Computational Fluid Dynamics (CFD) Study of Gas Separation Process for High Pressure Horizontal Separator

by

Muhammad Fasha Haiqal Bin Shamsudin

18300

Dissertation submitted in partial fulfilment of

the requirements for the

Bachelor of Engineering (Hons)

(Mechanical Engineering)

JANUARY 2016

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Computational Fluid Dynamics (CFD) Study of Gas Separation Process for High Pressure Horizontal Separator

by

Muhammad Fasha Haiqal Bin Shamsudin

18300

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,

____

(Dr. Tuan Mohammad Yusoff Shah Bin Tuan Ya)

UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK
January 2016

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.

MUHAMMAD FASHA HAIQAL BIN SHAMSUDIN

ACKNOWLEDGEMENT

First and foremost, I would like to praise Allah S.W.T, the Almighty, on whom we ultimately depend for guidance. Second, to my supervisor, Dr. Tuan Mohammad Yusoff Shah, whose supervision and constructive comments were vital. His efficient and timely contribution helped me to complete my project and I express my sincerest appreciation for his assistance in any way that I may have asked. I am also deeply thankful to Dr. Tamiru Alemu for his invaluable advices and comments throughout the whole duration of this project.

I also wish to thank Dr. Tuan Mohammad Yusoff Shah's CFD team for their guidance in using the ANSYS Fluent software, which in turn made this project a successful one. Special thanks, tribute and appreciation to all those that names do not appear here who have contributed. In which without their help, this project would not even past the starting line. The experience of using ANSYS Fluent will forever be useful for me.

I would also like to thank my parents for their support of fulfilling my dream of becoming a mechanical engineer and tirelessly endure my ups and downs in completing my studies. Next, I would like to thank my classmates and friends for their support in any way possible which ultimately helped in completing this project. Lastly, I would like to thank the Mechanical Engineering Department of Universiti Teknologi PETRONAS for giving me this priceless opportunity to do this project as part of completing my undergraduate studies.

ABSTRACT

Gas turbines generators on offshore platforms are essential for the power generation of the platform which would run its equipment, control systems and light up living quarters. In the last few decades, it is the most chosen equipment for power generation. Due to many platforms extract more than enough gas from oil and gas reservoirs, this makes the power generation option easier and more economical. Nonetheless, the gas needs to be in a state that has no presence of liquid hydrocarbon for it to function optimally and lengthily throughout its service life. Consequently, the issue of gas separation inside a HPS arisen. This is to ensure that gas turbines could maintain its efficiency and performance. The main equipment responsible for the separation of liquids and gas from hydrocarbon extracted from wells would be the separators. This project will only be focusing on the simulation of a 2-phase HPS, with only liquid and gas as the multiphase fluid. With running conditions and design specifications of a HPS be used, a CFD simulation is carried out to study the flow pattern of the multiphase fluid and to determine the gas outlet conditions. Initial study would include the use of water and air to substitute liquid hydrocarbon and gas, in order to achieve initial objectives. Nevertheless, realistic methods are later used to gather more realistic results. This project will hopefully result in a separation simulation between liquid and gas inside a HPS.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLE

LIST OF ABBREVIATIONS

CFD	Computational Fluid Dynamics
HPS	High Pressure Horizontal Separator
PTS	PETRONAS Technical Standards
EOR	Enhanced Oil Recovery
UTP	Universiti Teknologi PETRONAS
VOF	Volume of Fluid

CHAPTER 1: INTRODUCTION

1.1 Background

Oil and gas that is produced from the reservoir usually contains mud, oil, condensate and gas are flowed through the wellhead, safety and choke valves. Once it goes through the wellhead manifold, it is channeled into the main component of production process, the separators [1]. In most cases, the separation takes place in two stages. This is mainly due to their difference in pressure. The oil produced is either treated further to remove salt water and other impurities or sent to a storage tank until it is transported to a refinery.

Since water has higher density compared to the liquid hydrocarbon, water resolves at the bottom of the separator along with other condensates and impurities. Water from the separators goes through a sand cyclone to remove solid particles such as sand. The condensate and oil that is caught in the mixture is retrieved in a hydro cyclone where the oil and condensate is sent to a separate storage tank.

Water is normally processed until it passes local and international environmental laws before released back into the sea. The processed water could also be reinjected back to the reservoir to maintain the reservoir pressure. This is one of the examples of EOR method available today.

Gas produced that flows out of the separator is normally flowed through heat exchangers to reduce its temperature in order to ensure higher efficiency at compression stage. The gas is then flowed through scrubbers, to be dehydrated. This is to ensure that all the liquid droplets that might be present in the gas flow are removed. Meanwhile, further gas treatments to remove acidic gases could also be conducted before the compression stage. Metering and storage would be the final step before the oil and gas leaves the platform. Metering is important because the hydrocarbon content and energy value besides the pressure and temperature can be measured and analyzed.

1.2 Problem Statement

As more and more fields produce gas, the option to run on gas as a fuel source to power gas turbines has been increasing in popularity, due to its convenience and economic reasons. Nevertheless, with the increase in offshore production depths and floating platforms, weight and cost of maintenance are affecting the economics of using gas turbines.

On platforms, gas turbines have been found to encounter a higher rate of failure when running on gas recovered from the same well as crude oil. This is mainly due to the presence of liquid hydrocarbon in gas turbines. It is the silent killer that causes a drop in performance. Other than that, the buildup of liquid hydrocarbon in the gas turbine has also been found to be the reason for some of failures encountered.

In ensuring good quality of gas delivered to the gas turbine, the main equipment responsible is the separator. The separator is crucial in the separation process to ensure the gas and oil is separated. This study will focus on the efficiency of the separators in separating the oil and gas for offshore platforms.

1.3 Objective and Scope of Study

The objectives of this project are:

- To model full scale HPS adhering to PTS standards using modelling software.
- To conduct CFD simulation to study the hydrodynamics and multiphase fluid flow behaviour inside the HPS
- To determine flow characteristics and fluid velocities inside and at outlets of the HPS

The study of the flow of gas and fluid inside the HPS is this project main scope of study. Additionally, its separation due to gravity. The flow of the fluid within the separator is studied by vector contours produced in a CFD model of HPS used on common offshore platforms which is also bounded by regional and international standards.

Due to UTP having the availability of ANSYS, ANSYS Fluent will be used as the simulation software. This software will give a clear view of the movement of fluid within the HPS with respect to volume fraction distribution, its fluid/gas state and its velocity. ANSYS Fluent can also simulate the flow characteristics at the gas outlet of the HPS. Focus will be on the gravity settling region of oil and gas in the separator.

CHAPTER 2: LITERATURE REVIEW

In today's oil and gas business, most wells drilled produce a combination of gas, oil and water with other impurities that need to go through the separation process. These elements need to be separated according to its use. For most HPS, the extracted hydrocarbon is fed into it and retained for a period where the water resolves at the bottom, gas released from the top and the oil taken after the separation weir. Pressure reduction is done in stages by having multiple separators to avoid potential flash vaporization which could lead to instability [1].

Once the separator separates the gas, it normally goes through compression before it is flowed into pipeline or be used for power generation with a gas turbine. Fuel gas systems on offshore platforms are an essential part of the power generation to run the platform's equipment and living quarters for manned platforms, as gas turbine generator is one of the more common chosen equipment for power generation [1].

Many platforms produce gas in excess which is flared off as it is not economically viable to process and transport. However, this would be suitable to be used to power up gas turbines which are used for power generation on platforms. Gas turbines are highly engineered equipment that is sensitive and expensive. This piece of equipment is also sensitive to impurities and foreign objects which could cause damage to the turbine blade.

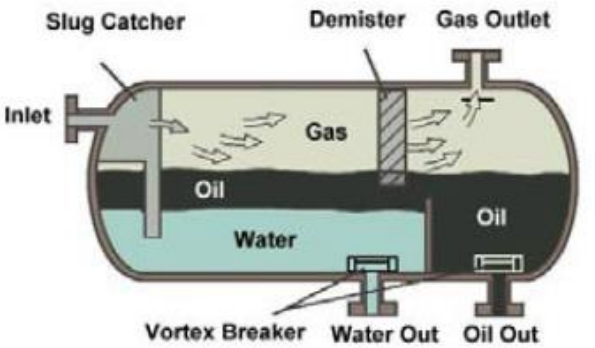

Figure 1: Schematic of a typical HPS

2.1 Causes and Effects of Crude Carry Over

Crude carry over to gas stream can be possible due to multiple reasons. The obvious one would be the failure of separator to meet design specifications based on fluid flow properties. Besides that, crude carry over into the gas stream problem has been observed in separators due to foaming [2]. The effect of foaming which occurs in the presence of a surfactant which can be present naturally or added to aid some other process dissolves in water can end up causing problems such as overhead fouling, reduced capacity of equipment and poor separation efficiency [2].

One way to reduce crude carry over would be the use of vane packs. Also known as wave plate mist eliminators. It is commonly used in chemical, oil and gas industries as it is effective in removing fine liquid droplets from gas flow [3]. The key in designing a suitable vane system lays the cut off liquid droplet size as the vanes are designed to trap the liquid when the gas is passed through the vane at high velocity [4].

Unfortunately, vane demister or vane packs will not be used in this project due to its complexity in designing and also would cost more time. Vane packs are also complicated to simulate, it would need another project to verify or simulate the results with it.

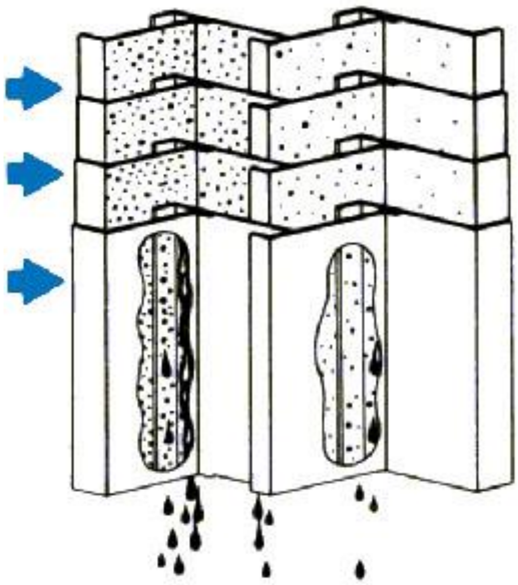

Figure 2: Horizontal vane demister

2.2 Modelling and CFD Software

Model of the horizontal gas separator is modelled using two different modelling software; ANSYS Design Modeler and CATIA. Best compatible design would then be used to be meshed in CFD software. Due to its ease to model when compared to other modelling software available in UTP such as AutoCAD, CATIA would be the preferred software to design the model before running it in the CFD software ANSYS Fluent. Nonetheless, it is advised to use ANSYS CFX in case that results obtained do not present a reasonable conclusion.

The CFD model developed will be focusing on simulating the fluid movement and separation process within the separator and at the outlets, especially the gas outlet. The simulation will focus on the volume fraction and velocity of the liquid stream. Though experiments and mathematical models have been developed, simulation could yet prove to unlock the key to this crude carry over issue in horizontal separators. By comparing experimental data to simulation results (Fluent), it predicts the phase separation and velocities better, while being able to simulate conditions as closely as possible to the exact situation [5].

