Numerical Simulation of Wave-Structure Interaction around an Obstacle

Sonia Ben Hamza[1], Sabra Habli[2], Nejla Mahjoub Saïd[2],
Hervé Bournot[3], and Georges Le Palec[3]

[1] LGM, National Engineering School of Monastir, University of Monastir, Tunisia
`soniabenhamza@yahoo.fr`
[2] LGM, Preparatory Institute for Engineering Studies, University of Monastir, Tunisia
`sabra.habli@fsm.rnu.tn, nejla.mahjoub@fsm.rnu.tn`
[3] IUSTI, UMR 7343, Faculty of Sciences, University of Aix-Marseille, France
`georges.lepalec@univ-amu.fr`

Abstract. In this work, we study a turbulent two-phase free surface flow around an obstacle in unsteady regime. A dynamic study relating to the formation of coherent vortex structures enables us to determine the shape of the flow and to clarify its main characteristics (shear layer, recirculation and reattachment). We determine first the dynamic structure of the flow through a numerical approach using the computer code ANSYS Fluent (closure model is k-ε). In the second part we study the impact of these vortices on such configurations. A series of numerical simulations have been conducted to further verify the applicability of this model for wave simulations interaction with vortex structures of various shapes.

Keywords: Turbulence, obstacle, free surface, wave-structure interaction, VOF.

1 Introduction

The wave-structure interaction (WSI) is one of the most important coastal and ocean engineering applications of free surface flow hydrodynamics. The physical understandings and robust numerical computations of the wave-structure interactions are crucial to assess the wave impact on structures as well as the structural responses to these wave attacks. In many numerical models, the wave-structure interaction problems can be solved by employing either a simultaneous solution or a partitioned solution, and the solid structures are often discretized into a Lagrangian form (Lin 2007, Liu et al. 2013, Maruzewski et al. 2009, Shao 2009). Although it is anticipated that the vortices generated around the submerged obstacle have only minor effects on the wave energy transmission (without wave breaking) and stability of the submerged obstacle, this localized rotational and dissipative vortices have a definite impact on the mixing process, sediment transport and scouring process.

The 3D computation of wave-structure interaction based on Navier-Stokes equation (NSE) can be very expensive, considering the fact that the model needs

M. Chouchane et al. (eds.), *Design and Modeling of Mechanical Systems - II*,
Lecture Notes in Mechanical Engineering, DOI: 10.1007/978-3-319-17527-0_68

to resolve the free surface, the bottom topography, and the surface of a body. As a matter of fact, the simpler problem of 3D wave propagation over an uneven bottom sometimes is challenging enough to a modeler if the domain size is too large. This was the reason that in the earlier age of wave simulation, the 3D model was developed in which the hydrostatic pressure is assumed and the total water depth is mapped into a σ-coordinate. For example, Casulli and Cheng 1992 presented a three-dimensional model. The model was used to simulate the flooding and drying of tidal waves. Such an approach has been adopted in many other ocean circulation models. It is realized, however, that all the above approaches are only applicable for relatively long waves or ocean currents for which the assumption of hydrostatic pressure is valid.

Interactions between water waves and submerged marine structures are important in the solution of many coastal engineering problems. With rapid advances in computing technology, more researchers and engineers are using numerical simulations to better understand the fluid-structure interactions. However, it is difficult to study numerically complex free-surface evolutions and irregular boundaries. The challenge is even higher if the structure is in motion (Shen and chan 2008).

Lin and Li 2002 made use of a σ-coordinate transformation to map a irregular physical domain to a computational domain of rectangular shape. Lin 2006 developed a three-dimensional (3D) multiple-layer σ-coordinate model to simulate surface wave interaction with various types of structures, including submerged, immersed, and floating structures. In general, such methodologies are capable of producing accurate predictions of free-surface displacements. A limitation, however, is the modeling of free surface of arbitrary configuration, such as in the case of wave breaking. Compared to the application of the coordinate transformation technique, it is still more convenient to carry out numerical simulations of wave-structure interactions in a cartesian coordinate system. Due to the complexity in modeling a general irregular solid boundary, many studies were limited to simpler cases such as wave flows over a submerged rectangular obstacle (Chang et al. 2001, Huang et al. 2001, Tang et al. 1998).

