# NUMERICAL MODELLING OF AIR BUBBLES IN ARTIFICIAL RIFFLE ZONE USING FLOW-3D

**KHOR HAO ZHE**

**UNIVERSITI SAINS MALAYSIA**

**2021**

# NUMERICAL MODELLING OF AIR BUBBLES IN ARTIFICIAL RIFFLE ZONE USING FLOW-3D

**by**

**KHOR HAO ZHE**

**Thesis submitted in fulfilment of the requirements for the diploma degree**

**August 2021**

**Appendix A8**

**SCHOOL OF CIVIL ENGINEERING**
**ACADEMIC SESSION 2020/2021**

**FINAL YEAR PROJECT EAA492/6**
**DISSERTATION ENDORSEMENT FORM**

Title: NUMERICAL SIMULATION OF AIR BUBBLES IN ARTIFICIAL RIFFLE ZONE USING FLOW-3D

Name of Student: KHOR HAO ZHE

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

Signature:

Date : 6/8/2021

Endorsed by:

(Signature of Supervisor)

N

Dr. Puay How Tion
Senior Lecturer
River Engineering and
Urban Drainage Research Centre (REDAC)
Engineering Campus
Universiti Sains Malaysia

D

4/8/2021

Approved by:

(Signature of Examiner)

Name of Examiner: 4/8/2021

ASSOC. PROF. DR. MOHD REMY ROZAINY MOHD ARIF ZAINOL
DEPUTY DIRECTOR
RIVER ENGINEERING & URBAN DRAINAGE RESEARCH CENTRE (REDAC)
ENGINERING CAMPUS, UNIVERSITI SAINS MALAYSIA
14300 NIBONG TEBAL
PENANG

Date:

(Important Note: This form can only be forwarded to examiners for his/her approval after endorsement has been obtained from supervisor)

**ACKNOWLEDGEMENT**

I would like to express my sincerest gratitude to every individual who has assisted me throughout the whole process of this project. First and foremost, I would like to thank my supervisor, Dr. Puay How Tion for inspiring my interest in numerical modeling and providing me the opportunity to carry out this project under his supervision. My upmost gratitude goes out to Dr. Puay for his guidance and encouragement throughout each stage of this project.

Next, I would like to acknowledge Mr. Lim Jia Jun and Mr. Ting Wen Kiat for their continuous support. Sharing of their past experiences has helped me greatly during the entirety of this process. I would also like to thank my friend, Mr Li Bai Hao who is also carrying out his research project under supervision of Dr. Puay for all his assistance and encouragement.

Moving forward, I would like to thank my parents for always being supportive towards my decisions. Without their love and upbringing, I would not be the same person as I stand today. I would also like to thank my siblings who are always there for me when I need them.

Lastly, I would like to apologize if I have missed out any names who have helped me in one way or another throughout this project.

TABLE OF CONTENTS

## LIST OF TABLES

## LIST OF FIGURES

# PEMODELAN NOMBOR GELUMBUNG UDARA DI ZON ARTIFICIAL RIFFLE MENGGUNAKAN FLOW-3D


## ABSTRAK

Struktur Jeram Buatan ialah salah satu daripada banyak kaedah yang digunakan untuk pemulihan sungai. ARS mempunyai pelbagai kelebihan seperti mewujudkan habitat kompleks, mengawal suhu air dan meningkatkan tahap oksigen terlarut (DO) di dalam air. Tahap DO dapat ditingkatkan di dalam air dengan meningkatkan jangka hayat gelembung udara dan isi padu udara teriring. Kajian ini bertujuan untuk mengkaji kesan bentuk permukaan unit ARS dan susunan unit-unit ARS terhadap tahap DO di dalam sungai. Prestasi peningkatan jangka hayat gelumbang udara dan isi padu udara dikaji dari perspektif tiga bentuk permukaan unit ARS yang berbeza iaitu permukaan rata, piramid dan sfera serta dua jenis susunan ARS yang berlainan. Model tiga dimensi (3D) setiap susunan dengan bentuk permukaan yang berbeza dihasilkan melalui perisian AutoCAD-3D. Sebuah perisian perkomputeran dinamik bendalir (CFD) komersial, iaitu Flow-3D digunakan untuk penyelakuan berangka aliran di atas zon ARS. Daripada hasil yang diperoleh, permukaan berbentuk sfera dan susunan ARS jenis kedua menghasilkan jangka hayat gelumbang udara dan isipada udara teriring yang tertinggi. Jangka hayat gelumbang udara tertinggi ialah 9.459 s dan isipada udara teriring tertinggi ialah 0.108024 $m^3$.

# NUMERICAL MODELLING OF AIR BUBBLES IN ARTIFICIAL RIFFLE ZONE USING FLOW-3D

## ABSTRACT

The Artificial Riffle Structures (ARS) is one of the many methods used in river rehabilitation projects. ARS provides various advantages such as creating complex habitats, regulating water temperature and increasing dissolved oxygen (DO) level in water. DO level can be increased in water by increasing the life-time of air bubbles and entrained air volume. This study serves to investigate the effect of surface shape of ARS unit and arrangement of ARS units in increasing DO level in water. The performance of the ARS in increasing the life-time of air bubble and entrained air volume under three different surface shapes of ARS unit i.e. the flat, pyramid and sphere surface and two different arrangements of ARS were investigated in this study. The three-dimensional (3D) model of each arrangement with different surface shape of ARS unit were produced using the AutoCAD-3D. A commercial computational fluid dynamics (CFD) software, i.e. the Flow-3D, was used to simulate the flow over the ARS zone. From the results obtained, the ARS unit with the sphere shaped surface and the second type of ARS arrangement produced the highest life-time of air bubbles of and entrained air volume. The highest life-time of air bubbles obtained was 9.459s and highest entrained air volume obtained was 0.108024 $m^3$.

# CHAPTER 1

# INTRODUCTION

## 1.1 General

Pool and riffles are natural river morphologies which play an important role in creating and supporting the river ecosystem. The pool-riffle sequence creates variability in flow conditions. A wide variety of flow will benefit different speicies of aquatic lives that have different preference for habitats. Studies have also shown that pool and riffle structures promote aeration in rivers which increases the amount of dissolved oxygen (DO). This is due to the increased number of air bubbles from air entrainment will cause an increase in rate of transfer between the air-water phases.

For the past few years, construction and development projects of buildings and infrastructures in Malaysia have significantly increased (Dehdasht et.al, 2021). Although these contributes towards development of a nation, but at the same time it causes destruction to natural habitats, jeopardizing various species of flora and fauna. To reverse these destruction, proper methods of preserving biodiversity can be carried out. One of the methods that was developed by conservationalists to preserve and improve quality of aquatic lives in rivers is by introducing artificial riffle structures (Howe, 1997).

Artificial riffle structure (ARS) is man-made structures that replicates the flow hydrodynamics found in natural pool and riffles. Amongst other river restoration and reclamation techniques, artificial riffle structure is gaining popularity researches on its features and benefits are widely documented. The popularity of this river rehabilitation method can be shown through its wide application in European countries such as Germany and Denmark (Howe, 1997). However, the application of ARS in river rehabilitation is still in its experimental stage, and there is continuous study and research to improve and increase the effeciency of the ARS design.

In Malaysia, the application of the ARS structures for rehabilitation project was piloted by the National Hydraulic Research Institute of Malaysia (NAHRIM). The ARS structures were installed in Sungai Jenderam, Selangor as shown in Figure 1.1.

