

Numerical Simulations of Plunging NACA 0012 Wing for MAV Propulsion

Ezzeddin M. Elarbi and Ning Qin

Department of Mechanical Engineering, the University of Sheffield, Sir Frederick Mappin Building, Mappin Street, Sheffield, S1 3JD, UK

Abstract

Computational simulations of a plunging NACA 0012 wing are conducted to study various flapping kinematics such as plunging frequencies and amplitudes for Garrick reduced frequency of 0.2 to 3. A coupling framework of Navier-Stokes equations based on control volume method and dynamic mesh techniques deploying layering and sliding interference concepts is used. The wall bounded and farfield computational domains with water/air are investigated in correlation with the validation study based on the water tunnel experiment by Heathcote et al. A reasonable agreement is found from quantitative and qualitative comparisons between the simulation with the experiment at Reynolds number of 30,000 and Garrick reduced frequency of 1.82. The results of the water confined domain are in closer agreement with the tunnel blockage of the experiment in contrast with other cases (the air wall bounded, air farfield and water farfield domains). The effect of the wall boundary should not be ignored in the validation study adopting experiments based on cramped sidewalls tunnel. For the flapping frequencies of 0.1916 - 2.8734Hz (Garrick reduced frequency of 0.2 to 3) with the amplitude-to-chord ratio of 17.5% (Strouhal number of 0.202763), a significant increase of the power input at frequencies higher than 1.7432Hz due to the enhance of the lift is detrimental to the propulsive efficiency. For flapping at the frequency of 1.7432Hz (Garrick reduced frequency of 1.82) and varying plunging amplitude-to-chord ratio from 7.5% - 27.5% (Strouhal number of 0.08689 - 0.31863), it does not show any further improvement of the efficiency even though the thrust significantly is enhanced. A propulsive efficiency as high as 18% is found due to the flapping at frequency of 1.7432Hz and amplitude-to-chord ratio of 17.5%.

Nomenclature

2D	two dimensional
3D	three dimensional
A_y	vertical flapping amplitude (m)
AFFD	air farfield domain
AWBD	air wall bounded domain
BC	boundary condition
c	chord (m)
CL	lift coefficient
$\overline{CP}$	mean power input coefficient
CT	thrust coefficient
$\overline{CT}$	mean thrust coefficient
f	wing beat frequency (Hz)
F_H	horizontal force (N)
F_V	vertical force (N)
FWMAVs	Flapping Wing Micro Aerial Vehicles
i, j and k	unit vectors in x, y and z directions
LE	leading edge
LEV	leading edge vortex
LS	lower surface
M	Mach number
MAVs	Micro Air Vehicles
ni	number of iterations (number of time steps)
N_C	number of flapping cycles
PIV	Particle Image Velocimetry
Re	Reynolds number
S	wing surface area (m ²)
St	Strouhal number
T	flapping period (sec)

TE	trailing edge
TEV	trailing edge vortex
t	physical simulation time (sec)
UDF	user defined function
US	upper surface
U_∞, u_∞	free stream speed, incoming air velocity (m/sec)
v	plunging velocity (m/sec)
WFFD	water farfield domain
WWBD	water wall bounded domain
y	plunging motion in vertical direction (m)
y_p	the distance between the wall surface and the adjacent cell centroid of the first normal layer (m)
α_{eff}	effective angle of attack (degree)
Δt	time step size (sec)
η	propulsive efficiency
μ	fluid viscosity (kg/(m·sec))
ρ	fluid density (kg/m ³)
τ	non-dimensional flapping time ($\tau = 0$ and 1 at the onset and at the end of the flapping cycle respectively)

Subscripts

$(\cdot)_d$	down-stroke
$(\cdot)_u$	up-stroke

1. Introduction

Miniature but propulsively efficient vehicles which can fly in the similar range of Reynolds number (Re) to small birds ($O(10^4)$) are still sought since the pioneering effort made by DARPA in 1996 to design the first prototype of micro air vehicles (MAVs). Conventional propellers offer moderately poor efficiency at these low scales. One of the design possibilities for MAV propulsion systems could be a pure plunging wing implementation to produce the required thrust to such vehicles in forward flight. A range of flapping kinematics based on Garrick reduced frequency (K) of 0.2 to 3 have been investigated to maximise the propulsive efficiency of flapping motion. Initially, five flapping frequencies (f) of 0.1916, 0.9578, 1.7432, 2.1072 and 2.8734Hz were studied with amplitudes-to-chord ratio (y/c) of 17.5%. The propulsive performance was also examined at other four cases of y/c of 7.5, 12.5, 22.5 and 27.5% based on the optimal f which shows the maximum efficiency in the initial study.

Motivated by the interest in further understanding the effect of boundary conditions (BC) on the accurate simulation of flapping wings in correspondence with the experimental results, this work is conducted to have an insight into how close numerical simulation results to experiment when wall bounded and farfield computational domain are used. Typically, farfield domain based on infinite fluid simulation was applied in most recent research on flapping wing concepts[1-5] in comparison with wall bounded tunnel used in the majority of experiments[6]. Noticeable disagreements were found between various CFD simulations and experimental measurements due to mismatching boundaries used during experimentations. For example, under-prediction of thrust force was observed in the simulation of plunging NACA 0012 wing as a result of applying farfield boundaries[1, 2]. The reason behind these discrepancies may be due to the difference between two approaches, i.e., experimental devices can generate errors leading to some uncertainty in the results and different numerical schemes produce various approximations. However, assuming these issues have minimal contributions to these discrepancies, the blockage in the experiment and the corresponding size of computational space are the obvious remaining factors which may influence the outcomes between the experiment and simulation.

The experiment conducted by Heathcote et al.[6] is considered in the validation part of this work. Figure. 1 schematically shows the experimental setup of water tunnel rig which was used to study the propulsion performance of a rectangular NACA 0012 wing flapping in a pure heaving at $K = 1.82$ and $Re = 30,000$. The wing has an aspect ratio of 3 where the chord and span of 100mm and 300mm respectively. The tunnel has a free-surface closed loop test section with a dimension of 381mm wide $\times$ 508mm deep. The wing was designed to be stiff in chordwise direction. One of the cases was considered by Heathcote and collaborators had an inflexible wing designed to be as stiff as possible. The heave amplitude-to-chord ratio of 17.5% was applied so that the Strouhal number (St) is 0.234. The wing was mounted vertically with a small clearance between its root and the splitter plate of 2mm and the gap between its tip and the floor of tunnel is $5c/3$. A force balance was used to measure the aerodynamic forces whereas a Particle Image Velocimetry (PIV) system was utilized to trace the flow field. Since the water tunnel used in the experiment can only drive a flow speed in the range of 0 to 0.45m/sec, which designates $Re < O(10^5)$, assuming a laminar flow in the simulation may be an adequate regime according to the experiment. Laminar flow assumption was also imposed in the previous simulations by Shyy et

al.[1, 2] and Liu[3]. Integrating various turbulence models in those studies did not show significant difference with the results obtained from the laminar solution.

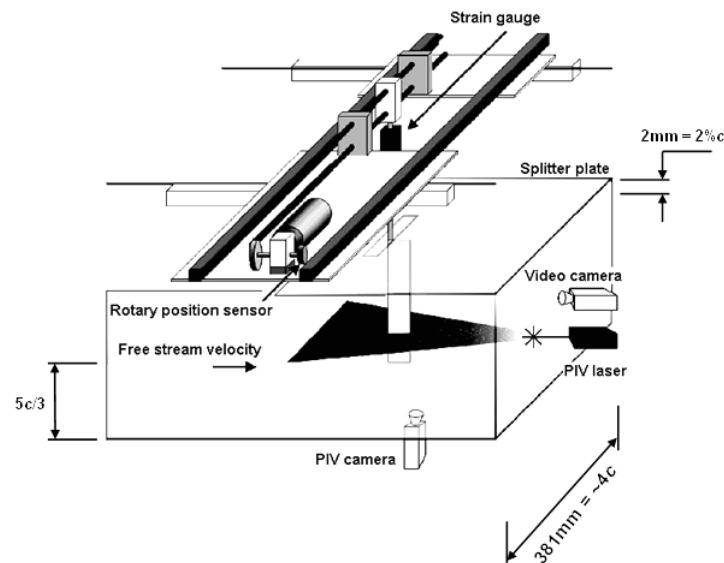


Figure 1 Water tunnel set-up and flapping wing rig system used to experiment on a pure heaving motion of three dimensional (3D) NACA 0012[6].

Lian[7] carried out a numerical investigation on boundary effects for the study of flapping wing undergoing pitching motion. A rectangular SD7003 wing with an aspect ratio of 3 was used. The side wall effect was discussed where the wing mounted wall to wall was constructed to compare to two dimensional (2D) simulation. No significant difference was found between 3D and 2D results obtained for the computations of lift and drag force coefficients. It is also revealed that the flow field at the middle of span is a fairly appropriate approximation to 2D case. In addition, the domain size influence was studied by using a farfield and wall bounded domain to simulate unbound and confined flow conditions. Two distinct conclusions were drawn from this assessment; the farfield domain should be used at the high reduced frequency, e.g. $K \geq 3.93$, to avoid results affected by the wrong size of computational space; and the instantaneous force calculations show a considerable sensitivity to the domain size, particularly at $Re = 10,000$, see Fig. 2. However, no comparison with experiments was shown in this study.

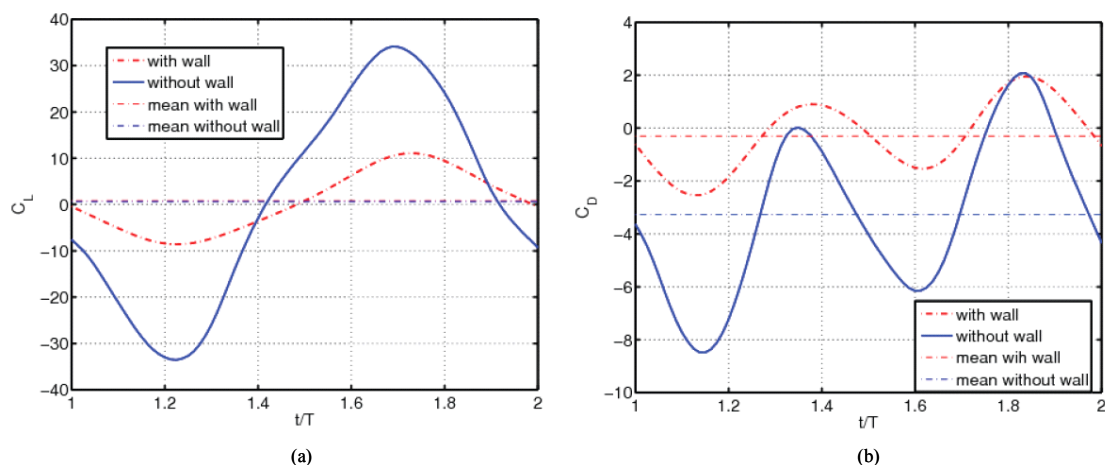


Figure 2 BC comparison without and with top and bottom walls **a)** Lift force coefficient **b)** Drag force coefficient[7].

