COMPUTATIONAL FLUID DYNAMICS SIMULATION OF INTERNAL FLUID FLOW: A CASE OF VENTURI EFFECT

KHO TUCK SING

FACULTY OF ENGINEERING
UNIVERSITY OF MALAYA
KUALA LUMPUR

2021

COMPUTATIONAL FLUID DYNAMICS SIMULATION OF INTERNAL FLUID FLOW: A CASE OF VENTURI EFFECT

KHO TUCK SING

**RESEARCH PROJECT SUBMITTED TO THE
FACULTY OF ENGINEERING, UNIVERSITY OF MALAYA,
IN PARTIAL FULFILMENT OF THE REQUIREMENTS
FOR THE DEGREE OF
MASTER OF MECHANICAL ENGINEERING**

2021

UNIVERSITY OF MALAYA

ORIGINAL LITERARY WORK DECLARATION

Name of Candidate: **Kho Tuck Sing**

Matric No: **17217788/1**

Name of Degree: **Master of Mechanical Engineering**

Title of ~~Project Paper~~ / Research Report / ~~Dissertation / Thesis~~ ("this Work"):

Computational Fluid Dynamics Simulation of Internal Fluid Flow: A Case of Venturi Effect

Field of Study: **Fluid Mechanics, Computational Fluid Dynamics**

I do solemnly and sincerely declare that: -

(1) I am the sole author/writer of this Work;
(2) This Work is original;
(3) Any use of any work in which copyright exists was done by way of fair dealing and for permitted purposes and any excerpt or extract from, or reference to or reproduction of any copyright work has been disclosed expressly and sufficiently and the title of the Work and its authorship have been acknowledged in this Work;
(4) I do not have any actual knowledge nor do I ought reasonably to know that the making of this work constitutes an infringement of any copyright work;
(5) I hereby assign all and every right in the copyright to this Work to the University of Malaya ("UM"), who henceforth shall be owner of the copyright in this Work and that any reproduction or use in any form or by any means whatsoever is prohibited without the written consent of UM having been first had and obtained;
(6) I am fully aware that if in the course of making this Work, I have infringed any copyright whether intentionally or otherwise, I may be subject to legal action or any other action as may be determined by UM.

Candidate's Signature Date:

Subscribed and solemnly declared before,

Witness's Signature Date:

Name:

Designation:

COMPUTATIONAL FLUID DYNAMICS SIMULATION OF INTERNAL FLUID FLOW: A CASE OF VENTURI EFFECT

ABSTRACT

This research project uses the Ansys Fluent software to conduct various Computational Fluid Dynamics (CFD) simulation of venturi effect occurs in an internal pipe flow based on the multi-phase flow model. In fluid dynamics, venturi effect is a common phenomenon occurs in an internal pipe flow, where a reduction in fluid pressure resultant from a fluid flow through a constricted section of a pipe. In accordance to Bernoulli's principle, an incompressible fluid's velocity increases as it passes through a constriction, in accordance to the principle of mass continuity, while its static pressure must decrease in accordance to the principle of conservation of mechanical energy. Hence, any gain in kinetic energy a fluid may attain by its increased velocity through a constriction is balanced by a drop in pressure. Physical experiments are time consuming and definitely will not be cost-effective to study the venturi-effect occurs within the pipelines. Therefore, CFD simulation is proposed to simulate the venturi effect occurs in an air ejector in this research project. Initially, a 2-D geometry modelling of the preliminary designed air ejector with water inlet, air inlet, converging section, throttling section, diverging section and outlet is constructed, conditioned, simulated and analyzed. The design and simulation process will be repeated to further improve the hydraulic conditions by modification of pre- and post-venturi effect piping arrangements. All simulation results will be plotted and analyzed from the pressure, velocity and volume fraction perspectives. Hypothetically, a streamline flow will enhance the hydraulic conditions and the mixing quality in this research project, through analyzing the simulation results of various improvement.

Keywords: Venturi Effect, Internal Fluid Flow, Computational Fluid Dynamics

SIMULASI KOMPUTASI BENDALIR DINAMIK BAGI ALIRAN BENDALIR DALAMAN: KESAN VENTURI

ABSTRAK

Projek penyelidikan ini menggunakan perisian Ansys Fluent untuk mejalankan pelbagai simulasi Computational Fluid Dynamics (CFD) terhadap kesan venturi yang berlaku dalam aliran dalaman paip berdasarkan model aliran pelbagai fasa. Dalam dinamik bendalir, kesan venturi adalah fenomena yang biasa terjadi pada aliran dalaman paip, di mana pengurangan tekanan bendalir disebabkan oleh aliran bendalir melalui bahagian paip yang tersekat. Menurut prinsip Bernoulli, kelajuan bendalir yang tidak dimampatkan akan meningkat ketika melalui penyempitan, berdasarkan dengan prinsip kesinambungan jisim; sementara tekanan statiknya harus menurun berdasarkan dengan prinsip pemuliharaan tenaga mekanikal. Sejurusnya, sebarang kenaikan tenaga kinetik cecair akan dicapai dengan peningkatan kelajuannya melalui penyempitan degan diimbangi oleh penurunan tekanan. Eksperimen fizikal memakan masa dan pasti tidak menjimatkan kos untuk mengkaji kesan venturi yang berlaku di dalam saluran paip. Oleh sedemikian, simulasi CFD dicadangkan untuk mensimulasikan kesan venturi yang berlaku pada ejektor udara dalam projek penyelidikan ini. Pada permulaannya, pemodelan geometri 2-D sebuah ejektor udara dibentukan dengan saluran masuk air, masuk udara, bahagian konvergensi, bahagian pendikit, bahagian penyimpangan dan saluran keluar dibina, dikondisikan, disimulasikan dan dianalisis. Proses reka bentuk dan simulasi akan diulangi untuk memperbaiki keadaan hidraulik dengan mengubahsuai susunan paip kesan pra- dan pasca-kesan venturi. Semua hasil simulasi akan diplot dan dianalisis dari perspektif tekanan, halaju dan pecahan isipadu Secara hipotetis, aliran arus akan meningkatkan keadaan hidraulik dan kualiti pencampuran dalam projek penyelidikan ini, melalui analisis hasil simulasi dari pelbagai pemajuaan.

Kata Kunci: Kesan Venturi, Aliran Bendalir Dalaman, Komputasi Dinamik Bendalir

ACKNOWLEDGEMENTS

First and foremost, I would like to express my sincere gratitude to my respectful supervisor Associate Professor Ir. Dr. Poo Balan A/L Ganesan for his continuous support, guidance and mentorship in completing this Research Project, as part of the fulfilment of my Master of Mechanical Engineering postgraduate studies and related researches. His patience, motivation, and immense knowledge have guided and assisted me to complete this project on time.

In general, I am very grateful and thankful to University of Malaya for continuous offering and facilitating the Master of Mechanical Engineering postgraduate program to inculcate the life-long learning practices and as well as provide working adults an opportunity to further study to update with new knowledge, while recapitulating lessons learned in undergraduate.

My sincere appreciation also extends to all my professors and lecturers for conducting and facilitating the postgraduate program professionally; and my fellow course-mates for sharing their thoughts constructively along the postgraduate learning journey. All of their advice and guidance are very useful indeed. Unfortunately, it is not possible to list all of them one by one in this limited space.

Last but not the least, my truthful appreciation goes to my family for their continuous encouragement and restlessness support throughout my postgraduate study. I am able to complete this research project and subsequently the postgraduate program as a whole, with the patience, trust and freedom given by them.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF SYMBOLS AND ABBREVIATIONS

2-D	:	Two-Dimensional
3-D	:	Three-Dimensional
A	:	Area
p	:	Pressure
v	:	Velocity
ρ	:	Density
CFD	:	Computational Fluid Dynamics
UDF	:	User Defined Function
SIMPLE	:	Semi-Implicit Method for Pressure-Linked Equations
SIMPLEC	:	SIMPLE-Consistence
PISO	:	Pressure-Implicit with Splitting of Operators
FSM	:	Fractional Step Method
COUPLED	:	Algorithm Solves Momentum and Continuity Equation
PRESTO!	:	Pressure Staggering Option

LIST OF APPENDICES

Not Applicable

CHAPTER 1: INTRODUCTION

1.1 Research Background

Venturi effect is a normal phenomenon occurs in an internal pipe flow as shown in Figure 1.1. It is an effect of a fluid pressure reduction, resultant from a fluid flows through a constricted throttling section of a pipe (Reader-Harris, 2015). According to the law of mechanical energy conservation, venturi effect is occurs in a pipeline based on a sudden flow velocity increase, due to the principle of mass continuity in a specific nozzle geometry that leads to the local static pressure decrease (Bogdanović-Jovanović, Stamenković, Spasić, Petrović, & Kocić, 2018). Operating characteristics of venturi nozzle depend on the flow characteristics and geometry of the nozzle. The application of the Venturi tubes in the process industry and the industry in general is huge and significant, therefore the determination of their operating characteristics is considerably important. Venturi-effect is used in various daily applications and in-depth study on this phenomenon is important to contribute for more engineered application in the future. The common applications of a venturi are in the measuring devices, where venturi meters used to measure the volumetric flow-rate of a specific fluid; mixing devices, where air ejector used to mix fluid with air; etc.

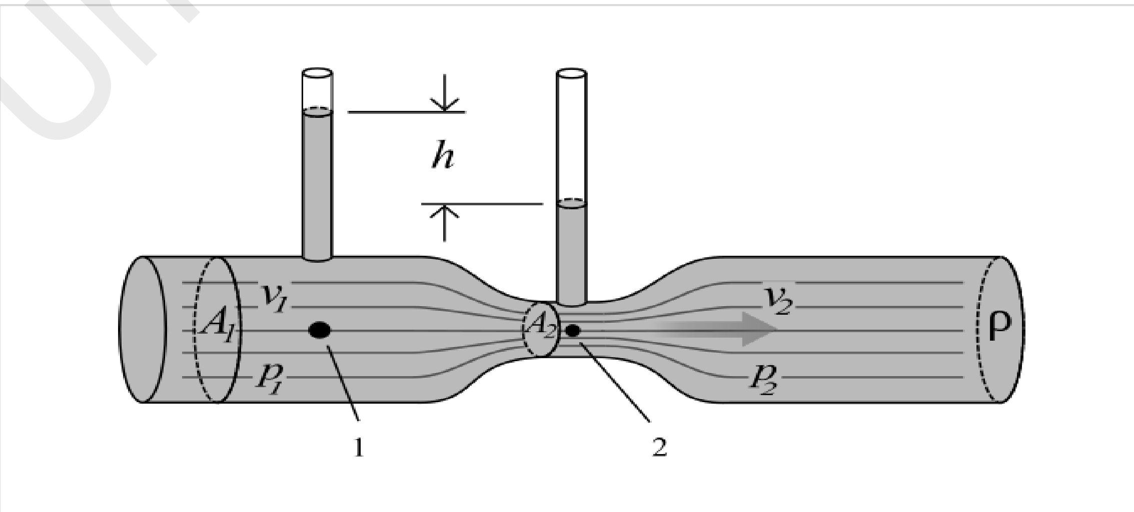

Figure 1.1 Venturi Effect (Bogdanović-Jovanović et al., 2018)

In the study of fluid dynamics, velocity increases in an incompressible fluid as it flowing through a constriction, in accordance to the principle of mass continuity, while its static pressure must decrease in accordance to the principle of conservation of mechanical energy based on the Bernoulli's principle. Therefore, any gain in kinetic energy, a fluid may attain by its increased velocity through a constriction is balanced by a drop in pressure, as shown in Figure 1.2 (Şcheaua, 2016). Conventionally, physical experiments are time consuming and definitely will not be cost-effective to study the venturi-effect occurs within the pipelines, where unnecessary time is wasted to redo the design, as well as resources to make prototyping or physical hydraulic modelling. Therefore, Computational Fluid Dynamics (CFD) is vastly deployed to model and simulate the phenomenon instead. Post-venturi effect pipe arrangements will be simulated based on various velocity-pressure models incorporated into a continuity equation by using further establishment of User Defined Function (UDF), which will reflect the variable wave speed of the venturi-effect flow and a related algorithms were established based on Navier-Stokes equation principle (Yeoh, Liu, Tu, & Timchenko, 2011). Simulation results then will be analyzed from the fluctuation of pressure, velocity and volume fraction perspectives.

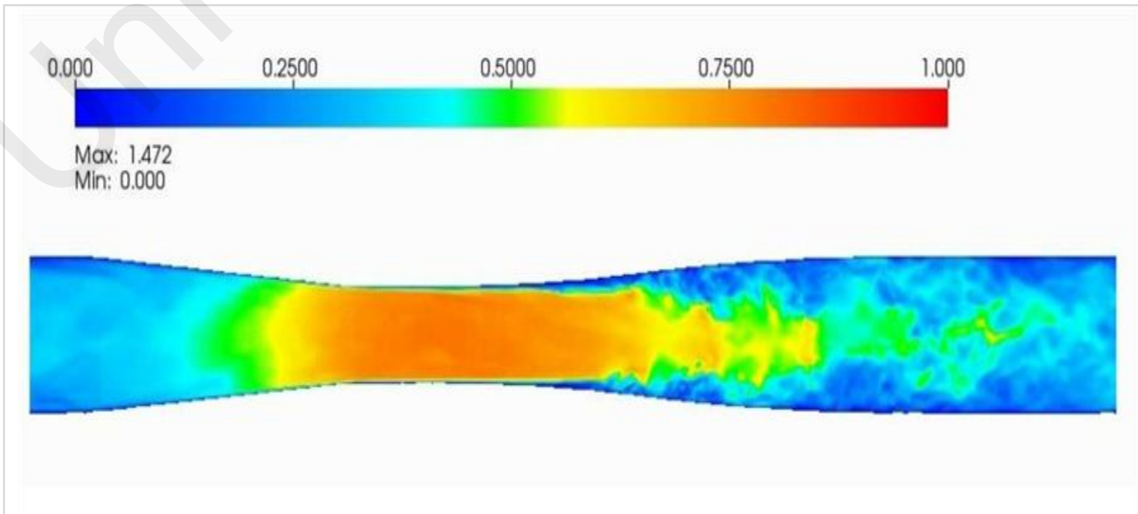

Figure 1.2 CFD Simulation of Venturi Effect (Şcheaua, 2016)

1.2 Research Gap and Problem Statement

CFD simulations that have been conducted in existing research to visualize and analyze the venturi effect are very rare, especially case studies applied in multi-phase flow have not been widely carried out, due to the maturity of the theoretical knowledge in design. This research project has been conducted to close the research gap through modelling a 2-D air ejector and conditioning to a multi-phase flow (mixture of water and air) to simulate the venturi effect, and subsequently visualize the flow distribution characteristics from velocity and pressure perspective; and the fluid mixing pattern through the volume fraction rendering. Multiple design improvements will be made based on the preliminary designed air ejector with same setup for calculation to visualize the results for comparison with the previous design and reference for future design. This research project is important for fluid dynamics study, all the simulation and analysis results are resourceful for improving the knowledge on venturi effect simulation and internal flow study as a whole ultimately. This research project will also contribute to the knowledge of mixing through the CFD simulation of multi-phase flow of a venturi effect and enhance this knowledge for future engineering application purposes. The success of this research project will spur the interest of postgraduate students or fluid mechanics researchers to conduct more researches on various applications of venturi effect through CFD simulation.

This research project is initiated to simulate and analyze the venturi effect occurs in a pipeline through conducting CFD simulations at various parameters. A preliminary design of a venturi air ejector is drafted as of Figure 1.3. Generally, a conical reducer is located at the contracting section converging the water inlet flow into the throttling section, with the air inlet located at vertically downward into the contracting section diverted by the reducer into the throttling section, following with a short throttling section and toward a broad and long diffusing section.

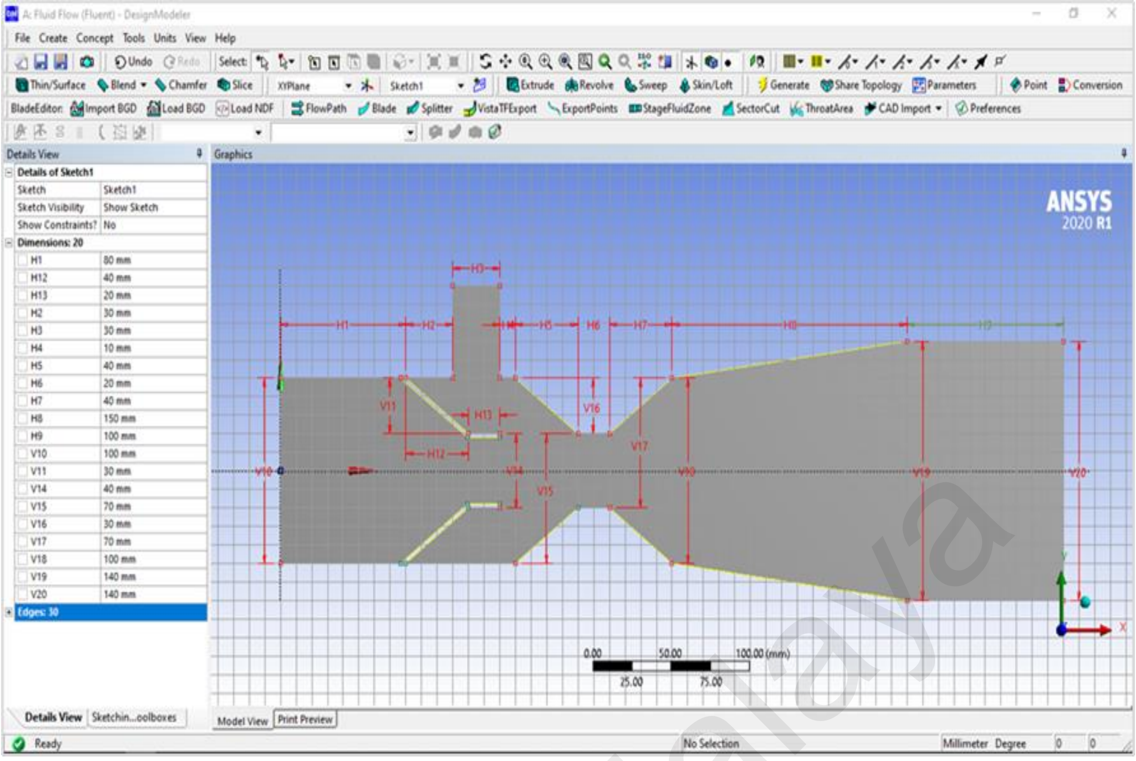

Figure 1.3 Preliminary Design of Venturi Air Ejector Mixer

Numerical method is chosen to investigate and analyze of venturi effect multiphase flow. Therefore, numerical method is deployed through CFD simulation in this research project, with the ANSYS FLUENT 2020 R1 Student Academic Version software is utilized to carry it out. CFD simulations visualize the changes of pressure and velocity, and the mixing process between the water and air through the air fraction output through multiphase flow modelling. The CFD simulation results are visualized and analyzed to determine optimization to be made to improve the mixing process. The preliminary design will be improved and simulated repeatedly, with several pre- and post-piping arrangement to demonstrate the flow characteristics, with the aim to improve the mixing quality through the volume fraction visualization. Finally, the relevant contours and graphs will be plotted and analyzed for design recommendations in the future. Hypothetically, improvement needs to be focused on hydraulics condition that will create a streamline flow in the pipeline to ensure the mixing quality will be increased.

1.3 Research Aim and Objectives

The aim of this research project is to simulate and analyze venturi effect occurs in pipeline through conducting CFD simulation on various pipe arrangements modelling to enhance the hydraulic conditions and multi-phase flow mixture (air-water) distribution subsequently, with the following highlighted research objectives: -

i. To simulate venturi-effect phenomenon occurs in an internal pipe flow through CFD simulation.
ii. To model various pre- and post-venturi effect pipe arrangements in an internal pipe flow through CFD simulation.
iii. To analyze the earlier modelled pre- and post-venturi effect pipe arrangements simulation results from the pressure, velocity and volume fraction perspectives.

1.4 Research Scopes and Limitation

The scopes this research project, as guidelines to achieve the research objectives and research aim ultimately, are identified and not limited to conduct CFD simulations based on a preliminary designed venturi air ejector arrangement to visualize the changes of pressure and velocity within the 2-Dimensional (2-D) model, as well as to model the mixing process between the water and air through the volume fraction visualization through multiphase flow modelling. The simulation process will be repeated to further improve the piping hydraulic design by relocation of the air inlet to throttling section; removal of the water inlet reducer at the converging or contracting section; extension the length of the throttling section with air inlet; and adding a flow straightener at the diverging or diffusing section immediately after the throttling section. In order to conclude the research study, comparison on the various models' design will be carried out to study the pressure and velocity changes through analyses of simulations results.

The limitation of this research project is identified as the directives to focus in meeting the research objectives. A low number of nodes has to be maintained in meshing the model for respective simulation, the main reason being is due to the computing power constraint of the computer used to perform the CFD simulations. Nevertheless, the CFD software used only limits maximum of 512,000 nodes for meshing. In order to achieve accurate simulation results, the meshing size needs to be well modelled with small number of nodes. However, this condition may require the time step size to be set at very small to prevent divergence of solution during calculation. Small time step size will tremendously increase the computational cost from both the simulation time and computation power perspectives. Due to the computer low computing power, mesh dependency test is unable to be carried out to ensure that simulation results are accurate and consistent. Therefore, all the models meshing size applied in this research project is maintained at a relatively small, yet sufficient to achieve accurate simulation results at reasonable simulation time.

1.5 Organization of Research

Overall, this research project report is structured and organized into 5 chapters following by references and appendixes. Firstly, Chapter 1 provides the overview of the research, through introducing the research background; identifying the research gap and problem statement; setting the research aim and objectives; defining the research scopes and limitations; and structuring the organization of research to introduce the research topic and present with its associated research framework, as a guideline for the research implementation. Following by, Chapter 2 discusses the fundamental theories about the Internal Flow, Venturi Effects and Computational Fluid Dynamics through reviewing of journals articles and reference books, to substantial the research topic and its associated research framework, as a reference for the research. Meanwhile, Chapter 3 proposes the research design, explaining how the modelling, meshing, simulation and data analysis are being carried out and the research tools to be used for simulation and data analyzing for the project implementation. Chapter 4 presents the simulation implemented and data analysis conducted to analyze the simulations, explanation of the simulation outcomes that are being carried; and makes recommendation for design improvements. Lastly, Chapter 5 as an overall conclusion for the research project that has been conducted: - concludes the research outcomes, and conclusion; summarizing the significance of research, and contributions to knowledge; and recommendation future works to be conducted for research.