## 1.2 Problem statement

This study is carried out due to the lack of information on how the surface shape of ARS unit and its arrangement will affect its air entrainment performance. There have been various past studies on how ARS is able to preserve biodiversity in the rivers effectively and sustainably. However, there have been less records on study of effectiveness of ARS in air entrainment. The effeciency of present ARS structures are not optimized as they all share the same structure of alternating rows of higher and lower ARS units. The lack of information might hinder final performance and effeciency of the constructed ARS zone.

Figure 1.1: ARS used in river rehabilitation project in Sungai Jenderam, Selangor

## 1.3 Objective of study

The objective of this study are as follows:

i. To investigate the effect of different surface shape of ARS unit on the residence time of air bubbles in the flow and the amount of entrained air.

ii. To determine the optimal arrangement of ARS unit in increasing the residence time of air bubbles and the amount of entrained air.

## 1.4 Scope of Study

This study focuses on investigating the effect of arrangement and surface shape on the effeciency of ARS in increasing residence time of air bubbles and entrained air in the flow. Numerical simulation will be used to model the flow in ARS zone. Flow-3D, a commercial software used in computational fluid dynamics will be used to carry out this simulation. The flow condition is set according to Froude number. Air bubbles will be injected into the ARS zone at the inflow of the domain to mimic the condition of air bubbles produced at upstream part of the ARS zone installed in the river. Residence time of the air bubbles in the flow will be recorded for different surface shape of ARS unit and different ARS arrangement.

## 1.5 Thesis Outline

This thesis is seperated into five chapters. Chapter one provides a brief introduction on the background of artificial riffle structures and its importance. The scope and objective of this study is also mentioned in this chapter.

Chapter two discusses about the literature review that has been done for this study. Background information about natural pool and riffles, artificial riffle structures, dissolved oxygen level, computational fluid dynamics and its governing equation are included in this chapter.

Chapter three covers the methodology used to achieve the objectives in this study. The procedures and details of the mesh convergence test, verification of numerical model, creation of three dimensional model and setup of numerical model for simulation are all explained in this chapter.

Chapter four presents the results obtained from this study. These include the results from mesh convergence test, verification of numerical model and simulation of flow over artificial riffle zone. The results for the simulation of flow over artificial riffle zone is discussed based on its efficiency in increasing life-time of air bubbles and increasing total air entrainment volume.

Chapter five serves as a summary for the outcomes obtained in this study and a conclusion is drawn based on it. Recommendations for future studies related to this study are also given.

# CHAPTER 2

# LITERATURE REVIEW

## 2.1 Natural Pool and Riffles

Pool and riffles are bedform undulations with deep and shallow regions. Pool and riffles are common morphologies found in rivers and they contribute towards preservation of biodiversity and quality of stream habitat. Pools are the deeper regions with slower and calmer flow. Riffles are the shallower regions with faster and more turbulent flow (Figure 2.1).

The variety in flow condition provides different microhabitats for aquatic organisms. The aquatic species chooses their microhabitat depending on abiotic conditions such as temperature, light intensity and dissolved oxygen. For pools, the depth it provides serves as refuge for aquatic life during dry conditions. Pools also offer protection against predators. The slower water flow in pool also allows settlement of organic debris as a food source for certain species of fishes. Riffles on the other hand are food source for most predator species. Some aquatic species wait for weaker preys to be carried to them through the rapid flow in riffles (Thompson, 2018). It works in the same manner as a conveyor belt, brining food to the predators. The only species that can survive in this harsher macrohabitat are animals that can cling to rocks very well. These animals such as caddisflies and stoneflies feed on bacteria and fungi on leaves that are carried along the flow in riffles (Cave, 1998).

Pool and riffle structures are also known to promote aeration in rivers. Pools mostly have slow and calm water which may lead to stagnant water and having low aeration. Riffles on the other hand have rapid and turbulent flow which promotes aeration through air entrainment at the water surface.

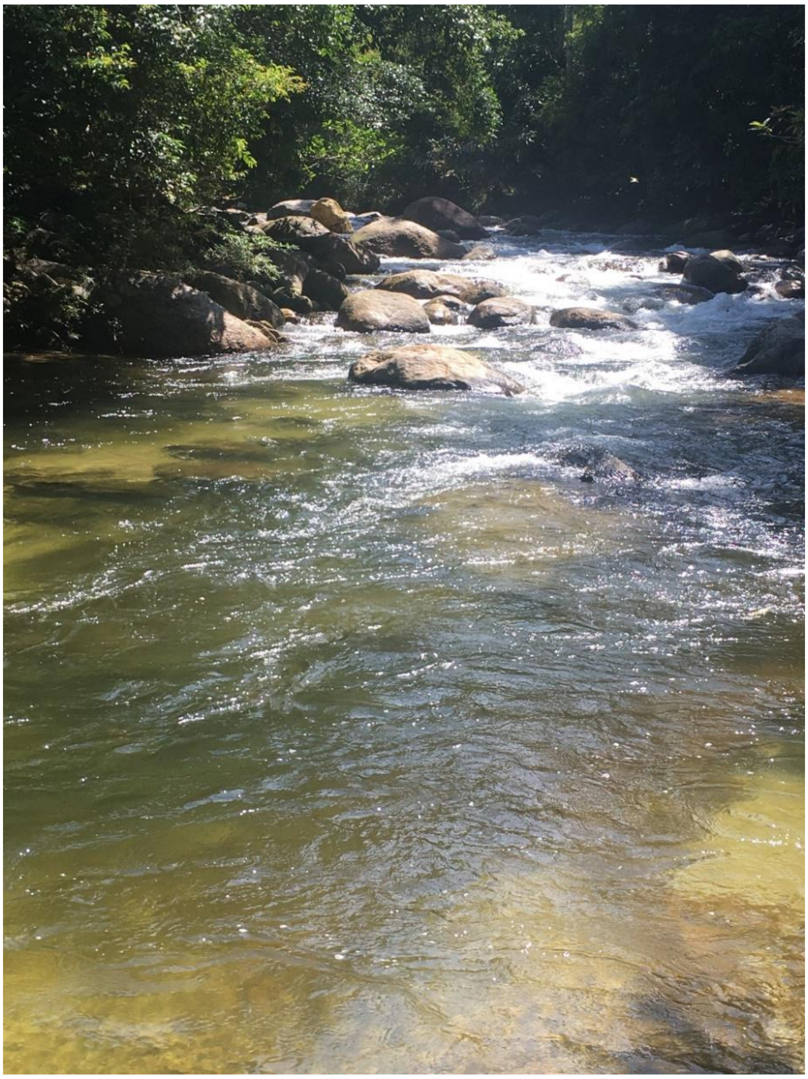

Figure 2.1: Natural pool and riffles found in Sungai Sedim, Kedah

## 2.2 Dissolved Oxygen Level

The level of dissolved oxygen in water can be increased through two mechanism, i.e. air entrainment and injection of air bubbles into water body.

### 2.2.1 Air entrainment

Generally, there are two ways for air entrainment to take place. The first way is through surface entrainment which air are entrained into the water due to turbulence on the water surface as depicted inFigure 2.2. The second way is through localized entrainment such as in hydraulic jumps or plunging jets at weirs as depicted in Figure 2.3.

