Computational modeling of unsteady low Reynolds number flapping wing aerodynamics

Dominic Denver John Chandar

2010

Dominic, D. J. C. (2010). Computational modeling of unsteady low reynolds number flapping wing aerodynamics. Doctoral thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/47219>

<https://doi.org/10.32657/10356/47219>

Nanyang Technological University

Downloaded on 26 Apr 2023 13:27:29 SGT

255
4X
2/Blue

COMPUTATIONAL MODELING OF UNSTEADY LOW REYNOLDS NUMBER FLAPPING WING AERODYNAMICS

DOMINIC DENVER JOHN CHANDAR

School of Mechanical and Aerospace Engineering

A thesis submitted to the Nanyang Technological University
in partial fulfillment of the requirement for the degree of
Doctor of Philosophy

2010

ABSTRACT

The unsteady low Reynolds number aerodynamics of biomimetic airfoils and wings stands out as one of the most challenging areas of research to the scientific community. As opposed to steady flow where substantial amount of research has been accomplished in understanding the concepts of separation and transition, apart from turbulence, unsteady low Reynolds number flows have not gained much attention except for the past few years owing to difficulties in modeling and simulating such complex flows on arbitrary domains of interest like insect and bird flight, flutter prediction etc. The basic underlying mechanisms of lift and thrust production by birds and insects are well known and have been established by early researchers through extensive experiments and lower order numerical computations, supported by classical theories of unsteady aerodynamics. Nevertheless, to date, experiments and numerical computations of this problem have been focused on predicting aerodynamic forces and moments with specified wing kinematics. The airfoil or wing under study is usually constrained to move along a particular direction with a fixed freestream velocity. The rigid body dynamics of plunging/pitching/flapping bodies have however, received less attention. In the present work, numerical computations have been carried out to simulate the dynamics of plunging airfoils and wings in incompressible flow by coupling rigid body dynamics to the governing equations of fluid motion. In this case, the body accelerates from the state of rest to the state of motion due to the aerodynamic forces and traces a particular trajectory. During the course of the study, the effect of prescribed wing flexibility on forward flight has also been analyzed. A fluid-structure coupling algorithm has been formulated to allow for airfoil and wing deformation, and critical numerical issues relating to the coupling are dealt with in detail. It is shown that, when the deformation of an airfoil is prescribed, under certain conditions, the airfoil with the highest deformation amplitude travels the fastest. However when the deformation is passive, or is obtained from the forces, the airfoil with a moderate flexibility travels the fastest.

Since the ambient flow of a flapping wing is filled with vortices of varying sizes depending on the wing kinematics, it is instructive to understand the origin of thrust from the viewpoint of these surrounding vortices. The standard expressions for the aerodynamic forces which are based on pressure and shear stress are not much useful. Hence, a non-conventional approach based on vorticity moments has been extended to prove that asymmetric vorticity generation is necessary for thrust generation, where the asymmetry here is between the vorticity produced at the leading and trailing edges of the wing. Different test cases corresponding to symmetric and asymmetric wing sections, with symmetric and asymmetric kinematics have also been investigated. All numerical computations addressed in this thesis have been performed by solving the unsteady incompressible Navier-Stokes equations coupled with rigid body dynamics and structural dynamics on overlapping or overset grids. Comparative analysis with existing experimental / computational data is also presented.

ACKNOWLEDGMENTS

This research project has been made possible by the collective help of many individuals. I cannot but initiate my statements without the all important mention about my supervisor, Dr Murali Damodaran. I gratefully record the fervent support and encouragement provided by him. When researchers face seemingly insurmountable problems in their quest for answers, it is the able and relentless guidance by the supervisors which brings laurels to their wards and dispels skepticism. I would like to thank him unconditionally for having given me a broad range of experience in overcoming such impediments as often as they arose during the formation of this work. I owe him my gratitude for allowing me, to present my work at different international conferences, providing thereby the immeasurable opportunity to meet people with similar research interests. I would also like to thank him for having given me an opportunity to work as a teaching assistant for the Singapore-MIT Alliance programme in computational engineering.

I wish to express my sincere gratitude to Dr William Henshaw and Dr Kyle Chand and the rest of *Overture* team at the Center for Applied Scientific Computing (CASC), Lawrence Livermore National Laboratory- California for their valuable suggestions and comments and for the extensive usage of the *Overture* code. I appreciate the timeless concern, the developers of the *Overture* have demonstrated, in sharing their ideas through an excellent forum.

I wish to thank the Computer Administration Team at the Bio-Informatics Research Centre (BIRC) who helped me with the installation of *Overture* as far back as 2006 and Mr Teo Hai Beng of CANES Lab for his relentless support in handling all administrative issues relating to the lab and the linux server.

A special thanks to Dr Kuan Tek Seang for having given me privileges to use his QUAD processor Linux computer, without which, I would not have been able to produce very quick results for the computations undertaken as a part of this research.

I would like to thank Prof Wei Shyy and Dr Hikaru Aono from the University of Michigan, U.S.A and Prof Yongsheng Lian from the University of Louisville, U.S.A for the initial discussions on the fluid structure interaction problem for flapping wings.

I would also like to thank my colleagues Ramesh, Shantanu, Raman, John and Pugal for making my stay very special and memorable.

I wish to acknowledge the loving support, moral encouragement and prayers of my parents Dr Alan and Olga. They have missed a lot due to my absence and have been very patient during the last few years. I feel very privileged to have wonderful parents who have allowed me to choose a career that I have always liked and I thank them for having dreamt my dreams!

I owe a special thanks to my wife, Ajitha who has been rather instrumental in helping me to cope up with various problems during stressful times of my research. We shared a lot of ideas technically and I have received valuable suggestions from her to make this work fruitful. I admire her effortless way of checking my thesis and other papers which I had written during my research.

My daughter Shruti Amelia, who was born on Oct 2009, has been a tremendous source of inspiration. It has been rewarding to have her amongst all of us especially during the last phase of my research.

I would not be at ease if I did not mention the grace I have received from God to pass through all the difficult moments during my research.

Financial support from the NTU graduate scholarship fund is gratefully acknowledged with due importance.

Dominic Denver John Chandar
Singapore, March 2010

TABLE OF CONTENTS

ABSTRACT	1
ACKNOWLEDGMENTS.....	3
LIST OF FIGURES	7
LIST OF TABLES	14
NOMENCLATURE	15
CHAPTER 1. INTRODUCTION.....	17
1.1. EARLY OBSERVATIONS AND CURRENT PROGRESS	19
1.2. MOTIVATION.....	25
1.2.1. <i>Passive Flight</i>	26
1.2.2. <i>Dynamic Stall and Lift Hysteresis</i>	26
1.2.3. <i>Vorticity Dynamics</i>	29
1.2.4. <i>Wing Deformation and Fluid-Structure Interaction</i>	29
1.3. OUTLINE	31
CHAPTER 2. OVERLAPPING GRID GENERATION	34
2.1. COMPOSITE GRID CONSTRUCTION	37
2.2. INTERPOLATION ON AN OVERLAPPING GRID	38
2.3. GRID GENERATION FOR DEFORMING BODIES	43
2.4. OVERLAPPING GRIDS USED FOR DIFFERENT COMPUTATIONS.....	49
2.4.1. <i>Overlapping Grid for an Airfoil</i>	49
2.4.2. <i>Overlapping Grid for a Deforming Airfoil</i>	51
2.4.3. <i>Overlapping Grid for a Deforming Tear-Drop Plate</i>	51
2.4.4. <i>Overlapping Grid for a Deforming Cantilever Beam in a Channel</i>	54
2.4.5. <i>Overlapping Grid for a Wing</i>	54
CHAPTER 3. COMPUTATIONAL FLOW MODELING	61
3.1. DISCRETIZATION OF THE GOVERNING EQUATIONS	62
3.2. BOUNDARY CONDITIONS	64
3.2.1. <i>Wall Boundary Conditions</i>	64
3.2.2. <i>Other Physical Boundaries</i>	65
3.3. RIGID BODY DYNAMICS: COUPLING WITH NAVIER STOKES EQUATIONS	68
3.4. ARTIFICIAL DISSIPATION	69
3.5. TIME DISCRETIZATION AND OVERALL SOLUTION PROCEDURE.....	72
CHAPTER 4. NUMERICAL VERIFICATION AND VALIDATION FOR UNSTEADY FLOW PROBLEMS	74
4.1. INTRODUCTION	74
4.2. NON-DIMENSIONAL PARAMETERS	74
4.2.1. <i>Expressions for Non-Dimensional Parameters for a Plunging and Pitching Wing</i>	75
4.2.2. <i>Expressions for Non-Dimensional Parameters for a Free Falling Body</i>	77
4.3. GRID REFINEMENT STUDY FOR A PLUNGING AIRFOIL.....	77
4.4. COMPARISON OF PLUNGING AIRFOIL WAKE STRUCTURES WITH EXPERIMENTS	82
4.5. RAPID PITCHING OF NACA 0015 AIRFOIL – SIMULATION OF DYNAMIC STALL	85
4.6. AUTOROTATION AND RIGID BODY FLUTTER OF RECTANGULAR/ELLIPTICAL PLATES	88
4.6.1. <i>Autorotation of an Elliptic Cylinder</i>	88
4.6.2. <i>Passive Rotation of Rectangular Plates</i>	94
4.6.3. <i>Free Fall of a Rectangular Plate</i>	94

4.7.	EFFECT OF THE LOCATION OF OUTER BOUNDARY ON AERODYNAMIC CHARACTERISTICS	99
4.8.	NACA 0012 CROSS-SECTION WING PLUNGING AT $Re=10000$	101
4.9.	RAPID ACCELERATED MOTION OF A WING	105
CHAPTER 5. AERODYNAMIC CHARACTERISTICS OF RIGID FLAPPING WINGS.....		107
5.1.	INTRODUCTION	107
5.2.	ACTIVE FLIGHT COMPUTATIONS.....	107
5.2.1.	<i>Plunging Airfoil – Effect of L.E-T.E Asymmetry</i>	<i>107</i>
5.2.2.	<i>Effect of Trailing-edge Topology on Thrust Generation</i>	<i>112</i>
5.2.3.	<i>Lift Hysteresis Reversal for a Rapidly Pitching Airfoil</i>	<i>121</i>
5.2.4.	<i>Reverse Hysteresis for a Pitching Elliptic Airfoil and Wing of Elliptic Cross- Section</i>	<i>135</i>
5.3.	PASSIVE FLIGHT COMPUTATIONS	139
5.3.1.	<i>Effect of Trailing Edge Sharpness on Trajectory of a Flapping Airfoil</i>	<i>139</i>
5.3.2.	<i>Effect of Wing Leading-edge/Trailing-edge Asymmetry and Asymmetric Kinematics on Forward Flight</i>	<i>142</i>
CHAPTER 6. COMPUTATIONAL STRUCTURAL DYNAMICS MODELING FOR FLUID-STRUCTURE INTERACTION		151
6.1.	BOUNDARY CONDITIONS	152
6.2.	DISCRETIZATION	153
6.2.1.	<i>Discretization of Boundary Conditions</i>	<i>155</i>
6.3.	SPATIAL AND TEMPORAL ACCURACY OF THE DISCRETIZED EULER-BERNOULLI BEAM EQUATION.....	156
6.4.	COUPLING OF FLUID AND STRUCTURAL SOLVERS	159
6.5.	NUMERICAL VALIDATION FOR DEFORMING BODY AERODYNAMICS	163
6.5.1.	<i>Plunging Motion of a Flexible NACA 0014 Airfoil</i>	<i>163</i>
6.5.2.	<i>Oscillation of a Thin Flexible Plate in an Incompressible Fluid.....</i>	<i>167</i>
CHAPTER 7. AERODYNAMIC CHARACTERISTICS OF FLEXIBLE FLAPPING WINGS.....		169
7.1.	INTRODUCTION	169
7.2.	FREE FLIGHT OF A DEFORMABLE TEAR DROP PLATE	169
7.3.	EFFECT OF DEFORMATION KINEMATICS ON THRUST GENERATION	173
7.3.1.	<i>Simple Flapping Mode</i>	<i>175</i>
7.3.2.	<i>Combined Flapping Mode.....</i>	<i>178</i>
7.4.	PLUNGING AND PASSIVELY DEFORMING AIRFOIL – EFFECT OF TIME STEP, RELAXATION AND DAMPING	182
7.5.	FORWARD FLIGHT OF A PLUNGING AND PASSIVELY DEFORMING AIRFOIL	186
7.6.	SPANWISE DEFORMATION OF A NACA0012 CROSS-SECTION WING	188
CHAPTER 8. CONCLUSIONS AND RECOMMENDATIONS FOR FUTURE WORK.....		195
8.1.	CONCLUDING REMARKS	195
8.2.	RECOMMENDATIONS FOR FUTURE WORK.....	197
REFERENCES		201
APPENDIX A1 - NAVIER-STOKES EQUATIONS AND NON-DIMENSIONAL PARAMETERS.....		212
APPENDIX A2 - SOLVER STATISTICS		214
APPENDIX A3 – SIMULATION VIDEOS.....		216

LIST OF FIGURES

Figure 1.1: Different Flying Species found in Nature, (a) Hummingbird (b) Honey Bee and (c) Barn Swallow..... 18

Figure 1.2: Speeds of various Flying Species, Mammals and Aircrafts in terms of Body Lengths..... 20

Figure 1.3: Man made Flapping Wing Robots (a) The I-Bird Toy and (b) Microfly 20

Figure 1.4: Various Lift Hysteresis Curves for Sinusoidal and Triangular type Pitching 27

Figure 2.1: Overlapping Grid in the Physical and Computational Space for a (a) Cylinder (b) Sphere 36

Figure 2.2: A Summary of the Overlapping Grid Generation Procedure 39

Figure 2.3: Individual Component Structured Grids 40

Figure 2.4: The Grid After Cutting Holes..... 41

Figure 2.5: The Grid After Classifying Discretization, Interpolation and Unused points 41

Figure 2.6: The Grid after Marking Required Interpolation Points 42

Figure 2.7: The Final Composite (Overlapping)Mesh..... 42

Figure 2.8: Interpolation on an Overlapping Grid 43

Figure 2.9: Hyperbolic Mappings and the Corresponding Overlapping Grid for a Deforming Ellipse at Various Time Instants 46

Figure 2.10: Portion of the Mesh Near the Trailing-edge for Different Marching Distances 47

Figure 2.11: Variation of Mesh Dilation as a Function of Time (a) At Point A and (b) At Point B as Indicated in Figure 2.10..... 48

Figure 2.12: NACA 0012 Airfoil Showing Various Edges and the Corresponding Flags 50

Figure 2.13: The Overlapping Grid for the NACA 0012 Airfoil..... 50

Figure 2.14: Hyperbolic Grid for a Deforming NACA 0014 Airfoil 52

Figure 2.15: Overlapping Grids for a Deforming NACA 0014 Airfoil..... 52

Figure 2.16: Overlapping Grids Surrounding a Tear-Drop Plate Showing the Rigid Fore Part and a Flexible Tail.	53
Figure 2.17: Problem Setup for the Cantilever Beam in the Channel.	55
Figure 2.18: Overlapping Grids for the Cantilever Beam in a Channel	55
Figure 2.19: Overlapping Grids for a NACA 0012 Cross-Section Wing (a) Surface Grid with a Singularity (b) Volume Grids for the Wing Tips and (c) Volume Grids for the Wing	57
Figure 2.20: Overlapping Grids for the Wing Showing the Inner Cartesian Grid (Green Boundary) and the Outer Cartesian Grid (Blue Boundary)	58
Figure 2.21: Spanwise Deflection Profiles for Different Values of Q	58
Figure 2.22: Overlapping Grid for a Wing Deforming in the Spanwise Direction	59
Figure 3.1: Application of Divergence Free Condition at a Wall Determines the Velocity at the Ghost Point.....	66
Figure 3.2: Other Boundary Conditions for the Flow Past a Wing in Three-Dimensions	66
Figure 4.1: Coordinate System and Wing Kinematical Parameters for Rotation about (a) X-axis and (b) Z-axis.....	76
Figure 4.2: (a) Lift Coefficient (b) Near View of the Lift Coefficient	79
Figure 4.3: (a) Drag Coefficient (b) Near View of the Drag Coefficient	80
Figure 4.4: Time Averaged Lift and Drag Coefficients as a Function of Grid Points.....	81
Figure 4.5: Plunging Airfoil Wake structures for $k=5.05$, $h_0=0.2$ (a) Current Computation (b) Experiments from Ref [122] (c) Experiments from Ref [123] at a Later Instant of Time(d) Computation with Zero Freestream Conditions and (e) Computation with Finite Freestream Conditions	83
Figure 4.6: Lift of a Plunging Airfoil with a Dual Mode Vortex Street , $k=5.05$, $h_0=0.2$	84
Figure 4.7: Computed Vorticity Contours for a Rapidly Pitching Airfoil at (a) $Re = 10^4$ on 200 x 100 Mesh and (b) $Re = 4.5 \times 10^4$ on 300x50 Mesh.....	86
Figure 4.8: Computed Aerodynamic Characteristics for a Rapidly Pitching Airfoil at (a) $Re = 10^4$ (b) $Re=4.5 \times 10^4$ on Various Grids	87
Figure 4.9: Overlapping Mesh for the Autorotation Case	90
Figure 4.10: Computed (a) Vorticity Contours from Lugt [128] (b) Vorticity Contours	

from the Present Computation and (c) λ_2 Contours from the Present Computation.....	90
Figure 4.11: Computed Moment Coefficients as a Function of Angle of Attack for (a) $S_p = 0.167$ (b) $S_p = 0.25$ (c) $S_p = 0.5$ (d) $S_p = 1.0$	92
Figure 4.12: Computed (a) Lift and (b) Drag Coefficients as a Function of Angle of Attack for $S_p = 0.1$	93
Figure 4.13: Time History of the Computed Lift Coefficient in Comparison with the Results of Mittal [132] for (a) $\beta = 0.5$ (b) $\beta = 0.2$	95
Figure 4.14: Free Fall of a Rectangular Plate Showing the Experimental [120] and Computational Setup.....	96
Figure 4.15: Free Fall of a Rectangular Plate Showing the Experimental [120] and Computational Trajectories.....	98
Figure 4.16: Aerodynamic Forces (a) F_x (b) F_y , Normalized with the Apparent Weight as a Function of Instantaneous Angle of Inclination	98
Figure 4.17: Effect of Outer Boundary Distance on (a) Thrust Coefficient and (b) Trajectory	100
Figure 4.18: Comparison of Computed Aerodynamic Coefficients with Available Data, Adapted from Heathcote et al. [36]	103
Figure 4.19: Comparison of Computed Thrust Coefficients with Experiments from Heathcote et al. [36].....	104
Figure 4.20: Computational Setup for the Accelerating Wing	106
Figure 4.21: Time History of the Aerodynamic Force Coefficients for the Accelerating Wing.....	106
Figure 5.1: Vorticity Contours for a Plunging Airfoil in Zero Freestream Conditions ..	108
Figure 5.2 : Comparison of Drag Coefficients for a Plunging Elliptical and NACA 0012 Airfoil in Zero Freestream Conditions.....	108
Figure 5.3: Illustration of the Region Occupied by the Wing and the Fluid Bounded at Infinity.....	109
Figure 5.4: Computed Variation of Q_v over one Cycle for the NACA 0012 Airfoil and the Elliptical Airfoil	111
Figure 5.5: Airfoil Trailing Edge Topology for Different Roundness Parameters.....	113
Figure 5.6: (a) Computed Wake Structures Behind a Plunging Airfoil (case 4) at $St = 1.01$	

(b) The Overlap Region Showing the Interpolation Points around the Airfoil Mesh and Vorticity Contours.....	114
Figure 5.7: Instantaneous Streamlines for a Plunging Airfoil (a) Base Airfoil (b) Airfoil case 4 at $t = 3.5$ s	115
Figure 5.8: Time History of the Drag Coefficient for Plunging Airfoils under (a) Finite and (b) Zero Freestream conditions for various T.E radii.....	117
Figure 5.9: Instantaneous Streamlines for a Plunging (a) Base Airfoil (b) Case 4 Airfoil at its Mean Position	118
Figure 5.10 (a) Plunge Displacement (b) Plunge Velocity	119
Figure 5.11: A Comparison of the Time History of the Drag Coefficient With and Without Ground Effect.....	119
Figure 5.12: Time Averaged Velocity Profiles Downstream of the Airfoil Located at 0.4c aft the Trailing Edge for Plunging in (a) Zero Freestream (b) Finite Freestream conditions	120
Figure 5.13: Computed Lift Hysteresis Curves for a Pitching Airfoil at $Re = 100$ for Reduced Frequencies (a) $k = 0.125 - 1.0$ and (b) $k = 2.0 - 5.0$	122
Figure 5.14: Hysteresis Loops of C_L vs. α Computed from the Mathematical Model for Different Phase Lags	125
Figure 5.15: Lift Coefficient and Angle of Attack during the Pitching Cycle for Various Reduced Frequencies	128
Figure 5.16 (a)-(c)Pressure and (d)-(f)Vorticity Contours for Reduced Frequency $k =$ 0.25 at Positions A, B, C in the Oscillation Cycle as Indicated in Figure 5.17(a)	129
Figure 5.17: (a)-(b) Pressure and (c)-(d) Vorticity Contours for Reduced Frequency $k =$ 0.25 at Positions D, E in the Oscillation Cycle as Indicated in Figure 5.17(a)	130
Figure 5.18 (a)-(c)Pressure and (d)-(f)Vorticity Contours for Reduced Frequency $k = 2.0$ at Positions A, B, C in the Oscillation Cycle as Indicated in Figure 5.17(b)	131
Figure 5.19: (a)-(b) Pressure and (c)-(d) Vorticity Contours for Reduced Frequency $k =$ 2.0 at Positions D, E in the Oscillation Cycle as Indicated in Figure 5.17(b)	132
Figure 5.20: (a)-(c)Pressure and (d)-(f)Vorticity Contours for Reduced Frequency $k =$ 5.0 at Positions A, B, C in the Oscillation Cycle as Indicated in Figure	

5.17(c)	133
Figure 5.21: (a)-(b) Pressure and (c)-(d) Vorticity Contours for Reduced Frequency $k = 5.0$ at Positions D, E in the Oscillation Cycle as Indicated in Figure 5.17(c)	134
Figure 5.22: Lift Coefficient per Unit Span for a Pitching Wing in Comparison with a Pitching Airfoil for a Reduced Frequency $k = 5.0$	136
Figure 5.23: Aerodynamic Force Coefficients per Unit Span for a Pitching Wing in Comparison with a Pitching Airfoil for a Reduced Frequency $k = 5.0$	137
Figure 5.24: (a), (b) Pressure Distributions at Mid-Span for a Pitching Wing for Reduced Frequency $k = 5.0$ (c) Corresponding Two-Dimensional Case	138
Figure 5.25: Streamlines w.r.t the Inertial Frame for Passive Flight at Various Time Instants and for Different Cases	140
Figure 5.26: (a) Trajectory of the Airfoil and (b) Forward Speed for Different Trailing-Edge Roundness Parameters	141
Figure 5.27: (a) Wing Kinematics About Various Co-ordinate Axes and the (b) Corresponding View in the XY Plane	143
Figure 5.28: Comparison of Computed Thrust Coefficients and Trajectories for a NACA 0012 Wing with an Elliptical Cross-Section Wing for Flapping about (a)-(b) X-axis (c)-(d) Z-axis (e)-(f) X- and Z-axes	144
Figure 5.29: Computed Vorticity Contours at the Beginning of the Cycle ($t = T$) and at Half Cycle ($t = T + \frac{1}{2} T$) for a Flapping NACA 0012 Cross-Section Wing for Rotation about (a) –(b) X-axis, (c)-(d) Z- axis (e)-(f) X- and Z-axes, (T being the Period of one Oscillation)	146
Figure 5.30: Computed Vorticity Contours at the Beginning of the Cycle ($t = T$) and at Half Cycle ($t = T + \frac{1}{2} T$) for a Flapping Elliptical Cross-Section Wing for Rotation about (a) –(b) X-axis, (c)-(d) Z- axis (e)-(f) X- and Z-axes, (T being the Period of One Oscillation)	147
Figure 5.31: Computed Variation of Normalized Positive and Negative Vorticity with Time for Rotation about Different Axes	149
Figure 5.32: Computed Variation of Q_v over one Cycle for Rotation about Different Axes for the NACA 0012 Cross-Section Wing and the Elliptical Cross-Section Wing	150
Figure 6.1: Discretization of (a) Euler Bernoulli Beam Equation using a 5 Point Stencil and (b) Kirchhoff Plate Equation using a 25 Point Stencil	154

Figure 6.2: Effect of Time Step on Tip Displacement for Forced Vibration Using the Euler-Bernoulli Beam Equation.....	158
Figure 6.3: Comparison of Interpolated Stresses on the Structural Dynamics Grid with the Actual Stresses on the Flow Solver Grid	160
Figure 6.4: Fixing up the Load on Unused Points of Grid C from Grid A and B through Interpolation.....	160
Figure 6.5: Interpolation on the Structural Grid from the Upper and Lower Surfaces of the Flow Solver Grid.....	162
Figure 6.6: Computed Drag Coefficient for (a) Rigid Plunging (b) Plunging with Deformation	165
Figure 6.7: Computed Contours of Pressure Coefficient from (a) Miao and Ho [143] and the (b) Present Computation for a Plunging and Deforming NACA 0014 Airfoil at its Mean Position.....	166
Figure 6.8: Time History of the Tip Displacement with Implicit Coupling (a) In Comparison with the Results of Shin et al. [80] (b) In Comparison with Explicit Coupling	168
Figure 7.1: Trajectory of the Tear-Drop Plate for Amplitudes (a) $A=0, 0.1, 0.2$ (b) $A=0.3, 0.4, 0.5$	170
Figure 7.2: (a)-(f) Computed Vorticity Contours for Different Deformation Amplitudes for a Plunging and Deforming Tear-Drop Plate at the Mean Position (Position of Maximum Deformation).....	171
Figure 7.3: Computed Time History of the Horizontal Force Coefficient C_x for Various Deformation Amplitudes	173
Figure 7.4: Trajectory of a Plunging and Deforming Tear-Drop Plate with and without Shear Stress for (a) $A = 0.3$, (b) $A = 0.5$	174
Figure 7.5: Computed Z-Vorticity Contours for a Deforming Wing at Different Instances of Times $t = T \frac{1}{4}$ to $t = T \frac{3}{4}$ (T being the Period of one Deformation Cycle = 2 s) For the Simple Flapping Mode	176
Figure 7.6: Characteristics of the Simple Flapping Mode Showing (a) Horizontal Force Coefficients Over One Cycle and (b) Trajectory of the Wing Tip	177
Figure 7.7: Wing Kinematics for Combined Flapping Modes at Different Time Instants (T being the Period of Oscillation = 2 s).....	179
Figure 7.8: Computed Z-Vorticity Contours for a Deforming Wing (Combined Flapping Mode) at Different Instances of Times $t = T \frac{1}{4}$ to $t = T \frac{3}{4}$ (T being the Period of One Oscillation = 2 s).....	180

Figure 7.9: Combined Flapping Characteristics Showing (a) Horizontal Force Coefficient Over one Cycle and (b) Trajectory of the Wing Tip.....	181
Figure 7.10: Time History of the Tip Displacement in Comparison with the results of Tang et al. [57].....	183
Figure 7.11: Time History of the Tip Displacement for Different Relaxation Coefficients (ω) and Time Steps	184
Figure 7.12: Trajectory of the Airfoil-Plate Combination for Various Airfoil Flexibilities	187
Figure 7.13: (a)-(c) Computed Vorticity Contours for Different Plate Flexibilities for a Plunging and Deforming Airfoil-Plate Combination at the Mean Position	187
Figure 7.14: Comparison of the Stresses (Pressure + Shear) for the CFD and the CSD Grids on the (a) Upper and (b) Lower Surfaces of the Wing.....	189
Figure 7.15: Computed (a) Thrust Coefficient in Comparison with the Experimental Results of Heathcote et al. [36] (b) Wing Displacement at the Root Showing Selected points A, B and C of interest	190
Figure 7.16: Computed (a) Spanwise Deflection and (b)-(d) Vorticity Contours at Selected Spanwise Stations (Root, Wing tip and Mid-span) for Plunging and Deforming NACA0012 Cross-section Wing at Selected Points of the Plunging Cycle as Indicated in Figure 7.15(b).	192
Figure 7.17: Streamlines on a Plane (a)-(b) ($y_{pl} = y_{LE} \pm 0.25c$) Parallel to the Wing Surface and (c)-(e) Perpendicular to the Span at point B of the Plunge Cycle, y_{LE} being the y-coordinate of the Leading Edge of the Wing Root	193
Figure 7.18: Streamlines on a Plane (a)-(b) ($y_{pl} = y_{LE} \pm 0.25c$) Parallel to the Wing Surface and (c)-(e) Perpendicular to the Span at Point C of the Plunge Cycle, y_{LE} being the y-coordinate of the Leading Edge of the Wing Root.	194

LIST OF TABLES

Table 3.1: Description of Boundary Condition Types for Different Regions in the Computational Domain 67

Table 4.1: Various Meshes Considered for Convergence Studies..... 78

Table 4.2: Distribution of Points for Different Grids with Reference to Figure 2.19..... 101

Table 5.1: Reduced Frequency (k) and Phase ϕ_1 for Pitching at $Re = 100$ 125

Table 7.1: Stability of the Fluid-Structure Interaction Problem with a Time Step of $\Delta t=10^{-3}$ for Various Relaxation and Numerical Damping Coefficients 185

Table 7.2: Stability of the Fluid-Structure Interaction Problem with a Time Step of $\Delta t=5 \times 10^{-4}$ for Various Relaxation and Numerical Damping Coefficients..... 185

Table 7.3: Stability of the Fluid-Structure Interaction Problem with a Time Step of $\Delta t=10^{-4}$ for Various Relaxation and Numerical Damping Coefficients..... 185

Table A2.1: Solver Statistics for a Two-Dimensional Computation, over 7018 time steps for a Plunging Airfoil.....214

Table A2.2: Solver Statistics for a Three-Dimensional Computation, over 2 time steps for a Flapping Wing.....215

Nomenclature

c	Chord length or mean aerodynamic chord
C_{dd}	Divergence damping coefficient
C_L	Lift coefficient
C_D	Drag coefficient
C_T	Thrust coefficient
C_p	Pressure coefficient
C_{pw}	Power input coefficient
E	Modulus of elasticity
f	Frequency of oscillation
g	Acceleration due to gravity
h_t	Thickness of airfoil / plate
h_0	Non-dimensional plunge amplitude
I^*	Non-dimensional moment of inertia
J	Jacobian of the coordinate transformation
L	Reference length scale
m_x	Mass per unit length / per unit area for the Euler-Bernoulli beam/Kirchhoff Plate
M	Mass of the rigid body
p	Pressure
R	Airfoil trailing edge roundness parameter
S	Wing span
T	Torque / moment of a force
U_s	Reference velocity
u, v, w	Components of the velocity vector
x, y, z	Components of physical coordinates
ξ, η, ζ	Components of computational coordinates
ρ_f	Density of the ambient fluid
ρ_b	Density of the structure
κ	Artificial dissipation parameter
κ_E	Explicit numerical damping coefficient

ν	Kinematic viscosity of the fluid
ν_p	Poissons ratio of the structure
ω_R	Angular frequency of oscillation
ω	Numerical relaxation for fluid-structure interaction

Vector Quantities

e	Principle axes
F	Force
G	Collection of composite grids
h_m	Angular momentum
I	Inertia tensor / moment of inertia
x	Physical coordinates
r	Computational coordinates
u	Velocity vector
ϖ	Vorticity

Non-Dimensional Quantities

Fr	Froude number = $\frac{U_s}{\sqrt{gL}}$
k	Reduced frequency = $\frac{\omega_R L}{U_s}$
Re	Reynolds number = $\frac{U_s L}{\nu}$
St	Strouhal number = kh_0
ζ_R	Density ratio = $\frac{\rho_b}{\rho_f}$

Chapter 1

Introduction

The flight of birds and insects has evoked a great deal of interest to the common man. With a highly evolved and complex biomechanical system, birds and insects have intrigued humans for thousands of years with their extraordinary flying capability and agility. According to Shyy et al. [1], there are about one million flying species on earth, out of which, 13000 of them are birds and mammals. Out of this, about 10000 species which include 9000 species of birds and 1000 species of bats fly. Schmidt-Nielsen [2] presents some qualitative estimates that demonstrate this fact. The energy costs of locomotion were compared for a given body weight. For the range of weights encountered in flying species which are of the order 10^{-3} - 10^3 grams, it was found that flying incurred less energy consumption compared to running. These stunning flying creatures also display some amazing feats in air. They can change directions rapidly and travel longer distances on their own effort as opposed to man. Figures 1.1(a)-(c) show some of these extraordinary flying creatures, such as the hummingbird, the honey bee and the barn swallow respectively. Each has a specific characteristic that distinguishes one from the other. The wing structure and morphology change accordingly. According to Rauzon [3], the hummingbird shown in Figure 1.1(a) for example, flaps in a near vertical plane at about 200 beats per second to support its weight and can even travel backwards. The honey bee shown in Figure 1.1(b) is relatively tiny, and flaps about 250 beats per second according to Micucci [4], and produces a buzzing sound. The barn swallow in Figure 1.1(c) distinguishes itself from other birds by having a very high rolling rate of about 5000 degrees per second, which is seven times more than a highly acrobatic aircraft such as the A4 Skyhawk [5]. Figure 1.2 shows a summary of the distances travelled per second relative to body size for different mammals and man made machines based on partial data adapted from Shyy et al. [1]. It can be seen that the Swift is unquestionably one of the fastest flying birds. Although it can be seen that humans have not evolved to master their locomotive characteristics, their evolution has been mainly focussed on mastering intelligence and development of the brain, which birds, insects and other

animals lack. Man, having acquired and mastered this intelligence, had entered the race to mimic the monarchs of the sky – birds and insects some 2000 years ago. From then, till the present day, the quest for achieving this feat of flying has partially paid off. Man has now been able to develop bird/insect like miniature robots that can take to the skies.

