Stepped Mound Breakwater Simulation by Using Flow 3D

Hasan Ibrahim Al Shaikhli*	Civil Engineering Department, College of Engineering, University of Basrah, Basrah, Iraq.
Prof. Dr. Saleh Issa Khassaf	E.hassanibraham@gmail.com, Saleh.Khassaf@uobasrah.edu.iq

ABSTRACT

Waves action is one of the most well-known hydraulic events in history. When waves hit rocks or other solid blocks, they form a breakwater. This made it necessary to build breakwaters, which fill the empty spaces with more rocks to create an equilibrium. Because experimental work is hard and models are expensive, CFD-based numerical simulations should be studied to show how well Flow 3D software can simulate the way waves behave. In this article five models has been studied to investigate the effect of stepped mound breakwater on the behavior of breaking waves with different main sea level above breakwater crest. The results show that the Flow 3D software is getting better because it can now make waves with different heights and lengths, also, the increase in steps cause an increase in dissipation of energy and an increase in wave transmission coefficient.

Keywords: Breaking waves, Wave Generation, Flow 3D, Numerical Analysis, CFD

Introduction

Most of what is known about using CFD to model breakwaters comes from studies that used different simulation software. The results of studies that used FLOW 3D to model different hydraulic structures showed a good match between the real-world results and the standard design rules. But many kinds of software packages were used to simulate the flow over the breakwater and study its hydraulic behavior. These include information on how CFD and Fluent software have been used successfully and comparisons of CFD and Fluent. Based on earlier research that dealt with hydraulic problems, FLOW 3D and Fluent were thought to be useful for solving advanced problems involving the interaction of fluids and solids. They could also provide important results.

Grilli et al.(1994) caused single waves to break over breakwaters in the lab and with a nonlinear potential model. Depending on the height of the wave that hits, waves can collapse above the crest or break backwards or forwards over submerged breakwaters. In experiments and tests, the wave transmission and reflection coefficients were found. Over submerged breakwaters, the transmission coefficient was seen to go up to (55–90%). For the breakwaters that are under water, nonlinear potential model calculations agree with lab data, especially for smaller wave heights. This model's calculations give a good estimate of how far a collapse could push over submerged breakwaters.

Petit et al. (1995) used the 2D Navier-Stokes equation to simulate plunging breakers. This numerical model talked about the SKYLLA approach, which was made to simulate breaking waves on coastal structures. The Volume of Fluid method is used to solve the Navier-Stokes equations in two dimensions. In a computational domain, weakly reflecting boundaries let water waves enter and leave the domain. So, impermeable barriers might be used. When compared to the results of physical model testing for waves on a submerged structure with a 1:20 slope, the two-dimensional Navier-Stokes equations can simulate waves over a low-crowned structure.

Hayakawa et al. (1999) used the SOLA-SURF method to simulate wave fields in two and three dimensions around a submerged breakwater. For the three-dimensional computation of the laboratory tank, the results of numerical models should be compared to the results of experiments, and the non-viscous side wall boundary condition, slip wall type, should be used for the non-viscous side wall boundary condition. The numerical approach shows how it can be used in the field.

Kawasaki (1999) said that a model combines a non-reflective wave generator with VOF approach and a dissipation zone with an open boundary to simulate the breaking and post-breaking wave deformation process around a submerged breakwater for a two-dimensional wave field in the vertical plane. In addition to how waves break over a submerged breakwater, the proposed model also looked at how waves change after they break. There were tests done in the lab to prove that the model was correct. Wave height and frequency of wave amplitude are looked at from different angles. Calculations and observations show that a circulating flow is made on the onshore side of the submerged breakwater when waves break. By comparing the proposed numerical wave model to experimental data, it is shown that it accurately shows how waves change shape before and after they break.

Hwang et al. (2004) chose the RANS model to simulate the impermeable submerged double breakwaters and to figure out how waves move forward, how vortices form, and how the energy they produce is lost. The k–e model and the height function are called the turbulence dynamics model and the free surface, respectively. The governing equations were broken up using a limited-volume method based on a grid system with different heights and widths. A series of suggested numerical answers were analyzed and developed, and then some experimental data was used to check the numerical model. The results of numerical models show that they agree with reality well enough to be thought of as a cheap way to simulate the flow of wave fields near coastal structures.

