REEF3D: an Open Source CFD Tool

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

October 26, 2023

REEF3D: An Open Source CFD Tool

Naveen Rao[1*], Arunakumar Hunasanahally Sathyanarayana[2], Pruthviraj U[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] National Institute of Technology Karnataka, Department of Water Resources and Ocean Engineering, Surathkal, India

[4] National Institute of Technology Karnataka, Department of Mechanical Engineering, Surathkal, India

* nrao19961@gmail.com

ABSTRACT

Computational Fluid Dynamics (CFD) to be simply put in layman's terms, defines the interaction of fluids with any obstacle. These interactions can be studied through experimentation and simulations. Although experimentation has been the traditional approach to study fluid flow and interactions, the way CFD is explored has undergone a massive shift with the advancements of simulation programs. For water and coastal modeling, 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 benefits of working with an open-source tool come from 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 itself as needed. This increases the capacity of the tool to be upgraded continuously. 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 major impact on the field of CFD.

Keywords: CFD; Numerical modelling; REEF3D; Open-source.

1 Introduction

Revisiting the lineage of modern-day CFD, it all started with digital computers in the 1950s [1][2]. While digital computers made it possible to use Finite Difference Method (FDM) and Finite Element Method (FEM) to solve the partial differential equation of fluid flow and CFD in particular. FDM and FEM 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 when Richardson presented a paper on FDM solution for stress analysis of a masonry dam.[2] In contrast, the first FEM work in the field of CFD was published in the Aeronautical Science Journal by Turner et. al for applications in aircraft stress analysis in 1956.[2] The application of FDM dominated the field of CFD in the initial days. FEM was found to be complex for applications in CFD. With digital computers getting better over the days FEMs are providing superior performance in the simulations.

Over the years, numerous tools have been developed to run CFD models and have been marketed to researchers. Most of these tools are developed as a product that can give monetary returns. While on the one hand, we have a research community that is constantly working on CFD simulation and is brimming with concepts. These concepts can be actualized by repeated updation of the modeling tools. However, the lack of access to the R&D of modeling tools sold as products hinders their pace to keep up with the problem statements that the research community would want to work on. Developing an open-source tool instead removes this obstacle and the whole research community in any place, at any time can work on modifying and updating the tool to simulate more complex numerical models and work on a wide range of problems.

REEF3D is a result of Professor 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 [5]. This in turn led to the creation of REEF3D, a new numerical model that has an integrated interface-capturing algorithm [5].

In the further sections of this paper, REEF3D's application in hydrodynamic modeling is explored through a case study of wave interactions with pile breakwaters.

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 [6].

$$\frac{\partial \mathrm{Ui}}{\partial \mathrm{xi}} = 0 \tag{1}$$

$$\frac{\partial \mathrm{Ui}}{\partial \mathrm{t}} + \mathrm{Uj}\frac{\partial \mathrm{Ui}}{\partial \mathrm{Xj}} = -\frac{1}{\rho}\frac{\partial \rho}{\partial \mathrm{Xi}} + \frac{\partial}{\partial \mathrm{Xj}}\left[\mathrm{v}\left(\frac{\partial \mathrm{Ui}}{\partial \mathrm{t}} + \frac{\partial \mathrm{Ui}}{\partial \mathrm{Xi}}\right) - \overline{u_\iota u_J}\right] + \mathrm{gi} \tag{2}$$

Where u is time-averaged velocity over time t, ρ is the density of the fluid, p is the pressure, ν is the kinematic viscosity, u is the velocity fluctuation over time, $\overline{u_\iota u_J}$and g is the acceleration due to gravity. Pressure is determined by the projection method, and the consequent Poisson equation is solved by BiGCGStab solver. The K–ω model is adopted for turbulence modelling. [4][6]

REEF3D has four parts to it. There is file called as ctrl.txt. used for instructing the program with instruction codes defined in the user guide a DIVEMesh file to be compiles and MPI libraries along with the graphic interface that gives an output of the simulation.

A ctrl.txt file is created by defining the boundary conditions of a wave flume and obstacles are also defined in the same file. Rao. N et. al [7] considers the efficiency of porous pile breakwater as opposed to traditional solid pile break waters. The numerical model validation was carried out by comparing the modeling result with laboratory results performed on a wave flume as seen in Figure 1. With consistent simulations producing results with mean average percentage error (MAPE) of below 5% the validation was considered successful and the new model to check wave attenuation with perforated pile breakwater of similar dimensions was carried out.

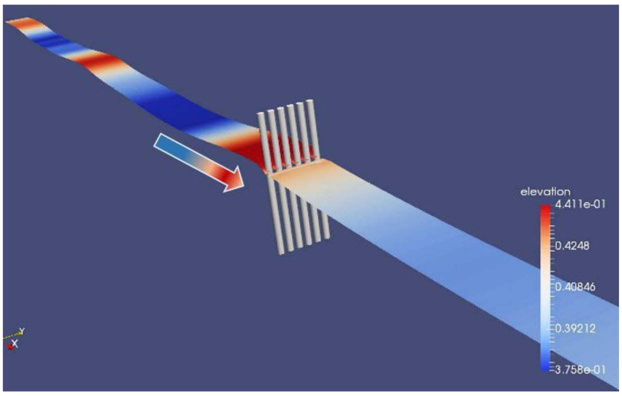

Figure 1. A numerical model of a single row of non-perforated pile breakwater. [7]

