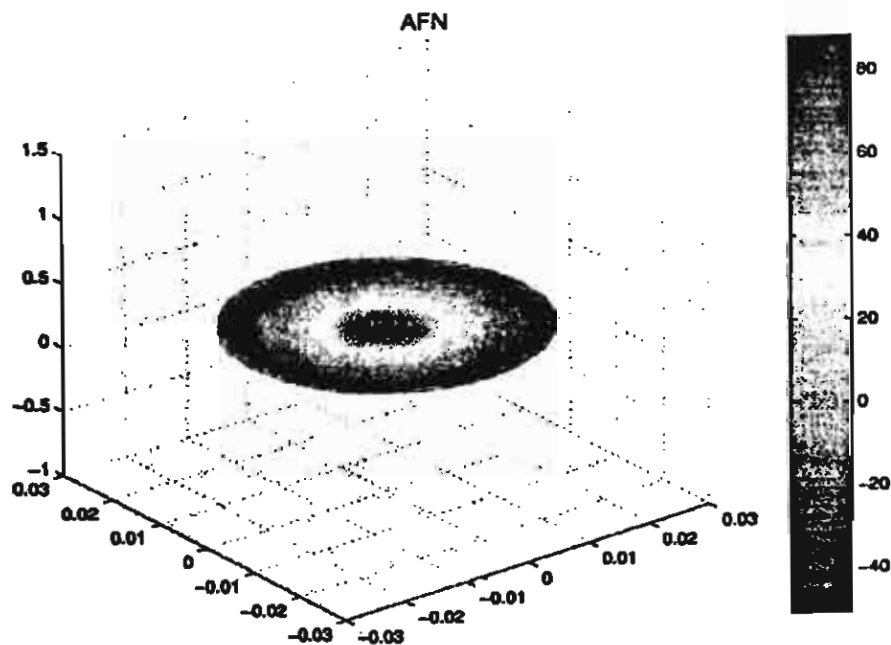


Fig. 7. Cross-section near the vortex finder showing the value of the angle calculated using AFN method [13].

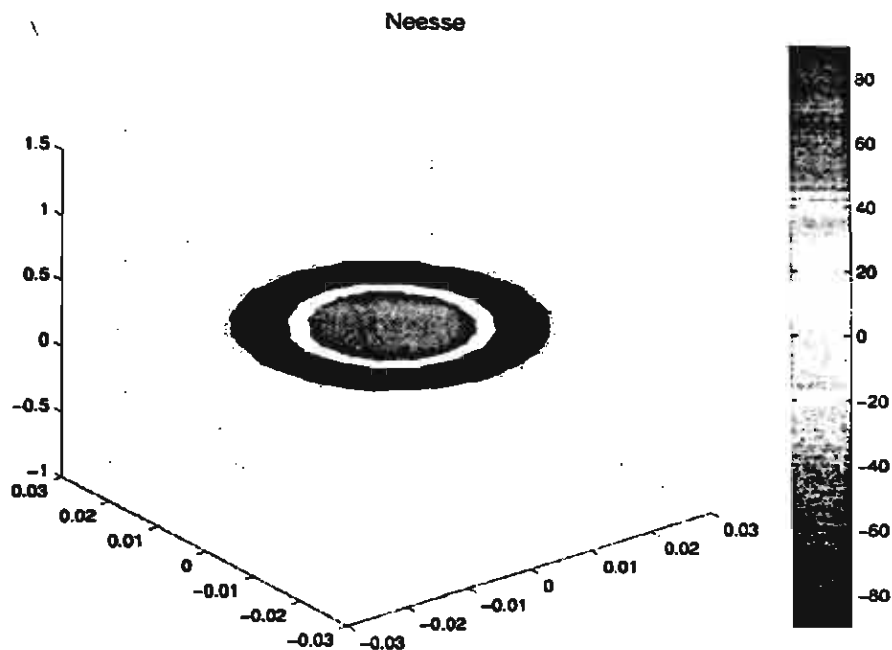


Fig. 8. Cross-section at the apex showing the value of the angle calculated using the Dueck method [11].

A comparison was made between the two different methods at the apex and right below the vortex finder exit, due to the complicated nature of the boundaries. The angles were calculated using both methods at the different cross-sections throughout the hydrocyclone. The following figures show two cross-sections at the apex using the AFN and Dueck methods.

The visual similarity between Figs. 5 and 6 are remarkable. Though at close examination of the legend, the figures illustrate that the gradient of the Dueck angle is steeper than the gradient of the AFN. However, our model predicts that at the center of the apex the actual discharge is in the upward direction, while Dueck's model does not take into account the other velocity vectors beside the tangential direction, w , and axial directions, z . The Dueck method does not predict an angle of discharge at the center of the apex, since the tangential velocity at the center is always zero for the case of axisymmetrical flow. A comparison at the top of the hydrocyclone provides additional insight into both methods.

Again, Figs. 7 and 8 show that the same basic trend of the flow is present. As shown previously, the gradient on the angle is more drastic in the Dueck model than the AFN model. An interesting occurrence happens in both cross-sections at the wall boundary. Both methods illustrate that the stresses from the boundary wall influence the angle. The predicted results qualitatively compared with the experimental results of Fisher and Flack [20], who showed that the shear stresses at the wall caused the velocity to approach or drop below zero.

4. Conclusion and future work

In conclusion, the study done on the prediction of the angle showed that both approaches tended to show similar an-

gle profiles. Even with the simplification of the equations from three- to two-dimensional, the Dueck method shows a comparable trend to the AFN method. The differences between the two methods is the small variations in the angle calculated with increasing viscosity, which may be due to the increased pressure at the high viscosity. Additionally, experimental verification will need to be done in order to confirm whether or not one method is more accurate than the other, though the Dueck method has already been shown to work for controlling an operational cyclone. However, the trend of both methods at the wall boundary condition coincides with experimental results from Fisher and Flack [20] makes the results look very promising for both methods. The advantage of using the proposed approach is that the velocity profile and split ratio is not explicitly specified initially, thereby allowing the flow field to develop naturally. In the future, the plan is to look at assuming the presence of a non-Newtonian fluid, which allows the viscosity to change through the system. The ultimate goal of the study is to use the analysis of the discharge angle as a tool to assist in hydrocyclone design.