Hur et al,. 2008, The researchers used numerical simulation to look at the nonlinear interactions between the ocean waves, the seabed, and the submerged breakwater under different wave conditions. Laminar flow is used to control how hard it is for fluid to move through the porous seabed in a numerical simulation. The flow results of how waves, the seabed, and the breakwater interact were tested for the cross-section of the submerged breakwater when the wave situation and pore water pressure changed. The numbers show that the breakwater helped protect against the flow field and the changes.

In 2009, Hajivalie and Bakhtary used a 2D numerical model to study the effect of turbulence based on the (RANS) equations and the steepness of the breakwater on the waves standing still and the steady stream. The k-turbulence model was used as a numerical model, and the Volume of Fluid (VOF) approach was used to watch the shape of the free surface waves. When it came to standing waves on a breakwater, the numerical results were compared to the experimental results. Three simulations were made for three different kinds of breakwaters, in addition to the wall on a vertical axis and sloped breakwaters with slopes of 1:2 and 2:1. The slope is not too steep, so the water flows steadily and the cell is recirculated, which might stop the scouring at the base of a breakwater.

Hur et al. (2011) said that permeable submerged breakwater studies on the effects of the slope gradient led to the same conclusions. Using a newly built numerical model, the flow through a porous material with inertial, linear, and nonlinear resistance is looked at. So, for a breakwater that is under water, the numerical model can simulate how the wave and seabed work together. In a 2-dimensional wave field, the LES turbulence model calculated the eddy viscosity. By comparing the generated to existing data for wave deformations caused by the rectangular, permeable, submerged breakwater, it was shown that the model was correct. When the slope gradient goes down, the transmission coefficient goes down and the waves that break on the submerged breakwater move toward the sea. Behind the submerged breakwater, the water flowed in a clockwise direction.

Uemura (2013) uses numerical simulation to study the erosion and wave transmission for a submerged breakwater. In order to solve these problems, breakwaters have been built to keep the coast from eroding too much. On the other hand, breakwaters have been made to lower both the average water level and the way waves travel. In terms of spectrum analysis, sea water level, and wave heights, the number calculation was done in two dimensions. submerged banks with many vertical, impermeable plates were figured out using numbers, and the results were compared to real data. The proposed submerged bank showed a decrease in both short and long wave transmission and a drop in water level behind the submerged breakwater.

Hajivalie et al. (2015) used numerical modeling to study how the size of a vertical breakwater affects the way waves move through the water and how vortices form around the breakwater. Two parameters without units, like Keulegan–Carpenter and the breakwater submergence, are made. The computational model was made up of the k–turbulence model, the (RANS) equations, and the VOF technique for tracing the free surface. For this study, ten different models were made, and each model had a different value for the depth of a submerged breakwater. As a/Hi goes up, the transmission coefficient goes up. For the waves that were looked at, the transmission coefficient goes up quickly. With this design, the width of the breakwater was made wider, and as a/Hi went up, the amount of turbulence on the sea side of the breakwater went down. The best a/Hi for a high rate of energy loss and a shallow scour depth.

Lianga (2015) studied numerically the transmission wave over double trapezoidal submerged breakwaters using non-hydrostatic wave. The model said that a non-hydrostatic wave model called SWASH (Simulating Waves until Shore) is used to simulate wave transmission. The numerical results were compared to the results of the physical model, so the SWASH model, which can predict wave transmission through double submerged breakwaters, was made and the effect of sea current on wave transmission was looked into. When the relative depth of the water stays at 1.0, the best relative breakwater spacing is around 1.11. When it comes to component dissipation, a superharmonic wave is more visible than a lowerharmonic wave.

Sasikumar, et al. (2018) search numerically for the effect of a submerged breakwater on adapting to climate change. It is known that mean sea level rises when the climate warms, which makes storms more common in coastal areas. The breakwaters will be hit by waves that are taller than what was planned. This will happen many times. Because of this, these effects may have an effect on how breakwaters work. Researchers used a numerical model to figure out the best size for a breakwater structure that will be submerged and used in coastal areas. The relationship between the submergence and width parameters was studied as the transmission coefficient changed. In the case of Kiberg, Norway, the results show that putting a submerged breakwater in front of an existing rubble mound barrier caused it to break. The wave prediction model CGWAVE used the finite element method on a local scale. This model gave the best shape to protect coastal areas.