This paper pays much attention to investigate the BC similarities between the simulations and experiment yielding very close results in the case of flapping wing studies. Thus, fully unsteady flow simulations have been

conducted using the coupling framework of Navier Stokes equations (NSE) solver integrating pressure-based finite volume method and dynamic mesh (DM) techniques deploying layering and sliding zone interference features. At the same Re and K of 30,000 and 1.82 respectively as those applied in the water tunnel experiment[6], a fully rigid NACA 0012 wing performing a pure plunging with forward flight speed based on this Re . A decent agreement was shown by the simulation to experiment when wall bounded computational domain considered with water. A degree of confidence is therefore conferred in the computation even though approximately 6% uncertainty in the efficiency data readings of both thrust and lift aerodynamic forces was found in the experiment[6]. A fairly close agreement has been obtained with the experiment for the shedding vortices downstream of the wing trailing edge during the onset and mid of down-stroke. A reasonable correspondence has been achieved from the second flapping cycle between the computed thrust with the experiment. The average thrust coefficient of 0.2215 is found from the computation compared to 0.21 measured in the experiment. A propulsive efficiency as high as 18% is found at $f = 1.7432\text{Hz}$ and $y/c = 17.5\%$. Heathcote S. et al[6] suggested that introducing an intermediate degree of spanwise flexibility as observed from birds or bats could be beneficial to minimise the power input and thus maximise the efficiency. Alternatively, plunging at some wing incidences (angle of attack > 0) may enhance the efficiency as a significant augmentation was found in both the thrust and lift[8, 9]. Propulsive efficiency of 40% was computed by Tuncer et al. with tandem configuration of flapping NACA 0012 wing at $K = 0.75$ and $y/c = 0.2$ [10]. However, the MAV propulsion systems may still benefit from plunging rigid wing mechanisms to produce the required thrust forces for forward flight at a reasonable implementation of flapping kinematics, not particularly greatly high frequency and amplitude to avoid the high power input requirements.

The present paper is organised as follows. Section 2 discusses the methodology followed here including the grid generation, flow solver numerical methods, dynamic mesh technique and the solution convergence. Section 3 gives the discussion of the results involving a verification study of a refinement of grid and time step, validation of aerodynamic forces and flow field against the experiment. Section 4 includes the study of flapping kinematics at different frequencies and amplitudes. Section 5 summarises the present findings showing the final conclusion statement reached of the whole paper.

2 Numerical methodology

This section gives a brief illustration of the method followed to simulate a plunging NACA 0012 wing based on the kinematics used in the experiment[6]. The aspects of computational grid are first discussed. Then, the flow solver methods are outlined in brief with short discussion of moving mesh technique, boundary conditions and convergence.

2.1 Grid details

Two sizes of computational space were made corresponding to farfield and wall bounded domains. The former is shown in Fig. 3 where the computational domain was made large enough so that the dimensions in the chord-, cross- and span-wise directions $20c \times 20c \times 11c$ respectively. Liu[3] used O-type structured grid with $18c$ computational domain with the fine grid of $45 \times 54 \times 91$ and grid spacing adjacent to the wing surface of 0.001. A large domain of $20c \times 20c \times 10c$ was also used by Shyy et al. with a structured multi-block C-O grid ($120 \times 56 \times 60$)[1,2]. A far field boundary condition was used to set the flow velocity of forward flight simulation.

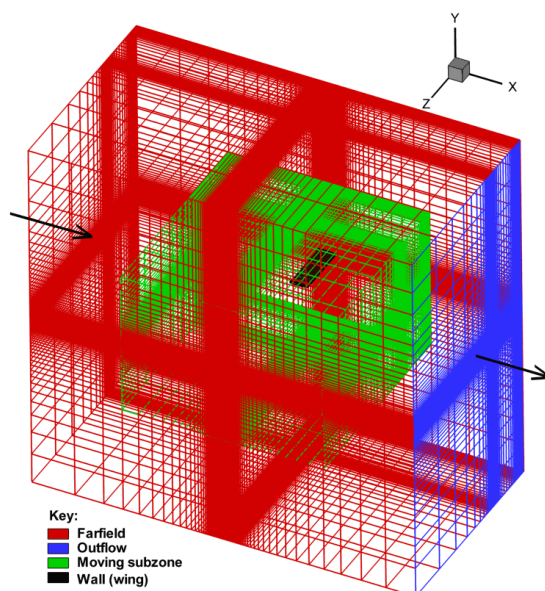


Figure 3 Farfield computational domain created with multi-block structured grid used to simulate infinite fluid boundary conditions similar to free flight.

The wall bounded domain is given in Fig. 4 where its size was made to be as consistent as possible to the tunnel-blockage of the experimentation[6]. Because the tight size of the computational domain, moving zone prevents the wing to be displayed clearly. Therefore, the computational domain is shown twice in Fig.4; one

includes the moving zone and the other does not. The dimension of approximately $20c \times 4c \times 5c$ was used in the chord-, cross- and span-wise directions respectively. The sidewall clearance was applied at the wing root of $2\%c$ and the wing tip of $56\%b$ ($5/3c$) in order to correspond with the tunnel setup used in the experiment[6], refer to Fig. 1. Because the tunnel has a close loop nature and the dimension in the streamwise direction is not explicitly given, the x- size of domains are made big enough to exclude the numerical error due to the reversed flow experienced at the outlet boundary. Also, it was aimed to make the same x-dimension as in the case of the farfield domain ($20c$).

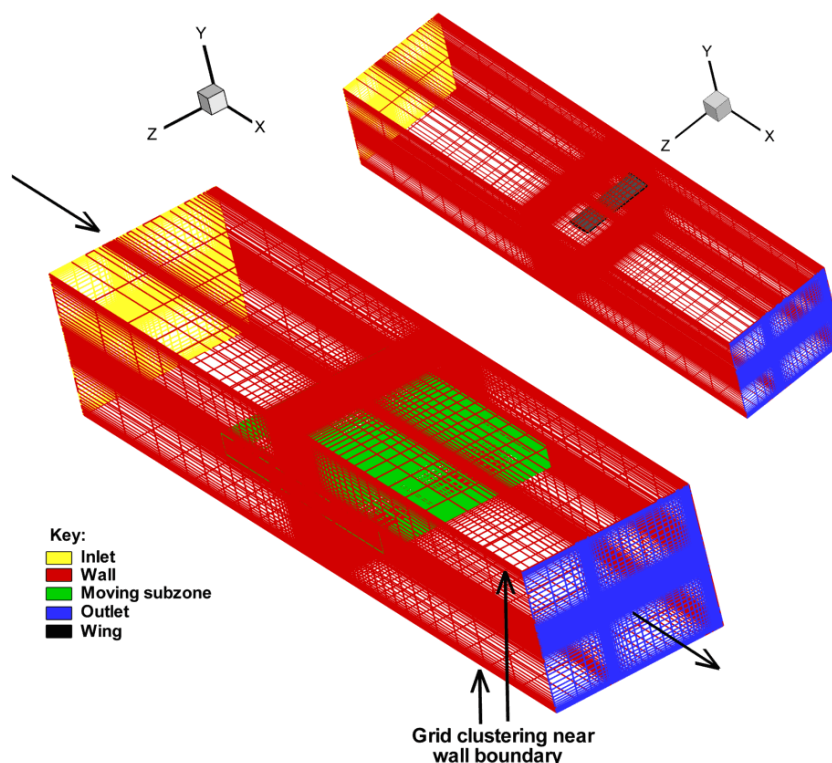


Figure 4 Wall bounded computational domain created with multi-block structured grid used to simulate confined fluid boundary conditions similar to the tunnel blockage[6].

To ease meshing the computational domains, they were decomposed into 154 and 108 subdomains for the farfield and wall bounded domains respectively. For the sake of convenience to use dynamic layering and sliding zone strategies of DM method, H-type grid was applied throughout all the computations with the relevant consistency of the mesh statistics. It consists of fully-structured hexahedral mesh and pave mesh (distorted hexahedron) along the span of the tip and root of the rectangular NACA 0012. No remeshing or deformation was used in conjunction with the DM technique. The computational domain was meshed by using Gambit 2.2.30 grid generator where the wing was placed at the centre of the computational space. A slightly fine grid was chosen by the process of grid sensitivity analysis which comprises about one million cells (912776 elements = 829400 hexahedrons plus 83376 paves). The hexahedral cells were created so that 130, 110 and 58 elements distributed in the parallel, transverse and lateral directions respectively with regard to the stream flow. 2400 cells were used to mesh the wing (60 chordwise $\times$ 40 spanwise elements). These elements were clustered near the tip, root, leading and trailing edges of the wing as shown in Figs. 5 and 7b.

Exponential tangent distribution was used along the radial direction which has less effect on the solution compared with hyperbolic approach[7]. The main requirement of generating the computational grid was to obtain a fine distribution of grid to minimise the change of flow variables from one cell to another due to the large flow gradient. Thus, algebraic equation was used for the boundary nodes. However, the density of grid uniformly decreased as moving far away from the wing surfaces using Laplace equation for the interior nodes for the seek of economising the computational resources with the expansion factor of 1.125. Two different density grids were generated and their influences were studied on the results obtained in this work, the reader is referred to section 3. The first cell (0.00058) in the boundary layer out of 20 points perpendicular to the wing surface was created within less $0.5\%c$ to adequately resolve the flow passage where there was expected a steep velocity

gradient at the wall when the wing is plunging. The resolution of the laminar boundary layer (i.e., spacing mesh adjacent to near wall) was chosen to obey the following relation to accurately compute wall shear stress[11].

$$y_p \frac{\sqrt{Re}}{x} \leq 1 \quad \dots(1a)$$

where y_p is the distance between the wall surface and the adjacent cell centroid, x is the distance along the wall which was taken to be c and $Re = \rho u_\infty c / \mu$. Note that, matching the same Re the mesh clustering near the wall surface in water/air computational domains is identical. Equation 1a may be written in another expanded form as

$$y_p \sqrt{\frac{\rho u_\infty}{\mu c}} \leq 1 \quad \dots(1b)$$

where u_∞ is the free stream velocity, μ and ρ are the fluid viscosity and density respectively. These are tabulated below for water and air

Input data	Air	Water
Free stream velocity u_∞ (m/sec)	4.38937	0.3009
Fluid density ρ (kg/m ³)	1.223	998.2
Fluid viscosity μ (kg/(m sec))	1.7894×10^{-5}	1.003×10^{-3}

Table 1 fluid properties and the free stream velocity based on $Re = 30,000$

The mesh was also clustered near the walls of wall bounded domain where the resolution of the grid normal to the side walls was obtained from Eq.(1a) as well. About 0.1% of the longitudinal dimension of the computational domain (i.e., $x = 20c$) was found adequate to model the first layers (0.001826) normal to the walls. 10 layers were made to solve the flow adjunct to the side walls with the expansion factor of 1.2.

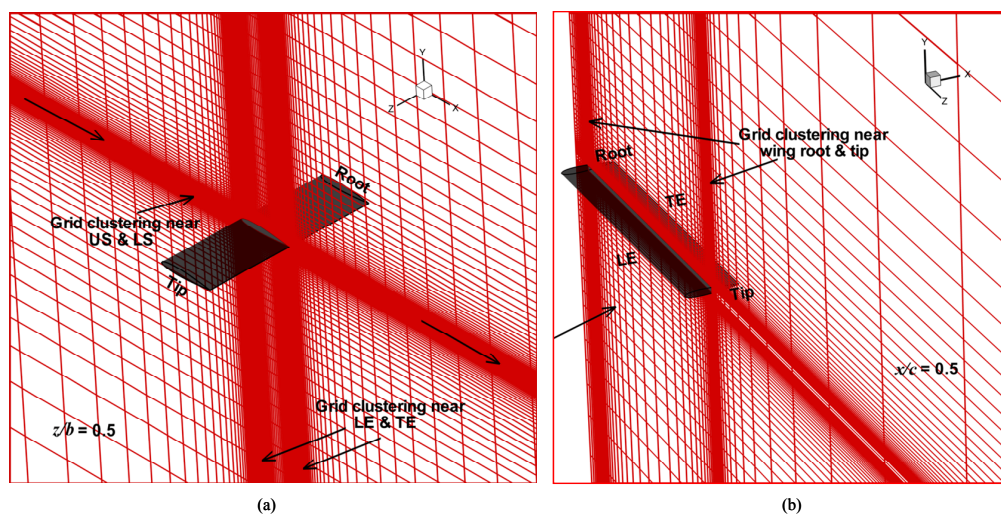


Figure 5 Zoomed-in view in the vicinity of the wing shown by (a) the z plane at mid span $z/b = 0.5$ and (b) the x plane at mid chord $x/b = 0.5$.

