

NUMERICAL SIMULATION OF AN OSCILLATING CONE AT THE WATER SURFACE USING COMPUTATIONAL FLUID DYNAMICS

Jan Westphalen¹, Deborah Greaves¹, Chris Williams², Kevin Drake³, Paul Taylor⁴

¹School of Engineering, University of Plymouth, Drake Circus, Plymouth, PL4 8AA, U.K.;

²Department of Architecture and Civil Engineering, University of Bath, Bath, BA2 7AY, U.K.;

³Department of Mechanical Engineering, University College London, London WC1E 7JE;

⁴University of Oxford, Department of Engineering Science, Parks Road, OX1 3PJ, Oxford, UK.

email: jan.westphalen@plymouth.ac.uk

Introduction

In coastal and offshore engineering fluid-structure interaction is of great interest. Realistic estimation of hydrodynamic forces and motions is essential in the safe and cost efficient design of wave energy converters and offshore structures. With recent developments in the mathematical models concerning the simulation of fluid flow and in the speed of computers, Computational Fluid Dynamics (CFD) has become an important part of the design process. CFD provides a large volume of results data and is capable of simulating complex structures and events (Jürgens *et al.* 2008). However, the highly complex case of fully nonlinear fluid-structure interaction with floating bodies is a challenging application and there are limited cases reported in the literature. Successful validations for focused waves and wave forces on fixed cylinders are described in Westphalen *et al.* (2008). In this work the problem considered is the fluid structure interaction resulting from the driven motion of an oscillating cone on the water surface. The motion of the cone is not influenced by the pressure and viscous forces from the water. The simulations involve mesh motion through deformation of the cells in the vertical direction as the cylinder oscillates vertically during the simulation.

Here, the fully nonlinear Navier-Stokes equations are solved using a control-volume Finite Element (CV-FE) approach on a mesh containing approximately 820,000 hexahedral cells. The results show very good agreement with physical experiments carried out by Drake *et al.* (2008).

Computational Domain

The simulations are performed in a three-dimensional domain with a length and width of 2.5m and a height of 2.0m. The cone is placed in the centre, as it can be seen in Figure 1. It has a top diameter of 0.6m and a steepness of the slope of 1:1. The slope itself is 0.3m high. The initial draught of the cone is 0.15m at a waterdepth of 1.0m.

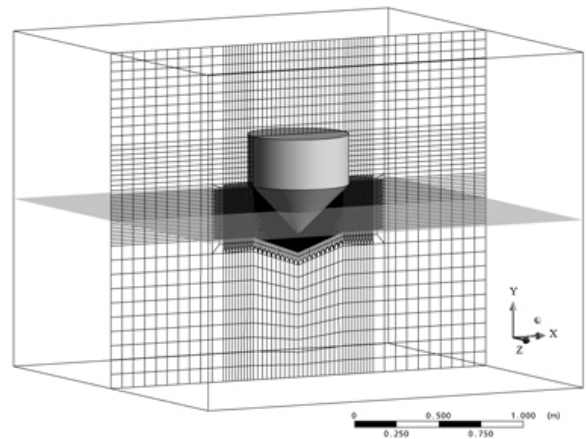


Figure 1: Computational domain

The cone is modelled as a cavity in the mesh. The outer boundaries, the bottom and the cone are modelled as free slip walls. The top boundary is defined as a pressure outlet with constant atmospheric pressure. The mesh consists of 820,000 hexahedral cells, where the regions around the water surface and the cone surface are highly refined to achieve cell edges of approximately 0.01m. The outer regions are relatively coarse to save computational resources

and encourage numerical damping, thus avoiding reflections from the walls.

The simulations were carried out using high performance computing on 16 CPUs. The timestep is 0.0005s. For one second of simulation time 28 hours of computing time on average was required.