Ahmed and Abo-Taha (2019) use FLOW 3D as a computer model for regular waves to study the hydrodynamic performance of half-pipe submerged breakwaters along the Egyptian coast. Three different diameters of precast half-pipes, two of which are horizontal and one of which is vertical, are used to figure out the relative structure height that lets the most energy escape. The models are analyzed numerically to find the wave energy dissipation, transmission coefficient, and reflection coefficient. The numerical results show that it is possible to predict how half-pipe submerged breakwaters will work in terms of hydrodynamics. Abd Alall, 2020, will study a numerical model of the hydrodynamic performance of double submerged breakwaters. This model used precast concrete half pipes submerged breakwater as natural coastal protection to stop beach erosion along the Egyptian coast. The separated half pipes submerged breakwater were looked at using a numerical model FLOW 3D software for linear waves as hydrodynamic performance. Using two models, the best distance between half-pipe breakwaters that are submerged was found. The transmission coefficient (Kt), the reflection coefficient (Kr), and the wave energy loss were all calculated using numbers. The experimental results approved the numerical results, which were then used to estimate how well submerged half pipes work from a hydrodynamic point of view. (Khassaf and Abbas, 2018 and 2019).

Flow 3D Approach

Flow 3D (V.11.2) software for solving cartesian coordinates 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 relatively small areas called cells and calculating the numerical flow value. The most important thing for all numerical models is to set up the right mesh domain, which reacts correctly to how the phenomenon behaves. (Al Shaikhlia and Khassaf, 2021)

To get different numerical approximations of the control equations, control volumes are made around each position of a variable. Surface fluxes, surface stresses, and body strengths can be calculated for each control volume based on the values of other variables in the area. Then, these amounts are added together to make an estimate for the protection laws that the movement equations show. (Ismaeel, etal., 2021) and (Dakheel, etal., 2022).

Numerical Model Set-Up

Overall, the way the model was set up was pretty similar to how all controlled breaking waves simulations were set up. In each case, the Global Tab was used to define a fluid, incompressible flow and a free surface or sharp contact. Also, for all simulations, the properties of the fluid were set to be those of water at 20°C. Several other parameters stayed the same, and the following parts will be looked at in more detail.

Physics

Even though there are many other physical options, only two of them had to be turned on in order to get accurate simulations of the data needed for this study. The gravity option was turned on when the acceleration of gravity in the vertical, or z-direction, reached -9.81 m/sec2. When Newtonian viscosity and a good turbulence model were used on the flow, the option for viscosity and turbulence was also turned on. As long as the two-equation (k-w) model is chosen, one turbulence model is used once the FLOW 3D model has been fully built. Based on comments in the FLOW 3D user's handbook, the (k-e) turbulent model is the best and best available model for simulating breaking waves in the software (2007).

Geometry

For other breaking waves that FLOW 3D had modeled before, the preparation for the geometry of the numerical model was very different. According to the experimental investigation, the geometry used in the simulations was given as a stereolithography (STL) picture made in AutoCAD and exported in STL format. If that happens, the right network can be built.

The usual concrete ruggedness value for all flume geometries was assumed to be ignored, and the geometry component remained the standard choice for all models done as part of this study.

Conditions at the start and at the border

An important and useful computer tool is the ability to figure out how much pressure is on one or more of the edges of the computing domain. Pressure limits include small fluid reservoirs, lab conditions, and the mechanical pressures that are put on the system. Most of the time, there are two kinds of pressure, which are called static pressures and stagnant pressures. In a static situation, the pressure on both sides of the border is more or less the same, and the speed limit is given a value based on a normal zero derivative state on both sides of the border. There are better outflow boundary conditions for a number of types of flow problems. For example, specific boundary therapies have been made for problems with waves spreading. These therapies try to figure out the speed and direction of waves as they approach the boundary and then set boundary conditions to allow the least amount of waves to reflect across the border. An early and useful example of this kind of treatment, which is sometimes called a "radiation limitation condition," was the use of x-rays (Orlanski, 1976).

