International Journal of
Advanced Thermofluid Research

ISSN 2455-1368
www.ijatr.org

An Improved Unsteady CFD Analysis of Plunging Airfoils by using OpenFoam

P. Srinivasa Murthy [1]* and M. R. Muralidharan [2]

[1]*Aeronautical Development Establishment, Bangalore-560075, INDIA.*
[2]*Supercomputer Education and Research Centre, Indian Institute of Science, Bangalore-560012, INDIA.*

Keywords
Navier Stokes Solvers • Parallel Performance • Plunging Airfoil • SnappyhexMesh • Unsteady Flow.

Received
Oct 15, 2016
Revised
Nov 27, 2016
Accepted
Nov 28, 2016
Published
Dec 01, 2016

Abstract

Unsteady Aerodynamics study of Plunging Airfoils at various frequencies and amplitude at low Reynolds Numbers of the order of Insect/Birds flight is of importance to the design and flying of Micro Aerial Vehicle to meet the requirement of Para-military forces, Border Security forces and other homeland service forces. A new methodology has been developed for the analysis of plunging airfoil based on CFD tool 'OpenFoam'. Since the experimental data available for NACA 0012 Airfoil is in plunging motion, conditions relevant to the experiment are chosen for the analysis. As plunging velocity increases there is an increase in thrust coefficient. Similar trends are observed from experiment and commercial CFD code, Fluent. However, OpenFoam agreement with experiment is better and comparable with commercial CFD code. Also, from the analysis of NACA 4412 airfoil, it has been found that Cl, Cd behavior is unimodal at low frequency and multimodal at high frequency plunging motions.

***Corresponding email**: dr_ps_murthy@yahoo.co.in (P. Srinivasa Murthy).

DOI: https://doi.org/10.51141/IJATR.2016.2.1.5

1. Introduction

There are several methods available to analyse unsteady flow starting from analytical method based on Theodorson (1935) to more complex CFD based Navier Stokes solvers (Ashraf et al. 2007). At low Reynolds Numbers of the order of insect/birds flight none of the analytical and quasi-steady methods are useful to predict the flow characteristics which are quite complex involving flow separation, transition, reattachment and unsteadiness. Hence CFD methods are the right choice although they are more costly and time-consuming, and require high performance computing systems. Since open source CFD code 'OpenFoam' is becoming more popular for implementing and evaluating High Performance Computing systems, GPU performance has been evaluated for bench mark test case and cavity flow problem, and compared with CPU performance. It has been found by

Murthy et al. (2013) that, GPU performance is 1.7 times faster than CPU performance. Although same set of Navier Stokes equations are solved in both the Openfoam and Fluent there are subtle differences in the way numerics are handled; the former is more accurate at the expense of cpu time than the later. Fluent data are used from Ashraf et al. (2007) for NACA 0012 airfoil analysis. This paper presents the solution of plunging NACA 0012 Airfoil at low Reynolds Numbers based on open source CFD code, 'OpenFoam'. The results are compared with the experimental data of Heathcote et al. (2006) and the Fluent's predictions of Ashraf et al. (2007).

2. Methods

2.1 Problem Setup and Method of Solution

Two typical airfoils, one symmetric (NACA 0012 airfoil) for which experimental and numerical data(Fluent) available for validation/comparison and the other cambered (NACA 4412 airfoil) which is a representative for practical applications in terms of designing camber for desired unsteady lift and drag characteristics are considered. These airfoils are exposed to plunging motions.

Case Analysed for NACA 0012 Airfoil: Velocity, U =0.2 m/s, Reduced frequency, Non-dimensional plunging velocity, kh=0.35, 1.0, 1.5, 2.0; Amplitude, h =0.175 m, Reynolds number, Re =20,000.

Case Analysed for NACA 4412 Airfoil: Velocity, U =50 m/s, frequency, f= 20Hz, 50Hz; Amplitude= 0.2m, 0.5m; Reynolds number, Re =2500.