Hur and Mizutani 2003 used a VOF-based model to simulate the interaction of waves and a permeable submerged breakwater and to estimate the wave force acting on it. Hus et al. 2002 included the volume averaged equations for porous flows derived by Van Gent 1995 into the VOF-based model proposed by Lin and Liu 1998, to study wave motions and turbulence flows in front of a composite breakwater. Comparisons of the numerical results and laboratory data showed a good agreement. Shen et al. 2004 used a VOF version of the SOLA-VOF code with a two-equation k-ε model to simulate the propagation of non-breaking waves over a submerged bar. Their simulated results showed a reasonable agreement with experimental data by Ohyama et al. 1995.

Other numerical studies based on the VOF-based two-phase flow model for the simulation of water wave motions have been reported. Hieu and Tanimoto 2002 developed a VOF-based two-phase flow model to study wave transmission over a submerged obstacle. Karim et al. 2003 developed a VOF-based two-phase flow model for wave-interactions with porous structures and studied the hydraulic performance of a rectangle porous structure against non-breaking waves. Their numerical results surely showed good agreements with experimental data. Hieu et al. 2004 simulated breaking waves in a surf zone using a VOF-based two-phase flow model. Their numerical results were compared with experimental data provided by Ting and Kirby 1994 for the spilling breaker on a sloping bottom. Their results agreed well with the experimental data. However, the wave motion in porous media and the non-reflective wave source method were not included in the model by Hieu et al. 2004.

The objective of the present study is to develop a volume of fluid (VOF) based two-phase flow model and to discuss the applicability of this model to the simulation of wave-structure interactions. The numerical VOF based two-phase flow model has been developed and applied to the simulations of wave interactions with a submerged obstacle. Numerical results are then exploited to verify the applicability of the numerical model to the simulations of complex interactions of waves and permeable vortex structures, including the effects of wave breaking. It is concluded that the two-phase flow model with the aid of the advanced VOF technique can provide with acceptably numerical results.

2 Mathematical Formulation

2.1 Assumptions

We considered a two-phase free surface flows in the turbulent regime, the surface tension effect are also taken into account. The considered two fluids are air and water, which have been assumed Newtonian and immiscible.

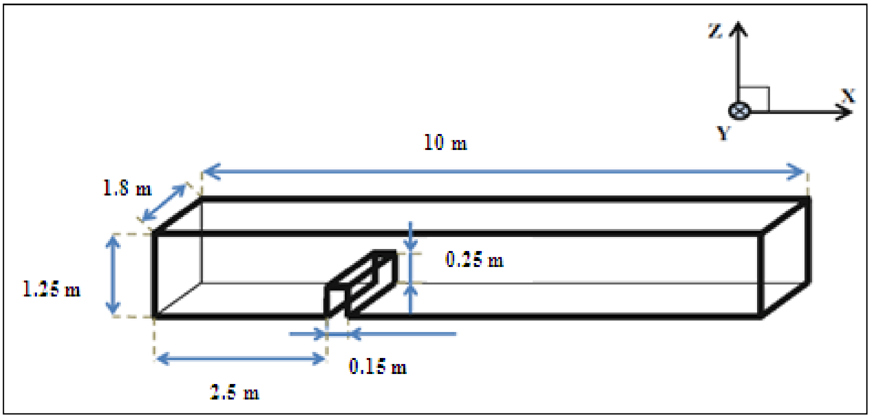

Fig. 1 Computational domain geometry

We select a portion of an urban sewerage network of Monastir in order to show the real condition of a water flow interaction. The domain was a rectangular 10 m long channel with a width of 1.8 m and maximum depth of 1.25 m. The water level was 0.35 m (Fig. 1). The rectangular obstacle has been placed at distance $x_{obst} = 2.5$ m from the channel inlet with a height of 0.25 m and a thickness of 0.15 m.

