CHAPTER 1

INTRODUCTION

1.6   Introduction

The design of structures to control waterways in Malaysia is a major concern for engineers. The options for flood control in urban areas, however, are limited. A large fraction of the ground surfaces is paved causing concentrated flood flow peaks. One of the practical methods of routing the water through the urban areas is via the use of high-velocity channels.

Hydraulic engineers often use the term "high-velocity channel" when referring to a control flood channel which was designed to discharge water as fast as possible to discharge point such as river or sea (Berger and Stockstill, 1995). High-velocity channels are often used for drainage purposes in urban regions where real estate is expensive. This kind of channels are normally constructed at a sufficient slope so that the flow is supercritical, thus reducing the flow area and concentration time.

The designer of these high-velocity channels is faced with many problems that cannot be solved easily. At the design level, two main concerns are the water depth and velocities of the flow. The depth must be known to determine sidewall heights and minimum bridge span elevations. Normally, a designer simply applies an

empirical equation such as the Manning's equation to obtain water depth with known discharge rate. However, determining the depth of flow is complicated by side inflows and boundary features such as contractions, expansions, curves, and obstructions. These boundary features in a supercritical channel cause flow disturbances that can result in a significant oscillation in flow.

Besides water depth, consideration should be given to flow velocity when designing a channel section. According to MASMA (Urban Stormwater Management Manual for Malaysia), flow velocity should be controlled within range 0.6 – 4.0 m/s to prevent sediments and to protect channel from bank corrosion.

For these design purposes, many methods have been used such as empirical equations, physical models and numerical model. A numerical model in handling shock capturing will be tested through this study.

1.7   Problem Statement

Open channel especially high-velocity channels are used for drainage in urban regions, since urban sprawl increase rainfall runoff due to altered land use. Flood control channels are designed and built to safely manage the anticipated hydrologic load. The desire is to minimize the water's time of residence in the urban area. The channels are designed to carry supercritical flow to reduce the water depths and the required route. Structures, such as bends and transitions cause flow to choke and form jumps. These hydraulic conditions generally necessitate higher walls, bridges and other costly containment structures. A poorly designed channel can cause bank erosion, damaged equipment, increased operating expense, and reduced efficiency (Berger and Stockstill, 1995). Furthermore, crossings may be washed out, and the town may be flooded.

Predicting the potential location of shocks and determining the elevation of water surface in channel are necessary to evaluate and decide the required sidewall heights. Normally empirical equations are often used in the channel design due to its

simple application. However, the presence of bends, contractions, transitions, confluences, bridge piers and access ramps can cause the flow to choke or to produce a series of standing waves and these all will complicate channel design.

In the past, applications of physical models are common for this water profile evaluation. Although physical model can reproduce a channel if properly conducted, but great care must be taken in model dimension and scale. A major drawback of physical models is the problem of scaling down a field situation to the dimensions of a laboratory model. Phenomena measured at the scale of a physical model are often different from conditions observed in the field. For example, the Reynolds number of the flow in site will not be the same as Reynolds number in the physical model.

Changes to the physical model require a "cut and try" technique that involves tearing down sections of the channel and rebuilding them with the new desired design. Due to the time and cost constraints of physical models, it is not practical to examine a wide range of designs. This could result in hydraulic performance that is only acceptable over a limited range.

Mathematical models have been developed to overcome the problem mentioned above. A mathematical model consists of a set of differential equations that are known to govern the flow of surface water. The reliability of predictions of models depends on how well the model approximates the field situation. Inevitably, simplifying assumptions must be made because the equations such as differential continuity equation and momentum equation are too complex to be solved. Usually, the assumptions necessary to solve a mathematical model analytically are fairly restrictive. To deal with more realistic situations, it is usually necessary to solve the mathematical model approximately using numerical techniques. Therefore, an inexpensive and a readily available model is needed. A numerical model is a logical approach.

An area of engineering design that can benefit the use of numerical model is the design and modification of high-velocity channels essential for the routing of floodwater through urban areas. The proper design of new channels and re-design of existing channels is required to avoid such things as bank erosion, damaged

equipment, increased operating expenses, flooding, and higher construction costs. By using numerical model, a better channel design can be produced with minimum cost and time.

1.8   Objective of the Study

The primary purpose of the research is to develop a methodology and ascertain the effectiveness of using a numerical model for open channel modeling. The challenges for this numerical model lie in representing supercritical transitions and, capturing the potential location and movement of the shocks. The specific objectives of the study are listed as the following:

1. to assess the practicality of using a two-dimensional numerical model to aid in the design of a realistic open channel, and
2. to evaluate the performance of the numerical model in handling shock capturing in various test cases through comparison with published results, laboratory tests and analytical solutions.

1.9   Scope of the Study

The purpose of this research is to describe the numerical flow model and to illustrate typical open flow fields that the model is capable of simulating. Only rectangular channel is focused in this research. A few test cases are conducted in laboratory using simple geometries. Numerical results are used for comparison with published laboratory results. Model parameters are tested to determine the model sensitivities. This reduces the number of parameters to only those that have major impact on the design. The model verification consists of comparing results computed using the numerical model with laboratory results and analytical solutions. However, studies will only focus on steady state flow. Model limitations will also be discussed.

The results can be used to determine the appropriate parameters to be optimized in the future.

1.5   Significance of Research

In surface water modelling, the most challenging part is to detect the location and water elevation of hydraulic jump or shock. The height of the jump is critical to the design of channel walls and bridges within high-velocity channel. And through this prediction, we can also define easily the critical location within the existing channel so that improvement can be done quickly before flood happens in that location. A lot of flow models used recently are not able to perform this task accurately. Some flow models have been developed specially for this shock capture purpose but most of them in one-dimensional (1D) mode.

There were some concerns to the adequacy of one-dimensional (1D) analysis of the flow conditions such as contractions, expansions, bends, hydraulic jumps and bridge piers which are commonly found in high-velocity channels. There was a question as to whether computing cross-sectional averaged flow variables provided a sufficiently accurate estimate of flow depths and velocities within these boundary features. Thus, a two-dimensional (2D) analysis was deemed necessary to evaluate these flow conditions.

A numerical model HIVEL2D is used to assess the design computationally before the construction of the physical model begins. Using a numerical model would accelerate this design process and lead to an improved initial physical model thus reducing the time spent on the physical model. This would allow for exploration of more design alternatives in a shorter length of time resulting in a more cost-effective solution.