Motion of the Cone

The motion of the cone is defined by the displacement $z(t)$ from the initial position at $t = 0$ s following the form of a Gaussian wave packet, which is described by

$$z(t) = A \sum_{n=1}^N Z(\omega_n) \cos[\omega_n(t-t_0) - h\pi/2] \Delta\omega_n, \quad (1)$$

where

$$Z(\omega_n) = \frac{1}{\frac{\omega_0}{2\pi} \sqrt{2\pi}} \exp\left[-(\omega_n - \omega_0)^2 / 2\left(\frac{\omega_0}{2\pi}\right)^2\right], \quad (2)$$

with $h = 0$ or 1 . A denotes the largest excursion from the still water level. N is the number of frequency components and ω_n is the appropriate circular frequency. The central circular frequency ω_0 [rad/s] is defined by

$$\omega_0 = m\pi/3$$

with m being an integer between 1 and 12.

The results presented in this paper are calculated with $h = 0$, $A = \pm 50$ mm and $m = 6$ and 9 . The frequency range, centred at ω_0 , is divided equally into $N = 50$ components.

Mathematical Model

A commercial CFD code is applied to solve the Navier- Stokes equations. These are discretised on a hexahedral mesh by means of a CV-FE approach, which combines the Finite Volume Method within a Finite Element framework. The equations are solved for both fluids, i.e. air and water, using a Volume of Fluid method. The interface between the fluids is captured sharply using the scheme developed by Barth and

Jespersion (1989) and described by Zwart (2005) and Zwart *et al.* (2003) to avoid smearing the surface across a large number of cells.

The cone is displaced and the mesh adapted at the beginning of each timestep depending on (1) and (2) by moving the nodes to deform the mesh without changing the connectivity. After the mesh is adapted the Navier–Stokes equations are solved for the displaced mesh in a fully coupled manner (Ansys, 2006).

Results

Simulations have been carried out in pairs in order to consider the positive direction cone displacement for a maximum excursion of $A = +50$ mm and the opposite negative displacement for $A = -50$ mm. By analysing the sum and difference of the wave elevation and forces for the paired tests, this allows the nonlinearity of the system to be considered. The surface elevations for the numerical calculations are extracted at the intersection of the cone surface and the water volume fraction of 0.5, which is generally accepted as representative of the water surface. The time and the surface elevations are non-dimensionalised by dividing by the period of the central frequency and A , respectively.

In Figure 2, the vertical forces for the downward maximum displacement case, $m = 9$, are shown. These are non-dimensionalised using the expression $F' = F/(\rho g \pi a^2 A)$. The measured force F is divided by the acceleration due to gravity g , the density of fresh water ρ , A and a , which is the radius of the cone at the waterline. Here, the numerically predicted fluid forces are in good agreement with the experiments. A small difference can be observed in the crests and troughs, especially for the extreme values.

To analyse the nonlinearity in the case, the time histories of the relative surface elevations for the paired tests, $m = 9$, have been subtracted and summed respectively and divided by 2. This enables results to be broken down into linear and higher order components and compared separately. The sum and difference elevations for $m = 9$ are plotted in Figure 3. For the central circular frequency corresponding to $m = 9$ the relative water surface elevation clearly contains a higher order component represented by the solid line.

Unlike the linear part the higher order component is not symmetric about the mean water line. It oscillates with double the frequency of the linear part around a slightly raised water level.

Applying the same analysis technique for the vertical forces, results in the plots shown in Figure 4. Here, the higher order parts, i.e. the sum terms, have a double frequency component superimposed on an asymmetric positive component.

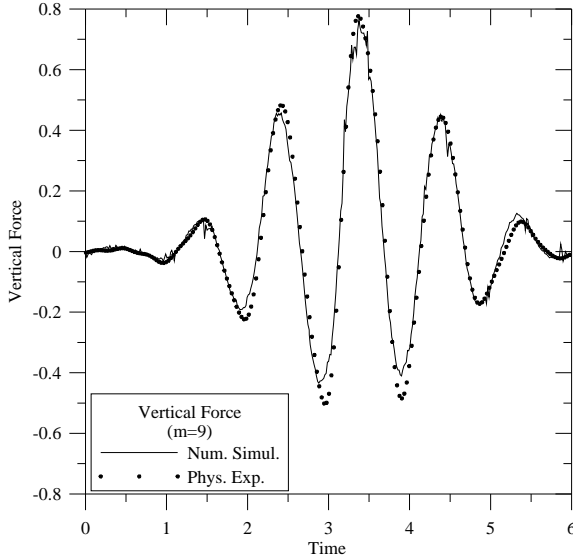


Figure 2 Non-dimensionalised vertical forces on cone: maximum excursion negative; Physical experiments by Drake *et al.* (2008)

