Computational Analysis of Pipe Flow in a Numerical Wave Tank

VIT
Vellore Institute of Technology
(Deemed to be University under section 3 of UGC Act, 1956)
CHENNAI

ROBERT GORDON UNIVERSITY ABERDEEN

Manimaran Renganathan
VIT, India

Mamdud Hossain
RGU, UK

Introduction

Wind energy has gained commercial uses in many countries. However, wind energy alone is not enough to provide all the energy demands. Energy from sea waves promises to be a key renewable energy source in near future due to its abundant sources throughout the globe.

The aim of the present study is to apply the CFD technique to study the hydrodynamics efficiency of differential pressure driven flow through submerged pipe wave energy device. The simulation has been carried out using open source CFD software OpenFoam.

Methodology

OpenFoam has built in interFoam solver that solves wave surface elevation using Navier Stokes equation coupled with a volume of fluid method. Wave was generated using Cnoidal.

Continuity:

$$\nabla . v = 0$$

Momentum:

- $$\rho\left(\frac{\partial v}{\partial t} + v.\nabla v\right) = -\nabla P + \mu\nabla^2 v + \rho g$$

Volume fraction:

- $$\frac{\partial \alpha}{\partial t} + \nabla.(\alpha U) = 0$$

Results

The wave tank was 20m long, 0.65m wide and the water depth was 0.42 m. Numerical regular wave was generated in the simulation for the time period T=1.0 second and wave height of H=0.02m. Figure 1 shows the free surface elevation prediction against the experimental data.

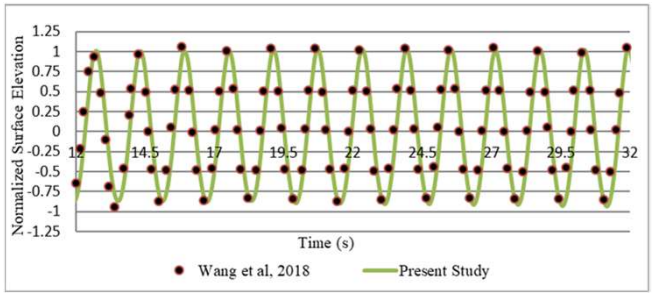

Figure 1: Normalized surface elevation history at 3 m downstream of the piston location

Figure 2 shows the contour plots of water volume fraction showing the generated wave and free surface elevation at the presence of pipe. Obviously, the presence of causes wave reflection and as a result the simulated wave height at the location of the centre of pipe (x=4.5 m from the tank boundary) is more than the imposed wave height of 0.02 m.

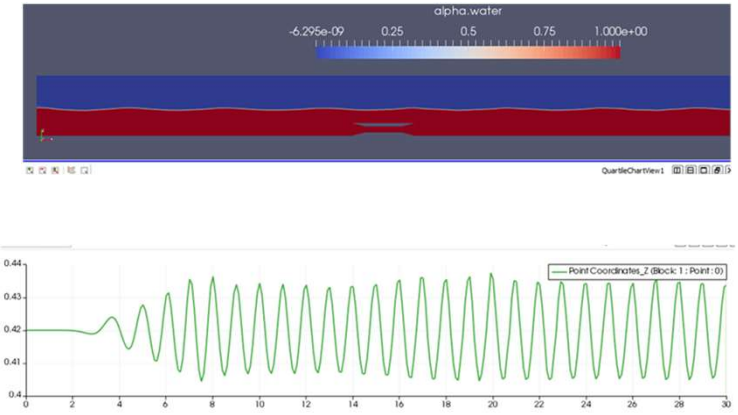

Figure 2: Contour plot of volume fraction of wave showing generated wave and free surface elevation in presence of pipe. Time period of wave, T=1.0 second and wave height, H=0.02 m.

Figure 3 shows the velocity vector in the tank with and without the presence of pipe. It is shown in the figure that the presence of pipe has large influence in the flow dynamics under the wave, then normal orbital flow is broken down and the fluid flows through the pipe.

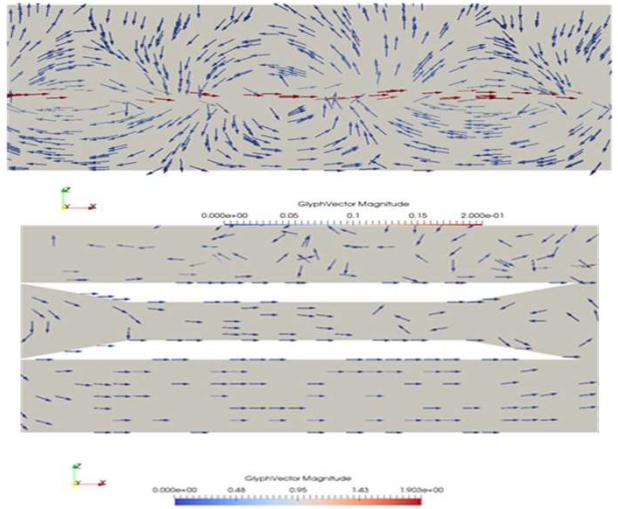

Figure 3: Vector plots showing (a) orbital flows in absence of pipe and (b) reversible flows inside the pipe. Time period of wave, T=1.0 second and wave height, H=0.02 m

Conclusions

A CFD modelling study of flow through a pipe under the influence of waves has been presented here. The simulation has been carried out using OpenFoam's interFoam submodel in order to prove a new concept of wave energy device. The simulated results show that the flow can take place within a submerged pipe.

Acknowledgment

The authors thank Global Challenges Research Fund for supporting visit by Hossain to VIT, India to setup the collaborative research