Figure 1.1: Different Flying Species found in Nature, (a) Hummingbird (b) Honey Bee and (c) Barn Swallow

Gaining a thorough understanding of the behavior of these flying creatures requires an integration of different disciplines. Issues dealing with wing morphology can be understood very well from the standpoint of anatomy and physiology. With regard to the size, shape of the wing, and its interaction with the surrounding fluid, aerodynamics and structural dynamics interaction, collectively known as aeroelasticity, holds the key in understanding how certain wings create lift and thrust with seemingly effortless kinematics. To complement biology and aeroelasticity, the response characteristics which represent the ability of a bird to control itself on a desired course also come into play. Biologists in association with fluid dynamicists have been trying to understand the mechanics of animal propulsion by means of experiments on model birds in wind tunnels and in water tunnels. Complementary to these flying creatures, there are aquatic animals such as fishes and micro-organisms that utilize some form of flapping or undulatory motion for propulsion. Since their characteristic length scales and forward speeds are very small, Lighthill [6] states that the Reynolds number, which is a non-dimensional parameter that quantifies the relative strengths of inertial and viscous forces, turns out to be low, in the range $10^{-3} - 10^4$ as opposed to other practical applications of interest such as the flow over an aircraft, where the Reynolds number is of the order 10^8 . At these low

Reynolds numbers, it was recognized that, flapping is a lot more advantageous than fixed wing propulsion due to the fact that forward flight cannot be sustained easily at low speeds using a fixed wing. The recent surge of investigations in this area of research can be attributed to development of Micro/Nano-Aerial Vehicles (MAV/NAV) and Unmanned Underwater vehicles (UUV) for military and urban applications as outlined in Abate et al. [7]. Figure 1.3(a) shows a typical flapping wing toy which has a wingspan of about 25cm and which uses both flapping and jet like propulsion to sustain forward flight. However the above model is still crude, lacks proper aerodynamic design, control and is far from realization as a well designed MAV. The research community on flapping wing propulsion has thus focussed on integrating the major disciplines such as aerodynamics, structural dynamics and control. Figure 1.3(b) shows a well designed robotic fly, also known as *microfly* from Harvard Microrobotics lab. Recent research on the robotic fly according to Ref [8] has suggested that a small amount of lift could be generated by exciting the body of a robotic insect using mechanical vibrations even without wings. Most of these MAVs are designed based on direct observations from bird and insect flight mostly by capturing their kinematics and replicating them on models, however the direct application of aerodynamics to the design of an MAV is still in its infant stage unlike its application to the design of aircrafts. A thorough understanding of the aerodynamics is thus required. A detailed overview of prior and current research activities pertaining to the aerodynamics of flapping wing fliers is presented below to motivate the current study in perspective.

1.1. Early Observations and Current Progress

Knoller [9] and Betz [10] first observed through experiments that a plunging airfoil creates an effective angle of attack which has both a forward component (thrust) as well as a normal component (lift). This effect was further verified by Katzmayr [11] in 1922 and came to be known as the Katzmayr effect. He noted that airfoils were able to utilize the upstream oscillatory flow condition to improve propulsive efficiency and also reduce drag. Ober [12] further verified that the Katzmayr effect can be helpful in reducing drag and demonstrated this effect using a simple mathematical model. One of the earliest

Figure 1.2: Speeds of various Flying Species, Mammals and Aircrafts in terms of Body Lengths

(a)

(b)

Figure 1.3: Man made Flapping Wing Robots (a) The I-Bird Toy and (b) Microfly

reported works in tackling unsteady low speed flows was undertaken by Prandtl as mentioned in Cebeci et al. [13] in which, the small perturbation approach for an incompressible flow past an airfoil executing small amplitude sinusoidal oscillations was explained. Further Birnbaum [14] explained in his doctoral thesis, the role of reduced frequency, a quantity which is the ratio between convective and rotational time scales for unsteady flows. Later, Wagner [15], Küssner [16] and Theodorsen [17] enhanced the understanding of this subject by following up Prandtl's small perturbation theory. Garrick [18] devised analytical expressions for the propelling force experienced by an airfoil oscillating in three degrees of freedom in a uniform air flow for a wide range of frequencies based on Wagner's functions for calculating the distribution of vorticity in the wake, and Theodorsen's inviscid incompressible oscillating flat plate theory. The advent of panel methods in 1960's allowed researchers to model inviscid steady incompressible flows which was further extended to handle unsteady flows. Cebeci [13] presents the state-of-the art unsteady panel methods and boundary layer methods to handle unsteady low Reynolds number problems. The recent past has seen considerable effort being spent in understanding the unsteady aerodynamic characteristics of plunging and pitching airfoils through experiments, numerical computations using high fidelity flow solvers, analytical methods and a combination of the above. The list is exhaustive, however the relevant articles are discussed briefly below.

Koochesfahani [19] performed water tunnel experiments on a NACA 0012 airfoil pitching at small amplitudes and stated that the structure of the wake can be controlled by the frequency, amplitude and the shape of the oscillating waveform. The presence of axial flow along the vortex cores was observed and was attributed to the no-slip wall boundary condition on the channel side walls. Complex vortex interactions were observed for non sinusoidal oscillations. Triantafyllou et al. [20] used experimental data from flapping foils and fish to describe the characteristics of the wake at the onset of thrust, and showed that the optimal efficiency of oscillating foils is obtained for Strouhal numbers in the range 0.25-0.35. Dickinson and Götz [21] demonstrated how the unsteady processes of vortex generation leads to the production of aerodynamic forces. Sunada et al. [22] had experimented different combinations of pitching and plunging and obtained

optimum combinations for maximum thrust and efficiency. Hong et al. [23] had examined the contribution of the spanwise flow to lift. They showed that a spanwise cambered wing generated a larger lift force. Isaac et al. [24] provided force measurements on semi-elliptic flapping wings and computed the inertial contribution of the wing to the aerodynamic forces. Parker et al. [25] conducted water tunnel experiments on a NACA 0030 cross-section wing and pointed out that three-dimensional effects due to wing tip vortices are negligible compared to the effects of leading- and trailing-edge vortices. Lai and Platzer [26] investigated the effect of sharp trailing edge based on the observation of Bennet et al. [27], that leading edge-trailing edge asymmetry is beneficial to thrust, however no generalized conclusion was made. Physical aspects relating to the formation and convection of the leading edge vortex (LEV) and its impact on lift generation can be found in Birch et al. [28] and in Birch and Dickinson [29]. The measured experimental data confirmed the hypothesis that the downwash induced by the tip vortices limits the growth of the LEV. Sane and Dickinson [30] devised a new mathematical model based on the quasi-steady approximation using the data obtained from a dynamically scaled model wing to account for the effect of rotational motion, and added inertia apart from translational motion. Ellington [31] provided a general approach to design efficient flapping wings based on observations from nature and available experimental data.

All experimental measurements/observations stated in the preceeding paragraph are based on active flight, which means that, the kinematics of the wing are prescribed and the wing is constrained to follow along a particular path not governed by the surrounding aerodynamic forces. If however, the wing is not constrained at least along one degree of freedom, then the motion of the wing along that direction is passive and is termed as free flight. The problem of free flight is relatively new in literature and has been addressed only by very few researchers experimentally. Fry et al. [32] through experiments on free flying fruit flies such as the *Drosophila*, obtained their wing kinematics using infrared three-dimensional high speed video and simulated them on a dynamically scaled robotic fly. The aerodynamic forces were measured, and it was found that these flies undertake rapid turns governed primarily by inertial forces rather than viscous forces.

Vandenbergh et al. [33] showed that a rigid symmetric wing undergoing plunge oscillations above a critical frequency in a static fluid, breaks symmetry and begins to rotate spontaneously. They observed a bifurcation point beyond which the angular velocity of the wing increases linearly with plunge frequency. Beal et al. [34] performed experiments on a dead trout placed downstream of a cylinder and demonstrated that under certain conditions, the trout can extract energy from the wake of the cylinder and propel itself upstream.

Experimental measurements on flexible flapping wings are sparse in literature and hence provide us very little understanding of the subject. Heathcote et al. [35] conducted water tunnel experiments on an airfoil/plate combination made of aluminum/steel respectively and analyzed extensively the role of chordwise flexibility of a flapping airfoil. It was shown that the flexibility of the airfoil has a direct impact on the vortex spacing and hence, on the generation of thrust. A moderate flexible airfoil was found to be more efficient in terms thrust generation. To analyze the effect of spanwise flexibility, Heathcote et al. [36] conducted water tunnel experiments on a NACA 0012 cross-section wing of varying degrees of stiffness and showed that spanwise flexibility benefits thrust and propulsive efficiency and that excessive flexibility can be detrimental to thrust production.

Computations of flapping/pitching/plunging rigid airfoils using full Navier-Stokes equations and potential flow codes are abundant in literature. A majority of them are two-dimensional and focus on non-accelerating free-stream flows. Tuncer and Platzer [37] verify the process of thrust generation on single/tandem airfoils using a potential flow code and a compressible Navier-Stokes solver and simulate the reverse Karman vortex street. The Knoller-Betz effect has been demonstrated computationally and experimentally by Jones et al. [38] identifying the bounds on Strouhal number for drag-thrust transition at a given Reynolds number. Young [39] has done extensive work relating to numerical simulation of flapping foils. The wake patterns corresponding to different Strouhal numbers, reduced frequency have been obtained in detail and the phenomenon of vortex lock-in explained. A very good agreement between laminar and turbulent flow computations were obtained. It was suggested that for large amplitude

plunging, both the laminar and turbulent wakes were very similar as the flow was governed by the large scale structures at the trailing edge. Artificial compressibility methods have been used by Liu and Kawachi [40] for computing unsteady aerodynamic forces on a moth, and qualitative flow features were compared with smoke flow visualizations. Ramamurti and Sandberg [41] used a Finite-Element based solver, to compute the flow past a pitching NACA 0012 airfoil and identified the Strouhal number as the governing non-dimensional parameter in thrust generation. An Immersed boundary (IB) method was successfully developed and tested by Wang [42] to model the flow around a flapping wing. Bozkurtas et al. [43] described a Cartesian grid method in conjunction with an immersed boundary method to simulate flows past flexible moving bodies. Tuncer and Platzer [44] used a compressible Navier-Stokes solver to predict the aerodynamic forces and compute particle traces for a pitching and plunging NACA 0012 airfoil. It was concluded that high propulsive efficiencies occur when there is attached flow for the entire cycle of oscillation. Sengupta et al. [45] had used a computational method using a vorticity stream function approach to compute flows past pitching and plunging airfoils, obtaining good comparisons with existing data from literature.

Recently, Shyy and Liu [46] used numerical computations to point out the importance of the LEV and showed that the delay of the LEV formation for low aspect ratio wings is beneficial to lift. A comprehensive review on numerical computations, and other aspects of flapping wings can be found in Shyy et al. [47]. A treatise of the vortex dynamics in the wake of a flapping wing in three-dimensions can be found in Dong et al. [48], Aono et al. [49].

The problem of free-flight has been addressed by very few researchers as mentioned earlier. Alben and Shelly [50] verified the observation of Vandenberghe et al. [33] that the transition from the state of rest to free-flight takes place at a critical plunge frequency. A vorticity stream-function approach in two-dimensions was adopted to study the dynamic performance of a plunging elliptical airfoil. Bergou et al. [51] used two-dimensional Direct Numerical Simulations (DNS) to demonstrate the conditions under which, the change in the angle of attack near stroke reversal takes place passively.

Time accurate flow computations over deforming bodies is gaining popularity due to the availability of cheaper computing resources. Zhu and Peskin [52] used an immersed boundary method to understand the passive deformation of a flexible flapping filament in a soap film. A similar problem was attempted by Farnell et al. [53] using the commercial solver FIDAP[®] [54]. Pederzani and Haj-Hariri [55] using a mixed Eulerian-Lagrangian approach with an immersed boundary method, modelled the interaction between an elastic airfoil and surrounding flow. A membrane model was used to predict structural displacements. Using an Arbitrary Lagrangian Eulerian (ALE) formulation with a discontinuous Galerkin method for the Navier-Stokes equations and a hyperelastic Neo-Hookean membrane formulation, Persson and Peraire [56] computed unsteady flow fields past a thin flapping membrane. Tang et al. in [57] and [58] demonstrated a variety of flexible airfoil/wing computations and concluded that a deforming airfoil was equivalent to a pitching airfoil which modified the local angle of attack, and showed how the aerodynamic characteristics were altered accordingly. It was also shown that pressure played an active part in deforming the airfoil rather than the shear stress. Different aspects relating to the aerodynamics of deforming wings have been addressed by Toomey and Elredge [59]. A viscous vortex particle method was formulated to analyse the role of flexibility in flapping flight. Results were compared with experiments and a substantial improvement in wing performance was achieved. Stanford et al. [60] performed static deformation analysis due to aerodynamic loads of a membrane wing. Singh and Chopra [61] developed robotic wings to analyse the effect of pitch and flexibility on thrust in hovering modes. Lian [62] has provided comprehensive information relating to the aerodynamics of rigid and flexible membrane wings at low Reynolds numbers. A shape optimization tool was also developed to improve the performance of membrane wings.

1.2. Motivation

Owing to a large number of existing computations and experiments on flapping foils and wings, it is not the intention of the present research to recompute all test cases, except for some cases for the purposes of code validation, but to identify and analyze critical areas that received less attention over the years. The bulk of this research is centered around

four areas namely, (i) passive flight (ii) dynamic stall and lift hysteresis (iii) vortex dynamics and (iv) wing deformation and fluid-structure interaction.

1.2.1. Passive Flight

The standard approach that has been adopted by the scientific community is to study the aerodynamic characteristics of flapping wings with a fixed freestream velocity. In this approach, a uniform stream of fluid flows past a plunging/pitching wing and the effect of kinematics, reduced frequency, k , Strouhal number, St , and Reynolds number, Re on the aerodynamic characteristics are analyzed. Based on the observed or computed thrust/lift/power coefficients, a suitable combination of non-dimensional parameters are identified as a best possible set that maximizes either thrust or propulsive efficiency. An alternative approach which is more realistic, is to set the body free to move under the action of aerodynamic forces with no incoming free-stream. In this case, the body accelerates from the state of rest to the state of motion and traces a particular trajectory. The Newton's law for a rigid body are solved simultaneously with the Navier-Stokes equations. Free or passive flight computations are sparse in literature and hence this aspect is dealt in detail in the present research.

1.2.2. Dynamic Stall and Lift Hysteresis

Unsteady airfoil stall or dynamic stall, has been studied extensively over the years in the context of rotorcraft, flapping wings, turbomachinery and wind turbines. An exhaustive review of this phenomenon can be found in Ericsson and Reding [63]. As stated by them, the delay in static stall due to a time lag or slow response of the flow and the transient nature of the separation point are the key attributes of unsteady airfoil stall. The methods described by the above authors on the prediction of dynamic stall involved semi-empirical methods based on available static and dynamic tests. Compressibility effects associated dynamic stall have been discussed by Carr & Chandrasekhara [64] with reference to helicopter blades at low Mach numbers ($M \sim 0.2$) in retreating motion, and also the possibility of localized supersonic flow due to rapid pitching of airfoils at low Mach numbers. Piziali [65] reported experimental results for airfoils and wings undergoing pitching motions representative of helicopter blades for a broad range of angle of attack, Mach number and frequency at $Re = 10^6$. Lift and moment hysteresis

loops were also computed corresponding to these flow conditions. One major observation that can be made from the data pertaining to airfoils and wings undergoing rapid periodic motion is the size, shape and direction of the lift and moment hysteresis loops. The hysteresis loops can vary in size, shape and direction depending on the Reynolds number and reduced frequency. In most of the cases, the usual clockwise hysteresis loops have been observed for airfoils pitched near the static stall angle. By clockwise, it means that the lift coefficient is higher during a pitch up movement compared to the quasi steady value and the pitch down movement at the same angle of attack. Counterclockwise hysteresis loops have also been observed in literature. Figure 1.4 shows the different types of lift hysteresis curves.

Figure 1.4: Various Lift Hysteresis Curves for Sinusoidal and Triangular type Pitching

One of the earliest works related to the observation of counterclockwise lift hysteresis loops for airfoils pitched at angles of attack near stall was by Moore [66]. The observation that the lift is relatively higher on the downstroke compared to the quasi-steady value has been attributed to the quasi steady movement of the separation point on the upper surface. Different forcing functions for the angle of attack produced different shapes for the hysteresis loop. For example, it is shown that if the variation in angle of attack was a saw-tooth function (triangular waveform), the hysteresis loop was nearly rectangular as observed in Figure 1.4. However Takallu and James [67] pointed out, that the analysis of Moore [66] did not contain the effect of boundary layer separation and pressure distribution in the separated layer, and further went on to show that at small angles of attack, the hysteresis loop was anticlockwise and that beyond the static stall angle, the hysteresis loop was clockwise. They also observed a reverse hysteresis loop

where lift decreases with increase in angle of attack vice-versa when pitched about the static stall angle. Johansen [68] also computed various lift hysteresis loops using Beddoes-Leishman model [69] coupled with aeroelasticity and showed how the lift hysteresis changed direction to a reverse hysteresis loop, when an airfoil was pitched beyond the static stall angle. The transition between clockwise and anticlockwise moment hysteresis loops has been reported in Ekaterineris and Platzer [70] in which, a detailed review of the various numerical methods including panel methods coupled with boundary layer and full Navier Stokes computation of airfoil dynamic stall is presented. Detailed discussions on the effect of free stream Mach number, reduced frequency, turbulence models on lift and pitching moment hysteresis loops have also been presented. Computations of dynamic stall in three-dimensions are also sparse in literature and can be found in Spentzos et al. [71] and [72], Le Papu and Le Canu [73].

In the present work, lift hysteresis curves are computed for airfoils and wings pitching about a zero angle of attack for a wide range of reduced frequencies. Unlike other instances reported in literature where the lift hysteresis curves change direction only when the mean pitching angle is near the static stall angle, results from the present work indicate that, the change in direction can also occur when the airfoil or wing is pitched about a zero mean angle of attack, but at higher reduced frequencies. This phenomenon might not have gained attention previously due to the fact that most of the practical applications of interest involve airfoils and wings pitching about some mean angle of attack such as the static stall angle at high Reynolds numbers. Studies on pitching about a zero mean angle of attack in low Reynolds numbers might be useful to complement research on aquatic and insect flight. To support the observation, a mathematical model is formulated for an airfoil undergoing harmonic oscillations about a zero mean angle of attack. It is proved that a gradual transition from the usual hysteresis loop to a reverse hysteresis loop is attained at a particular reduced frequency which is a function of the phase lag associated with the response of the flow with the forcing function. It is also shown that reverse hysteresis is present when there is no asymmetry between the leading and trailing edges of the airfoil.

1.2.3. Vorticity Dynamics

Thrust generation by flapping airfoils at low Reynolds numbers is usually associated with the formation of a reverse Karman vortex street as outlined in Jones et al. [38]. For example, when there is an incoming free-stream, Anderson et al. [74] showed that for high efficiency, the leading edge vortex is convected downstream and interacts with the trailing edge vortex to produce a reverse Kaman vortex street. However, when there is no incoming free-stream, the vortices generated from the leading edge of the wing are not convected downstream and hence the reverse Karman vortex street might not be visible in certain cases although thrust is being generated. Based on these observations, there is a need to resort to non-conventional ways to analyze the origin of these forces. The approach that is used in the present study is based on the analysis of Wu [75] who derived expressions for the aerodynamic forces acting on a solid body as a function of vorticity moments and inertial forces. For the flapping wing problem, regions of positive and negative vorticity are identified and their corresponding moments, computed. Based on vorticity moments, it will be shown that the interplay between positive and negative vorticity determines the level of thrust generated.

1.2.4. Wing Deformation and Fluid-Structure Interaction

As reviewed earlier, active and passive wing deformation offers greater prospects for thrust generation when compared to a rigid wing. A good deal of literature exists for two-dimensional flows, from which qualitative conclusions could be made, however, high resolution three-dimensional computations are still limited by today's computational resources. Within the framework of available computational algorithms and hardware, an attempt is made in this study to simulate various flexible modes of a flapping wing and to obtain quantitative estimates for aerodynamic forces and trajectories. When the wing deformation is passive or obtained from the forces, the structural dynamics equations must be solved simultaneously for wing deformation with the Navier-Stokes equations. This constitutes a fluid-structure interaction problem. The fluid-structure coupling algorithm must be robust and independent of the geometry in order to ensure stable and meaningful flow calculations. Presently, a certain class of coupling algorithm known as the partitioned algorithm for fluid-structure interaction is formulated and a variety of

computations are performed. Although partitioned methods are quite popular, they have posed numerous difficulties limiting its analysis only to a narrow range of problems as described in Causin et al. [76].

Partitioned algorithms are well known for their extensive code modularity, yet these class of methods exhibit numerical instabilities for a certain choice of geometric and/or fluid properties. For example, numerical instabilities in the coupling have been observed when the density of the body is less or marginally higher compared to the density of the ambient fluid. Causin et al. [76] reported possible solutions to handle these class of problems by introducing proper under-relaxation to the structural equations. Some of the other relevant quantities that could cause instabilities are the time step used for advancing the solution in time and numerical damping which arises out of the time discretization of the governing equations. All the above mentioned aspects have not received detailed attention despite the availability of computational results on flapping airfoils when the ratio of the density of the solid to the ambient fluid is not very high (~ 7.85) as in Tang et al. [57] and [58]. Zhu [77] used the boundary element method for fluid dynamics and a non-linear thin plate model for the structural dynamic equations to analyze the effect of fluid density on the chord-wise and span-wise deformation of a plunging wing. An implicit scheme was used for the structural discretization with inner fluid-structure coupling iterations. However, the time step employed for the structural iteration was smaller than the time step used in the fluid dynamics computation by an order $O(10^2)$ to guarantee stability despite of using an implicit discretization. Chimakurthi et al. [78] compared explicit and implicit coupling approaches for a plunging and deforming wing and demonstrated excellent agreement with the results of Heathcote et al. [36]. Although the effect of time step was discussed, it was limited to a narrow range of $1 \times 10^{-3} - 5 \times 10^{-3}$ s. Gluck et al. [79] and Shin et al. [80] used extremely high density ratios (1000-6000) with an explicit time integration scheme for the structural equations and did not encounter numerical instability. Another example of using an explicit coupling method can be found in Balint et al. [81]. Recently, Wood et al. [82] using a compressible flow solver, described a partitioned approach with implicit coupling for the analysis of fluid-structure interaction problems on unstructured grids. In the present work, the test case of

Shin et al. [80] is considered for the purposes of validation. The structure under consideration is a beam having a very high density ratio of 6667, for which a comparison of the explicit and implicit coupling approaches can be made. Another test case in which a low density ratio of 7.85 is chosen for the present study since this corresponds to a steel plate flapping in water as in Heathcote et al. [35] and Tang et al. [57]. Using this density ratio, the effect of time step on the structural dynamics equation is analyzed. It will be shown that the numerical discretization adds some damping to the computed solutions. The implicit damping however, is not like material damping where the damping force is proportional to the velocity, but, proportional to the rate of change of velocity. In spite of this artificial damping being present in the system, it is proved that additional damping, higher time step, and low relaxation factors are needed to achieve stability of the coupled problem.

All computations of flapping wings in this study are performed by solving the unsteady incompressible Navier Stokes equations on moving and overlapping grids using the open source solver *cgins/OverBlown* [83]. The solver is a subset of *Overture* [84], which is a large framework for solving partial differential equations on overlapping grids. A grid generation tool *Ogen* [85], also a part of *Overture* [84] is used to generate overlapping grids. *Overture* [84] is built on *A++* objects [86], which enable faster computation by vector processing of arrays. Overlapping grids are preferred for the choice of problems investigated in the present research as moving bodies are easily treated without having the need to displace a large number of grid points. Only the grids surrounding the bodies that are in motion are transformed or regenerated and the boundary between the grids is handled by appropriate interpolation which is performed at each time step.

1.3. Outline

This thesis is organized as follows:

Chapter 2 describes the computational methodology for overlapping grid generation using *Ogen* [85], the overlapping grid generator. Grid generation using a hyperbolic algorithm is also discussed in the context of deforming bodies. Grid generation for

deforming bodies requires the grid points to move in a particular fashion to satisfy the Geometric Conservation Law (GCL) [87], the violation of which, will lead to erroneous sources of mass in the solution. In the context of finite difference discretization, an analysis is made to determine the effect of GCL on computed solutions and the conditions in which the GCL can be avoided, yet maintaining a reasonable solution. Various aspects specific to overlapping grids, such as interpolation between grids, the quality of interpolation and the various grids used in different computations are explained.

Chapter 3 discusses the flow solution procedure of the Incompressible Navier-Stokes equations coupled with rigid body dynamics using the flow solver *cgins* [83]. Discretization of convection, viscous and pressure terms are also discussed.

Chapter 4 presents in-depth validation studies for unsteady flow problems and unsteady flow problems coupled with rigid body dynamics. Accuracy and convergence issues are also addressed in detail.

Chapter 5 introduces the concept of active and passive flight for flapping rigid airfoils/wings. For active flight conditions, the airfoil/wing is constrained to move along a particular direction and the aerodynamic characteristics are studied. The effect of sharp trailing edge and lift hysteresis reversal are also investigated. For passive flight conditions, the airfoil/wing is set free to move by virtue of the aerodynamic forces. Trajectories are then computed for various trailing edge topologies in two-dimensions. Various types of rotational motions are tested in three-dimensions for a NACA0012 wing and an elliptical cross-section wing. Wu's analysis [75] is applied to justify the thrust generated for different types of rotations.

Chapter 6 deals with the governing equations of structural dynamics, its discretization, coupling procedure with the Navier-Stokes equations and numerical validation of the fluid-structure coupling problem. Effect of time step on computed oscillation modes and fluid-structure load interpolation are discussed.

Chapter 7 aims at understanding the effect of different deformation methodologies on thrust and trajectories. The effect of neglecting shear stress in the total force computation is analyzed. Moreover, numerical stability issues that are a function of time step, numerical relaxation and damping are discussed in the context of fluid-structure interaction.

Chapter 8 concludes and summarizes the work, and provides an insight into further work to be performed.

Chapter 2

Overlapping Grid Generation

Grid generation is an important aspect of the numerical solution of partial differential equations. The accuracy of the numerical method is limited by the quality of the grid. There are numerous of grid generation techniques available today, each technique being suited for a particular application. The basic approaches to grid generation fall under the category of structured, unstructured and mesh free methods. A detailed review of structured and unstructured methods with explanation to various algorithms and applications can be found in Liseikin [88]. Mesh free methods are relatively a new class of methods which strictly do not require a mesh but operate on a cloud of points. Computations using mesh free methods can be found in the context of finite element method in Liu [89] and finite difference method for hyperbolic conservation laws in Praveen [90]. Another class of methods which form a subset of these three methods is the overlapping/overset/chimera tools where a set of meshes are generated for individual components, like multi-body problems and they overlap in a finite region. Overlapping grids can be generated using structured as in Chesshire [91], unstructured in Nakahashi et al. [92] and mesh free methods using individual chimera clouds in Deshpande [93]. NASA's *Pegasus* code [94] and Petersson's *Chalmesh* code [95] are other overlapping grid generation codes. A recent article by Chan [96] reviews the development of overset grid methodologies at NASA. Although unstructured/mesh free methods offer more flexibility in generating meshes/points around complex structures, structured meshes are more efficient in handling data structures and connectivity information and allows easy implementation using finite difference methods. The mesh generation tool *Ogen* [85] used in this work is based on *Overture* [84], a framework for the analysis of partial differential equations on overlapping grids. The overlapping grid algorithm is based on *CMPGRD* [91] and handles structured grids only. These overlapping structured meshes are quite similar to block structured meshes except for the fact that block structured meshes match boundaries on neighboring component grids.

To generate a set of overlapping grids, a set of structured curvilinear grids are generated initially across each physical boundary of interest. Each of those curvilinear grids in the physical space can be transformed to a unit rectangle in the computational space where the governing equations are discretized. In the overlap region, the solutions are obtained by appropriate interpolation. Consider for example the problem of generating an overlapping grid for a cylinder in two-dimensions and a sphere in three-dimensions. Figure 2.1(a) shows the overlapping grid for the cylinder along with the interpolation points. There are two structured mapped grids as seen in the physical space, one surrounding the body of interest which is colored green is the cylinder grid and the other which is colored blue, contains the the cylinder grid is a background Cartesian grid. The overlap region between the grids is identified by interpolation points colored in green and blue corresponding to the background and the cylinder grid respectively. Similarly, in three-dimensions, as in the case of a sphere, Figure 2.1(b) shows the overlapping grid for a quarter sphere and the interpolation points in the physical and computational space. Each point in the physical space denoted by x , including the interpolation points corresponds to a particular point in the computational space denoted by r which is Cartesian. Numerical discretization of a desired partial differential equation at an arbitrary point which is not an interpolation point is performed using standard discretization procedures. At interpolation points, interpolation is performed in the computational space and the obtained set of equations are solved together with the discretized equations to obtain the final solution.

Each of these component structured grids can be generated using an elliptical, hyperbolic or data-point mapping – a mapping generated based on a user input of grid points or from a different grid generation tool. The foremost advantage of using overlapping grids is that each component grid can be treated independently and hence multi-component systems can be handled efficiently. For moving bodies without deformation, the component grids are displaced/rotated rigidly at every time step thereby regenerating only the interpolation points. If the co-ordinates of the grid before and after moving are denoted by x and x' respectively, then $x' = R' x + T$, where R' is a rotation matrix and T is a displacement vector. This is in contrast to many other techniques that use grid deformation or remeshing at each time step even for rigid body motion.

(a) Cylinder

(b) Sphere

Figure 2.1 : Overlapping Grid in the Physical and Computational Space for a (a) Cylinder (b) Sphere

However, for deforming body problems, remeshing is inevitable since the boundary changes shape and is a function of both space and time. The component grids in this case, are regenerated at each time step using a hyperbolic mapping. Further details can be found in Sec.2.3. Moving boundary problems using the *Overture* [84] framework have been tested successfully by Pan and Damodaran [97] and Guerrero [98]. Higher order computations using overset grids for moving bodies can be found in Alabi and Ladeinde [99]. In the next section, the overlapping grid algorithm is outlined along with a discussion relating to interpolation and its accuracy.

2.1. Composite Grid Construction

The composite grid algorithm as outlined in Chesshire and Henshaw [91] is centred around classifying the grid points into discretization, interpolation and exterior/unused points. A discretization point is one, where the governing equations are discretized to the desired level of accuracy. Interpolation points are those points where the flow variables are interpolated from neighboring grids and/or the same grid. They are classified into explicit and implicit interpolation respectively. When a grid point is neither a discretization nor an interpolation point, and is not used in the computation, it is classified as an exterior/unused point. As an example, consider the problem of generating composite grids for three tandem cylinders. Figure 2.2 briefly outlines the procedure to generate a composite grid. A composite grid $\mathbf{G}$ consists of individual component grids g_k which are logically rectangular. These individual component grids are generated first as illustrated in step-A in Figure 2.2. In the current example, there are four component grids in total, a background Cartesian grid and three cylinder grids as shown in Figure 2.3.

The set of component grids are then arranged in order of increasing priority for a process known as hole cutting as described in step-B in Figure 2.2. Grids that have the higher priority will cut holes on lower priority grids thereby creating a number of unused points. The order of priority for the grids in the current example is as follows: background grid - (0), left cylinder - (1), right cylinder - (2) and bottom cylinder - (3). Figure 2.4 shows the grid after the holes have been cut. All grid points that are inside the cylinder are flagged as unused points, and are not used in the computation. Following the hole cutting

procedure, the interpolation points that govern the connectivity between grids are determined as in step-C of Figure 2.2. By a local search algorithm, starting with the highest priority grid, each grid point is examined whether it can be interpolated from a lower or higher priority grid. A point on a given grid can interpolate from a neighboring grid if it has the correct stencil of points. If the points can be interpolated, they are marked as interpolation points. If the point is not a discretization point and cannot be interpolated, it is classified as an exterior or unused point. At this stage each grid point has a flag that determines whether it is a discretization, interpolation or an unused point. Figure 2.5 shows the grid at this point of time. In step-D as in Figure 2.2, points on lower priority grids that are definitely needed for interpolation are marked. These set of points are highlighted in black for the left most cylinder in Figure 2.6. Finally, these interpolation points are retained and the rest are assigned as unused points. At this stage, the overlapping grid has been computed and is shown in Figure 2.7. According to Chesshire and Henshaw [88], the number of operations to construct a composite grid is of the order $n_g N^2$, where n_g denotes the number of component grids and N , the total number of grid points.

2.2. Interpolation on an Overlapping Grid

Each component grid $g_k \in \mathbf{G}$ has a set of interpolation points, where the flow variables are interpolated from adjacent grid points. Consider a point x belonging to grid 1 in the physical space which is to be interpolated from the neighboring grid, which is grid 0 and the corresponding point r in the computational space as illustrated in Figure 2.8. The interpolation stencil for this point consists of points from grid 0 highlighted in red. The point r is interpolated in the computational space using an interpolation of the form:

$$W'(r) = \sum_{k=1}^9 \alpha_k W'_k \quad (2.1)$$

where $W'(r)$ denotes the variable to be interpolated based on the stencil defined by W'_k .

The interpolation weights α change according to the type of interpolation, which can be lagrange, bilinear or quadratic. When the *R.H.S* of Eq. (2.1) contains interpolation points, the interpolation is implicit and Eq. (2.1) represents a system of

Figure 2.2: A Summary of the Overlapping Grid Generation Procedure

(a) Background Cartesian Grid

(b) Cylinder Grids

Figure 2.3: Individual Component Structured Grids

Figure 2.4: The Grid After Cutting Holes

Figure 2.5: The Grid After Classifying Discretization, Interpolation and Unused points

Figure 2.6: The Grid after Marking Required Interpolation Points

Figure 2.7: The Final Composite (Overlapping)Mesh

Figure 2.8: Interpolation on an Overlapping Grid

linear equations with interpolation points as unknowns. The set of linear equations are then solved for each interpolation point to obtain the unknown $W'(r)$. Concerning the order and accuracy of interpolation, according to Chesshire and Henshaw [91], for a $2\bar{p}^{th}$ order boundary value problem, the width of the interpolation formula is given by $q = 2\bar{p} + 1$ where $2\bar{p}$ is the order of accuracy of discretization. For a second order accurate discretization, $q = 3$ hence quadratic type of interpolation is performed similar to the stencil defined in Figure 2.8. This type of interpolation results in accurate solutions if the solutions are smooth although the discretization is non-conservative. For non smooth solutions, Chesshire and Henshaw [100] have demonstrated a conservative interpolation scheme and applied it to a two-dimensional viscous Burgers' equation with a shock.

