

UC Berkeley

UC Berkeley Electronic Theses and Dissertations

Title

High-Performance Discrete-Vortex Algorithms for Unsteady Viscous-Fluid Flows near Moving Boundaries

Permalink

<https://escholarship.org/uc/item/11b3z1t9>

Author

Wang, Lu

Publication Date

2016

Peer reviewed|Thesis/dissertation

**High-Performance Discrete-Vortex Algorithms for Unsteady Viscous-Fluid
Flows near Moving Boundaries**

by

Lu Wang

A dissertation submitted in partial satisfaction of the

requirements for the degree of

Doctor of Philosophy

in

Engineering - Mechanical Engineering

in the

Graduate Division

of the

University of California, Berkeley

Committee in charge:

Professor Ronald W. Yeung, Chair

Professor Stephen Morris

Associate Professor Per-Olof Persson

Fall 2016

**High-Performance Discrete-Vortex Algorithms for Unsteady Viscous-Fluid
Flows near Moving Boundaries**

Copyright 2016
by
Lu Wang

Abstract

High-Performance Discrete-Vortex Algorithms for Unsteady Viscous-Fluid Flows near Moving Boundaries

by

Lu Wang

Doctor of Philosophy in Engineering - Mechanical Engineering

University of California, Berkeley

Professor Ronald W. Yeung, Chair

A high-performance Discrete-Vortex Method (DVM) is successfully developed and implemented to directly simulate two-dimensional low to medium-Reynolds number flows around multiple arbitrarily shaped bodies undergoing either prescribed or free motions. The deterministic Viscous-Vortex-Domain (VVD) formulation is adopted to simulate vorticity diffusion. Through the use of CPU and Graphics-Processing-Unit (GPU) parallel computing, significant speedup of the simulation compared to a serial implementation on a CPU is achieved. The validity of the present DVM simulation is confirmed by comparing the present results with published ones for a variety of test cases. The current implementation of DVM has been used to study two novel flow problems of practical interest and has led to significant findings.

First, the full and partial ground effects on the lift generation of a flapping (air)foil in normal hovering mode are investigated. To achieve full ground effect, the foil of chord c is made to hover above the center of a finite-sized platform of length $10c$. The computed force-enhancement, force-reduction, and force-recovery regimes at low, medium, and high ground clearances are observed to be in line with existing literature. This research puts special focus on partial ground effects when the foil is hovering near the edge of the platform. Lift-modifying mechanisms not previously observed under full ground effect have been discovered. When stroke reversal of the flapping occurs near the edge of the platform, a relatively stationary strong vortex may form above the platform edge. This strong vortex can either increase or decrease the instantaneous lift force on the foil depending on the position of the foil relative to the platform edge. Further, the platform edge may lead to the formation of an additional vortex pair which increases the instantaneous lift force as the foil sweeps past the edge under certain suitable conditions. Lastly, the platform edge can lead to the formation of a reverse von Kármán vortex street that extends well below the stroke plane under suitable geometric arrangements.

Second, the flow past a Bach-type vertical-axis wind or current turbine is simulated using the DVM at a Reynolds number of 1,500. The main purpose of the study is to evaluate the

suitability of Bach-type turbines for use as micro-scale energy harvesters that can be applied to power, for example, sensor nodes of a Wireless Sensor Network. Through simulations, the maximum power coefficient of the turbine operating at a prescribed constant tip-speed ratio is found to be 0.18, which is comparable to the performance of a turbine of the same geometry at much higher Reynolds numbers. This indicates that there is only minimal performance penalty for miniaturization. The angular velocity of the turbine has a strong influence on the evolution of vortical flow structures. A new wake-capturing mechanism that boosts the performance of the turbine is discovered from the simulations for a certain range of tip-speed ratios where the vortex shed by the advancing blade helps drive the returning blade. In addition to the condition of prescribed rotation, free rotation of a steel Bach-type turbine under a steady current in water is also investigated. Significant fluctuation in angular velocity over one period of rotation is observed. This speed fluctuation is found to be detrimental to energy extraction, reducing the maximum power coefficient to approximately 0.16. This level of power-generation capability implies that such micro-scale turbines can significantly extend the life expectancy of a wireless sensor node or even maintain the node in a low-power state indefinitely.

