Materials Today: Proceedings 49 (2022) 2907–2911

Contents lists available at ScienceDirect

Materials Today: Proceedings

journal homepage: www.elsevier.com/locate/matpr

Using of flow 3d as CFD materials approach in waves generation

Hasan Ibrahim Al Shaikhli *, Saleh Issa Khassaf

Civil Engineering Department, College of Engineering, University of Basrah, Basrah, Iraq

ARTICLE INFO

Article history:
Available online 3 November 2021

Keywords:
Breaking waves
Wave generation
Flow 3D
Numerical analysis
CFD

ABSTRACT

Waves action considering one of the most hydraulic phenomenon that covered by the history, the waves action hits the stones or any concert blocks that formed breakwater then the development of breakwaters became necessary so an equilibrium shape by filling the voids with more stones occur. Because of the complex of experimental work and the higher cost of models, the numerical simulations using CFD approach should be studied to show the ability of Flow 3D software to simulate the wave behavior phenomenon. The results present an improvement that the Flow 3D software able to generate the waves with different heights and periods.

Selection and peer-review under responsibility of the scientific committee of the Indo-UK International Virtual Conference on Advanced Nanomaterials for Energy and Environmental Applications (ICANEE-2020).

1. Introduction

The greater part of existing literature on applying CFD to breakwater modeling came from studies using different simulation software. Results from studies employing FLOW 3D to simulate various hydraulic structures that informed a reasonable concord with the experimental results and the standard design guide lines. However, many types of software packages used to simulate the flow over breakwater and studied its hydraulic behavior, these consist of an information on the successful uses of CFD, Fluent software and comparisons of both CFD and Fluent. Based on the earlier researches that deal with hydraulic problems, FLOW 3D and Fluent considered to be applied effectively for advanced problems involving fluid–solid interaction with taking into account the significant results that could be supplied. [3], induced the breaking of single waves over breakwaters in the laboratory and nonlinear potential model. Based on the incident wave height, waves may collapse above the crest or break backward or forward over submerged breakwaters. Wave transmission and reflection coefficients established in experiments test. Transmission coefficient observed increasing to reach (55–90) percent over submerged breakwaters. For the submerged breakwaters, nonlinear potential model computations show agreement with laboratory data, especially for smaller waves heights. This model computation properly estimates the limit of collapse overtopping submerged breakwaters. [11], used 2D Navier-Stokes equation for simulating plunging breakers, this numerical model discussed SKYLLA approach which created to simulate breaking waves on coastal structures. The Volume of Fluid technique used to solve two-dimensional Navier-stokes equations. In computational domain, water waves enter and leave the domain due to weakly reflecting boundary, So, impermeable barriers might be used. The results show that the two-dimensional Navier-stokes equations has simulated waves over a low crested structure when compared to those of physical model testing for waves on a submerged structure with a 1:20 slope

Hayakawa, et al., [6], used the SOLA-SURF method to simulate wave fields around a submerged breakwater numerically in two and three dimensions. The results of numerical models compared with experimental results and the non-viscous side wall boundary condition, slip wall type, for the three-dimensional computation of the laboratory tank should be employed. The numerical approach shows a practical implement of field situation. [9], asserted that numerical simulation of breaking and post-breaking wave deformation process around a submerged breakwater for a two-dimensional wave field in the vertical plane, the model combines a non-reflective wave generator with VOF approach and dissipation zone with open boundary. The proposed model examined post-breaking wave deformation in additional to wave breaking over submerged breakwater. Laboratory tests were carried out to confirm the model's validity. Various incident wave height and fre-

* Corresponding author.
E-mail addresses: e.hassanibraham@gmail.com (H. Ibrahim Al Shaikhli), Saleh.Khassaf@uobasrah.edu.iq (S. Issa Khassaf).

https://doi.org/10.1016/j.matpr.2021.10.282
2214-7853/
Selection and peer-review under responsibility of the scientific committee of the Indo-UK International Virtual Conference on Advanced Nanomaterials for Energy and Environmental Applications (ICANEE-2020).

