A STUDY ON THE FLOW CHARACTERISTIC IN THE ARTIFICIAL RIFFLE ZONE

LI, BAI HAO

SCHOOL OF CIVIL ENGINEERING
UNIVERSITI SAINS MALAYSIA

2021

A STUDY ON THE FLOW CHARACTERISTIC IN THE ARTIFICIAL RIFFLE ZONE

by

LI, BAI HAO

Thesis submitted in fulfilment of the requirements for the honours degree of Civil Engineering

August 2021

ACKNOWLEDGEMENT

We would like to express our gratitude to USM for giving us the opportunity and every teacher who has been imparting knowledge. Very special gratitude goes out to my dissertation supervisor - Dr. Puay How Tion, for his guidance, encouragement, constructive criticism throughout my whole research work. Support to my study, and concern for my life. He deserves my greatest gratitude. Moreover, in completing the dissertation, I had to take the help and guideline of some respected persons, thanks to my senior Lim Jia Jun, Ting Wen Kiat, and classmate Khor Hao Zhe from the hydraulic REDAC. Last but not least, I would like to thank all examiners for their help. Thank you for the tremendous positive feedbacks given along with the continuous support received.

TABLE OF CONTENTS

LIST OF TABLES

Page

LIST OF FIGURES

Page

LIST OF ABBREVIATIONS

ARS	Artificial Riffle Structure
CFD	Computational Fluid Dynamics
DO	Dissolved Oxygen
FAVOR	Fractional Area-Volume Obstacle Representation
TKE	Turbulent Kinetic Energy
VOF	Volume Of Fluid

ABSTRAK

Dengan perkembangan pesat dunia termasuk Malaysia, banyak sungai semula jadi telah dimusnahkan. Walaupun pemodenan bandar telah membawa perubahan pada persekitaran semula jadi, sungai yang rosak boleh dipulihkan dengan menggunakan struktur jeram buatan. Struktur jeram buatan dapat membantu mengembalikan bentuk sungai, kualiti air dan habitat akuatik. Maka, dalam tesis ini, ciri-ciri aliran di zon struktur jeram penggunaan struktur jeram buatan akan dikaji dengan permodelan berangka. Pemodelan berangka ialah kaedah yang sesuai digunakan untuk mengkaji struktur aliran air di zon jeram buatan. Sebagai perisian simulasi berangka yang boleh dipercayai, perisian komersial Flow-3D digunakan untuk mensimulasikan aliran di zon jeram buatan. Simulasi dengan menggunakan nombor Froude aliran yang berbeza digunakan untuk menilai ciri-ciri tenaga kinetik aliran gelora di dalam aliran di zon jeram buatan. Hasil simulasi berangka menunjukkan perbezaan ketara tenaga kinetik aliran gelora di paras kedalaman air yang berbeza di dalam zon jeram buatan.

A STUDY ON THE FLOW CHARACTERISTIC IN THE ARTIFICIAL RIFFLE ZONE

ABSTRACT

With the rapid development of the world as well as in Malaysia, many natural rivers have been destroyed due to urbanisation. Urban modernisation has brought changes to the natural environment; therefor'e, people should ameliorate the damaged river using artificial riffle structures. The artificial riffle structure helps restore the shape of the river, water quality, and aquatic habitat. Therefore, in this thesis, the characteristics of flow in the artificial riffle zone are studied with numerical modeling method. Numerical modeling is a convenient tool for studying flow in artificial riffle zone. As a reliable commercial software, the Flow-3D model is used to simulate flow over the artificial riffle zone. The simulation was carried out with different flow Froude numbers to evaluate the turbulent kinetic energy in the artificial riffle zone. Numerical simulation results show that the is distinct difference in the turbulent kinetic energy at different flow depths in the artificial riffle zone.

CHAPTER 1

INTRODUCTION

1.1 Background

Riffle is an area of stream characterised by the deposits in the riverbed below the surface of the water, as shown in Figure 1.1. Due to the change of the riverbed flow velocity, the erosion and accumulation of the water flow alternate, resulting in the longitudinal profile of the riverbed to undulate with riffle and pool alternately distributed along the river as shown in Figure 1.2. The accumulation part is the riffle, and the eroded part is the pool. In the riffle zone, the water depth is shallow the water flows faster. The pool is characterised by deep water, and the water flow is slow.

The most developed section of the riffle is in the wide riverbed or near the mouth of a tributary estuary. Here, due to the slowing of the water flow, the sediment is easy to silt up, often resulting in a riffle.

The pool-riffle sequence has some features: promote aeration, increase Dissolved Oxygen (DO) level; provide shelter for aquatic living things; the pools act as a temperature regulator; and the pool has self-cleansing ability. It will be elaborated in the following chapter.

ARS is an acronym for Artificial Riffle Structure. The ARS is widely used material in river restoration. The ARS units are arranged in the river to create an artificial riffle zone. A typical ARS unit is shown in Figure 1.3 The presence of ARS increases the overall channel bed roughness and the flow characteristic. Therefore, the understanding of the flow characteristic, such as the turbulent kinetic energy (TKE) is important for design purposes to prevent unwanted erosion and sediment deposition in the artificial riffle zone.

Figure 1.1: Natural riffle in stream

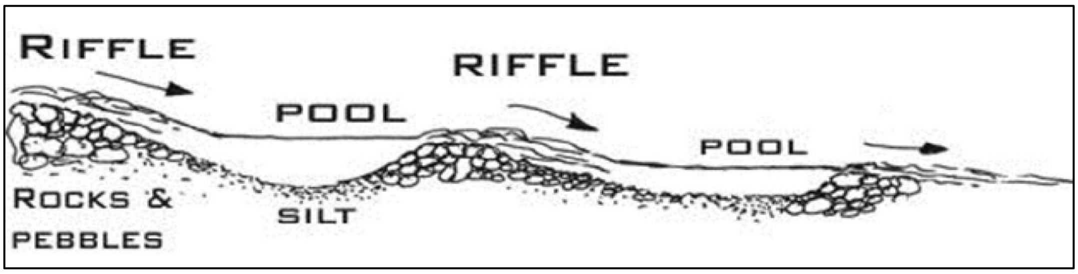

Figure 1.2: A cross-section of a stream and the riffles and pools that have formed in its bed