2.2 Computational flow solver methods

With the considerations of solving 3D flow, NSEs governing unsteady flow were solved using the finite volume discretisation method (FVM) in an inertial reference frame fixed to the moving wing. Since Mach number $M < 0.3$ and $Re = 30,000$, the assumption of incompressible laminar flow may be here justified to the simulations intended. These equations were on the basis of the Arbitrary Eulerian-Lagrangian framework which allows the flow computations dealt with on a fixed Eulerian mesh and the grid deformation obtained in the fashion of lagrangian approach. In order to eliminate artificial mass sources, the deformation of cell volume due to the moving boundary was enforced to fulfil Geometry Conservation Law (GCL)[1, 2] where the grid velocity was considered throughout the solution of NSEs.

Second-order discretisations were used for the spatial part, and the pressure-velocity coupling was via the fully-implicit scheme for the pressure-linked equations (the SIMPLE algorithm which is widely applied in many past studies[12]). The time derivative was discretised using two-point backward differences in first order implicit formulation. A coupled NSEs and DM framework was utilised to solve the unsteady flow caused by flapping wings based on the Advanced Time Scheme (ATS) provided in the pressure based solver (Fluent 6.3.2). The momentum equations were solved and the resulting flow field was corrected by pressure to fulfil the continuity constraints. Then, the solution was advanced to the next time step and so on until periodic solution or a maximum time step is reached. The smoothness and orthogonality of structured grid is compatible with the iterative procedure. Thus, no heavy under relaxation parameters were required to encourage the convergence of solution which could affect the rate of computational efforts.

2.3 Boundary conditions

Setting the boundary condition (BC) appropriately at each part of the domain boundary is the initial task toward solving the governing equations, comprising the model, of flow past oscillating wing. A free turbulence was imposed at the incoming flow due to laminar flow assumed. To eliminate the effect of the downstream boundary on the upstream flow field, outflow BC was set at the exit boundary which located at suitably far distance from the object (wing) expecting no circulating flow at the exit boundary. The extrapolation from the inner solution is used to provide the flow field properties at the exit. A time-varying BC of rigid body wing was achieved via user defined function (UDF) integrated in DM method. The no- penetration boundary condition[3] is satisfied by

$$V_c = v(t) \quad \dots(2)$$

where V_c is the velocity of the boundary curve of wing surface and $v(t)$ is the instant velocity of moving wing.

The no-slip BC was applied on the wing surface and the side wall of the confined domain [13]

$$u_\infty = 0 \text{ and } \nabla \cdot u_\infty = 0 \quad \dots(3)$$

Constant free stream velocity was imposed on the far field boundary of the farfield domain and on the inlet of wall bounded domain. Therefore, the initial flow field condition of numerical simulation[3] is applied as

$$U_{t=0} = U_\infty = u_\infty i + v_\infty j + w_\infty k \quad \dots(4)$$

where i , j and k are the unit vectors of Cartesian coordinates in the x , y and k directions respectively.

$U_{t=0} = U_\infty = u_\infty$ and $v_\infty = w_\infty = 0$.

2.4 Plunging wing kinematics

A DM technique was required to be integrated with the flow solver in order to model the flapping wing undergoing a pure heaving motion (angle of attack $\alpha = 0$) in the field of uniform constant velocity simulating forward flight. Table 2 shows the flapping kinematics based on physical flow characteristics used in the experiment[6].

Kinematics and flow properties	air	water
Flapping frequency f (cycle/sec)	25.3871	1.7439
Flapping period T (sec)	0.0394	0.5734
Flapping amplitude A_f (m)	0.0175	
Reynolds number ($Re = \rho u_\infty c / \mu$)	30,0000	
Carrick reduced frequency ($K = \pi f c / u_\infty$)	1.82	
Strouhal number ($St = 2f A_f / u_\infty$)	0.234	

Table 2 flapping kinematics and physical flow characteristics

UDF C code script was written to manage a moving mesh with the prescribed kinematics. Expert motion-tracing software (RealViz MatchMover Pro 3.1) proved that a cosine fit representation defines the motion of plunging wing[6]. However, the motion of wing was prescribed in the simulation by Shyy et al. with a sinusoidal harmonic function[1,2]. The centre of gravity of moving wing was updated at each time step based on the linear velocities which were used in the macro DEFINE_CG_MOTION. These were defined in UDF associated with DM feature. The equation was applied to swap the wing in the y direction is given by

$$v(t) = -\omega A_y \sin(2\pi ft) \quad \dots(5)$$

where A_y is the vertical flapping amplitude, $\omega = 2\pi f$ (the angular flapping frequency), f is shown in Table 2 for water and air and t is the flapping instant where the flapping period $T = 1/f$. Note that, the integration of Eq. (5) yields the plunging translation along the stroke plane (vertical plane y).

$$y(t) = \int v(t) dt = A_y \cos(2\pi ft) \quad \dots(6)$$

The minus was shown in Eq. (5) to steer the wing to first undergo the downstroke phase of motion. Figure 6 shows the flapping trajectory for heaving motion of NACA 0012 wing.

The physical time of simulation was updated each time step based on the following expression

$$\left. \begin{aligned} t &= t + \Delta t \\ \Delta t &= \frac{T}{ni} \end{aligned} \right\} \quad \dots(7)$$

where Δt is the time step needed to be cautiously selected for the consideration of mesh deformation issues, ni is the number of iterations (number of time steps) required for the flapping simulation, which is typically chosen from the time step sensitivity study. At the beginning of the simulation, $t = t_0 = 0$ then t sets to the previous time. However, Δt should also met the CFL stability condition regarding with the nonlinearity of PDEs when explicit scheme formulation is applied.

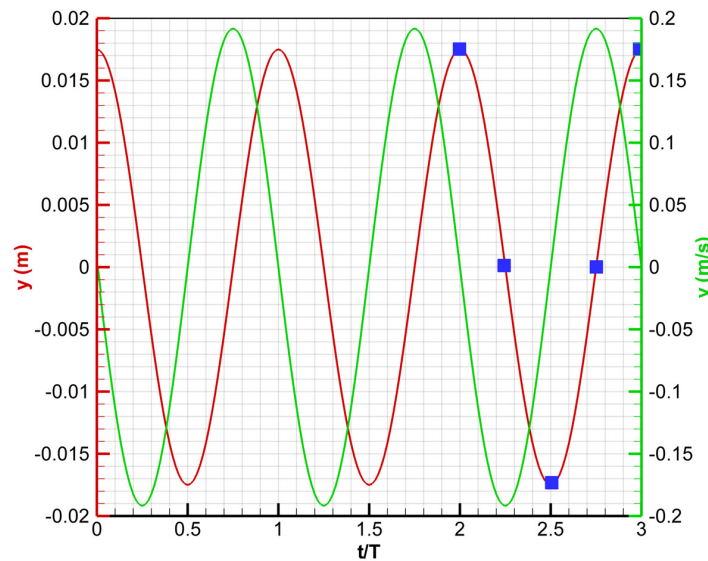


Figure 6 Plunging velocity and amplitude of a vertically heaving NACA 0012 wing given as a function of the normalised flapping period t/T for three cycles. Blue squares indicate the instants at which flow field structure extracted, see Figs. 21 and 22.

On the basis of the prescribed kinematics given in Fig. 6, the grid was warped to accommodate with the plunging rigid wing. Figure 7b represents grid elements clustering around 3D NACA 0012 wing. The utmost position of moving grid and wing ($y = 0.0175\text{m}$) is shown by Fig. 7a whereas the least position of motion ($y = -0.0175\text{m}$) appears in Fig. 7c. These are the nearest positions reached by the wing with respect to the top and bottom boundaries of the computational domain. These are in harmony with the positive and negative peaks illustrated by the red line in Fig. 6. It was found that for proper moving wing (avoiding the deterioration of grid cells and thus preserving the integrity of grid) the translation in one time step should not be more than half the cell size adjacent to the moving boundary.

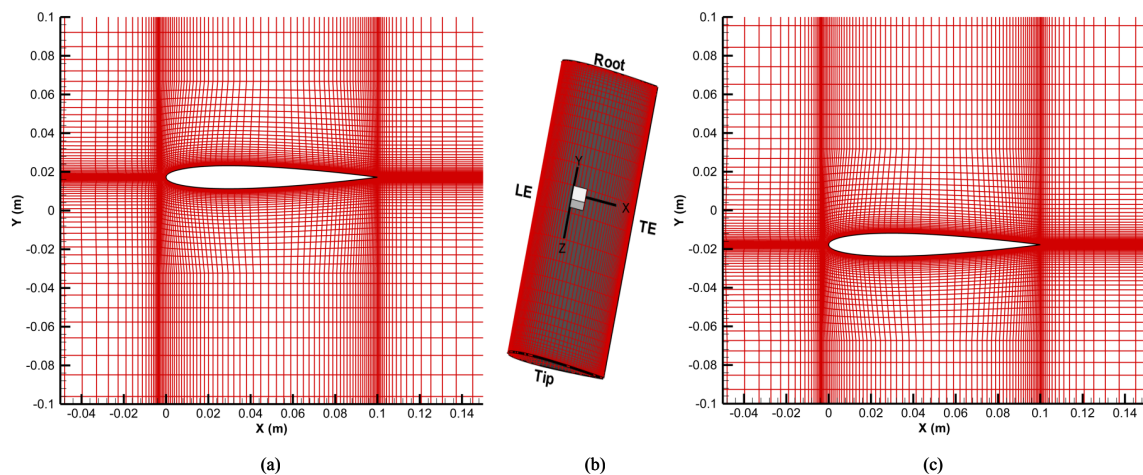


Figure 7 Grid of z plane at $z/b = 0.5$ shown for $y/c = 0.175$ a) At the onset of downstroke motion (positive maximum position) b) 3D NACA 0012 surface mesh c) At the end of the downstroke (negative maximum position).

2.5 Dynamic mesh method

A combination of two techniques of DM method was applied, namely, dynamic layering and sliding zone features, as schematically shown in Fig. 8. The volume mesh is updated in the deforming regions subject to the motion defined the boundaries.

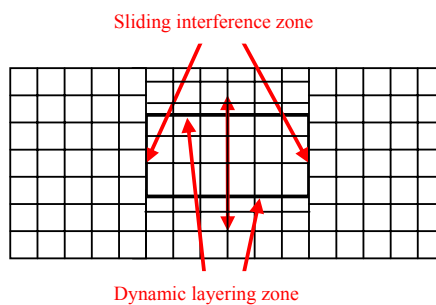


Figure 8 Coupling feature of dynamic layering and sliding interference zones for moving grid.

As demonstrated in Fig. 8, a sliding interference capability of DM method was applied along the sideways of the direction of motion between the moving zone and its shadow. This feature allows to transition from the adjacent non-deforming cells to the deforming cells. Dynamic layering was wall bounded to function only in the direction of motion. This means that it needs to be applied to cells adjacent to a moving boundary to maintain the quality of deforming grid by adding or removing layers of cells. This technique was found suitable for quadrilateral/hexahedral type of grid. Therefore, the layer of every moving zone either expands or collapses on the same side. Knowing the height of the layer adjacent to the moving surface as diagramed by Fig. 9, the following two expressions are applied in this approach to split and/or merge that layer with the layer of cells next to it.

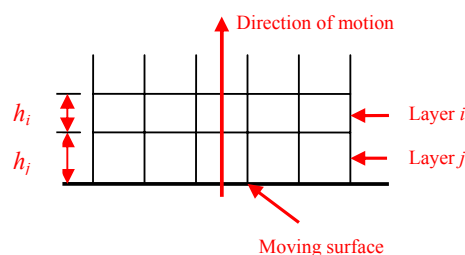


Figure 9 Illustration of dynamic layering method.