2.3 Multiphase flow theory

Multiphase flow is flow with simultaneous presence of different phases, where phase refers to solid, liquid or vapor state of matter. There are four main categories of multiphase flows; gas-liquid, gas-solid, liquid-solid and three-phase flows. Further characterization is commonly done according to the visual appearance of the flow as separated, mixed or dispersed flow. These are called flow patterns or flow regimes and the categorization of a multiphase flow in a certain flow regime is comparable to the importance of knowing if a flow is laminar or turbulent in single-phase flow analysis [6]. A flow pattern describes the geometrical distribution of the phases and the flow pattern greatly affects phase distribution, velocity distribution and etcetera for a certain flow situation [5]. A number of flow regimes exist and the possible flow patterns differ depending on the geometry of the flow domain. For some simple shapes, for example horizontal and vertical pipes, the flow patterns that occur for different phase velocities etcetera have been summarized in a so called flow map. The two extremes on a flow map is dispersed flow and separated flow. In separated flow there is a distinct boundary between the phases. Examples of separated flow is stratified flow where one phase is flowing on top of another or annular flow in a pipe with a liquid film along the pipe and a gas core in the middle. Dispersed flow is flow where one phase is widely distributed as solid particles or bubbles in another continuous phase. Several intermediate regimes also exist, which contain both separated and dispersed phases such as for example annular bubbly flow [5]. Due to growing instabilities in one regime, transition to another regime can occur. This phenomenon complicates the modelling of multiphase flow even further as the transition is unpredictable and the different flow regimes are to some extent governed by different physics.

2.3.1 Multiphase modelling approach

Models are used to be able to describe and predict the physics of multiphase flow. As previously mentioned, modelling of multiphase flow is very complex. In addition, there are also limitations in time, computer capacity etc. when performing numerical studies. This has led to the development of models that can account for different levels of information, meaning different levels of accuracy, and are suitable for different multiphase flow applications [5]. Some of these modelling approaches are presented below.

- Euler-Lagrange approach

In the Euler-Lagrange approach, particles are tracked on the level of a single particle where particle refers to either a solid particle or a gas/fluid bubble/droplet. Conservation equations are solved for the continuous phase and the particle phase is tracked by solving the equations of motion for each particle.

$$\frac{\partial \alpha_f \rho_f}{\partial t} + \nabla \cdot (\alpha_f \rho_f \mathbf{u}_f) = \mathbf{S}_{mass}$$

$$\frac{\partial \alpha_f \rho_f}{\partial t} + \nabla \cdot (\alpha_f \rho_f \mathbf{u}_f \mathbf{u}_f) = \alpha_f \nabla \mathbf{p} - \alpha_f \nabla \cdot \boldsymbol{\tau}_f - \mathbf{S}_p + \alpha_f \rho_f \mathbf{g} = 0$$

$$\frac{\partial \mathbf{u}_p}{\partial \mathrm{t}} = \sum \mathbf{F}$$

Conservation equations are solved for the continuous phase and the particle phase is tracked by solving the equations of motion for each particle. The forces acting on particles vary depending on the flow situation. The drag force is generally included and other forces that can be of importance are for example lift force, virtual mass force and/or history force.

- Euler-Euler approach

In Euler-Euler models all phases are treated as continuous. For that reason, these models are often also called multi-fluid models. Multi-fluid models are appropriate for separated flows where both phases can be described as a continuum. However, the Euler-Euler approach can also be used to model dispersed flows when the overall motion of particles is of interest rather than tracking individual particles. The dispersed phase equations are averaged in each computational cell to achieve mean fields. To be able to describe a dispersed phase as a continuum, the volume fraction should be high and hence this approach is suitable for dense flows. The phases are treated separately and one set of conservation equations are solved for each phase [5]. The interphase exchange coefficients need to be modelled. Just as in the Euler-Lagrange approach it is up to the modeler to decide which interphase phenomena to include. In addition to the regular transport equations, a transport equation for the volume fraction is also solved for each phase. The sum of the volume fractions should be equal to one. The governing equations for a two-fluid model with two continuous phases are shown below.

$$\frac{\partial \alpha_k \rho_k}{\partial t} + \nabla \cdot (\alpha_k \rho_k \mathbf{U}_k) = 0$$

$$\frac{\partial \alpha_k \rho_k \mathbf{U}_k}{\partial t} + \nabla \cdot (\alpha_k \rho_k \mathbf{U}_k \mathbf{U}_k) = -\alpha_k \nabla \mathbf{P} + \alpha_k \nabla \cdot \boldsymbol{\tau}_k + \alpha_k \rho_k \mathbf{g}_k + \mathbf{S}_k = 0$$

$$\frac{\partial \alpha_k}{\partial t} + \nabla \cdot (\alpha_k \mathbf{U}_k) = 0$$

A mixture model is a simplified version of an Euler-Euler model. As in the Euler-Euler models both phases are treated as interpenetrating continua but in the mixture model the transport equations are based on mixture properties, such as mixture velocity, mixture viscosity etc. To track the different phases, a transport equation for the volume fraction is also solved. The phases are allowed to move with different velocities by using the concept of slip velocity, which in turn includes further modelling.

- Volume of fluid approach

A third modelling approach is the volume of fluid (VOF) method. VOF belongs to the Euler-Euler framework where all phases are treated as continuous, but in contrary to the previous presented models the VOF model does not allow the phases to be interpenetrating. The VOF method uses a phase indicator function, sometimes also called a color function, to track the interface between two or more phases. The indicator function has value one or zero when a control volume is entirely filled with one of the phases and a value between one and zero if an interface is present in the control volume. Hence, the phase indicator function has the properties of volume fraction. The transport equations are solved for mixture properties without slip velocity, meaning that all field variables are assumed to be shared between the phases [5]. To track the interface, an advection equation for the indicator function is solved. In order to obtain a sharp interface the discretization of the indicator function equation is crucial. Different techniques have been proposed for this. The equations solved in the VOF method are shown below.

$$\frac{\partial \rho_m}{\partial t} + \nabla \cdot \left(\rho_m \mathbf{u}\right) = 0$$

$$\frac{\partial \rho_m \mathbf{u}}{\partial t} + \nabla\left(\rho_m \mathbf{u}\mathbf{u}\right) = -\nabla P + \nabla \boldsymbol{\tau} + \rho_m \mathbf{g} + \mathbf{S} = 0$$

$$\frac{\partial \alpha}{\partial t} + \nabla\left(\alpha \mathbf{u}\right) = 0$$

Here $\rho_m = \sum \alpha_k \rho_k$. The subscript m refers to mixture properties.

As the focus of the VOF method is to track the interface between two or more phases it is suitable for flows with sharp interfaces, such as slug, stratified or free-surface flows

CHAPTER 3: METHODOLOGY

3.1 Flow Chart

The project begins with problem identification and literature review. The HPS is modelled using ANSYS Design Modeler & CATIA while the simulation is run on ANSYS Fluent. The projected flow of the project is as shown in the figure below.

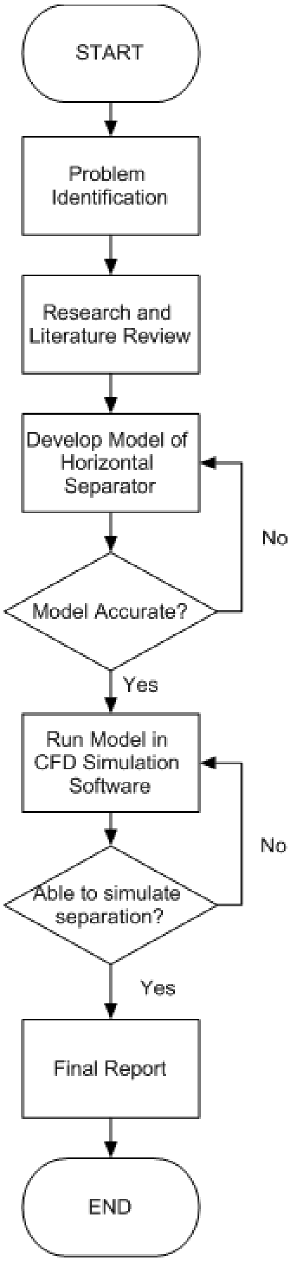

Figure 3: Project flow chart

3.2 Project Gantt Chart

Month / Year		2015																2016															
		Sept				October				November				December				January				February				March				April			
Topic	Week	1	2	3	4	1	2	3	4	1	2	3	4	1	2	3	4	1	2	3	4	1	2	3	4	1	2	3	4	1	2	3	4
	FYP			1	2	3	4	5	6	7	8	9	10	11	12	13	14			15	16	17	18	19	20	21	22	23	24	25	26	27	28
	I			1	2	3	4	5	6	7	8	9	10	11	12	13	14																
	II																			1	2	3	4	5	6	7	8	9	10	11	12	13	14
Selection of topic																																	
Preliminary Research Work																																	
Extended Proposal										Δ																							
Design Modelling																																	
Proposal Defense												Δ																					
Improve Design																																	
Study of Solver Methods																																	
Study of Setup Parameters																																	
Interim Report																	Δ																
Final Geometry Mesh																																	
Preliminary Simulation																																	
Progress Report																										Δ							
Further Simulations																																	
Pre-SEDEX																													Δ				
Final Simulation																																	
CFD Analysis																																	
Finalize Report																																	
Dissertation Draft																															Δ		
Viva																															Δ		
Hardbound Submission																																	Δ

Legend: Process Δ Key Milestone

3.3 Model Design

In order to have a realistic design, a realistic design parameter would be needed. The HPS is divided into many main parts, including the inlet, inlet deflector, liquid outlet, gas outlet, vane pack and vessel. Design parameter is obtained from a major operator for offshore production in this region. Physical data of the HPS are as shown below:

Table 1: HPS physical data

	Internal Diameter (inch)	Internal Diameter (mm)
Vessel Inside Diameter	90.55	2300
Vessel Height	236.22	6000
Inlet	18	457.2
Liquid Outlet	9.56	242.824
Gas Outlet	11.38	289.052

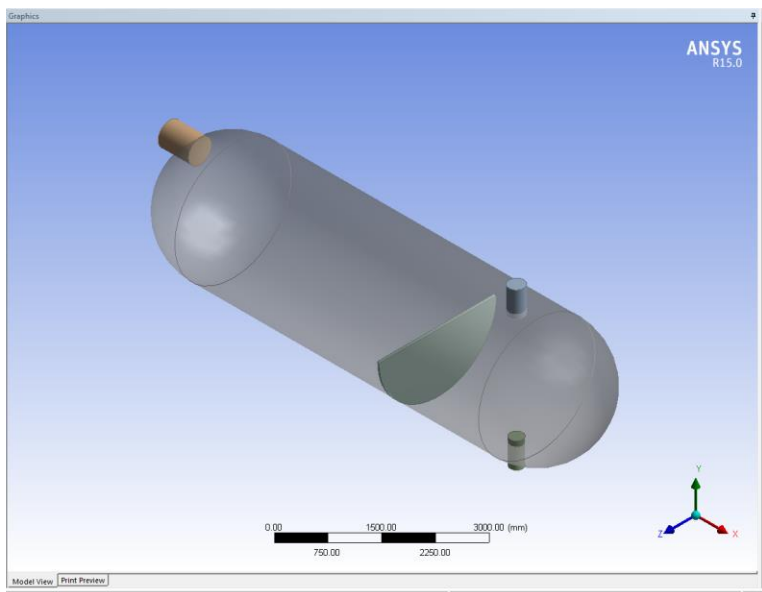

Figure 4: HPS first ANSYS DesignModeler model

The first model is designed with ANSYS DesignModeler. Initially found to be good. But after running it in ANSYS Fluent, it is found that this model could not be meshed. This is mainly due to the model only being modeled as “skin” and had some defects in its inlet and outlet. Additionally, it has an unused weir in its geometry as this project will be focusing on a 2-phase HPS.

A second model is then developed, with the weir removed. This model also cannot be meshed in Fluent. After troubleshooting, it is decided that modeling using design modeler would not produce anything good and thus changed the modeling software to CATIA.

