Separator Internal Design Optimization to Enhance the Separation Efficiency of Horizontal Separator by Using CFD

By

Zaihasfikrie B. Zainuddin

Dissertation submitted in partial fulfillment of

The requirement for the

Bachelor of Engineering (Hons)

(PETROLEUM ENGINEERING)

MAY 2013

Universiti Teknologi PETRONAS,

Bandar Seri Iskandar,

31750 Tronoh,

Perak Darul Ridzuan.

CERTIFICATION OF APPROVAL

Separator Internal Design Optimization to Enhance the Separation Efficiency of Horizontal Separator by Using CFD

By

Zaihasfikrie B. Zainuddin

A project dissertation submitted to the

Petroleum Engineering Programme

Universiti Teknologi PETRONAS

In partial fulfillment of the requirement for the

Bachelor of Engineering (Hons)

(PETROLEUM ENGINEERING)

Approved by,

(Ir. Dr. Shiraz Aris)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

MAY 2013

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 reference and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

(Zaihasfikrie B. Zainuddin)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

MAY 2013

ABSTRACT

Efficient and effective production fluid separation is required for a success of many production operations. Most of major producers of oil and gas faced the difficulty regarding the production fluid separation equipment known as separator. The separator does not perform 100% efficiency which leads to the uneconomical production of fluid. One of the reasons for this problem to happen, is the separator internal (inlet and baffle perforated plate) does not control the flow uniformity very well. Therefore the optimization of the separator internal is needed in order to meet the required separation process. Different types of inlet designs and type baffle designs were proposed to optimize the separation efficiency in this project. The design proposed based on the engineering point of view and to improve the existing design. The Resak field data were used as a case study for this project. The Computational Fluid Dynamics (CFD) method is used in optimizing the separator internal for better separation efficiency. In applying this method there are several step that have to be done to ensure the simulations are correct. The first step is by doing the meshing validation study. This study is to determine the minimum size of mesh to produce the most accurate result. From the research, for a three meter horizontal separator, the minimum size of mesh that must be used for meshing generation part is 0.02m. This is the critical size of mesh to get the most accurate result. Then move on to the second step in which the model validation of CFD modeling. In CFD modeling there are many fluid models that need to be chosen with respect to the purpose of modeling. For model validation, the CFD modeling of the separator is by duplicating the experimental work from the previous researchers. From the research for the horizontal separator, the fluid model that appropriate to the horizontal separator modeling is Multiphase model (Free surface model) and turbulent model (k-epsilon) with a value of turbulent Schmidt number is 35. In selection of improve inlet design, Inlet design 4 shows the highest separation efficiency which is **99.56%.** This is because the structure of the inlet design decreases the velocity of fluid which makes the separation process easier to happen. The large surface area in contact with the fluid at the end of inlet design and the present of porous plate give the advantages to inlet design 4 to reduce the velocity as much as possible. On the other hand, in the analysis of baffle improvement, baffle design 2 shows the improvement in term of separation efficiency. The improvement

by 1.81% efficiency from the baffle design 1 as the curvy baffle (Baffle Design 2) is more effective in stabilize the flow to reduce the fluid velocity compared to common vertical baffle (Baffle Design 1).

ACKNOWLEDGEMENT

I would like to express my highest gratitude to my supervisor, Ir. Dr. Shiraz Aris for giving me such a tremendous helps, advices, opinions, courage and support all the way through the completion of this project. I really appreciated the time allocates for me to sit and discuss the problems and any related issues regarding my project although he quiet busy with his work as Deputy Head of Petroleum Department.

My highest appreciation also goes to Final Year Project Coordinator, Mr. Muhamamad Aslam B Md Yusof for his efforts guiding me and all Petroleum Engineering students in completing our projects. A million thanks for his effort in assuring the smoothness of this course.

Special thanks to postgraduate student, Mr. Sanny and Mr. Fadhil for guiding me in conducting the simulation using Computational Fluid Dynamics (CFD) Software. Last but not least, A million thanks to my parents, my friends and everyone who directly and indirectly in helping me along the project.

Table of Contents

ABSTRACT ii
ACKNOWLEDGEMENT iv
List of Figures vii
List of Tables.. viii
Abbreviation and Nomenclatures ix

Chapter 1: Introduction... 1
1.1 Background of Study 1
1.2 Problem Statement.. 1
1.3 Objective and Scope of Study.. 2
1.3.1 Objective.. 2
1.3.2 Scope of Study.. 2
1.4 Relevancy of the Project 2
1.5 Feasibility of the Project 2

Chapter 2: Literature Review 3
2.1 Separator 3
2.2 Theory of separation 3
2.2.1 Momentum Separation 4
2.2.2 Momentum Control.. 4
2.3 Separation Efficiency.. 5
2.4 Computational Fluid Dynamics 6

Chapter 3: Methodology 11
3.1 Project Activities... 11
3.2 Research Methodology 11
3.3 Key Milestones 13
3.4 Gant Chart... 15

Chapter 4: Result & Discussion 16
4.1 Data Gathering and Analysis 16

4.2 Result & Discussion... 17
4.2.1 Meshing Validation (Sensitivity Study).. 17
4.2.2 Model Validation (Model Sensitivity Study).. 22
4.2.3 Phase 1: Inlet Design Improvement.. 26
4.2.3 Phase 2: Baffle Design Improvement 32

Chapter 5: Conclusion 34

References 35
Appendices 36

List of Figures

Figure 1: Procedure of implementing the CFD 8
Figure 2: Project Activities.. 11
Figure 3: Key Milestones 13
Figure 4: Complete Geometry development 17
Figure 5: Auto size mesh generation 18
Figure 6: Location of points in the separator.. 18
Figure 7: Pressure measurement at Point 1 with respect to different mesh number . 19
Figure 8: Pressure measurement at Point 2 with respect to different mesh number . 20
Figure 9: Comparison of Pressure Measurement at point 1 Between Existing Mesh and Modified Mesh.. 21
Figure 10: Comparison of Pressure Measurement at point 2 Between Existing Mesh and Modified Mesh.. 21
Figure 11: Geometry details based on research paper 22
Figure 12: Geometry development in CFD 23
Figure 13: Completed mesh generation for model validation 23
Figure 14: Position of points measured in Plane 1 24
Figure 15: Standard deviation of water velocity versus the turbulent Schmidt number 26
Figure 16: Inlet design separation efficiency 27
Figure 17: Inlet Design 1 27
Figure 18: Inlet Design 2 28
Figure 19: Inlet Design 3 29
Figure 20: Inlet Design 4 30
Figure 21: Fluid Velocity at inlet 31
Figure 22: Baffle design separation efficiency... 32
Figure 23: Baffle Design 1 33
Figure 24: Baffle Design 2 33
Figure 25: Mesh size of 0.05 m 36
Figure 26: Mesh size of 0.04 m 36
Figure 27: Mesh size of 0.03 m 37
Figure 28: Mesh size of 0.02 m 37
Figure 29: Mesh size of 0.0175 m 38
Figure 30: Mesh size of 0.015 m 38

Figure 31: Mesh size of 0.0125 m 39
Figure 32: Mesh size of 0.01 m 39
Figure 33: Geometry of Inlet Design 1 40
Figure 34: Mesh of Inlet Design 1 40
Figure 35: Geometry of Inlet Design 41
Figure 36: Mesh of Inlet Design 2 41
Figure 37: Geometry of Inlet Design 3 42
Figure 38: Mesh of Inlet Design 2 42
Figure 39: Geometry of Inlet Design 4 43
Figure 40: Mesh of Inlet Design 4 43
Figure 41: Geometry of Baffle Design 1 44
Figure 42: Mesh of Baffle Design 1 44
Figure 43: Geometry of Baffle Design 2 45
Figure 44: Mesh of Baffle Design 2 45
Figure 45: Isometric View 46
Figure 46: Front View 46
Figure 47: Back View 46
Figure 48: Side view 47

List of Tables

Table 1: Project Gant Chart 15
Table 2: Resak field data 16
Table 3: Two phase simulation data 16
Table 4: Coordinated of points measured 18
Table 5: Result of simulations 19
Table 6: Result of simulations for modified mesh 20
Table 7: Result for model validation 25
Table 8: Inlet design separation efficiency 26
Table 9: Baffle design separation efficiency 32

Abbreviation and Nomenclatures

3D	- CFD Software Tool
ANSYS FLUENT	- CFD Software Tool
ANSYS Spaceclaim	- Geometry development tool.
ART	- Actual residence time
Autocad	- Geometry development tool
CFD	- Computational Fluid Dynamics
HYSYS	- Process Simulation Tool
ICEM	- Geometry and meshing development tool
iCON	- The International Communication and Negotiation Simulations
OpenFoam	- CFD Software Tool
PETRONAS	- Petroliam Nasional Berhad
Petrosim	- Geometry and meshing development tool
Solidwork	- Geometry development tool
STAR CCM+	- CFD Software Tool
TransAT	- CFD Software Tool
TRT	- Theoretical residence time

Chapter 1: Introduction

1.1 Background of Study

In oil field terminology, the separator is the pressure vessel designed to separate well fluid produced from the oil and gas well into liquid and gaseous component. Separators used on the offshore platforms, separating bulk flows of water, oil and gas are vessels in which gravity settling due to density differences of the fluids taking place. The production rates and compositions of the fluids through the separators are frequently changing and hence this has affected the separation efficiency and production optimization efforts. The key issues to produce the economical production of oil and gas are primarily based on the separator and its performance. Therefore it is essential to ensure the separator achieve maximum efficiency in order to produce oil and gas economically.

There are a lot factors that contributing in the separator efficiency. One of them is because of the flow of the fluid is not uniform and the velocity of the fluid is too great resulting the separation is difficult to achieve. The separator internal is responsible to decrease the velocity of the fluid for making the separation easier to occur. However the improper design of the separator may not decrease the fluid velocity.

The CFD analysis was done to improve the efficiency of liquid-liquid horizontal separator by try to reduce the fluid velocity. This analysis mainly focuses on the design of the inlet and the configuration of baffles.

1.2 Problem Statement

The high efficiency of horizontal gravity separator will produce the oil and gas economically. However, not all separators will achieve the intended efficiency. One of the factors that lead to this phenomenon is the flow of fluid inside the separator is the fluid velocity is too high. When the velocity of fluid is high, the multiphase fluid is hard to separate out to their phase. Sometime, the fluid may slosh inside the separator as the velocity of the fluid is extremely high which make the separation hard to happen.

The influence of separator internal such as inlet and baffle can decrease the fluid velocity and provide the good separation efficiency. When the fluid flow is extremely high, the inlet design and baffles configuration may not effective enough to decrease the fluid velocity. Therefore improved designs for separator internals must be proposed to accommodate this situation. In this project, the aims to optimize the separator design which focus on the inlet design and baffle configuration to improve the flow of the fluid that enhanced separator efficiency and separation process of the liquid-liquid separator.

1.3 Objective and Scope of Study

1.3.1 Objective

The following is the objectives for this project.

- To conduct CFD analysis of the liquid-liquid separator using RESAK field data.
- To identify the improved inlet design that decreases the fluid velocity.
- To identify the improved baffle configuration that enhanced the separation efficiency.

1.3.2 Scope of Study

The scopes of study for this project are as follow..

- Development of fluid model in CFD analysis.
- The CFD analysis on different types of the inlet design.
- The CFD analysis on different types of baffles configuration.

1.4 Relevancy of the Project

This project is relevant to the separator located at the field (Development stage) where there are plan in the future to drill the well and tie in the production into the processing system (separator).

1.5 Feasibility of the Project

The project takes about 28 weeks to be completed.

Chapter 2: Literature Review

2.1 Separator

The separator used to separate the production fluid from the well into its individual phases. There are different types of separator available in the market nowadays. The types of separator can be classified into three categories which are separator based configuration; vertical, horizontal and spherical separator, separator based on number of phases involves; two phases or three phases, and separator based on its function; knockout-out vessel or scrubber (Khuzaimah, 2009). This project directed our focus more on the horizontal two phase separator (gravity separator).

The function of the separator is to separate the fluid, therefore the separation efficiency must be higher in order to produce the production fluid economically. The separation efficiency is very essential to be monitor time to time in order to decrease the cost of additional separation later on. The separator performance can be determined by look at the separation efficiency. The separation process can be affected by:

- The production fluid properties; temperature, pressure, viscosity, density etc.
- The flow properties; the types of flow, the flow rate.
- The design of the separator determine the efficiency of separation for a given fluid properties and operating conditions; Internal design and the size of the separator relative to the fluid flow properties.
- The liquid droplet sizes formed, if they are too small, they cannot settle out under the gravity alone, they must be coalesced with use of extractors or centrifugal force.

2.2 Theory of separation

The gravity separator has three main principles in separating the fluid into their individual component. These principles are separation based on gravity settling, the change in momentum and the coalescence of liquid droplet. As the aim for the project is to improve the design of inlet and baffle, the focuses is more on the momentum separation.

2.2.1 Momentum Separation

This principle explains about the change in velocity and direction of the fluid which lead to the change in momentum. These phenomena will lead to the separation of the fluid into their individual phase. The fluid with different densities will have different momentum. As the fluid stream change rapidly, the greater momentum will not allow the particles of the heavier phase to turn as rapidly as the lighter fluid. This may lead to separation to occur. The separation of different density can be affected with the sudden change in velocity. This is due to the difference in inertia between the fluids. The decreasing in velocity of high density fluid, the higher the inertia for the fluid to move away from the low density fluid.

The momentum is usually a technique to employ the bulk separation of the multiphase fluid in the stream (PETRONAS, 2009). The internal of the separator usually applied the separation based on momentum such as separator inlet and perforated baffle. The perforated baffle and inlet configuration improved the separation by enhance the uniformity of the flow across the cross sectional area of the separator (Mee & Mohamad Nor, 2011) by applying the momentum separation principle. Therefore the design of inlet and perforated baffle are very important to ensure the uniformity of the flow which lead the maximum separation.

2.2.2 Momentum Control

Inside the separator, the momentum of the fluid flow is control by its internal; the inlet and baffles perforated plates. These components are responsible to reduce the momentum of the flow which leads to the separation between the fluid phases.

2.2.2.1 Inlet Momentum Control

In the horizontal separator, several alternatives are available for controlling inlet momentum. Splash plates, dished heads, the Porta-Test Involute and the Port-Test Revolution™ are available from NATCO Group. The inlet device controls the inlet momentum by redirecting the inlet stream and dissipating the energy of the inlet fluid. The Porta-Test Revolution additionally utilizes the energy of the incoming fluid to eliminate foam (Chugh, 2011). The new inlet designs were proposed by increase the inlet surfaces area in contact with the fluid to decrease the velocity of the fluid.

2.2.2.2 Baffle momentum control

The design of the baffles is governed principally by the structural support required to resist the impact-momentum load. When the fluid flow and hit the baffle, the fluid will change it direction due to the barrier created by the baffle. At the same time the velocity of the fluid decreases. As velocity of fluid decreases, the higher inertia of the phases cause then to separated from each other (Smith, 1987).

2.3 Separation Efficiency

The separation efficiency of must be calculated in order to know the performance of the separator. There a several ways to determine the separator efficiency. One of them is by a measure of the weight of fluid separated out to the total weight of fluid (Spirax Sarco, 2013). The following is the formula of separator efficiency.

$$Separator\ Efficiency, \% = \frac{The\ weight\ of\ fluid\ separated\ out}{The\ total\ of\ weight\ of\ fluid\ in\ the\ stream} \times 100$$

For this project, the formula was modified by replace the weight with the mass flow rate of the fluid and assuming the mass flowrate move at 1 second. The difference between the weight and mass is the gravitational constant. When the value of gravitational constant at the denominator and numerator and can easily cancel it out.

The separation efficiency can be determined by calculate the volumetric utilization analysis of the separator. The volumetric utilization is basically used as indicator to determine the effectiveness baffles perforated plated design and inlet design on the liquid hydraulics. The volumetric data indicate the percentage of each phase volume being utilized compared to ideal case. The volumetric utilization can be calculated by using actual residence time (ART) and theoretical residence time (TRT). The residence time is the average amount of time for a fluid spends in a separator. ART can be measured in the experiment or in CFD result while the TRT need to be calculated by simply dividing the dividing the oil and water volume present in the vessel by corresponding oil and water flow rate entering into the vessel.(Lee, Khan, & Phelps, 2008)

The volumetric utilization can be calculated by using the following formula:

$$Volumemetric\ Utilization = \frac{ART}{TRT} \times 100$$

The separation efficiency is expected to be higher with larger residence time and volumetric utilization.

In the other way, the separation efficiency can determine by the reduction of velocity after flows through internal. In one research paper written by Vilagines and Akhras, they used the velocity reduction as indicator to determine the effectiveness of the baffles perforated plates and the separation efficiency.

2.4 Computational Fluid Dynamics

The Computational Fluid Dynamics (CFD) is a tool used to carry out the project. CFD is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process. CFD is a branch of Fluid Mechanics that uses Numerical Methods and Algorithms to solve and analyze problem involving fluid flows. CFD provide detailed insight of fluid flow in simple and complex 3D geometries, complementing other process simulation tools such as iCON, HYSIS and PETROSIM. The result of CFD analyses provide a relevant engineering data used in conceptual studies of new designs, detailed product development, troubleshooting, and redesign. The advantages using CFD are reduced the total effort required in the laboratory, reducing the total cost required for experimentation and provide comprehensive flow visualization.

