

# Numerical Study of Water Wave Striking Different Shapes with the Help of OpenFOAM

Akhil Joshi<sup>1</sup>, Ahmed Khatib<sup>2</sup>, Dr. Sachin Borse<sup>3</sup>

P.G. Student, Department of Mechanical Engineering, Rajarshi Shahu College of Engineering, Pune, Maharashtra, India<sup>1</sup>

Software Engineer, VJ Coresoft Limited, Pune, Maharashtra India<sup>2</sup>

Professor, Department of Mechanical Engineering, Rajarshi Shahu College of Engineering, Pune, Maharashtra, India<sup>3</sup>

**ABSTRACT:** This study investigates effect of shape and size of wall facing wave, created by collapsing water column. Here VoF fluid method is used using interFoam module of OpenFoam software. Different leading wall shapes considered are converging, flat and rounded. Three different wall thicknesses were considered attached to these shapes. Effect of water wave is dominant on flat leading wall as compared to converging and rounded leading wall. Converging shape found better to dissipate impact of wave.

**KEYWORDS:** water wave; CFD; OpenFoam; VoF; interFoam.

## I. INTRODUCTION

Computational Fluid Dynamics popularly known by the acronym CFD is a numerical tool used for analysis of systems involving all types of flow like fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer based numerical simulation techniques. The physical laws governing a fluid flow problem are represented by a system of partial differential equations, the Navier-Stokes equation, Continuity equation and any other conservation equations. Some of the available software package tools are AnsysFluent, Ansys CFX, OpenFOAM etc.

Man has been building structures in water bodies for ages. Dams, bunds, bridges, embankments, breakwater structures, waterbarriers, piers etc. are some examples. The effect of behavior of water on the design of these structures makes for an interesting study. The dissipation of energy stored in the flowing water is an important aspect of these structures. Experimental analysis for problems like these is complicated and expensive procedure. Application of computational fluid dynamics on the other hand provides easy solutions for a wide range of data sets.

In the study and design of coastal structures and marine engineering applications, it is imperative to get the behaviour of water wave right. At the same time, the impact of water becomes doubly important for further analyses. In such a scenario, the use of computational fluid dynamics (CFD) provides an advantage over traditional models. With the escalating needs of speed and accuracy at the same time changing nature of real life problems in marine engineering, CFD solutions have gained importance.

Morgan et al. have used the rasinterFoam for verifying the experimental data of Beji (1993) of propagation of monochromatic waves over sand bar. The results of Beji were further replicated on a double scale by Luth (1997). The highlight of Morgan et al. CFD model is creation of the computational domain for simulation of a three dimensional water wave.

The current study uses the interFoam module from OpenFOAM software involving volume of fluid method (VoF). This two phase problem with water and air as phases. Here water wave is created by collapsing water column. Height of water column indicates total head in water.

## II. METHODOLOGY

For the computational purpose, domain considered here is 3D. Meshing is done using OpenFoam mesh blockMesh. Different boundaries and boundary conditions are shown in Fig. 1. Here water column is collapsed to generate the wave impacting on the wall surface. Entire geometry is generated in openFOAM using multiple blocks to form structured mesh. The left wall as shown in Fig. 1 represents the water column which is used for the simulation of wave. The water wave thus simulated strikes the right wall beginning with the leading surface. The entire domain is essentially a rectangular shaped box which has two phases (. water and air). Red column in domain shows initial water column before collapsing. On the right hand side divider wall of rounded leading shape is shown.