quency of wave amplitude are investigated. The calculated and observed results show that a breaking wave-induced circulating flow forms on the onshore side of the submerged breakwater. Through comparison with experimental data, it is shown that the proposed numerical wave model accurately simulates wave deformation before and after wave breaking. Hwang, et al., 2004, selected RANS model in order to simulate the impermeable submerged double breakwaters and to understand the behavior of waves progressive, vortex formation and the produced dissipation. The k–e model and height function characterized as turbulence dynamics model and free surface, respectively. the governing equations discretized by limited-volume technique that revolves around a grid system witch varying in height and width. A chain of suggested numerical answers evaluated and developed analytically, then the numerical model verified by some of experimental data. The results of numerical models show an acceptable agreements and can be considered as an economical tool for simulating flow wave fields nearly coastal structures. [7], Simulate the nonlinear dynamic waves on submerged breakwater under different wave conditions, the researchers used numerical simulation for nonlinear interaction between the ocean waves, seabed, and submerged breakwater. The flow condition in the numerical simulation of seabed is laminar flow to control the resistance of fluid through the porous medium. The flow results of wave–seabed – breakwater interaction subjected to the change of incident wave situations and the change in pore water pressure tested for cross-section of the submerged breakwater. The numerical results show that the breakwater protected against the flow field and the fluctuation. Hajivalie and Bakhtiary, [4], study the effect of turbulence based on the (RANS) equations using 2D numerical model to investigate the impact of steepness breakwater on the waves standing and the steady streaming. Numerical model employed called k-turbulence model and the shape of the free surface waves monitored using the Volume of Fluid (VOF) approach. The numerical results compared with experimental results for standing waves of breakwater. Three simulations created for different three breakwaters, that is in addition to the wall on vertical axis such 1:2 and 2:1 sloped breakwater. The moderate slope causes steady flowing that is recirculates the cell over which might be controlling the scouring at toe of a breakwater. [8], stated that the conclusions of permeable submerged breakwater investigations on the impacts of the slope gradient. The flow through a porous material with inertial, linear, and nonlinear resistance components is considered using a newly constructed numerical model. As a result, for a submerged breakwater, the numerical model can simulate the interaction between the wave and seabed. The LES turbulence model calculated the eddy viscosity in a 2-dimensional wave field. Comparison between model and experimental data, the generated to existing for wave deformations induced by the rectangular permeable submerged breakwater, the model shown to be accurate. The transmission coefficient decrease and the wave breaking of the submerged breakwater migrate towards sea when the slope gradient decreases. Also, a clockwise circulation flow created behind the submerged breakwater. [13], investigate the erosion and wave transmission for submerged breakwater using numerical simulation. For solving these issues, breakwaters have been built to minimize coastal erosion, on the other hand, breakwaters developed, in order to reduce both mean water level and wave transmission. The numerical calculation, in terms of spectrum analysis, sea water level and wave heights, in two dimensions was performed. Submerged banks with numerous vertical impermeable plates calculated numerically and the obtained findings were compared with actual data. The proposed submerged bank showed reducing in short and long wave transmission while decreasing in water level behind submerged breakwater. [5], used numerical modeling to investigate the influence of vertical breakwater size on the hydrodynamics of waves in addition to the vortex formation surrounding the breakwater. Two dimensionless parameters is developed such as Keulegan–Carpenter and the breakwater submergence. The computational model included k–turbulence model and the (RANS) equations, the free surface traced by VOF technique. Ten different models conducted for this study, each model has different values submerged breakwater depths. The transmission coefficient increases as the a/Hi increase. The transmission coefficient for the waves examined increases rapidly. This design proposed for breakwater width, so the increases in a/Hi caused in decreases of turbulence intensity on the breakwater seaside. The optimum a/Hi for both high energy dissipation rate and minimal scour depth. Lianga, [10], study numerically the transmission wave over double trapezoidal submerged breakwaters using non-hydrostatic wave, the model stated that a non-hydrostatic wave model called SWASH (Simulating Waves until Shore) is utilized to simulate wave transmission. The numerical outcomes compared to the physical model's results, so, the SWASH model capable of predicting wave transmission through double submerged breakwaters and the influence of sea current on wave transmission investigated. The optimum relative breakwater spacing is around 1.11 when the relative submerged depth remains at 1.0. Super harmonic wave shows more visibility than lower harmonic wave in component dissipation. [12], search numerically the effect of a submerged breakwater for climate change adaptation, as known, mean sea level rise when the climate warming cause more frequent storm occurrences in coastal regions. The breakwaters will be exposed to many times of overtopping waves with more incident waves height than the design state. As a result, these effects may influence on the operation of breakwaters structures. Researchers used a numerical model to determine the optimum dimensions of a submerged breakwater that will be operated as breakwater structure in coastal regions. The relative in submergence and width parameters investigated with variation in transmission coefficient. The results show that in case of Kiberg, Norway, submerged breakwater destroyed when placed in front of an existing rubble mound barrier. The wave prediction model CGWAVE used as local-scale of finite element method, this model gave the optimum shape produced to protect coastal regions. [2], investigate the hydrodynamic performance of half pipes submerged breakwater in Egyptian coastlines using FLOW 3D software as computer model for regular waves. Three different diameters of a precast in half pipe shape, two half pipes are in horizontal positions and one in vertical position, used to determine the relative structure height that gives maximum energy dissipation. The models are numerically analyzed to calculate the wave energy dissipation, transmission coefficient and reflection coefficient, the numerical results show an ability of predicting the hydrodynamic performance of half pipe submerged breakwaters. [1], investigate numerical model of hydrodynamic performance of double submerged breakwaters, this model used precast concrete half pipes submerged breakwater as nature coastal protection witch prevent beach erosion in Egyptian coastline. The separated half pipes submerged breakwater investigated using a numerical model FLOW 3D software for linear waves as hydrodynamic performance. The optimum space between half pipes submerged breakwaters determined using two models. The transmission coefficient (Kt), the reflection coefficient (Kr), and the wave energy dissipation calculated numerically. The numerical results approved by the experimental and used to estimate the hydrodynamic performance of submerged half pipes.