2.2 Governing Equations

The Reynolds-averaged Navier-Stokes equations are computationally solved, for an unsteady and incompressible turbulent fluid. The mass (1) and the momentum (2) conservation equations are presented in the Cartesian coordinates system as follow:

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial U_i}{\partial t} + \frac{\partial}{\partial x_k}(U_k U_i) = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_k}\left[\nu\left(\frac{\partial U_i}{\partial x_k} + \frac{\partial U_k}{\partial x_i}\right) - \overline{u_i' u_k'}\right] + g_i \tag{2}$$

Where $\overline{u_i' u_k'}$ is the Reynolds stresses and i, k = 1, 2, 3 refer to x, y and z respectively. U_i is the mean velocity and x_i is the coordinate in the i direction.

The turbulence model that was used for the calculations is the k-ε model. It is considered one of the most efficient model, and this because it is essentially a model that assumes the existence of non-isotropic turbulence.

The multiphase model used in this work is the Volume Of Fluid model (VOF). It allows locating the interface between two different media fields. It is applicable to immiscible fluids (water, air) when there is no interpenetration between them.

2.3 Boundary Conditions

A water velocity of 5m/s and a superficial air velocity of 1m/s were chosen for the CFD calculations. The model inlet was divided into two parts: in the lower 50% of the inlet cross section, water was injected and in the upper 50% air (Fig. 2).

Both phases have been treated as isothermal and incompressible at 25°C and at a reference pressure of 1 bar. A hydrostatic pressure was assumed for the liquid phase.

At the inlet, the turbulence properties were equivalent to a turbulence intensity of 5% (Chang et al. 2001) in both phases. The inner surface of the channel walls has been defined as hydraulically smooth with a non-slip boundary condition applied to both gaseous and liquid phases. The channel outlet was modeled with a pressure outlet boundary condition.

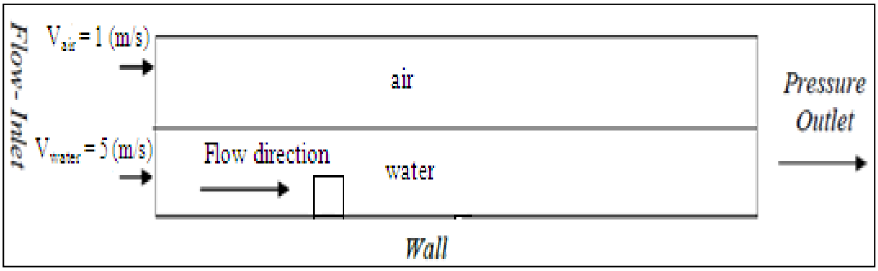

Fig. 2 Domain geometry with boundary conditions

3 Numerical Method

Numerical computations were carried out using ANSYS Fluent which is based on the finite volume approach.

The discretized equations, along with the initial and boundary conditions, were solved using the segregated solution method. Using the segregated solver, the governing equations were solved sequentially.

In order to improve accuracy, the second order upwind scheme was used. The SIMPLE method was used to calculate the pressure–velocity coupling. It uses a relationship between velocity and pressure corrections to enforce mass conservation and obtain the pressure field. The maximum residual of all variables was 10^{-3} in the converged solution.

The mesh was periodically refined and the computation was repeated until the variation in results was adequate. It was found, that the optimal number of grid points is approximately 150 000.

4 Results and Discussion

4.1 Dynamic Study of the Flow Structure around an Obstacle

To see how the wave height varies as a solitary wave passes over an obstacle, Figure 3 shows the maximum value of the water elevation at different times in the channel. In this figure, the rectangle indicates the location of the obstacle. The rise of the wave height near the leading edge of the obstacle can be explained as due to the shallow effect. Consequently, the wave height decreases slightly as part of the wave is positively reflected due to the energy dissipation caused by the wave and structure interaction and reaches a minimum value at the trailing edge. Afterwards, the wave height increases a little, as the negatively reflected wave is generated. The positively and negatively reflected waves can be seen in this figure with a longer distance between them. This result is proved by Huang et al. 2003.

