A STUDY ON THE EFFECT OF SPUR DIKE ON THE FORMATION OF SAND BAR IN A RECTANGULAR CHANNEL

MUHAMMAD AZFAR BIN HAMIDI

SCHOOL OF CIVIL ENGINEERING
UNIVERSITI SAINS MALAYSIA

2022

A STUDY ON THE EFFECT OF SPUR DIKE ON THE FORMATION OF SAND BAR IN A RECTANGULAR CHANNEL

By

MUHAMMAD AZFAR BIN HAMIDI

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 JULY 2022

Appendix A8

UNIVERSITI SAINS MALAYSIA

SCHOOL OF CIVIL ENGINEERING
ACADEMIC SESSION 2021/2022

FINAL YEAR PROJECT EAA492/6
DISSERTATION ENDORSEMENT FORM

Title: Mechanical Properties of Concrete with Different Types of Glass Waste as Cement Replacement Material

Name of Student: Muhammad Azfar bin Hamidi

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: 10/08/2022

Approved by:

____

(Signature of Supervisor)

Name of Supervisor: Dr. Puay How Tion

Date :

Approved by:

____

(Signature of Examiner)

Name of Examiner :

Date :

ACKNOWLEDGEMENT

I'd like to offer my heartfelt gratitude to everyone who has helped me with this project throughout its whole. First and foremost, I'd want to express my gratitude to my supervisor, Dr. Puay How Tion, for igniting my interest in numerical modeling and allowing me to work on this project under his guidance. My heartfelt thanks to Dr. Puay for his support and encouragement at every level of this effort.

Next, I'd want to express my gratitude to Mr. Lim Jia Jun and Khor Hao Zhe for their unwavering support. Sharing their previous experiences has been extremely beneficial to me throughout this process. I'd also like to thank Mr. Yeoh Chun Xian, a friend of mine who is also working on a research project under Dr. Puay's supervision, for all of his help and support.

Moving forward, I'd like to express my gratitude to my parents for always being supportive of my choices. I would not be the same person I am now if it weren't for their love and upbringing. I'd like to express my gratitude to my siblings, who are always there for me when I need them.

Finally, please accept my apologies if I have left anyone out who has assisted me in some way during this effort.

KAJIAN KESAN BENTENG HAKISAN TERHADAP PEMBENTUKAN BENTENG PASIR DALAM SALURAN SEGI EMPAT

ABSTRAK

Benteng hakisan adalah salah satu daripada teknologi yang digunakan dalam operasi pemulihan sungai. Benteng hakisan memberikan banyak kelebihan seperti mewujudkan habitat akuatik, menavigasi aliran air dan melindungi tebing sungai. Kedalaman air boleh dinaikkan dengan membina benteng berserenjang. Halaju aliran juga boleh menjadi rendah di belakang benteng. Kebolehan benteng hakisan dalam mendorong pembentukan bar pasir disiasat di bawah tiga sudut benteng yang berbeza. Model tiga dimensi (3D) telah dibina menggunakan Flow-3D untuk menyiasat kedalaman dan halaju aliran di sekitar spur dike. Berdasarkan taburan halaju, pengaruh sudut taji terhadap pembentukan bar pasir telah dijelaskan. Kepekatan skalar digunakan untuk mewakili sedimen terampai seperti dalam simulasi, dan pembentukan bar pasir diwakili oleh kepekatan skalar. Daripada keputusan berangka, didapati benteng sudut 90 darjah mempengaruhi bar pasir yang lebih besar dan lebih panjang berbanding dengan benteng yang sudut 45 darjah dan 135 darjah.

A STUDY ON THE EFFECT OF SPUR DIKE ON THE FORMATION OF SAND BAR IN A RECTANGULAR CHANNEL

ABSTRACT

Spur dike is one of the various technologies utilized in river rehabilitation. Spur dike provides many advantages such as promoting aquatic habitat, improving channel for navigation and protecting the stream banks. Water depth can be raised by constructing perpendicular spur dike. Flow velocity also is low behind the spur dike. This study serves to evaluate the effect of a spur dike on the formation of sand bar in a straight rectangular channel. The performance of the spur dike in inducing the formation of spur dike is investigated under three different spur dike angles. A three-dimensional (3D) model was built using Flow-3D to investigate the flow depth and velocity around the spur dike. Based on the velocity distribution, the influence of spur dike angle on the formation of sand bar was explained. A scalar concentration is used to represent the suspended sediment as in the simulation, and the formation of sand bar is represented by the scalar concentration. From the numerical results, it was found that the 90 degree angle spur dike induced larger and longer sand bar compared to the 45 degree and 135 degree angle spur dike.

