



## TWO-DIMENSIONAL CFD SIMULATION FOR VISUALIZATION OF FLAPPING WING ORNITHOPTER STUDIES

Harijono Djojodihardjo and Khairul Afiq A. Rahim

Department of Aerospace Engineering, Faculty of Engineering, University Putra Malaysia, Malaysia

E-Mail: [harijono.djojodihardjo@yahoo.com](mailto:harijono.djojodihardjo@yahoo.com)

### ABSTRACT

A Two-Dimensional Computational Fluid Dynamic (2D-CFD) study is carried out on an oscillating flat plate at various flow situations in order to gain insight on the flow field and forces acting on a generic flapping wing in a viscous flow within a certain flying conditions. The dimensions and parameters associated with the amplitude and frequency of the oscillating flat plate simulating flapping wing are designed meticulously with reference to those suggested from literature study and observation. The dimensions and parameters associated with the amplitude and frequency of the oscillating flat plate simulating flapping wing are designed meticulously with reference to those suggested from literature study and observation. The dimensions and parameters associated with the amplitude and frequency of the oscillating flat plate simulating flapping wing are designed meticulously with reference to those suggested from literature study and observation. The dimensions and parameters associated with the amplitude and frequency of the oscillating flat plate simulating flapping wing are designed meticulously with reference to those suggested from literature study and observation.

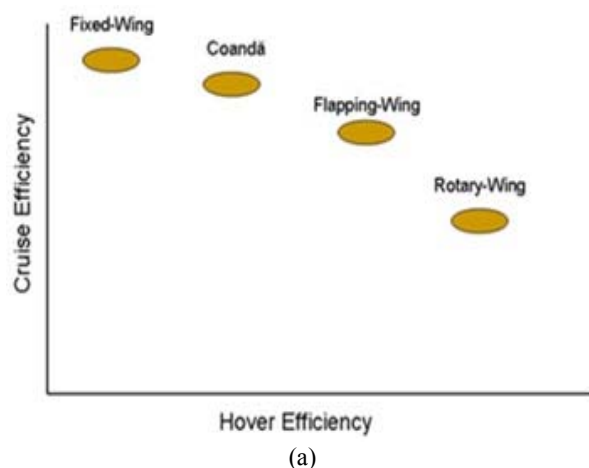
**Keywords:** CFD analysis, flapping wing, fluid dynamics, ornithopter.

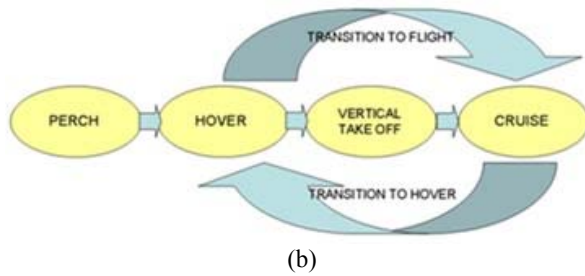
### INTRODUCTION

Flying animal generates lift, thrust, and performing astonishing manoeuvres by flapping its wings. By studying the flight characteristics of natural flyers such as birds and bats, the motion of a two-dimensional model can be identified, and the associated unsteady aerodynamics can be modelled and simulated using CFD. Instructive examples can be developed by looking into simplified and generic motion of flapping and pitching, which may later be utilized as guide lines in the design a flapping wing ornithopter model. Therefore, to have a better insight for the optimization of the flapping wing kinematics effect on the generated aerodynamic forces, CFD approach has been widely used as it give promising results in addition to several other benefits like relatively low cost and low turn-around time, and ease in implementation. However, there are several aspects that need to be understood before one could embark on a computational simulation of the flapping wings ornithopter. For example, Carson et. al. [3] stipulated, that the optimization of a flapping wing model by using Direct Numerical Method (DNS) may accrue high computational cost which is one of the drawbacks, along with the uncertainty of the best optimization approach to be used. Therefore, the present approach utilizes Reynolds Averaged Navier-Stokes (RANS) based CFD software, as well as 2-D wing model (flat plate or various appropriate NACA airfoil), prior to more involved 3-D wing models, to obtain various information of the flapping wing with certain kinematical characteristics without considering the high computational cost issue.

Currently, there are three main propulsion and lifting means for UAV flight; these are fixed wing, rotary wing, and flapping wing. Each one of them offers desirable performance trade-offs, depending on its mission requirements, as schematically depicted in Figure-1 [17]

[4]. As the wings are reduced in size, a transition from high to low Reynolds number aerodynamics occurs, resulting in the decrease of the performance of the wings, rotor or airfoil. Due to the scaling effects, fixed wing ornithopters require high forward speeds to generate the lift necessary for flight. Therefore, it is difficult to perform complex manoeuvres needed for successful indoor flight. Furthermore, fixed wing ornithopters are not well suited to short take-offs and landings, obstacle avoidance, or other behaviours dependent on low flight speeds. While for rotary wing ornithopters have higher capability of performing intricate manoeuvres, such as vertical take-off and landing, well controlled hover, zero-radius turns, and obstacle avoidance.





**Figure-1.** (a) An impression of the qualitative performance of propulsion and lifting system in various UAV/MAV systems (adapted from [17][4]); (b) Flight Manoeuvring Structure (adapted from [8]).