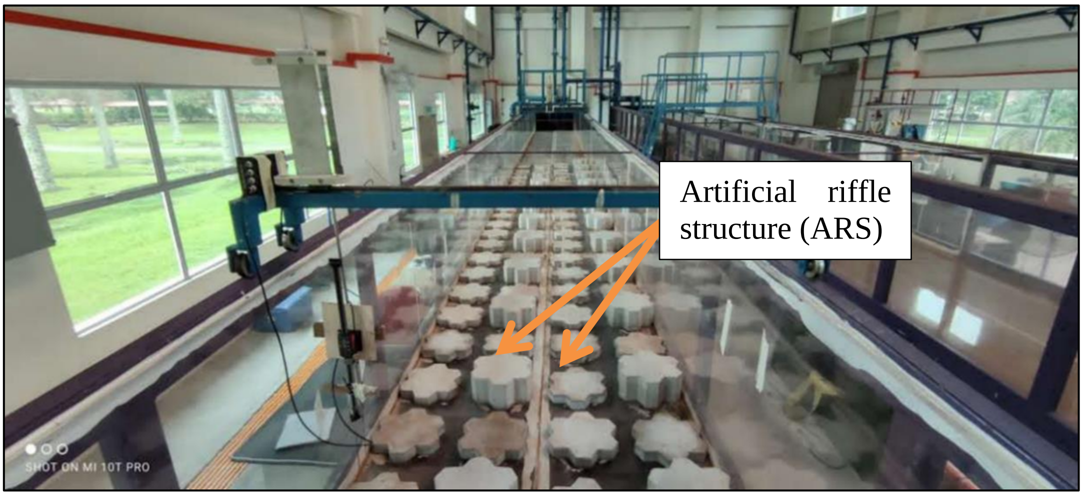

Figure 1.3: ARS units arranged in the hydraulic laboratory flume

1.2 Problem Statement

With the development of human society, rivers are changed intentionally or unintentionally. The channelisation of the river has changed the basic form of the meandering of the river. The pattern of rapids, slow flows, bends, and shoals have disappeared, and the geometric regularisation of the cross-section has also changed the situation of pool and riffle. The structure and function of the system will change accordingly, especially the decrease of the biodiversity of the biological community, which may cause the degradation of the freshwater ecosystem.

Studies on trends in the fauna of freshwater fish worldwide reveal that most of the fauna are severely declining and require immediate protection. The species most likely to be endangered either live exclusively in large rivers or are endemic species with extremely small distributions. It is conservatively estimated that 20% of the world's freshwater fish (about 1,800 species) have become extinct or severely reduced. (Moyle. and Leidy, 1992)

In order to improve the quality of the stream habitat and the local biodiversity, Artificial Riffle Structure (ARS) is used to restore the ecosystem to a certain extent.

Artificial Riffle Structures (ARS) are installed in the river for rehabilitation purposes. However, the presence of ARS changes the flow characteristics, especially the turbulent kinetic energy. The amount of TKE will affect the sediment transport process in the artificial riffle zone and at the upstream and downstream of the artificial riffle zone. Therefore, the flow characteristic, especially the TKE in the artificial riffle zone, is investigated in this study important for design purposes.

1.3 Objectives

The objectives of this study are as follows:

To investigate the effect of Froude number on the turbulent kinetic energy in the artificial riffle zone.

To establish the relationship between Froude number and turbulent kinetic energy in the artificial riffle zone.

1.4 Scope of Work

In this study, the effect of inflow Froude number towards the turbulent kinetic energy in the artificial riffle zone is investigated by using a three-dimensional numerical model. The scope of work is shown as follows:

a) Numerical simulation

Numerical simulation is carried out by using Flow-3D, proprietary software to solve the free surface flow problems. Continuity and Navier-Stokes equations are solved in the numerical simulation to measure the TKE in the flow over the Artificial Riffle zone.

b) Verification of numerical model

Verification of numerical model is done by comparing the flow front of the numerical solution with the experimental result done by Lobovský et al., 2014 for two-dimensional dam-break flow problem.

c) Range of Froude number

In this study, flow over artificial riffle zone will be investigated in the range of Froude number from $Fr = 0.4$ to $Fr = 1.2$. The inflow Froude number, Fr is defined as $Fr = \frac{V}{\sqrt{gh}}$, where V is the flow velocity, g is the gravitational acceleration and h is the flow depth. For $Fr < 1$ the flow is called a subcritical flow, further for $Fr > 1$ the flow is characterised as supercritical flow. When $Fr \approx 1$ the flow is denoted as critical flow.

1.5 Dissertation Overview

I organised the thesis content by the following order:

Chapter 1 introduces the background, problem statement, objectives, and scope of work. The direction and purpose of the study are explained in this chapter

Chapter 2 provides a comprehensive literature study on the previous work done by other researchers. These studies are related to previous work in ARS and numerical simulations.

Chapter 3 describes the methodology of this study. It provides mathematical principles and a detailed numerical simulation process.

Chapter 4 publishes the study results and discussion on the results. The results of different flow conditions are compared.

Chapter 5 summarises this study and put forward some suggestions for the future.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

The literature review is organised into 5 sections. The first section describes the natural riffle and pool. The second section describes the advantage of the riffle. The ARS is explained in the third section. In section four, the literature review on turbulent kinetic energy is presented. In section five, a review of the computational fluid dynamics (CFD) and the numerical model is presented.

2.2 Natural Pool and Riffle

One of the most obvious characteristics of a flowing river is the appearance of periodic form oscillations in the riverbed, such as dunes, meandering, and ripples. Usually, in gravel rivers, relatively shallow waters to deeper waters alternate, which is called the pool-riffle sequence (Scott N. et al., 2008).

The pool-riffle sequence is the basic part of dynamic morphological adjustment in the vertical and horizontal directions due to sediment migration (Thompson, 1986; Carling and Orr, 2000; Church 2006).