The first expression is concern with the heights of expanding cells in layer j (h_j) neighboring to moving surface (wing) which are permitted to increase as long as the next condition hold

$$h_j < (1 + \alpha_s) h_{j0} \quad \dots(8)$$

where h_{j0} is the initial height of cells in layer j and α_s is the layer split factor. However, the cells are split when the above condition is void due to the large deformation so that the ratio of new cell heights is α_s everywhere.

The second expression is dealing with the heights of compressing cells in layer j (h_j) neighboring to moving surface (wing) which are permitted to compress on condition that

$$h_j > \alpha_c h_{j0} \quad \dots(9)$$

where α_c is the layer collapse factor. However, the cells are merged into the layer of cells (layer i) next to the compressed cells when the above condition is invalid due to the large deformation.

2.6 Simulation convergence

The convergence histories of unsteady simulations conducted to investigate the domain boundary issues are shown in Fig. 10. It is intended to present the comparison of residuals of flow field among wall bounded and farfield domain when air and/or water were used. Overall, reasonable convergences were achieved with all the simulation cases of flapping wings. As much as possible reduction in the simulation residuals was painstakingly obtained, which was found in many cases a drop until a five order of magnitude at each time step.

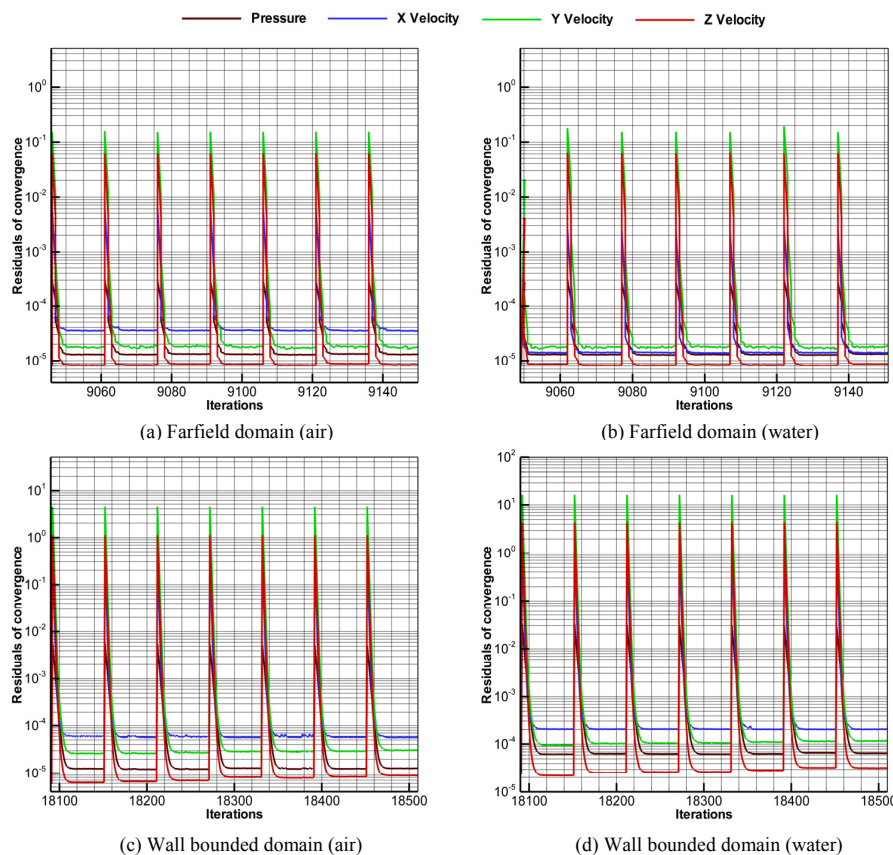


Figure 10 Residual histories of simulation convergences during the second period of plunging NACA 0012 wing with four cases of domain boundary studies.

As may be noticed, the convergence of simulation of using the farfield domain (Figs 10a and b) consumes less an amount of iterations than with applying wall bounded domain (Figs 10c and d). In fact, there was 30 pseudo iterations needed to enforce the simulation residuals to converge at each time step in the case of farfield

domain compared with about 60 pseudo iterations in the state of the wall bounded domain. The physical iterations of 300 (baseline case after time step effect study) was found adequate to obtain converged solution. It seems that the quality of convergence was better in the case of the farfield domain than blockage-tunnel like domain. This may refer that the wing at the end of each stroke is very close to the wall side of the wall bounded domain which affects the convergence of flow quantities, particularly x- and y-velocities. It is also shown that the fluid media (air/water) has a considerable influence on the reduction in residuals.

3 Results and discussions

This section discusses in brief the verification and validation of the results. Followed by further illustrative analyses of the significant findings are given.

3.1 Grid and time-step effects

To exclude numerical artifacts from the solution, the effects of grid resolution and time step size refinements on solutions were studied as shown in Fig. 11. Two different density grids of 912776 and 723691 cells were used as given in Fig. 11a with 300 time steps (physical simulation time = 0.00192 sec/time step). Figure 11b shows the effect of increasing the time step by 100 steps (200 - 400 time steps) based on using the fine grid. Force coefficients were used to assess the independence of numerical solution with respect to grid density and time step refinements. Drag/thrust (horizontal) and lift (vertical) force coefficients (CT and CL), measured in the inertial coordinates x and y respectively, are defined by

$$CT = \frac{2F_H}{\rho U_\infty^2 S} \quad \dots(10)$$

$$CL = \frac{2F_V}{\rho U_\infty^2 S} \quad \dots(11)$$

where F_V and F_H are the vertical and horizontal force components respectively which were computed by summing the pressure and viscous forces. S is wing surface area ($S = c \times b = 0.1 \times 0.3 = 0.03 \text{ m}^2$).

The integration of the thrust coefficient was conducted by averaging over a number of converged flapping cycles. Time-averaged (mean) drag/thrust force coefficients $\overline{CT}$ is defined by

$$\overline{CT} = \frac{1}{N_C T} \int_0^{N_C T} CT dt \quad \dots(12)$$

where N_C is the number of flapping cycles.

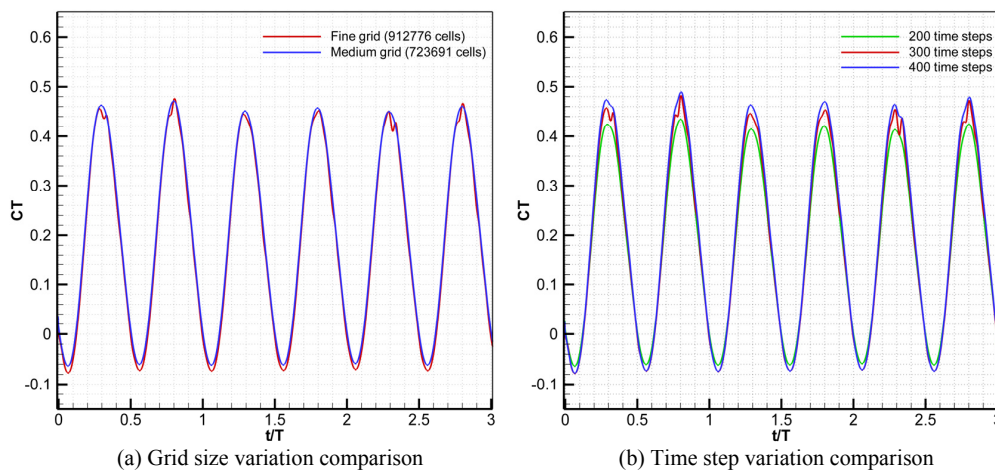


Figure 11 The variation effects of time step and grid size on thrust force coefficient.

An asymptotic behaviour was obtained from executing flapping wing simulation for three cycles. As may be clear from Figure 11, the overall pattern of the force calculations over three flapping wing cycles were not very sensitive to mesh density and time step refinements. Note that from Table 3, the mean thrust force coefficients $\overline{CT}$ calculated from various grid resolution and time step refinements show insignificant variations.

The fine grid was employed in the following computations in order to obtain high quality visualisations of the flow field structure during the flapping motion. Because of the CPU requirement, the medium number of time steps of 300 (time step of 0.00192sec) was deemed adequate considering computational costs. However, this seems much finer than those was applied by Liu (time step of 0.01sec)[3] and by Shyy et al. (287 time steps, 0.002sec) per cycle of oscillation of the wing[1, 2].

Grid size (# cells)		# Time steps	
$\overline{CT}$		$\overline{CT}$	
912776		200	0.19463
723691		300	0.21905
		400	0.21914
(a) Grid resolution effects		(b) Time step effects	

Table 3 Mean thrust coefficient comparisons among various refinements of time step and grid resolution

3.2 Force coefficient validations

This section discusses the accuracy of the results achieved from the simulations applied here in terms of validations with other experimental and computational findings. The boundary effects on the solution were investigated by applying farfield and wall bounded domains with air and water for each case. The following terms AFFD, AWBD, WFFD and WWBD are used to identify air farfield domain, air wall bounded domain, water farfield domain and water wall bounded domain respectively.

First of all, the instantaneous aerodynamic force (lift and thrust) coefficients found here were compared with those computed and measured values by Shyy et al. and Heathcote et al. respectively. Figure 12 presents comparisons of the thrust coefficients whereas Fig. 13 shows comparisons of the lift coefficients. Two observations can be drawn from the current predictions:- (1) the results obtained based on using AWBD and WWBD (particularly WWBD) are asymptotically close to the experimental measurements by Heathcote et al.[6]; (2) the use of AFFD and WFFD (specifically AFFD) show a reasonable correspondence with the past computations by Shyy et al.[1, 2]. These observations may be much clear in Fig. 13 (the lift coefficient vs. normalised flapping cycle). However, a slight discrepancy between WWBD (the closest agreement found) and the experiment might be seen in Fig. 12 (the thrust coefficient vs. normalised flapping cycle), particularly at $\pm$ peaks of the instantaneous thrust coefficient. This could refer to the effect of inertie force in the real water experiment where it was measured that inflexible (infinitely stiff) wing shows the tip-to-root amplitude ratio of 1.13 and phase lag between the tip and root of -4.2° [6]. This is the obvious reason of asymmetry in the experimental data. The computations of Shyy et al.[1, 2] and Liu [3] were based on fully rigid wing, i.e., the tip-to-root amplitude ratio of 0 and phase lag between the tip and root of 0° were applied.

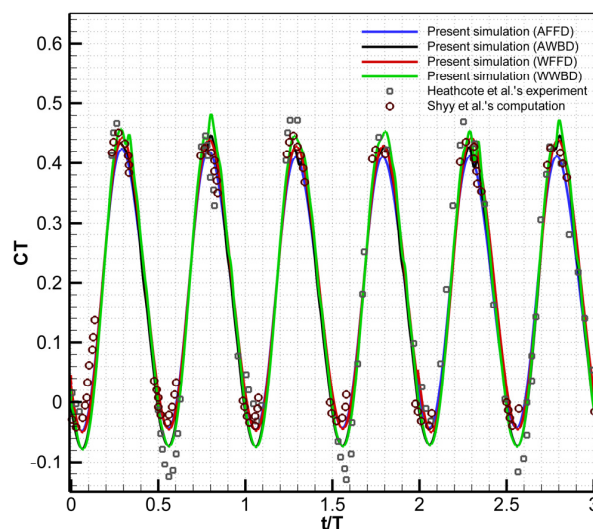


Figure.12 Thrust force coefficient comparisons between the present simulation based on wall bounded and farfield domain (air/water) and the past computation[1, 2] and experiment[6].