References

- [1] K. Heiskanen, Experimental hydrocyclone roping models, *Chem. Eng. J.* 80 (2000) 289–293.
- [2] L. Plitt, A mathematical model of the hydrocyclone classifier, *Miner. Process. (CIM Bull.)* (1976) 114–123.
- [3] A. Lynch, T. Rao, Studies on the operating characteristics of hydrocyclone classifiers, *Indian J. Technol.* 6 (1968) 106–114.
- [4] Nageswararao, K., Wiseman, D., Napier-Nunn, T., 2003. Two empirical hydrocyclone models revisited. In: *Hydrocyclones 2003* in Capetown, South Africa.
- [5] L. Chu, Q. Luo, Hydrocyclone with high sharpness of separation, *Filtr. Sep.* 31 (1994) 733–736.

- [6] F. Concha, A. Barrientos, J. Montero, R. Sampaio, Air core and roping in hydrocyclones, *Int. J. Miner. Process.* 44–45 (1996) 743–749.
- [7] T. Neesse, M. Schneider, J. Dueck, V. Golyk, S. Buntentbach, H. Tiefel, Hydrocyclone operation at the transition point rope/spray discharge, *Miner. Eng.* 17 (5) (2004) 733–737.
- [8] Van Latum, L., 1992. Computed tomographic imaging of a dense media cyclone. In: *Applied Research in the Minerals Industry (JKMRC)*.
- [9] R. Williams, Measurement and modelling of slurry mixing using resistance tomography, in: *Proceedings of the XIX International Mineral Processing Congress, Comminution and Simulation and Control*, Society for Mining, Metallurgy, and Exploration, Inc. (AIME), 1995.
- [10] K. Petersen, C. Aldrich, J. Van Deventer, C. McInnes, W. Stange, Hydrocyclone underflow monitoring using image processing methods, *Miner. Eng.* 9 (3) (1996) 301–315.
- [11] T. Neesse, M. Schneider, V. Golyk, H. Tiefel, Measuring the operating state of a hydrocyclone, *Miner. Eng.* 17 (5) (2004) 697–703.
- [12] J. Van Deventer, D. Feng, K. Petersen, C. Aldrich, Modelling of hydrocyclone performance based on spray profile analysis, *Int. J. Miner. Process.* 70 (2003) 183–203.
- [13] A. Nowakowski, T. Dyakowski, Investigation of swirling flow structure in hydrocyclones, *Trans. IChemE, Part A, Chem. Eng., Res. Des.* 81 (A4) (2003) 862–873.
- [14] P. Gresho, Incompressible fluid dynamics: some fundamental issues, *Annu. Rev. Fluid Mech.* 23 (1991) 413–453.
- [15] Nowakowski, A., Kraipech, W., Dyakowski, T., 2003. Analysis and Simulation of Multifield Problems. Springer, Ch. Performance of some finite elements in numerical simulation of complex incompressible three dimensional flow.
- [16] P. Gresho, R. Sani, *Incompressible Flow and the Finite Element Method*, Wiley, 1998.
- [17] V. Haroutunian, M. Engelman, I. Hasbani, Segregated finite element algorithms for the numerical solution of large-scale incompressible flow problems, *Int. J. Numer. Methods Fluids* 17 (1993) 323–348.
- [18] P. He, M. Salcudean, I. Gartshore, A numerical simulation of hydrocyclones, *Trans. IChemE* 77 (1999) 429–441.
- [19] L. Ma, D. Ingham, X. Wen, A numerical technique for dealing with the axis in simulating the fluid flows in polar cylindrical coordinates, *Numer. Methods Laminar Turbulent Flows* 10 (1997) 203–214.
- [20] M. Fisher, R. Flack, Velocity distributions in a hydrocyclone separator, *Exp. Fluids* 32 (2002) 302–312.

Appendix C

Title: The experimental study of the flow within a transparent hydrocyclone with a vortex finder attachment

Authors: Wanwilai Kraipech and Andrew F. Nowakowski

Presentation: American Filtration and Separations Society Annual Conference 2005 at Atlanta, GA, 10-13 April 2005.

THE EXPERIMENTAL STUDY OF THE FLOW WITHIN A TRANSPARENT HYDROCYCLONE WITH A VORTEX FINDER ATTACHMENT

Wanwilai Kraipech ^{1*} and Andrew F. Nowakowski²

¹ Department of Chemical Engineering, Srinakharinwirot University Ongkharuk, Nakornnayok 26120, Thailand. Tel.: +66(0)6 999 8908, Fax: 0066(0)37 322 608, E-mail: wanwilai@swu.ac.th

² Department of Mechanical Engineering, University of Sheffield, Mappin Street, Sheffield, S1 3JD, England. Tel.: +44(0)114 222 7812, E-mail: a.f.nowakowski@sheffield.ac.uk

ABSTRACT

The experimental studies of the flow behaviour within 50-mm hydrocyclones with and without a vortex finder attachment (Fin) have been performed. The flow characteristics within both hydrocyclones were observed visually and recorded both photographically and on video as the hydrocyclones are made of a transparent material. The development of the double-helical flow was observed; also the shape and diameter of the air-core have been examined for different values of operating pressure drop. It was found that the air-core is unstable and its size, shape and position are unfixed because of the instability of the gas-liquid interface. The results also showed that the air core is related to the shape of the underflow discharge and is a function of the pressure drop. A regular spray discharge from the spigot was observed. The roping discharge was observed when an extension pipe was connected to the underflow outlet. An increase in pressure drop leads to an increase in air-core diameter and the angle of the underflow profile. However, the relationships between these three parameters are non-linear. The knowledge of the size and location of the air-core relative to the operating pressure drop can assist in the validation of computational fluid dynamics (CFD) simulation work.

The separation performance of these two transparent hydrocyclones for a CaCO₃ in water system has been investigated. In this study, the results showed that a fin-type vortex finder attachment could not increase the sharpness of separation but it could help in the thickening process. It can be seen that the separation performance does not depend only on the geometry and operating conditions, but also on slurry concentration and particle size distribution. The concentration of the feed slurry and the operating pressure drop strongly affect the sharpness of separation.