The total force may be decomposed into its hydrostatic and hydrodynamic parts. The hydrostatic part results from the buoyancy force, which is subtracted from the total force to obtain the hydrodynamic contribution. Figure 5 shows the non-dimensionalised vertical forces for $m = 9$ for the maximum excursion negative case, decomposed into dynamic and hydrostatic components. For the lower frequency case, $m = 6$, the same plot is shown in Figure 6. Here, the hydrodynamic force is a much smaller component of the total force.

This is because, due to the higher central circular frequency for larger m , the Keulegan-Carpenter number, K_C , given by

$$K_C = \frac{AT_{\omega 0}}{0.3}, \quad (3)$$

reduces. A is the maximum excursion, $T_{\omega 0}$ the period corresponding to the central frequency and 0.3 is the diameter of the cone at still water. K_C describes the relationship between the drag forces over the inertia. For lower K_C the inertia dominates the force contribution. This can be seen in the results. For case $m = 9$, with $K_C = 0.11$, the dynamic force component, which is related to the inertia of the cone is more developed than for case $m = 6$, with $K_C = 0.16$.

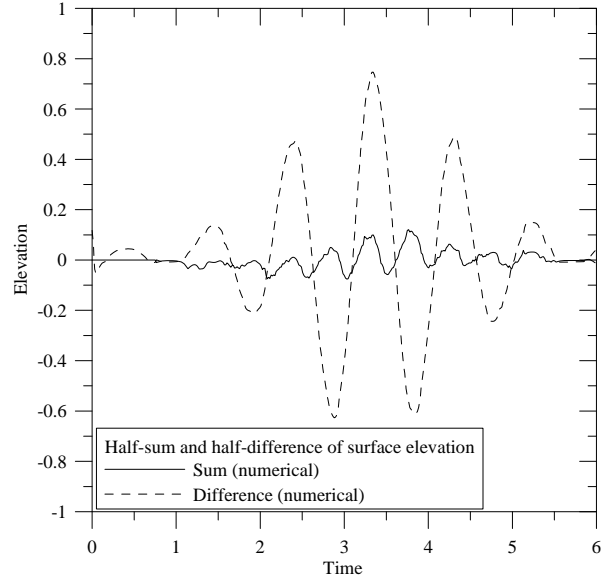


Figure 3 Surface elevation: half-sum and half-difference comparison ($m = 9$)

Conclusions

Preliminary CFD results for the study of nonlinear wave-structure interaction resulting from the forced motion of a cone near the water surface have been presented. The Navier-Stokes solver performed well within this challenging case involving multi-phase fluid flow with mesh motion and deformation. In this application the CFD code predicts well high order force components on a moving structure, which makes such packages very interesting for the design of e.g. offshore structures, where this knowledge is crucial for safe and economical solutions.

Further tests with different central circular frequencies for the Gaussian wave packet will be carried out to further investigate the behaviour of the applied CFD code.

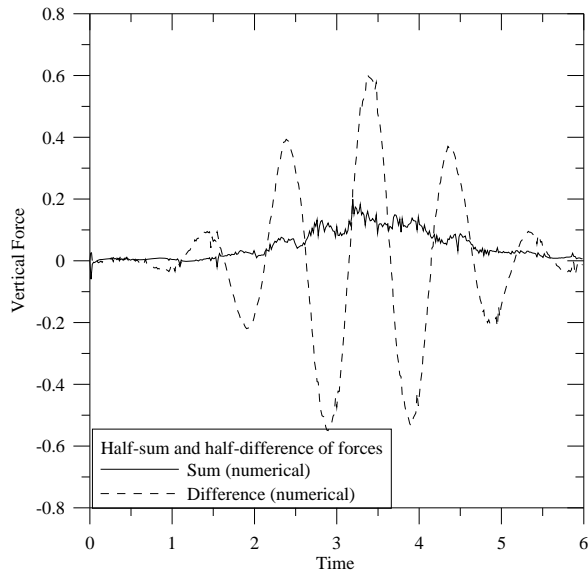


Figure 4 Non-dimensionalised vertical forces: Half-sum and half-difference comparison ($m = 9$)