The CFD analysis is a mathematical tool capable of simulating a wide range of fluid flows by solving Navier-Stokes equations. There are three mains governing equation used in CFD analysis which are:

- The continuity equation

$$\frac{\partial p}{\partial t} + \nabla .(\rho U) = 0$$

- The momentum equations

$$\frac{\partial(\rho U)}{\partial t} + \nabla.(\rho U \otimes U) = -\nabla p + \nabla.\tau + S_M$$

- The total energy equation

$$\frac{\partial(\rho h_{tot})}{\partial t} = \nabla.(\lambda \nabla T) + \nabla.(U.\tau) + U.S_M + S_E$$

CFD is a methodology and there are several software that used to run CFD simulation which are ANSYS-CFX (commercial), ANSYS-Fluent (commercial), STAR CCM+ (commercial), TransAT (commercial), and OpenFoam (open source). For this case, ANSYS-CFX software was used to run the simulation. Typically, a CFD software package consists of three main groups of software, a pre-processor, a solver and post-processor.

i. **Pre-processing**

Pre-processing includes geometry and mesh generation, flow specification, and setting solver control parameters. Once the geometry has been generated and meshed, the fluid properties, flow models and solver control parameters are specified and boundary and initial conditions applied. These steps are usually carried out through a graphical interface.

ii. **Solving the equations**

All the data defined in the pre-processing step are fed into the solver program in the form of a data file. The solver is a specialized program that solves the numerical equations based on the data specified in the data file. The results obtained by the solver are written to a results file for examination using the post-processor software.

iii. **Post-processing**

In this software, the data obtained by the solver can be visualized and displayed using a variety of graphical methods such as contour, plane, vector and line plots. Calculations can also be made to obtain the values of scalar and vector variables, such as pressure and velocity, at different locations.

Error! Reference source not found. shows the procedure of implementing the computational fluid dynamics.

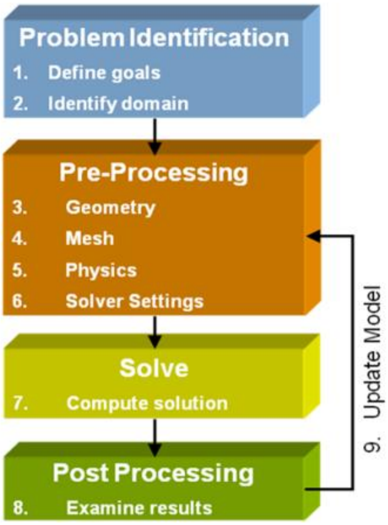

Figure 1: Procedure of implementing the CFD

a) Problem Identification

Define the problem and understand the purpose of the simulation. It is very important to understand as much as possible about the problem being formulate. At this stage, all necessary data required for simulation are collected including geometry details, fluid properties, flow specification and boundary and initial conditions.

b) Geometry development

Develop the geometry of the object of interest (domain) using 3D drawing software such as ANSYS Spaceclaim, Autocad, Solidwork or ICEM.

c) Meshing development

In this section, the domain is discretized into a finite set of control volumes or cells. The discritized domain is called the "grid" or the "mesh". For mesh generation, present software tools provide some predefined building units in a variety of forms, such as tetrahedral, pyramidal, hexahedral, and recently, polyhedral blocks. Variable gradients are generally more accurately calculated on a fine mesh than on a coarse one. A fine mesh is therefore particularly important in regions where large variations in the flow variables are expected. A fine mesh, however, requires more computational power and time. The mesh size is optimized by conducting a mesh-independence test whereby, starting with a coarse mesh, the mesh size is refined until the simulation results are no longer affected by any further refinement.

d) Flow specification

This stage involved; defining the fluid physical properties, selection of appropriate physical models, defining the boundary and initial condition, and prescribe operating conditions. The solver control is set up in this stage such as the convergence criteria and the number of iterations for the CFD analysis. In this case, all this information is set up in the Pre-processing section in the ANSYS-CFX software.

e) Calculation of the numerical solution

When all the information required for simulation has been specified, the CFD software performs iterative calculations to arrive at a solution to the numerical equations representing the flow. A number of iterations are usually required to reach a converged solution. Convergence is reached when changes in solution variables from one iteration to the next are negligible, residuals provide a mechanism to help monitor this trend or overall property conservation is achieved. The accuracy of a converged solution is dependent upon appropriateness and accuracy of the physical models, grid resolution and independence and problem setup. The user needs also to provide the information that will control the numerical solution process such as the advection scheme and convergence criteria. In this case, the discritized conservation equation is solved iteratively in the Solver section in the ANSYS-CFX software.

f) Result analysis

Once a converged solution is achieved, the user can then analyze the results in order to check that the solution is satisfactory and to determine the required flow data. In order to visualize these CFD simulation results and obtain qualitative aspects of the system, the post-processing tool of CFD software is used. If the results obtained are unsatisfactory, the possible source of error needs to be identified, which can be an incorrect flow specification, a poor mesh quality, or a conceptual mistake in the formulation of the problem.

CFD techniques have been applied on a broad scale in the process industry to gain insight into various flow phenomena, examine different equipment designs or compare performance under different operating conditions. CFD techniques are applicable in chemical process, aerospace, agriculture, automotive and many other fields as long as it involve fluid. The examples of CFD applications in the chemical process industry include drying, combustion, separation, heat exchange, mass transfer, pipeline flow, reaction, mixing, multiphase systems and material processing. CFD has also been applied to a number of food processing operations such as drying, refrigeration, sterilization, mixing and heat exchangers. For example, CFD has been used to predict the air flow and velocity. CFD has also been successfully used in modeling various multiphase flow systems, particularly gas-solid mixtures, although some limitations still exist. Multiphase CFD models can help understand the complex interactions between the different phases and provide detailed 3D transient information that experimental approaches may not be able to provide.

Chapter 3: Methodology

3.1 Project Activities

Figure 2 shows the overall project activities threw out the project.

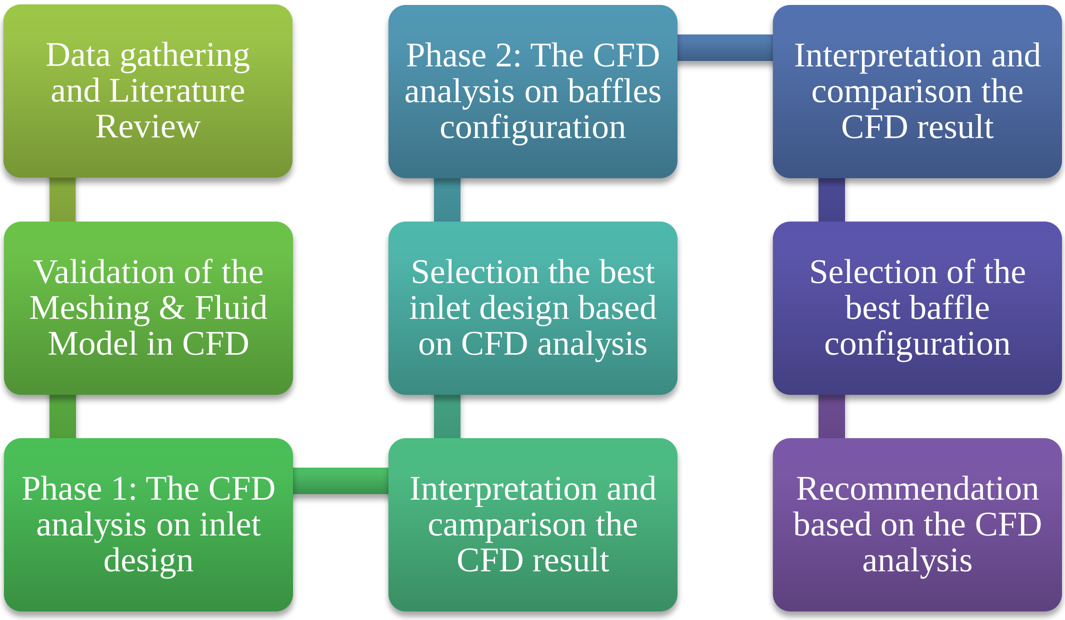

Figure 2: Project Activities

3.2 Research Methodology

The research methodology begins with the **meshing validation**. The purpose of this activity is to determine the appropriate number of meshing for the separator. According to Andre Bakker (2012) meshing is the discretization of the domain of interest into a finite set of control volumes. The dicretized domain called mesh or grid. The higher number of mesh the more accurate the simulation. However, the higher number of mesh may consume more time to reach the convergence of the calculation. Therefore the meshing validation is very importance step to determine the minimum number of mesh with the most accurate result.

The steps for meshing validation are as follow:

1) Develop one domain of interest (separator).
2) Generate the mesh to the separator using the auto size and measure the maximum size of the mesh produce.
3) Define all the fluid properties, the physical fluid model and the boundary condition of the separator.

4) Compute the simulation.
5) After simulation has complete, the measurement of a pressure at one point was taken somewhere in the separator.
6) Results were recorded.
7) Then, reduce the mesh maximum size of the separator.
8) Repeat step 3 to 7 until obtained a consistence measured of pressure at a point.
9) Lastly, decide the mesh maximum size of the separator based on the result.

Then the project proceeds with the **validation of the fluid model** used in CFD. The purpose of this activity is to determine the appropriate and accurate fluid model for the simulation of separator. After validated the model, then changes to the design and study about their efficiency can be made. The fluid model validation is implementing by carry out the CFD simulation on a particular separator and compares the simulation result to the experimental result and/or existing validated CFD result.

The **Phase 1** begins after the validation of meshing and fluid model have completed. In phase 1, different types of the inlet designs were proposed to improve the flow pattern and uniformity. The procedures of phase 1 are as follow:

1) The geometry of inlet design will be developed.
2) Generate the mesh to the completed geometry development.
3) Define the fluid properties, physical fluid model and boundary condition based on Resak Field data.
4) Compute the simulation.
5) After the simulation completed, the separation efficiency will be calculated and the velocity reduction inside the separator will be measured.
6) Then the phase 1 continues by repeating step 1 to 5 for the other two inlet designs.
7) Lastly, select the best inlet design with higher separation efficiency and low pressure drop.

After the selection of the inlet design, the **Phase 2** is initiated. In Phase 2, different types of baffle perforated plate design will be analyzed. The procedure for this phase is approximately the same as phase 1. The Phase 2 procedures are as follow:

1) The geometry of inlet design will be developed.
2) Generate the mesh to the completed geometry development.
3) Define the fluid properties, physical fluid model and boundary condition based on Resak Field data.
4) Compute the simulation.
8) After the simulation completed, the separation efficiency will be calculated and the velocity reduction inside the separator will be measured..
5) Then the phase 1 continues by repeating step 1 to 5 for the other two baffle perforated plate designs.
6) Lastly, select the best baffle perforated plate design with higher separation efficiency and low pressure drop.

3.3 Key Milestones

Figure 3 shows the key milestones of the project.

Figure 3: Key Milestones

1. Literature review
    - Deliverability
        - Gather information about the separator.
        - Gather the separation theory and principle of separation in separator.
        - Gather the information about the separation efficiency.
        - Gather the information about the Computational Fluid Dynamics (CFD).
2. Validation of fluid model
    - Deliverability
        - Validating the meshing of the domain of interest (separator).
        - Validating the fluid model in CFD used for the separator by comparing the result with the experimental result and/or validated CFD result.

3. Phase 1 (CFD analysis on inlet designs)
    - Deliverability
        - Geometry development of different inlet designs.
        - Meshing generation of the inlet designs.
        - Defining the boundary condition, fluid properties and the physical fluid model
        - Run a simulation using ANSYS CFX
4. Phase 2 (CFD analysis on baffle perforated plate design)
    - Deliverability
        - Geometry development of different baffles perforated plate designs.
        - Meshing generation of the baffles perforated plate designs.
        - Defining the boundary condition, fluid properties and the physical fluid model.
        - Run a simulation using ANSYS CFX
5. Documentation
    - Deliverability
        - Prepare the project technical report
        - Present the result of the project.

3.4 Gant Chart

Table 1: Project Gant Chart

		FYP 1																FYP 2															
No.	Details/Weeks	1	2	3	4	5	6	7	Mid semester Break	8	9	10	11	12	13	14	Semester Break	1	2	3	4	5	6	7	Mid Semester Break	8	9	10	11	12	13	14
1	Selection of Project Topic																															
2	Preliminary Research: i. Literature Review ii. Intro. to CFD																															
3	Extended Proposal Submission																															
4	Proposal Defence																															
5	Project work: i. Literature Review (continue) ii. Meshing Validation																															
6	Interim Draft Report Submission																															
7	Interim Report Submission																															
8	CFD Fluid model Validation																															
9	CFD analysis of Phase 1																															
10	CFD analysis of Phase 2																															
11	Result interpretation																															
12	Documentation																															

Chapter 4: Result & Discussion

4.1 Data Gathering and Analysis

The data used for all CFD modeling are based on the real field data collected at the RESAK field. All the fluid properties and flow properties are the real fluid properties produced from one well at RESAK field. **Table 2** shows the RESAK field data used for the simulations.

Table 2: Resak field data

	Water	Condensate
Bulk Mass Flowrate (kg/s)	20	
Density (kg/m3)	995.1	770
Viscosity (Cp)	0.65	0.57
Surface Tension (Dyne/cm)	37.34	17.05
Volume Fraction	0.53	0.47

However, for model validation, the data used for the simulation is based on research paper entitled "Baffle plate configurations to enhance separation in horizontal primary separators".

Table 3 shows the details of simulation data on development of CFD model.

Table 3: Two phase simulation data

Inlet Superficial Velocity	0.011m/s
Baffles Free surface area	100%
Air volume fraction	0.5
Water volume fraction	0.5
The water standard deviation (output)	0.0267m/s

For three phase simulation, all the data are the same as the two phase simulation except the fluid volume fraction. For three phases, the oil (density 805 kg/m^3, viscosity 0.014 Pa s) was introduced into the separator. The volume fraction for air, water and oil are 0.5, 0.4 and 0.1 respectively.

4.2 Result & Discussion

This project involves the use of CFD analysis and a few softwares were used in order to carry out the analysis such as Design modeler, ICEM and ANSYS CFX.

4.2.1 Meshing Validation (Sensitivity Study)

The meshing sensitivity study starts with the geometry development by using the Design Modeler. Since there is function computing a simulation for a symmetry object in CFD, the geometry of the separator design half from the actual geometry. The reason behind is to decrease the time of simulation. **Error! Reference source not found.** shows the completed geometry development for the study.

After completed the geometry development, generated the mesh on it by using the ICEM software. For first cases, used auto size meshing and determine the maximum size of the mesh. **Error! Reference source not found.** shows the meshing generated for the auto size mesh.

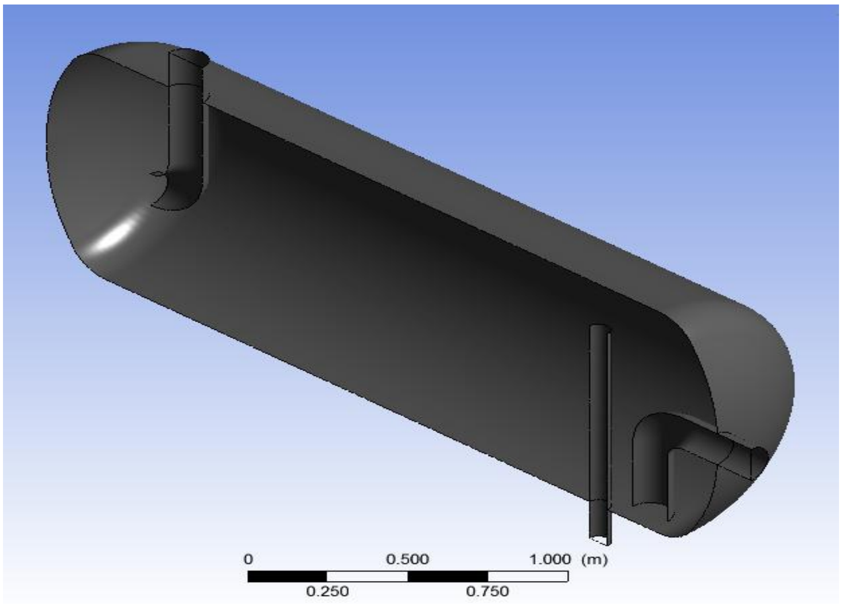

Figure 4: Complete Geometry development

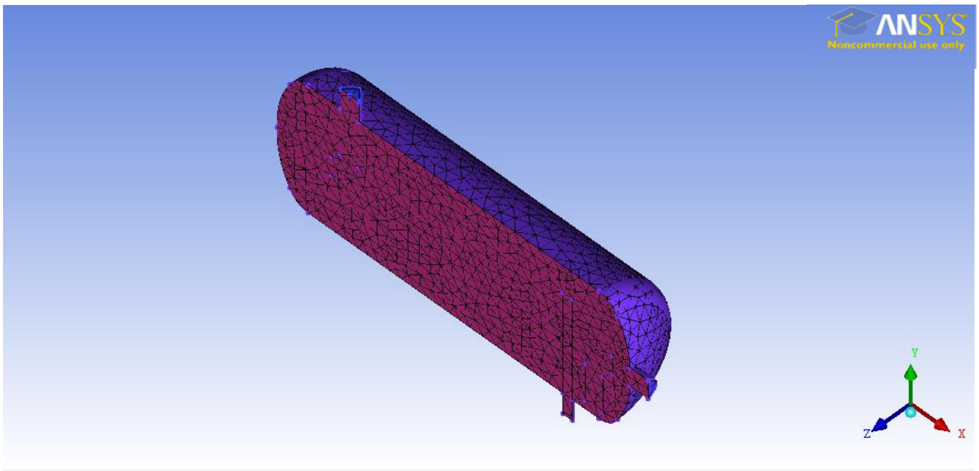

Figure 5: Auto size mesh generation

The maximum size of the mesh is **0.11 m** and the total amount of mesh generated using this size is **21184 elements**. Using the meshing generated to run a simulation. After completed the simulation, the pressure from 2 different points in separator were measured.