2. Flow 3D approach

Flow 3D (V.11.2) software for solving cartesian co-ordinates of the Naiver-Stock equations that alternated a grid functions by dividing the flow field into the rectangular-form-groove sub division mesh of the flow in comparatively tiny areas nominated as cells and calculating the numerical flow value. The key requirement of all numerical models is the establishment of the appropriate mesh domain, which reacts properly to phenomenon behavior.

Control volumes around each variable position are created to develop different numerical approximations to the control equations. In relation to the surrounding variable values for each control volume, surface fluxes, surface stresses and body strengths may be calculated. These amounts are then merged in an estimate for the protection legislation indicated by movement equations.

3. Numerical model Set-Up

The overall model set-up was fairly identical to all controlled breaking waves simulations. In every example, a fluid, incompressible flux and free surface or sharp contact were defined on the Global Tab. Furthermore, for all simulations the fluid characteristics were defined as those for water at 20 °C. Several additional parameters remained unchanged in general, and the following parts will be addressed more. The only two selections needed to be activated to produce accurate simulations of the data requested in this study despite the fact that there are many other physical alternatives. With gravity acceleration in the vertical or z-direction reaching negative 9.81 m/sec2 the gravity option was activated. The option of viscosity and turbulence was also activated when Newtonian viscosity and selection of a suitable turbulence model were applied to the flow. Once the FLOW 3D model is fully constructed as long as the two-equation (k-w) model is selected, one turbulence model is applied. The choice of (k-e) turbulent model is the best and best available model for breaking waves simulation in the software, based on comments in FLOW's 3D user's handbook (2007).