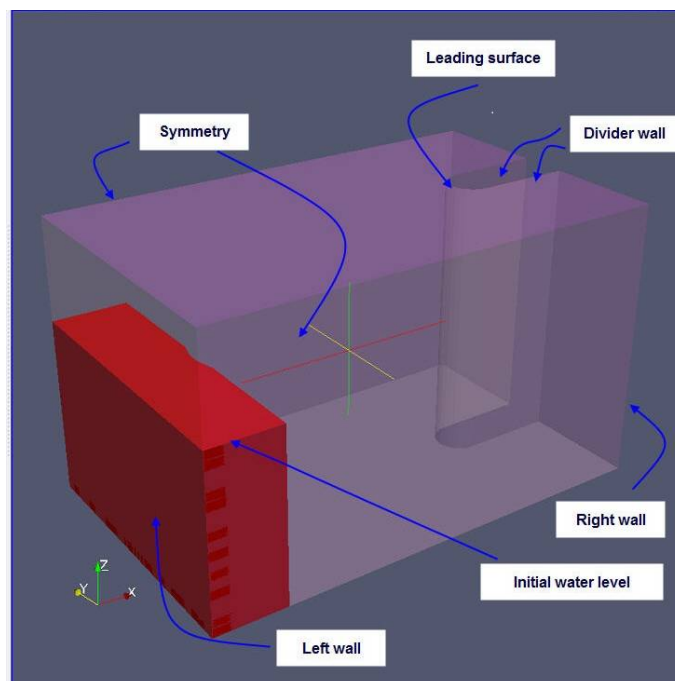


Figure 1. Computational domain

The wave was impacted upon different shapes of leading edge. The wave impact is studied for six different cases viz a flat surface, a round wall surface, and four different angles for a converging wall surface. In each case two widths of the extended surface i.e. 0.5 m and 1m are considered and calculations are carried out for them. Effectively twelve different wall geometries are considered here.

## III. NUMERICAL TEST

The wave when generated travels through the channel before impacting on the divider structure. The boundary conditions of the side wall are solid boundary. When the wave impacts the divider structure it rises up the structure. The successive impact on the right wall is of a wave which has been weakened by the divider wall and leading surface structure. The measure of the height up to which the water rises on the leading surface gives the extent of weakening of the wave. Figure 2 to figure 7 water phase in red colour with its height indicating total head.

**Case 1:** Leading edge Flat wall:

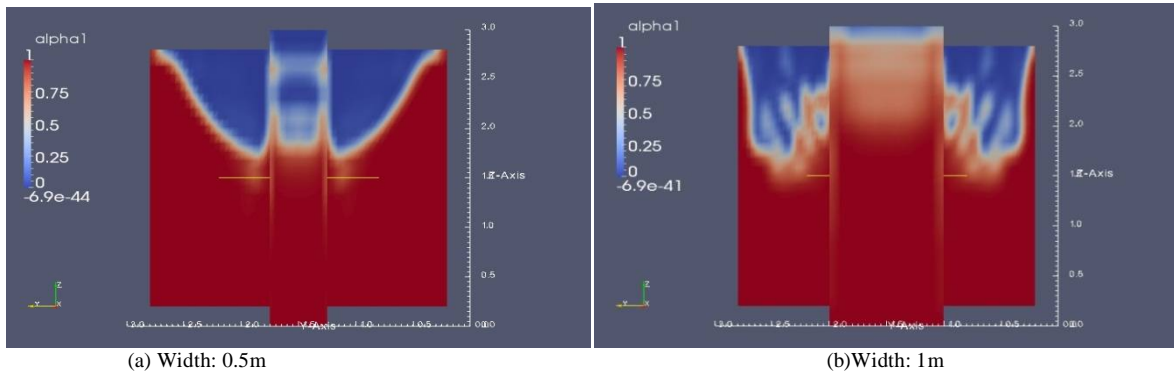


Figure 2 Wave impact on wall with leading edge flat wall.