TABLE OF CONTENTS

Page

LIST OF FIGURES

Page

LIST OF TABLES

Page

CHAPTER 1

INTRODUCTION

1.1 General

For a long time, rivers and channels were the primary source for water supply. Humans have traditionally lived close to rivers, and they've developed canals to transport water to locations that aren't as close to the rivers. As a result, canals and rivers must be protected against erosion and damage by flowing water. An investigation of river and channel behavior through time has a unique position in terms of scientific and technological foundations for optimal water resource utilization, as evidenced by the efforts of researchers in several scientific fields such as hydraulic and hydrology, sedimentology, and geology.

Spur dike is commonly employed for river and channel training. The benefits of the spur dikes are well-documented. Spur dikes are used to constrict flow route and as a result, raise the flow depth near the structure and increase the average velocity in open channel. Amongst various river restoration, spur dikes are gaining popularity due to the advantages. The popularity of this river rehabilitation technology may be proved via its vast implementation in European countries (Howe, 1997).

In Malaysia, a storm drain is often large and used to channel a large amount of runoff to the downstream receiving body. However, during the dry season, the flow discharge in the storm drain is low, causing low-velocity flow and sometimes stagnant water. Shallow and low-velocity flow promote the growth of algae and phytoplankton which can increase the Biological Oxygen Demand (BOD) and cause bad odor. Low-velocity flow also causes accumulation of pollutants and rubbish.

1.2 Problem Statement

The storm drain in Malaysia is often oversized causing shallow flow depth and low flow velocity during dry period. This condition encourages algae growth which causes bad ordor and also causes poor water quality due to high BOD and concentration of pollutants. Storm drain, which is often made of concrete, also lacks in biodiversity and aesthetic value. In general, society has a negative perception on storm drain. A polluted storm water channel also brings adverse economic impact on its surroundings due to poor water quality and poor appreciation value.

Sand bar can be used for storm drain improvement by constricting the channel width to increase flow depth and velocity during dry period. At the same time, sand bar provides a platform for riparian species to grow, and thus improve the biodiversity in the storm drain.

Although spur dikes have been used to control flow and erosion of river bank, there have been few records units' application in storm drain rehabilitation in Malaysia. Therefore, in this study, the effect of spur dike on the formation of sand bar will be investigated. The knowledge gain from this study will help the design of sand bar in storm drain improvement purpose in the future.

Figure 1.1: Example of an oversized storm drain at Upcycle Park, Perda, Pulau Pinang

1.3 Objective of the study

The objectives of this study are as follows:

i. To investigate the effect of a spur dike on the formation of sand bar in a straight rectangular channel.
ii. To investigate the effect of spur dike under different angle on the flow velocity and flow depth around the spur dike in a straight rectangular channel.

1.4 Scope of study

This study focuses on the effect of spur dike on the formation of the sand bar in a straight rectangular channel. A numerical model is used to simulate the flow and the formation of sand bar around the spur dike. The numerical model used in this study is Flow3D. The numerical model will solve the three-dimensional flow problem with injection of a scalar concentration. A scalar concentration will be injected from the upstream of the channel to mimic the inflow of suspended sediment. In this study the bed of the channel is considered immobile. This is because storm drain is usually made of concrete.

This study will only focus on the problem with only one spur dike. However, three different spur dike angles will be investigated.

1.5 Thesis Outline

This thesis consists of five chapters. An overview of the history and significance of the spur dikes system is provided in Chapter One. This chapter also discusses the scope and objective of the study.

Chapter Two addresses the study's review of the existing literature. This chapter provides background information on the accumulation of sediment, spur dikes structures, the numerical model, and the governing equations in the numerical model.

Chapter Three presents the methodology of the study. This chapter explains the data collection and the setup of the numerical model for simulation. The numerical verification process is also explained in this chapter.

