Study of Mist Elimination System

By

Mohd Safuan Abd Rahman

A project dissertation submitted in partial fulfillment of
the requirements for the
Bachelor of Engineering (Hons)
(Mechanical Engineering)

JAN 2009

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Study of Mist Elimination System

by

Mohd Safuan Abd Rahman

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfilment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(MECHANICAL ENGINEERING)

Approved by,

(Prof. Vijay Raghavan)
Project Supervisor

Vijay R. Raghavan
Professor
Mechanical Engineering Department
Universiti Teknologi PETRONAS

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

January 2009

CERTIFICATION OF ORIGINALITY

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

MOHD SAFUAN ABD RAHMAN

ACKNOWLEDGEMENT

First and foremost, thanks to God because of His bless, the Final Year Project is completed successfully within the designated time frame. The experiences, knowledge and skills gained from this project are very useful and beneficial in a journey to be a wise and charismatic engineer in future.

Next, a special gratitude is expressed to Prof. Vijay Raghavan for his guidance, patience and supervision in completing this project. This special gratitude is also forwarded to Dr. Ahmed Maher for sharing his experiences and lends me their expertise especially in Fluent.

Not to be forgotten, to the entire author's friends who had shared their experiences and giving a lot of advices for the whole period of project. To those who had assist direct or indirectly in making this project into success, highest appreciation is given to all of them.

Finally, the author dedicates this work to family for their support, encouragement and understanding until completion of this dissertation report. Wassalam.

ABSTRACT

A mist eliminator or demister is to be designed and developed to remove excessive moisture content from a vapor stream in a certain flow or process. Mist eliminator consists of a series of profiled baffle plates that remove liquid droplets from a vapor stream by the inertial impaction of the droplets on the baffle [1] while the gas negotiates the zigzag flow path between the baffles. This study is to understand the mechanism of mist elimination system by a detailed study of the flow by computational fluid dynamics (CFD) and at the end, propose a good demister design. The proposed demister design should provide efficient droplets removal with acceptable pressure drop. Due to the complex geometry of mist eliminator design, purely analytical work is impossible, hence leaving only experiments and numerical analysis as the method of choice. The scope of study in this project is to make use of CFD software available to analyze fluid dynamics through the complex geometry of the vanes in a mist eliminator. In this study, a computational fluid dynamic study will be carried out using commercial CFD software FLUENT 6.3 while the geometry is created using GAMBIT 2.2.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1
INTRODUCTION

1.1 BACKGROUND OF STUDY

It is sometimes vital to remove droplets from gas streams as this liquid drops could cause damage to downstream equipment. Demisters are devices that can accomplish this job effectively. Demisters are commonly used in gas-liquid operations such as distillation, absorption, and evaporation. This study will focus on analyzing and upgrading a commercial model of demister, which is in used in humidity conditions to remove water droplets from entering the air intake of gas turbine (GT) compressor.

The GT compressors that operate in high humidity areas (offshore) require demister to remove entrained liquid in the air to avoid soaking of the air filters at their intakes. The demisters are designed in such a way that humid air will have to travel through angled passages so that water droplets can be trapped, usually by inertial impingement [1] and drained out. A good design is necessary to ensure efficient trapping of water droplets without compromising the pressure drop.

The performance of demister depends on many variables such as orientation of the device, vapor velocity, material of construction and design of the passages. The latter will be the subject of interest in this study. To assess the performance of demister, two variables are taken for close look, which are the demister's efficiency in term of percentage of accumulated liquid and the pressure drop penalty. Both will be predicted by CFD.

1.2 PROBLEM STATEMENT

Removal of liquid droplets in a humid area by the proposed mist elimination system by means of CFD analysis and optimize it in terms of liquid removal efficiencies and its pressure penalty.

1.3 SIGNIFICANCE OF STUDY

The demister efficiency, η, is the separation of liquid from gas that occurs due to the effect of the design of the passages. This is defined by Eq. (1)

$$\eta = \frac{M_{in} - M_{out}}{M_{in}} \times 100 \qquad (1)$$

The other quantitative demister performance is pressure drop or decrease in pressure from the inlet to the outlet pressure. Large pressure drop across a demister will require large energy to be used to pump or blow in the gas to flow through a demister.

1.4 OBJECTIVES

The aim of this project is first, to understand the mechanism of mist elimination system and physics behind it through literature review of past papers. Second is to study the efficiency of an existing demister design and further develop it without compromise its pressure drop. The latter will be achieved by the mean of CFD. Since the simulation involving multiphase flow which is new in UTP, hopefully it will lead further study by others.

CHAPTER 2
THEORY/ LITERATURE REVIEW

2.1 SCOPE OF WORK

Geometry of a demister's vanes from a commercial model will be constructed in a similar demister tower as used in R. Rahimi, D. Abbaspour in their research [7]. It will be then analyzed using CFD and its performance will be evaluated by two variables; pressure drop and the efficiency of accumulated liquid from the gas that flows through it. The result will then be compared to R. Rahimi, D. Abbaspour research [7]. The geometry of the vanes will then be changed to study the effect to the efficiency as well as the pressure drops.

2.2 IMPORTANCE OF MIST REMOVAL

In some processes in the chemical, oil and gas industries, it is sometimes necessary to remove liquid droplets from gas or vapor stream [1]. Separation between this two phases is vital for several reasons, such as to improve product purity, to reduce environmental pollution, to protect downstream equipment from corrosion, to increase recovery of valuable liquids dispersed in a process gas stream, to eliminate hazardous liquid from reactive gases, in general cutting the operating cost [2][9][10].

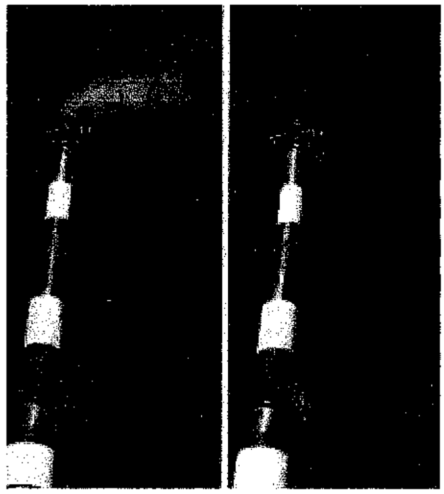

Figure 1: After Installation of Mist Eliminator (no more visible stack plume)

2.3 MIST ELIMINATOR

Mist eliminator has been used since 1947 to remove mist from gas streams for previously stated reasons[10]. Mist eliminator can be categorized into six groups which are settling tanks, fiber filtering candles, electrostatic precipitators, cyclones, impingment van separator, and wire mesh as shown in Figure 1.[10]. They are typically used in distillation of fractionation, absorption, evaporation and boiling, gas scrubbing, evaporative cooling, trickle equipment for sewage[7][10]. A good mist eliminator should have low cost, minimum pressure drop, high separation efficiency, high capacity, low tendency of flooding phenomena, and small in size.

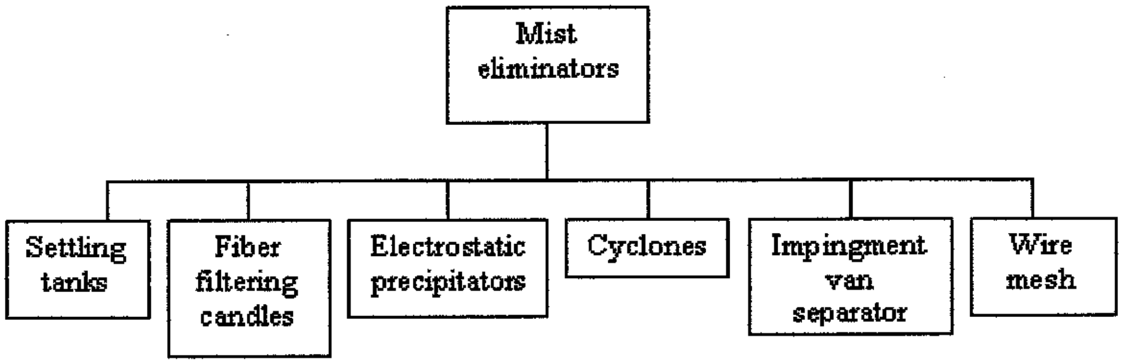

Figure 2: Mist Eliminator Types

2.3.1 WAVE-PLATE MIST ELIMINATORS

Wave-plate mist eliminator is also known as vane or blade type demister. Its generally made of several wavy or zig-zag shaped plates which being put equally spaced between each other along the gas stream. It can be placed in either vertical or horizontal flow. In mist eliminator for horizontal flow, the collected liquid drains perpendicular to the gas flow, while for vertical or upward gas flow, the drainage liquids counter-flow to the rising gas.[2][9]

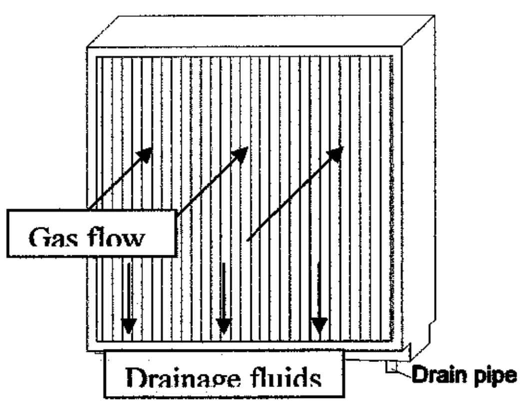

Figure 3: Mist Eliminator in Horizontal Gas Flows

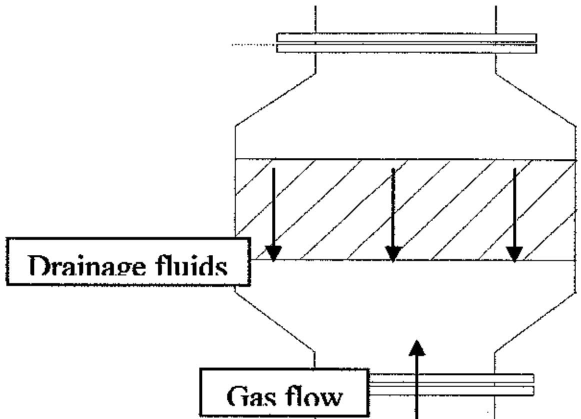

Figure 4: Mist Eliminator in Vertical Flow

This type of mist eliminator work on principle of direct impaction due to inertia and impaction due to centrifugal force [13]. The physics of direct impaction is summarized into three steps; firstly, the liquid droplets impact on the surface of the vanes that block their way. As the gas flow through the vanes, the droplets with too high kinetic energy in the gas stream are not able to follow its streamline or "path" may collide on the vanes. There is where the second stage happen which is the coalescence of the liquid droplets. Lastly, when the accumulated droplets become sufficiently heavy/big, it will detached from the vanes and drain out under the gravity force.[10][13]

Second principle which is led by Burkholz is based on the concept that the turning of the gas could centrifuge droplets out of the flow[1]. When the droplets in the gas stream is forced to travel in a zig-zag manner through the wavy plates, its repeatedly need to change its flow direction. However, the entrained liquids that have higher inertia could not follow these changes will deviate from the main gas stream and impinge on to the solid surfaces. This process will then be followed by the last two stages as desribed earlier. [2][9]

2.4 COMPUTATIONAL FLUID DYNAMICS (CFD)

CFD is a computational technology that uses numerical methods and algorithms to study the dynamics of things that flow. By using CFD, a computational model can be build to represents a system or device that we want to study. Then, apply the fluid flow physics and chemistry to this virtual prototype, and the software will perform the calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. Therefore, CFD is a sophisticated computationally based design and analysis technique. CFD software enables us to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction, and acoustics through computer modeling. [11] [12]

Examples of CFD software commercially available are PHOENICS, FLUENT AND CFX. This particular study will be carried out using FLUENT 6.3, while the geometry is created using GAMBIT 2.2 as both of the software are available in UTP.