Table 4: Coordinated of points measured

Position	Coordinates (x,y,z)
Point 1	(0.4,0,0)
Point 2	(-0.25,-0.7,0)

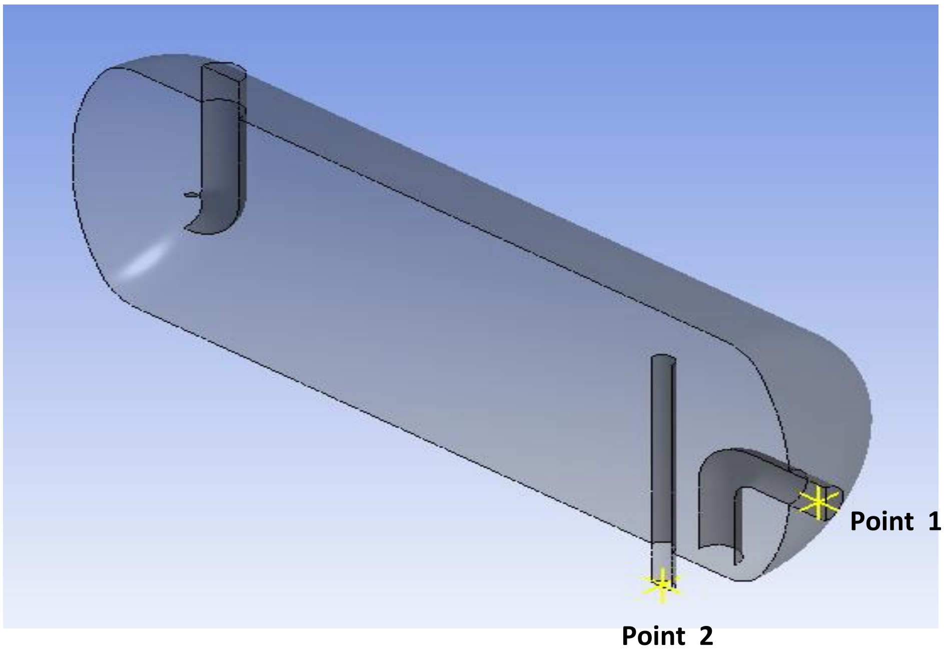

Figure 6: Location of points in the separator

Then reduce the maximum mesh size accordingly with relative to the result of the simulation. The reduction of mesh size will cause the number of mesh increases. Refer **Appendix 1** to see the meshing generated for all the size of the mesh used in this study. **Table 5** shows the result of the simulation.

Table 5: Result of simulations

Max size of mesh (m)	No. of Mesh	Pressure at point 1 (pa)	Pressure at point 2 (pa)
0.0500	80348	2129970.070	2129973.247
0.0400	147590	2129974.826	2129978.917
0.0300	225039	2129985.235	2129987.091
0.0200	634424	2129988.728	2129990.061
0.0175	730050	2129988.964	2129990.397
0.0150	1068309	2129990.351	2129993.289
0.0140	1238506	2129990.915	2129993.800
0.0125	1467403	2129991.937	2129994.735
0.0100	2473977	2129992.191	2129994.893

From the result obtained in the table, the graphs of pressure versus number of mesh are plotted.

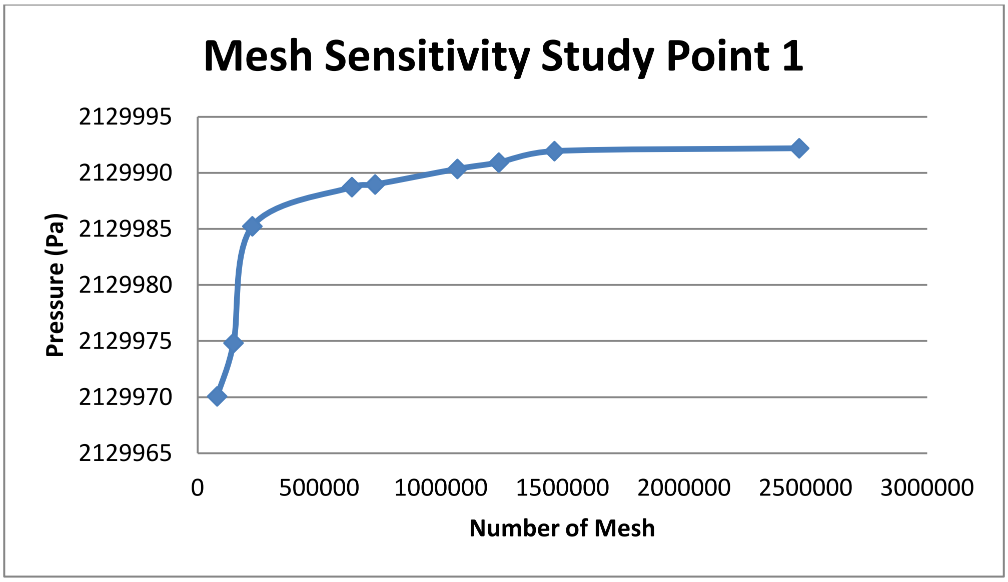

Figure 7: Pressure measurement at Point 1 with respect to different mesh number

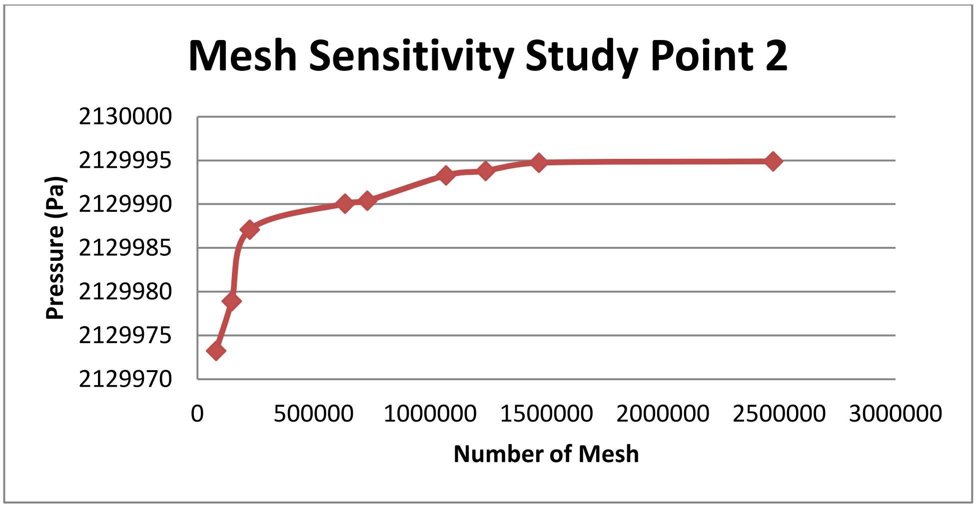

Figure 8: Pressure measurement at Point 2 with respect to different mesh number

From **Error! Reference source not found.**, and **Error! Reference source not found.** its clearly shows that the pressure at these three points and the average pressure at Plane 1 are approximately the same after reach the 1500000 number of mesh which respect to 0.0125 m maximum size of mesh. These mean that the pressure is does not effected anymore by the increasing number of mesh. However, the number of mesh is too large which may take longer computational time. Therefore, the modification of mesh must be done. For the modification, the plan is to vary the size of mesh throughout the geometry. The finer mesh were computed at the specific area which where the important points were measured. Then for the other part of the geometry, the mesh is coarser. Refer **Appendix 3** to see how the variation of mesh size looks like. The sensitivity study was done to select the appropriate variation of mesh size throughout the separator and the results were compared to the previous mesh sensitivity study.

Table 6: Result of simulations for modified mesh

Max size of mesh (m)	No. of Mesh	Pressure at point 1 (pa)	Pressure at point 2 (pa)
0.2000	125503	2129976.98	2129991
0.1000	160051	2129986.79	2129990
0.0500	309613	2129988.28	2129994
0.0400	506892	2129990.669	2129994
0.0300	781248	2129991.079	2129995
0.0200	1050990	2129992.199	2129995
0.0100	2420247	2129992.3	2129995

Table 6 Shows the simulation result for the modified mesh.

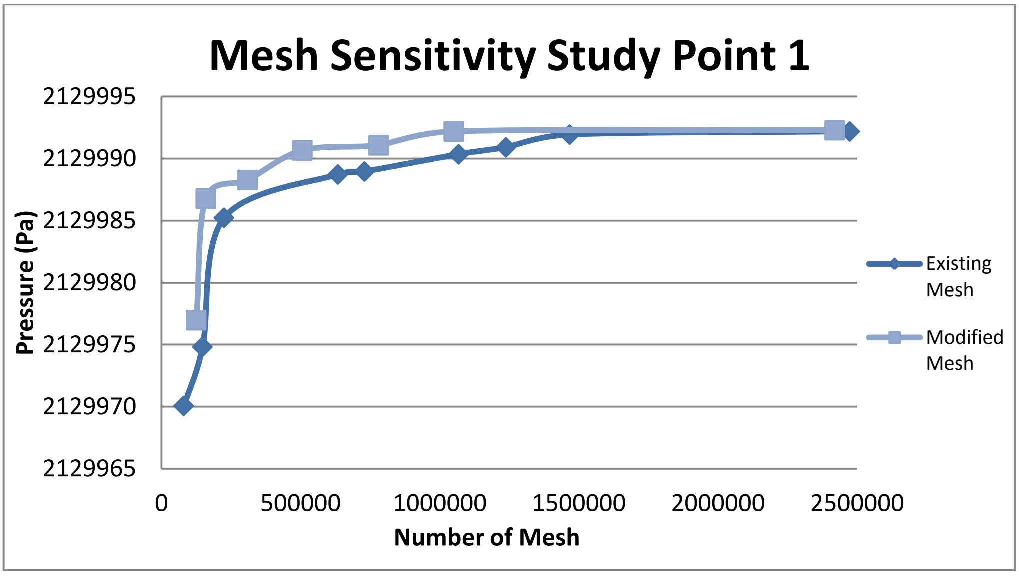

Figure 9: Comparison of Pressure Measurement at point 1 Between Existing Mesh and Modified Mesh

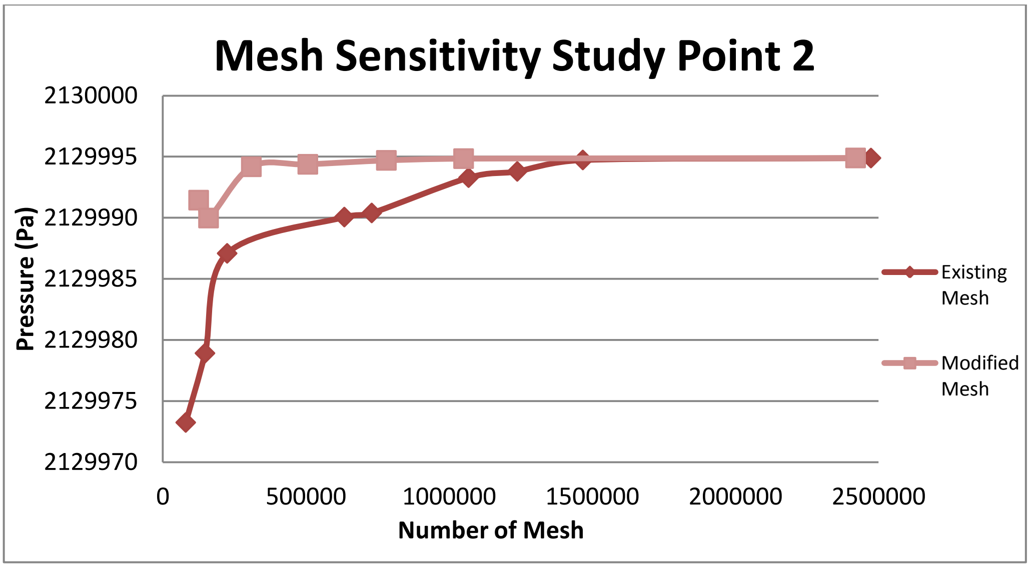

Figure 10: Comparison of Pressure Measurement at point 2 Between Existing Mesh and Modified Mesh

Figure 9 and **Figure 10** show the comparison pressure measurement at point 1 and point 2 respectively between existing mesh and modified mesh. For modified mesh, the graph converged faster compared to the existing mesh. When using modified mesh, at point 1 and 2, the graph start to converge at 1000000 number of mesh while the existing mesh converged at number of mesh of 1500000. Therefore by using the modified mesh, the mesh number can be saved up to 33% of previous mesh. The mesh size for 10000000 meshes is 0.02m.

It concluded that **0.02 m** is critical mesh size with the minimum number of the mesh that gives the optimum result.

4.2.2 Model Validation (Model Sensitivity Study)

The purpose of model validation is to find the appropriate model in CFD for the horizontal separator. The model validation was done by duplicating the experimental work from research paper entitle the Baffle plate configurations to enhance separation in horizontal primary separators by Derek Wilkinson, Brian Waldie, M.I. Mohamad Nor, and Hsio Yen Lee in 1999. All input for the simulation was taken from the research paper. **Figure 11** and **Figure 12** show the geometry details based on their research paper and the completed geometry development in the CFD respectively.

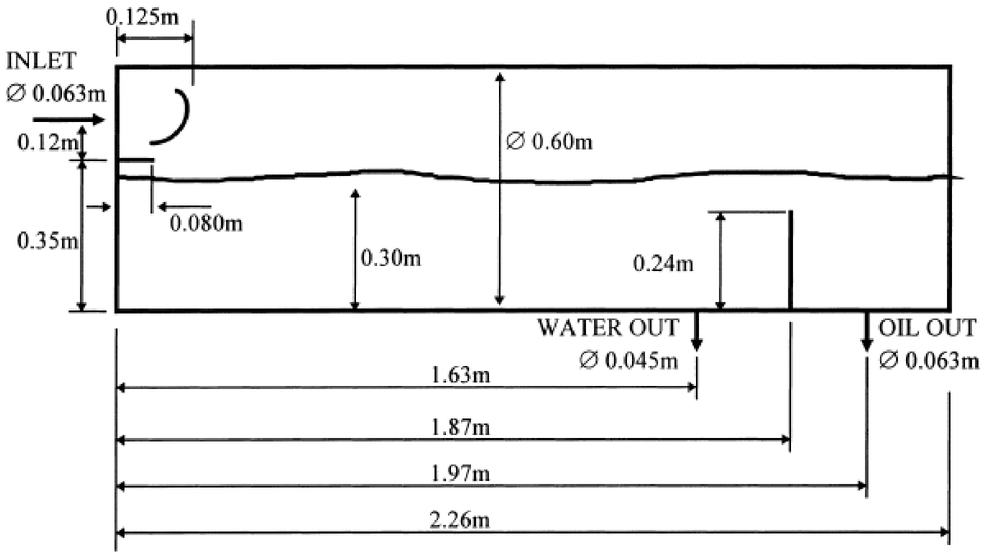

Figure 11: Geometry details based on research paper

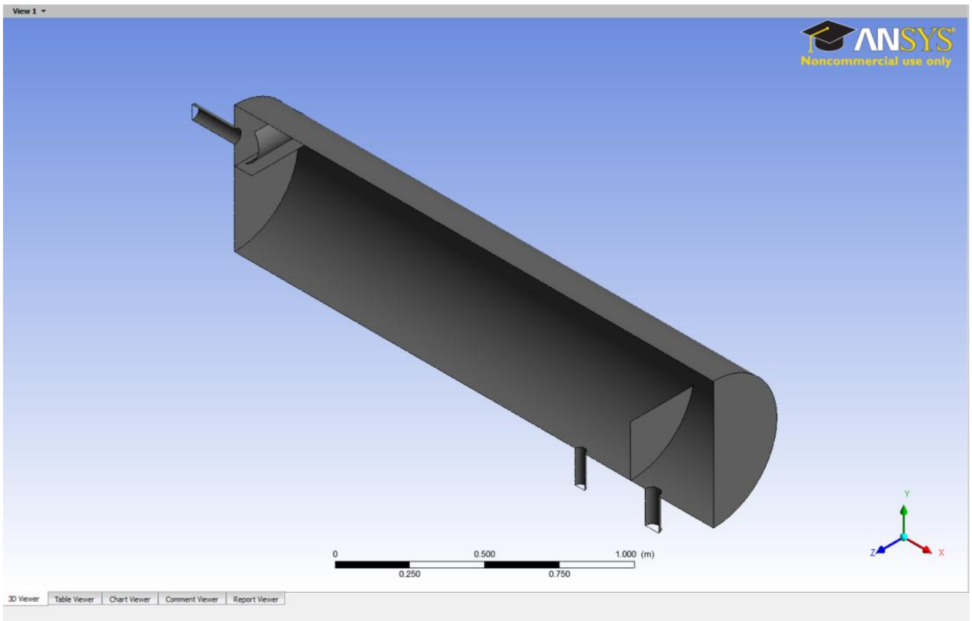

Figure 12: Geometry development in CFD

Then, the completed geometry was meshed using a 0.0125 m maximum number of mesh based on the result obtained from the meshing validation part. The total number of mesh generated is 718989.