The study's findings are discussed in detail in Chapter Four. This chapter presents the data collected from the site, result of the verification of the numerical model. The simulation result of flow around the spur dike for three different angles are also presented in this chapter. The effectiveness of the spur dike in different angle, in enhancing the rate at which the suspended sediment accumulates and forms the sand bar is explored from the numerical findings.

The findings of this study are summarized in Chapter 5, and a conclusion is taken from them. Suggestions for improvement in future research are given in this chapter.

CHAPTER 2

LITERATURE REVIEW

2.1 Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) uses computer to solve and simulate fluid dynamic problems numerically. Governing equations for fluid flow are cast in partial differential form in CFD. Using CFD model, we will obtain a list of numbers representing the flow variables, as the result of the simulation. Since the goal of most engineering analyses is to get a quantitative representation of the problem, the application of CFD models are essential (Anderson et al., 2009).

High-speed digital computers have played a significant role in the development of computational fluid dynamics (CFD) (Chung, 2009). For inviscid flows, CFD was first employed for simpler flow geometry, and this is sometimes referred to as the method's first generation. The CFD solutions currently in use are from the second generation, and they deal with more complicated fluid dynamics issues that cannot be solved without the use of processing power. For example, boundary layer approximations are not suitable for issues involving mixed subsonic-supersonic flows or viscous flows. Fluid dynamics predictions can be made using CFD as much as they can be made using pure experiment and pure theory (Anderson et al., 2009). CFD's brilliance rests in its ability to assist and complement both experimental and theoretical approaches to solving problems.

The finite difference method (FDM) and the finite element method (FEM) are the two most used CFD approaches (FEM). The finite volume technique (FVM), which is derived from FDM or FEM, has been increasingly popular in recent years. In terms of FDM, FEM, and FVM, CFD formulations and methodologies continue to be improved and refined to this day (Chung, 2009).

2.2 Spur Dikes

Conservationists and river engineers design spur dikes for river restoration initiatives. Numerous research have been conducted over the last several decades on the analysis and modeling of river hydrodynamic dynamics. The spur dike is a classic canal alteration because it narrows the water cross-section, increasing flow velocity and flushing the riverbed. At the same time, it safeguards natural variety and enhances river habitat. Diverse forms of spur dikes may have a significant effect on the relationship between flow structure and local geomorphology, hence influencing the evolution of river aquatic ecosystems.(Huang, and et al ,2019). An example of the application of spur dike in river to improve navigation in River Rhine is shown in Figure 2.1.

Figure 2.1: Spur dike in Rhine River.(Zhenghua Gu and et al, 2016)

2.3 Riparian Vegetation

Vegetated buffers are a well-recognized means of safeguarding aquatic resources from human activity, such as rivers and wetlands. However, because of the numerous limitations associated with urban growth, creating vegetated buffers for aquatic resources is extremely difficult in urbanized development. (Jin, K., & Chen, J, 2020).In storm drain made of concrete, there is no platform for the riparian vegetation to grow. Therefore, sand bar plays an important role in providing platform for the riparian vegetation to grow.

2.3.1 River Restoration

River restoration is the process of restoring biodiversity through restoring natural processes in rivers, which helps both people and wildlife. Natural processes can be reintroduced to modify rivers and ensure their long-term viability by providing the range of habitats essential for a healthy river ecosystem recovery by addressing the problem's underlying source. A good example of river rehabilitation is shown in Figure 2.2 where River Wandel in London was rehabilitated by restoring the river biodiversity Trout spawning was observed for the first time in 80 years in the river (Ulrika Aberg, 2010)

The river rehabilitation method can be used to improve the condition in storm drain by creating or improving the biodiversity. However, it is a challenge to create biodiversity in a concrete storm drain because of the lack of platform for vegetation to grow. One of the solution is to induce sand bar in concrete storm drain by using spur dike. The application of spur dike in inducing the formation of sand bar can be seen in storm drain rehabilitation in Fresnes-en-Woevre, France as shown in Figure 2.3.

Figure 2.2: Restoration of the River Wandle, London.
(Ulrika Aberg, 2010)

Figure 2.3: Application of spur dike to induce sand bar formation for storm drain improvement in Fresnes-en-Woevre, France (Pansera Jean-Noël, 2018)

2.4 FLOW-3D

FLOW-3D is a commerical CFD modeling program that is well recognized for advancing the three-dimensional Volume of Fluid (VOF) algorithm and is frequently used in free surface flow research. It's also notable for its ability to perform in-depth analysis on even small-scale simulations (Hu et al., 2018).