For riffles, the air is mostly entrained through surface entrainment. The turbulent water surface will continuously renew the surface film of water and increase difference in oxygen concentration between air and the water surface (Tucker, 2015). This will lead to an increase in rate of diffusion of oxygen between air and water as depicted in the equation below:

$$\frac{dC}{dt} = K_L \frac{A}{V} \left(C_S - C_m\right) \tag{2.1}$$

where $C$ is the oxygen concentration, $C_s$ is the dissolved oxygen concentration when wtaer is saturated with oxygen, $C_m$ is the measured dissolved oxygen concentration, $K_L$ is the liquid -film coefficient and $\frac{A}{V}$ is the ratio of the air-water interfacial area ($A$) to water volume ($V$). Turbulence also tends to break the water surface, hence trapping air and bringing it back to the water (Souders and Hert, 2004).

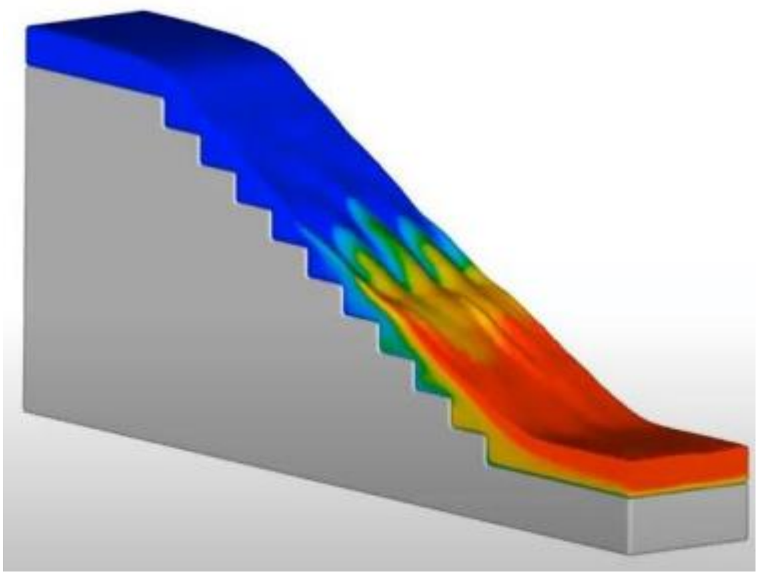

Figure 2.2: Air entrainment through turbulent on free surface (FLOW-3D, 2021)

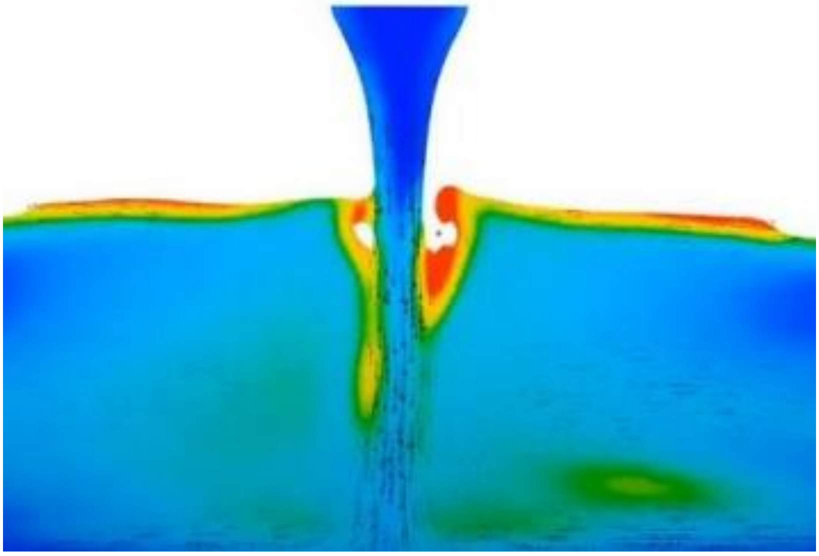

Figure 2.3: Localized air entrainment by plunging jet (FLOW-3D, 2021)

### 2.2.2 Air bubbles

The usage of fine bubble aeration in water treatment facilities to promote aeration has been prominent for the past few decades (Brooks and Imboden, 1992). Oxygen transfer occurs via diffusion from regions of higher concentration to regions of lower concentration through the surface of air bubbles (Tucker, 2005). It has been proven that this method is able to effectively increase oxygenation performance and hence increasing DO in water (Favolle et.al, 2007). The higher the residence time of air bubbles in the flow, the greater the rate of oxygenation as there is a higher chance for mass transfer across bubble surface to take place (Navisa et.al, 2014). Hence, in this study, the residence time of air bubbles in the flow will be used to measure the efficiency of ARS in increasing DO in the water.

## 2.3 Artificial Riffle Structures (ARS)

Artificial riffle structures are developed by conservationalists and river engineers for river rehabilitation projects. In the past few decades, there have been various studies done on analyzing and modeling of the hydrodynamic processes in pool and riffles, hence the information obtained were used to create models to describe pool-riffle morphodynamics (Almeida, 2011). From the conceptual models, artificial riffle structures that replicate the function of natural pool and riffles were constructed. Due to its ability to recreate hydrodynamic viability of natural pool and riffles, ARS has then been use widely in river restoration projects (Rodriguez and Garcia, 2013).

Natural pool and riffles are usually arranged in sequence with pools having a skewed center at its deepest point. (Rodriguez and Garcia, 2013). Hence, ARS are usually modelled in a similar manner as natural pool and riffles (Figure 2.4). Rows of man-made structures at a designed height are constructed and placed into the river to replicate natural riffles, the region in between the rows will serve as the pool.

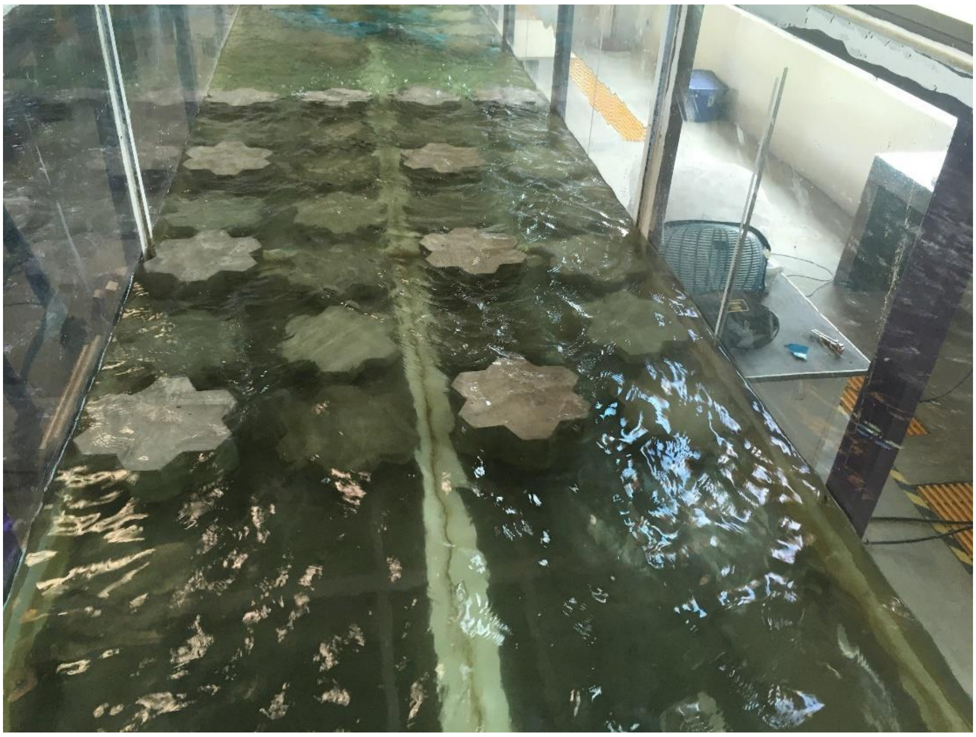

Figure 2.4: Artificial Riffle Structures (ARS) modelled in similar fashion to natural pool and riffles.

