« Research Note »

Calibration of Water Distribution Hydraulic Models Considering Different Decision Variables and Consumption Scenarios

M. Tabesh[1*], M. Jamasb[2], R. Moeini[3]
1. Associate professor, Centre of Excellence for Engineering and Management of Infrastructures, Department of Civil Engineering, University of Tehran
2. M. Sc. Student, Department of Civil Engineering, University of Tehran
3. PhD Student, Department of Civil Engineering, Iran University of Science and Technology

mtabesh@ut.ac.ir

Abstract
In recent years, hydraulic models are widely used to simulate hydraulic performance of water distribution systems. They play a significant role in evaluation and operational management of these systems. Calibration is necessary for improvement of model function and reduction of the uncertainties during different consumption scenarios in the system. At the moment just a few commercial models are capable of calibration calculation besides the hydraulic performance evaluation. However, the popular and freely available model, EPANET can not be calibrated. In this paper, an optimization procedure is proposed using Genetic Algorithm (GA) for calibration of the hydraulic simulator. For this problem, the pipe roughness (Hazen Williams) coefficients, nodal demands and pipe diameters are considered as decision variables of the optimization problem. The procedure is implemented for four different consumption scenarios: normal, minimum, maximum and fire flow. Finally, to evaluate the proposed methodology, a test network at different consumption scenarios is calibrated and the results are compared. Comparison of the results shows that the best result with minimum error has been obtained when the fire consumption with considering both variables of Hazen-William coefficient and nodal demand was selected.

Key words: Calibration, Hydraulic Simulation Model, Water Distribution Network, EPANET, Genetic Algorithm.

Comparison of Flow Characteristics in a Physical Model with Three Numerical Models in a River Reach, under two Different Training Schemes

L. Jalili[1] , M. Yasi[2*]
1. Former M.Sc. Student of Hydraulic Structures, Urmia University, Urmia.
2. Assistant Professor of River Engineering, Urmia University, Urmia.

m.yasi@urmia.ac.ir

Abstract
Application of numerical models in river engineering projects is unavoidable. However, the reliability of model, minimum requirement for field data, and fewer computational processes are of major concern. The main purpose of the present study was to test the reliability of some well-known numerical models in the simulation of flow characteristics in different river planforms. Comparison were made between the flow characteristics in a river physical model and the simulation results from three numerical models, under two different river training schemes, with identical initial and boundary conditions.
In this study, a reach of the Nazloo River in Urmia, Iran, was selected. A fixed-bed physical model of the River Reach (1200 m long, with horizontal scale of 1:100 and vertical scale of 1:20, and with the Nazloo cross bridge included within the reach) was constructed, calibrated and verified. Two training schemes were planned, using longitudinal and transversal structures, i.e, levees and groynes. Flow parameters (such as,: depth and point velocity) were measured for four different flows. Three river models: 1D model HEC-RAS, Quasi-2D model BRI-STARS, and 2D model FAST-2D were selected. Six flow parameters (i.e. river flow capacity, water surface elevation, mean flow depth, mean velocity, mean bed shear stress and Froude number) were compared between the physical model and the three numerical models. Four different flow conditions were examined, and the results were compared in three sub-reaches along the river reach (from downstream to upstream of the bridge).
The averaged predictive errors from these three models were determined for corresponding flow parameters. The HEC-RAS, FAST-2D, and BRI-STARS models are considered to be the best fitted models with the true physical model, respectively. Simulation results from the HEC-RAS model are well adapted to the river flows confined between the two-sided levees, where the geometry of the river reach is more uniformly defined, and in minor flood flows. The prediction from the FAST-2D model is superior along the river reach with groynes involved, particularly in higher flow levels. The application of each of the three models is recommended in river projects subject to the inclusion of the order of certainties provided by the present study.

Key words: HEC-RAS, BRI-STARS, FAST-2D, Nazloo River, Groynes, Levees.