However, rotary wing UAV's usually suffer from similar disadvantage relating to airfoil breakdown at low Reynolds numbers. As the size decreases, the lifting capacity of the rotor significantly degrades. Due to the continuous movements of the rotor blades, there will be a distinct noise generated in flight that can be detected easily, thus preventing stealthy operation [18]. In addition, with the rotor blades spinning at great speeds, a rotary wing could be potentially hazardous to nearby people. A flapping wing mechanism used by ornithopter offers combined potential from many of the beneficial properties of the other two modes of flight, while diminishing its undesirable properties.

Observation on the wing kinematics of natural flying animals (as illustrated by Figure-2) suggests that highly complex and precise maneuvers are possible with flapping wings, even at very small sizes. Animal-inspired flight provides the following advantages over traditional forms of UAV propulsion.



**Figure-2.** Illustration of a bird in flapping motion, extracted Google images.

Natural flyer animals are capable of extremely agile flight manoeuvres that would translate to useful behaviour such as perching, hovering, navigating through tight spaces, and maintaining stability in the presence of strong variable disturbances. They have the capability of modifying their flight mode depending on the changing aerodynamic demands. The ability of natural flyer to

perform these kinds of flight was achieved through variation in angle of attack, wingtip trace pattern, wing area, and complex adjustments to feather orientation. Because of the wide range of abilities that the natural flyers can demonstrate, a flapping wing UAV offers high potential of great versatility in flight. Behaviours such as hovering, low speed flight, short take-off and landing, and efficient gliding would make flapping wing ornithopter such a useful compromise between fixed wing and rotary wing UAV's. One of the benefits of flapping wings is the favorable scaling relative to fixed wings and rotorcraft. A very small scale wings also have the possibility to create flight by increasing the flapping frequency.

An important benefit of wing flapping is that the wings can operate at a significantly lower frequency than a propeller or a rotor, so the noise would be difficult to detect, resulting in very stealthy operation. While the primary disadvantage of flapping wing UAV is the limitation of payload capabilities which contributes to insufficient functionality and operational range for practical applications.

## MODELING APPROACH

### Governing equation

The results of CFD analysis are relevant in the conceptual studies of new designs and the detailed product development process. Basically, the working principle of CFD is based on the principles of conservation in fluid mechanics and numerical method. In the numerical computation, the domain is discretized into a finite set of control volumes and the general conservation equations for mass, momentum, energy and else are solved on the set of finite control volumes. From a macroscopic point of view, starting from a general control volume  $V$  with surface boundary  $A$ , the conservation principle of any quantity  $\phi$  can be expressed as:

$$\frac{\partial}{\partial t} \int_V \rho \phi dV + \underbrace{\oint_A \rho \phi V \cdot dA}_{\text{convection}} = \underbrace{\oint_A \Gamma_\phi \nabla \phi \cdot dA}_{\text{diffusion}} + \underbrace{\int_V S \phi dV}_{\text{generation}} \quad (1)$$

$\phi$  – mass, momentum, energy

where  $\phi$  represent mass, momentum and energy. Equation (1) is the generic transport equation in fluid dynamics in the form of an integro-differential equation [13]. From microscopic point of view, equation (1) is also valid at any small finite volume  $\delta V$ . The commercial CFD code utilized, i.e. ANSYS FLUENT, is also based on the integral form of the governing equations equation (1), and for viscous flow is elaborated in the form of Reynolds Averaged Navier Stokes Equations. The latter are discretized into a system of algebraic equations and these equations are then solved numerically to render the solution field. For further reference in the numerical simulation of the problem considered in this work, relevant forms of the equations utilized for CFD analysis will be highlighted. The Conservation form of the generalized Navier-Stokes equation suitable for CFD,



comprising the conservation of mass, momentum and energy equation can be expressed as (Anderson [1], which should be resorted to for detailed definition of the variables):

$$\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{\partial H}{\partial z} = J \quad (2)$$

where U, F, G, H and J are the column vectors, given by

$$U = \begin{Bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho \left( e + \frac{V^2}{2} \right) \end{Bmatrix} \quad (3a)$$

$$F = \begin{Bmatrix} \rho u \\ \rho u^2 + p - \tau_{xx} \\ \rho uv - \tau_{xy} \\ \rho wu - \tau_{xz} \\ \rho \left( e + \frac{V^2}{2} \right) u + pu - k \frac{\partial T}{\partial x} - u\tau_{xx} - v\tau_{xy} - w\tau_{xz} \end{Bmatrix} \quad (3b)$$

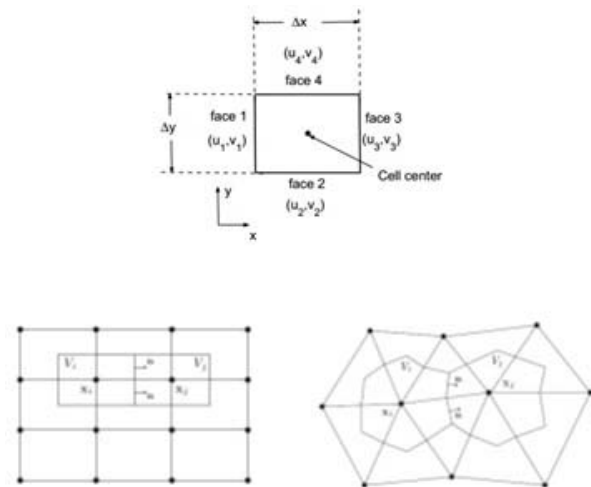
$$G = \begin{Bmatrix} \rho v \\ \rho uv - \tau_{yx} \\ \rho v^2 + p - \tau_{yy} \\ \rho vw - \tau_{yz} \\ \rho \left( e + \frac{V^2}{2} \right) v + pv - k \frac{\partial T}{\partial y} - u\tau_{yx} - v\tau_{yy} - w\tau_{yz} \end{Bmatrix} \quad (3c)$$

