Large-eddy simulation of an internal ship air cavity

Timofey Mukha and Rickard E. Bensow
Chalmers University of Technology, Department of Mechanics and Maritime Sciences, Hörselgången 4, SE-412 96 Gothenburg, tel. 46704191850, timofey@chalmers.se

Introduction

Air lubrication is a promising technology potentially allowing for a 15-20% reduction in the drag of a ship hull through covering a part of the wetted surface with an air layer [1]. Depending on the hull type, different approaches to air lubrication are applicable. For large displacement hulls, the most attractive alternative appears to be introducing one or several air-filled cavities in the bottom of the hull. The challenge is designing the cavity in such a way that it remains stable and leaks a minimal amount of air.

Using computational fluid dynamics (CFD) to evaluate the performance of cavity designs has been attempted, but chiefly using potential flow solvers (see e.g. [1]). While this approach allows to predict certain properties of the wave system formed under the cavity, it is not adequate for capturing the mechanisms driving air leakage and thus fully characterizing the cavity's performance. Here, we present results from an implicit large-eddy simulation (LES) of an internal air cavity in model scale. A detailed characterization of the flow in the cavity's closure region is obtained allowing to identify the mechanism behind the air leakage that has previously not been reported in the context of air lubrication.

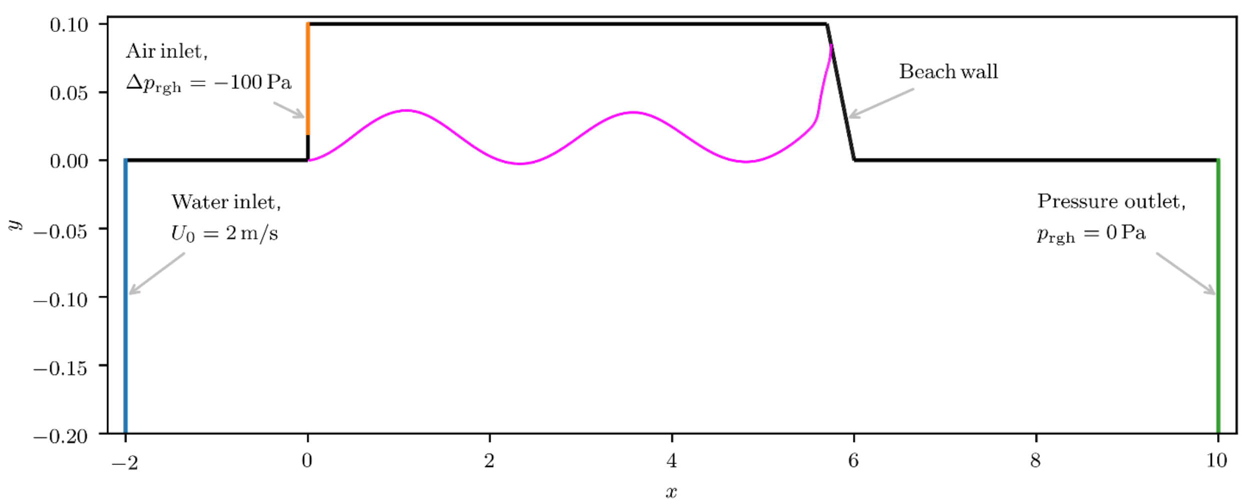

Figure 1. Side-view of the computational domain featuring the employed boundary conditions. The magenta line indicates the $\langle \alpha \rangle = 0.5$ isoline obtained in the simulation.

Case set-up and CFD methods

The simulation reproduces the experimental set-up used in [2]. A side-view of the computational domain is shown in Figure 1 along with the employed boundary conditions. The considered cavity is 6 m long and 0.1 m high. In the spanwise direction, the length of the domain is 0.25 m, and periodic boundary conditions are applied. A wave system is formed under the cavity, with the wave-length $\lambda \approx 2.56$ m. The flow's Froude and cavity pressure difference numbers are, respectively, $Fn \approx 0.26$ and $En \approx 0.05$.