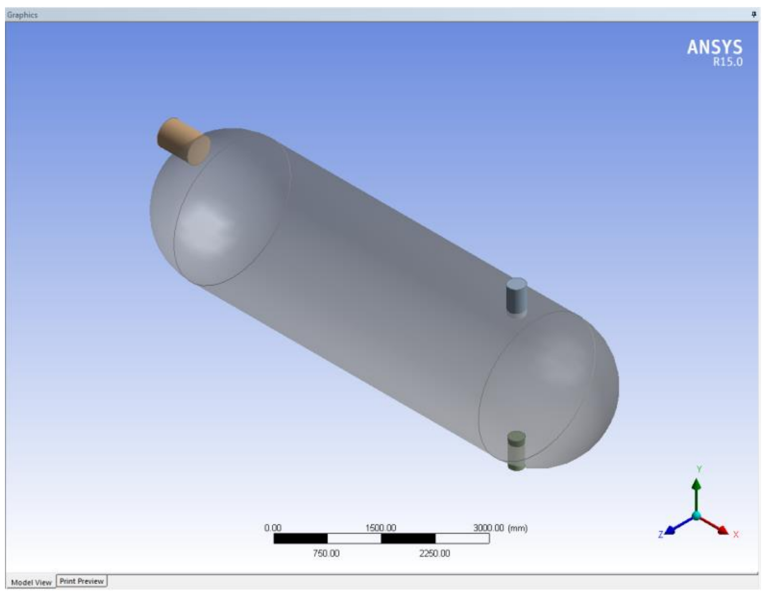

Figure 5: HPS second ANSYS DesignModeler model

The first CATIA model is developed sometime later in the first project phase. Though looked quite accurate at first, it was found that the geometry used for its inner diameter in this model was wrong. The supposed inner diameter was used as radius and thus had a very large size. Adjustments are made to the first CATIA model to get the model to be used to simulate the preliminary result.

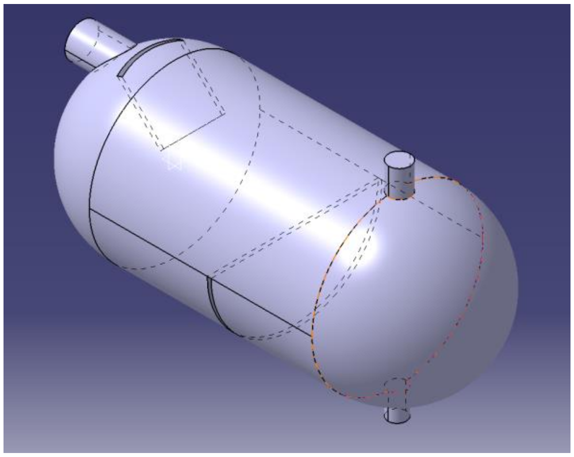

Figure 6: HPS first model modeled using CATIA

After many changes, the updated design model was produced. This model has all the required components except the vane packs. It is decided that the vane pack is a complicated feature and is not included in this project.

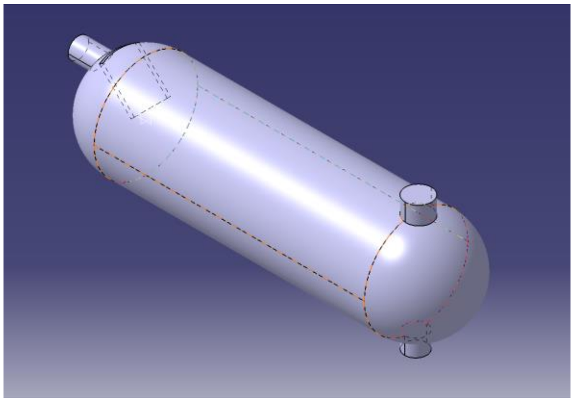

Figure 7: HPS second CATIA model

After many adjustments, the model is finalized. The updated model has a deflector near the inlet pipe with a downward deflection of 45° and has a side blocker to ensure that the liquid and gas entering the HPS is correctly deflected after coming through the inlet.

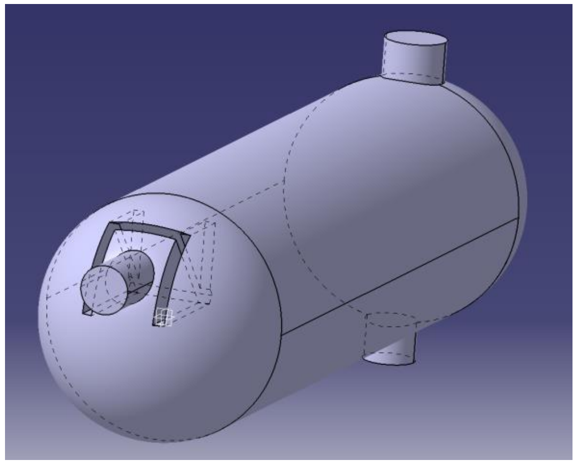

Figure 8: HPS final CATIA model

CATIA model proved to be the better of the two modeling software. This is mainly due to the fact that the modelling software is more user-friendly and more versatile. The ANSYS DesignModeler has very limited capability and thus making it harder to model with. CATIA has a very broad tutorial online with many other users using this software to develop models. Thus, when in need, help is not too far to reach. Author also has a very limited experience with ANSYS DesignModeler, but better experience with CATIA. Therefore, that might be one of the reasons for the mentioned justification.

3.4 Meshing

The CFD software used in this project is ANSYS Fluent. The software will focus on simulating the fluid movement and separation process within the separator and its outlet especially the gas outlet.

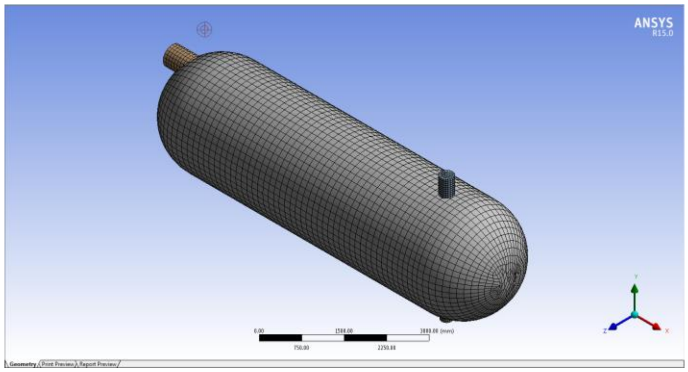

Figure 9: Mesh produced using second ANSYS DesignModeler model

The first mesh is the mesh of the second ANSYS DesignModeler model using "Automatic Method". This model proved to be wrong in many ways. Mainly is due to its geometry only covers the external or skin of the model. The inner side of the model is hollow.

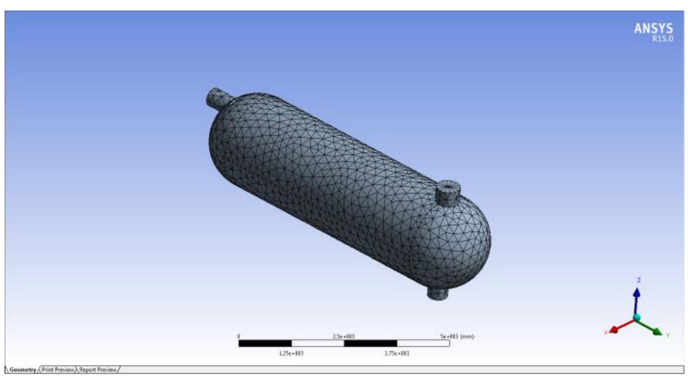

Figure 10: Initial mesh of final CATIA model

The second CATIA model is found to be better as it meshed beautifully using "Automatic Method". Nevertheless, the mesh values are still coarse; with 13828 Nodes and 8263 Elements. This mesh model is used to run the preliminary simulation.

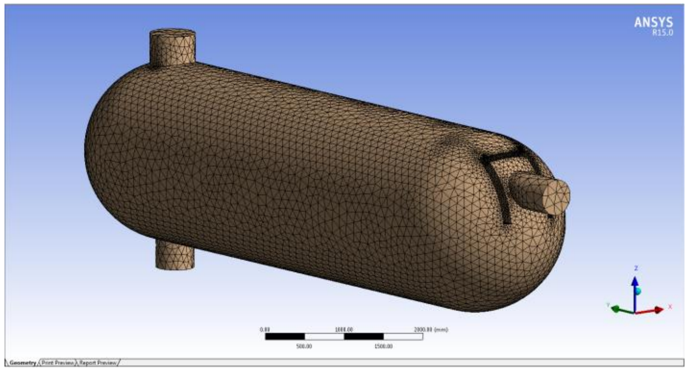

Figure 11: Mesh produced using final CATIA model

The final CATIA model proved to be the right model as it meshed beautifully using “Patch Independent” method with further adjustments made by adjusting its method from “automatic” to “tetrahedrons” and its algorithm from “patch conforming” to “patch independent”. The mesh values are very fine with 338704 Nodes, 237636 Elements. Thus becoming the best mesh produced for this project. Further enhancement of mesh is done and found irrelevant due to more time needed to mesh and run calculations.

3.5 Setup

3.5.1 Preliminary Simulation Setup

The solver used for the preliminary simulation is pressure-based with absolute velocity formulation. The gravity acceleration was set to -9.81m/s2 in the z-direction. This is due to the design being modelled in the Z-Y coordinate, thus needing to adjust accordingly. Steady time was selected due to this being a preliminary simulation to see if the iteration could converge.

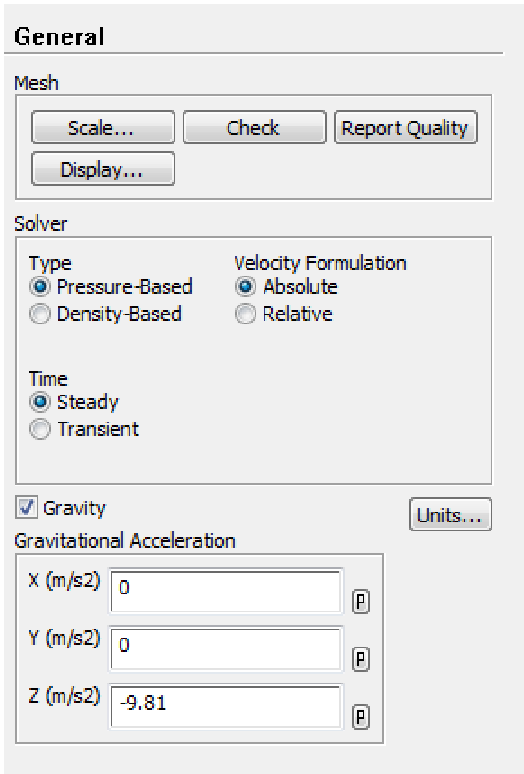

Figure 12: Preliminary simulation general setup parameters

In models setup tab, multiphase mixture model is turned on with 2 Eulerian phases. These will later be explained more in the materials selection. This setup is recommended for oil and gas separation by software developer’s tutorial.

The viscous model is selected because it is the most suitable model for turbulent flow in a mixture. This setting will probably be maintained for all setups after this, unless found irrelevant.

