Environmental Modeling of Storm Water Channels

L. Grinis

Abstract**—**Turbulent flow in complex geometries receives considerable attention due to its importance in many engineering applications. It has been the subject of interest for many researchers. Some of these interests include the design of storm water channels. The design of these channels requires testing through physical models. The main practical limitation of physical models is the so called "scale effect", that is, the fact that in many cases only primary physical mechanisms can be correctly represented, while secondary mechanisms are often distorted. These observations form the basis of our study, which centered on problems associated with the design of storm water channels near the Dead Sea, in Israel. To help reach a final design decision we used different physical models. Our research showed good coincidence with the results of laboratory tests and theoretical calculations, and allowed us to study different effects of fluid flow in an open channel. We determined that problems of this nature cannot be solved only by means of theoretical calculation and computer simulation. This study demonstrates the use of physical models to help resolve very complicated problems of fluid flow through baffles and similar structures. The study applies these models and observations to different construction and multiphase water flows, among them, those that include sand and stone particles, a significant attempt to bring to the testing laboratory a closer association with reality.

Keywords**—**Baffles, open channel, physical modeling.

I. Introduction

HIGH-VELOCITY channels are flood control channels that have certain characteristics. They are typically lined and are designed to discharge supercritical flow through specific reaches. Designers of these channels are primarily concerned with the depth of flow for the designed discharge. Depth determination is complicated by side inflows and boundary features such as contractions, expansions, curves, and obstructions to the flow, such as bridge piers. These boundary features in a supercritical channel cause flow disturbances that can result in a significant increase in local flow depth [1]. Hydraulic structures like canals and bridges are very expensive from the standpoints of construction and maintenance. Because it is often very difficult for design or field engineers to understand the proper functioning of such structures during operation, it therefore becomes essential to test models of such structures hydraulically, in the laboratory, before their actual construction [2]. Physical modeling is the specific branch of scientific research that enables observation of complex phenomena and processes that take place at and upon actual constructions, albeit that these occur on usually models, including their interdependences.

L. Grinis is with the SCE -Shamoon College of Engineering, Basel/Bialik Streets. Beer Sheva, 84100 Israel (e-mail: grinis@sce.ac.il).

Physical hydraulic models are commonly used during design stages to optimize a structure and to ensure its safe operation. Models have another important role to assist non-engineering people during the decision-making process. A hydraulic model may help decision-makers visualize and picture the flow field, giving them information that will help them select a suitable design. In civil engineering applications, a physical hydraulic model is usually a smaller-size representation of the prototype. In particular, for free surface flows the gravitational driving forces–the primary mechanism–must be correctly scaled in relation to inertia (Froude scaling) [3]. A distorted model is a physical model in which the geometric scale is different between each main direction. For example, channel models are usually designed with a larger scaling ratio in the horizontal directions than in the vertical direction: X > Z. The scale distortion does not significantly distort the flow pattern and it usually gives good results [4]. Froude-scaled modeling is based on the concept that complete similarity between model and prototype is achieved when the model displays geometric, kinematic, and dynamic similitude. Geometric similitude implies that all homologous geometric ratios are equal. Kinematic similitude implies that the ratios of homologous particle path lengths to homologous travel times are all equal, i.e., that flow patterns in the model parallel those of the prototype. Dynamic similitude implies that all homologous forces, work magnitudes, and power rates are in the same proportion. Water and sediment movement in a channel is caused by energy differentials and is resisted by shear stresses within the water, between water and sediment, and along the channel boundaries. Their dynamic balance is controlled by the shape and roughness of the channel boundary and material properties such as density, viscosity, and surface tension [5]. Based on the above-mentioned criteria, the scales of the model used in this study were 1:100 in the horizontal directions and 1:50 in the vertical direction. Setting the model and prototype the Froude number equal, results relations between the dimensions and hydraulic. A model can be calibrated to simulate, with approximate accuracy. In this research project we studied problems of storm water multiphase flow through an open channel that included baffles. The storm water channel model we used was built in the hydraulics laboratory of SCE - Shamoon College of Engineering, in Israel. The objectives of the present study were: to construct a physically distorted scale model of a specific storm water channel, to introduce designed baffle elements into the channel, to study the behavior of multiphase water flow with inclusion of sand and stone particles, and to compare the results of the model study with those of the prototype.