To mesh the domain, the mesh generator *blockMesh*, found in OpenFOAM®, is employed. The mesh is structured apart from a refinement in the spanwise direction introduced at $x = 5.5$ m, which is necessary to better resolve the flow near the beach wall. The total number of cells is $\approx 11.5 \cdot 10^6$.

The Volume of Fluid-based multiphase solver *interFoam*, distributed as part of OpenFOAM®, is used to conduct the LES. In *interFoam*, the phase-incompressible Navier-Stokes equations are solved using the PIMPLE algorithm, and a transport equation for the phase volume fraction, α, is solved to capture the location of the interface. Further details can be found in [3] and [4].

Results

Upstream of the closure region, the flow is essentially two-dimensional and steady. By contrast, near the beach wall, it becomes three-dimensional, highly turbulent, and transient. As visualized in Figure 1, the presence of the wall results in the location of the air-water interface shifting upwards. An instantaneous snapshot of the interface is shown in Figure 2. A complicated three-dimensional structure is observed. Note that the flow direction at the interface is reversed with respect to the incoming wave. Packets of discharged air can be observed in the figure. The main mechanism for air entrainment appears to be the plunging of breaking waves that appear across the interface. Air packets get trapped below the waves and subsequently discharge into the water, where they get transported further downstream. This type of entrainment has previously been observed in [5], where the flow in a hydraulic jump is investigated.

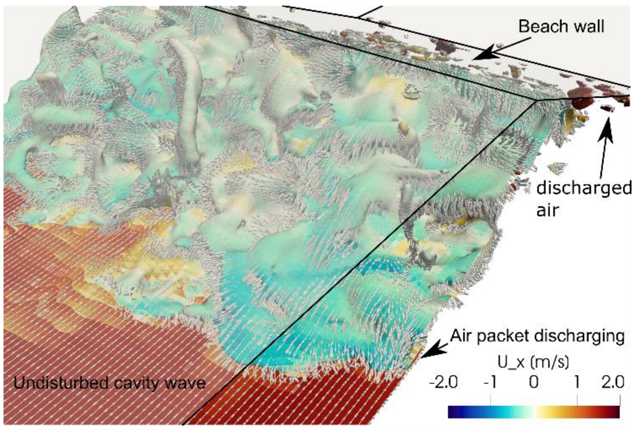

Figure 2. Snapshot of the $\alpha = 0.5$ isosurface colored by streamwise velocity magnitude. Vectors show the direction of the velocity field.

More insight in the flow's behavior can be gained by further investigating the direction of the mean flow, see Figure 3. It is observed that the flow separates and forms a recirculation region, which encompasses the air-water interface, and therefore drives the processes leading to air entrainment. It can consequently be expected that the flow in the closure region is sensitive to both the amplitude and the phase of the linear wave as it approaches the beach wall. This entails that the length of the cavity and the cavity air pressure can serve as effective control parameters.

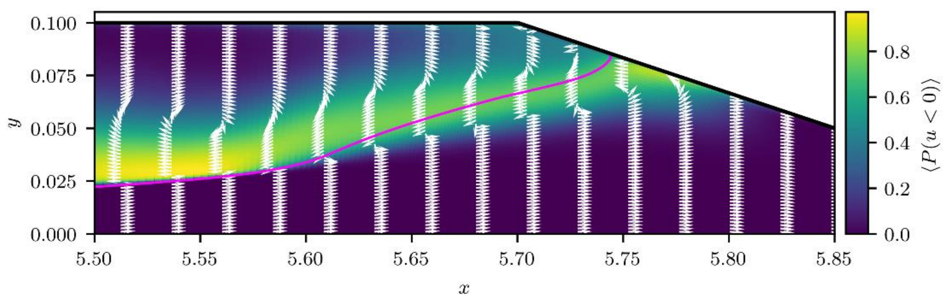

Figure 3. Distribution of the probability of backflow near the beach wall. White arrows show the direction of the mean velocity vectors. The magenta line indicates the $\langle\alpha\rangle = 0.5$ isoline.