**Case 2:** Leading edge Converging surface angle:30<sup>0</sup>

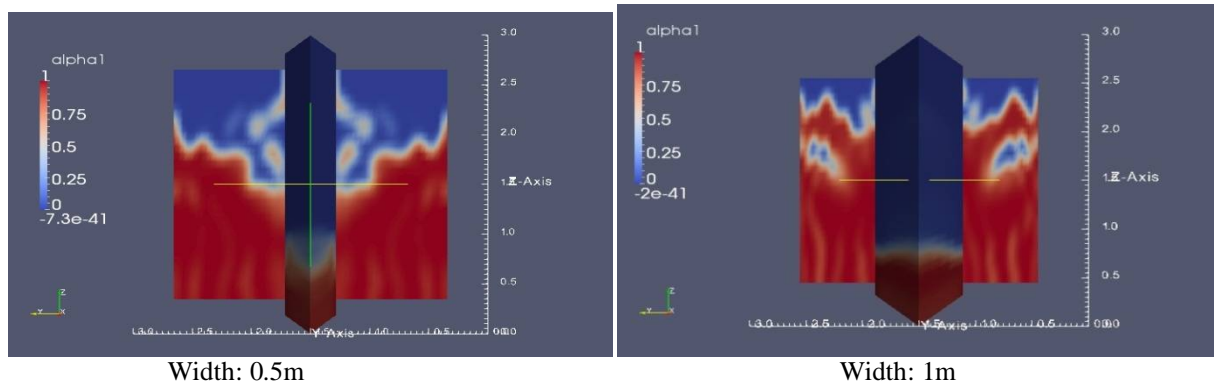


Figure 3 Wave impact on wall with leading edge 30° converging surface.

**Case 3:** Leading edge Converging surface angle:45<sup>0</sup>

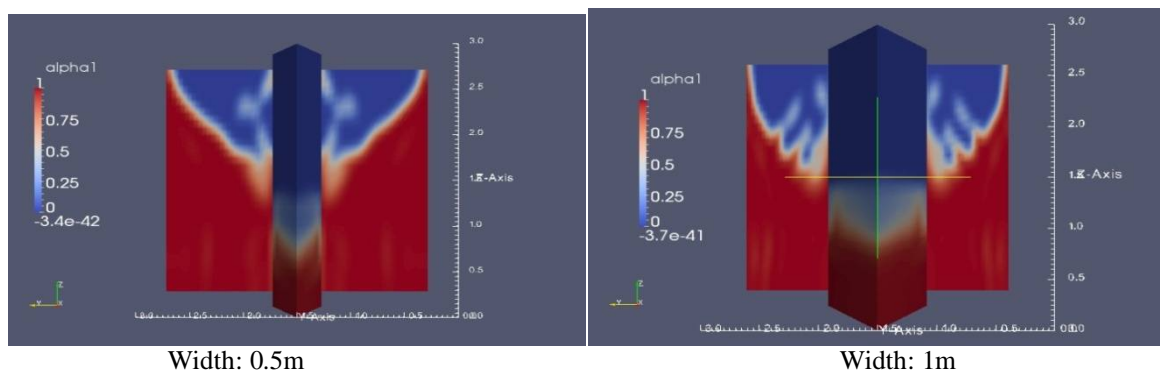
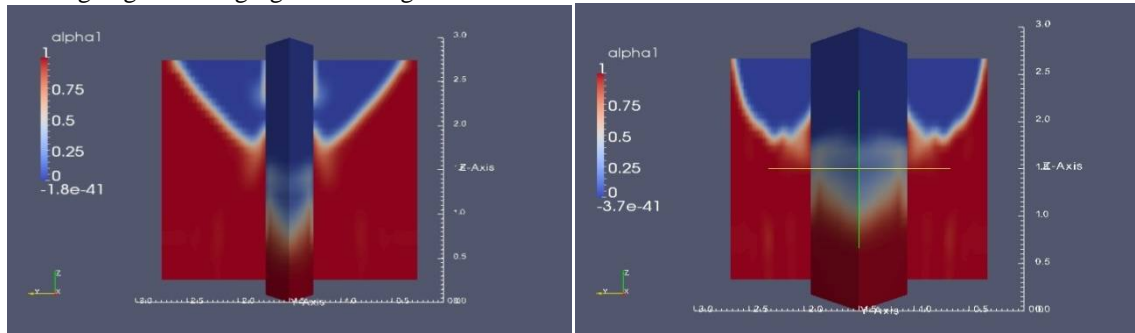


Figure 4 Wave impact on wall with leading edge 45° converging surface.