Numerical Simulation of Submerged Vanes at 180° Bend with Intake in order to Identify the Optimum Vane's Angle and Height

M. Rostamabadi[1], S.A.A. Salehi Neyshabouri[2*], H. Montaseri[3].

1. PhD Student of Civil Engineering, Tarbiat Modares University
2. Professor of Civil Engineering, Water Engineering Research Inst., Tarbiat Modares University
3. Assistant professor of Civil Engineering, Yasooj University

salehi@modares.ac.ir

Abstract

Submerged vanes are plane structures placed on river bed with an angle to the approach flow which cause secondary flow. They are used to reduce sediment entrainment to water intakes structures. The vane's geometric parameters affect the flow pattern around the vanes. Numerical solution of the three-dimensional Reynolds-Averaged Navier-Stokes equations for turbulent flows around submerged vanes placed in front of the intake in curved channel with 180° bend using the Fluent software in order to identify the optimum vane's angle and height and the last vane position is the aim of this paper. The results of numerical model are compared with available experimental data.

Investigations show that the optimum angle of attack for the first vane is 20° to approach flow and the optimum ratio of the vane height to the flow depth is 36%. Also the last vane position just off the downstream corner of the intake is recommended.

Key words: Submerged Vanes, 180° Bend, Intake, Optimum Angle, Optimum Height.

Application of Genetic Algorithm for Determining the Existent Friction Coefficients in the Real-life Pipe Networks as an Inverse Problem

J. M. V. Samani[1*], M. Shahbazi[2]
1. Associate Professor, Department of Water Structures, College of Agriculture, Tarbiat Modares University
2. M.Sc. Student of Water Structures, Department of Water Structures, Tarbiat Modares University

[*] **Samani_J@modares.ac.ir**

Abstract

An integrated approach for determining the existent friction coefficients in water networks is proposed. As pressurized systems age, the carrying capacity of network decreases because the internal roughness increases with the aging of pipes. This can lead to loss of satisfactory performance and uneconomic operation. Therefore, rehabilitation of an existing network becomes very important problem in water industry. Determining the real physical characteristics of pipes for network analysis is a regular component of the rehabilitation process. Pipe friction coefficients cannot be determined explicitly by direct measurement, they are determined implicitly, as an inverse problem, from measured model outputs (pressures). Values of friction coefficients are determined in a way that they should yield a reasonable match between measured and predicated pressures in the network. One problem associated with the Re-calibration of real-life pipe networks is the lack of field measurements, which can sometimes, lead to the formulation of an ill-posed inverse problem. In this study certain methods have been utilized to tackle this problem. The hydraulic analysis of steady and quasi-steady flow and optimization process are combined to develop a program. By selecting a proper optimization method (Genetic Algorithm) the inverse model was developed and verified successfully in a case study.

Key words: Optimization, Genetic Algorithm, Inverse Problem Technique, Friction Coefficient, Pipe Networks.

Numerical Investigation of Air Detrainment Process in Chute Flows

A. Bahrami[1]; G. A. Barani[2*]
1. M. Sc. in Water Engineering, Shahid Bahonar University of Kerman
2. Professor of Civil Engineering Department, Shahid Bahonar University of Kerman

[*] **gab@mail.uk.ac.ir**

Abstract
One of the most important factors in designing spillways is to consider sufficient discharge capacity. Possible creation of negative pressure over spillway during high flood conditions can be a potential for occurrence of cavitation damage. Flow aeration might be one of the best ways to prevent this phenomenon. Realizing the air detrainment process along the chutes is a key problem in designing aerators. Although experiments on hydraulic models have their own advantages, but these methods are expensive and time consuming. In this study the CFD code FLOW-3D® has been used to determine the spillway flow air concentration in case of aeration by means of deflectors. Comparison of obtained results with the Kramer's experimental results shows good agreement. It has been realized that the flow bottom air concentration downstream of deflectors reduces exponentially.

Key words: Spillway, Cavitation, Flow Aeration, FLOW-3D, Air Concentration.