Flow-3D is believed to be more user-friendly than other CFD software such as OpenFoam or Mike 3FM. The software's workspace is user-friendly and lead the user through each phase of the modeling process. It's also easy to create solid boundaries and mesh blocks. FLOW-3D's FAVOR technique also allows for the modeling of more complicated geometric regions.

2.5 Governing Equation

In Flow-3D, the incompressible three-dimensional continuity and Navier-Stokes equations are employed to solve fluid motion. The equations are written in Einstein's summation convention notation, as seen below:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{2.1}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \mathrm{P}}{\partial x_i} + g_i + \frac{1}{\rho}\frac{\partial \tau_{ij}}{\partial x_i} \tag{2.2}$$

$$\tau_{ij} = 2\mu S_{ij},\ S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\ with\ (i,j = 1,2,3) \tag{2.3}$$

Here, g_i indicates gravitational acceleration, where u_i indicatesvelocity, ρ denotes fluid density and P is defined as pressure, and τ_{ij} denotes the deviatoric stress.

2.6 Turbulence Model

The turbulence stress in the model is calculated using the two-equation Re-Normalisation Group (RNG) k-ω turbulence model. When compared to the normal k-ϵ model, the turbulence diffusion coefficient is explicitly calculated in the RNG k- ω model (where the turbulence diffusion coefficient is evaluated based on the specified turbulence length scale in the k-ϵ model). Because it can explain low-intensity turbulence, the RNG model has a broader range of applications. Since the RNG k-ω turbulence model is the RANS (Reynold-Averaged Navier-Stokes) type turbulence model, the governing equation in Eq. (2.1) and (2.2) are recast into the time-averaged form .

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \tag{2.4}$$

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \bar{P}}{\partial x_i} + g_i + \frac{1}{\rho}\frac{\partial \bar{\tau}_{ij}}{\partial x_i} - \frac{\partial}{\partial x_j}(\overline{u'_i u'_j}) \tag{2.5}$$

$$\bar{\tau}_{ij} = 2\mu \bar{S}_{ij},\ \bar{S}_{ij} = \frac{1}{2}\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)\ with\ (i, j = 1,2,3) \tag{2.6}$$

Here, the bar accent over the variable are the time-averaged variables, e.g. $\bar{u}_i$ is the time-averaged velocity. The term $\overline{u'_i u'_j}$ is the Reynold's shear stress, which is modelled by the turbulence model.

2.7 Volume of Fluid (VOF) Method

FLOW-3D employs the Volume of Fluid (VOF) method for free surface configuration (Hirt, 1993). In the staggered mesh system, the fluid fraction in each cell is specified as a function F. When the cell is filled with water, the value of F is 1.0; when the cell is empty, the value of F is 0 (Barkudarov, 2004). If the value of F is between 0 and 1, the cell is only partially filled, implying that there will be a free border surface. Figure 2.3 displays a state in which cells are filled, half filled, or vacant. The advection equation for VOF function F is as follow:

$$\frac{\partial F}{\partial t} + u\frac{\partial F}{\partial x} + v\frac{\partial F}{\partial y} + w\frac{\partial F}{\partial z} = 0 \tag{2.7}$$

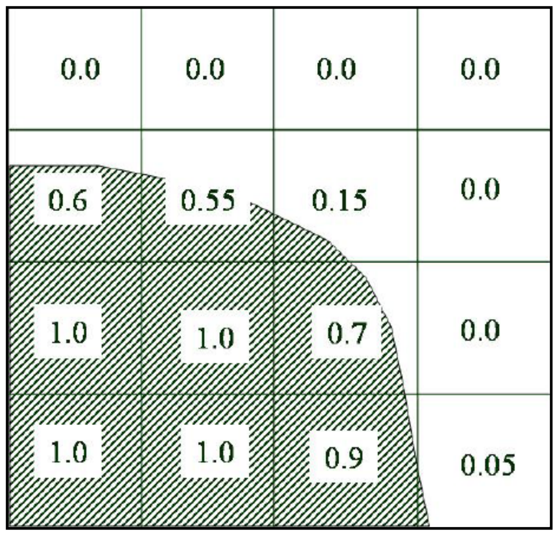

Figure 2.4: Value of function F in filled, half filled and vacant cells in the VOF method