2.5 LITERATURE REVIEW

Mist eliminator is devices that can remove entrained liquid from a gas flow. The entrained liquid removal is by the mean of inertial impingement [1] or in other word by impaction. When the gas is forced to change direction negotiating zigzag passages formed the shape of the vanes, larger liquid droplets with higher inertia will tend to continue in a straight line and coalesce on the vane surface that blocks their path [1]. There they coalesce, and accumulate, when the amount of liquid is large enough, it will drains down under gravity [2].

Prior CFD work similar to this study has been done by R. Rahimi and D. Abbaspour (2007) which is quiet straight forward where a simple wire mesh mist eliminator being analyzed by using CFD and the pressure drop across it was determined.

From the study, predicted pressure drop in demister by CFD have a good relationship compared to experimental data and empirical method with about 14-21% of deviation.

The inertia of the drop and the flowing of the gas cause the motion of drops in zigzag passages. Drag of the gas usually leads to re-entrainment of the drops into the gas stream [2]. Wang and Davies (1996), Wang and James (1998, 1999) has used CFD method to predict fluid deposition in demister. Both experimental and theory they practiced showed that higher separation efficiencies could be achieved by increasing the droplets inertia through its velocity however this will increased the pressure drop across the demister [3] [4] [5]. Furthermore, pioneer on this particular study reported that there will be a dramatically decrement on separation efficiency when the velocity of the gas over a critical value. This was due to re-entrainment of the accumulated fluid [6].

Orientation of the demister also contributes to the efficiency of the device. For vertical demister where gas flow upwards and liquid drained downward, decrease in efficiency typically because of upwards gas flow that prevent accumulated fluids on the vanes to flow downward. The disturbing of the draining film due to upward shear of the gas is known as flooding [8]. On horizontal wave-plate demister, recent works by B. J. Azzopardi and K. S. Sanaullah (2002) reports on its overall efficiency, variation of drop size along the demister and its corresponding pressure drop [2].

Table 1: Some literature on mist eliminators.

References	Types of mist eliminator	Droplet size	Gas velocity	Types of study	CFD code being used
H.T.El-Dessouky (2000)	Wire mesh	1-5mm	0.98-7.5m/s	Experiment	None
R. Rahimi and D. Abbaspour (2007)	Wire mesh	0.1mm	1- 7m/s	CFD, compare with exp. Data and emperical model.	Fluent 6.0
C.Galletti,E. Brunazzi,L. Tognotti	Wave-plate	3μm and 6μm	2- 5m/s	CFD, compare with exp. Data	Self

CHAPTER 3
METHODOLOGY

3.1 METHODOLOGY

In order to study the efficiency of a demister that is currently being used in a company, the geometry of the demister's vanes will be constructed in a similar tower and working condition being used by R. Rahimi and D. Abbaspour in their research [7]. The main reason for this will be; validation of result at the end of this study.

Geometrical details of the test chamber are taken and modified from R. Rahimi and D. Abbaspour research are shown in Figure 3. Their work were determination of pressure drop in wire mesh mist eliminator, while this study replace the wire mesh with wave-plates.

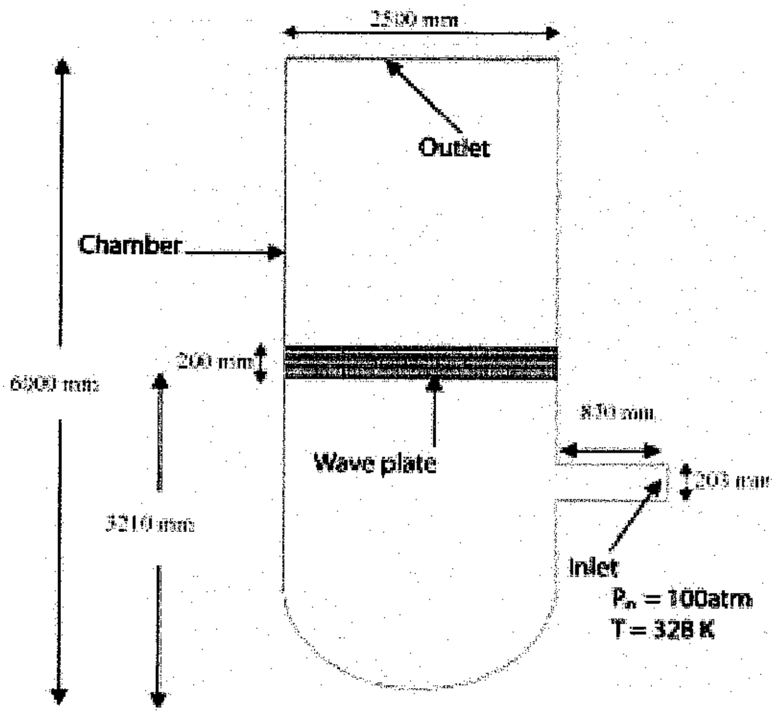

Figure 5: Test Chamber Configuration

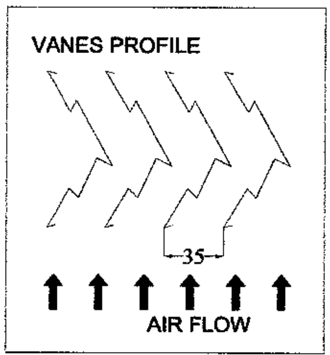

Figure 6: Direction of Airflow

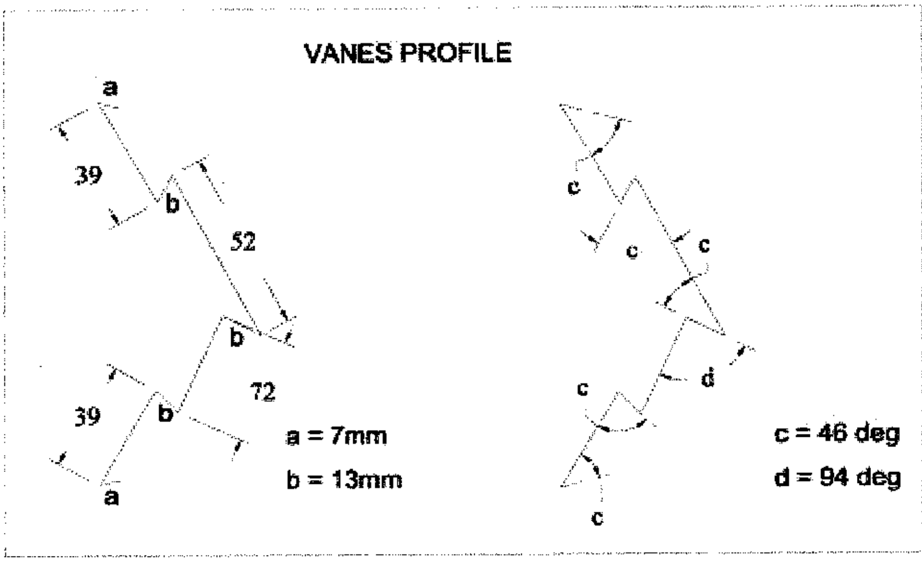

Figure 7: Vanes Profile (thickness of the vanes = 1mm)

The method to find demister efficiency, η, is defined by Eq. (1) while the pressure drop can be obtained by calculating the pressure difference at the outlet and inlet of demister. The corresponding pressure can be determined from data predicted by CFD.

Software used to create the geometry of the demister is Gambit 2.2.30. This software works closely with Fluent.

Several effects will be put into consideration during the simulation of CFD. One of it is gravity, the simulation later should be done in two different ways, one is gravity acts co-flow with the gas (for vertical type of demister), and another one is perpendicular to the gas flow (horizontal type of demister). However, the configuration or design of the vanes will be firstly come into consideration in order to increase the demister efficiency.

3.2 CFD PROCESS

There are essentially three stages to every CFD simulation process: preprocessing, solving and postprocessing. In CFD, continuous and Navier-Stokes equations were solved iteratively.

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \qquad (2)$$

3.2.1 Preprocessing

In this preprocessing step, Gambit 2.2.30 is used to create and mesh the geometry. Gambit is the Fluent's own preprocessor, therefore the CAD geometries will be easily imported and adapted for CFD solutions by Fluent's solver later. Other software that will assist in this step would be AutoCAD and CATIA

After the geometry or the physical bounds of the problem are defined, the geometry of the test chamber with the vanes has been created in Gambit as shown in Figure 7. The area that represent the fluid should be divided into discrete cells (mesh). Quad and pave options have been used for the mesh scheme. However, the computational time required to mesh the geometry is too long. Therefore, it is necessary to simplify the model.

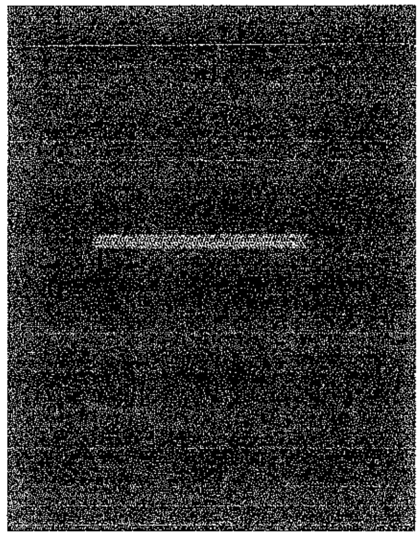

Figure 8: Test Chamber before Meshing

Since the array of the vanes is translationally periodic, periodic boundary approach has been used to simplify the model. Hence, boundary types for the virtual planes between the vanes have been assigned as PERIODIC as shown in Figure 8 below. Boundary types for the remaining edges were shown in the Table 2.