2.3 Advantage of Riffle

i) Riffle can promote ventilation and increase the dissolved oxygen in the water.

In the riffle area, water flows at a higher speed and is more turbulent than the flow in the pool. During the flow process, ripples will be generated due to the collision of obstacles or the slope of the river bed. Common obstacles are gravel, pebbles, gravel, and branches. They are arranged at different intervals. Because riffle has a higher flow rate, it has a higher Froude number. The Froude number range is basically between 0.5 and 0.8, and occasionally greater than 1 is called supercritical water flow.

The corrugation of the flow surface due to the surface ripples increases the area in contact with the air and brings oxygen into the water by entraining the air, thereby increasing the oxygen content in the water. The tumbling, spiral, and turbulence in the pool will accelerate this process. Oxygen is very important for the activities and reproduction of fish, plants, and various microorganisms. Pool-riffle can also provide shelter for aquatic life.

Habitat is the natural environment in which organisms live. The zone of the riffle is often accompanied by a stable substrate. The gravel substrate provides a safe shelter and productive habitat for aquatic organisms. The grooves provide a lot of pebbles and gravel that can be hidden. Algae also attach to the gravel to provide food for other aquatic insects and fish. Fish that are small and cannot compete in the pool, such as carps prefer the riffle zone (Cary institute of ecosystem studies, 2012). The pool is deep and the water flow is slow, and organic debris will settle into it. Suitable for the life of slightly larger fish such as trout. Another advantage of the pool is that if the water level starts to drop, it does not have to be moved to other zones (Cave, and Fisheries, 1998). Most fish stay in the riffle during the daytime and rest in the pool at nighttime. (Kim, 2015).

ii) The pool acts as a temperature regulator.

Dallas and Rivers-Moore (2011) inspected the micro-scale heterogeneity of water temperature on 6 highland sites in the Western Cape of South Africa and found that the depth of the pool affects the water temperature, and the deeper the pool, the more stable the thermal environment. Groundwater dependence affects the difference in water temperature, and the fewer groundwater-dependent rivers, the greater the daily maximum temperature difference (Dallas and Rivers-Moore, 2011). And because of the large specific heat capacity of water, a certain amount of water accumulation in the pool will alleviate sudden external temperature changes.

The temperature has a major impact on biological activity and growth. Stream temperature can affect species composition through biological processes (metabolic rate) and ecosystem processes (leaf decomposition, nutrient absorption). Temperature also affects the chemical properties of water. The rate of chemical reactions generally increases at higher temperatures.

iii) Pool-riffle has a strong self-cleaning ability.

The staggered distribution of riffle-pool, rapid and slow current changes, create a variety of different habitats. This continuous sequence is also a prerequisite for the rapid and complete dissolution of contaminants. Abundant and diverse microorganisms can easily digest and decompose pollutants and purify the environment. (Mihov and Hristov, 2011).

Recently, people have realised that the restoration of rivers in ecological habitats is very important, so it is necessary to conduct hydraulic research on the ecological characteristics of pool-riffle.

2.4 Artificial Riffle Structure (ARS)

ARS (Artificial Riffle Structure) is a tool widely used in river rehabilitation. It can simulate natural riffles and play a similar role. However, the use of ARS is still in the experimental stage. On the basis of the case study, the ARS was studied and described in the following literature. The research on the pool-riffle sequence and its influence on the evolution of the river still needs to be improved.

A study by Favata et al. (2018) showed that the implementation of ARS, scouring keys, and coarse boulder substrate promoted the landform stability of Kickapoo Creek

(East Central Illinois). These habitat changes led to significant changes in the structure of fish communities and initially restored degraded biological integrity. They proved that the artificial shoals and river structures used in the channelised hot water flow can effectively reduce degradation and can also support the level of fish biodiversity. (Favata et al., 2018).

Experiments by Ebrahimnezhad and Harper (1997) in river restoration have proved the effectiveness of ARS in increasing the abundance and diversity of macroinvertebrates. It has an effect similar to that of a natural riffle. (Ebrahimnezhad and Harper, 1997).

In river rehabilitation, Harper et al. (1998), by measuring the number of functional habitats in the riffle, confirmed the importance of the rapid flow of shallow depths in the midstream leading to changes in the microenvironment and explained the simplicity of linking geomorphology and ecology The ability of technology to be used more widely. (Harper et al., 1998).

2.5 Turbulent Kinetic Energy (TKE)

TKE is an acronym for Turbulent Kinetic Energy. In fluid dynamics, TKE is the average kinetic energy per unit mass associated with eddy currents in turbulence. The TKE k, can be evaluated using the turbulent velocity component as follows,

$$k = \frac{1}{2}[\overline{(u')^2} + \overline{(v')^2} + \overline{(w')^2}] \tag{1.1}$$

Here, $\overline{(u')^2}$, $\overline{(v')^2}$ and $\overline{(w')^2}$ are the variance of the turbulent velocity component in x, y and z component respectively.

TKE is one of the most common physical quantities in turbulence models. The turbulence of rivers and estuaries affects water quality, ecosystem processes, and river morphology by accelerating various elements. The generation and dissipation of TKE control the evolution of tides and the density structure of the estuary on a large scale. (Friedrichs and Aubrey, 1994, Cread, 2004). On the water surface (air-water interface), the dissipation of TKE in the water controls the gas exchange between water and air, and therefore determines the refilling of oxygen-depleted water and the exchange of carbon dioxide (Zappa et al., 2007).

Although TKE is very important, it is difficult to accurately perform in-situ measurement of turbulence by means of field experiments such as the use of current meters, scalar tracers, and other techniques, which often deform the flow field, and there is often a lack of evaluation methods for complex flow. The desired spatial resolution mode (Chickadel et al., 2011). Therefore, we adopt a numerical simulation method for better analysis.

2.6 Computational Fluid Dynamics (CFD)

CFD is an acronym for Computational Fluid Dynamics. CFD is a combination of modern fluid mechanics, numerical mathematics, and computer science. It is an interdisciplinary science with strong vitality. It uses the fast-computing power of a computer to obtain an approximate solution to the fluid governing equation.