2.8 Fractional Area Volume Obstacle Representation (FAVOR) Method

In FLOW-3D, the FAVOR technique is utilized to represent barriers in the domain (Hirt, 1993). The benefit of FAVOR is its capacity to represent complicated geometrical structures. The FAVOR approach may be applied to both structured and unstructured grids. However, because fractional areas and volumes are easily determined, structured and rectangular grids are frequently selected. The FAVOR equation for incompressible, inviscid fluids may be expressed as follows:

$$\frac{\partial V}{\partial t} + \nabla \cdot (Au) = 0 \tag{2.8}$$

where A is the open area fraction, V is the open volume fraction, and u is the velocity vector. When an impediment completely fills a cell, the function V is zero. When a cell contains the border of an obstacle, it implies that the function V will have a value ranging from 0 to 1.

2.9 Simulation of Suspended Sediment

The data from the river survey at the storm drain at site (Upcycle Park, Perda, Pulau Pinang) found the lack of bed load material. Suspended load is dominant in the flow. Therefore, a scalar concentration can be used to represent the transport of suspended sediment in the flow (Wei et. al., 2014). The concentration is assumed to be uniform in a given cell and is coupled with the fluid cell density and viscosity. The suspended sediment concentration is calculated by solving a transport equation. The transport equation for the suspended sediment or scalar concentration is given as follows (Wei et. al., 2014),

$$\frac{\partial C_s}{\partial t} + \nabla \cdot (C_s \boldsymbol{u_s}) = \nabla \cdot \nabla (DC_s) \tag{2.6}$$

The symbol C_s represents the suspended sediment or scalar concentration in mass per volume of fluid-sediment mixture, D is the diffusivity and $\boldsymbol{u_s}$ is the velocity (in vector form) of the sediment or concentration.

CHAPTER 3

METHODOLOGY

3.1 Introduction

This chapter discusses the research methodology with river survey, validation, and simulation model setup for the study. The research methodology flow chart is shown in Figure 3.1. First, as described in Chapter 2, a literature review was conducted. The range of flow velocity, flow depth, and suspended load in a typical concrete storm drain were measured for the river study at Upcycle Park in Perda, Pulau Pinang.. The flow velocity and flow depth were used as a reference value for the numerical model. The river survey methodology is presented in Section 3.2. Section 3.3 presents the setup of the numerical model. The numerical model is first setup using Flow-3D for validation purpose. The model is verified against the result from Holtz's experiment (Holtz, 1991). In the validation process, two tests were carried out, i.e, Test 1 for rigid lid condition and Test 2 for free surface condition. The verified model is then used to to simulate flow around a spur dike in a rectangular open channel. Three different configuration of spur dike, i.e. 45, 90 and 135 degree angle were used in the simulation to investigate the effect of spur dike angle on the flow depth and velocity around the spur dike. A scalar concentration is also injected into the simulation domain to see the effect of spur dike angle on the formation of sand bar. The simulation results are visualized using FlowSight and presented in Chapter 4.