**Case 4:**

Leading edge Converging surface angle:60°

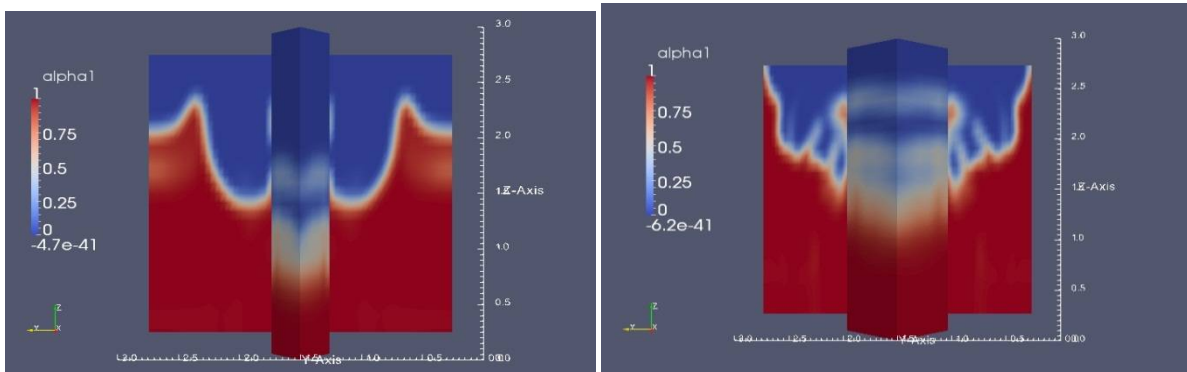


Width: 0.5m

Width: 1m

Figure 5 Wave impact on wall with leading edge 60° converging surface.

**Case 5:**Leading edge Converging surface angle:90°

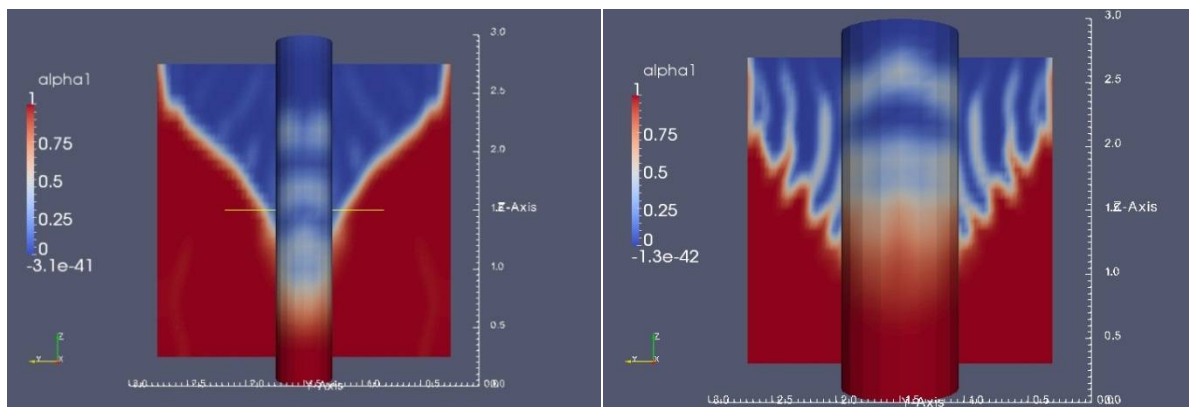


Width: 0.5m

Width: 1m

Figure 6 Wave impact on wall with leading edge 90° converging surface.

**Case 6:**Leading edge Rounded wall



Width: 0.5m

Width: 1m

Figure 7 Wave impact on wall with leading edge rounded wall.