Contents

Contents	i
List of Figures	iii
List of Tables	ix
1 Introduction	1
1.1 Discrete-Vortex Method	2
1.1.1 Random-Vortex Method	2
1.1.2 Deterministic models for diffusion	3
1.2 Current DVM implementation	5
1.3 Applications of DVM simulations	6
1.3.1 Flapping foil with ground effects	6
1.3.2 Micro-scale Bach-type turbine	7
1.4 Research objectives	9
2 Description of numerical method	10
2.1 Problem description and general setup	10
2.2 Choice of vortex blob structure	11
2.3 Motion of vortex blobs	12
2.3.1 Treatment of advection process	13
2.3.2 Treatment of diffusion process	18
2.4 Creation of vortex blobs and enforcement of boundary conditions	25
2.5 Hydrodynamic force and moment	30
2.5.1 Pressure force and moment	30
2.5.2 Viscous force and moment	32
2.6 Free motion of rigid bodies	33
2.7 Merging and decay of vortex blobs	35
2.7.1 Merging of vortex blobs	35
2.7.2 Decay of vortex blobs	37
2.8 General numerical procedure	37
3 Validation of the DVM algorithm	40
3.1 Flow past a translating cylinder	40
3.2 Flow past two translating cylinders in tandem	46
3.3 Flow past a translating and rotating ellipse	49
3.4 Flow near a flapping foil in normal hovering mode	51
3.5 Clap-and-fling motion	53

3.6	Two-dimensional Vortex-Induced Vibration of an elastically mounted circular cylinder	56
4	Investigation of full and partial ground effects on a flapping foil hovering above a finite-sized platform	63
4.1	Problem description	63
4.2	Numerical results	64
4.2.1	Full ground effect	66
4.2.2	Partial ground effect	71
5	Performance of a micro-scale Bach-type turbine	86
5.1	Problem description	86
5.2	Numerical results	88
5.2.1	Prescribed rotation	89
5.2.2	Free rotation	100
5.3	Potential applications	104
6	Conclusions	106
6.1	Implementation and validation of the Discrete-Vortex Method	106
6.2	Flapping wing with ground effects	107
6.3	Micro-scale Bach-type turbine	108
6.4	Summary of major contributions	109
	Bibliography	110

List of Figures

2.1	Schematic of the viscous-fluid flow problem with submerged bodies.	11
2.2	Example of quadtree construction for the boundary layer and wake of a translating cylinder. The blue dots represent vortex blobs, and the black boxes are childless nodes (boxes) of the quadtree.	15
2.3	Block for diffusion velocity (Dots represent vortex blobs and the “+” sign marks the calculation point) [80].	22
2.4	k -d tree and quadtree data structures for neighbor search.	24
2.5	Flowchart of the DVM simulation with prescribed body motions.	38
2.6	Flowchart of the DVM simulation with free body motions. Differences from the procedure for prescribed motions are highlighted in red.	39
3.1	A translating circular cylinder.	40
3.2	Time history of the drag coefficient of a rapidly started circular cylinder at $Re = 100$	42
3.3	Time history of the lift coefficient of a rapidly started circular cylinder at $Re = 100$	42
3.4	Time-averaged pressure coefficient as a function of angular position of a steadily translating circular cylinder at $Re = 100$. The forward stagnation point corresponds to $\theta = 0$	43
3.5	Time histories of the lift and drag coefficients of a rapidly started circular cylinder at $Re = 1,000$	43
3.6	Time histories of the lift and drag coefficients of a rapidly started circular cylinder at $Re = 9,500$. Cylinder contour consists of 800 panels. The numerically computed mean drag coefficient from [89] is included for comparison.	44
3.7	Non-dimensional vorticity fields ($\omega D/U_0$) at various time instants after the rapid start of the cylinder. $Re = 100$	45
3.8	Non-dimensional vorticity fields ($\omega D/U_0$) at various time instants after the rapid start of the cylinder. $Re = 1,000$	45
3.9	Non-dimensional vorticity fields ($\omega D/U_0$) at various time instants after the rapid start of the cylinder. $Re = 9,500$. Cylinder contour consists of 1,200 panels.	46
3.10	Non-dimensional vorticity fields ($\omega D/U_0$) of the wake of a translating cylinder at (a) $Re = 100$, (b) $Re = 1,000$, and (c) $Re = 9,500$. For $Re = 9,500$, 800 panels are used.	47
3.11	Two circular cylinders translating in tandem.	48
3.12	Non-dimensional vorticity field ($\omega D/U_0$) of the wake of two translating cylinders in tandem at $Re = 100$ and $U_0 t/D = 175$. Center-to-center separation between the two cylinders is $s/D = 2$	48