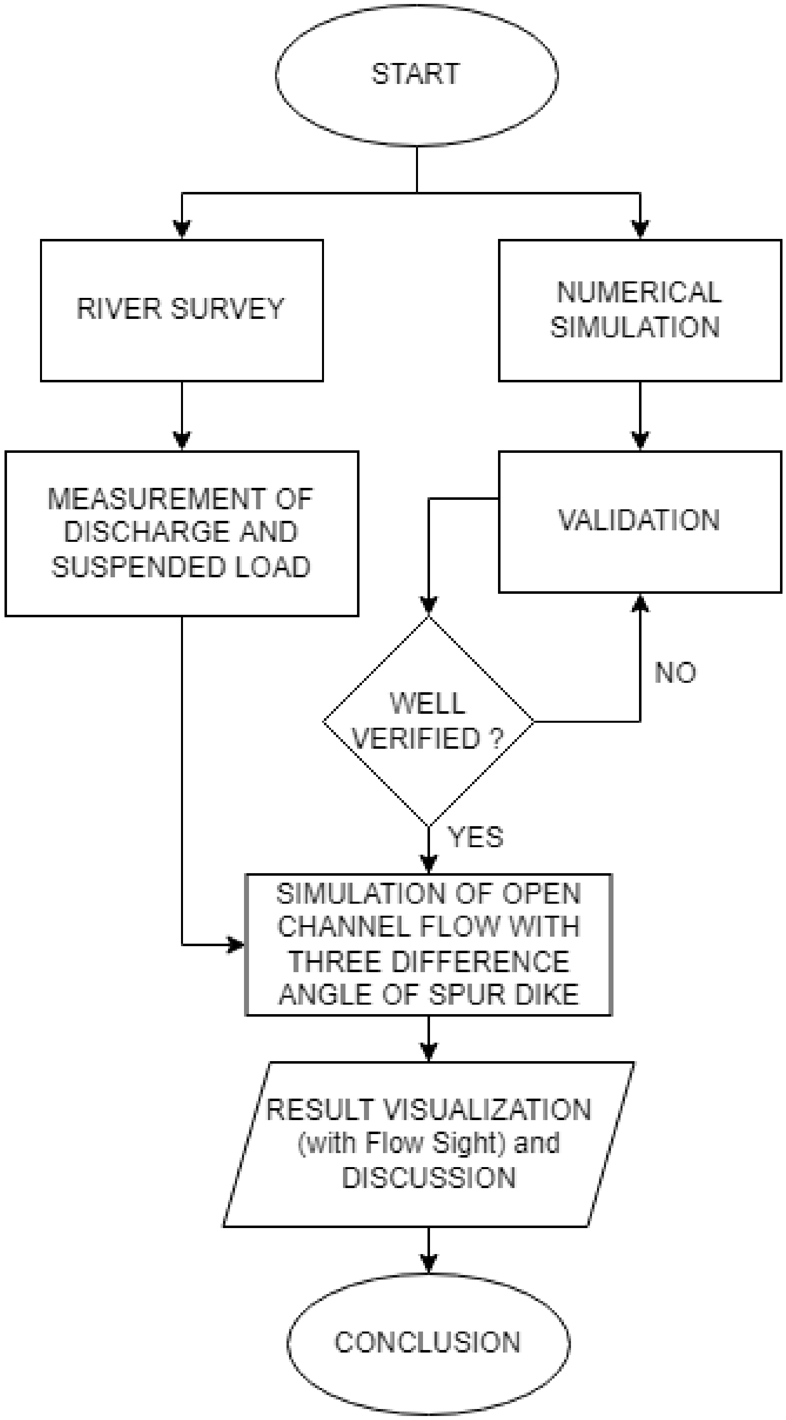

Figure 3.1: Methodology of Research

3.2 River Survey

A river survey was conducted in the storm drain located at the Upcycle Park, Perda, Pulau Pinang on the 22th March 2022. The purpose of the river survey is to gather data on the flow depth, flow velocity and suspended load. The river survey data will be used as reference data for the numerical simulation. River survey was conducted based on the Hydrological Procedure No. 15 from JPS (HP, 1976). River survey was conducted at three different cross sections. The location of the cross sections is shown in Figure 3.2 and their longitudinal and latitude are given in Table 3.1. The distance between CS1-CS2 and CS2-CS3 is around 30 m.

Table 3.1: Latitude and Longitude for each cross section point at location study

	Latitude	Longitude
CS1	5°22'14.18"N	100°25'52.56"E
CS2	5°22'13.70"N	100°25'50.96"E
CS3	5°22'13.16"N	100°25'49.25"E

Figure 3.2: Location of cross section CS1, CS2 and CS3 (Picture from Google Earth)

3.2.1 Discharge Measurement

To estimate the flow discharge, flow velocity and cross sectional area of the rectangular channel are needed. The mean velocity at a point is calculated by taking the average of the velocities measured at defined depths. The velocities are measured using an electromagnetic current meter and a data logger as shown in Figure 3.3. Velocity distribution between the water surface and the stream bed approximates a parabola, reaching zero velocity at bed level, and maximum velocity at roughly one third of the depth below the surface. For area measurement, we measured width of the rectangular channel and flow depth at site.

