Numerical simulation of a turbulent boundary layer at the gas/liquid interface

A E Goltsman and I I Saushin
Institute of Power Engineering and Advanced Technologies, FRC Kazan Scientific Center, Russian Academy of Sciences, 2/31 Lobachevsky Street, Kazan, Tatarstan, 420111, Russia, PO Box 261.
E-mail: an116ya@mail.ru

Abstract. This paper is the numerical study of the boundary layer evolution at gas streamlining of the surface of a standing liquid. The paper presents the results of numerical simulation of a two-phase flow at a moderate dynamic head of gas (about 10 Pa) insufficient for significant deformation of the interphase boundary. Comparison with the nature of the boundary layer in the flow around a solid plate has revealed a number of significant differences. Three characteristic regions of the boundary layer evolution depending on Re in the range of values from 0 to 320,000 have been obtained and described.

1. Introduction
The classical theory of the boundary layer considers two extreme cases of a continuous medium flow past a body: real and ideal fluids. In the first case, due to the action of tangential forces on the body surface, the so-called boundary no-slip condition is assumed, while for the ideal fluid flow on the wall, there is a continuous medium slip condition along the body surface. These two fundamental problems are investigated in detail both analytically and experimentally. The problem of the boundary layer along the surface of a moving body at a given velocity is also well known. In the overwhelming majority of such problem statements, the surface of the body is considered as solid and the velocity of surface motion is known. However, in practice, the flow of the gas medium along the free surface of a liquid is more common. The study of such a flow has actual practical significance, e.g. at liquid film entrainment by gas flow [1] and oil slick spread on the basin surface [2-3], for estimating the probability of drop entrainment in separation engineering systems [4], and while cooling liquid media with a gas flow in the chemical and energy industries. However, detailed experimental measurements of velocity profiles in the boundary layer of gas when the latter flows along the surface of a steady fluid were not found in any of the papers. When real gas flows on the streamlined body surface, tangential stresses arise due to the no-slip condition; and due to the absence of elastic tangential stresses in the fluid, this induces the movement of the upper layers of the fluid. Thus, the law of motion of a free surface is determined at once by several interrelated factors — the forces of viscosity in gas and liquid, and the forces of surface tension at the phase interface. With an increase in the dynamic head of the gas, unsteady wave disturbances on the liquid surface contributing to the separation of the boundary layer will begin to appear.

Several scientifically interesting issues arise at once for the above-mentioned problem statements. First, the fundamentals on the development of laminar and turbulent boundary layers in flow past a flat plate were formulated; criteria for estimating the boundary-layer thickness, displacement thickness and
momentum thickness depending on the Re number and length from the initial part of the plate were obtained for a long time ago. However, to date, there are no experimental data obtained for the boundary layer in the flow around the free surface of the liquid; so, the question of the evolution of such a boundary layer on the streamlined body length remains open. Second, whether fundamental postulates on the structure of the velocity profile and turbulent pulsations in the laminar and turbulent boundary layers will be preserved; how the range of the Re number characterizing the laminar-to-turbulent transition flow regime in the boundary layer will change; at what value of the Re number the significant deformations of the free surface leading to a significant restructuring of the boundary layer and its possible separation will occur.

Before conducting experimental studies, especially in the case of the multiphase flow, it is important to know the flow pattern beforehand. Therefore, the present paper shows the numerical simulation results of one of the flow regimes, which is planned to be studied experimentally using a Smoke Image Velocimetry technique (SIV) [5] for estimating instantaneous vector fields.

2. Problem statement and Computational Fluid Dynamics (CFD)

Figure 1 shows the geometric model for calculations, which is identical to the planned experimental setup. The test section was a channel of 1460 mm length with a rectangular cavity (1200 mm length and 120 mm height) filled with water; while the right wall of the cavity had a height of 145 mm, which should prevent water flowing out of the cavity due to the wave appearance on the water surface. For similar reasons, a small 5-mm long inlet section was realized in front of the cavity.

The forming of a zero-gradient 2D boundary layer of gas (air) flow on the free surface washed by liquid (water) was studied. In order to compare the boundary layers in the flow around a solid plate and on the interface with a liquid, the case of the cavity with the inserted solid body is also considered. When conducting experiments, it was planned to install a smooth inlet before the inlet section of the channel to create the required uniform velocity profile. Therefore, a constant velocity of 4 m/s was used as the boundary condition in the inlet section (inlet, Fig. 1); the turbulent characteristics were described by a hydraulic diameter of 260 mm (two channel heights) and a turbulence intensity of 5%.

According to different sources, the active effect of the gas flow on the flow of a liquid, up to the interfacial boundary rupture, can occur at the dynamic pressure more 15-30 Pa [7] or 39 Pa [8].

The simulations was conducted with Ansys Fluent 19.2 software. At this stage of our study, we used a two-dimensional formulation. The two-phase flow was described by unsteady Reynolds-averaged Navier-Stokes equations (URANS) with a k-ω SST isotropic turbulence model closure. Solver parameters were chosen based on the general practice of solving such problems: pressure-velocity coupling - coupled scheme, spatial discretization: volume fraction - QUICK, momentum, turbulent kinetic energy and turbulent dissipation rate - third-order MUSCL. The Eulerian multiphase model is used in Fluent simulations. The Eulerian model is the most complex multiphase model used in Fluent. It solves a set of n momentum equations for each phase.