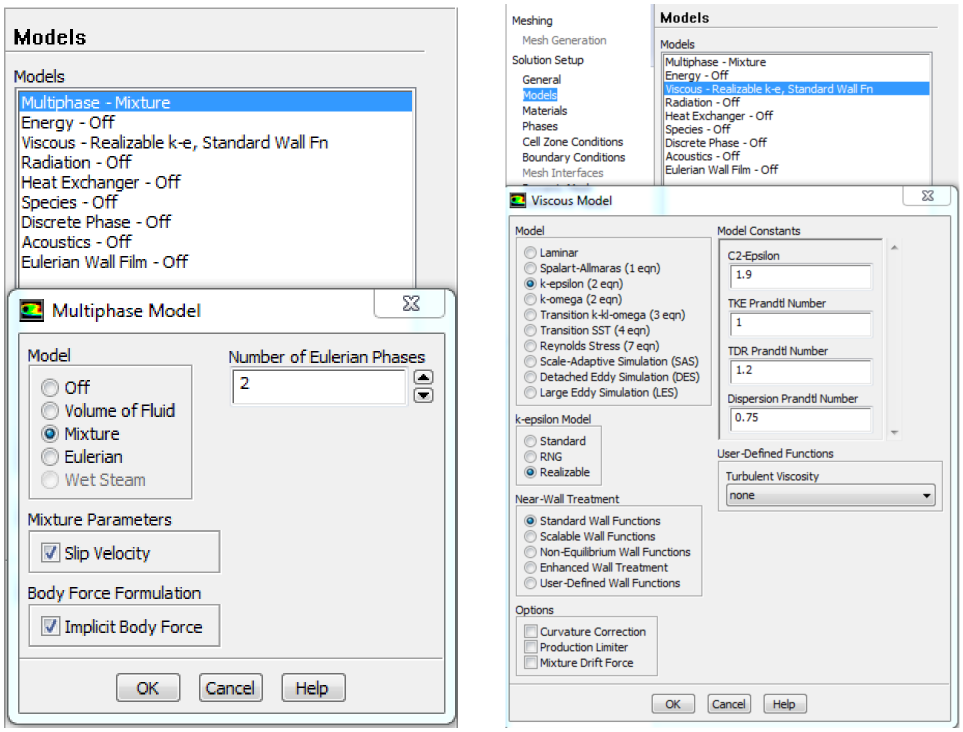

Figure 13: Preliminary models setup parameters

The preliminary simulation is aimed to achieve objectives. Thus, the material selection consists of water-liquid and air for the fluid and aluminum as solid.

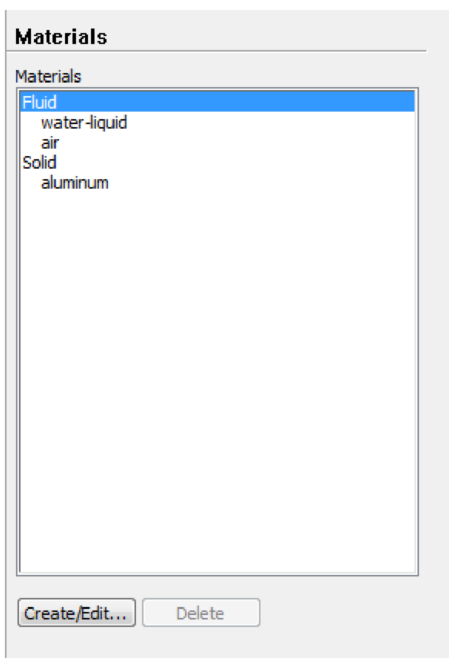

Figure 14: Preliminary simulation material setup

Next is the boundary conditions setup. This setup mainly focuses on the inlet and outlet conditions. The inlet is set with the value of 1.2 MPa and 0 MPa at the outlets. This is to ensure that the flow will move from the input to the corresponding outputs.

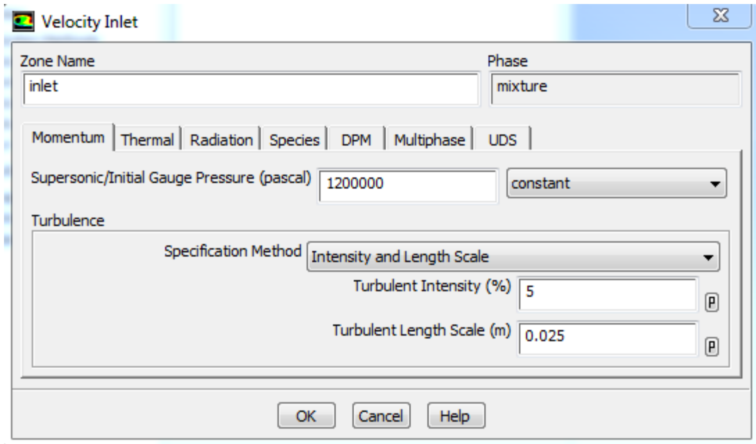

Figure 15: Preliminary simulation inlet boundary condition setup for mixture

Solution method is set to be as most conservative possible. This is partly to achieve preliminary simulation objective.

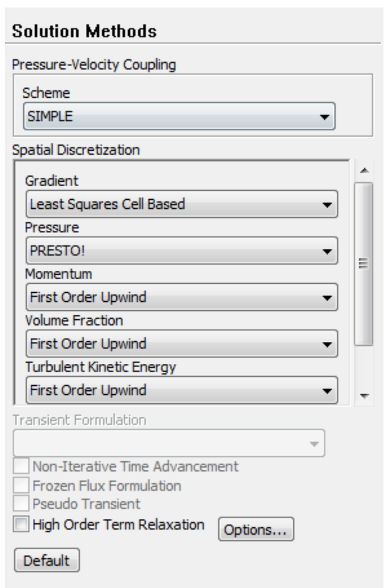

Figure 16: Preliminary simulation solution methods setup

Solution is then initialized using hybrid initialization method. Calculation is set to 1000 iterations and then calculation is started. The solution converges.

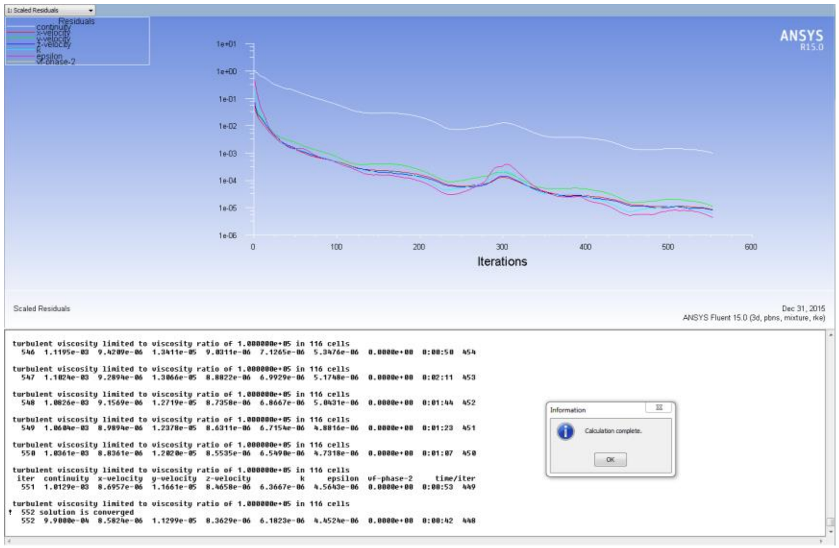

Figure 17: Convergence of solution

3.5.2 Secondary Simulation Setup

The solver used for secondary simulation is pressure-based with absolute velocity formulation. The gravity acceleration was set to -9.81m/s^2 in the z-direction. Transient was selected due to the expected turbulence effect with the simulation.

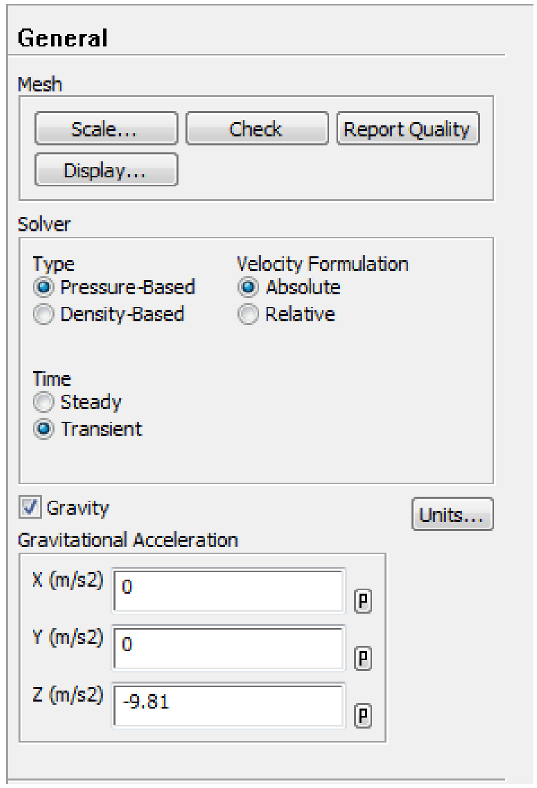

Figure 18: Secondary simulation general setup parameters

In models setup tab, multiphase mixture model is also turned on with 2 Eulerian phases. The viscous model is selected because it is the most suitable model for turbulent flow in mixture.

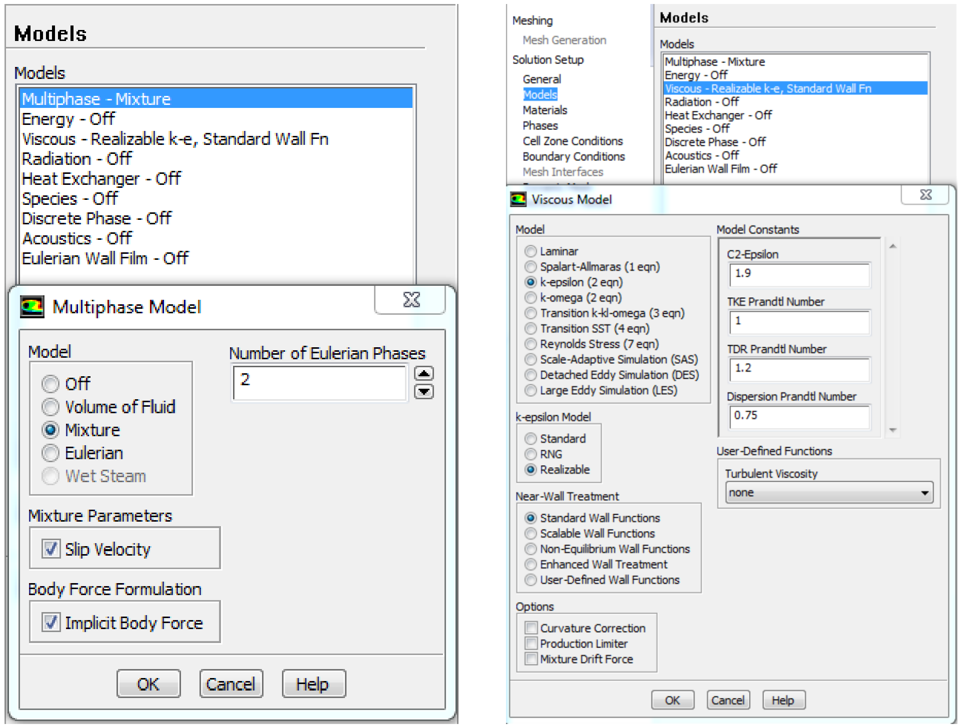

Figure 19: Secondary simulation models setup parameters

The material section for the secondary simulation still consists of water-liquid and air. This is mainly due to achieve further simulation objectives of running with different boundary conditions and mathematical models.