$$H = \begin{Bmatrix} \rho w \\ \rho uw - \tau_{zx} \\ \rho vw - \tau_{zy} \\ \rho w^2 + p - \tau_{zz} \\ \rho \left( e + \frac{V^2}{2} \right) w + pw - k \frac{\partial T}{\partial z} - u\tau_{zx} - v\tau_{zy} - w\tau_{zz} \end{Bmatrix} \quad (3d)$$

$$J = \begin{Bmatrix} 0 \\ \rho f_x \\ \rho f_y \\ \rho f_z \\ \rho (uf_x + vf_y + wf_z) + \rho \dot{q} \end{Bmatrix} \quad (3e)$$

where the column vector F, G, and H are the flux terms, and J represents a 'source term' (which is zero if body forces are negligible).

These governing equations are utilized in the numerical algorithm incorporated in the Computational Fluid Dynamics software, discretizing the computational domain into a set of elements. The discretization method used in the CFD procedure adopted is carried out using the Finite-Volume method. As a baseline in the Finite-Volume method, quadrilateral elements can be utilized and is commonly referred to as a "cell" and a grid point as a "node". In the two-dimensional (2D) version, one could also have triangular or rectangular cells as typically shown in Figure-3.



**Figure-3.** Illustration of Control volumes for a vertex-centered Finite Volume Method in two dimensions in the discretization process (adapted from Kuzmin [13])

Further details are elaborated in the particular software used, which are carefully consulted to ensure the correct application of the problem. The spatial discretization of the computational domain into a large number of finite elements is a specialized topic that will not be addressed here, although the choice of the meshing is very critical to the success of the CFD simulation. In the present work, some guidelines from the software for spatial as well as time discretization will a-priori be judiciously followed to ensure plausible CFD results, which should also be checked a-posteriori.

### Geometry and boundary conditions

The present work is mainly addressed for two-dimensional heaving motion, which corresponds to the flapping motion of a generic tree-dimensional wing. This approach is considered to be relevant, in view of results presented in the literature (such as [5][6][11][12][16][19]) on the generation of thrust and the influence of the Strouhal number. However, the study is also carried out at different pitching angle of the airfoil. Accordingly, the heaving simulation is carried out at different flapping wing amplitudes, frequencies and free-stream velocities. Each



set of simulations will also be carried out at different values of angle of attack.

Further details are elaborated in the particular software used, which are carefully consulted to ensure the correct application of the problem. The spatial discretization of the computational domain into a large number of finite elements is a specialized topic that will not be addressed here, although the choice of the meshing is very critical to the success of the CFD simulation. In the present work, some guidelines from the software for spatial as well as time discretization will a-priori be judiciously followed to ensure plausible CFD results, which should also be checked a-posteriori.

Two airfoil models were originally chosen for simulation; these are a flat plate wing and a NACA 0012 airfoil. Upon preliminary comparison of the simulation results of the two models, for convenience and without losing generalities, the flat plate model is chosen for further simulation. Such choice is considered to be plausible within the present limited objectives, based on observation and literature reviews. For 2-D CFD simulations, the geometry is given by the following dimensions: (i) Wingspan,  $b$ , 20cm, (ii) wing chord length,  $c$  is 8cm, (iii) MAC is 4% of the chord length which is 0.32cm. The wingspan (or aspect ratio) is considered to be suitable for two-dimensional CFD simulation, which is based on 3D code.

The boundary conditions and primary estimation on the airflow field will be further discussed. For the case of forward flapping flight, the inflow and outflow of the domain boundaries are defined (Figure-4). At the inflow boundary the velocity is defined as fixed-value and the pressure as zero-gradient. On the other hand, at the outflow boundary, the pressure has to be fixed-value and the velocity zero-gradient.

For modelling of the flow in a confined space (such as in a wind-tunnel), at the stationary wall the no-slip condition needs to be guaranteed, and accordingly a fixed-value velocity ( $u=0$ ) is specified. Otherwise, in a free air, the velocity is not specified. If the boundary of the wall moves, then the proper boundary condition is the moving wall-velocity which introduces an extra velocity in order to maintain the no-slip condition and ensures a zero-flux through the moving boundary. For the relatively small blockage ratio, the results in the vicinity of the airfoil will not differ significantly. The flow around flapping wings, at the scale relevant to birds, is very unsteady and vortical, described by the unsteady incompressible Reynolds Averaged Navier-Stokes equations. The flow at the Reynolds number of the flow considered,  $Re = 10^4$  is in the turbulence region. In the present CFD simulation, without delving further into further details and for convenience,  $k$ -Epsilon turbulence modelling is chosen. The boundary conditions and primary estimation on the airflow field will be discussed accordingly.