**IV. RESULTS**

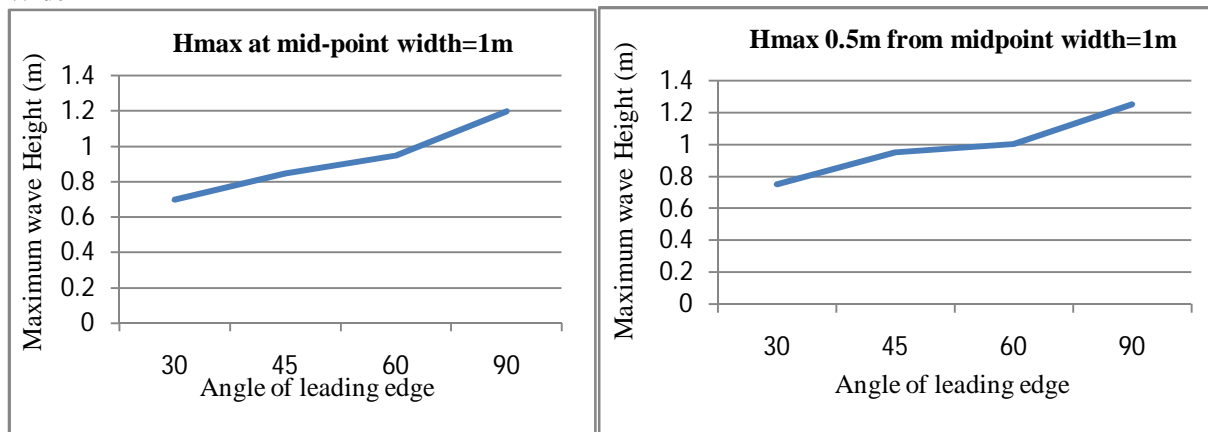
After running the simulation for each case result is collected by measuring height of water a certain data points. For the sub class of wall with width 0.5m the maximum height of water on the leading surface is measured at two points viz. one at the central line of leading surface and other at 0.25m from the central line. For wall with leading surface width of 1m, the measurement is similar to the previous sub case. Additionally height is measured at 0.5m from central line as well. Results are tabulated in Table 1. Figure 9 shows effect of angle of leading edge on water column height at different axial location on leading edge. Figure 10 shows effect on water column at leading wall at axial distance from centerline.

Table 1 Maximum height of watercolumn on leading surface

	Maximum height of wave in metres				
	Width=0.5m		Width=1m		
	Mid Point	0.25m from mid point	Mid Point	0.5m from mid point	0.25m from mid point
<b>Flat Edge</b>	<b>1.7</b>	<b>2.35</b>	<b>2.8</b>	<b>2.9</b>	<b>2.87</b>
<b>30</b>	<b>0.6</b>	<b>0.65</b>	<b>0.7</b>	<b>0.75</b>	<b>0.8</b>
<b>45</b>	<b>0.7</b>	<b>0.65</b>	<b>0.85</b>	<b>0.95</b>	<b>1.05</b>
<b>60</b>	<b>0.7</b>	<b>0.75</b>	<b>0.95</b>	<b>1.0</b>	<b>1.25</b>
<b>90</b>	<b>0.75</b>	<b>0.7</b>	<b>1.2</b>	<b>1.25</b>	<b>1.4</b>
<b>Round</b>	<b>0.65</b>	<b>0.75</b>	<b>1.55</b>	<b>1.75</b>	<b>1.3</b>

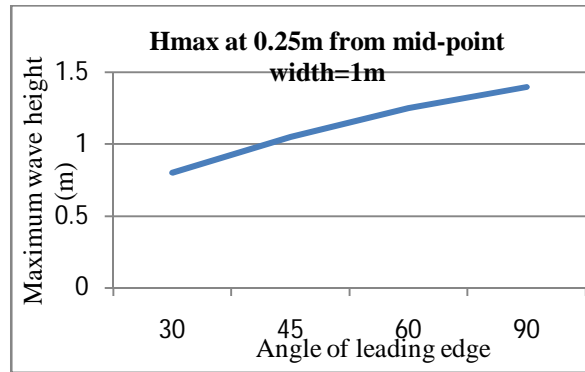
Graphs are plotted for the three heights for converging leading surfaces.

