APPLICATION OF COMPUTATIONAL FLUID DYNAMICS IN AN OUTER CHANNEL OF ORBAL BIOLOGICAL SYSTEM

NURZULAIKHA BINTI MOHD KAMAL

SCHOOL OF CIVIL ENGINEERING
UNIVERSITI SAINS MALAYSIA
2019

APPLLICATION OF COMPUTATIONAL FLUID DYNAMICS IN AN OUTER CHANNEL OF ORBAL BIOLOGICAL SYSTEM

By

NURZULAIKHA BINTI MOHD KAMAL

This dissertation is submitted to

UNIVERSITI SAINS MALAYSIA

As partial fulfilment of requirement for the degree of

BACHELOR OF ENGINEERING (HONS.)
(CIVIL ENGINEERING)

School of Civil Engineering,
Universiti Sains Malaysia

June 2019

SCHOOL OF CIVIL ENGINEERING
ACADEMIC SESSION 2018/2019

FINAL YEAR PROJECT EAA492/6
DISSERTATION ENDORSEMENT FORM

Title:

Name of Student:

I hereby declare that all corrections and comments made by the supervisor(s) and examiner have been taken into consideration and rectified accordingly.

Signature:

Approved by:

____

____

(Signature of Supervisor)

Date :

Name of Supervisor :

Date :

Approved by:

____

(Signature of Examiner)

Name of Examiner:

Date :

ACKNOWLEDGEMENT

I would like to express my deepest appreciation to all those who provided me the possibility to finish my Final Year Project. A special gratitude I give to my supervisor, Dr Noor Aida binti Saad, whose contribution in stimulating suggestions and encouragement, and helped me a lot to finish this FYP.

Beside my supervisor I would like to thank my parents, Mohd Kamal bin Arbain and Norna'aimah binti Omar for always give me support and take good care of me. My sincere thanks also goes to REDAC Physical Lab which let me use the apparatus during on-site measurement for this project. Without all the support and encouragement, it would not be possible to finish this project.

ABSTRAK

Sistem Orbal Biologi (OBS) merupakan salah satu daripada parit pengoksidaan yang telah diubah suai, didakwa cekap tenaga, paling selamat dan mempunyai tahap penyenggaraan yang rendah. OBS boleh diubah suai dengan mudah untuk memenuhi pelbagai syarat influen dan keperluan effluen. OBS mempunyai tiga saluran iaitu saluran luar, saluran pertengahan dan saluran dalaman. Saluran luar OBS mempunyai isi padu terbesar di antara ketiga- tiga saluran tersebut, iaitu sekitar 50% daripada isi padu keseluruhan sistem. Kajian sistem operasi saluran luar OBS amat penting kerana ia menyumbang kepada penggunaan tenaga tertinggi dalam sistem. Hal ini demikian kerana sistem rawatan air sisa pada masa kini, piawaian kualiti efluen bukan hanya tertumpu pada kriteria, tetapi aspek-aspek lain seperti ekonomi juga diambil sebagai pertimbangan. Oleh itu, kajian ini dijalankan untuk memahami fungsi dan kesan bilangan cakera pengudaraan di saluran luar OBS. Model *Computational Fluid Dynamics* (CFD) dibangunkan untuk mempunyai gambaran OBS yang lebih baik. Simulasi dalam CFD digunakan untuk mengkaji tingkah laku yang mempengaruhi saluran luar OBS dan pengagihan halaju untuk keadaan operasi yang berlainan di saluran luar OBS. Hasil dari model CFD digunakan untuk mengkaji hubungan antara halaju dan jumlah cakera pengudaraan yang digunakan. Berdasarkan kajian ini, dengan mematikan beberapa batang di saluran luar OBS menyumbang kepada penggunaan tenaga yang kurang. Daripada penyelidikan ini, penyelewengan purata bagi model CFD adalah 23%, yang memerlukan peningkatan dalam penjanaan grid. Penyelidikan ini memberikan pemahaman yang lebih baik mengenai kelakuan aliran model sistem rawatan biologi menggunakan alat simulasi. Ia boleh menyumbang kepada reka bentuk model rawatan air buangan yang lebih baik pada masa akan datang.

ABSTRACT

Orbal Biological System (OBS) is one of the modified oxidation ditches which claimed as the most energy efficient, safest and has the lowest maintenance. The OBS can be easily modified to meet wide assortment of influent conditions and effluent requirements. There are three channels in OBS, outer channel, middle channel and inner channel. The outer channel of OBS has the largest volume among the three channels, which is about 50% of the total volume of the system. It is crucial to study the operating system of the outer channel of OBS as it contribute to the highest energy consumption in the system. This is because in the wastewater treatment system nowadays, the quality standards of the effluent are not only the criteria to be focused on, but other aspects such as economic also are taken as considerations. Therefore, the study was conducted to understand the function and the effect of the number of aeration discs in the outer channel of the OBS. The Computational Fluid Dynamics (CFD) model is developed to have a better picture of the OBS. The simulation in CFD were used to study the behaviour of the influent in the outer channel of OBS and the velocity distribution for different operating conditions in the outer channel of OBS. The result of the CFD model were used to study the relationship between the velocity and the number of aeration discs being used. Based on the study, by turning off some shaft in the outer channel of OBS was contributing to less energy consumption. From the research, the average deviation for the CFD model is 23%, which need improvement in grid generation. The research provides a better understanding of model flow behaviour of a biological treatment system using the simulation tool. It can contribute to a better design of wastewater treatment process models in the future.

TABLE OF CONTENTS

APPENDICES

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATIONS

CFD	Computational Fluid Dynamics
DO	Dissolved Oxygen
GMP	Good Modelling Practice
IWK	Indah Water Konsortium
OBS	Orbal Biological System
OD	Oxidation Ditch
RANS	Reynolds Averaged Navier-Stokes
SND	Simultaneous Nitrification and Denitrification
WWT	Wastewater Treatment
WWTP	Wastewater Treatment Plant

CHAPTER 1

INTRODUCTION

1.1 Background

Wastewater is a mixture of pure water with large number of chemicals, including organic and inorganic, and heavy metals which can be produced from domestic, industrial and commercial activities, in addition to storm water, surface water and ground water (Dixit et al., 2011).