It approximates the integral and differential terms in the governing equations of fluid mechanics as discrete algebraic forms, making them a system of algebraic equations, and then solving these discrete algebraic equations through a computer to obtain numerical values at discrete time/space points solution.

2.6.1.1 Governing equations

The governing equations for the simulation consists are the continuity and momentum equation for an incompressible fluid. The momentum equation for incompressible Newtonian fluid is also known as the Navier-Stokes equation.

i) Continuity equation

ii) The three-dimensional continuity is expressed as follows,

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{2.1}$$

iii) Navier-Stokes equation

The Navier-Stokes equation (i.e. the N-S equation.) are equation of motion describing the conservation of momentum of viscous incompressible fluids. The motion equation of viscous fluid was first proposed by Navier (Claude-Louis-Marie-Henri Navier) in 1827 and only considered the flow of incompressible fluids. Poisson (Simeon-Denis Poisson) proposed the motion equation of compressible fluid in 1831. Saint-Venant (Adhémar Jean Claude Barré de Saint-Venant) and Stokes (George Gabriel Stokes) independently proposed the form of a constant viscosity coefficient in 1845. The N-S equation summarises the general laws of the flow of viscous incompressible fluids, so it has special significance in fluid mechanics. The N-S equations in x, y, and z directions are given as follows,

i) x-component,

$$\frac{\partial u}{\partial t}+\frac{\partial (uu)}{\partial x}+\frac{\partial (uv)}{\partial y}+\frac{\partial (uw)}{\partial z} = g_x+\frac{1}{\rho}\frac{\partial p}{\partial x}+\nu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z}\right) \tag{2.2}$$

ii) y-component,

$$\frac{\partial v}{\partial t}+\frac{\partial (vu)}{\partial x}+\frac{\partial (vv)}{\partial y}+\frac{\partial (vw)}{\partial z} = g_y+\frac{1}{\rho}\frac{\partial p}{\partial y}+\nu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z}\right) \tag{2.3}$$

iii) z-component,

$$\frac{\partial w}{\partial t}+\frac{\partial(wu)}{\partial x}+\frac{\partial(wv)}{\partial y}+\frac{\partial(ww)}{\partial z} = g_z+\frac{1}{\rho}\frac{\partial p}{\partial z}+\nu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z}\right) \tag{2.4}$$

The definitions for the symbols used in Equation (2.1) to (2.4) are as follows, t is time, $u, v, and\ w$ are the velocity component in $x, y,$ and z direction respectively, p is the flow pressure, ρ is the density of the fluid, ν is the kinematic viscosity and g is the gravity acceleration.

2.6.1.2 Volume of Fluid (VOF) method

In CFD, the VOF is a free surface modeling technique, that is, a numerical technique used to track and locate a free surface (or fluid-fluid interface). It belongs to a category of the Eulerian method, which is characterised by a fixed grid or moving in a certain way to adapt to the changing shape of the interface. Therefore, VOF is an advection scheme. It is a numerical formula that allows programmers to track the shape and position of the interface, but it is not an independent flow solving algorithm. The Navier-Stokes equation describing the movement of the flow must be solved separately. The same is true for all other advection algorithms.

2.6.1.3 Fractional Area-Volume Obstacle Representation (FAVOR) method

We know that a perfectly suitable coordinate grid can be difficult to construct, and even with the best grid generator, it still takes a lot of time to build a viable and well-behaved grid.

The simple rectangular structure of the FAVOR method grid makes it very easy to generate. The fractional area and fractional volume must be calculated to define obstacles placed in the grid. These calculations are well-defined and can be easily automated using simple algorithms.

The numerical advantages of the FAVOR method cannot be ignored, which are inherent in the structured, smoothly changing, and strictly orthogonal grids. In addition, the ability to automatically represent porous media is another reason for choosing the FAVOR method as the basis for Flow-3D.

2.7 Dam-Break Flow Problem

Dam-break flow is a common benchmark test used as a means of validating numerical models with free surfaces. This is because it includes a variety of rapid free surface deformations. Many researchers conducted experimental studies on dam-break flow used to verify numerical models (Martin and Moyce, 1952, Koshizuka and Oka, 1996 Hu and Sueyoshi, 2010 and Lobovsky et al., 2014).

CHAPTER 3

METHODOLOGY

3.1 Introduction

This chapter introduces the methodology of this study. The following flowchart in Figure 3.1 summarises the research methodology. Section 3.2 describes the verification of the numerical model in order to know the reliability and accuracy of the model setup in Flow-3D. The verification problem is the three-dimensional dam-break flow problem. Chapter 3.3 describes the arrangement of the ARS module in the ARS zone. Section 3.4 describes the setup of the numerical model for the simulation of flow in the ARS zone. Section 3.5 describes the simulation cases.