Figure 3.3: Data logger and electromagnetic current meter

First and foremost, the width of the rectangular channel is measured by measuring tape from the storm drain right bank to the left bank. The measured width of the storm drain is 15 meter. The width is then divided into 8 parts to get the equal spacing for each 7 points, the distance between two neighboring points is 1.9 meter (round up to two significant figures). Rope and tagline or tape are used to mark each point to make sure the spacing distance is secured.

Next, the flow depth is measured with the wading rod as shown in Figure 3.4. Reading of the flow depth is made to the nearest 5mm. During flow depth measurement, the baseplate is made sure to rest flat on the bottom of the rectangular channel. Depth was measured by sliding the index finger down the rod and positioning the fingertip on the side of the rod at the water surface level. The flow depth was read by raising the rod while holding the finger steady.

After the flow depth was recorded, the current meter was adjusted to the 0.6d. Here d is the flow depth. This is based on the guidelines set by HP No. 15 (HP ,1976) for flow depth less than 0.25m, flow velocity is measured at only one point, at 0.6d.

The rod, together with the current meter is put back into the water and the base plat is put to rest at the bottom of the channel. The current meter is reset, and reading is made after 10 to 20 seconds. The reading on the current meter is recorded, and an example of data at CS1 as shown in Table 3.2. Measurement of the flow depth and flow velocity are carried out based on the same procedure at CS1, CS2 and CS3.

Table 3.2: Example of recorded data at CS 1.

Point	Distance from right bank (m)	Depth d (cm)	Method of measure	Depth from surface for velocity measurement (cm)	Velocity at point (m/s)			Average velocity (m/s)
					1st	2nd	3rd	
0	15	0	0.6d	0	0	0	0	0
1	13.3	12	0.6d	7.2	0.02	0.02	0.02	0.02
2	11.4	11	0.6d	6.6	0.03	0.03	0.03	0.03
3	9.5	13	0.6d	7.8	0.06	0.06	0.05	0.06
4	7.3	15	0.6d	9	0.07	0.06	0.06	0.06
5	5.7	13	0.6d	7.8	0.05	0.05	0.05	0.05
6	3.8	12	0.6d	7.2	0.05	0.06	0.06	0.06
7	1.9	14	0.6d	8.4	0.06	0.06	0.06	0.06

Figure 3.4: Flow discharge measurement at the site

3.2.2 Measurement of Suspended Load

The sediment load in the storm drain at the site is consisted of mostly suspended load. Measurement of bed load with the Halley-Smith bed load sampler (Figure 3.5) showed negligible amount of bed load in the channel. This is because the channel is made of concrete (thus immobile bed). In addition, the inflow source into the storm drain is mostly from the upstream retention pond, and some smaller perimeter drains. The suspended load is measured using the DH-48 sampler (Figure 3.6). The general technique for taking sediment samples is first splitting the cross section to be sampled into sub-sections by selecting several vertical sampling sections across the cross section, referred to as verticals. The optimal subdivision is to place the verticals in such a way that their limits divide the cross section into equal discharge sub-sections. Each vertical is then sampled, either by integrating throughout its depth or by integrating at one or more representative places in the vertical over a brief period. The optimum sample is one that fills the sample vial to about two-thirds of its capacity as shown in Figure 3.7. Under no circumstances should the bottle be filled. Take three sample points (point 1, point 2 and point 3) for each cross-section. Before sending the samples to the laboratory, all important information about the sampling is entered on the appropriate form and the samples are properly identified.