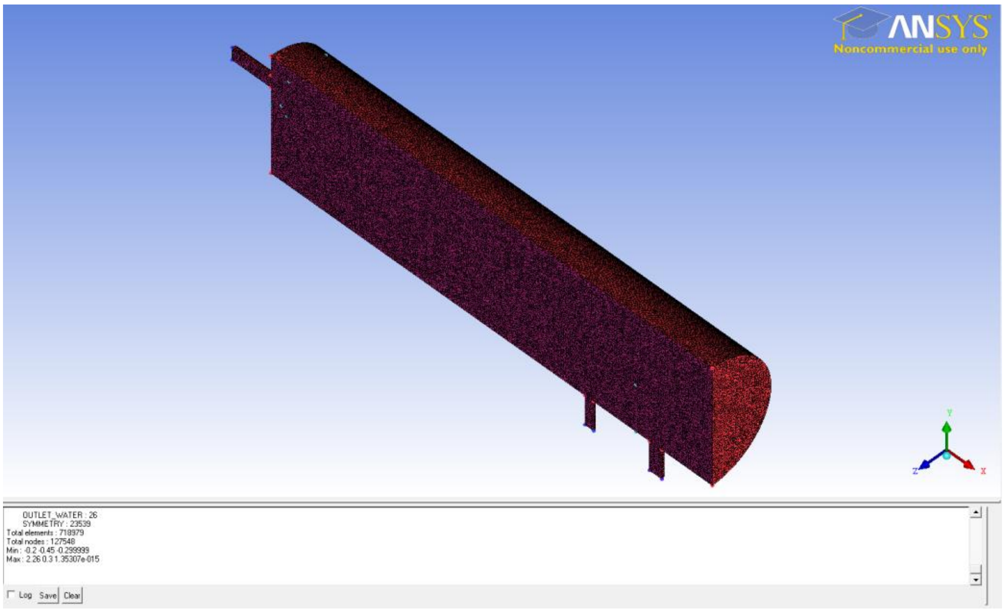

Figure 13: Completed mesh generation for model validation

When the meshing part completed, the model is ready for simulation. For the simulation, the physical fluid model was used is multiphase model (Free Surface Model) and turbulent model (k-epsilon). The Free Surface model was used because in the separator there is separation between the fluids with different density due to the gravity forces. Therefore it is more appropriate to use the free surface model instead of mixture model in the CFD. For the turbulent model, the k-epsilon model as chosen. The reason behind is because local velocities were very much higher than the superficial velocity (cf. the inlet velocity and superficial velocity) and the Reynolds number of flow through baffle holes was turbulent in many cases **(Wilkinson, Wildie, Mohamad Nor, & Lee, 1999)**. Besides, most of the researcher of CFD analysis of gravity separator used k-epsilon as their turbulent model. However, the value of kinetic energy and epsilon (dissipation energy) plays important roles in expressing how the flow of fluid inside the separator. Thus the kinetic energy value and dissipation energy value were varied. In ANSYS-CFX the value of both parameters is control by turbulent Schmidt Number.

In model validation the water velocity standard deviation at 0.6 m from the inlet will be compared between experimental result and simulation result. The experimental result shows the value water velocity of standard deviation is 0.0267 m/s. Therefore our goal is to play around with the value of turbulent Schmidt Number until the result of simulation close to the experimental result. In CFD work, the water standard deviation was determined by measuring 30 point at the plane 0.6 m from the inlet. **Figure 14** shows the position of a group points being measured at Plane 1.

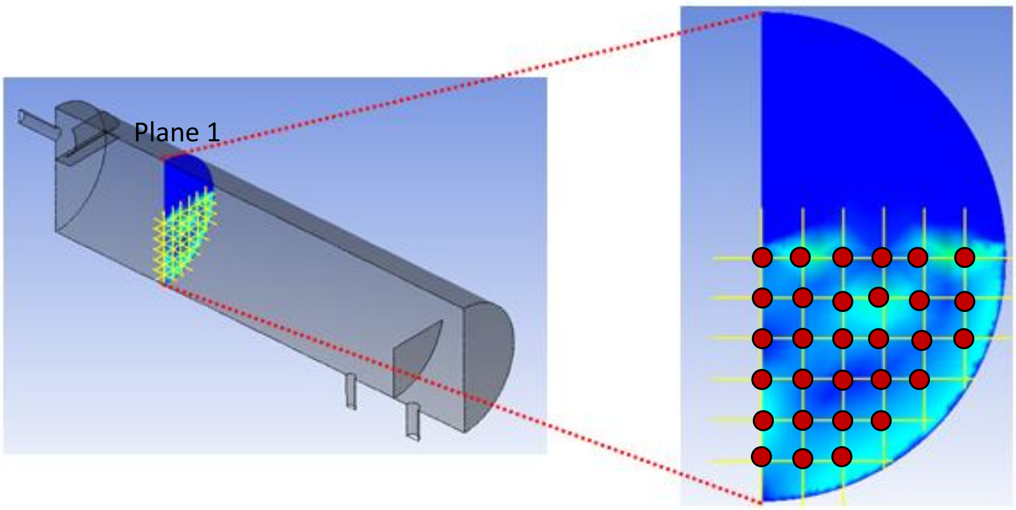

Figure 14: Position of points measured in Plane 1

The default value of turbulent Schmidt number which is 1 is set for the first simulation. The result from the first simulation show that the value of the water standard deviation is far away from the experimental. Therefore, the decisions to increase the value of turbulent Schmidt number were made. As the value increase, the water standard deviation is closer to the experimental value. The simulation was run with increasing number of Schmidt number until the result close to the experimental. **Table 7** shows the result of the simulation

Table 7: Result for model validation

Turbulent Schmidt Number	Water Standard Deviation (m/s)
1	0.05778929
2	0.04486700
3	0.04121540
4	0.04579200
5	0.04358500
6	0.04177343
7	0.04057105
8	0.03924700
9	0.03822800
10	0.03700000
11	0.03604289
12	0.03565100
13	0.03569213
14	0.03400000
15	0.03380697
16	0.03360800
17	0.03244700
18	0.03214879
19	0.03165322
20	0.03136900
21	0.03095800
22	0.03059172
23	0.03029217
24	0.03982362
25	0.02997800
26	0.02979821
27	0.02926373
28	0.02956783
29	0.02856253
30	0.02861193

31	0.02861200
32	0.02820000
33	0.02780520
34	0.02780540
35	0.02648897

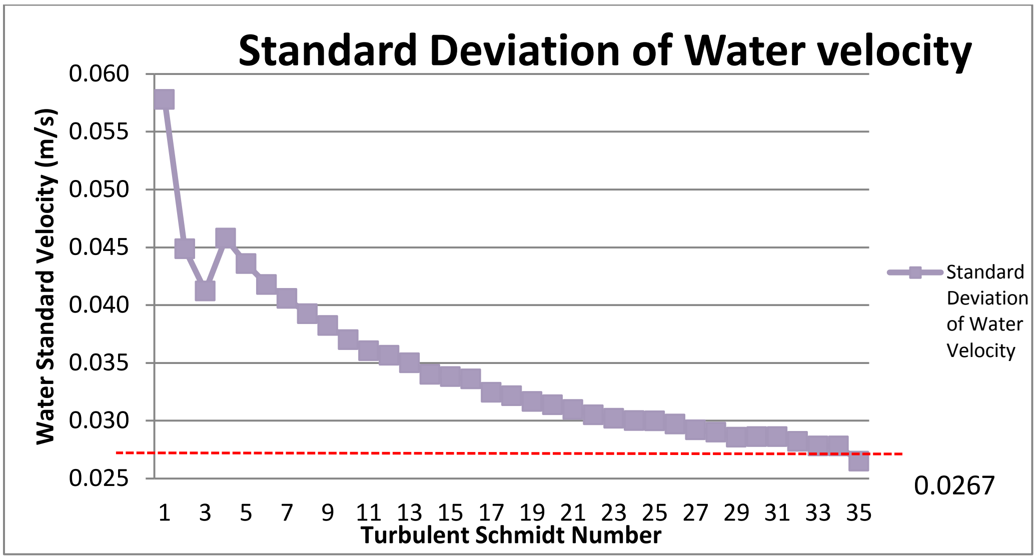

Figure 15: Standard deviation of water velocity versus the turbulent Schmidt number

From the result of a series simulation with a different turbulent Schmidt number, the best result that obtained is **0.02648897 m/s** for water standard deviation which is the closes to the experimental result **0.0267 m/s**. The error calculated is only **0.79%.** As a conclusion, the appropriate turbulent Schmidt number is 35.

4.2.3 Phase 1: Inlet Design Improvement

After completing the sensitivity study on meshing validation and model validation in CFD, the Phase 1 is initiated. Phase 1 generally about the improvement of the inlet to obtain the higher separation efficiency of the separator. **Table 8** shows the result from the simulation on inlet design.

Table 8: Inlet design separation efficiency

Inlet Design Types	Oil Separation efficiency, %	Water Separation efficiency, %	Average Separation Efficiency, %
1	56.59443921	99.8935663	78.24
2	72.59235784	96.1116643	84.35
3	84.79551359	90.1188764	87.46
4	99.20952651	99.910012	99.56

The separator efficiency of oil and gas are calculated by using the following formula:

$$\text{Separation Efficiency,\%} = \frac{\text{The mass flow rate of fluid separated out}}{\text{The total of mass flow rate of fluid in the stream}} \times 100$$

From the individual separation efficiency, the average separation efficiency was calculated for all the design proposed.

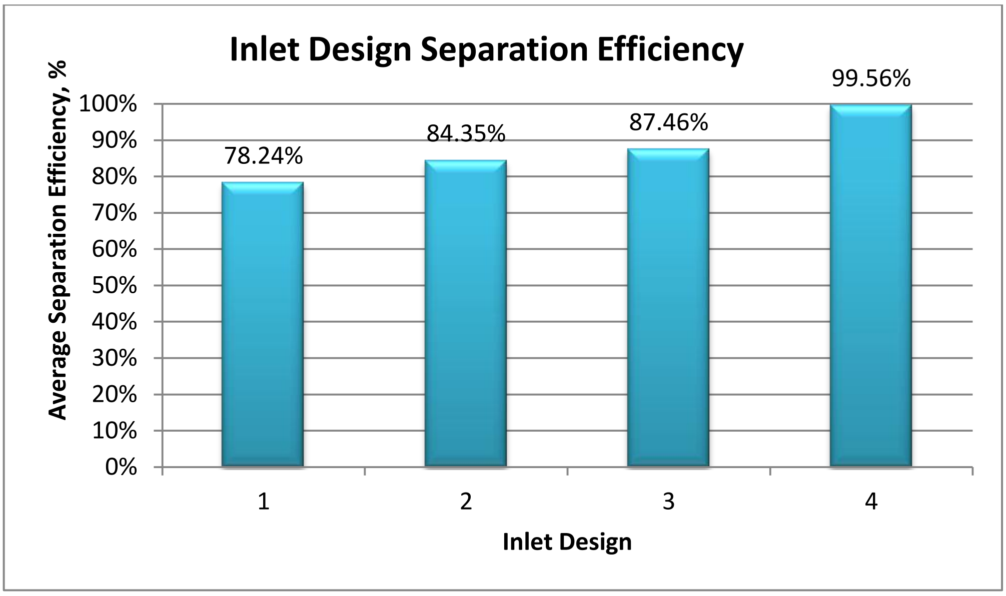

Figure 16: Inlet design separation efficiency

Phase 1 begins with simulation of Inlet Design 1 which is pipe-looked structure as shown in **Figure 17**. Refer **Appendix 2** for completed geometry, mesh generated and simulation of inlet design 1.

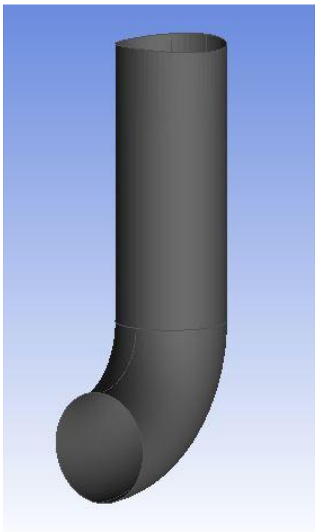

Figure 17: Inlet Design 1

From the CFD analysis, Inlet Design 1 gives the average separation efficiency of **78.24%**. Then, Inlet Design 2 was proposed to increase the separation efficiency of the separator. Inlet Design 2 was designed by increasing the area at the end point of the inlet. **Figure 18** shows the Inlet Design 2.

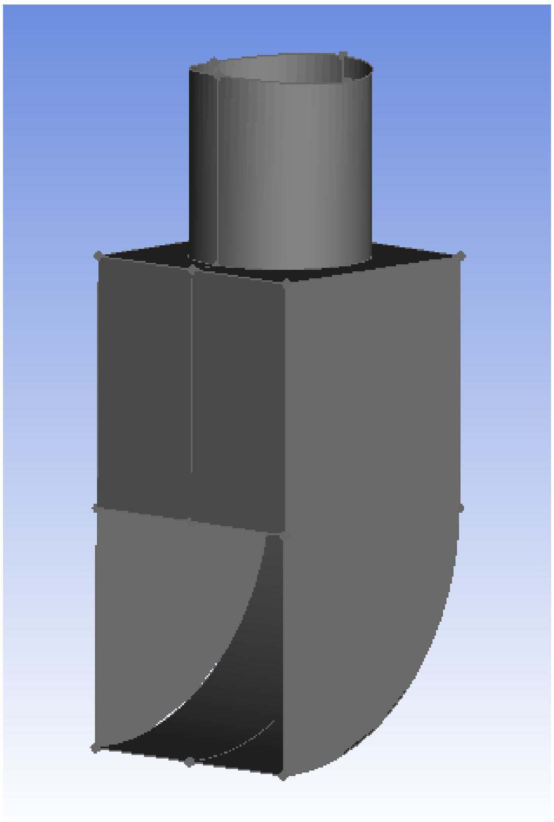

Figure 18: Inlet Design 2

From the simulation, Inlet Design 2 shows increasing in separation efficiency by **6.11%.** The increasing in area of inlet gives more surface area in contact with the liquid flow into the separator. As the surface area in contact increase, the flow's energy decrease as it distribute equally to the larger surface area. This may lead to decreasing in velocity. From the literature review part, the decreasing in velocity may cause the separation of two or more phase fluid with different density. Then, with the same concept apply to design the Inlet Design 2, Inlet Design 3 was proposed by increases further the surface area. The geometry for the Inlet Design 3 is shown in **Figure** 19**.**

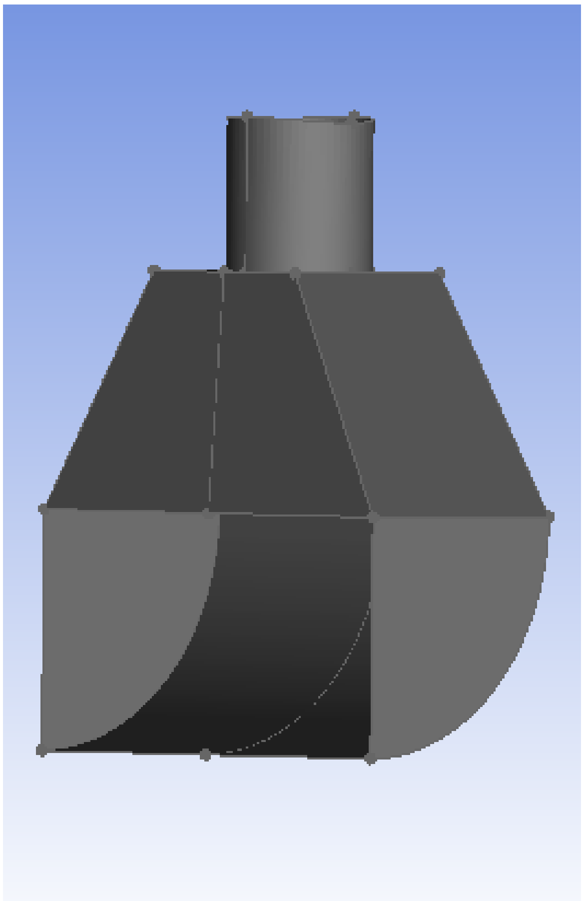

Figure 19: Inlet Design 3

From Simulation, Inlet Design 3 shows separation efficiency of **87.46%.** After completed analyzing the result for Inlet Design 3, Inlet Design 4 was proposed. Inlet Design 4 is the improvised design of Inlet Design 3 by adding a flow restriction at the end point of the inlet as shown in **Figure 20**. The restriction consist a plate with holes for the fluid to flow. This plate was used to create a change in momentum of the fluid lead to decrease more the velocity of the fluid. The simulation shows the separation efficiency up to **99.56%.**

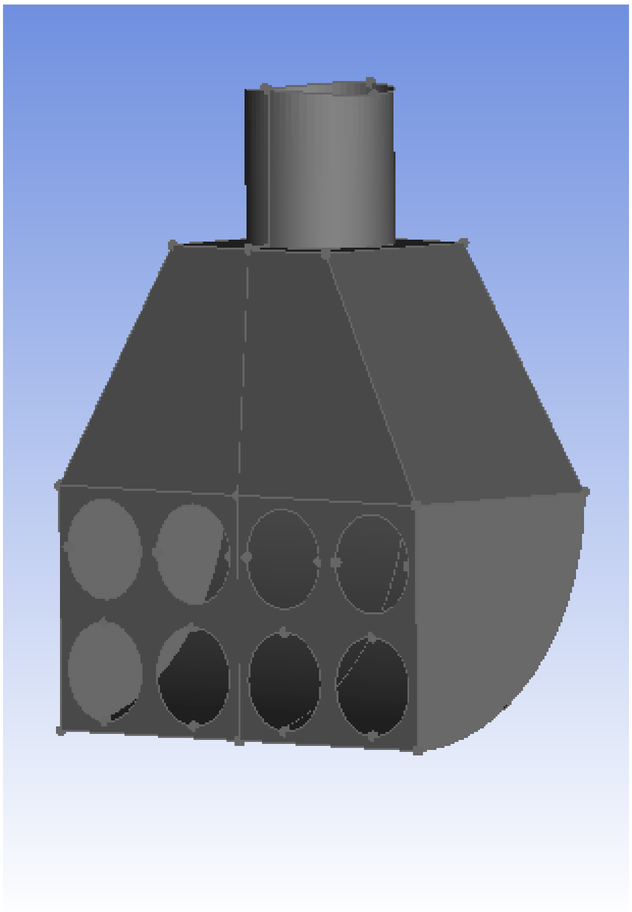

Figure 20: Inlet Design 4

Figure 21 shows the visual of the fluid velocity at the inlet region. Clearly it shows that the velocity of fluid decrease as the increasing the area at the end of the inlet. At point A for Inlet Design 1 the average fluid velocity is almost 2.5 m/s, while at point A for Inlet Design 2 the average fluid velocity is about 1.8 m/s. This is due to the bigger area of Inlet Design 2 compared to Inlet Design 1. The Inlet Design 3 on the other hand, has the lowest fluid velocity at point A around 1 m/s compared to the previous inlet design. Inlet design 3 has more surface area in contact to the fluid which decreasing the fluid velocity. For inlet design 4, the velocity is less at point A compared to the others with the value around 0.6 m/s. The installation of plate with a hole proved it capability to decrease further the fluid velocity at the end of inlet.