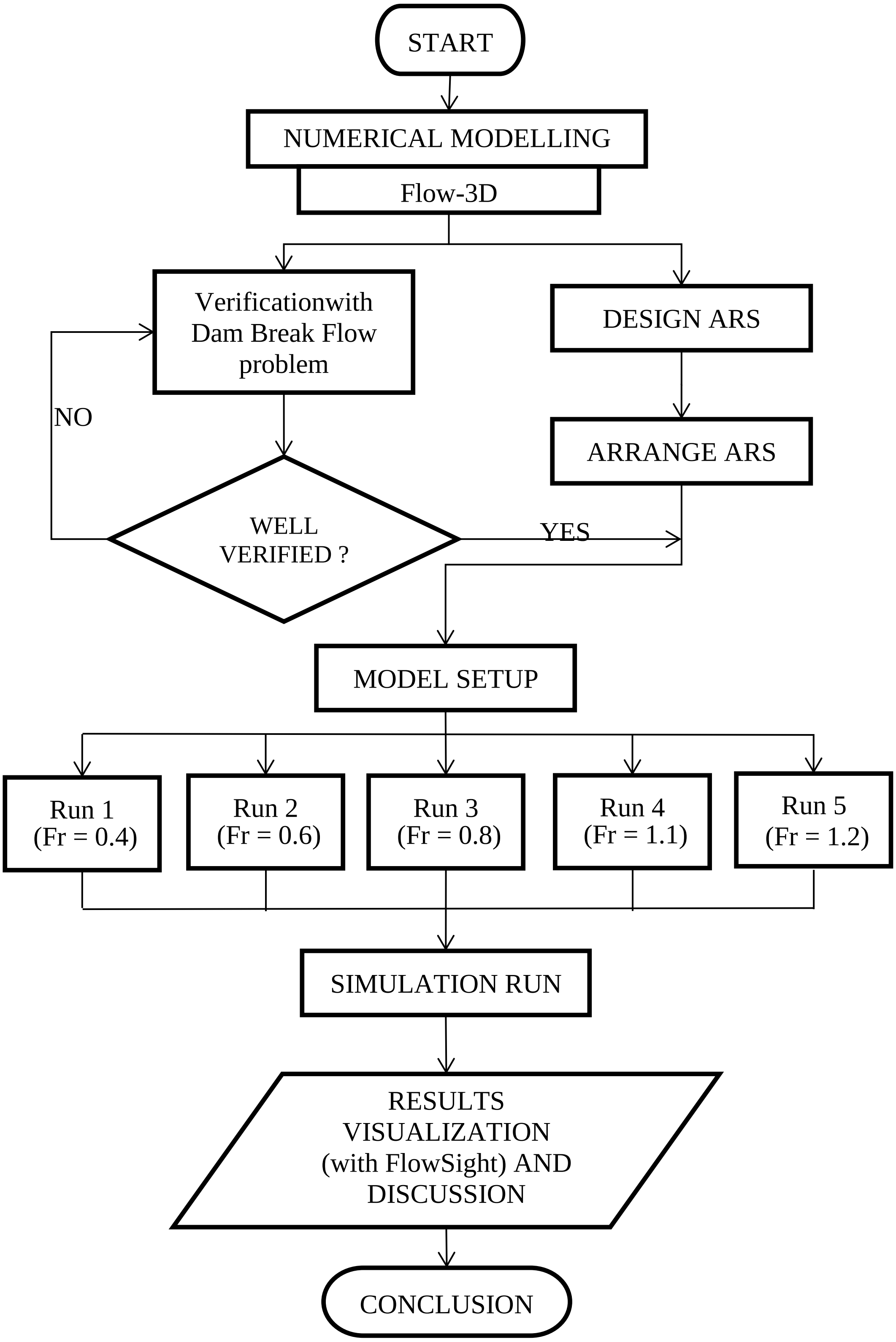

Figure 3.1: Research Methodology

3.2 Verification of numerical model

This section analyses the wave propagation on the flat dry bottom after the dam break. Mathematical models have been used and compared with each other to simulate this dam break situation, as shown in Figure 3.2 and Figure 3.3.

With the help of verification, the accuracy and stability of the numerical model can be judged. If the performance of the operation is not good, fine adjustments will be made accordingly. These verification procedures are important when developing numerical models. The verification process also ensures that the model is setup correctly. The final verification result is in the following chapter Figure 4.2. The value of the parameters, initial and boundary conditions are summarised in Table 3.1.

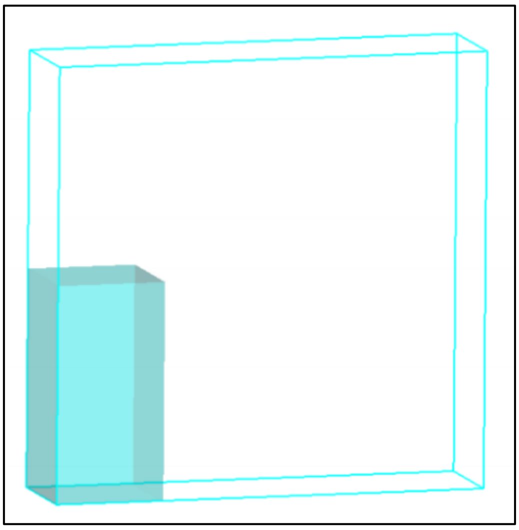

Figure 3.2: Verification of numerical models – 3D

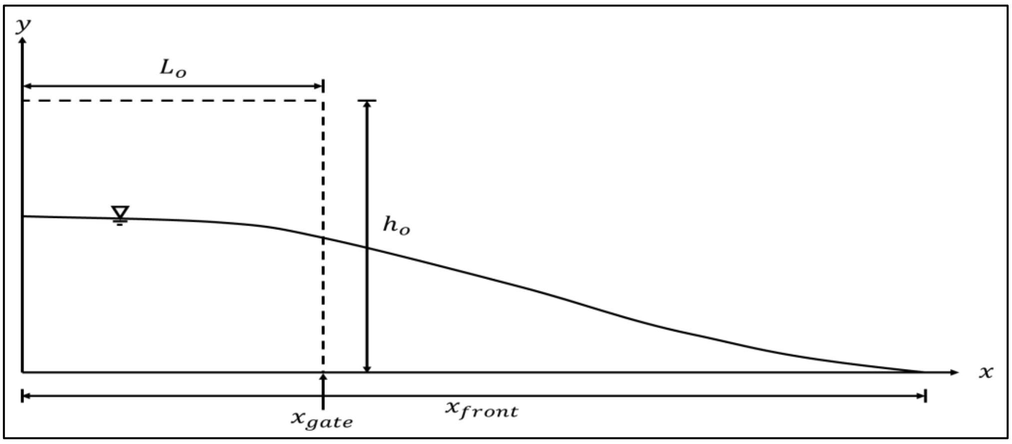

Figure 3.3: Verification of numerical models – 2D

Table 3.1: Simulation parameter, initial and boundary conditions for the dam break flow verification problem

Parameter	**Value**
Cell size,Δx, Δy, Δz	0.04 m
Time increment Δt	Automatically adjusted
Finish time	10 s
Gravity acceleration	z component = -9.81 m/s^2
Density	1000 kg/m^3
Viscosity	0.001 kg/m/s
Initial condition	**Value**
Width of water reservoir, L_o	600
Height of water reservor, h_o	600
Boundary condition	
Side walls, bed and ceiling	Non slip, zero normal velocity

3.3 Development of 3D model for the simulation of flow over Artificial Riffle Structures (ARS)

Two types of ARS are used; the higher type has a height of 0.21m and the lower type has a height of 0.11 m, as shown in Figure 3.4. The ARS units are arranged in a certain order to simulate the riverbed environment under different inflow conditions. The following are the specific dimensions of the ARS, and they are arranged on the riverbed as shown in Figure 3.5.