Fig. 3 also shows that as the waves propagate over the obstacle, higher harmonics are generated due to the nonlinear effect. Hence, they are gradually detached from the main crest, as it can be seen. As the waves propagate into the deep water region, the wave non-linearity becomes so weak that the bound waves are

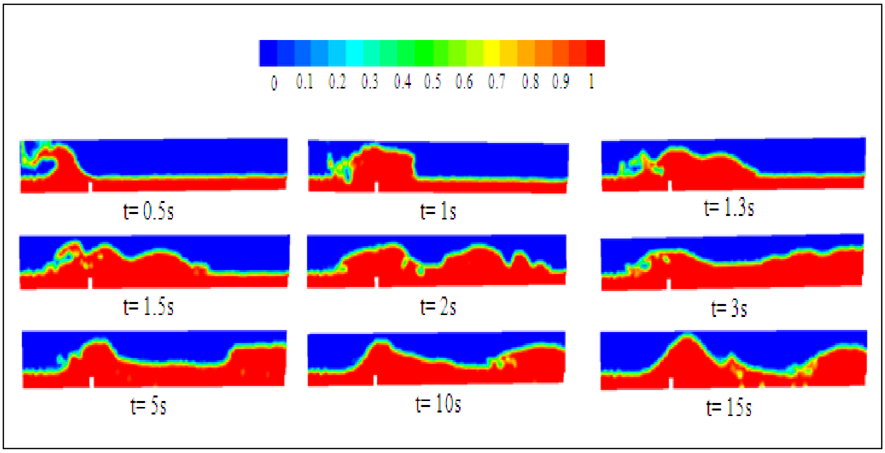

Fig. 3 Water volume fraction contours in presence of a rectangular obstacle at different times in the channel

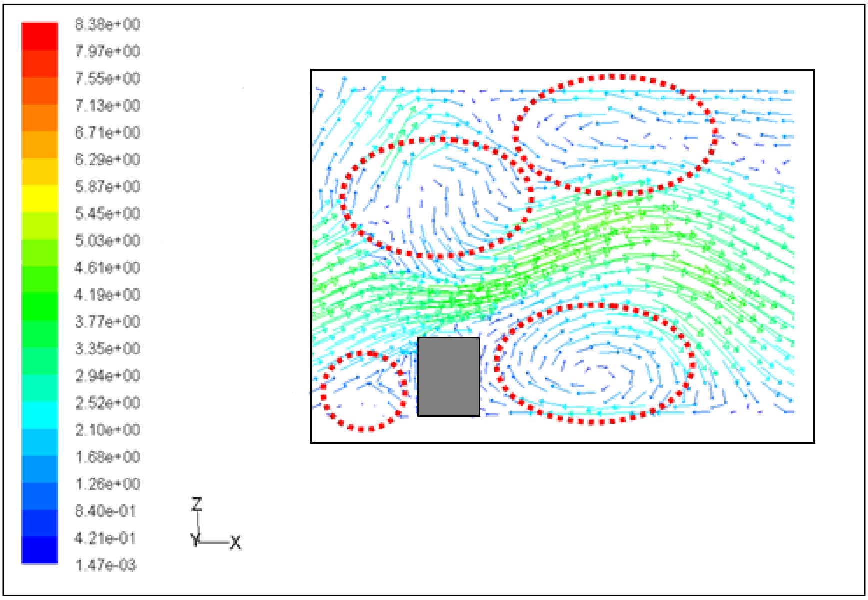

Fig. 4 Velocity vectors at t = 15s

substantially reduced and a large amount of energy is transformed from the bound waves into the free waves (Huang and Dong 2001).

By analyzing the flow through the Fig. 4, we can identify four vortex structure, the first one is upstream the obstacle characterized by low intensity, a second one is just above the obstacle with a small size but without reattachment as it was confirmed by Bouteraa et al. 2002. The mixing layer formed downstream of the obstacle, shows the existence of two counter-rotating vortices type Kelvin-Helmotz, one small size trapped between the wall and the ground and other of larger size.