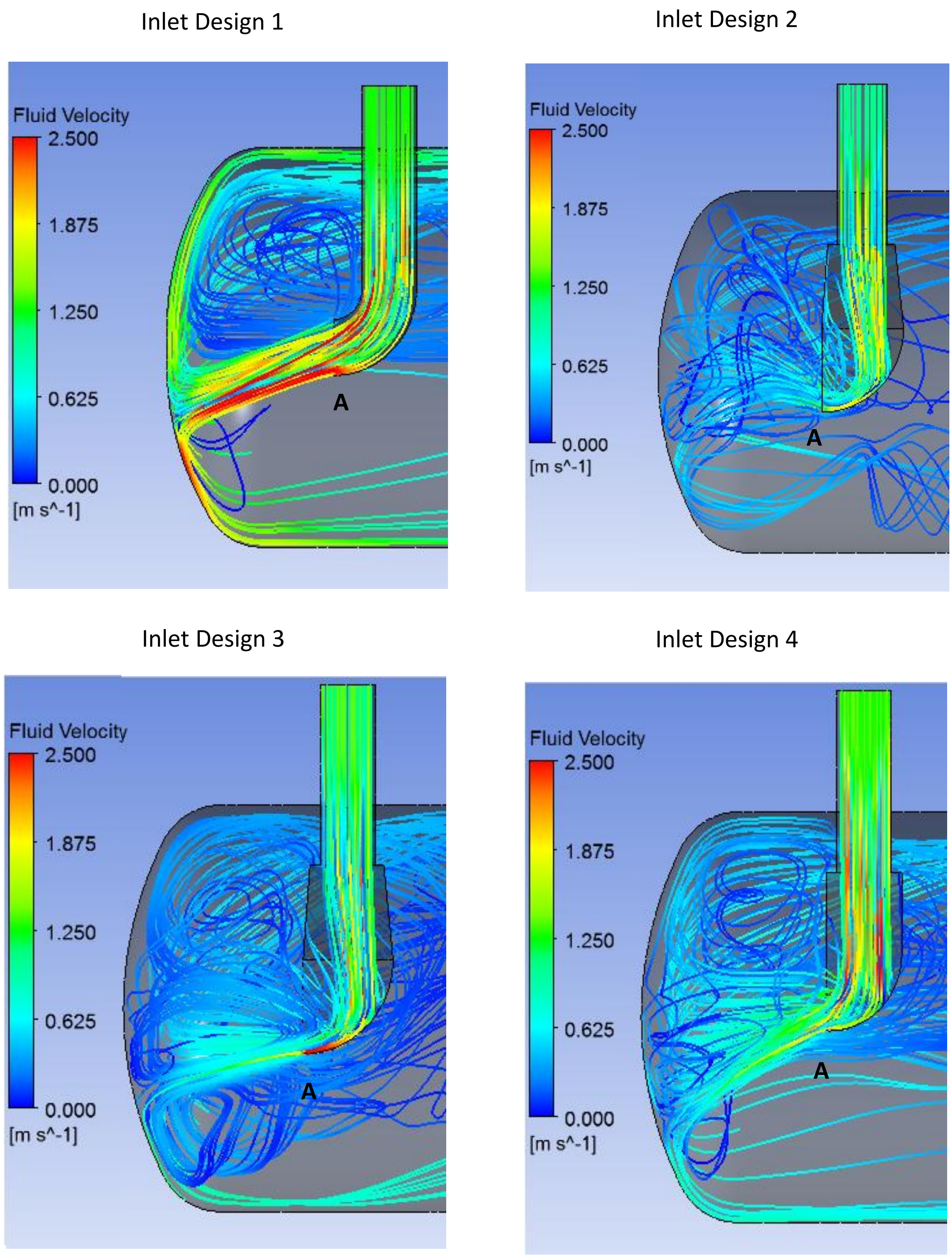

Figure 21: Fluid Velocity at inlet

As a conclusion, inlet design 4 is the best inlet design compared to the others as it creates the highest separation efficiency, 99.56%. The fluid flow in contact with a large surface area at the end of inlet of inlet design 4 causes the decreasing in the fluid energy. As the energy is decrease, the fluid velocity with decrease as well. The plate was installed at the end of the inlet end to restrict the fluid flow. This restriction may lead reduction on the fluid velocity. The reduction of velocity leads to the better separation efficiency.

Refer **Appendix 2** to see the completed geometry development and generated mesh of all the inlet designs.

4.2.3 Phase 2: Baffle Design Improvement

The baffle improvement phase was initiated as soon as the inlet design improvement phase completed. There are two baffle were analyze based on their contribution on separation efficiency and pressure drop across the baffle. **Table 8** shows the result for baffle design improvement analysis.

Table 9: Baffle design separation efficiency

Inlet Design Types	Oil Separation efficiency, %	Water Separation efficiency, %	Average Separation Efficiency, %
1	73.56057	99.96004	86.76
2	77.82032	99.32267	88.57

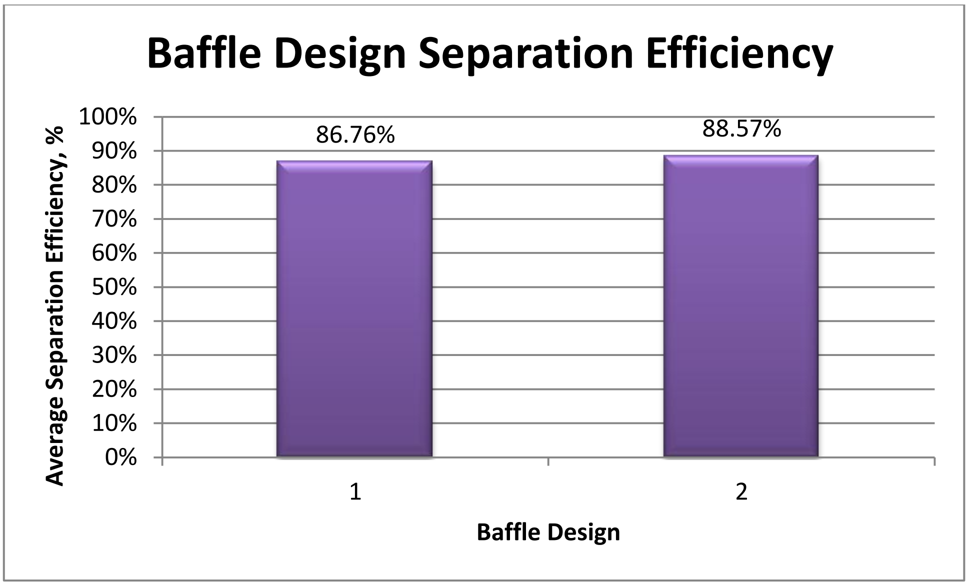

Figure 22: Baffle design separation efficiency

From **Figure 22,** it shows that the baffle design 2 contributed a higher separation efficiency compare to baffle design 1. Baffle design 1 is the common baffle used in the oil and gas separator. The average separation efficiency is 86.76%. The geometry of baffle design 1 is shown in **Figure 23.**

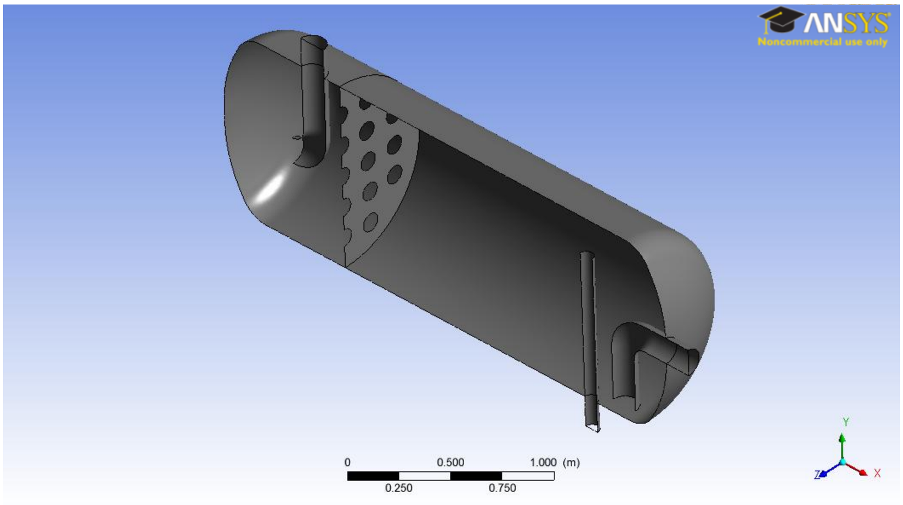

Figure 23: Baffle Design 1

Baffle Design 2 was proposed as a curvy baffle as shown in **Figure** 24. The reasons behind is to make the flow more stable. When stability of the fluid achieved, the flow will have a low velocity. The low fluid velocity is a good sign for the good separations process occurred. Besides, the advantage using curvy baffle is that they create less disturbance compared to flat plate and cutting down the re-entrainment problem. (Steward & Arnold, 2011). The CFD analyses of baffle design 2 shows that the separation efficiency increases by **1.81 %.**

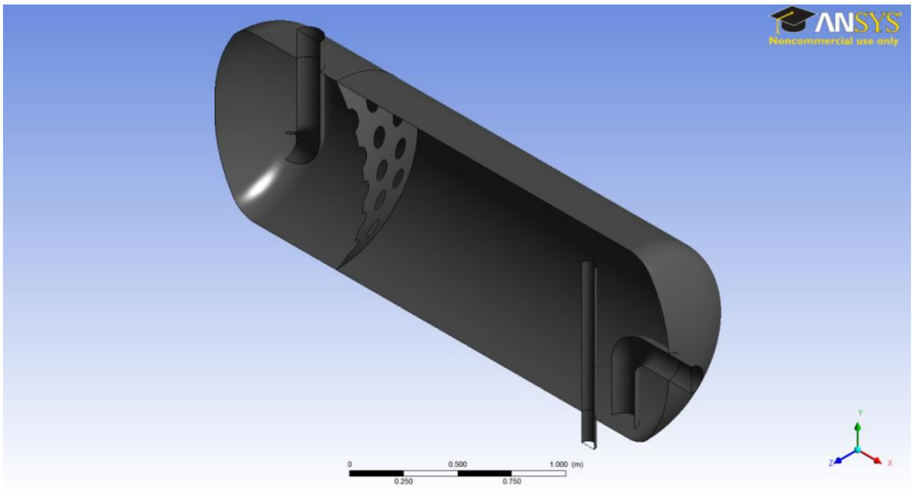

Figure 24: Baffle Design 2

As a conclusion, the curvy baffle (Baffle Design 2) is more effective in stabilize the flow to reduce the fluid velocity compared to common vertical baffle (Baffle Design 1).

Chapter 5: Conclusion

The mesh validation study is very important in order to carry out the CFD analysis on the optimization of separator internal. This is because the high number of mesh will give more accurate result compare to the low number of mesh. However, high number of mesh will take a longer time for the simulation. Therefore, the mesh validation study is conducted to find the minimum number of mesh that can give the most accurate results. This will save the time for the simulation. From this research the minimum size of mesh is 0.02m that will give the most accurate result. This is called the critical size of mesh.

The model validation is also very important in CFD analysis. In CFD modeling there are many fluid models and the chosen of the right one for particular purpose of simulation. In the research, **the multiphase model (Free surface model)** and **the turbulent model (k-epsilon)** were chosen. In turbulent model, the value of **turbulent Schmidt number is 35** was specified. By using this value, the result of simulation is closer to the experimental result with the error of **0.79%.**

For phase 1, inlet design 4 shows the highest separation efficiency which is **99.56%.** This is because the structure of the inlet design decreases the velocity of fluid which makes the separation process easier to happen. The large surface area in contact with the fluid at the end of inlet design and the present of porous plate give the advantages to inlet design 4 to reduce the velocity as much as possible.

While in phase 2, baffle design 2 with a curvy structure shows the improvement in term of separation efficiency. The improvement by 1.81% efficiency from the baffle design 1. This is because the curvy baffle (Baffle Design 2) is more effective in stabilize the flow to reduce the fluid velocity compared to common vertical baffle (Baffle Design 1).

References

Bakker, A. (2012, March 4). *Applied Computational Fluid Dynamics.* Retrieved March 21, 2013, from The Colourful fluid Mixing Gallery: http://www.bakker.org

Chugh, A. (2011). *Flow Separators Types, Working and Function*. Retrieved February 24, 2013, from PIPING GUIDE: http://www.pipingguide.net/2009/04/flow-separator.html

Khuzaimah. (2009, December 29). Retrieved April 5, 2013, from PETRONAS AXIS Enterprise Search: http://pww.axis.petronas.com.my/cop/cop_PETRONAS%20Sabah/SBOKM/Surface/SKG%2016.1%20Oil%20and%20Water%20Handling%20L2/Microsoft%20PowerPoint%20-%20spf1-pmC4-Separator%20and%20separation.pdf

Lee, J. M., Khan, R. I., & Phelps, D. W. (2008). Debottlenecking and CFD Studies of High and Low Pressure Production Separators. *SPE 115735. The paper was presented in Denver, Colorado, USA*, 11.

Lu, Y., Lee, J. M., Phelps, D., & Chase, R. (2007). Effect of Internal Baffles on Volumetric Utilization of an FWKO-CFD Evaluation. *SPE 109944. Presented at the 2007 SPE Annual Technical Conference and Exhibition held in ANaheim, California, U.S.A*, 6.

Mee, C. G., & Mohamad Nor, M. I. (2011). *Flow Pattern In a Horizontal Primary Separator With a Perforated Baffle.* Melaka: Multimedia University.

PETRONAS. (2009). *Surface Production oil/water Handling : Separation Process and Separator.* Retrieved February 23, 2013, from PETRONAS AXIS Enterprise Search: http://pww.axis.petronas.com.my/cop/cop_PETRONAS%20Sabah/SBOKM/Surface/SKG%2016.1%20Oil%20and%20Water%20Handling%20L2/Microsoft%20PowerPoint%20-%20spf1-pmC4-Separator%20and%20separation.pdf

Smith, H. (1987). Oil and Gas Separators (1987 PEH Chapter 12). In H. V. Smith, *Petroleum Engineering Handbook* (p. 44). Texas: Society of Petroleum Engineers.

Spirax Sarco. (2013). *Separator*. Retrieved February 24, 2013, from Spirax Sarco: http://www.spiraxsarco.com/resources/steam-engineering-tutorials/pipeline-ancillaries/separators.asp

Vilagines, R. D., & Akhras, A. R. (2010). Three-Phase Flows Simulation for Improving Design of Gravity Separation Vessels. *SPE 134090. Paper is presented at SPE Annual Technical Conference and Exhibition held in Florence, Italy*, 11.

Wilkinson, D., Wildie, B., Mohamad Nor, M., & Lee, H. Y. (1999). Baffle plate configurations to enhance separation in. *Chemical Engineering Journal*, 6.

Appendices

Appendix 1

The generated meshing for mesh size of 0.05 m.

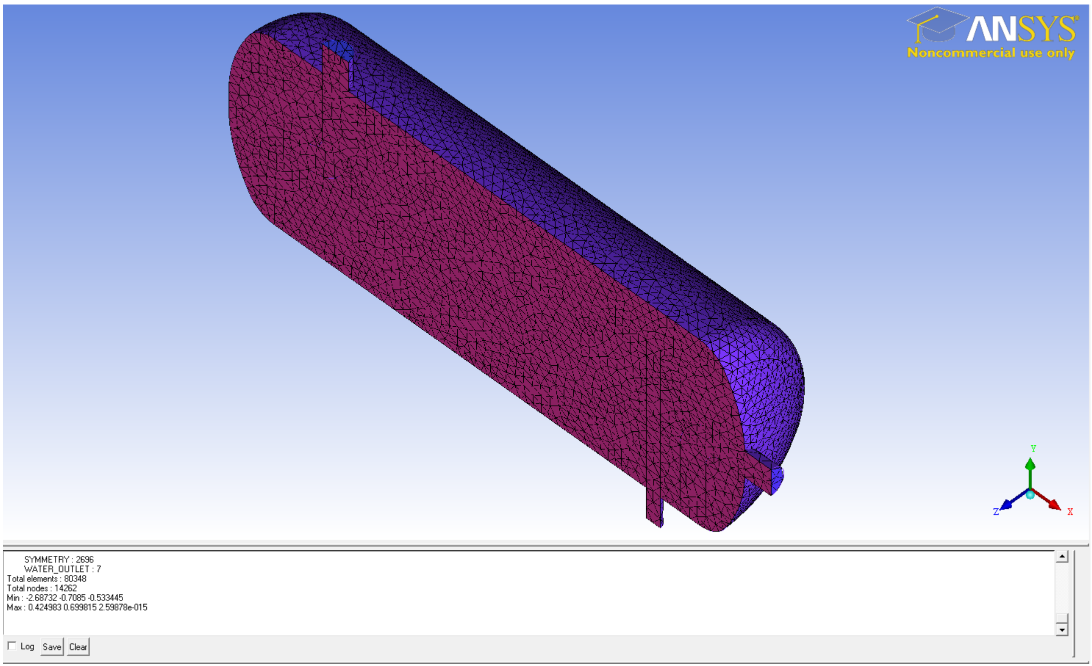

Figure 25: Mesh size of 0.05 m

The generated meshing for mesh size of 0.04 m.

