Computers & Fluids 108 (2015) 142–155

Contents lists available at ScienceDirect

Computers & Fluids

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

# Fluid structures of flapping airfoil with elliptical motion trajectory

J.A. Esfahani [a,*], E. Barati [b], H.R. Karbasian [a]

[a] Center of Excellence on Modelling and Control Systems (CEMCS), Department of Mechanical Engineering, Ferdowsi University of Mashhad, Mashhad 91775-1111, Iran
[b] Department of Mechanical Engineering, Faculty of Eng., Khayyam University, Mashhad, Iran

ARTICLE INFO

*Article history:*
Received 22 October 2013
Received in revised form 25 October 2014
Accepted 2 December 2014
Available online 6 December 2014

*Keywords:*
Flapping airfoil
Motion trajectory
Fluid structures
Biomimetic propulsion

ABSTRACT

In the present study, the effect of elliptical motion trajectory on the aerodynamic characteristics and propulsive performance of a flapping airfoil is evaluated. A periodic horizontal motion (forward/backward) is combined with vertical motion (upward/downward) of the airfoil to introduce a new kinematic parameter, and of course, an elliptical motion trajectory for flapping airfoil. Similar kinematics is also observed in the flying of birds and swimming of the penguins or turtles. For this modeling, the Navier–Stokes equations are used to simulate the unsteady flow field over a two dimensional NACA0012 airfoil. The Navier–Stokes equations are discretized based on the finite volume method and are solved with a pressure-based algorithm. The flow is assumed to be laminar and incompressible and transient terms are conducted using a second order Euler implicit scheme. It is shown that the combination of horizontal and vertical motions for the flapping airfoil changes the kinematics, motion trajectory and hence the effective angle of attack profile during the flapping cycle. The elliptical motion trajectory will also influence on the fluid structures and change the vortex shedding pattern and wake zone behind the airfoil. Additionally, the introduced kinematics may influence significantly on the aerodynamics and propulsive performance of either pure plunging or pitching/plunging airfoil.



## 1. Introduction

A successful understanding of flapping airfoil propulsion requires the contribution of various disciplines including aerospace and biology. On the point of high level propulsive efficiency, stability and maneuverability, flapping airfoil with pure pitching, pure heaving or combination of them have potential applications in the design of high efficient Micro Air Vehicles (MAVs) [1]. Many experimental, theoretical and computational works have been carried out to understand the flapping airfoil aerodynamics. Anderson et al. [2] and Koochesfahani [3] performed experiments on various types of airfoil in order to consideration of flapping foil performance. They observed that for heaving airfoil, a thrust-type vortex street is formed by a weak leading edge separation, which moves toward the trailing edge of airfoil and constructively merges with the trailing edge vortex. Anderson [4] used Digital Particle Image Velocimetry (DPIV) on an airfoil oscillating in both pitch and heave, and observed that at least six distinct vortex formation regimes exist for the range of parameters, but a region of optimal wake formation, characterized by two vortices per cycle in a reverse Von Karman pattern.

To understand the optimum kinematic and aero/hydrodynamic conditions many scientists have done many experimental, theoretical and numerical studies, which some of highlighted researches are briefly provided here. Hover et al. [5] in their experimental investigations found that in the most of cases the high efficiencies for flapping foil propulsion happened in low Strouhal numbers and, of course, in low thrust coefficients. On the other hand, high thrust coefficients limited in low efficiencies. Detailed experimental analysis related to the effects of kinematics as well as the wake visualization behind the flapping airfoils can be found in past works by Read and Hover [6] and Schouveiler et al. [7].

Based on the airfoil's configurations, the results of Lehmann [8], Lentink and Gerritsma [9] and Rozhdestvensky and Ryzhov [1] indicated that thick airfoils can improve plunging airfoil performance, whereas Vandenberghe et al. [10] and Wang [11] suggested that thin airfoils perform better, and the inviscid analysis of Cebeci et al. [12] concludes no influence of airfoil thickness on plunging airfoil propulsion. Amiralaei et al. [13] developed the 2D Navier–Stokes which is associated with Finite Volume Method (FVM) simulating the flapping-wing in low Reynolds number flows. They have demonstrated that the importance of pitch amplitude and phase angle difference between plunging and pitching are more than Reynolds and Strouhal numbers. They also announced that the best aerodynamic performance occurs in symmetrical oscilla-

* Corresponding author. Tel.: +98 915 134 4361; fax: +98 511 77 240 488.
*E-mail addresses:* Jaesfahani@gmail.com, Abolfazl@um.ac.ir (J.A. Esfahani).

http://dx.doi.org/10.1016/j.compfluid.2014.12.002
0045-7930/

tions. Esfahani et al. [14] studied numerically the hydrodynamics of fish tail and the caudal effect on swimming kinematics. They found that the fish's caudal can influence on kinematics of flapping tail and affects significantly on overall propulsive performance. In recent studies, Lu et al. [15] numerically investigated the asymmetric sinusoidal motion effect on aerodynamic performance of pitching NACA0012 airfoil using Computational Fluid Dynamic (CFD) method. They found that the asymmetry parameter can significantly influences on the aerodynamic performance including instantaneous lift, maximum lift and drag and hysteresis loop. They also reported the significant impact of asymmetry motion on the flow structures and development of vortices.