Solver: SimpleFoam, Incompressible turbulent flow solver

CPU: 1 to 3 days for one flow simulation for 1.6 sec duration

Grid size: 1, 17,696 cells generated from open source BlockMesh and SnappyHexMesh which are part of the OpenFoam Library.

2.2 Boundary and Initial conditions

At the inflow boundary, the velocity is specified (velocity Dirichlet boundary condition) based on the desired Re, and the pressure is restricted to the zero-gradient condition (Neumann boundary condition). At the outflow boundary, the pressure is set to the freestream value (Dirichlet boundary condition), while the velocity is set to the zero-gradient condition (Neumann boundary condition). The far-field boundary is set to symmetry boundary condition, and is placed at an appropriate distance from the moving body surface to minimize its undesired effects on the airfoil's surrounding flow field. The stationary airfoils are set to no-slip boundary condition with fixed velocity (U = 0) and zero gradient pressure boundary condition. The unsteady moving airfoils are set to moving wall velocity boundary condition. This boundary condition guarantees the no-slip boundary condition by introducing an extra velocity to keep the flux through the moving boundary equal to zero. The solution of N-S equations also needs the initial fluid properties (velocity, pressure, kinematic viscosity, k and omega) to be specified at the start of the simulations. The steady state solutions are used as the initial conditions for the time-marching unsteady calculations with free stream velocity.

2.3 Turbulence Model

k-omega SST turbulence model is used in the incompressible Reynolds Averaged Navier Stokes solver.

2.4 New Methodology

A New methodology has been developed by means of numerical experimentation of critical parameters such as relaxation parameters, relative tolerance level, number of sub iterations, stability parameters (courant numbers) and smoothing parameters to achieve converged solution in cases of severe flow gradients. In order to get more accurate unsteady solution one need to do more rigorous analysis in terms of grid refinement in the region of severe flow gradients with higher order numerical schemes with better spatial and temporal accuracy. This requires computationally more intensive computing system with large memory and high speed.

To study unsteady variations of pressure or entropy in both time and space involves spectral analysis of pressure at each point on the surface of the airfoil. Since the requirements in time bound project were to get unsteady characteristics of lift and drag over the airfoil doing plunging motions, the analysis were confined to gross flow characteristics. Effect of angle of attack on the unsteady flow characteristics of the airfoils can be considered by imposing pitching motion over the plunging motion. Combination of pitching and heaving motions are not considered in this paper. But it will be considered at a later stage as an extension of the present work.

3. Results and Discussion

Fig. 1 shows Mean Thrust coefficient versus kh for OpenFoam, Fluent due to Ashraf et al. (2007) and Experiment due to Heathcote et al. (2006). Grid size of 91,001 hexahedral cells was used in Fluent due to Ashraf et al. (2007). Grid size of 1,17,696 hexahedral cells is used in OpenFoam. As flapping velocity increases there is an increase in thrust coefficient. Similar trends are observed from experiment and commercial CFD code, Fluent. However, OpenFoam agreement with experiment is better and comparable with commercial CFD code. The reason could be due to the better grid distribution generated by snappyhexmesh grid generator and the numerical algorithm designed in OpenFoam.

Fig 2 shows Mean Propulsive efficiency versus kh for OpenFoam, Fluent due to Ashraf et al. (2007) and Experiment due to Heathcote et al. (2006). As flapping velocity increases propulsive efficiency decreases. Similar trends are observed from experiment and commercial CFD code Fluent. However, OpenFoam agreement with experiment is better and comparable with commercial CFD code. The reason could be due to the better grid distribution generated by snappyhexmesh grid generator and numerical algorithm designed in OpenFoam.