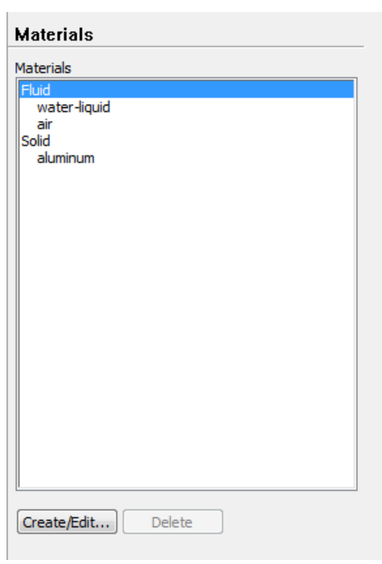

Figure 20: Secondary simulation material setup

Next is the boundary conditions setup. Here, the inlet condition is set as:

Gas phase: Velocity Specification Method: Magnitude and Direction, with 0.3 in Y-component & 0.7 in the Z-component.

Liquid phase: Velocity Specification Method: Magnitude and Direction, with 0.3 in Y-component & -0.7 in the Z-component.

This is to ensure that the flow each phase to move corresponding to its natural vectors, while the outlet is maintained as pressure-outlet.

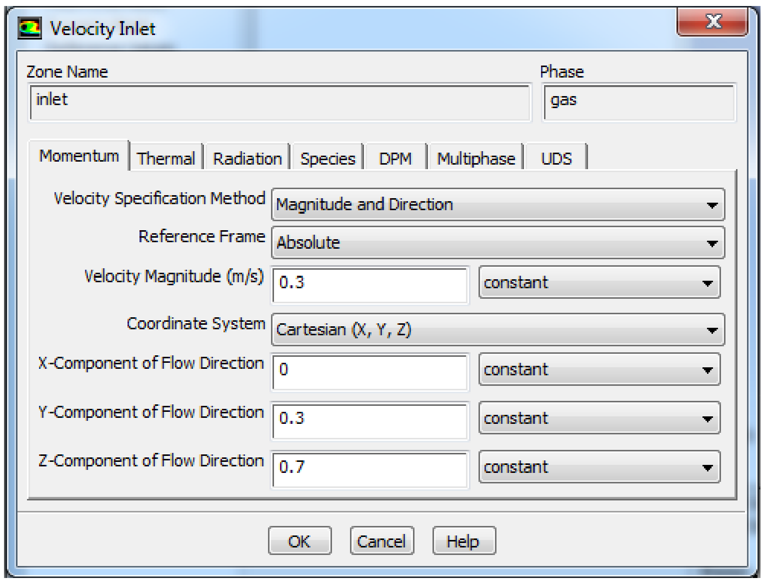

Figure 21: Secondary simulation inlet boundary condition setup for gas

Solution method for the further simulation is as shown in the figure below. This is partly to achieve further simulation objective.

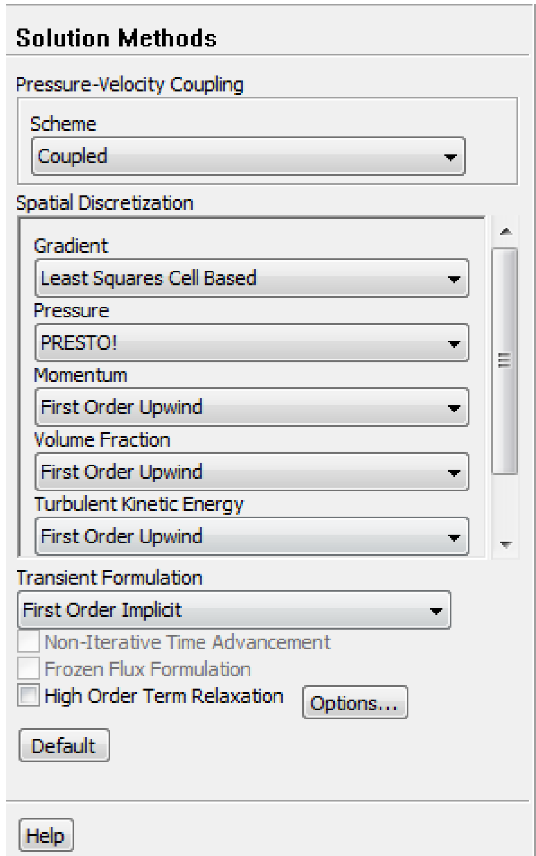

Figure 22: Secondary simulation solution methods setup

Solution is then initialized using hybrid initialization method and fluid is completely patched in the mesh. Calculation is set to 1000 iterations and then calculation is started.

Many try and error method is done in this phase of the project. Varieties of simulation methods were used, which inevitably resulted in a better simulation to obtain the best simulation result.

3.5.3 Final simulation Setup

The solver used for final simulation is still pressure-based with absolute velocity formulation. The gravity acceleration was set to -9.81m/s^2 in the z-direction. Transient setting maintained.

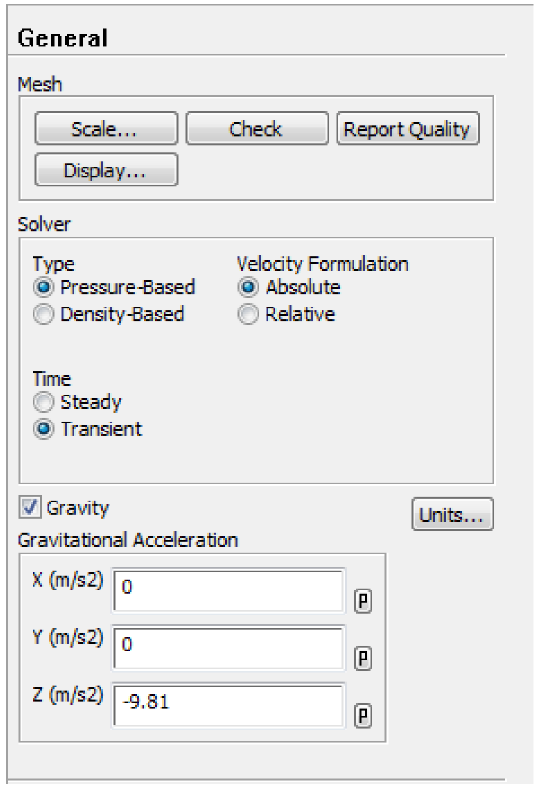

Figure 23: Final simulation general setup parameters

In models setup tab, Eulerian model is turned on with 2 Eulerian phases. The Eulerian parameter of "Multi-Fluid VOF Model" is also turned on with "implicit" scheme. The viscous model is selected because it is the most suitable model for turbulent flow in mixture.

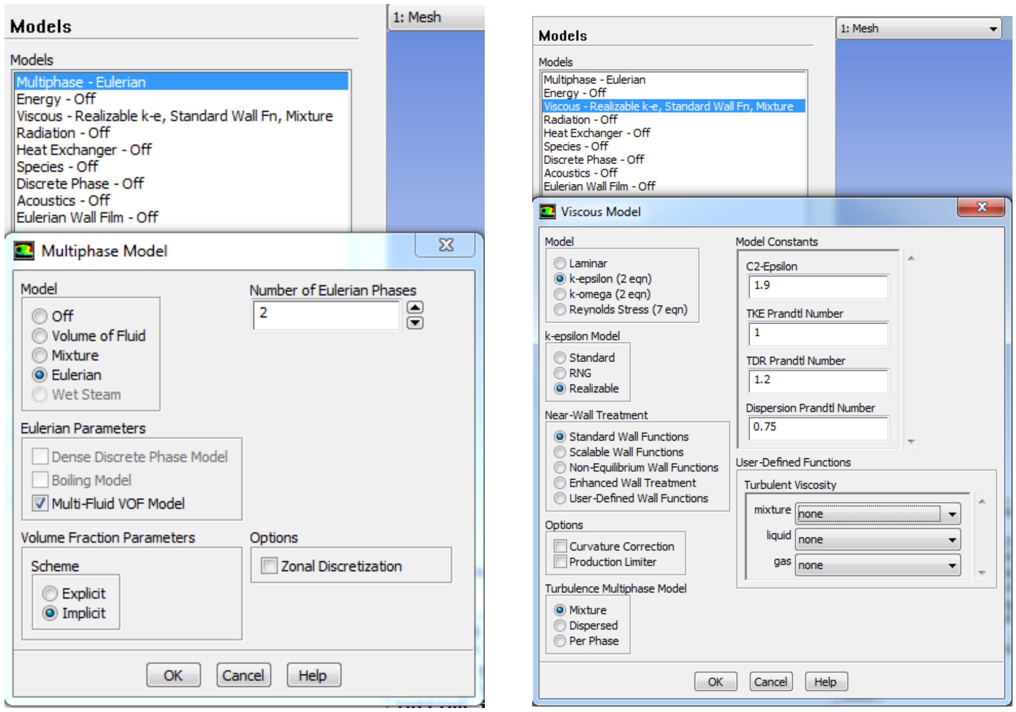

Figure 24: Final simulation models setup parameters

The material section for the final simulation consists of more realistic materials, that is fuel-oil-liquid and methane gas. Both of which were extracted from the Fluent Material Database.

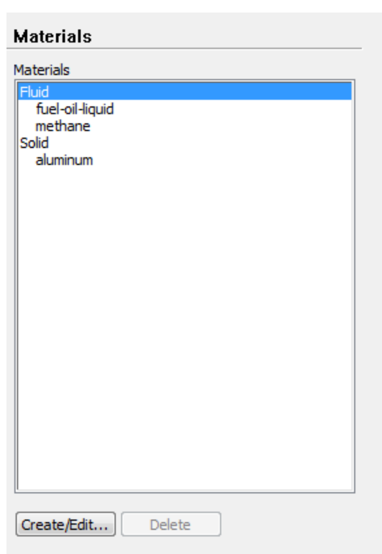

Figure 25: Final simulation material setup

Next is the boundary conditions setup. Here, the inlet condition for mixture is set as 1.2MPa of pressure and 2.9454m/s for both velocities of gas and liquid.

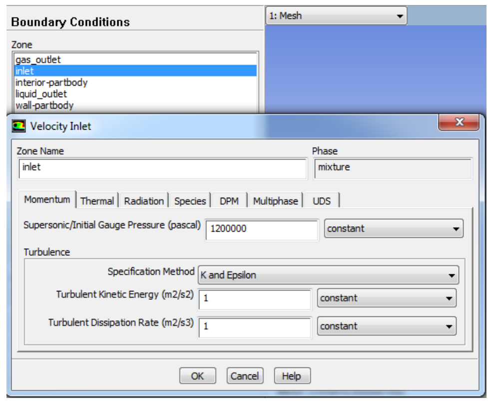

Figure 26: Mixture velocity inlet settings

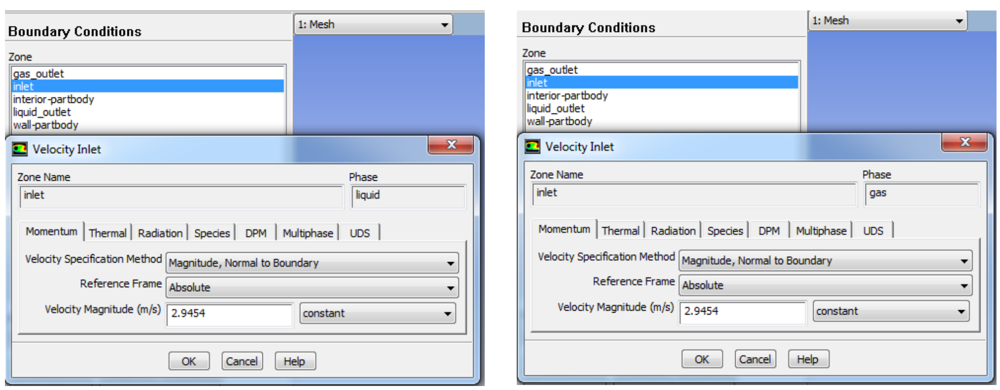

Figure 27: Liquid and gas setting at inlet

The pressures at both the outlets are kept the same at 1.0 MPa. This is to ensure that the flow will go out at the corresponding outlets. The outlet pressure data was just a try and error guess due to not being able to find a reasonably good operating data for it.