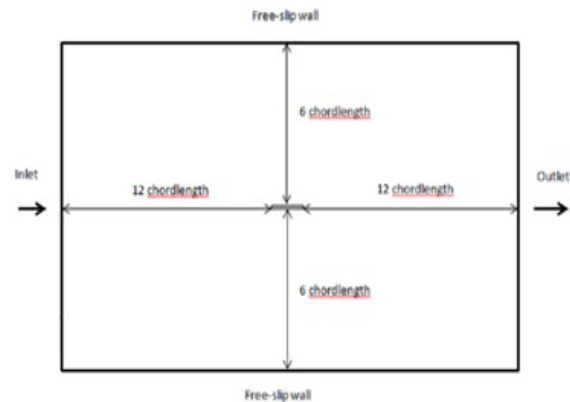


Figure-4. Boundary condition modelling.

### Meshing

When dealing with moving objects, it is possible to solve this mathematical case by changing the boundary conditions as if the boundary was deforming to complete mesh. The ANSYS FLUENT CFD code incorporates different mesh motion techniques in order to change the location of the internal mesh points according to the varying domain shape. Preservation of high mesh quality is necessary to solve the flow in an accurate and efficient way. When using a mesh motion solver, the computational mesh points are moved in order to keep track of the changing location of boundary points. In order to assess the quality of a mesh motion solver, three different aspect need to be formulated, quality, efficiency and robustness. With all these considerations, smoothing dynamic mesh motion solver in the ANSYS FLUENT CFD code has been utilized.

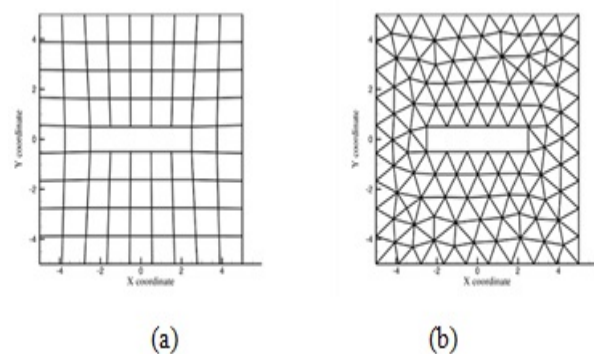


Figure-5. Schematic of the (a) Structured mesh and (b) Unstructured mesh.

Suitable method to discretise these equations was chosen to solve the incompressible Reynolds Averages Navier-Stokes equations. In the field of CFD, three methods are commonly used, the finite difference, finite element and finite volume method. There are three types of grids, structured, block-structured and unstructured grids. Figure-5 taken from Bos [2] above shows an example of a structured and an unstructured grid. When



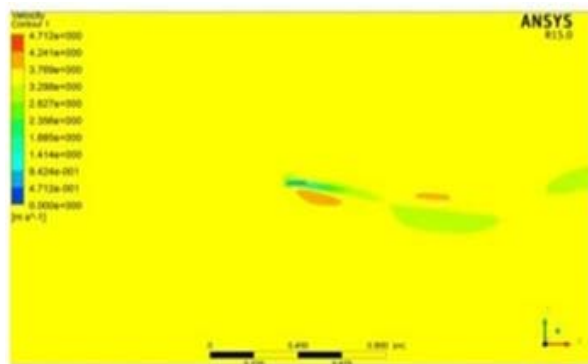
using the structured grids, the cell ordering is fairly straightforward such that the flow solver uses this fact to solve the system in a more efficient way. A drawback of a structured grid is that it is more difficult to create around complex geometries. This is the more important asset of unstructured grids. Besides the type of grid, the cell shape can be varied from tetrahedral (three corners in two dimensions), hexahedral (four faces) to polyhedral (arbitrary number of corners) cells. For less complex geometries, a structured grid is more favourable in terms of accuracy and efficiency of the flow solver. In addition to the spatial discretisation, the time is discretised as well, which is necessary to perform unsteady simulations. The ANSYS FLUENT CFD codes employed in the present simulation work utilizes Reynolds Averages Navier-Stokes equations and finite volume discretization.

## RESULTS AND VALIDATION

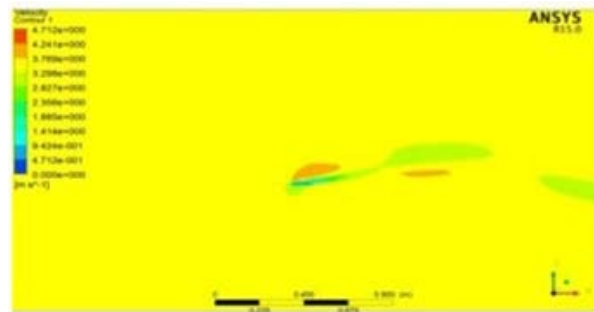
The flow behaviour from Figure-6 and Figure-7 shows the instantaneous vorticity contours, which reveals the presence of the von Kármán vortex street behind the flapping wing flat plate for the heaving motion case. The alternating vortex shedding pattern obtained from the simulation leads to periodic force variation which can be visualized using CL-CD limit cycles. Different values of  $St$  provide the results of the simulation as shown. Noting the results obtained, the CL - CD curve is observed to increase with the increasing Strouhal number,  $St$ . The Strouhal number is defined as:

$$St = \frac{fL}{V_{\infty}}$$

where  $f$  is the frequency of oscillation,  $c$  the chord length of the airfoil, and  $V$  is the free-stream velocity. The frequency of oscillation has been varied from 8 to 20 Hz. Without losing generalities and limiting to attached flow case, the angle of attack  $\alpha$  has been varied from  $-10^\circ$  to  $+10^\circ$ .