In developing countries, the water sector is a major contributor to municipal energy use. Wastewater treatment plants stand for a major part of the energy- demanding processes related to water. A significant amount of energy is consumed in the aeration processes, where oxygen is provided for a biological system (Odlare, 2014). Wastewater treatment plants can be classified as energy-intensive facilities, as they account for up to 35% of municipal energy consumption. Pumps and aeration systems consume a significant portion of energy within the wastewater plants in particular(Masłoń, 2017). Process of aeration represents the largest proportion of the plant energy consumption, ranging from 45 to 75% of the plant energy expenditure(Sánchez et al., 2018).

One of the modified oxidation ditches used known as the Orbal Biological System (OBS), which designed to provide an optimized approach of the biological treatment process. The OBS can be easily modified to meet a wide assortment of influent conditions and effluent requirements. There are three channels in OBS, outer channel, middle channel, and an inner channel. The outer channel of OBS has 50-55% of the total volume in the system (Daigger and Littleton, 2018). The actual oxygen demand of the first channel might be as high as 75% of the total, the aeration discs allotted to this channel supply only 30-60% of the system’s overall oxygen requirements to ensure a constant oxygen deficit condition and an operating dissolves oxygen of zero throughout the channel. Delivery and power of aeration disc vary with changes in immersion, changes in speed or with discs added on or removed from individual assemblies (Siemen, 2006).

Computational Fluid Dynamics (CFD) is a branch of physics that deals with the study of the mechanics of fluid: liquid, plasmas, gas and forces acting on them. CFD is a popular modelling technique which is widely used to simulate and analyze the performance of various processes(Bitog et al., 2014). The most powerful use of the Computational Fluid Dynamics is to simulate integrated physical, chemical and/or biological processes involves in wastewater treatment (WWT) design and operation, but to date, it should be understood for unit process design and troubleshooting (Samstag et al., 2016a).

1.2 Problem Statement

The OBS is claimed as the most energy efficient, lowest maintenance and the safest oxidation ditch available. The Orbal process requires less power to operate than other oxidation ditch system that available (Siemen, 2006). However, in reality the energy consumption in OBS is higher than it was claimed.

The outer channel of the OBS has the largest volume among the three channels that available in OBS. As the volume of the outer channel in OBS is the largest, the outer channel is targeted to have the largest energy consumption. The energy consumption in OBS can be varied due to the changes in velocity of the aeration disc or the addition and removal of the discs. Besides, the number of discs placed of the shaft may also allowing the relative distribution of oxygen supply to be varied.

The velocity and behaviour of the influent in an outer channel of OBS can be determined by developing a CFD model. The simulation in CFD is specifically used as a tool to model flow behaviour of a system and to predict the events which control the behaviour.

1.3 Objectives

The purpose of this research is to develop a model that can represent a real scale of OBS for a better understanding of wastewater behaviour as well as increase the knowledge on the function of aeration discs in OBS. In order to achieve the aim, the following objectives will be performed:

1. To study the application of CFD in biological treatment system.
2. To develop a CFD model for an outer channel of OBS.
3. To investigate the relationship between velocity and the number of aeration disc.

1.4 Scope of Work

The system of OBS, where the simultaneous nitrification/denitrification (SND) process takes place, is claimed as the most energy efficient, lowest maintenance and the safest oxidation ditch available nowadays (Siemen, 2006). In reality, the energy consumption in OBS is the highest in a wastewater treatment plant when compared to other processes. CFD model developed can be used to determine the flow properties such as velocity at any location inside an outer channel of the OBS.

A three-dimensional CFD model has been developed to understand the flow pattern in an outer channel of OBS. By developing a CFD model of OBS, the properties of flow such as velocity at any location inside an outer channel of OBS can be determined. Besides, the study of the number of aeration discs used in the outer channel can be linked to the contribution of energy in OBS.

In general, the number of aeration discs used contribute to the energy consumption of OBS. The model developed can be used to determine the optimum number of aeration discs is required in an outer channel of OBS. However, the velocity and the number of aeration discs used have to be validated and calibrated in the real model of OBS to obtain the real value on site.

1.5 Dissertation Outline

This dissertation is presented in 5 chapters. The first chapter is the introductory, this chapter provides the overview of the problems, the description of the problem, the objectives and the scope of works. Chapter two is the literature review that covering the objectives. Chapter three portrays the research methodology of the project. In this chapter, a detailed of CFD simulation that includes the case set up and solution is presented. Chapter four presents the results and discussion for the CFD model. The discussion also include the findings of the project. Chapter five gives the conclusions and recommendations based on the findings in this project. These include the recommendations for the future research on wastewater industry.

CHAPTER 2

LITERATURE REVIEW

2.1 Overview

Wastewater is a complex mixture of inorganic and organic materials, and heavy metals. Wastewater also known as sewage, can be divided into domestic wastewater, industrial wastewater, and municipal wastewater (Odlare, 2014). Wastewaters consist of water in which solids exist as settleable particles, dispersed as colloids, which are materials that do not settle readily, or solids in a dissolved state (Muralikrishna and Manickam, 2017). Wastewater treatment is a process which is used to convert wastewater into a useful effluent with no negligible health and environmental issues, which is then returned back to the water-cycle or in other words, it can be reused (Ameta, 2018). The aim of treating the wastewater is to reduce the concentrations of pollutants to the specified level before it is discharged. However, wastewater treatment is energy-intensive, with such processes as aeration consuming large amounts of energy (McIntosh and Pontius, 2017). Recently, wastewater treatment is becoming increasingly complex because of the wide spectrum of pollutants generated by industries including chemical, biomedical, pharmaceutical, textile, and other sectors(Bhuta, 2014).

Wastewater treatment plants are designed to have distinct water treatment process combinations with varied water treatment facilities depending on the influent's water quality and the effluent discharge standard (Ou and Zeng, 2018). Biological treatment methods use microorganisms, mostly bacteria, in the biochemical decomposition of wastewaters to stable end products. More microorganisms, or sludges, are formed, and a portion of the waste is converted to carbon dioxide, water, and other end products (Muralikrishna and Manickam, 2017). Generally, there are two methods of biological treatment, which is aerobic and anaerobic methods, the difference is based on the availability of the dissolved oxygen.

2.2 The Orbal Biological System

The Orbal Biological System is one of the oxidation ditch used in Malaysia. An oxidation ditch is a modified activated sludge biological treatment process that uses long solids retention times (SRTs) to remove biodegradable organics, the typical oxidation ditch is equipped with aeration rotors or brushes that provide aeration and circulation (Shammas and Wang, 2009). An oxidation ditch (OD) is a modified activated sludge treatment process, which has been used for many years all over the world, a typical oxidation ditch consists of a single- or multi-channel configuration within a ring, oval, or horseshoe-shaped basin (U.S.EPA, 2000).