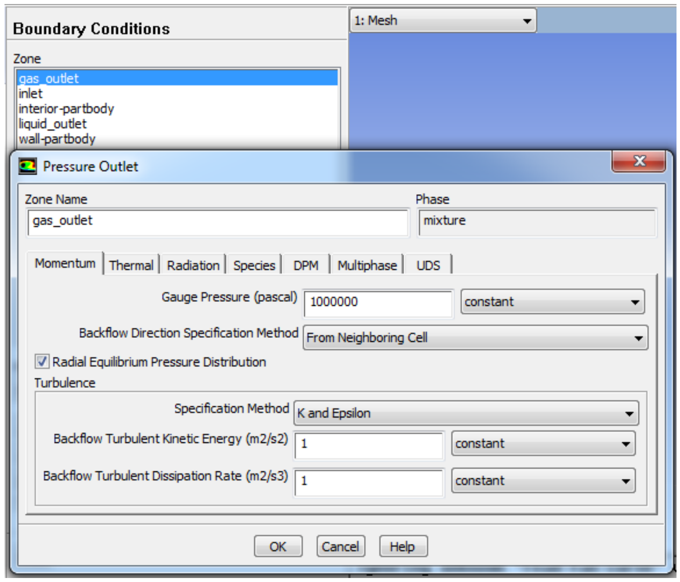

Figure 28: Pressure outlets pressure kept at 1.0MPa

Solution method for the final simulation is as shown in the figure below.

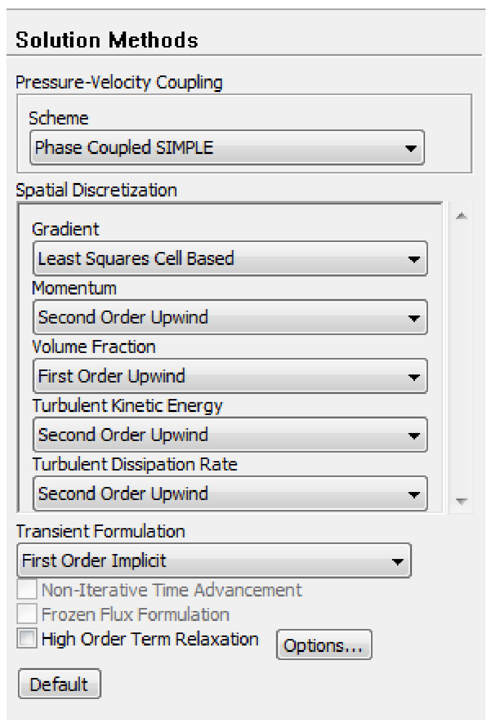

Figure 29: Final simulation solution methods setup

Solution is then initialized using standard initialization method and to be computed from the inlet.

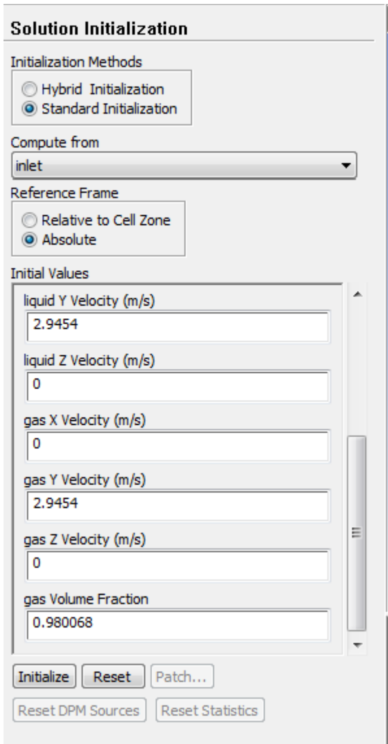

Figure 30: Initialized using standard initialization method

Model is then patched to a height of 0.5m from the bottom base with the liquid, fuel-oil-liquid.

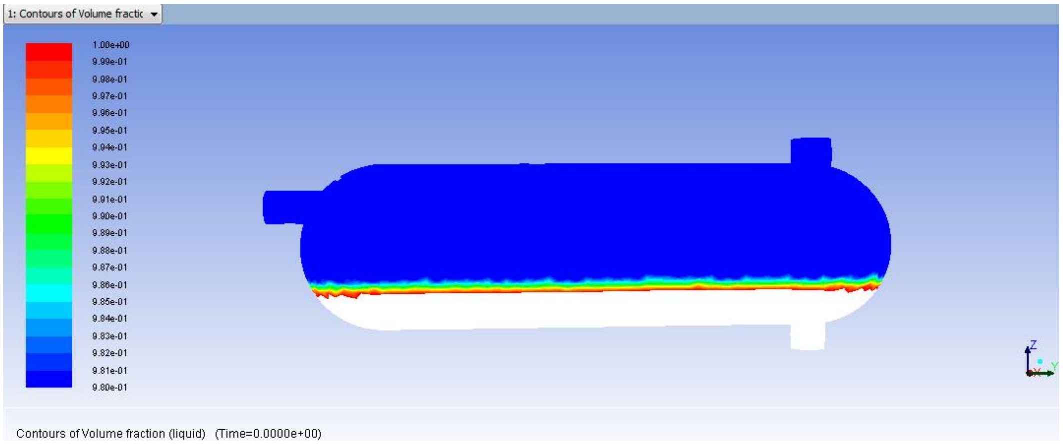

Figure 31: Bottom 0.5m of HPS marked and filled with fuel-oil-liquid

Calculation is set to 0.01s time step with 300 time steps. This in turn gives a total simulation time of 3s. Time steps were set to a maximum of 500 to hasten the calculation.

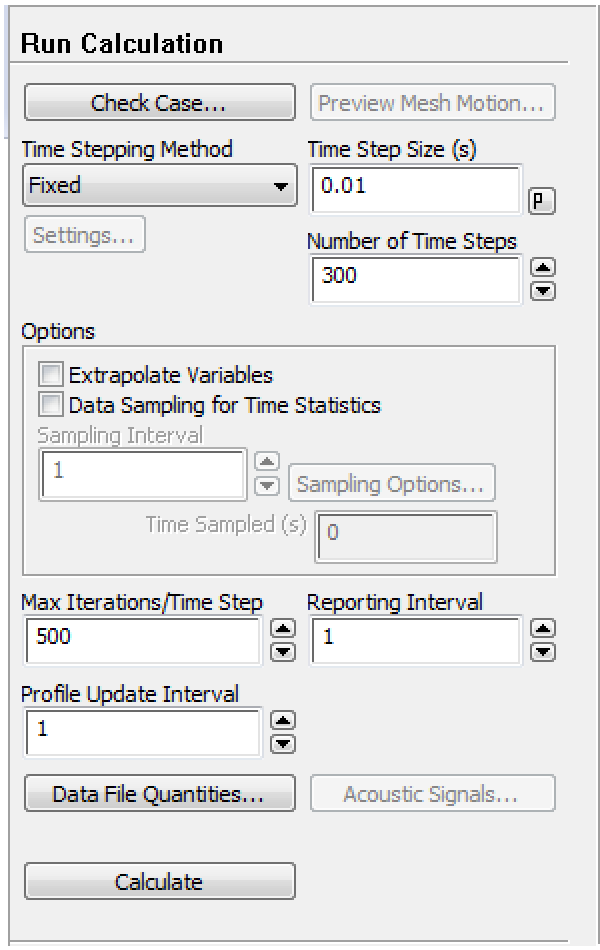

Figure 32: Setup used in this simulation

CHAPTER 4: RESULTS AND DISCUSSION

4.1 Preliminary Simulation Results

Preliminary simulation results are presented in the form of graphs, contours, vectors, etc. The most anticipated result would be the vector profile and is as shown below:

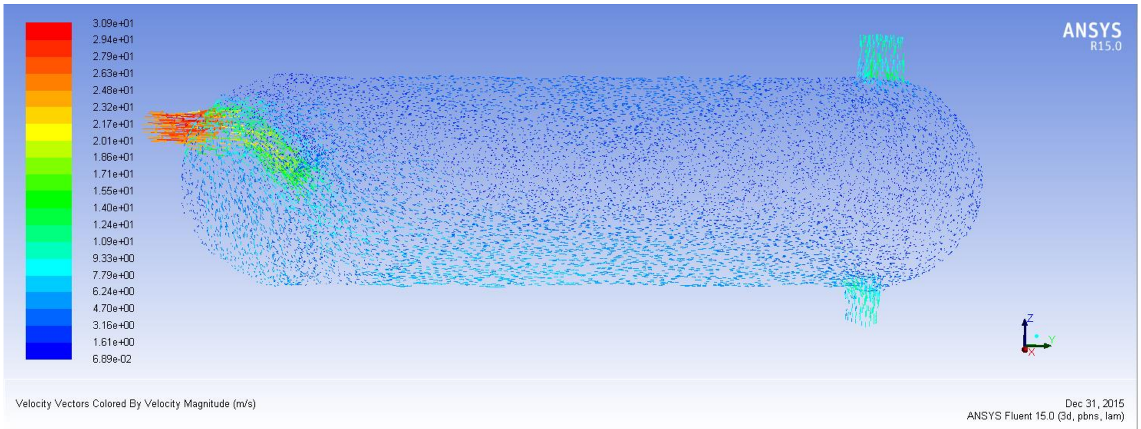

Figure 33: Velocity vector of the preliminary simulation

Due to the fact that the liquid and gas is understood by the software as a single fluid, these fluids do not have any difference in individual characteristics. In figure above, it is seen that mixture entering does not have any characteristics shown of it being liquid or gas, but more likely as one mixture. More adjustments are made which eventually lead up to a better result in further simulations.

4.2 Secondary Simulation Results

The best result of the secondary simulation is presented in velocity contours, streamlines and vectors of both gas & liquid phases, as shown below.