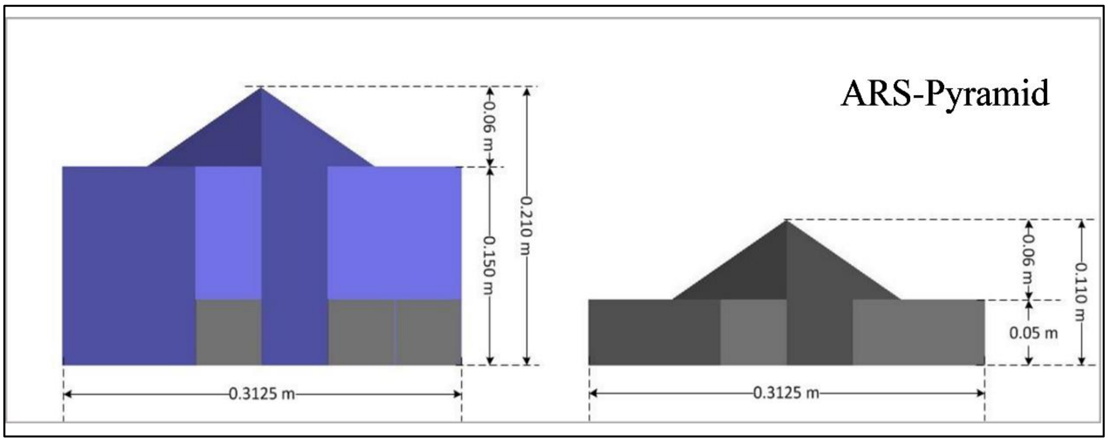

Figure 3.4: Dimension of high and low ARS

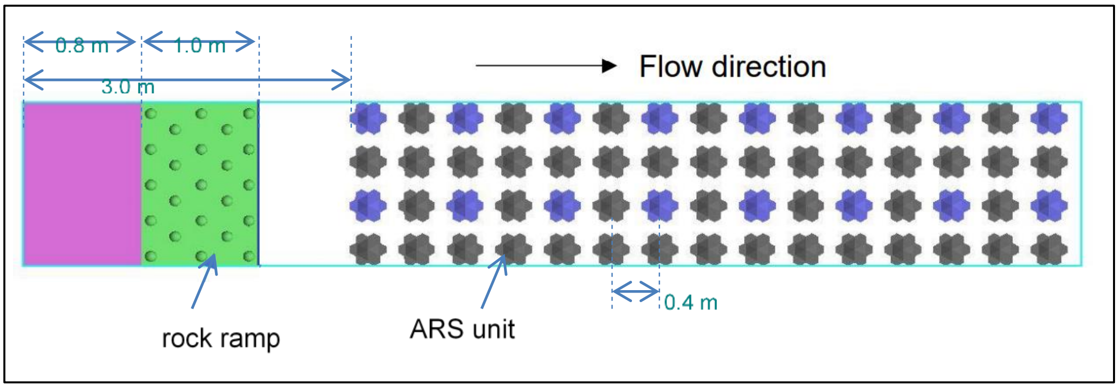

Figure 3.5: Arrangement of ARS unit in the ARS zone

3.4 Setup of numerical model for simulation the flow over ARS zone

The Flow-3D software is used to simulate the flow over ARS zone under different inflow Froude number. The advantage of using Flow-3D is that it can carry out the simulation in the order of setup. The procedure in setting up the Flow-3D model is as follows:

a) Step 1 – Under the 'Simulation Manager' tab, a new simulation is created as shown in Figure 3.7.

b) Step 2 – Under the ‘Model Setup’ tab, the finish time is set to 60.0 s as shown in Figure 3.8.

c) Step 3 – Under the ‘Model Setup’ tab, the ‘Physics’ tab is selected. Under the “Physics” tab, the most basic and objective physical law, such as gravity and gravity direction, are defined. The properties of ‘Gravity and Non-inertial Reference Frame’ and ‘Viscosity and Turbulence’ are defined under this tab, as shown in Figure 3.9. The settings of ‘Gravity and Non-inertial Reference Frame’ are shown in Figure 3.10, and the settings of ‘Viscosity and Turbulence’ are shown in Figure 3.11.

d) Step 4 - At the "Fluid" tab, the properties of water (at 20°C) such as density, viscosity, and others are defined as shown Figure 3.12.

e) Step 5 - Under the ‘Model Setup’ tab, ‘Meshing & Geometry’ is selected. The physical entity and the base that constrains the fluid is defined as shown in Figure 3.13 to Figure 3.21.

f) Step 6 - At the ‘Meshing & Geometry’ tab, the boundary conditions such as water inlet, outlet, and wall boundary are set up. The symbol "V" indicates the inflow condition, while the symbol "O" define the outlet of the test flume (Figure 3.22). The boundary setting of Mesh Block 1 is shown in Figure 3.23 and the overall boundary condition of the model set up is shown in Figure 3.24.

The inlet flow condition is set to a uniform flow condition as shown in Figure 3.25 and the inflow velocity is set to 0.524 m/s, 0.786 m/s, 1.048 m/s, 1.441 m/s, and 1.572 m/s for different scenarios. These correspond to the different Froude numbers of the following simulation cases: $Fr = 0.4$ (RUN 1), $Fr = 0.6$ (RUN 2), $Fr = 0.8$ (RUN 3), $Fr = 1.1$ (RUN 4), $Fr = 1.2$ (RUN 5). The inflow fluid elevation is assumed at 1 cm below the top of the ARS unit, therefore by considering the height of the ARS, the fluid elevation is set to 0.375 m (Figure 3.25). The setting for the simulation condition is shown in Table 3.2 and the setting for inflow condition is shown in Table 3.3. The numerical models of finite extent are shown in Figure 3.6. The boundary conditions are summarised in Table 3.4.

g) After the numerical model is set up, the simulations are carried out accordingly under the ‘Simulation Manager’ tab. The calculation running time is proportional to the simulation time.