2.3. Grid Generation for Deforming Bodies

As mentioned earlier, a hyperbolic mapping is used every time step to regenerate the grid of a deforming body. The description of the method can be found in Henshaw [101]. Applications using the hyperbolic mapping in *Overture* [84] can be found in Fast and Henshaw [102], Fast and Shelly [103]. In order to generate a hyperbolic mapping

surrounding an arbitrary closed boundary, the initial boundary profile which is stored as an array of coordinates (x, y) , the marching distance normal to and away from the boundary, and the number of co-ordinate lines along the co-ordinate axis must be known. A proper combination of marching distance and marching spacing will yield a good quality mesh. For large deformation problems where the boundary profile may become concave, the hyperbolic grid generation algorithm will fail if the marching distance is too large as the grid lines may cross. To overcome these possibilities for specific deformation problems, the meshes are first computed apriori to check for any errors over a complete deformation cycle so as to ensure that the algorithm will not fail while computing the actual flow past the deforming body. Figure 2.9 shows a hyperbolic grid generated surrounding a deforming ellipse at different time instants. Only the grids surrounding the ellipse are regenerated at each time instant. The interpolation points also change accordingly. Another aspect of deforming body computations is the satisfaction of space conservation law or the geometric conservation law (GCL) as outlined in Thomas and Lombard [87]. When there is appreciable grid deformation, it has been observed by Demirdzic and Peric [104], that the violation of the GCL results in erroneous solutions due to artificial mass sources. The GCL is given by an evolution equation for the Jacobian J of the transformation $(\xi, \eta, \zeta) \rightarrow (x, y, z)$, where (ξ, η, ζ) are the computational co-ordinates and (x, y, z) are the corresponding physical co-ordinates:

$$\frac{\partial J}{\partial t} + \frac{\partial}{\partial \xi} \left(J \frac{\partial \xi}{\partial t} \right) + \frac{\partial}{\partial \eta} \left(J \frac{\partial \eta}{\partial t} \right) + \frac{\partial}{\partial \zeta} \left(J \frac{\partial \zeta}{\partial t} \right) = 0, \quad J = \frac{\partial(x, y, z)}{\partial(\xi, \eta, \zeta)} = \begin{pmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial x}{\partial \eta} & \frac{\partial x}{\partial \zeta} \\ \frac{\partial y}{\partial \xi} & \frac{\partial y}{\partial \eta} & \frac{\partial y}{\partial \zeta} \\ \frac{\partial z}{\partial \xi} & \frac{\partial z}{\partial \eta} & \frac{\partial z}{\partial \zeta} \end{pmatrix} \quad (2.2)$$

From Thomas and Lombard [87], the terms $J \frac{\partial \xi}{\partial t}$, $J \frac{\partial \eta}{\partial t}$ and $J \frac{\partial \zeta}{\partial t}$ can be computed using the following identities:

$$\begin{aligned} J \frac{\partial \xi}{\partial t} &= -(\xi_x \dot{x} + \xi_y \dot{y} + \xi_z \dot{z}), \quad \xi_x = y_\zeta z_\eta - y_\eta z_\zeta, \quad \xi_y = z_\zeta x_\eta - z_\eta x_\zeta, \quad \xi_z = x_\zeta y_\eta - x_\eta y_\zeta \\ J \frac{\partial \eta}{\partial t} &= -(\eta_x \dot{x} + \eta_y \dot{y} + \eta_z \dot{z}), \quad \eta_x = y_\xi z_\zeta - y_\zeta z_\xi, \quad \eta_y = z_\xi x_\zeta - z_\zeta x_\xi, \quad \eta_z = x_\xi y_\zeta - x_\zeta y_\xi \\ J \frac{\partial \zeta}{\partial t} &= -(\zeta_x \dot{x} + \zeta_y \dot{y} + \zeta_z \dot{z}), \quad \zeta_x = y_\eta z_\xi - y_\xi z_\eta, \quad \zeta_y = z_\eta x_\xi - z_\xi x_\eta, \quad \zeta_z = x_\eta y_\xi - x_\xi y_\eta \end{aligned} \quad (2.3)$$

Second order central differences are used to approximate the metrics in Eq. (2.2). The terms $\dot{x}, \dot{y}, \dot{z}$ are components of the grid velocity which are computed using a backward Euler discretization as follows:

$$\dot{x}^{n+1} = \frac{x^{n+1} - x^n}{\Delta t}, \quad \dot{y}^{n+1} = \frac{y^{n+1} - y^n}{\Delta t}, \quad \dot{z}^{n+1} = \frac{z^{n+1} - z^n}{\Delta t} \quad (2.4)$$

The vertex Jacobian in Eq. (2.2) is then updated using the same backward Euler discretization as in Eq. (2.3). The quantity $\nabla_r \cdot \left(J \frac{\partial r}{\partial t} \right)$ in Eq. (2.2) is a geometric and

kinematic quantity and provides us with the information about the dilation of the mesh. If this quantity is very small, the use of GCL can be avoided hence computation time, which amounts to 5-10% of the total time can be reduced. Numerical estimates of

$\nabla_r \cdot \left(J \frac{\partial r}{\partial t} \right)$ are obtained at selected points on the mesh as a function of time for a

deforming airfoil for three different marching distances. Figure 2.10 shows a portion of the trailing edge of an airfoil where maximum deformation occurs. At points A and B, the

mesh dilation $\nabla_r \cdot \left(J \frac{\partial r}{\partial t} \right)$ is computed for all time steps and shown in Figure 2.11. It is

seen that the mesh dilation is of the order $O(10^{-3})$ at point A for all marching distances and $O(1)$ at point B for a marching distance of $0.08c$. When the marching distance is reduced, the variation in the mesh dilation also reduces. Hence GCL has less impact on the computed solutions when the marching distance is very small. The above estimates are computed for a deformation frequency and tip amplitude of 2 Hz and $0.2c$ respectively. However for low frequencies and amplitudes, it is found that a higher marching distance can be used. In the standard deforming grid framework for structured meshes, the grid is not regenerated but grid points are displaced along the coordinate

(a) $t = 0$

(b) $t = t_1$

(c) $t = t_3$

Figure 2.9: Hyperbolic Mappings and the Corresponding Overlapping Grid for a Deforming Ellipse at Various Time Instants

(a) Marching Distance = $0.08c$

(b) Marching Distance = $0.06c$

(c) Marching Distance = $0.03c$

Figure 2.10: Portion of the Mesh Near the Trailing-edge for Different Marching Distances

Figure 2.11: Variation of Mesh Dilation as a Function of Time (a) At Point A and (b) At Point B as Indicated in Figure 2.10

directions using the spring analogy. The grid velocities are computed based on the displaced points and the Jacobian updated using the GCL. However in the present approach, hyperbolic grid generation is used at each time step to regenerate the grid according to the shape of the body. In this case, the grid points are not displaced along the coordinate directions but undergo infinitesimal rigid body rotations conforming to the body surface. Since the GCL is automatically satisfied for rigid body motions, it is speculated that this manner of deforming body motion maintains the Jacobian according to the GCL. Based on the above fact, the use of GCL is avoided when a grid is generated with a smaller marching distance ($< 0.05c$).

2.4. Overlapping Grids Used for Different Computations

Different grids used in various computations are described in this section. As mentioned earlier, all overlapping grids have been generated using *Ogen* [85] the overlapping grid generator. Individual component grids are generated using an elliptical or hyperbolic mapping depending on the type of problem in hand. For two-dimensional problems, the grid for an airfoil/ellipse/rectangular plate is constructed and for three-dimensional problems, a grid for a NACA 0012 cross-section wing /elliptical cross-section wing is constructed. Grids for deforming airfoils and wings are also presented.

2.4.1. Overlapping Grid for an Airfoil

The overlapping grid for an airfoil consists of at least two component grids, a curvilinear grid for the airfoil and a Cartesian grid in which the airfoil is contained. For moving body problems without deformation, the airfoil grid is generated once before the start of the computation using transfinite interpolation and then smoothed using an elliptical smoothing technique [105]. Each of these two-dimensional grids in the computational space consists of four boundaries, each having a particular flag to indicate the type of boundary. A flag value of 1 indicates a physical boundary, 0 an interpolation boundary and -1, a periodic boundary. Figure 2.12 shows the curvilinear grid for a NACA 0012 airfoil with the different boundaries in the physical and computational space and the corresponding flags. This airfoil grid is then overlapped on to a background Cartesian grid. The complete overlapping grid is shown in Figure 2.13. The above grid may be

Figure 2.12 : NACA 0012 Airfoil Showing Various Edges and the Corresponding Flags

Figure 2.13: The Overlapping Grid for the NACA 0012 Airfoil

supplemented by additional grids depending on the type of problem. For example, to capture the vortices in the wake accurately, another grid could be added behind the airfoil as in the test case described in Chapter 4. The grids shown in Figure 2.12 and Figure 2.13 are for the purpose of illustration only. The actual grids with different grid density and domain dimensions vary according to the problem.

2.4.2. Overlapping Grid for a Deforming Airfoil

The elliptic type of grid used in Sec 2.4.1 for a NACA 0012 airfoil is used for rigid body motions. The outer boundary given by edge D in Figure 2.12, in this case has to be prescribed for the grid to be generated. When the body deforms, and if the outer boundary is specified, the grid points must also be deformed and this may result in the loss of grid quality. To circumvent this problem, the grid surrounding the airfoil is constructed using a hyperbolic mapping. This requires only the specification of the airfoil surface, and the extent to which the marching has to take place. When the airfoil deforms, the grid surrounding the airfoil is regenerated at each time instant. Another aspect of using a hyperbolic mapping is that near orthogonal grids can be generated. Figure 2.14 and 2.15 shows the grid surrounding a deformed NACA 0014 airfoil using the hyperbolic mapping. An animation showing the deformation of the grid surrounding a NACA 0014 airfoil is shown in Appendix A3.2.1. This grid is used for a validation test case of a deforming airfoil in Chapter 6.

2.4.3. Overlapping Grid for a Deforming Tear-Drop Plate

A tear-drop plate has been studied experimentally by Heathcote and Gursul [35] and Tang et al. [57]. It consists of a rigid symmetrical airfoil attached to a flexible tail and resembles a tear-drop. Similar to the grid generated for the NACA 0014 airfoil, a hyperbolic mapping is used for the present grid. The extent of the grid generated is limited to a small distance from the boundary since it keeps the mesh dilation small as discussed in Sec. 2.3. Figure 2.16 shows the overlapping grid for a tear-drop plate in the deformed state. This grid is used for both forced and passive deformation computational cases considered in Chapters 6 and 7.

Figure 2.14: Hyperbolic Grid for a Deforming NACA 0014 Airfoil

Figure 2.15: Overlapping Grids for a Deforming NACA 0014 Airfoil

Figure 2.16: Overlapping Grids Surrounding a Tear-Drop Plate Showing the Rigid Fore Part and a Flexible Tail.

2.4.4. Overlapping Grid for a Deforming Cantilever Beam in a Channel

For testing the fluid-structure coupling algorithm in Chapter 6, the test case of an oscillating cantilever beam in incompressible flow is studied. A hyperbolic mapping similar to the ones used for the previous grids is used to construct the grid surrounding the beam. However there is a minor difference in the manner in which grids are generated for the present case when compared to the earlier cases. All cases considered previously had the airfoil grid completely immersed in the background grid. The outer boundary was usually an interpolation boundary. For the present problem, one end of the beam is clamped to the lower end of the channel and hence the wall boundary of the channel coincides with boundary A, B of the cantilever grid – lower boundary as shown in Figure 2.17. If not handled properly, the cantilever grid will try to cut a hole in the background grid along the lower boundary. Since the lower boundary of the background grid is also a physical boundary; a wall, the overlapping grid algorithm will fail. To avoid such a situation, the boundaries that are all coinciding are given a unique flag known as a share flag which instructs the grid generator not to cut holes. Figure 2.18 shows the overlapping grid for this case in the deformed state.

2.4.5. Overlapping Grid for a Wing

The first step in generating an overlapping grid for a wing is to generate a mapping from a set of two-dimensional airfoil profiles. If the wing is rectangular, as with the cases considered here, a minimum of two cross-section profiles are required. The mapping will then represent the wing surface from which, volume grids will be built. Since flow solutions need to be obtained on wing tips, the structured grid methodology will fail, since all grid lines along the spanwise and chordwise direction will converge to a point forming a singularity at the wing tip. The resulting grid along with the singularity is illustrated in Figure 2.19(a). To overcome this problem, the region surrounding the singularity is patched using an orthographic transform as described in Henshaw [105]. Near the singularity, a transformation to spherical co-ordinates (ϕ, θ) of the form

$$(r_1, r_2) \rightarrow (t_1, t_2) = \left(\frac{\phi}{\pi}, \frac{\theta}{2\pi} \right)$$

is applied. Here r and t denote the co-ordinates before and after the transformation.

Figure 2.17: Problem Setup for the Cantilever Beam in the Channel.

Figure 2.18: Overlapping Grids for the Cantilever Beam in a Channel

The co-ordinates (ϕ, θ) are given by:

$$\tan \phi = \frac{2\sigma}{1-\sigma^2}, \quad \tan \theta = \frac{s_1}{s_2}; \quad s_1 = \left(r_1 - \frac{1}{2}\right)s_a, \quad s_2 = \left(r_2 - \frac{1}{2}\right)s_b, \quad \sigma^2 = s_1^2 + s_2^2 \quad (2.5)$$

In the above expression, s_a and s_b denote the dimensions of the plane that is projected on to the wing tip. For the present problem a value of 0.2 is assigned to s_a and s_b . Using the above procedure, both wing tips are re-parameterized to produce separate mappings from which volume grids are built using a hyperbolic mapping algorithm. The resulting set of grids are shown in Figure 2.19(b). The volume grids for the wing tips and the surface grid for wing are identified by A,B and C respectively. At this stage the volume grid for the wing has not yet been built. Without taking the singularity into consideration, a volume grid is built using a hyperbolic mapping from the wing surface. Figure 2.19(c) shows this volume grid. For the overlapping grid algorithm to work, a share flag is introduced for the grids A,B and C similar to the cantilever grid described in Sec 2.4.4, since one side of the grids A,B, the wing tips coincide with the surface of grid C, the wing grid. The grids A,B and C are then enclosed in two background Cartesian grids, one of which is fine and has all of its sides as interpolation boundaries and the other which is relatively coarse and resembles a far-field boundary. The final set of overlapping grids is shown in Figure 2.20. When there is an arbitrary wing deformation, the hyperbolic grid algorithm is utilized during each time step to reconstruct the grid surrounding the wing. The non moving grids such as the background Cartesian grid is held fixed. As an example, one can consider the spanwise time dependent deformation of a wing with the following kinematics with reference to Figure 2.19(b):

$$y(x, z, t) = y_0(x, z) + A(z - z_c)^n \cos(\omega_R t + 2\pi Qz) \quad (2.6)$$

where the deformation y in Eq. (2.6) is expressed as a function of the chordwise co-ordinates, x , spanwise co-ordinates, z , time t and the initial undeformed state is identified by y_0 . A and p are a measure of the deformation amplitude; z_c is a point along the span where there is no deformation and ω_R is the deformation frequency. To add a wave like behavior to the wing deformation, a factor Qz is added, Q being a quantity that determines how many cycles of deformation are present per unit span. Figure 2.21 shows different spanwise deflection profiles for an arbitrary wing for $y_0=1$, $A=0.5$, $n=2$, $z_c=0$,

Figure 2.19: Overlapping Grids for a NACA 0012 Cross-Section Wing (a) Surface Grid with a Singularity (b) Volume Grids for the Wing Tips and (c) Volume Grids for the Wing

Figure 2.20: Overlapping Grids for the Wing Showing the Inner Cartesian Grid (Green Boundary) and the Outer Cartesian Grid (Blue Boundary)

Figure 2.21: Spanwise Deflection Profiles for Different Values of Q

Figure 2.22: Overlapping Grid for a Wing Deforming in the Spanwise Direction

$\omega_R=12.56$, $t=0.5$ s and Q varying from 0.7 to 1.5. Figure 2.22 shows an example of a wing in the deformed state. An animation of the dynamic motion of the wing and the grid can be found in Appendices A3.2.2 and A3.2.3.

Chapter 3

Computational Flow Modeling

The governing equations for fluid flow are the unsteady incompressible Navier-Stokes Equations written in “Velocity-Divergence” form,

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} + \frac{\nabla p}{\rho_f} = \nu \Delta \mathbf{u} + \mathbf{F} \quad (3.1a)$$

$$\nabla \cdot \mathbf{u} = 0 \quad (3.1b)$$

where $\mathbf{u}$ is the velocity vector, p , the pressure and $\mathbf{F}$, the external forces, e.g. gravity if any. The momentum Equation, Eq. 3.1(a) has an intrinsic divergence free condition incorporated, yet it is supplemented by Eq. 3.1(b) so that the divergence free condition acts as a constraint for the numerical solution of pressure at all times. A variety of numerical methods exist in literature for solving the above set of equations. Those methods that solve directly for the pressure in the elliptic step are termed as pressure-poisson methods as in Henshaw [106] and those that solve an auxiliary equation for the pressure are projection methods as in Bell et al. [107], Brown and Minion [108]. A detailed review of the existing methods, including issues on boundary conditions can be found in Brown and Minion [108], Minion and Brown [109], Brown et al. [110]. The features of *cgins/OverBlown* [83], the Incompressible Navier Stokes Solver which uses a “Pressure-Poisson” method are described in this Chapter. The Navier-Stokes Equations form an Initial Boundary Value problem with Initial conditions, $\mathbf{u}(\mathbf{x}, 0) = \mathbf{f}(\mathbf{x})$ and boundary conditions $B(\mathbf{u}, p) = \mathbf{g}_b$. The initial conditions must satisfy $\nabla \cdot \mathbf{f} = 0$. In the above form, the velocity and pressure are strongly coupled, rendering the conventional numerical methods useless. An alternative form is the pressure-poisson system “Velocity-Pressure” formulation in which the pressure can be obtained explicitly from the velocity. This is given by

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} + \frac{\nabla p}{\rho_f} - \nu \Delta \mathbf{u} - \mathbf{F} = 0 \quad (3.2a)$$

$$\Delta P - (\nabla u_x \cdot \mathbf{u}_x + \nabla v_y \cdot \mathbf{u}_y + \nabla w_z \cdot \mathbf{u}_z) - C_{dd}(\nu) \nabla \cdot \mathbf{u} - \nabla \cdot \mathbf{F} = 0 \quad (3.2b)$$

Eq. 3.2(b) is a consequence of taking divergence of the momentum equation Eq. 3.2(a). Here $P = p/\rho_f$. An extra term $C_{dd}(\nu)\nabla\cdot\mathbf{u}$ is added to the pressure equation to damp the divergence. As stated by Henshaw [111], this does not have any effect in the continuous case but, in the discrete case, this term will keep the divergence small. The above set of equations are discretized in space on a set moving of overlapping meshes as described in Chapter 2. As of now, consider a set of meshes $\mathbf{M}_i$ which overlap each other. Out of these meshes, some of them will enclose the body such as an airfoil in two-dimensions or a wing in three-dimensions. One of them will form a background mesh that will define the extent of the computational domain. It is now required to discretize the governing equations, Eq. 3.2(a)-(b) on these meshes and solve them using a higher order accurate stable scheme. The method used in *cgins/OverBlown* [83] is the split-step or fractional-step approach is described in the following sections.

3.1. Discretization of the Governing Equations

On a stationary system of meshes, each component mesh transforms to a unit rectangular mesh using the transformation $\mathbf{x}=\mathbf{G}_t(\mathbf{r})$ where $\mathbf{x}$ and $\mathbf{r}$ denote co-ordinates of the points on the physical and computational space respectively. The corresponding transformation for a moving grid is given by $\mathbf{x}=\mathbf{G}_t(\mathbf{r}, t)$. In the frame moving with the grid, the Navier Stokes equations are given by

$$\frac{\partial \mathbf{U}}{\partial t} + \left[(\mathbf{U} - \dot{\mathbf{G}}_t) \cdot \tilde{\nabla} \right] \mathbf{U} + \tilde{\nabla} P = \nu \tilde{\Delta} \mathbf{U} \quad (3.3)$$

$$\tilde{\Delta} P + (\tilde{\nabla} U_x U_x + \tilde{\nabla} V U_y + \tilde{\nabla} W U_z) - C_{dd}(\nu) \tilde{\nabla} \cdot \mathbf{U} - \nabla \cdot \mathbf{F} = 0 \quad (3.4)$$

Here $P = \frac{p}{\rho}$ and the terms U_x , U_y and U_z are derivatives of fluid velocity vector in the moving frame with respect to co-ordinates of the inertial frame and can be written in index notation as:

$$\frac{\partial U}{\partial x_i} = \frac{\partial U(r, t)}{\partial x_i} = \frac{\partial U}{\partial r_j} \frac{\partial r_j}{\partial x_i} \quad (3.5)$$

The modified gradient operator $\tilde{\nabla}$ is a function of the co-ordinate transformation parameters and is given in indicial notation by:

$$\tilde{\nabla}_k = \frac{\partial r^j}{\partial x_k} \frac{\partial}{\partial r^j} \quad (3.6)$$

In component form, we can write Eq. (3.6) as

$$\tilde{\nabla}_1 = \frac{\partial r^1}{\partial x_1} \frac{\partial}{\partial r^1} + \frac{\partial r^2}{\partial x_1} \frac{\partial}{\partial r^2} + \frac{\partial r^3}{\partial x_1} \frac{\partial}{\partial r^3} \quad (3.7)$$

$$\tilde{\nabla}_2 = \frac{\partial r^1}{\partial x_2} \frac{\partial}{\partial r^1} + \frac{\partial r^2}{\partial x_2} \frac{\partial}{\partial r^2} + \frac{\partial r^3}{\partial x_2} \frac{\partial}{\partial r^3} \quad (3.8)$$

$$\tilde{\nabla}_3 = \frac{\partial r^1}{\partial x_3} \frac{\partial}{\partial r^1} + \frac{\partial r^2}{\partial x_3} \frac{\partial}{\partial r^2} + \frac{\partial r^3}{\partial x_3} \frac{\partial}{\partial r^3} \quad (3.9)$$

Similarly, the operator, $\tilde{\Delta}$ can be written as

$$\tilde{\Delta} = \frac{\partial}{\partial r^m} \left[\frac{\partial r^j}{\partial x_q} \frac{\partial}{\partial r^j} \right] \frac{\partial r^m}{\partial x_q} \quad (3.10)$$

$$= \left(\frac{\partial r^m}{\partial x^q} \frac{\partial^2 r^j}{\partial r^m \partial x_q} \right) \frac{\partial}{\partial r^j} + \left(\frac{\partial r^j}{\partial x_q} \frac{\partial r^m}{\partial x_q} \right) \frac{\partial^2}{\partial r^m \partial r^j} \quad (3.11)$$

Note the summation over j, q, m leading to a scalar operator. On a rectangular grid, let $U_i(t)$, $P_i(t)$ denote the discrete approximation to $\mathbf{U}(\mathbf{r}, t)$, $P(\mathbf{r}, t)$ and $\tilde{\nabla}_h$, $\tilde{\Delta}_h$ denote discrete operators. Let the mesh spacing be h_m in each coordinate direction m . The NS equations are now discretized using discrete operators replacing continuous operators in Eq. (3.3) and Eq. (3.4). The expressions for the derivatives of velocity are given by the standard central difference approximation :

$$\frac{\partial U_{i_m}}{\partial r^m} = D_{0m} U_{i_m} = \frac{U_{i_m+1} - U_{i_m-1}}{2h_m} \quad (3.12)$$

$$D_{+m} U_{i_m} = \frac{U_{i_m+1} - U_{i_m}}{h_m} \quad (3.13)$$

$$D_{-m} U_{i_m} = \frac{U_{i_m} - U_{i_m-1}}{h_m} \quad (3.14)$$

$$\frac{\partial^2 U_i}{\partial r^m \partial r^j} = D_{+m} D_{-m} U_{i_m} \quad (3.15)$$

In the above equations, m denotes the component direction as described before, the suffix 0,+,- denotes central, forward and backward differences respectively and i_m denotes the co-ordinate index along direction m . Before proceeding to the time discretization, the boundary conditions are explained and will serve as a supplement to the time discretization procedures.

3.2. Boundary Conditions

To aid in discretizing the governing equations at a boundary, one/two ghost points are added for second/fourth order accurate discretization respectively.

3.2.1. Wall boundary Conditions

For the velocity components, the Dirichlet boundary condition $\mathbf{U}=\mathbf{U}_w$ is used, where $\mathbf{U}_w$ is the velocity of the wall. However, for the pressure equation, the implementation is not straightforward and has resulted in lots of controversies over the years. A detailed discussion of this topic can be found in Gresho [112]. The boundary condition for pressure which is used in the Laplacian, must come from the governing equations and must introduce new information into the solution. It turns out that at the first instance, the appropriate boundary condition for pressure is obtained by taking the normal component of the momentum equation on the boundary. From Eq. (3.4), we have

$$\mathbf{n} \cdot \tilde{\nabla} P = \mathbf{n} \cdot (-\ddot{\mathbf{G}}_t + \nu \tilde{\Delta} \mathbf{U}) \quad (3.16)$$

where $\ddot{\mathbf{G}}_t$ is the acceleration of the boundary. Eq. (3.16) however does not add new information into the solution as stated in Henshaw [111]. A more robust boundary condition which incorporates a divergence free velocity field at the boundary can be obtained by using the simple vector identity

$$\tilde{\Delta} \mathbf{U} = \tilde{\nabla}(\tilde{\nabla} \cdot \mathbf{U}) - \tilde{\nabla} \times \tilde{\nabla} \times \mathbf{U} \quad (3.17)$$

Setting $\tilde{\nabla}(\tilde{\nabla} \cdot \mathbf{U}) = 0$, in Eq. (3.17), the “curl-curl” boundary condition is given by

$$\mathbf{n} \cdot \tilde{\nabla} P = \mathbf{n} \cdot (-\ddot{\mathbf{G}}_t - \tilde{\nabla} \times \tilde{\nabla} \times \mathbf{U}) \quad (3.18)$$

It is shown in Henshaw [111] that the above boundary condition along with the initial condition $\tilde{\nabla} \cdot \mathbf{U} = 0$ will enforce $\tilde{\nabla} \cdot \mathbf{U} = 0$ at all times. To discretize the above boundary conditions, the velocity gradients at the boundary in the above expression require values

at the ghost points. These values are obtained enforcing the divergence free condition at the boundary. As illustrated in Figure 3.1, if u, v are the velocity components of the vector U in two-dimensions, for a rectangular grid, the value at the ghost point is obtained as:

$$\begin{aligned}\nabla \cdot U_b &= 0 \\ \Rightarrow \frac{u_1 - u_{-1}}{2h_1} &= -\frac{v_{j+1} - v_{j-1}}{2h_2} \\ \Rightarrow u_{-1} &= u_1 + \frac{h_1}{h_2} (v_{j+1} - v_{j-1})\end{aligned}\tag{3.19}$$

where the velocity vector at the boundary point, first grid point inside the boundary and the ghost point are identified by U_b , U_1 and U_{-1} respectively. The tangential component of velocity v is extrapolated to the ghost point.

3.2.2. Other Physical Boundaries

Other physical boundaries that are relevant to the problem are inflow, outflow and a slip wall boundary. The mathematical equivalent of these boundaries are summarised in Table 3.1 with reference to Figure 3.2 which describes a sample problem for the flow past a wing in three-dimensions. The discussion on interpolation boundary which is highlighted in green in Figure 3.2 can be found in Chapter 2. Other boundary conditions appearing in Table 3.1 are explained below:

3.2.2.1 Inflow Boundary : At the inflow (PQRS), a Dirichlet boundary condition on U is specified. If the values at ghost points are also required, the divergence free condition is applied along this boundary, and will result in a Neumann condition for u in the r_1 direction since the other components v and w are the same across the inflow boundary under the assumption that the inflow boundary is undisturbed by perturbations due to the flow past the body.

Figure 3.1: Application of Divergence Free Condition at a Wall Determines the Velocity at the Ghost Point

Figure 3.2: Other Boundary Conditions for the Flow Past a Wing in Three-Dimensions

Table 3.1: Description of Boundary Condition Types for Different Regions in the Computational Domain with Reference to Figure 3.2

Boundary condition type	Region	Boundary condition
Wall (No slip)	Wing surface	$u = 0, \nabla \cdot u = 0, \frac{\partial p}{\partial n} = n \cdot (-\ddot{G}_t - v \nabla \times \nabla \times u)$
Slip wall	P-R-R1-P1-P Q-S-S1-Q1-Q Q-P-P1-Q1-Q S-R-R1-S1-S	$n \cdot u = 0, \frac{\partial}{\partial n} (t_m \cdot u) = 0, \nabla \cdot u = 0$
Inflow	P-Q-S-R-P	$u = u_s \text{ (velocity specified)}, \frac{\partial p}{\partial n} = 0$
Outflow	P1-Q1-S1-R1-P1	Extrapolate $u, \frac{\partial p}{\partial n} = 0$

The other components namely v and w are extrapolated using a second order extrapolation. For the pressure, a homogeneous Neumann condition is applied. This directly follows from the momentum equation with the assumption of uniform flow at the inlet.

3.2.2.2 Outflow Boundary : At the outflow boundary (P1-Q1-S1-R1-P1) and at the ghost points adjacent to it, the values are extrapolated from interior using the same second order extrapolation mentioned previously. Physically this requires, the outflow boundary to be as far away as possible. A discussion on this aspect will be shown in Chapter 4. For the pressure, a homogeneous Neumann condition is applied.

3.2.2.3 Slip Wall Boundaries: Here the normal derivative of the tangential velocity are set to zero along with the normal component of velocity and this forces the flow to be parallel to the wall. Having found the components along the wall, the velocity in the r_2 direction at the ghost point can be obtained from the divergence free condition. Other components are extrapolated. For the pressure, the same homogeneous Neumann condition is applied.

3.3. Rigid Body Dynamics: Coupling with Navier Stokes Equations

Most of the problems considered in this work involve computing the trajectory of a deformable airfoil/wing due to thrust generation with at least one degree of freedom. Using the computed aerodynamic forces and moments, the position and orientation of the body are determined at each time t by solving the equations of motion for a rigid body given by

$$M \frac{d^2 \mathbf{x}_{cm}}{dt^2} = \mathbf{F}_{Ap} + \varepsilon \mathbf{F}_{Av} \quad (3.20)$$

$$\frac{d\mathbf{h}_m}{dt} = \mathbf{T} \quad (3.21)$$

where $\mathbf{x}_{cm}$ is the position of the centre of mass, $\mathbf{h}_m$ is the angular momentum, $\mathbf{F}_{Ap}$, $\mathbf{F}_{Av}$ are the pressure and viscous forces acting on the centre of mass respectively and is obtained as $\mathbf{F}_{Ap} = \int_{d\Omega} p n_i dS$, $\mathbf{F}_{Av} = - \int_{d\Omega} n_k \tau_{ki} dS$, $\mathbf{T}$ is the torque defined by

$$\mathbf{T} = \int_{d\Omega} (\mathbf{r}' - \mathbf{x}_{cm}) d\mathbf{F}_A \quad \text{where } \mathbf{r}' \text{ is any point on the surface of the body where the}$$

differential $d\mathbf{F}_A$ is calculated and ε is a switch which takes either 1 or 0 to decide whether both the pressure and viscous forces are required for the computation of the position of the body. The position of the center of mass can be obtained by integrating Eq. (3.20) using a second order predictor-corrector algorithm as in Henshaw [113]. The orientation of the rigid body with respect to the initial axes can be obtained by tracking the orientation of the principle axes $\mathbf{e}_i(t)$ with respect to its initial position $\mathbf{e}_i(0)$ by relating it to the angular momentum vector $\mathbf{h} = \mathbf{I} \cdot \boldsymbol{\Omega}$, where $\mathbf{I}$ is the inertia tensor, which is a 3 x 3 matrix and $\boldsymbol{\Omega}$ is the angular velocity vector about the body fixed axes. In the reference frame moving with the rigid body, the off-diagonal terms of the moment of inertia tensor vanish and the angular momentum can be expressed in a simpler form as $h_k = I_k \Omega_k \mathbf{e}_k$ where $\mathbf{e}_k$ denotes the unit vectors along the principal axes. Using this in Eq. (3.21), the torque along axis i , is given by

$$\mathbf{T} \cdot \mathbf{e}_i = I_i \dot{\Omega}_i \quad (3.22)$$

The principal axes also satisfy the relation,

$$\frac{d\mathbf{e}_i}{dt} = \boldsymbol{\Omega} \times \mathbf{e}_i \quad (3.23)$$

where $\boldsymbol{\Omega} \times$ is an antisymmetric tensor given by

$$\boldsymbol{\Omega} \times = \begin{pmatrix} 0 & -\Omega_3 & \Omega_2 \\ \Omega_3 & 0 & -\Omega_1 \\ -\Omega_2 & \Omega_1 & 0 \end{pmatrix} \quad (3.24)$$

The principal axes $\mathbf{e}_i(t)$ can be determined by solving Eq. (3.22) and Eq. (3.23) using a second order predictor-corrector algorithm as in Henshaw [113]. Noting that each of the principal axes $\mathbf{e}_i(t)$ can be expressed as $\mathbf{e}_i(t) = \mathbf{R}\mathbf{e}_i(0)$, the rotation matrix $\mathbf{R}$ is obtained as $\mathbf{R} = \mathbf{E}(t)\mathbf{E}(0)^{-1}$ where $\mathbf{E}$ is now a matrix whose columns are the components of the i^{th} unit vector. The rotation matrix $\mathbf{R}$ holds the information about the angular orientations of all three axes with respect to the initial position. This rotation matrix is then used to update the position, velocity and acceleration of any point on the rigid body including the boundaries. The boundary acceleration obtained is then used in the boundary condition for pressure in Eq. (3.18). Using the above procedure to integrate Eq. (3.20) and Eq. (3.21), induces a small lag between the flow and rigid body dynamics since the forces and moments at the previous time step are used to obtain the position and orientation at the new time step. This type of coupling is termed as weak coupling. The weakly coupled system can be converted to a strongly coupled system by introducing an inner iteration loop or by adding multiple corrector steps in the predictor-corrector algorithm as described in Sec 3.5. In the current implementation of the algorithm, multiple corrector steps are used only when the ratio of the density of the body to that of the fluid falls below a certain limit depending on the shape of the body for maintaining stability. However for other cases, a single predictor and corrector step was found to be sufficient to guarantee stability of the coupled algorithm. More discussions on the coupling with respect to fluid-structure interaction problems can be found in Chapter 6, 7.

3.4. Artificial Dissipation

It is a well stated fact that solutions to Navier-Stokes equations with small viscosity are usually *turbulent* and possess a variety of scales that change with space and time. These type of flows are characterised as high Reynolds number flows. High Reynolds number

two- and three-dimensional simulations have been the centre of attention for many researchers worldwide as it is characterized by a wide range of features. However resolving a turbulent flow completely is difficult owing to the nature of the Navier-Stokes equations and “nature” as well. As the Reynolds number increases, the range of scales in the flow increase, and the size of the smallest scale reduces. This requires additional grid points to resolve the flow and it is a daunting task to determine the number of grid points required to resolve a flow completely at a given Reynolds number. Another well known fact is that when an attempt is made to simulate a viscous flow problem on a grid that is relatively coarse, non-physical oscillations or numerical turbulence governs the solution. This will ultimately lead to bad or no convergence at all. Turbulence models have been used to partially model a wide range of scales. However for large amplitude flapping in low Reynolds number flow conditions, Young [39] has shown that, both laminar and turbulent wakes are very similar and that under certain cases, addition of a turbulence model is not very beneficial. In order to understand the nature of scales at low viscosities, many researchers have developed theoretical estimates for the smallest scales in a turbulent flow, to name a few, Lilly [114], Orzag [115], Henshaw et al. [116]. Using the expression for the smallest scale derived by Henshaw et al. [116], which have been incorporated in *cgins* [83], An expression that relates the amount of artificial dissipation required for stability with the kinematic properties of the airfoil motion is obtained. In Henshaw et al. [116], the minimum scale of the flow λ_{min} is related to the global bounds of the velocity gradient $\overline{|Du|}_{\infty}$ by

$$\lambda_{min} = \sqrt{\frac{\nu}{\overline{|Du|}_{\infty}}} \quad (3.25)$$

The minimum scale based on energy dissipation rate $\varepsilon = 2\nu \int \|\mathbf{u}\|^2 dV$, where V is the volume bounded by the fluid, is given by the Kolmogoroff dissipation scale

$$\lambda_{3D} = \frac{\nu^{3/4}}{\varepsilon^{1/4}} \quad (3.26)$$

It is found that for two dimensional flows, bounds of $\overline{|Du|}_{\infty}$ can be found in terms of vorticity. However this is not the case with three dimensional flows where it has to be assumed. In Henshaw et al. [116] it is shown that the minimum scale defined by Eq.