Figure 3.5: Halley-Smith bed load sampler

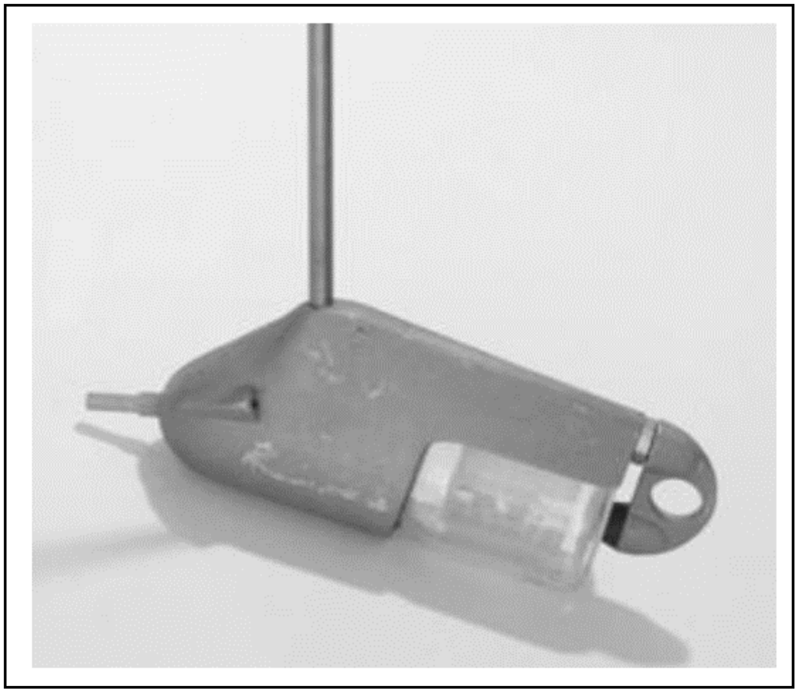

Figure 3.6: DH-48 Sampler to collect suspended sediment

Figure 3.7: Water sample for each cross section (CS1 and CS2)

3.2.3 Laboratory test for Suspended Load

First, a graduated cylinder has used to add 100 ml of deionized water to the filtering flask. A vacuum was applied to the filtered water until all of the water has been filtered. Aluminum foil cup with the fiber filter disc was preheated in the drying oven at 103–105 °C (217–221 °F) for 1 hour. The fiber filter disc was dried to a consistent weight in an oven to calculate the weight of the empty disc. A well-mixed filtered sample was dried in the same fiber filter disc to a consistent weight in an oven at 102-105 °C (217–221 °F). The weight difference between the empty disc and the disc with the remaining materials reveals the Total Non-filterable Solids. The weight difference between the disc and the disc with remaining materials displays the Volatile Non-filterable Solids. The preparation for the measurement of the suspended load is shown in Figure 3.7.

Figure 3.7: Preparation for suspended load laboratory test.

The reagent and apparatus used in the suspended load measurement is listed below:

i. Aspirator, vacuum pump

ii. Balance

iii. Bottle, wash

iv. Cylinder, graduated

v. Filter discs, glass fiber, 47 mm 1 100/ρ kg 253000

vi. Filter holder, magnetic base

vii. Distilled water

3.3 Numerical Simulation

Validation is required to demonstrate the model set using Flow-3D is correct and capable of producing acceptable numerical results .Validation a three-dimensional spur dike system is placed in a straight rectangular channel for validation against the experimental work by Holtz (Holtz, 1991). Validation of the verified model is used to simulate flow around spur dike under different configurations of spur dike, i.e. 45, 90 and 135 degree angle. For each case, a scalar concentration is injected at the upstream to represent the supply of suspended load. The flow depth, flow velocity and distribution of scalar concentration in the simulation domain are observed.

3.3.1 Validation of Numerical Model

1. Numerical model setup of validation

Figure 3.8 illustrates the setup of a model. A three-dimensional spur dike is placed at one side of a straight rectangular channel. A constant inflow is setup at upstream, and a free outflow is setup at the downstream of the channel. This setup is similar to Holtz's experiment (Holtz, 1991). Therefore, the numerical model will be validated against the experiment result of Holtz's. To simulate the flow around the spur dike in an open channel, a rigid lid condition is conducted first (Test 1). The steady flow result from Test 1 is then used as the initial condition for a free surface flow condition (Test 2). Such procedure is used so that the steady state flow condition in the free surface model can be achieved in shorter time (Holtz, 1991). In the undisturbed zone, the average water depth was 0.23 m and the average flow velocity was 0.345 m/s. In Figure 3.8 shows the domain used in Holtz's experiment.

The setup of the validation model is explained below.

a) Setup of fluid properties

The fluid chosen for simulation is water at 20 degrees Celcius. Although the temperature of water at the site is more than 20 degree and less than 30 degree, the density of water does not differ much. From 20 degrees Celcius to 30 degree Celcius the water temperature changes is only around 0.00292 kg/m3 (USGS, 2018). In addition, since incompressible fluid is considered in the simulation, the changes of density is neglected.The properties of the fluid used in the simulation is shown in Table 3.3. Figure 3.9 shows the default properties of this fluid in Flow-3D.