« Research Note »

The Effect of Nonlinear Form in Direct Numerical Simulation of Channel Flow Analysis

E. Rajabi[1], M.R. Kavianpoor[2*]
1. Ph.D. Candidate, Khaje Nasir Toosi University of Technology
2. Associate Professor, Department of Civil Engineering, Khaje Nasir Toosi University of Technology

[*] **kavianpour@kntu.ac.ir**

Abstract
Analysis of turbulent plane channel flow is performed by direct numerical simulation. In this paper common forms of the nonlinear term are applied to solve Navier-Stokes equations by using pseudo spectral method. The Navier-Stokes equations are expanded with Chebychev and Fourier series in PS method and commonly used forms of the nonlinear terms $u_{tot}.\nabla u_{tot}$ are explained and applied, referred to as divergence, convection, skew-symmetric, rotational, alternating and linearized form. The results of channel flow analysis are presented due to those six forms of the nonlinear terms .The computational grids of 128×65×128 in computational domain of Ω=[0,4π]×[-1,1]×[0,2π] are used in the x, y and z directions, respectively. The friction Reynolds number for channel flow is set to be Re_τ=175.
The comparison is made between turbulent quantities such as the turbulent statistics $\overline{uu}, \overline{vv}, \overline{ww}$ The results show that that the rotational and linearized forms are more appropriate than other schemes, especially than skew-symmetric form. But it must be noted that linearized form is not energy-conserving so it is not stable and applicable in all situations. Hence the choice of nonlinear term reduces to the rotational form in analysis of pseudo spectral method which satisfies robustness and accuracy and reduces the computational cost at about 62% (compared to other schemes). From practical point of view, different types of nonlinear term do affect in reduction of total running time at higher Reynolds number and greater domain in fluid flow, which provides decreased cost and time in direct numerical simulation of channel flow if final decision rests on economics.

Keywords: Channel Flow, Pseudo Spectral Method, DNS, Nonlinear Term Forms.

Hydrodynamic Performance of Floating Breakwaters under Regular Wave

E. Kakavand[1], M. Kolahdoozan[2*]
1. Graduate Student, Department of Civil and Environmental Engineering, Amir Kabir University
2. Assistant Professor, Department of Civil and Environmental Engineering, Amir Kabir University

[*] **mklhdzan@aut.ac.ir**

Abstract

Floating breakwaters (FB) represent an alternative solution to protect coastal areas from wave attack and erosion. The major advantages of using floating breakwater in comparison to conventional fixed ones are lower cost of construction, using in severe hydrodynamic conditions and high speed installation. Therefore, engineers lead to design and use floating breakwaters in those situation in which this type of breakwater is more suitable to construct. To predict floating breakwater behavior, it is essential to investigate flow characteristics in the vicinity of structure. In this regard, numerical modeling is presented based on the solutions of 2DV Reynolds Averaged Navier-Stokes (RANS) equations and the VOF method using in Fluent Software.

To generate wave, a sub-model has been developed and hooked to the main software. Results obtained from the numerical model then validated by experimental data cited in the literature to represent the capability of model in simulation of complicated problems. Numerical modeling shows that the single fixed FB operates well but, the attached plate in the front part of the FB significantly enhances the efficiency of the structure and with regard to cost-effectiveness, the configuration of the FB with the attached plate should be considered for design purposes. Also, double FBs and trapezoidal ones with inclined faces of different angles are simulated. It is found that the trapezoidal FBs are the most efficient in comparison to the other types of FBs.

Keywords: Floating Breakwater, Regular Wave, Numerical Model, Turbulent Navier-Stokes Equation, VOF, Transmission Coefficient.