As mentioned, many investigations have been done to find out the optimum kinematic and aero/hydrodynamic parameters in the best operation condition toward having a favorable propulsive performance. But until recently, there is not enough focus on the influence of motion trajectory on the aerodynamics and propulsive performance of a flapping airfoil. The main objective of this paper is to evaluate the effect of elliptical motion trajectory on the aerodynamic parameters and propulsive performance of conventional flapping airfoil. For this purpose, a periodic horizontal (forward/backward) motion is added to vertical (upward/downward) motion of airfoil to change direction of a conventional flapping airfoil and change its motion trajectory. This change will be conducted for both pure plunging and pitching/plunging airfoils. The unsteady flow field over the two dimensional NACA0012 airfoil is simulated using the Navier–Stokes equations. The flow is assumed to be laminar and incompressible, and transient terms are conducted using a second order Euler implicit scheme.

## 2. Governing equations

In the present study the flow field interactions with a moving NACA0012 airfoil is numerically simulated. The flow is also assumed to be incompressible and laminar. Corresponding simulations have been addressed by Young [16]. He showed little difference between laminar and turbulent results. The flow field over a two dimensional airfoil in relatively low Reynolds number is simulated using Navier–Stokes equations, which can be expressed in non-dimensional form as following [13]:

$$\nabla \cdot U = 0 \tag{1}$$

$$\left(\frac{k}{\pi}\right)\frac{\partial U}{\partial \tau} + (U \cdot \nabla)U = -\nabla p + \frac{1}{\mathrm{Re}}\nabla^2 U \tag{2}$$

where none-dimensional terms are $U = (u, v)$, $\tau = t/T$ and $p = \text{pressure}/\rho U^2$ (where $\rho$ is density). Additionally, $k$ and Re are reduced frequency and Reynolds number, respectively. These parameters can be defined as following equations:

$$k = \frac{\pi f c}{U_\infty} \tag{3}$$

$$\mathrm{Re} = \frac{U_\infty c}{\upsilon} \tag{4}$$

where $U_\infty$, $c$, $f$, $\upsilon$ are reference velocity, airfoil chord length, frequency and fluid's kinematic viscosity, respectively.

## 3. Computational model

The solver is pressure-based segregated solver that computes the continuity and momentum equations. The Navier–Stokes equations are discretized based on the Finite Volume Method (FVM) within the open source CFD software OpenFOAM [17]. The FVM is a discretization technique used commonly in CFD modeling of flapping foil. Amiralaei et al. [13,18], Lu et al. [15,19], Young et al. [20,21] and Wang et al. [11,22,23] are the example of researchers used FVM in their CFD simulations to study flapping foil performance. The convective and diffusive terms are discretized using a second order central differencing scheme. The transient terms are also conducted using a second order Euler implicit scheme. The Semi Implicit Method for Pressure Linked Equations (SIMPLE) algorithm [24] is used for pressure–velocity coupling. The SIMPLE algorithm is a widely used numerical procedure to solve the set of discretized Navier–Stokes equations and more details about this method can be found in [25] and [26]. The convergence criteria for residuals (continuity and momentum) are selected $10^{-5}$ in each iteration and the number of iterations in each time step is also chosen 150 to ensure satisfactory results. To reach the stable and reliable results in the present simulation the results are reported after 5 flapping cycle. This policy is chosen because the unstable vortex shedding are not formed in the appropriate manner at the early steps of simulation [23]. The motion of the airfoil is modeled using dynamic mesh technique and more details can be found in [17]. Since the quality of mesh around the airfoil to be kept suitable and minimize the skewness factor, the airfoil and whole computational domain moves simultaneously. The skewness factor is one of the important characteristics in structured mesh that must be taken into account due to its sensitive effects on the vortex structure and forces acting on the airfoil [27]. In order to set dynamic motion of the airfoil, the reference point for the airfoil oscillation is selected around 1/3 of chord length.

## 4. Specifications of computational domain

The computational domain is discretized using C-type mesh. This domain contains $6 \times 10^4$ cells and 1000 time steps per period. Additionally, 900 nodes are distributed on the airfoil surface. The schematic of gird used for present simulation is displayed in Fig. 1. The remarked time step and grid size is specified based on the extensive investigations on grid and time independencies, which will be presented in the Section 7 in details. The boundary conditions and domain specifications are schematically shown in Fig. 2.The steady state solutions are used as initial condition for unsteady computations at corresponding conditions. The airfoil surface is set to no-slip condition. The upstream inlet velocity, downstream pressure outlet and across the flow boundaries are set to 25$c$, 30$c$ and 25$c$ from the airfoil, respectively. The inlet velocity is specified based on the desired Reynolds number and the outflow boundary is set to pressure outlet.

## 5. Kinematics of flapping foil

In most of the cases a conventional kinematics of flapping foil propulsion is introduced by two motions. One of them is translational motion (plunge), which is normal to direction of reference

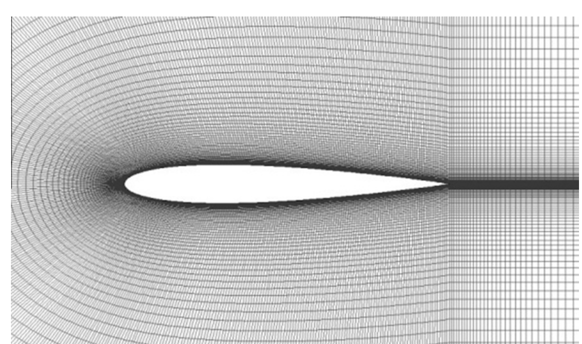

**Fig. 1.** Schematic of C-type grid around the NACA0012 airfoil used for present simulation.