ARTICLE IN PRESS

Journal of Fluids and Structures ∎ (∎∎∎∎) ∎∎∎–∎∎∎

Contents lists available at ScienceDirect

Journal of Fluids and Structures

journal homepage: www.elsevier.com/locate/jfs

Numerical study of flapping filaments in a uniform fluid flow

Julien Favier [a,*], Alistair Revell [b], Alfredo Pinelli [c]

[a] Aix Marseille Université, CNRS, Centrale Marseille, M2P2 UMR 7340, 13451 Marseille, France
[b] School of Mechanical, Aerospace and Civil Engineering (MACE), University of Manchester, United Kingdom
[c] School of Engineering and Mathematical Sciences (EMS), City University, London, United Kingdom

ARTICLE INFO

Article history:
Received 8 October 2013
Accepted 30 November 2014

Keywords:
Beating filaments
Flapping flags
Inverted hydrodynamic drafting
Immersed Boundary
Lattice Boltzmann

ABSTRACT

The coupled dynamics of multiple flexible filaments (also called monodimensional flags) flapping in a uniform fluid flow is studied numerically for the cases of a side-by-side arrangement, and an in-line configuration. The modal behaviour and hydrodynamical properties of the sets of filaments are studied using a Lattice Boltzmann–Immersed Boundary method. The fluid momentum equations are solved on a Cartesian uniform lattice while the beating filaments are tracked through a series of markers, whose dynamics are functions of the forces exerted by the fluid, the filaments flexural rigidity and the tension. The instantaneous wall conditions on the filaments are imposed via a system of singular body forces, consistently discretised on the lattice of the Boltzmann equation. The results exhibit several flapping modes for two and three filaments placed side-by-side and are compared with experimental and theoretical studies. The hydrodynamical drafting, observed so far only experimentally on configurations of in-line flexible bodies, is also revisited numerically in this work, and the associated physical mechanism is identified. In certain geometrical and structural configuration, it is found that the upstream body experiences a reduced drag compared to the downstream body, which is the contrary of what is encountered on rigid bodies (cars, bicycles).

1. Introduction

The scope of this work is the physical analysis of the dynamics of flapping filaments in a streaming ambient fluid, which has a large spectrum of applications in aeronautics, civil engineering or biological flows. From the theoretical side, this fluid structure interaction problem is particularly challenging as it involves non-linear effects as well as large structural deformations (Païdoussis, 2004; Shelley and Zhang, 2011). The present study is particularly inspired by various experiments on flapping filaments realised in soap films (Zhang et al., 2000; Zhu and Peskin, 2000; Ristroph and Zhang, 2008). Indeed, soap film experiments associated with thin-film interferometry for flow visualisation can be considered as a reasonable approximations of 2D fluid structure interaction scenarios, thus suitable for the validation of the results obtained with our 2D numerical approach.

In our simulations, we consider a 2D incoming incompressible flow modelled using a Lattice Boltzmann method, coupled to a model of infinitely thin and inextensible filament experiencing tension, gravity, fluid forces and flexural rigidity (i.e. a bending term in the form of a 4th derivative with respect to the curvilinear coordinate describing the filament). Also, at all

* Corresponding author.
E-mail address: Julien.Favier@univ-amu.fr (J. Favier).

http://dx.doi.org/10.1016/j.jfluidstructs.2014.11.010
0889-9746/

Please cite this article as: Favier, J., et al., Numerical study of flapping filaments in a uniform fluid flow. Journal of Fluids and Structures (2014), http://dx.doi.org/10.1016/j.jfluidstructs.2014.11.010

time instants tension forces are determined to maintain the inextensibility of the structure. In this simple model the energy balance of the system is driven by the bending forces and fluid forces, as the structure is controlled by an inextensibility constraint which prohibits stretching or elongation motions that would dissipate energy. This system encompasses all the essential ingredients of a complex fluid–structure interaction problem: large deformations, slender flexible body, competition between bending versus fluid forces, inextensibility and effect of the filament tips on the surrounding flow as vorticity generators.

To enforce the presence of the solid on the fluid lattice, we use a variant of the Immersed Boundary method previously developed by the authors (Pinelli et al., 2010) on finite difference and finite volume Navier Stokes solvers. In this work we use the same algorithm as in Pinelli et al. (2010) to impose the Immersed Boundary forces, but we adapt it to a Lattice Boltzmann solver. This approach for imposing the forces has shown to be order 2 in space, computationally cheap and directly provides for the forces exerted on the fluid by the filaments without the introduction of any empirical parameter. Using the Lattice Boltzmann method in conjunction with an Immersed Boundary technique to solve the motion of an incompressible fluid also allows for a clean imposition of the boundary conditions on the solid since it does not suffer from errors originating from the projection step, as it is the case when associated with unsteady incompressible Navier Stokes solvers (Domenichini, 2008).