Upon the completion of simulation, the required visualisation option is defined under the ‘Analyze’ tab as shown in Figure 3.26, and the simulation can be view under the ‘Display’ tab as shown in Figure 3.27.

Alternatively, the “FlowSight” software can be used to check, analyse and produce graphical representation of the simulation results.

Table 3.2: Setting of parameter for simulation of flow over artificial riffle zone

Mesh Block 1	
Cell size, Δx, Δy, Δz	0.04 m
Mesh Block 2	
Cell size, Δx, Δy, Δz	0.02 m
General	
Finish time	12 s
Time increment, Δt	Automatically adjusted
Physics	
Gravity	Z component = -9.81 m/s^2
Fluids	
Material name	Water at 20 ℃
Density	1000 kg/m^3
Viscosity	0.001 kg/m/s

Table 3.3: Setting of inflow condition for the simulation of flow over artificial riffle zone

Case	Inflow Froude number, Fr	Inlet flow elevation, h (m)	Flow velocity, V (m/s)
RUN 1	0.4	0.375	0.524
RUN 2	0.6	0.375	0.786
RUN 3	0.8	0.375	1.048
RUN 4	1.1	0.375	1.441
RUN 5	1.2	0.375	1.572

Table 3.4: Boundary conditions for the simulation of flow over artificial riffle zone

Boundary	Conditions
Bed	Non-slip and zero normal velocity
Side walls (all)	Non-slip and zero normal velocity
Ceiling	Non-slip and zero normal velocity
Inflow boundary	Constant flow depth and velocity
Outflow boundary	Zero gradient boundary