3.13	Time histories of the lift and drag coefficients of two rapidly started circular cylinders in tandem at $Re = 100$. Center-to-center separation between the two cylinders is $s/D = 2$	48
3.14	Time-averaged pressure coefficients as functions of angular position of two steadily translating circular cylinders in tandem at $Re = 100$. Center-to-center separation between the two cylinders is $s/D = 2$. Curves are current DVM results, and symbols are results from Sharman <i>et al.</i> [85].	49
3.15	A translating and rotating elliptic cylinder.	50
3.16	Time histories of the lift and drag coefficients of a translating and rotating elliptic cylinder. (a) Time histories from impulsive start to the end of the tenth rotation cycle. (b) The variations of drag and lift coefficients over the tenth period. Curves are current solutions, and symbols are results of Zhang & Liang [93]. $Re = 200$ and $\lambda = \pi/4$	50
3.17	Non-dimensional vorticity fields ($\omega D/U_0$) during the tenth revolution of the translating and rotating ellipse. $Re = 200$ and $\lambda = \pi/4$	51
3.18	An elliptic foil flapping in normal hovering mode. The x -axis coincides with the stroke plane. (a) Downstroke. (b) Upstroke. The foil orientation shown corresponds to symmetric rotation with $\phi = 0$	52
3.19	Time histories of the hydrodynamic force coefficients on the flapping foil in hovering. (a) Lift coefficient. (b) Drag coefficient. $Re = 75$, $\phi = 0$	53
3.20	Time histories of the hydrodynamic force coefficients on the flapping foil in hovering. (a) Lift coefficient. (b) Drag coefficient. $Re = 75$, $\phi = \pi/4$	54
3.21	Non-dimensional vorticity fields ($\omega c/U_{max}$) during the fourth flapping cycle. $Re = 75$ and $\phi = 0$	55
3.22	Non-dimensional vorticity fields ($\omega c/U_{max}$) during the fourth flapping cycle. $Re = 75$ and $\phi = \pi/4$	56
3.23	Two elliptic foils performing the clap-and-fling motion. (a) Fling. (b) Clap. . . .	57
3.24	The non-dimensional translational and rotational velocities of the right foil in clap-and-fling motion. The translational velocity and rotational velocity are normalized by U_{max} and U_{max}/c , respectively. During the fling phase, the foil pivots about the trailing edge (TE), and the translational velocity of the trailing edge is given. Similarly, during the clap phase, the foil rotates about the leading edge (LE), and the translation velocity of the leading edge is shown instead. The foil motion shown here is identical to the case with rotational overlap $\xi = 50\%$ in [28].	58
3.25	Time history of the lift coefficient of the right foil over a single clap-and-fling cycle at $Re = 128$. The lift coefficient of the left wing is identical to that of the right. The result from [28] is used for comparison.	59
3.26	Instantaneous vorticity fields near the foil at several time instants during the fling motion: $t/T =$ (a) 0.35, (b) 0.41, and (c) 0.44. Panels (d)-(f) show the vorticity fields at approximately the same time instants extracted from [28].	59