CHAPTER 2: LITERATURE REVIEWS

2.1 Overview

In this chapter, technical literatures on journal, articles, and reference books related to Venturi Effect, Air Ejector and Computational Fluid Dynamics (CFD) are being reviewed, highlighted and presented. Firstly, the theoretical reviews on Internal Flow and Venturi Effect are discussed. Following by the technical reviews on CFD simulations conducted for internal flow will be evaluated. Lastly, this chapter will be concluded with a concise summary to explain the correlation between both earlier discussed topics, which forms major core of the research gap.

2.2 Venturi Effect

Giovanni Battista Venturi (1746-1822), an Italian scientist in the late 18th century has discovered that water flowing through a throttling section in a hydraulic energy line will develop an unexpected pressure drop (Tang, Juárez, & Li, 2019). This effect is commonly using in modern fluid systems, nevertheless to measure flow rates and produce partial vacuums in a particular process pipeline. He has conducted numerous laboratory tests to determine the best method to drain water from low-lying marshy areas. Figure 2.1 shows in one of the experiments, he used a large tank with the volume of several gallons and was connected to a smaller open tank with a pipe from the bottom (Reader-Harris, 2015). As the water flows in gravity and rapidly into the smaller tank, a ramp is used to drag water from it. This method was successfully used the technique to drain local marsh lands and subsequently developed into jet pump in the modern days. He has then expanded his hydraulic experiments by using a pipe with a throttling section, which he defined it as a contracted vein; connected to the bottom of the same large experimental tank. A small glass indicator tube was connected to the top of the throttling section from a container of colored water at bottom. As water flows from the tank, it passed though the throttling section, the velocity increases with pressure drops at the same time, and drag up the colored water. This velocity-pressure characteristic is named as the Venturi Effect (Reader-Harris, 2015).

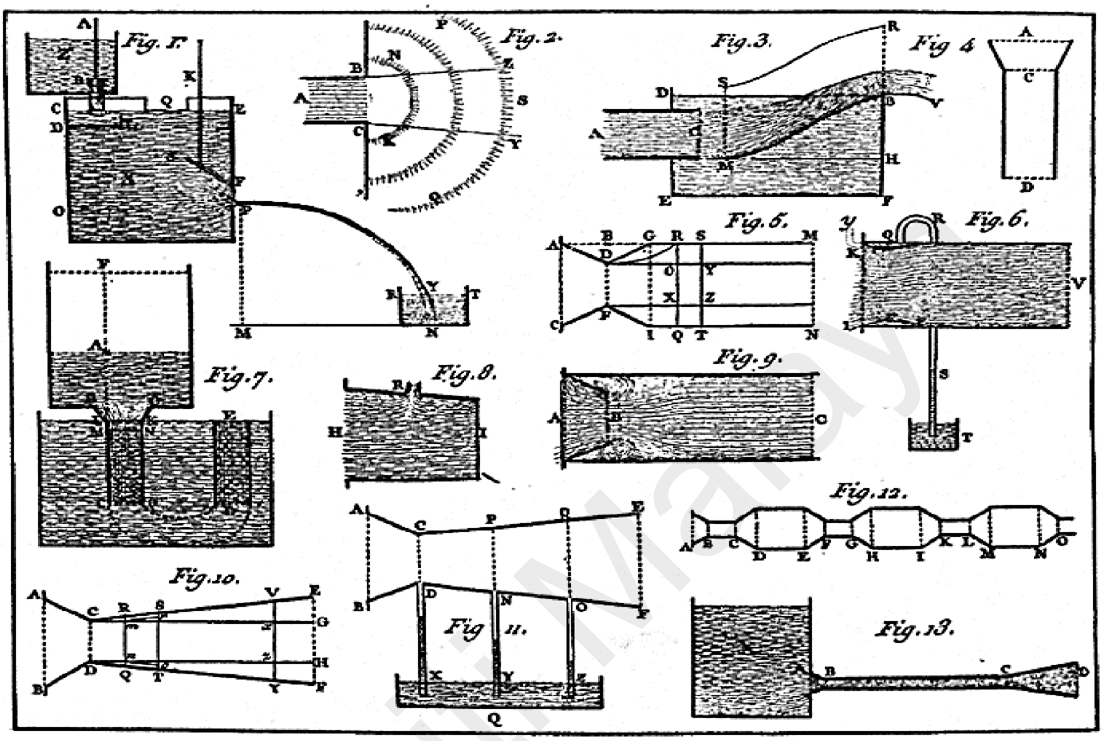

Figure 2.1 Development of Venturi Effect in 1797 (Reader-Harris, 2015)

The Venturi effect highlighted that flow in a situation with constant mechanical energy, fluid velocity flowing through a narrow area will increase and its static pressure will decrease. Both the principle of continuity and the principle of conservation of mechanical energy are used to derive this hydraulic effect (Shinde, Chaudhari, Patil, & Marathe). As the fluid flowing through the throttling section, fluid molecules will accelerate due the cross sectional area us constricted (Zheng, Li, & Qin, 2018). The fluid molecules must accelerate in the throttling section, as the total flow rate is still remaining the same. The amount of fluid molecules flowing through the pipe in a given time must be same as the amount of fluid molecules rushing through throttling section and flowing out from the other end. Since, the cross sectional area for the throttling area is smaller, therefore the fluid molecules must accelerate in order for sufficient fluid molecules to rush through in the specified time. (Xu, Liu, & Pang, 2016) states the static pressure decreases when a fluid in the subsonic regime is forced through a pipe with a smaller cross-sectional area. Therefore, an ideal, incompressible and inviscid form of the Bernoulli equation reasonably describes that the relationship between velocity and pressure. Figure 2.2 and Equation (1) shows that, the velocity increases in the event of a pressure drops, and vice versa. This characteristic is observed in the phenomenon of viscous and weakly compressible flows.

$$p_1 - p_2 = \frac{\rho}{2} - (v_2 - v_1) \tag{1}$$

$$where\ A = Area;\ p = Pressure;\ v = Velocity;\ \rho = Density$$

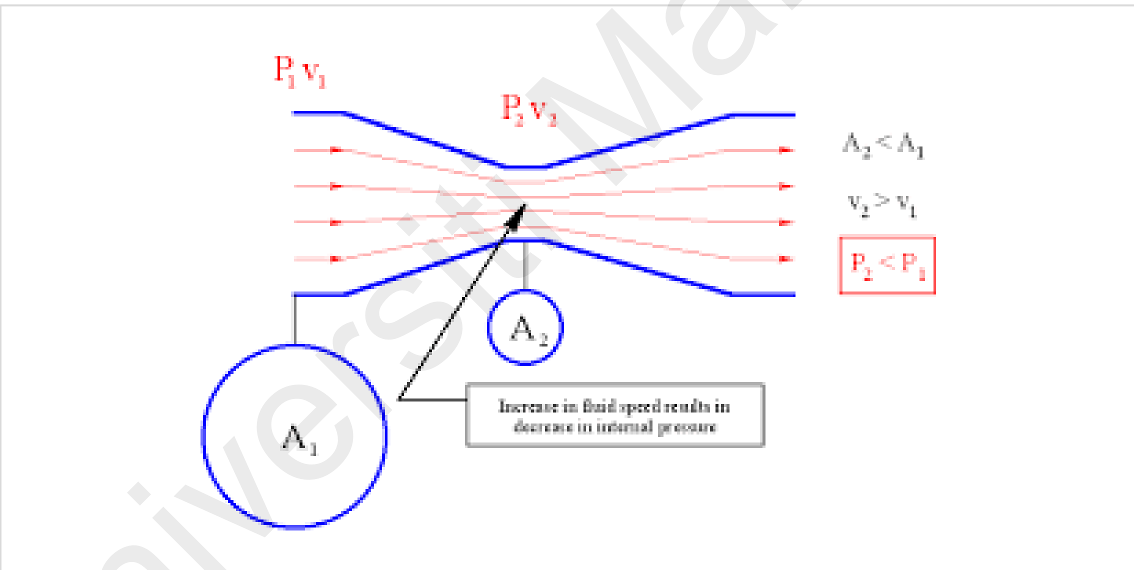

Figure 2.2 Venturi Effects (Xu et al., 2016)

Understanding of the internal flow field mechanism for venturi effect is important to enhance venturi jet mixer performance. (Djebedjian, Abdalla, & Rayan, 2000) discussed that the further away the air nozzle exit, a more uniform velocity profile will be formed across the flow cross section, due to the jet fluid transports slower as the distance increases and its viscous action transfers its kinetic energy to the environment as shown in Figure 2.3. (CROFT, 1976) studied the ejector internal flow behavior, specifically at the mixing process in between the primary flow and secondary flow. The energy contours presented defined the mixing point, whereby a higher rate of thermal energy is generated due to the high turbulence length scale for the mixing position; and nevertheless the turbulent length scale decreases gradually throughout the throat section (Sundararaj & Selladurai, 2013). This phenomenon indicates that energy is transfers to the propelled stream from the motive stream rapidly. The turbulence length scale is a physical quantity quantifies the large eddies with energy in the turbulent flows and the pipe diameter will restrict the turbulence length scale in fully developed flows of any particular pipe (Reader-Harris, 2015).

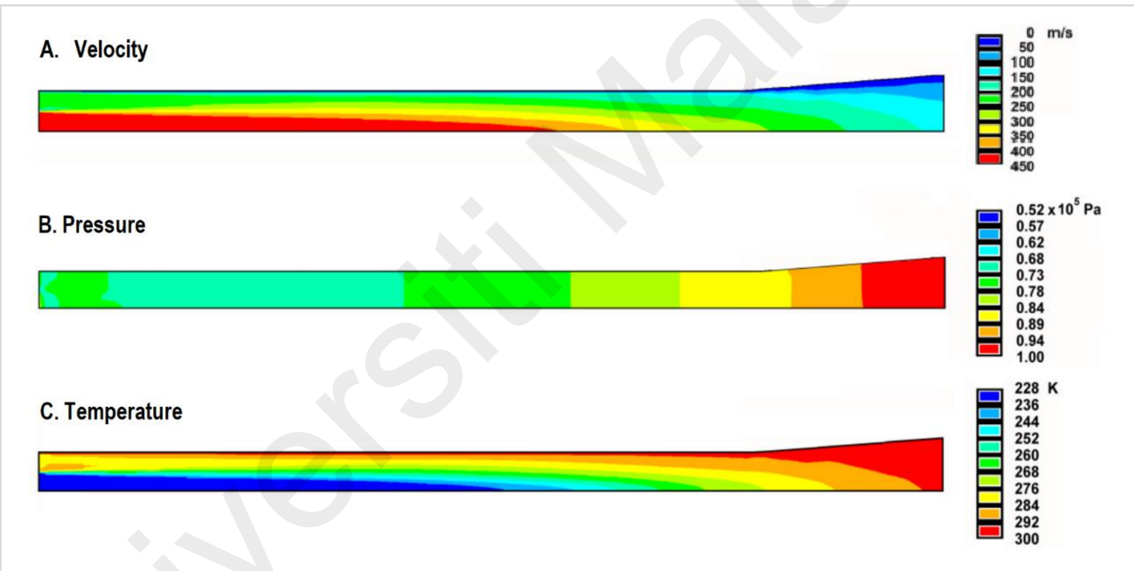

Figure 2.3 Velocity, Pressure and Temperature Distribution Profiles along Throttling Section (Djebedjian et al., 2000)

Figure 2.3 shows that the ejector performance at the throttling section with its effect of these parameters on velocity, pressure and temperature were studied by (Djebedjian et al., 2000). The flow velocity distribution indicates the degree of mixing between both the motive and propelled streams, and the quantity of entrained fluid. The length of the throttling section contributes a significant effect in producing a uniform velocity profile to the divergence section entrance. Figure 2.3 (A) shows that the fluid velocity profile is decreases along the throttling section to the divergence section, whereas Figure 2.3 (B) shows the pressure increases significantly from the throttling section towards the divergence section. Figure 2.3 (C) shows the fluid temperature profile along the throttling section. As the fluid velocity decreases, the static temperature increases. The fluid temperature increases in an exchange of energy process, as the heat is generated from the kinetic energy losses. Both the fluid velocity and the static temperature profiles are identical but in an inverse magnitude.

2.3 Air Ejector

An air ejector allows a primary flow as the motive fluid to entrain another fluid flow, applicable to both compressible and incompressible flow, either as a single-phase or two-phase mixture (Galanis & Sorin, 2016). Figure 2.4. shows a cross-sectional illustration of an air ejector consists of four (4) main components, namely primary nozzle, secondary chamber, a mixing chamber and a diffuser (Tamhankar, Pandhare, Joglekar, & Bansode, 2014). An air ejector is designed based on Bernoulli's Principle, states that a high pressure motive fluid flow accelerated through a nozzle will exit the nozzle at higher velocity, due to its kinetic energy increases and pressure reduces (Huang & Chang, 1999).

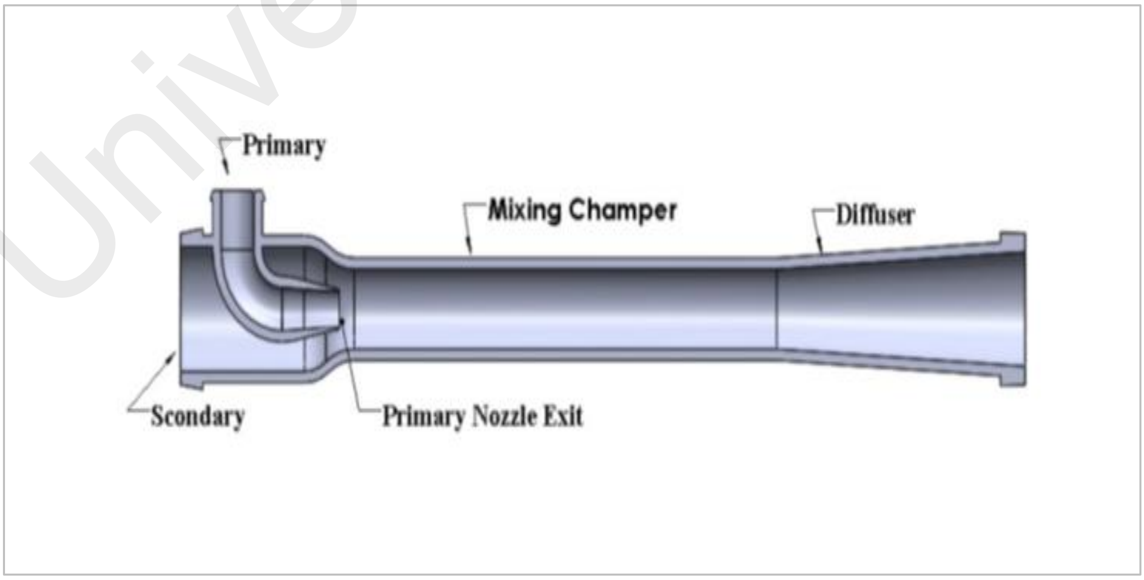

Figure 2.4 Cross Sectional Drawing of an Air Ejector (Tamhankar et al., 2014)

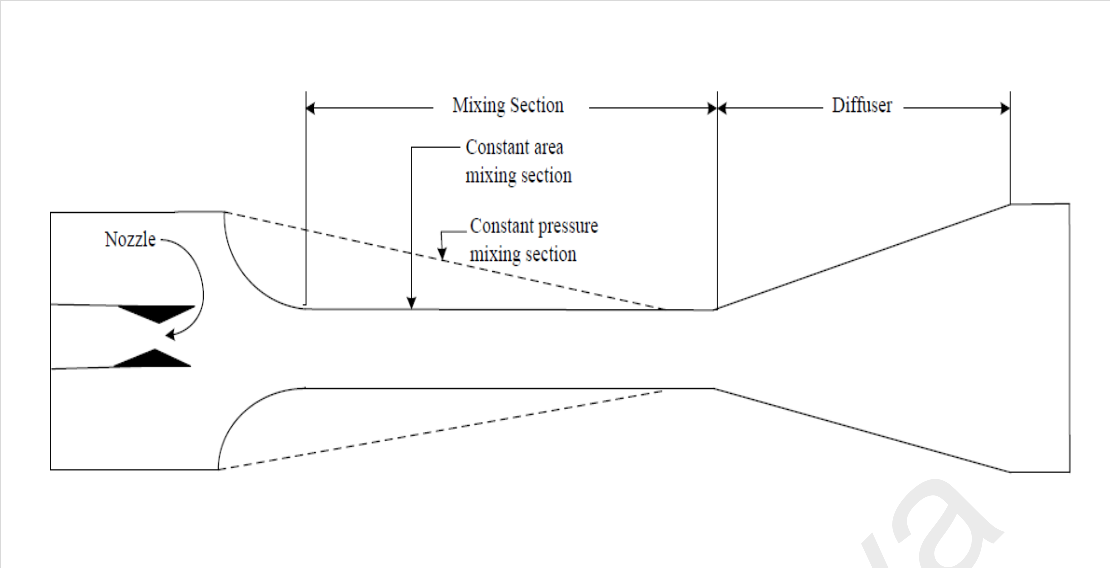

Figure 2.5 Types of Air Ejector (Tamhankar et al., 2014)

Air ejector is categorized as either constant pressure jet ejector or constant area jet ejector, depending on their convergence configuration. Figure 2.5 shows the difference between both types of ejector. (Huang & Chang, 1999) states that since the motive fluid flow creates a vacuum at the nozzle exit due to the pressure drop, a low-pressure and low-velocity secondary fluid flow induces. Both the motive fluid (liquid or gaseous) and secondary fluid integrate in the mixing chamber and form a mixture of flow. (Zhang, 2017) states that there will be two types of fluid at different pressures and velocities, the fluid flow energy and momentum will transfer from high to low pressure, and high possibility that turbulence will occur in the mixing chamber. Generally, the mixture of fluid is assumed to be fully developed at the end of the mixing chamber, as the pressure partially recovering and the flow is decelerating at the same time (Vijay & Subrahmanyam, 2014). As most of the energy is assumed to be lost in the mixing chamber, the mixture of fluid will be assumed as homogenous and no slip of velocity as well. The mixture of fluid further decelerates and gains more pressure, due to the energy conversion happens at the diffuser which forms the last section of the air ejector, whereby the kinetic energy is converted into potential energy (Hemidi, Henry, Leclaire, Seynhaeve, & Bartosiewicz, 2009). Figure 2.6 and 2.7 show the pressure distribution characteristics within an air ejector and pressure-velocity variation from inlet nozzles to mixing chamber and later to diffuser respectively (Wang, 2017).

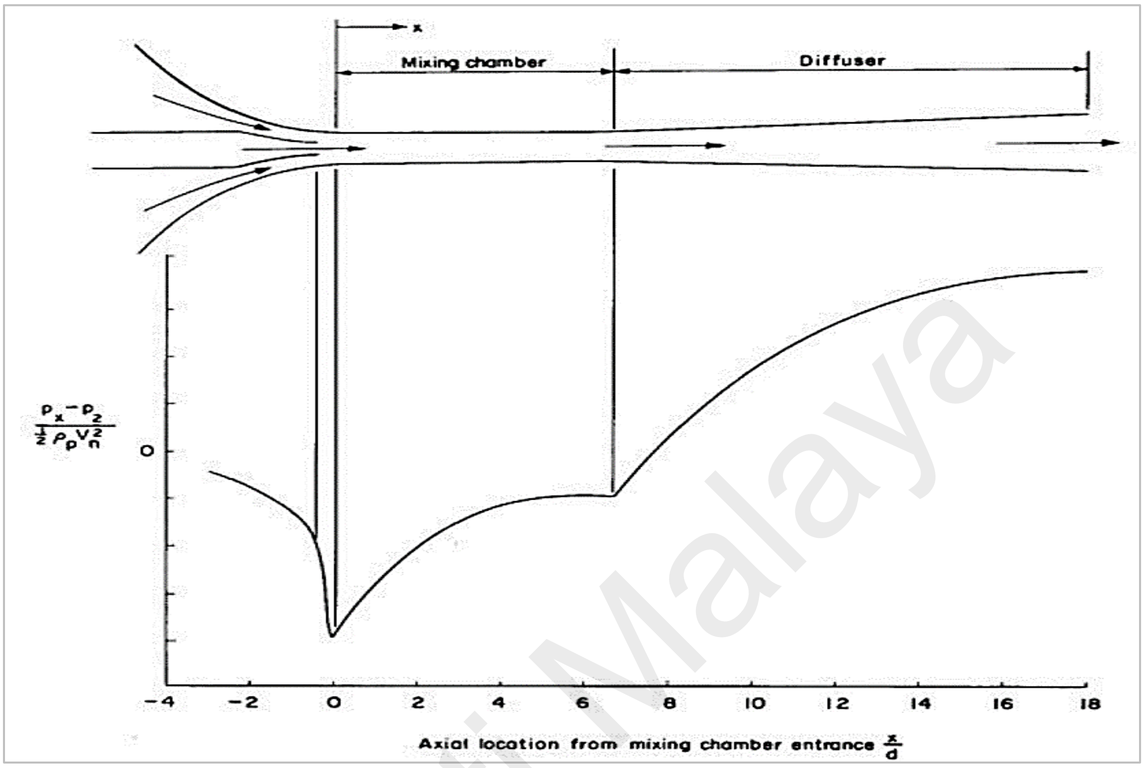

Figure 2.6 Pressure Distribution within an Air Ejector (Wang, 2017)

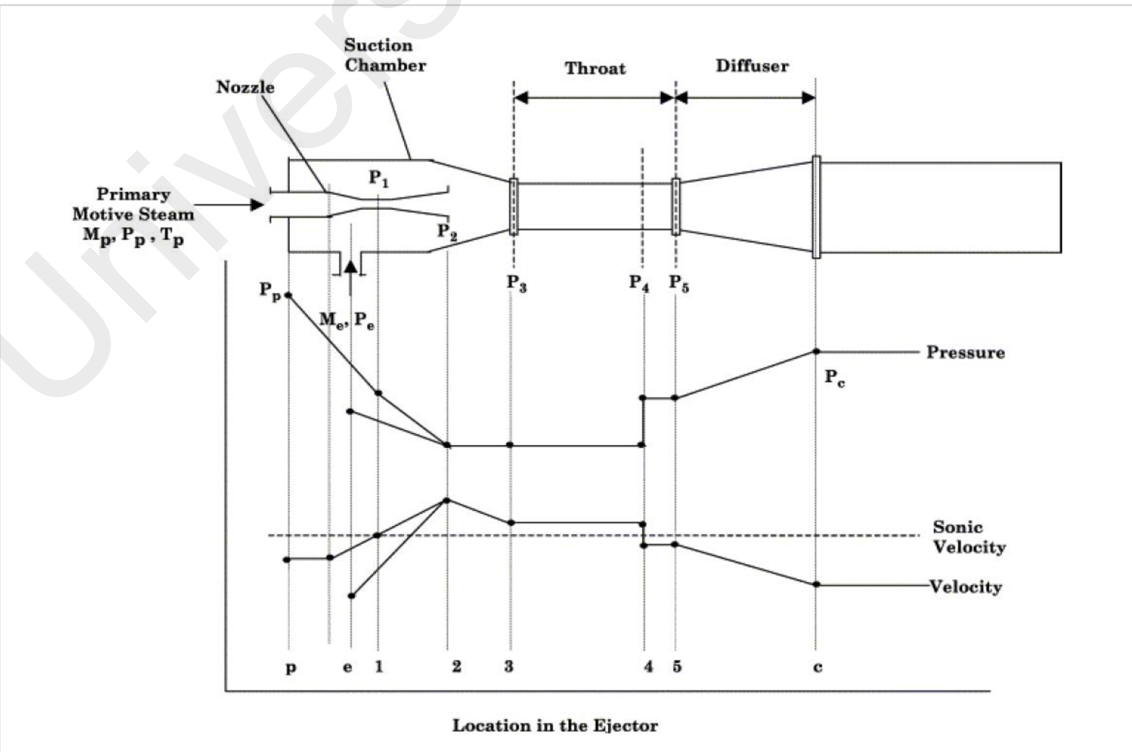

Figure 2.7 Pressure-Velocity Variation within an Air Ejector (Wang, 2017)

An air ejector performance can be determined through the phenomenon of mixing, separation, friction, turbulence, and energy consumption in the propelled suction of the stream. (Forney & Kwon, 1979) states that a turbulent mixing shall be created or enhanced in order to maximize jet ejector performance. The nozzle geometry should be well-designed to the right dimensions. in order to boost the tangential shear interaction between both the motive and propelled streams (Bhatkar & Ban, 2019). An air ejector should be tactfully designed in details to diminish turbulence effects, to ensure both streams should blend and mix thoroughly inside the throttling section (Singhal & Parveen, 2013). There are two most important parameters, namely flow ratio and pressure ratio used to describe the performance of an ejector. Multiplication of both ratios describes the ejector performance or overall efficiency, whereby the flow ratio is the secondary flow rate over to the primary flow-rate, and the pressure ratio is the secondary pressure rise over to the primary pressure drop. An ejector performance normally is determined based on either a single-phase (liquid or gaseous) or a multi-phase mixture of fluids (liquid and/or gaseous) (Zheng et al., 2018). In this research project, a multi-phase mixture of water-liquid and air-gaseous will be modelled and simulated for results and discussion.