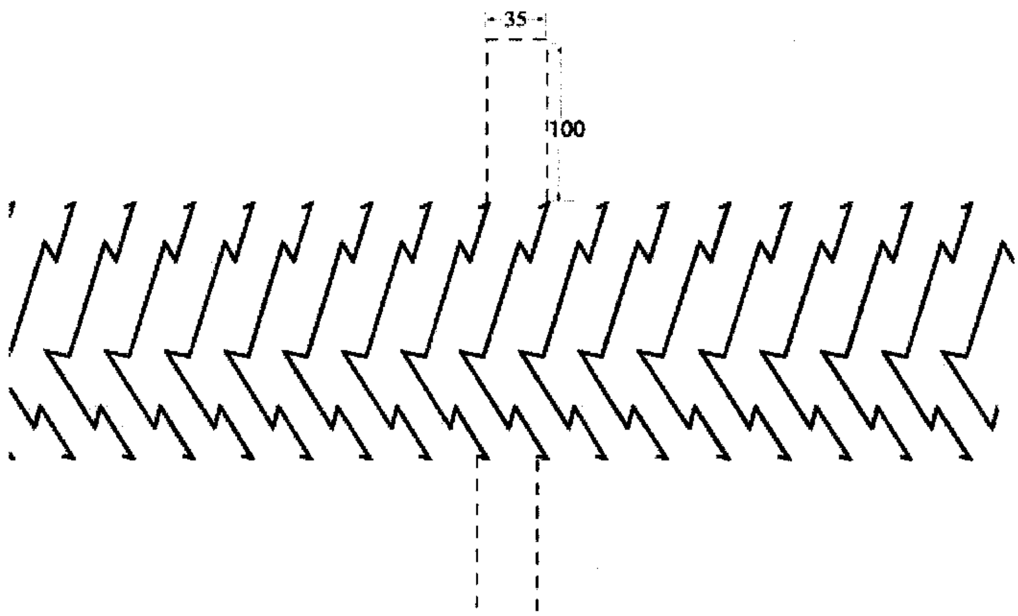

Figure 9: Simplified model (dashed lines)

Table 2: Boundary Types for the Edges.

Name	Type
Outlet	PRESSURE_OUTLET
Inlet	PRESSURE_INLET
Vanes	WALL
Left boundaries	PERIODIC
Right boundaries	PERIODIC

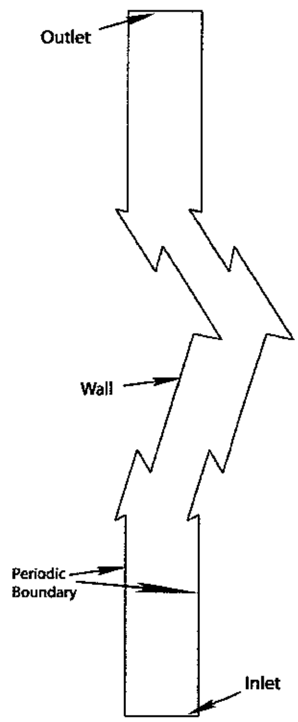

Figure 10: Boundary Types of Each Edge

The simplified model then, has been meshed with quad and pave scheme with interval size of 1mm. This size is small enough to cover the thickness of the vanes which is 1mm. Interval size of less than 1mm will just taking more of CPU time. The structured meshes are showed in Figure 9.

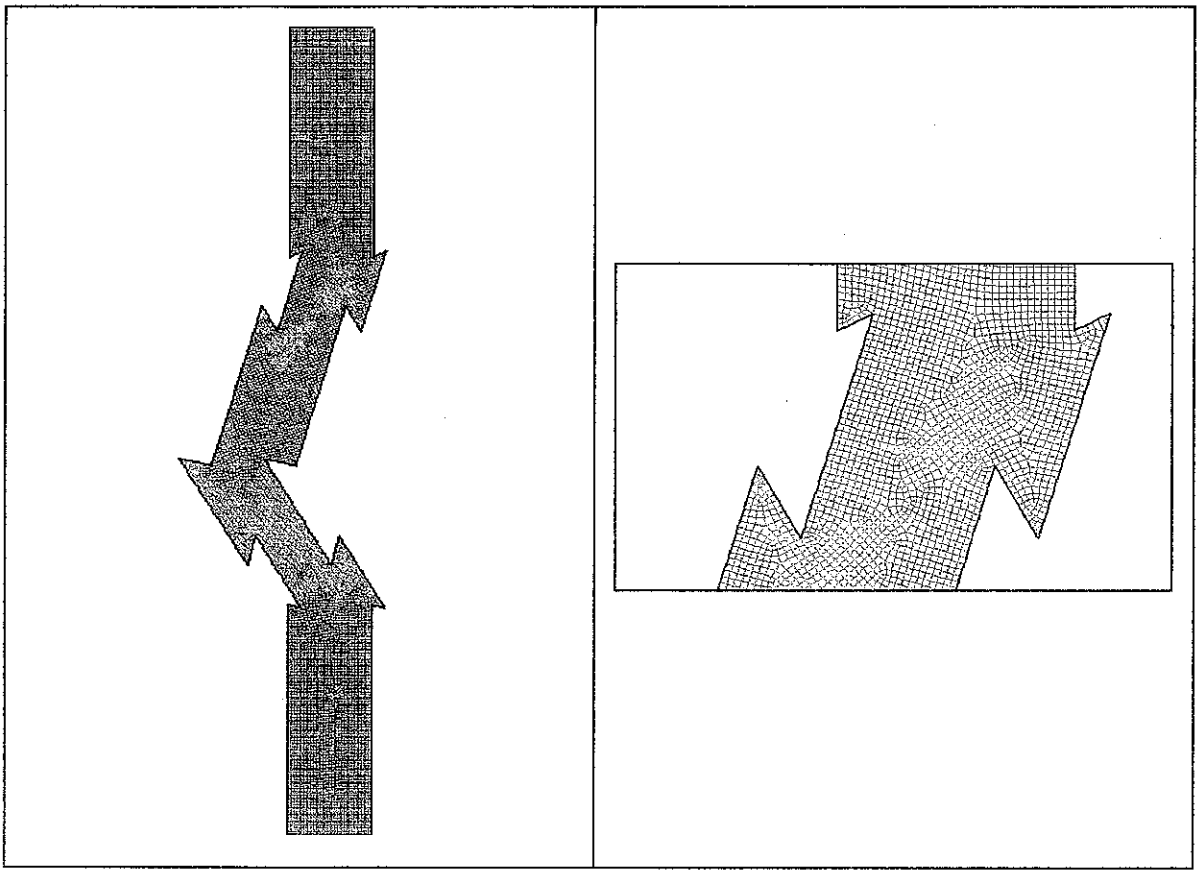

Figure 11: Simplified Model After Meshed

At the end of this step, boundary conditions will be defined. This involves specifying the fluid behaviour (e.g. turbulent) and properties at the boundaries of the problem.

3.2.2 Solving

In this step, the CFD solver that is Fluent will do the flow calculations where all the equations will be solved iteratively and the results will be produced. The model will be solved using segregated solver. In order to treat turbulence phenomena in both phases (liquid and gas), Realizable k-ε model has been used. Energy equation was used in the calculation too.

The fluid through the vanes is a mixture of liquid and saturated gas. The latter will be the primary phase with phase fraction of 0.9964. Physical properties of the fluids are in Table 3 below.

Table 3: Properties of Phases at T= 328K and P=100atm.

	Gas (Primary-phase)	Liquid (Secondary-phase)
Phase Fraction	0.996	0.004
Density, ρ(kg/m^3)	1.225	998.20001
Specific Heat, C_p (J/ kg.K)	1006.43	4182
Thermal Conductivity, W (W/m.K)	0.0242	0.60000002
Viscosity, μ (cP)	1.7894001e-05	0.001003

This simulation was carried out with pressure different of 50-100 Pa while the diameter of the liquid droplets is 0.1mm.

3.2.3 Postprocessing

This is the final step in CFD analysis, and it involves the interpretation of the predicted flow data and the production of CFD images and animations. Images and animations (visualization) will be produced during this stage to enables us understand the result in a quick and effective manner.

Two conditions have been expected for the result. First is the terminal velocity. From R. Rahimi and D. Abbaspour in their research, the predicted terminal velocity is 6 m/s. Theoritically, wave type of mist eliminator has larger terminal velocity than wire mesh type. Hence, the terminal velocity of this study should be greater than 6 m/s. Secondly, pressure drop predicted from this CFD should be less than predicted in R. Rahimi and D. Abbaspour research.

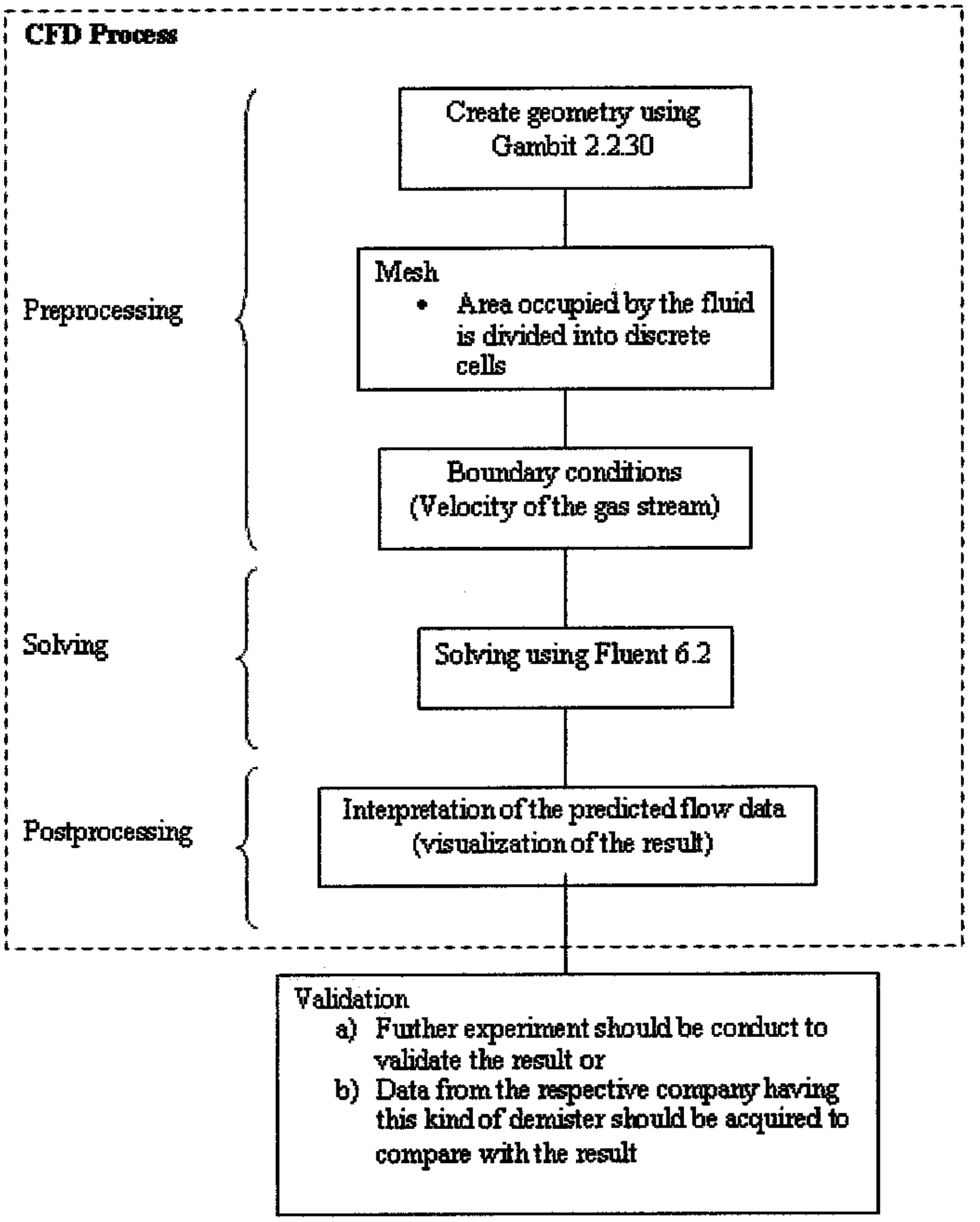

Figure 12: CFD Flow Process

CHAPTER 4

RESULT AND DISCUSSION

4.1 RESULT

After meshing and defining the boundary conditions, the modelling file is exported to mesh file and ready to be used in FLUENT. The initial grids of the flowing area have been showed in Figure16 below.