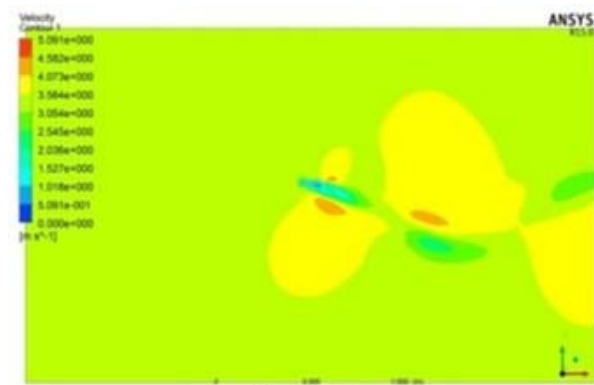


(a)

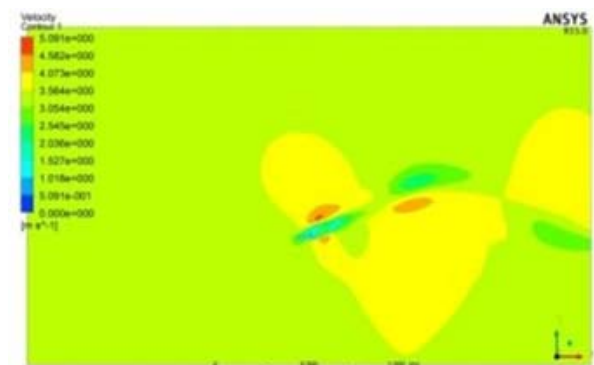


(b)

**Figure-6.** Heaving: (a) Velocity contour for the flat plate during upstroke (1s); (b) Velocity contour for the flat plate during down-stroke (3s), with the free-stream velocity,  $V_{\infty}$  of  $3.5 \text{ ms}^{-1}$ , and the flapping amplitude,  $A$  is  $0.0866 \text{ m}$ .

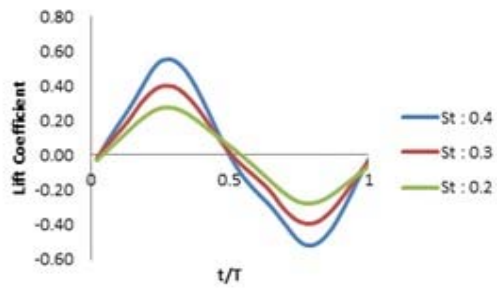


(a)

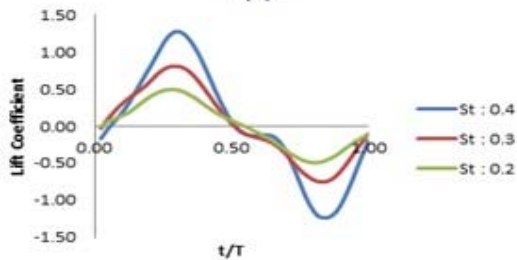


(b)

**Figure-7.** (a) Velocity contour for the flat plate during upstroke (1s); (b) Velocity contour for the flat plate during down-stroke (3s), with the free-stream velocity,  $V_{\infty}$  of  $3.5 \text{ ms}^{-1}$ , and the flapping amplitude,  $A$  is  $0.1732 \text{ m}$ .

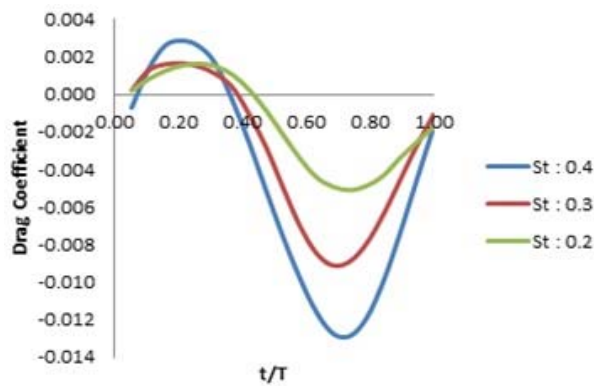


(a)

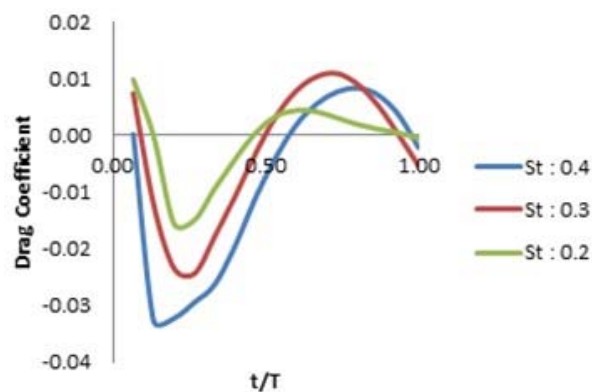


(b)