A computational grid with 2 mm square cell height was generated in the geometric area. Validation of the grid adaptation was performed in three stages. At the first stage, after obtaining the solution on the initial grid, the adaptation of the near-wall cells at the solid walls and the mesh refinement up to the condition y’<1 were carried out. At the second stage, the computational domains with a variable value of the volumetric phase concentrations (or the area of the interface location) were adapted. After obtaining a solution that is invariant with respect to the size of the cells at the interface, all the nodes of the computational domain were adapted; the results confirmed the sufficient spatial scale of the computational domains for all areas of the geometric model. The paper presents the solution obtained on the final computational grid after all stages of adaptations.
3. Results

In order to validate the CFD data, the results on the flow around a solid surface were analyzed first. Figure 2 shows the evolution of the boundary-layer thickness $\delta$, displacement thickness $\delta^*$ and momentum thickness of the turbulent boundary layer at streamlining of a smooth fixed plate. These results obtained from the results of numerical simulation (solid blue line) are in good agreement with the estimates of these values using (1-3) [9] (dashed lines).

$$\delta(x) = 0.37x\left(\frac{U_\infty x}{v}\right)^{-0.2},$$  
$$\delta^*(x) = \frac{\delta}{8},$$  
$$\delta''(x) = \frac{7}{72}\delta,$$  

where $x$ is the distance from the beginning of the plate, $U_\infty$ is the velocity at the boundary layer edge, and $v$ is the kinematic viscosity. The velocity profiles (Figure 3) in the wall coordinates also agree well with the results of the direct numerical simulation at the similar Re$_\tau$ [10].

The smooth solid plate was replaced with a water tank in the next stage of the CFD. As expected, the dynamic pressure of the air flow was insufficient for severe deformation of the interphase boundary. However, the evolution of the boundary layer seems to be completely different compared to the flow past the plate.

There are three cases of the boundary-layer thickness growth depending on the Reynolds number calculated by the distance from the beginning of the plate (Re$_x$) in Figure 2:

At the initial section $0<\text{Re}_x<70000$, the growth rate of the boundary-layer thickness at the interface more than doubles the one in the flow past the plate (Figure 2). The $U^+$ profiles retain self-similarity on the viscous and buffer sections of the boundary layer. The velocity profile is similar to the one with a streamwise adverse pressure gradient on the logarithmic section (Figure 3).

The range of $70000<\text{Re}_x<150000$ is characterized by the constant boundary-layer thickness, while the self-similarity of the $U^+$ profile is preserved.

At the $\text{Re}_x>150000$, the boundary layer began to grow again and its growth rate became close in magnitude to that flowing past the plate (Figure 3). The velocity profile instability in the viscous sublayer of the boundary layer is also observed.

One of the reasons for these three flow regimes is obviously the interfacial behavior caused by the mutual influence of the fluids on each other. Figure 4 shows the air velocity at the boundary layer edge $U_a$ normalized to the water velocity at the interface $U_{\text{water}}$. If the graph is divided into three sections according to the Re$_x$ number from the description of the flow regimes, the correlation between the
water velocity and the growth rate of the boundary layer is clearly seen. Interval I is characterized by large amplitudes of $U_{\text{water}}/U_0$, which is essentially an evidence of disturbances or, in other words, waves on the water surface. These waves are similar to the roughness on a smooth plate and, as a result, contribute to the increase of the boundary-layer thickness. In interval II, the amplitude $U_{\text{water}}/U_0$ decreases, but the magnitude of the streamwise liquid velocity is sufficient to maintain a constant boundary-layer thickness. Finally, in interval III, the drop in the amplitude and streamwise water velocity leads to the similarity of the growth rate of the boundary-layer thickness in the case of a flow past a solid plate.

**Conclusion**

The results of the numerical simulation have revealed several phenomena that are interesting from a scientific point of view. First, of course, it is a significant difference in the evolution of the boundary layer in the flow at the interphase boundary and that in the flow past a solid plate. Moreover, the effect of $Re_x$ on the growth rate of the boundary-layer thickness has been found. As noted earlier, the present work is the first stage of a large-scale study on air streamlining of a water surface. The results obtained at this stage will be verified experimentally, which will allow, firstly, validating the numerical simulation problem statement, and secondly, proving the reliability of the present results.

**Figure 2.** The evolution of the characteristic boundary-layer thicknesses.

**Figure 3.** $U^+(y^+)$ velocity profiles in wall coordinates.
Figure 4. Normalized liquid velocity at the interface depending on the Reynolds number $Re_x$.

Acknowledgments
The study was supported by the President of Russian Federation grant for the support of young Russian scientists (Project No. MK-1256.2019.1). Computational resources for numerical simulation of the present study were carried out with financial support from the government assignment for FRC Kazan Scientific Center of RAS.

References
[1] Vozhakov I S, Arkhipov D G, Tsvelodub O Y 2015 Thermophysics and Aeromechanics 22 (2) 191–202
[2] Kim T H, Yang C S, Oh J H, Ouchi K 2014 PloS one 9 (1) e87393
[3] Guze S, Mazurek J, Smolarek L 2016 Journal of KONES 23
[4] Hreiz R, Gentric C, Midoux N, Lainé R, Fünfschilling D 2014 Chemical engineering research and design 92 (11) 2231–46
[5] Mikheev N I, Goltsman A E, Saushin I I, Dushina O A 2017 Experiments in fluids 58 (8) 97
[6] Ishii M, Grolmes M A 1975 AIChE Journal 21 (2) 308–18
[7] Kasatkin A G 1973 Basic processes and apparatures of chemical technology (in Russian).
[8] Ng S Y, Priestman G H, Allen R W K 2006 Chemical Engineering Research and Design 84 (10) 884–94.
[9] Schlichting H, Gersten K 2017 Boundary-layer theory. 9th edition (Springer-Verlag: Berlin, Heidelberg) p 805
[10] Schlatter P, Örlü R, Li Q, Brethouwer G, Fransson J.H.M., Johansson A.V., Alfredsson P H, Henningson D S 2009 Phys Fluids 21 (5) 051702