II. Experimental Apparatus

A schematic description of the experimental setup is presented in Fig. 1. The system consists of the following components: storage tank (1), centrifugal pump (2), throttle valve (3), flow meter (4), storm water channel (5), baffles (6) and bridge (7). The fluid (in our case water) is circulated from tank (1) through storm water channel (5) by centrifugal pump (2). The flow rate was controlled by throttle valve (3) and measured by flow meter (4).

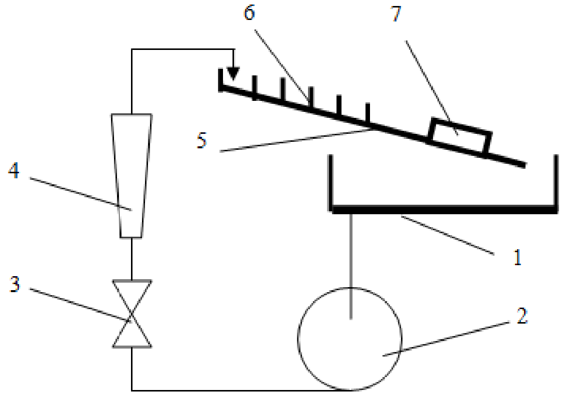

Fig. 1 Experimental setup

Experiments were performed in a 4.8 m long laboratory channel with a 0.1m x 0.4m rectangular cross section. The model of the storm water channel, constructed with a wood bottom and sidewalls, is presented in Fig. 2.

Fig. 2 Model of a storm water channel

We prepared a series of 16 rows of 0.013 m x 0.05 m wood baffles. Each row, which was perpendicular to and traversed the channel from side to side, consisted of approximately five such baffles, each securely anchored in order to prevent being dislocated by the force of the storm water flow. Each row was slightly offset from the row the preceded it.

A photograph of the installed baffles is presented in Fig. 3.

Free flow conditions through the baffles were considered for all the experiments, this condition being common in channels. Water level measurements into the channel were carried out by mechanical gauges (error of 0.05 mm). Inlet flow rate was variable but did not exceed 0.017 m^3/s , which corresponds to the maximum (600 m^3/s) flow rate in the prototype of storm water channel.

Fig. 3 Photograph of installed baffles (Taken from downstream perspective)

This construction of the storm water model, allowed us to simplify the explanation (and increase the understanding) of the hydraulic phenomena occurring during the experiments.

Therefore, the physical model is usually based on the concept that complete similarity between model and prototype is achieved when the model displays geometric, kinematic, and dynamic similitude.

Geometric similitude implies that all homologous geometric ratios are equal. Kinematic similitude requires that the ratio of inertial forces to gravitational force in the model is equal to those of the prototype. Dynamic similitude implies that all homologous forces, work magnitudes, and power rates are in the same proportion.

Water and sediment movement in a channel is caused by energy differentials and is resisted by shear stresses within the water, between water and sediment, and along the channel boundaries. Their dynamic balance is controlled by the shape and roughness of the channel boundary and material properties. The model used in the study was built in accordance with the above-mentioned criteria. During the experimental test, the discharge of water was variable for different case, and the level of the water surface was measured by using a device. When the Froude number is equal for both the model and the prototype, the relationships between the

International Journal of Earth, Energy and Environmental Sciences
ISSN: 2517-942X
Vol:8, No:9, 2014

dimensions and hydraulic characteristics are as shown in Table I.

TABLE I
SCALE RELATION FOR COMPLETE SIMULATION

Characteristics	Dimension
length	X, Z
area	$X \cdot Z$
volume	$X^2 \cdot Z$
velocity	$Z^{1/2}$
discharge	$Z^{3/2} \cdot X$
time	$X/Z^{1/2}$
force	$\rho \cdot Z \cdot X^2$
pressure	$\rho \cdot Z$
manning coefficient	$Z^{2/3}/X^{1/2}$

III. EXPERIMENTAL RESULTS

The experimental apparatus allows us to explore different conditions of the flow-through channel. Our goal was to use baffles in a storm water channel. We studied this model for two cases, one with and one without baffles. The design utilized a water flow rate through the bridge at 400 m^3/s. We also tested the possibility of a flow rated up to 600 m^3/s of water through the bridge.

