Mechanical Testing and Diagnosis
ISSN 2247 – 9635, 2013 (III), Volume 1, pp. 32-37

WAVE RUN-UP SIMULATIONS AND COMPARISON WITH EXPERIMENTAL DATA ON A SEMI-SUBMERSIBLE

Sorin DIACONU

University "Dunarea de Jos" of Galati, Mechanical Engineering,
Applied Mechanics Department, ROMANIA
sorin.diaconu@ugal.ro

ABSTRACT

Use of CFD tools for industrial offshore applications is a great interest nowadays, therefore is the need for validation of such tools against experimental results. This paper presents wave run up estimations with a fixed large volume semi-submersible performed with the CFD code ComFlow. The predicted results of numerical simulations are compared with experimental data at two locations: on the first and the second column of the semi-submersible. These codes are used to simulate the flow around a semi-submersible offshore structure due to an incoming regular wave under the influence of high steepness regular waves. Wave basin tests were performed with small scale model of a large volume semi-submersible. These tests evidenced some issues of the run-up effects along a semi-submersible body and it will be shown that the simulation results are quite similar with the experimental data.

Keywords: experimental data, offshore, semi-submersible, run-up

1. INTRODUCTION

During lifetime, an offshore structure encounters numerous loading conditions. The conditions of such structures dominate the design of the structure regarding fatigue, wave and structure motion in the long term. In heavy storms, these motions can become so large that solid amounts of (green-colored) seawater flow over the deck and inadequate assessment may result in an unnecessary elevation with serious consequences from an economic standpoint.So, wave run-up and overtopping, but also the wave-induced flow patterns near, intake and outlet structures are only a few important phenomena that occur in the marine environment. Insight on these issues plays a specialconcern in studies on safety, water quality, maintenance and optimization of design.

Nowadays, is still a challenge for the designers predict wave elevation. Theinherent complexity correlated with the non-linear hydrodynamic effects of the wave run up on the body of the structure, need a characterization of the steepest waves in more sea conditions.

Model scale tests are performed at the final stages of the design to verify the seakeeping behavior of the system. Moreover, the experimental approach will not be practical at the early stages and therefore, the use of numerical methods that can estimate the value of parameters from the vicinity of the structure becomes attractive.

The evolution of the free water surface is described by an adapted and (highly) improved version of the Volume-of-Fluid method (VOF) designed originally by Hirt and Nichols [5].

The mathematical model for complex water flow dates from the first half of the 19th century already and is known as the Navier–Stokes equations, but, it is only for about a decade that these field equations can be solved for large-scale complex free-surface flow problems.

The numerical tool used in this paper is a hydrodynamic flow model based on the Navier–Stokes equations to simulate the steep waves near and around fixed and floating structures, offshore platforms, coastal breakwaters and it can be used in an early stage of the design process.

2. THE COMFLOW MODEL

The numerical method COMFLOW is based on the Navier- Stokes equations, in which a VOF method is applied to describe the evolution of the free surface. A local height function near the free surface yields improved performance in terms of mass conservation and the number of disconnected droplets, compared to the original VOF method.

COMFLOW simulate extreme wave impact events, as if the structure were out at sea. For reasons of efficiency, the domain should ideally be confined to the direct surroundings of the structure. An accurate representation of the wave system near the structure the wave reflections from the boundaries of the computational domain need to be prevented. Especially in 3D representation and in case of long waves, the incorporation of absorbing layers or numerical damping zones and the requirement of multiple wavelengths to effectively dissipate wave energy and prevent reflections is very important in order to keep the domain size only slightly larger than the structure itself. So, it will be used an absorbing boundary condition, which has been developed for a better evaluation of extreme wave impact simulations with COMFLOW.

The main topic of this paper is thepresentation of ComFLOW numerical simulations results of the wave run-up along the two legs of a semi-submersible platform. The CFD results are compared with data obtained from model experiments of the platform, for regular incoming waves.

3. GOVERNING EQUATIONS

This paper uses the single phase ComFLOW model, and the fluid is considered incompressible and viscous (like the sea water), while void fills the remaining space of the computational domain.