KEY WORDS: Hydrocyclone, Solid-liquid system, Separation, Air-core

1. INTRODUCTION

Hydrocyclones are widely used in the mineral, chemical, coal, powder-processing industries for de-watering, or classifying according to a required particle size or density separation. The

reasons for this popularity lie in their simplicity of design and operation, high throughput, low maintenance, low operating cost and small physical size of the unit. Hydrocyclones are physically simple and robust separation devices with no moveable parts. A typical hydrocyclone consists of three sections; a cylindrical section, conical section and a tangential inlet section. The tangential inlet section is attached to the cylindrical section. The vortex finder is located centrally through the lid and at the top of the cylindrical section. Figure 1a shows a conventional hydrocyclone, which is inherently simple in construction. The design variables which influence the performance of the unit operation are the inlet and outlet diameters. The separation process is achieved as a result of the centrifugal force acting on the slurry and its components. However, there is a secondary flow down the outside wall of the vortex finder to the overflow, which is called a short-circuit flow. This flow was reported to cause low particle separation efficiency [1]. Svarovsky [2] suggested that a redesigned-vortex finder with a skirt can eliminate this inefficiency problem.

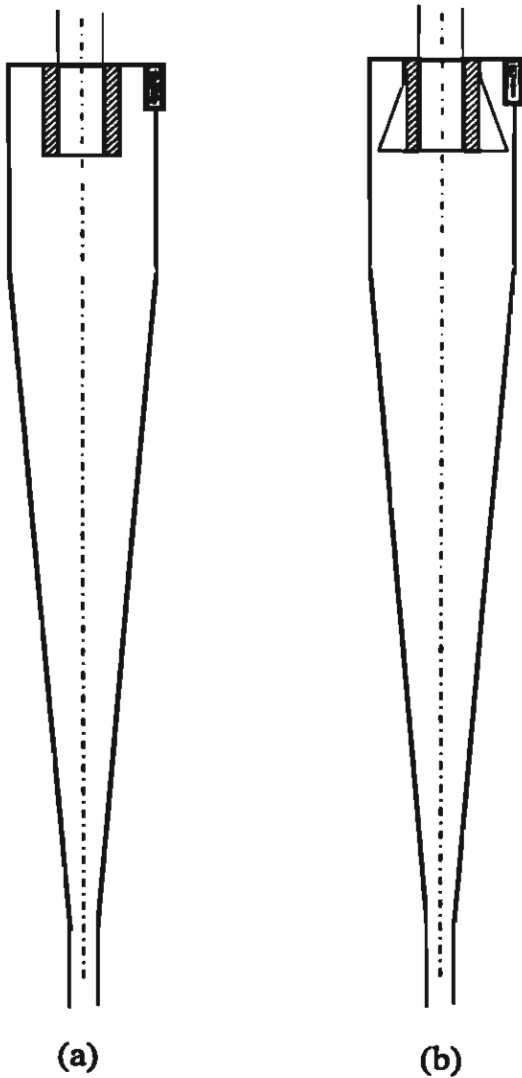
Despite its apparent simplicity, the elucidation of the mechanism of separation and detailed models of fluid flow with hydrocyclone separators has yet to be achieved. The main reasons for these short comings are: the complexity of solving Navier-Stokes equations for non-Newtonian anisotropic fluids that contain high concentrations of interacting particles, The complex high swirling and varying turbulence conditions prevailing in the separator, the presence of an interface (the air-core) inside the separator and the lack of on-online measurement methods to provide reliable experimental data for model development under credible process conditions [3].

An attempt to model the flow in a hydrocyclone has been made by using a theoretical model, which is based on solving the Navier-Stokes equations. This approach provides a physical insight into the fundamental causes of the observed phenomena. The earlier theoretical models were presented by [4], [5], [6], [7] and [8]. These steady state and 2D-axisymmetrical models have been considered, but the results are limited to dilute flow only. In addition, it is difficult to describe the behaviour of high turbulent swirling flow caused by the 3D flow entry. Therefore, more advanced modelling is needed that allows, for example, the study of such phenomena as an adjustment of three-dimensional flow to axisymmetrical, particle-fluid, particle-particle and particle-wall interactions. Such models will allow the description of particle effects on suppressing or generating turbulence and non-Newtonian slurry flows. Additionally, in the context of modelling turbulence, a physical model is needed to show how a fluid turbulent deformation characterises swirl flows and the deformation of the air-core inside the separator. Advanced theoretical and experimental techniques are needed to obtain a better understanding of the complex physical phenomena affecting the performance of hydrocyclones.

The purpose of this work is to study the flow behaviour, focusing on the air-core and underflow discharge characteristics, by using 50-mm transparent hydrocyclones with and without a vortex finder attachment (Fin). The experiments of these hydrocyclones treating CaCO_3 in water system are carried out. The knowledge of the underflow discharge characteristics relative to the operating pressure drop and feed property affecting the separation performance is investigated.

2. EXPERIMENTAL SYSTEMS

Figure 1 shows the schematic diagram of both 50-mm diameter acrylic hydrocyclones with and without a vortex finder attachment. They were equipped and set up with a feed pump and pressure gauge to measure the feed inlet pressure (see Figure 2). The vortex finder and apex diameters were kept constant at 14mm and 10 mm. Hydrocyclone overflows and underflows were directed back to the sump for recirculation. To study the flow behaviour and the characteristics of the air core, water was used. The operating pressure drop was varied from 10 psi to 35 psi by varying the inlet water flow rate. The density and the viscosity of water are 1031 kg/m^3 and 1.330 cp , respectively. The size and the location of the air core were observed relative to the pressure drop. From each experiment, samples were collected from the feed, overflow and underflow streams. The collected samples were weighed and mass flow rates determined.



