Aerospace Science and Technology 55 (2016) 242–251

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Symmetrical and non-symmetrical 3D wing deformation of flapping micro aerial vehicles

W.B. Tay [1]

Temasek Laboratories, National University of Singapore, 5A Engineering Drive 1, 117411, Republic of Singapore

ARTICLE INFO

Article history:
Received 3 January 2016
Received in revised form 19 April 2016
Accepted 25 May 2016
Available online 30 May 2016

Keywords:
Flapping wings
Simulations
Deformations
MAV
Immersed boundary method

ABSTRACT

Wing deformation can improve the performance of flapping wing MAV. Although this can be achieved through passive wing deformation, it is not possible to tune the deformation easily to obtain maximum performance. With the advancement in smart materials, prescribed deformation to improve maximum performance is becoming more practical. In this study, we study different forms of deformation, including a unique single-sided deformation to improve the wing's performance. This study extends the previous 2D study to 3D, and investigates if the favorable results in the earlier study are also applicable in 3D. Results show that we can obtain improvements as much as 111% for thrust and 125% for efficiency through careful selection of parameters. Positive lift coefficient of up to 1.45 is also observed when non-symmetrical single sided flexing is used. This study will be helpful in the design of future flapping wing MAV.

1. Introduction

Wing flexibility in flapping micro aerial vehicles has been investigated in many ways [1–7] due to its potential to improve efficiency, lift and thrust. These investigations can in general be classified into two types: prescribed (active) flexibility [1,4,7], or fluid structure interaction (FSI) induced (passive) flexibility [2,3,5]. In the first scenario, Miao and Ho [1] used FLUENT to investigate the effect of chord-wise flexibility on the aerodynamic characteristics for a flapping airfoil with various combinations of Reynolds number (Re) and reduced frequency (k). Results show the formation of leading edge vortex (LEV), with improvement in both the efficiency and thrust. Tay and Lim [4] extended the previous works by investigating additional factors such as flexing center location, standard two-sided flexing as well as a type of single-sided flexing using their own in-house Navier–Stokes solver. Results show that with the correct parameters, efficiency and thrust coefficient increase to as high as 0.76 and 3.57 respectively. The new single-sided flexing also produces up to a lift coefficient of 4.61 for the S1020 airfoil. However, the study was performed only in 2D. Ghommem et al. [7] used the unsteady vortex lattice method together with a gradient-based optimizer to obtain optimized wing shapes that give maximum efficiency. It was also found that the optimal wing shapes are highly dependent on the reduced frequency.

The advantage of prescribed flexibility is that the amount of flexibility on the wing can be controlled at all times to achieve the highest possible efficiency, lift and thrust. Moreover, the prescribed deformation will change depending on the mission objective, such as high payload or high speed mission through high lift or thrust configuration respectively. In comparison, passive flexibility depends on the forces generated by the wing which in turn determines the amount of flexibility. Hence, when flow conditions change, it is difficult to predict in advance the amount of flexibility as well as their effect on the force output. Despite the clear advantage of prescribed flexibility, the means to achieve it is not so straightforward. However, with the advent of smart materials such as shape memory alloys [8] and dielectric elastomers [9], it is now possible to control the wing's deformation much more easily. Currently, practical efficiency of the dielectric elastomers now ranges from 18 to 26%, with the possibility of reaching 60% through charge recovery [9].

In the paper by Tay and Lim [4], their 2D flapping wing simulations show great improvements in the efficiency, lift and thrust with their prescribed symmetrical and non-symmetrical flexibility. However, their study is restricted to 2D only. In this study, we intend to use similar flexibility strategies as used in their paper and extend them to 3D. The objective of the current paper is to improve the performance of flapping MAV (FMAV) through prescribed flexibility. The Re is fixed at 5,000 since we are only interested in FMAV in this regime. Due to the large computational requirement

E-mail address: tsltaywb@nus.edu.sg.
[1] Temasek Laboratories, T-Lab Building 5A, Engineering Drive 1, #02-02 Singapore 117411.

http://dx.doi.org/10.1016/j.ast.2016.05.026
1270-9638/

Nomenclature

a_f	flex degree either at the leading or trailing edge	t	time
$a_{lf/tf}$	flex degree at the leading / trailing edge	t_0	time the wing starts to flap
c	chord length	T	period
c_l	lift coefficient	Tr	torque
c_t	thrust coefficient	u	velocity vector
dx	minimum grid length	U_∞	far field incoming velocity
f	non-dimensional flapping frequency	x_{fc}	flexing center location
F	output force from solver	x_{rot}	center of pitch rotation
fc	forcing term	x_{tf}	distance from flexing center to point on wing
h_{lf}	deformed displacement at the leading edge	α_0	pitching amplitude
h_{tf}	deformed displacement at the trailing edge	β_{tf}	flex angle for the trailing edge
p	pressure	η	propulsive efficiency
P_{in}	power input	θ_0	flapping amplitude
PI	performance index	ϕ	phase angle
Re	Reynolds number	ω	angular velocity
S	wing area	ρ	density of air