For other breaking waves modeled earlier by FLOW 3D, the preparation for the numerical model geometry was considerably different. The geometry utilized in the simulations was given as a stereo lithography (STL) picture produced in Auto CAD and exported in STL-format, according to the information available from the experimental investigation. In case the proper network can then be built. The usual concrete ruggedness value for all flume geometry was assumed to be disregarded and the geometry component remained the standard choice for all models conducted as part of this study.

4. Boundary and initial conditions

The ability to recognize the pressure situation on one or more borders of the computing domain is an essential and valuable computer tool. Pressure limits include compact fluid reservoirs, environment laboratory conditions as well as the mechanical pressures imposed. Typically, there are two sorts of pressures, known as static or stagnant pressures. In a static situation, the pressure across the border is more or less continuous, and a value based on a normal zero derivative state across the border is allocated to the speed on the limit. For several types of flow problems, better outflow boundary conditions exist. For example, specific boundary therapies have been developed for wave spreading issues that aim to detect the speed and direction of waves approaching the boundary and then establish boundary conditions so as to permit a minimum reflection across the border. An early and useful example of this sort of therapy, sometimes referred to as a radiation limitation condition (Orlanski, 1976).

5. Turbulence model boundary conditions

The energy of turbulence and the dissipation functions are addressed in parallel at all borders, save hard no-slip barriers, with other cell centric variables such as density. There are no specific requirements at symmetry limits since there are zero-speed derivatives across the frontier, thus the creation of zero turbulence. There is also a zero flow region, so that no advective or diffusive flows are automatically ensured. Specific considerations are nonetheless essential for stiff no-slip borders, as the numerical resolution is generally too low to fix the features of a laminar border layer area. A turbulent velocity profile has been used for the wall shear-stress model. In order to be consistent, the wall limits for the turbulence energy and the dissipation of turbulence functions must be determined. FAVOR technique in which solid walls can be cut across a mesh cell at any angle, it is uncertain how these limits can be met. So, this technique should have picked up. In all cells with a portion or fully blocked no-slip, stiff border, turbulence energy and dissipation values are defined. From the velocity profile assumed (logarithmic law approximation) and assuming a local balance between turbulent production and decay processes, the Wall boundary values are derived.

6. Free-Surface boundaries

The pressure setting technique presented in the Incompressible SOR method meets normal stress and the required pressure, condition on the free surface. (Only single fluid problems have free-surface boundaries requirements). The free-surface tangent stresses are zero because all speed derivatives involving components of velocity outside the surface are nil. Speeds should be adjusted at every cell line between surface cell and empty cell however the fluid advection should be properly taken into consideration. This may have done in two stages. First, the value on the opposite face of the surface cell is set for every speed component on a side next to the empty cell. In the second stage, the surface cell will be adjusted virtually to try driving the cell's velocity divergence to zero. Only, speeds are changed in this procedure on the sides open for empty cells. The divergence in cell velocity cannot be pushed to nil, as the correction is also proportional to the fractional fluid in the cell. The above limits are applied to ensure the flow consists of internal impediments. The establishment of suitable boundary conditions has a significant influence on whether the results from the numerical model match the actual scenario one attempted to replicate. Data from free fluxes of surface were requested in the mesh block (1) in this case and hence the top border was set as atmospheric pressure while the bottom border was set as wall. In order to capture the channel bed, the lower border was placed directly below the input form of the models geometry, whilst the top edge was set just above the highest water level in the vertical or the z-direction grid. The change of mean water depths and wave heights was made by the determination of the upstream limit as a certain stress with a fluid height, to make the simulation comparable with the experimental work. The simulation also chose to set the downstream border to wall, but various additional boundary choices, such as defined wave with specified length and height, this wave options selected under the limits of Fourier series method and the solution of Stokes and Cnoidal, were provided in the software that used in upstream boundary. A symmetry option has been modified for the mesh in the y-direction. The initial condition was defined as the fluid area at the top border of the mesh (1) and hydrostatic pressure was established at the top border.