Geometry	Value (mm)
underflow internal diameter	10
vortex finder internal diameter	14
vortex finder outer diameter overflow	28
cylindrical section diameter	45
conical section length	270
cylindrical section length	40
width of inlet orifice	22
length of inlet orifice	40
height of inlet orifice	16
length of fin	6
height of fin	20

Figure 1. Schematic diagram of 50-mm diameter acrylic hydrocyclones (a) conventional hydrocyclone and (b) with a vortex finder attachment (Fin).

The separation performance was examined by using the solution of CaCO_3 in water as feed slurry. The density of CaCO_3 is 2700 kg/m^3 . The flow rate and concentration of the feed was varied. In this case, the solid phase in each stream was also weighed to determine the concentration. The particle size distribution of each stream was measured by using the laser-diffraction size-analysis technique.

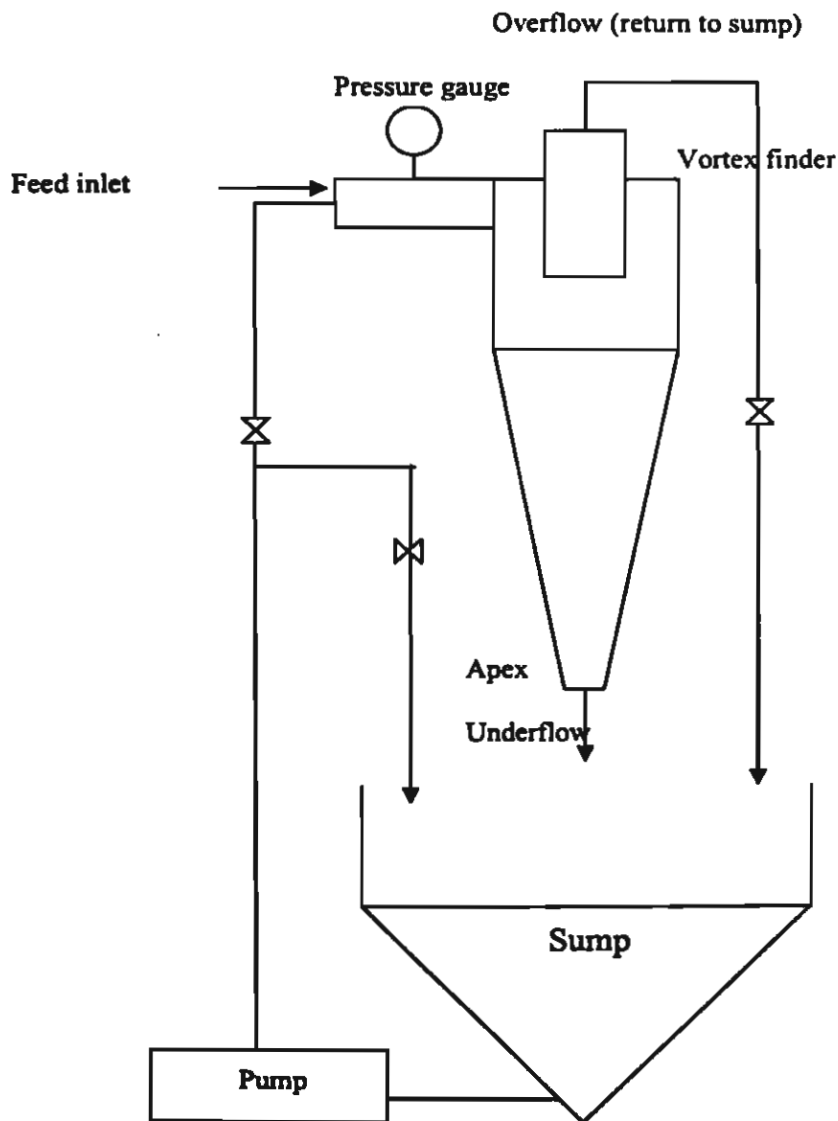
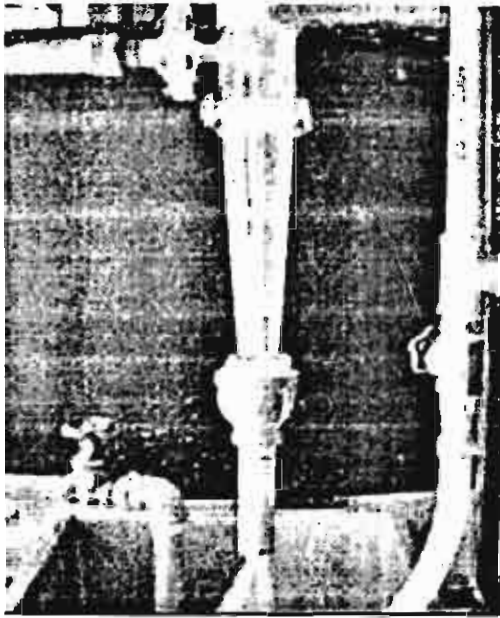
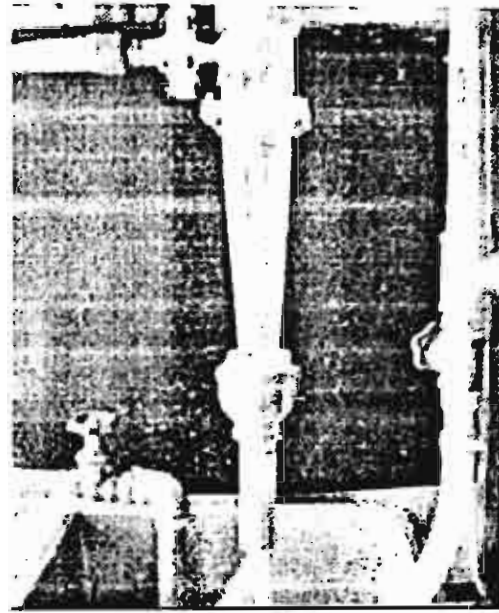


Figure 2. Hydrocyclone apparatus.