**Figure-8.** (a)  $C_L$  against time-step for  $A=0.0866m$ ; (b)  $C_L$  against time-step for  $A=0.1732m$ .



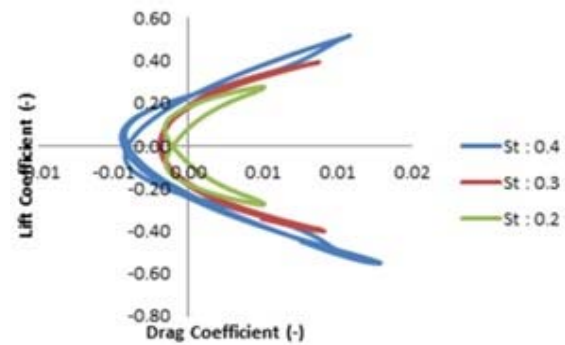
(a)



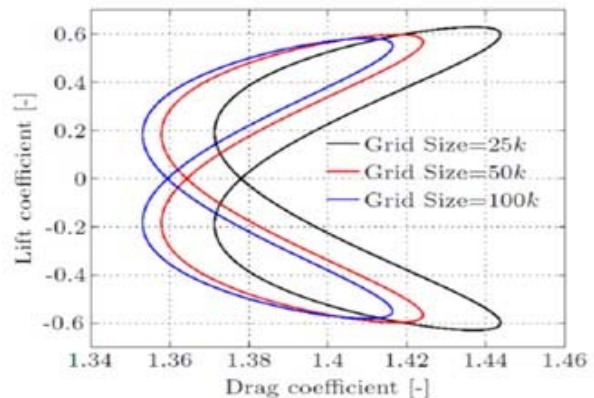
(b)

**Figure-9.** (a)  $C_L$  against time-step for  $A=0.0866m$ , (b)  $C_L$  against time-step for  $A=0.1732m$ .

While for the  $C_L$ - $C_D$  limit cycle pattern, it is observable that the periodicity of the flow of the flapping wing is significantly steady for both flapping amplitude. In order to assess whether the present result falls within reasonable bound, a qualitative comparison is shown in Figure-13(a) and (b). What its exhibit is the polar curve for the Strouhal number,  $St$  of 0.2, 0.3 and 0.4, while Bos has calculated this polar curve but by using Courant number,  $Co$  as the parameter [2]. By inspection, the two curves have qualitative similarity. Work is in progress to produce similar curve as obtained by Bos [2].

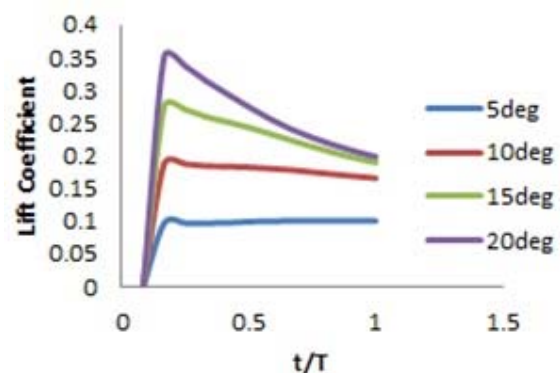


(a)

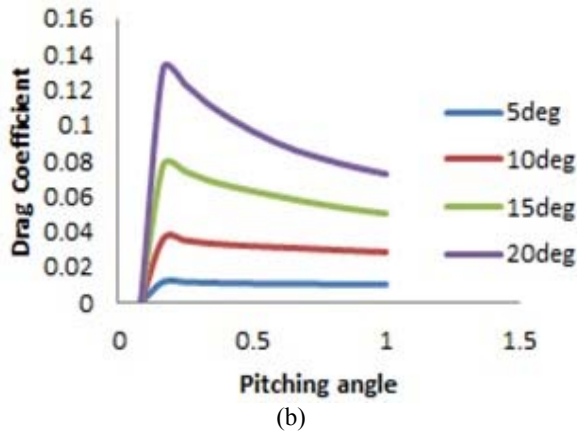


(b)

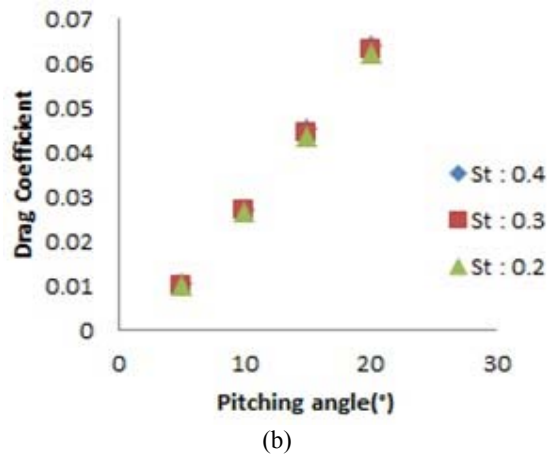
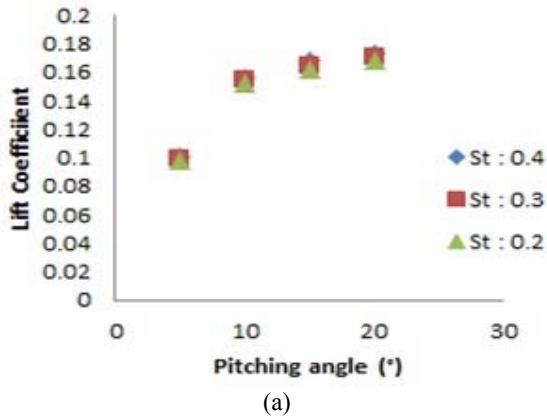
**Figure-10.** (a)  $C_L$  against  $C_D$  for  $A=0.0866m$ ; (b)  $C_L$  against  $C_D$  for  $A=0.1732m$ ,  $Co$  max = 2.0.



