NUMERICAL SIMULATION OF WAVE RUN-UP OF A SUBMERSIBLE PLATFORM USING OPENFOAM

YANG LIN[1], LIAO KANG PING[2], MA QING WEI[3]
[1] College of Shipbuilding Engineering, Harbin Engineering University, China, yanglin205@163.com
[2] College of Shipbuilding Engineering, Harbin Engineering University, China, liaokangping@hrbeu.edu.cn
[3] School of Engineering and Mathematical Sciences, City University London, United Kingdom qw_ma2004@yahoo.co.uk

Keywords: *wave run-up, wave loads, openFOAM.*

Abstract:
A numerical simulation of wave run-ups and wave loads on a model scale semi-submersible offshore platform was conducted using interFoam solver combining the wave module in openFOAM 5.0.For interFoam simulation, a high-quality block-based structured grid is generated by ANSYS-ICEM 17.0 and then exported to openFOAM. Grids between the inlet and the platform are carefully refined to minimize numerical dissipations avoiding unphysical wave height reductions. A vertical damping technique is adopted for wave relaxation which applying an explicit damping force to components of the vector field in the direction of gravity. Wave elevation at different locations are monitored by interfaceHeight utility in openFOAM 5.0 and the pressure loads are probed at target points lies on the columns and under deck of the platform.

0.introduction

The wave run-up is a complex nonlinear phenomenon due to the interactions between marine structures and waves, currents, which is widely encountered in offshore engineering applications. This paper tries to validate the suitability of the incompressible multiphase flow solver interFoam in openFOAM 5.0[1] with the newly added wave module to simulate wave run-up phenomenon.

Two extensively concerned physical index of wave run-up are wave elevations at specific positions and wave impacting loads on marine structures. These numerical results are obtained by using interfaceHeight utility and probe utility in OpenFOAM 5.0 with a properly positioned virtual monitoring points and wave gauge locations as can be seen in Fig.1(a).

1.Geometries and numerical set up

This model scale submersible platform has dimensions of 4.58×0.35×0.36×0.32[m](Length×Width×Height×Draft), and simulated in the numerical computational domain with the size of 25×2×2[m] of which inlet is approximately one wave length upstream the fixed platform. The platform geometries and the computational domain are illustrated in Fig.1.

A block based structured grids (depicted in Fig.2) generated by ANSYS ICEM 17.0 instead of snappyHexMesh are utilized in this paper.The incident wave is set to be 5^{th}-stokes wave with wave length of 5.49 m and wave height of 0.4m. The length of wave relaxation zone is set to be 11m (approximately two wave length) in the outlet region to avoid reflections. The damping type and damping coefficient Lambda are set to be verticalDamping and 1.0, respectivrly.