## 2.4 Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is a modern mathematical method used in modelling fluid flow cases and solve the problem numerically with computers. One of the main part of CFD involves substituting the governing equation for fluid flow, which is cast in partial differential form with numbers. The result obtained through the manipulation of equations will be a collection of numbers. Numbers are very crucial in any engineering analysis, as in most cases, the objective is to acquire the quantitative description of the problem (Anderson et al., 2009).

The development of CFD is indeed contributed by the advancement of high-speed digital computers since 1950s (Chung, 2009). CFD were initially used for simpler flow geometry for inviscid flows, which is usually termed as the first generation of this method. The second generation of CFD solutions are those that are applied today, which most of the time involves complex fluid dynamic problems that are inherently not solvable without utilizing computational prowess. For instance, problems involving mixed subsonic-supersonic flows or viscous flows are harder to solve as they are not applicable to the boundary layer approximations. The significance of CFD in modern day engineering predictions in fluid dynamics as it can be seen to be as important as pure experiment and pure theory (Anderson et al., 2009). The beauty of CFD lies within its ability to support and complement both pure experiment and pure theory.

There are traditionally two main forms of CFD methods, which are the finite difference method (FDM) and finite element method (FEM). However, in recent years, finite volume method (FVM), which is formulated from FDM or FEM has been used more extensively. Formulations and methods for CFD are constantly improvised and developed until today in terms of FDM, FEM and FVM (Chung, 2009).

## 2.5 FLOW-3D

FLOW-3D is a Computational Fluid Dynamics modelling software which is well known for in the advancement of three dimensional volume of fluid (VOF) algorithm and is frequently used in free surface flow analysis. It is also known for its ability to provide detailed analysis on even small-scaled simulations (Hu et al., 2018).

In comparison to other CFD softwares such as OpenFoam or Mike 3FM, Flow-3D is considered to be more user friendly. The workspace in the software is able to guide its user step by step through each phase of modelling easily. It is also able to create solid boundaries and develop mesh blocks with ease. The FAVOR method provided by FLOW-3D also allows modeling for more complex geometric regions.

## 2.6 Governing Equations

The incompressible three dimensional continuity and Navier-stokes equation are used to solve motion of fluids in Flow-3D. The equations are denoted in Einstein's summation convention form as shown below:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{2.2}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + g_i + \frac{1}{\rho}\frac{\partial \tau_{ij}}{\partial x_j} \tag{2.3}$$

$$\tau_{ij} = 2\mu S_{ij},\ \ S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \text{with } (i, j = 1,2,3) \tag{2.4}$$

Where $u$ is the velocity, $g$ is the gravitational acceleration, $\rho$ is the fluid density, $P$ is the pressure and $\tau$ is the deviatoric stress. The two-equations RNG k-epsilon turbulence model is used to model the turbulence stress. In the RNG k-epsilon model, the turbulence diffusion coefficient is evaluated explicitly as compared to the standard k-epsilon model (where the turbulence diffusion coefficient is evaluated based on the specified turbulence length scale). The RNG model has wider application as it can describe low intensity turbulence.

### 2.7 Volume of Fluid (VOF) Method

Volume of Fluid (VOF) Method is used in FLOW-3D for free surface configuration (Hirt, 1993). The fluid fraction in a each cell in the staggered mesh system is defined as a function *F*. When the cell is fully filled with water, the value of *F* will be 1.0, when the cell is void, the value of *F* will be 0 (Barkudarov, 2004). If the value of *F* is in between 0 and 1, it indicates that the cell is only partially filled, which means there will be free boundary surface. Figure 2.5 depicts the condition where cells are either completely filled, partially filled or void.

The advection equation for VOF function *F* is as follow:

$$\frac{\partial F}{\partial t} + u\frac{\partial F}{\partial x} + v\frac{\partial F}{\partial y} + w\frac{\partial F}{\partial z} \tag{2.5}$$

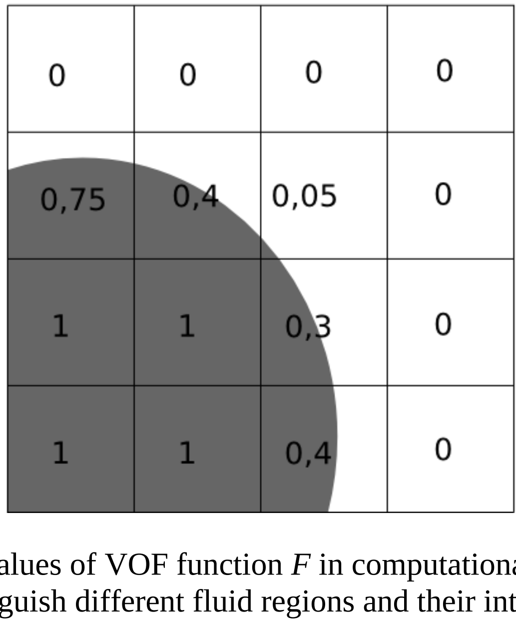


Figure 2.5: The values of VOF function *F* in computational cells are used to distinguish different fluid regions and their interface

## 2.8 Fractional Area Volume Obstacle Representation (FAVOR) method

FAVOR method is used in representing obstacles in the domain in FLOW-3D (Hirt, 1993). FAVOR's advantage lies within its ability to model complex geometrical structures. FAVOR method can be used for both structured or unstructured grid. However, structured and rectangular grids are usually preferred to be used as the fractional areas and volumes are easily computed. For incompressible, inviscid fluids, FAVOR equation can be written as follows:

$$\frac{\partial V}{\partial t} + \nabla \cdot (\boldsymbol{Au}) = 0 \tag{2.6}$$

where $A$ is the open area fraction, $V$ is an open volume fraction and $\boldsymbol{u}$ is the velocity vector. When a cell is fully occupied by an obstacle, the function $V$ will be zero. When the cell contains boundary of an obstacle, it indicates function $V$ will have a value between 0 to 1.

## 2.9 Dam Break Flow Problem

Dam break flow problem is commonly known as a benchmarking problem in the determination of accuracy and viability of a numerical model. (Ji et al., 2013). It is widely used for tests for computations that involves multiphase flows. (Kawara et al., 2011). This is due to the fact that dam break flow problems are widely documented with proof of experimental data available. Dam break flow is also considered as a challenging free-surface problem to be simulated. (Arnold, n.d.). Hence, verification of a numerical model may be done by assuring the results produced comes to an agreement with the experimental data.

## 2.10 Summary

This chapter discusses all the background information on natural and artificial pool and riffles alongside concepts and factors affecting dissolved oxygen level. The ltierature studies on computational fluid dynamics and its governing equations have provided a deeper understanding on the basics on how commercial CFD software operates. Literature studies on Flow-3D and its relevant configuration methods granted a more solid reason to choose it as the software to be used for numerical simulations in this study.

# CHAPTER 3

# METHODOLOGY

## 3.1 Introduction

This chapter discusses the research methodology of this study. The research methodology flow chart is shown in Figure 3.1. Firstly, literature review has been done as discussed in Chapter 2. Section 3.2 discusses the verification of the numerical model of with a three-dimensional dam break flow problem. This dam break flow problem is used only for verification purposes. The development of ARS models will be described in Section 3.3. Section 3.4 presents the setup of the numerical model for the simulation of flow over the ARS zone using Flow-3D.