3.27	Instantaneous vorticity fields near the foil at several time instants during the clap motion: $t/T =$ (a) 0.55, (b) 0.59, (c) 0.63, and (d) 0.97. Panels (e)-(h) show the vorticity fields at approximately the same time instants extracted from [28].	60
3.28	A circular cylinder translating in the x -direction undergoing VIV in the y -direction.	60
3.29	Maximum excursion of a two-dimensional spring-mounted circular cylinder undergoing VIV. $Re = 150$, $M_{red} = 2$, and $\zeta = 0$. The results from [95, 96] are used for comparison.	61
3.30	Phase relation between lift and excursion at periodic steady state of a two-dimensional spring-mounted circular cylinder undergoing VIV. $Re = 150$, $M_{red} = 2$, and $\zeta = 0$	61
3.31	Normalized VIV frequency as a function of the reduced velocity, U_{red} . For current DVM solution, $M_{red} = 2$, $Re = 150$, and $\zeta = 0$. The results for $M_{red} = 1.57$, $Re = 150$, and $\zeta = 0.007$ from [97] are included for comparison.	62
3.32	Non-dimensional vorticity field ($\omega D/U_0$) of the wake of a circular cylinder undergoing VIV. The cylinder is near maximum excursion. The dashed line shows the range of cylinder center excursion. $M_{red} = 2$, $U_{red} = 4$, $\zeta = 0$, and $Re = 150$. The horizontal and vertical scales are x/D and y/D , respectively.	62
4.1	An elliptic airfoil flapping in normal hovering mode above the edge of a flat platform. The dashed line represents the stroke plane. (a) Downstroke. (b) Upstroke.	64
4.2	Time-dependent (a) lift and (b) drag coefficients over the first seven periods for $S/c = 0$ and $D/c = 1$. The configurations of the coarse, medium, and fine cases are listed in Table 4.1.	65
4.3	Time histories of the lift and drag coefficients for $S/c = 0.25$ and $D/c = 1$. The top curve is the lift coefficient.	66
4.4	Time-averaged lift and drag coefficients, $\overline{C_L}$ and $\overline{C_D}$, of the flapping elliptic airfoil in normal hovering mode above the center of the platform ($S/c = -5$). The time-averaged lift and drag coefficients of the same airfoil in infinite fluid are $\overline{C_{L\infty}}$ and $\overline{C_{D\infty}}$, respectively.	67
4.5	Time-dependent (a) lift coefficient and (b) drag coefficient of the flapping elliptic airfoil over one period of flapping motion under full ground effect ($S/c = -5$). Downstroke occurs over $t/T = 0$ to 0.5, and upstroke occurs over $t/T = 0.5$ to 1. Stroke reversals occur at $t/T = 0(1)$ and 0.5.	68
4.6	Non-dimensional vorticity fields ($\omega c/U_{max}$) during downstroke for $S/c = -5$ and (a) $D/c = 1$, (b) $D/c = 3$, and (c) $D/c = 5$	69
4.7	Non-dimensional vorticity fields ($\omega c/U_{max}$) during upstroke for $S/c = -5$ and (a) $D/c = 1$, (b) $D/c = 3$, and (c) $D/c = 5$	70
4.8	Time-averaged (a) lift coefficients and (b) drag coefficients of a flapping elliptic airfoil in normal hovering mode near the edge of a platform.	72
4.9	Time-dependent lift coefficient of the flapping elliptic airfoil over one period of flapping motion in the force-enhancement regime ($D/c = 1$).	73