In the same figure which shows the velocity vectors, we observe that the detachment of the shear layer causing the appearance of a vortex. It is interesting to note the tendency to the organization into elongated structures. The wake structure downstream of the obstacle reveals a driveway swirling consisting of counter-rotating structures, in accordance with existing experimental and numerical results of Bouterra et al. 2002. These lines circumvent the obstacle and espouse its geometry by restricting three characteristic zones of the flow: stagnation, separation and recirculation.

The stagnation zone upstream of the obstacle is low intensity. The separation zone above the obstacle with a small thickness is dominated by strong shear present without gluing as was observed by Zhang 1991. The mixture layer formed downstream of the obstacle, is characterized by two counter rotating vortex structure. The first, small size is trapped between the wall and soil. The second structure, of elongated shape, and is larger behind the wall. These results confirm the experimental work of Eaton and Johnston 1980.

4.2 Flow Disturbed by the Presence of Two Obstacles

In the case of the presence of two obstacles, Fig. 5 shows the existence of four regimes based on the size of the obstacles and the distance between them. When the

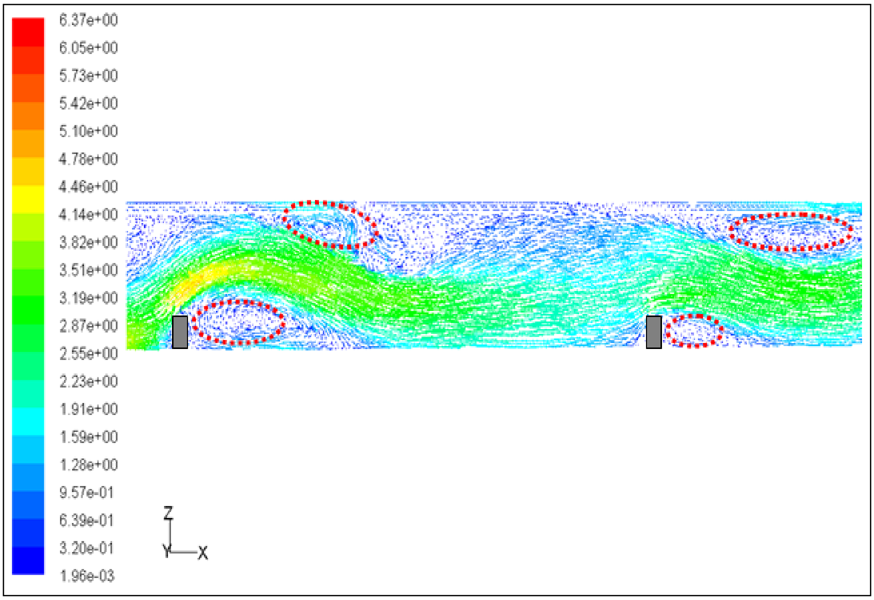

Fig. 5 Structure of the flow behind two obstacles at t= 15s

obstacles are well spaced from each other, the flow is like a succession of identical structures around obstacles isolated and the wakes are disturbed. The flow downstream of an obstacle is reinforced by a deflection at the upstream side of the obstacle adjacent, so the flow will close and a recirculation zone is established at spacing identical to the other. This result is confirmed by Abderrahmane 2012.

5 Conclusion

In this work, we study the dynamic of the flow field around an obstacle. We note that the formation of coherent structures provides information about the pace of the flow and specify its principal features (shear layer, recirculation and reattachment). The aim of this study is to understand the fundamental mechanisms which govern the flow.

Note also from this paper, due to the nonlinear effect, the wave height above the obstacle increases continually which may eventually result in wave breaking. The presence of these instabilities provides a new perspective in the description of turbulent flows.

References

Abderrahmane, N.: Etude des écoulements à surface libre en présence des obstacles dans le fond. PhD Thesis, University of Abou Bekr Belkaid Tlemcen (2012)

Bouterra, M., El Cafsi, A., Laatar, A.H., Belghith, A., Le Quéré, P.: Numerical simulation of a stratified two-dimensional turbulent flow around an obstacle. Int. J. Therm. Sci. 41, 281–293 (2002)

Casulli, V., Cheng, R.T.: Semi-implicit finite difference methods for three-dimensional shallow water flow. Int. J. Numer. Methods Fluids 15, 629–648 (1992)

