Evaluating the Hydraulic Performance of Conical Pile Head Breakwater Using REEF3D: An Open Source CFD Tool

Naveen Rao, Arunakumar Hunasanahally Sathyanarayana,
U Pruthviraj and Shrikantha S Rao

October 31, 2023

Evaluating the Hydraulic Performance of Conical Pile Head Breakwater Using REEF3D: An Open Source CFD Tool

Naveen Rao[1*], Arunakumar Hunasanahally Sathyanarayana[2], Pruthviraj Umesh[3], Shrikantha S. Rao[4]

[1] University of New South Wales, School of Civil and Environmental Science, Sydney, Australia
[2] CSIR – National Institute of Oceanography (NIO), Goa, India
[3] Department of Water Resources and Ocean Engineering, National Institute of Technology Karnataka, Surathkal, India
[4] Department of Mechanical Engineering, National Institute of Technology Karnataka, Surathkal, India
[*] nrao19961@gmail.com

ABSTRACT

Computational Fluid Dynamics (CFD) employs numerical methods and computer simulations to analyze and predict fluid flow behaviors and their interactions with surfaces. These interactions can be studied through experiments and simulations. Although experimentation has been the traditional approach to studying fluid flow and interactions, CFD has undergone a massive shift with the advancements of simulation programs. Various water modeling tools have been developed and released into the market that can run multiple and repetitive simulations all whilst allowing for the possibility of changing the initial parameters. The development of such tools incurs a cost and comes with a cost to the user. With the advent of open-source tools which is a result of increased access to the internet and connectivity between scholars around the globe, numerical modeling tools are available as open-source programs rather than as a product. The benefit of working with an open-source tool is the plausibility of any researcher from any part of the world being able to understand the core of the program and make any additions to the program as needed. This constantly upgrades the tool. This paper aims to show the vast potential that the tool REEF3D holds through a case study of a CFD simulation all while highlighting how open-source tools can create a significant impact on the field of CFD.
Keywords: CFD; Numerical modeling; REEF3D; Open source.

1 Introduction

Computational fluid dynamics (CFD) involves identifying and defining a problem as a mathematical model and solving the math model using a numerical model [1]. The numerical model is usually a computer simulation. The mathematical model in the field of water engineering generally uses the Navier-Stokes equation since they are the most accurate representation of fluid behaviour [1]. The mathematical models define the CFD problems as Partial Differential Equations (PDEs). There are three principal approaches to solving these PDEs namely the Finite Difference Method (FDM), the Finite Volume Method (FVM), and the Finite Element Method (FEM) [1]. The continuous fluid problem domain is replaced by discrete domains using a selected mesh or grid [2]. At these grid points, the relevant flow variables are solved directly and the values at other locations are derived by interpolation of the values at the grid points [2].

FDM and FEM, FVM have had a stark difference in the timeline from when they came into being. FDM work in CFD was first published in 1910 at the Royal Society of London on FDM solution for stress analysis of a masonry dam [3]. In contrast, the first FEM work in the field of CFD in aircraft stress analysis in 1956 [3]. The application of FDM dominated the field of CFD in the initial days. With digital computers getting better over the days FEM and FVMs are providing superior performance in the simulations. Over the years, numerous tools have been developed to run CFD models and marketed to researchers. Most of these tools are developed as a product that can give monetary returns. The research community is constantly working on CFD simulation and is brimming with concepts. These concepts can be actualized by repeatedly updating the modeling tools. REEF3D is a result of Prof. Hans Bihs's (NTNU Norway) efforts during his time as a Ph.D. scholar working on local scouring. Due to the prominent role of hydrodynamics in local scouring, his research demanded a more sophisticated turbulence model and free surface algorithm. This led him to impose an Explicit Algebraic Reynolds stress model into SSIM [4]. This in turn led to the creation of REEF3D, a new numerical model that has an integrated interface-capturing algorithm. In the further sections of this paper, the application of REEF3D in hydrodynamic modeling is explored through a case study of wave interactions with conical pile head breakwater.

Conventional breakwaters are gravity-based rubble mounds or caisson or composite walls in the waterbody. They mainly function as barriers that reduce wave activities by completely blocking them off or by attenuation then by dissipating wave energy by reducing its turbulence or by reflection. Pile breakwaters fall under non-gravity-based breakwaters made of single or multiple rows of prismatic piles placed closely [5]. It is generally an emergent structure that majorly attenuates the wave energy by dissipating turbulence and a minor amount by reflection [6]. It is evident from the literature review that a minimal study has been undertaken by varying the cross-sectional area of the piles at the water surface level. Therefore, this study intends to bridge the knowledge gap. The concept of increasing the area of the piles at the upper portion in a conical shape is to have a larger area of structure at the water surface, which obstructs the predominant wave energy. The typical numerical wave tank with a conical pile head breakwater (CPHB) structure is presented in Figure 1. This study evaluates the performance characteristics, viz., the wave transmission coefficient (K_t), the reflection coefficient (K_r), and the dissipation coefficient (K_d) of the new concept of CPHB through numerical modeling open-source CFD tool REEF3D.