4.10	Non-dimensional vorticity fields ($\omega c/U_{max}$) for $S/c = 0.75$ and $D/c = 1$ at $t/T =$ (a) 0.35, (b) 0.55, and (c) 0.80.	73
4.11	Non-dimensional velocity magnitude normalized by U_{max} (color) and streamlines (dark curves) for $D/c = 1$ with (a) $S/c = 0.75$ and (b) $S/c = 2$ at $t/T = 0.35$. .	74
4.12	Non-dimensional vorticity fields ($\omega c/U_{max}$) for $S/c = -0.75$ and $D/c = 1$ at $t/T =$ (a) 0.1, (b) 0.55, and (c) 0.80.	74
4.13	Non-dimensional vorticity field ($\omega c/U_{max}$) for $S/c = -1.75$ and $D/c = 1$ at $t/T = 0.35$. The positive TEV shed from the current downstroke is merging with the TEV shed from the previous downstroke.	75
4.14	Non-dimensional vorticity fields ($\omega c/U_{max}$) at $t/T = 0.6$ for $D/c = 1$ and $S/c =$ (a) -1.25 , (b) -1.75 , and (c) -3	75
4.15	Non-dimensional velocity magnitude normalized by U_{max} (color) and streamlines (dark curves) for $D/c = 1$ with (a) $S/c = -1.25$ and (b) $S/c = -1.75$ at $t/T = 0.6$. 76	76
4.16	Time-dependent lift coefficient of the flapping elliptic airfoil over one period of flapping motion in the force-reduction regime ($D/c = 2$).	77
4.17	Non-dimensional vorticity fields ($\omega c/U_{max}$) at $t/T = 0.3$ for $D/c = 2$ and $S/c =$ (a) 1 and (b) 2.	77
4.18	Non-dimensional vorticity fields ($\omega c/U_{max}$) for $S/c = 0.5$ and $D/c = 2$ at $t/T =$ (a) 0.1, (b) 0.3, and (c) 0.6.	78
4.19	Non-dimensional vorticity fields ($\omega c/U_{max}$) for $S/c =$ (a) -2 and (b) 0 and $D/c =$ 2 at $t/T = 0$ and 0.3.	78
4.20	Non-dimensional velocity magnitude normalized by U_{max} (color) and streamlines (dark curves) for $D/c = 2$ with (a) $S/c = -2$ and (b) $S/c = 0$ at $t/T = 0$	79
4.21	Time-dependent lift coefficient of the flapping elliptic airfoil over one period of flapping motion in the force-reduction regime ($D/c = 3$).	80
4.22	Non-dimensional vorticity fields ($\omega c/U_{max}$) for $S/c =$ (a) 0 and (b) -2 and $D/c =$ 3 at $t/T = 0.1, 0.3$, and 0.8.	81
4.23	Efficiencies of lift generation in terms of the ratio $\overline{C_L}/\overline{C_P}$ for various values of S/c and D/c	82
4.24	Time-averaged lateral-force coefficient $\overline{C_x}$ of a flapping elliptic airfoil in normal hovering mode near the edge of a platform.	83
4.25	Non-dimensional vorticity fields ($\omega c/U_{max}$) of the wake of the flapping airfoil in normal hovering mode for $S/c =$ (a) 0.5, (b) 1, and (c) 2 and $D/c = 1$ at $t/T = 0$. Increment of contour level is 0.25 for the solid and dashed contour lines. The influence of the platform edge leads to the formation of a reverse von Kármán vortex street.	84
4.26	Non-dimensional vorticity field ($\omega c/U_{max}$) of the wake of the flapping airfoil in normal hovering mode in an infinite fluid at $t/T = 0$. Increment of contour level is 0.25 for the solid and dashed contour lines. The wake is dominated by close vortex pairs.	85

5.1	Flow past a rotating Bach-type vertical-axis turbine. The turbine rotates in the clockwise direction about point O . The cross-sectional area of the rotor is $A \approx 2d(p + \varphi q)$	87
5.2	Time-step convergence check using prescribed rotation with $\lambda = 1$	89
5.3	Time-averaged (a) torque and (b) power coefficients, $\overline{C_T}$ and $\overline{C_P}$, of the Bach-type turbine with $\varphi = 124^\circ$ and $p/q = 0.2$. The current numerical DVM solutions at $Re = 1,500$ is compared to the experimental results at $Re = 80,000$ [98].	90
5.4	Instantaneous torque coefficient C_T and the horizontal and vertical force coefficients C_x and C_y over one period of rotation for $\lambda = 0.9$ at $Re = 1,500$	92
5.5	Non-dimensional vorticity ($\omega D/U$) contour plots of the flow and non-dimensional relative pressure p^* on the blade surface at $t/T = 0.233$ and $t/T = 0.519$ for $\lambda = 0.9$. Vorticity contour level increment is 5. Minimum torque is observed around $t/T = 0.233$ shown in (a) and (b). Maximum torque is observed around $t/T = 0.519$ shown in (c) and (d). The non-dimensional relative pressure p^* is shown as a function of the non-dimensional arclength s/D along the blade measured from the edge marked by a “•” in the insets (zero relative pressure at this point).	93
5.6	Non-dimensional vorticity ($\omega D/U$) contour plots of the flow around the Bach-type turbine under prescribed rotation with $\lambda = 0.9$ and $Re = 1,500$. Horizontal scale is x/D , and vertical scale is y/D . Solid contour lines represent positive vorticity; dashed contour lines represent negative vorticity. Contour level increment is 5.	95
5.7	Non-dimensional vorticity ($\omega D/U$) contour plots of the flow around the Bach-type turbine under prescribed rotation operating at $\lambda = 0.5$ with $Re = 1,500$. Horizontal scale is x/D , and vertical scale is y/D . Solid contour lines represent positive vorticity; dashed contour lines represent negative vorticity. Contour level increment is 3.5.	96
5.8	Non-dimensional vorticity ($\omega D/U$) contour plots of the flow around the Bach-type turbine under prescribed rotation operating at $\lambda = 1.2$ with $Re = 1,500$. Horizontal scale is x/D , and vertical scale is y/D . Solid contour lines represent positive vorticity; dashed contour lines represent negative vorticity. Contour level increment is 5.	97
5.9	Torque coefficient over one period of rotation of the Bach-type turbine operating at $\lambda = 0.725$ and 0.75	98
5.10	Non-dimensional relative pressure p^* on the turbine blade at $t/T = 0.98$ for $\lambda = 0.725$ and 0.75 . The value of p^* is plotted against the non-dimensional arclength s/D along the blade measured from the edge marked by a “•” in the inset (zero relative pressure at this point).	99
5.11	Non-dimensional vorticity ($\omega D/U$) contour plots of the flow around the Bach-type turbine in prescribed rotation at approximately $t/T = 0.98$ for $\lambda =$ (a) 0.725 and (b) 0.75 . Horizontal scale is x/D , and vertical scale is y/D . Solid contour lines represent positive vorticity; dashed contour lines represent negative vorticity. Contour level increment is 3.5.	99