Chang, K.A., Hsu, T.J.: Vortex generation and evolution in water waves propagating over a submerged rectangular obstacle: Part I. Solitary waves. Coast. Eng. 44, 13–36 (2001)

Eaton, J.K., Johnston, J.P.: Turbulent flow reattachment: an experimental study of the flow and structure behind a backward-facing step. Stanford University Rep. 39 (1980)

Huang, C.J., Dong, C.M.: On the interaction of a solitary wave and a submerged dike. Coastal Engineering 43, 265–286 (2001)

Hieu, P.D., Tanimoto, K.: A two-phase flow model for simulation of wave transformation in shallow water. In: Proceedings of the Fourth International Summer Symposium, Kyoto, JSCE, pp. 179–182 (2002)

Hus, T.J., Sakakiyama, T., Liu, P.L.F.: A numerical model for wave motions and turbulence flows in front of a composite breakwater. Coastal Engineering 46, 25–50 (2002)

Huang, C.-J., Chang, H.-H., Hwung, H.-H.: Structural permeability effects on the interaction of a solitary wave and a submerged breakwater. Coastal Engineering 49, 1–24 (2003)

Hur, D.S., Mizutani, N.: Numerical estimation of the wave forces acting on a three-dimensional body on submerged breakwater. Coastal Engineering 47, 329–345 (2003)

Hieu, P.D., Tanimoto, K., Ca, V.T.: Numerical simulation of breaking waves using a two-phase flow model. Applied Mathematical Modeling 28, 983–1005 (2004)

Karim, M.F., Tanimoto, K., Hieu, P.D.: Simulation of wave transformation in vertical permeable structure. In: Proceedings of the 13th International Offshore and Polar Engineering Conference, Hawaii, USA, pp. 727–733 (2003)

Lin, P., Liu, P.L.-F.: A numerical study of breaking waves in the surf zone. Journal of Fluid Mechanics 359, 239–264 (1998)

Lin, P., Li, C.W.: A σ-coordinate three-dimensional numerical model for surface wave propagation. International Journal for Numerical Methods in Fluids 38, 1045–1068 (2002)

Lin, P.: A multiple-layer σ-coordinate model for simulation of wave-structure interaction. Computers and Fluids 35, 147–167 (2006)

Lin, P.Z.: A fixed-grid model for simulation of a moving body in free surface flows. Comput. Fluids 36, 549–561 (2007)

Liu, X., Xu, H., Shao, S., Lin, P.: An improved incompressible SPH model for simulation of wave–structure interaction. Computers & Fluids 71, 113–123 (2013)

Maruzewski, P., Le Touze, D., Oger, G., Avellan, F.: SPH high-performance computing simulations of rigid solids impacting the free-surface of water. J. Hydraul. Res. 47, 126–134 (2009)

Ohyama, T., Kioka, W., Tada, A.: Applicability of numerical models to nonlinear dispersive waves. Coastal Engineering 24, 297–313 (1995)

Shen, Y.M., Ng, C.O., Zheng, Y.H.: Simulation of wave propagation over a submerged bar using the VOF method with a two-equation k–ε turbulence modeling. Ocean Engineering 31, 87–95 (2004)

Shen, L., Chan, E.-S.: Numerical simulation of fluid–structure interaction using a combined volume of fluid and immersed boundary method. Ocean Engineering 35, 939–952 (2008)

Shao, S.D.: Incompressible SPH simulation of water entry of a free-falling object. Int. J. Numer. Methods Fluids 59, 91–115 (2009)

Ting, F.C.K., Kirby, J.T.: Observation of undertow and turbulence in a laboratory surf zone. Coastal Engineering 24, 51–80 (1994)

Tang, C.J., Chang, J.H.: Flow separation during solitary wave passing over submerged obstacle. Journal of Hydraulic Engineering 124, 742–749 (1998)

Van Gent, M.R.A.: Wave interaction with permeable coastal structures. PhD Thesis, Technical University Delft (1995)

Zhang, C.X.: Simulation numérique d'écoulements turbulents autour d'un obstacle. PhD Thesis, University of Nante (1991)