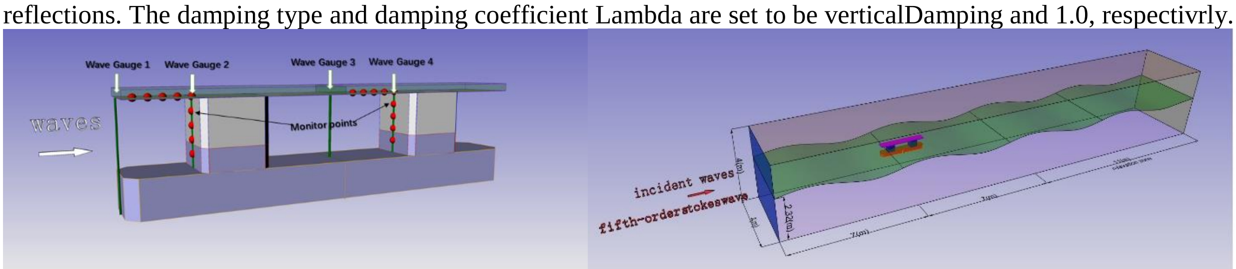


Fig.1 (a) Geometries and virtual monitoring points (b) computational domain

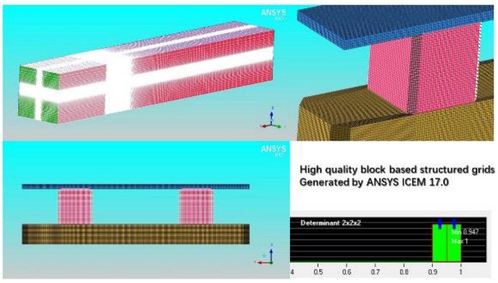


Fig.2 Computational mesh

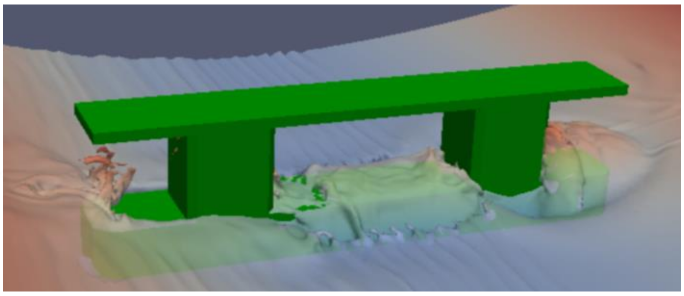


Fig.3 wave impact and breaking waves

2.Numerical cases and results

2.1 grid convergent study

The numerical diffusion origins from low order discretization scheme of the convection term and insufficient grid size in the vicinity of the free surface can cause severe wave height reduction which is a major shortcoming compared with traditional potential solvers. For the discretization of convection terms, the Gauss linearUpwindV is used for momentum equations and Gauss vanleer for VOF equation. Numerical cases with different grids in a wave length and a wave height are conducted to investigate the relation between wave height reduction and grid size. The elevation at the position one wave length away from the inlet are monitored and illustrated in Fig 4 and Fig5. For a grid with 100 grids in a wave length and 10 grids in a wave amplitude (20 grids in a wave height) it is donated as X100A10 in this paper. Since the main purpose of this part is to exam the grid size’s effect on wave height reduction, all cases are running without the submersible platform and the left/right patches are set to empty boundary condition to run the interFoam solver in 2D mode.

As can be seen from Fig.4 and Fig.5, given 100 grids in every wave length, refining the mesh in a wave amplitude from 10 to 40 brings little change to the wave height reduction and similar results are observed in refining the mesh in wave length direction from 100 to 300 with 40 grids in every wave amplitude, both of which all suffers severer wave height reduction. Specifically, the observed wave height for X100A40 and X300A40 are 0.245 (m) and 0.279(m), respectively, corresponding to wave height reductions of 38.75% and 30.28% since the initialized wave height is 0.4 (m). These results are actually unexpected and the reasons remained unclear since the wave module in openFOAM 5.0 is quite new. A screen shot of the wave profile at the end of the run time simulation of X100A10 is exhibited in Fig.6 showing that the wave is effectively relaxed with the explicit damping technique.