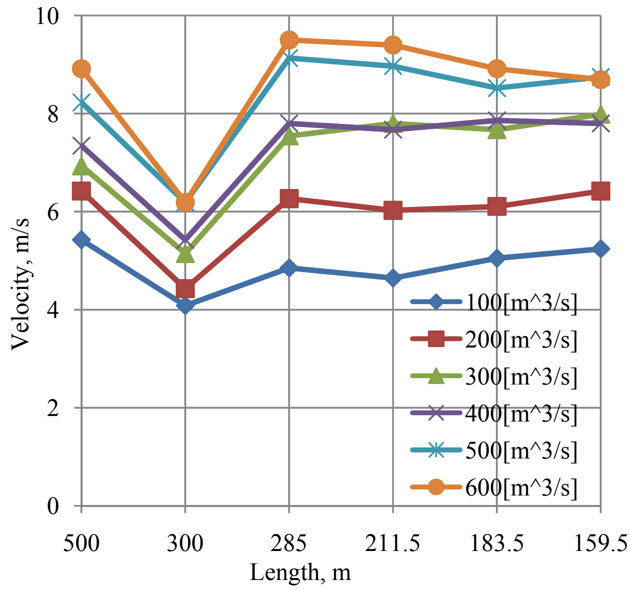

Fig. 4 Velocity of the water in the experiment versus the length of the channel, with baffles, for different discharges

The results of the measurement of the velocity of the water in the experiment versus the length of the channel, with baffles, for different discharges are presented in Fig. 4. It is evident that the velocity of the water depends on the flow rate and length of the channel. The experiments showed that the velocity of the water was greatly reduced when it was moving through the baffles. This can be seen in the Fig. 4, when the velocity of water through the bridge does not exceed 8 m/s.

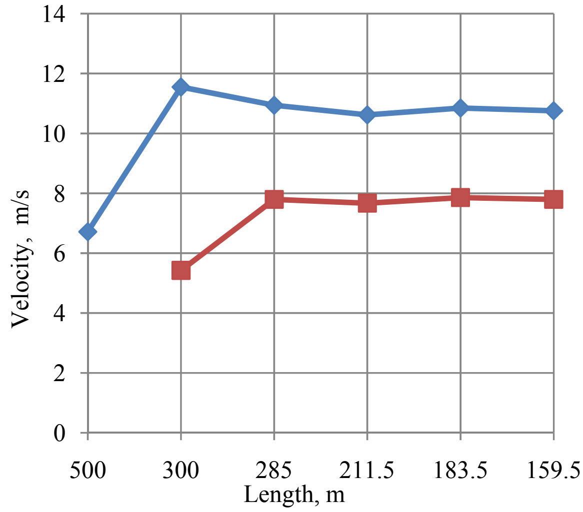

Fig. 5 Velocity of the water in the experiment versus the length of the channel, both with and without baffles (flow rate 400 m^3/s)

Fig. 5 represents the results of the measurements of the velocity of the water in the experiment versus the length of the channel, both with and without baffles (flow rate 400 m^3/s). The line graphs in Fig. 5 show that baffles significantly reduce the velocity of water flow through the bridge.

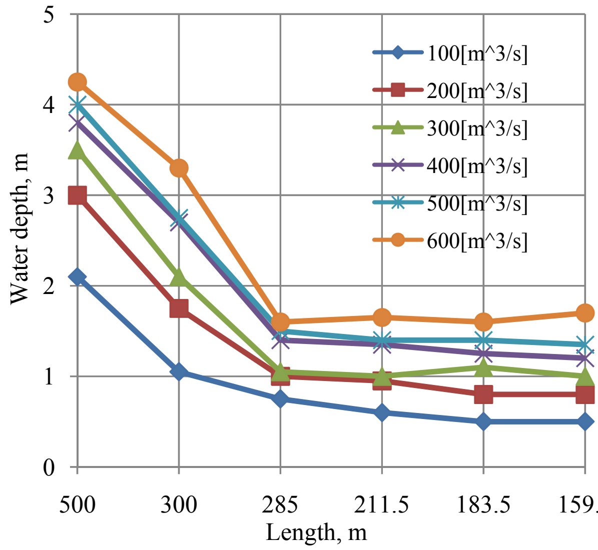

Fig. 6 Water depth in the experiment versus the length of the channel, with baffles for different discharges

Fig. 6 shows the height of water flowing through the bridge. The experimental results showed that water depth does not exceed the height of 1.4 m, when the design flow flowing via the bridge. Studies have shown that the structure of the bridge allows pass the flow to 600 m^3/s.

Comparison (Fig. 7) of the theoretical calculation with experimental data, showing that the difference between the

International Journal of Earth, Energy and Environmental Sciences
ISSN: 2517-942X
Vol:8, No:9, 2014

theoretical calculations and the experiments does not exceed 13.6%. For the studied model, this is quite sufficient accuracy.