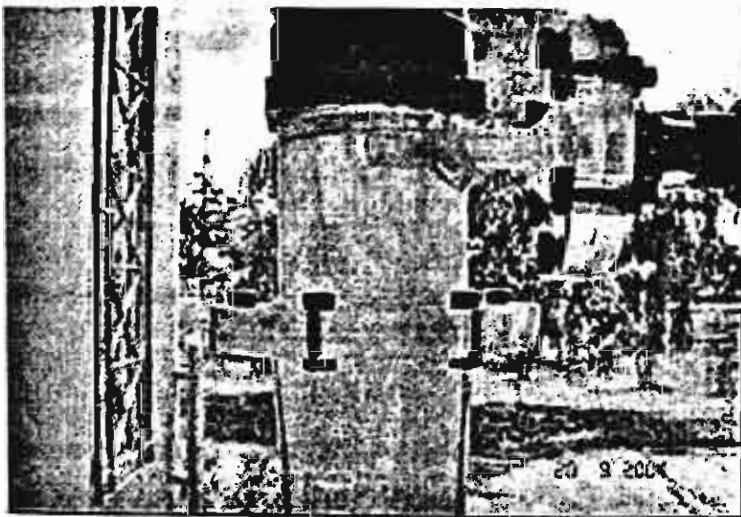


(a)

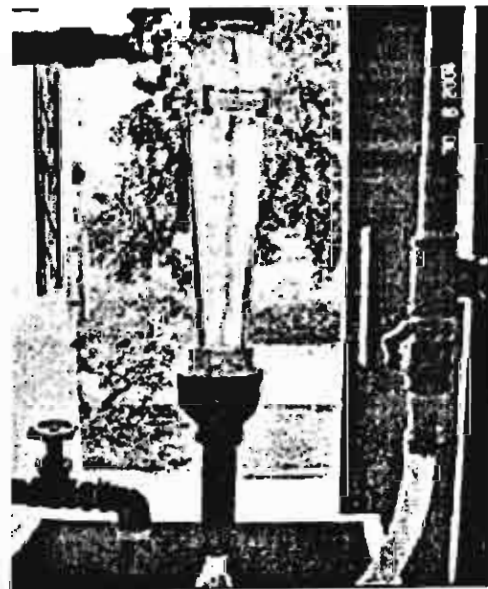


(b)

Figure 3. The flow pattern within the conventional hydrocyclone operating at: a) 10 psi b) 30 psi.



(a)



(b)

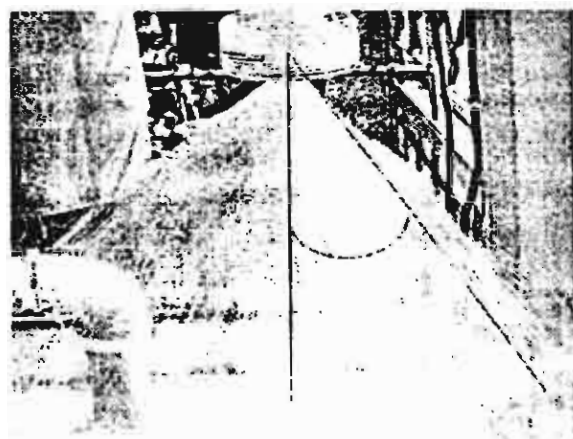
Figure 4. The flow pattern within the hydrocyclone with a vortex finder attachment operating at 10 psi.

The diameter of the air-core of a conventional hydrocyclone for water-only flow was measured and found to be increased due to an increase in the pressure drop. It is also related to the angle of the underflow profile.

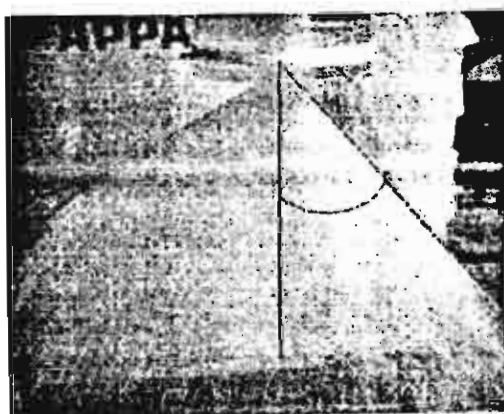
In addition, the angle of the underflow discharge of a conventional hydrocyclone treating CaCO_3 slurries was observed for 1%, 2% and 5% by volume of solid in feed as shown in Figure 5.



(a)



(b)



(c)

Figure 5. The underflow discharge of the hydrocyclone operating at 30 psi for feed solid concentrations of: a) 1% b) 2% and c) 5% by volume.

A regular spray discharge from the spigot was observed in every experiment. The shape of the underflow discharge was found to be altered according to the change of feed-solid concentration and operating pressure drop as shown in Figure 6. Figure 6 also depicts the relationships between the air-core diameter, the angle of the underflow discharge and the pressure drop. It was found that an increase in pressure drop leads to an increase in the air-core diameter. However, in low-solid concentration cases, which are 1 and 2% by volume of

solid in feed, it was found that an increase in pressure drop leads to an increase in the angle of the underflow profile. For higher solid concentration, which is 5% by volume, it was found that above a certain operating pressure (25 psi), the discharge angle decreases due to the increase in pressure drop. This is due to the sedimentation of solids in the underflow section caused by high operating pressure in dense-flow separation.