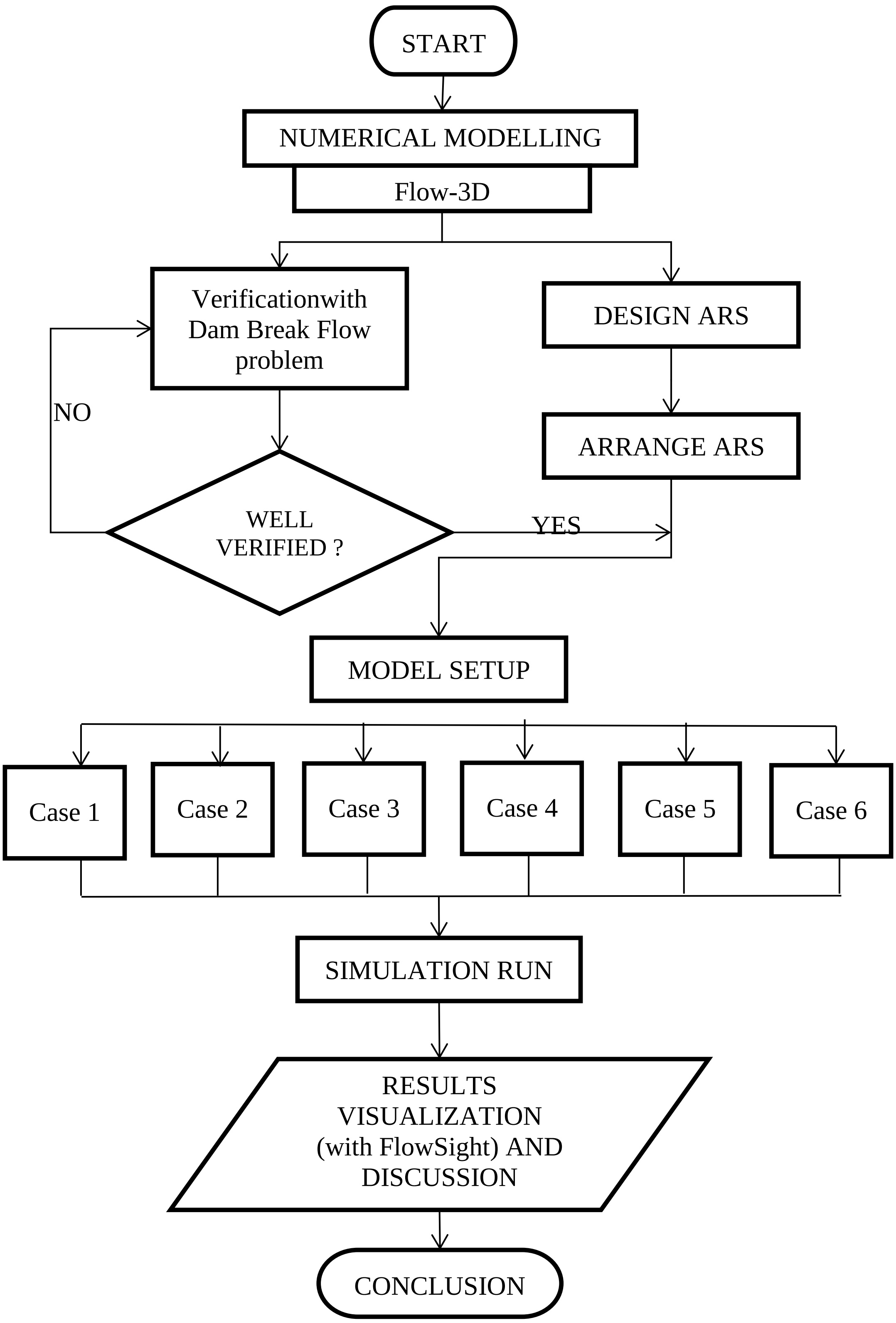


Figure 3.1: Research Methodology

### 3.2 Verification of Numerical Model

Viability of Flow-3D in developing three-dimensional numerical models is first verified before carrying out actual simulation of this study. Verification is necessary to prove that Flow-3D is capable of developing satisfactory numerical models. Secondly, this verification ensures that a free surface model is properly setup before it is used for the simulation of flow over ARS zone. For this study, verification of numerical model is done with three-dimensional dam break problem. A numerical model was setup with Flow-3D to simulate the three-dimensional dam break flow problem. Verification for the Flow-3D model was carried out by comparing to the numerical results with the results obtained from the experiment carried out by Lobovský in 2014. (Lobovský, 2014).

#### 3.2.1 Setup of Three Dimensional Dam Break Flow Problem

The fluid selected for the simulation is water at 20°C. Properties of this type of fluid defined defaultly by Flow-3D is shown in Figure 3.2. Two physics model were considered for the simulation. Gravity and non-inertial reference frame model was used to compute and include gravitational force in the simulation. Viscosity and turbulence model was used to define the type of flow and the model used for modelling turbulent fluctuations. K-epsilon model was selected for the simulation. Settings for the physics model are shown in Figure 3.3 and Figure 3.4. The simulation was setup in a domain with the size of 1.61m × 0.15m × 0.6m. Non-conforming Cartesian mesh with cell size of 0.02m were used for the simulation. Initial fluid region is also introduced with parameters defined as shown in Figure 3.5. Figure 3.6 shows the final meshing and geometry setup for the model, where red lines denote the outlines for boundary of the domain and the fluid region is represented by the cyan colored region. The settings of the simulation parameter, initial and boundary conditions are summarized in Table 3.1.