(3.25) and the Kolmogoroff scale defined by Eq. (3.26) are equivalent if, in three-dimensions, one assumes that $\overline{|Du|}_\infty : \nu^{-1/2}$, and the energy dissipation rate, $\varepsilon : O(1)$. Eq. (3.25) then reduces to the Kolmogoroff scale $\lambda_{min} = \nu^{3/4}$. In the present case, a flapping foil is considered and the velocity gradient based on the boundary velocity is taken into consideration to compute $\overline{|Du|}_\infty$. Let us suppose that the airfoil is executing plunge oscillations at a frequency f and amplitude h_0 . The body now follows a simple periodic motion given by

$$h = h' + h_0 \sin(2\pi ft) \quad (3.27)$$

Here h' is a constant. The maximum possible boundary velocity is thus $u_{max} = 2\pi fh_0$ and this occurs at time $t = 0$. Now, the velocity gradient is given by the following expression:

$$\overline{|Du|}_\infty \leq \frac{2\pi fh_0}{h_m} \quad (3.28)$$

where h_m is the shortest distance from the boundary to the first grid point. Consider now the case when the minimum scale $\lambda_{min} < h_m$. This would correspond to an under-resolved flow which might give rise to unphysical oscillations. By adding artificial dissipation to the momentum equations of the form $d = \nu_a \Delta u$, the minimum scale can be captured by the grid [116]. The coefficient ν_a is given by

$$\nu_a = (\kappa_{21} + \kappa_{22} |\nabla u|) \quad (3.29)$$

For simplicity $\kappa_{21} = \kappa_{22} = \kappa$ is assumed. These coefficients are now determined for the specific case of a plunging airfoil. Our assumption that the minimum scale being less than the shortest boundary distance to the first grid point, $\lambda_{min} < h_m$ is still valid. Let artificial dissipation ν_a be added so that the minimum scale is enlarged and fits exactly the shortest boundary distance to the first grid point. i.e. $\lambda_{min} \rightarrow h_m$. Hence Eq. (3.25) becomes

$$h_m^2 = \frac{\nu + \nu_a}{|\nabla u|} \quad (3.30)$$

Now eliminating ν_a from Eq. (3.29) and Eq. (3.30), the artificial dissipation coefficient is obtained as

$$\kappa \sim \frac{h_m^2 |\nabla u| - \nu}{1 + |\nabla u|} \quad (3.31)$$

More generally in terms of frequency of oscillation, from Eq. (3.28) at a given Reynolds number,

$$\kappa \sim \frac{f}{1 + f} \quad (3.32)$$

Eq. (3.31) and Eq. (3.32) tell us that if the mesh is refined, κ has to be decreased and if the frequency is increased, κ has to be increased. The above artificial dissipation is added to the momentum equations. Although most of the flow features are captured by adding this quantity, it also results in loss of accuracy if too much of it is added. There exists an optimum range of κ below which the solution is unstable and above which it is highly dissipative. Typical values of κ are 0.1-1.0 for low Reynolds number flows of the order 10^3 and 2.0-5.0 for high Reynolds number flows of the order 10^4 .

3.5. Time Discretization and Overall Solution Procedure

The solver uses a multi-step predictor-corrector algorithm to integrate Eq. (3.3) and Eq. (3.20). This method is outlined in detail in Henshaw [113]. Starting with the momentum equation, Eq. (3.3), an implicit predictor step is applied for the solution of the predicted velocity ($\mathbf{u}^p$). This results in a set of linear equations of the form

$$\nu \Delta \mathbf{u}^p - \frac{2}{\Delta t} \mathbf{u}^p = 3(\mathbf{u} \cdot \nabla \mathbf{u})^n - (\mathbf{u} \cdot \nabla \mathbf{u})^{n-1} + 3\nabla p^n - \nabla p^{n-1} - \frac{2}{\Delta t} \mathbf{u}^n - \nu \Delta \mathbf{u}^n + \mathbf{d}^n \quad (3.33)$$

where $\mathbf{d}^n = (\kappa_{21} + \kappa_{22} |\nabla \mathbf{u}^n|) \Delta \mathbf{u}^n$ represents artificial dissipation. At this stage, the forces ($\mathbf{F}_A$) and moments ($\mathbf{T}$) are determined and the predictor step for the rigid body equations is called. The predicted linear and angular velocities, principle axes and displacements are calculated using a centered leap-frog scheme as outlined in Henshaw [113].

$$\left(\frac{d\mathbf{x}_{cm}}{dt} \right)^p = \left(\frac{d\mathbf{x}_{cm}}{dt} \right)^{n-1} + 2\Delta t \left(\frac{\mathbf{F}_A}{M} \right)^n \quad (3.34)$$

$$\mathbf{x}_{cm}^p = 2\mathbf{x}_{cm}^n - \mathbf{x}_{cm}^{n-1} + \Delta t^2 \left(\frac{\mathbf{F}_A}{M} \right)^n \quad (3.35)$$

$$\Omega^p = \Omega^{n-1} + 2\Delta t \left(\frac{\mathbf{T} \cdot \mathbf{e}_i}{I_i} \right)^n \quad (3.36)$$

$$e_i^p = e_i^{n-1} + 2\Delta t \left(\frac{d\mathbf{e}_i}{dt} \right)^n \quad (3.37)$$

The boundary acceleration is then computed at this intermediate step. Using this information, the pressure equation is solved as

$$\Delta p^p = -\tilde{\nabla} \cdot (\mathbf{u} \cdot \tilde{\nabla} \mathbf{u})^p, \quad \frac{\partial p}{\partial n} \Big|_{\partial \Omega} = n \cdot (-\ddot{\mathbf{G}}_t - \nu \tilde{\nabla} \times \tilde{\nabla} \times \mathbf{u})^p \quad (3.38)$$

The corrector step is executed using a trapezoidal-rule [113] as follows:

$$\nu \Delta \mathbf{u}^{n+1} - \frac{2}{\Delta t} \mathbf{u}^{n+1} = (\mathbf{u} \cdot \nabla \mathbf{u})^p + (\mathbf{u} \cdot \nabla \mathbf{u})^n + \nabla p^p + \nabla p^n - \frac{2}{\Delta t} \mathbf{u}^n - \nu \Delta \mathbf{u}^n + \mathbf{d}^p + \mathbf{d}^n \quad (3.39)$$

$$\left(\frac{d\mathbf{x}_{cm}}{dt} \right)^{n+1} = \left(\frac{d\mathbf{x}_{cm}}{dt} \right)^n + \frac{\Delta t}{2} \left(\frac{F_A^n}{M} + \frac{F_A^p}{M} \right) \quad (3.40)$$

$$\mathbf{x}_{cm}^{n+1} = \mathbf{x}_{cm}^n + \frac{\Delta t}{2} \left(\frac{d\mathbf{x}_{cm}^n}{dt} + \frac{d\mathbf{x}_{cm}^p}{dt} \right) \quad (3.41)$$

$$\Omega^{n+1} = \Omega^n + \frac{\Delta t}{2} \left(\frac{\mathbf{T}^n \cdot \mathbf{e}_i^n}{I_i} + \frac{\mathbf{T}^p \cdot \mathbf{e}_i^p}{I_i} \right) \quad (3.42)$$

$$e_i^{n+1} = e_i^n + \frac{\Delta t}{2} \left(\frac{d\mathbf{e}_i^n}{dt} + \frac{d\mathbf{e}_i^p}{dt} \right) \quad (3.43)$$

The boundary acceleration is computed once again and the pressure equation is solved to determine the updated value at time t^{n+1} . In the above solution procedure, the implicit discretization for the velocity and the discretization of the pressure equation results in a set of linear equations which can be solved by direct or iterative algorithms. When the grid size is small, a direct method such as LU decomposition is used. For large three-dimensional problems, the Krylov subspace solver, Bi-Conjugate Gradient Stabilized (Bi-CGSTAB) method from *PETSc* [117] is used.

Chapter 4

Numerical Verification and Validation for Unsteady Flow Problems

4.1. Introduction

Verification and validation are two terms that always appear in the numerical solution of partial differential equations. Verification is defined as the process of determining that a model implementation accurately represents the developer's conceptual description of the actual physical model and the solution to the model. According to AIAA standards [118], this means – solving the equations right. Verification can be used to probe the code for possible bugs in the structure and model implementation, and to investigate possible errors due to numerical discretization. Validation is defined as the process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model. It also means – solving the right equations [118]. Validation includes both the verification of the computation as described earlier as well as quantification of errors with respect to an experimental calculation. In this chapter, both verification and validation studies are performed for a variety of unsteady flow computations at low Reynolds numbers. Grid refinement studies are carried out and computations are compared with available computational and experimental data. The effect of the outer fictitious boundary on computed solutions is also quantified.

4.2. Non-Dimensional Parameters

Based on the analysis in Appendix A, one can conclude that, the relevant non-dimensional parameter governing unsteady fluid flow for a non-moving body is the Reynolds number. When a body is in motion, depending on the relevant time scale and a fixed reference velocity U_S , there arises two different non-dimensional parameters, the reduced frequency k , for oscillating bodies and the Froude number Fr , for free falling bodies. If the reference velocity is a function of reduced frequency, then the Strouhal Number St is also a governing non-dimensional parameter for oscillating bodies. The expressions for these non-dimensional parameters vary in literature, however in the

present work, we choose to define the reduced frequency as $k = \frac{\pi f c}{U_S}$, the Strouhal number as $St = k h_0$, the Reynolds number as $Re = \frac{U_S c}{\nu}$ and the Froude number as $Fr = \frac{U_S}{\sqrt{g c}}$, where f is the frequency of oscillation, c is the chord, h_0 is the plunge amplitude scaled by chord, ν is the kinematic viscosity of the fluid and g the acceleration due to gravity. For simple problems that involve pure plunging, such as, oscillations about an axis perpendicular to the chord, the relevant non dimensional parameters are straightforward to compute. However, it is not the case for the type of kinematics that involves both pitching and plunging in three-dimensions as in Chapter 5. In the following sections, variants of the non-dimensional quantities are presented based on the problem in hand.

4.2.1. Expressions for Non-Dimensional Parameters for a Plunging and Pitching Wing

Many problems considered involve plunging/pitching of a wing in zero freestream conditions. Since there is no incoming freestream, the maximum translational velocity is taken as the velocity scale. When a wing flaps about the X-axis as outlined in Figure 4.1(a), the reference velocity $U_S = 2\pi f \theta_0 S$ and $h_0 = \frac{S \sin \theta_0}{c}$, where θ_0 is the flap angle amplitude and S is the span. The corresponding reduced frequency and Strouhal number are given by:

$$k = \frac{c}{2\theta_0 S}, St = \frac{1}{2} \frac{\sin \theta_0}{\theta_0} \quad (4.1)$$

For flapping about the Z-axis as described in Figure 4.1(b), the velocity scale $U_S = \pi f \theta_0 c$ and amplitude $h_0 = \frac{1}{2} \sin \theta_0$. The corresponding reduced frequency and Strouhal number are given by

$$k = \frac{1}{\theta_0}, St = \frac{1}{2} \frac{\sin \theta_0}{\theta_0} \quad (4.2)$$

Figure 4.1: Coordinate System and Wing Kinematical Parameters for Rotation about (a) X-axis and (b) Z-axis

4.2.2. Expressions for Non-Dimensional Parameters for a Free Falling Body

For free falling bodies, the velocity scale is based on the terminal velocity of the body and is given by $U_s \approx \sqrt{\zeta_R g h_t}$, where ζ_R is the ratio of the density of the body to the density of the ambient fluid. Using this expression, in the definition of Froude number defined earlier, we obtain:

$$Fr^2 = \zeta_R \beta \quad (4.3)$$

where β is called the thickness to chord ratio. This expression for Froude number was also used by Belmonte et al. [119] to analyze free falling plates. Andersen [120] uses also the non-dimensional moment of inertia

$$I^* = \frac{I}{\rho_f c^4} = \frac{\rho_b h_t c (h_t^2 + c^2)}{12 \rho_f c^4} = \frac{1}{12} \frac{\rho_b}{\rho_f} \frac{h_t}{c} \left(\frac{h_t^2}{c^2} + 1 \right) = \frac{1}{12} \zeta_R \beta (\beta^2 + 1) \quad (4.4)$$

where I , h_t , c , ρ_b are the moment of inertia, thickness, chord and density of the plate respectively and ρ_f is the density of the ambient fluid. Interestingly, it can be seen that the non-dimensional moment of inertia and the Froude number are equivalent.

Comparing those expressions, we can arrive at $I^* \propto Fr^2$.

4.3. Grid Refinement Study for a Plunging Airfoil

The effect of mesh size on the numerical accuracy is an important aspect in assessing the quality of a flow solution. Here an airfoil undergoing plunge oscillations, or oscillations along a plane perpendicular to the chord at zero angle of attack is considered with different mesh sizes, and the aerodynamic force coefficients are computed. A sinusoidal displacement $h = h_0 \sin(2\pi ft)$ with an amplitude $h_0 = 0.12$ and frequency $f = 0.979$ Hz is prescribed for the airfoil. The flow Reynolds number is set to $Re = 500$ and the Strouhal number to $St = 0.75$. Four mesh systems are considered, each mesh system consisting of three overlapping meshes, a background Cartesian mesh, an airfoil curvilinear mesh and a wake Cartesian mesh. The details of the mesh are summarized in Table 4.1. When the airfoil moves, the airfoil mesh and the wake mesh move rigidly with the airfoil.

Table 4.1: Various Meshes Considered for Convergence Studies

Component Mesh	Mesh System	Mesh System	Mesh System	Mesh System
	1	2	3	4
Square(Background)	30 x 30	30 x 30	41 x 41	81 x 81
Wake	20 x 20	30 x 30	61 x 61	121 x 121
Airfoil	30 x 30	60 x 35	121 x 71	242 x 142

The flow is initialized with a steady flow field and the solutions are computed at regular intervals of time. Figure 4.2 and Figure 4.3 show the computed time histories of aerodynamic lift and drag coefficients on the four mesh systems along with a close-up view of the peak lift and drag coefficients. It can be seen that on the coarse mesh, the minimum drag coefficient is under-predicted or the thrust is maximum. This has a direct implication on the forward flight arising due to plunging motion. An airfoil set in motion will travel faster when the solutions are computed on a coarse mesh due to higher thrust coefficients. Defining average lift and drag coefficients is not appropriate for unsteady flows in general, whether they are periodic or not. For example when the forward flight of an insect or a bird is considered, it is the instantaneous lift/thrust coefficient which dictates the entire motion and not the average force coefficients. However for the purposes of obtaining quantitative results for mesh dependency study, the force coefficients are averaged out over one cycle. For the present computation, the average value of lift and drag coefficients are obtained in the interval $6 < t < 7$ seconds. These average quantities are plotted as a function of $\log(1/N)$ where N is the total number of mesh points used for a given mesh system and shown in Figure 4.4. The lift and drag coefficients reach a value of 0.069 and -0.0497 respectively in an asymptotic manner. The negative drag coefficient indicates that the airfoil is producing thrust.

Using the airfoil mesh as a reference, and mesh refinement ratio $r_m = 2$, the observed order of convergence as outlined in Roache [121] is estimated using three values of the lift coefficient $C_{L_1} = 0.069$, $C_{L_2} = 0.0773$, $C_{L_3} = 0.119$ where C_{L_1} , C_{L_2} and C_{L_3} are the lift coefficients obtained on mesh system 4, 3, 2 respectively.

(a)

(b)

Figure 4.2: (a) Lift Coefficient (b) Near View of the Lift Coefficient

(a)

(b)

Figure 4.3: (a) Drag Coefficient (b) Near View of the Drag Coefficient

Figure 4.4: Time Averaged Lift and Drag Coefficients as a Function of Grid Points

The observed order of convergence turns out to be $p = \frac{1}{\ln(r_m)} \ln \left(\frac{C_{L3} - C_{L2}}{C_{L2} - C_{L1}} \right) = 2.06$. To verify whether the solutions are in the asymptotic range of convergence, the Grid Convergence Index (GCI) which is a measure of the deviation from the asymptotic value is computed for the fine grid. An estimate of the GCI as outlined in Roache [121] for the fine mesh which is mesh system 4 is given by $GCI = \frac{F_s |\epsilon|}{r_m^p - 1}$ where $F_s = 1.25$, is a factor of safety while comparing three or more meshes and $\epsilon = \frac{C_{L2} - C_{L1}}{C_{L1}}$. The computed value of the GCI turns out to be 0.0483. The small value of GCI indicates that the solution is in the asymptotic range of convergence. Similar values of GCI were obtained for a wide choice Reynolds numbers of the order 10^2 - 10^4 . Mesh system 4 is thus used as a reference mesh which dictates the minimum number of grid points required for reasonable solutions to be obtained.

4.4. Comparison of Plunging Airfoil Wake Structures with Experiments

The plunging performance of an airfoil in zero and finite freestream conditions with prescribed kinematics is investigated. Computed wake structures of a plunging NACA 0012 airfoil are first compared for a reduced frequency $k = 5.05$ and plunge amplitude $h_0 = 0.2$ with the experimental results of Jones et al. [122]. The Reynolds number for this case is 500. Computations are performed using a unit freestream velocity specified at the inlet, from which the fluid viscosity is obtained based on the Reynolds number. The vertical displacement of the airfoil is as per $h = h_0 (1 - \cos(2\pi ft))$. The mesh system used in the present problem is as per Sec. 4.3. It consists of three meshes, a moving airfoil mesh that moves rigidly with the airfoil, a Cartesian wake mesh and a background Cartesian mesh. Under these conditions, a deflected and a dual mode vortex street, was observed experimentally, and the direction of the jet depended on initial conditions. Figure 4.5(a) shows the computed vortex structures at $t = 2.2$ s and is compared with the corresponding experimental measurements in Figure 4.5(b). The agreement between the computational and experimentally visualized flow structures can be seen to be quite good in terms of the initial jet deflection.

A comparison of the wake structures is also made at a higher Reynolds number ($Re = 20000$) using Heathcote and Gursul's [123] experimental result. The distinguishing part of this test case was that, they observed the jet to be switching directions periodically irrespective of the initial plunge direction, when plunged in zero freestream conditions. Figure 4.5(c) shows the vortex structures obtained by Heathcote and Gursul [123] at the instant the jet is deflected downwards. For the current study, computed vortex structures in zero freestream and finite freestream conditions as shown in Figures 4.5(d)-(e) also show this phenomenon. This also results in the lift coefficient to be periodic about a positive value for some period of time and then negative, for rest of the time after 50 plunge cycles as shown in Figure 4.6. An animation of the jet switching phenomenon can be found in Appendix A3.4.1.

Figure 4.5: Plunging Airfoil Wake structures for $k=5.05$, $h_0=0.2$ (a) Current Computation (b) Experiments from Ref [122] (c) Experiments from Ref [123] at a Later Instant of Time (d) Computation with Zero Freestream Conditions and (e) Computation with Finite Freestream Conditions

Figure 4.6: Lift of a Plunging Airfoil with a Dual Mode Vortex Street , $k=5.05$, $h_0=0.2$

4.5. Rapid Pitching of NACA 0015 airfoil – Simulation of Dynamic Stall

Dynamic stall in airfoils has been the subject of many researchers for the past few decades. A wide range of experiments reported in McCroskey et al. [124] and numerical computations by Liu and Kawachi [40], Visbal and Gordnier [125], Vandyken et al. [126], Visbal and Shang [127] have been performed to study the unsteady separation and the formation of dynamic stall vortex and its impact on lift augmentation in pitching airfoils. Following the lines of Visbal and Shang [127] and Liu and Kawachi [40], the rapid pitching of a NACA 0015 airfoil about the quarter chord position from a zero degree angle of attack to a maximum of 45 degrees is demonstrated. A temporal variation of angular velocity is prescribed and is given by $\Omega(t) = \Omega_0 (1 - e^{-4.6t/t_0})$, $t \geq 0$ where $t_0 = 0.5$ is the time taken to reach 99% of the final angular velocity $\Omega_0 = 0.2$. For $Re = 10^4$, computations have been carried out on a 200×100 mesh around the airfoil and 80×80 background mesh with initial conditions projected so that the divergence condition $\nabla \cdot U = 0$ is satisfied at $t=0$. For $Re = 4.5 \times 10^4$, computations have been carried out on a 200×100 , 400×100 airfoil mesh with $\nabla \cdot U = 0$ as the initial condition and 300×50 airfoil mesh with steady flow initial conditions. For $Re=10^4$, the present computational results are compared with that of Liu and Kawachi [40] and for $Re=4.5 \times 10^4$, with the experimental results cited in Visbal and Shang [127].

Figure 4.7 shows the computed vorticity contours for the two cases of Reynolds numbers at various angles of attack $\alpha = 5, 15, 30$, and 35 degrees respectively. The flow features are very similar for both these Reynolds numbers except for the fact that finer structures are observed for the higher Reynolds number case. The leading edge vortex is seen to develop and convect downstream of the airfoil. Figure 4.8(a) shows the comparison of the computed variations of C_L and C_D with angle of attack for $Re=10^4$ with the computational results of Liu and Kawachi [40]. It can be seen that the results of the present computation are in good agreement with that of Liu and Kawachi [40]. Figure 4.8(b) shows the corresponding lift and drag characteristics for $Re = 4.5 \times 10^4$. The lift coefficient maintains a very high value till about 35 degrees for $Re = 4.5 \times 10^4$ and 30 degrees for $Re = 10^4$.

Figure 4.7: Computed Vorticity Contours for a Rapidly Pitching Airfoil at (a) $Re = 10^4$ on 200×100 Mesh and (b) $Re = 4.5 \times 10^4$ on 300×50 Mesh

(a)

(b)

Figure 4.8: Computed Aerodynamic Characteristics for a Rapidly Pitching Airfoil at
(a) $Re=10^4$ (b) $Re=4.5 \times 10^4$ on Various Grids

This can also be seen from Figure 4.7 that the vortex detachment is accompanied by a loss in lift, also observed in Figure 4.8. The position of $C_{L_{\max}}$ is in good agreement with the experimental results when the steady flow initial conditions were used. This is expected since the experiment was also conducted in a similar manner. The results from an arbitrary divergence free condition do not converge to the experimental conditions as it can be seen from Figure 4.8(b) that by mesh refinement, the computations converge to a wrong solution. The dynamic motion of the vortices can be visualized as a movie in Appendix A3.4.2 -A3.4.3.

4.6. Autorotation and Rigid Body Flutter of Rectangular/Elliptical Plates

When an object whose centre of mass is pinned to a stationary reference frame is placed in a stream of flowing fluid, it will either rotate continuously or oscillate with a particular frequency. When it rotates continuously, it is best described by the term autorotation, and when it oscillates about a particular point, it is termed as flutter. The term flutter is not to be confused with the instability observed as a result of self sustained oscillations as in conventional aeroelasticity. Here the plate continues to oscillate with constant amplitude and high frequency. Both autorotation and flutter are easily observed in many practical applications such as the windmill and store separation in aircrafts. The fluttering and free fall of leaves are some of the naturally occurring examples of these phenomena and complement research on flapping flight. A recent article by Bergou et al. [51] describes the conditions in which insect wings undergo passive pitch reversal during pronation or supination, a phenomenon very similar to autorotation. These computations also help in verifying the implementation of the coupled fluid dynamics - rigid body motion model. In this section, various computational cases corresponding to autorotation and flutter and described.

4.6.1. Autorotation of an Elliptic Cylinder

The autorotating elliptic cylinder test case considered by Lugt [128] is used in the present computation for verification purposes. In his paper, he assumed a constant angular velocity for the cylinder and maps out the condition for autorotation based on the

observation of Riabouchinsky [129] which is $d\mathbf{T} / d\bar{\Omega} > 0, \bar{T} = 0$ where $\bar{T}$ and $\bar{\Omega}$ are the average torque and angular velocity over one cycle respectively. This is done to avoid performing repeated trials to induce autorotation. The dimensionless parameters used in this case are the spin parameter, S_p , Reynolds number, Re , lift coefficient, C_L , drag coefficient, C_D and the moment coefficient, C_M which are defined in Lugt [128] as follows:

$$S_p = \frac{\bar{\Omega}L}{2U_\infty} \quad (4.5)$$

$$Re = \frac{U_\infty L}{\nu} \quad (4.6)$$

$$C_L = \frac{2F_{Ay}}{\rho U_\infty^2 (L/2)} \quad (4.7)$$

$$C_D = -\frac{2F_{Ax}}{\rho U_\infty^2 (L/2)} \quad (4.8)$$

$$C_M = \frac{2T}{\rho U_\infty^2 (L/2)^2} \quad (4.9)$$

where L is the chord of the elliptic cylinder, U_∞ is the freestream velocity. The quantities F_{Ay} and F_{Ax} represent the aerodynamic forces normal to, and along the freestream direction. The aerodynamic torque T , is computed about the center of mass of the elliptic cylinder. Computations have been performed for $S_p = 0.167, 0.25, 0.5, 1.0$ at $Re = 200$. Fixing the angular velocity of the cylinder to be $\bar{\Omega} = 1.5$ rad/s (anticlockwise), the free-stream velocity and fluid viscosity are computed for each S using a thickness ratio $\beta = 0.1$ and chord $L=1$ cm. The mesh used for this problem is shown in Figure 4.9. It consists of two meshes, a Cartesian background mesh (100 x100) and a curvilinear mesh (120 x 70) with stretching at the wall boundaries. The region of overlap is handled by appropriate interpolation. At the left boundary, a uniform velocity is specified and at the right boundary, an outflow boundary condition is specified where the flow velocities are extrapolated from interior. On the cylinder wall, the no-slip condition is imposed. The flow in the vicinity of the top and bottom boundaries is assumed to be parallel to the corresponding boundaries; hence a slip wall boundary condition is used. The pressure boundary condition for the pressure equation is obtained from the momentum equation and contains the rotational component of the boundary acceleration. With all the given

Figure 4.9: Overlapping Mesh for the Autorotation Case

Figure 4.10: Computed (a) Vorticity Contours from Lugt [128] (b) Vorticity Contours from the Present Computation and (c) λ_2 Contours from the Present Computation

conditions, the solution is computed for 20 cycles of rotation. Vorticity contours from Lugt [128] for the case $S_p = 0.25$ when the cylinder is inclined at 135° to the horizontal is shown in Figure 4.10(a) with a comparison from the present computation in Figure 4.10(b). For identifying the vortex cores correctly, the λ_2 method of Jeong and Hussain [130] has been implemented. If $\tilde{S}, \tilde{\Omega}$ denote the symmetric and anti-symmetric parts of the tensor $\nabla \mathbf{u}$, and $\Pi = \tilde{S}^2 + \tilde{\Omega}^2$, a vortex core exists if the second largest Eigen value of Π is negative. Denoting λ_2 to be this Eigen value, the contours of constant λ_2 ($\lambda_2 < 0$) are plotted in Figure 4.10(c) with the same number of contour levels as for the vorticity plot. The vortex cores are easily identified and show a good correlation with the results of Lugt [128]. Figure 4.11 shows the computed moment coefficient as a function of angle of attack $\alpha = \bar{\omega}t$ for various spin parameters, with the corresponding values from Lugt [128].

For $S_p = 0.167, 0.25$ in Figure 4.11(a)-(b), it is evident that the mean moment coefficient is negative. In this region, the aerodynamic torque supports the applied torque due to rotation and the plate is said to be ‘autorotative’ [129]. Any external torque applied in this region will make the cylinder unstable. The phenomenon of lock-in occurs during this period wherein the frequency of vortex shedding will be in resonance with that of the forced rotation. For $S_p = 0.5, 1.0$ in Figure 4.11(c)-(d) the aerodynamic torque is positive, in the pitch up sense despite the rotation of the cylinder being counterclockwise. In this region, the cylinder ceases to autorotate and will keep oscillating about a pivot. All these results show a good agreement with the observation of Lugt [128] and Riabouchinsky [129]. Also plotted is the variation of lift and drag coefficients over half a cycle of rotation in Figure 4.12. The results are compared with the computed results of Lugt [128] and the experimental results for an autorotating wing at $Re = 90000$ from Smith [131] with $\beta = 0.15$, $S_p \sim 0.8-1.0$. The computations verify the observation of Lugt [128] that the Reynolds number has little influence on the aerodynamic force coefficients in the range $200 \leq Re \leq 90000$.

(a) $S_p = 0.167$

(b) $S_p = 0.25$

(c) $S_p = 0.5$

(d) $S_p = 1.0$

Figure 4.11: Computed Moment Coefficients as a Function of Angle of Attack for
(a) $S_p = 0.167$ (b) $S_p = 0.25$ (c) $S_p = 0.5$ (d) $S_p = 1.0$

(a) Lift Coefficient

(b) Drag Coefficient

Figure 4.12: Computed (a) Lift and (b) Drag Coefficients as a Function of Angle of Attack for $S_p = 0.1$

4.6.2. Passive Rotation of Rectangular Plates

Flutter and autorotation is studied for rectangular plates whose centre of mass is pinned or has a fixed point of rotation. The computational results of Mittal et al. [132] are used for reference. A series of computations at a Reynolds number $Re=300$ and non-dimensional moment of inertia $I^*=0.17$, with varying thickness is considered. The plate under consideration is pinned to its centre of mass and is free to oscillate by virtue of the aerodynamic forces. All computations for pinned configurations are performed on overlapping meshes consisting of a background Cartesian mesh (100 x 100) and a curvilinear mesh for the rectangular plate (120 x 70) similar to the mesh in Figure 4.9. A uniform flow of velocity U_∞ is specified at the left boundary and the plate is free to rotate about its fixed centre of mass. The plate will either oscillate about a mean angle which depends on the initial angle or attain the state of constant rotation after a long duration. An initial angle of inclination though not required, will perturb the plate to achieve autorotation faster. When there is no initial angle between the plate and the direction of motion, an initial impulse is necessary to obtain the state of autorotation. If neither an initial angle nor an initial impulse or asymmetry is present, the plate is always in a stable state and will keep oscillating with low amplitude. The time history of the lift coefficient for $\beta = 0.5, 0.2$ is shown in Figures 4.13(a)-(b) along with the computations of Mittal et al. [132]. A reasonable comparison is obtained with the computations of Mittal et al. [132]. The state of rotation for $\beta = 0.5$ was classified as a fluttering mode, and for $\beta = 0.2$ was classified as autorotation. For both these cases, I^* is held fixed ($I^*=0.17$) by adjusting the density ratio. Appendix A3.4.4 shows an animation of $\log(\text{Vorticity})$ contours for the case of a plate experiencing autorotation initially, followed by flutter.

4.6.3. Free Fall of a Rectangular Plate

To validate the coupled flow-rigid body problem, the experimental test case of Andersen et al. [120] is considered. A rectangular plate of length $L = 0.648$ cms with a thickness ratio $\beta = 0.125$ and density ratio $\zeta_R = 2.7$ is released from rest in a quiescent fluid of kinematic viscosity $= 0.0089 \text{ cm}^2\text{s}^{-1}$ at an angle of 11.5° to the horizontal. Using the mean vertical velocity of the plate as a relevant velocity scale, the Reynolds number is given by 837. The aspect ratio of the plate in the experiment was large enough (~ 30) for three-

(a)

(b)

Figure 4.13: Time History of the Computed Lift Coefficient in Comparison with the Results of Mittal [132] for (a) $\beta = 0.5$ (b) $\beta = 0.2$

Figure 4.14: Free Fall of a Rectangular Plate Showing the Experimental [120] and Computational Setup

dimensional effects to be negligible. Hence a two-dimensional computation is carried out for the present case but with similar experimental conditions. Figure 4.14 shows a comparison of the experimental and computational setup. A 512×256 mesh is used in this computation. Using Eq. (4.4), the non-dimensional moment of inertia is given by $I^*=0.028$. The numerical value of I^* is different compared to the one stated in the experiment, where it is stated as $I^*=0.29$, since there is an additional factor $32/\pi$ in their definition of I^* . At time $t = 0$, the velocity field is initialized to zero since the fluid is at rest and the boundary acceleration which is obtained from the plate's apparent weight serves as the boundary condition for the pressure equation for the solution of the initial pressure. The computed pressure distribution then starts the iterative process. Since the coupling between the fluid and the rigid body is performed using a partitioned algorithm, the stability of the scheme is affected for small density ratios. This can be partially improved by using multiple corrector steps, typically 8-10 per time step. However for very small density ratios less than 2.7, 80-100 corrector steps are required to maintain stability. This incurs a large amount of computational time. More discussions on this instability can be found in Wei [133].

For the moment of inertia in the present case which is $I^*=0.028$, the plate initially executes high amplitude side to side oscillations, termed as flutter due to the asymmetry of the shed vortices from the leading and trailing edges of the plate. After a few cycles of fluttering, the combined effect of plate inertia and the fluttering motion renders the plate to start rotating continuously. This motion is termed as tumbling or autorotation. Figure 4.15 shows the computed trajectory of the plate during the tumbling sequence in comparison with experiments from Andersen et al. [120]. The computed trajectory has a minor phase lag corresponding to the points where the plate overturns compared to the experiments but overall they are similar. It is hardly possible to recover the same trajectory as it is not easy to simulate the exact experimental conditions computationally. Added to these are the errors due to numerical discretization. Appendix A3.4.6 shows an animation of the vortex structures released from the falling plate. Figure 4.16 shows a comparison of the aerodynamic forces, in terms of the apparent weight $m'g = (\rho_b - \rho_f) h_t L g$ at different angles of inclination with experiments. Discrepancies do

Figure 4.15: Free Fall of a Rectangular Plate Showing the Experimental [120] and Computational Trajectories

Figure 4.16: Aerodynamic Forces (a) F_x and (b) F_y , Normalized with the Apparent Weight as a Function of Instantaneous Angle of Inclination

exist at turning points where a vortex pair is shed but the overall comparison is satisfactory.

4.7. Effect of the Location of Outer Boundary on Aerodynamic Characteristics

In order to have accurate solutions for external flow problems, the outer boundary needs to be placed at large distances from the airfoil/wing so that the disturbances created by truncating the computational domain do not affect the solutions in the vicinity of the airfoil/wing. The following assessment is carried out to study the effect of the far-field outer boundary location on the computed aerodynamic characteristics and trajectories of a plunging and pitching 10 % thick elliptical cross-section airfoil. Three different extents of the outer boundary locations are considered. The width (W) and height (H) of the computational domain for these cases are given by (a) $W=24c$, $H=16c$ (b) $W = 12c$, $H=6c$ and (c) $W=6c$, $H=4c$ where c is the airfoil chord. The mesh density is maintained the same in all cases and the minimum mesh spacing from the wall is $O(10^{-4})$. An airfoil having a density $\rho_b = 10\rho_f$ undergoes a combined plunge and pitch oscillation analogous to a flapping wing as described in Chapter 5. The plunge and pitch are governed by the expressions, $h = -h_0(1 - \cos 2\pi ft)$, $\theta_i = \theta_0(1 - \cos 2\pi ft)$ respectively with $h_0=0.34c$, $\theta_0=10^\circ$, $k=1.47$, $St=0.5$ and $Re = 320$. The reduced frequency, k , Strouhal number, St , and Reynolds number, Re have been calculated based on the velocity scale $U_s = 2\pi fh_0$. From the aerodynamic forces, the position of the centre of mass is then obtained by integrating Newton's second law for a rigid body using Eq. (3.20) and Eq. (3.21). Figure 4.17(a) shows the variation of the thrust coefficient (C_T) during the second cycle for the three cases. Computations show that these forces are either overestimated or underestimated on a smaller domain. The forces corresponding to case (b) do not differ much from that of case (a) which corresponds to a very large domain, hence for both two- and three-dimensional computations, the extent of the outer boundary along X- and Y-axes is decided based on this fact. The computed trajectory, which is expressed in terms of the co-ordinates of the centre of mass corresponding to these three cases are shown in Figure 4.17(b). It can be seen that the trajectory corresponding to case(c) has a larger wavelength compared to that of case(a) and case(b).