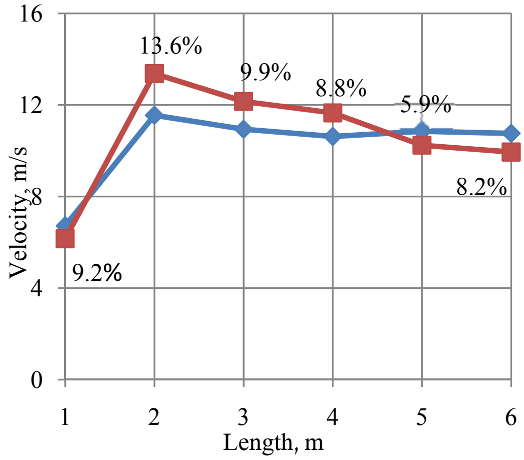

Fig. 7 Comparison of the theoretical calculation with experiment

The study also tested the effect of water flow rate and time of the change, on the self-cleaning of the channel in which the baffles are installed. Because baffles reduce the passage of sediment through the channel, there is a probability of sediments being deposited in the channel. We needed, therefore, to test the effect of the flow rate on the mechanism of sediment deposition, as well as the effect of the time of the changes of water flow on the quantity of the sediment that remained in the channel. In Fig. 8 shows the quantity of sediment that remained after a flow of water through the prototype, with a discharge flow rate velocity that did not exceed 30 m^3/s. This experiment shows that under certain water in the channel remain sediments which must be removed.

Fig. 8 Self-cleaning channel in percentage of the sediment (Taken from upstream perspective)

Fig. 9 represents the results of the measurements of the self-cleaning channel, showing the percentage of sediment produced under various conditions of discharge. The figure shows that the self-cleaning benefits of channels with baffles do not begin until water flow exceeds 40 m^3/s. Thus, when there is a low rate of flow, the sediments remain in the channel. This requires the use of additional equipment and personnel to physically clean the channel. It is important to note that channel self-cleaning also depends not only on the rate of flow of the storm water, but also on the length of time that the flow was sustained at that rate.

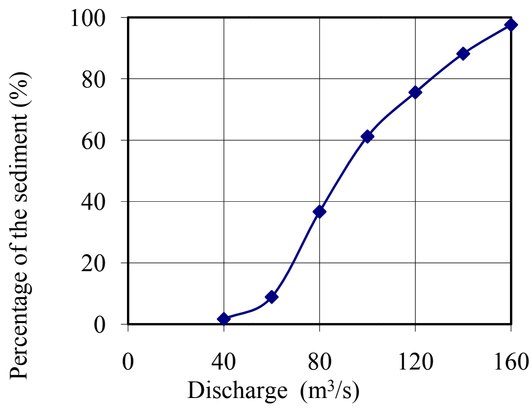

Fig. 9 Self-cleaning channel in percentage of the sediment versus flow rate

Not to be disregarded are the equally important but contrary implications of our study. Among these is the suggestion that while maintaining a 40 m^3/s or lower discharge rate enhances operations that are designed to reduce the amount of downstream sedimentation, a significantly higher rate acts to maximize the fluid movement.

IV. Conclusions

In this paper the results of experimental investigations of models for a storm water channel are presented. The experiments were performed in a laboratory using different models. The theory used to scale the field prototype is discussed, the construction and scaling criteria of the models are presented, and the results obtained from the model channel are compared with the prototype channel. A self-cleaning channel that removes sediment from the flow is presented. This work demonstrates the possibility of using physical models to study very complicated effects in water storm channels.

References

[1] R.Ettema, M. Muste , "Scale-Effect Trends in Hydraulic Models of Flow at a Dike on a Flat-Bed Channel". *Hydroscience & Engineering College of Engineering The University of Iowa*. Technical Report No. 426, 2002.
[2] R. Arndt, P. Robers, and T. Wahl, "Hydraulic modeling concepts and practice". ASCE Manual No.97, 2000.
[3] H. Chanson, C. Gonzales. "Physical modelling and scale effects of air-water flows on stepped spillways". *J. of Zhejiang University Science*, 1009-3095, 2005.
[4] H. Chanson . *The Hydraulics of Open Channel Flow*. Arnold, 338 Euston Road, London NW1 3BH, UK, 1999.
[5] N. Wallerstein. "Distorted Froude-Scaled Flume Analysis of Large Woody Debris". *Earth Surf. Process. Landforms* 26, 1265–1283, 2001.