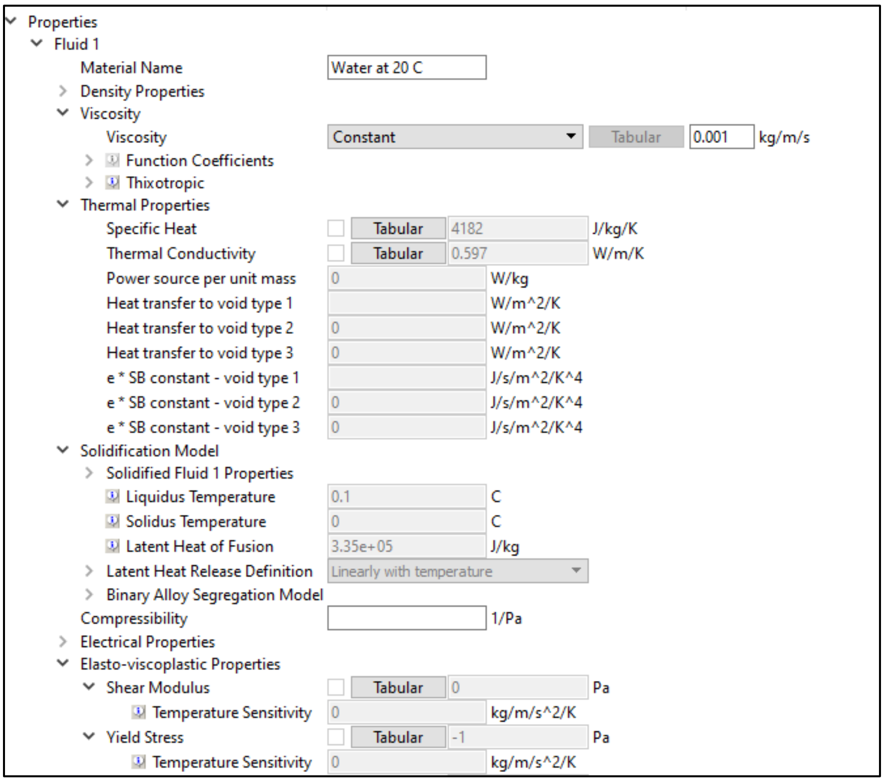


Figure 3.2: Fluid properties for water at 20$^{\circ}$C

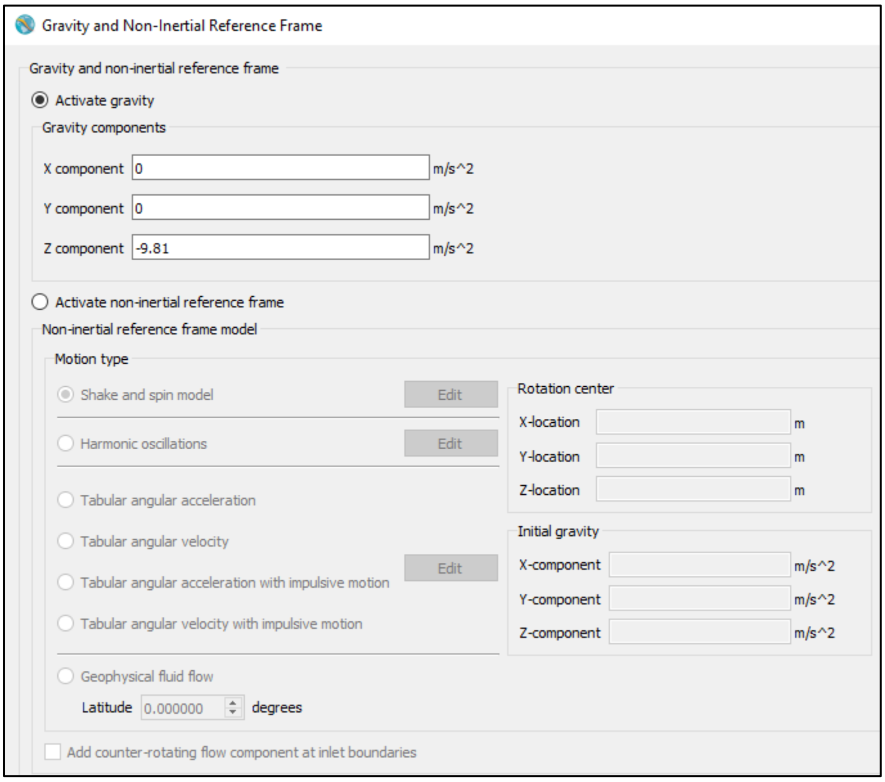


Figure 3.3: Gravity and Non-Inertial Reference Frame model settings

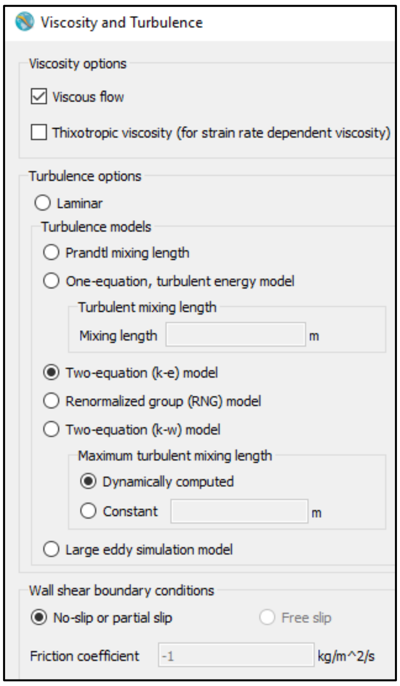


Figure 3.4: Viscosity and Turbulence model settings

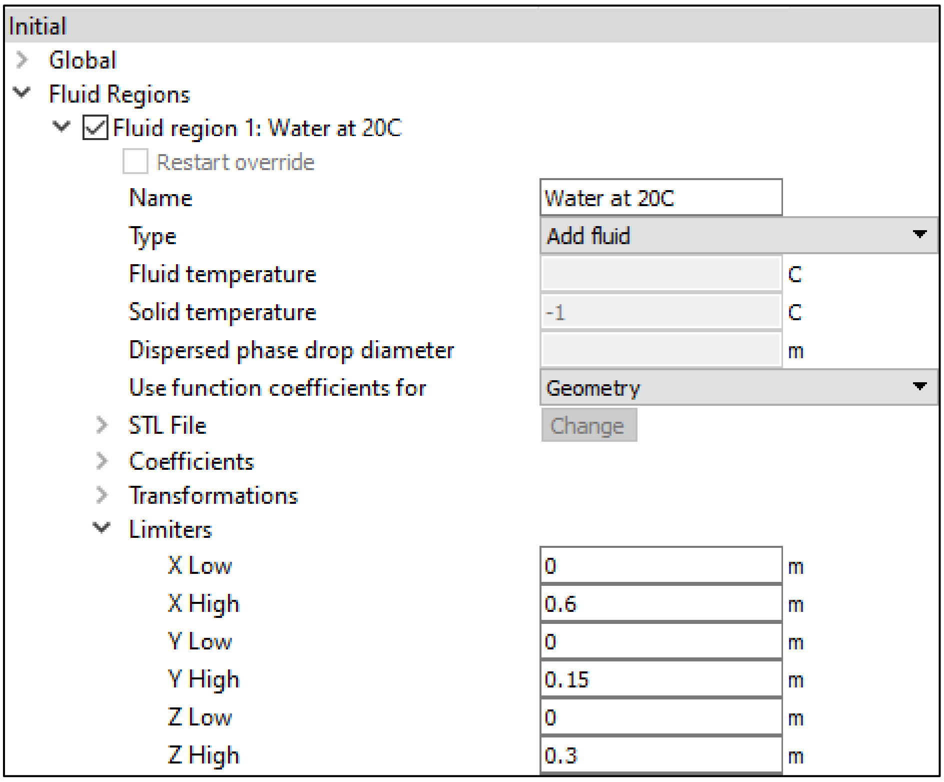


Figure 3.5: Parameters for initial fluid region

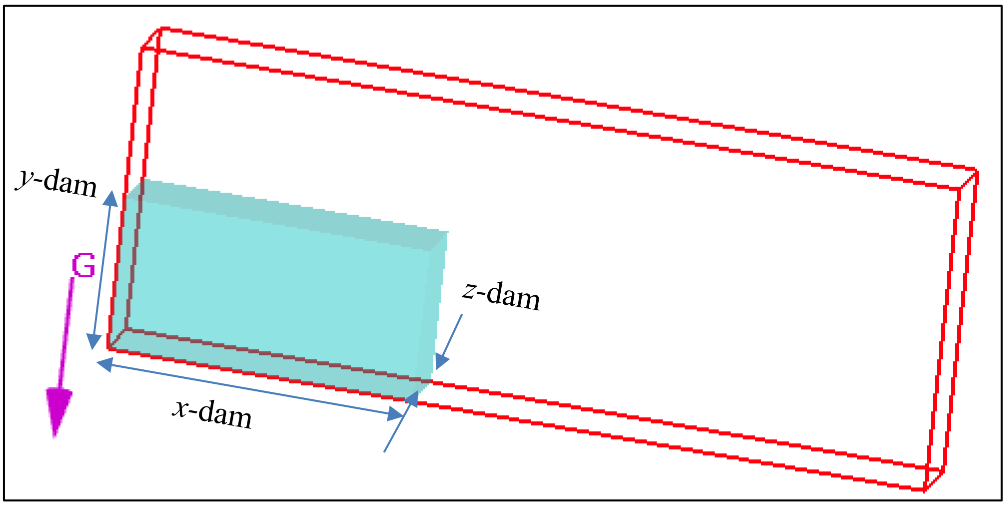


Figure 3.6: Final meshing and geometry setup

Table 3.1: Simulation parameter and the initial and boundary condition for the three-dimensional dam break flow problem