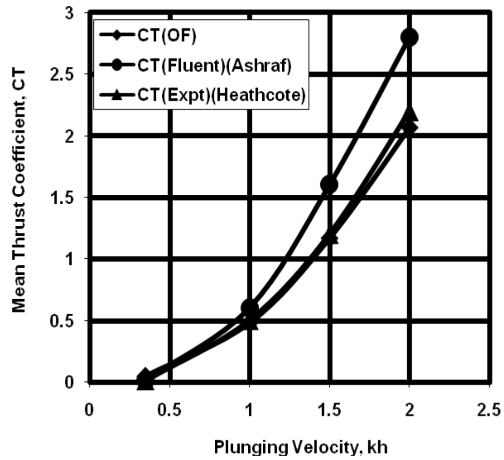

Fig. 1. Time-averaged CT versus kh, NACA0012 Airfoil, U=0.2 m/s, R= 20.000, h=0.175, Grid 1, 17,696 cells.

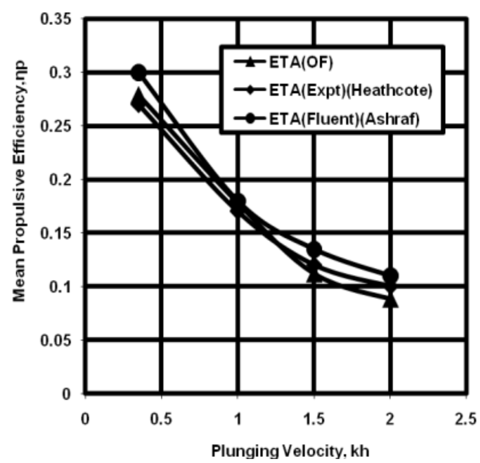

Fig. 2. ηp (Mean propulsive efficiency) versus kh (non-dimensional plunging velocity), NACA0012 Airfoil, U=0.2 m/s, R= 20.000, h=0.175,Grid 1,17,696 cells.

Figs. 3-4 show Cl and Cd time history plot of plunging NACA 4412 Airfoil for U=50 m/s, Frequency=20Hz, h=0.2m, Reynolds number =2500 and Strouhal number 0.08. Mean Cl is 0.04418. This is the unsteady lift generated at zero angle of attack (plunging axis is horizontal). Effect of positive camber in producing positive lift is evident from this plot. Mean Cd is 0.00059. This is the unsteady drag generated at zero angle of attack (plunging axis is horizontal). This is the lift dependent drag due to camber. An interesting observation from the Figs. 3-4 is Cl and Cd variations are unimodal with constant amplitude and frequency. The reason could be due to the vortex shedding periodically with constant amplitude and frequency. But this happens at plunging frequency of 20 Hz and amplitude of 0.2m.

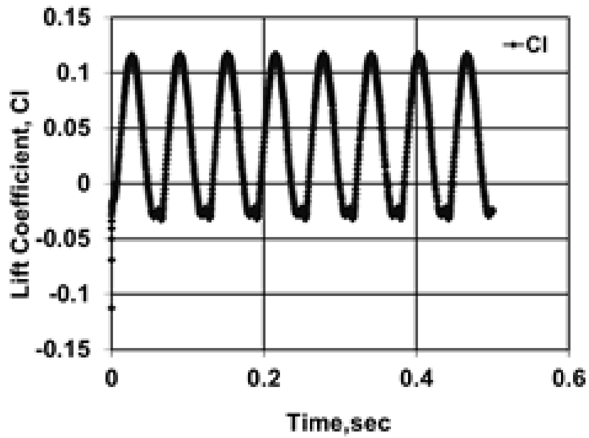

Fig. 3. Cl versus time, NACA4412 Airfoil, U=50 m/s, Amplitude=0.2m, Frequency= 20Hz, R=2500, St No.=0.08, Av Cl = 0.04418.

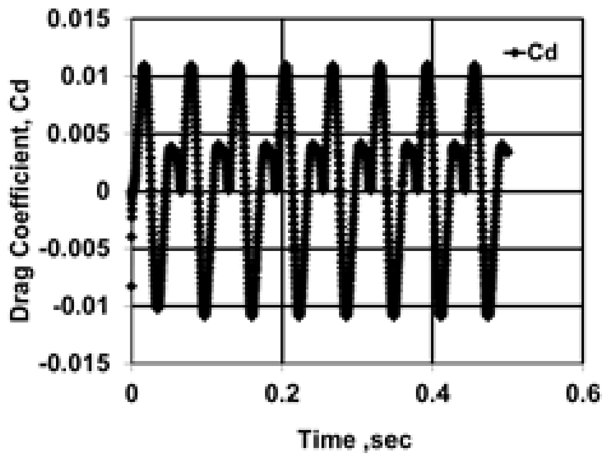

Fig. 4. Cd versus time, NACA4412 Airfoil, U=50 m/s, Amplitude=0.2m, Frequency= 20Hz, R=2500, St No.=0.08, Av Cd = 0.00059.