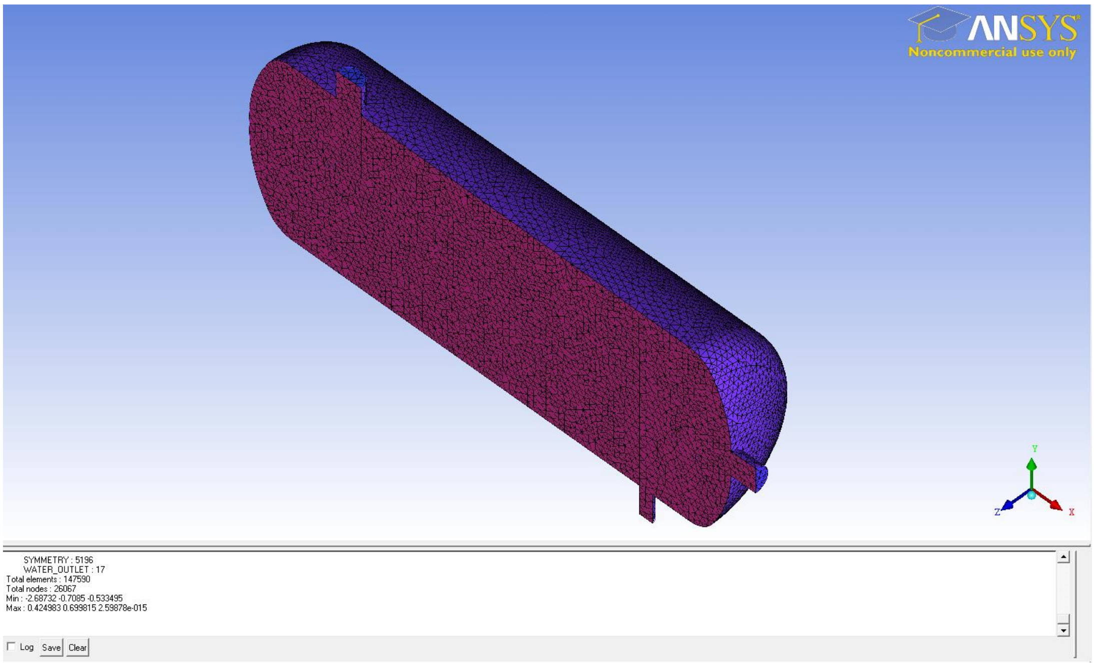

Figure 26: Mesh size of 0.04 m

The generated meshing for mesh size of 0.03 m.

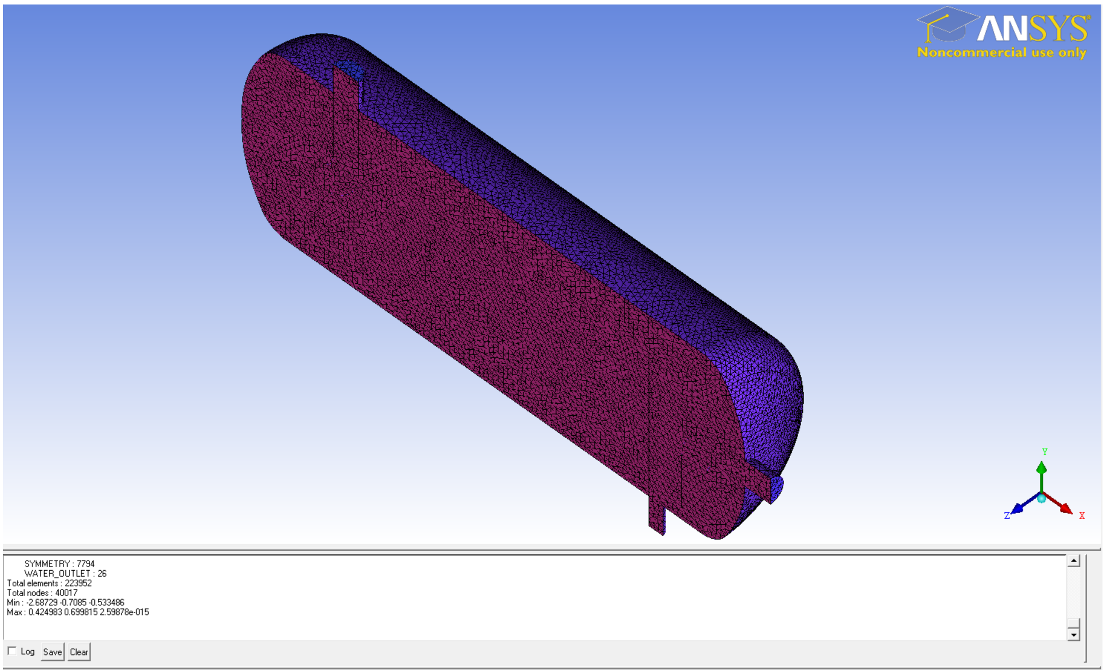

Figure 27: Mesh size of 0.03 m

The generated meshing for mesh size of 0.02 m.

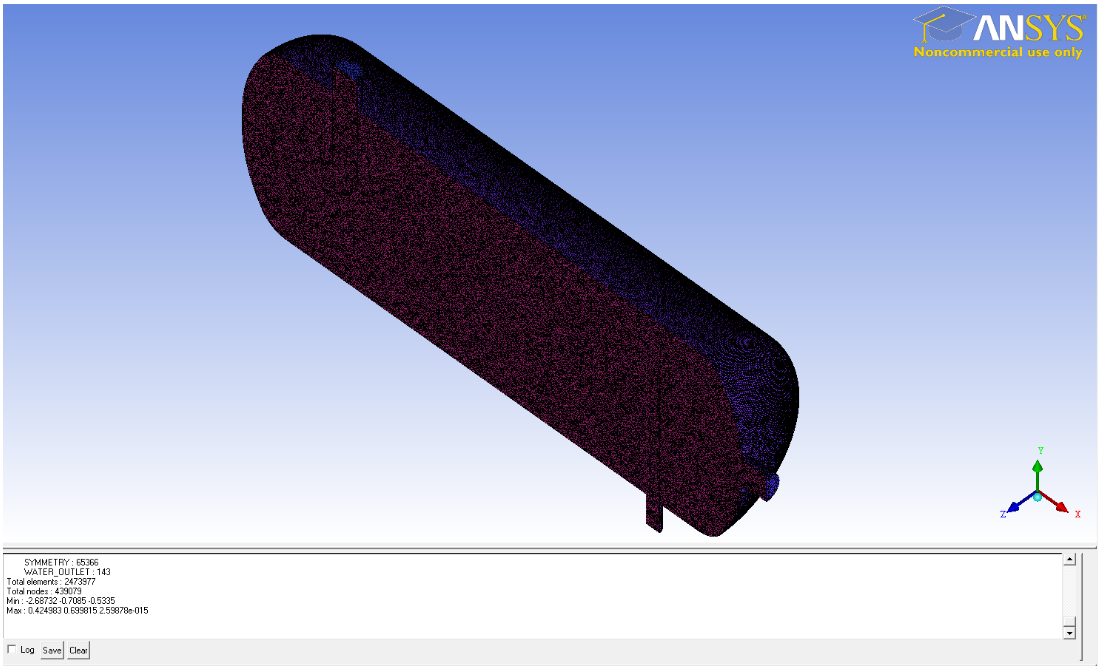

Figure 28: Mesh size of 0.02 m

The generated meshing for mesh size of 0.0175 m.

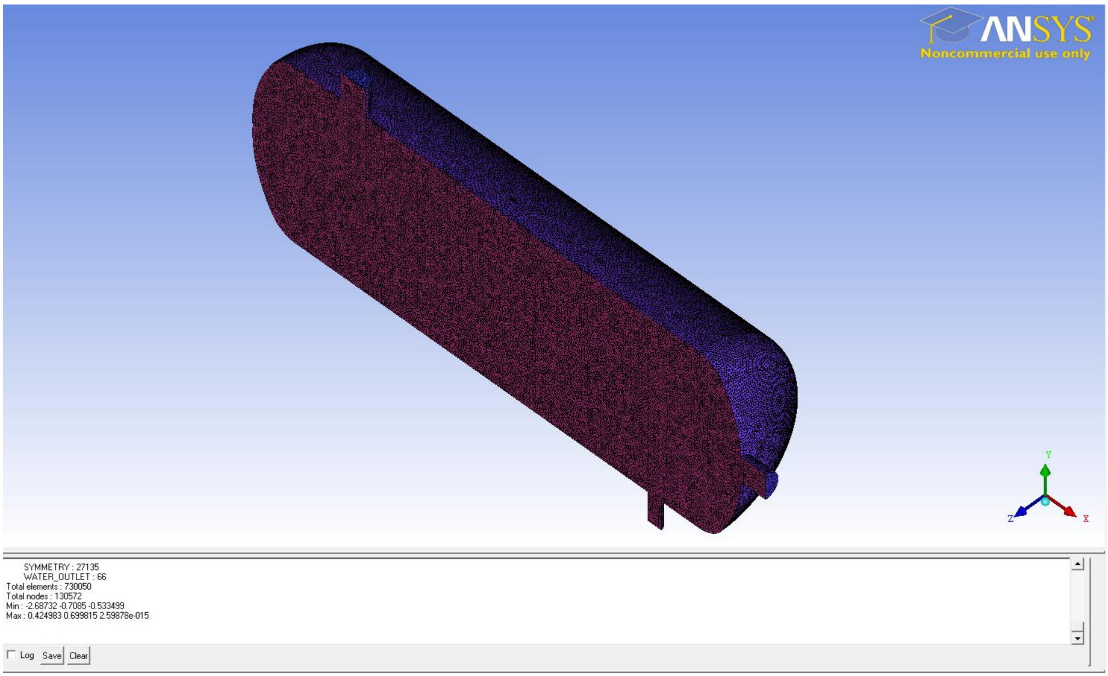

Figure 29: Mesh size of 0.0175 m

The generated meshing for mesh size of 0.015 m.

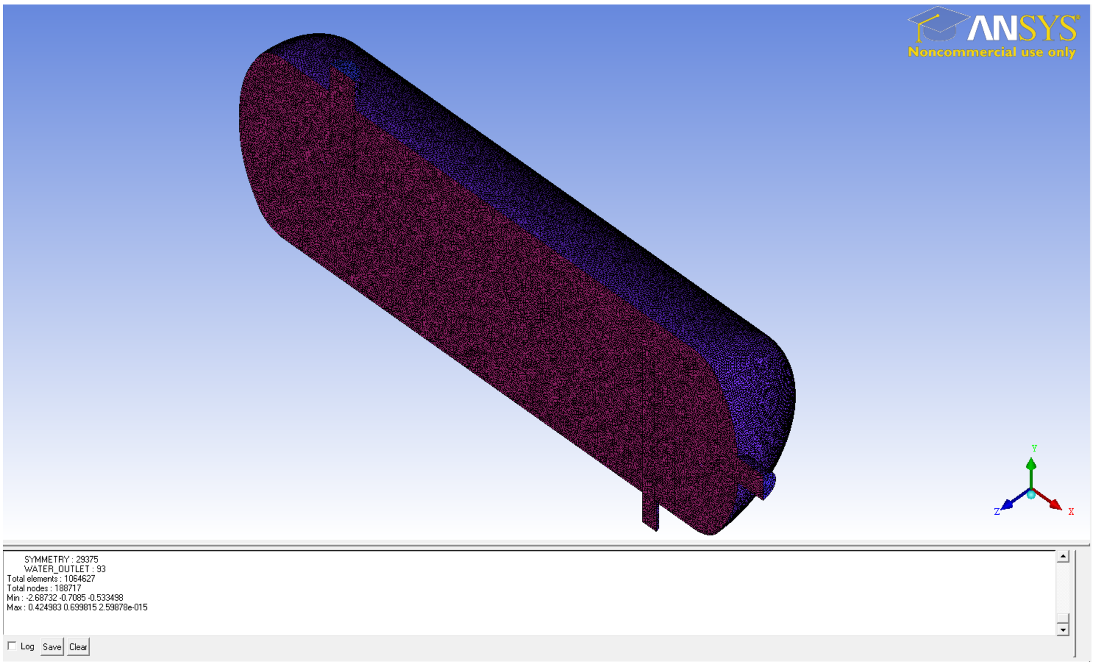

Figure 30: Mesh size of 0.015 m

The generated meshing for mesh size of 0.0125 m.

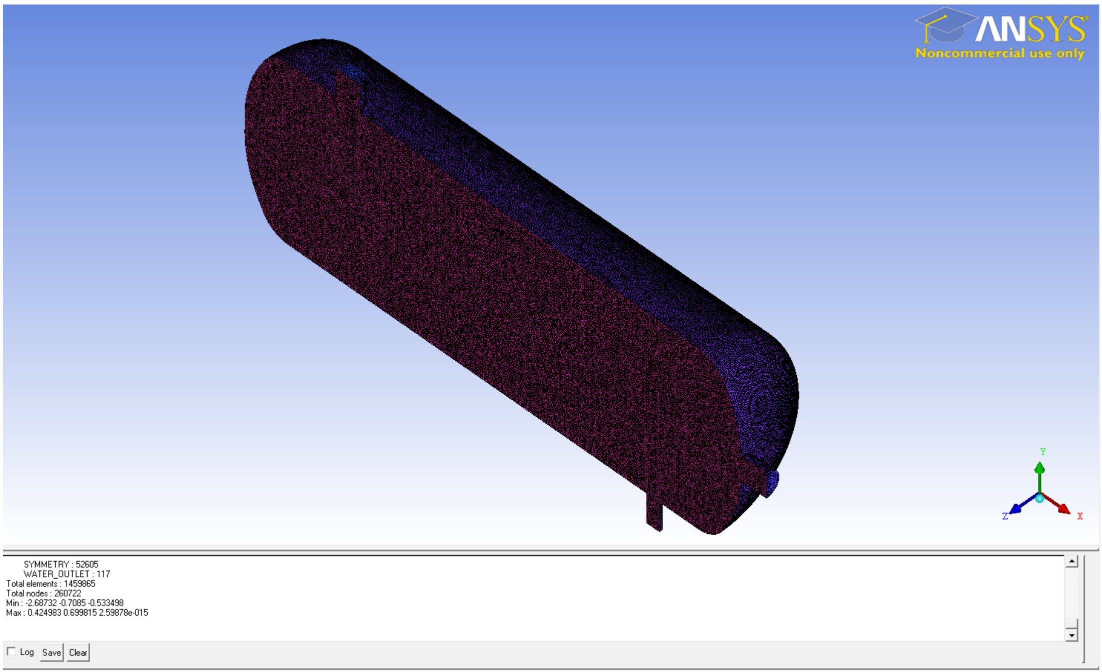

Figure 31: Mesh size of 0.0125 m

The generated meshing for mesh size of 0.01 m.