(a)

(b)

Figure 4.17: Effect of Outer Boundary Distance on (a) Thrust Coefficient and (b) Trajectory

This shows that on smaller domains, numerical errors will cause the airfoil to travel faster. An animation showing the dynamic motion of the airfoil for the present case can be found in Appendix A3.4.7 and A3.4.8.

4.8. NACA 0012 Cross-section Wing Plunging at $Re=10000$

The test case of a plunging airfoil/ wing has been studied by numerous authors through numerical computation and experiments. A compilation of these, for a wide range of Reynolds numbers can be found in Ashraf et al. [134]. Presently, comparisons are made for a plunging NACA 0012 cross-section wing having aspect ratios 3 and 4 with existing experimental work of Heathcote et al. [36] and numerical work of Young [39]. Also mentioned is a comparison with the linear theory of Garrick [18] and the panel code of Young [39]. Following the lines of Heathcote et al. [36], a NACA 0012 cross-section wing having a rectangular planform with an aspect ratio of 4 is considered. Overlapping grids are generated surrounding the wing as shown in Figure 2.19 by the procedure outlined in Chapter 2. A grid refinement study was initially performed and the solutions were considered to be grid independent at this Reynolds number for the grid size mentioned in Table 4.2.

Table 4.2: Distribution of Points for Different Grids with Reference to Figure 2.19

Grid	Number of Points (chordwise x spanwise x normal)
Outer cube	51 x 51x 51
Inner cube	180 x 80 x 90
Wing (C)	172x91x43
Wing Tip (A)	172x61x43
Wing Tip (B)	172x61x43

A sinusoidal displacement $h = -h_0(1 - \cos 2\pi ft)$ with $h_0=0.175c$ is prescribed for the wing. Solutions are computed for different reduced frequencies, $k=0.5, 1.0, 1.82, 2.5, 3.5, 4.0$ for five cycles of plunging with a time step of $\Delta t = 0.001$. The time averaged thrust coefficient $\bar{C}_T$, power input coefficient $\bar{C}_{pw}$, and efficiency η are computed using the following expressions for each reduced frequency.

$$\bar{C}_T = \frac{2\bar{F}_{Ax}}{\rho U_s^2 S c} \quad (4.10)$$

$$\bar{C}_{pw} = \frac{2\bar{F}_{Ay}\dot{h}}{\rho U_s^3 S c} \quad (4.11)$$

$$\eta = \frac{\bar{C}_T}{\bar{C}_{pw}} \quad (4.12)$$

where the aerodynamic forces normal to, and along the freestream direction is represented by F_{Ay} and F_{Ax} respectively, and the freestream velocity U_s is taken as a reference velocity scale. The quantities S and c represent the span and chord of the wing respectively. Figures 4.18(a)-(c) depict the variation of thrust coefficient, power input coefficient and propulsive efficiency as a function of reduced frequency. A very good comparison is obtained with the results of Heathcote et al. [36] and Young [39]. An alternative set of computations were performed with an aspect ratio = 3 at $Re = 10000$ and $k=1.82$. Figure 4.19 shows a comparison of the thrust coefficient over two cycles of plunging with the experimental results of Heathcote et al. [36]. Visible differences exist at points when the stroke reverses ($t/T = 0.5, 1.0, 1.5$), however, on an average, a close agreement is obtained, indicating that the solutions are computed to a reasonable level of accuracy.

(a) Thrust Coefficient

(b) Power Input Coefficient

(c) Propulsive Efficiency

Figure 4.18: Comparison of Computed Aerodynamic Coefficients with Available Data , Adapted from Heathcote et al. [36]

Figure 4.19: Comparison of Computed Thrust Coefficients with Experiments from Heathcote et al. [36]

4.9. Rapid Accelerated Motion of a Wing

The autorotation of a missile, flutter of an aircraft wing or the rapid maneuvers that insects display are practical examples of unsteady flows involving rapid body movement. Accelerated wing motions especially in insects results in a large time lag between flow response and body motion. Added to this is the presence of strong attached vortices which result in high aerodynamic forces which enables the insect to maneuver rapidly. A computational test case relevant to the present discussion was studied by Lan and Sun [135] and is used in the present work for the purpose of verification. The work aims to quantify the aerodynamic forces obtained as a result of large wing accelerations through rotations. A wing of elliptical cross-section is accelerated from rest and then rotates with constant angular velocity as shown in Figure 4.20. The angle of attack of the wing is set to 35° and the wing tip that is rotating slower is located at a distance of 0.5 units from the origin of the coordinate system, which represents the point of rotation. Figure 4.21 shows the time history of lift and drag coefficients. The steady state lift and drag coefficients, $C_L = 1.08$, $C_D = 1.06$ are comparable to the computations of Lan and Sun [135] which are $C_L = 1.00$, $C_D = 1.01$. The higher lift and drag coefficients during the acceleration phase can be attributed to the presence of the dynamic stall vortex on the upper side of the wing. The fact that C_L and C_D do not show dependence with time when the wing is rotating with constant speed, was attributed to the stabilization of the dynamic stall vortex due to spanwise flow. An animation of the iso-surfaces of vorticity surrounding the wing can be found in Appendix A3.5.8.

Figure 4.20: Computational Setup for the Accelerating Wing

Figure 4.21: Time History of the Aerodynamic Force Coefficients for the Accelerating Wing at $Re = 100$

Chapter 5

Aerodynamic Characteristics of Rigid Flapping Wings

5.1. Introduction

What is well known from existing literature is that, when an airfoil or a wing executes periodic plunge or pitch oscillations with an incoming free-stream, above a critical reduced frequency, thrust is generated. It has been observed that the leading edge vortex is convected downstream and interacts with the trailing edge vortex and a reverse Karman vortex street is observed [38]. What is not quite well known is that, an airfoil or a wing plunged in zero free-stream conditions produces thrust only when there is asymmetry between the leading and trailing edges. Based on this observation, two test cases using different airfoil profiles, a symmetric NACA 0012 airfoil and a symmetric 12% thick elliptical airfoil are investigated in detail, to assess the role of leading edge-trailing edge asymmetry in thrust generation in two- and three-dimensions. Also, the trailing edge radius of a NACA 0012 airfoil is progressively increased to study its impact on thrust generation. For three-dimensional computations, the wing is set free to move under the action of aerodynamic forces with prescribed kinematics. During the course of investigation, it will also be demonstrated that a pitching airfoil / wing above a critical reduced frequency results in the reversal of the lift hysteresis curve. A mathematical model is also framed to support the observation. First, computations using prescribed kinematics or active flight computations will be discussed, followed by passive flight computations where the aerodynamic forces drive the airfoil/wing.

5.2. Active Flight Computations

5.2.1. Plunging Airfoil – Effect of L.E-T.E Asymmetry

The present analysis is carried out to investigate the effect of airfoil leading edge-trailing edge asymmetry on thrust generation in zero free-stream conditions. Two types of airfoils are considered. A NACA 0012 airfoil and a 12% thick elliptical airfoil is plunged with amplitude $h_0=0.175$, reduced frequency $k=5.71$ at a Reynolds number $Re=110$.

Figure 5.1: Vorticity Contours for a Plunging Airfoil in Zero Freestream Conditions

Figure 5.2 : Comparison of Drag Coefficients for a Plunging Elliptical and NACA 0012 Airfoil in Zero Freestream Conditions

As the airfoil plunges in zero free-stream conditions, the maximum plunge velocity is taken as reference velocity scale. Figures 5.1 (a)-(b) shows a comparison of the vorticity contours between a NACA 0012 airfoil and elliptical airfoil at the end of the plunge cycle after 10 cycles of oscillation. There clearly exists a visible difference in the vortex structures due to an asymmetry in the airfoil geometry. The Karman vortex street is not clearly visible for the elliptical airfoil and the thrust generated is almost non-existent as seen from Figure 5.2. Based on vortex dynamics, it is now explained why this vortex asymmetry leads to thrust generation. As mentioned in Chapter 1, the theory developed by Wu [75] is used to express the net aerodynamic force acting on a solid body as a function of vorticity moments and inertial forces. Similar ideas to explain lift generation in flapping airfoils had been attempted by Miller and Peskin [136]. Consider a region R_f occupied by an infinite viscous fluid bounded by R_∞ as shown in Figure 5.3 and assume that the solid body occupies a region R_b . According to Wu [75], the aerodynamic force acting on a two-dimensional object is given by

$$F = -\frac{d\alpha}{dt} + \rho_f \frac{d}{dt} \int_{R_b} u(r', t) dR \quad (5.1)$$

where $\mathbf{u}$ is the velocity of the solid body and α , the first moment of vorticity ϖ , given by

$$\alpha = \int_{R_\infty} \mathbf{r}' \times \varpi dR \quad (5.2)$$

Figure 5.3: Illustration of the Region Occupied by the Wing and the Fluid Bounded at Infinity

Since the focus is on understanding thrust generation, only the first component of Eq. (5.1) and Eq. (5.2) is considered. Moreover, only the Z-component of vorticity is taken since the wing flaps in the X-Y plane. This reduces Eq. (5.2) to

$$\alpha = \int_{R_\infty} y \varpi_z dR \quad (5.3)$$

During each half stroke, both positive and negative vortices are shed from the wing. Based on this fact, the vorticity moment in Eq. (5.3) is split into a sum of positive and negative vorticity moments as follows.

$$\begin{aligned} \alpha &= \int_{R_p} y |\varpi_{zp}| dR - \int_{R_n} y |\varpi_{zn}| dR \\ &= \alpha_p - \alpha_n \end{aligned} \quad (5.4)$$

where R_p and R_n denote regions of positive (ϖ_p) and negative (ϖ_n) vorticity respectively. To make sense of Eq. (5.4), the quantities y_{pv} and y_{nv} which are the ordinates of the geometric centre of the region R_p and R_n weighted by the vorticity are defined as follows:

$$y_{pv} = \frac{\int_{R_p} y |\varpi_{zp}| dR}{\int_{R_p} |\varpi_{zp}| dR}, \quad y_{nv} = \frac{\int_{R_n} y |\varpi_{zn}| dR}{\int_{R_n} |\varpi_{zn}| dR} \quad (5.5)$$

Differentiating Eq. (5.5) once w.r.t time and using Eq. (5.4), it can be shown that the rate of change of the first moment of positive/negative vorticity $\frac{d\alpha_{p/n}}{dt}$ is related to the ‘velocity’ of the positive and negative vortices as follows:

$$\begin{aligned} \dot{\alpha}_p &= \dot{y}_{pv} \int_{R_p} |\varpi_{zp}| dR + y_{pv} \frac{d}{dt} \int_{R_p} |\varpi_{zp}| dR \\ \dot{\alpha}_n &= \dot{y}_{nv} \int_{R_n} |\varpi_{zn}| dR + y_{nv} \frac{d}{dt} \int_{R_n} |\varpi_{zn}| dR \end{aligned} \quad (5.6)$$

Using the principle of conservation of total vorticity in R_∞ , it can be shown that

$$\int_{R_p} |\varpi_p| dR = \int_{R_n} |\varpi_n| dR \quad (5.7)$$

Using Eq. (5.6) and Eq. (5.7) in Eq. (5.1), the horizontal force acting on the body is given by

$$F_1 = -\rho_f \left[\left(\dot{y}_p - \dot{y}_n \right) \int_{R_p} |\varpi_{zp}| dR + \left(y_p - y_n \right) \frac{d}{dt} \int_{R_p} |\varpi_{zp}| dR \right] + \rho_f \frac{d}{dt} \int_{R_b} u(r, t) dR \quad (5.8)$$

$$= -\rho_f Q_v + \rho_f \frac{d}{dt} \int_{R_b} u(r, t) dR$$

In the above expression for the force, the first term Q_v represents the relative rate at which the vortices are convected away from the wing and the second term represents the inertial contribution of the displaced fluid. Since there is no horizontal motion, the horizontal component of the inertial term is absent and hence, thrust generation purely depends on the relative contribution of positive and negative vorticity convection rates. Thus it can be concluded that thrust is generated when $Q_v > 0$, i.e., when positive vorticity moves away from the wing at a faster rate than negative vorticity. This is further verified in Figure 5.4 by plotting the variation of Q_v over one cycle of oscillation and its corresponding mean value for both the elliptical airfoil and the NACA 0012 airfoil. It is observed that Q_v for the NACA 0012 airfoil is greater than that for the elliptical airfoil, hence the thrust generated by the NACA 0012 airfoil is greater than that for the elliptical airfoil.

Figure 5.4: Computed Variation of Q_v over one Cycle for the NACA 0012 Airfoil and the Elliptical Airfoil

5.2.2. Effect of Trailing-edge Topology on Thrust Generation

The effect of trailing edge topology on the performance of plunging airfoils is investigated. A base airfoil such as NACA 0012 with a finite trailing edge angle is considered. Assuming that $y = y(x)$ describes the shape of the base airfoil and R ($R \neq 0$) denotes the trailing edge roundness parameter [105], then the portion of the airfoil in the range $1-R \leq x \leq 1$ is altered according to $Y(x) = y(x) + \frac{x-(1-R)}{R} (R\sqrt{1-x} - y(x))$ where $Y(x)$ denotes the shape of the new airfoil with the altered trailing edge. Computations are carried out at a Reynolds number $Re = 500$, reduced frequency $k = 5.05$, 6.15 and plunge amplitude $h_0 = 0.2$, 0.06 . The Strouhal number in each of these cases is 1.01 and 0.37 respectively. In Sec 4.4, a plunging airfoil at a Strouhal number $St = 1.01$ was considered and the wake structures were compared with experiments. Hence, using the same Strouhal number, a comparison of wake structures is made between an airfoil having a blunt trailing edge with $R = 0.18$ and an airfoil having a sharp trailing edge with $R \sim 0$. For a Strouhal number of $St = 0.37$, computations are performed for $R = 0.08$, 0.12 , 0.15 and 0.18 and the corresponding thrust coefficients are studied. Figure 5.5 shows the airfoil profiles with different trailing edge radii.

For a Strouhal number $St = 1.01$, Figure 5.6(a) shows the computed airfoil wake structures at $t = 2.2s$. When compared with the wake structures in Figure 4.4(a)-(b), one can conclude that, by increasing the trailing edge radius to $R = 0.18$ (case 4), there is a reduction in the wake deflection angle by a small amount. Figure 5.6(b) shows a portion of the interpolation zone when the vortex is in the overlapping region. It can be seen that the contours are continuous across the overlap region confirming a proper implementation of the algorithm on the system of overlapping meshes. Instantaneous streamlines in the reference frame of the moving airfoil, for the original base airfoil with $R \sim 0$ as well as for the modified airfoil with $R = 0.18$ are shown in Figure 5.7 after the instant when the airfoil has just reached its maximum position at $t = 3.5$ s. For the base airfoil, it can be seen that the vortex at the trailing edge has already

Figure 5.5: Airfoil Trailing Edge Topology for Different Roundness Parameters

(a)

(b)

Figure 5.6: (a) Computed Wake Structures Behind a Plunging Airfoil (case 4) at $St = 1.01$
(b) The Overlap Region Showing the Interpolation Points around the Airfoil Mesh and Vorticity Contours

(a)

(b)

Figure 5.7: Instantaneous Streamlines for a Plunging Airfoil
(a) Base Airfoil (b) Airfoil case 4 at $t = 3.5$ s

separated and some amount of fluid has made its way to the upper side on the trailing edge. However for the modified airfoil, the trailing edge vortex is still attached to the airfoil giving rise to a larger effective body thickness. This results in increased drag or lower thrust.

For a Strouhal number $St=0.37$ and Reynolds number $Re=500$, instantaneous drag coefficients have been estimated for different trailing edge radii. Figure 5.8 shows the instantaneous drag coefficients for plunging in finite and zero freestream conditions. It is clearly observed that as the trailing edge radius is reduced, the drag curve shifts vertically downwards indicative of more thrust. The corresponding streamlines for the base airfoil and the airfoil of case 4 are shown in Figure 5.9 at the instant the airfoil is in the mean position and is moving upwards. These figures are consistent with the observation in Figure 5.7 that, effective body thickness increases the drag. The lift coefficient is not shown due to the fact that it remains almost symmetric during the upper and lower halves of the plunging cycle, giving rise to a zero time averaged lift coefficient, unlike for the case involving a dual mode vortex street as in Figure 4.5.

More computations are carried out to assess the effect of a solid wall or a ground, which is present near the plunging airfoil. The airfoil under consideration is placed at a location of 1.5 units above the lower boundary of the computational domain at $t=0$. It is observed that, plunging in ground effect either increases or decreases the thrust at selected points in the plunging cycle. Figure 5.10 shows the plunge displacement and velocity with time and Figure 5.11 shows a comparison of the drag coefficient with and without ground effect in the presence of a finite freestream flow. It is observed that drag increases with ground effect at end points of the cycle, i.e. when the airfoil crosses $h=0$ and $h=2h_0$, precisely during the upstroke. But during the downstroke, as the airfoil passes through its mean position ($h=h_0$), when velocity is maximum, the presence of the ground increases thrust. Time averaged velocity profiles downstream of the airfoil at a distance $0.4c$ from the trailing edge of the airfoil are plotted in Figure 5.12 for the base airfoil as well and the airfoil for case 4. These results show that the sharper T.E airfoil produces a jet with higher velocity and hence higher thrust.

(a)

(b)

Figure 5.8: Time History of the Drag Coefficient for Plunging Airfoils under (a) Finite and (b) Zero Freestream conditions for various T.E radii

(a)

(b)

Figure 5.9: Instantaneous Streamlines for a Plunging
(a) Base Airfoil (b) Case 4 Airfoil at its Mean Position

Figure 5.10 (a) Plunge Displacement (b) Plunge Velocity

Figure 5.11: A Comparison of the Time History of the Drag Coefficient With and Without Ground Effect

Figure 5.12: Time Averaged Velocity Profiles Downstream of the Airfoil Located at $0.4c$ aft the Trailing Edge for Plunging in (a) Zero Freestream (b) Finite Freestream conditions

5.2.3. Lift Hysteresis Reversal for a Rapidly Pitching Airfoil

Lift hysteresis curves, which express the variation of the lift coefficient C_L with the instantaneous angle of attack α , are obtained for a pitching NACA 0012 airfoil about the quarter chord position at a Reynolds number, $Re = 100$ for reduced frequencies $k=0.125, 0.25, 0.5, 1.0, 2.0, 3.0, 4.0$, and 5.0 . The airfoil is initially placed at zero angle of attack and the steady flow solution is computed. Using this solution as the initial condition, the airfoil executes pitch oscillations of the form $\alpha = \alpha_0 \sin(\omega t)$ with $\alpha_0 = 10^\circ$. After ten cycles of pitching, the hysteresis curves are plotted for different reduced frequencies. Figure 5.13(a) and Figure 5.13(b) shows these results for $k = 0.125-1.0$ and $k = 2.0-5.0$ respectively. For $k=0.125 - 1.0$, lift increases with increase in angle of attack with a certain amount phase lag and decreases with decrease in angle of attack. The major axis of the hysteresis curve lies on the first and the third quadrant. However for $k=2.0 - 5.0$, the hysteresis curves flip over such that its major axis lies on the second and fourth quadrant. In this condition, lift drops with increasing angle of attack vice-versa. This phenomenon has been observed before, only for airfoils pitching near the static stall angle. However in the present computation, the airfoil is pitched about a zero angle of attack. By analyzing the flow-field, and by framing a simple mathematical model, this phenomenon is now explained in detail.

The instantaneous angle of attack, defined with respect to free-stream conditions can be expressed as:

$$\alpha = \alpha_0 \sin(\omega t + \phi_2) \quad (5.9)$$

where α_0 is the maximum pitch angle, ω the angular frequency of oscillation and ϕ_2 an arbitrary phase angle.

(a)

(b)

Figure 5.13: Computed Lift Hysteresis Curves for a Pitching Airfoil at $Re = 100$ for Reduced Frequencies (a) $k = 0.125 - 1.0$ and (b) $k = 2.0-5.0$

Over a large range of numerical experiments conducted, and also from the observations of Sankar and Tang [137], the response of the flow to periodic changes in angle of attack can also be expressed in a similar form. For the lift coefficient, it can be expressed as:

$$C_L = A(k) \sin(\xi t + \phi_1(k)) \quad (5.10)$$

Normally $\xi = \omega$ for the ranges of reduced frequency and Reynolds number considered. However, in high Reynolds number flows, there may be a wide range of scales which might result in multiple harmonics in the lift history. But for the present analysis it is justified from the results that $\xi = \omega$. The amplitude A is the maximum lift coefficient

$C_{L,\max}$ for a given cycle and ϕ_1 is the phase associated with the response. It is to be noted that, these quantities are functions of the reduced frequency $k = \frac{\pi f c}{U_\infty}$. Eliminating t

from Eq. (5.9) and Eq. (5.10), the lift coefficient can be expressed as:

$$C_L = A(k) \sin\left(\phi_1(k) + \sin^{-1}\left(\frac{\alpha}{\alpha_0}\right) - \phi_2\right) \quad (5.11)$$

Without loss in generality, taking $\phi_2 = 0$ and differentiating Eq. (5.11) once with respect to α , the lift curve slope can be expressed as:

$$\frac{dC_L}{d\alpha} = \frac{A(k)}{\sqrt{\alpha_0^2 - \alpha^2}} \cos\left(\phi_1(k) + \sin^{-1}\left(\frac{\alpha}{\alpha_0}\right)\right) \quad (5.12)$$

There is a singularity at $\alpha = \alpha_0$ indicating a change in the lift curve slope at this point. Eq.

(5.11) describes an ellipse with semi-major and semi-minor axis a_m and b_m respectively

with a slope of $\tan 2\beta = 2 \cos \phi_1(k) \frac{\alpha_0 A(k)}{\alpha_0^2 - A(k)^2}$. Here a_m and b_m are related by the

relation $\frac{1}{a_m^2} + \frac{1}{b_m^2} = \frac{1}{\alpha_0^2} + \frac{1}{\alpha_0^2 \tan^2 \phi_1} + \frac{1}{A(k)^2 \sin^2 \phi_1}$. Eq. (5.12) is the starting point of

our analysis. Whether or not, the lift hysteresis will display a regular or a reversed loop depends on the phase ϕ_1 . From Eq. (5.12), the lift curve slope at $\alpha = 0$, the mean pitch angle is given by,

$$\frac{dC_L}{d\alpha} \bigg|_{\alpha=0} = \frac{A(k)}{\alpha_0} \cos(\phi_1(k) + n\pi) \quad (5.13)$$

and since $A(k)$ and α_0 are always positive, the following conditions can be derived:

For a regular hysteresis loop, which implies, lift increasing with angle of attack at the mean position:

$$\frac{dC_L}{d\alpha} > 0 \Rightarrow (4\zeta - 1)\frac{\pi}{2} - n\pi < \phi_1(k) < (4\zeta + 1)\frac{\pi}{2} - n\pi, \quad \zeta = 0, 1, 2, \dots; n = 0, 1, 2, \dots \quad (5.14)$$

For a reverse hysteresis loop, which implies, lift decreasing with angle of attack at the mean position:

$$\frac{dC_L}{d\alpha} < 0 \Rightarrow (4\zeta + 1)\frac{\pi}{2} - n\pi < \phi_1(k) < (4\zeta + 3)\frac{\pi}{2} - n\pi, \quad \zeta = 0, 1, 2, \dots; n = 0, 1, 2, \dots \quad (5.15)$$

These conditions are evaluated at time $t=0$ or when the airfoil is about to pitch up from the mean position. Taking the first mode ($\zeta = 0, n = 0$), the conditions for regular hysteresis are given by

$$-\frac{\pi}{2} < \phi_1(k) < \frac{\pi}{2} \quad (5.16)$$

and for reverse hysteresis,

$$\frac{\pi}{2} < \phi_1(k) < \frac{3\pi}{2} \quad (5.17)$$

At $\phi_1(k) = \pi/2$, a transition is observed with the lift hysteresis loop resembling an ellipse. If $\phi_1(k) = 0$, the lift curve slope is linear all through the cycle with a slope of $A(k)/\alpha_0$. In this condition, the response of the flow is very immediate. Since the time lag associated with the flow response is the major contribution to a delayed stall, this phenomenon will not occur if $\phi_1(k) = 0$. This state may also correspond to an airfoil pitching very slowly ($k \sim 0$) as it will be seen from the analysis. Figure 5.14 shows the various hysteresis loops obtained for different values of $\phi_1(k)$ with $\alpha_0 = 10$. These loops are quite similar to the ones obtained from the numerical computation in Figure 5.13. For each value of reduced frequency k , the phase ϕ_1 is determined. The values of reduced frequency and the phase ϕ_1 are summarized in Table 5.1.

Figure 5.14: Hysteresis Loops of C_L vs. α Computed from the Mathematical Model for Different Phase Lags

Table 5.1: Reduced Frequency (k) and Phase ϕ_1 for Pitching at $Re = 100$

Reduced frequency (k)	Phase ϕ_1
0.25	0.1π
0.5	0.15π
1.0	0.28π
2.0	0.55π
5.0	0.7π

The hysteresis curves corresponding to values of ϕ_1 obtained through numerical computations agrees well with the conditions postulated by Eq. (5.16) and Eq. (5.17). It is seen that the transition from regular to reverse hysteresis takes place in the range $1 < k < 2$. Since it is known that the critical phase angle for this transition is 0.5π , the critical reduced frequency can be estimated as $k = 1.6$ from Table 5.1 using a simple curve fit. The key point is that, if the lift response while pitched at zero mean angle of attack has the same frequency as that of the forcing function, then reverse hysteresis is bound to occur by the conditions postulated by Eq. (5.17) depending on the Reynolds number. The variation of the phase ϕ_1 with k depends on Reynolds number and hence the transition k changes. Nevertheless, it's not surprising that the transition k is associated with the phase $\phi_1(k) = \pi/2$. The question at this stage that needs to be addressed is, why the lift coefficient decreases with an increase in the angle of attack as opposed to conventional steady flow theory or what is responsible for the phase lag at a given reduced frequency? To answer this question partly, the pressure and vorticity fields are examined at different stages of pitching. Considering the results pertaining to the case when $k=0.25$ and shown in Figure 5.15(a), it is clear that the lift lags the angle of attack by a very small phase. At point A when the airfoil is at zero angle of attack, the lift is almost zero (<0) like the steady flow field shown in Figure 5.16(a). A decrease in angle of attack brings it to point B and then to point C. During this process the suction region in the lower surface near the leading edge of the airfoil is responsible for the drop in the lift as shown in Figures 5.16(b)-(c). At points D and E, when the angle of attack is increasing, the suction region is present on the upper surface of the airfoil and the lift increases as shown in Figures 5.17(a)-(b). The vorticity contour plots do not predict vortex shedding close to the airfoil at this frequency. This is also evident from the corresponding animation reported in Appendix A3.5.1. For this reduced frequency, it can be concluded that the flow responds almost instantly, to a change in angle of attack, with a small phase lag. It is also hypothesized that vortex shedding has a minor impact on the aerodynamic forces. The effect of vortex shedding will be dominant when the airfoil is pitched at higher reduced frequencies. For $k=2$ from Figure 5.15(b), it is clear that the lift lags the angle of attack by an amount which is greater than that for $k=0.25$. Consider the effect of increasing the

angle of attack as the airfoil passes through points A and B. At point A, the lift is positive. An increase in angle of attack causes a strong counter-clockwise vortex to be shed from the trailing edge as seen in Figures 5.18(d)-(e). This vortex induces a local velocity field thereby increasing the pressure on the upper surface marginally. This is then accompanied by a loss in lift. Increasing the angle of attack to point C and then decreasing it to point D as shown in Figure 5.18(f) and Figure 5.19(c), the lift however keeps reducing due to the formation of a strong pressure field near the trailing edge of the airfoil on the upper side which was set up by the vortex. At this point the airfoil has already reversed its motion and the angle of attack is decreasing. The strength of the suction region near the nose in the lower side of the airfoil is offset by another clockwise vortex shown in Figure 5.19(d) shed from the trailing edge. The pressures induced by this vortex will create a lifting force on the airfoil. Thus for pitching down, the lift increases. This phenomena is clearly observed when pitched at $k=5$. Figure 5.15(c) shows the pitching motion and the corresponding lift coefficient for $k=5$. There is a large amount of lag between the lift and the angle of attack. Considering the point A when the angle of attack is increasing, the lift coefficient is positive at this point. Increasing the angle of attack to the point B causes a strong counter-clockwise vortex to form on the upper side near the trailing edge of the airfoil as shown in Figures 5.20(d)-(e). The extra vortex which was produced from the previous oscillation as seen in figure is absent for $k=0.25$ and is hardly visible for $k=2.0$. Instead of causing the lift to increase, the pressures induced on the upper side causes it to drop despite the growth of the leading edge vortex. At point C, the vortex at the trailing edge is about to “spill” while maintaining a high pressure in the vicinity of the trailing edge on the upper surface of the airfoil as shown in Figure 5.20(f). At points D and E, as the angle of attack starts to drop, i.e. the airfoil starts pitching down, the clockwise vortex shed from the trailing edge induces a strong pressure field on the lower surface of the airfoil, causing the lift to increase as shown in Figures 5.21(c)-(d). Hence the presence of the trailing edge vortex in the vicinity of the airfoil for longer periods of time without being swept by the freestream at high reduced frequencies causes the reversal of lift hysteresis loop. The corresponding animation for $k=5$ can be found in Appendix A3.5.2. At low reduced frequencies, the vortices from the trailing edge are swept away before the airfoil motion reverses.

(a) $f=0.08$ Hz, $k=0.25$, $\Delta\theta = 0.1\pi$

(b) $f=0.64$ Hz, $k=2.0$, $\Delta\theta = 0.556\pi$

(c) $f=1.6$ Hz, $k=5.0$, $\Delta\theta = 0.704\pi$

Figure 5.15: Lift Coefficient and Angle of Attack during the Pitching Cycle for Various Reduced Frequencies

Figure 5.16 (a)-(c)Pressure and (d)-(f)Vorticity Contours for Reduced Frequency $k = 0.25$ at Positions A, B, C in the Oscillation Cycle as Indicated in Figure 5.17(a)

Figure 5.17: (a)-(b) Pressure and (c)-(d) Vorticity Contours for Reduced Frequency $k = 0.25$ at Positions D, E in the Oscillation Cycle as Indicated in Figure 5.17(a)

Figure 5.18 (a)-(c) Pressure and (d)-(f) Vorticity Contours for Reduced Frequency $k = 2.0$ at Positions A, B, C in the Oscillation Cycle as Indicated in Figure 5.17(b)

Figure 5.19 (a)-(b) Pressure and (c)-(d) Vorticity Contours for Reduced Frequency $k = 2.0$ at Positions D, E in the Oscillation Cycle as Indicated in Figure 5.17(b)

Figure 5.20 (a)-(c) Pressure and (d)-(f) Vorticity Contours for Reduced Frequency $k = 5.0$ at Positions A, B, C in the Oscillation Cycle as Indicated in Figure 5.17(c)

Figure 5.21: (a)-(b) Pressure and (c)-(d) Vorticity Contours for Reduced Frequency $k = 5.0$ at Positions D, E in the Oscillation Cycle as Indicated in Figure 5.17(c)

5.2.4. Reverse Hysteresis for a Pitching Elliptic Airfoil and Wing of Elliptic Cross-Section

To confirm whether asymmetry between the leading and trailing edges influence the onset of reverse hysteresis, numerical experiments on elliptical airfoils and wings have been computed for a reduced frequency $k=5$ and Reynolds number $Re = 100$ on meshes identical to the ones described in Chapter 2. For two-dimensional pitching, the mesh in Figure 2.13 is regenerated for an ellipse. For three-dimensional pitching, a wing with aspect ratio $=3$, similar to Figure 2.19 is considered and the mesh consisting of 1.38 million points is generated. Figure 5.22 shows a comparison of the lift hysteresis curves for the two- and three-dimensional cases. It can be seen that both the lift curves exhibit reverse hysteresis.

Figure 5.23 compares the variation of the aerodynamic force coefficients with time for two- and three-dimensional pitching. It can be seen that the three-dimensional lift coefficient is less than its two-dimensional counterpart during pitch down and higher during pitch up. This can be readily explained by observing the surface pressure distribution on the wing at an arbitrary time $t = 1.7$ s when the wing is pitching up 9.05 degrees and about to reach its maximum angular displacement as shown in Figure 5.24. In two- and three-dimensions, the pressure on the upper side towards the trailing edge is high, and low on the corresponding lower side. This is clearly indicative of negative lift which is consistent with Figure 5.22. However in three-dimensions, the two wing tips act as a source of three-dimensional flow, which are associated with wing tip vortices, which then, reduce the pressure marginally on the upper surface towards the wing tip regions. However, this effect is absent in the two-dimensional case and thus the lift coefficient is higher during this phase in three-dimensions. A similar explanation can be given for the case of the wing pitching down. The evolution of the trailing edge vortices from the wing can be seen in Appendix A3.5.3

Figure 5.22: Lift Coefficient per Unit Span for a Pitching Wing in Comparison with a Pitching Airfoil for a Reduced Frequency $k = 5$

(a) C_L

(b) C_D

Figure 5.23: Aerodynamic Force Coefficients per Unit Span for a Pitching Wing in Comparison with a Pitching Airfoil for a Reduced Frequency $k = 5$

Figure 5.24: (a), (b) Pressure Distributions at Mid-Span for a Pitching Wing for Reduced Frequency $k = 5$ (c) Corresponding Two-Dimensional Case

5.3. Passive Flight Computations

5.3.1. Effect of Trailing Edge Sharpness on Trajectory of a Flapping Airfoil

Numerical computations on a NACA 0012 airfoil, with different trailing edge radii were discussed in Sec 5.2.2. It was demonstrated that airfoils having sharper trailing edges and round leading edges produced higher thrust. In this section, the motion of the airfoil is relaxed, by allowing it to move under the influence of the surrounding aerodynamic forces. The Navier-Stokes equations are coupled with the dynamical equations of motion for a rigid body as described in Chapter 3. Gravity is not employed in all these test cases since the effect of plunging on thrust/lift production is studied. The Reynolds number for these cases varied from $Re = 2000$ -4500 based on the forward speed. The airfoil which is initially located at $0.75c$ units above the ground ($y = -0.75$), is plunged at a frequency of $f = 0.979$ Hz with an amplitude of $h_0 = 0.12$, and follows a sinusoidal displacement $h = h_0 \sin(2\pi ft)$ in a reference frame fixed to the moving airfoil. The mesh system consists of a mesh surrounding the airfoil overlapping a background Cartesian mesh as discussed previously in Chapter 2. The mesh surrounding the airfoil moves rigidly with the airfoil and the background Cartesian mesh is held fixed. As the airfoil plunges, the rigid body motion is simultaneously determined at each time step from the forces using Eq. (3.20).

