In Short

- Hydrophobic structured surfaces are able to retain air when submerged in water.
- When applied on ship hulls these surfaces can lead to drag reduction.
- To understand the influence of the structure size on drag reduction and turbulence flow, a high-resolution DNS simulation is carried out.
- The surface is modeled by slip/no-slip boundary conditions.

Organisms like the water moss *Salvinia molesta* and the backswimmer *Notonecta glauca* are able to retain air on their surface when submerged in water [1,2]. An analysis of their surfaces show hydrophobic hairlike structures. In addition, *S. molesta* features a small hydrophilic area at the tip of the hair structure to stabilize the water-air interface. Figure 1 shows the eggbeater like hair structure of *S. molesta*. Due to the combination of hydrophobicity and structure a superhydrophobic surface arises [1].

If such an air retaining surface is overflowed by water a reduction in drag in comparison to a smooth plate can be observed. Due to the lower viscosity of the air in comparison to water a drag reduction can be achieved [3]. This has been shown for laminar and turbulent flow as well as in experiment and numerical simulation in literature. Depending on the structure size, geometry or Reynolds number reductions of up to 69% are reported [4]. In these cases the geometry of the structure mostly consists of pillars or grooves. An example of pillars and air-water interface in laminar channel flow is given in Figure 2.

Technical application of this phenomenon are surfaces which are overflowed by water such as water tubes or ship hulls. In case of container ships the part of frictional drag can amount up to 60% [5]. A substantial reduction in drag would reduce the energy supply and the emission of greenhouse gases. The project AIRCOAT (Air Induced friction Reducing ship COATing), funded by the European Commission in the frame of Horizon2020, targets the development of such structures including their large scale application on ships. The project started on 1st of May 2018, the duration is three years. The University of Applied Sciences Bremen is involved in the hydrodynamic characterisation of such structures.

Different approaches can be found in literature to model air retaining surfaces within a numerical simulation: (a) homogenous slip length [6], (b) slip/no-slip condition [7], (c) slip/no-slip condition with deformable air-water interface [4] (d) two-phase flow with pinned air-water-interface at the pillars [8]. In (b) and (c) slip considers the air-water interface and no-slip the tip of the pillars. In case of turbulent flows, all the mentioned approaches are applied on channel flows using direct numerical simulation (DNS).

Open questions are still: (a) Which influence do different structure sizes performs on Reynolds stress tensor and the flow topology? (b) Do the two modeling approaches for the air retaining surface – slip/no-slip and two-phase flow simulation – deliver comparable results? (c) How big is the discrepancy between results of a high resolution second order numerical simulation in comparison to a higher order one? It is planed to answer the above mentioned questions in three parts. For all of them turbulent channel flow with a DNS and \( Re_T = 300 \) will be applied to consider the turbulent flow around ship hulls and to enable a simulation with feasible computational costs. Part I:
Figure 3: Two phase flow numerical simulation of air loss in laminar channel flow. The surface structure is abstracted from Salvinia molesta. The color on the interface indicates the normalized flow velocity (blue 0, red 1).

The chosen grid spacings in streamwise and spanwise directions amount to $\Delta x^+ = \Delta z^+ = 0.75$ for structure sizes of $L^+ = 25; 50; 100$ which is more than factor two lower in comparison to [4]. We assume that this resolution delivers sufficient precise data for the flow topology to evaluate the influence of the different boundary conditions. Furthermore, the higher spatial resolution in $x$ and $z$ is necessary to resolve the liquid air interface, treated in the next part. Part II: By using the same spatial resolution but different modeling of the air retaining surface (slip/no-slip, two-phase flow approach) a quantitative comparison of the two approaches is possible. If similar results are found, the use of slip/no-slip would mean significant lower computational costs. It has to be mentioned that the two-phase approach is the only one which delivers information on air losses. An example for air loss in a laminar flow is given in Figure 3. For the two-phase flow simulation the code OpenFOAM will be applied, which enables in maximum a second order discretization in space. This is disadvantageous for accuracy of a DNS. However, the implemented volume of fluid method allows the investigation of interactions between air-water interface with turbulence and the prediction of air losses. The latter are crucial for development of air retaining surfaces [4]. Part III: Comparable simulations to part I with the higher order code Nek5000 will be carried out to estimate the accuracy of OpenFOAM results. In this proposal part I is concerned.

WWW

https://aircoat.eu/

More Information


Funding

The AIRCOAT project has received funding from the European Unions Horizons 2020 research and innovation programme under grant agreement N° 764553.