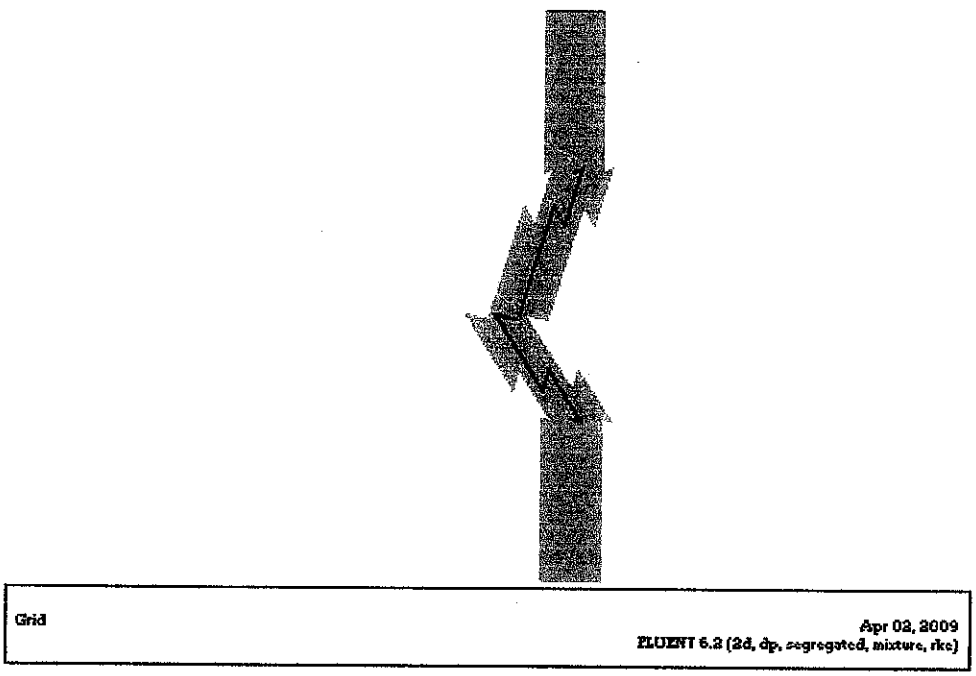

Figure 13: Initial Grids of the Flowing Area

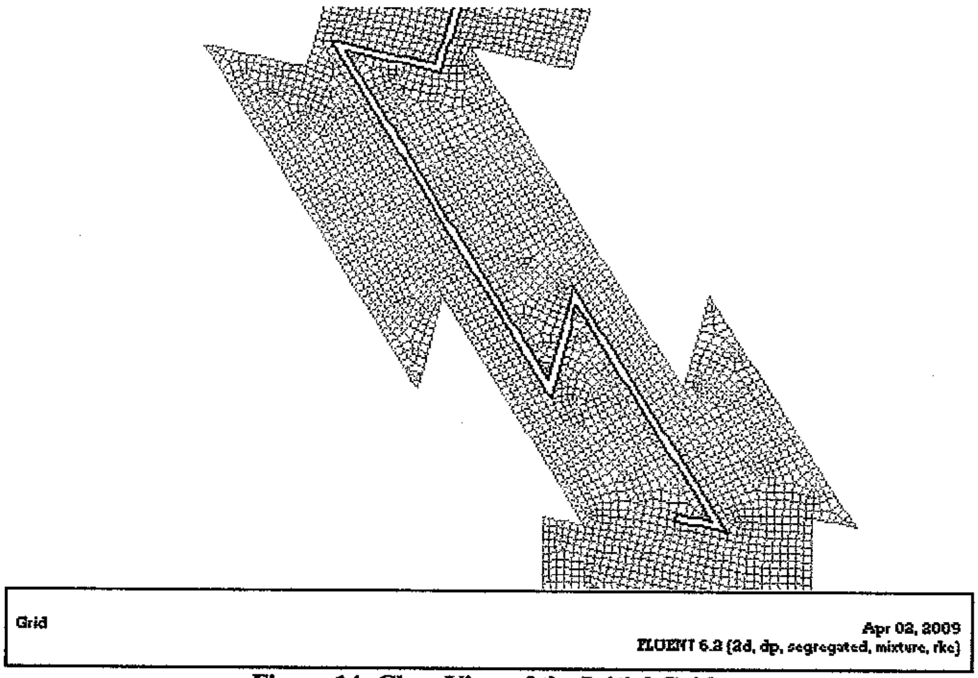

Figure 14: Close View of the Initial Grid.

4.1.1 Initial solution (150 iterations)

The calculation have been started with 150 iterations by using first-order discretization method, the residual of the first 150 iterations is shown in Figure 18. Figure 19 show the predicted velocity distribution. The miminum velocity is 0.0005 m/s and the velocity reach up to 2.6224 m/s at the smallest cross sectional area (red spot).

The magnified velocity vectors for the mixture is shown in Figure 20. The vanes' wall are in black line and the periodic boundaries are blue in color. Density distribution of the mixture are shown in Figure 21. The minimum density is 80.99998 kg/m^3 while the maximum density is 81.00002 kg/ m^3. Mass flow rate across the inlet and outlet are shown in Figure 22 with the imbalance of 0.009 kg/s. This value represent the trapped fluid between the vanes.