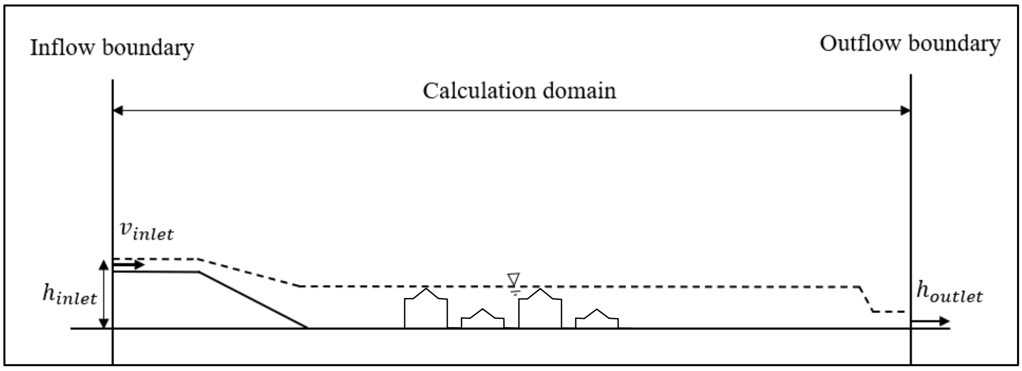

Figure 3.6: Numerical models of finite extent

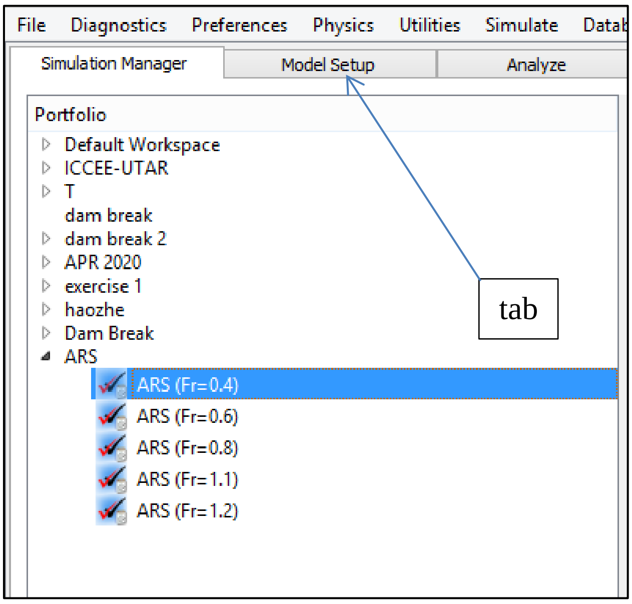

Figure 3.7: Create a new simulation in the first 'simulation manager' tab

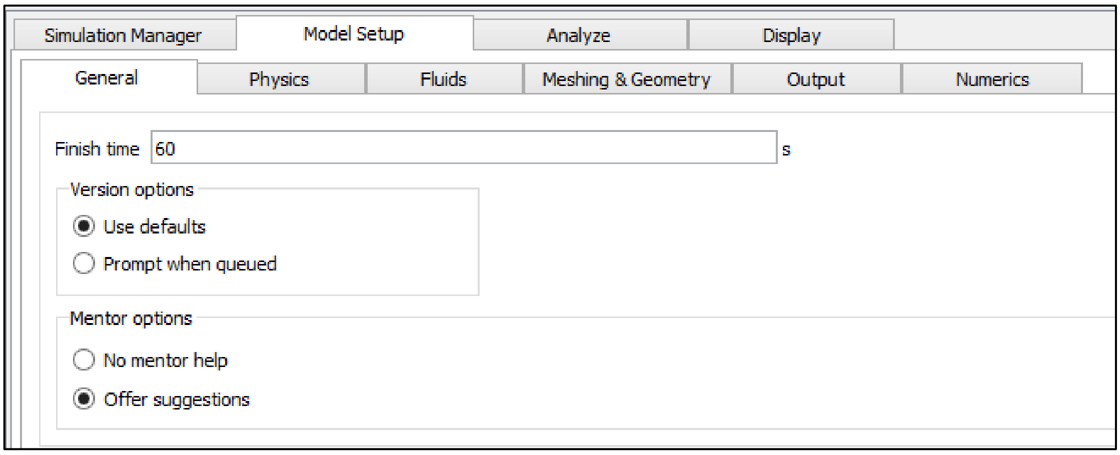

Figure 3.8: Setting of simulation time for the model

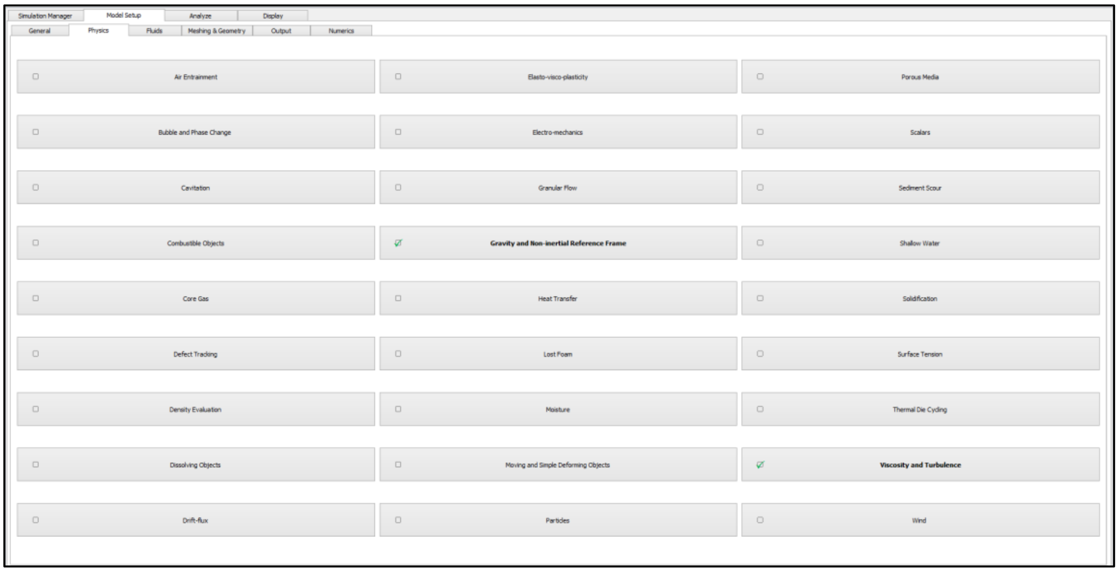

Figure 3.9: Selecting the basic physical rules in the ‘Physics’ tab

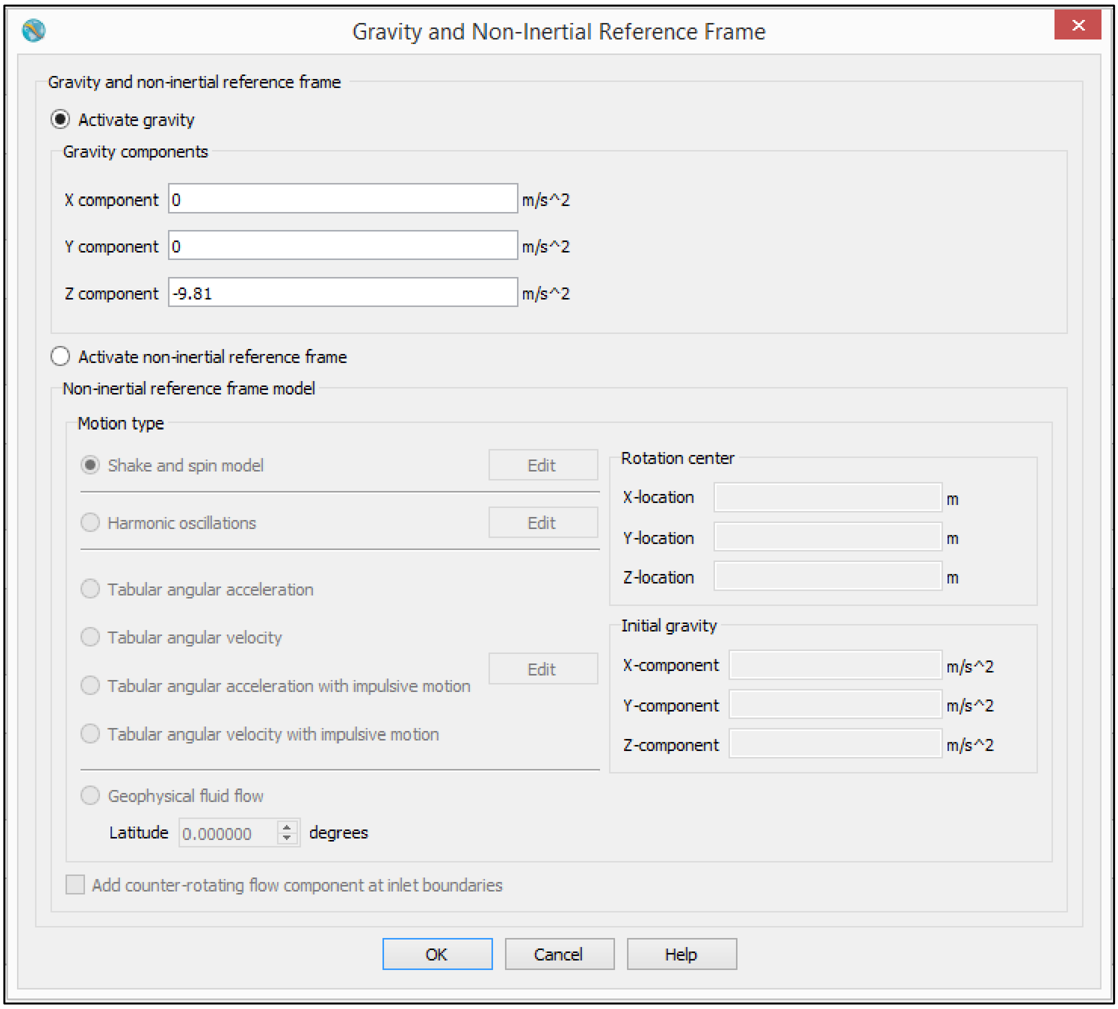

Figure 3.10: Setting of gravity and non-inertial reference frame