Study of Dynamic Pressure Distribution over the Bed of Compound Flip Buckets

O. AminoroayaieYamini[1*], M.R. Kavianpour[2]

1. M.Sc. Student, K.N. Toosi University of Technology, Tehran, Iran
2. Associate Professor, K.N. Toosi University of Technology, Tehran, Iran

[*] **O.aminoroaya@sina.kntu.ac.ir**

Abstract

Flip buckets are usually placed at the end of chute spillways or outlet conduits of high dams. The projected jets from these buckets jump into plunge pools to dissipate the destructive energy of plunging jets. Compound flip bucket is a special bucket with a non-zero degree of cross sectional slope, followed by a curvature in its longitudinal direction. These structures are very suitable for restricted geometries and conditional topographies. Although, studies of flow over flip buckets started many years ago and valuable information is available for similar buckets, still there are uncertainties regarding the flow over compound flip buckets. Therefore, in this study the hydraulic parameters such as pressure, velocity and depth of flow on these structures were evaluated. Two chute spillways with their flip buckets were used to check the pressure distribution and its variation with different hydraulic characteristic along the compound flip buckets. The scaled model of Gotvand dam (south of Iran) at Water Research Institute of Iran was used for this study. The Froude number varies in the range of 3.5 to 7.5. The results are presented in the form of dimensionless equations, which are calibrated based on the present results. The location of maximum dynamic pressure on the bed of the compound flip bucket is also determined.

Keywords: Flip Bucket, Compound Bucket, Pressure Distribution, Dynamic Pressure, Physical Model.

Armoring Effect on Local Scouring around the Bridge Pier under (Part1 :) Steady Flow

A.R. Kabiri-Samani
Associatet Professor, Civil Engineering Department, Isfahan University of Technology, Isfahan, Iran

akabiri@cc.iut.ac.ir

Abstract

In this study armoring effect on local scouring around bridge pier under steady flow condition was investigated based on model experimentation. Different combinations of bed-armor layers were employed. Three circular piers of 22, 33 and 42 mm were used. Results show that while small bed particles are under the protective effect of armor layer, increasing the total scouring depth, yields an increase in the secondary armor layer depth until the armor layer itself is not influenced by the horse-shoe vortices. The minimum stabilized secondary armor layer is identical to the armor grain size. Results showed that there is not meaningful correlation between total scouring depth and B/d (where B is the pier diameter and d is the mean bed grain sizes) for B=33 and 42 mm. However for B=22 mm, increasing B/d tends to increase the total scouring depth. For the armor layer particles size of 2.36 mm the armoring effect for B=22 mm is vanished. The most important parameters in evolution of scouring depth are the armor layer grain size and the armor layer thickness.

Keywords: Local scouring, Bridge pier, Armor layer, Steady flow.

Development of a Numerical Model for Predicting Dynamic Pressures in Open Channel Flow using Non-Orthogonal Curvilinear Coordinate Grid

Zohre Dehghan Bahabadi[1], Mohammad Reza Hadian[2*], Amir Reza Zarrati[3]
1. M.Sc. Student, Department of Civil Engineering, Yazd University, Yazd, Iran.
2. Assistant Professor, Department of Civil Engineering, Yazd University, Yazd, Iran.
3. Professor, Dept. of Civil & Environmental Engineering, Amirkabir University of Technology, Tehran, Iran.

[*] **mr_hadian@yazduni.ac.ir**

Abstract

A 2DV numerical model is developed for solving unsteady Navier-Stokes equations with the ability for calculating dynamic pressures in non-orthogonal curvilinear coordinate grids for free surface flows. The model is based on fractional step method (or projection method). The free surface elevation is calculated using continuity equation. The present method uses the advantages of shallow water models for calculating free surface elevation (even in cases with large changes) and benefits the flexibility of moving non-orthogonal grids. Moreover, this method presents the ability of developing a shallow water solver to take into account the dynamic pressures.

The simulation of water elevation in a gradually varied flow, showed the ability of mesh adaption in moving and curved boundaries. Comparing the results of the model with experimental data and other numerical models, for flow passing over a hump and trench, confirms the ability of the developed model for simulating the free surface flows. Having the ability of calculating free surface and dynamic pressures, the model can therefore be applied to rapidly varied flows.

Keywords: Free Surface, Projection Method, Dynamic Pressure Equation, Momentum Interpolation, Curvilinear Coordinate System.