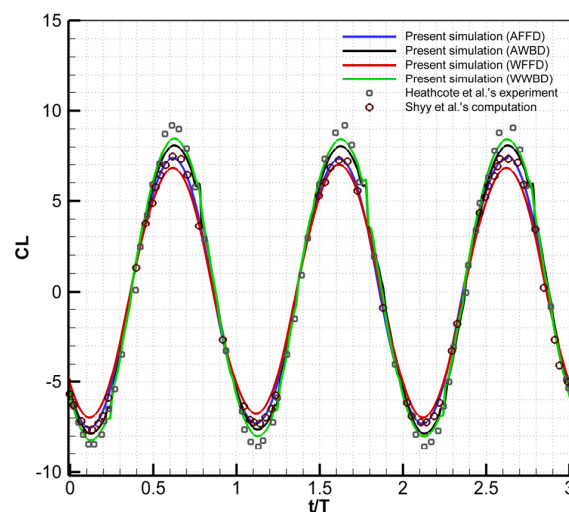


Figure 13 Lift force coefficient comparisons between the present simulation based on wall bounded and farfield domain (air/water) and the past computation[1, 2] and experiment[6].

Note that from Table 4, the mean thrust force coefficients $\overline{CT}$ and percentage of difference $\% \overline{CT}$ with respect to the experiment (baseline case) calculated from various simulation conditions show insignificant variations. A further detailed demonstration is given in the subsequent subsection to understand the reason of differences in the thrust and/or lift obtained from those various cases (AWBD, WWBD, AFFD and WFFD). Overall, the significant difference between various domain boundaries studied is found in instantaneous results, particularly at the peaks of the force calculations as shown in Figs. 12 and 13. The average results are quite comparable as seen from Table 4.

Category of study	$\overline{CT}$	$\% \overline{CT}$
Present simulation (AFFD)	0.1975	5.953
Present simulation (AWBD)	0.1947	7.286
Present simulation (WFFD)	0.19817	5.653
Present simulation (WWBD)	0.21905	-4.309
Heathcote et al. water tunnel experiment[6] (baseline case)	0.21	—
Shyy et al. past computation[1, 2]	0.20	4.762

Table 4 Mean thrust coefficient and percentage of difference with respect to the experiment (baseline case) comparisons between the present simulation based on wall bounded and farfield domain (air/water) and the past computation[1, 2] and experiment[6]

3.3 Flow field validations

In correlation with the force coefficient calculations in Sec. 3.2, the results obtained from different domain boundary investigations are slightly distinguishable apart from those obtained from WWBD which shows better comparable thrust coefficient to the experiment[6], see Table 4. After those quantitative analyses are given in the preceding section, this section presents qualitative comparisons of vortical flow field at two time instants, i.e., at the onset and middle of downstroke phases in the correspondence with the positive peak and zero amplitude positions respectively, see Fig. 6. At these instants of the plunging wing as shown in Figs. 12 and 13, the instantaneous force coefficients (CL and CT) computed by the present simulation are very similar to the experiment and past computation. For this reason the qualitative comparisons are required to see whether farfield (Shyy et al.'s simulation) and wall bounded domains (Heathcote et al.'s experiment, current simulation) result in the same flow field. Instantaneous vortical flow structures of the present simulation based on wall bounded domain with water for the plane at the mid-span location ($z/b = 0.5$) are presented in Fig. 14.

Intentionally, it was aimed to compare the shedding of vortices around the wake (the region behind the trailing edge TE) obtained of this simulation with the experiment[6] and past simulation[1]. A fairly reasonable correspondence amongst these cases was found in the size, location and rotational direction of the trailing-edge vortex TEV shedding downstream the direction of flight, although white and black lines were used for clarity purposes to separate positive and negative vorticity in the experiment. Gray scale was used in the experiment and present simulation whereas rainbow scale with black colour indicating zero vorticity was used by Shyy et al.'s

simulation. A strong pattern of continuously alternating vortices shedding is seen downstream TE. Heathcote et al. found less strong pattern of vorticity for highly flexible wing[6]. Two vortices of opposite sign are visible. The red and white colours of the maximum positive vorticity scales of the Shyy et al.'s simulation and the experiment/present simulation respectively designate the clockwise vorticity. The blue and black colours of the maximum negative vorticity scales indicate the anticlockwise vorticity.

Figure 14a shows spanwise vortical flow at the positive peak position (the beginning of downstroke). This is equivalent to $\tau = 0, 1$ and 2 ($\tau = t/T$, the normalized period) at the first, second and third stroke respectively, see Fig. 6. Leading edge vortex LEV starts shedding in the clockwise direction on the lower surface LS (pressure side) and second separation appears close to the mid chord on LS. No visible large-scale separation occurs on the upper surface US (suction side). TEV shedding observed in the anticlockwise direction. In correspondence with the time history of force coefficients shown in Figs. 12 and 13, CD ($-CT$) increases until the peak value reached (approximately -0.07468) and as the wing moves farther down negative lift increases till the CL peak value reached (approximately -8.0645).

Figure 14b shows spanwise vortical flow at the zero amplitude position (the middle of downstroke). This is equivalent to $\tau = 0.25, 1.25$ and 2.25 at the first, second and third stroke respectively, see Fig. 6. LEV shedding at the beginning of downstroke proceeds toward the mid chord on LS. As the wing reached the middle of downstroke, the shedding vortices downstream TE fades further away which enhances the thrust and lift forces. Again, no visible large-scale separation occurs on US. Correlating this with the time history of force coefficients shown in Figs. 12 and 13, CD increases until the peak value reached (approximately 0.4836) and as the wing moves farther down negative lift increases till the CL peak value reached (approximately 8.4912).

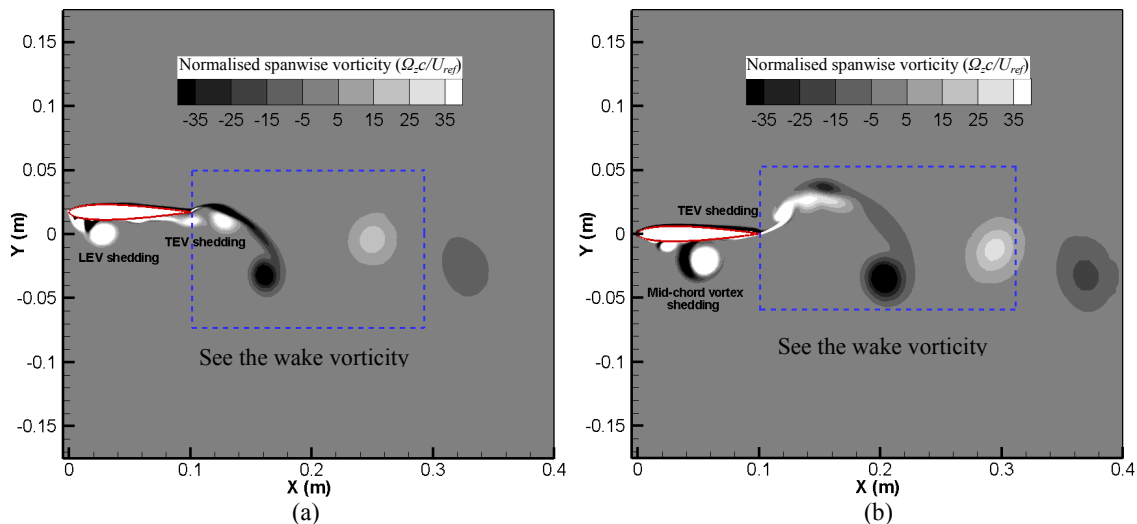


Figure 14 Spanwise vortical flow structures of the WWBD simulation (a) at the beginning of downstroke (b) at the middle of downstroke. The spanwise vorticity Ω_z (or ω_z) is normalized by c/U_{ref} to compare the shedding of vortices with experiment[6] and past simulation[1] around the wake where $U_{ref} = U_\infty$.

Figure 15 shows comparison of spanwise vortical flow downstream TE at the beginning of downstroke between farfield domain (Shyy et al.'s simulation), Heathcote et al.'s water tunnel experiment and WWBD current simulation. As can be seen from this figure, the results given by the experiment and wall bounded domain (present work) are in a good harmony particularly the clockwise vorticity shedding far away downstream TE in comparison with that based on farfield domain. Shyy et al.'s vorticity structure shows slightly larger size of TEV. The existence of side wall boundary of wall bounded domain could have the main effect on pushing the vortices shedding behind the wing to be in a good correspondence with the experiment [6] which is not the case in the farfield domain used in Shyy et al.'s simulation. It seems that this explanation is greatly true because the wing is at this instant at the closest position to the top side wall whereas the wing is far away from the farfield boundary by $10c$ length in Shyy et al.'s simulation.

Figure 16 shows comparison of spanwise vortical flow downstream TE at the middle of downstroke between farfield domain (Shyy et al.'s simulation), Heathcote et al.'s water tunnel experiment and WWBD current simulation. Once more, the vorticity flow field obtained from the present simulation with WWBD simulation is very comparable with the experiment[6]. It seems that the Shyy et al.'s qualitative results were affected by the domain size and mismatching the effect of wall blockage in the water experiment[6], which is observed again at this instant for the size of anticlockwise TEV shedding in the wing wake.

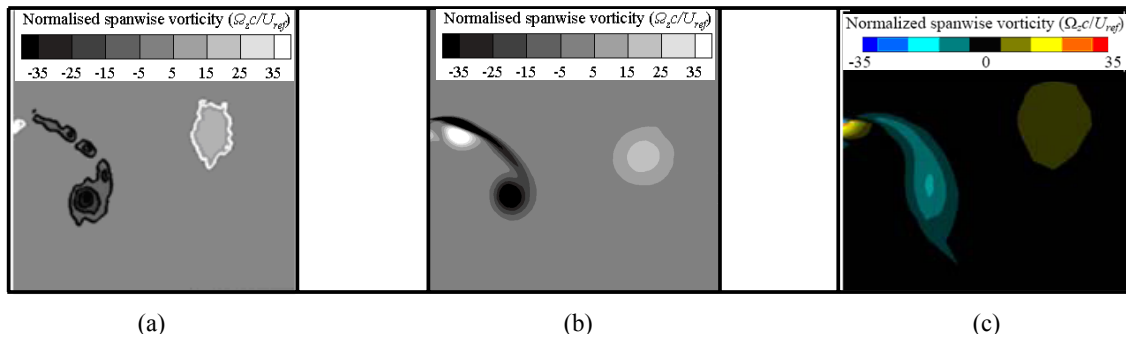


Figure 15 Spanwise vortical flow comparisons at the beginning of downstroke around the wake (a) experiment[6] (b) the present simulation (c) the past computation[1].

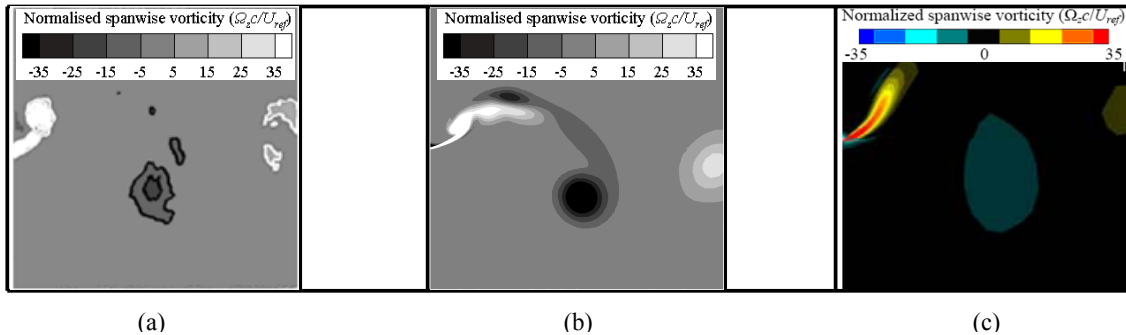


Figure 16 Spanwise vortical flow comparisons at the middle of downstroke around the wake (a) experiment[6] (b) the present simulation (c) the past computation[1].