2.4 Computational Fluid Dynamics (CFD)

CFD has been emerging in engineering simulation application purposes ever since the 1950s, due to the huge improvement of speed of computers and its memory capacity. CFD is primarily established as a simulation tool for flow-based modelling, structural design, process evaluation, etc. With a properly design implementation plan, CFD will be a cost effective, rapid, non-intrusive, parametric test method. (Sudakhar, 2017) states that as CFD is principally a design tool that encourages developments with greater repeatability, as well as reliability; CFD only consumes at a fraction of the cost and time as compared to the traditional design approaches, that involve empiricism, followed by prototyping and testing. In fluid dynamics, CFD based on numerical method has five major advantages as compared with the experimental method:- significant lead time reduction in design and development; simulate flow conditions that is not reproducible in experimental model tests; more detailed comprehensive calculation and information; more cost-effective as compared to physical modelling and testing; and lower energy consumption (Baylar, Aydin, Unsal, & Ozkan, 2009). With the rapid development of computers, CFD is capable to solve even more complex problems that demanding for more details at very high precision. Figure 2.8 shows a 3-D CFD simulation of a venturi effect model has been implemented to visualize the phenomenon.

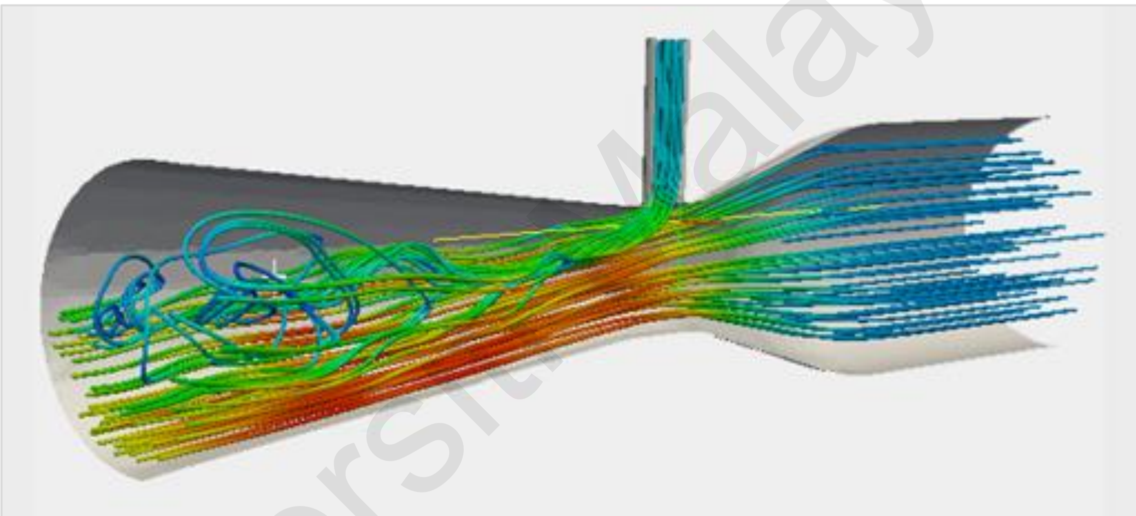

Figure 2.8 3-D CFD Simulation of a Venturi Effect (Sudakhar, 2017)

ANSYS Fluent module is a state-of-the-art CFD simulation software used to predict fluid-flow, heat and mass transfer, chemical reactions and other related phenomena. It is a well-recognized in delivering the accurate solutions for advanced physics modeling capabilities, nonetheless cutting-edge turbulence models, multi-phase flows, heat-transfer, combustion, shape optimization, multi-physics, etc. Figure 2.9 shows the two-stage processes to obtain the computational solution in ANSYS Fluent (Arias & Shedd, 2007). The first stage is the Discretization process, where the continuous partial differential equations are converted into a discrete system of algebraic equations. The detail of discretization is explained in the following section. Discretization is a process that converts the governing partial differential equations to a system of algebraic equations. The second stage is the Equation/Numerical Solving process, whereby a solver (Segregated or Coupled) is selected to solve the discrete system obtaining from the first stage and the solution of the system of algebraic equations will be obtained subsequently (Saharan, 2016).

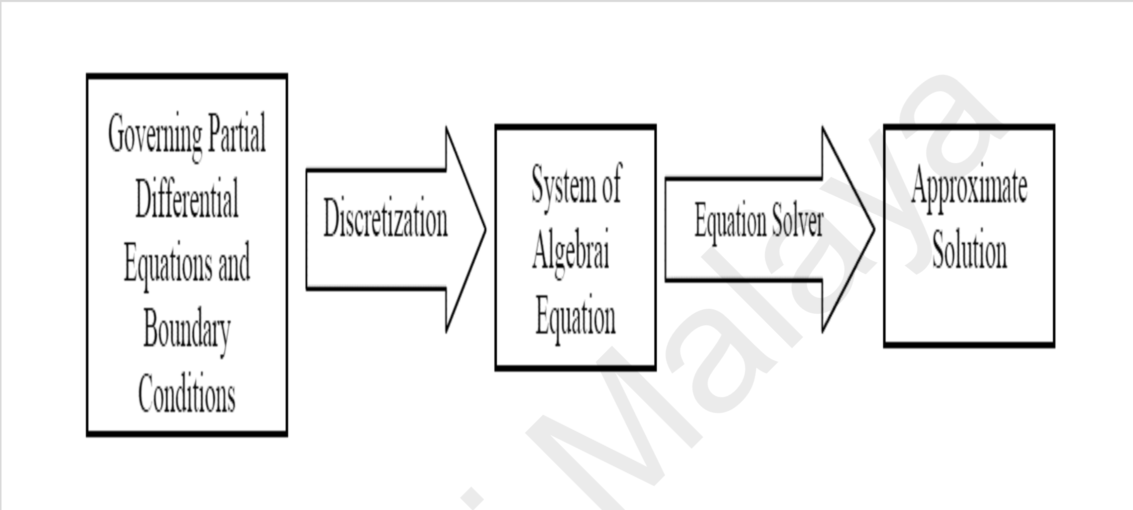

Figure 2.9 Overview of Computational Solving Processes (Arias & Shedd, 2007)

In ANSYS Fluent, there are a total of five (5) options available for the pressure-velocity coupling algorithms, which are SIMPLE (Semi-Implicit Method for Pressure-Linked Equations); SIMPLEC (SIMPLE-Consistence); PISO (Pressure-Implicit with Splitting of Operators); COUPLED and FSM (Fractional Step Method). All algorithms in principle should derive the same results. In actual application, all of them often will get results at very closed proximity, but don't give the exact same values, due to uncertain or unforeseen reasons. Theoretically, the COUPLED scheme should give the most accurate results, due to it is basically works on brute force, but the COUPLED solver scheme is computationally inefficient and caused this solution the upmost expensive. PISO and COUPLED solver schemes generally are more aggressive and converge faster as compared to SIMPLE per iteration cycle. However, SIMPLE is computationally faster and the overall computational cost allows to perform more iterations as compared to use

PISO or COUPLED. Both PISO and COUPLED solver schemes are preferably apply for transient simulations, due to its aggressive convergence behavior. However, transient simulation might use SIMPLE, as it is computationally faster to offset with the slower convergence through calculating more iterations. The selection between solver schemes is basically based on the computational efficiency, where COUPLED can be vastly different as compared to SIMPLE/PISO that highly nonlinear (pressure and temperature dependent) property changes. Since the equations are fully coupled, hence there will be no bias of the solution. In this research project, the SIMPLE algorithm is chosen as it uses a relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field.

2.5 Summary

Generally, venturi effect is a phenomenon widely adopted for designing an air ejector, based in the principle of velocity increases and pressure drops when steady flow passing through a constriction. CFD simulations are commonly conducted to visualize and analyze flow in fluid dynamics study. However, CFD simulation on venturi effect, especially applied in multi-phase flow case studies have not been widely carried out, due to the maturity of the conventional theoretical knowledge in design the equipment. Therefore, there is a need in research to conduct more studies on air ejector CFD simulations with the multi-phase flow (mixture of water and air) model, to visualize the flow distribution characteristics from velocity and pressure perspectives; and subsequently the fluid mixing pattern through the volume fraction rendering.

CHAPTER 3: RESEARCH METHODOLOGY

3.1 Research Design

This research project with the focus on CFD simulation on Venturi Effect is planned and designed to be implemented in four (4) phases, namely first phase – modelling and meshing; second phase – setup and simulation; third phase – evaluation and improvement; and fourth phase – data analysis, based on the problem statement and research scopes set on the earlier chapter. With the successful implementation of all phases of research design, research aim and objectives will be achieved as a whole.

A preliminary designed 2-D air ejector arrangement model with water inlet, air inlet, converging section, throttling section, diverging section and outlet will be modelled and setting up for calculation and simulation to visualize the changes of pressure and velocity, as well as to model the mixing process between the water and air through the air fraction visualization through multiphase flow modelling.

3.2 Modelling and Meshing

Initially, 2-D geometry modelling of the preliminary designed air ejector (Model 1) with water inlet, air inlet, converging section, throttling section, diverging section and outlet as shown in Figure 3.1 is constructed using ANSYS DesignModeler based on the given dimensions, following with meshing using ANSYS Meshing on the modelled geometry; and then setup with various settings for calculation and simulation The design and simulation process will be repeated to further improve the piping hydraulic design by relocation of the air inlet to throttling section (Model 2) as shown in Figure 3.3; removal of the water inlet reducer at the converging or contracting section (Model 3) as shown in Figure 3.5; extension the length of the throttling section with air inlet (Model 4) as shown in Figure 3.7; and adding a flow straightener at the diverging or diffusing section immediately after the throttling section (Model 5) as shown in Figure 3.9.

3.2.1 Model 1 – Preliminary Design

3.2.1.1 2-D Geometry

Preliminary air ejector (Model 1) design with water inlet, air inlet, converging section, throttling section, diverging section and outlet is drafted with the dimensions shown in Figure 3.1.

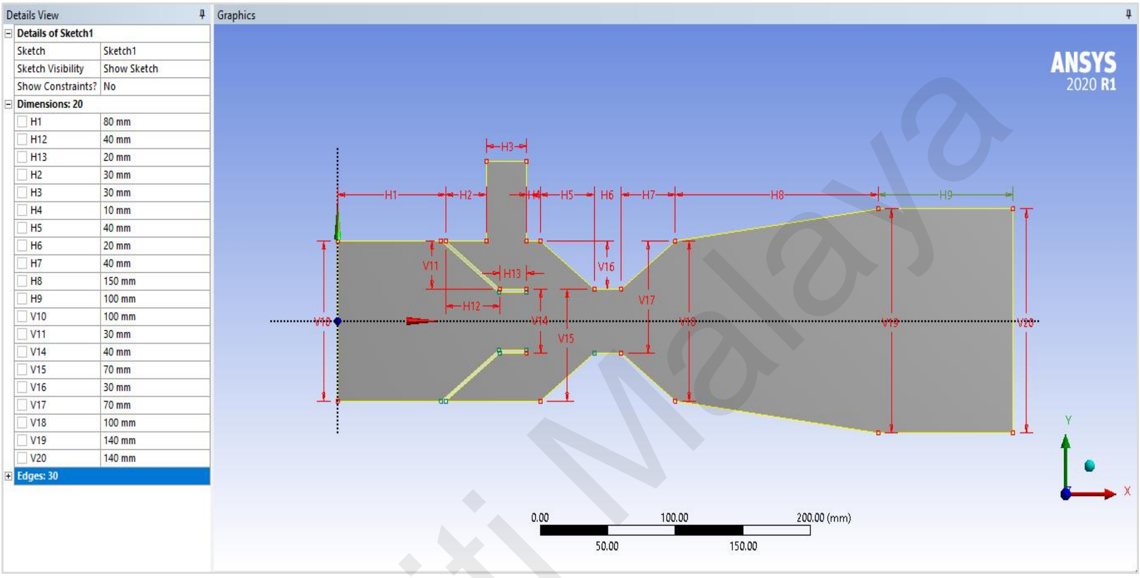

Figure 3.1 Model 1 with Dimensions

3.2.1.2 Meshing

Mesh shown in Figure 3.2 been created for the earlier modelled preliminary air ejector (Model 1) design with sizing resolution, element size, inflation growth rate, quantity of nodes and elements shown in Table 3.1.

Table 3-1 Model 1 Meshing Settings and Statistics

Sizing Resolution	2
Element Size	2.5 mm
Inflation Growth Rate	1.1
Nodes	10305
Elements	9977

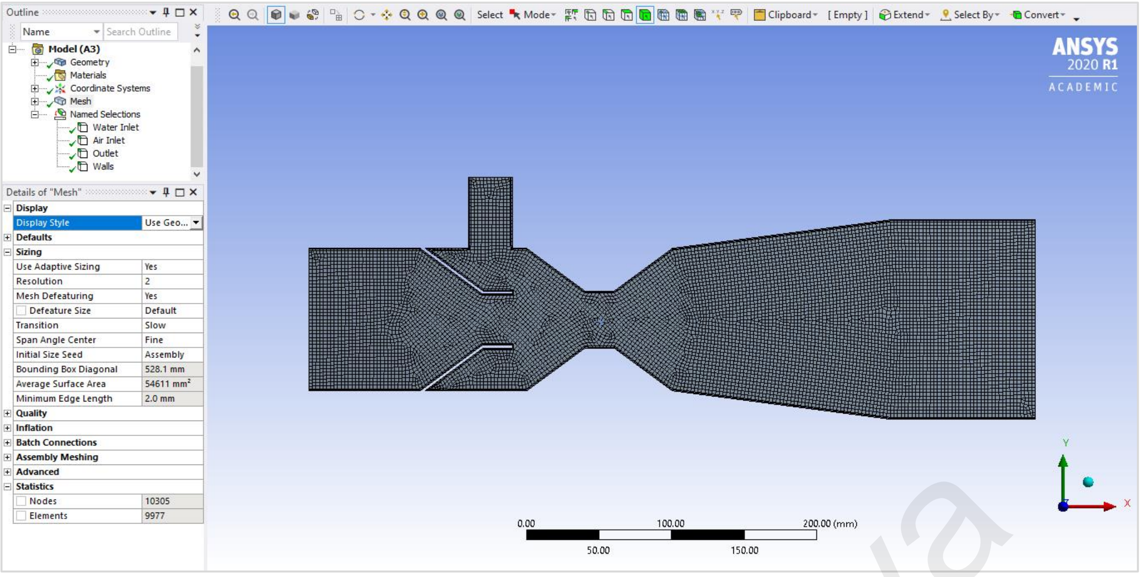

Figure 3.2 Model 1 with Meshing

Boundaries for water inlet, air inlet, walls and outlets has been named for later simulation boundary conditions settings purposes as shown in Figure 3.3.

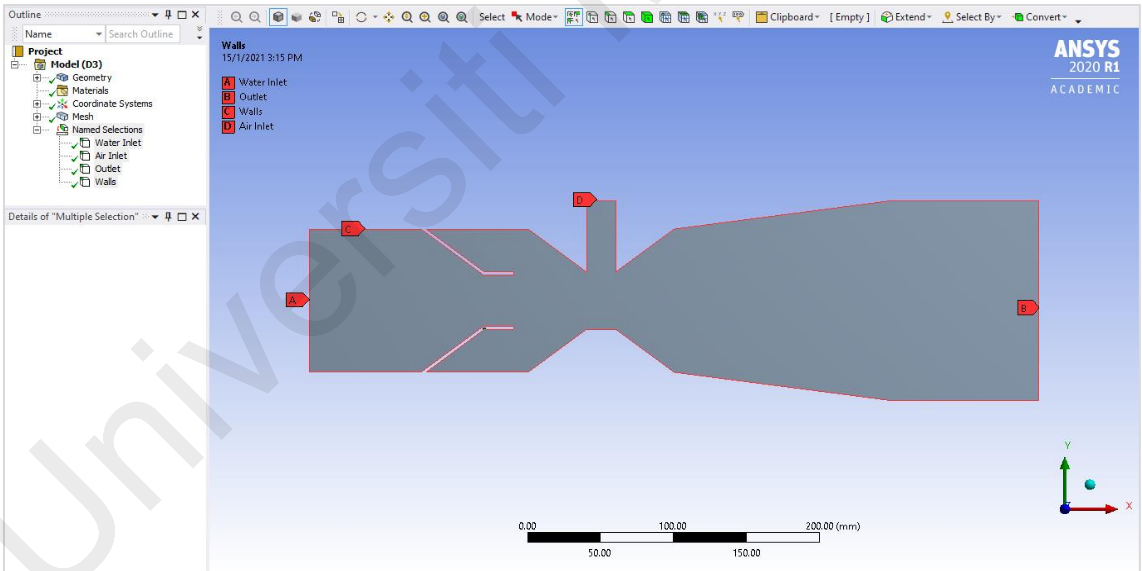

Figure 3.3 Model 1 with Named Boundaries

3.2.2 Model 2 – Improved Design 1

3.2.2.1 2-D Geometry

Improved air ejector (Model 2) design with relocation of the air inlet to throttling section is drafted with the dimensions shown in Figure 3.4.

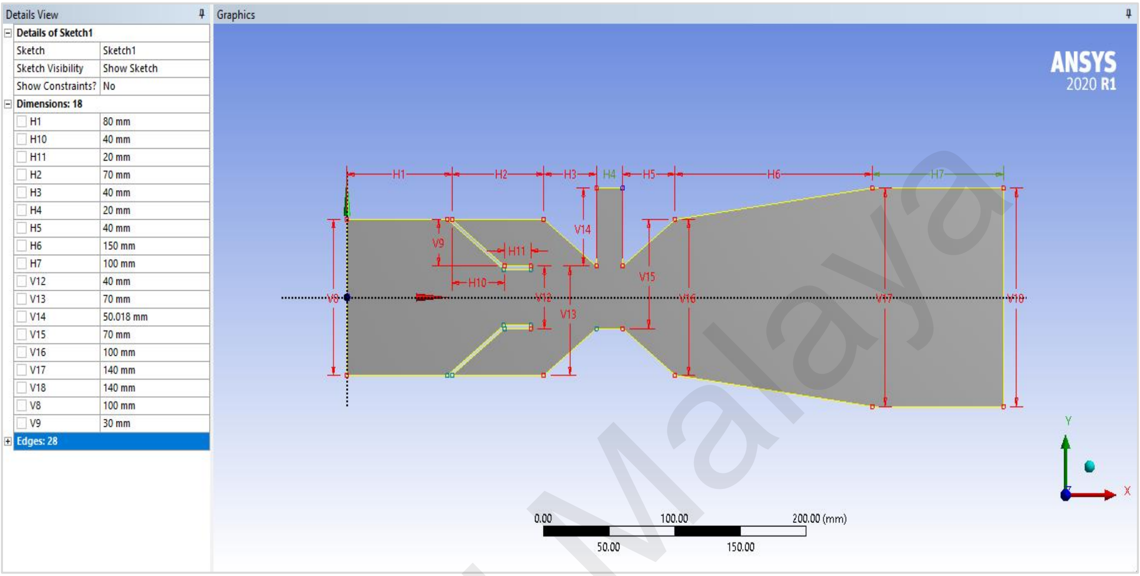

Figure 3.4 Model 2 with Dimensions

3.2.2.2 Meshing

Mesh shown in Figure 3.5 has been created for the earlier modelled improved air ejector (Model 2) design with sizing resolution, element size, inflation growth rate, quantity of nodes and elements shown in Table 3.2.