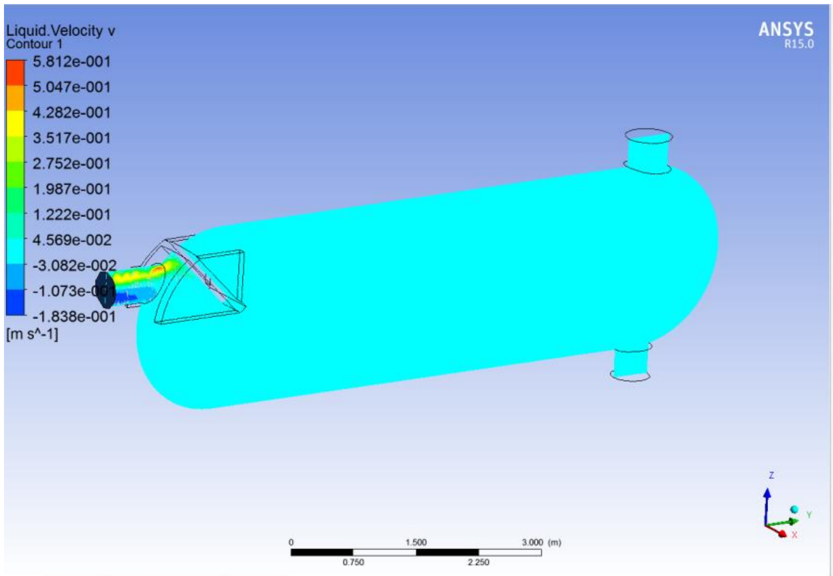

Figure 34: Liquid velocity contour

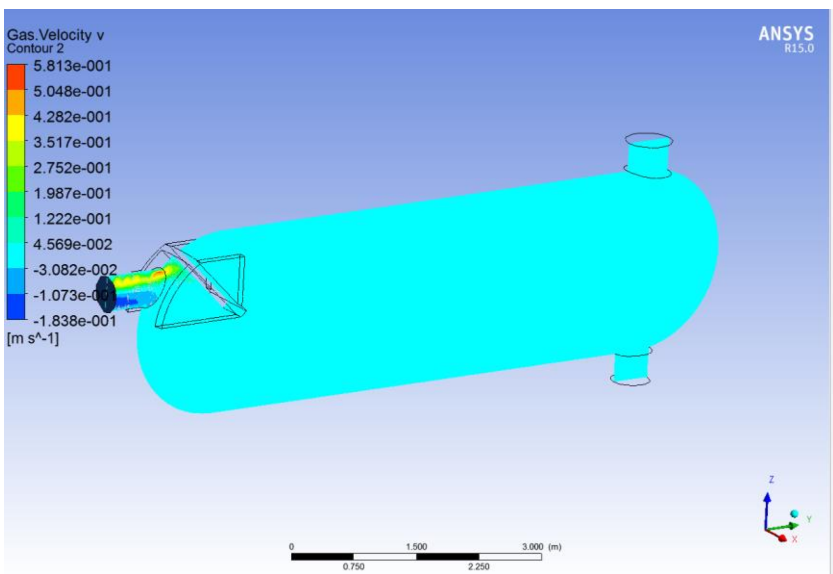

Figure 35: Gas velocity contour

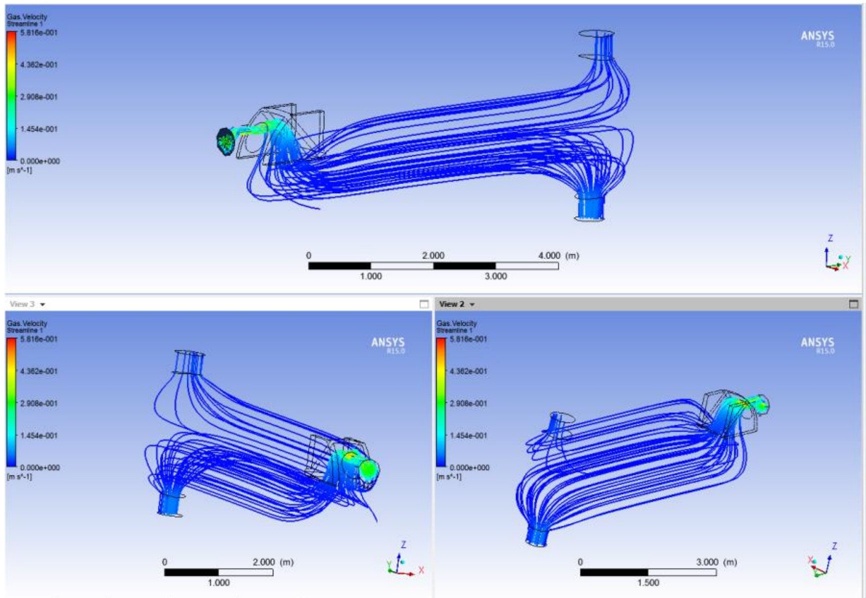

Figure 36: Gas velocity streamline

Gas streamline shows a good separation. Despite that, it can be seen that many gas particles went out the liquid outlet which is not quite right. There should only be traces of gas flowing out the liquid outlet, while majority of the gas goes out through the gas outlet.

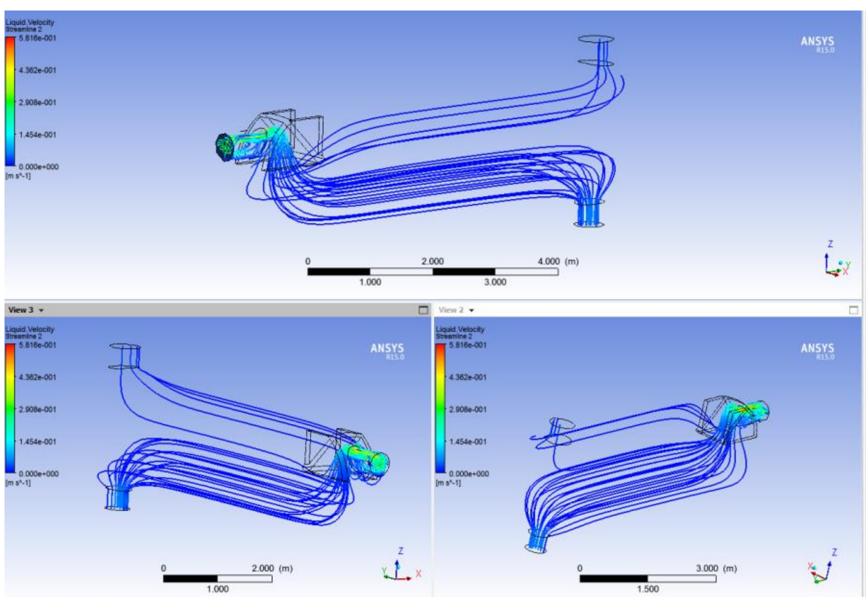

Figure 37: Liquid velocity streamline

This was also observed in the liquid velocity streamline, but in a sense that is correct nonetheless. Majority of liquid is seen to exit at the liquid outlet and only a few particles goes out at the gas outlet. This finding ultimately improves the understanding of multiphase simulation.

Both gas and liquid velocity vectors also show similar results to the streamline. Though, it is found that most particles lose their velocity way before reaching their corresponding outlets. This is seen to be inaccurate and thus, opted for another simulation to be carried out.

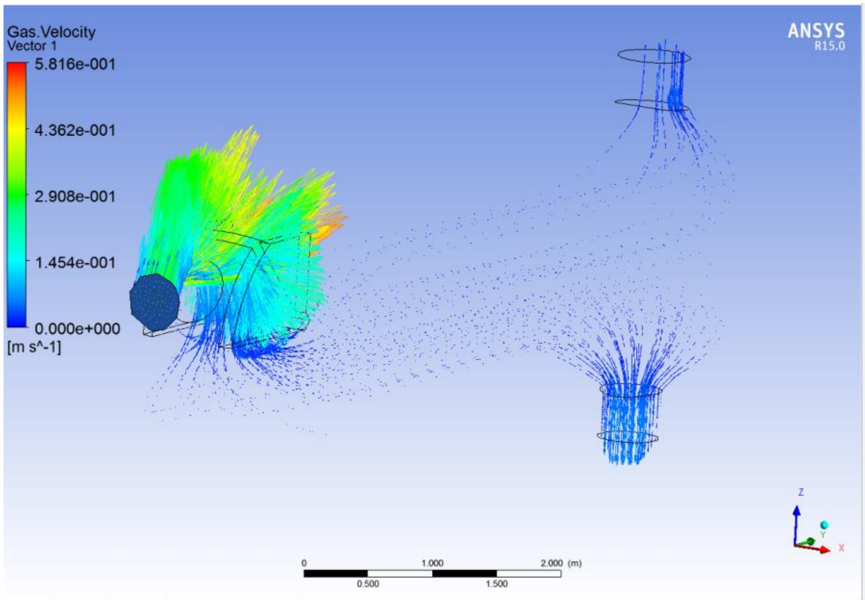

Figure 38: Gas velocity vector

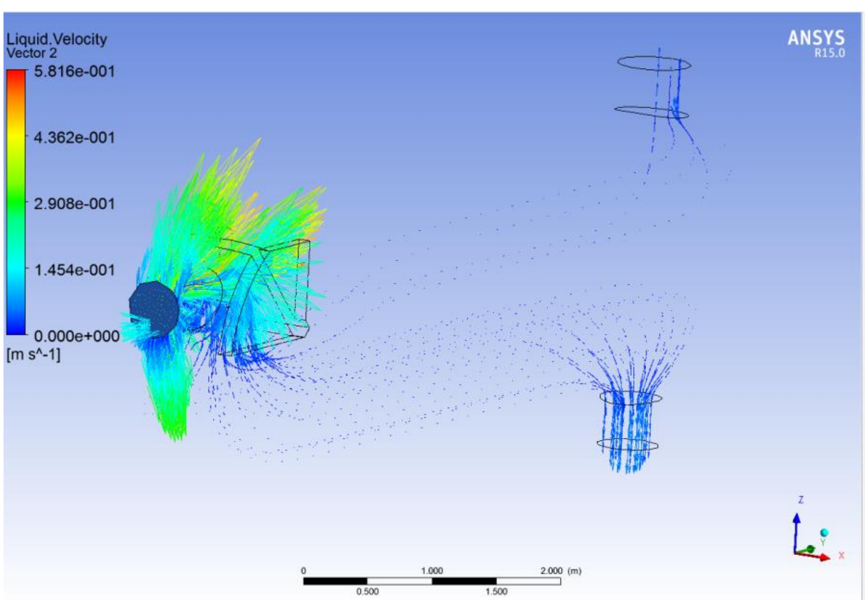

Figure 39: Liquid velocity vector

4.2 Final Simulation Results

The best result of the final simulation is presented in velocity contours, streamlines and vectors of both gas & liquid phases, as shown below.

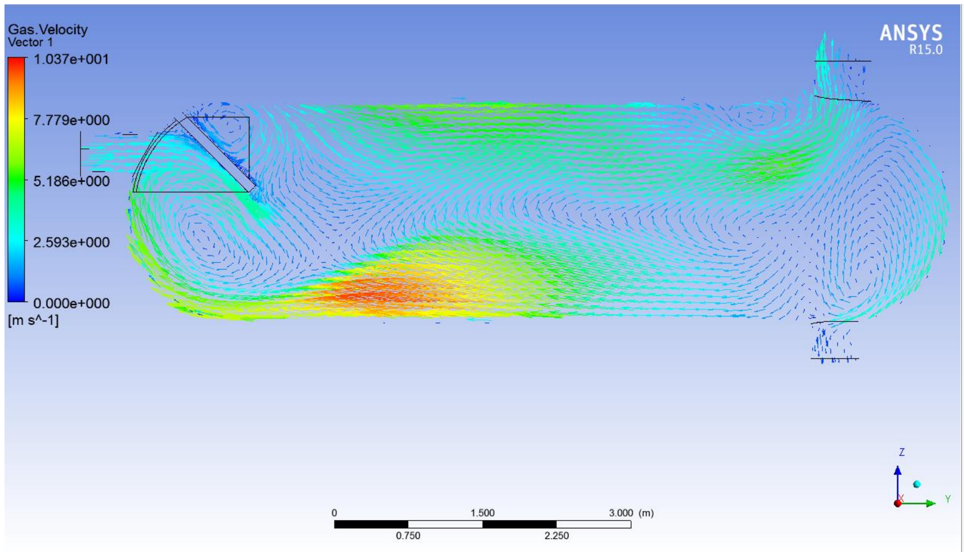

Figure 40: Gas velocity vector

Here it is clearly seen that the gas velocity is as what has been set up in the setup, which is 2.9454m/s at the inlet. It can also be seen that the high gas velocity created a lot of turbulence in the model and also pushed the patched liquid at the bottom part of the HPS to the side. Probably due to the small time step, the movement of the patched liquid cannot be fully simulated. Gas particles were seen to be exiting the outlets at both ends, but since the bottom part of the HPS is patched with fuel-oil-liquid, it is understood that the pressure created by the gas particles only pushes the liquid more towards the liquid outlet and no gas could pass the liquid barrier.