Figs. 5-6 show Cl and Cd time history plot of NACA 4412 Airfoil for U=50m/s, Frequency=50Hz, h=0.5m, R No. 2500, St No. 0.5. Mean Cl is -0.39047. This is the unsteady lift generated at zero angle of attack (plunging axis is horizontal). Effect of positive camber in producing positive lift is not evident from this plot because of plunging frequency, 50 Hz and amplitude 0.5m. Mean Cd is -0.133. This is the unsteady drag generated at zero angle of attack (plunging axis is horizontal). This is the lift dependent drag due to camber at plunging frequency, 50Hz and amplitude 0.5m. It is to be noticed that Cd is negative which means airfoil is producing thrust at the plunging frequency of 50Hz and amplitude 0.5m An interesting observation from the Figs. 5-6 is Cl and Cd variations are multi-modal with varying amplitudes and frequencies. The reason could be due to the vortices of various sizes shedding periodically with varying amplitudes and frequencies at different instants of time. But this happens at plunging frequency of 50 Hz and amplitude 0.5m. It has been found that at low plunging frequency, Cl and Cd behaviour is uni-modal with constant amplitude. But at high plunging frequency, Cl and Cd behaviour is multi-modal with varying amplitude. This important finding is useful to design cambered airfoil to get the desired unsteady results

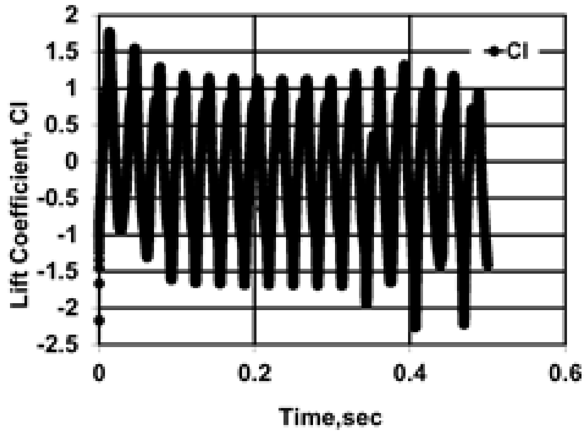

Fig. 5. Cl versus time, NACA4412 Airfoil, U=50 m/s, Amplitude=0.5m, Frequency= 50Hz, R=2500, St No.=0.5, Av Cl = -0. 39047.

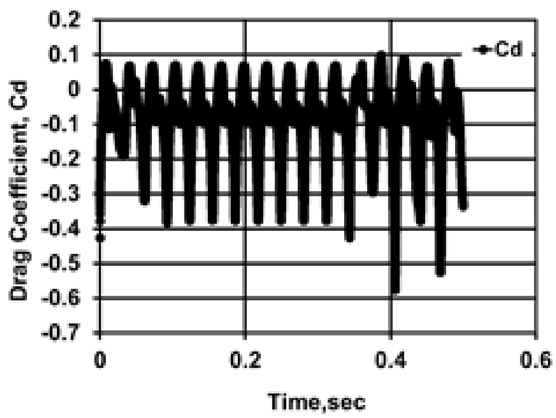

Fig. 6. Cd versus time, NACA4412 Airfoil, U=50 m/s, Amplitude=0.5m, Frequency= 50Hz, R=2500, St No. =0.5, Av Cd = -0.133.

