Commun. Comput. Phys.
doi: 10.4208/cicp.OA-2019-0124

Vol. **28**, No. 3, pp. 1133-1146
September 2020

A Novel Strong-Coupling Pseudo-Spectral Method for Numerical Studies of Two-Layer Turbulent Channel Flows

Tong Wu[1,2], Zixuan Yang[1,3,*], Shizhao Wang[1,3] and Guowei He[1,3]

[1] *State Key Laboratory of Nonlinear Mechanics, Institute of Mechanics, Chinese Academy of Sciences, China.*
[2] *Laboratoire de Mécanique Physique, École Centrale de Pékin, Beihang University, China.*
[3] *School of Engineering Sciences, University of Chinese Academy of Sciences, China.*

Received 21 July 2019; Accepted (in revised version) 8 January 2020

Abstract. A new method is proposed to simulate a coupled air-water two-layer turbulent channel flow. A numerically effective dynamic viscosity is implemented to calculate the viscous momentum flux at the interface, leading to a strong-coupling scheme for the evolution of air and water motions. The direct numerical simulation results are compared with those in the literature obtained from a weak-coupling scheme. It is discovered that while the turbulence statistics of the air phase based on the strong- and weak-coupling schemes are close to each other, the results on the water side are influenced by the coupling approach, especially near the water surface.

AMS subject classifications: 76F65

Key words: Direct numerical simulation, coupled air-water turbulence, interface condition.

1 Introduction

The investigations of air-water interactions are foundations of many applications in ocean science, environmental engineering, and chemical engineering. The computational fluid dynamics (CFD) are used broadly for studying these problems. During the last two decades, various physical models of air-water interactions have been studied through CFD based on different numerical schemes.

CFD studies of air-water interface can be categorized into one-fluid and two-fluid simulations. In the one-fluid simulation, the governing equations of only one fluid phase

*Corresponding author. *Email addresses:* yangzx@imech.ac.cn (Z. Yang), wutong95@buaa.edu.cn (T. Wu), wangsz@lnm.imech.ac.cn (S. Wang), hgw@lnm.imech.ac.cn (G. He)

are solved, while the effect of the other phase is modeled through appropriate boundary conditions. Specifically, in the numerical studies of air flows over a water surface, the water surface is usually treated as a flat or wavy boundary with prescribed surface roughness and velocity [1–4], while in a large number of simulations of water flows, the effect of air is imposed through a given shear stress at the water surface [5–9]. The one-fluid simulation is usually conducted to study physical problems without strong two-way interactions between the two fluid phases.

In the two-fluid simulation, it is challenging and expensive to track or capture the deformation of the interface between two fluid phases [10–12]. A simplified physical model for two-fluid simulation is a two-layer channel flow [13,14], which has been studied for fundamental research of turbulent motions in the vicinity of the interface. In the two-layer channel flow, the interface between two fluid phases is assumed to be flat [14], corresponding to flow conditions with large gravity or large surface tension. The two-layer channel flow features a simple interface geometry, which makes it possible to solve the motions of two fluid phases by using a pseudo-spectral scheme [15]. The high accuracy and high efficiency of the pseudo-spectral method are desired features of numerical algorithms [16–19].

While the numerical method for simulating one-layer channel flow is mature, the key of the numerical method for simulating two-layer channel flow is the implementation of the interface condition. Lombardi *et al.* [13] proposed a staggered advancement method. In the first-half timestep of this method, the velocity at the water surface is used as the boundary condition of the air flow, while in the second-half timestep, the shear stress at the air bottom is used as a momentum source to drive the water motion. This method is known as a weak-coupling approach as the continuities in velocity and stress are not satisfied simultaneously.

To develop a strong-coupling method, the use of inner iteration is an option [14,20]. However, this method significantly increases the computational cost. Based on the literature reviewed above, we have developed a strong-coupling method for two-layer turbulent channel flow by implementing an effective viscosity at the interface, such that the motions of two fluid phases are evolved synchronically without applying any inner iteration. The proposed method is then tested in the context of a low-Reynolds-number two-layer turbulent Couette flow. The turbulent statistics are compared with the results based on the weak-coupling method of Liu *et al.* [14]. The remainder of this paper is organized as follows. In Section 2, the numerical methods are introduced. Then, the results are presented and discussed in Section 3, followed by the conclusions in Section 4.

2 Numerical method

2.1 Computational domain and governing equations

Fig. 1 shows the computational domain of a two-layer channel flow. As shown, x_1, x_2, and x_3 represent the streamwise, spanwise, and vertical directions, respectively. The air-