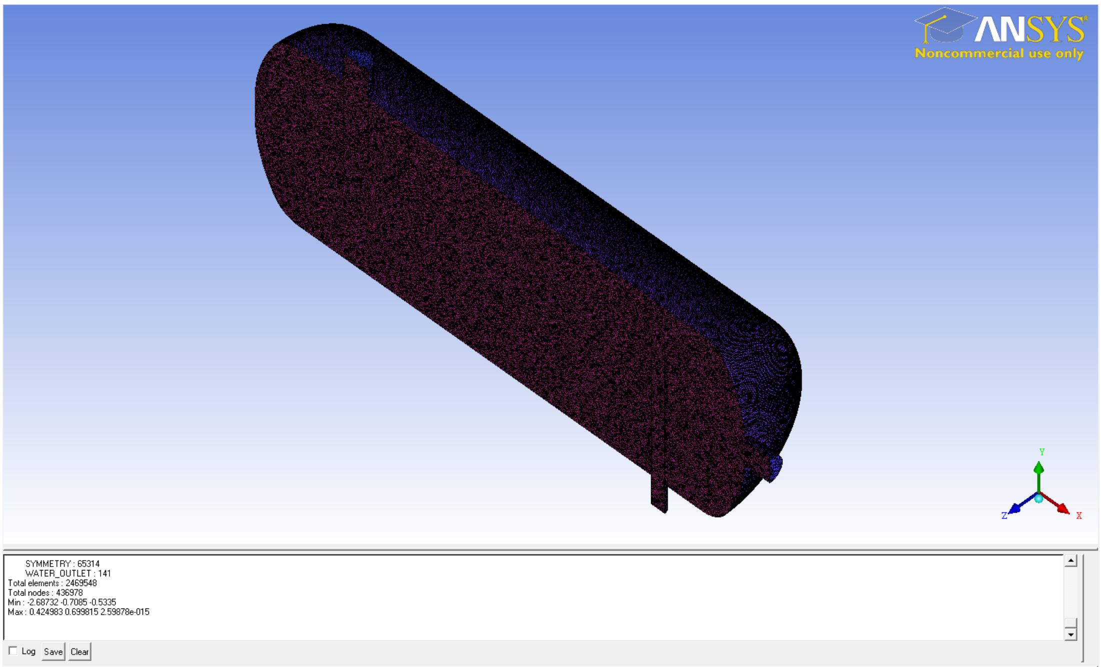

Figure 32: Mesh size of 0.01 m

Appendix 2

<u>Inlet Design 1</u>

Geometry

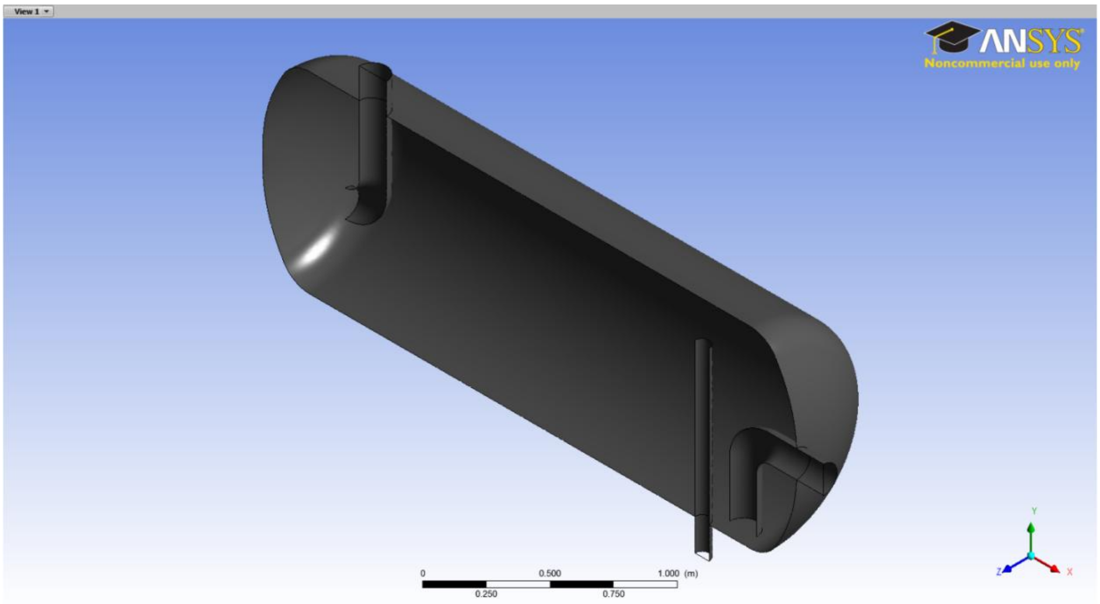

Figure 33: Geometry of Inlet Design 1

Mesh Generation

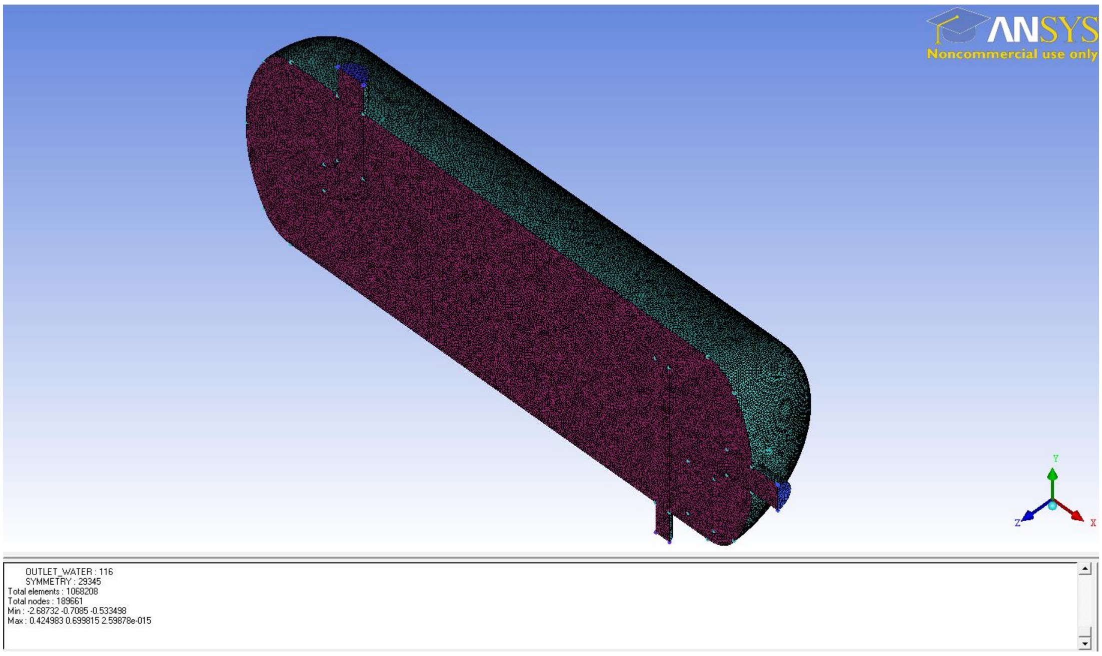

Figure 34: Mesh of Inlet Design 1

<u>Inlet Design 2</u>

Geometry

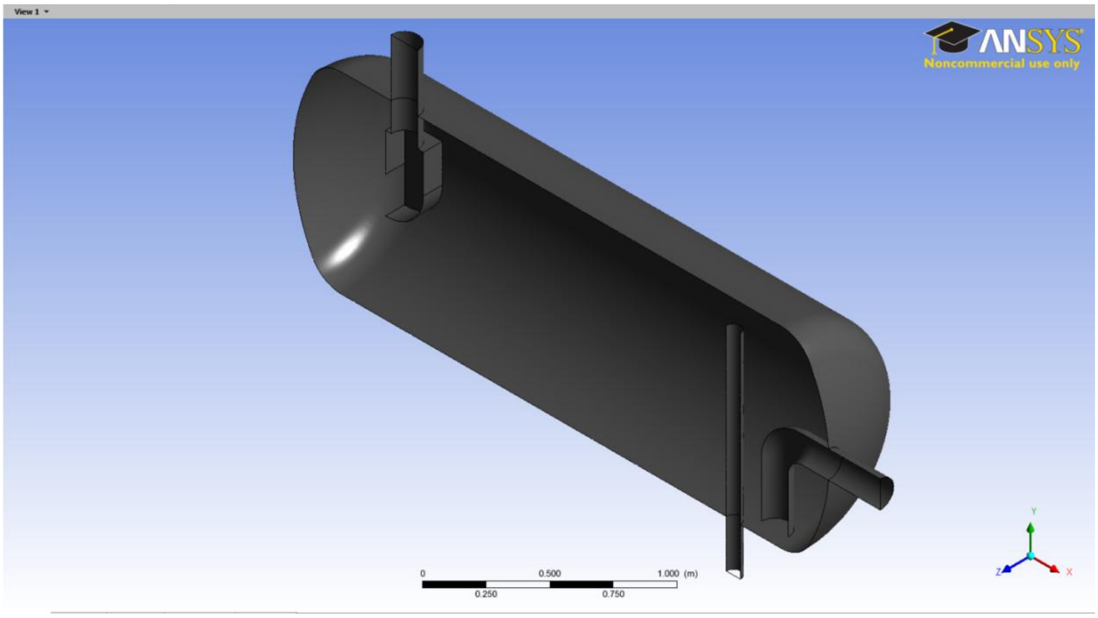

Figure 35: Geometry of Inlet Design

Mesh Generation

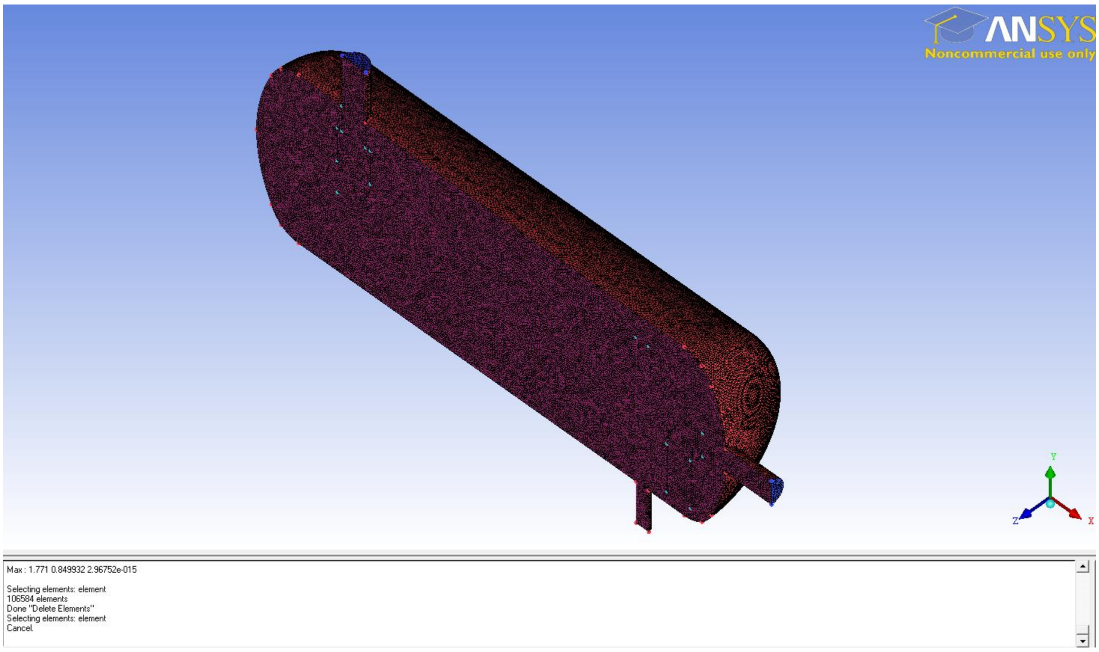

Figure 36: Mesh of Inlet Design 2

Inlet Design 3

Geometry

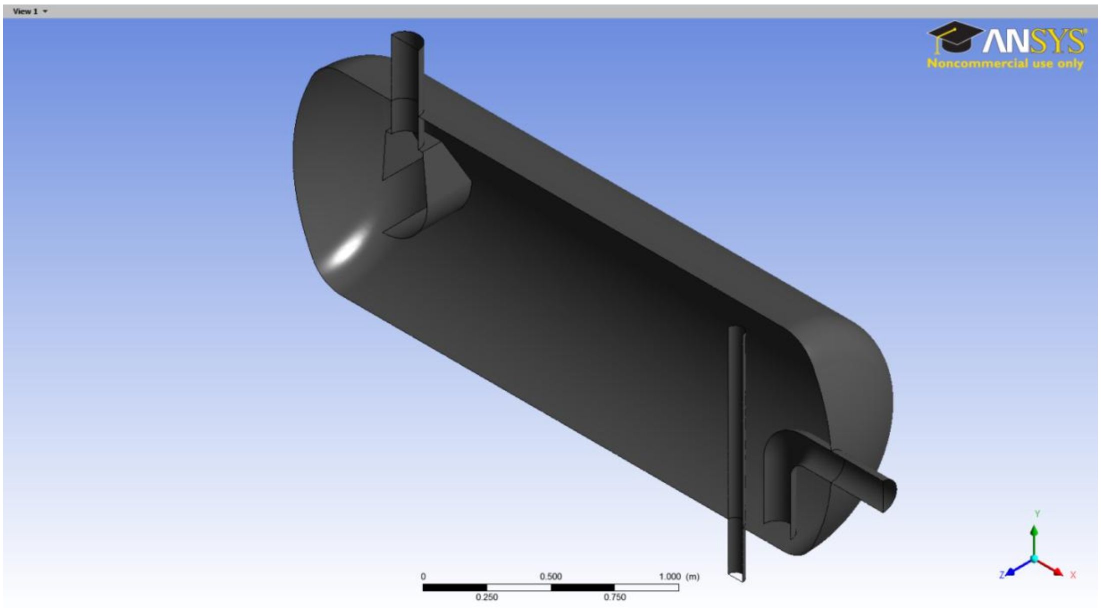

Figure 37: Geometry of Inlet Design 3

Mesh Generation

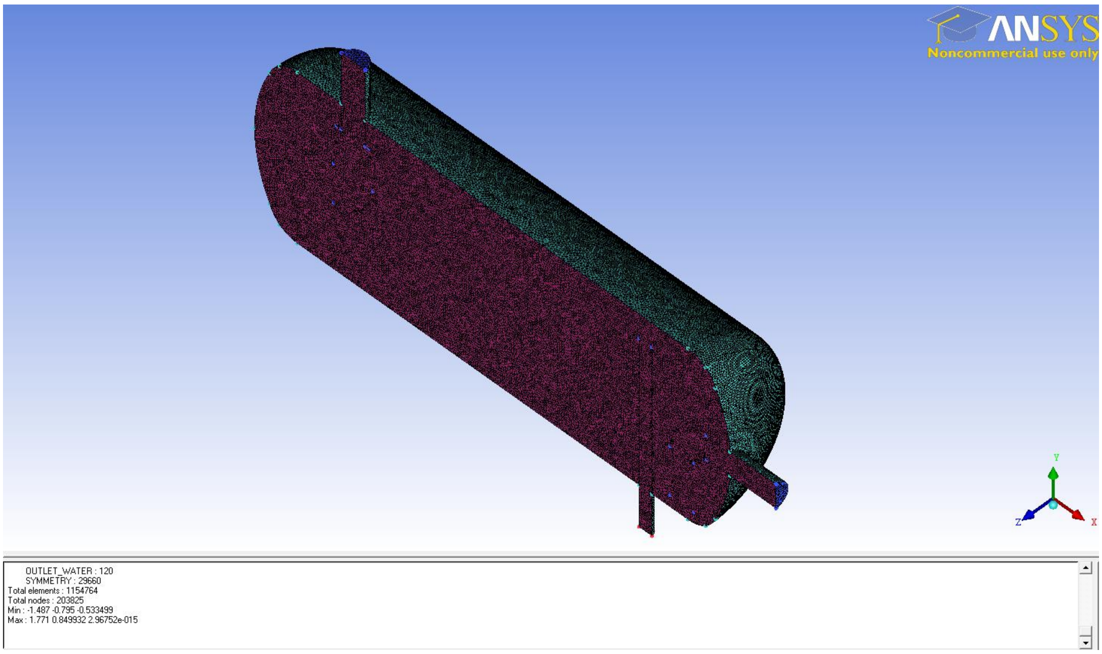

Figure 38: Mesh of Inlet Design 2

<u>Inlet Design 4</u>

Geometry

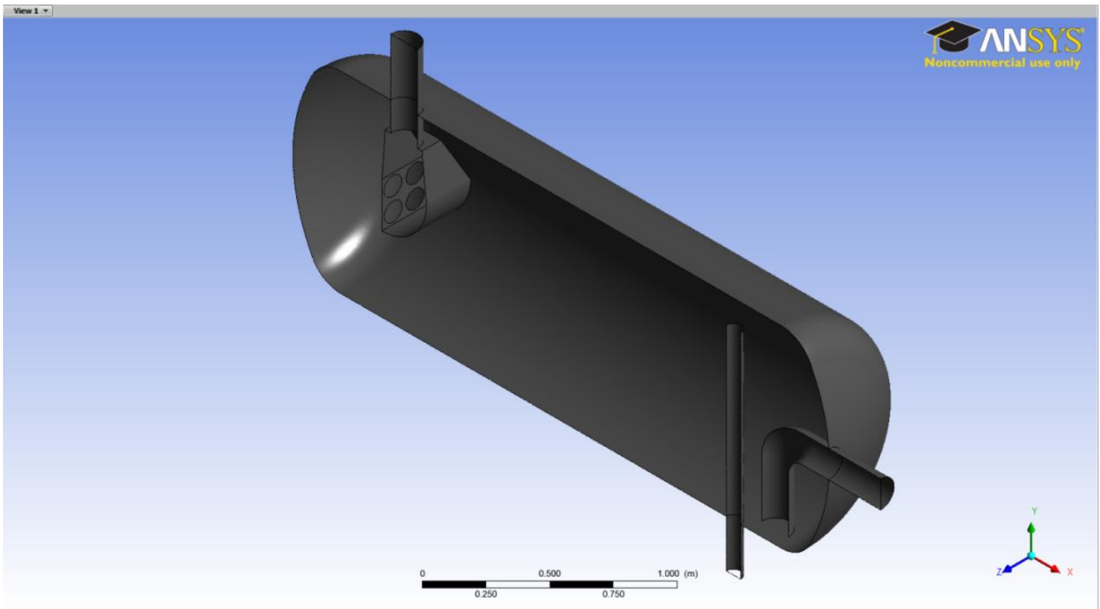

Figure 39: Geometry of Inlet Design 4

Mesh Generation