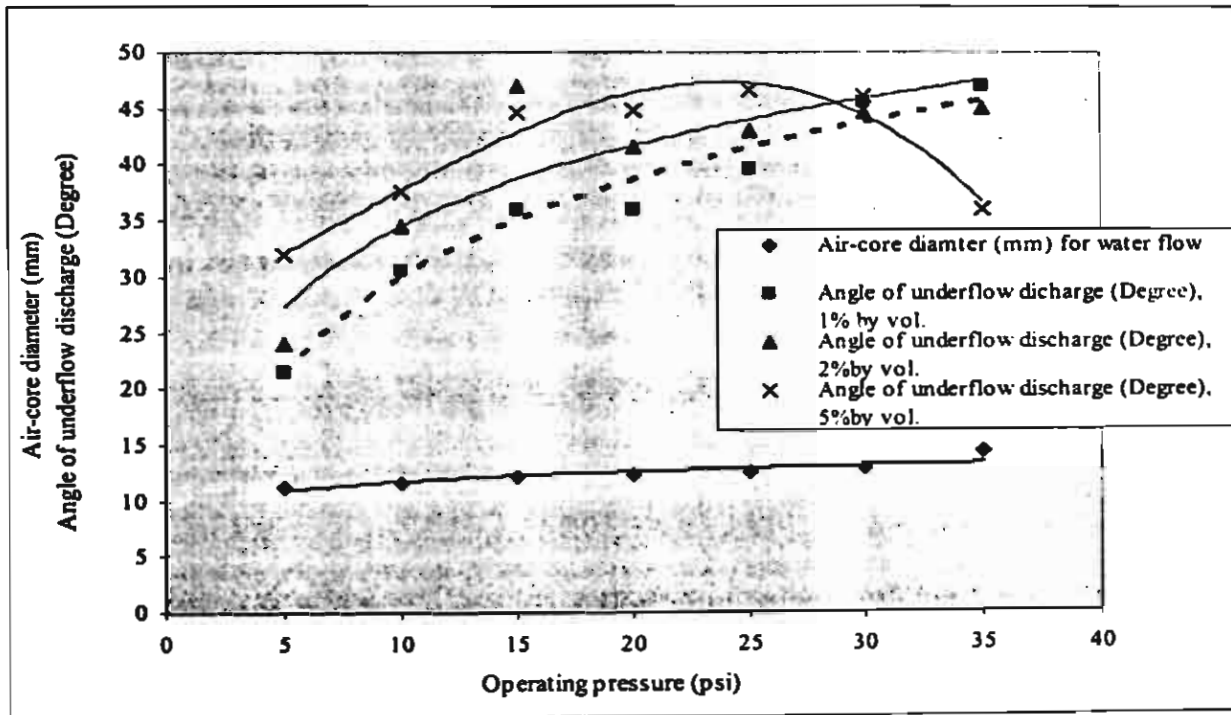


Figure 6. The relationship between the pressure drop, the air-core diameter and the angle of underflow discharge.

The solid concentration has a very strong effect on the angle of underflow discharge. The relationships between these three parameters are non-linear.

3. 2. Separation performance

The separation performance of this hydrocyclone with the extension pipe, treating CaCO_3 slurry, was examined. The operational data (pressure drop), the concentration and flow rate of each stream, the cut size and throughput ratio (R_f) of each test are shown in Table 1. The throughput ratio, R_f , is the ratio between the volumetric flow rate of the underflow and that of the feed. The particle size distribution of the feed is shown in Figure 7. The selectivity curves are presented in Figures 8-11.

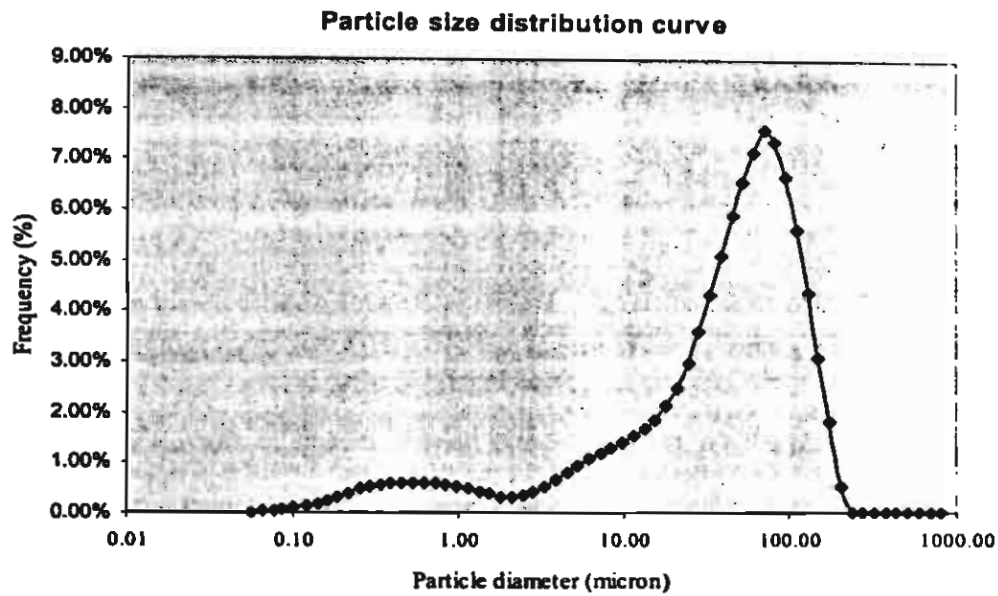
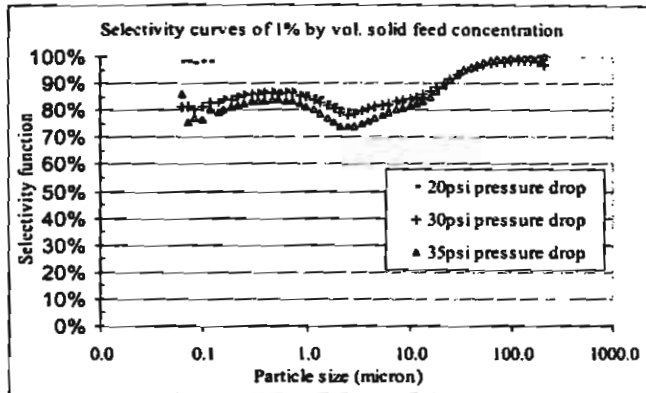


Figure 7. Particle size distribution curves.

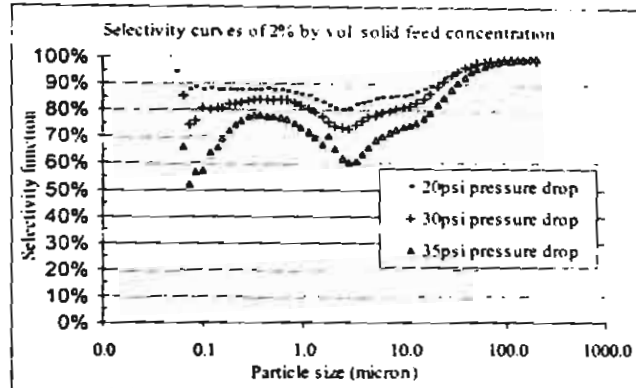
Table 1. Experimental results.