for 3D simulations in this study, it will be restricted to chordwise flexibility and the parameters involved are the flex location and the degree of flexibility. The bulk of the simulations will also be run at lower resolution, as will be explained later. The solver used is the 3D immersed boundary method (IBM) [10] Navier–Stokes solver. The motivation for using IBM is that the simulations require large wing deformation, a situation which the IBM is very suited for. Details about the methodology will be discussed in the later section. The result will be analyzed in terms of the thrust (c_t), lift (c_l) coefficient, propulsive efficiency (η) and power input, while the flow itself will be visualized by means of pressure and Q criterion contour plots.

2. Numerical method

2.1. Solver

The solver used in this study is the 3D IBM Navier–Stokes solver. The IBM approach is chosen because the wing undergoes large motion and deformation. If using an Arbitrary Lagrangian–Eulerian (ALE) formulation, it may be difficult to maintain grid quality under this condition. In IBM, stationary Cartesian grids are used and the body of interest simply cuts through the grid. To simulate the presence of the body, the Navier–Stokes momentum equation is modified through the addition of a forcing term fc, as shown in Eqn. (1).

$$\frac{\partial u}{\partial t} = -u \cdot \nabla u + \frac{1}{\mathrm{Re}}\nabla^2 u - \nabla p + fc, \tag{1}$$

where $\boldsymbol{u}$ is the velocity vector, t is the time, p is the pressure and Re is the Reynolds number. Equation (1) has been non-dimensionalized using the far field incoming velocity (U_∞) and root chord length (c) as the reference velocity and length. Accordingly, the calculation of Re is based on U_∞ and c.

There are many ways [10] to calculate the value of fc. We have chosen the discrete forcing approach based on a combination of the methods developed by Yang and Balaras [11], Kim et al. [12] and Liao et al. [13], whereby fc is provisionally calculated explicitly using the 1st order forward Euler and 2nd order Adams Bashforth (AB2) schemes for the viscous and convective terms, respectively, to give:

$$fc^{n+1} = \frac{u_f - u^n}{\Delta t} + \left(\frac{3}{2}C^n - \frac{1}{2}C^{n-1}\right) - D^n + \nabla p^n, \tag{2}$$

where C and D are spatial operators containing the convective $\nabla \cdot uu$ and viscous $\nabla^2 u/\mathrm{Re}$ terms respectively.

To solve the modified non-dimensionalized incompressible Navier–Stokes equations (Eqn. (1) and Eqn. (3)), we use the finite volume fractional step method, which is based on an improved projection method [14].

$$\nabla \cdot u = 0 \tag{3}$$

For the time integration scheme, the convective and viscous terms use the second order AB2 and Crank Nicolson (CN2) discretization respectively. The convective and viscous spatial derivatives are discretized using the second order central differencing on a staggered grid. In this fractional step method, we first solve the momentum equation to obtain a non-divergence free velocity field. Using this non-divergence free velocity, we then solve the Poisson equation to obtain the pressure field, which in turn updates the velocity to be divergence free. The momentum and Poisson equations are solved using the open source linear equation solvers PETSc [15] and HYPRE [16] respectively. No turbulence model has been added to the solver due to the relatively low Re used in the simulations.

2.2. Force and power calculation

Because the body is now not aligned with the grid, the force coefficients on the body are calculated using the forcing term fc^{n+1} obtained earlier [12]:

$$F_i = -\int_{solid} fc_i^{n+1} dV + \int_{solid} \left(\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j}\right) dV, \tag{4}$$

where V is the volume of the wing.

In this study, we assume that the forces F_i obtained are equivalent to the 3D force coefficients $c_t(\frac{F_t}{0.5\rho U_\infty^2 S})$ and $c_l(\frac{F_l}{0.5\rho U_\infty^2 S})$ for simplicity since comparisons are only performed within the study. Power input for the flapping is calculated based on the angular velocity of the wing and the torque it produces. Torque (Tr) is given by

$$Tr = \int (r \times F) dV, \tag{5}$$

where $\boldsymbol{r}$ and $\boldsymbol{F}$ are the distance of the point from the rotating axis and the output forces respectively. All the grid cells' torques within the volume of the wing are summed up to give the total torque Tr.

Power input is then given by:

$$P_{in} = -Tr \cdot \omega, \tag{6}$$

where $\boldsymbol{\omega}$ is the angular velocity of the wing.