The Effect of Wastewater on the Non-Settling Velocity and Shear Stress of Cohesive Sediments

M. Khastar-Borujeni[1], H. Samadi[2*], K. Esmaili[3]
1. M.Sc. student of Water Engineering Department, Ferdowsi University, Mashhah, Iran
2. Assistant Professor, Water Engineering Department, Shahrekord University, Iran
3. Assistant Professor, Water Engineering Department, Ferdowsi University, Mashhah, Iran

[*] **Samadi153@yahoo.com**

Abstract
Sedimentation in open channels is one of the basic problems in the management of irrigation networks. Complexity of behavior of cohesive sediment transport makes determination of the criteria for open channel design to be difficult. The purpose of this study is to determine non-settling velocity and non-settling shear stress for open channel flow containing wastewater and suspended cohesive sediment. Experiments were carried out in an annular flume located in the hydraulic laboratory of Shahrekord University, Iran. The sediment samples were taken from Pirbalut dam reservoir, located upstream of Northern Karun basin. The result showed that wastewater content in flow causes the suspended sediment deposition and flocculation to be likely increased. It also caused the threshold of non-settling velocity to be increased at a rate of 10% compared to pure water. A similar result was also obtained for threshold of shear stress. The results obtained from this study were compared and the results showed that the empirical methods were not capable in calculating non-settling velocity for open channel flow containing waste water and suspended cohesive sediments. The results also showed that the Froud number should be more than 0.47 for non-settling condition of suspended cohesive sediments.

Keywords: Cohesive Sediment Deposition, Shear Stress Threshold for Deposition, Annular Flume, Acoustic Doppler Velocimeter.

Coupled Analysis of Fluid Transients and Structural Dynamic Responses in Pipelines by MOC-FDM

S. M. Jazayeri Moghaddas[1*], H. Mohamadvali Samani[2]
1. Instructor, Hydraulic Structures, Islamic Azad University, Shoushtar Branch, Iran
2. Hydraulic Professor, Shahid Chamran University of Ahvaz, Iran

[*] **mahmoodjazayeri@yahoo.com**

Abstract

Water hammer is a type of transient flow in pipelines which may in general be produced due to valve closure or sudden shut down of pumps. Transient flow creates sudden variation in the velocity and pressure of fluid flow in the pipelines which in effect may induce some dynamic vibrations in the structure of pipe. Furthermore, such structural vibrations can affect the fluid flow. Taking into account them is necessary for modeling of fluid-structure interaction. In this paper the fluid-structural interaction (FSI) of water hammer in pipeline systems are investigated. Two sets of hydraulic and structural equations with common variables have been numerically analyzed through MOC-FDM simultaneous solving process which has been prepared by a computer-assisted program. Some examples have been solved by this method, with satisfactory results. This indicates that fluid-structure interaction can have critical effect on the value of fluid pressure.

Keywords: Pipeline, Transient Flow, Water Hammer, Fluid-Structure Interaction, Coupled Analysis.

Investigation of Effective Parameters on the Hydrofoil Performance near the Free Surface of Water

M.H. Javareshkian[1], A. Esmaeili[2]
1- Associate Professor, Ferdowsi University of Mashhad, Iran.
2- M.Sc. student, Ferdowsi University of Mashhad, Iran.

[*] **javareshkian@ferdowsi.um.ac.ir**

Abstract

In this study, a numerical method is used to assess the effect of thickness, camber, submergence depth and flow velocity around a submerged hydrofoil. A pressure based algorithm is used to solve Navier-stokes equations, and Volume of Fluid (VOF) approach is applied to simulate two-phase fluid (water and air). The $k-\varepsilon$ model and second order upwind scheme have been used for turbulence models and differential schemes, respectively. For verifying, a part of results is compared with published experimental results. The numerical and experimental results show excellent agreement; thus, the process of numerical simulations is confirmed. The results of this study illustrate that the lift and drag coefficients increase with rising the thickness of the hydrofoil, but the lift to drag ratio has a downward trend. On the other hand, the lift and drag coefficients increase with growing the camber of the hydrofoil, but lift to drag ratio has upward trend. Therefore, to opt the best section of the hydrofoil, the less thickness and the high camber are more desirable to getting high hydrodynamic performance. The switched of the hydrofoil section obviously affects the airflow and boundary layer formed on the free surface of water. Moreover, lift and drag coefficients become high by increasing the submergence depth and angle of attack, as a result lift to drag ratio increases. Lift and drag coefficients are decreased by increasing flow velocity, and lift to drag ratio has a downward trend.

Keywords: Hydrofoil, Wave, Lift, Drag, Hydrodynamic Performance.