Boundary Conditions for the Turbulence Model

At all borders except for hard, non-slip ones, the energy of turbulence and the dissipation functions are looked at at the same time, along with other cell-centered variables like density. At symmetry limits, there are no specific rules because there are zero-speed derivatives across the boundary. This means that there is no turbulence. There is also a region where there is no flow, so neither advective nor diffusive flows are guaranteed. Still, there are some things that need to be taken into account for stiff, non-slip borders, since the numerical resolution is usually not high enough to fix the features of a laminar border layer area. For the wall shear-stress model, a rough velocity profile was used. To be consistent, the wall limits for the turbulence energy and the dissipation of turbulence functions must be set.

FAVOR technique, in which solid walls can be cut across a mesh cell at any angle, it is not clear how these limits can be met. So, this method should be used. Turbulence energy and dissipation values are set for all cells with a part or fully blocked no-slip, stiff edge. The Wall boundary values are found by starting with the assumed velocity profile (logarithmic law approximation) and assuming that there is a local balance between turbulent production and decay.

Boundaries of Free Surface

The Incompressible SOR method shows how to set pressure using normal stress and the required pressure on the free surface (Only single fluid problems have free-surface boundaries requirements). The tangent stresses of a free surface are zero because there are no speed derivatives with velocity components outside the surface that are not zero. Speeds should be changed at every cell line between a surface cell and an empty cell, but the flow of the fluid should also be taken into account. There are two ways to do this. First, for each speed component on a side next to the empty cell, the value on the opposite side of the surface cell is set. In the second stage, the surface cell will be changed to try to get the cell's speed to be the same everywhere. Only, in this method, the speeds are changed on the sides where there are empty cells. The difference in cell speed can't be made equal to zero because the correction is also based on how much fluid is in the cell.

The above limits are put in place to make sure that the flow is made up of internal obstacles. Setting up good boundary conditions has a big effect on whether or not the results of the numerical model match the real situation that was tried to be duplicated. In this case, the mesh block (1) asked for data from surface free fluxes, so the top border was set to atmospheric pressure and the bottom border was set to wall. To get the channel bed, the bottom edge was put right below the model's input form, and the top edge was put right above the highest water level in the vertical or z-direction grid.

To make the simulation similar to the real-world work, the upstream limit was set as a certain stress with a certain fluid height. This changed the mean water depth and wave height. The simulation also chose to set the downstream boundary to a wall, but the software used in the upstream boundary gave the user other boundary options, such as a defined wave with a set length and height. This wave option was chosen within the limits of the Fourier series method and the solution of Stokes and Cnoidal. In the y direction, a symmetry setting for the mesh has been changed. The fluid area at the top edge of the mesh (1) was set as the initial condition, and hydrostatic pressure was set at the top edge.

Options and Results of Numerical Simulations

As was said above, the numerical tab of setting up the FLOW 3D model gives you a lot of options. These choices have shown that the Reynold equations, which are the most important equations for FLOW 3D, can be changed into averaged Navier Stokes (RANS) equations. Most simulations that have been finished have used the default settings.

The time step settings were left at their defaults, and the simulation did not crash with an error message saying that the time step was less than the minimum. In this case, sometimes a shorter time to finish was tried to get closer to a solution. Simulations of pressure resolution with Generalized Minimum Residual (GMRES) using the default parameters. By default, explicit solvers were used to run simulations. The difference is that an explicit solution is found gradually in each computer cell by taking steps through time, while the time step is limited to meet the stability conditions. Implicit solution is done at each step using knowledge of a different phase. There are no time limits, but iterative or matrix solutions need to be more refined. Most simulations were done with the default button selected in the volume of fluids advection section of the Numeric tab. This way, the solver software would automatically choose the single fluid free surface option based on the parameters given in the global tab. All simulations were also done during the solution of both continuity and momentum equations and with first order momentums. The results of the final simulation model are shown in figures 1 to 5 the Flow 3D results of all model, figure 6 to 9 the results of Kt against number of steps for all heads