Table 3-2 Model 2 Meshing Settings and Statistics

Sizing Resolution	2
Element Size	2.5 mm
Inflation Growth Rate	1.1
Nodes	10259
Elements	9936

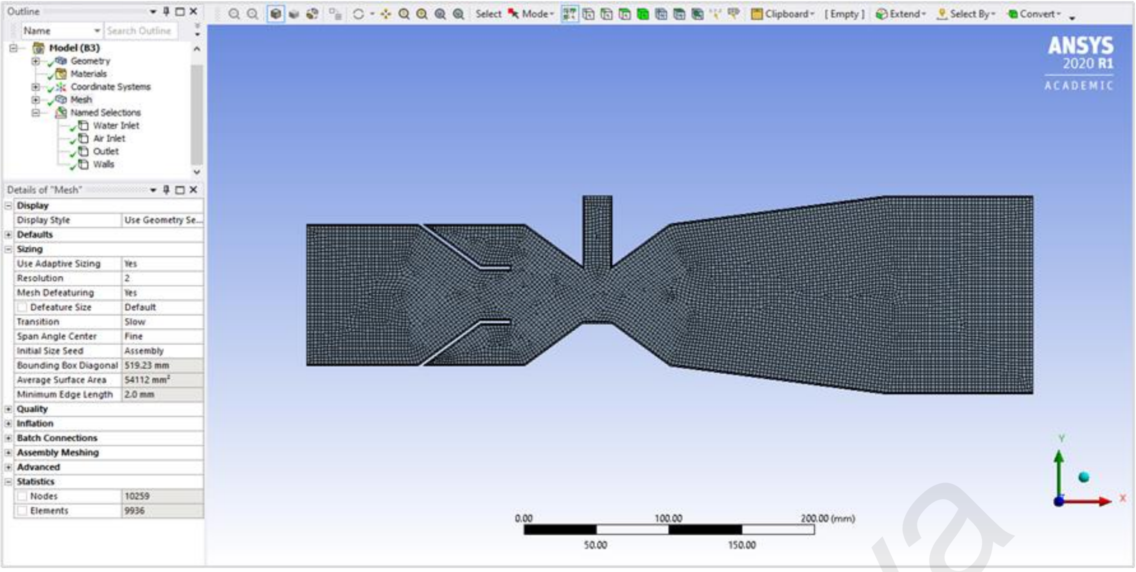

Figure 3.5 Model 2 with Meshing

Boundaries for water inlet, air inlet, walls and outlets has been named for later simulation boundary conditions settings purposes as shown in Figure 3.6.

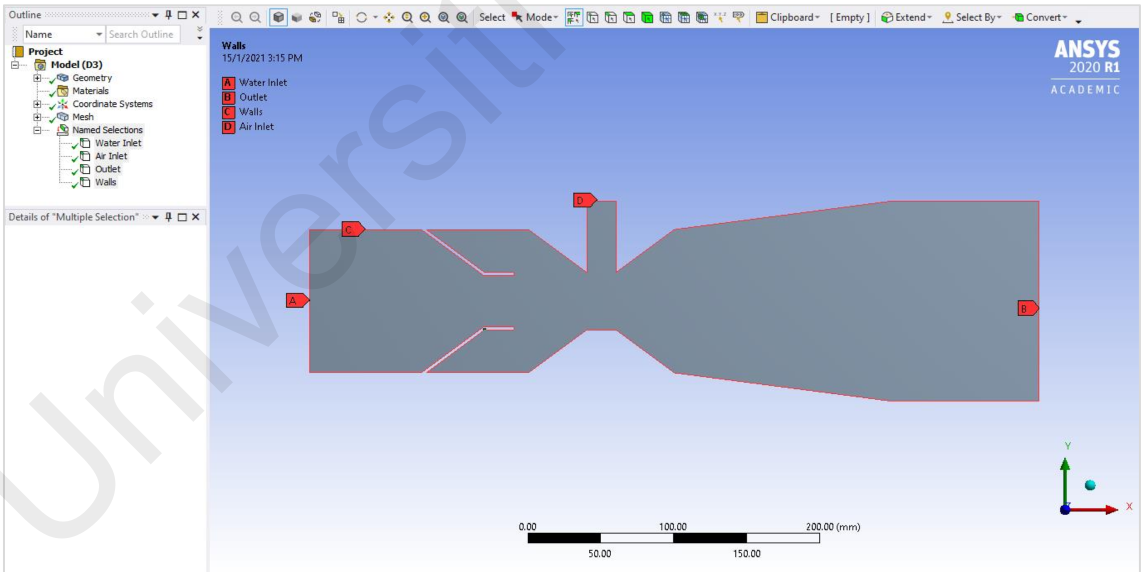

Figure 3.6 Model 2 with Named Boundaries

3.2.3 Model 3 – Improved Design 2

3.2.3.1 2-D Geometry

Improved air ejector (Model 3) design with removal of the water inlet reducer at the converging or contracting section with the dimensions shown in Figure 3.7.

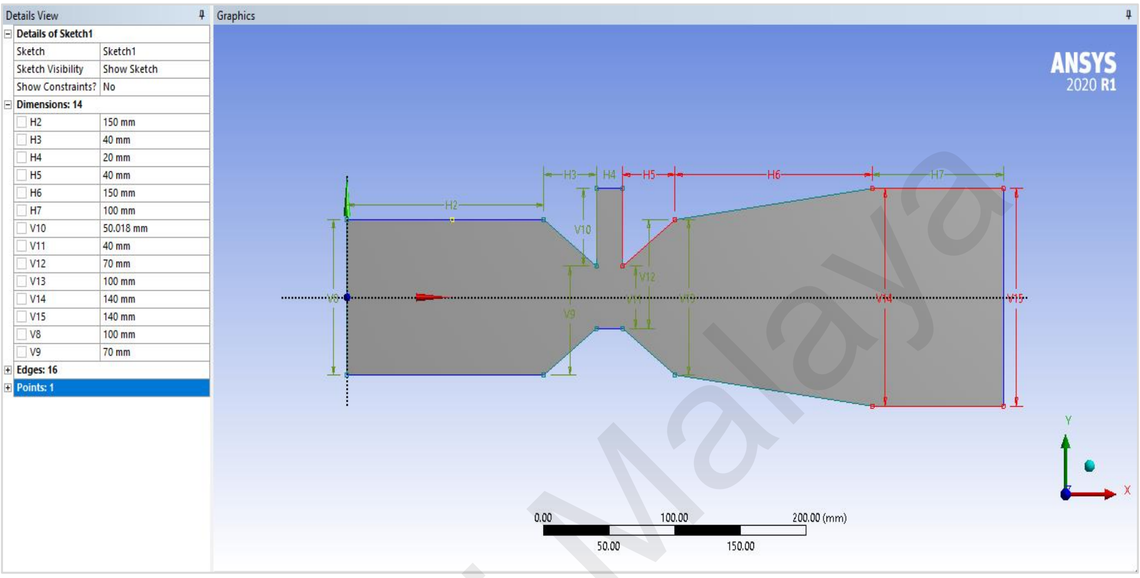

Figure 3.7 Model 2 with Dimensions

3.2.3.2 Meshing

Mesh shown in Figure 3.8 has been created for the earlier modelled improved air ejector (Model 3) design with sizing resolution, element size, inflation growth rate, quantity of nodes and elements shown in Table 3.3.

Table 3-3 Model 3 Meshing Settings and Statistics

Sizing Resolution	2
Element Size	2.5 mm
Inflation Growth Rate	1.1
Nodes	39044
Elements	38504

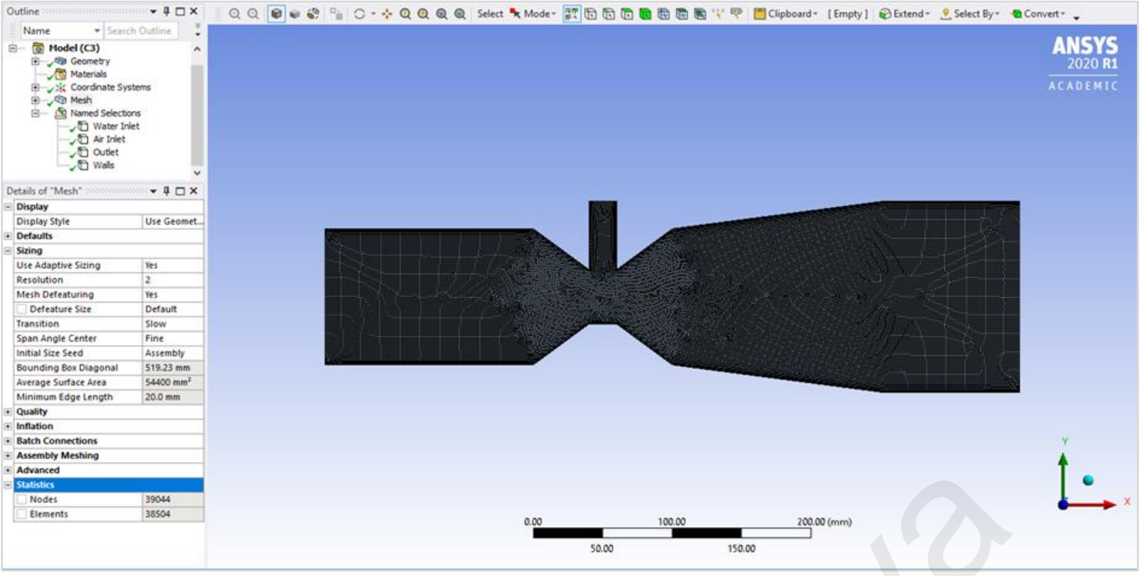

Figure 3.8 Model 3 with Meshing

Boundaries for water inlet, air inlet, walls and outlets has been named for later simulation boundary conditions settings purposes as shown in Figure 3.6.

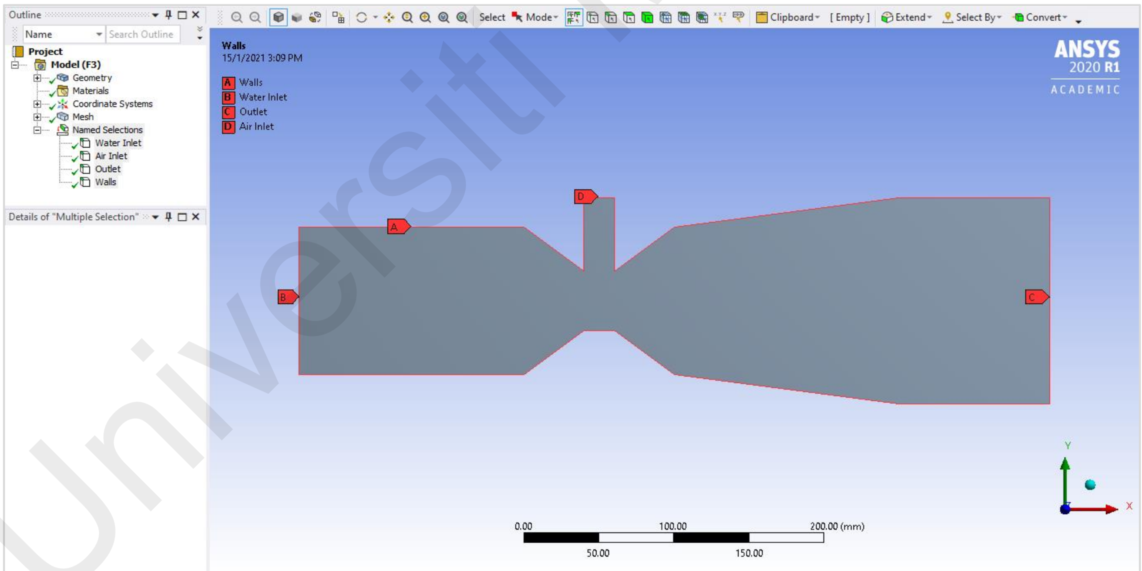

Figure 3.9 Model 3 with Named Boundaries

3.2.4 Model 4– Improved Design 3

3.2.4.1 2-D Geometry

Improved air ejector (Model 4) design with extension the length of the throttling section with air inlet with the dimensions shown in Figure 3.10.

Figure 3.10 Model 3 with Dimensions

3.2.4.2 Meshing

Mesh shown in Figure 3.11 has been created for the earlier modelled improved air ejector (Model 3) design with sizing resolution, element size, inflation growth rate, quantity of nodes and elements shown in Table 3.4.

Table 3-4 Model 4 Meshing Settings and Statistics

Sizing Resolution	2
Element Size	2.5 mm
Inflation Growth Rate	1.1
Nodes	40225
Elements	39654

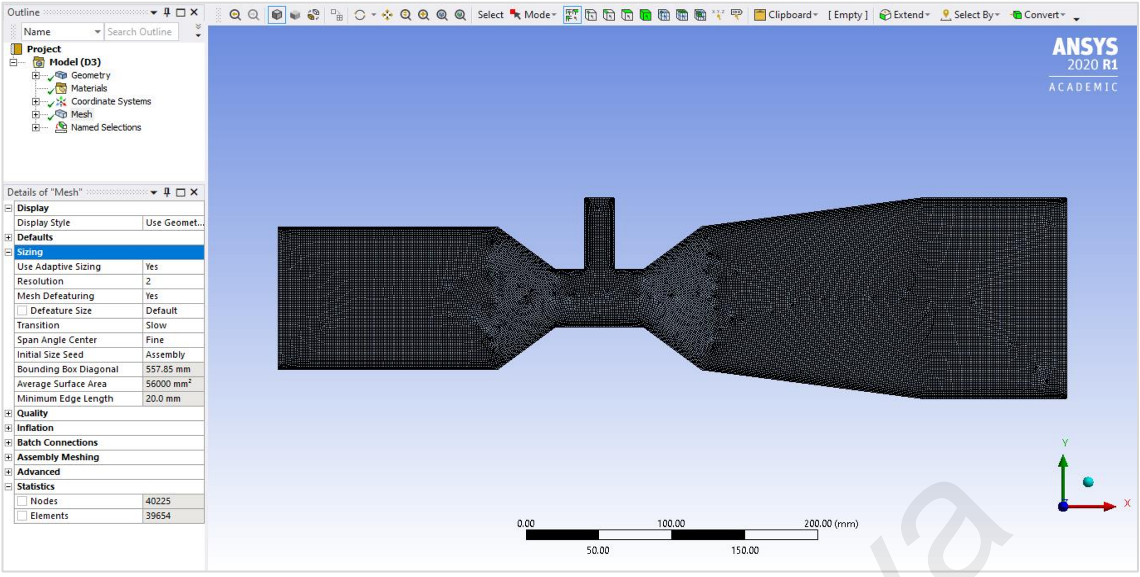

Figure 3.11 Model 4 with Meshing

Boundaries for water inlet, air inlet, walls and outlets has been named for later simulation boundary conditions settings purposes as shown in Figure 3.6.

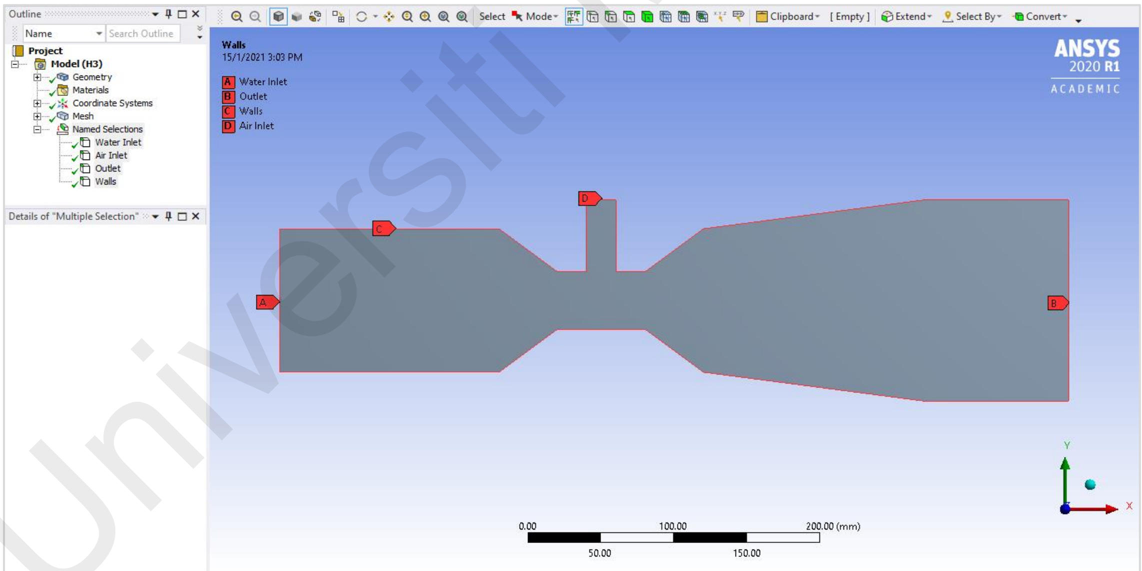

Figure 3.12 Model 4 with Named Boundaries

3.2.5 Model 5– Improved Design 4

3.2.5.1 2-D Geometry

Improved air ejector (Model 5) design with adding a flow straightener at the diverging or diffusing section immediately after the throttling section with the dimensions shown in Figure 3.10.

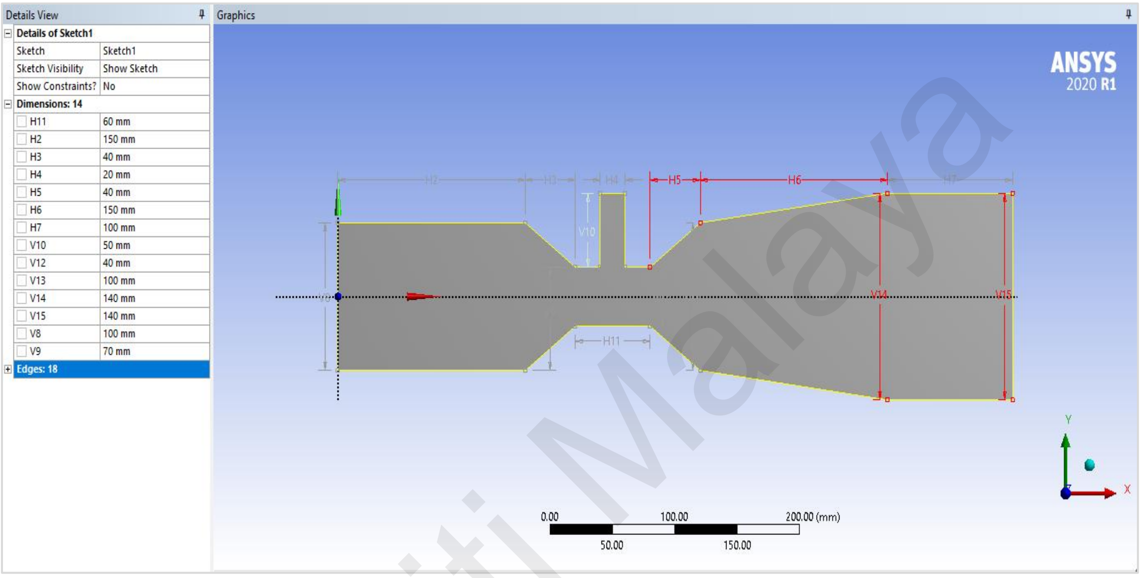

Figure 3.13 Model 5 with Dimensions

3.2.5.2 Meshing

Mesh shown in Figure 3.15 has been created for the earlier modelled improved air ejector (Model 3) design with sizing resolution, element size, inflation growth rate, quantity of nodes and elements shown in Table 3.5.

Table 3-5 Model 5 Meshing Settings and Statistics

Sizing Resolution	2
Element Size	2.5 mm
Inflation Growth Rate	1.1
Nodes	11774
Elements	11048

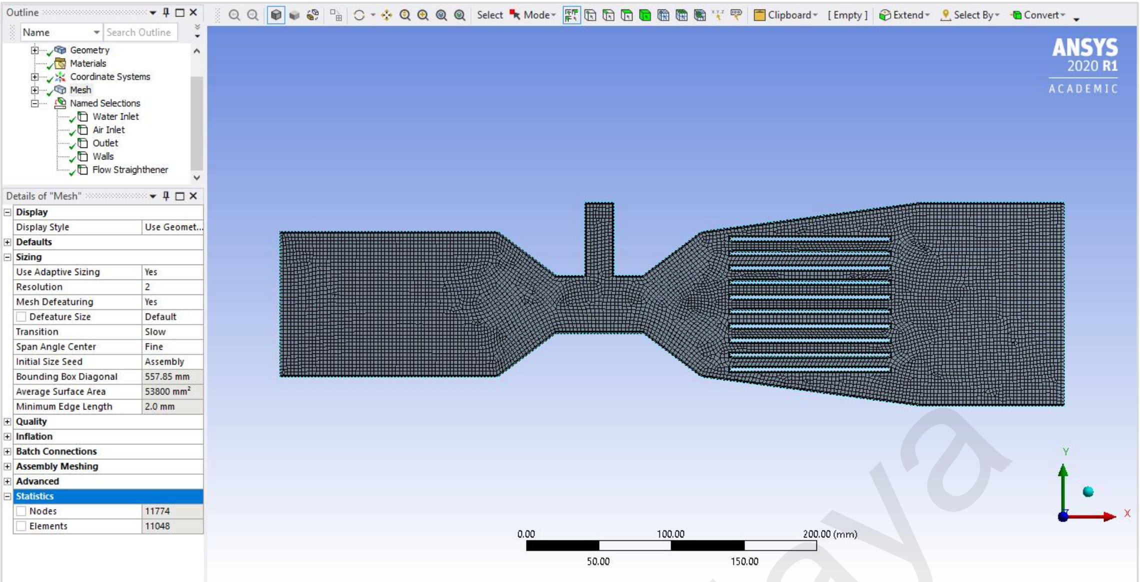

Figure 3.14 Model 5 with Meshing

Boundaries for water inlet, air inlet, walls, flow straightener and outlets has been named for later simulation boundary conditions settings purposes as shown in Figure 3.6.