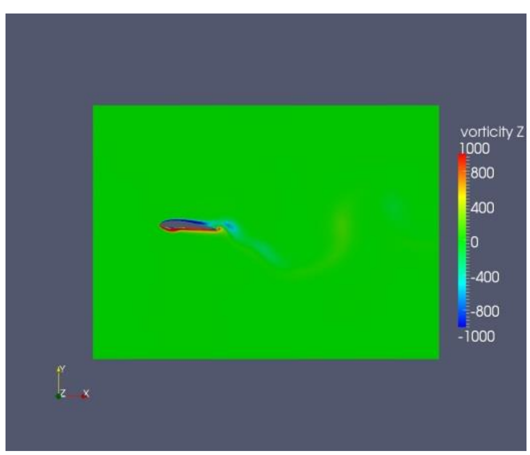

Fig. 7. NACA 4412 Airfoil, Vorticity-Z Contours, t=0.07 sec, U=50 m/s Amplitude=0.2m, Frequency=20Hz, R=2500, St No=0.08, Grid 12,559.

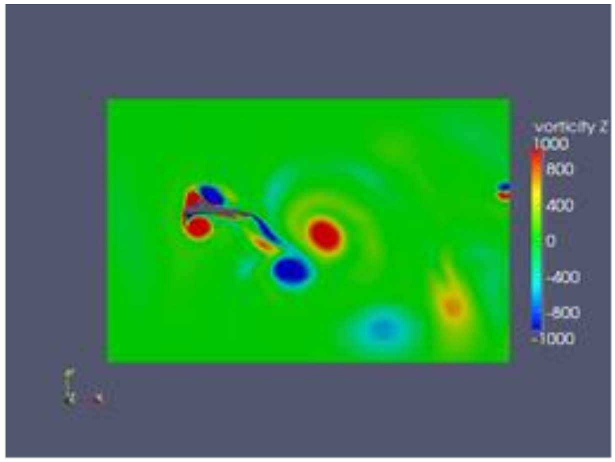

Fig. 8. NACA 4412 Airfoil, Vorticity-Z Contours, t=0.07 sec, U=50 m/s Amplitude=0.5m, Frequency =50Hz, R=2500, St No=0.5, Grid 12,559.

Figs. 7-8 show Vorticity-Z contours for NACA 4412 Airfoil at low and high frequency plunging motions at t=0.07 sec. In Fig. 7 (which is a low frequency low amplitude plunging motion) the vortex shedding pattern is confined to bottom surface and trailing edge region with small amount of vorticity. But at high frequency and high amplitude plunging motion (Fig. 8), the vortex pattern is distributed all over the surface from leading edge to trailing edge. Large amount of vorticity is generated all around the surface.

4. Conclusion

CFD Analysis of plunging NACA0012 Airfoil has been carried out by using OpenFoam incompressible turbulent flow solver SimpleFoam, and the results are compared with Fluent and Experimental data. OpenFoam solutions match the experiment and show good agreement with Fluent. NACA 4412 Airfoil unsteady results indicate that Cl, Cd behavior is uni-modal at low frequency plunging motion while it is multi-modal at high frequency plunging motion.

Acknowledgments

The authors would like to express gratitude for the funding received by the AR&DB project for carrying out the work and are grateful to the project team for their support and cooperation.

References

Ashraf, MA, Lai, JCS and Young, J. (2007). Numerical Analysis of Flapping Wing Aerodynamics, 16th Australasian Fluid Mechanics Conference, Crown Plaza, Gold Coast, Australia, 2-7 December.

Heathcote, S and Gursul, I. (2006). Effect of Spanwise Flexibility on Flapping Wing Propulsion, 35th AIAA Fluid Dynamics Conference and Exhibit.

Murthy SP, Muralidharan MR, Rao SU, Prasanti T. (2013). Parallel Performance of GPU and CPU in Computational Fluid Dynamics, The 14th Asian Congress of Fluid Mechanics – 14ACFM, October 15-19, 2013, Hanoi and Halong, Vietnam.

Theodorsen, T. (1935). General Theory of Aerodynamic Instability and the Mechanism of Flutter, NASA Report No. 496.