Water Hammer in Polyethylene Pipes

A.R. Keramat[1], A. Haghighi[2*]
1. Assistant Professor, Civil Engineering Faculty, Jundi-Shapur University of Technology
2. Assistant Professor, Civil Engineering Department, Faculty of Engineering, Shahid Chamran University

*** a.haghighi@scu.ac.ir**

Abstract

This research aims to model the dynamic viscoelastic effects of polyethylene pipes under water-hammer pressures. As known, following the water hammer event, a cyclic loading and unloading due to the pressure fluctuations occurs. Loading on polymeric materials results in deformations that unlike the elastic materials do not immediately return after unloading. This retarded behaviour gradually reduces the wave speed causing to a different transient response. Herein, the viscoelastic effects of polyethylene pipes on the transient flow is numerically simulated and studied using the Kelvin-Voigt model. The developed model is then verified using some experimental data from the literature. Afterwards, a simple hypothetical reservoir-pipe-valve system is defined to investigate several aspects of viscoelasticity. It is concluded that in a standard water hammer caused by valve closure, the minimum pressure head is more affected by viscoelasticity than the maximum one. Furthermore, with the increase of initial flow, viscoelasticity takes part more to damp out the destructive dynamic effects of water hammer. This makes the viscoelastic properties of polyethylene pipes to be figured out as an adaptive procedure to automatically alleviate the water hammer issues in water supply systems.

Keyword: Water Hammer, Polyethylene, Viscoelastic, Numerical Models.

Determination of Flood Flow Depth and Velocity at the Threshold of Crop Rapture and Position of Rupture Point Using Analytical and Laboratory Analysis (Case Study: Rice)

Z. Ganji Noroozi[1], A.R. Shokoohi[2], J. Mohammad Vali Samani[3]
1. PhD. Student, Hydraulic Structures Department, Tarbiat Modares Uinversity, Tehran, Iran.
2. Associate Professor, Water Engineering Department, International University of Imam Khomeiny, Qazvin, Iran.
3. Professor, Hydraulic Structures Department, Tarbiat Modares Uinversity, Tehran, Iran.

* **samani_j@modares.ac.ir**

Abstract

Investigation on the behavior of plants, when exposed to hydrodynamic forces, is of vital importance in flood analysis and evaluation of its consequent losses, design of channels and water resources management. Because of the important effects of flow depth and velocity on plants' bending and rapture point, the evaluation of which is necessary for flood loss estimation and its management across the watershed. Studying the bending of plants and their probable rapture is not easy on account of complicated interaction between plant and flow characteristics. In this paper, an analytical method is presented for plant rapture in an un-submerged condition and static state and the outcomes are compared with laboratory results. In this study, the multiplication of depth by flow velocity is used as an index for the analysis of the plant rapture. The differences between the results of the analytical solution and laboratory tests for the rapture index and the maximum tensile stress were less than 0.5 and 0.1 percents respectively. The results of the analytical solution and laboratory tests reveals that under the same hydraulic conditions the rapture point for rigid plants with fixed diameter is located in the plant's base, but this point will move upward for plants of variable diameter. In this regard, if the reduction of the plant's diameter is relatively high, this point will be closer to the water surface.

Keywords: Maximum Tensile Stress, Rapture Point, Flow Hydraulics, Rice, Flood.

The Effect of Flow Diversion on Sediment Entering Intake Located at Section 52^o in Sinuous Rivers

M. Abolghasemi[1*], M. Ghodsian[2], H. Sanginabadi[3]

1. Assistant Professor, Water Research Institute.
2. Professor, Faculty of Civil and Environmental Engineering, Tarbiat Modares University.
3. M.Sc. in Civil Engineering, Ghazvin Water District Company.

[*] **ms_abolghasemi@yahoo.com**

Abstract

The outer bank in meandering rivers is suitable place for flow diversion. Flow diversion from the outer bank of channel lead to formation of a transverse flow component having an opposite alignment with the secondary current in bed and consequently transferring a part of the sediment bed load to intake. The diversion of flow will change the flow pattern in the river bend and decrease the energy of the secondary current in this location. In this paper the effect of flow diversion to sediment entering the intake in a 52 degree diversion angle was investigated. The results have shown that the low ratios of diverted flow has a little effect on the secondary current, and also this current causes the repelling of sediments from the outer bend of the river. Along with the increase of the ratio of diverted flow, the energy of secondary currents decrease and the ratio of sediment diversion increases. In a specific value of flow diversion, the secondary flow was fully damped. After that, the sediments entering the intake have a higher increase compared to the previous situation. The results also showed that the trend of variations of diverted sediments with the flow diversion generally follows the S shape.

