Available online at www.sciencedirect.com
ScienceDirect
Journal of Bionic Engineering 14 (2017) 88–98

Flow Confinement Effects on the Wake Structure behind a Pitching Airfoil: A Numerical Study Using an Immersed Boundary Method

Ali Akbar Hosseinjani, Ali Ashrafizadeh

Faculty of Mechanical Engineering, *K. N. Toosi University of Technology*, *Tehran P.O.B.* 19395-1999, *Iran*

Abstract

The flow patterns and wake structures behind a pitching airfoil in an un-bounded domain have been studied extensively. In contrast, the flow phenomena associated with a pitching airfoil near a solid boundary have not been adequately studied or reported. This paper aims at filling this research gap by considering the flow confinement effects on the flow pattern around a pitching airfoil. To achieve this goal, the flow fields around a flapping airfoil in un-bounded, bounded and semi-bounded domains are studied and compared. Numerical simulations are carried out at a fixed Reynolds number, Re = 255, and at a fixed oscillation frequency corresponding to St = 0.22. An accurate immersed boundary method is employed to calculate the unsteady flow fields around the airfoil at various flapping amplitudes. It is argued that two flow mechanisms, here called “the interaction effect” and “the induced reverse flow effect” are responsible for the variations of the flow field due to the presence of a nearby solid boundary.

Keywords: immersed boundary method, pitching airfoil, vortex street, induced momentum, aligned vortex

doi: 10.1016/S1672-6529(16)60380-3

1 Introduction

Locomotion, by means of flapping, is an important biological mechanism for flying and swimming. Scientists, engineers and biomechanical researchers have all been interested in the phenomena associated with the flapping wings. It has been found, for example, that the flapping mechanism is more efficient, as compared to the classical fixed wing flight mechanism, at low Reynolds numbers[1].

The flow field around a flapping airfoil is very complex due to the transient discontinuous interactions between the generated vortices. The vortical flow structures behind a pitching airfoil in an un-bounded domain have been studied extensively using theoretical[1,2], experimental[3–5] and numerical[6–10] methods. A review of research activities in this field has been provided by Hosseinjani and Ashrafizadeh[11] who have also studied the wake structure and thrust/lift generation of a pitching airfoil at low Reynolds number using a direct forcing Immersed Boundary Method (IBM).

Previous studies on the flow fields around pitching airfoils in an un-bounded domain have made it possible to better understand and explain the effects of parameters such as frequency, amplitude and flapping mode on the lift/thrust generation and momentum loss in the flow field. These studies have shown that there are distinguished flow regimes associated with different wake patterns. In particular, Kármán Vortex Street (KVS) and Reverse Kármán Vortex Street (RKVS) are two flow regimes that are associated with the enhancement of the adverse and favorable momentum transfer, respectively. In contrast to the KVS regime, which only provides a symmetric flow pattern, the flow patterns corresponding to RKVS regime can be either symmetric or un-symmetric[8].

The flow phenomena associated with a pitching airfoil near a solid boundary, however, have not been adequately studied or reported in the literature. Moryossef and Levy[12] studied the flow field around a plunging airfoil near the ground numerically. They reported that when the airfoil was close to the ground, viscous flow effects were dominant only at low frequencies and inviscid flow characteristics took over at

Corresponding author: Ali Akbar Hosseinjani
E-mail: aahosseinjani@mail.kntu.ac.ir

high frequencies. Gao and Lu[13] investigated the ground effects on hovering insects at Re = 100 and reported force enhancement, reduction and recovery as three important effects in that particular flow situation and Reynolds number. Moliana and Zhang[14] studied the aerodynamic behavior of an inverted airfoil with heaving motion near the ground. They reported that three different flow regimes, *i.e.* flow regimes due to the ground effect, incidence effect and added mass effect, could be distinguished. Wu and Zhao[15] numerically studied a plunging airfoil near the ground. The high frequency oscillations and the small distance between the airfoil and the ground were considered responsible for the generation of thrust and lift forces.

Van Truong *et al.*[16] investigated the aerodynamic forces and flow structures of a single flapping wing near the ground. Liang *et al.*[17] studied a hovering airfoil in ground effect. They assumed that the flow is inviscid and incompressible and reported that for the forced oscillating airfoil, high heaving frequencies resulted in larger time-averaged values and amplitudes of the lift coefficient. Wu *et al.*[18] considered the flow field around a flapping insect near the ground. They reported that drag coefficient increased at low frequency and decreased at high frequency and the lift coefficient increased at both low and high frequencies and reduced at moderate frequencies.