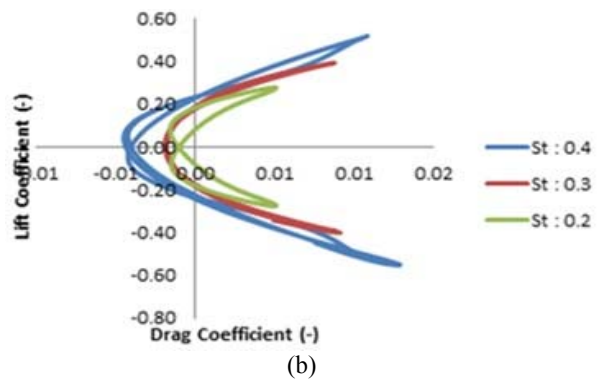
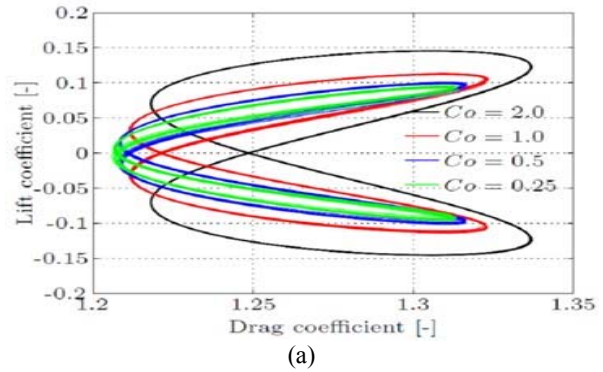
(a)



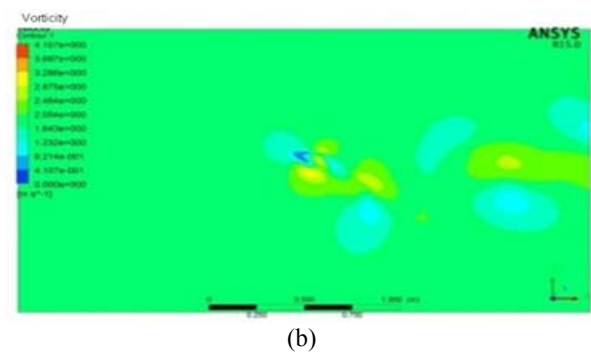
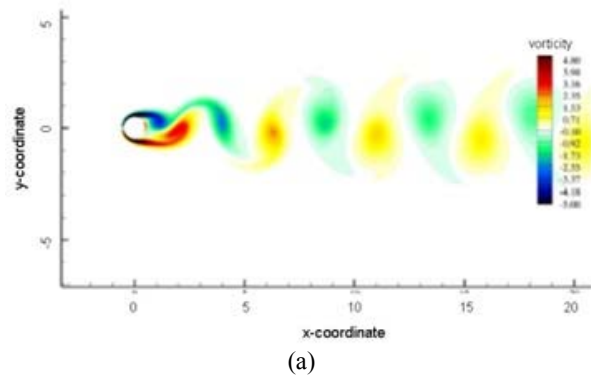
**Figure-11.** (a) $C_L$  against time-step and (b) $C_D$  against time-step for the Static Pitch simulation case.



**Figure-12.** (a)  $C_L$  against pitching angle; (b)  $C_D$  against pitching angle, for the Static Pitch simulation case.



**Figure-13.**  $C_L$ - $C_D$  comparison between results from (a) previous work by Bos (2010), with (b) current work.



**Figure-14.** Velocity contour comparison between results from (a) previous work by Bos [2], with (b) current work.



Even though the variation parameters were different, the vortex shedding for both velocity contours associated with periodic forces flow variation shown in Figure-14(a) and 13(b) have the same pattern similarities. Next, for case (ii): Static pitch, parametric study approach has been performed in order to determine the behavior of the airflow passed through a flat plate with a certain range of angle of attack variations. The trend curve was achieved by considering the angle of incidence,  $\alpha$  with respect to the free-stream velocity, which for this case is in the horizontal direction, with the heaving (representing flapping) velocity,  $v$ . In addition, it can be seen that the CL of the flat plate tends to increase with the increased heaving amplitude; the same trend is also indicated by the CD curve.