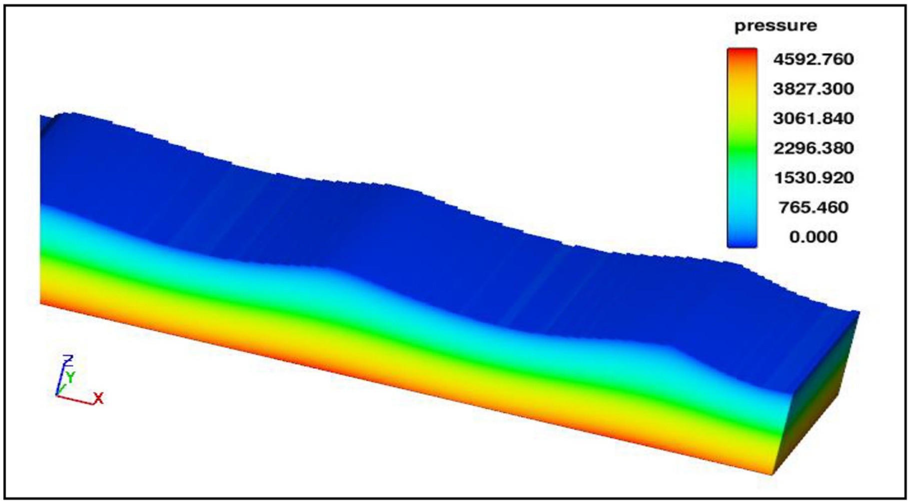

Fig. 1. FLOW 3D results in three dimension.

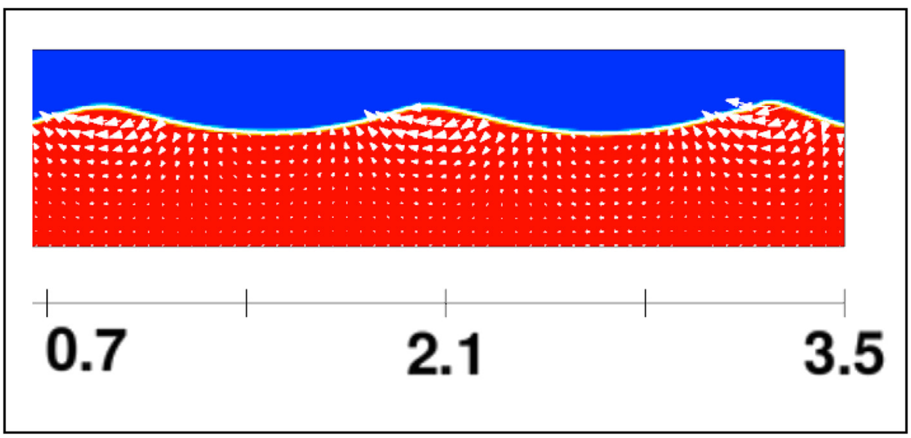

Fig. 2. FLOW 3D results in two dimension.

7. Numerical simulations options and results

As mentioned above, the numerical tab of the FLOW 3D model set-up has a wide range of choices. These choices have shown an adaptation of the Reynold, which are the essential underlying equations of the FLOW 3D, in Navier Stokes (RANS) averaged equations. The default selections have been utilized for the majority of completed simulations.

The time step settings were left as standard without the simulation crashing with the error message indicating the time step was less than the minimum. Sometimes a smaller finishing time was tried to get a convergent solution in this situation. Generalized Minimum Residual (GMRES) pressure resolution simulations using the default parameters. Simulations were typically finished by default explicit solver choices, the difference is that an express solution is gradually resolved in each computer cell by stepping over time, while the time step is limited to fulfill the stability conditions. Implicit solution performed at each step utilizing knowledge of a different phase, which does not impose any time constraints but needs more refined iterative or matrix solutions. Most simulations were conducted with a chosen default button in the volume of the fluids advection part of the Numeric tab, so that the solver of software would automatically choose the single fluid free surface choice, according to the parameters given in the global tab. Found in the 3D handbook of FLOW 3D users (2007), all simulations were also performed during solution of both continuity and momentum equations and with first order momentums, the results of final simulation model shown in Figs. 1 and 2, for three and two dimensions, respectively.