The mechanism of nitrogen removal in an oxidation ditch is based on the presence of aerobic and anoxic zones within the channel (Daijun et al., 2010). Although operational conditions such as low loading rate, high internal recycle ratio of the mixed liquor, and long sludge age are responsible for promoting nitrogen removal, control of aeration seems the easiest and most economic operational strategy (Liu et al., 1996). Aerobic and anoxic zones can be formed within a single channel through the control of oxygen supply, with nitrogen removal achieved via simultaneous nitrification and denitrification (SND) (Daigger and Xiaoxin Littleton, 2000).

The outer channel of an Orbal oxidation ditch plays an important role in TN removal which is quite difficult, however, to ascertain how much oxygen is needed within the outer channel due to the different DO and organic substrate requirements between nitrification and denitrification (Liu et al., 1996). The oxygen provided in the outer channel is usually less than the actual oxygen demand to promote SND under the low oxygen condition (Siemen, 2006).

The outer channel of OBS contains about 50-55 percent, middle channel about 30-35 percent, and the inner channel about 15-20 percent of the total bioreactor volume. A typical design and operating strategy for the orbal process is to provide less oxygen to the outer channel than would be required to meet the full process oxygen demand. The oxygen supply in channel 1 is typically 50-70 percent of the calculated demand. This is done to allow nitrification and denitrification to occur in the outer channel (Siemen, 2006).

2.3 Sewage Treatment Plant in Malaysia

Based on the data provided by the Malaysia Department of Statistics (Q4, 2018), Malaysia’s population is 32.6 million. The population growth effects wastewater treatment plant in terms of plant design, operational capacity (flow rate) and other treatment process constraints (Teklehaimanot et al., 2015). Other than that, the population growth causing more pressure on the environment and threatens the sources of fresh water supplies due to waste management problems (Saad, 2016). Malaysia’s fresh water supply is 98% comes from surface water, and raw surface water becomes contaminated as a result of an excessive and indiscriminate discharge of wastewater directly from households or factories to drains into rivers with minimal or no treatment.

Indah Water Konsortium (IWK) is responsible for planning and rationalizing the public sewerage facilities to reduce the number of treatment plants using the "multipoint concept" or regionalization in Malaysia. Then, sewerage pipeline networks will be laid in urban areas currently serviced by IST to convey the domestic sewage to modern secondary treatment facilities. In Malaysia, 38% of public sewage treatment plants in the country are mechanical plants, they operate using mechanical equipment that accelerates sewage break down. The usage of public sewage treatment plants in shown in Table 2.1. According to IWK (the country’s main sewerage operator), the focus has been providing a basic standard of preliminary, primary, and secondary treatment. The dominant wastewater treatment types are preliminary (removal of rags, rubbish, grit, oil, grease), primary (removal of settleable and floatable materials), and secondary treatment (biological treatment to remove organic and suspended solids). At the moment, there is no plan to build tertiary treatment systems in Malaysia (Mat et al.).

In Malaysia, the treatment that is commonly used in an urban area are activated sludge, aerated lagoons, rotating biological contractors and trickling filters. That is because in urban areas, sophisticated treatment systems have evolved, and can produce a high quality effluent. While for small communities’ area, the populations are less dense, and the receiving environment is able to cope with the lesser level of treatment, therefore only primary treatment will be provided. That is the reason on using a simpler systems have been used, however, to increase the environment standards the installation of the better treatment systems should be done. The major biological sewage treatment processes and the commonly used treatment systems for small communities are shown in Table 2.2 and Table 2.3.

Table 2.1 : Public Sewage Treatment Plants in Malaysia by IWK

No.	Types of Sewage Treatment Plant	As At Oct 2014	Population Equivalent
1	Imhoff Tank	679	507,648
2	Oxidation Ponds	403	1,681,176
3	Mechanical Plants	4,902	18,665,408
4	Network Pump Stations	982	4,852,844
	TOTAL	6,966	25,707,076
	COMMUNAL SEPTIC TANK	3,625	405,432

Table 2.2: Major Biological Sewage Treatment Processes

Aerobic Processes	Suspended Growth	Activated Sludge
		• Plug flow • Complete mix • Sequencing batch reactor • Extended aeration* • Oxidation ditch* • Deep shaft* • Aerated lagoons*
	Attached Growth	Trickling Filters • Low rate • High rate* • Rotating biological contactors* • Submerged biological contactors*
	Combines	Biofilter activated sludge • Trickling filter activated sludge*
Anaerobic Process	Attached Growth	• Anaerobic contact • Anaerobic filter • Expanded bed
Pond Processes		• Aerobic Stabilization (oxidation) • Facultative

		• Anaerobic

*Systems used in Malaysia

Table 2.3: Commonly used Treatment Systems for Small Communities

Primary Treatment	• Individual septic tanks • Communal septic tank • Imhoff tanks
Secondary Treatment	Package (pre-fabricated) Plants • Sequencing batch reactors • Contact stabilization • Rotating biological contactors Individually Designed Plants • Activated sludge systems • Oxidation pons • Sequencing batch reactors • Rotating biological contactors • Trickling filter • Facultation lagoons • Aerated lagoons

2.4 Computational Fluid Dynamics (CFD)

Computational fluid dynamics (CFD) models have been extensively and successfully used for many decades in a wide range of engineering disciplines including, but not limited to, automotive engineering, chemical engineering, mechanical engineering and aerospace engineering (Wicklein et al., 2015). CFD tools and methods are often used to analyse the energy distribution in a domain. CFD is, in part, the art of replacing the governal partial differential equations of fluid flow with numbers, and advancing these numbers in time or space to obtain a final numerical description of the complete flow field of interest (Wendt, 2008). The software is based on the fundamental equations of continuity, momentum, and energy (McClure et al., 2014). CFD is a highfidelity solver based on Reynolds Averaged Navier-Stokes (RANS) equations which has been proved giving accurate flow prediction (Lin et al., 2019).