Figure 5.25 shows the computed streamlines in the inertial frame of reference for $R = 0.08$ and 0.12 at times $t = 6, 12$ seconds respectively. Figure 5.26 shows the path of the airfoil and forward speed attained for different trailing edge roundness parameters. As a result of thrust generation, it can be seen that the airfoil takes off and travels in the negative X -direction. The periodic behavior in the trajectory is obtained approximately after 10 cycles. It is seen that with decreasing roundness, more thrust is generated and the airfoil travels faster. It is also observed that the airfoil plunges to a higher altitude to take up a stable position with increasing trailing edge sharpness due to the fact that, the initial C_{Lmax} for sharper trailing edges was found to be relatively higher during the first cycle when compared to airfoils having blunt trailing edges. This could be possible due to the presence of the ground. These initial transients are clearly observed in the simulations shown in Appendix A3.5.4-A3.5.6. It is unknown however whether these initial transients are physical or numerical.

Figure 5.25: Streamlines w.r.t the Inertial Frame for Passive Flight at Various Time Instants and for Different Cases

(a)

(b)

Figure 5.26: (a) Trajectory of the Airfoil and (b) Forward speed for Different Trailing-Edge Roundness Parameters

5.3.2. Effect of Wing Leading-edge/Trailing-edge asymmetry and asymmetric kinematics on Forward Flight

The analysis in Sec 5.2.1 provided insight into the effect of leading-edge/trailing-edge asymmetry and vortex asymmetry on thrust. This analysis is now extended to predict aerodynamic forces and trajectories on a free flapping wing. An elliptical airfoil and a NACA airfoil of 12% thickness are considered as suitable cross-section profiles for a wing of aspect ratio 2. Composite overlapping meshes are generated surrounding the wing as shown in Figure 2.19 and Figure 2.20. The masses of both elliptical and NACA wing are assumed to be 1.20. A rotational motion of the form $\theta_i = -\theta_0 (1 - \cos 2\pi ft)$ is specified about a co-ordinate axis i as explained in Figure 5.27. Using this kinematics, three different modes of rotation are investigated to study the effect of independent and coupled rotations on forward flight at a reduced frequency $k = 1.43$, Strouhal number $St = 0.49$ and Reynolds number $Re = 329$.

For independent rotations, the wing is rotated only about a single axis (X- or Z- axes) whereas, for coupled rotations, at least two simultaneous rotations (X- and Z- axes) are performed. At all times, $t \geq 0$, the wing flaps with no incoming free-stream and the aerodynamic forces are integrated to obtain the position of the wing. For all cases considered here, the wing is constrained to move along the X- axis and the solutions are computed till the wing reaches the boundary of the inner Cartesian grid, which is marked by the inner cube in Figure 2.20. A comparison of the thrust coefficients for the elliptical cross section wing and NACA 0012 wing for rotations about different axes are shown in Figures 5.28(a), Figure 5.28(c) and Figure 5.28(e) and the corresponding trajectories in Figures 5.28(b), Figure 5.28(d) and Figure 5.28(f). For rotation about the X- or Z-axes as in Figures 5.28(a)-(d), it is observed that the thrust coefficients are higher for the NACA 0012 wing when compared with the elliptical cross-section wing and hence the NACA 0012 wing accelerates faster than the elliptical cross-section wing. This clearly indicates that asymmetry between the leading and trailing edge of the wing plays a significant role in thrust generation. For simultaneous rotation about X- and Z-axes as in Figures 5.28(e)-(f),

Figure 5.27: (a) Wing Kinematics About Various Co-ordinate Axes and the
(b) Corresponding View in the XY Plane

Figure 5.28: Comparison of Computed Thrust Coefficients and Trajectories for a NACA 0012 Wing with an Elliptical Cross-Section Wing for Flapping about X-axis (a)-(b) X-axis (c)-(d) Z-axis (e)-(f) X- and Z-axes

it is observed that both the NACA 0012 wing and the elliptical cross-section wing produce similar characteristics and they accelerate with almost the same speed. Comparing Figure 5.28(e) with Figure 5.28(a), we can also infer that the simultaneous rotation mode produces larger amounts of thrust compared to rotation about a single axis. This draws us to the conclusion that thrust can be generated through proper (a) geometric asymmetry i.e., when there is asymmetry between the leading and trailing edges of the wing and (b) coupled kinematics i.e., when there is a coupled symmetric motion of the wing. For both the above mentioned cases, the vortex shedding patterns are investigated. Figure 5.29 and Figure 5.30 show the Z-vorticity component for the NACA 0012 wing and the elliptical cross-section wing respectively. Two different time instances are considered corresponding to when the wing is at the start of the as cycle as in Figure 5.29(a),(c),(e) and Figure 5.30(a),(c),(e) and when it has completed half cycle as in Figure 5.29(b),(d),(f) and Figure 5.30(b),(d),(f). When there is a rotational motion either about the X- or Z- axes, the vortex structures generated from the trailing edge are symmetric with respect to that generated at the leading edge for an elliptical cross-section wing, whereas they are asymmetric for the NACA 0012 wing. However for the coupled rotation mode, both types of wings produce asymmetric vortex structures. Hence it can be concluded that the presence of geometrical asymmetry and/or coupled kinematics can influence the structure of the shed vortices which in turn can lead to thrust generation.

Using vorticity moments as dealt in Sec 5.2.1, the role of asymmetry in thrust generation is explained. Before that, we prove the principle of total vorticity conservation given by Eq. (5.7) for all modes of rotation, for both the NACA 0012 cross-section wing and the elliptical cross-section wing. Figure 5.31(a)-(b) shows the variation of normalized positive and negative Z-component of vorticity over one cycle of flapping for rotation about the X-axis (X), Z-axis (Z), X- and Z- axes (XZ). Only the Z-component is considered since the flapping takes place in the XY plane. It can be seen that both positive and negative vorticity overlap for all rotation modes. This proves that the net vorticity is conserved, and hence the computations are being performed accurately. For rotation about X-axis as seen in Figure 5.32(a), the mean value of Q_v from Eq. (5.8) for the NACA 0012 wing over one cycle of oscillation is greater than that for the elliptical cross-section wing. Thrust is generated as a result of geometric asymmetry.

Figure 5.29: Computed Vorticity Contours at the Beginning of the Cycle ($t = T$) and at Half Cycle ($t = T + \frac{1}{2} T$) for a Flapping NACA 0012 Cross-Section Wing for Rotation about (a)–(b) X-axis, (c)–(d) Z- axis (e)–(f) X- and Z-axes, (T being the Period of one Oscillation)

Figure 5.30: Computed Vorticity Contours at the Beginning of the Cycle ($t = T$) and at Half Cycle ($t = T + \frac{1}{2} T$) for a Flapping Elliptical Cross-Section Wing for Rotation about (a) –(b) X-axis, (c)-(d) Z- axis (e)-(f) X- and Z-axes, (T being the Period of One Oscillation)

A similar situation occurs for rotation about Z-axis as in Figure 5.32(b). However Q_v is found to be two orders of magnitude smaller than the previous case. This is because both positive and negative vortices are convected away from the wing at almost the same rate. For simultaneous rotation about X- and Z- axes as in Figure 5.32(c), the level of asymmetry created in the vortical structures are similar for the elliptical cross-section wing and the NACA 0012 wing, with the elliptical cross-section wing having a higher relative vorticity convection rate than the NACA 0012 wing. The thrust generated by the elliptical cross-section wing is higher than that generated by the NACA 0012 wing as a result, the former travels faster than the latter. This is also consistent from Figure 5.28(e)-(f) where the elliptical cross-section wing has a higher time averaged thrust coefficient and travels faster than the NACA 0012 cross-section wing.

(a) Wing with NACA 0012 Cross-Section

(b) Wing with Elliptical Cross-Section

Figure 5.31: Computed Variation of Normalized Positive and Negative Vorticity with Time for Rotation about Different Axes.

Figure 5.32: Computed Variation of Q_v over one Cycle for Rotation about Different Axes for the NACA 0012 Cross-Section Wing and the Elliptical Cross-Section Wing

Chapter 6

Computational Structural Dynamics Modeling for Fluid-Structure Interaction

The flapping motion of the wing results in considerable traverse deflection along the spanwise direction. When the wing is thin, this deflection can be modeled using the classical thin plate theory of Kirchhoff under the following assumptions, namely (i) the plate is elastic, homogenous and isotropic, (ii) the deflection of the mid-plane of the plate is small compared with the thickness of the plate which results in the slope and square of the slope to be small quantities, (iii) straight lines which are initially normal to the middle plane prior to bending remain straight and normal after bending, as a result, the normal shear strains vanish, and (iv) the mid-plane is unstrained after bending as stated in Ventsel and Krauthammer [138]. Under these assumptions, the actual three-dimensional problem is reduced to a two-dimensional problem and the equation governing the deflection of a plate is given by a fourth order partial differential equation as follows:

$$m_x \frac{\partial^2 y_s}{\partial t^2} + D \nabla^4 y_s = q(x, z, t), \quad D = \frac{Eh^3}{12(1-\nu_p^2)} \quad (6.1)$$

where E is the modulus of elasticity, ν_p the Poisson's ratio, m_x is the mass per unit area and the distributed load $q(x, z, t) = \Pi_u n_u + \Pi_l n_l$, Π being the stress tensor $\Pi = -p\delta_{ij} + \tau_{ij}$ and n_u and n_l are the normals to the boundary on the upper and lower surface of the wing respectively. For two-dimensional flow problems, i.e., for flow past a flexible airfoil, the structural dynamics model of Eq. (6.1) can be reduced further to the well known Euler-Bernoulli beam equation in one dimension as follows:

$$m_x \frac{\partial^2 y_s}{\partial t^2} + EI \frac{\partial^4 y_s}{\partial x^4} = q(x, t) \quad (6.2)$$

where I is the moment of inertia of the plate cross-section. It is to be noted that the thin plate assumption results in a drastic simplification of the actual physical model and can be applied when $8 \leq \frac{L}{h_t} \leq 100$ as mentioned in Ventsel and Krauthammer [138].

Although the lower limit of this condition cannot be violated, the upper limit can be as high as 200 for the beam theory to be applicable. For $L/h_t > 200$, a membrane model would be the most appropriate choice, in which case, the governing equation reduces to a second order partial differential equation as described by Stanford et al. [60].

6.1. Boundary Conditions

The fourth order PDE given by Eq. (6.1) and Eq. (6.2) requires the specification of two boundary conditions per boundary point. For the problems investigated in the present work, two types of boundary points namely, (i) fixed and (ii) free boundary point are considered. For the plate equation, i.e. Eq. (6.1), at a fixed boundary point, the boundary conditions are given by

$$y_s = y_{sp} \quad \text{and} \quad \frac{\partial y_s}{\partial n} = 0 \quad (6.3a)$$

where n is in the direction normal to the edge of the plate under consideration. For the Euler-Bernoulli beam equation i.e., Eq. (6.2), at a fixed boundary point, the deflection of the beam is specified and the slope is zero:

$$y_s = y_{sp} \quad \text{and} \quad \frac{\partial y_s}{\partial x} = 0 \quad (6.3b)$$

where y_{sp} is a specified displacement. When the airfoil as shown in Figure 2.16 executes plunge oscillations while allowing the flexible tail to deform passively, the governing equations Eq. (6.1) and Eq. (6.2) are solved in the reference frame moving with the body, and at a fixed boundary node, y_s is set to zero. At a free boundary point, the moment and shear stress vanish. For the plate equation,

$$M_i = EI \left(\frac{\partial^2 y_s}{\partial x_i^2} + \nu_p \frac{\partial^2 y_s}{\partial x_j^2} \right) = 0 \quad \text{and} \quad V_i = -EI \left(\frac{\partial^3 y_s}{\partial x_i^3} + (2 - \nu_p) \frac{\partial^3 y_s}{\partial x_i \partial x_j^2} \right) = 0 \quad (6.4a)$$

where i, j are normal to, and along the edges of the plate respectively, and for the Euler-Bernoulli beam equation, the free boundary point satisfies

$$M_x = EI \frac{\partial^2 y_s}{\partial x^2} = 0 \quad \text{and} \quad V_x = -EI \frac{\partial^3 y_s}{\partial x^3} = 0 \quad (6.4b)$$

6.2. Discretization

For the discretization of the Euler-Bernoulli beam equation, consider a flexible airfoil which is modeled by a beam of finite length L and discretized into N points such that the x -coordinate is given by $x=ih$ where $i = 0.1.2....N-1$, $h = L/(N-1)$ as in Figure 6.1(a). At any point $x=ih$ excluding the left boundary, Eq. (6.2) is discretized using an implicit backward difference method in time and a central difference in space as follows:

$$m_x \frac{y_i^{n+1} - 2y_i^n + y_i^{n-1}}{\Delta t^2} + EI \frac{y_{i+2}^{n+1} - 4y_{i+1}^{n+1} + 6y_i^{n+1} - 4y_{i-1}^{n+1} + y_{i-2}^{n+1}}{\Delta x^4} = q(x, t)^{n+1} \quad (6.5a)$$

For the Kirchhoff plate equation, the flexible wing is modeled using the thin plate assumption as in Figure 6.1(b). Using the notation in Figure 6.1(b) to denote the various grid points in the domain, and utilizing a similar discretization procedure as that of the Euler-Bernoulli beam equation, one can arrive at the following discretized form of the static part $\nabla^4 y_s$ of the plate equation.

$$\nabla^4 y_s = \sum_{i=1}^{25} A_i y_{is} \quad (6.5b)$$

where the coefficients A_i , are constants that depend on the mesh spacing. By adding the inertial term to Eq. 6.5(b), the final set of discretized equations is given by

$$\left(I + \frac{\Delta t^2 D}{m_x} [A] \right) y_s^{n+1} = \frac{\Delta t^2 q(x, z, t)^{n+1}}{m_x} + 2y_s^n - y_s^{n-1} \quad (6.5c)$$

where I is an identity matrix. Eq. 6.5(a) and Eq. 6.5(c) leads to a system of linear equations of the form $[W] y_s^{n+1} = f(y_s^n, q^{n+1})$ which is of the order $O(\Delta t, \Delta x^2)$. The first order discretization in time, as it will be seen adds a small amount of numerical damping to Eq. (6.2) for the time steps used in the computation. Since the matrix W does not depend on any of the flow properties, its inverse needs to be computed just once and stored in memory. At each time step, a matrix vector multiplication is then performed to obtain the plate displacements y_s . The load vector $q(x, z, t)$ which is defined at time level $n+1$ is generally not available at the beginning of each time step for fluid-structure interaction problems, hence Eq. 6.5(c) is solved iteratively with proper under-relaxation, the details of which are highlighted in Sec 6.4.

(a)

(b)

Figure 6.1: Discretization of (a) Euler Bernoulli Beam Equation using a 5 Point Stencil and (b) Kirchhoff Plate Equation using a 25 Point Stencil

6.2.1. Discretization of Boundary Conditions

Fixed Boundary Point : For the Euler-Bernoulli beam equation, the Dirichlet boundary condition $y_s=y_{sp}$ is imposed by blanking the matrix entry corresponding to the first point to unity and setting the right hand side of the discretized equation to y_{sp} . The boundary condition $\frac{\partial y_s}{\partial x}=0$ is discretized using a second order central difference expression as follows:

$$\frac{y_1 - y_{-1}}{2\Delta x} = 0 \quad (6.6)$$

$$\Rightarrow y_{-1} = y_1 \quad (6.7)$$

Using the above two boundary conditions, the grid point y_1 can be discretized easily using Eq. 6.3(b). For the Kirchhoff plate equation, the Dirichlet boundary condition $y_s=y_{sp}$ and the Neumann boundary condition $\frac{\partial y_s}{\partial n}=0$ are imposed on a given edge.

Free Boundary Point: Discretizing the Euler-Bernoulli beam equation at point y_{N-1} as shown in Figure 6.1(a) requires the deflection at two ghost points y_N and y_{N+1} which can be obtained by discretizing Eq. 6.4(b) at y_{N-1} .

$$\frac{\partial^2 y_s}{\partial x^2} = \frac{y_N - 2y_{N-1} + y_{N-2}}{\Delta x^2} = 0, \Rightarrow y_N = 2y_{N-1} - y_{N-2} \quad (6.8a)$$

$$\frac{\partial^3 y_s}{\partial x^3} = \frac{y_{N+1} - 2y_N + 2y_{N-2} - y_{N-3}}{2\Delta x^3}, \Rightarrow y_{N+1} = 4y_{N-1} - 4y_{N-2} + y_{N-3} \quad (6.8b)$$

Similarly for the Kirchhoff plate equation, discretization on a boundary point G as shown in Figure 6.1(b) requires the ghost points D, H, L and I to be known. The solution at point H and I can be obtained by discretizing the boundary condition Eq. 6.4(a) at point G .

$$\frac{\partial^2 y_s}{\partial x_i^2} + \nu_p \frac{\partial^2 y_s}{\partial x_j^2} = 0 \Rightarrow y_{sH} = [\alpha \ -1 \ 2-2\alpha \ \alpha] \begin{bmatrix} y_{sC} \\ y_{sF} \\ y_{sG} \\ y_{sK} \end{bmatrix}, \quad \alpha = -\nu_p \frac{h_i^2}{h_j^2} \quad (6.9a)$$

$$\frac{\partial^3 y_s}{\partial x_i^3} + (2 - \nu_p) \frac{\partial^3 y_s}{\partial x_i \partial x_j^2} = 0 \Rightarrow y_{sl} = [\beta \quad -\beta \quad 1 \quad -2(1+\beta) \quad 2(1+\beta) \quad \beta \quad -\beta] \begin{bmatrix} y_{sB} \\ y_{sD} \\ y_{sE} \\ y_{sF} \\ y_{sH} \\ y_{sJ} \\ y_{sL} \end{bmatrix}, \quad \beta = (2 - \nu_p) \frac{h_i^2}{h_j^2} \quad (6.9b)$$

where h_i, h_j denote the mesh spacing along directions i, j respectively. Similarly the solution at points D and L can be obtained. The above set of equations, are then assembled in the matrix which arises out of the discretization of Eq. 6.5(c) at point G .

6.3. Spatial and Temporal Accuracy of the Discretized Euler-Bernoulli Beam Equation

A straightforward analysis can prove that the discretized Euler-Bernoulli beam equation Eq. 6.5(a), is first order accurate in time and second order accurate in space. With reference to spatial accuracy, for a cantilever beam of unit length vibrating in its natural mode, a very minor reduction of about 0.02% in amplitude and 0.05 rad in phase was observed when the number of points on the beam was reduced from 201 to 31. Hence 31 points was deemed sufficient to capture the mode shapes corresponding to a given time instant. However with reference to time accuracy, the discretization is only first order accurate and adds numerical damping which might be comparable to the aerodynamic damping which is implicit in Eq. 6.5(a). A modified equation analysis is performed and it is inferred that solving Eq. 6.2 numerically using the discretization represented in Eq. 7.5(a), is akin to solving the following PDE exactly.

$$m_x \frac{\partial^2 y_s}{\partial t^2} + EI \frac{\partial^4 y_s}{\partial x^4} = \Delta t m_x \frac{\partial^3 y_s}{\partial t^3} + q(x, t) \quad (6.10)$$

Taking $q(x, t) = 0$ for a beam vibrating in its natural mode and using the method of separation of variables, one can substitute $y_s(x, t) = \psi(t)\Phi(x)$ to decouple Eq. (6.10) into the following two ODE's.

$$\Delta t \frac{d^3 \psi}{dt^3} - \frac{d^2 \psi}{dt^2} - \tilde{\omega}^2 \psi = 0, \quad \frac{d^2 \Phi}{dx^2} - \frac{m_x \Phi \tilde{\omega}^2}{EI} = 0 \quad (6.11)$$

Here $\tilde{\omega}$ is the undamped natural frequency of the beam which is obtained by solving the Eigen value problem in Φ with appropriate boundary conditions, the details of which can be found in Voltera [139]. The Eigen functions $\Phi(x)$ represent only mode shapes and hence are not relevant in the discussion of temporal accuracy. Eq. (6.11) can be solved for $\psi(t)$ and the solution is given by

$$\psi(t) = \exp(\sigma_1 t) (c_1 \cos \sigma_2 t + c_2 \sin \sigma_2 t) \quad (6.12)$$

Here σ_2 is the damped natural frequency. At time $t=0$, $\psi=0$. Hence, the constant $c_1=0$. The ratio of the amplitudes between an arbitrary time $t=t_1$ and $t=t_1+2\pi/\sigma_2$ over one cycle of oscillation is given by

$$\delta = \frac{\psi(t+2\pi/\sigma_2)}{\psi(t)} = \exp\left(2\pi \frac{\sigma_1}{\sigma_2}\right) \quad (6.13)$$

where σ_1 and σ_2 are given by

$$\sigma_1 = \frac{1}{12\Delta t} \left(-\beta - \frac{4}{\beta} + 4 \right), \quad \sigma_2 = \frac{\sqrt{3}}{12\Delta t} \left(\beta - \frac{4}{\beta} \right), \quad \beta^3 = 8 + 108\tilde{\omega}^2 \Delta t^2 + 12\sqrt{3}\tilde{\omega}\Delta t \sqrt{27\tilde{\omega}^2 \Delta t^2 + 4} \quad (6.14)$$

For a plate of thickness $h_f=0.003$, chord $c=1$, $E=2 \times 10^{10}$, the amplitude drop δ is computed for different time steps. Similar geometric parameters were used by Shin et al. [80] to study the dynamics of a thin flexible plate in a fluid. From a wide range of time steps, it was concluded that an optimum time step $\Delta t=10^{-3}$ can be used since the amplitude drop $\delta \sim 0.983$ and $\sigma_2/\omega \sim 0.999$. Moreover at this time step, the higher modes of oscillation do not appear which makes it favorable for the fluid flow computation as higher modes result in very high frequencies and might render the fluid flow computation unstable. Figure 6.2 shows a portion of the tip response of a thin plate for the above mentioned geometrical parameters with a periodic external force for different time steps. It is clearly observed that smaller time steps capture higher modes of oscillation.

Figure 6.2: Effect of Time Step on Tip Displacement for Forced Vibration Using the Euler-Bernoulli Beam Equation

6.4. Coupling of Fluid and Structural Solvers

The coupling of fluid and structural solvers follows the standard *Dirichlet-Neumann* approach described in Causin et al. [76] and Kuttler and Wall [140] where the fluid dynamic equations are first solved and the computed loads are transferred to the structural equations. If only one fluid and structural solution is sought per time step, the coupling is explicit and weakly coupled. An implicit and strong coupling results if multiple fluid and structural solutions are performed iteratively within a time step. The following is a summary of the coupling procedure:

1. An implicit predictor is used to discretize the momentum equation as follows:

$$\frac{U^p - U^n}{\Delta t} = \frac{3}{2} X^n - \frac{1}{2} X^{n-1} + \frac{1}{2} (IU^{n+1} + IU^n), \quad X = -(u \cdot \nabla) u - \nabla p, \quad IU = \nu \Delta u \quad (6.15)$$

2. Solve for pressure based on predicted velocities:

$$\tilde{\Delta} p^p + \rho \sum_i \tilde{\nabla} u_i^p \cdot \frac{\partial u^p}{\partial x_i} = 0 \quad (6.16)$$

3. Using the predicted pressures which are computed on the flow solver grid, the net load that is to be imposed on the structure, i.e., $\Pi = -p\delta_{ij} + \tau_{ij}$ is computed at each grid point on the flow solver grid. However, the grid points on the flow solver grid and the structural grid do not match. Hence the loads from the flow solver grid are transferred to the structural grid by interpolation. For the case of the plunging airfoil with a flexible tail as in Figure 2.16, the interpolation is straightforward and is performed on a one-dimensional grid using a second order Lagrange interpolation. Figure 6.3 shows a comparison of the interpolated stresses on the surface of a flexible airfoil with that of the original stress distribution. In three-dimensions, the wing is composed of three distinct grids, a grid which defines the actual wing surface, and two grids that cover the wing tips. In practice, the area on the actual wing surface which is covered by the wing tip grids, is not used for computation and consists of a number of unused points. Since it is easier to interpolate between the actual wing surface and the structural grid directly, the loads on the unused grid points on the actual wing surface at the tips need to be computed prior to interpolating the loads on the structural grid.

Figure 6.3: Comparison of Interpolated Stresses on the Structural Dynamics Grid with the Actual Stresses on the Flow Solver Grid

Figure 6.4: Fixing up the Load on Unused Points of Grid C from Grid A and B through Interpolation

Figure 6.4 illustrates this situation. Grid C corresponds to the actual wing surface, and the loads at the grid points covered by the wing tip grids A and B are interpolated from grids A and B. After this step, all grid points of grid C have the corresponding load Π . Following this, the grid is separated into two halves, each half corresponding to grid points on the upper and lower surfaces of the wing respectively as described in Figure 6.5. Looping around each grid point in the structural grid, the load is interpolated from both the upper and lower surface of the flow solver grid. The net load acting on the structural grid is then obtained by summing up the interpolated values from the upper and lower surfaces as follows:

$$q(x, z, t) = \Pi_u n_u + \Pi_l n_l \quad (6.17)$$

where x, z represent co-ordinates of the structural grid points, n represents the normal to the boundary, and the suffixes u and l denote upper and lower surface respectively. Using the structural interpolated loads, the structural displacement y_s^p is obtained from Eq. 6.5(a) or Eq. 6.5(c). The updated shape of the plate is interpolated back to the flow solver grid. The grid is regenerated and the grid velocities and accelerations are also updated.

4. Using predicted values of flow velocity and grid velocity, an implicit corrector, applied to the momentum equation yields:

$$\frac{U^c - U^p}{\Delta t} = \frac{1}{2} X^p + \frac{1}{2} X^n + \frac{1}{2} (IU^c + IU^n) \quad (6.18)$$

5. The corrected pressure is solved based on corrected velocities:

$$\tilde{\Delta} p^c + \rho \sum_i \tilde{\nabla} u_i^c \cdot \frac{\partial \mathbf{u}^c}{\partial x_i} = 0 \quad (6.19)$$

6. Using the corrected pressure, the corrected loads are computed, and the corresponding displacements $\tilde{y}_s^c$ are obtained. This displacement is then relaxed using an under-relaxation factor ω as follows:

$$y_s^c = \omega \tilde{y}_s^c + (1 - \omega) y_s^p, \quad 0 < \omega < 1 \quad (6.20)$$

7. Check for convergence:

$$\text{if } |y_s^c - \tilde{y}_s^p| \leq 10^{-9}, \text{ set } y_s^{n+1} = y_s^c, \quad U^{n+1} = U^c, \quad p^{n+1} = p^c \quad (6.21)$$

Figure 6.5: Interpolation on the Structural Grid from the Upper and Lower Surfaces of the Flow Solver Grid

If the above conditions are not satisfied, steps 4-7 are carried out in sequence till convergence is attained. The number of corrector steps required for convergence depends on the type of problem in hand. For the present computations, typically 15-30 steps were required for convergence. When convergence is achieved, Eq. (3.1) and Eq. (6.1) will represent a fully coupled system of equations.

6.5. Numerical Validation for Deforming Body Aerodynamics

In this section, relevant test cases involving deforming boundaries are investigated for the purpose of validation. The first test case focuses on active or prescribed deformation whereas in the second test case, structural deformation due to aerodynamic forces using a partitioned type implicit algorithm is considered.

6.5.1. Plunging Motion of a Flexible NACA 0014 airfoil

Miao and Ho [141] studied the effect of chord-wise flexure amplitude on the aerodynamic forces for a NACA 0014 airfoil using the popular commercial flow solver *Fluent*[®] [54] using conformal hybrid meshes where the mesh surrounding the airfoil is structured and the background mesh is unstructured and concluded that there exists a maximum flexural amplitude beyond which thrust producing vortical signatures cease to exist. The same test case is used in the present computation to validate the deforming body framework and to study the implication of the Geometric Conservation Law (GCL) on aerodynamic force coefficients. A generalized deformation equation for any object which includes translation and deformation can be represented as

$$y(x,t) = y_0(x) + A(x - x_c)^n \cos(\omega_R t + Qx + \phi) + B(1 - \cos(\omega t)) \quad (6.22)$$

where the first term y_0 is the initial airfoil profile, the second term emulates a wave-like profile, the third term represents a rigid body motion. Here x_c denotes the point about which the deformation is zero, $A(c - x_c)^n$, the deformation amplitude with respect to the airfoil leading edge, Q , a parameter analogous to the wave number in the wave equation, ϕ the phase difference between deformation and plunge and B is the plunging amplitude. For plunging motion with cantilever type deformation considered by Miao and Ho [141], x_c represents the x -coordinate of the leading edge of the airfoil and $Q=0$. This represents a

deforming airfoil executing plunging oscillations. For a flexible phase angle of $\phi = \pi / 2$, Miao and Ho [141] observed a maximum propulsive efficiency of 30% and this phase angle has been used for all the present computations. For the present study, individual mapped grids are generated in regions surrounding the airfoil and in a background rectangle, and these are overlapped to form a composite grid as shown in Figure 2.15. The grid around the airfoil consists of 221×21 grid points with a minimum mesh spacing of 3×10^{-4} in the normal direction and the background grid is of size 221×221 . The deforming body framework is initially tested by forcing a pure plunging motion by setting $A = 0$, $B = 0.4$ in Eq. (6.22). Computations are performed for a Reynolds number $Re = 10^4$ and $h_0 = 0.4$, $k = 1$ and the results compared with the computations of Tuncer and Kaya [142]. The solution is initialized with a steady solution corresponding to the same Reynolds number and computations are performed for less than four plunge cycles. Figure 6.6(a) compares the computed time variation of the drag coefficients, which are indicative of thrust, with the results of Tuncer and Kaya [142]. A very good agreement is obtained. For $A = 0.3$, $n = 2$ and $x_c = 0$ as in Figure 6.6(b), the airfoil has chord-wise deformation and the computed time variation of the drag coefficient is shown and compared with the results from Miao and Ho [141]. Minor differences do exist and these can be attributed to issues related to grid quality, solver type etc. All these computations have been carried out with the GCL implementation. An alternative set of calculations have been performed without the GCL and it has been observed that the numerical value of the Jacobian differed only in the fourth decimal for reasons described in Chapter 2 and hence the aerodynamic force coefficients remained unaltered. In Figure 6.7, the computed pressure coefficient contours from the present study is compared qualitatively with the results of Miao and Ho [141] when the airfoil is at the mean position. There are minor variations in the minimum and maximum values which could be due to the fact that an exact comparison could not be made since all pressure coefficient plots for different deformation amplitudes have the same minimum and maximum values in the legend in Miao and Ho [141] whereas the local minimum and maximum values of pressure coefficient have been used in Figure 6.7(b). It is also seen that the relative position and size of the vortices are in good comparison with that of Miao and Ho [141]. The evolution of vortices from the airfoil, for the present case can be visualized in Appendix A3.6.1

(a)

(b)

Figure 6.6: Computed Drag Coefficient for (a) Rigid Plunging (b) Plunging with Deformation

Figure 6.7: Computed Contours of Pressure Coefficient from (a) Miao and Ho [141] and the (b) Present Computation for a Plunging and Deforming NACA 0014 Airfoil at its Mean Position

6.5.2. Oscillation of a Thin Flexible Plate in an Incompressible Fluid

Shin et al. [80] using a hybrid/Cartesian immersed boundary method, discussed the behavior of a thin vertical plate oscillating in a fluid at rest using an explicit coupling approach. A thin vertical plate of chord $c = 1$ cm, thickness $h_t = 0.003c$, density $\rho_b = 6667\rho$ and modulus of elasticity $E = 2 \times 10^{10}$ fixed to a horizontal base is considered. Based on a

non-dimensional quantity $\Gamma = \frac{EI}{\rho_b U^2 h_t^2 c^2} = 2$ defined by Shin et al. [80], and Reynolds number $Re = 500$, the plate is initially deflected in its first mode of oscillation for $1/4^{\text{th}}$ of a cycle with frequency $\tilde{\omega} = 5.284 \sqrt{\frac{EI}{\rho_b h_t}}$ and then set free. Boundary conditions for the

plate are $y_s(y = 0, t) = 0$, $y_{s,x}(y = 0, t) = 0$, $y_{s,xx}(y = c, t) = 0$, $y_{s,xxx}(y = c, t) = 0$. These correspond to zero displacement and slope/rotation at the fixed end, zero bending moment and shear at the free end respectively, as described in the beginning of this chapter. Overlapping grids are generated for this configuration and are shown in Figure 2.18. The mesh surrounding the plate has 4000 points with a minimum mesh spacing of 10^{-3} . The total number of points including a background Cartesian grid is about 157,000. Solutions are computed using both explicit and implicit coupling approaches with a numerical relaxation of $\omega = 0.9$. Figure 6.8(a) shows the computed tip displacement as a function of time. The results show a good agreement with that of Shin et al. [80]. Appendix A3.6.2 shows an animation of the vorticity contours for the present case. Figure 6.8(b) shows a comparison of the tip displacement as a function of time for explicit and implicit coupling. The explicit coupling suffers from a small phase lag and drop in amplitude compared to an implicit coupling approach. From Figure 6.8(a), the amplitude drop over one cycle of oscillation can be estimated to be $\delta_A = 0.92$ and for a plate oscillating in its natural mode with a time step of 10^{-3} , one can estimate the amplitude drop due to numerical damping based on the analysis in Sec 6.3 to be $\delta_N = 0.983$. From this, one can roughly conclude that the aerodynamic damping relative to an undamped system is $1 - \delta_A/\delta_N = 6.4\%$. As the density ratio is quite high, it is observed that the displacement of the plate is dominated by inertial forces. Also numerical instabilities due to added mass effects [76] are avoided when the motion of the plate is mainly driven by inertia.

(a)

(b)

Figure 6.8: Time History of the Tip Displacement with Implicit Coupling (a) In Comparison with the Results of Shin et al. [80] (b) In Comparison with Explicit Coupling

Chapter 7

Aerodynamic Characteristics of Flexible Flapping Wings

7.1. Introduction

This chapter focuses on the effect of airfoil/wing flexibility on thrust and trajectories. Both active/prescribed and passive deformation cases are studied. For both these types of deformation, analysis of forward flight is carried out by treating the deforming body as a rigid body at a given time instant and integrating the forces to obtain the position of its centre of mass. For active deformation cases, the effect of deformation amplitude and kinematics on thrust is investigated. The consequence of neglecting shear stress in the computation of total force is explored. For passive deformation cases, numerical issues relating to the coupling of the fluid-structure interaction algorithm and the effect of airfoil stiffness are discussed.