Test No.	Pressure Drop Δp (psi)	Feed		Over flow		Under flow		d_{50} (μm)	R_f
		% by vol.	(gal/min)	% by vol.	(gal/min)	% by vol.	(gal/min)		
I	20	1	8.98	0.096	5.38	2.548	3.60	-	0.40
II	30	1	10.89	0.133	6.68	3.559	4.21	-	0.38
III	35	1	11.74	0.107	7.35	3.362	4.39	-	0.37
IV	20	2	8.64	0.207	5.08	5.088	3.56	-	0.41
V	30	2	10.53	0.218	6.54	6.321	3.99	-	0.38
VII	35	2	11.81	0.216	7.42	3.632	4.39	-	0.37
VII	20	5	8.47	3.030	4.91	11.340	3.56	40	0.42
VIII	30	5	11.28	3.415	6.93	14.163	4.35	11	0.38
IX	35	5	11.56	0.978	7.21	8.200	4.35	2	0.37
X*	30	0.20	9.62	3.74	3.15	3.95	6.47	-	0.67
XI*	30	0.37	9.32	0.78	1.45	3.92	7.87	-	0.84
XII*	35	1	10.01	0.22	3.62	0.87	6.39	-	0.64
XIII*	35	2	10.61	0.38	3.95	1.43	6.66	-	0.63

* Hydrocyclone with vortex finder attachment (Fin).



(a)



(b)

Figure 8. Selectivity curves of the feed solid concentration: a) 1% and b) 2% by volume for 20, 30 and 35psi.

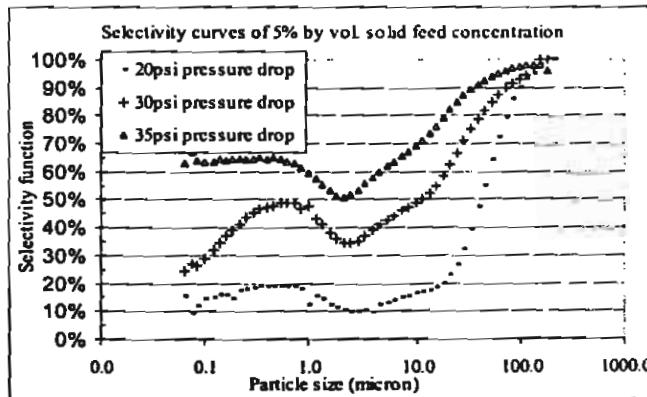


Figure 9. Selectivity curves of the feed solid concentration of 3% by volume for 20, 30 and 35psi.

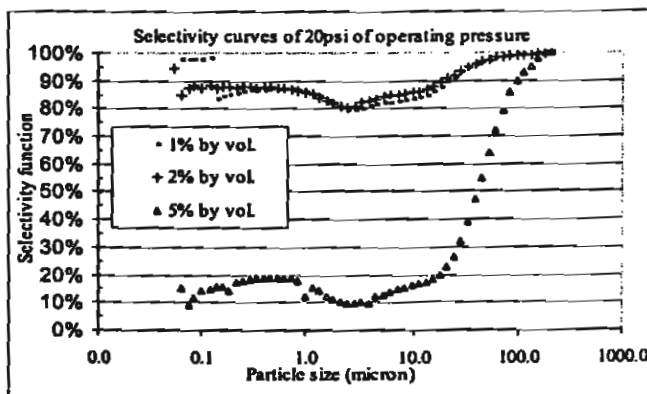


Figure 10. Selectivity curves of the operating pressure of 20 psi for 1, 2 and 5% by volume of solid in feed.

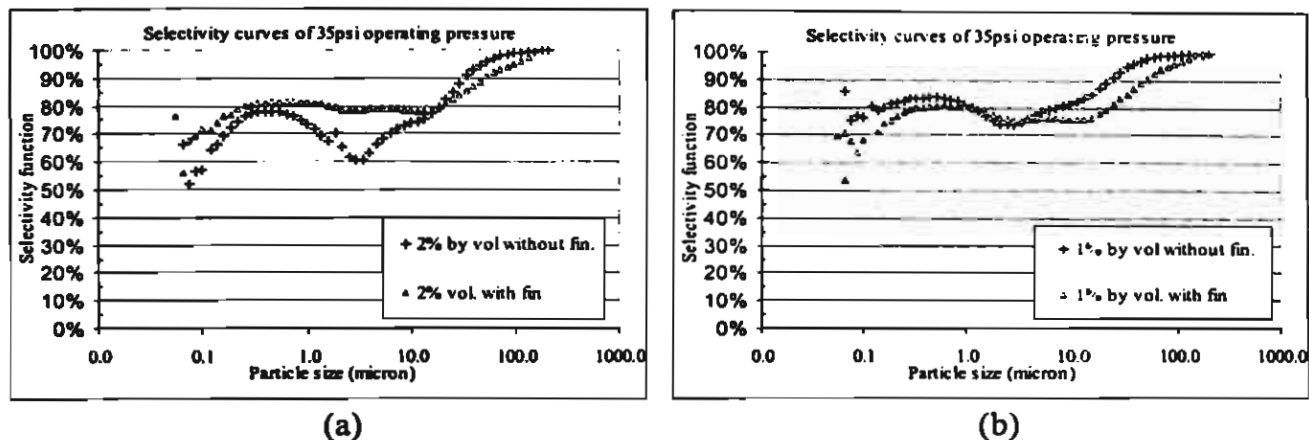


Figure 11. Selectivity curves of the operating pressure of 35 psi for: a) 1% and b) 2% by volume of solid in feed for the hydrocyclone operated with and without fin.

In this study, the operating pressure is limited. The highest pressure drop is 35 psi. In this range, it was found that the performance of the 50-mm hydrocyclone unit treating CaCO_3 slurry at 1 and 2% by volume was poor. The cut size cannot be obtained as can be seen in Figure 8a and 8b. However, the solid concentration in the underflow was three times higher than the concentration in the feed. The separation performance of the hydrocyclone treating the feed in this concentration range could be improved for a higher pressure drop.