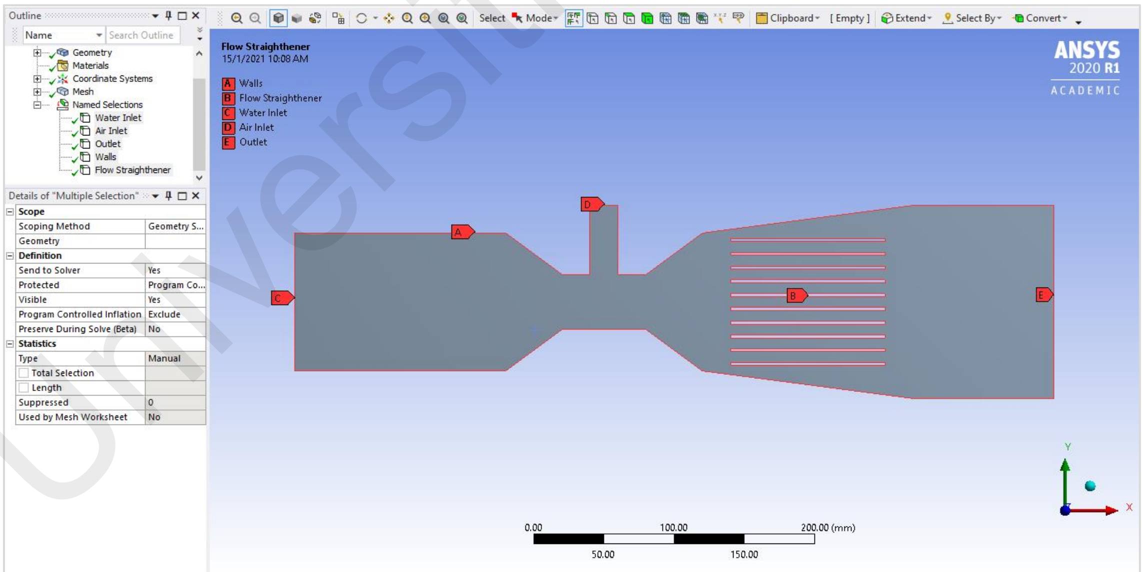

Figure 3.15 Model 5 with Named Boundaries

3.3 Setup and Simulation

Earlier model 2-D geometry and meshing will be set according to Table 3-6 settings to setup, initiate, run the calculation and subsequently simulate the flow pressure, velocity, and air fraction. The phase is crucial and important that will determine the accuracy and relevancy of the simulation outcomes, as well as the time and power consumption to complete the simulation.

Table 3-6 Simulation Setup Settings

Methods	
Models	Multiphase (On), Viscous (Laminar)
Multiphase Model	
Homogeneous Model	Mixture
No. of Eulerian Phases	2
Mixture Parameters	Slip Velocity
Phases	Primary - Water Secondary - Air
Materials	Fluid (Air & Water-Liquid)
Density	Air (1.225 kg/m^3) Water (998.2 kg/m^3)
Viscosity	Air (1.7894×10^{-5} kg/ms) Water (1.003×10^3 kg/ms)
Boundary Conditions	
Air Inlet	Water: 0, Air: 1; v = 3.5368 m/s; p = 101,325 Pa
Water Inlet	Air: 0, Water: 1; v = 0.3183 m/s
Outlet	Volume Fraction = Air : Water = 0.1 : 0.9

Solution	
Pressure-Velocity Coupling	
Scheme	SIMPLE
Spatial Discretization	
Gradient	Least Squares Cell Based
Pressure	PRESTO!
Momentum	First Order Upwind
Volume Fraction	First Order Upwind
Transient Fraction	First Order Implicit
Solution Initialization	
Initialization Methods	Standard Initialization
Compute from	Water Inlet
Reference Frame	Relative to Cell Zone
Initial Values	X-Velocity: 0.3183 m/s; Air Volume Fraction: 0
Run Calculation	
Time Advancement	
Type	Fixed
Method	User-Specified
Parameters	
Number of Time Steps	1000
Time Step Size	0.01
Max. Iteration/Time Step	25
Reporting Interval	1

The earlier constructed 2-D model will be defined as a multiphase model, with water as the primary phase flow and air as the secondary phase flow with the density of 998.2 kg/m^3 and 1.225 kg/m^3; and viscosity of 1.003×10^3 kg/ms and 1.7894×10^{-5} kg/ms respectively. The water inlet velocity of 0.3183 m/s is defined based on the flow rate of 150 liters/minute over the inlet diameter of 100mm (Velocity, V = Flow, Q ÷ Cross-Sectional Area, A). Based on the water inlet velocity over the throttling section, a low-pressure region is developing and causing the atmospheric pressure at 101,325 Pascal pushing the air into the system over the air inlet diameter of 30mm at the air inlet velocity of 3.5368 m/s.

The simulation is using the SIMPLE pressure-velocity coupling scheme for the generating solution with spatial discretization PRESTO and Least Squares Cell Based for pressure and gradient settings respectively. The calculation will be initiated from the water inlet with the air volume fraction of zero based on standard initialization method. The convergence criteria are set based on 1000 number of time steps, based on the time step size of 0.01 at maximum 25 iteration/time step with reporting interval of 1, to ensure simulation result is generated based on the optimum capability (software and computing power limitation) and resources (time and power consumption).

3.4 Evaluation and Improvement

The simulation results will be evaluated based on the flow distribution on velocity, pressure and water-air volume fraction contour. A new design will be improved with the proposed piping arrangements based on the earlier model results, to ensure better mixing quality ultimately. All new designs with improvements will be undergone the first, second and third phase of the research design repeatedly and thoroughly, until a satisfactory model is achieved. This research project limits to only four (4) improvements will be made based on the initial model and total five (5) models will be simulated and analyzed accordingly.

3.5 Data Analysis

After running various simulation for all models at the specified setup settings, all results will be plotted into flow contours for velocity and pressure; and water-air volume fraction for analyses and comparison. It is a crucial and important process to diagnose the hydraulic conditions in order to make conclusion for further design references, as well as recommendations for future research purposes.

CHAPTER 4: RESULT AND DISCUSSION

4.1 Model 1 Simulation

4.1.1 Results

All simulation results on Model 1 will be plotted as following: - (Figure 4.1) Velocity Contour; (Figure 4.2) Velocity Volume Rendering; (Figure 4.3) Pressure Contour; (Figure 4.4) Pressure Volume Rendering; (Figure 4.5) Air Volume Fraction; and (Figure 4.6) Water Volume Fraction.

4.1.1.1 Velocity

Figure 4.1 and 4.2 show that water as the primary phase fluid flowing from the water inlet through the pipe at flowrate of 150 liter per minute, passing through the conical reducer towards the throttling section, creating lower pressure as compared to the atmospheric pressure and drawing air as the secondary phase fluid from the air inlet at a higher velocity due to smaller air inlet diameter.

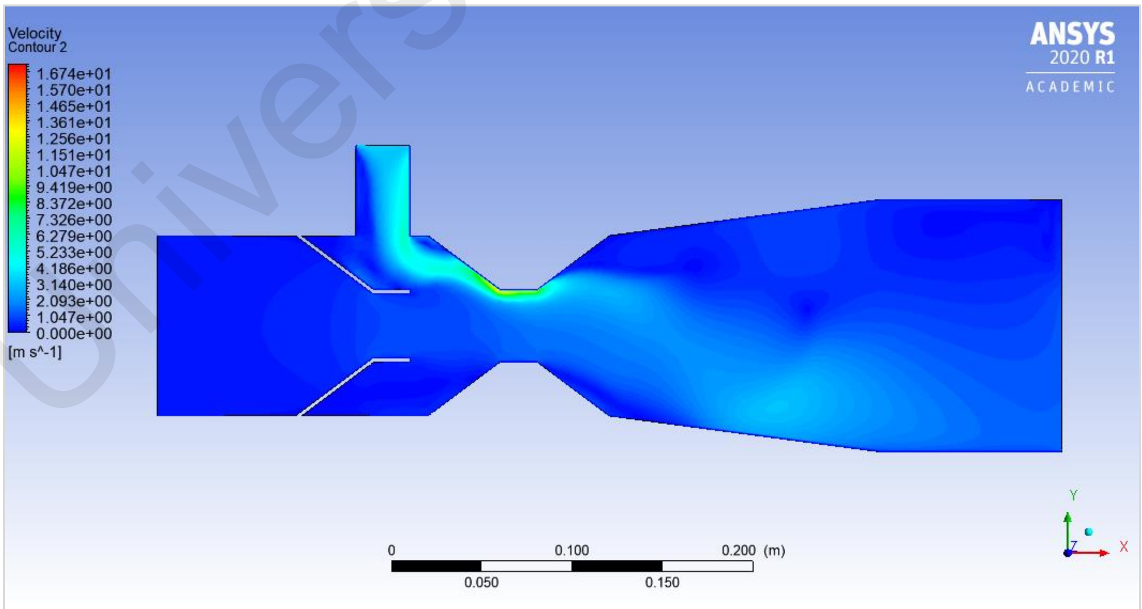

Figure 4.1 Model 1 Velocity Contour

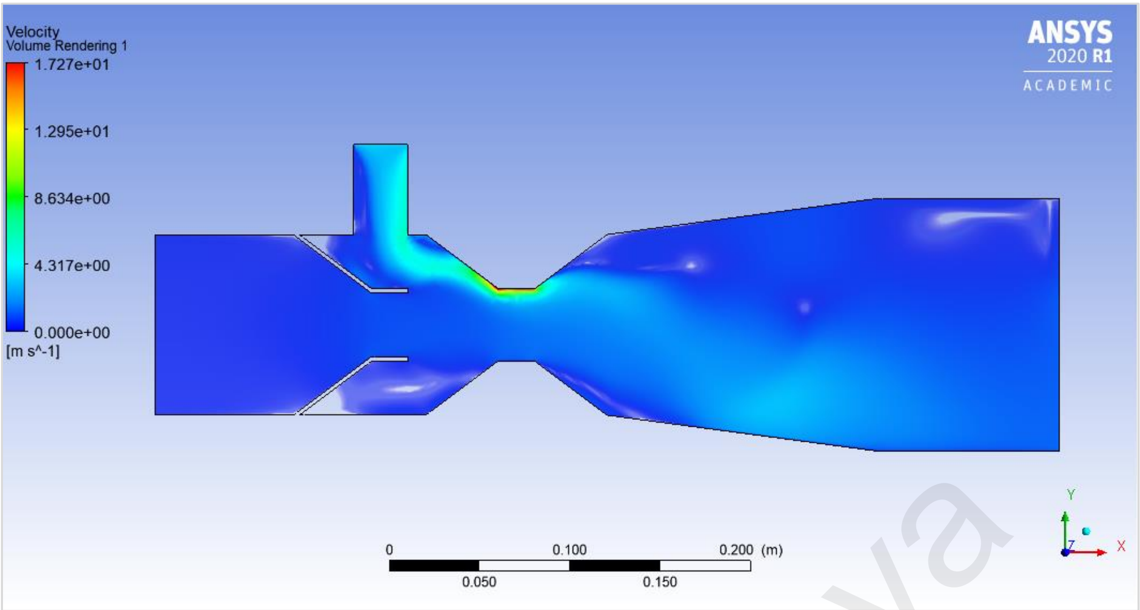

Figure 4.2 Model 1 Velocity Volume Rendering

4.1.1.2 Pressure

Figure 4.3 and 4.4 show that water flowing from the water inlet at a bigger diameter pipe passing through the conical reducer building higher pressure at the bottom part of the pipe due to intrusion of air inflow from the air inlet at a higher velocity; and lower pressure is observed after the narrow throttling section connecting to a bigger diameter diffusing section, creating lower pressure at the downstream.

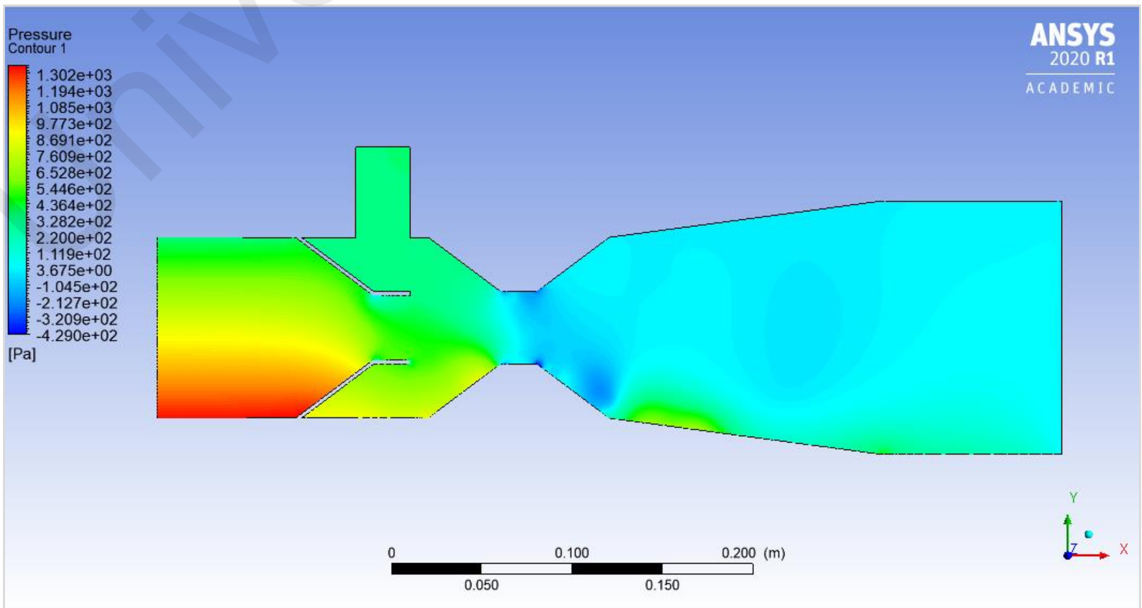

Figure 4.3 Model 1 Pressure Contour

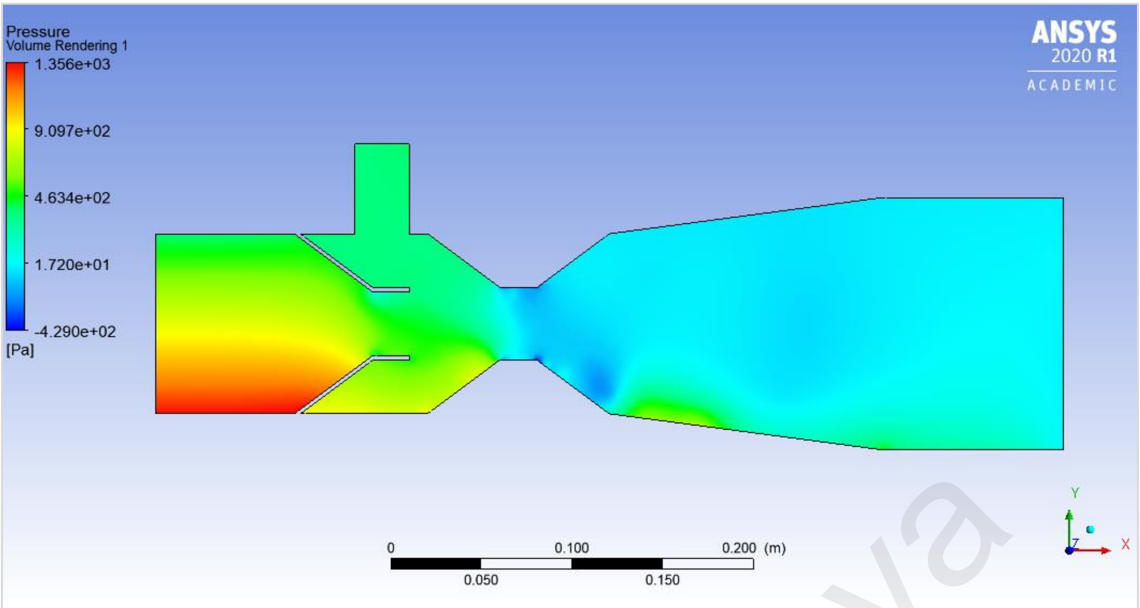

Figure 4.4 Model 1 Pressure Volume Rendering

4.1.1.3 Volume Fraction

Figure 4.5 and 4.6 show that the water (primary phase flow) fraction is observed to be higher at the upstream and air (secondary phase flow) is observed to be at the higher fraction at the downstream, mainly due the conical reducer limited the lower velocity water from flowing through the throttling; and caused the lower pressure at the diffusing section drew more air in at higher velocity subsequently.

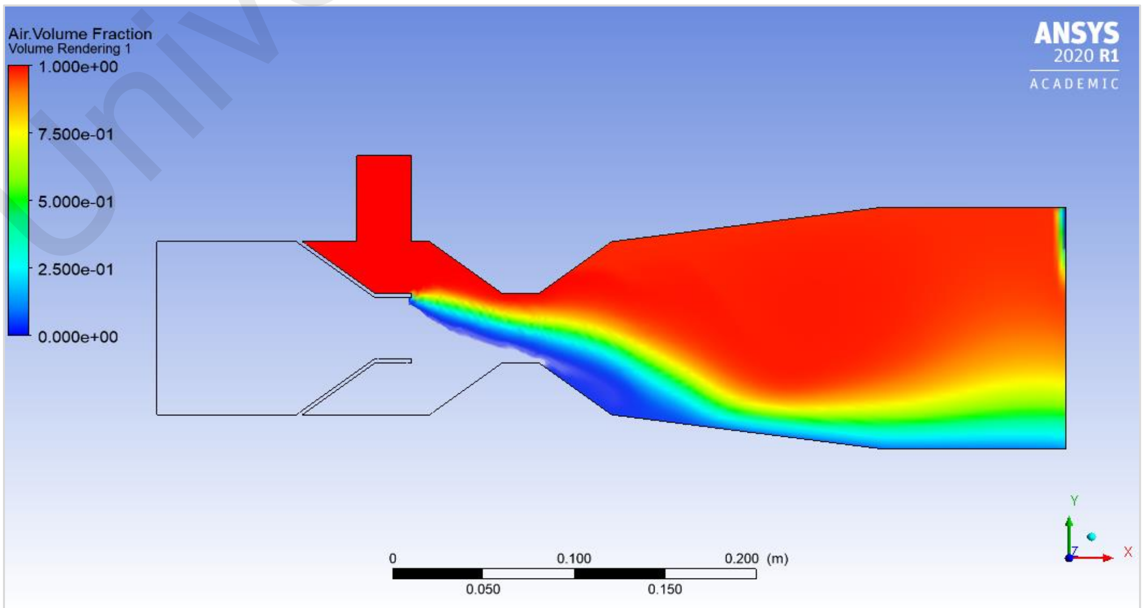

Figure 4.5 Model 1 Air Volume Fraction

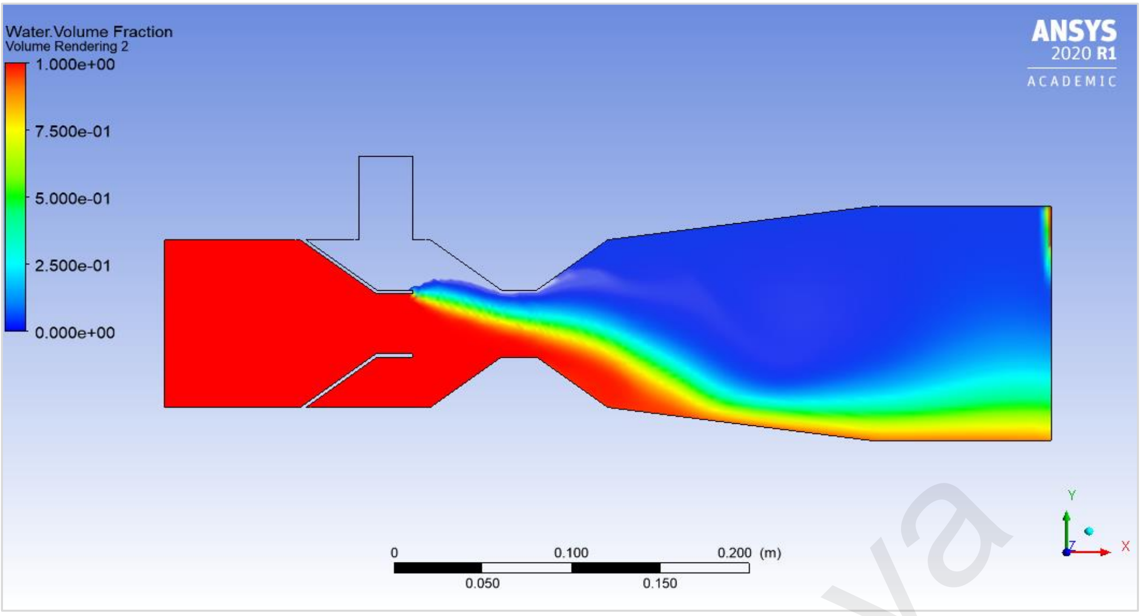

Figure 4.6 Model 1 Water Volume Fraction

4.1.2 Discussion

From the observation of all Model 1 simulations, the air from the inlet seems is draw in directly into the throttling section at higher velocity blocking the water flowing through at a lower velocity. Recommendation to relocate the air inlet to the middle of the throttling section at a higher velocity and condition better mixing at the diffusing section.

4.2 Model 2 Simulation

4.2.1 Results

All simulation results on Model 2 will be plotted as following: - (Figure 4.7) Velocity Contour; (Figure 4.8) Velocity Volume Rendering; (Figure 4.9) Pressure Contour; (Figure 4.10) Pressure Volume Rendering; (Figure 4.11) Air Volume Fraction; and (Figure 4.12) Water Volume Fraction.

4.2.1.1 Velocity

Figure 4.7 and 4.8 show that water flow from the water inlet converging through the conical reducer towards the throttling section, velocity increases passing through the conical reducer and integrates with the air flow from the air inlet forming higher velocity flow towards the bottom part of the diffusing section.