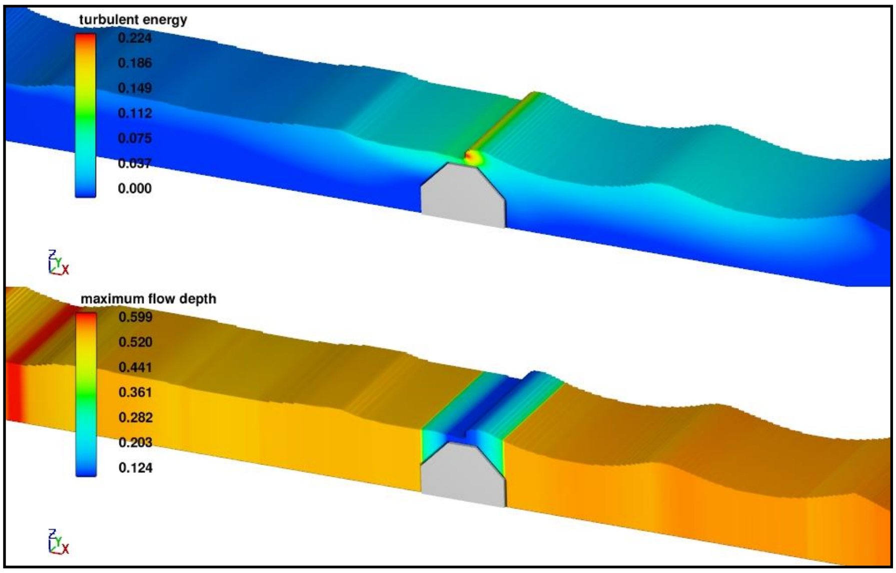

Figure 1: FLOW 3D results in three dimension

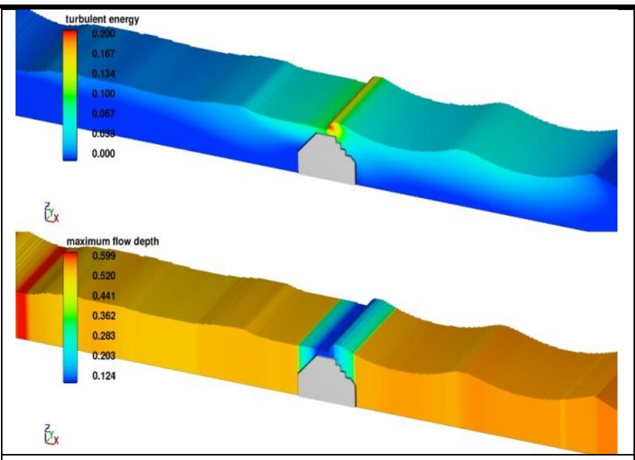

Figure 2: results of 4 steps

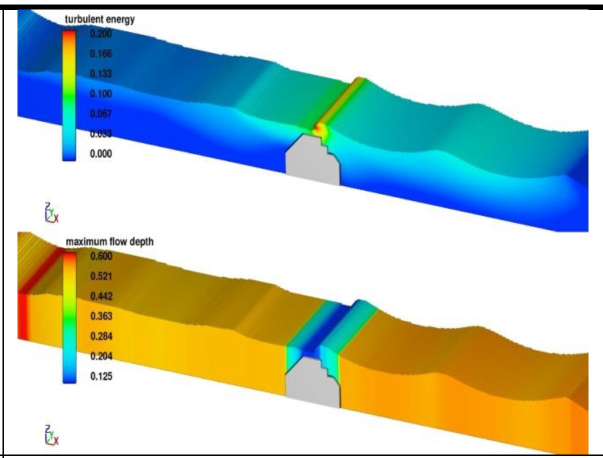

Figure 3: results of 3 steps

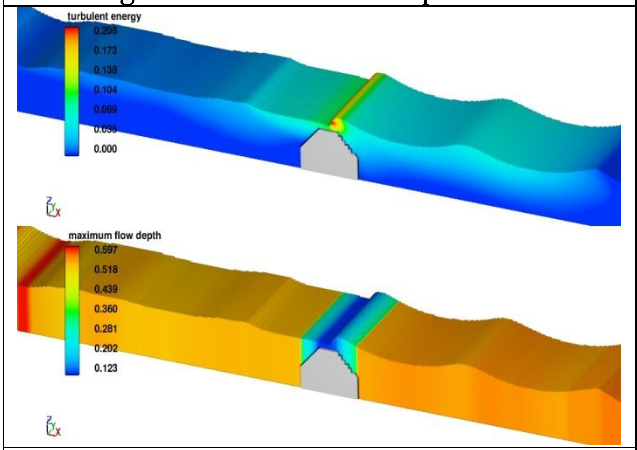

Figure 4: results of 6 steps

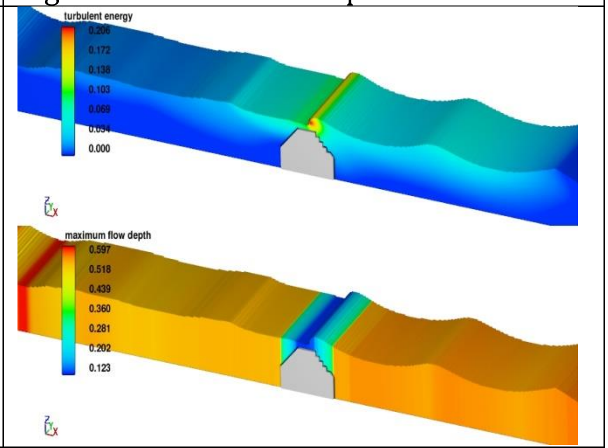

Figure 5: results of 5 steps

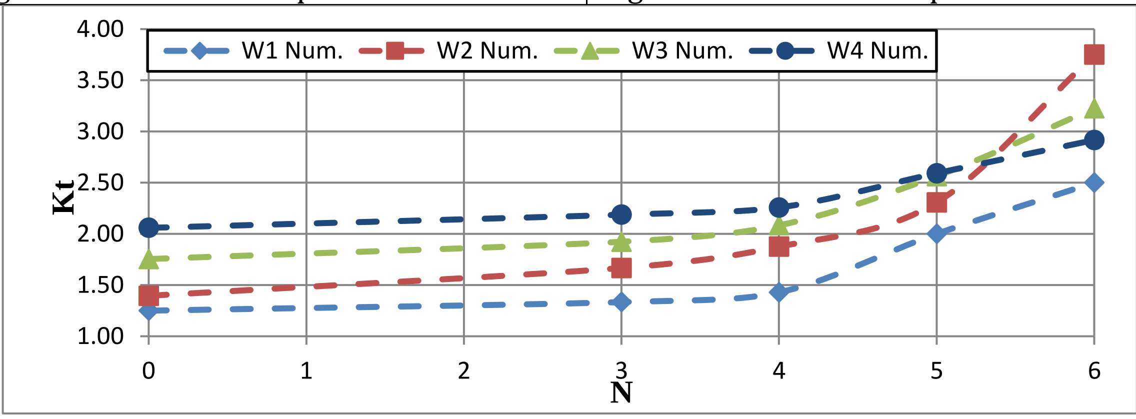

Figure 6: Kt Vs N for head =0 cm

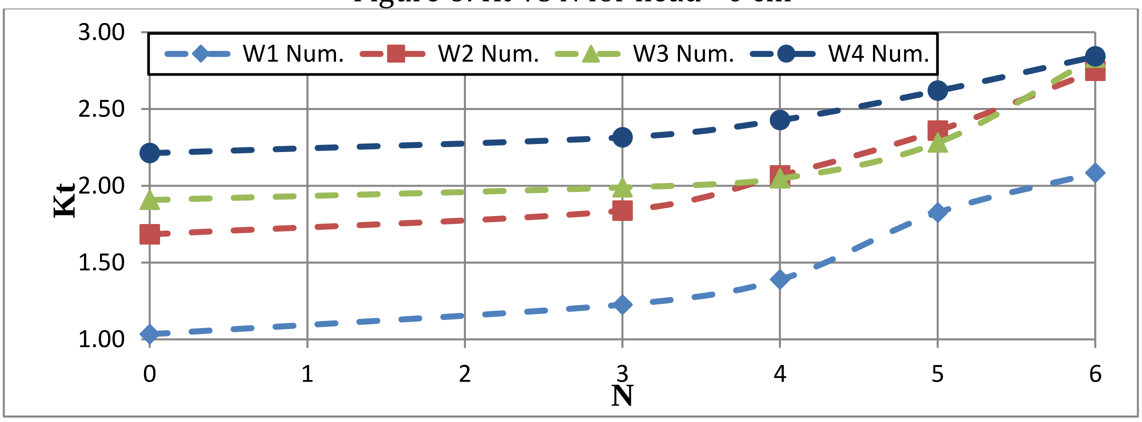