The selectivity curves of the operation at the feed concentration of 5% by volume are shown in Figure 9. The separation performance of the hydrocyclone on this feed solid concentration is good. The results showed that the operating pressure of 35psi gave the smallest cut size. It is shown that the change in pressure drop obviously affects the hydrocyclone separation performance since an increase in pressure drop increases all velocities throughout the hydrocyclone. Therefore, an increase in pressure drop causes a decrease in the cut size.

The pressure drop used in practice usually depends on economic considerations. Operating at high-pressure means less units are required to treat a given flow. Lower capital costs, finer cut sizes and sharper separations can be obtained. These benefits must be offset against drawbacks including higher pumping cost and increased abrasion. Furthermore, the increase in feed flow-rate and the decrease in cut size tend to decline above a certain pressure drop. This is thought to be due to resistance effects within the hydrocyclone [9]. More details about the effects of the feed properties and the operational conditions on the separation performance can be found in [10].

The cut size of the 50-mm hydrocyclone unit treating CaCO_3 slurry also strongly depends on the feed concentration. It also appeared that the angle of the underflow discharge at an operating pressure of 35 psi was the smallest angle compared to those at 20 and 30 psi. The discharge of the separator contains the information which could possibly be used for better operational control, assuming that two distinct operational states of the hydrocyclone exist, which are the following: spray and roping. The states differ by the discharge profile from the

apex. Roping tends to form a rope-like discharge, while spray resembles an umbrella. Concha et al. [11] points out that the best separation occurs near the formation of rope discharge. Similarly, Neesse et al. [12] states on a broader note that a hydrocyclone achieves the best separation at a transitional discharge phase between roping and spraying. Therefore, with the calculation of the angle, determination of the operating performance of the hydrocyclone can be predicted. It is of interest to establish a procedure for improving the operation of hydrocyclones based on computational simulations.

A characteristic dip or fish-hook effect was observed in all experiments. It started to occur at particle sizes of approximately 2 microns. The analysis of this phenomenon is described in [10].

The separation performance of the hydrocyclone with the fin attachment was investigated for the feed concentration of 0.20, 0.37, 1 and 2% by volume. The results of the dilute feed (0.20 and 0.37% by volume) showed that the concentration of the underflow stream was ten to twenty times higher than that of the feed, but the selectivity curve showed low sharpness of separation and the cut size could not be obtained. The fin attachment causes an increase in the throughput ratio. The results of the 1 and 2% by volume of the feed concentration showed very poor results as shown in Figure 11. The operation with 5% by volume of the feed slurry was also examined, and it was found that there was no overflow stream. This is due to the effect of the fin attachment forcing the flow swirling down to the apex tip and the high viscosity of the high concentration feed causing higher drag force, which reduces the effect of the centrifugal force. It can be summarised that the fin-type vortex finder attachment could not increase the sharpness of separation but it could help in the thickening process for a dilute feed system.

4. CONCLUSIONS

The results demonstrate that the air core is related to the type of underflow discharge and is a function of the pressure drop, which has an effect on the separation performance of the hydrocyclone. It can be seen that the separation performance does not depend only on the geometry and operating conditions, but also on slurry concentration. The results of this study are useful since they offer an ideal for improving hydrocyclone design and will assist in the validation of computational fluid dynamics (CFD) simulations.

5. ACKNOWLEDGMENTS

Acknowledgements are due to Ms. Nareerut Ju and Ms. Doungjai Tumma for experimental work and to the Thailand Research Fund for financial support.

6. REFERENCES

- [1] Kelsall, D. F. "A study of the motion of solid particles in a hydraulic cyclone", Transactions of the Institution of Chemical Engineers, Vol. 30, pp. 87-108, 1952.
- [2] Svarovsky, L. "A short course in cyclones", Manual to Course at the Dow Chemical Co. in Freeport/Texas, January 10-11, 1994.
- [3] Bond, J., Cullivan, J. C., Climpson, N., Dyakowski, T., Faulks, I., Jia, X., Kostuch, J. A., Payton, D., Wang, M., Wang, S. J., West, R. M. and Williams, R. M. "Industrial monitoring of hydrocyclone operation using Electrical Resistance Tomography", 1st World Congress on Industrial Process Tomography Buxton, Greater Manchester, April 14-17, 1999.
- [4] Bloor, M. I. And Ingham, D. B. "Theoretical investigation of the flow in a conical", Transactions of the Institution of Chemical Engineers, Vol. 51, pp. 36-41, 1973.
- [5] Bloor, M. I. And Ingham, D. B. "Turbulent spin in a hydrocyclone", Transactions of the Institution of Chemical Engineers, Vol. 53, pp. 1-6, 1975.
- [6] Bloor, M. I. And Ingham, D. B. "The flow in industrial cyclones", J. Fluid Mech, Vol. 178, pp. 507-519, 1987.
- [7] Pericleous, K. A. and Rhodes, N. "The hydrocyclone classifier: A numerical approach", International Journal of Mineral Processing, Vol. 17, pp. 23-43, 1986.
- [8] Hsieh, K. T. and Rajamani, R. K. "Mathematical model of hydrocyclone based on physics of fluid flow", AIChE Journal, Vol. 37. No. 5, pp. 735-745, 1991.
- [9] Bradley, D. "The Hydrocyclone", Oxford: Pergamon Press, 1965.
- [10] Kraipech, W. "Studying performance of industrial hydrocyclones", Ph.D. Thesis, UMIST, UK, 2002.
- [11] Concha, F., Barientos, A., Munoz, L., Bustamante, O. and Castro, O. "A phenomenological model of a hydrocyclone", in Claxton, D., Svarovsky, L. and Thew, M. (Eds), *Hydrocyclones '96*, London: Mechanical Engineering Publications Limited, pp. 63-82, 1996.
- [12] Neesse, T., Schneider, M., Dueck, J., Golyk, V., Buntenbach, S., Tiefel, H. "Hydrocyclone operation at the transition point rope/spray discharge". In: *Hydrocyclones 2004 in South Africa*, 2003.