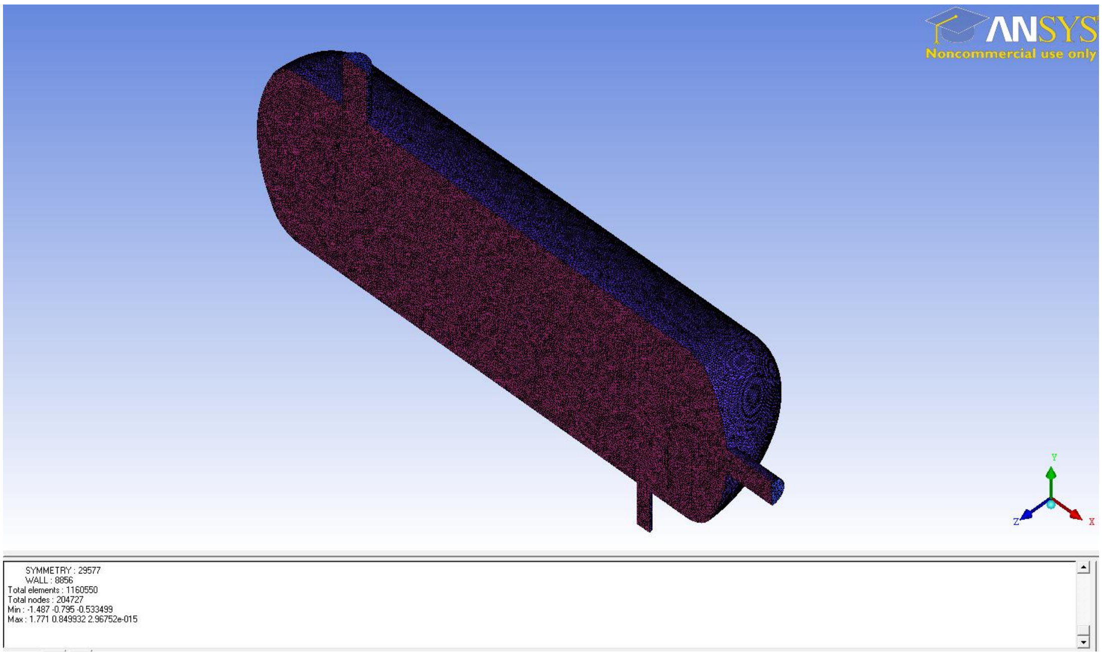

Figure 40: Mesh of Inlet Design 4

Baffle Design 1

Geometry

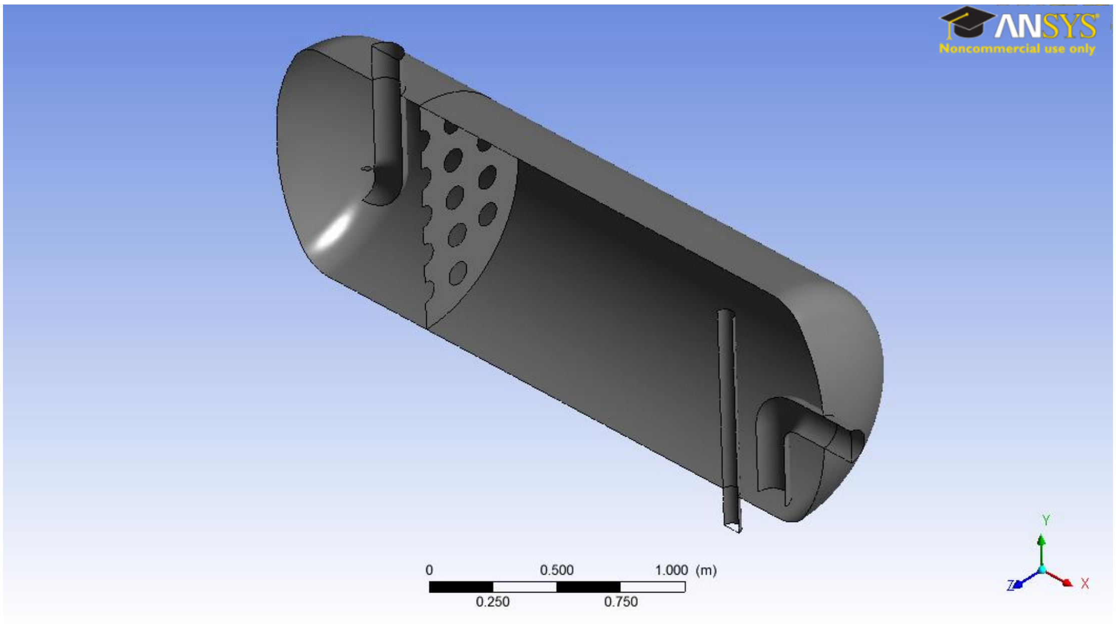

Figure 41: Geometry of Baffle Design 1

Mesh Generation

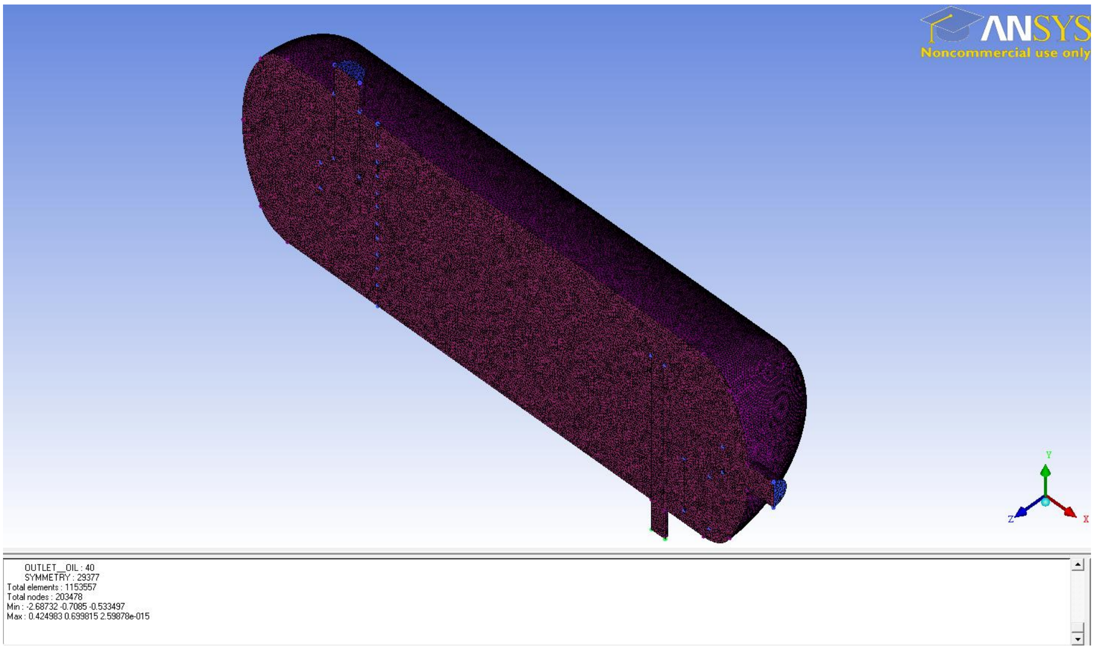

Figure 42: Mesh of Baffle Design 1

Baffle Design 2

Geometry

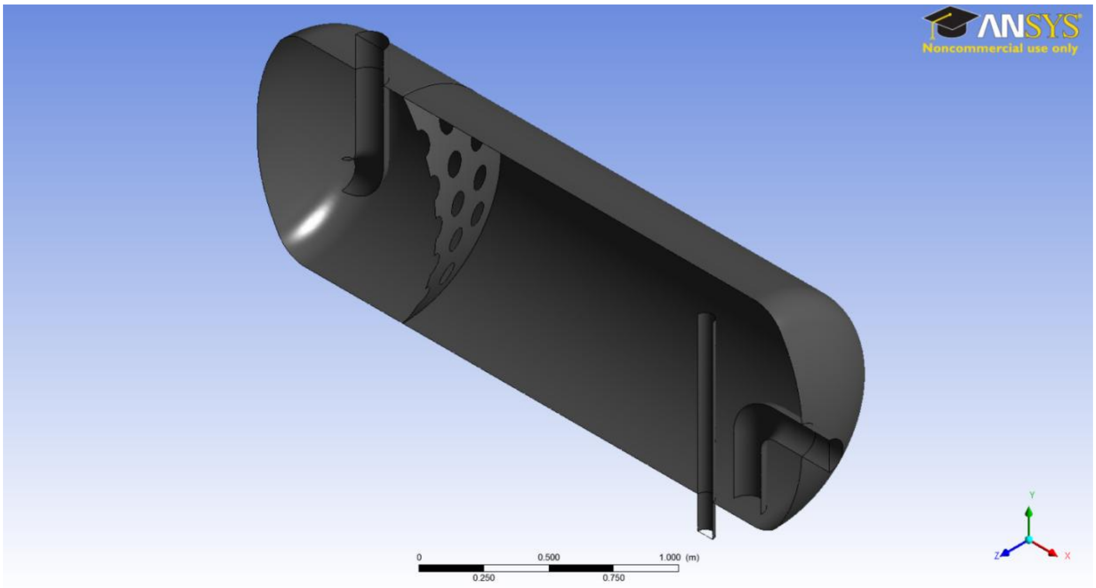

Figure 43: Geometry of Baffle Design 2

Mesh Generation

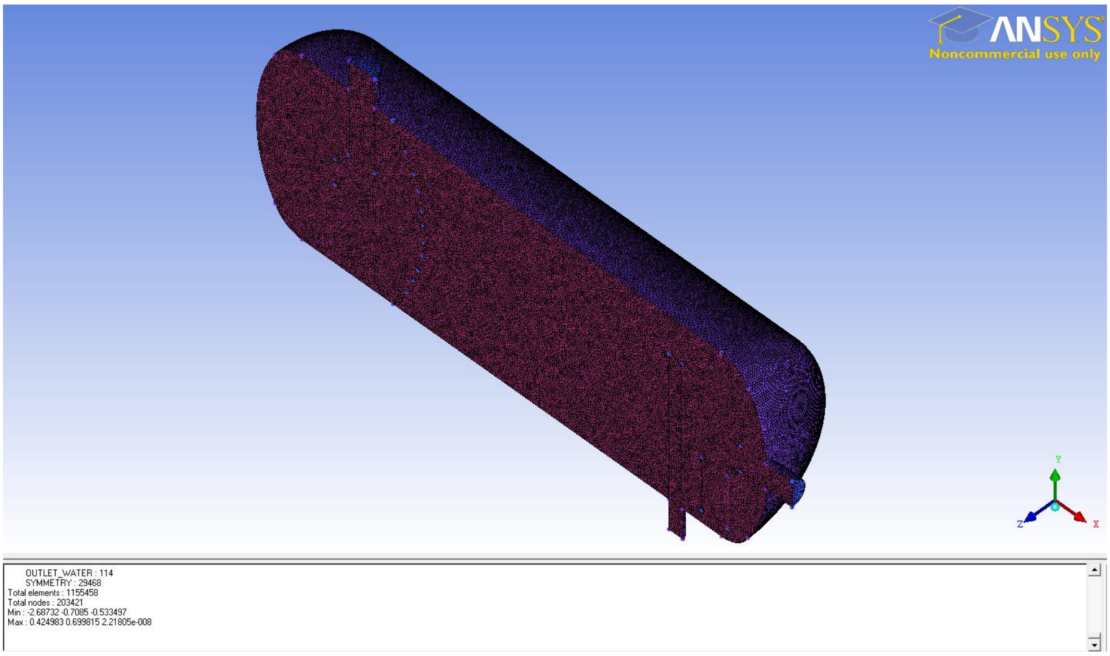

Figure 44: Mesh of Baffle Design 2

Appendix 3

<u>Meshing Modification</u> (Mesh size of 0.05m for the whole body mesh)

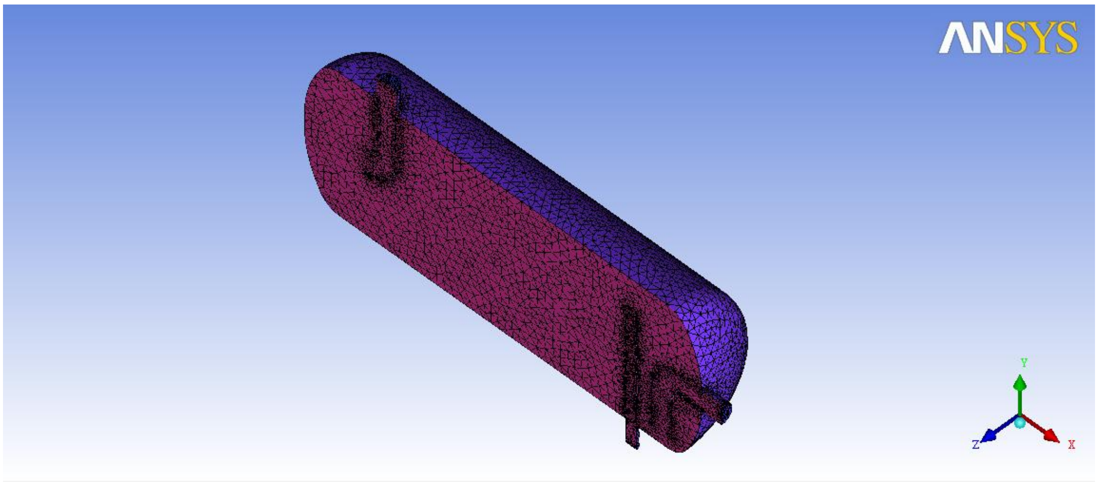

Figure 45: Isometric View

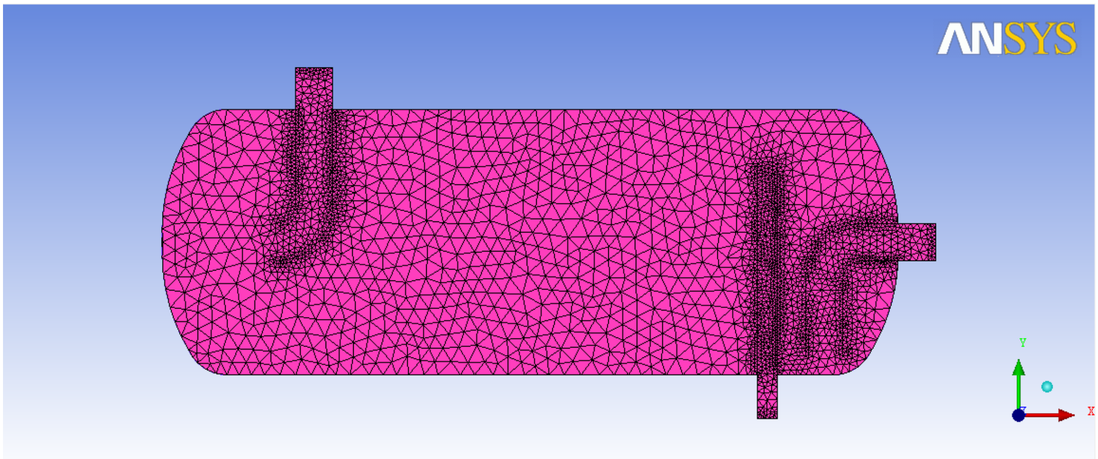

Figure 46: Front View

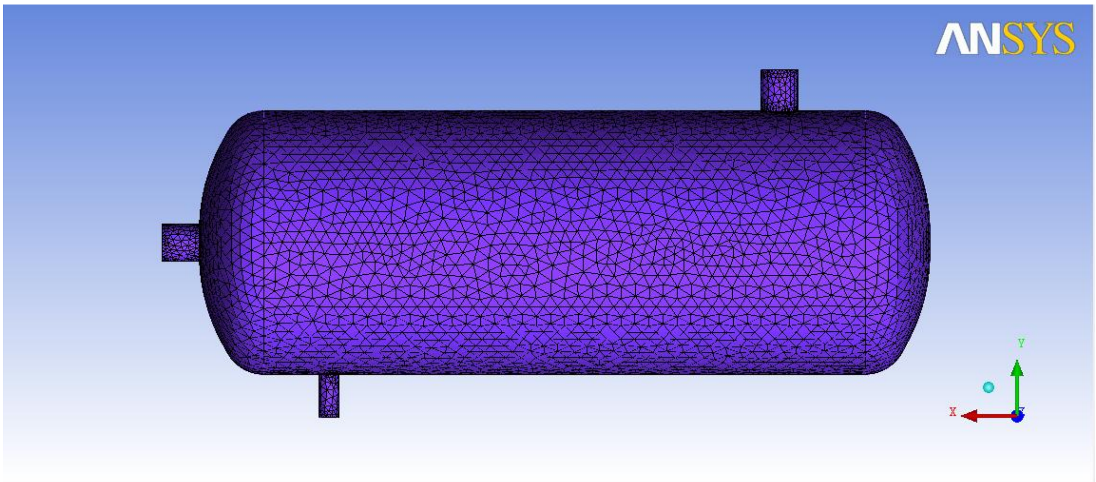

Figure 47: Back View

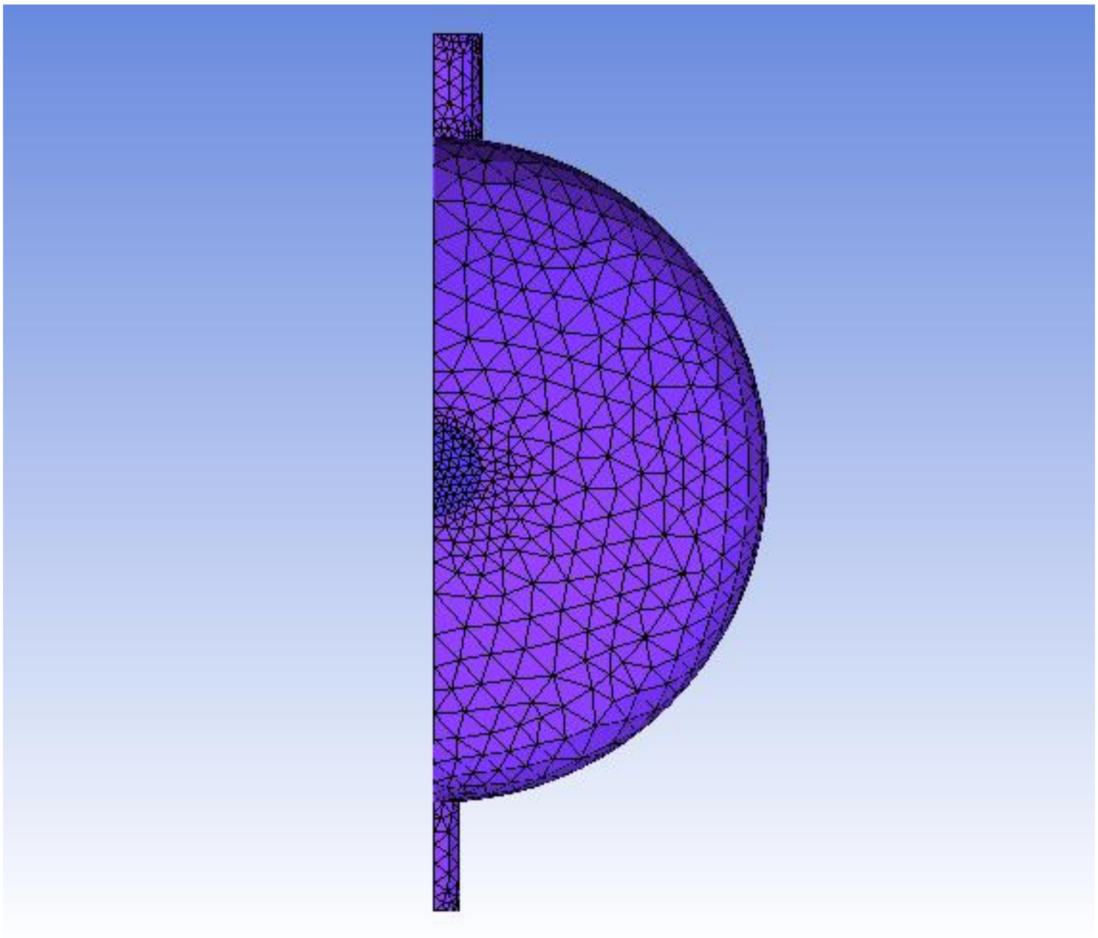

Figure 48: Side view