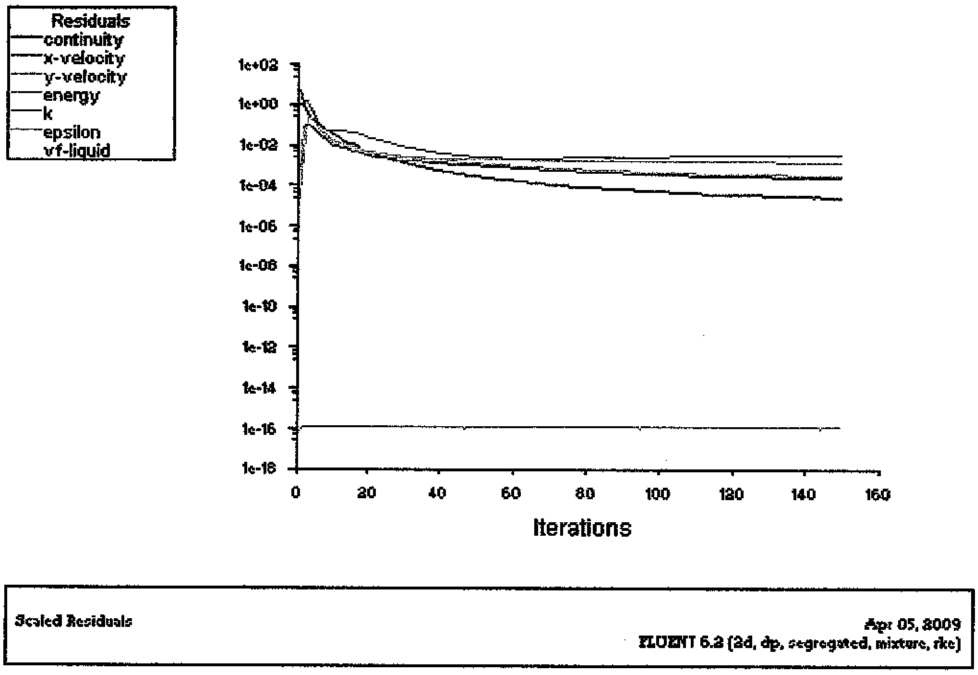

Figure 15: Residuals for the First 150 iterations

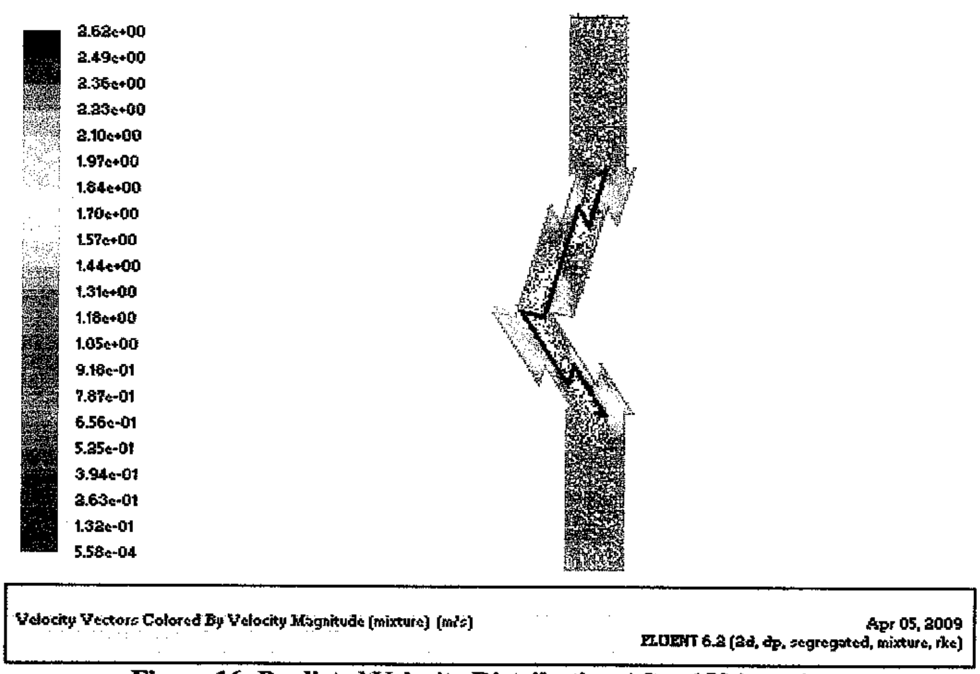

Figure 16: Predicted Velocity Distribution After 150 iterations

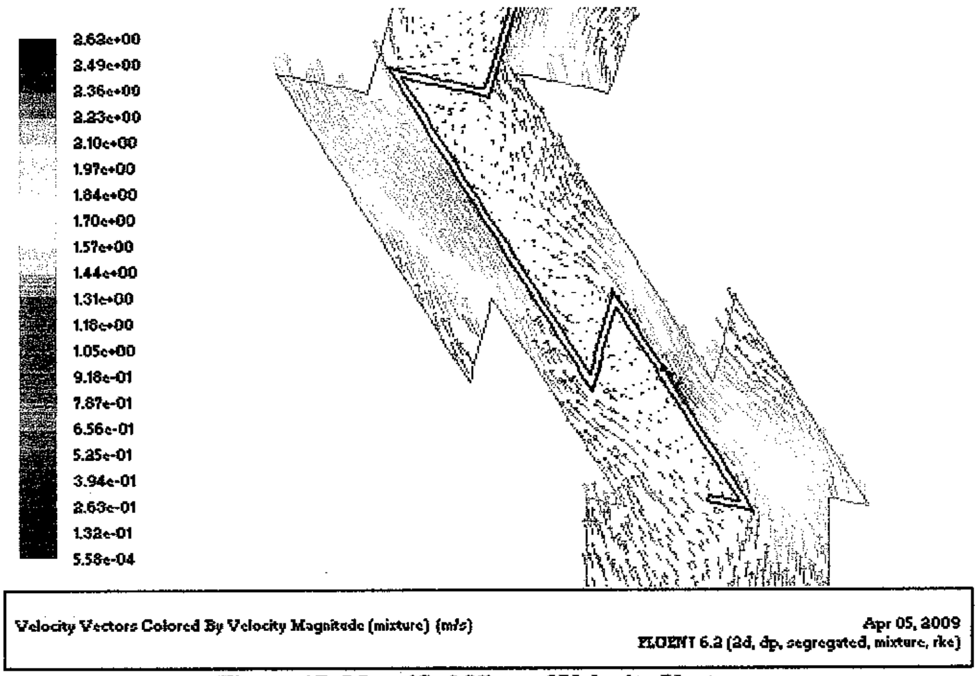

Figure 17: Magnified View of Velocity Vectors.

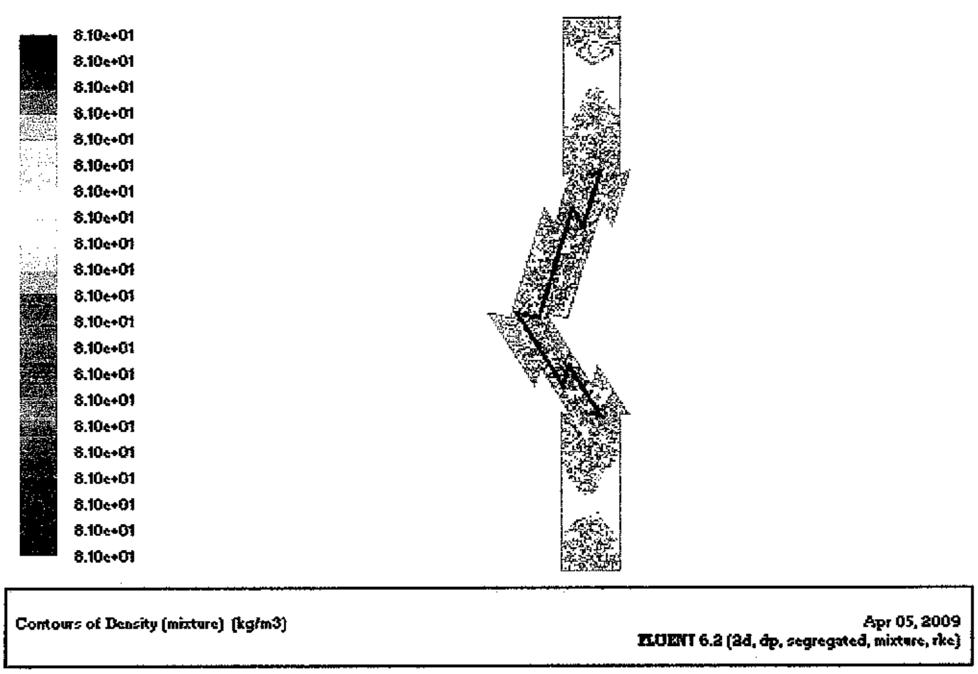

Figure 18: Predicted Density Distribution after 150 iterations

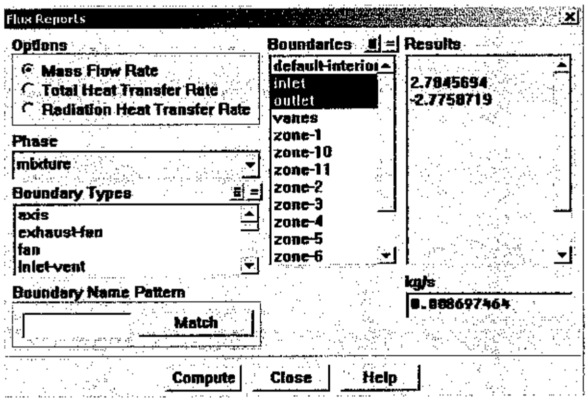

Figure 19: Mass Flow Rate through the Inlet and the Outlet

4.1.2 Second-order and 150 more iterations

The solution computed before uses first-order discretization. The resulting solution can be improved further by using second-order discretization , the calculation are continued with another 150 iterations. The residuals are shown in Figure 23 below.

The predicted velocity distribution are shown in Figure 24 with minimum velocity of 7.89e-6 m/s and maximum velocity of 3.09 m/s (red spot). Figure 26 showed the predicted density distribution with the minimum value of 80.99998 kg/m^3 and maximum value of 81.00002 kg/m^3. The imbalance mass flow rate after 300 iterations is 0.011 kg/s.