| | |
|---|---|
| 1. Gravitational acceleration | |
| In x-direction, $g_x$ | 0 |
| In y-direction, $g_y$ | 0 |
| In z-direction, $g_z$ | -9.81 $ms^{-2}$ |
| 2. Mesh size | |
| Mesh size in *x*-direction | 0.02 m |
| Mesh size in *y*-direction | 0.02 m |
| Mesh size in *z*-direction | 0.02 m |
| 3. Domain size | |
| Length of domain in *x*-direction | 1.61 m |
| Length of domain in *y*-direction | 0.15 m |
| Length of domain in *z*-direction | 0.6 m |
| 4. Initial condition | |
| Length of water reservoir, *x*-dam | 0.6 m |
| Length of water reservoir, *y*-dam | 0.15 m |
| Length of water reservoir, *z*-dam | 0.3 m |
| 5. Boundary condition | |
| All walls, ceiling and floor | Zero normal velocity on the wall ( no inflow or outflow and non-slip type. |

## 3.3 The 3D model of Artificial Riffle Structures

Artificial Riffle Structures (ARS) units are drawn and prepared using AUTOCAD 3D software. The shape of the base of each unit is modelled based on NUCLeuS ARS model developed by National Hydraulics Research Insitute of Malaysia (NAHRIM) as shown in Figure 3.7. Two different heights of ARS were used as shown in Figure 3.8, where the red colored ARS is 5 cm and the yellow colored ARS is 15 cm.

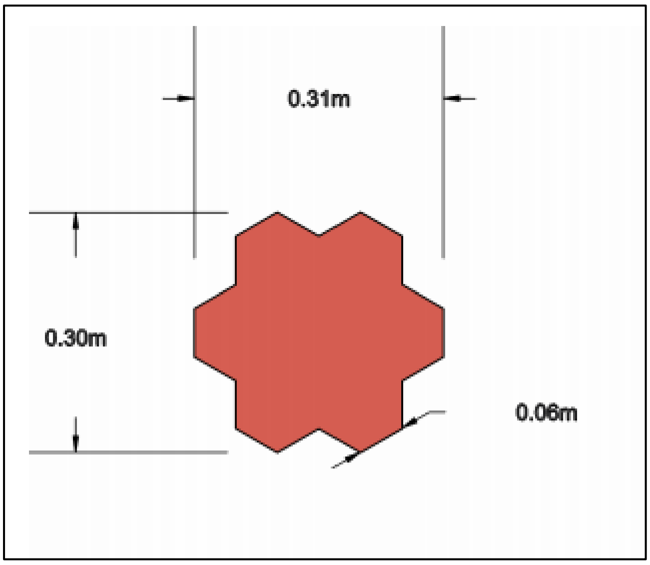


Figure 3.7: Base shape of a single ARS unit

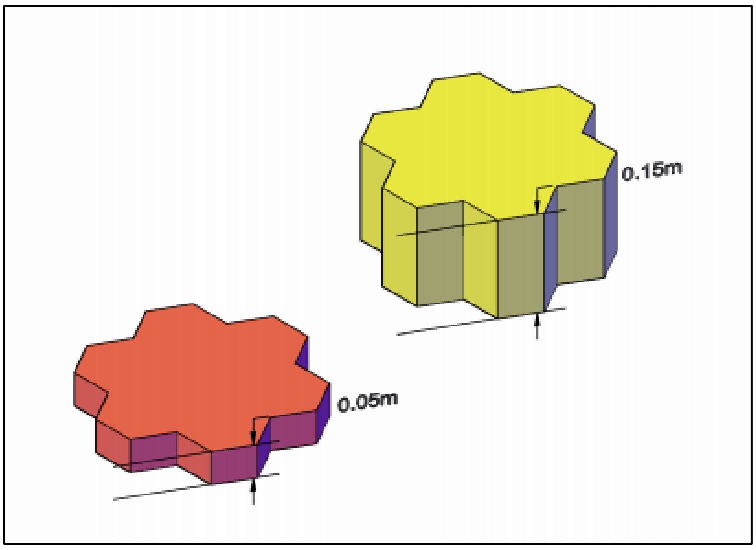


Figure 3.8: Two different ARS heights

### 3.3.1 Arrangement of Artificial Riffle Zone

The ARS units are then arranged in 2 distinct manners in a rectangular domain to form 2 different artificial riffle zones. The plan view of the proposed arrangements are shown in Figure 3.9 and Figure 3.10. Arrangement 1 is modelled based on natural pool and riffle formations but arranged in a more orderly way. Arrangement 2 is modelled based on J-hook vane formations in rivers. J-hook vanes are chosen as the river morphology to be based on due to its feature to form a pool in its concave section after water flows over it. Multiple models of this feature are then arranged