In this paper, the vortex structure behind a sinusoidal pitching symmetric airfoil in a bounded domain (channel flow) is numerically studied over a range of oscillation amplitudes. In all previous studies only the ground effect, corresponding to flow near a single wall, was studied and no study regarding the flow field around a flapping airfoil in a channel was published to the best knowledge of the authors. Also, the effects of the oscillation amplitude of a near the ground pitching airfoil on the wake structure and force coefficients have not yet been reported in the open literature.

2 Governing equations

The dimensionless forms of the governing equations for an incompressible flow in the computational domain Ω are:

$$\frac{\partial \boldsymbol{u}^*}{\partial t} + \boldsymbol{u}^*.\nabla \boldsymbol{u}^* = -\nabla p^* + \left(\frac{1}{Re}\right)\nabla^2 \boldsymbol{u}^* + \boldsymbol{f}^*, \tag{1}$$

$$\nabla.\boldsymbol{u}^* = 0, \tag{2}$$

where, $\boldsymbol{u}^*$ is the dimensionless fluid velocity vector, p^* is the dimensionless pressure and $\boldsymbol{f}^*$ is the dimensionless external momentum source used to impose the no-slip boundary condition. Also, $Re = u_0 L_0/v$, is the Reynolds number, where u_0 and L_0 are the problem-dependent velocity and length scales, respectively and v is the kinematic viscosity of the fluid.

3 The numerical solution approach

3.1 The immersed boundary treatment

An iterative direct forcing immersed boundary methods, proposed by Ji *et al.*[19], is implemented here. One important feature of this method is that the pressure and the source term are calculated simultaneously in each time step. The method using two types of nodal points is in the discrete computational domain. Fixed background points are called the Eulerian points (Eu points) and the possibly moving points, which define the immersed boundary, are called the Lagrangian points (IB points). The variables at Eu and IB points are related via discrete delta functions.

Two operators, $I(\phi)$ and $D(\Phi)$, are defined which represent the interpolation and distribution operators respectively[19]. The parameter ϕ represents variables such as $\boldsymbol{u}^*$, p^*, $\boldsymbol{f}^*$, at the Eu points, and the parameter Φ indicates the variables at IB point, *i.e.* $\boldsymbol{U}^*$, P^*, $\boldsymbol{F}^*$. Using the interpolation operator, any variable at an IB point, *i.e.* $\Phi(X_{IB})$, can be expressed as:

$$\Phi(\boldsymbol{X}_{\mathrm{IB}}) = I(\phi) = \sum\nolimits_{\boldsymbol{x}\in g_h} \phi(\boldsymbol{x})\delta(\boldsymbol{x} - \boldsymbol{X}_{\mathrm{IB}})\Delta x^2, \tag{3}$$

where g_h represents a set of Cartesian grid points at both sides of IB point, $\boldsymbol{X}_{\mathrm{IB}}$, here called the active grid points. Non-dimensional coordinates $\boldsymbol{X}_{\mathrm{IB}} = (X_{\mathrm{IB}}, Y_{\mathrm{IB}})$ and $\boldsymbol{x} = (x, y)$ represent IB and Eu points, respectively and Δx is the non-dimensional uniform Cartesian grid size. The discrete delta function (δ) is similar to the function used in Ref. [20].

Likewise, using the distribution operator, any variable at an Eu point, *i.e.* $\phi(\boldsymbol{x})$, can be expressed as:

$$\phi(x) = D(\Phi) = \sum_{\mathrm{IB}=1}^{N_b} \Phi(\boldsymbol{X}_{\mathrm{IB}})\delta(\boldsymbol{x} - \boldsymbol{X}_{\mathrm{IB}})\Delta V_{\mathrm{IB}}, \tag{4}$$

where, N_b is the total number of IB points and $\Delta V_{\mathrm{IB}} = \Delta x \times \Delta S_{\mathrm{IB}}$ is the discrete volume (cell area in 2D) around the IB point and ΔS_{IB} is the distance between any two consecutive IB points.