Simulation such as CFD has been established as a helpful tool. An example is the cheaper setup for experiments compared to real experimental setups take a lot of effort to build and are cost intensive. In addition, observability of critical processes as well as the breakdown into simpler segments is an advantage of simulation (Stefan and Gunther, 2016). The CFD model typically consists of a description of flow geometry, a set of differential equations describing the physics of the flow, boundary and initial conditions, and mesh points at which these equations are solved (Warsi, 2005).

General components of good modelling practice (GMP) for any modelling exercise consist of: (1) clearly defining the objective of the modelling task with all project parties involved; (2) collection of data, defining the model configuration (e.g. level of detail of the model, spatial discretization or mesh generation) and simulation settings; (3) running the simulations (steady state, dynamic); and (4) post-processing, calibration and validation as shown in Figure 2.4 (Cameron and Hangos, 2001). The development of computational fluid dynamics (CFD), flow patterns in oxidation ditches have been simulated by a 3D turbulence model to simulate the flow field in an integrative oxidation ditch aerated with one set of brush aerators (Yang et al., 2010).

CFD can be regarded as a numerical simulation of the flow under the control of the basic equations of flow (mass conservation equation, momentum conservation equation, energy conservation equation), CFD uses the computer as a simulation method to use some computational techniques to find the complex problems of fluid mechanics and discrete numerical solution (Wei, 2017). Numerical solution is a discrete approximation of the calculation method, which generally follow the steps shown in Figure 2.1.