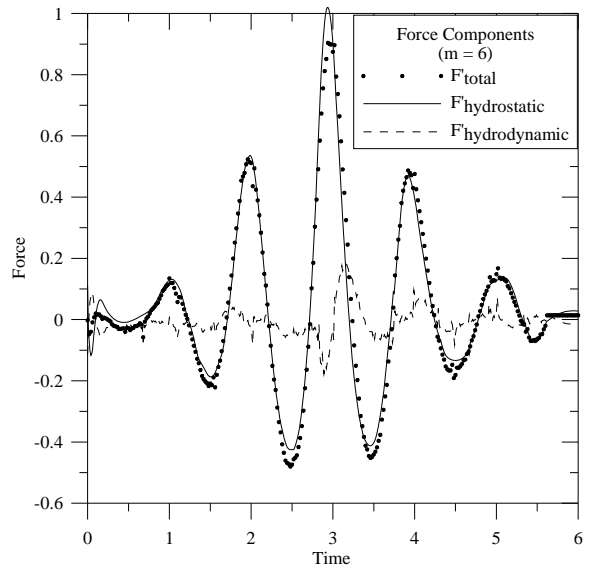


Figure 6 Non-dimensionalised force components for $m = 6$: maximum excursion negative

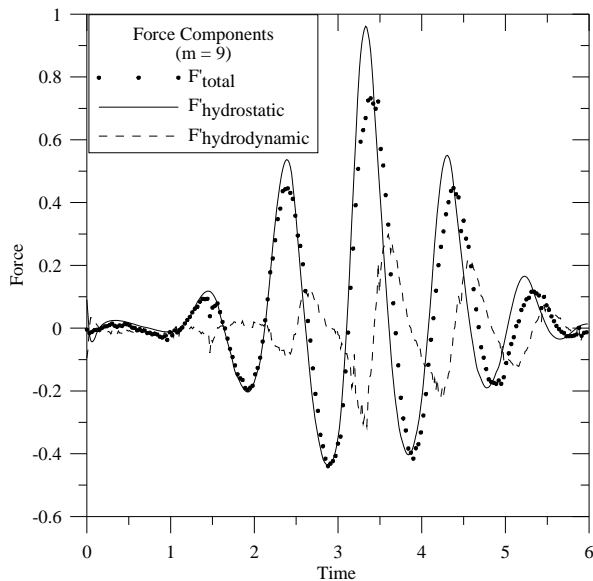


Figure 5 Non-dimensionalised force components for $m = 9$: maximum excursion negative

Acknowledgements

The authors would like to acknowledge the support of the Engineering and Physical Sciences Research Council.

References

- Ansys (2006), Ansys CFX-Solver Theory Guide, Ansys CFX Release 11.0, Ansys, Inc., 2006
- Barth, T. J., Jespersen, D. C. (1989), The design and application of upwind schemes on unstructured meshes, AIAA Paper 89-0366
- Drake, K., Eatock Taylor, R., Taylor, P. and Bai, W. (2008), On the hydrodynamics of bobbing cones, submitted for publication.
- Jürgens, D., Palm, M., Peric, M., Schreck, E. (2008), Prediction of Resistance of Floating Vessels, Proceedings of Marine CFD 2008, Southampton, 26.-27. March 2008
- Westphalen, J., Greaves, D., Williams, C., Zang, J., Taylor, P. H. (2008), Numerical Simulation of extreme free surface waves, Proceedings of the 18th International Offshore and Polar Engineering Conference Vancouver, Canada, 6-11 July, 2008
- Zwart, P. J. (2005), Numerical modelling of free surface flows, VKI Lecture Series
- Zwart, P. J., Scheuerer, M., Bogner, M. (2003), Free surface modelling of an impinging jet, ASTAR International Workshop on Advanced Numerical Methods for Multidimensional Simulation of two-phase Flow, Sep. 15.-16. 2003, GRS Garching, Germany