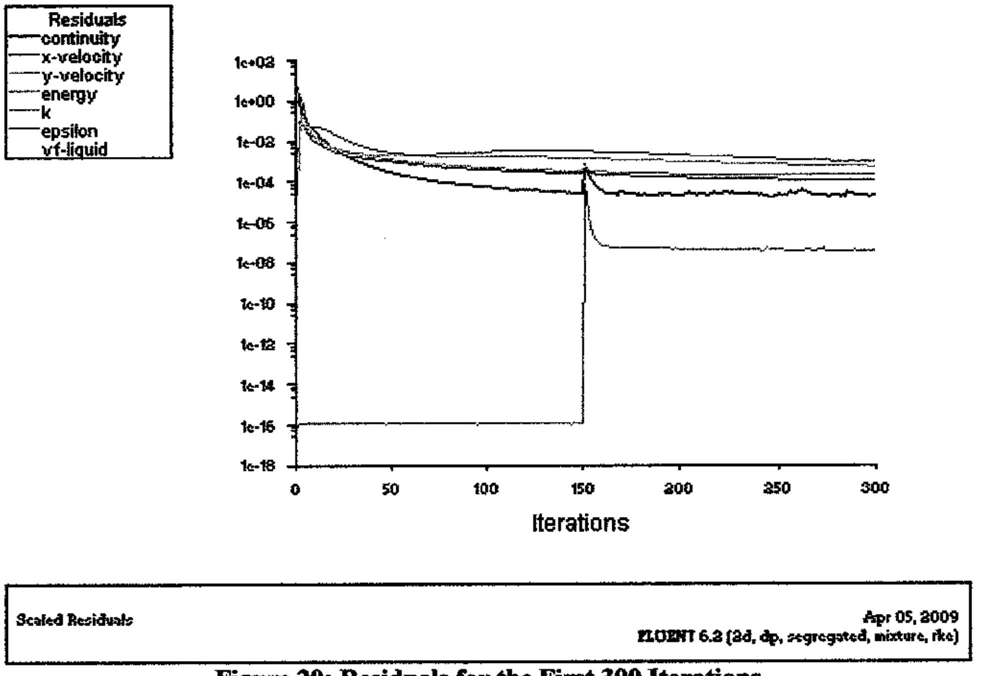

Figure 20: Residuals for the First 300 Iterations

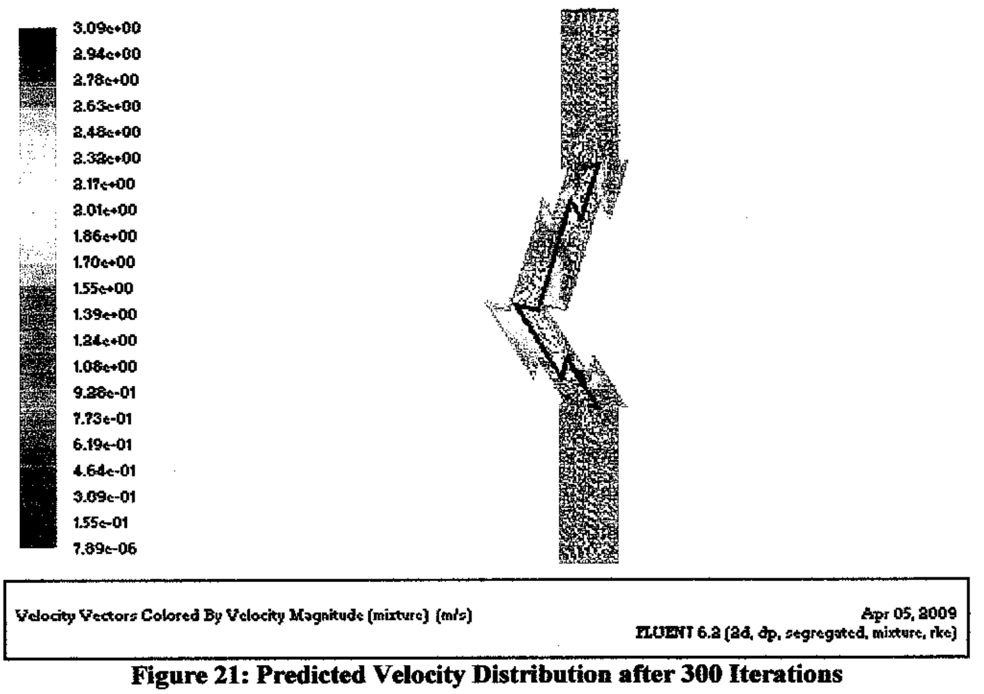

Figure 21: Predicted Velocity Distribution after 300 Iterations

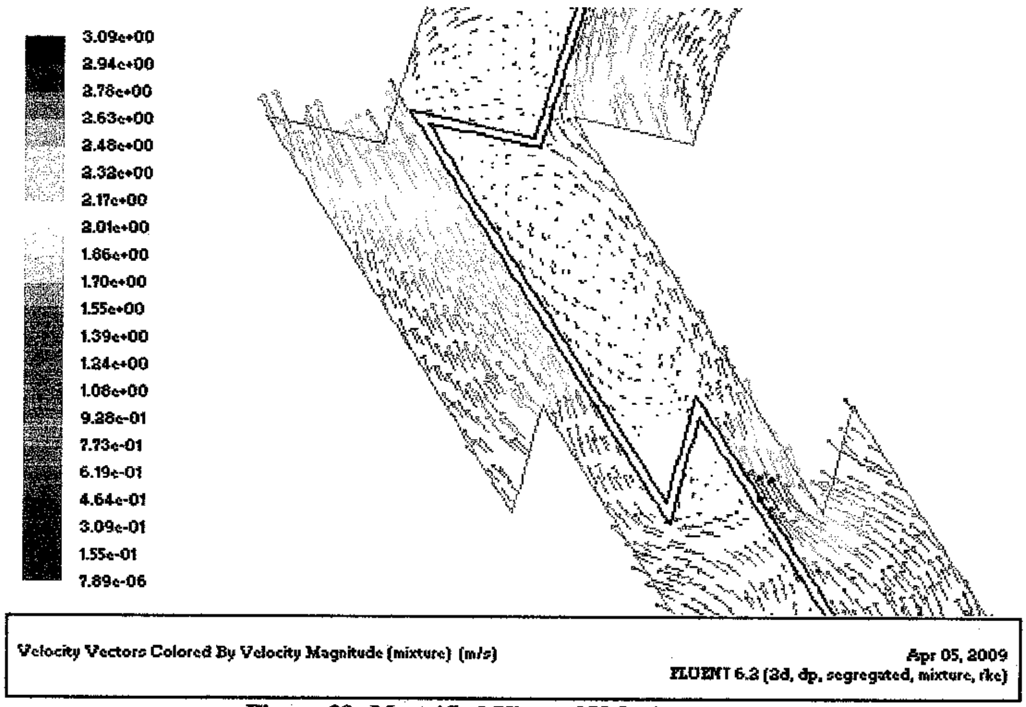

Figure 22: Magnified View of Velocity Vectors

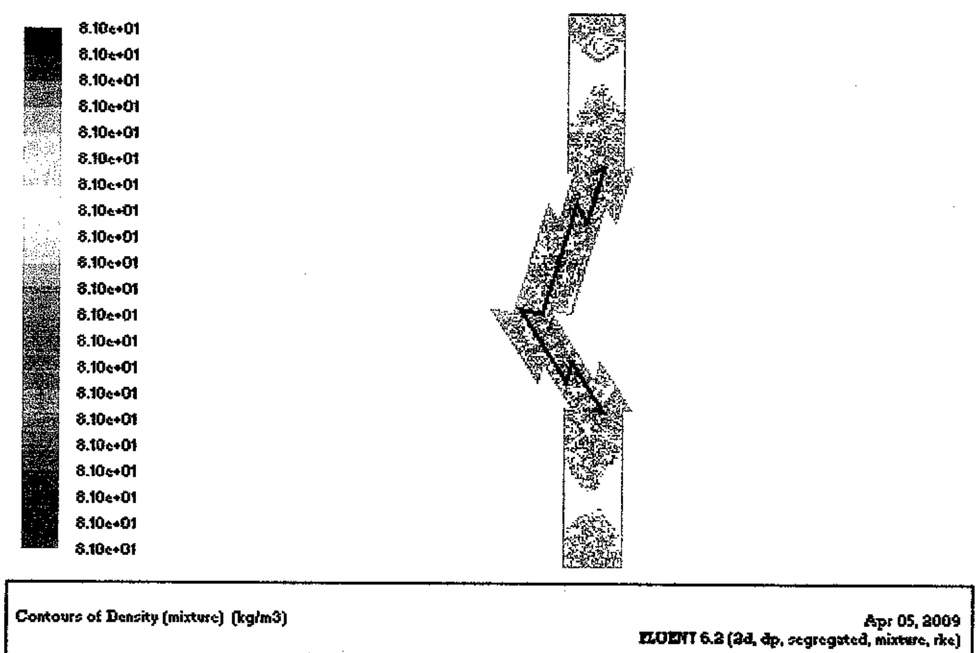

Figure 23: Predicted Density Distribution after 300 iterations

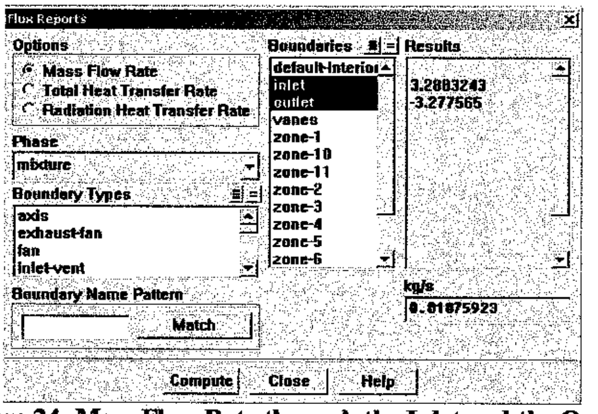

Figure 24: Mass Flow Rate through the Inlet and the Outlet

4.1.3 Refining the grid and 150 more iterations

The solution can be improved further by refining the grid to better resolve the flow details. In this steps, the grid will adapted based on the pressure gradient of the current solution. Figure 28 shows the 443 red marked cells for adaption or refinement in this step. The adapted grids are shown in Figure 29 and can be recognised by the small cells inside the marked cells in Figure 28.

Once the grid has been refined, the calculation will be continued with another 150 iterations. The residuals are shown for the calculations are shown in Figure 30. Figure 31 shows the predicted velocity distribution after 450 iterations with minimum value of 3.75e-5 m/s and maximum value of 3.25 m/s. The magnified view of the velocity vectors are shown in Figure 32.

The predicted density distribution are shown in figure 33 with the minimum density is 80.99998kg/m^3 and maximum density of 81.00002kg/m^3. The mass flow rate imbalance is 0.002 kg/s.