The fluid flow can be found by solving the continuity equation, together with equations which are describing the conservation of momentum:

$$\nabla \cdot u = 0 \tag{1}$$

u representing the fluid’s velocity vector.

In VOF-like methods a function F(x,y, z, t) is introduced, with values between zero and one, to indicate the fractional volumeof a computational cell which is occupied by fluid. Evolution of the VOF function is given by Equation (2):

$$\frac{\partial F}{\partial t} + (u \cdot \nabla) F = 0 \tag{2}$$

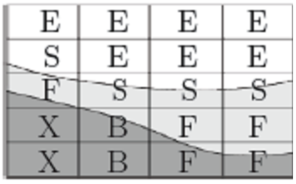

Fig. 1. Two dimensional grid-cell labeling for an arbitrary geometry and liquid configuration

Every grid cell is given a label to distinguish between fluid, air and boundary. These labels depend on the variables F_b and F_s, resulting that the fraction of a grid cell is open forthe fluidd and the fraction of a cell filled with the liquid phase. The interior cells containing no fluid, when $F_b > 0$ and $F_s = 0$, are labeled as E(empty) cells, which are truly empty and are left out of the computations. At the same time, non-empty cells ($F_s > 0$) adjacent to E cells are labeled as S (surface) cells, as they represent the part of the free surface. The remaining non-empty cells are labeled as F (full) cells and cells which are satisfying $F_s = F_b = 0$ are called B (boundary) cells when they are adjacent to an interior cell,otherwisee they are labeled as X (exterior). In Fig. 1, an example of a label configuration is shown.

4. THE SEMI-SUBMERSIBLE ESPERIMENTS

The semi-submersible model experiments were performed by MARIN of The Netherlands and in this paper are presented only some results of these experiments.

The main dimensions of the semi-submersible platform at full scale are presented in Fig. 2. Regular waves, with the height of 8m and a period of 9.0 seconds have been chosen for numerical analysis with ComFLOW and presented the results of wave elevation in two wave points (WP1 and WP2).

The 1:5 model of the semi-submersible consist of two columns and a pontoon under water (so actually one halve of a semi-submersible)and the experimental basin has dimensions of 200/4/3.6 m. The high speed basin is equipped with a flap-type wave generator at one end and a beach at the other end.

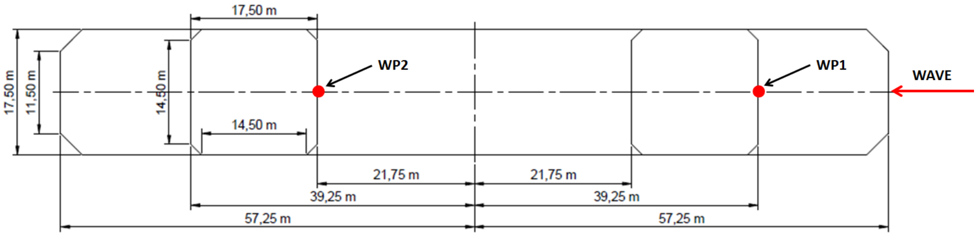

Fig. 2. Main dimensions of the semi-submersible (full scale) and the location of the wave-probes (WP1, WP2)

The center of the test set-up was placed at a considerable distance(100 meters prototype value) from the wave flap. During wave calibration and during the tests with the semi-submersible, the motions of the wave flap were measured. This enables synchronization of the undisturbed wave measurements with the wave run-up measurements.

Except where explicitly stated otherwise, all test results presented in this report are given with respect to the "zero values". Prior to the start of the tests, all signals are set to zero and during the test, the difference with respect to the zero value is measured. This means for example that, when a transducer is located below the calm water surface, the data is not included in the measurement signal.The measurement continued until it was clear that reflections from the outer ends of the basin started to arrive at the test set-up and the ratio ρ = 1.025 between the specific weight of salt water and the fresh water in the basin means which was included indicates that all test results apply to seawater.

During the experiment, surface elevations were measured at the two wave points.

5. WAVE ELEVATION SUMULATIONS

The fluid flow domain included only the platform's neighborhoodof a reasonable size. The platform model was placed at the origin of the coordinate system. Such domain is symmetrical in the longitudinal direction, with the same amount of space upstream and downstream of the platform.