### Nomenclature

$b$	= wingspan
$c$	= chord
$C_d$	= drag coefficient
$C_L$	= lift coefficient
$L$	= total lift
MAC	= mean aerodynamic chord
St	= Strouhal number
$V_\infty$	= free-stream velocity
$v$	= flapping velocity
$\alpha$	= free-stream velocity angle
$\Gamma$	= circulation
$\rho$	= air density

### CONCLUSIONS

This present work has been performed in order to understand the effects of the flow field and forces acting on an oscillating flat plate, as a generic flapping wing, in viscous flow at certain conditions. A two-dimensional wing model has been considered, and a CFD computational simulation using ANSYS FLUENT software has been carried out. An analysis on the CFD simulation has also been elaborated to show various characteristics of the oscillating airfoil, such as the effect of flapping amplitude  $A$  to the 2-D flapping wing generated aerodynamic forces, and the influence of the flapping frequency as well as the Strouhal number, St. The influence of the kinematics of the flapping motion on the flow field and generated forces were assessed by comparison with other established results. From this CFD simulation work, 2-D visualization can to some extent offer useful insight related to the behaviour of flapping wing ornithopter. Even more information could be gained by extending the present work to 3-D model. The CFD simulation study on a generic 2-D airfoil has demonstrated its usefulness in exhibiting the basic fluid flow

characteristics. The study will be useful in providing guidelines in carrying out 3-D CFD simulation studies, further experimental work as well as preliminary considerations of ornithopter development, since the CFD analysis could reveal relevant characteristics of the fluid flow behavior.

### REFERENCES

- [1] Anderson J.D. 2009. Governing Equations of Fluid Dynamics. Chapter 2, In: J.F. Wendt (Ed.), Computational Fluid Dynamics, 3<sup>rd</sup> Ed., 15 c Springer-Verlag Berlin Heidelberg.
- [2] Bos F. M. 2010. Numerical simulations of flapping foil and wing aerodynamics; Mesh deformation using radial basis functions, PhD Thesis, Technical University Delft, The Netherlands.
- [3] Carson C., Bo Y., Luo H. 2013. San Diego CFD based optimization of a flexible flapping wing. Proceedings, 21st AIAA Computational Fluid Dynamics Conference 2013, San Diego, CA, USA, ISBN: 978-1-62748-892-1
- [4] Djojodihardjo H., Ahmed R.I., Abu-Talib A. R., Mohd-Rafie A.S. 2015a. Analytical and CFD visualization studies of Coandă MAV. for presentation at the 13<sup>th</sup> Asian Symposium on Visualization, Novosibirsk, Russia.
- [5] Djojodihardjo H., Bari M.A.A. 2014. Kinematic And Unsteady Aerodynamic Modelling Of Flapping Bi-And Quad-Wing Ornithopter, ICAS paper 0433, Proceeding, the International Congress of the Aeronautical Sciences, St Petersburg.
- [6] Djojodihardjo H., Ramli A.S.S., Bari M.A.A. 2015b. Kinematic And Unsteady Aerodynamic Study On Bi-And Quad-Wing Ornithopter, Aeroelasticity and Structural Dynamics Journal, in print
- [7] Ferziger J. H. and Peric M. 2002. Computational methods for fluids dynamics. Berlin: Springer-Verlag, Third edition.
- [8] Frank A., McGrew J., Valentiz M., Levine D. and How J.P. 2007. Hover, transition, and level flight control design for a single-propeller indoor airplane. Tech. Report, MIT, OMB No. 0704-0188.
- [9] Gerdes J. 2010. Design, analysis, and testing of a flapping wing miniature air vehicle. MSc Thesis, Maryland, USA.
- [10] Ho S., Nassef H., Pornsinsirak N., Tai Y.-C., Ho, C.-M. 2010. Unsteady aerodynamics and flow control for flapping wing flyers. Progress in Aerospace Science 39.



- [11] Hover F.S., Haugsdal Ø., Triantafyllou M.S. 2004. Effect of angle of attack profiles in flapping foil propulsion, *Journal of Fluids and Structures* Vol. 19, pp. 37-47.
- [12] Jones K. D., Platzer F. M. 2006. Bio-inspired design of flapping wing micro air vehicles - an engineer's perspective. AIAA Aerospace Sciences Meeting and Exhibit, Nevada, USA.
- [13] Kuzmin D. 2010, A Guide to Numerical Methods for Transport Equations, Friedrich-Alexander Universität Erlangen Nürnberg, [www.mathematik.uni-dortmund.de/~kuzmin/Transport .pdf](http://www.mathematik.uni-dortmund.de/~kuzmin/Transport.pdf), retrieved 22 July 2015.
- [14] Malik M. A. and Ahmad F. 2010. Effect of different design parameters on lift, thrust and drag of an ornithopter. World Congress on Engineering, 2010, Vol. 2, London, United Kingdom.
- [15] Pitt Ford C.W. and Babinsky H. 2013. Lift and the leading-edge vortex. Cambridge University Press, 2013, Vol. 720. 10.1017, United Kingdom.
- [16] D.A. Read, F.S. Hover, M.S. Triantafyllou. 2003. Forces on oscillating foils for propulsion and maneuvering *Journal of Fluids and Structures* Vol. 17, pp. 163-183.
- [17] Schroyen M. and Van Tooren M. 2009. MAV propulsion system using the Coandă effect// 45thAIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, Denver.
- [18] Shyy W., Aono H., Chimakurthi S.K., Trizila P., Kang C. K., Cesnik, C.E.S. and Liu H. 2010. Current progress in flapping wing aerodynamics and aeroelasticity. Elsevier, Vol. 46, pp. 284-327.
- [19] Triantafyllou M. S., Triantafyllou G. S. and Gopalkrishnan R. 1991, "Wake Mechanics for Thrust Generation in Oscillating Foils," *Physics of Fluids*, Vol. 3, p. 2835.