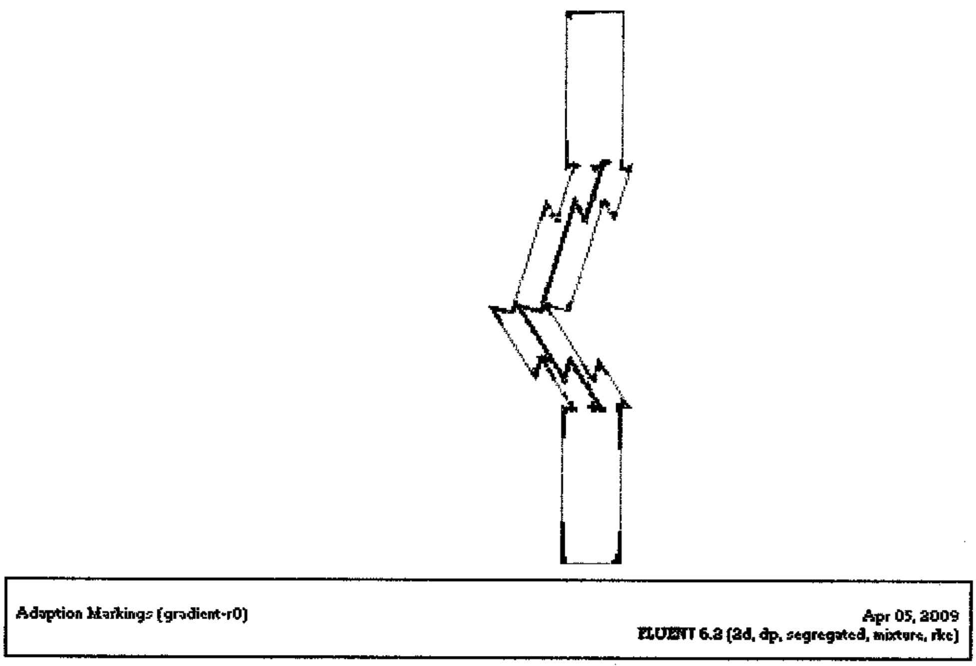

Figure 25: Cell marked for Adaption

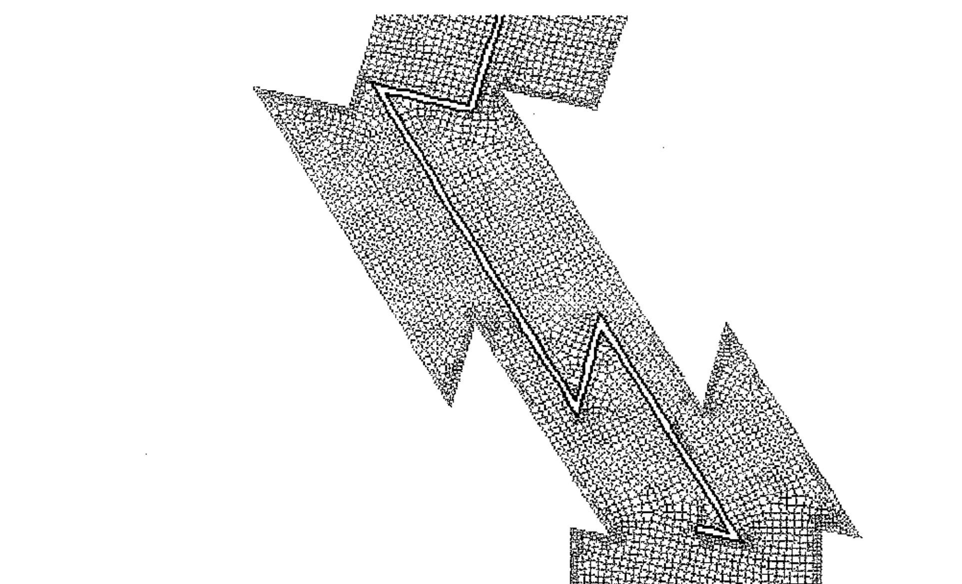

Figure 26: The adapted grid

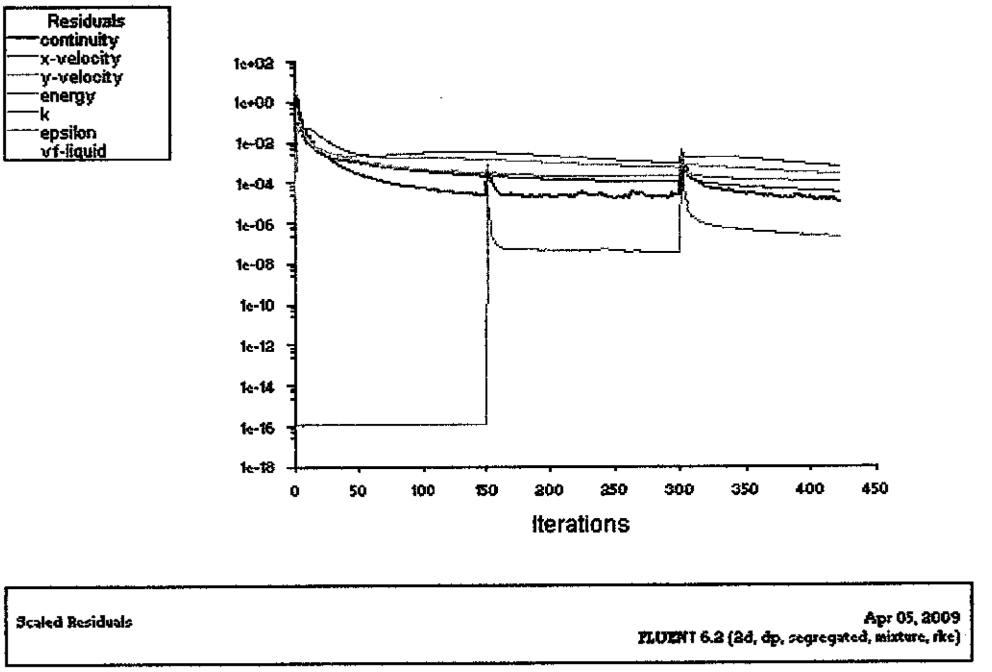

Figure 27: Residuals for 450 iterations

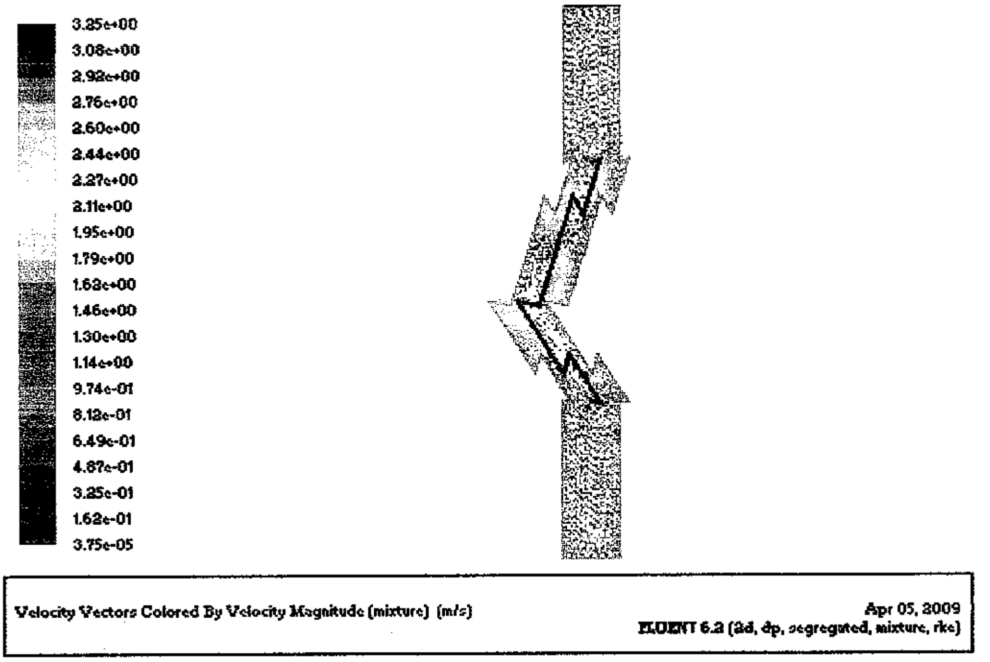

Figure 28: Predicted velocity distribution after 450 iterations

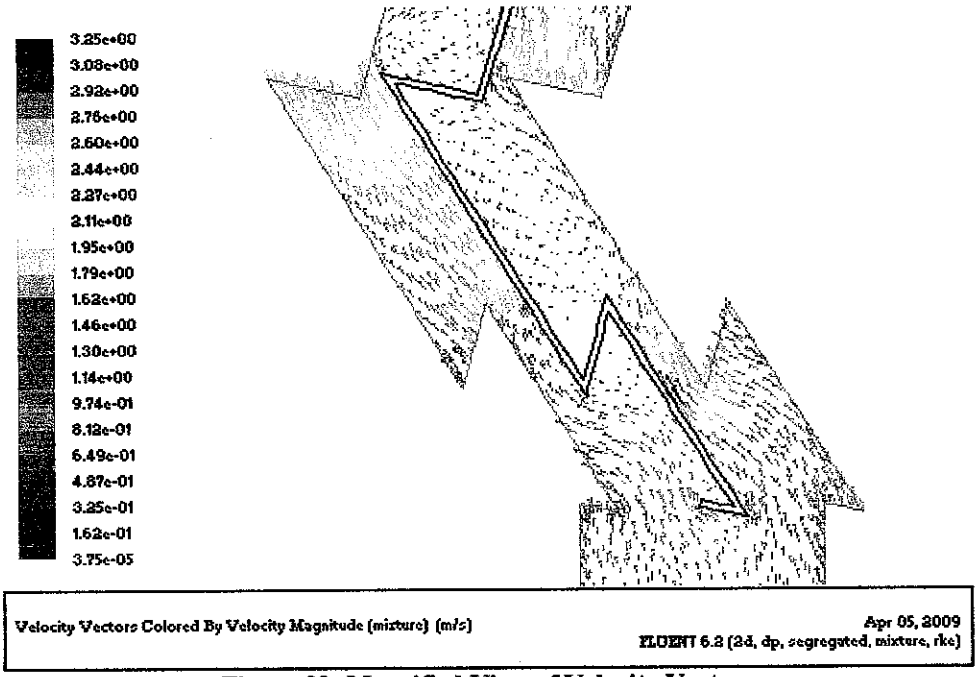

Figure 29: Magnified View of Velocity Vectors

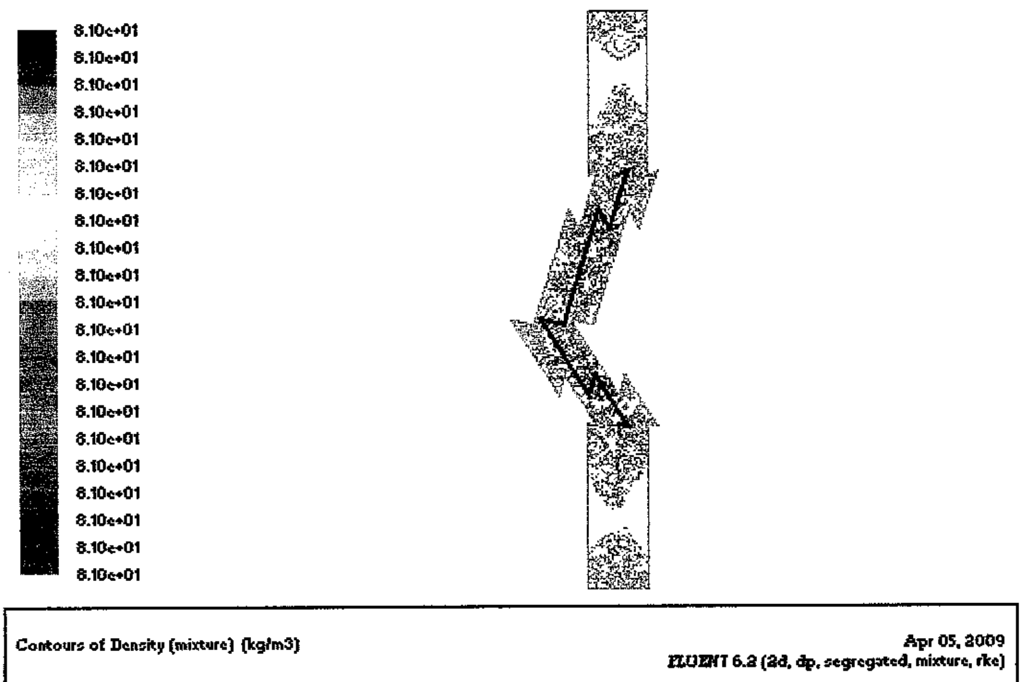

Figure 30: Predicted Density Distribution after 450 iterations

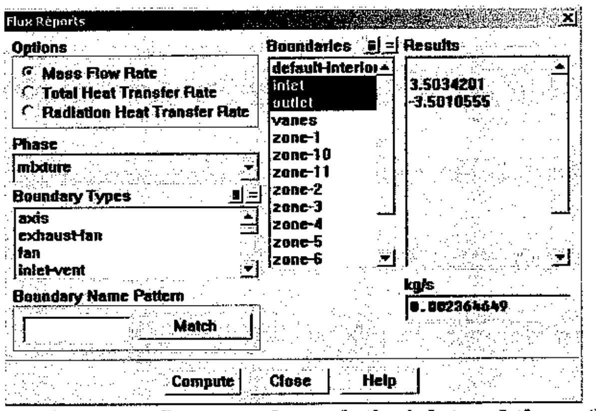

Figure 31: Mass flow rate through the inlet and the outlet

The simulation were repeated with different pressure drop as in the table below.

Table 4: Simulation results.