After this brief discussion given above, closer comparison in the vortical flow which was shown between the current simulation and experiment compared with farfield domain[1] validates the quantitative agreement in the force coefficients between the experiment and WWBD present simulation.

3.4 Propulsion performance

The farfield domain with air is used to conduct a parametric study of flapping kinematics as the averaged results obtained from the other cases to some extent are very close as shown in Table 4. It is also practical to use the farfield domain to study large flapping amplitude with less numerical errors expected from simulations due to the use of confined domain. The instantaneous results of the farfield domain however in good agreement with Shyy et al.'s computations[1, 2]. In relation with whether the flapping performance is economically acceptable choice, Wang et al.[14] found that 27% of the aerodynamic power required in steady flight can be saved by optimised flapping wing motions.

1) Effect of flapping frequency:- propulsion performance of the plunging NACA 0012 wing in terms of the flapping efficiency was first investigated at $f = 0.1916, 0.9578, 1.7432, 2.1072$ and 2.8734Hz with $y/c = 17.5\%$. Figure 17 shows the effect of $K = 0.2, 1, 1.82, 2.2$ and 3 on the force coefficients which corresponds with $f = 0.1916, 0.9578, 1.7432, 2.1072$ and 2.8734Hz respectively. It may be noticed that drag force was produced at $K = 0.2$ ($f = 0.19155\text{Hz}$). Refer to Table 2 for the definition of K where f was calculated based on $Re = 30,000$ and $c = 0.1\text{m}$. The average of instantaneous thrust coefficients $\overline{C_T}$ (Eq. 15) are tabulated in Table 5. In addition, the power input $\overline{C_P}$ and propulsive efficiency η were calculated from below equations at various St corresponding to the above values of K or f (refer to Table 2 for the definition of St for the range of f).

$$\overline{C_P} = \frac{\overline{F_r v}}{\frac{1}{2} \rho U_\infty^3 S} \quad \dots(13)$$

where v is the instantaneous velocity of plunging wing as shown in Fig. 6.

$$\eta = \frac{\text{power out}}{\text{power input}} = \frac{\overline{CT}}{\overline{CP}} \quad \dots(14)$$

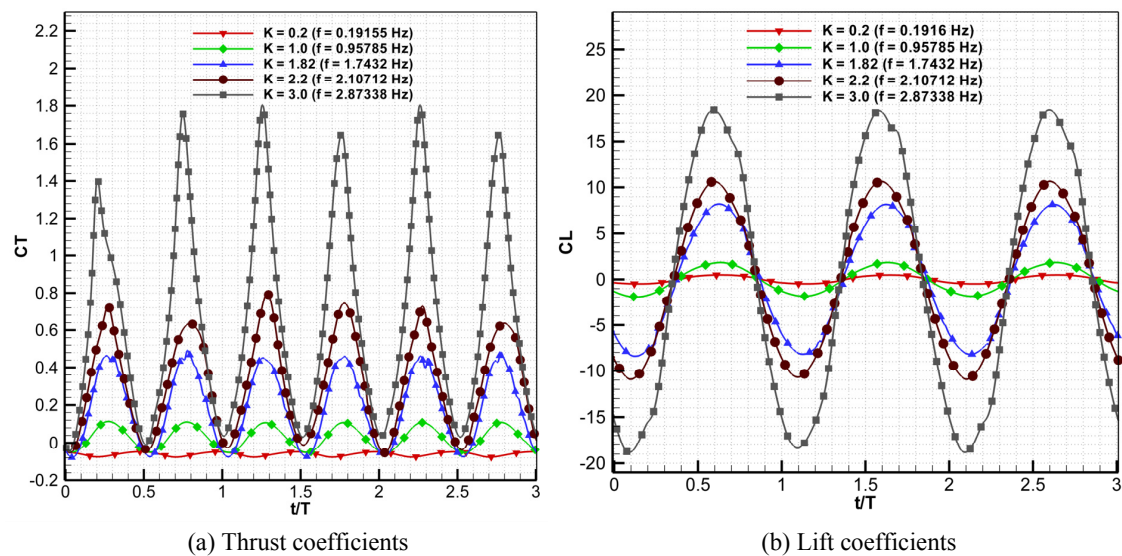


Figure 17 Effect of K at 0.2, 1, 1.82, 2.2 and 3 in correspondence with $f = 0.1916, 0.9578, 1.7432, 2.1072$ and 2.8734 Hz respectively on the force coefficients with $y/c = 17.5\%$ ($St = 0.2237634$).

K	f (Hz)	T (sec)	St	$\overline{CT}$	$\overline{CL}$	η
0.2	0.19155889	5.220327	0.02228169	-4.05E-02	0.1131	-0.35817
1	0.957794448	1.044065	0.11140846	5.38E-02	0.307947	0.176705
1.82	1.743185894	0.573662	0.2237634	0.21905	1.222392	0.179198
2.2	2.107147785	0.474575	0.24509861	3.46E-01	2.275197	0.152075
3	2.873383343	0.348022	0.33422538	6.96E-01	5.504892	0.126433

Table 5 Trade-off analysis of the plunging performance based on studying different flapping frequencies with amplitude of $0.0175m$

Figure 18 shows the analysis of propulsion performance based on St and K with $f = 0.1916, 0.9578, 1.7432, 2.1072$ and 2.8734 Hz respectively at $y/c = 17.5\%$. Comparison with the available data of Heathcote et al.'s experiment[6] is made. Figs. 18a and b indicate that the mean power input and thrust exponentially increase as St raises. Fig. 18c gives the effect of K on the mean thrust. Likewise, the mean thrust enhanced with raising K . However, no enhancement in the efficiency above 18% was noticed as shown in Fig. 18d due to the increase of the mean power input and thrust coefficients with St . Overall, a good agreement was found in comparison with the experiment[6]. Forth order polynomial curve fitting was created for the simulation data with minute maximum normal residuals ($\|Residuals\| < 10^{-13}$).

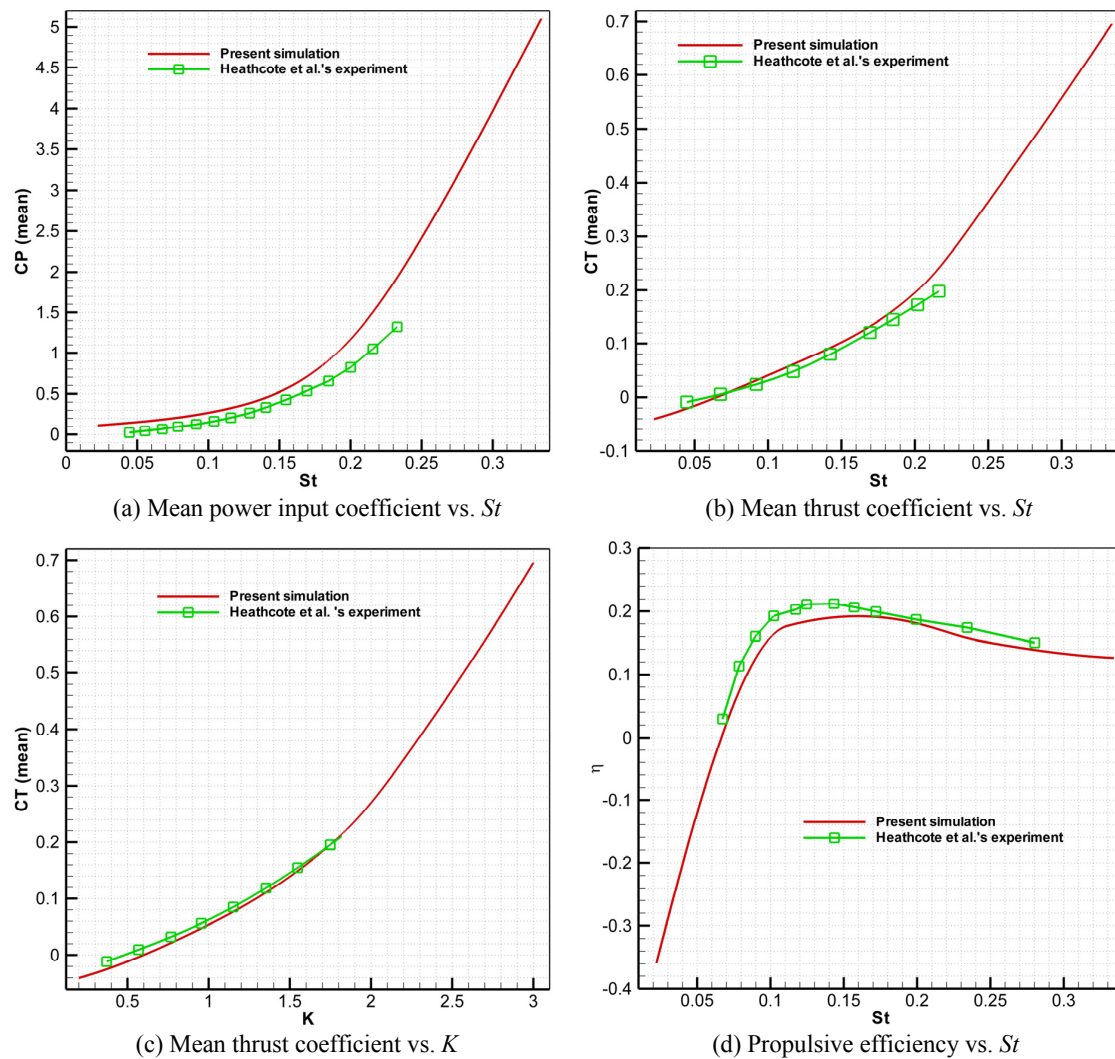


Figure 18 Propulsion performance analysis based on St and K with $f = 0.1916, 0.9578, 1.7432, 2.1072$ and 2.8734Hz respectively at $y/c = 17.5\%$. Comparison with Heathcote et al.'s experiment attached.

2) **Effect of flapping amplitude:-** Similarly, the propulsive efficiency of plunging NACA 0012 wing was then examined at $y/c = 7.5, 12.5, 17.5, 22.5$ and 27.5% based on the optimal f representing the maximum efficiency in the initial investigation ($f = 1.7432\text{Hz}$). Figure 19 shows the effect of plunging at higher amplitude on the force coefficients which corresponds with $St = 0.0869, 0.1448, 0.2027, 0.2607$ and 0.3186 . No surprises were found in the forces because the trends are proportional with raising the flapping amplitudes. The average of instantaneous thrust coefficients $\overline{CT}$ (Eq. 12) are tabulated in Table 6. In addition, $\overline{CP}$ and η were calculated from Eqs. 13 and 14 at various St corresponding to y/c (refer to Table 2 for the definition of St for the range of A_y where $f = 1.743186\text{Hz}$).

Figure 20 shows the analysis of propulsion performance based on various St at $K = 1.82$ ($f = 1.7432\text{Hz}$). Fig. 20a indicates that the mean power input and thrust exponentially increase as St raises. However, no enhancement in the efficiency above 18% was noticed as shown in Fig. 20b due to the high mean power input found as St increased. As may be useful, it is here again forth order polynomial curve fitting was created for the simulation data with minute maximum normal residuals ($\|Residuals\| \ll 10^{-13}$).