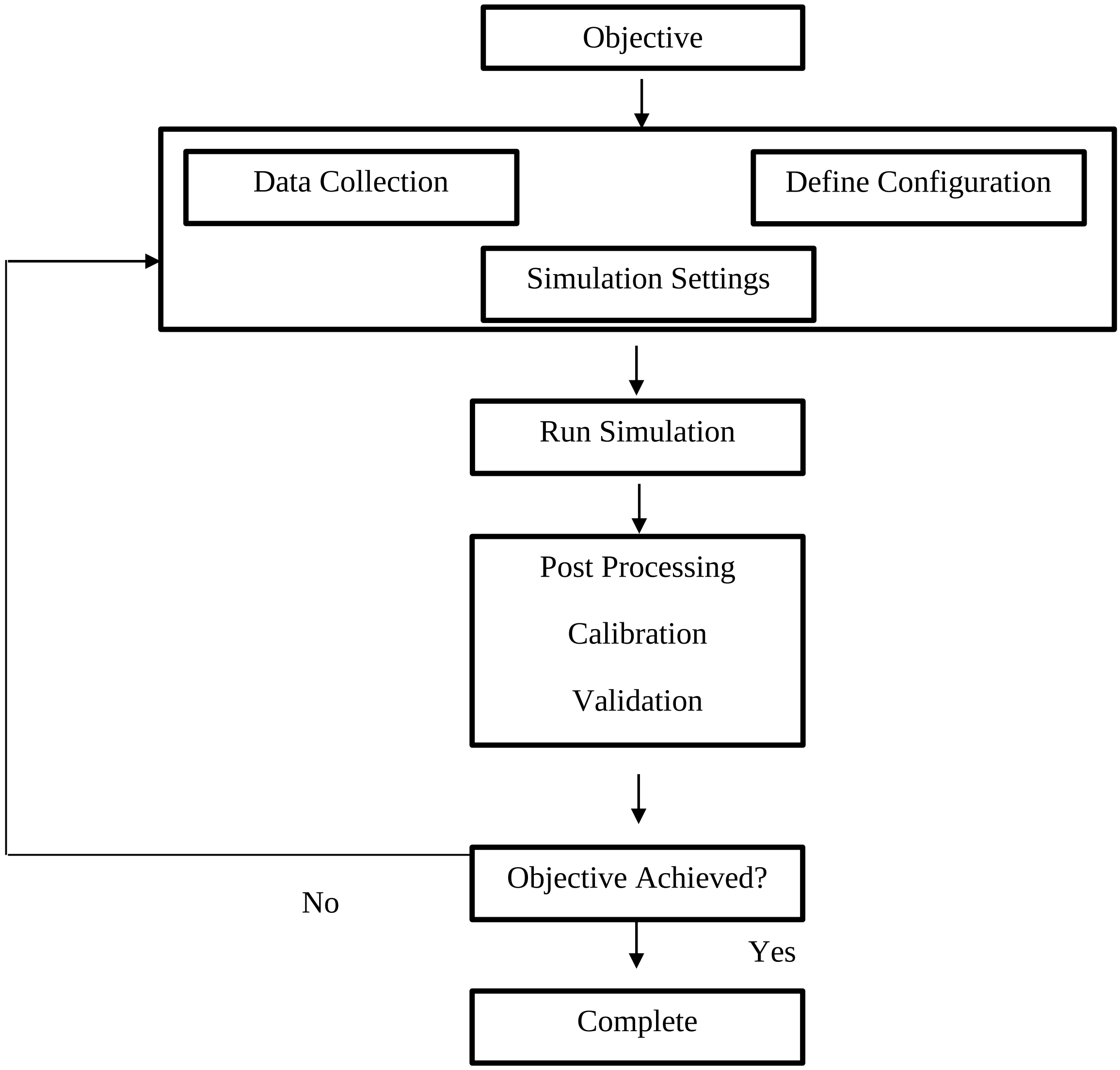

Figure 2.1: Overview of The Different Components of a general Modelling Process

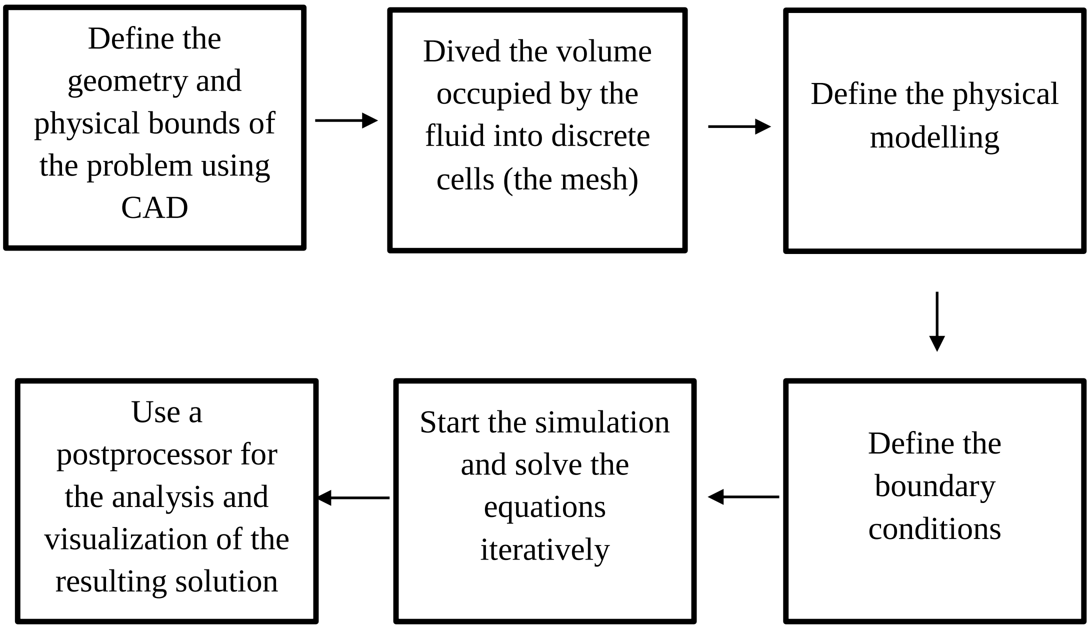

Figure 2.2: CFD numerical method for solving flowchart (Wei, 2017)

2.5 Application of CFD Modelling in WWTP and Previous Case Studies

CFD has become an accepted method for process analysis of fluid flows in many industries including analysis of hydraulic problems in water and wastewater treatment (WWT). There are substantial financial and risk drivers to conduct CFD for better wastewater design (Wicklein et al., 2016). CFD is described as a supplement to use the simpler model (Laurent et al., 2014). However, there are lacks an analysis of unit operations, including historic analysis of work focusing on the development of specific unit process approaches (Samstag et al., 2016a).

One of the application of CFD in wastewater treatment facilities is to identify flow splitting and evaluation of head losses. Flow splitting is defined as a critical unit operation that enables balanced flow to multiple units across a range of flows. CFD has been used relatively extensively in engineering practice for analysis of flow splitting between process units and for estimation of complex head losses in hydraulic profiles (Marques, 2015a). CFD is also used to indicate the influent direction relative to the channel that may impact the results in (Knatz, 2005). Besides, CFD is used to investigate balancing the flow through a wide range of manifold geometries, and the results showed that the optimal flow uniformity can be achieved by the expansion of the cross-sectional area, linear or non-linear tapering of the distribution manifold or varying the cross-sectional area of the outflow channels modifications in (Tong et al., 2009).

CFD is also used in grit chamber to evaluate the grit removal that goes back to the work and developing a rational approach for sizing of grit removal channels based on ideal settling behaviour. CFD provides an alternative to reliance on manufacturer's claims, particularly where the hydraulic configuration mixes gravity and centrifugal sedimentation, such as in a vortex separator (Samstag et al., 2016b). A comprehensive evaluation of three different types of grit removal tank geometries is also done for forced vortex, detritor and lamella (McNamara et al., 2012). The best work appears to have included analysis of solids transport and discrete settling in the grit chamber.

Primary sedimentation was one of the first areas in wastewater treatment that used the application of CFD. Earlier, CFD in primary sedimentation was done in 2-D and in relatively coarse grids in custom programs, with a key focus on neutral density hydraulics rather than sedimentation performance. Previously, there are report on a comprehensive evaluation of two primary sedimentation tanks evaluated with 2-D and 3-D CFD models (Griborio et al., 2014) and used to estimate velocity profiles in the inlet and outlet (Griborio, 2004). Other than that, CFD is also used for 3-D model, such as effects of flocculation (Griborio et al., 2014) and involve multiple mechanisms, including sedimentation, mechanical-hydraulic coupling, and multiphase interactions (Paulsrud et al., 2014).

In biological processes, a 3-D single phase CFD simulation of an oxidation ditch configuration was implemented to determine RTD characteristics (Pereira et al., 2012). Besides, 3-D commercial CFD model to simulate fluid flow in a closed loop reactor with aeration and consumption of dissolved oxygen (DO) simulated using a supplemental equation with a goal of identifying zones where simultaneous nitrification–denitrification was likely, and the CFD model was validated with field data obtained from a test tank and a full-scale tank (Littleton et al., 2001).

In general, the usage of CFD in WWT has identified that the sophistication of the technique is highly variable between different units. Particularly, CFD simulation of secondary sedimentation is highly mature, while work in anaerobic digestion is less well developed for the application of CFD in wastewater treatment plant.

CHAPTER 3

METHODOLOGY

This chapter presents the procedures and the methods that involved in order to develop a 3-Dimensional CFD model.

3.1 CFD Modelling Framework

The 3-Dimensional CFD model is developed to discover the relationship between velocity and the number of aeration disc in an outer channel of OBS.

The flow in the OBS in complex and usually cannot be described by using the simple techniques. In order to get a clearer picture of the real conditions inside the outer channel of OBS, a 3-Dimensional CFD model is developed. From the model, the system can be described better.

In general, there are five major steps involved in CFD model development as shown in Figure 3.1. Firstly, all of the information needed to develop the model is gathered. The geometry model of OBS was created according to the design layout and dimensions that provided by IWK. X-axis corresponded to the fluid flow direction, Y-axis indicated the width of the outer channel, while Z-axis represents the depth of the fluid inside the OBS.

After the geometrical model was done, the grid generation or meshing part was started. Mesh generation is trivial, and the physical location of any mesh point and the identity of its neighbours are known implicitly and need not to be stored (Mavriplis, 1996). A Cartesian mesh is used as it is the simplest gridding approach, and the mesh points are regularly distributed at equal distances from one another throughout the flow field. For the OBS computational model, a grid independence study was conducted for different size of mesh. By the grid independency analysis, the grid is independent of any effects due to the grid size.

Then, appropriate boundary conditions at the domain boundary is specified after the meshing was done. Geometry creation, grid generation and boundary conditions setup is classified as pre-processing stage. Pre-processing stage is the part before solver and later the post-processing stage took place. After the case had been set up at the pre-processing stage was computed and iterated using the Fluent software. Pre-processing and post-processing stages both can be done in Fluent software. CFD results were visualised, examined and processed during the post-processing works in FLUENT. The whole development's procedures of the CFD model is illustrated in Figure 3.2 and 3.3.