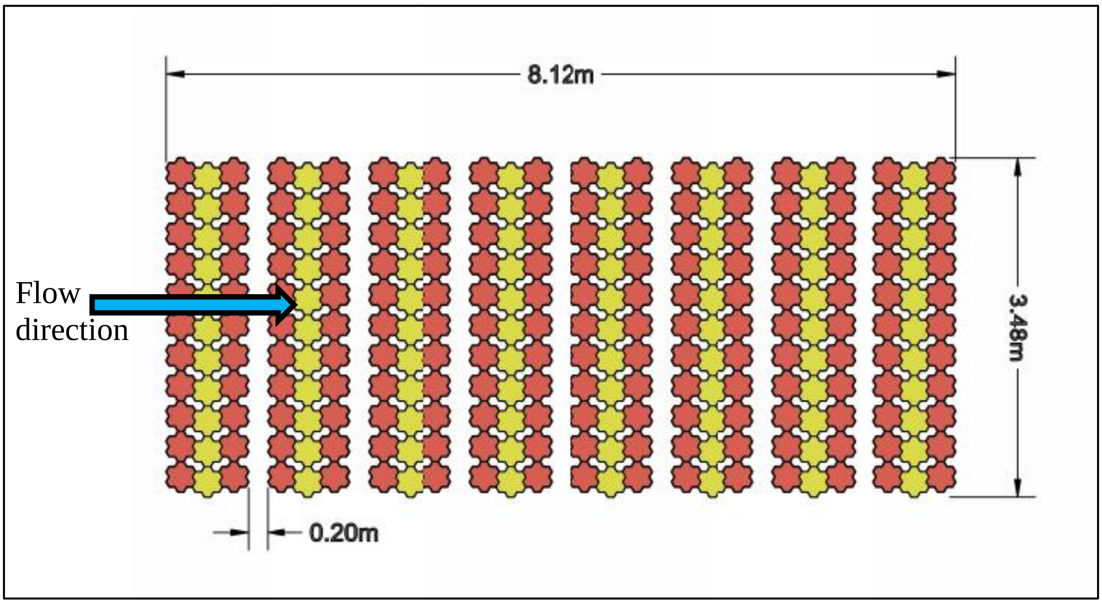


Figure 3.9: Arrangement 1 in artificial riffle zone

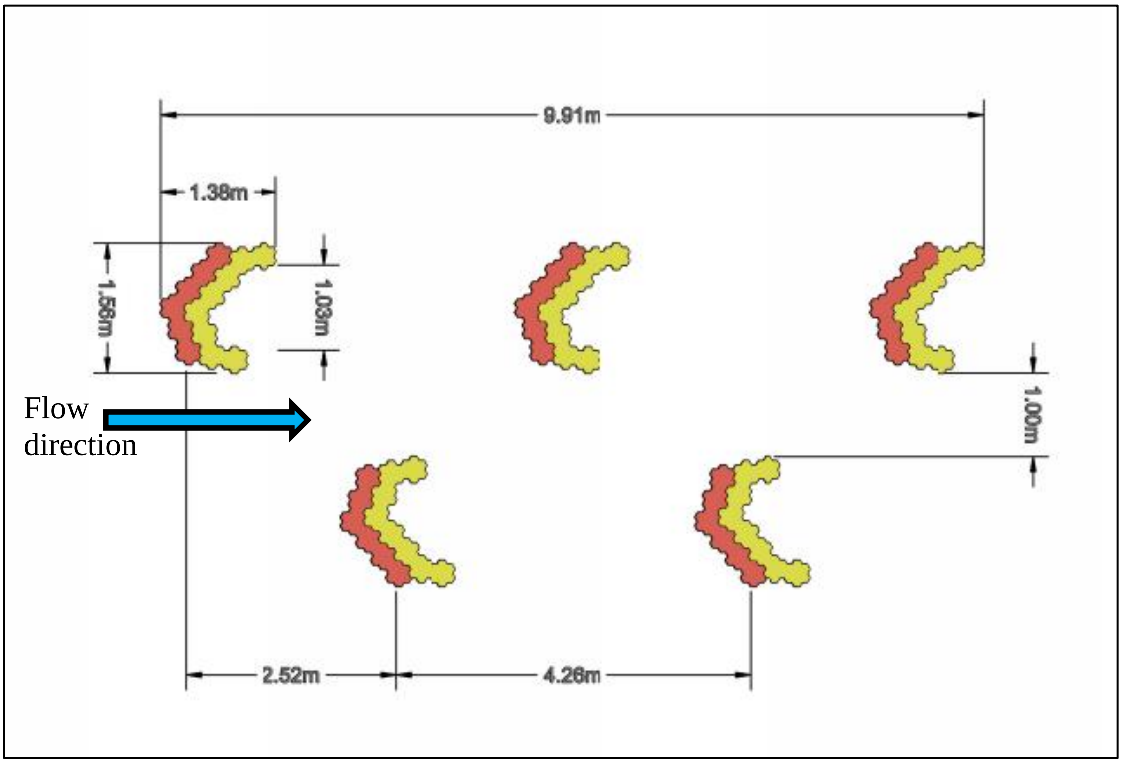


Figure 3.10: Arrangement 2 in artificial riffle zone

### 3.3.2 Surface Shape of Artificial Riffle Structure Unit

Three different surface shape of ARS unit were also considered for each arrangement. The surface shape considered are flat surface, pyramid surface and spherical surface. The top view and side view for each surface shapes are shown in Figure 3.11, Figure 3.12 and Figure 3.13.

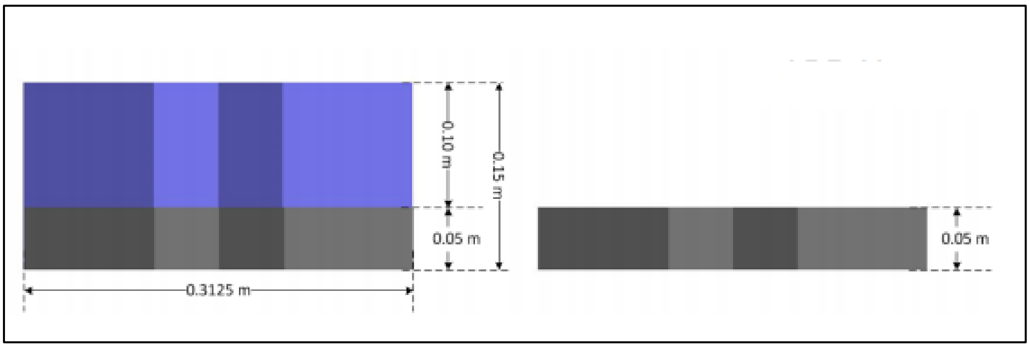


Figure 3.11: Top and side view of flat surfaced ARS unit

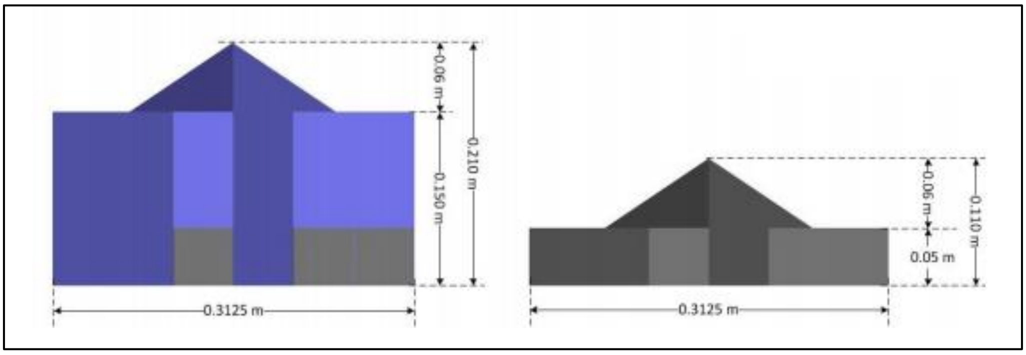


Figure 3.12: Top and side view of pyramid surfaced ARS unit

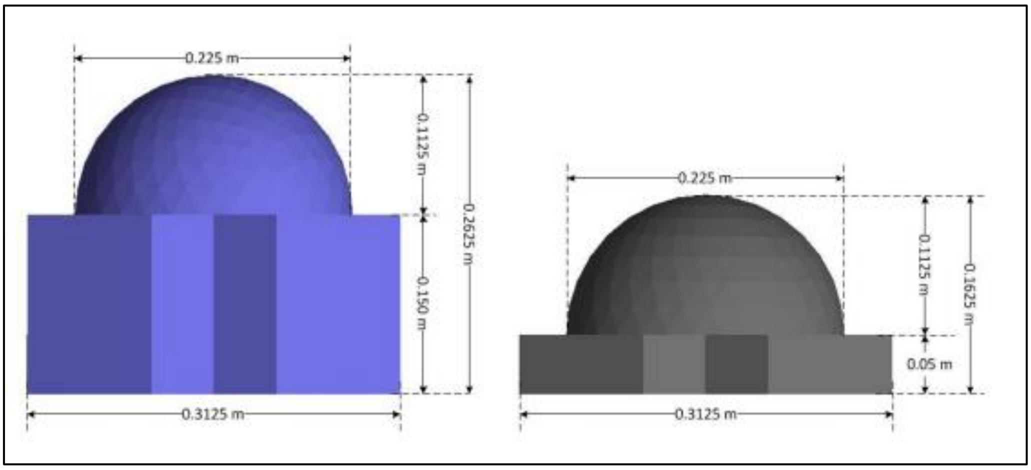


Figure 3.13: Top and side view of spherical surfaced ARS unit