**Width=1m**



(a)

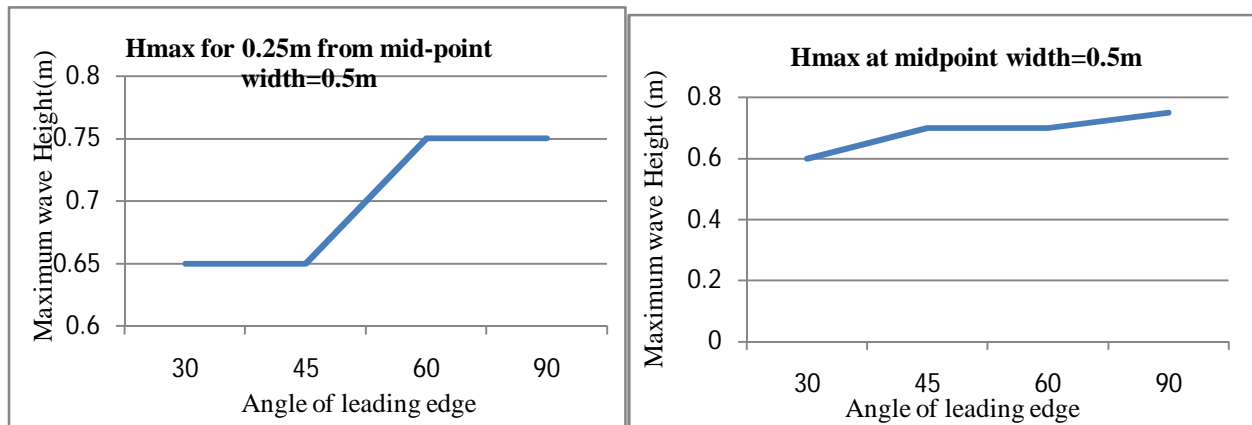
(b)



(c)

Figure 9 Graph of Maximum height versus angle of convergence of leading surface width 1m

Width=0.5m



(a)

(b)

Figure 10 Graph of Maximum height versus angle of convergence of leading surface width 0.5m .

## V. CONCLUSION

It is observed that the effect of the divider wall width is pronounced in each case. The water wave is disturbed by the wall. As regards to the shape of the divider, the maximum height reached by the water wave is inversely proportional to the angle between the outward normal from the surface and the direction of the water flow. Therefore the flat wall case shows maximum water height and hence more chances of damage to it. For the converging surfaces, the height increases with the angle of convergence. The rounded wall provides optimum water height and break of the waves among all the cases.

While it serves to have a wider dividing wall for having more chances of damage as have more height, a rounded leading surface trumps over a converging surface.

## REFERENCES

- [1] R. Luppés, B. Duz, H. J. L. van der Heiden, P. van der Plas and A. E. P. Veldman, "Numerical Solution of extreme wave impact on offshore platforms and coastal constructions", International Conference on computational methods in marine engineering, 2011.
- [2] G. C. Morgan, J. Zang, D. Greaves, A. Heath, C. D. Whitlow and J. R. Young, "Using rasinterFoam CFD model for wave transformation and coastal modeling.", 32<sup>nd</sup> Conference on Coastal Engineering, Shanghai, China, 2010
- [3] Beji, S. and J. A. Battjes. 1993. Experimental investigation of wave propagation over a bar, *Coastal Engineering*, 19, 151-162
- [4] Luth, H. R., G. Kloman, and N. Kitou. 1994. Kinematics of wave breaking partially on an offshore bar; LDV measurements of waves with and without a net onshore current, *Delft Hydraulics*.