5.12	Time-averaged (a) torque and (b) power coefficients, $\overline{C_T}$ and $\overline{C_P}$, as a function of the mean tip-speed ratio $\bar{\lambda}$ of the freely rotating Bach-type turbine with $\varphi = 124^\circ$ and $p/q = 0.2$ at $Re = 1,500$. The numerical value next to each data point is the non-dimensional generator damping B_g^*	101
5.13	Dependence of $\bar{\lambda}$, $\overline{C_T}$, and $\overline{C_P}$ on the non-dimensional generator damping B_g^* of the freely rotating Bach-type turbine with $\varphi = 124^\circ$ and $p/q = 0.2$ at $Re = 1,500$	102
5.14	The instantaneous (a) tip-speed ratio λ , torque coefficient C_T , and (b) the force coefficients C_x and C_y over one period of rotation for $B_g^* = 0.176$. The mean tip-speed ratio and torque coefficient are $\bar{\lambda} = 0.941$ and $\overline{C_T} = 0.166$, respectively. Because of the unsteady rotation, the turbine angular position θ is also shown for reference.	102
5.15	Non-dimensional vorticity ($\omega D/U$) contour plots of the flow around the freely rotating Bach-type turbine with $B_g^* = 0.2$ and $Re = 1,500$. For this case, $\bar{\lambda} = 0.88$. Horizontal scale is x/D , and vertical scale is y/D . Solid contour lines represent positive vorticity; dashed contour lines represent negative vorticity. Contour level increment is 5.	103

List of Tables

2.1	Performance and accuracy of the Fast Multipole Method (FMM) subroutine: single-thread versus GPU-accelerated	17
4.1	Configurations for convergence tests	64
5.1	Turbine blade geometric parameters	86
5.2	Shaft power estimation of a micro-scale flow turbine in air and water	104

Acknowledgments

I would like to first express my gratitude to Professor Ronald W. Yeung, my research advisor, for his guidance on my Ph.D. dissertation work and his help throughout my study at the University of California, Berkeley. I first encountered Professor Yeung when taking a course he taught on Ocean-Environment Mechanics. The fascinating lectures he has given were what initially drew my attention towards the field of ocean engineering and fluid mechanics in general. This encounter led me to join Professor Yeung's Berkeley Marine Mechanics Laboratory (BMML). During my time in BMML, Professor Yeung has suggested various exciting research topics to me which have led to fruitful scientific pursuits and eventually publications. His guidance has been instrumental in helping me successfully complete my Ph.D. work. In addition to academic support, Professor Yeung has also been very attentive to my career development. He encouraged me to present my work in conferences and to join professional organizations whenever an opportunity arose. Lastly, I am most grateful for the friendship Professor Yeung has offered to me which makes me feel comfortable to freely consult him about any concerns or hurdles I have had.