It is interesting to see whether the air discharge process is periodic. To that end, the total amount of air present within a box located downstream of the recirculation zone has been sampled. Figure 4 shows the autocorrelation function (ACF) of this signal (blue line), which exhibits a clear peak at the frequency $f \approx 6.3$ Hz. Interestingly, the signal of the total force applied to the beach wall reveals periodicity at the same frequency (orange line in Figure 4). While this finding is perhaps not surprising, further analysis is needed to fully understand what happens at each stage of the entrainment cycle.

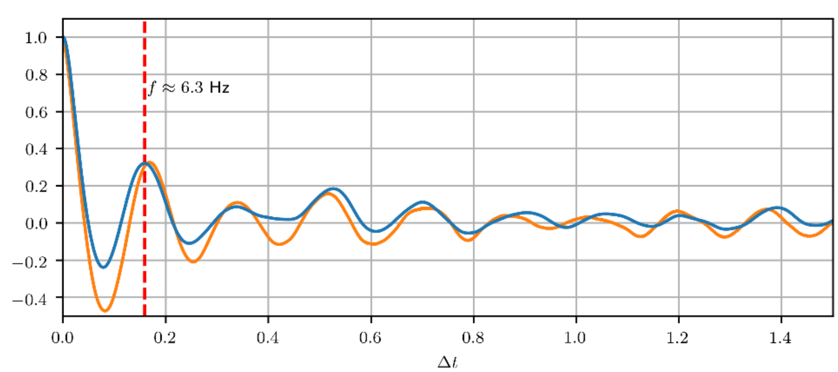

Figure 4. Blue line: The ACF of the total amount of air present in the box $x \in [5.8; 5.825], y \in [0.025; 0.1], z \in [0.0; 0.25]$. Orange line: The ACF of the total force on the beach wall.

Conclusions and future work

This work presents results from an implicit LES of an internal air ship cavity in model scale. To the best of the authors' knowledge, this is the first application of LES to this type of flow. Significant insight into the flow behavior in the closure region has been attained. In particular, an air entrainment mechanism via plunging of breaking waves has been identified. Furthermore, reference data for several flow quantities has been gathered, such as statistical moments of the velocity and pressure fields (not shown here for brevity).

Plans for future work include reproducing the conducted simulation on a denser grid, as well as investigating the flow behavior for different Fn, En, and beach wall angles. More advanced grid construction strategies, including adaptive mesh refinement, are also being considered. Another research direction is using the LES data to drive development of lower-fidelity models capable of accurately predicting the main characteristics of the flow.

References

[1] K. I. Matveev, "Three-dimensional wave patterns in long air cavities on a horizontal plane," *Ocean Engineering,* vol. 34, no. 13, pp. 1882-1891, 2007.

[2] A. Shiri, M. Leer-Andersen, R. E. Bensow and J. Norrby, "Hydrodynamics of a displacement air cavity ship," in *29th Symposium on Naval Hydrodynamics*, Gothenburg, Sweden, 2012.

[3] H. Rusche, "Computational fluid dynamics of dispersed two-phase flows at high phase fractions," PhD Thesis. Imperial College of Science, Technology and Medicine, 2002.

[4] S. Márquez Damián, "An extended mixture model for the simultaneous treatment of short and long scale interfaces," PhD thesis. Universidad Nacional del Litoral, 2013.

[5] M. Mortazavi, V. Le Chenadec, P. Moin and A. Mani, "Direct numerical simulation of a turbulent hydraulic jump: Turbulence statistics and air entrainment," *Journal of Fluid Mechanics,* vol. 797, pp. 60-94, 2016.