The simulation starts with the initial fluid configuration of a fully developed wave field. The Fig.3 show the fluid configuration at perspective view (Fig. 3a) andthe side view (Fig. 3b).

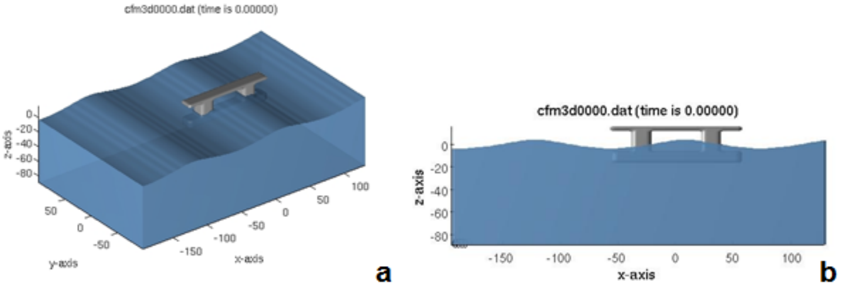

Fig. 3. Semi-submersible modeled in ComFlow and initial condition for simulation: perspective view (a) and side view (b)

Computational domain was symmetrical with respect to the platform's symmetry plane and the domain depth was set to 60 m (that is, 1/3 of the experimental basin depth). The reason was to limit the number of the computational cells. The representative waves were modeled as Stokes 5th order waves and the fluid flow was simulated for 3-4 wave periods. Parameters of the representative incoming waves derived from numerical analysis are a wave height of 7.6m and a period of 9.0 s. For a better perspective view in Fig. 4 shows two time-synchronized video frames composed from the experiment and the calculation.

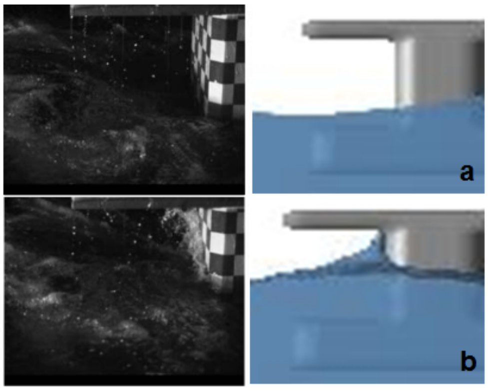

Fig. 4. Synchronized video frames with experiment vs. simulation: wave front is running onto the forward pontoon (a) and wave front runs up against the forward column (b)

The comparison between experimental data and numerical simulations registered at the first column (WP1) and second column (WP2), respectively, are presented in Figs 5 and 6. It can be seen that the experimental signal and the ComFLOW result agree quite well at the incoming wave fluid height probe location. ComFLOW reproduces the incoming wave crest height and also at the wave troughs a very good agreement at the location of the two columns.

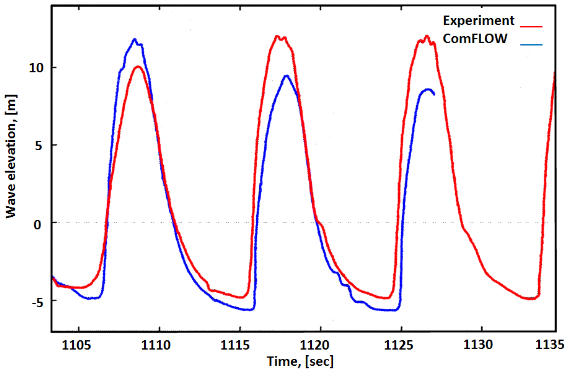

Fig. 5. Relative wave elevation at the first column. Comparation between experiment and ComFlow simulation results