7.2. Free Flight of a Deformable Tear Drop Plate

A deformable tear drop plate similar to the ones used by Heathcote et al. [35] and Tang et al. [57] is used in the present work. The plate has a leading edge in the shape of an airfoil with chord $0.4c$ and maximum thickness $0.11c$ and the trailing edge in the shape of a very thin flat plate with a chord of $0.6c$ and thickness $0.005c$. The leading edge is constructed using a half-ellipse for the forward portion and a triangle for the rear portion. The mass of the plate is assumed to be unity. A mapping is generated for the tear drop plate to create a mesh around it and is overlapped with a background mapping to form a composite mesh as in Figure 2.16. The tear drop plate mesh is of size 500×11 with a minimum mesh spacing of 2×10^{-3} . The background mapping is of size 200×150 . Computations are performed at a Reynolds number of 1000 and a reduced frequency $k=1$. The parameters in the deformation function from Eq. (6.22) are given as $B=0.4$, $x_c=0.4c$, $\phi = \pi / 2$, $n=2$, and $Q = 0$. The deformation amplitude A is varied from 0 to 0.5 in steps of 0.1. Using this kinematics, the plate plunges in zero free-stream velocity and deforms with a maximum

(a)

(b)

Figure 7.1: Trajectory of the Tear-Drop Plate for Amplitudes
(a) $A=0, 0.1, 0.2$ (b) $A=0.3, 0.4, 0.5$

Figure 7.2: (a)-(f) Computed Vorticity Contours for Different Deformation Amplitudes for a Plunging and Deforming Tear-Drop Plate at the Mean Position (Position of Maximum Deformation)

deformation at 25% of the plunge cycle. The aerodynamic forces are then integrated to obtain the position of the center of mass at each instant of time. Figures 7.1(a)-(b) shows the computed trajectory of the plate centre of mass for different values of amplitudes. Figure 7.2 shows the computed vorticity contours for each of the different deformation amplitudes for a plunging and deforming tear drop plate at the mean position which corresponds to the position of maximum deformation. It is observed that increasing amplitude increases the wavelength of the trajectory thereby indicating that the most flexible plate has traveled faster than the rigid plate. Reference is made to Appendix A3.7.1-A3.7.3 for a visualization of the airfoil motion and the surrounding flow field. It is seen that the plate deformation induces a local angle of incidence similar to a combined plunging and pitching motion. When the plate plunges in the negative Y -direction, the positive deformation of the plate which is accompanied by a counter-clockwise rotation of the plate towards the trailing edge produces a small clockwise vortex which becomes evident when the plate reaches its mean position as shown in Figure 7.2. This vortex is absent when there is pure plunging motion and becomes pronounced with higher deformation. Together with the counter-clockwise vortex released at the trailing edge due to the plunging motion, this vortex pair produces large amounts of thrust which propels the body forward.

Figure 7.3 shows the time history of the variation of the horizontal force coefficient corresponding to different values of the amplitude of the deformation and from this it can be seen that when the airfoil reaches its maximum deformation near $t = 0.78$ seconds, large peaks of thrust occur. This corresponds to the situation shown in Figures 7.2 (d)-(f) where the extra vortex is also responsible for thrust production. After few cycles of motion, the plate tends to reach a periodic state where the net horizontal force over one cycle would vanish and the plate would move forward as a result of the interplay between inertia and the aerodynamic forces. As mentioned in Tang et al. [57], Lian et al. [62], even at low Reynolds numbers, the force acting on a plunging and deforming airfoil is dominated by pressure. It is unknown however whether this assumption is suitable for trajectory computations as viscous shear forces can also influence the trajectory. Figures 7.4(a)-(b) show the computed trajectory of the centre of mass of the flexible tear drop plate for selected values of the deformation amplitudes.

Figure 7.3: Computed Time History of the Horizontal Force Coefficient C_x for Various Deformation Amplitudes

When only pressure forces are considered in setting the body in motion, it is observed that during the initial times corresponding to the inception of the motion, there exists minor differences in the trajectory for reasonable deformation amplitudes such as $A=0.3$, but these differences amplify as the body moves forward as can be seen in Figure 7.4(a). For the case corresponding to a large deformation amplitude, i.e. $A=0.5$, the difference is evident and a phase lag is introduced between the two trajectories influenced by pressure and viscous forces as can be seen in Figure 7.4(b). Hence it is clear that both pressure and shear forces must be taken into account for computing the long term behavior of a plunging and deforming airfoil.

7.3. Effect of Deformation Kinematics on Thrust Generation

Three-dimensional flexible wing kinematics is more complex from the view point of grid generation and requires prior knowledge of the functioning of the hyperbolic grid generation algorithm for various mesh control parameters such as the marching distance,

(a)

(b)

Figure 7.4: Trajectory of a Plunging and Deforming Tear-Drop Plate with and without Shear Stress for (a) $A = 0.3$, (b) $A = 0.5$

mesh spacing, smoothing etc. Hence a computation is performed on an initial mesh over one cycle of deformation to check for any errors induced in the mesh generation process. The flow solver is then run for longer time after no errors have been confirmed during the mesh generation process. This will reduce the overall computational overhead while solving the actual problem using the flow solver. The wing considered in this computation has an elliptical cross-section which is 10 % thick and a rectangular planform with unit chord and of aspect ratio 2. The density ratio of the wing (ρ_b/ρ_f) is assumed to be 12.7. It is fixed at the mid span and free to deform in the span-wise and chord-wise direction. The composite mesh with 0.35 Million points, surrounding the wing consists of three hyperbolic mappings, one for the main wing and two for the wing caps similar to the one shown in Figure 2.19. These three mappings are made to overlap a background Cartesian mapping which is rectangular. When the body deforms, only the meshes surrounding the body are regenerated. For this section, we consider the forced deformation of a wing at a deformation frequency of 0.5 Hz. At this frequency, the marching distance from the surface of the wing can be as high as $0.2c$ for the mesh dilation to be very small. Both active and passive flight characteristics with two different types of flapping modes have been investigated for this configuration with the wing deforming in zero free-stream conditions.

7.3.1. Simple Flapping Mode

For this mode, one uses $A = 0.25c$, $Q=0$, $B=0$, $\phi = \pi / 2$, $n=2$, and $f=0.5$ Hz in Eq. (6.22). This results in a symmetric span-wise deformation along Z axis as in Figure 2.19(c). For the case with active flight where there is no forward motion, computations are carried out for five cycles of deformation. Figure 7.5 shows the computed Z-component of vorticity contours in the vicinity of the wing tips at various times during the deformation cycle. Since the wing is deforming in zero free-stream velocity, the vortices are not convected downstream of the flow. The effect of these vortices will be realized only when the passive flight where the forward motion is computed simultaneously similar to the computations shown in Sec 5.3. The vortices generated as a result of the motion of the wing are symmetrical and are consistent with the kinematics of the problem.

Figure 7.5: Computed Z-Vorticity Contours for a Deforming Wing at Different Instances of Times $t = T_{1/4}$ to $t = T_{3/4}$ (T being the Period of one Deformation Cycle = 2 s) For the Simple Flapping Mode

Figure 7.6: Characteristics of the Simple Flapping Mode Showing (a) Horizontal Force Coefficients Over One Cycle and (b) Trajectory of the Wing Tip

The horizontal force coefficients which are computed based on the maximum tip velocity of $U_T = 2\pi fA = 0.78$ are shown in Figure 7.6(a). Figure 7.6(b) shows the path of the wing tip when the wing is set free to move by virtue of thrust. It can be seen that the wing has traveled less than 0.001 chord lengths indicating that this mode of flapping might not be suitable for forward flight.

7.3.2. Combined Flapping Mode

In this mode, wing deformation is prescribed about all three co-ordinate axes. The wing kinematics shown Figure 7.7 is described by the following set of equations:

$$\begin{aligned} X &= x(x) - AZ^2 \sin(2\pi ft) \\ Y &= y(x) + AX^2 \sin(2\pi ft + \pi/2) + AZ^2 \sin(2\pi ft) \\ Z &= z(x, y) \end{aligned} \quad (7.1)$$

where the co-ordinates in the lower (x, y, z) and upper case (X, Y, Z) correspond to undeformed and deformed states respectively. The amplitude and frequency are chosen as $A=0.2c$ and $f=0.5$ Hz respectively. At the beginning of the flapping cycle, the wing is in the undeformed state as in Figure 7.7(a). During each phase of the cycle, deformations about all three axes occur simultaneously. Figure 7.8 shows the computed Z-component of vorticity in the vicinity of the wing tips at different instances of times. During the first half of the cycle as shown in Figures 7.7(a)-(f) and Figures 7.8(a)-(c), the negative vortex generated at the trailing edge is considerably stronger, by considering the size of the vortex, compared to the positive vortex at the leading edge. This is in contrast to the previous case of a simple flapping mode where in Figure 7.5, both the leading and trailing edges produce symmetric vortices for the entire cycle. During the second half of the cycle, the positive vortex generated at the trailing edge is stronger than the corresponding negative vortex at the leading edge. This asymmetry in the size of vortices results in a higher thrust compared to the simple flapping mode as observed in Figure 7.9(a). The higher thrust can also be explained by means of the vorticity moment equations described in Chapter 5. When the wing is set free to move under the influence of thrust, it is observed from Figure 7.9(b), that for every 5 cycles of deformation, the wing moves by a distance of $0.18c$. An animation of the surface pressure contours for the present case can be found in Appendix A3.7.4 and A3.7.5.

Figure 7.7: Wing Kinematics for Combined Flapping Modes at Different Time Instants (T being the Period of Oscillation = 2 s)

Figure 7.8: Computed Z-Vorticity Contours for a Deforming Wing (Combined Flapping Mode) at Different Instances of Times $t = T/4$ to $t = T/4$ (T being the Period of One Oscillation = 2 s)

Figure 7.9: Combined Flapping Characteristics Showing (a) Horizontal Force Coefficient Over one Cycle and (b) Trajectory of the Wing Tip

7.4. Plunging and Passively Deforming Airfoil – Effect of Time Step, Relaxation and Damping

Tang et al. [57] through numerical computations and Heathcote et al. [35] through experiments have studied the effect of chord-wise flexibility on the aerodynamic forces of a plunging airfoil. Utilizing a similar geometry adopted by the above authors, computations are performed to demonstrate that stable solutions are obtained with a proper combination of time step, numerical relaxation and explicit numerical damping proportional to the velocity of the body, $\kappa_E \dot{y}_s$. The geometry under consideration is an airfoil-plate combination of a net chord $c=1.0$ cm. The airfoil of chord $0.4c$ is assumed to be rigid, and the plate of length $0.6c$ is assumed to be thin and flexible with a thickness ratio of $h/c=0.005$. Overlapping grids are generated using a hyperbolic marching algorithm as shown in Figure 2.16. The grid surrounding the airfoil has 500×11 points with a minimum spacing $O(10^{-3})$. The total number of grid points including the background Cartesian mesh is about 58300. The flexible plate has a density of $\rho_b = 7.85\rho$ and modulus of elasticity $E = 2.05 \times 10^{10} \rho_f U^2$. The airfoil-plate combination undergoes a plunging motion $h = -0.194(1 - \cos 8.8t)$ in a uniform stream of fluid with velocity $U=1$ cm/s. This corresponds to a Strouhal number of $St=0.85$ and Reynolds number $Re = 9000$. Stable solutions are obtained using implicit coupling with a time step of 10^{-3} , relaxation of 0.01 and explicit damping coefficient $\kappa_E = 10$. Figure 7.10 shows the time history of the tip displacement in comparison with the results of Tang et al. [57]. Exact reproduction of their results could not be obtained due to differences in the coupling algorithm. However the comparison is satisfactory. An animation of the vorticity contours for the present test case can be found in Appendix A3.7.6. With the same damping coefficient $\kappa_E=10$, time steps ranging from 10^{-4} to 10^{-3} and relaxation varying from 0.01 to 0.05, the behavior of the coupled problem is analyzed. Figure 7.11(a)-(c) shows the time history of tip displacement with the iteration index, which is defined as the product of local and global iteration index. The iterations within one time step are identified by the region where the tip displacement hardly changes or oscillates. Successive time steps ($t, t+\Delta t, t+2\Delta t, \dots$) are identified by large gradients in the tip displacement. The point of instability is identified by the rapid growth of displacement.

In Figure 7.11(a), when the time step is 10^{-3} , oscillations occur within the inner iteration for a relaxation of 0.04.

Figure 7.10: Time History of the Tip Displacement in Comparison with the results of Tang et al. [57]

However they are damped while the iteration proceeds. But for a relaxation of 0.05, the coupling is unstable. When the time step is decreased further as in Figure 7.11(b)-(c), instability appears at a lower relaxation levels. It can be concluded that more relaxation is desirable when the time step is decreased in order to have a stable solution. Similar computations were performed for a damping coefficient of $\kappa_E=15, 20$. The results of which are summarized in Tables 7.1-7.3. If the combination of damping, relaxation and time step results in a stable solution, it is identified by (S)-Stable else by (U)-Unstable. If the method is neutrally stable, it is denoted S/U. In this case, the displacements keep oscillating about a mean value within the inner iteration loop. Based on the observations from the above computations, the following conclusions can be derived; (i) For a given damping and time step, instability is reached with increasing relaxation, (ii) For a given damping and relaxation, instability is reached with decreasing time step, and (iii) For a given time step and relaxation, instability is reached with decreasing damping.

(a) $\Delta t = 1e-3$

(b) $\Delta t = 5e-4$

(c) $\Delta t = 1e-4$

Figure 7.11: Time History of the Tip Displacement for Different Relaxation Coefficients (ω) and Time Steps

Table 7.1: Stability of the Fluid-Structure Interaction Problem with a Time Step of $\Delta t=10^{-3}$ for Various Relaxation and Numerical Damping Coefficients

$\omega \backslash \kappa_E$	0.01	0.015	0.0165	0.0169	0.017	0.018	0.0185	0.019	0.02	0.03	0.04	0.05	0.06	0.075
10.0	S	S	S	S	S	S	S	S	S	S	S	U	U	U
15.0	S	S	S	S	S	S	S	S	S	S	S	S	S	U
20.0	S	S	S	S	S	S	S	S	S	S	S	S	S	S/U

Table 7.2: Stability of the Fluid-Structure Interaction Problem with a Time Step of $\Delta t=5 \times 10^{-4}$ for Various Relaxation and Numerical Damping Coefficients

$\omega \backslash \kappa_E$	0.01	0.015	0.0165	0.0169	0.017	0.018	0.0185	0.019	0.02	0.03	0.04	0.05	0.06	0.075
10.0	S	S	S	S	S	S	S	S	S	S	U	U	U	U
15.0	S	S	S	S	S	S	S	S	S	S	U	U	U	U
20.0	S	S	S	S	S	S	S	S	S	S	S	U	U	U

Table 7.3: Stability of the Fluid-Structure Interaction Problem with a Time Step of $\Delta t=10^{-4}$ for Various Relaxation and Numerical Damping Coefficients

$\omega \backslash \kappa_E$	0.01	0.015	0.0165	0.0169	0.017	0.018	0.0185	0.019	0.02	0.03	0.04	0.05	0.06	0.075
10.0	S	S	U	U	U	U	U	U	U	U	U	U	U	U
15.0	S	S	S	U	U	U	U	U	U	U	U	U	U	U
20.0	S	S	S	S	S	S	U	U	U	U	U	U	U	U

7.5. Forward Flight of a Plunging and Passively Deforming Airfoil

To study the effect of flexibility on forward flight, three different cases of plate stiffness is considered corresponding to $E=2 \times 10^{10}$, 10×10^{10} and 20×10^{10} . The airfoil-plate combination is plunged at zero free-stream conditions with the same kinematics described in the previous section. Once the forces are computed at each time step, the horizontal component of the centre of mass is recomputed by solving Eq. (3.20). Although the pressure is a dominating factor in deforming and moving the airfoil, the absence of shear stress results in a minor phase lag in the position of the airfoil as discussed earlier. Hence both pressure and shear are taken into consideration for computing the position of the airfoil.

Figure 7.12 shows the computed position of the centre of mass for different moduli of elasticity. It is seen that for all cases, the plunging and deforming motion of the airfoil creates a thrust force that propels the body forward in the negative X-direction. While it was observed previously that the airfoil with the highest deformation traveled the fastest when the forced deformation is out of phase by $\pi/2$ with the plunging motion, it is not the case when the deformation is passive. It is seen that a moderate flexible airfoil with $E=10 \times 10^{10}$ outperforms the least and the most flexible airfoil by traveling the fastest. An animation of the computed trajectory and vorticity contours for different values of E can be found in Appendix A3.7.7-A3.7.9. Vorticity contours for all three configurations are plotted in Figures 7.13(a)-(c) at the instant when the airfoil has crossed the mean position $h = -0.194$ while plunging downward. A counter-clockwise vortex appears at the trailing edge of the airfoil which is favorable for thrust production. However, upstream of this vortex, is seen a small clockwise vortex. The magnitudes of both these vortices increase with reducing stiffness. However, it is also observed that the rate of growth of the clockwise vortex with reducing stiffness is greater than that of the counter-clockwise vortex. This is observed consistently for every period of oscillation when the airfoil passes its mean position. Based on this observation, it can be concluded that there must be an optimum stiffness for which the net effect of both these vortices will not be detrimental to thrust. This could be one of the possible reasons as to why the moderate flexible airfoil produces more thrust.

Figure 7.12: Trajectory of the Airfoil-Plate Combination for Various Airfoil Flexibilities

Figure 7.13: (a)-(c) Computed Vorticity Contours for Different Plate Flexibilities for a Plunging and Deforming Airfoil-Plate Combination at the Mean Position

7.6. Spanwise Deformation of a NACA0012 Cross-Section Wing

Heathcote et al. [36] and Chimakurthi et al. [78] performed experiments and computations respectively on a rectangular NACA0012 cross-section wing undergoing plunging motion. The plunging motion of the wing results in a spanwise deformation that varies with wing stiffness. Different wing stiffness corresponding to a very flexible, flexible and rigid wing were considered and the aerodynamic characteristics were obtained for different flow conditions at a Reynolds number $Re=3 \times 10^4$. Here the 'Flexible' wing case, with $E=200\text{Gpa}$ considered in Heathcote et al. [36] is considered for the validation study. The wing is plunged at a reduced frequency $k=1.82$, Strouhal number $St=0.31$ and Reynolds number $Re=10^4$. As the density ratio is small (~ 7.8), the dominant deformation mechanism is the ambient aerodynamic force rather than inertial force, hence, the coupling is very sensitive to explicit damping (κ_E), time step (Δt) and relaxation factor (ω). Typical choices for these parameters for the present problem are $\kappa=10$, $\Delta t=2.5 \times 10^{-3}$, $\omega=0.01$. The number of inner iterations required for convergence of Eq. (6.20) varies from 20 to 30. To verify whether the loads have been transferred from the fluid to the structure accurately, the total stress distribution on the fluid dynamics grid is compared with the structural dynamics grid for both the upper and lower surfaces in Figure 7.14(a)-(b) at an arbitrary instant of time. Good comparisons are obtained indicating that interpolation of the stresses on the structural grid is being performed accurately. This comparison is similar to Figure 6.3 where the total stresses were compared for the fluid dynamics and structural dynamics grid in two-dimensions.

Solutions are computed for two cycles of oscillation and the computed thrust coefficients (C_T) are compared with the experimental results of Heathcote et al. [36] shown in Figure 7.15(a). A very good comparison is obtained although the Reynolds number is different in the experiment. The fact that the Reynolds number has little impact on the aerodynamic forces in the range $10^4 < Re < 3 \times 10^4$ has also been observed in Heathcote et al. [36] for rigid plunging wings. Figure 7.16(a) shows the position of the wing at different points of the plunge cycle i.e., points A, B and C as indicated in Figure 7.15(b).

Figure 7.14: Comparison of the Stresses (Pressure + Shear) for the CFD and the CSD Grids on the (a) Upper and (b) Lower Surfaces of the Wing

Figure 7.15: Computed (a) Thrust Coefficient in Comparison with the Experimental Results of Heathcote et al. [36] (b) Wing Displacement at the Root Showing Selected points A, B and C of interest

The portion of the wing at the origin of the axes is the wing root. At point A of the plunge cycle as shown in Figure 7.16(a) the wing is at the topmost position and is moving downwards. The wing tip is deflected downwards and starts to move in a direction opposite to that of the plunge motion. Figures 7.16(b)-(d) show the contours of the Z-component of vorticity at $z = 0, \pm 1.5$. There is a distinct difference between the vortex structures at the wing root and wing tip. At point B, the wing is in the horizontal position and the vorticity structures are similar throughout the wing span. However at point C, when the wing is at the lowest position of the cycle, the concentration of vorticity on the wing surface is not symmetrical and is biased towards the root of the wing. The size of the leading-edge vortex appears to be larger towards the wing root. Figure 7.17 and Figure 7.18 show the computed streamlines relative to the reference frame of the moving wing at two different instants corresponding to points B and C of the plunge cycle shown in Figure 7.15(b). The computed streamlines are shown on two x - z planes, and three x - y planes. One of the x - z plane is located above the wing at a distance of $y_{pl} = y_{LE} + 0.25c$ and the other, below the wing at a distance of $y_{pl} = y_{LE} - 0.25c$, y_{LE} being the y -coordinate of the leading edge of the wing root. The selected x - y planes are located at different spanwise locations, $z = -1.5, 0$ and 1.5 which correspond to the wing root, mid-span and tip respectively. Pressure contours are also plotted along the streamline plots at the different spanwise directions. When the wing is at point B of the plunge cycle, as can be seen from Figure 7.17, the streamlines appear to be almost symmetric about the mid-span and the fluid is convected from the lower side of the wing tips to the upper side as shown in Figures 7.17(a)-(b). The upper surface shows a leading edge vortex consistent with Figure 7.16(b) and Figures 7.17 (c)-(e). When the wing moves to point C as can be seen in Figure 7.18, the wing tip deflection results in an asymmetric leading edge vortex about the mid span as in Figure 7.18(b). The size of the vortex gradually increases from tip to root. This requires the fluid velocity in the spanwise direction to be higher at the tip compared to the root. A collection of animations pertaining to the present test case can be found in Appendix A3.7.10-A3.7.15.

Figure 7.16: Computed (a) Spanwise Deflection and (b)-(d) Vorticity Contours at Selected Spanwise Stations (Root, Wing tip and Mid-span) for Plunging and Deforming NACA0012 Cross-section Wing at Selected Points of the Plunging Cycle as Indicated in Figure 7.15(b).

Figure 7.17: Streamlines on a Plane (a)-(b) ($y_{pl} = y_{LE} \pm 0.25c$) Parallel to the Wing Surface and (c)-(e) Perpendicular to the Span at point B of the Plunge Cycle, y_{LE} being the y-coordinate of the Leading Edge of the Wing Root

Figure 7.18: Streamlines on a Plane (a)-(b) ($y_{pl} = y_{LE} \pm 0.25c$) Parallel to the Wing Surface and (c)-(e) Perpendicular to the Span at Point C of the Plunge Cycle, y_{LE} being the y-coordinate of the Leading Edge of the Wing Root

Chapter 8

Conclusions and Recommendations for Future Work

In this chapter, the key conclusions which are derived from the present research work are highlighted. A brief overview of potential research work to be undertaken in the future is also presented to motivate further research on flapping wing aerodynamics.

8.1 Concluding Remarks

Computation of unsteady low Reynolds number flow-field past moving airfoils and wings have been carried out by solving the unsteady incompressible Navier-Stokes Equations on a system of overlapping and moving grids with the specific purpose of understanding the aerodynamic characteristics of low Reynolds number flapping wings. Certain numerical considerations such as verification and validation of the flow solver have been performed by solving various benchmark problems. The effect of grid size and location of the far-field boundary are discussed in the context of unsteady flows in two dimensions. Owing to numerical errors on coarse grids, an airfoil set into motion due to the forces is seen to travel faster compared to solutions on finer grids. When the location of the outer or far field boundary is very near to the oscillating body, it is observed that the numerical boundary conditions have a tendency to accelerate the airfoil. The jet switching phenomenon which was observed by earlier researchers through experiments and computations is also demonstrated.

An important result from the present research is the effect of leading edge – trailing edge asymmetry on thrust generation both under active and passive flight conditions. Using the general aerodynamic theory based on vorticity moments, it is demonstrated that under zero freestream conditions, asymmetry of the shed vortices is required for thrust generation. Based on two different airfoil sections, a NACA 0012 and an elliptical airfoil, numerical evidence suggests that the relative convection rate of positive and negative vortices is a key factor that contributes to thrust generation. It is also proved that NACA 0012 airfoils having sharper trailing edges are beneficial in terms of thrust production. When the airfoils are set free to move under the action of

aerodynamic forces, it is observed that the airfoil having the sharpest trailing-edge travels the fastest. To investigate further the role of asymmetry, asymmetric kinematics was tested on symmetric airfoil sections. When the kinematics are symmetric, symmetric vortex structures are released from the wing giving rise to zero thrust, whereas for asymmetric kinematics, a finite thrust is produced due to asymmetry in the vortex structures. For pitching airfoils and wings below the static stall angle and above a certain critical frequency, it is shown that the lift hysteresis loops can flip over such that the lift can decrease with increase in angle of attack vice-versa. A mathematical model is also framed to support the observation.

The effect of active and passive wing flexibility has also been investigated in detail. The effect of the Geometric Conservation Law (GCL) for deforming bodies on computed solutions is also presented. When the grids are generated using a hyperbolic mapping, it is observed that the usage of the GCL can be avoided when the marching distance from the airfoil surface is small. A generalized deforming grid framework is formulated such that the body deforms and simultaneously moves forward due to the aerodynamic forces. The effect of prescribed deformation amplitude and kinematics on thrust is explored. The most flexible airfoil is seen to travel faster compared to a rigid airfoil of the same size. Most importantly, the effect of neglecting shear stress for the force computation is investigated. When the airfoil deformation is large, it is seen that a phase difference in the trajectory is observed corresponding to the cases with and without shear stress, hence both pressure and shear must account for the forward motion of the airfoil. Another aspect that was given attention was the effect of deformation kinematics on thrust. Similar to the discussion on rigid wings dealing with asymmetry of the vortices, it is observed that a symmetric flapping mode gives rise to no thrust at all, whereas asymmetric wing deformation which included deformation about all axes gives rise to a higher thrust.

To model the interaction between the fluid and the structure, a partitioned coupling algorithm is formulated and successfully tested using the *Overture* framework. By treating the airfoil as a thin beam and using the Euler-Bernoulli beam equation, the structural equations are solved using a finite difference method. The flow solver *cgin*s is

modified so as to accommodate multiple corrector steps with grid regeneration within each inner step for the coupling. The algorithm is successfully tested on a limited number of test cases. Numerical aspects such as the effect of time step on the numerical damping are assessed. It is observed that when the density of the structure is very small, a proper combination of time step, relaxation and numerical damping are essential to achieve stability. Free flight computations on plunging and passively deforming airfoils show that there is an optimum flexibility for which thrust is maximum. Three-dimensional computations showing the deformation of a wing due to the surrounding aerodynamic forces was attempted. The governing equations of fluid flow were coupled with Kirchhoff's plate equations, an approximate model governing the deformation of a thin plate under an external load. Using a partitioned approach, the fluid flow equations and the structural dynamical equations were solved to compute the thrust generated as a result of a plunging motion. The computed results were compared with experimental results available in literature.

8.2 Recommendations for Future Work

Having done extensive two-dimensional computations and few three-dimensional computations, it is desirable from a design point of view that more number of three dimensional computations be made. The three dimensional computations of flapping wings coupled with rigid body dynamics will stimulate a lot of interest in the design department. Although the motion of the wing in the present study is constrained to move passively along a particular direction, the motion can be relaxed about all directions since the code is generalized to handle six degrees of freedom motions. This however requires a very large computational domain and these computations cannot be achieved without having efficient computing hardware / software. Hence, further realistic computations could be made in parallel computing environments and on faster architectures. Parallelization can be achieved in many ways. One way which is usually followed by most practitioners is to decompose the domain into several small regions and let each processor handle that region. This is commonly exploited when scalar indexing of variables is used. In this context, conventional codes loop over each mesh point to update the solution to the next time level. Hence by letting each processor handle a section of the

domain, substantial savings in time can be obtained. However while using vector addressing, like the one used in *Overture*, variables in memory are updated in the so called “data parallel” mode. Hence parallelizing a vectorized code will provide more computational savings when compared to the former method.

An alternative way which might be very helpful in reducing the overall computation time is to use parallel linear solvers for the Pressure Poisson equation. In Appendix B, a break up of the computational time spent for each process is given for a two- and three-dimensional computation. It can be seen from Table A2.1 and Table A2.2 that almost 35%, 50 % of the time is spent in solving for the pressure in two- and three-dimensions respectively. Hence parallelizing this portion might help reduce computational time. It might seem at the first instance that this method might be easy to set up. However, the parallel linear solvers do require strictly a parallel code for calling these routines. The parallel linear solvers from the PETSc library require the matrix to be partitioned before the solver is called. Hence changes to the *cgins* code are required to be made to implement this procedure. A combination of parallel linear solvers with a parallel vectorized code would be the best choice for three-dimensional unsteady problems. Using a parallel code, one can investigate the effect of flapping kinematics on trajectories in three dimensions. For example, the sensitivity to changing the frequency and/or amplitude while the wing is flapping on the trajectory can be estimated by a series of computations. Similar to aerodynamic derivatives which appear in the study of fixed wing aircrafts, frequency and amplitude derivatives can be formulated. This then will represent a control problem for the path of the wing. A wing plunging at a given frequency and amplitude can be steered easily to a desired destination on a desired path. Solving the above problem will also require a series of computations to be performed for a wide range of flapping frequencies and amplitudes. Using this data, a reduced order model can be framed which then will represent the behavior of the same system for different frequencies / amplitudes. The forces and moments acting on the wing can be expressed in terms of instantaneous wing velocity, acceleration (both linear and angular). For a new set of velocities and accelerations, the reduced order model may be called to

obtain the forces. By this procedure the solution to the fluid dynamics equations are avoided and hence will reduce the overall computational time.

So far, many computations and experiments have laid their focus on single wings or a pair of wings. The effect of wing body interaction, and tail-wing (if any) interaction is missing. The effect of vortices on the tail is important since it contributes to the stability of the entire micro air vehicle. Hence with the advent of computational power, more focus can be directed to these types of problems. Low Reynolds number dynamic stall in three-dimensions has not been investigated in the literature in detail; hence lot of attention needs to be given to these phenomena. Studies on unsteady separation and its impact on both thrust and lift generation will benefit the development of Micro Air Vehicles (MAVs).

The other area which deserves more attention is the effect of wing flexure on aerodynamic performance in flapping. As seen from the present research, a flexible wing is shown to provide substantial improvement in thrust generation. The lift characteristics and propulsive efficiency were not studied. As there are very few details relating to this area, it will be worthwhile to analyze the role of spanwise and chordwise flexibility on lift and propulsive efficiency. To account for structural deformation due to the aerodynamic forces, a partitioned algorithm which is developed as a part of this work is not very robust compared to fully coupled algorithms. Due to the disparity in the time scales of fluid and structural motion, partitioned algorithms fail for certain geometric and numerical parameters with the appearance of instabilities as observed earlier. However these instabilities can be overcome when one uses a fully coupled algorithm. Since the grids involved are overlapping, the structural dynamics equations need to be solved in the same overlapping grid framework as that of the fluid flow, which poses a challenge. Passive structural deformation coupled with free flight also poses problems with the size of the computational domain. Since the direction of free flight is not known apriori, it is necessary to have a bigger domain to track the motion of the wing. This incurs a lot of computational overhead. Possible solutions to this problem could be the subject of future investigations. Improvements to the constitutive equation of the structure to handle large deformations using a Mooney-Rivlin model would also be very advantageous.

Much work has to be done in the area of airfoil shape optimization under unsteady low Reynolds number flow conditions. Some of the parameters that can be optimized based on the airfoil shape are thrust, lift, propulsive efficiency and forward speed during a dynamic motion. The obtained airfoil shape can also change with Reynolds number. Optimization can also be performed by holding the geometry fixed, and varying the kinematical parameters such as amplitude, frequency and phase.

Turbulence is another open question to the scientific community. At high Reynolds numbers, resolving a flow completely till the smallest possible scale using **Direct Numerical Simulations (DNS)** is still not possible with the present day supercomputers. Hence the need for turbulence models was realized. For low Reynolds number applications involving flapping wing flight or aquatic flight, one can resort to DNS through efficient parallel algorithms. However this again takes considerable computation time before one realizes whether the obtained solutions are physically realizable. There have been many instances where the use of turbulence models are avoided for computing low Reynolds number unsteady flows. Some of the questions that one can ask is, are these justifiable at all times? Under what conditions can we resort to avoiding the use of a turbulence model? Research can be focused on performing a DNS using higher order accurate methods on low Reynolds number unsteady flow configurations and comparing the results with that of lower order accurate methods without a turbulence model.

Finally, compressibility effects on flapping wings need to be addressed in detail. Generally, low Reynolds number flow computations use the incompressible flow assumption to predict aerodynamic forces and moments. However, there might be a possibility that under high frequency flapping, the flow velocities near the wing tips may approach the compressible regime. When that happens, the structure of the vortices may get altered. The resulting aerodynamic forces, moments and the trajectory will be influenced under these conditions. Using a compressible solver, one might also be interested in computing the acoustic field generated due to vortex shedding, which is crucial in understanding the stealth properties of flapping wing MAVs.

References

- [1] Shyy, W., Berg, M., and Ljungqvist, D., Flapping and Flexible Wings for Biological And Micro Air Vehicles, *Progress in Aerospace Sciences*, Vol. 35, 1999, pp. 455-505
- [2] Schmidt-Nielsen, K., Locomotion: Energy Cost of Swimming, Flying, and Running, *Science*, Vol. 177, No. 4045, 1972, pp. 222-228.
- [3] Rauzon, Mark J. *Hummingbirds*. Ulster, PA. Grolier 1997: 12-13.
- [4] Micucci, Charles. *The Life and Times of the Honeybee*. United States: Houghton Mifflin, 1995.
- [5] Aerosphere, Air and Space Magazine
- [6] Lighthill, J., Mathematical Biofluidynamics, *Regional Conference Series in Applied Mathematics, SIAM*, 1975.
- [7] Abate, G., Ol, M., and Shyy, W., Introduction:Biologically Inspired Aerodynamics, *ALAA Journal*, Vol. 46, No. 9, 2008, pp. 2113-2114.
- [8] Milano, M., Jurjevich, D., and Adrian, R., The Flying Brick: A Cautionary Note on Testing Flying Robots Using Guided Wires, *IEEE Transactions on Robotics*, Vol. 25, No. 2, 2009, pp. 426-428.
- [9] Knoller, R., Die Gesetze des Luftwiderstandes, *Flug- und Motortechnik(Wien)*, Vol. 3, No. 21, 1909, pp. 1-7.
- [10] Betz, A., Ein Beitrag zur Erkl ä rung des Segelfluges, *Zeitschrift fü r Flugtechnik und Motorluftschiffahrt*, Vol. 3, 1912, pp. 269-272.
- [11] Katzmayer, R., Effect of Periodic Changes of Angle of Attack on Behavior of Airfoils. Translation from “Zeitschrift fur Flugtechnik und Motorluft-Schiffahrt”, *NACA TM*, No. 147, 1922.
- [12] Ober, S., Note on the Katzmayer Effect on Airfoil Drag, *NACA-TN-214*, 1925.
- [13] Cebeci, T., Platzer, M.F., Chen, H., Chang, K.-C., and Shao, J.P., Analysis of Low-Speed Unsteady Airfoil Flows, *Springer Verlag* - Germany, 2005.
- [14] Birnbaum, W., Das Ebene Problem des Schagenden Flügels, *Zeitschrift für angewandte Mathematik and Mechanik (ZAMM)*, Vol. 4, pp. 277-292, 1924.