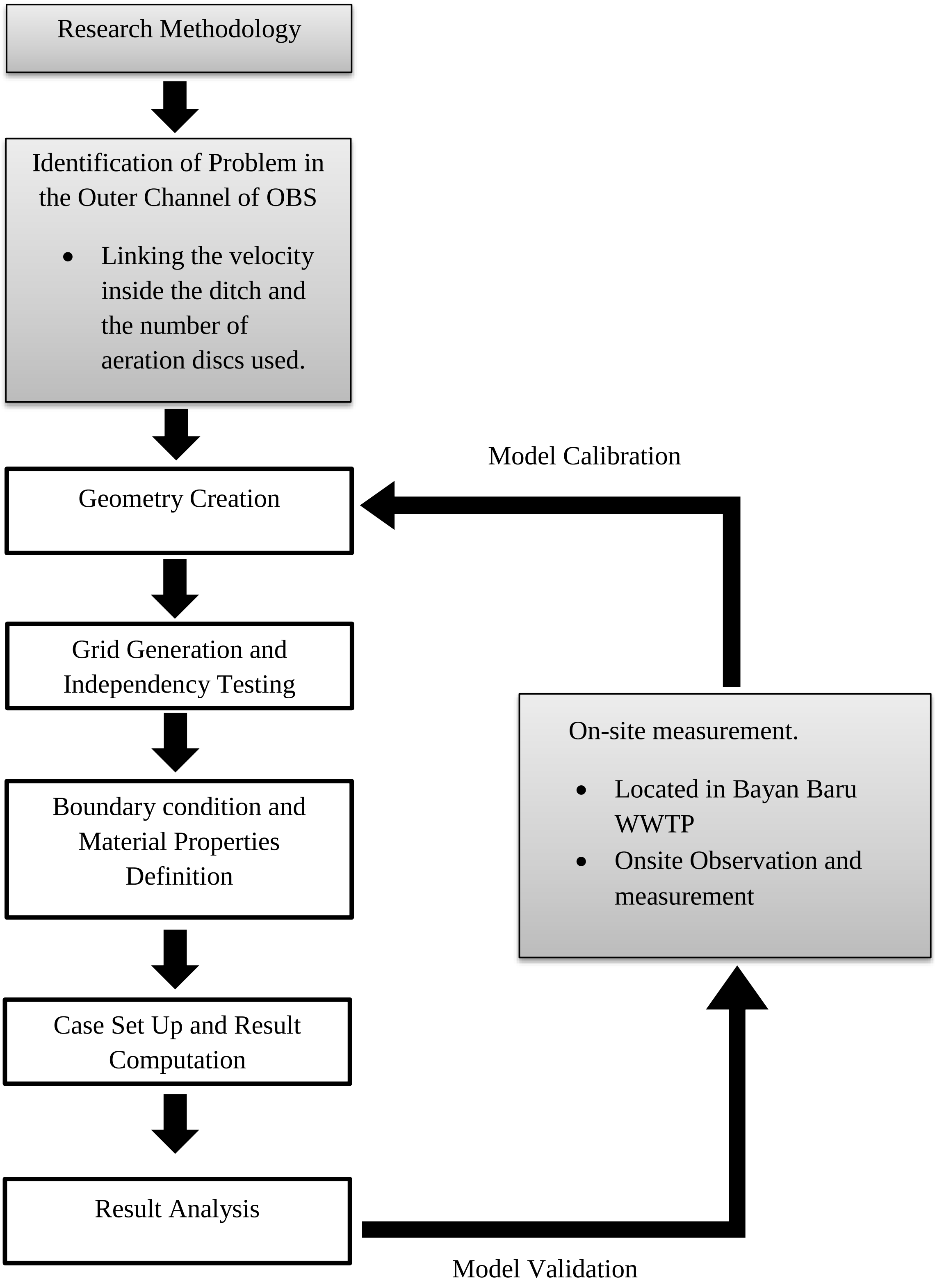

Figure 3.1: Flow chart for overall methodology

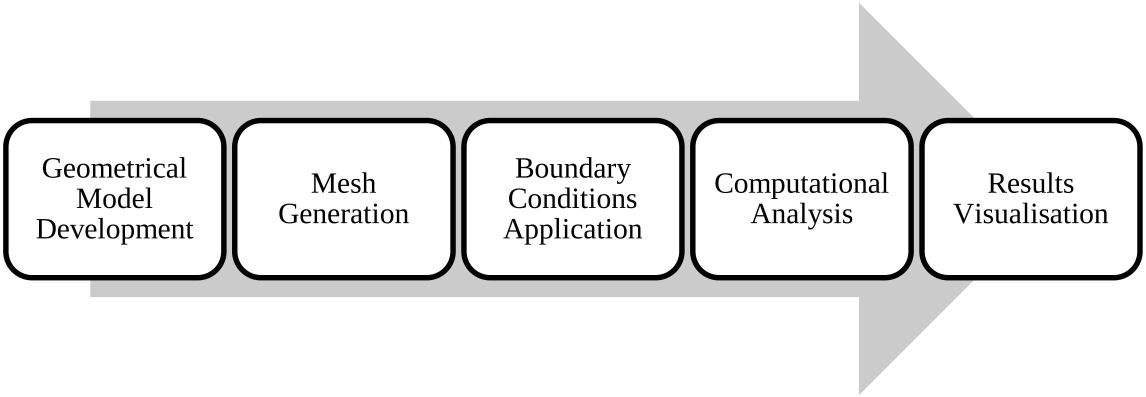

Figure 3.2: Five major steps of CFD simulation

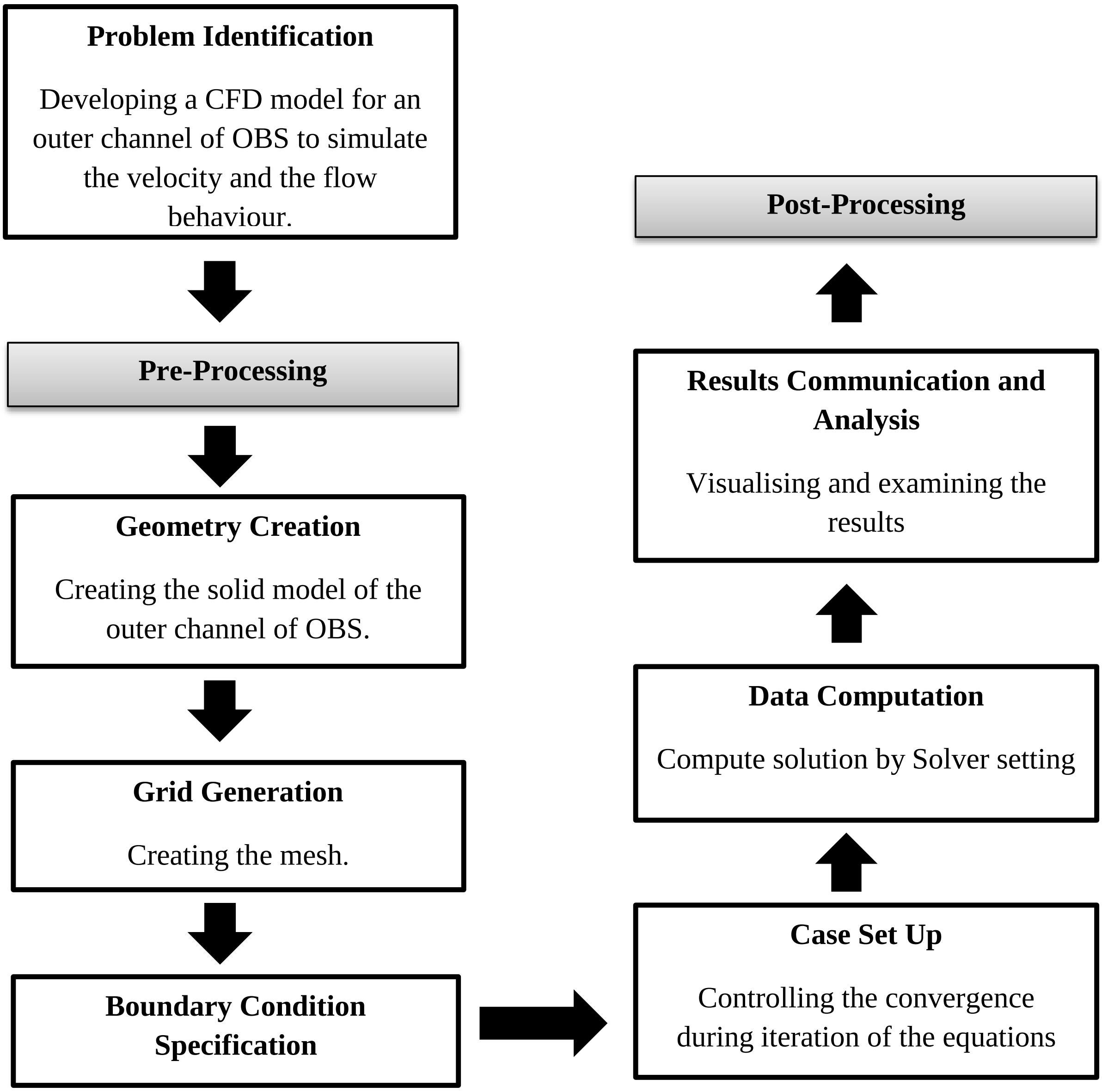

Figure 3.3: Procedures of CFD model development

3.1.1 Geometry Creation

Firstly, the geometrical model was developed by using DesignModeler in ANSYS Fluent. DesignModeler is used to create and modify the geometry in preparation for analysis in ANSYS Workbench.

The overall length of the model is 91.90 m, while the overall width of the model is 54.56 m as shown in Figure 3.4. This is based on the actual values of the OBS onsite. The geometrical developed containing 3 channels that are outer, middle, and inner channel. However, the CFD model developed is only for the outer channel of OBS. The outer channel is the largest and where the influent is located. The width of outer channel is 12.50 m, the thickness of the wall is 0.36 m, and the length of long axis straight section is 37.34 m. The depth of water in this system is 4.4 m. The water from the distribution enters the system through the outer channel, move along the outer channel then enters middle channel throughout the effluent penstock.