Pressure Inlet	Pressure Outlet	delta p	M_{in}-M_{out}	M_{in}	Vane type	vel	vel	Wire mesh
10132500	10132000	500	0,00582	1,464208	27,85	1,36	1	62
10132500	10131900	600	0,00819	1,887869	30,37	1,76	2	68
10132500	10131800	700	0,01040	2,267878	32,11	2,13	3	71
10132500	10131700	800	0,01205	2,610283	32,33	2,46	4	74
10132500	10131600	900	0,01329	2,923198	31,85	2,77	5	78
10132500	10131500	1000	0,01429	3,212568	31,16	3,06	6	81
10132500	10131400	1100	0,01511	3,484193	30,37	3,34	7	78

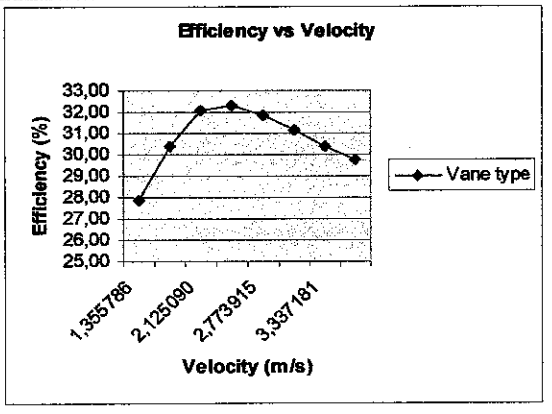

Figure: Effeciency versus velocity for vane-type mist eliminator

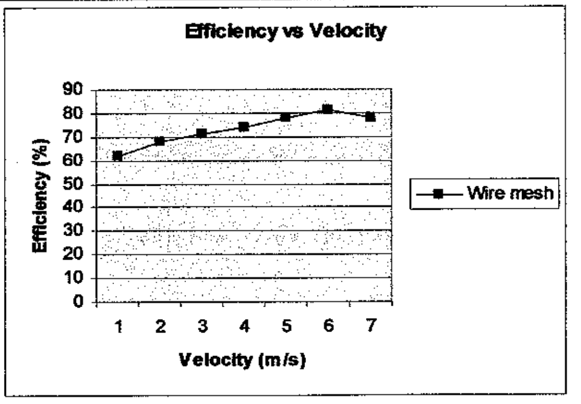

Figure: Effeciency versus velocity for wire mesh-type mist eliminator

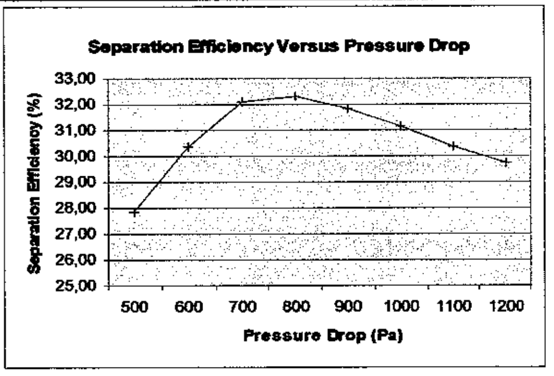

Figure: Effeciency versus pressure drop for vane-type mist eliminator

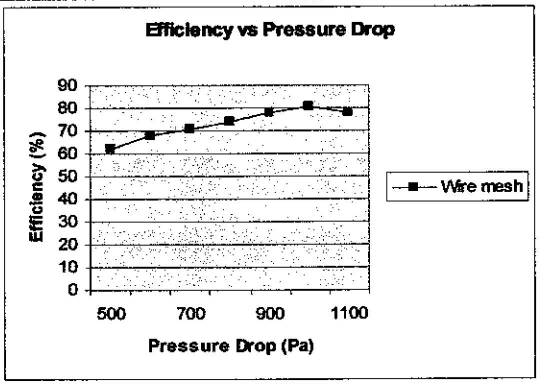

Figure: Effeciency versus pressure drop for wire mesh-type mist eliminator

CHAPTER 5
CONCLUSION

4.1 CONCLUSION

In this study, a 2D vane has been created in GAMBIT and a simulation of a steady-state turbulent flow over the vane was performed in FLUENT. The computed velocity and density distributions through the gas stream were shown in the result.

From this study, maximum efficiency that can be achieved with the proposed configuration is 32% with a gas velocity of 2.78 m/s and the pressure drop is 850pa.
The mechanism of the mist elimination system is well understood and CFD is a great tool to find the efficient velocity with pre-determined configuration.

More reliable result were achieved with second-order discretization and grid adaption approach. This study should promote further CFD analysis study on optimizing the design of mist eliminator.

References:

[1] A. Buerkholz, Droplet Separation, VCH, New York (1989).

[2] B. J. Azzopardi and K. S. Sanaullah, Re-entrainment in wave-plate mist eliminators, *Chem. Eng. Sc* (2002).

[3] W. Wang and G. A. Davies, CFD studies of separation of mists from gases using vane-type separators. *Transactions of the Institute of Chemical Engineers* (1996).

[4] Y. Wang and P. W. James, The calculation of wave-plate demister efficiencies using numerical simulation of the flow field and droplet motion. *Transactions of the Institute of Chemical Engineers* (1998).

[5] Y. Wang and P. W. James, Assessment of an Eddy-interaction model and its refinement sing predictions of droplet deposition in a wave plate demister. . *Transactions of the Institute of Chemical Engineers* (1999).

[6] H. G. Houghton and W. H. Radford, Measurement on eliminators and the development of a new type for use at high gas velocities. *Transaction of the American Chemical Engineers* (1939).

[7] R. Rahimi and D. Abbaspour, Determination of pressure drop in wire mesh mist eliminator by CFD, *Chem. Eng. Process* (2007)

[8] C. C. J. Verlaan, Performance of novel mist eliminators, Ph. D. thesis, University of Technology, Delft (1991).

[9]C. Galletti, E. Brunazzi, L. Tognotti, A numerical model for gas flow and droplet motion in wave-plate mist eliminators with drainage channels, *Chem. Eng. Process* (2008)

[10]H. T. El-Dessouky, Performance of wire mesh mist eliminator, , *Chem. Eng. Process* (2000)

[11]J. Zhao, B. Jin, Z. Zhong, Study of the separation efficiency of a demister vane with response surface methodology, *Journal of Hazardous Material* (2007)

[12] About Mist Eliminators, http://www.amistco.com/products/eliminators/index.html as on 1st November 2008

[13] About CFD, http://www.fluent.com/solutions/whatcfd.htm as on 1st November 2008

[14] About CFD, http://en.wikipedia.org/wiki/Computational_fluid_dynamics as on 1st November 2008

[15] Armatec Environmental Ltd, http://www.armatec.co.nz as on 11th September 2008

Appendix 1: Vanes configuration.

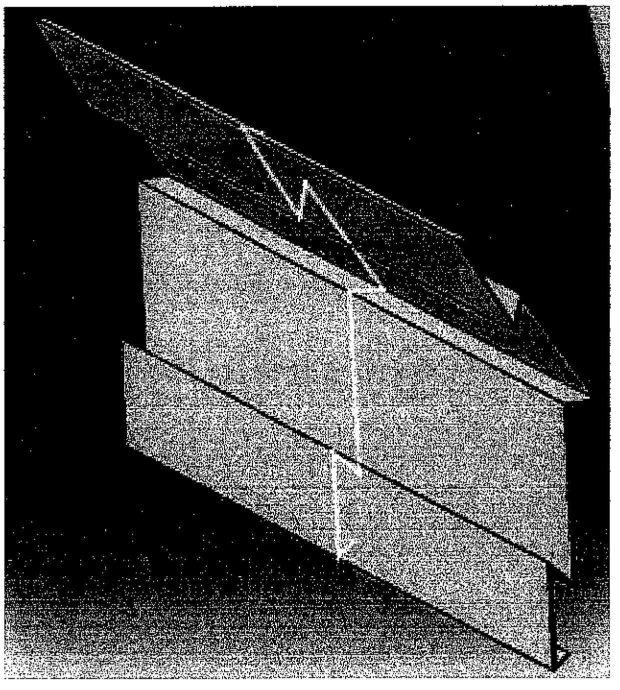

Figure 32: 3D view of a vane

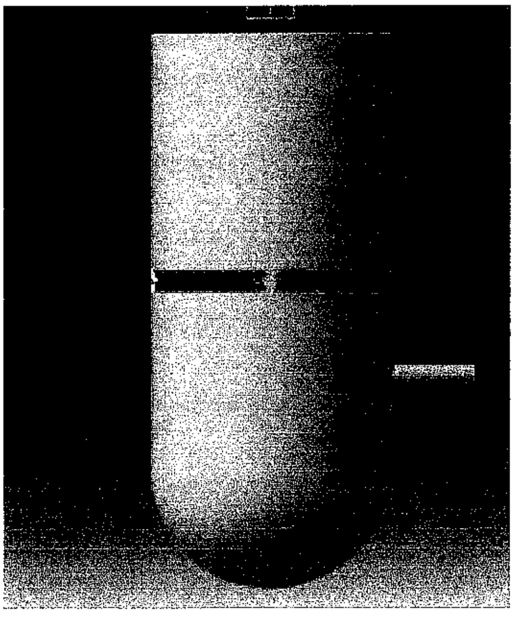

Figure 33: Front view of the test chamber

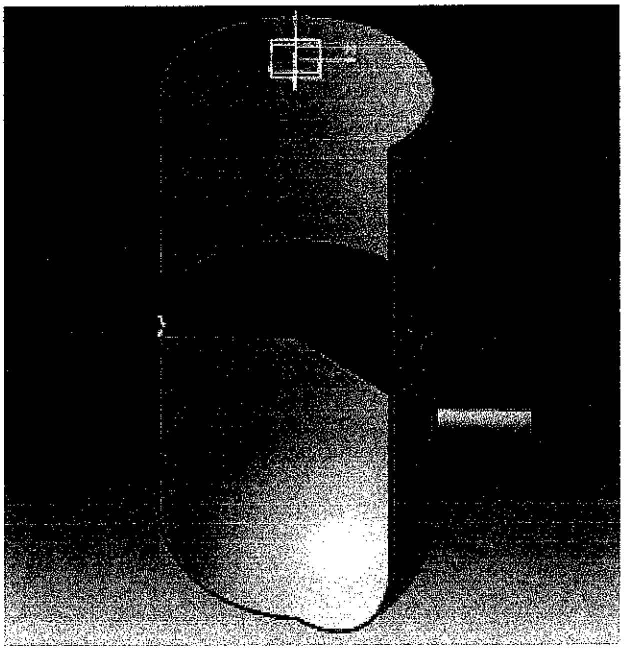

Figure 34: Cross section of the test chamber

Appendix 2: Gantt chart.

MOHD SAFUAN BIN ABD RAHMAN 8320 MILESTONE FOR FYP 1																
No.	Detail/ Week	1	2	3	4	5	6	7	8	9	10		11	12	13	14
1	Selection of Project Topic	■	■													
2	Preliminary Research Work ➢ Do literatures study on Mist Eliminator. ➢ Familiarization with Computational Fluid Dynamics (CFD) software. ➢ Modelling the mist eliminator in Gambit2.2		■	■	■	■	■	■	■	■	■	Mid-semester break	■	■		
3	Submission of Preliminary Report ➢ Write an abstract ➢ Identify the problem statement, objectives and scope of the study ➢ Write the literature review ➢ Explain the methodology to be used in the study				●											
7	Submission of Progress Report								●							
8	Seminar (compulsory)									●						
9	Familiarization with CFD (Fluent 6.2).											■				
10	Modelling of mist eliminator ➢ Using Gambit												■	■	■	■
13	Submission of Interim Report Final Draft														●	
14	Oral Presentation															●

Milestone for the Second Semester

No.	Detail/ Week	1	2	3	4	5	6	7		8	9	10	11	12	13	14
1	Project Work Continue • Meshing and pre-define the boundary type in Gambit.								Mid-Semester Break							
2	Submission of Progress Report 1				X											
3	Project Work Continue • Define boundary types. • Define boundary condition.															
4	Submission of Progress Report 2									X						
5	Seminar (compulsory)									X						
5	Project work continue • Solving using Fluent 6.2 and evaluate the result (post-processing)															
6	Poster Exhibition											X				
7	Submission of Dissertation (soft bound)													X		
8	Oral Presentation														X	
9	Submission of Project Dissertation (Hard Bound)															X