- [15] Wagner, H., Über die Entstehung des dynamischen Auftriebs an Tragflügeln, *Zeitschrift für angewandte Mathematik und Mechanik (ZAMM)*, Vol. 5, 1925 pp.17-35.
- [16] Küssner, H.G., Zusammenfassender Bericht über den instationären Auftrieb von Flügeln, *Luftfahrt-Forschung*, Vol. 13, 1936, pp. 410-424.
- [17] Theodorsen, T., General Theory of Aerodynamic Instability and the Mechanism of Flutter, *NACA report 496*, 1935.
- [18] Garrick, I.E., Propulsion of a Flapping and Oscillating Airfoil, *NACA-TR-567* 1936.
- [19] Koochesfahani, M., Vortical Patterns in the Wake of an Oscillating Airfoil, *AIAA Journal*, Vol. 27, No. 9, 1989, pp. 1200-1205.
- [20] Triantafyllou, M.S., Triantafyllou, G.S., and Gopalkrishnan, R., Wake Mechanics for Thrust Generation in Oscillating Foils, *Physics of Fluids A*, Vol. 3, No. 12, 1991, pp. 2835-2837.
- [21] Dickinson, M.H., and Gotz, K.G., Unsteady Aerodynamic Performance of Model Wings at Low Reynolds Numbers, *The Journal of Experimental Biology*, Vol. 174, 1993, pp. 45-64.
- [22] Sunada, S., Kawachi, K., Matsumoto, A., and Sakaguchi, A., Unsteady Forces on a Two-Dimensional Wing in Plunging and Pitching Motions, *AIAA Journal*, Vol. 39, No. 7, 2001, pp. 1230-1239.
- [23] Hong, Y., and Altman, A., An Experimental Study on Lift Force Generation Resulting from Spanwise flow in Flapping Wings, 44th *AIAA Aerospace Sciences Meeting and Exhibit*, Reno, NV, AIAA paper 2006-448, 2006.
- [24] Isaac, K.M., Colozza, A., and Rolwes, J., Force Measurements on a Flapping and Pitching Wing at Low Reynolds Numbers, 44th *AIAA Aerospace Sciences Meeting and Exhibit*, Reno, NV, AIAA paper 2006-450, 2006.
- [25] Parker, K., Soria, J., and Ellenrieder, K.D., Thrust Measurements From a Finite-Span Flapping Wing, *AIAA Journal*, Vol. 45, No. 1, 2007, pp. 58-70.
- [26] Lai, J.C.S., Platzer, M.F., Characteristics of a Plunging Airfoil at Zero Freestream Velocity, *AIAA Journal*, Vol. 39, No. 3, 2000, pp. 531-534.
- [27] Bennet, A.G., Obye, R.C., and Jeglum, P.M., Ornithopter Aerodynamic Experiments, Swimming and Flying in *Nature*, Vol. 2, 1975, pp. 985-1000.
- [28] Birch, J.M., Dickson, W.B., and Dickinson, M.H., Force Production and Flow Structure of the Leading Edge Vortex on Flapping Wings at High and Low

Reynolds Numbers, *The Journal of Experimental Biology*, Vol. 207, pp. 1063-1072.

- [29] Birch, J.M., and Dickinson, M.H., Spanwise Flow and the Attachment of the Leading-edge Vortex on Insect Wings, *Nature*, Vol. 412, 2001, pp. 729-733
- [30] Sane, S.P., and Dickinson, M.H., The Aerodynamic Effects of Wing Rotation and a Revised Quasi-steady Model of Flapping Flight, *The Journal of Experimental Biology*, Vol. 205, 2002, pp. 1087-1096.
- [31] Ellington, C.P., The Novel Aerodynamics of Insect Flight: Applications to Micro-Air Vehicles, *The Journal of Experimental Biology*, Vol. 202, 1999, pp. 3439-3448.
- [32] Fry, S.N., Sayaman, R., and Dickinson, M.H., The Aerodynamics of Free-Flight Maneuvers in *Drosophila*, *Science*, Vol. 300, No. 5618, 2003, pp. 495-498.
- [33] Vandenberghe, N., Zhang, J., and Childress, S., Symmetry-Breaking Leads to Forward Flapping Flight, *Journal of Fluid Mechanics*, Vol. 506, 2004, pp. 147-155.
- [34] Beal, D.N., Hover, F.S., Triantafyllou, M.S., Liao, J.C., and Lauder, G.V., Passive Propulsion in Vortex Wakes, *Journal of Fluid Mechanics*, Vol. 549, 2006, pp. 385-402.
- [35] Heathcote, S., Martin, D., and Gursul, I., Flexible Flapping Airfoil Propulsion at Zero Freestream Velocity, *AIAA Journal*, Vol. 42, No. 11, 2004, pp. 2196-2204.
- [36] Heathcote, S., Wang, Z., and Gursul, I., Effect of Spanwise Flexibility on Flapping Wing Propulsion, *Journal of Fluids and Structures*, Vol. 24, 2008, pp. 183-199.
- [37] Tuncer, I.H., and Platzer, M.F., Thrust Generation due to Airfoil Flapping, *AIAA Journal*, Vol. 34, No.2, 1996, pp. 324-331.
- [38] Jones, K.D., Dohring, C.M., and Platzer, M.F., Experimental and Computational Investigation of the Knoller-Betz Effect, *AIAA Journal*, Vol. 36, No. 7, 1998, pp. 1240-1246.
- [39] Young, J., Numerical Simulation of the Unsteady Aerodynamics of Flapping Foils, *Ph.D Thesis*, School of Aerospace, Civil and Mechanical Engineering, University of New South Wales, Australia, 2005
- [40] Liu, H., and Kawachi, K., A Numerical Study of Insect Flight, *Journal of Computational Physics*, Vol. 146, 1998, pp. 124-156.
- [41] Ramamurti, R., and Sandberg, W., Simulation of Flow About Flapping Airfoils Using Finite Element Incompressible Flow Solver, *AIAA Journal*, Vol. 39, No.2, 2001, pp.253-352.

- [42] Wang, Z.J., and Xu, S., An Immersed Interface Method for Simulating the Interaction of a Fluid with Moving Boundaries, *Journal of Computational Physics*, Vol. 216, No. 2, 2006, pp. 454-493..
- [43] Bozkurttas, M., Dong, H., Seshadri, V., Mittal, R., and Najjar, F., Towards Numerical Simulation of Flapping Foils on Fixed Cartesian grids, 43rd *AIAA Aerospace Sciences Meeting and Exhibit*, Reno, NV, AIAA paper 2005-79, 2005.
- [44] Tuncer, I.H., and Platzler, M.F., Computational Study of Flapping Foil Aerodynamics, *Journal of Aircraft*, Vol. 37, No. 3, 2000, pp. 514-520.
- [45] Sengupta, T.K., Vikas, V., and Johri, A., An Improved Method for Calculating Flow Past Flapping and Hovering Airfoils, *Theo. Comput. Fluid Dyn*, Vol. 19, No. 6, 2005, pp. 417-440.
- [46] Shyy, W., and Liu, H., Flapping Wings and Aerodynamic Lift : The Role of Leading-Edge Vortices, *AIAA Journal*, Vol. 45, No. 12, 2007, pp. 2817-2819.
- [47] Shyy, W., Lian, Y., Tang, J., Viieru, D., and Liu, H., Aerodynamics of Low Reynolds Number Flyers, *Cambridge University Press*, New York, 2008.
- [48] Dong, H., Harff, M., and Mittal, R., Vortex Structures and Aerodynamic Performance of Finite-Aspect-Ratio Flapping Wings in Hovering Motion, 37th *AIAA Fluid Dynamics Conference and Exhibit*, Miami, FL, AIAA paper 2007-4210, 2007.
- [49] Aono, H., Shyy, W., and Liu, H., Vortex Dynamics in the Near Wake of a Hovering Hawkmoth, 46th *AIAA Aerospace Sciences Meeting and Exhibit*, Reno, NV, AIAA paper 2008-583, 2008.
- [50] Alben, S., and Shelly, M., Coherent Locomotion as an Attracting State for a Free Flapping Body, *Proc.Natl.Acad.Sci*, Vol. 102, No. 32, 2005, pp. 11163-11166.
- [51] Bergou, A.J., Sheng, X., and Wang, Z.J., Passive Wing Pitch Reversal in Insect Flight, *Journal of Fluid Mechanics*, Vol. 591, 2007, pp. 321-337.
- [52] Zhu, L., and Peskin, C.S., Simulation of a Flapping Flexible Filament in a Flowing Soap Film by the Immersed Boundary Method, *Journal of Computational Physics*, Vol. 179, 2002, pp. 452-468.
- [53] Farnell, D.J.J., David, T., and Barton, D.C., Numerical Simulations of a Filament in a Flowing Soap Film, *International Journal for Numerical Methods in Fluids*, Vol. 44, 2004, pp. 313-330.
- [54] ANSYS® FLUENT Software Package, Release 12, ANSYS, Inc.
- [55] Pederzani, J., and Haj-Hariri, H., A Numerical Method for the Analysis of Flexible

Bodies in Unsteady Viscous Flows, *International Journal for Numerical Methods in Engineering*, Vol. 68, 2006, pp. 1096-1112.

- [56] Persson, P.-O., and Peraire, J., A Higher Order Discontinuous Galerkin Method for Fluid-Structure Interaction, 45th *AIAA Aerospace Sciences Meeting and Exhibit*, Reno, NV, AIAA paper 2007-0513, 2007.
- [57] Tang, J., Viieru, D., and Shyy, W., A Study of Aerodynamics of Low Reynolds Number Flexible Airfoils, 37th *AIAA Fluid Dynamics Conference*, Miami, FL, AIAA paper 2007-4212, 2007.
- [58] Tang, J., Chimakurthi, S., Palacios, R., Cesnik, E.S.C., and Shyy, W., Computational Fluid-Structure Interaction of a Deformable Flapping Wing for Micro Air Vehicle Applications, 46th *AIAA Aerospace Sciences Meeting and Exhibit*, Reno, NV, AIAA Paper 2008-615, 2008.
- [59] Toomey, J., and Elredge, J.D., Numerical and Experimental Investigation of the Role of Flexibility in Flapping Wing Flight, 36th *AIAA Fluid Dynamics Conference and Exhibit*, San Fransisco, CA, AIAA paper 2006-3211, 2006.
- [60] Stanford, B., Vieeru, D., Albertani, R., Shyy, W., and Ifju, P., A Numerical and Experimental Investigation of Flexible Micro Air Vehicle Wing Deformation, 44th *AIAA Aerospace Sciences Meeting and Exhibit*, Reno, NV, AIAA paper 2006-440, 2006.
- [61] Singh, B., and Chopra, I., Insect-Based Hover-Capable Flapping Wings for Micro Air Vehicles : Experiments and Analysis, *AIAA Journal*, Vol. 46, No. 9, 2008, pp. 2115-2135.
- [62] Lian, Y., Membrane and Adaptively Shaped Wings For Micro-Air Vehicles, *Ph.D Thesis*, University of Florida, 2003.
- [63] Ericsson, L.E., and Reding, J.P., Fluid Dynamics of Unsteady Separated Flow, Part II, Lifting Surfaces, *Progress in Aerospace Sciences*, Vol. 24, 1987, pp. 249-356.
- [64] Carr, L.W., and Chandrasekhara, M.S., Compressibility Effects on Dynamic Stall, *Progress in Aerospace Sciences*, Vol. 32, No. 6, 1996, pp. 523-573.
- [65] Piziali, R.A., 2-D and 3-D Oscillating Wing Aerodynamics for a Range of Angles of Attack Including Stall, *NASA-TM-4632*, 1994.
- [66] Moore, F.K., Lift Hysteresis at Stall as an Unsteady Boundary Layer Phenomenon, *NACA-TR-1291*, 1956.
- [67] Takallu, M.A., and Williams, J.C., Lift Hysteresis of an Oscillating Slender Ellipse, *AIAA Journal*, Vol. 22, No. 12, 1984, pp. 1733-1741.

- [68] Johansen, J., Unsteady Airfoil Flows with Application to Aeroelastic Stability, *Riso national laboratories, Report – 1116*, 1999.
- [69] Leishman, J.G., and Beddoes, T.S., A Semi-Empirical Model for Dynamic Stall, *Journal of the American Helicopter Society*, Vol. 34, pp. 3-17.
- [70] Ekaterinaris, J.A., and Platzer, M.F., Computational Prediction of Airfoil Dynamic Stall, *Progress in Aerospace Sciences*, Vol. 33, No. 11, 1998, pp. 759-846.
- [71] Spentzos, A. et al., CFD Study of 3D Dynamic Stall, *43rd AIAA Aerospace Sciences Meeting and Exhibit*, Reno, NV, AIAA Paper 2005-1107, 2005.
- [72] Spentzos, A. et al., Modeling 3D Dynamic Stall using CFD and Neural Networks, *44th AIAA Aerospace Sciences Meeting and Exhibit*, Reno, NV, AIAA Paper, 2006-1071, 2006.
- [73] A. Le Papu., and J. Le Canu., 3D Navier Stokes Computations of a Stall Regulated Wind Turbine, *Wind Energy*, Vol. 7, No. 4, 2004, pp. 309-324.
- [74] Anderson, J.M., Streitlien, K., Barret, D.S., and Triantafyllou, M.S., Oscillating Foils of High Propulsive Efficiency, *Journal of Fluid Mechanics*, Vol. 360, 1998, pp. 41-72.
- [75] Wu, J.C., Theory for Aerodynamic Force and Moment in Viscous Flows, *AIAA Journal*, Vol. 19, No. 4, 1981, pp. 432-441.
- [76] Causin, P., Gerbeau, J.F., and Nobile, F., Added-Mass Effect in the Design of Partitioned Algorithms for Fluid-Structure Problems, *Comput. Methods Appl. Mech. Engrg.*, Vol. 194, 2005, pp. 4506-4527.
- [77] Zhu, Q., Numerical Simulation of a Flapping Foil with Chordwise or Spanwise Flexibility, *AIAA Journal*, Vol. 45, No. 10, 2007, pp. 2448-2457.
- [78] Chimakurthi, S.K., Tang, J., Palacios, R., Cesnik, C.E.S., and Shyy, W., Computational Aeroelasticity Framework for Analyzing Flapping Wing Micro Air Vehicles, *49th AIAA Structures, Structural Dynamics and Materials Conference*, Schaumburg, IL, AIAA Paper 2008-1814.
- [79] Gluck, M., Breuer, M., Durst, F., Halfmann, A., and Rank, E., Computation of Fluid-Structure Interaction on Lightweight Structures, *Journal of Wing Engineering and Industrial Aerodynamics*, Vol. 89, 2001, pp. 1351-1368.
- [80] Shin, S., Bae, S.Y., Kim, I.C., Kim, Y.J., and Goo, J.S., Computations of Flow over a Flexible Plate Using the Hybrid Cartesian/Immersed Boundary Method, *International Journal for Numerical Methods in Fluids*, Vol. 55, 2007, pp. 263-282.

- [81] Balint, T.S., and Lucey, A.D., Instability of a Cantilevered Flexible Plate in Viscous Channel Flow, *Journal of Fluids and Structures*, Vol. 20, 2005, pp. 893-912.
- [82] Wood, C., Gil, A. J., Hassan, O., and Bonet, J., A Partitioned Coupling Approach for Dynamic Fluid-Structure Interaction with Applications to Biological Membranes, *International Journal for Numerical Methods in Fluids*, Vol .57, No. 5, 2008, pp. 555-581.
- [83] Henshaw, W.D., OverBlown/cgins: A Fluid Flow Solver for Overlapping grids, Reference Guide, Version 1.0, Univ. of California Radiation Lab., Paper MA-134289, 2003, pp. 1-51.
- [84] Brown, D.L., Chesshire, G.S., Henshaw, W.D., and Quinlan, D.J., Overture: An Object Oriented Software System for Solving Partial Differential Equations in Serial and Parallel Environments, *Eighth Conference on Parallel Processing for Scientific Computing*, Society for Industrial and Applied Mathematics Paper CP97, 1997.
- [85] Henshaw, W.D., Ogen : An Overlapping Grid Generator for Overture, UCRL-MA-132237, 2003, pp. 1-78.
- [86] Quinlan, D., A++/P++ Manual, UCRL-MA-136511, 2000, pp. 1-111.
- [87] Thomas, P.D., and Lombard, C.K., Geometric Conservation Law and Its Application to Flow Computations on Moving Grids, *ALAA Journal*, Vol. 17, No. 10, 1979, pp. 1030-1037.
- [88] Liseikin, V.D., Grid Generation methods, (in) *Scientific Computation, Numerical Grid Generation*, Springer, Berlin, 1999.
- [89] Liu, G.R., Mesh Free Methods: Moving beyond the Finite Element Method, *CRC Press*, 2003.
- [90] Praveen, C., and Deshpande, S.M., A New Grid-Free Method for Conservation Laws, *Proceedings of the International Conference on Computational Fluid Dynamics 2*, Springer Verlag, 2002
- [91] Chesshire, G., and Henshaw, W.D., Composite Overlapping Meshes for the Solution of Partial Differential Equations, *Journal of Computational Physics*, Vol. 90, No.1, 1990, pp. 1-64.
- [92] Nakahashi, K., Ito, Y., and Togashi, F., Some Challenges of Realistic Flow Simulations by Unstructured Grid CFD, *International Journal for Numerical Methods in Fluids*, 43, 2003, pp. 769-783.

- [93] Deshpande, S.M., Kulkarni, P.S., and Ghosh, A.K., New Developments in Kinetic Schemes. *Computers and Mathematics with Applications*, Vol. 35, No.s 1-2, 1998, pp. 75-93
- [94] Suhs, Norman, E., Rogers, Stuart, E., and Dietz, W.E., PEGASUS 5 : An Automatic Pre-Processor for Overset-Grid CFD, 32nd *AIAA Fluid Dynamics Conference*, St.Louis, MI, AIAA paper 2002-3186, 2002.
- [95] Petersson, N.A., An Algorithm for Assembling Overlapping Grid Systems, *SIAM Journal of Scientific Computing*, Vol. 20, No. 6, 1999, pp. 1995-2022.
- [96] Chan, W.M., Overset Grid Technology Development at NASA Ames Research Center, *Computers and Fluids*, Vol. 38, No.3, 2009, pp. 496-503.
- [97] Pan, H., and Damodaran, M., Computational Hydrodynamics of Moving Underwater Rigid Bodies using Godunov-Projection method on Overlapping Grids, *Ph.D Thesis*, School of Mechanical and Aerospace Engineering, Nanyang Technological University, Singapore 2000.
- [98] Guerrero, J., Numerical Simulation of the Unsteady Aerodynamics of Flapping Flight, *Ph.D Thesis*, Department of Environmental Engineering, University of Genova, Italy, 2009.
- [99] Alabi, K., and Ladeinde, F., Higher Order Dynamic Overset Procedure Applied to Moving Body Calculations, 45th *AIAA Aerospace Sciences Meeting and Exhibit*, Reno, NV, AIAA paper 2007-248, 2007.
- [100] Chesshire, G., and Henshaw, W.D., A Scheme for Conservative Interpolation on Overlapping Grids, *SIAM Journal of Scientific Computing*, Vol. 15, No. 4, 1994, pp. 819-845.
- [101] Henshaw, W.D., The Overture Hyperbolic Grid Generator, User Guide Version 1.0, UCRL-MA-134240, 2003, pp. 1-35.
- [102] Fast, P., and Henshaw, W.D., Time Accurate Simulation of Viscous Flow Around Deforming Bodies Using Overset Grids, 15th *AIAA Computational Fluid Dynamics Conference*, Anaheim, CA, AIAA Paper 2001-2604, 2001.
- [103] Fast, P., and Shelley, M.J., A Moving Overset Grid Method for Interface Dynamics Applied to a Non-Newtonian Hele-Shaw Flow, *Journal of Computational Physics*, Vol. 195, 2004, pp. 117-142.
- [104] Demirdzic, I., and Peric, M., Space Conservation Law in Finite Volume Calculations of Fluid Flow, *International Journal for Numerical Methods in Fluids*, Vol. 8, 1988, pp. 1037-1050

- [105] Henshaw, W.D., Mappings for Overture: A Description of the Mapping Class and Documentation for Many Useful Mappings, UCRL-MA-132239, 2003, pp. 1-284.
- [106] Henshaw, W.D., A Fourth Order Accurate Method For the Incompressible Navier-Stokes Equations on Overlapping Grids, *Journal of Computational Physics*, Vol. 113, No. 13, 1994, pp. 13-25.
- [107] Bell, J.B., Colella, P., and Glaz, H.M., A Second-Order Projection Method for the Incompressible Navier-Stokes Equations, *Journal of Computational Physics*, Vol. 85, 1989, pp. 257-283.
- [108] Brown, D.L., and Minion, M.L., Performance of Under-Resolved Two-Dimensional Incompressible Flow Simulations, *Journal of Computational Physics*, Vol. 122, 1995, pp. 165-183
- [109] Minion, M.L., and Brown, D.L., Performance of Under-Resolved Two-Dimensional Incompressible Flow Simulations, II, *Journal of Computational Physics*, Vol. 138, 1997, pp. 734-765.
- [110] Brown, D.L., Cortez, R., and Minion, M.L., Accurate Projection Methods For The Incompressible Navier-Stokes Equations, *Journal of Computational Physics*, Vol. 168, 2001, pp. 464-499.
- [111] Henshaw, W.D., and Petersson, N.A., A Split-Step Scheme for the Incompressible Navier-Stokes Equations, *UCRL-JC-144040*, 2001.
- [112] Gresho, P.M., and Sani, R.L., *Incompressible Flow and the Finite Element Method*, New York, Wiley, 2000.
- [113] Henshaw, W.D., and Schwendeman, D.W., Moving Overlapping Grids with Adaptive Mesh Refinement for High-Speed Reactive and Non-Reactive Flow, *Journal of Computational Physics*, Vol. 216, No. 2, 2006, pp. 744-779.
- [114] Lilly, D.K., Numerical Simulation of Developing and Decaying of Two Dimensional Turbulence, *Journal of Fluid Mechanics*, Vol. 45, 1971, pp.395-415.
- [115] Orzag, S.A., Analytical Theories of Turbulence, *Journal of Fluid Mechanics*, Vol. 41, 1970, pp.363-386.
- [116] Henshaw, W.D., Kreiss, H.O., and Reyna, L.G., On the Smallest Scale for the Incompressible Navier-Stokes Equations, *NASA-CR-181628*, ICASE report NO. 88-8, 1988.
- [117] Balay, S., Buschelman, K., Eijkhout, V., Gropp, W.D., Kaushik, D., Knepley, M.G., McInnes, L.C., Smith, B.F., and Zhang, H., "PETSc Users Manual", ANL-95/11 - Revision 3.0.0, Argonne National laboratory, 2008.

- [118] Guide for Verification and Validation of Computational Fluid Dynamics Simulation, *AIAA G-077*, 1998.
- [119] Belmonte, A., Eisenberg, H., and Moses, E., From Flutter to Tumble: Inertial Drag and Froude Similarity in Falling Paper, *Physical Review Letters*, Vol. 81, No. 2, 1998, pp. 345-348.
- [120] Andersen, A., Pesavento, U., and Wang, Z.J., Unsteady Aerodynamics of Fluttering and Tumbling Plates, *Journal of Fluid Mechanics*, Vol. 541, 2005, pp. 65-90.
- [121] Roache, P.J., Verification and Validation in Computational Science and Engineering, Hermosa publishers, Albuquerque, NM, 1998.
- [122] Jones, J.D., Dohring, C.M., and Platzer, M.F., Wake Structures Behind Plunging Airfoils: A Comparison of Numerical and Experimental Results, 34th *AIAA Aerospace Sciences Meeting and Exhibit*, Reno, NV, AIAA Paper 96-0078, 1996.
- [123] Heathcote, S., and Gursul, I., Jet Switching phenomena for a Plunging Airfoil, 34th *AIAA Fluid Dynamics Conference and Exhibit*, Portland, OR, AIAA Paper 2004-2150, 2004.
- [124] McCroskey, Carr, W.J., and Mcalister, L.W., Dynamic Stall Experiments on Oscillating Foils, 13th *AIAA Aerospace Sciences Meeting and Exhibit*, AIAA paper 1975-125, 1975.
- [125] Visbal, M.R., and Gordnier, R.E., A Higher Order Flow Solver for Deforming and Moving Meshes, *Fluids 2000 Conference and Exhibit*, AIAA Paper 2000-2619, 2000.
- [126] Vandyken, R.D., Ekaterinaris, J.A., Chandrasekhara, M.S., and Platzer, M.F., Analysis of Compressible Light Dynamic Stall Flow at Transitional Reynolds Numbers, *AIAA Journal*, Vol. 34, No. 7, 1996, pp. 1420-1427.
- [127] Visbal, M.R., Shang, J.S., Investigation of the Flow Structure Around a Rapidly Pitching Airfoil, *AIAA Journal*, Vol. 27, No. 8, 1987, pp.1044-1051.
- [128] Lugt, H.I., Autorotation of an Elliptic Cylinder about an Axis Perpendicular to the Flow, *Journal of Fluid Mechanics*, Vol. 99, 1980, pp. 817-840.
- [129] Riabouchinsky, D.P., Thirty Years of Theoretical and Experimental Research in Fluid Mechanics, *Journal of the Royal Aeronautical Society*, Vol. 39, 1935, pp. 282-348.
- [130] Jeong, J., and Hussain, F., On the Identification of a Vortex, *Journal of Fluid Mechanics*, Vol. 285, 1995, pp. 69-94.

- [131] Smith, E.H., Autorotating Wings: An Experimental Investigation, *Journal of Fluid Mechanics*, Vol. 50, 1971, pp. 513-534.
- [132] Mittal, R., Seshadri, V., and Udaykumar, H., Flutter, Tumble and Vortex Induced Autorotation, *Theoretical and Computational Fluid Dynamics*, Vol. 17, 2004, pp. 165-170.
- [133] Wei, G., An Implicit Method to Solve Problems of Rigid Body Motion Coupled with Fluid Flow, FLOW-3D, FloSci Report TN-76, 2006, pp. 1-18;
- [134] Ashraf, M.A., Lai, J.C.S., and Platzer, M.F., Numerical Analysis of Flapping Wing Aerodynamics, *16th Australasian Fluid Mechanics Conference*, Gold Coast, Australia, Dec 2007.
- [135] Lan, S.L., Sun M., Aerodynamic Properties of a Wing Performing Unsteady Rotational Motions at Low Reynolds Numbers, *Acta Mech*, Vol. 13, 2001, pp. 135-147.
- [136] Miller, L.A., and Peskin, C.S., When Vortices Stick: An Aerodynamic Transition in Tiny Insect Flight, *The Journal of Experimental Biology*, Vol. 207, pp. 3073-3088, 2004.
- [137] Sankar, and N.L., Tang, W., Numerical Solution of Unsteady Viscous Flow Past Rotor Sections, *23rd AIAA Aerospace Sciences Meeting and Exhibit*, Reno, NV, AIAA Paper 85-0129, 1985.
- [138] Ventsel, E., and Krauthammer, T., *Thin Plates and Shells: Theory: Analysis, and Applications*, Marcel Dekker Inc, New York, 2001
- [139] Voltera, E., and Zachmanoglou, E. C. *Dynamics of Vibrations*. Columbus, Charles E. Merrill Books, Inc., 1965.
- [140] Kuttler, U., and Wall, W.A., "The Dilemma of Domain Decomposition Approaches in Fluid-Structure Interactions with Fully Enclosed Incompressible Fluids", *Fluid Structure Interaction- Modeling, Simulation and Optimization*, Lecture Notes in Computational Science and Engineering, Vol. 53, 2006, pp. 575-582.
- [141] Miao, J.-M., and Ho.,-M., Effect of Flexure On Aerodynamic Propulsive Efficiency of Flapping Flexible Airfoil, *Journal of Fluids and Structures*, Vol. 22, 2006, pp. 401-419.
- [142] Tuncer, I.H., and Kaya, M., Thrust Generation Caused By Flapping Airfoils in a Biplane Configuration, *Journal of Aircraft*, Vol. 40, 2003, pp. 509-515.

Appendix A1 - Navier-Stokes Equations and Non-Dimensional Parameters

The governing equations for unsteady, incompressible fluid flow are the Navier-Stokes equations given by,

$$\frac{\partial u}{\partial t} + u \cdot \nabla u = -\frac{\nabla p}{\rho} + \nu \Delta u \quad (\text{A1.1})$$

where u is the flow velocity, p , the static pressure, ρ the fluid density and ν , the fluid viscosity. To obtain the relevant non-dimensional parameters governing fluid motion, let us define the reference velocity, length and time scales as U_s , the freestream velocity, L , the chord length and t_s respectively. The non-dimensional parameters thus are defined by

$$\bar{u} = \frac{u}{U_s}, \quad \bar{x} = \frac{x}{L}, \quad \bar{t} = \frac{t}{t_s}, \quad \bar{p} = \frac{p}{\rho U_s^2} \quad (\text{A1.2})$$

Using Eq. (A1.2) in Eq. (A1.1), we obtain

$$\frac{\partial \bar{u}}{\partial \bar{t}} + \left(\frac{t_s U_s}{L} \right) \bar{u} \cdot \nabla \bar{u} = - \left(\frac{t_s U_s}{L} \right) \frac{\bar{\nabla} \bar{p}}{\rho} + \left(\frac{\nu t_s}{L^2} \right) \bar{\Delta} \bar{u} \quad (\text{A1.3})$$

When the body of interest is not moving, the obvious choice for the time scale would be $t_s = \frac{L}{U_s}$, which then gives the non-dimensional incompressible Navier-Stokes equation

that depend on Reynolds number $Re = \frac{U_s L}{\nu}$:

$$\frac{\partial \bar{u}}{\partial \bar{t}} + \bar{u} \cdot \nabla \bar{u} = -\frac{\bar{\nabla} \bar{p}}{\rho} + \left(\frac{1}{Re} \right) \bar{\Delta} \bar{u} \quad (\text{A1.4})$$

When there is a flapping / periodic motion, there exists a more relevant time scale which is associated with the periodic motion, the flapping frequency $\omega_R = 2\pi f$. If the time scale corresponding to the flapping motion $t_s = \frac{1}{\omega_R}$ is used in Eq. (A1.3), the non-

dimensional equations are given by:

$$k \frac{\partial \bar{u}}{\partial \bar{t}} + \bar{u} \cdot \nabla \bar{u} = -\frac{\bar{\nabla} \bar{p}}{\rho} + \left(\frac{1}{Re} \right) \bar{\Delta} \bar{u} \quad (\text{A1.5})$$

where $k = \frac{\omega_R L}{U_s}$ is defined as the reduced frequency. Unless otherwise stated elsewhere in this thesis, $k = \frac{\omega_R L}{2U_s}$ is used to define the reduced frequency of an object oscillating with a particular frequency to maintain consistency with existing data available in literature. Thus, there are two parameters, the Reynolds number and reduced frequency that govern flows around flapping wings. However, the above only assumes that the reference velocity U_s is constant and independent of the flapping frequency / amplitude. When there is a plunging motion of the airfoil / wing, it is more appropriate to consider the resultant velocity as the reference velocity, which is $U_s = \sqrt{U^2 + \omega_R^2 L^2 h_0^2}$ where U is the freestream velocity, and h_0 the non-dimensional plunge amplitude. If the governing equations are non-dimensionalized using the above velocity, we obtain,

$$\frac{k}{\sqrt{1+St^2}} \frac{\partial \bar{u}}{\partial t} + \bar{u} \cdot \nabla \bar{u} = -\frac{\bar{\nabla} \bar{p}}{\rho} + \left(\frac{1}{Re \sqrt{1+St^2}} \right) \bar{\Delta} \bar{u} \quad (A1.6)$$

where $St = kh_0$ is the Strouhal number and $k = \frac{\omega_R L}{U}$. It is to be noted that these non-dimensional numbers have been non-dimensionalized using a constant freestream velocity U and not U_s to maintain consistency with existing literature. With the above expression Eq. (A1.6), the freestream Reynolds number, reduced frequency and Strouhal number govern the fluid flow surrounding a flapping wing.

For the case of a falling body in an incompressible flow, the relevant time scale corresponds to that of an oscillating pendulum, $t_s = \sqrt{\frac{L}{g}}$, where g is the acceleration due to gravity. Using this expression in Eq. (A1.3), we obtain :

$$\frac{1}{Fr} \frac{\partial \bar{u}}{\partial t} + \bar{u} \cdot \nabla \bar{u} = -\frac{\bar{\nabla} \bar{p}}{\rho} + \left(\frac{1}{Re} \right) \bar{\Delta} \bar{u} \quad (A1.7)$$

where $Fr = \frac{U_s}{\sqrt{gL}}$ is the Froude number.

Appendix A2 - Solver Statistics

Some of the solver statistics for two- and three-dimensional computations are summarized below. The data includes a break up of the CPU time in seconds spent for each process in the computation.

Table A2.1: Solver Statistics for a Two-Dimensional Computation, over 7018 time steps for a Plunging Airfoil

Timings	Sec	Sec/Step	Sec/Step/Point	Percentage
Setup and initialize	1.204e+00	1.716e-04	7.159e-08	0.018
Compute du/dt	8.893e+02	1.267e-01	5.286e-05	13.32
Add forcing	9.671e-02	1.378e-05	5.749e-09	0.001
Boundary Conditions	1.138e+02	1.621e-02	6.762e-06	1.704
Interpolation	1.273e-01	1.814e-05	7.568e-09	0.0022
Moving Grids	5.827e+02	8.302e-02	3.464e-05	8.730
Moving update grids	6.997e+00	9.970e-04	4.160e-07	0.1049
Moving update operators	1.789e+02	2.550e-02	1.064e-05	2.681
Moving update Interpolant	1.534e+00	2.186e-04	9.119e-08	0.023
Interpolate exposed pts	3.248e+01	4.629e-03	1.931e-06	0.487
Implicit time stepping	1.670e+03	2.380e-01	9.929e-05	25.027
Compute dt	7.450e+01	1.062e-02	4.429e-06	1.115
Pressure Solve	2.363e+03	3.367e-01	1.405e-04	35.410
Output File	7.542e+02	1.075e-01	4.483e-05	11.30
Total time	6.673e+03	9.508e-01	3.858e-04	100

Table A2.2: Solver Statistics for a Three-Dimensional Computation, over 2 time steps for a Flapping Wing

Timings	Sec	Sec/Step	Sec/Step/Point	Percentage
Setup and initialize	1.856e+02	9.278e+01	3.555e-04	13.98
Compute du/dt	1.643e+02	8.213e+01	3.147e-04	12.37
Add forcing	7.629e-05	3.815e-05	1.462e-10	0.000
Boundary Conditions	6.838e+00	3.419e+00	1.310e-05	0.514
Interpolation	3.768e+00	1.884e+00	7.219e-06	0.284
Moving Grids	7.760e+01	3.880e+01	1.487e-04	5.845
Moving update grids	5.005e-03	2.502e-03	9.589e-09	0.000
Moving update operators	8.682e+00	4.341e+00	1.663e-05	0.653
Moving update Interpolant	6.714e-04	3.357e-04	1.286e-09	0.000
Interpolate exposed pts	1.513e+01	7.565e+00	2.899e-05	1.139
Implicit time stepping	1.821e+02	9.106e+01	3.489e-04	13.72
Compute dt	2.109e+01	1.055e+01	4.041e-05	1.588
Pressure Solve	6.583e+02	3.291e+02	1.261e-03	49.58
Output File	8.681e+00	4.341e+00	1.663e-05	0.653
Total time	1.332e+03	6.66e+02	2.543e-03	100