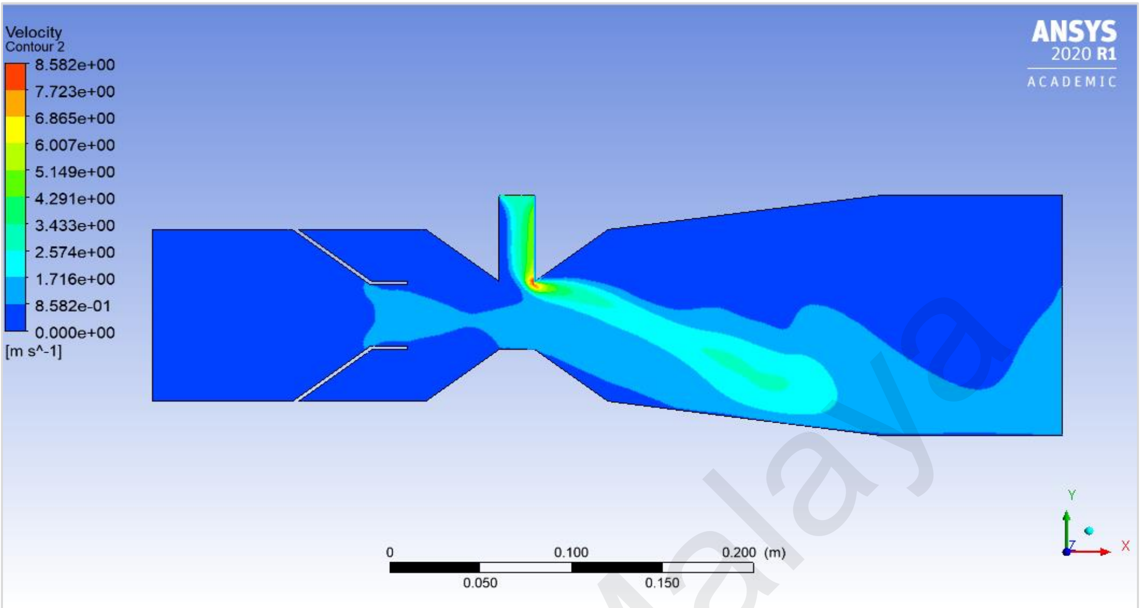

Figure 4.7 Model 2 Velocity Contour

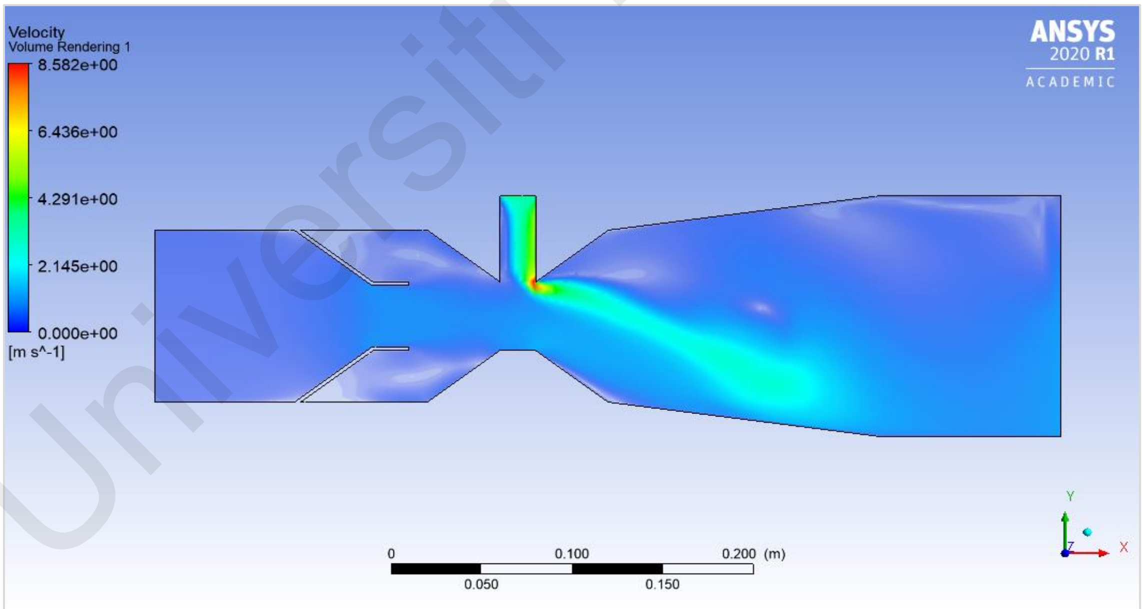

Figure 4.8 Model 2 Velocity Volume Rendering

4.2.1.2 Pressure

Figure 4.9 and 4.10 show that water flow from the water inlet converging through the conical reducer towards the throttling section, pressure is observed to be higher at the bottom part before the conical reducer and top of the converging section after the conical reducer; and pressure gradually reduces through the diffusing part towards the outlet.

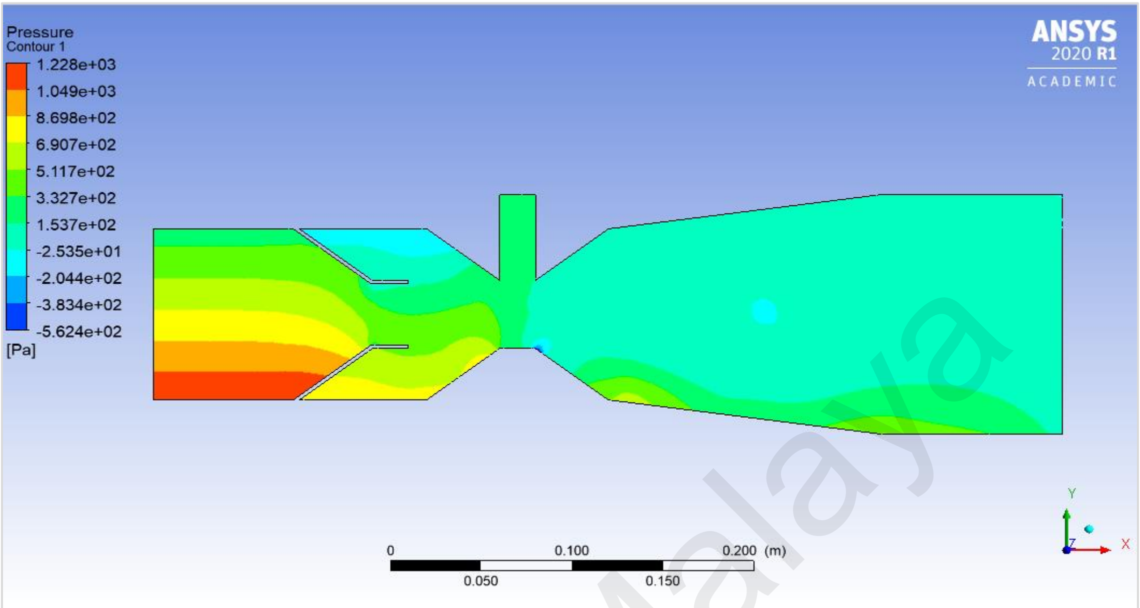

Figure 4.9 Model 2 Velocity Contour

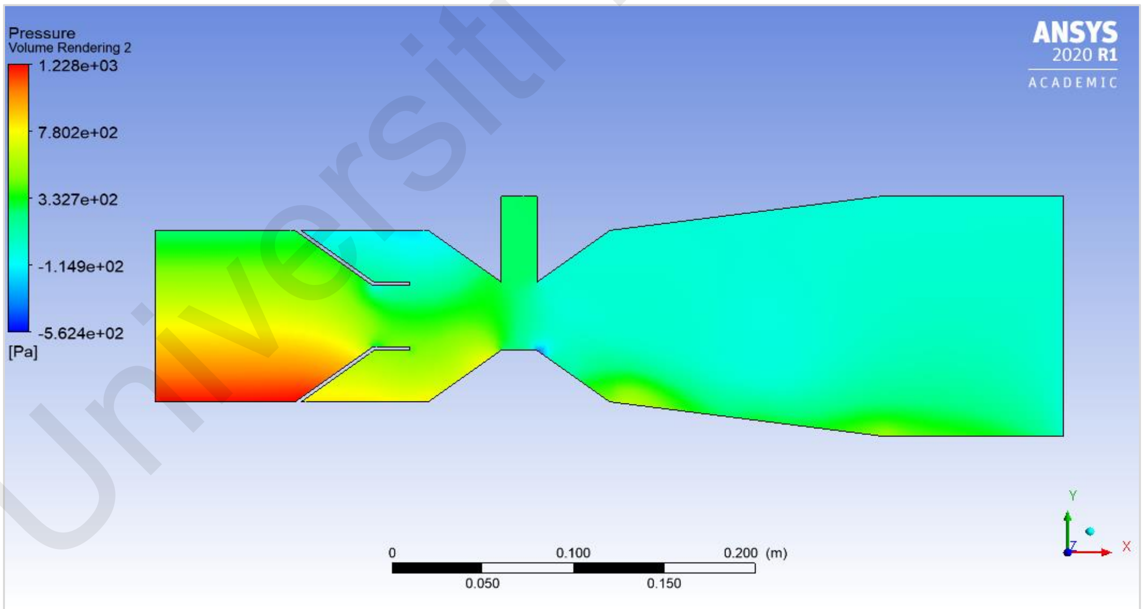

Figure 4.10 Model 2 Velocity Volume Rendering

4.2.1.3 Volume Fraction

Figure 4.11 and 4.12 show that water fraction is observed to be slightly higher at the upstream, due shifting of the air inlet to the throttling section; and air fraction is still observed to be relatively higher as compared to the water fraction at the bottom part of the downstream.

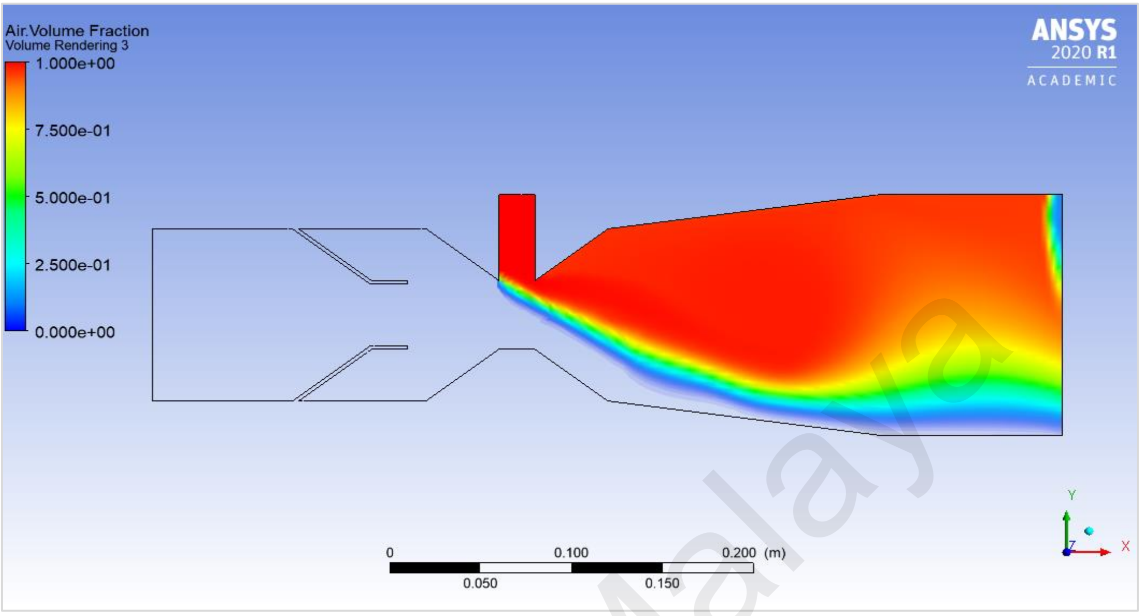

Figure 4.11 Model 2 Air Volume Fraction

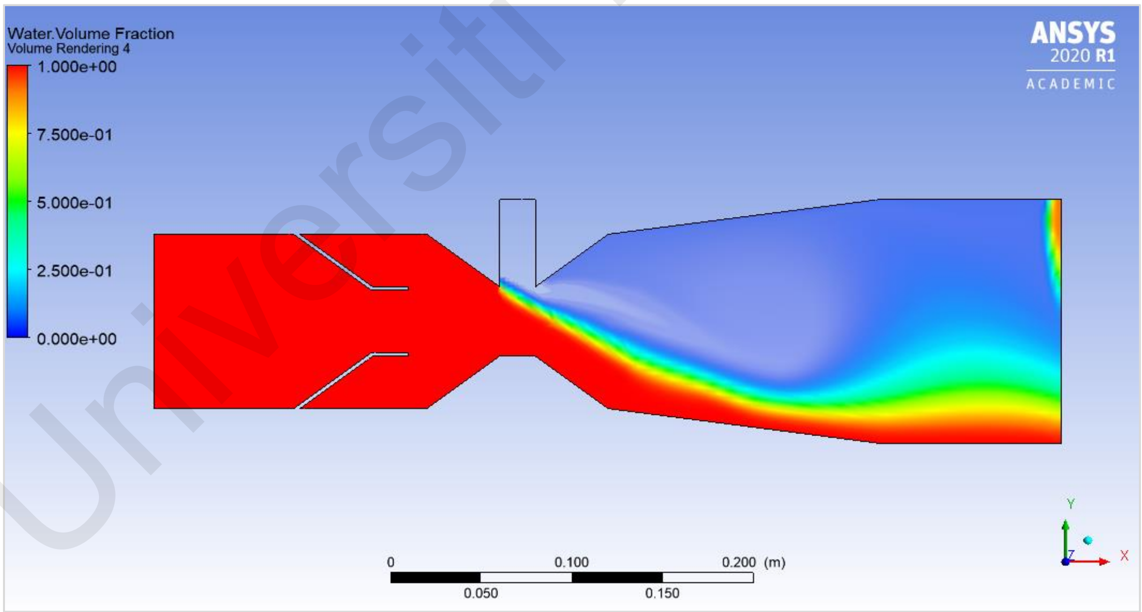

Figure 4.12 Model 2 Water Volume Fraction

4.2.2 Discussion

From the observation of all Model 2 simulations, shifting of the air inlet to the throttling section is obstructing the flow from the water inlet and creating higher back pressure before the conical reducer. Recommendation to remove the conical code to reduce the back pressure built up at the bottom part of the upstream.

4.3 Model 3 Simulation

4.3.1 Results

All simulation results on Model 3 will be plotted as following: - (Figure 4.13) Velocity Contour; (Figure 4.14) Velocity Volume Rendering; (Figure 4.15) Pressure Contour; (Figure 4.16) Pressure Volume Rendering; (Figure 4.17) Air Volume Fraction; and (Figure 4.18) Water Volume Fraction.

4.3.1.1 Velocity

Figure 4.13 and 4.14 show that water flow from the water inlet converging through the towards the throttling section, velocity increases passing through the throttling section and integrates with the higher velocity air flow from air inlet forming flow towards the bottom part of the diffusing section, and homogenize to lower velocity in the later stage.

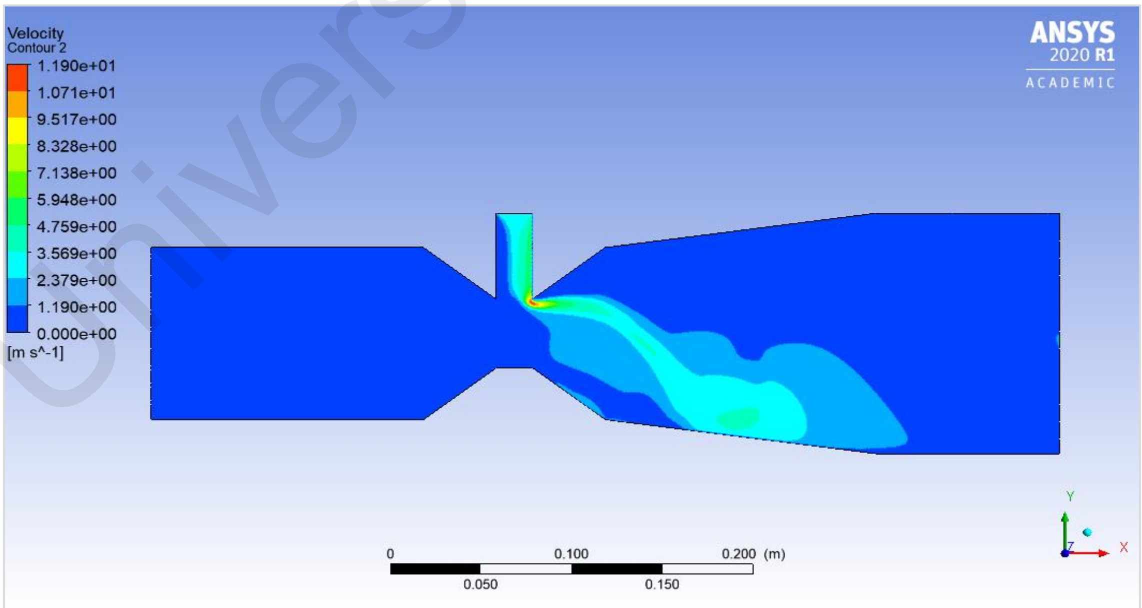

Figure 4.13 Model 3 Velocity Contour

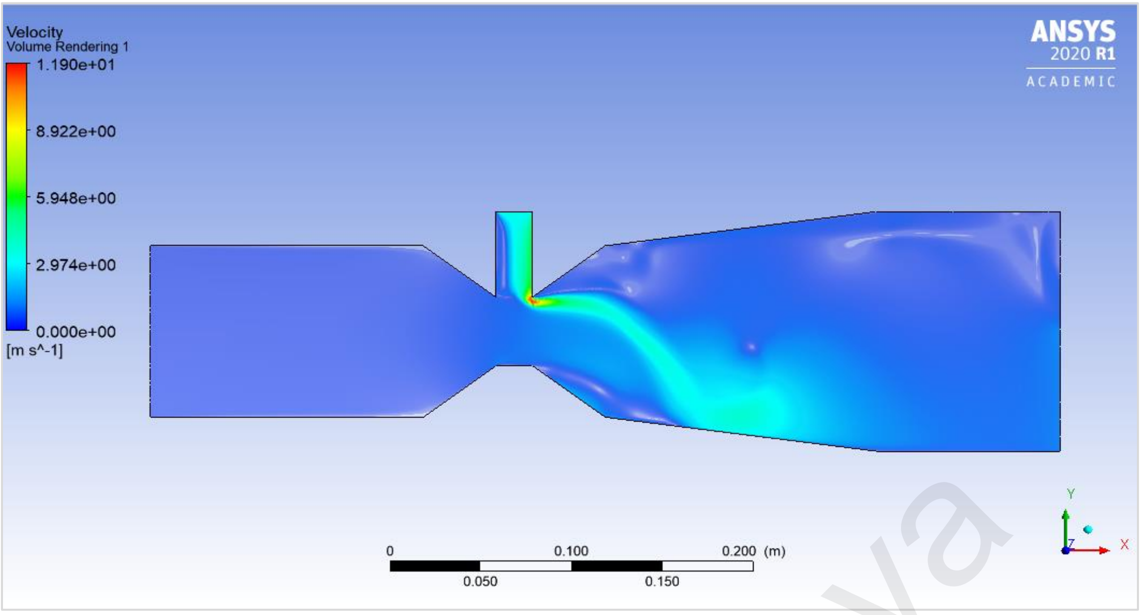

Figure 4.14 Model 3 Velocity Volume Rendering

4.3.1.2 Pressure

Figure 4.15 and 4.16 show that high back pressure caused by the conical reducer is eliminated and pressure is observed averagely throughout all the sections, only a spot of negative-pressure region is observed after the throttling section due to the air flow jetting towards the diffusing section after the sharp edge of the throttling section.

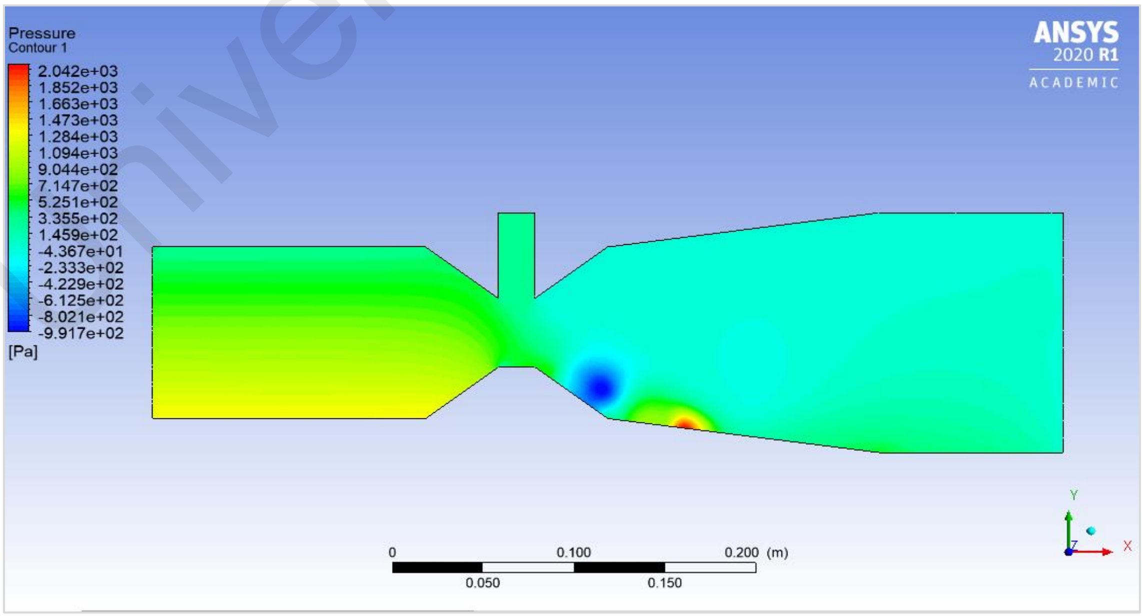

Figure 4.15 Model 3 Pressure Contour

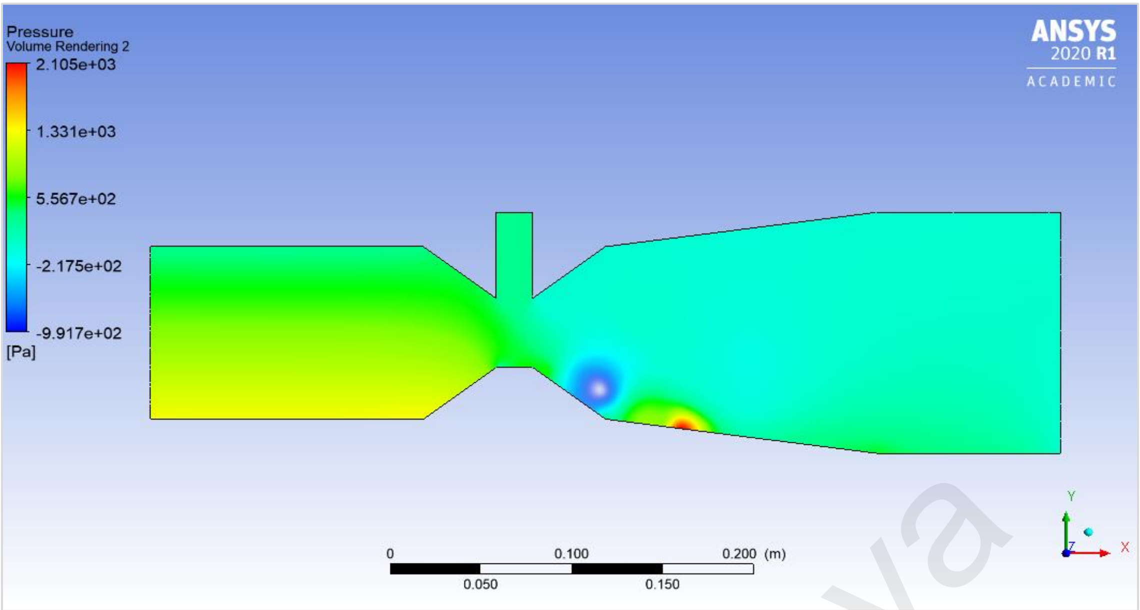

Figure 4.16 Model 3 Pressure Volume Rendering

4.3.1.3 Air-Fraction

Figure 4.17 and 4.18 show that water fraction is observed to be slightly higher at the upstream towards the throttling section with the removal of the conical reducer; and air fraction is still observed to be relatively higher as compared to the water fraction at the bottom part of the downstream.