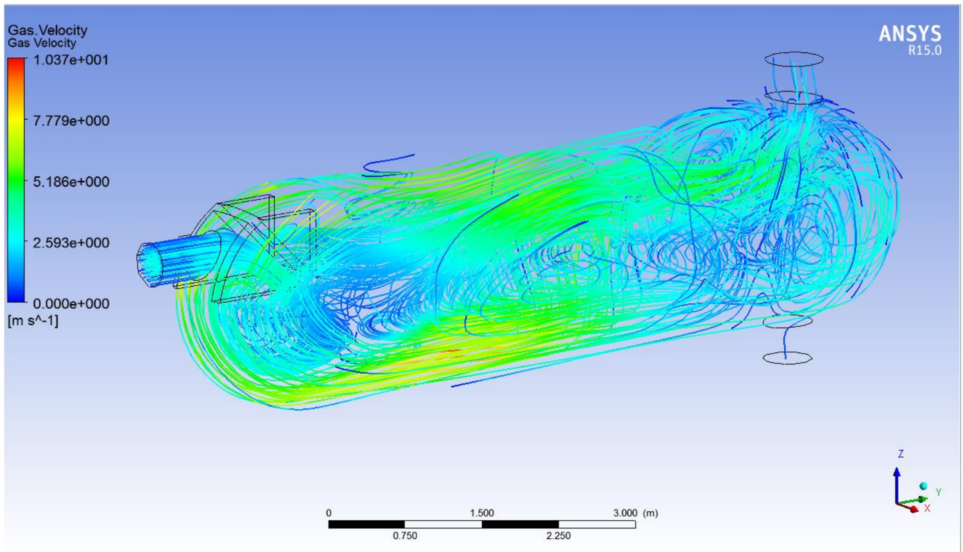

Figure 41: Gas velocity streamline

As seen above, the gas streamline shows a very convincing graphic. The particles is seen to be in turbulence inside the HPS and that is very much expected. Out of the 100 particles released in the HPS inlet, 8 went out the gas outlet and only 1 headed to the liquid outlet. This presents a notable result.

Other streamlines were found to either end at the bottom of the HPS of somewhere inside it. It is concluded that that is where the particles lose their energies and in this case, the gas would then be pushed by other gas particles which circulates inside the HPS and due to its very low density, it will float to the gas outlet located at the top right corner.

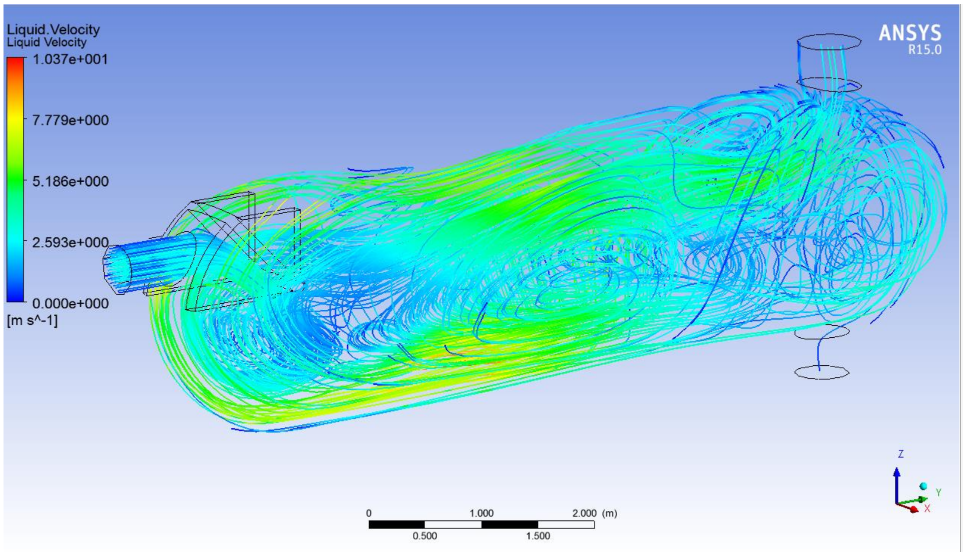

Figure 42: Liquid velocity streamline

On the other hand, the liquid velocity streamline shows a very convincing result. It can be seen that the particles flow turbulently in the HPS and out of the 100 particles put at the inlet, 9 went to the gas outlet. This gives an assumption of 9% of liquid particles of the mixture end up flowing out the gas outlet. This result is very much anticipated from the very beginning.

The possibility of the liquid percentage flowing out could be dramatically reduced if a vane pack were to be present before the gas outlet. Its zig-zag design should be able to reduce the liquid velocity and thus preventing it from going out the gas outlet. Nevertheless, 9% of liquid flowing through to the gas outlet is a very big number to be considered, hence the reason why liquid carry-over to the gas turbine occurs.

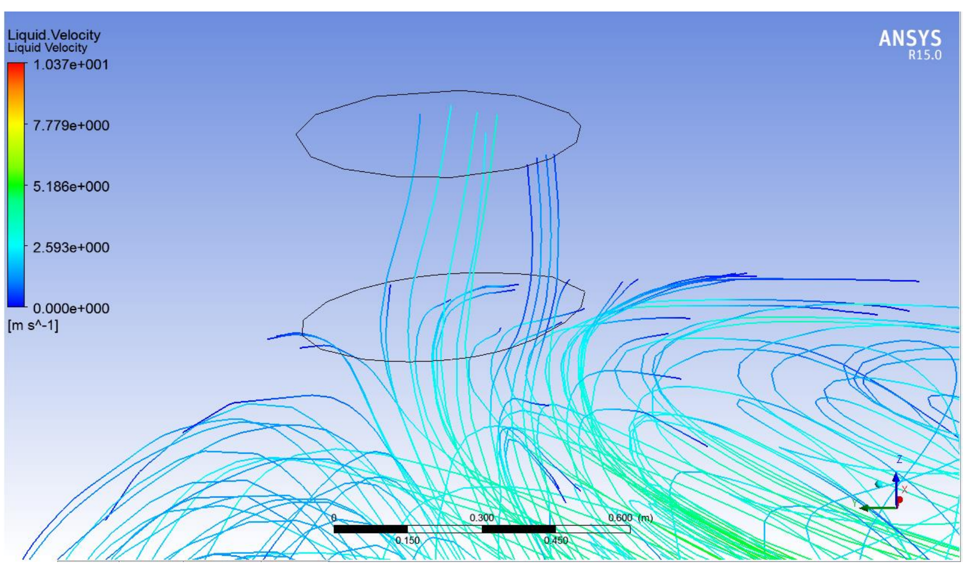

Figure 43: 9 liquid particles seen to be exiting the gas outlet

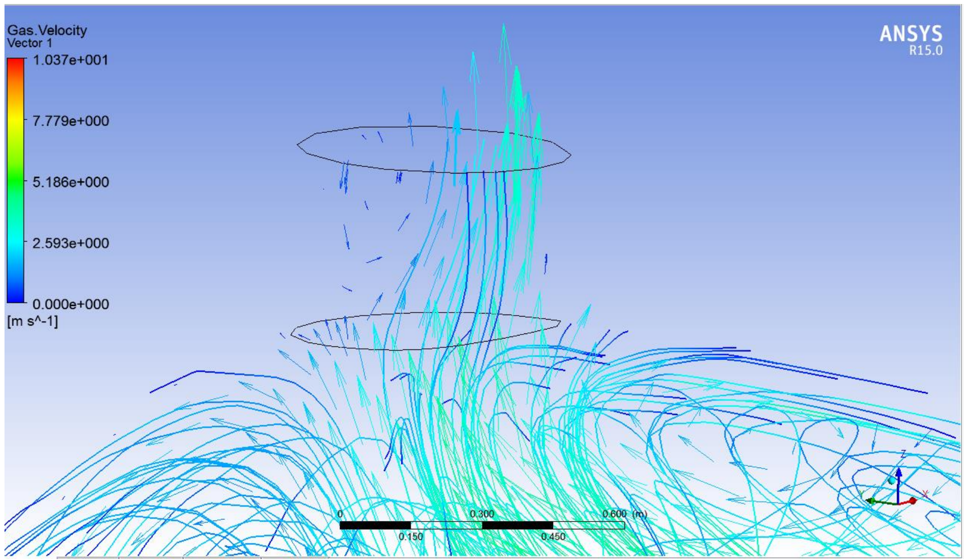

Figure 44: Gas velocity vector and liquid streamline

Another concern is regarding the gas velocity and liquid streamline. A local industry expert suggests that liquid carry-over is mainly due to the speed of the gas particles travelling in a HPS. In the industry, the mixture at the inlet is normally in the form of wet gas. The main constituent of it is mainly gas, such as in this project that has a volume fraction of gas of 98%. Hence when liquid is kept at a level at the bottom of the HPS; as patched in simulation, it is predicted that the high velocity gas would hit the liquid and some traces of liquid will be carried over with the gas to the gas outlet. This project indeed agrees to the statement and the above figure supports the claim.

CHAPTER 5: CONCLUSION AND RECOMMENDATION

This project is seen to be a success. With the CFD project result and model view, it would make the separator to be easily understood and be open for further developments or improvements in the future. Based on the results achieved, it is concluded that project objectives are achieved. Flow pattern of a multiphase flow in an HPS can be simulated. But due to its complexity, more studies need to be made so that more concrete results can be presented.

The main area of focus of the multiphase separation, has been modelled, simulated and studied to understand its behavior. The separation is understood to be due to the gravitational effect on the liquid and the very low density of gas.

The case of the liquid carry over is seen to be a major obstacle in the HPS. This project strongly agrees with that statement. But due to the absence of the vane pack, the result would not be as accurate as hoped. Further simulation with more number of elements of meshes would be recommended to enhance the results, but with that, more time will be needed as computational hardware plays a vital role.

The design of the deflector could also be revised so as to be deflected to the top side of the HPS. This is so that the gas would not be in direct contact with the liquid at the bottom of the HPS. That may indeed reduce the impact of crude carry-over by gas. More studies would be suggested for this method. A weir might also play a vital role in preventing splashes of liquid in the HPS. These theories must first be simulated before any change could be introduced to the industry.

REFERENCES

[1] H. Devold, ABB Oil and Gas, 3rd. ed., Oslo, 2013.

[2] H. I. Shaban, “A study of foaming and carry-over problems in oil and gas separators”. Elsevier Science Ltd., 1995.

[3] F. Kavousi, Y. Behjat, S. Shahhosseini, “Optimal design of drainage channel geometry parameters in vane demister liquid-gas separators”, Chemical Engineering Research and Design, 2013.

[4] E. Narimani, S. Shahhosseini, “Optimization of vane mist eliminators”, Applied Thermal Engineering, 2010, pp. 188-193.

[5] E. Stenmark, “Multiphase Flow Models in ANSYS CFD Software”, Göteborg: Chalmers University of Technology, 2013.

[6] Thome, J.R. (2004), “Engineering Data Book III”, Wolverine Tube Inc., Decatur, Alabama, USA

[7] A. J. Jaworski, G. Meng, “On-line measurement of separation dynamics in primary gas/oil/water separators: Challenges and technical solutions – A review. Journal of Petroleum Science and Engineering, 2009, pp. 47-59.

[8] H. I. Shaban, “A study of foaming and carry-over problems in oil and gas separators, 1995. Great Britain: Elsevier Science Ltd.

[9] M. J. Simmons, J. A. Wilson, B. J. Azzopardi, “Interpretation of the flow characteristics of a primary oil-water separator from the residence time distribution. Trans IChemE, 2002, pp. 471-481.