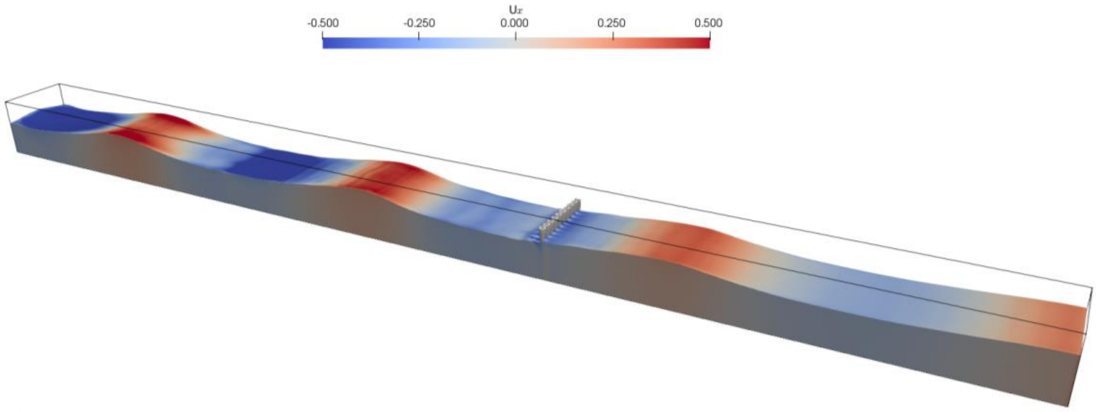

Figure 1. Numerical wave tank with conical pile head breakwater.

To validate the numerical results of CPHB, the current work is compared with the physical model studies carried out on the non-perforated CPHB using experimental facilities at NITK Surathkal, India [7]. The structural and wave parameters of the numerical model are in accordance with this physical model study.

2 Numerical Methodology

REEF3D runs by applying incompressible Reynolds-Averaged Navier–Stokes (RANS) equations in line with the continuity equation to solve fluid flow problems [5].

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

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\left(\nu + \nu_t\right)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + g_i \tag{2}$$

The terms u_i stands for the averaged velocity throughout time t, ρ represents the density of water, ν stands for kinematic viscosity, ν_t is the eddy viscosity, p represents pressure and g is the acceleration due to gravity. The projection method proposed by *Chorin* in the Numerical solution of the Navier-stokes is used to solve the pressure terms in the RANS equation [8]. The Poisson equation for pressure is solved using the BiCGStab algorithm [9]. The convection terms of the RANS equation are discretized using the fifth order Weighted Essentially Non- Oscillatory scheme (WENO) [10]. The third-order TVD Runge-Kutta scheme discretized the time [11]. *Brackbill et al's* continuum surface force (CSF) model suggests a method to calculate the material characteristics of the two phases for the numerical domain [12]. *Wicox's k-ω* model is applied to model turbulence where k stands for kinetic energy and ω stands for specific turbulence dissipation rate [13]. To maintain the numerical stability during the runtime of the simulation, optimal time steps are determined using the Courant-Friedrichs-Lewy (CFL) criterion. A Message Passing Interface (MPI) maximizes the efficiency of the numerical model by carrying out parallel computation between multiple cores. The level set method differentiates the free surface between air and water [14], [15].

3 Results and Discussion

To simulate field conditions of wave height, wave period, and pile head diameter, a geometrical scale of 1:30 is selected. The parameters considered for the investigation are listed in Table 1. After running the simulations, the results are compared with laboratory experiments that were conducted in the past using the same dimensions of the piles and the wave flume including the spacing between the piles. The wave interaction of the CPHB is illustrated in Figure 2. For a typical case of CPHB (with $D/H_{max} = 0.4$, $Y/H_{max} = 1.0$, $b/D = 0.1$), the comparison of numerical results with the physical modeling data is illustrated in Figure 3. Also, an error analysis is performed to quantify the differences between the numerical and physical modeling results. RMSE value is calculated to quantify the results. RMSE values obtained for K_t, K_r, and K_d are 0.045, 0.045, and 0.05, respectively. The numerical model using REEF3D produced the desired results within the margin of error and is proven to be useful in running further complicated models. The same model can be further optimized by changing the arrangement and introducing the perforations on the conical pile head surface.