Keywords: Lateral Intake, Flow Rate Diversion, Sediment Rate Diversion, 52 Deg. Diversion Angle, Sinuous River.

Hydraulics of Flow over the Weir of Subsurface Dams

M.R. Shams[1], A.R. Kabiri-Samani[2*], M.K. Beirami[3], and H.R. Safavi[4]
1. M.Sc. Student, Civil Engineering Department, Isfahan University of Technology
2. Associate Professor, Civil Engineering Department, Isfahan University of Technology
3. Associate Professor, Stahban Azad University
4. Associate Professor, Civil Engineering Department, Isfahan University of Technology

[*] **akabiri@cc.iut.ac.ir**

Abstract
One of the most substantial water resources in water engineering is the groundwater, stored in aquifers. In arid and tropical regions having monsoon climate, the groundwater accumulated under the ground and used later during dry seasons. The use of subsurface dams in aquifers to store water behind the dam is one of the common techniques in recent years. But a weir should be constructed in subsurface dam body to convey the excess water from the aquifer. Without an appropriate weir to withdraw the excess water, groundwater level will reach the ground surface and in addition to disrupting the lands application, will cause inevitable damages to the environment. In this study, the effects of characteristics of weir together with the aquifer hydrodynamic properties on groundwater flow discharge are studied. The effects of different hydraulic parameters on the flow rate through the weir of the dam are also investigated. An experimental model of the aquifer, including underground reservoir, dam and weir is built and the effects of different parameters such as hydraulic conductivity of the aquifer, head above the weir, height of the weir and weir crest length on the discharge through the weir in steady state condition are tested. The results show that increasing the hydraulic conductivity of the aquifer, yields an increase of the flow discharge. By increasing the weir height, the head above the weir or the weir crest length, the discharge increases. In order to evaluate all the influential parameters and analyze their effects on flow discharge through the subsurface weir, the Seep3D software is used. The results of numerical models are verified using the measured data. Then different models of underground dams and were prepared and the effect of each influential parameter on the groundwater discharge through the weir was studied. Results of numerical models showed that the hydraulic conductivity of aquifer, head above the weir and the effective width of the aquifer are the most significant parameters to establish a certain groundwater discharge through the weir.

Keywords: Aquifer, Subsurface Dam, Flow Hydraulics, Weir, Hydraulic Conductivity.

Evaluation of the Various Methods of Shear Stress Determination in a Sharp Channel Bend with Developed Topography

J. Mozaffari[1*], E. Amiri-Tokaldani[2], M. Khademi[3]

1. Assistant Professor, Water Engineering Group, Arak University.
2. Assistant Professor, Irrigation Group, Tehran University.
3. PhD. Student of Hydraulic Structures, Tehran University.

[*] **Javad_370@yahoo.com**

Abstract

Shear stress is one of the important parameters in river flow studies and has a key role on river pattern and also estimating the amount of erosion and sedimentation of fluvial material within the river. Therefore, estimating this parameter in different positions of the field flow, especially next to the river bed and banks, is vital. Despite doing intensive studies by many researchers on the amount of shear stress and its distribution in straight and curved channels, due to the complex flow regime within the bends, more studies have to be carried out. In this research, the results obtained from various methods of shear stress estimation are compared with those obtained by using of mean velocities and their fluctuations taken by an ADVP instrument. The compared methods include Reynolds shear stress, Turbulent Kinematic Energy, and Logarithmic Profile, have been previously used and compared in straight channels, but never have been used and compared in a sharp bend. The results obtained from the study on a 193^0 bend with developed topography showed that unlike the straight channels, the strong 3D flow changes the Reynolds stress patterns in sharp bend, so that there is no specific order to get a reliable estimation of bed shear stress in this type of channels. Moreover, because of the effects of strong secondary current, the longitudinal velocity profiles in sharp bends are flatter than those in straight channels, and hence there are some uncertainty regarding using logarithmic profile distribution. The results of this study showed that in general, the Logarithmic method to estimate the bed shear stress gives better results among the available mentioned methods.

Keywords: Bed Shear Stress, Reynolds Stress, Logarithmic Velocity Distribution, Turbulent Kinematic Energy, Sharp Bend.