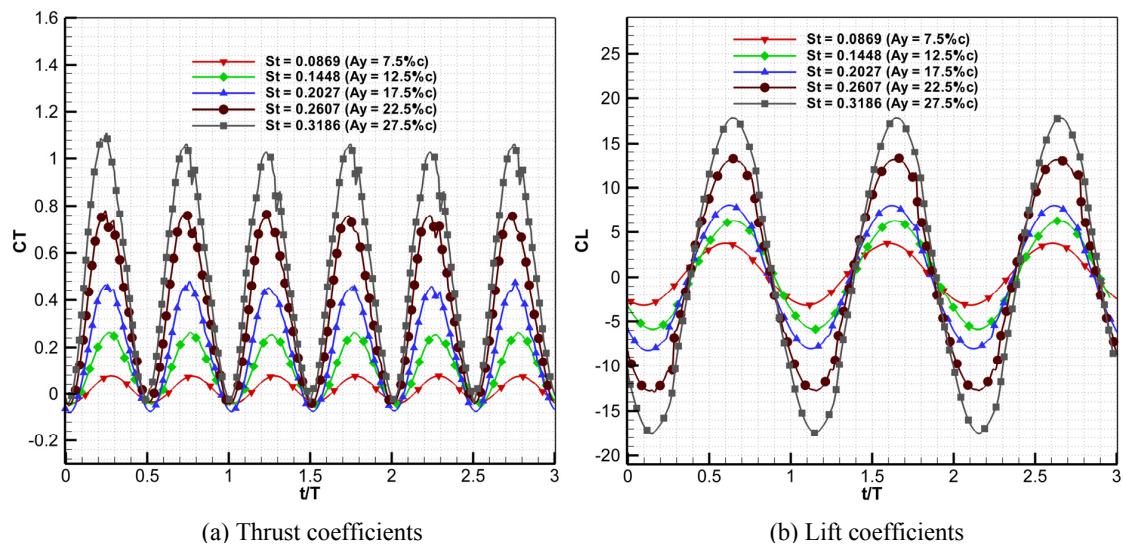


Figure 19 Effect of flapping amplitudes-to-chord ratio at 7.5, 12.5, 17.5, 22.5 and 27.5% in correspondence with $St = 0.0869, 0.1448, 0.2027, 0.2607$ and 0.3186 respectively with $f = 1.7432\text{Hz}$ on the force coefficients.

$A_y/c\%$	$A_y(m)$	St	$\overline{CT}$	$\overline{CP}$	η
7.5	0.0075	0.086899	8.95E-03	0.17204766	0.0520205
12.5	0.0125	0.144831	9.90E-02	0.56613239	0.1748708
17.5	0.0175	0.202763	0.21905	1.2223924	0.1791978
22.5	0.0225	0.260696	3.56E-01	3.05803682	0.1164146
27.5	0.0275	0.318628	5.22E-01	5.51664871	0.0946227

Table 6 Trade-off analysis of the plunging performance based on studying different flapping amplitudes (St) at $f = 1.7432\text{Hz}$ ($K = 1.82$)

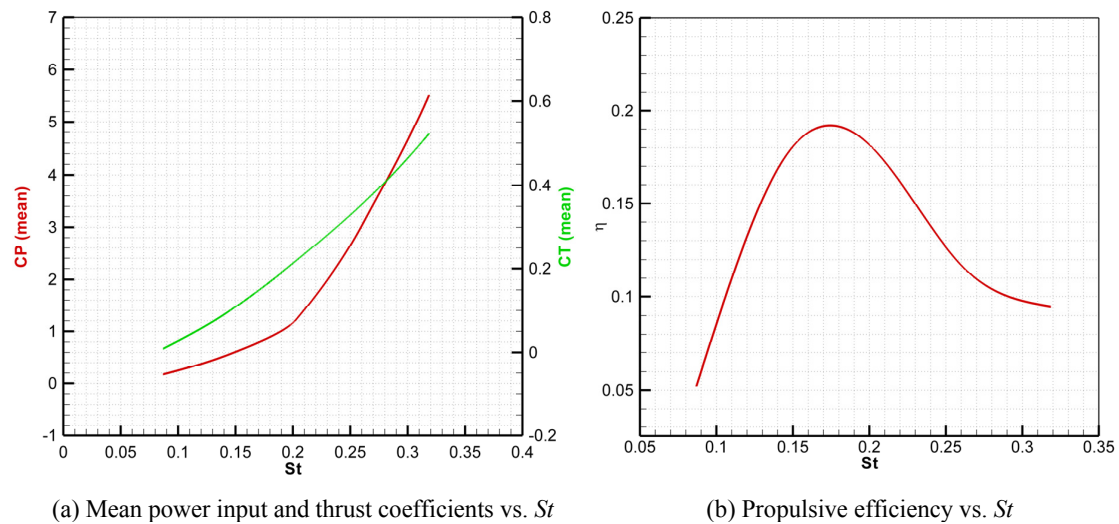


Figure 20 Propulsion performance analysis based on St with various $y/c = 7.5, 12.5, 17.5, 22.5$ and 27.5% at $f = 1.7432\text{Hz}$.

2) Unsteady vortical flow field

Instantaneous iso-vorticity contours of unsteady flow field along wing span (z-vorticity) are shown in Fig. 21 at various locations of plunging stroke (these locations are indicated by the blue squares in Fig. 6). Different patterns of wake vortex structure were found during these various instants.

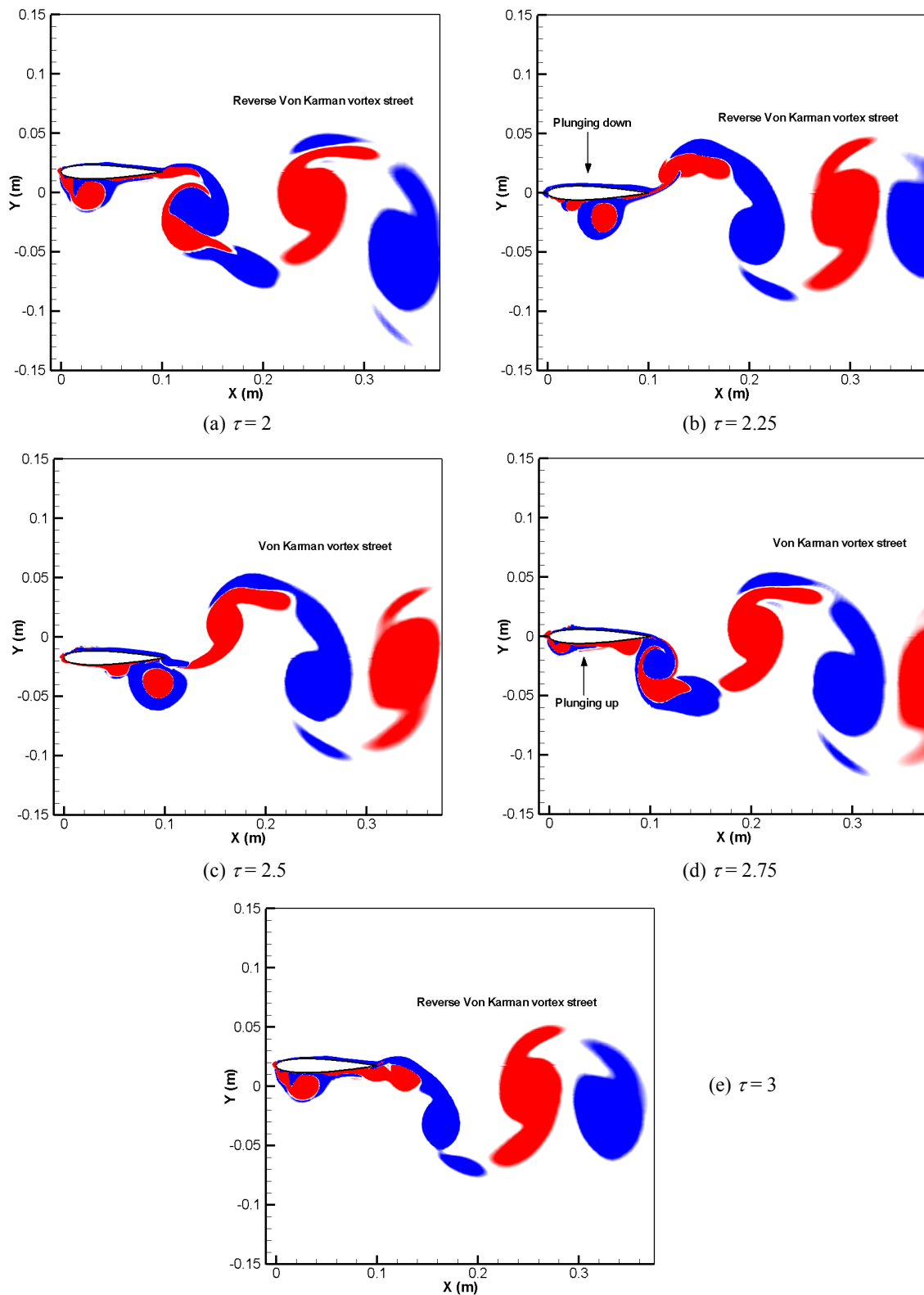


Figure 21 Spanwise iso-vorticity contours at $z/b = 0.5$ of NACA 0012 rigid wing during the third flapping cycle shown by various normalised period. The red vortices refer to the anticlockwise directions, and vice versa.

At the onset of downstroke $\tau = 2$ (Fig. 21a), Von Karman street which formed in the previous stroke transits to reverse Von Karman street. A momentum loss is here expected in the wake compared to the upstream flow which signifies drag-indicative vortex configuration[10, 15 and 16] as shown in Fig. 17a. As the wing moves further down at $\tau = 2.25$ (Fig. 21b), LEV starts to slip along LS and reverse Von Karman street is observed. This is known as thrust-indicative vortex configuration[10, 15 and 16] where flow velocity in the wake is higher than upstream (maximum thrust was found, see Fig. 17a). At $\tau = 2.5$ (Fig. 21c), LEV reaches TE and the flow leaves TE smoothly causing Von Karman street of shedding vortices in the wake. At this instant some drag was produced as shown in Fig. 17a. Slightly deflected vortex street was dominated the flow structure at this high frequency flapping which explains the thrust production at higher frequencies, this is in accordance with Refs. 13 and 15. Fig. 21d shows the prominent interaction of LEV and TEV of forming deflected vortex sheet and new LEV was created due to which the maximum unsteady thrust and lift noticed, as shown in Figs. 17a and b respectively. The small deflection of vortex wake may refer to the minute instantaneous effective angle of attack[2, 7] which defined as

$$\alpha_{eff} = \tan^{-1} \left(-\frac{v(t)}{u_{\infty}} \right) \quad \dots(15)$$

Yang et al.[8] mentioned the effect of the so-called nominal angle of attack or α_{eff} and reported that the thrust keep increases as α_{eff} increases. Note that, $v(t)$ is always perpendicular to u_{∞} . $\alpha_{eff} = 0, 33, 0, -33$ and 0 degrees are computed at $\tau = 0, 0.25, 0.5, 0.75$ and 1 respectively, likewise for the next flapping periods. In between, α_{eff} either increases or decreases gradually in the form of sinusoidal harmonic variation with respect to τ . Apparently, the maximum unsteady forces (Figs. 17a and b) are due to the large induced vertical plunging velocity and consequently the effective angle of attacks. At the mid of upstroke $\tau = 2.75$ (Fig. 21d), TEV shedding appears and wake vortex array is deformed gradually to reversed Von Karman street due to the high effective angle of attack of almost -33 degrees. The thrust force was enhanced again and maximum value was reached at this instant. At the end of upstroke $\tau = 3$ (Fig. 21e), the vortices fades away in the wake as the wing reaches the utmost position. LEV slips along LS to about $30\%c$. This is a significant similarity between this instant with $\tau = 2$, particularly for the flow structure around the wing LS until about $80\%c$. However, a slight correspondence was observed in the vortex wake because at $\tau = 2$ was affected by the instantaneous flow from the second stroke. The effect of viscous on US of the wing is pronounced at all instants of plunging, i.e., no separation flow was noticed on US through various flapping times.