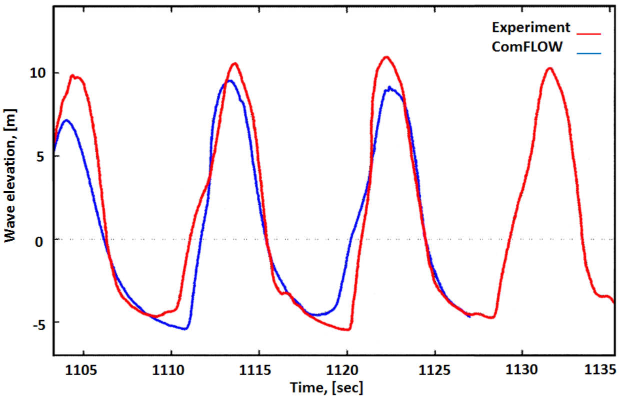

Fig. 6. Relative wave elevation at the second column. Comparation between experiment and ComFlow simulation results

5. CONCLUSIONS

The ComFLOW program can be applied for numerical simulation of wave run-up on a marine structure. The proposed method include high-order potential solutions, particle image velocimetry experiments and CFD calculations with Volume of Fluid (VOF) approaches used to track movement of the fluid's free surface.

The absorbing boundary conditions have been applied in 3D numerical simulations of wave impacts on a semi-submersible model and special attention have been given on local fluid flow details in the immediate vicinity of the two columns of the platform.

Regarding the comparison between experimental data and numerical simulations of surface elevation, results that the numerical method ComFLOW is suitable for the simulation of two-phase flow in wave impact events and the free surface can assume any shape. The overall agreement between simulations and experiments is good.

ACKNOWLEDGMENT

The work of the first author has been made in the scope of the project SOP HRD - EFICIENT 61445/2009 (Management System for the Fellowships Granted to the PhD Students).

REFERENCES

[1] **Buchman B., Ferrant, P., Skourup J.,** 2000, Runup on a Body in Waves and Current. Fully Nonlinear and Finite-order Solutions, *Applied Ocean Research*, 22, pp. 349–360.

[2] **Kristiansen T., Baarholm, R., Rørtveit, G.J., Hansen, E.W., Stansberg, C.T.,** 2005. Kinematics in a Diffracted Wave Field: Particle Image Velocimetry (PIV) and Numerical Models, Proceedings, *24-th Intern. OMAE Conference*, Halkidiki, Greece. Paper OMAE2005-67176.

[3] **Loots E., Buchner B.,** 2004, Wave Run Up As Important Hydrodynamic Issue For Gravity Based Structures, *Proceedings of the 23-rd International OMAE Conference*, Vancouver, Canada. Paper OMAE2004-51084.

[4] **Danmeier D.G., Seah R.K.M., Finnigan T., Abault A., Vache, M., Imamura J.T.,** 2008, “Validation of Wave Run-up Calculation Methods for a Gravity Based Structure”. In Proceedings, 27-th International OMAE Conference, Estoril, Portugal. Paper OMAE2008-57625.

[5] **Hirt C.W., Nichols B.D.,** 1981, Volume Of Fluid (VOF) Method for the Dynamics of Free Boundaries. Journal of Computational Physics, 39, pp. 201–225.

[6] Iwanowski, B., 2008, ComFLOW-2 JIP: Benchmarks of the ComFLOWProgram. Technical Report TR-2008-0049, FORCE Technology Norway AS, Sandvika, Norway, October.

[7] **Veldman A.E.P., Gerrits J., Luppes R., Helder J.A., Vreeburg J.P.B.,** 2007, The Numerical Simulation of Liquid Sloshing On Board Spacecraft, *J. Comp. Phys.,* 224, pp. 82-99.

[8] **Kleefsman K.M.T., Fekken G., Veldman A.E.P., Iwanowski B., Buchner B.,** 2005, A Volume-Of-Fluid Based Simulation Method for Wave Impact Problems, *J. Comp. Phys.,* 206(1), pp. 363-393.

[9] **Kleefsman K.M.T., Fekken G., Veldman A.E.P., Buchner B., Bunnik T., Iwanowski B.,** 2002, Prediction of green water and wave loading using a Navier-Stokes based simulation tool, *the 21st Int. Conf.*, OMAE2002-28480.

[10] **Kleefsman K.M.T., Veldman A.E.P., Bunnik T.,** 2004, An improved Volume-Of-Fluid (IVOF) method for wave impact type problems, Spec. *Symp. on FPSO Integrity,* OMAE-FPSO04-0066.