CFD Analysis of Flow past a Rigid Body Executing a General Two Dimensional Motion

By

SYED FAHAD ANWAR
Department of Applied Mechanics

Submitted
in fulfillment of the requirements of the degree of Doctor of Philosophy
to the

Indian Institute of Technology Delhi

January'2007

· Computational fluid dynamics
· Numerical analysis
· Computational mathematics

Certificate

This is to certify that the thesis entitled **CFD Analysis of Flow past a Rigid Body Executing a General Two Dimensional Motion** being submitted by **Syed Fahad Anwer** to the **Indian Institute of Technology New Delhi (India)** for the award of the degree of Doctor of Philosophy in Department of Applied Mechanics is a bonafide research work carried out by him under my supervision and guidance. The research results reports and results presented in this thesis have not been submitted in parts or in full to any other University or Institute for the award of any degree or diploma.

Sanjeev Sanghi, Ph. D.

Professor
Department of Applied Mechanics
Indian Institute of Technology
New Delhi -110016
India

Sudipto Mukherjee, Ph. D.

Professor
Department of Mechanical Engineering
Indian Institute of Technology
New Delhi -110016
India

Acknowledgement

I would like to take this opportunity to thank those who helped and supported me throughout this effort. In the capacity of my research supervisors Prof. Sanjeev Sanghi and Prof. Sudipto Mukherjee gave me total freedom and their able guidance to develop me into a researcher. The presentation of my thesis has been largely due to their efforts and guidance. I would like to thanks my mentor and my teacher Dr. Nadeem Hasan for his expert suggestions without which this work was not possible. I would also like to thank Prof. Krishnan V. Pagalthivarthi and Prof. V. Sheshadri to be part of my research committee and his critical comments which were invaluable suggestions which helped me in improving the quality of the work.

I would also like to thank my Lab colleagues who weathered my anger and supported me in all ways. I would also like to thank my mother because it is she who wished that I should have a doctorate and still confidence and patience in me for carryout this work. Above all, thanks to the Almighty for showing me the light in the moments of darkness.

Syed Fahad Anwer

Abstract

The primary objective of this work is to develop a computational fluid dynamic tool which is computationally efficient and accurate in predicting fluid dynamics generated by or due to motion of body or boundaries. In this process, a mesh movement algorithm and an outflow boundary condition is developed. The mesh movement algorithm and the outflow boundary condition are then combined with a structured body fitted grid based flow solver for incompressible laminar flows. The computational methodology developed herein is applied to two flow situations pertaining to bio-fluid dynamics for which results were not known earlier. The flow generated by an elliptical airfoil executing normal flapping and rowing motion for different parameters are explored.

The moving mesh in the physical domain is mapped to a regular fixed mesh in the computational domain through a time dependent transformation between the physical and computational co-ordinates. The governing equations of laminar incompressible flow are transformed in the computational plane by incorporating the time dependent transformation which naturally accounts for the mesh velocities. The transformed equations are discretized on the structured, collocated, o-type elliptic grid using the finite difference methodology. The unsteady equations are marched in time by using a semi-implicit pressure correction (projection) scheme. Along with the time marching of the governing equations, the mesh points are also moved by utilizing the mesh velocities and the forward Euler time integration. A number of test problems, involving flows around bodies executing two dimensional translational and rotational motion have been employed to assess the robustness and accuracy of the overall computational procedure. The results of the bench mark cases show that the procedure is quite robust and yields accurate results.

This work also presents a new procedure for extrapolating velocities at the outflow boundary in the computations of incompressible flows around rigid bodies. The extrapolation procedure is based on the radial variation of the velocity field at large distances from the rigid body, which can be inferred from mass conservation and vorticity considerations. Since the extrapolation is based on these physical considerations, the proposed boundary condition is considered to be physically consistent. It has been demonstrated, via numerical simulations of 2D, laminar, incompressible, viscous flow past a circular and a square cylinder at Re = 100, that the application of the proposed boundary condition allows one to limit the unbounded domain to a small size (6–8 times the characteristic size of the body) without any significant change in the flow characteristics like the lift coefficient (C_L) and the Strouhal number (St). Thus, the proposed boundary conditions can enhance the computational efficiency of this class of flows.

The flow generated by the interaction of a fluid with a body enacting a flapping motion is investigated. Lift and Drag characteristics are calculated as a function of time and flow changes are related to the shedding of leading edge vortex (LEV). The angle of attack of the airfoil is varied from 45° and 70° and the Reynolds number (Re) is varied from 20 to 128. On the basis of vortex shedding, the flow can be divided into two regimes. In the first regime, there is no vortex shedding during the translational phase of the wing's motion from the leading edge or the trailing edge. In the second regime the vortices are shed from the trailing edge in the translational phase to generate a vortex street. The transition from one regime to the other occurs at Re = 40. Further a new measure of efficacy is proposed which shows a maxima at Re = 60 while the steady state efficiency measure of coefficient of lift to drag ratio shows a monotonic variation. So it can be concluded that the power efficiency could be suggested as an effective measure for evaluating a wing stroke.

Elliptic airfoil model which mimics the biological locomotion is studied. The airfoil undergoes a combined plunging and translation at low Reynolds number. For a given frequency of plunging a thrust is generated when the Re exceeds a critical value (critical Reynolds number, Re_c). The optimal frequency which corresponds to peak thrust is found to occur at a reduced frequency parameter, St_c, of 0.7 is independent of Re. The mode of vortex shedding is same at Re = 100 and 200 for $St_c = 0.7$. A new mode of vortex shedding is also reported in which vortices are shed as dipoles from trailing as well as leading edge.

A study of a plunging and pitching elliptic airfoil incident with a uniform inflow is also presented. This flow situation is same as the flow generated by a translating, pitching and plunging airfoil in quiescent fluid. Numerical simulations are carried for three different set of Re and frequency. The vortices arrange themselves as a deflected von Karman wake. The flow shows no transition from periodic to quasi-periodic with an increase in pitch amplitude as the power spectrum shows periodic flow while for Re = 500, flow develops a thrust at all pitch amplitudes.

Table of Contents