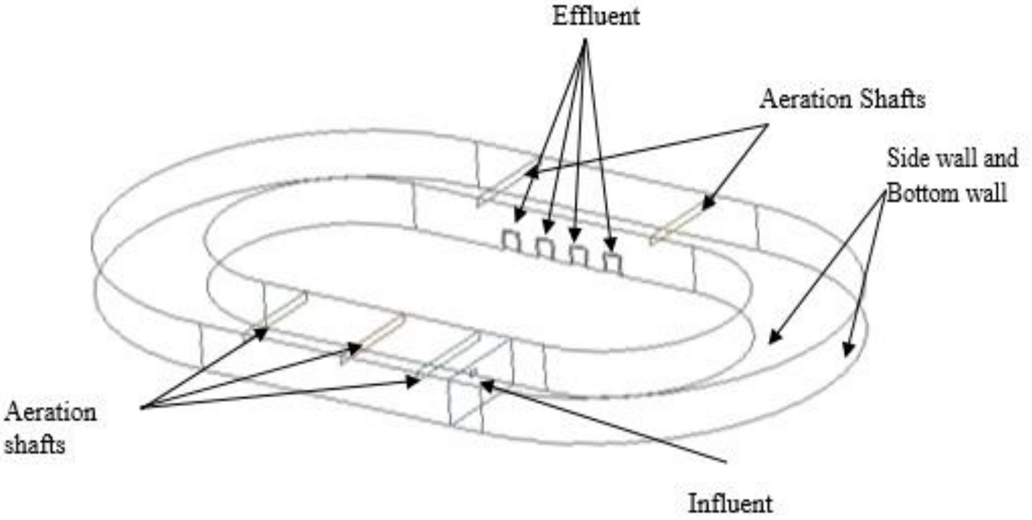

Figure 3.4: Geometrical layout of 3D model

3.1.2 Grid Generation

The existence of a mesh is to determine the locations in the field at which the flow quantities will be evaluated as well as the stencil of the resulting discrete equations. A mesh is simply an artificial geometric construction that facilitates the spatial discretization of the governing equations to be solved (Mavriplis, 1996). Meshing part is done by using ANSYS Meshing. ANSYS Meshing is a general-purpose, intelligent, automated high-performance product. It produces the most appropriate mesh for accurate, efficient multiphysics solutions. A mesh well suited for a specific analysis can be generated with a single mouse click for all parts in a model. Full controls over the options used to generate the mesh are available for the expert user who wants to fine-tune it.

There are three types of grids, which are: (1) structured grids; unstructured grids; and (3) block structured grids. Structured grids is the simplest one, because all nodes have the same number of elements around it. This type of grids can be described and stored easily. However, this type of grid is suitable for simple domain only.