Figure 7: Kt Vs N for head =2 cm

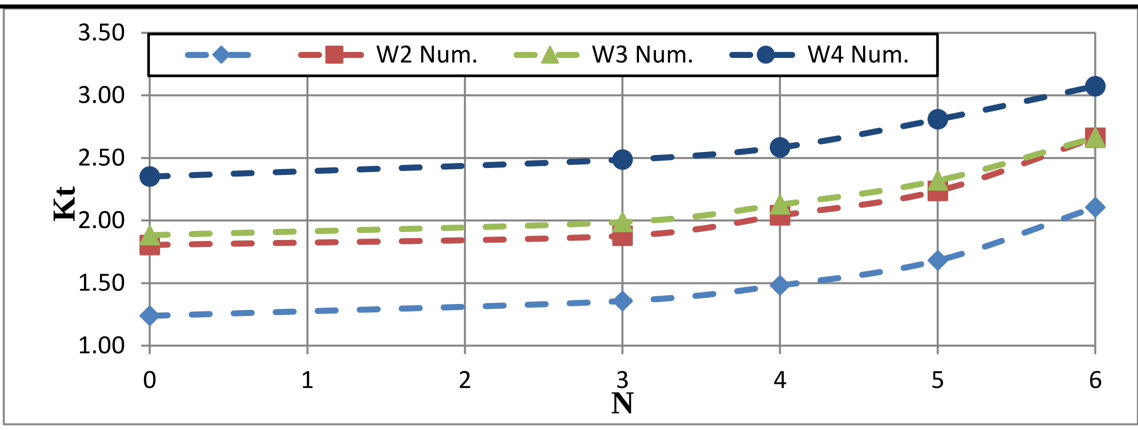

Figure 8: Kt Vs N for head =4 cm

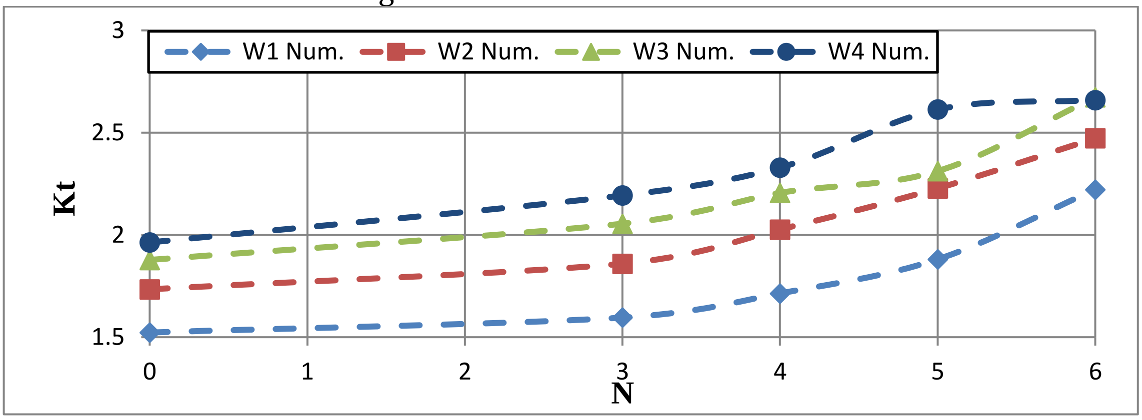

Figure 9: Kt Vs N for head =6 cm

Conclusions

Since the CFD methodology was utilized, there has been an increase in the ability of the Flow 3D software to create waves and their breaking behavior with varying heights and periods. This was discovered through the numerical simulation of wave behavior based on choices made in Flow 3D and the method of calculating the Fourier series.The increase in wave incident height cause an increase in transmission height of wave for (0, 2, 4, 6, 8 and 10) cm water level above breakwater crest. For (30, 45 and 60) degree of side inclination, the increase in number of steps cause an increase in wave transmission coefficient for height of wave (0, 2, 4 and 6) cm, until the flood point occur at head more than or equal (8) cm above breakwater crest.