I would also like to thank the members of my dissertation committee: Professor Stephen Morris and Associate Professor Per-Olof Persson. Professor Morris taught me the fundamentals of fluid mechanics and many more advanced materials after that. His attention to the actual physics behind the equations has shaped the way I approach research problems. The lectures Professor Persson has given on numerical computations have helped me greatly in completing my dissertation work. The valuable feedback provided by Professor Morris and Professor Persson on my dissertation is greatly appreciated. It is a great honor to have them on my dissertation committee.

In research, a different and fresh perspective is often needed. In this respect, I am lucky to have my fellow students in the BMML. Discussions with them have often led to new inspirations and ideas to attempt. The friendship they have offered makes my time in BMML an unforgettable experience.

Doctoral studies have been the most challenging phase of my student life. However, the difficulties have been made that much more manageable by the never-waning support provided by my parents and my wife. Their patience over the years is greatly appreciated.

In the end, I want to acknowledge the *American Bureau of Shipping Ocean Technology Fellowship* and the *John V. Wehausen Graduate Scholarship* awarded to me. This work would not have been possible without the financial support.

Lu Wang
November 8, 2016

Chapter 1

Introduction

Currently, the mainstream Computational Fluid Dynamics (CFD) methods such as the finite-volume method or the finite-element method are mesh-based. These methods have been tested on a wide range of practical flow problems and are arguably more mature. The main difficulty of applying these numerical methods lies in the mesh generation. Generation of high quality meshes for problems involving complicated moving boundaries is not trivial both in terms of implementation and computational workload. Nevertheless, the challenge with mesh generation is gradually being overcome for certain classes of problems with new techniques such as Sliding Mesh, Chimera Mesh, and the Arbitrary Lagrangian-Eulerian (ALE) formulation.

Despite the relative maturity of mesh-based CFD methods, alternative mesh-free numerical algorithms are also being pursued. The two well-known examples are the Smoothed-Particle Hydrodynamics (SPH) and Discrete-Vortex Method (DVM). The mesh-free nature of these methods makes them particularly suitable for certain niche problems. For instance, SPH is naturally suited for wave simulations with violent free-surface activities such as breaking waves and wave-structure interactions. However, SPH often suffers from excessive noise issues and large numerical viscosity. The latter limits the effective Reynolds number that SPH can simulate. SPH is also inefficient in the sense that regions with little to no flow activities also need to be occupied by fine fluid particles. The DVM, on the other hand, is very convenient for simulating viscous-fluid flows near moving bodies with complex geometries in two dimensions. Mesh generation is not needed, and the computational elements (vortex blobs) naturally concentrate in regions of interest occupied by rotational flow. No vortex blob is needed in regions occupied by irrotational flow, leading to efficient use of computational resources. However, generalization of two-dimensional DVM to three dimensions, although possible (see, *e.g.*, [1]), can be very challenging and comes at the cost of significantly increased computational time; the gradient of velocity needs to be computed to account for the stretching and tilting of vorticity.

In this work, a high-performance DVM algorithm for two-dimensional viscous-fluid flow near boundaries is implemented. Efforts were made to improve the performance of the algorithm through both CPU and Graphics-Processing-Unit (GPU) based parallel computing. The details of the formulation are presented in Chapter 2. The DVM algorithm is validated through a range of tests shown in Chapter 3 and applied to the analyses of two novel flow problems of practical interest (Chapters 4 and 5).

1.1 Discrete-Vortex Method

In DVM, the flow is determined by solving the vorticity transport equation. For an incompressible and isotropic fluid under conservative body forces, the vorticity transport equation is given by

$$\frac{D\boldsymbol{\omega}}{Dt} = (\boldsymbol{\omega} \cdot \nabla)\mathbf{u} + \nu \nabla^2 \boldsymbol{\omega} \quad (1.1)$$

where $\boldsymbol{\omega}$, t , $\mathbf{u}$, and ν are the vorticity, time, flow velocity, and kinematic viscosity, respectively. There are certain advantages and disadvantages associated with solving the vorticity equation instead of the Navier-Stokes equation. By taking the curl of the Navier-Stokes equation, pressure is eliminated from the problem. Furthermore, if we restrict the problem to two dimensions, the first term on the right-hand side (RHS) of Eq. (1.1) is eliminated because stretching and tilting of vorticity does not occur in a two-dimensional flow. In this case, we have the following relation:

$$\frac{\partial \omega}{\partial t} + (\mathbf{u} \cdot \nabla)\omega = \nu \nabla^2 \omega \quad (1.2)$$

In Eq. (1.2), the scalar vorticity ω is the vorticity component perpendicular to the two-dimensional plane. The problem is simplified to the advection and diffusion of a single scalar quantity. However, this simplification also means that the primitive variables (*i.e.* velocity and pressure) are not part of the solution explicitly and require considerable post-processing effort to be reconstructed from the vorticity field.