3 Results and Discussion

3.1 Simulations

AutoCAD model of the piles was imported to REEF3D. Three pores per horizontal layer are modeled with a vertical clear spacing of 0.5D provided between the perforations. Piles with a pore diameter of 0.15D, 0.20D, and 0.25D are provided which give porosity of 2.65%, 4.35%, and 6.25% respectively. At any given cross-section of the pile along the perforations, three numbers of circular perforations are provided at right angles to the direction of the wave attack. The simulations are conducted for different test conditions by varying different parameters viz. diameter of pores (Dp), water depth (d), wave height (Hi), and time-period of waves (T) as described in Table 1. [7]

Table 1. Experiential conditions for a single row of perforated pile breakwater used in the case [7]

Parameters	Water depth (d) = 0.4m	Water depth (d) = 0.5m
Diameter of piles (D)	0.0335m	0.0335m
Clear spacing between piles (b)	0.5D	0.5D
Pore diameter (Dp)	0.15D, 0.20D and 0.25D	0.15D, 0.20D and 0.25D
Incident wave height (Hi)	0.08m to 0.16m	0.08m to 0.16m
Wave time period (T)	1.5sec and 2sec	1.5sec and 2sec

3.2 Analysis

On deriving the result of the simulations the results were 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 pile. The results of the numerical model were concurrent with the results of the experiment by Rao et al. [8]

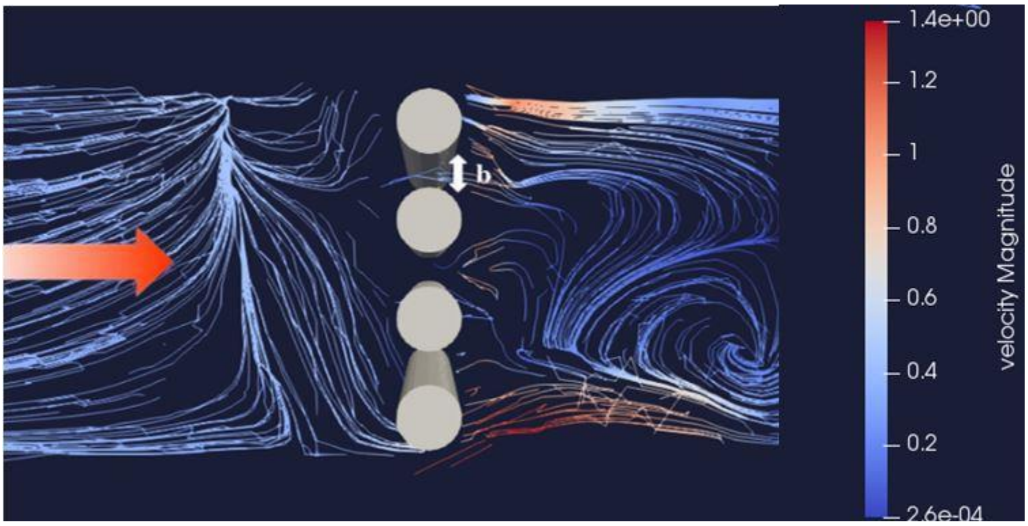

Figure 2. Streamline tracing for pile breakwater simulation as seen on REEF3D's graphic interface [7]

All the data of the simulation can be accessed as numbers stored in the .txt files in the REEF3D folder. REEF3D graphic interface tools further provide various options for inferring the results. The simulation can be visualized to see how the water waves interact with the piles and the flume in the numerical model. Streamline tracing as seen in Figure 2 can be visualized with varying scales.

The numerical model using REEF3D produced the desired results within the margin of error and was proven to be useful in running further complicated models. The same model can be further optimized by changing the arrangement of the perforations or the diameter of the perforations to produce better wave attenuations. This would be a very tedious task to perform physically in a wave flume.

4 Conclusions

REEF3D has proven to be a validated CFD tool for modeling hydrodynamics, hydraulics, environmental engineering, and offshore, coastal, and marine CFD. [6][7]. 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 team would like to express its gratitude to Prof. Hans Bihs and Dr. Arun Kamth, NTNU Norway for developing and upgrading the open-source software. The team that is monitoring the development of the software, Centre for System Design at NITK led by Dr. K. V. Gangadharan and Dr. Pruthviraj U. Special thanks to Dr. Pruthviraj U for providing the team with access to REEF3D and NTNU research team under Prof. Hans Bihs and Dr. Arun Kamath.

References

[1] Pankova, E. O., & Spalding, D. B. (2017). Introduction to CFD. *HEDH Multimedia*, 1–21. https://doi.org/10.1615/hedhme.a.000102

[2] Tanjung, P. (2022). Computational Fluid Dynamics Notes: The Introduction. *SSRN Electronic Journal*, 1–7. https://doi.org/10.2139/ssrn.3959220

[3] W, V. H. K. M. (1995). *CFD 1era Edimalalasekhara*.

[4] Guide, U. (n.d.). *REEF3D :: User Guide*, 2012.

[5] Hans Bihs, Three-Dimensional Numerical Modeling of Local Scouring in Open Channel Flow

[6] Sathyanarayana, A. H., Suvarna, P. S., Umesh, P., Shirlal, K. G., Bihs, H., & Kamath, A. (2022). Numerical Modelling of an Innovative Conical Pile Head Breakwater. *Water (Switzerland)*, *14*(24), 1–21. https://doi.org/10.3390/w14244087

[7] Rao, N., Suryanarayana Barimar Rao, P., Nayak, K., Kishor Pal, S., Hunasanahally Sathyanarayana, A., Suvarna, P., & Umesh, P. (2019). Numerical investigation on wave transmission characteristics of perforated and non-perforated pile breakwater. *Journal of Physics: Conference Series*, *1276*(1). https://doi.org/10.1088/1742-6596/1276/1/012021

[8] Rao and Rao N B S 1999 Laboratory Investigation on Wave Reflection Characteristics ofSuspended Perforated Pipe Breakwater *ISH J. Hydraul. Eng.* 5 22–32