Making use of the outlined Lattice Boltzmann–Immersed Boundary approach, we consider the coupled dynamics of systems made of highly deformable flexible filaments, as introduced by Favier et al. (2014). No artificial contact force is introduced between the filaments, in order to preserve a purely hydrodynamical interaction. We focus in this work on the modal behaviour of a set of two and three side-by-side filaments, by varying the spacing between them. The obtained results confirm the theoretical predictions and experimental observations mentioned in the literature. The so-called anomalous hydrodynamic drafting pointed out experimentally in Ristroph and Zhang (2008) is recovered here numerically and a physical mechanism is proposed to explain this phenomenon.

2. Coupled Lattice Boltzmann–Immersed Boundary method

This fluid–structure problem is tackled using an Immersed Boundary method coupled with a Lattice Boltzmann solver. In the following we provide a summary of the numerical technique while details of the methodology can be found in Favier et al. (2014).

The fluid flow is modelled by advancing in time the Lattice Boltzmann equation which governs the transport of particles density distribution f (probability of finding a particle in a certain location with a certain velocity). It is often classified as a mesoscopic method, where the macroscopic variables, namely mass and momentum, are derived from the distribution functions f. An excellent review of the method can be found in Succi (2001).

Using the classical BGK approach (Bhatnagar et al., 1954), and after an appropriate discretisation process (Malaspinas, 2009), the Boltzmann transport equation for the distribution function $f=f(\mathbf{x},\mathbf{e},t)$ at a node $\mathbf{x}$ and at time t with particle velocity vector $\mathbf{e}$ is given as follows:

$$f_i(\mathbf{x}+\mathbf{e_i}\Delta t, t+\Delta t)-f_i(\mathbf{x},t)=-\frac{\Delta t}{\tau}\left(f(\mathbf{x},t)-f^{(eq)}(\mathbf{x},t)\right)+\Delta t F_i. \tag{1}$$

In this formulation, $\mathbf{x}$ are the space coordinates, $\mathbf{e}_i$ is the particle velocity in the ith direction of the lattice and F_i accounts for the body force applied to the fluid, which conveys the information between the fluid and the flexible structure. The local particle distributions relax towards an equilibrium state $f^{(eq)}$ in a single time scale τ. Eq. (1) governs the collision of particles relaxing toward equilibrium (first term of the r.h.s.) together with their streaming which drives the data shifting between lattice cells (l.h.s of the equation). The rate of approach to equilibrium is controlled by the relaxation time τ, which is related to the kinematic viscosity of the fluid by $\nu=(\tau-1/2)/3$. Eq. (1) is approximated on a Cartesian uniform grid by assigning to each cell of the lattice a finite number of discrete velocity vectors. In particular, we use the D2Q9 model, which refers to two-dimensional and nine discrete velocities per lattice node (corresponding to the directions east, west, north, south, centre, and the 4 diagonal directions as given by Eq. (2)), where the subscript i refers to these discrete particle directions. As is usually done, a convenient normalisation is employed so that the spatial and temporal discretisation in the lattice are set to unity, and thus the discrete velocities are defined as follows:

$$\mathbf{e_i}=c\begin{pmatrix} 0 & 1 & -1 & 0 & 0 & 1 & -1 & 1 & -1 \\ 0 & 0 & 0 & 1 & -1 & 1 & -1 & -1 & 1 \end{pmatrix} \quad (i=0,1,\dots,8), \tag{2}$$

where c is the lattice speed which defined by $c=\Delta x/\Delta t=1$ with the current normalisation. The equilibrium function $f^{(eq)}(\mathbf{x},t)$ can be obtained by Hermite series expansion of the Maxwell–Boltzmann equilibrium distribution (Qian et al., 1992):

$$f_i^{(eq)}=\rho\omega_i\left[1+\frac{\mathbf{e_i}\cdot\mathbf{u}}{c_s^2}+\frac{(\mathbf{e_i}\cdot\mathbf{u})^2}{2c_s^4}-\frac{\mathbf{u}^2}{2c_s^2}\right]. \tag{3}$$

In Eq. (3), c_s is the speed of sound $c_s=1/\sqrt{3}$ and the weight coefficient ω_i are $\omega_0=4/9$, $\omega_i=1/9$, $i=1\dots4$ and $\omega_5=1/36$, $i=5\dots8$ according to the current normalisation. The macroscopic velocity $\mathbf{u}$ in Eq. (3) must satisfy the requirement for low Mach number, M, i.e. that $|\mathbf{u}|/c_s\approx M\ll 1$. This stands as the equivalent of the CFL number for classical

Please cite this article as: Favier, J., et al., Numerical study of flapping filaments in a uniform fluid flow. Journal of Fluids and Structures (2014), http://dx.doi.org/10.1016/j.jfluidstructs.2014.11.010