3) Flow characteristics and pressure distribution

Streamlines at three spanwise positions ($z/b = 0, 0.5$ and 1) along with z-vorticity at various across streamwise planes ($x/c = 0.01, 0.5, 1, 1.5, 2$ and 2.5) were shown at four instants ($\tau = 0, 0.25, 0.5$ and 0.75) of pure plunging NACA 0012 wing (Figs. 22a, b, c and d). Pressure distribution on the wing surface was also included on Figs. 22a, b, c and d. Generally, various flow structure patterns are generated as a consequence of oscillating wing and a brief discussion on significant features found is given below.

At the onset of downstroke $\tau = 0$ (Fig. 22a), the resultant negative pressure acting on the wing surfaces is higher due to the high negative pressure exerting on US. At the middle of downstroke $\tau = 0.25$ (Fig. 22b), apart from negative pressure acts along LE the rest of wing surfaces has positive pressure distribution which causes small negative resultant pressure over the wing surface. At the end of downstroke $\tau = 0.5$ (Fig. 22c), the resultant positive pressure acting on the wing surfaces is higher due to the high positive pressure exerting on US. At the middle of upstroke $\tau = 0.75$ (Fig. 22d), the resultant positive pressure acting on the wing surfaces is higher due to the high positive pressure exerting on US.

The vortices shedding along chordwise direction and in the wake are visualised and the growth of these vortices are visible, particularly at the tip and root due to small clearance made in relation to the water tunnel blockage modeling[6]. It seems that the flow structure at $\tau = 0$ (Fig. 22a) and $\tau = 0.5$ (Fig. 22c) is somewhat reverse of each other. i.e., the pressure distribution on US of Fig. 22c represents the pressure distribution on LS of Fig. 22a, and vice versa. Similarly, the opposite flow structure was mentioned previously is also observed at $\tau = 0.25$ (Fig. 22b) and $\tau = 0.75$ (Fig. 22d), i.e., the pressure distribution on US of Fig. 22d represents the pressure distribution on LS of Fig. 22b, and vice versa. Counterclockwise vortices of Fig. 22a and Fig. 22b appear respectively as clockwise vortices of Fig. 22c and Fig. 22d, and vice versa. The stream lines generated around the wing may verify these hypotheses.

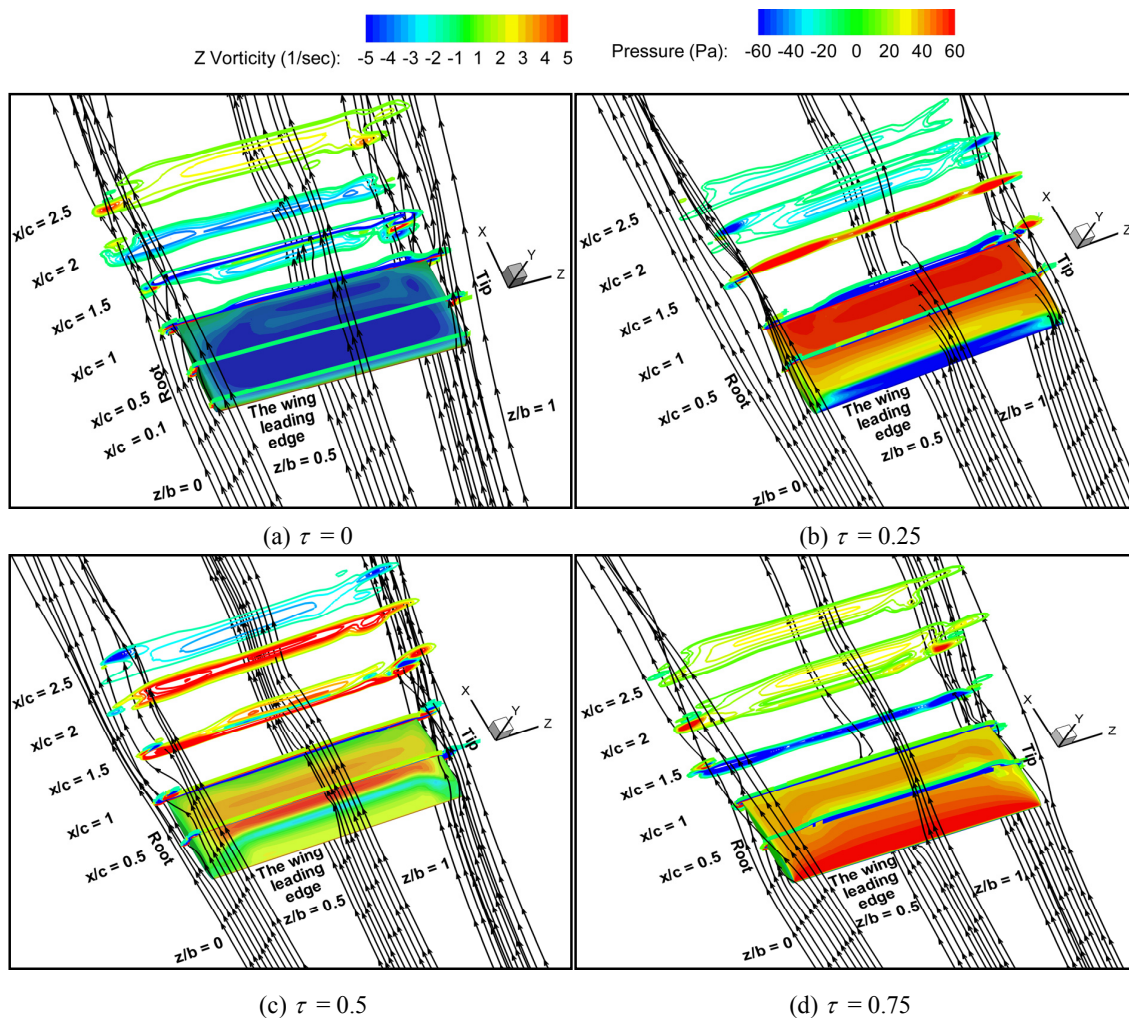


Figure 22 Flow structure shown by z-vorticity contours at various across streamwise planes ($x/c = 0.01, 0.5, 1, 1.5, 2$ and 2.5) and streamlines at three spanwise positions ($z/b = 0, 0.5$ and 1). Pressure distribution on the wing surface given at four significant instants of pure plunging NACA 0012 wing ($\tau = 0, 0.25, 0.5$ and 0.75).

4 Conclusion

Promising results correlated with the studies of flapping wings are obtained based on the unsteady numerical simulations. A rational degree of confidence has been conferred by comparing the prediction with measurements with two sets of domain boundaries with water and/or air for each, namely, wall bounded and farfield domains to model the tunnel blockage of experiment and free flight respectively. A high-fidelity coupled framework of dynamic mesh technique (layering and sliding) with the flow solver is used. Decent qualitative and quantitative agreements are found between the experiment and wall bounded domain simulation. A fairly close agreement has been obtained for the shedding vortices downstream of the wing trailing edge during the onset and mid of down-stroke. A moderately stronger trailing edge vortex structure has been observed. A train of shedding vortices opposing sign has been detected from the leading and trailing edges in correlation with the thrust production. The strength of these vorticity patterns becomes weaker as they travel farther away in the wing wake. Apart from the asymmetric thrust shown in the experiment between the downstroke and upstroke phases, reasonable correspondence has been achieved from the second flapping cycle between the computed thrust with the experiment. The average thrust coefficient of 0.2215 is found from the computation compared to 0.21 measured in the experiment. The following concluding remarks can be drawn from this study:-

- (I) The instantaneous results achieved from water confined domain are in closer agreement with the tunnel blockage of water tunnel test compared with other cases (air wall bounded domain, air farfield domain and water farfield domain). However, no significant differences are observed from the averaged forces.

- (II) The effect of wall boundary should not be ignored in the validation study adopting experiments based on cramped sidewalls tunnel.
- (II) The effects of flapping frequency and amplitude on propulsive performance of plunging NACA 0012 wing are studied separately based on Garrick reduced frequency of 0.2 – 3. The propulsive efficiency as high as 18% is found due to the flapping at frequency of 1.7432Hz and amplitude-to-chord ratio of 17.5% corresponding with Garrick reduced frequency of 1.82 and Strouhal number of 0.202763. No further improvement of the efficiency can be found at other flapping kinematics because the power input is significantly increased. Heathcote et al suggested that introducing an intermediate degree of spanwise flexibility as observed from birds or bats could be beneficial to minimise the power input and thus maximise the efficiency.

Future work will focus on modelling 3D dragonfly corrugated wing undergoing combined motion (pitch and plunge excitation) and carrying out a similar parametric study of the aerodynamic performance of their flapping kinematics.

References

- 1] Aono, H., et al. Computational Modelling of Spanwise Flexibility Effects on Flapping Wing Aerodynamics. *47th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*. 2009. Orlando, Florida: AIAA.
- 2] Chimakurthi, S.K., et al., *Computational Aeroelasticity Framework for Analysing Flapping Wing Micro Air Vehicles*. *AIAA Journal*, 2009. **47**(8).
- 3] Liu, H., *Integrated Modeling of Insect Flight: from Morphology Kinematics to Aerodynamics*. *Journal of Computational Physics*, 2009. **228**: p. 439-459.
- 4] Tang, J., et al. Computational Fluid-Structure Interaction of a Deformable Flapping Wing for Micro Air Vehicle Applications. *46th AIAA Aerospace Sciences Meeting and Exhibit*. 2008. Reno, Nevada.
- 5] Anderson, J.M., et al., *Oscillating Foils of High Propulsive Efficiency*. *J. Fluid Mech.*, 1998. 360: p. 41-72.
- 6] Heathcote, S., Z. Wang, and I. Gursul, *Effect of Spanwise Flexibility on Flapping Wing Propulsion*. *Journal of Fluids and Structures*, 2008. **24**(2): p. 183-199.
- 7] Lian, Y. Numerical Investigation of Boundary Effects on Flapping Wing Study. *47th AIAA Aerospace Science Meeting including the New Horizons Forum and Aerospace Exposition*. 2009. Orlando, Florida: AIAA, Inc.
- 8] Yang, T. and M. Wei. Numerical Study of Flexible Flapping Wing Propulsion. *48th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*. 2010. Orlando, Florida.
- 9] K.D.Jones and M.F.Platzer. Flapping-Wing Propulsion for a Micro Air Vehicle. *38th Aerospace Sciences Meeting & Exhibit*. 2000 Reno, NV.
- 10] Tuncer, I.H. and M.F. Platzer, *Thrust Generation due to Airfoil Flapping*. *AIAA Journal*, 1996. **34**(2).
- 11] Schlichting, H. *Boundary-Layer Theory*, McGraw-Hill, New York, 1979.
- 12] Ahmed N., Yilbas B.S., and Budair M.O., *Computational Study into the Flow Field Developed around a Cascade of NACA 0012 Airfoils*. *Comput. Methods Appl. Mech. Engrg.*, 1998. **167** p. 17-32.
- 13] Chandar, D.D.J. and M. Damodaran, *Computation of Unsteady Low Reynolds Number Free-Flight Aerodynamics of Flapping Wings*. *Journal of Aircraft*, 2010. **47**(1).
- 14] Pesavento, U. and J. Wang, *Flapping Wing Flight Can Save Aerodynamic Power Compared to Steady Flight*. *Physical Review Letters*, 2009. **103**(4).
- 15] Ashraf, M. A., J.C.S. Lai, and J. Young. Numerical Analysis of Flapping Wing Aerodynamics. *16th Australasian Fluid Mechanics Conference*. 2007. Crown Plaza, Gold Coast, Australia.
- 16] Koochesfahani, M.M., *Vortical Patterns in the Wake of an Oscillating Airfoil*. *AIAA J.*, 1989, **27** (9): p. 1200-1205.