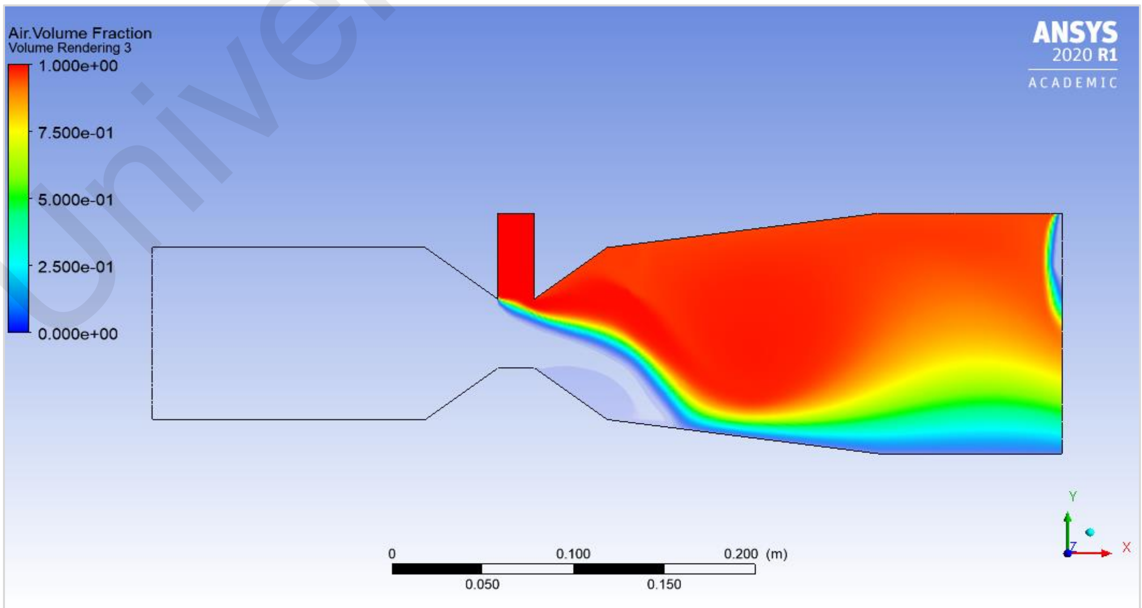

Figure 4.17 Model 3 Air Volume Fraction

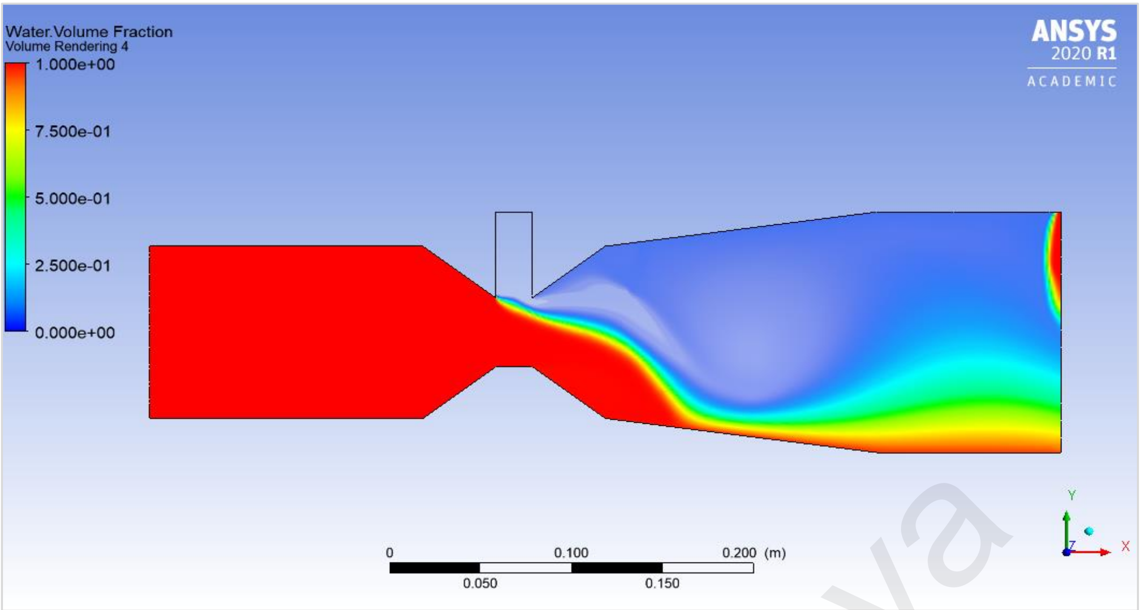

Figure 4.18 Model 3 Water Volume Fraction

4.3.2 Discussion

From the observation of all Model 3 simulations, removing conical reducer is smoothening the flow from the water inlet creating better flow at the upstream and air flow jetting in is slightly obstructing the water flow. Recommendation to lengthen the throttling section to smoothen the air flow integrating with the water flow.

4.4 Model 4 Simulation

4.4.1 Results

All simulation results on Model 4 will be plotted as following: - (Figure 4.19) Velocity Contour; (Figure 4.20) Velocity Volume Rendering; (Figure 4.21) Pressure Contour; (Figure 4.22) Pressure Volume Rendering; (Figure 4.23) Air Volume Fraction; and (Figure 4.24) Water Volume Fraction.

4.4.1.1 Velocity

Figure 4.19 and 4.20 show that water flow from the water inlet converging through the towards the throttling section, velocity is observed to be homogenize along the converging section to the throttling section and the air flow jetting in at higher velocity is gradually reduced with the lengthen throttling section, towards the diffusing section.

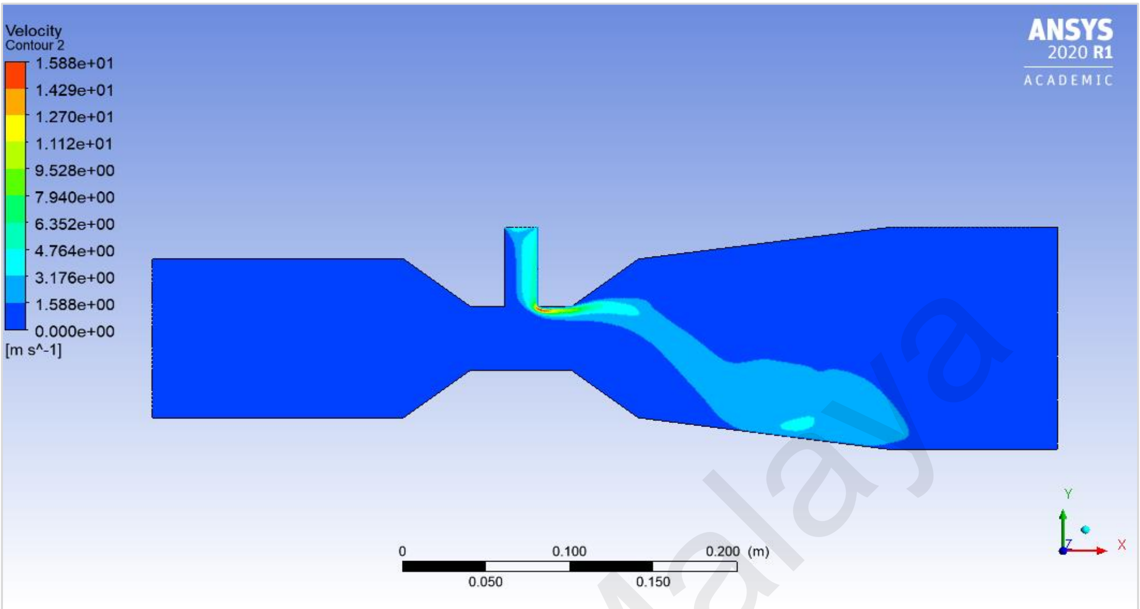

Figure 4.19 Model 4 Velocity Contour

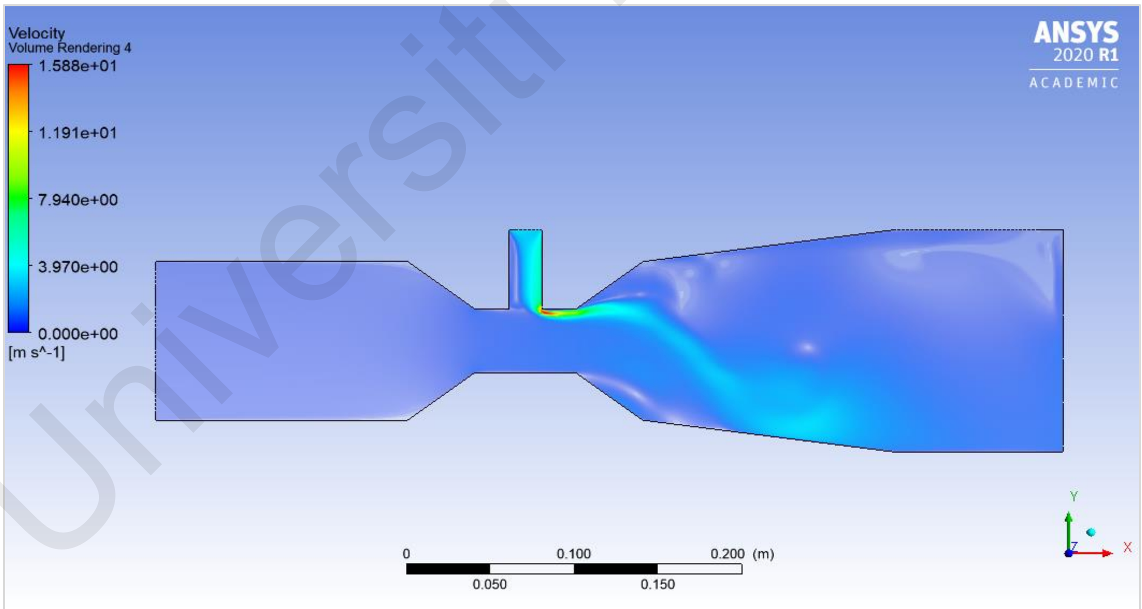

Figure 4.20 Model 4 Velocity Volume Rendering

4.4.1.2 Pressure

Figure 4.21 and 4.22 show that higher pressure build-up at the bottom part of the converging section after lengthening the throttling section. Spots of uneven high and low pressure are observed at the diffusing section after the throttling section, due to the higher air flow velocity from the throttling section towards the diffusing section and the negative-pressure region is still persist.

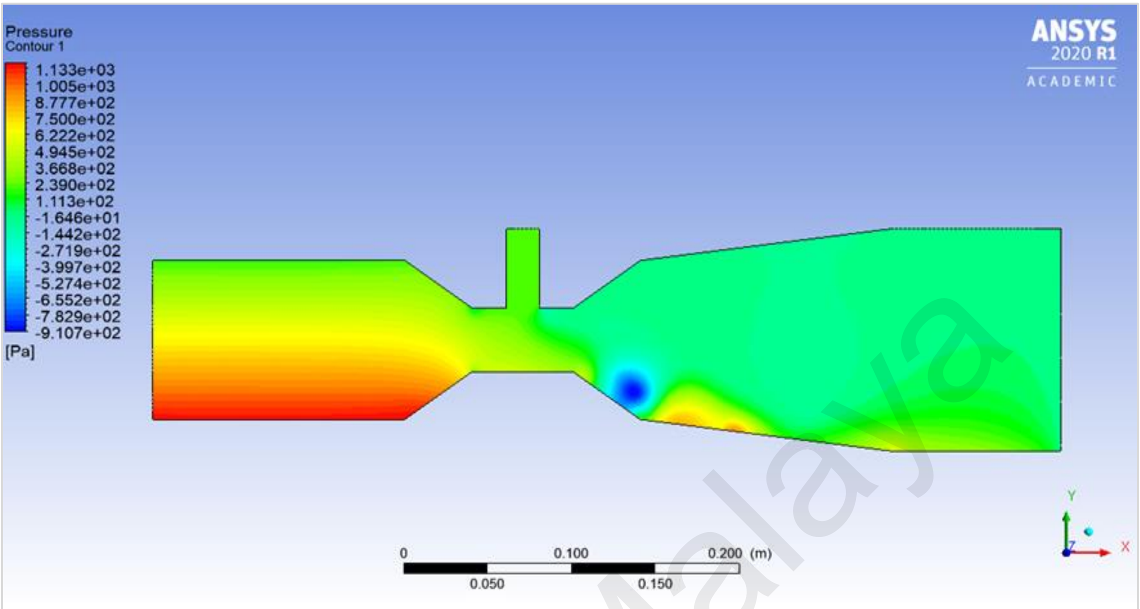

Figure 4.21 Model 4 Pressure Contour

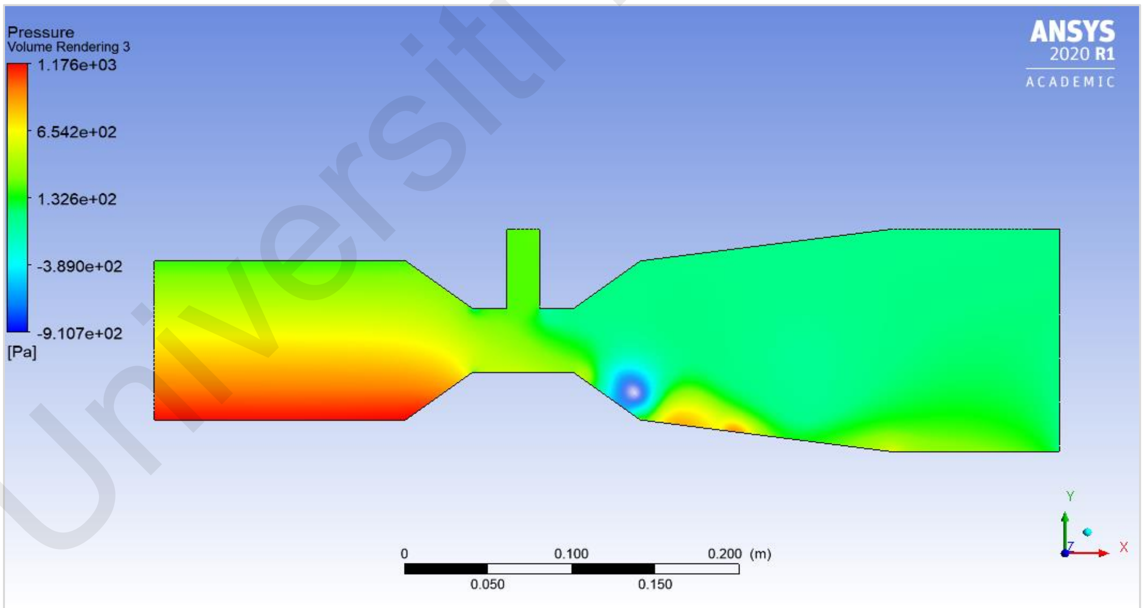

Figure 4.22 Model 4 Pressure Volume Rendering

4.4.1.3 Volume Fraction

Figure 4.23 and 4.24 show that water-air volume fraction is observed quite similar to the earlier model with water fraction slightly pushing the air fraction towards the diffusing section, maintaining the air fraction relatively higher at the top part, as compared to the water fraction at the bottom part of the downstream.

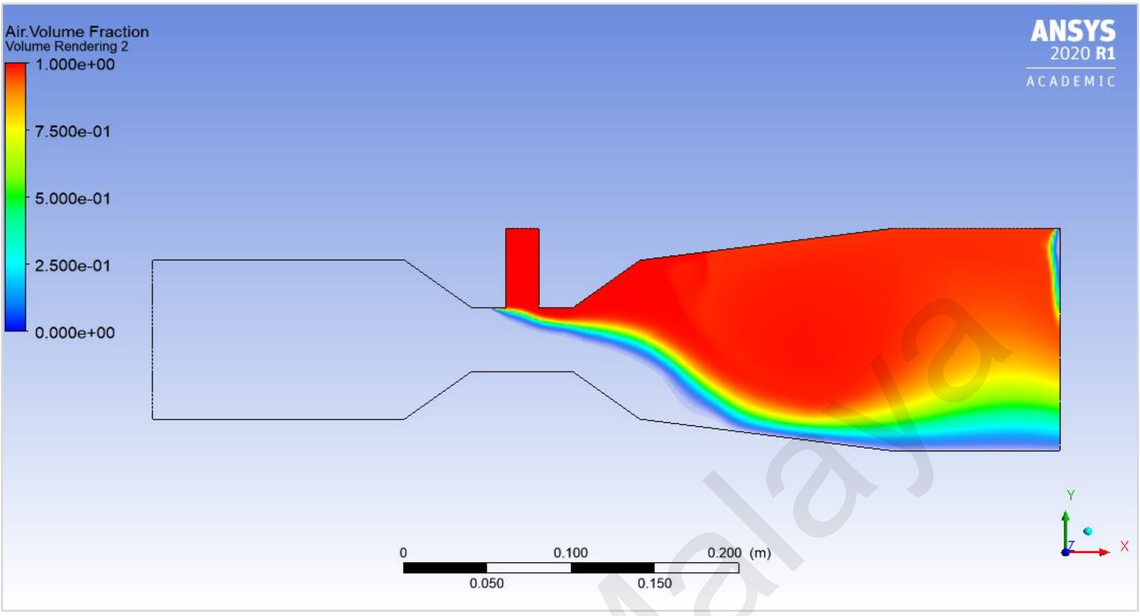

Figure 4.23 Model 4 Air Volume Fraction

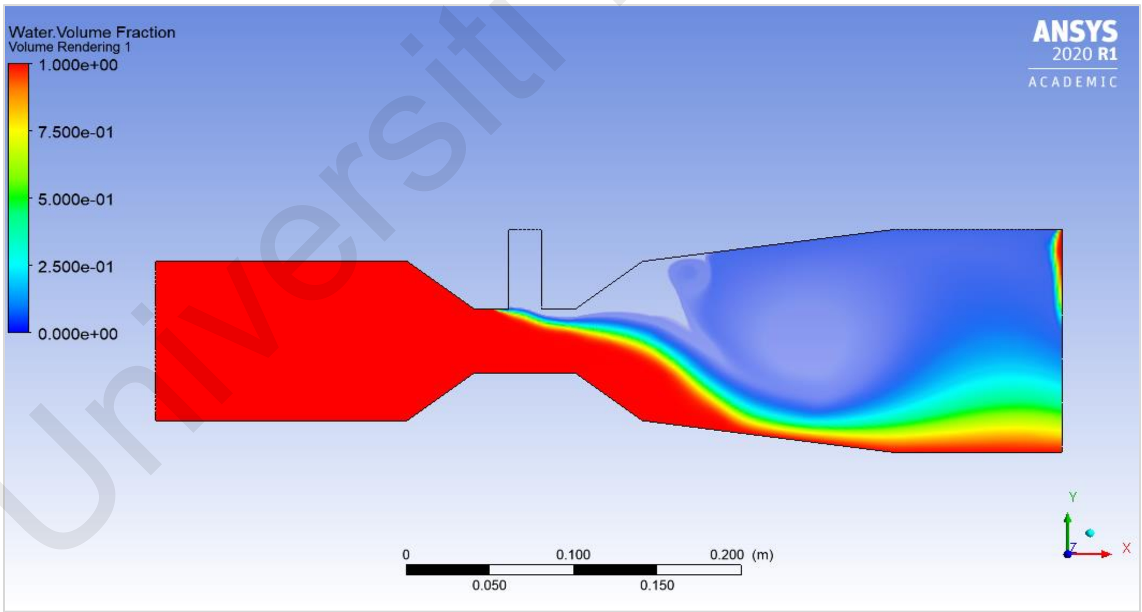

Figure 4.24 Model 4 Water Volume Fraction

4.4.2 Discussion

From the observation of all Model 4 simulations, lengthening throttling section slightly smoothening the flow, but the air flow jetting in effect is still persists obstructing the water flow. Recommendation to add flow straightener across the diffusing section to smoothen the flow in the downstream and improved the upstream flow subsequently.

4.5 Model 5 Simulation

4.5.1 Results

All simulation results on Model 5 will be plotted as following: - (Figure 4.25) Velocity Contour; (Figure 4.26) Velocity Volume Rendering; (Figure 4.27) Pressure Contour; (Figure 4.28) Pressure Volume Rendering; (Figure 4.29) Air Volume Fraction; and (Figure 4.30) Water Volume Fraction.

4.5.1.1 Velocity

Figure 4.25 and 4.26 show that water flow from the water inlet converging through the towards the throttling section, the velocity is observed to be homogenized along the converging section to the throttling section and the air flow is observed to be at higher towards the diffusing section with the addition of flow straightener.