8. Conclusions

Since the CFD approach used as numerical simulations with Flow 3D software, the simulate of the wave behavior phenomenon by numerical technique depending on Flow 3D options and Fourier series method shows an improvement that the Flow 3D software able to generate waves and its breaking behavior with different heights and periods. The numerical simulations using CFD approach should be studied to show the ability of Flow 3D software to simulate the wave behavior phenomenon. The results present an improvement that the Flow 3D software able to generate the waves with different heights and periods.

Declaration of Competing Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

References

[1] Abd Alall, Mostafa. "Numerical Investigation of hydrodynamic Performance of Double Submerged Breakwaters", International Journal of Scientific & Engineering Research 11(3), (2020). ISSN 2229-5518

[2] Ahmed, Hany and Abo-Taha, M. "Numerical Investigation of Regular Waves Interaction with Submerged Breakwater", International Journal of Scientific & Engineering Research 10(11), (2019). ISSN 2229-5518.

[3] S.T. Grilli, M.A. Losada, F. Martin, Characteristics of solitary wave breaking induced by breakwaters, J. Waterway, Port, Coastal, Ocean Eng. 120 (1) (1994) 74–92.

[4] F. Hajivalie, A. Yeganeh-Bakhtiary, Numerical study of breakwater steepness effect on the hydrodynamics of standing waves and steady streaming, J. Coastal Res. (2009) 658–662.

[5] F. Hajivalie, A. Yeganeh-Bakhtiary, J.D. Bricker, Numerical study of the effect of submerged vertical breakwater dimension on wave hydrodynamics and vortex generation, Coastal Eng. J. 57 (3) (2015) 1550009-1–1550009-21.

[6] Hayakawa, Norio, Tokuzo Hosoyamada, Shigeru Yoshida, and Gozo Tsujimoto. "Numerical simulation of wave fields around the submerged breakwater with SOLA-SURF method." In Coastal Engieering 1998, pp. 843-852. (1999).

[7] D.-S. Hur, C.-H. Kim, D.-S. Kim, J.-S. Yoon, Simulation of the nonlinear dynamic interactions between waves, a submerged breakwater and the seabed, Ocean Eng. 35 (5-6) (2008) 511–522.

[8] D.-S. Hur, K.-H. Lee, D.-S. Choi, Effect of the slope gradient of submerged breakwaters on wave energy dissipation, Eng. Appl. Comput. Fluid Mechanics 5 (1) (2011) 83–98.

[9] K. Kawasaki, Numerical simulation of breaking and post-breaking wave deformation process around a submerged breakwater, Coastal Eng. J. 41 (3-4) (1999) 201–223.

[10] B. Liang, G. Wu, F. Liu, H. Fan, H. Li, Numerical study of wave transmission over double submerged breakwaters using non-hydrostatic wave model, Oceanologia 57 (4) (2015) 308–317.

[11] H.A.H. Petit, P. Tönjes, M.R.A. Van Gent, P. Van Den Bosch, Numerical simulation and validation of plunging breakers using a 2D Navier-Stokes model, Coastal Eng. 1994 (1995) 511–524.

[12] A. Sasikumar, A. Kamath, O. Musch, A. Erling Lothe, H. Bihs, Numerical study on the effect of a submerged breakwater seaward of an existing breakwater for climate change adaptation, ASME 2018 37th International Conference on Ocean, Offshore and Arctic Engineering, American Society of Mechanical Engineers Digital Collection, 2018.

[13] Takahiro Uemura, A numerical simulation of the shape of submerged breakwater to minimize mean water level rise and wave transmission, TVVR13/5004 (2013).