Table 1. Parameters considered for numerical study.

Variables	Expression	Parameter Range
Conical pile head diameter (m)	D	0.064
Height of conical pile head (m)	Y	0.160
Wave height (m)	H_i	0.06,0.08,0.10,0.12,0.14,0.16
Maximum wave height	H_{max}	0.16
Wave period (s)	T	1.4, 1.6, 1.8, 2
Water depth (m)	d	0.40
Relative pile head diameter	D/H_{max}	0.4
Relative pile head height	Y/H_{max}	1.0
Relative clear spacing between piles	b/D	0.1, 0.2

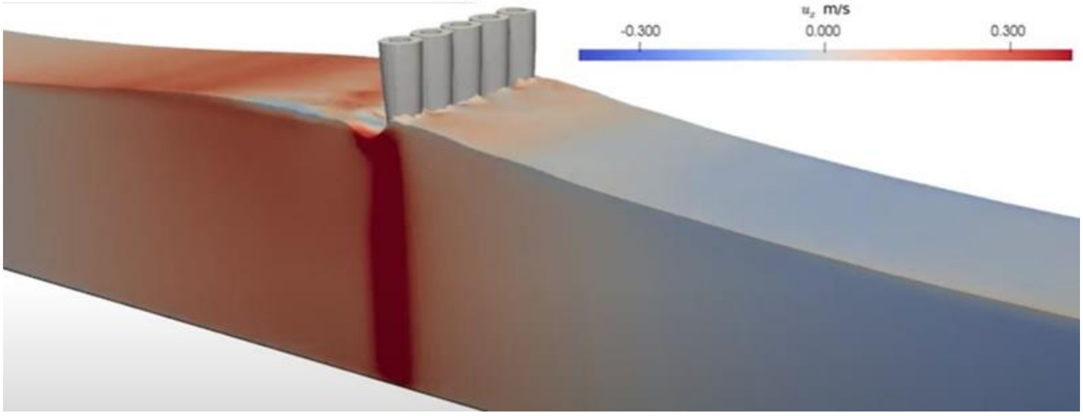

Figure 2. Wave interaction with the CPHB.

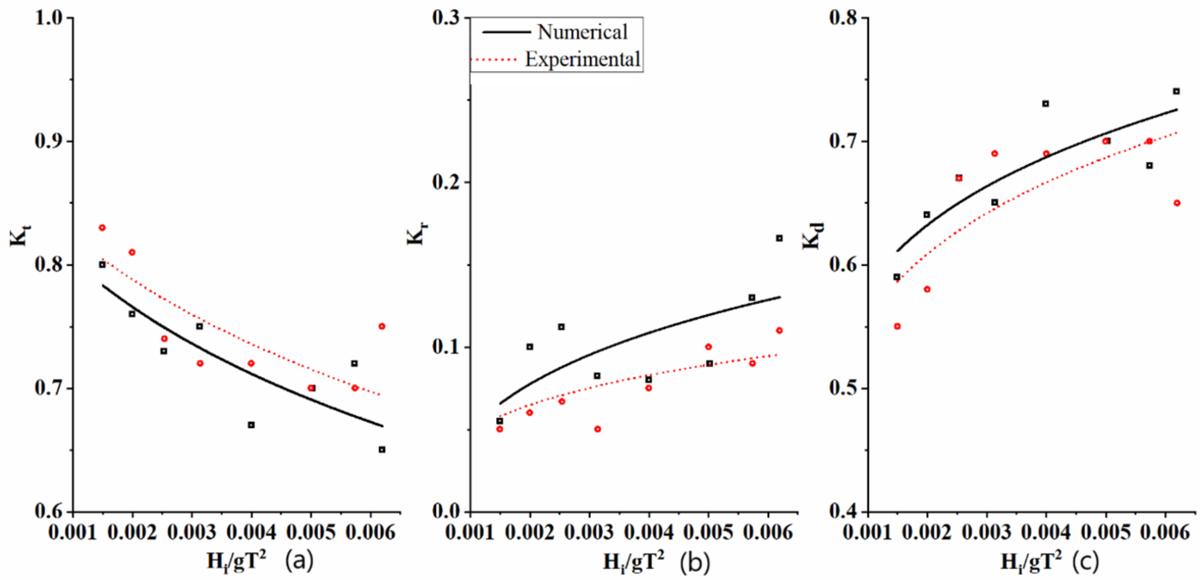

Figure 3. Comparison of numerical and physical modeling results for $D/H_{max} = 0.4$, $Y/H_{max} = 1.0$, $b/D = 0.1$ and $h = 0.40$ m