As the outer channel of OBS is a complex domain, unstructured grid is used. The flow near the object is very important and complex, therefore very fine grid is needed at this region. The coarse grid can be used at the region that have comparably simple flow. Generally, unstructured grid is suitable for all geometries. The disadvantage is that because the data structure is irregular, it is more difficult to describe and store them.

For this OBS model, 493315 number of elements was created (Figure 3.5). These number of elements was generated from grid independence study. The average skewness for this number of elements is 0.26, maximum skewness is 0.90. The mesh quality is considered low quality when the maximum skewness is exceeding 0.98. A grid independence study has been performed for OBS computational model. The grid independence study was started at 341961 number of elements. The model was meshed using tetrahedrons elements.

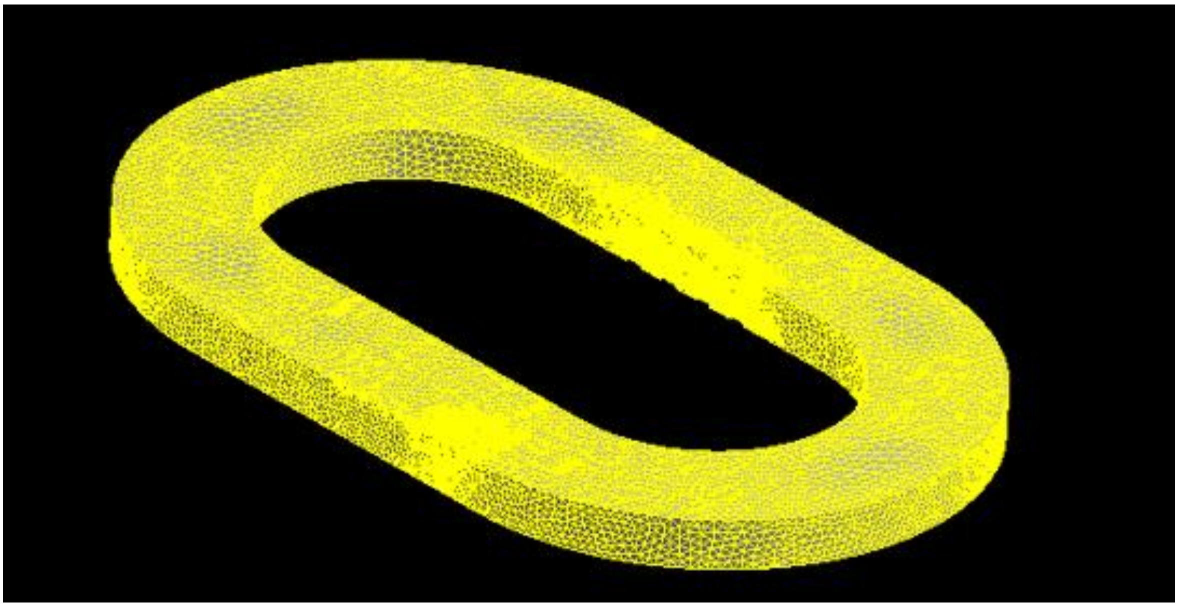

Figure 3.5: 3D grid layout

The grid independence test has been conducted for different size of mesh. As the size of mesh is varies, the number of elements also varies. Table 3.1 show the number of elements and their velocity magnitude in at 4 sampling points, and their analysis is shown in Figure 3.6. Each adaption in meshing gives a different number of elements and variation of velocity magnitude.

The results obtained show that the velocity is increasing at 493315 elements, but decreasing at 583244 elements However, the difference between the velocity tested using different numbers of elements is low. Due to this reason, the number of elements 493315 was chosen for this computational model. The meshing part is crucial to ensure the iteration part after the case set up is not too long. As the number of elements increase, the time of iterations may also increase. Therefore, the number of elements chosen should suit the geometry created. The outcome of this grid independence study shows that the grid developed was independent due to the effects of the grid size. Besides, the grid independency tests were performed to ensure the grid size selection for the computational model is optimal and suitable.

Table 3.1: Grid adaptation (m/s)

Sampling Point	**341961**	**493315**	**583244**
1	0.96	0.97	0.98
2	0.44	0.56	0.48
3	0.22	0.32	0.39
4	0.38	0.41	0.49

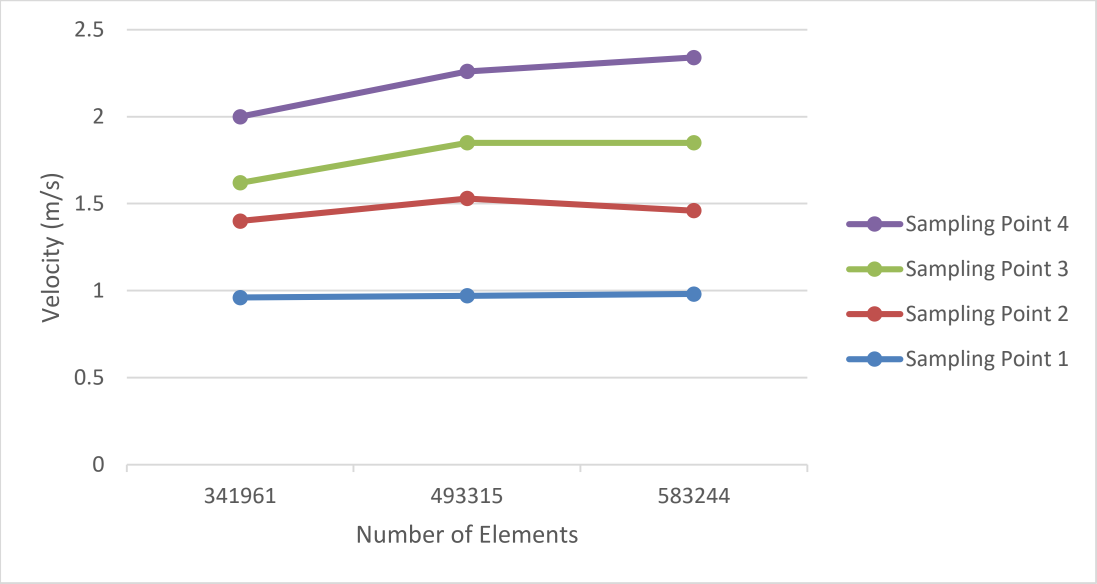

Figure 3.6: Grid Independency Analysis