References

1. Abd Alall, Mostafa. “Numerical Investigation of hydrodynamic Performance of Double Submerged Breakwaters”, International Journal of Scientific & Engineering Research Volume 11, Issue 3, (March-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 Volume 10, Issue 11,(2019). ISSN 2229-5518
3. Grilli, Stephan T., Miguel A. Losada, and Francisco Martin. "Characteristics of solitary wave breaking induced by breakwaters." Journal of Waterway, Port, Coastal, and Ocean Engineering 120, no. 1 (1994): 74-92.
4. Hajivalie, F., and A. Yeganeh-Bakhtiary. "Numerical study of breakwater steepness effect on the hydrodynamics of standing waves and steady streaming." Journal of Coastal Research (2009): 658-62.
5. Hajivalie, Fatemeh, Abbas Yeganeh-Bakhtiary, and Jeremy D. Bricker. "Numerical study of the effect of submerged vertical breakwater

dimension on wave hydrodynamics and vortex generation." Coastal Engineering Journal 57, no. 03 (2015): 1550009.

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 Engi
7. eering 1998, pp. 843-852. (1999).
8. Hsu, Tai-Wen, Chih-Min Hsieh, and Robert R. Hwang. "Using RANS to simulate vortex generation and dissipation around impermeable submerged double breakwaters." Coastal Engineering 51, no. 7 (2004): 557-579.
9. Hur, Dong-Soo, Chang-Hoon Kim, Do-Sam Kim, and Jong-Sung Yoon. "Simulation of the nonlinear dynamic interactions between waves, a submerged breakwater and the seabed." Ocean Engineering 35, no. 5-6 (2008): 511-522.
10. Hur, Dong-Soo, Kwang-Ho Lee, and Dong-Seok Choi. "Effect of the slope gradient of submerged breakwaters on wave energy dissipation." Engineering Applications of Computational Fluid Mechanics 5, no. 1 (2011): 83-98.
11. Kawasaki, Koji. "Numerical simulation of breaking and post-breaking wave deformation process around a submerged breakwater." Coastal Engineering Journal 41, no. 3-4 (1999): 201-223.
12. Liang, Bingchen, Guoxiang Wu, Fushun Liu, Hairong Fan, and Huajun Li. "Numerical study of wave transmission over double submerged breakwaters using non-hydrostatic wave model." Oceanologia 57, no. 4 (2015): 308-317.
13. Petit, H. A. H., P. Tönjes, M. R. A. Van Gent, and P. van Den Bosch. "Numerical simulation and validation of plunging breakers using a 2D Navier-Stokes model." In Coastal Engineering 1994, pp. 511-524. (1995).
14. Sasikumar, A., Kamath, A., Musch, O., Erling Lothe, A., & Bihs, H. (2018). Numerical study on the effect of a submerged breakwater seaward of an existing breakwater for climate change adaptation. In ASME 2018 37th International Conference on Ocean, Offshore and Arctic Engineering. American Society of Mechanical Engineers Digital Collection.
15. Uemura, Takahiro. "A numerical simulation of the shape of submerged breakwater to minimize mean water level rise and wave transmission." TVVR13/5004 (2013).
16. Abbas, H.A. and S.I. Khassaf, “Local scour evaluation around non-submerged curved groynes”, International Journal of Civil Engineering and Technology (IJCIET), Vol. 10, Issue1, pp. 155-166, January (2019).
17. Ahmed A. Dakheel, Ali H. Al-Aboodi, Sarmad A. Abbas, (2022), Assessment of Annual Sediment Load Using Mike 21 Model in Khour Al-Zubair Port, South of Iraq, Basrah Journal for Engineering Sciences, Vol. 22, No. 1, (2022), 108-114.