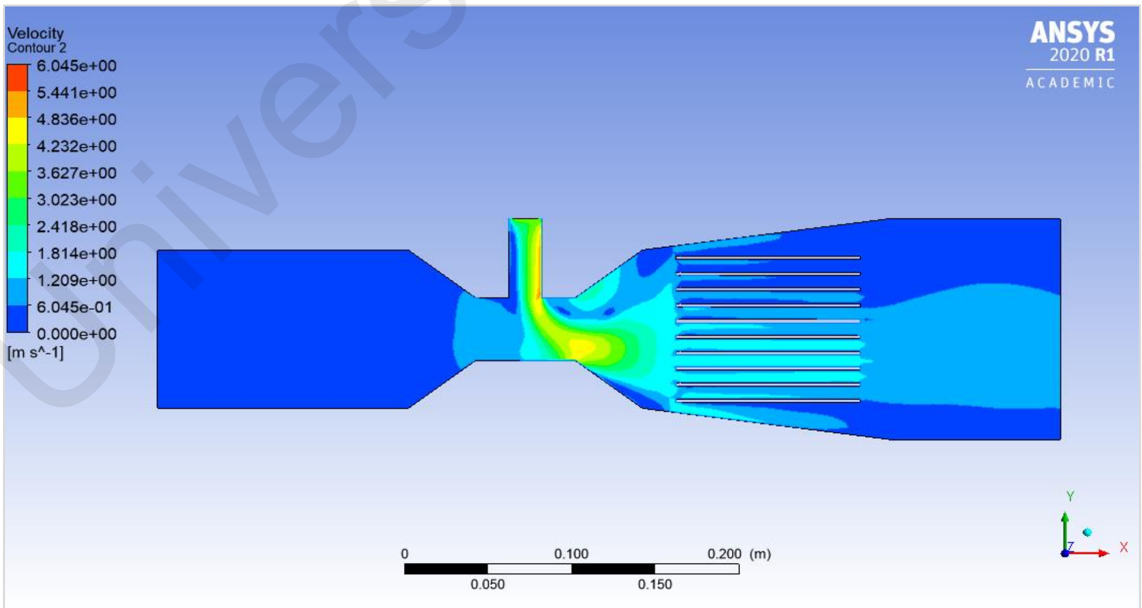

Figure 4.25 Model 5 Velocity Contour

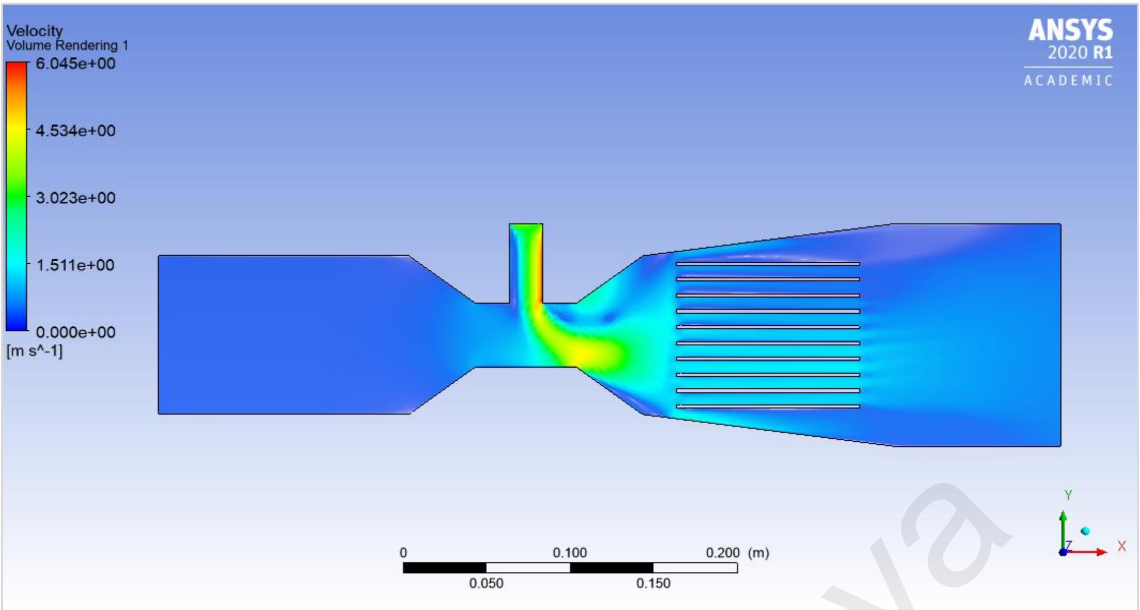

Figure 4.26 Model 5 Velocity Volume Rendering

4.5.1.2 Pressure

Figure 4.27 and 4.28 show that pressure is homogenized both the converging and diffusing section after adding the flow straightener. Spots of uneven high and low pressure at the diffusing section are resolved, where the mixture of water-air has been redistributing into the diffusing section.

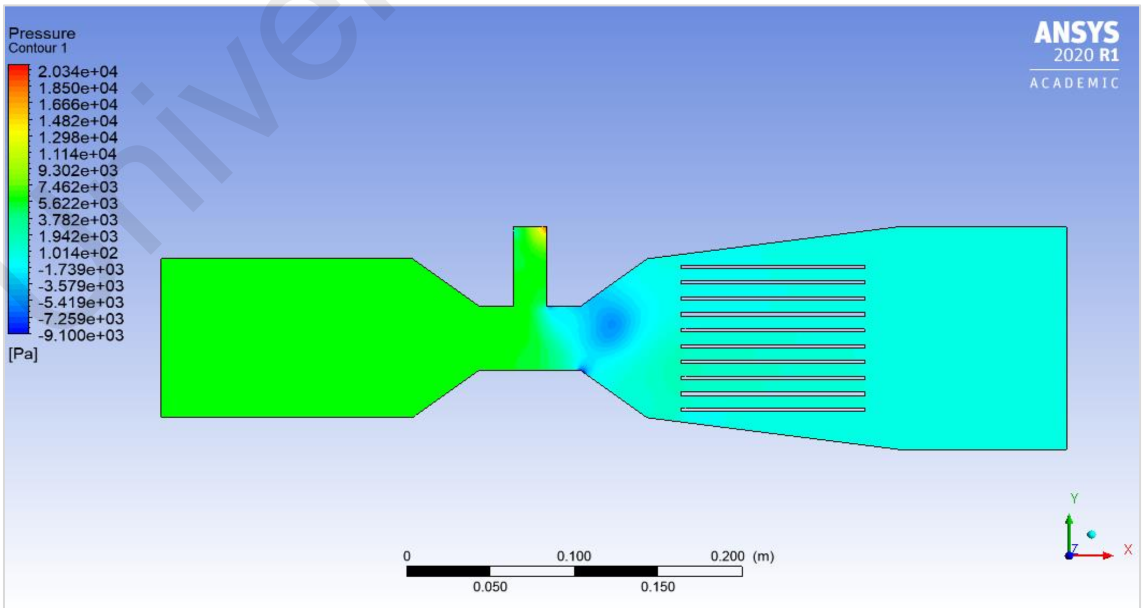

Figure 4.27 Model 5 Pressure Contour

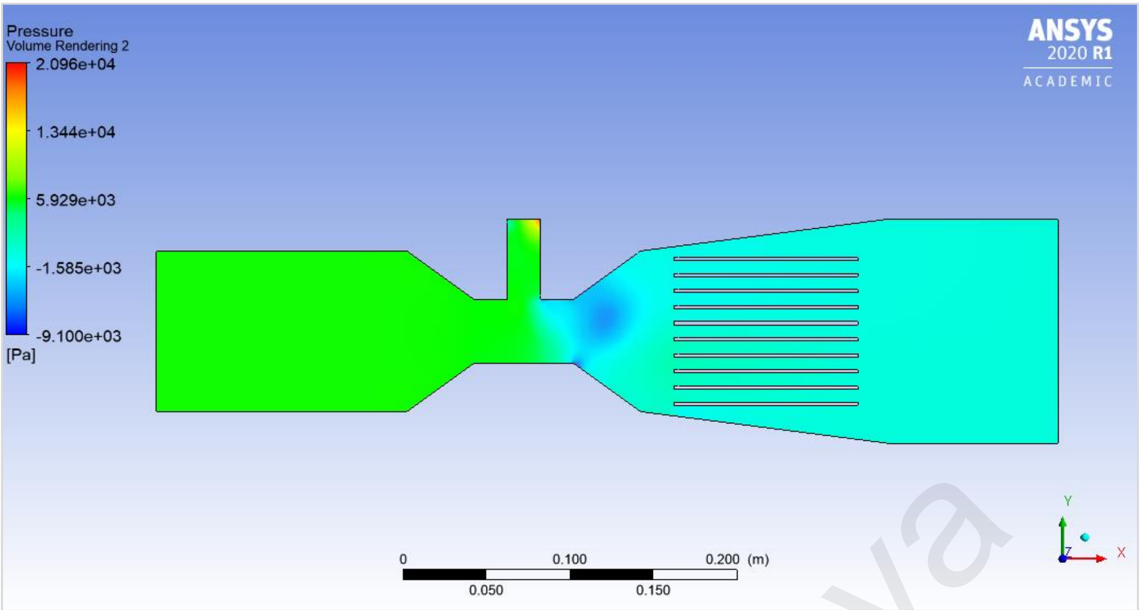

Figure 4.28 Model 5 Pressure Volume Rendering

4.5.1.3 Volume Fraction

Figure 4.29 and 4.30 show that water-air volume fraction is observed quite similar to the earlier model with water fraction at the diffusing section is slightly lifting up, maintaining the air fraction relatively higher at the top part, as compared to the water fraction at the bottom part of the downstream.

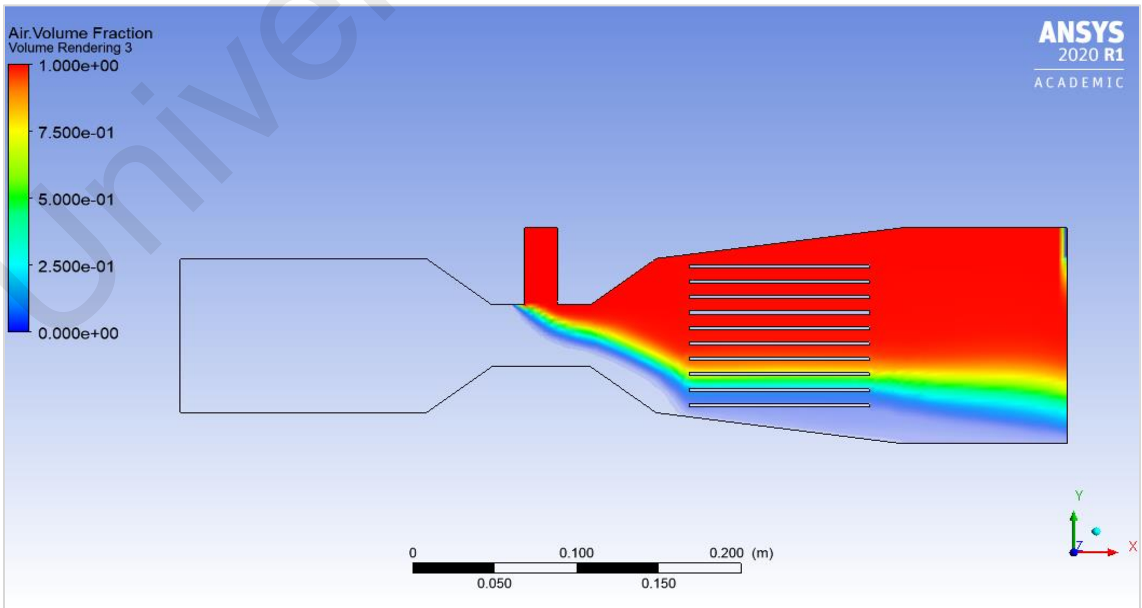

Figure 4.29 Model 5 Air Volume Fraction

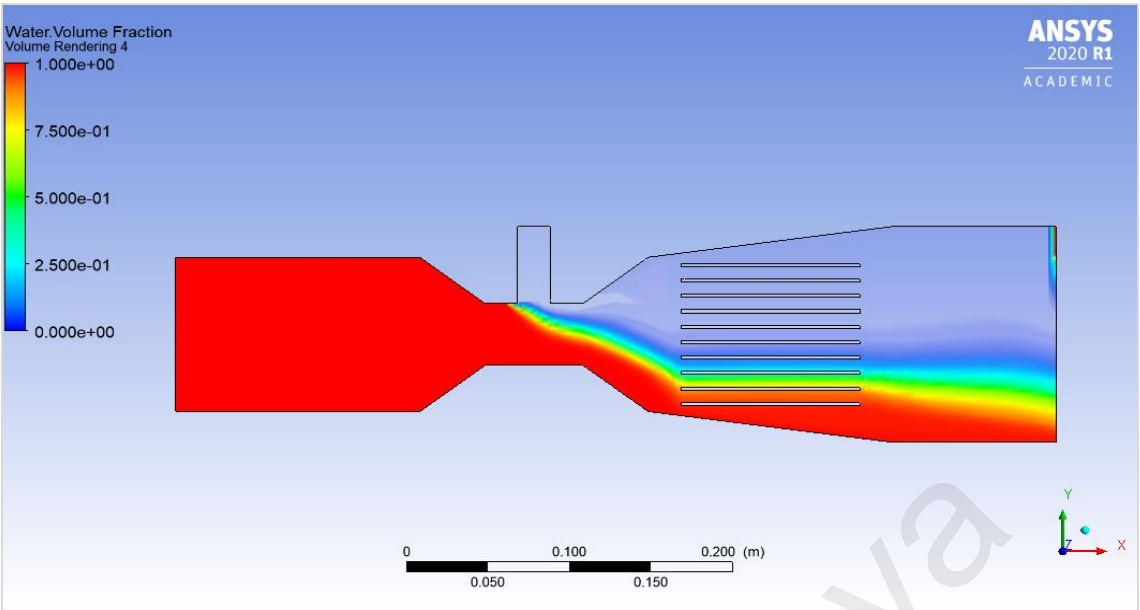

Figure 4.30 Model 5 Water Volume Fraction

4.5.2 Discussion

From the observation of all Model 5 simulations, adding the flow straightener smoothening the flow in the downstream and improving the upstream flow subsequently However, the water-air mixture volume fraction is still maintaining the air at the top, whereas water at the bottom, due to the fact that water is denser than the air, the earlier preset boundary conditions at outlet might not be achieved. Another reason is that the water flow is relatively low as compare to pipe diameter makes the velocity slower as compare to the air inlet. Recommendation to replace the flow straightener with a static mixer, in order to diffuse the mixture flow into all angles to improve the mixing across the diffusing section and the air inlet diameter shall be reduced with additional air inlet from other angles. Lastly, the primary phase flow velocity from water inlet shall be significantly increased and air inlet diameter shall be reduced to limit the secondary phase flow volume to achieve the mixing ratio of primary phase to secondary phase at 9:1.

CHAPTER 5: CONCLUSION AND RECOMMENDATION

5.1 Research Outcomes

This research project focused on the venturi effect occurs in pipeline system, specifically from an air ejector perspective through numerical analysis with the application of CFD simulation. Theoretical knowledge with regards to internal flow, especially on venturi effect and air ejector have been revisited, summarized and recorded. Numerous CFD simulations have been successfully conducted in this project, in order to study the venturi effect through an air ejector preliminary design and its improvement processes to visualize and compare the changes. Nonetheless, this research project improves the CFD simulation skills and experiences for other flow simulation purposes in the future study in fluid dynamics.

5.2 Conclusion

This research project has successfully implemented and the research objectives have been achieved to meet the aim as a whole. This research project concluded that the CFD simulation is an important tool for fluid dynamic studies, whereby a complex numerical calculation on a small-scale simulation can be done in a reasonable time to obtain accurate result, to make instant decision, particularly changes on design, setting condition, etc., without spending unnecessary time to redo the design and resources to make prototyping or physical hydraulic modelling for this case.

Hypothesis that a streamline flow will enhance the hydraulic conditions and the mixing quality is proven in this research project, through comparing and analyzing the simulation results of various improvement made. Throughout the simulation experience in this research project, future recommendation to conduct similar simulation on a venturi effect model, it is advisable to set the primary phase flow at a high velocity is advisable, to ensure better mixing of multiphase flow dominant by the primary phase will be achieved.

Nevertheless, the 2-D venturi effect model shall be transformed into a 3-D model to enhance the simulation results with more realistic results of the actual conditions.

5.3 Significance of Research

This research project has showed case the CFD simulation implementation in simulating the venturi effect occurs in an air ejector and the similar approach can also be applied to other process equipment, such as injector, inductor, aerator, etc. which has the similar working principle that utilizing venturi effects in industry. Therefore, this research project research methodology can be a reference to model other process equipment.

5.4 Contribution to Knowledge

With the successful implementation of this research project, this research project report will contribute to the application of CFD simulations in the studies of fluid dynamics as a case study for future research reference, expand the study to more in-depth and subsequently expanding the knowledge in this field.

5.5 Future Works

Future works beyond this research project should further demonstrate the extensibility of the simulation by implementing additional variants of multi-phase flow systems and also to simulate the model in 3-D, where it will simulate the unforeseen hydraulic circumstances in a 2-D model with the introduction of recirculation flow and more accurate assumptions.

Further to this research project, more extensive optimization of the shall be continued to simulate and study various hydrodynamics conditions to enhance the mixing quality through venturi effect, where additional parameters e.g., temperature, etc. can be added to the settings. Suggestion for the continuing of this research also to be extended to the designing and simulation process of an air ejector applies to an aeration process that is generally used in biological waste water treatment, whereby more actual industry application parameters can be utilized to simulate and visualize the equipment operation, as well as to make improvement through the design customization.

Future recommendations might consider to compare the simulation results with a physical scale-up hydraulic modelling. This will not only able to visualize the actual venturi effect phenomenon, but also to confirm the CFD simulation results. Adverse hydraulic condition, such as vortices, recirculation, etc. might be easily identified through physical modelling as compared to CFD simulation.

REFERENCES

Arias, D. A., & Shedd, T. A. (2007). CFD analysis of compressible flow across a complex geometry venturi.

Baylar, A., Aydin, M. C., Unsal, M., & Ozkan, F. (2009). Numerical modeling of venturi flows for determining air injection rates using FLUENT V6. 2. *Mathematical and Computational Applications, 14*(2), 97-108.

Bhatkar, M. R., & Ban, P. V. (2019). Review Study on Analysis of Venturimeter using Computational Fluid Dynamics (CFD) for Performance Improvement'.

Bogdanović-Jovanović, J., Stamenković, Ž., Spasić, Ž., Petrović, J., & Kocić, M. (2018). NUMERICAL INVESTIGATION OF CAVITATING FLOW IN VENTURI NOZZLE. *Annals of the Faculty of Engineering Hunedoara, 16*(2), 165-168.

CROFT, D. (1976). *Jet pump design and performance analysis.* Paper presented at the 14th Aerospace Sciences Meeting.

Djebedjian, B., Abdalla, S., & Rayan, M. (2000). *Parametric Investigation of Boost Jet Pump Performance.* Paper presented at the Proceedings of FEDSM, ASME Fluids Engineering Summer Conference.

Forney, L. J., & Kwon, T. (1979). Efficient single ‐ jet mixing in turbulent tube flow. *AIChE Journal, 25*(4), 623-630.

Galanis, N., & Sorin, M. (2016). Ejector design and performance prediction. *International Journal of Thermal Sciences, 104*, 315-329.

Hemidi, A., Henry, F., Leclaire, S., Seynhaeve, J.-M., & Bartosiewicz, Y. (2009). CFD analysis of a supersonic air ejector. Part I: Experimental validation of single-phase and two-phase operation. *Applied Thermal Engineering, 29*(8-9), 1523-1531.

Huang, B., & Chang, J. (1999). Empirical correlation for ejector design. *International journal of Refrigeration, 22*(5), 379-388.

Reader-Harris, M. (2015). *Orifice plates and venturi tubes*: Springer.

Saharan, V. K. (2016). Computational study of different venturi and orifice type hydrodynamic cavitating devices. *Journal of Hydrodynamics, Ser. B, 28*(2), 293-305.

Șcheaua, F. D. (2016). Theoretical approaches regarding the VENTURI effect. *Coefficient of Small Orifices*, 69.

Shinde, P. R., Chaudhari, R. H., Patil, P. S., & Marathe, S. S. Modelling and Simulation of Venturi Parameters in Relation to Geometries and Discharge Coefficient with Computational Fluid Dynamics.

Singhal, A., & Parveen, M. (2013). Air flow optimization via a venturi type air restrictor. *London UK, WCE*.

Sudakhar, K. (2017). CFD Analysis On Different Geometries Of Venturimeter By Using Fluent'. *Indian journal of Research, 6*(7), 2250-1991.

Sundararaj, S., & Selladurai, V. (2013). Flow and mixing pattern of transverse turbulent jet in venturi-jet mixer. *Arabian Journal for Science and Engineering, 38*(12), 3563-3573.

Tamhankar, N., Pandhare, A., Joglekar, A., & Bansode, V. (2014). Experimental and cfd analysis of flow through venturimeter to determine the coefficient of discharge. *International Journal of Latest Trends in Engineering and Technology (IJLTET), 3*(4), 194-200.

Tang, P., Juárez, J. M., & Li, H. (2019). Investigation on the effect of structural parameters on cavitation characteristics for the Venturi tube using the CFD method. *Water, 11*(10), 2194.

Vijay, P. H., & Subrahmanyam, V. (2014). Cfd simulation on different geometries of venturimeter. *Momentum, 1*(A2v2), 4.

Wang, W. (2017). Simulation of Venturi Tube Design for Column Flotation Using Computational Fluid Dynamics.

Xu, J., Liu, X., & Pang, M. (2016). Numerical and experimental studies on transport properties of powder ejector based on double venturi effect. *Vacuum, 134*, 92-98.

Yeoh, G. H., Liu, C., Tu, J., & Timchenko, V. (2011). advances in computational fluid dynamics and its applications. In: Hindawi.

Zhang, J. (2017). Analysis on the effect of venturi tube structural parameters on fluid flow. *AIP Advances, 7*(6), 065315.

Zheng, P., Li, B., & Qin, J. (2018). CFD simulation of two-phase ejector performance influenced by different operation conditions. *Energy, 155*, 1129-1145.