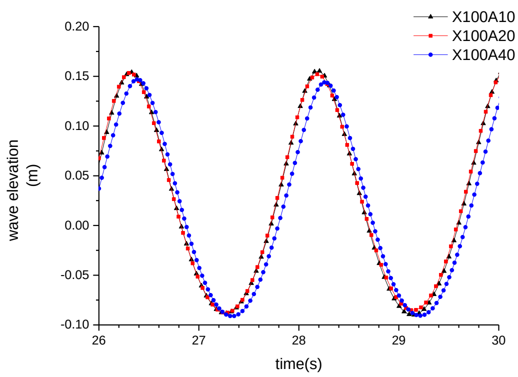


Fig.4 refining mesh in wave length direction

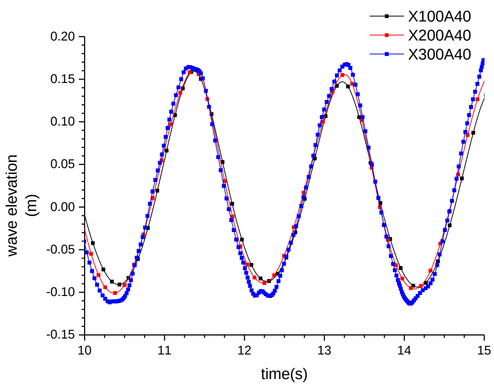
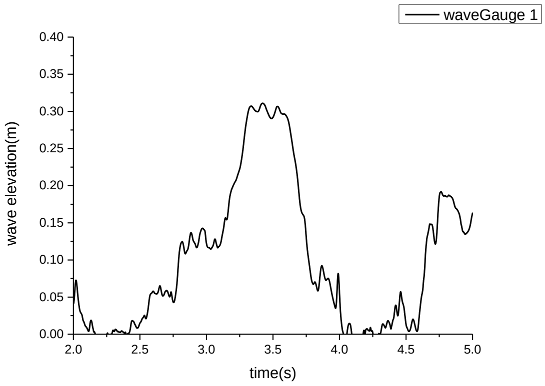


Fig.5 refining mesh in wave height direction

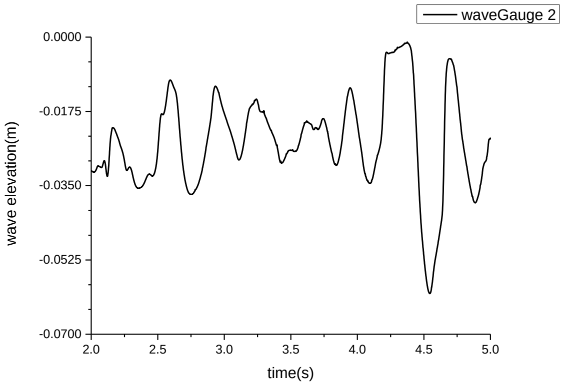
Fig.6 wave profile of X100A10

2.2 wave run-up results of openFOAM

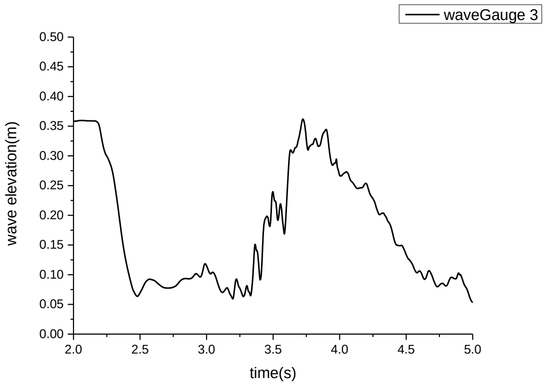
In this submitted abstract, a numerical case with approximately 5 million hexahedrons is conducted to provides some tender results which definitely need further improvements either in mesh refinements or algorithm justification. The wave run-up and wave breaking phenomenon are observed in Fig.3. The wave elevation above the platform bottom at 4 gauges can be seen in Fig.7.



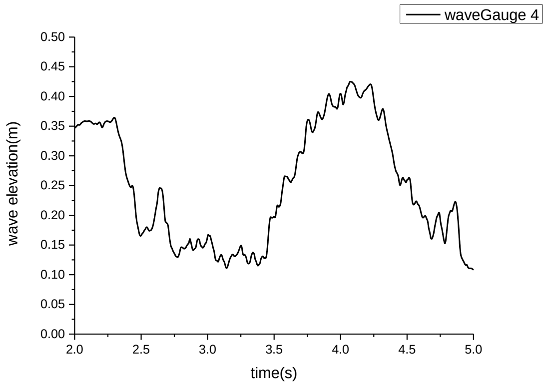
(a)



(b)



(c)



(d)

Fig.7 wave run up height above the bottom of the platform

Acknowledgements

The authors thank all those involved in the organisation of OFW13 and to all the contributors that will enrich this event.

References

[1] OpenCFD, OpenFOAM: The Open Source CFD Toolbox. User Gide Version 5.0, OpenCFD Limited. Reading UK, July. 2017.