4 Conclusions

REEF3D has proven to be a validated CFD tool for modeling hydrodynamics, hydraulics, environmental engineering, and offshore, coastal, and marine CFD. The open-source tool is already being used by a large community of researchers. The tool is continuously undergoing upgradation to accommodate more and more complex CFD problems. There are various features of the software that are yet to be developed. This is slowly being achieved by the increasing number of researchers being ready to use the tool to run simulations for various CFD cases. The main motivation behind increasing the user base of REEF3D is the cost it saves by being a free tool and the flexibility the tool offers in terms of the user having the freedom to access and modify the tool to suit their need. These modifications keep adding new features to the tool. Presently the tool is majorly used in marine research but is highly scalable to be used in compressible fluid modelling. Furthermore, REEF3D has a dedicated team that can work with any researcher wanting to upgrade the software for a specific use and isn't able to do so individually thus adding to the reliability of the open-source CFD software.

Acknowledgments

The authors would like to express their gratitude to Prof. Hans Bihs and Dr. Arun Kamath, NTNU, Norway for developing and upgrading the open-source software REEF3D. The authors acknowledge the computational resources provided by the Centre for System Design at NITK led by Prof. Gangadharan K V.

References

[1] Peiro J, Sherwin S. Finite difference, finite element and finite volume methods for partial differential equations. Handbook of materials modeling. 2415-46

[2] Pankova EO, Spalding DB. Introduction to CFD. HEDH Multimed 2017:1–21. https://doi.org/https://doi.org/10.1615/hedhme.a.000102.

[3] P. T. Computational Fluid Dynamics Notes: The Introduction. SSRN Electron J 2022:1–7. https://doi.org/https://doi.org/10.2139/ssrn.3959220.

[4] Bihs H, Kamath A, Alagan Chella M, Aggarwal A, Arntsen ØA. A New Level Set Numerical Wave Tank with Improved Density Interpolation for Complex Wave Hydrodynamics. Comput Fluids 2016;140:191–208. https://doi.org/10.1016/j.compfluid.2016.09.012.

[5] Rao N, Suryanarayana Barimar Rao P, Nayak K, Kishor Pal S, Hunasanahally Sathyanarayana A, Suvarna P, et al. Numerical investigation on wave transmission characteristics of perforated and non-perforated pile breakwater. J Phys Conf Ser 2019;1276. https://doi.org/10.1088/1742-6596/1276/1/012021.

[6] Sathyanarayana AH, Suvarna PS, Umesh P, Shirlal KG, Bihs H, Kamath A. Numerical Modelling of an Innovative Conical Pile Head Breakwater. Water (Switzerland) 2022;14. https://doi.org/10.3390/w14244087.

[7] Sathyanarayana AH, Suvarna PS, Umesh P, Shirlal KG. Performance characteristics of a conical pile head breakwater : An experimental study. Ocean Eng 2021;235:1–16. https://doi.org/10.1016/j.oceaneng.2021.109395.

[8] Chorin AJ. Numerical Solution of the Navier-Stokes. Math Comput 1968;22:745–762.

[9] H. Van Der Vorst. BiCGStab: A fast and smoothly converging variant of Bi-CG for the solution of nonsymmetric linear systems. SIAM J Sci Stat Comput 1992;13:631–44.

[10] Jiang GS, Shu CW. Efficient implementation of weighted ENO schemes. J Comput Phys 1996;126:202–28. https://doi.org/10.1006/jcph.1996.0130.

[11] Shu C-W, Osher S. Efficient Implementation of Essentially Non-oscillatory Shock-Capturing Schemes. J Comput Phys 1988;77:439–71.

[12] Brackbill JU, Kothe DB, Zemach C. A continuum method for modeling surface tension. J Comput Phys 1992;100:335–54. https://doi.org/10.1016/0021-9991(92)90240-Y.

[13] Wilcox DC. Turbulence Modelling for CFD. La Canada, California: 1994.

[14] Osher S, Sethian JA. Fronts propagating with curvature-dependent speed: Algorithms based on Hamilton-Jacobi formulations. J Comput Phys 1988;79:12–49. https://doi.org/10.1016/0021-9991(88)90002-2.

[15] Peng D, Merriman B, Osher S, Zhao H, Kang M. A PDE-Based Fast Local Level Set Method. J Comput Phys 1999;155:410–38. https://doi.org/10.1006/jcph.1999.6345.