A closed system of equations can be formed using Eq. (1.2) and the following two relations:

$$\nabla^2 \psi = -\omega \quad (1.3)$$

$$\mathbf{u} = \nabla \times (\psi \mathbf{k}) \quad (1.4)$$

where ψ is the stream function and $\mathbf{k}$ is the unit vector perpendicular to the two-dimensional plane. By applying appropriate boundary conditions, the above system of equations can be solved for the time-dependent vorticity field.

In DVM, the vorticity field is represented by the superposition of a large number of vortex blobs which act as the basic computational elements of DVM. The advection and diffusion of vorticity are represented by the motion and, in some cases, deformation of the vortex blobs.

1.1.1 Random-Vortex Method

Although there are considerable differences between the current DVM formulation and the Random-Vortex Method (RVM) of Chorin [2], it is beneficial to briefly describe Chorin's original formulation here to illustrate the principal ideas behind DVM. In [2], the vorticity equation, Eq. (1.2), is solved using a fractional step approximation, also called operator splitting, which separates the advection and diffusion of vorticity into two different steps:

$$\frac{\partial \omega}{\partial t} + (\mathbf{u} \cdot \nabla) \omega = 0 \quad (1.5)$$

$$\frac{\partial \omega}{\partial t} = \nu \nabla^2 \omega \quad (1.6)$$

Equation (1.5) is the advection step and indicates that vorticity is transported according to the local flow velocity. Equation (1.6) is the diffusion step which shows that vorticity is diffused according to the local vorticity gradient.

Numerically, the advection step is achieved by moving each vortex blob in the flow field according to the flow velocity at the center of the blob. Flow velocity is calculated by summing the induced velocity of all vortex blobs present and the velocity induced by any rotating solid bodies. The diffusion step is simulated through random walk of the vortex blobs. At each time step, random displacements in both x and y directions are added to each vortex blob on top of advection. The displacements are random variables following a Gaussian distribution with zero mean and a standard deviation of $\sqrt{2\nu\Delta t}$ where Δt is the time step.

If a body is present, a potential flow component is added to satisfy the no-penetration boundary condition on the body surface, and new vortex blobs are created near the surface to cancel the slip velocity. This procedure simulates the physical process of vorticity creation on boundaries.

Despite its apparent simplicity, RVM can be considered as the workhorse vortex method used in engineering [3] and has been applied by several authors to good effect. For instance, Yeung and Jiang successfully incorporated RVM to simulate the roll motion of ships with bilge keels in three dimensions using the Slender-Ship Free-Surface Random-Vortex Method (SS-FSRVM) [4,5]. This method accepts arbitrary body or hull shapes and includes free-surface effects. The computed ship-roll motions show excellent agreement with tow-tank experiments. However, RVM has significant drawbacks. At lower Reynolds numbers, the large random displacements of the vortex blobs lead to excessive noise in the solution. In addition, because of the presence of a stochastic process, conventional high-order time-integration schemes cannot be applied directly to RVM [6,7], leading to the use of very small time steps. This issue is more prominent at higher Reynolds numbers where the error from advection can easily dominate over random walk [8]. RVM also has a slow convergence rate, requiring a large number of vortex blobs to achieve reasonable accuracy [9]. The fractional step approach also introduces additional splitting error. These drawbacks motivated the adoption of a different diffusion model for the current work.

1.1.2 Deterministic models for diffusion

The various drawbacks of RVM all stem from the use of a stochastic process. In order to overcome these drawbacks, a better deterministic method is needed for incorporating the effect of diffusion without having to introduce a mesh to solve Eq. (1.6). To this end, several

deterministic models for vorticity diffusion have been developed over the years to replace the random-walk model. Some of the more well-known mesh-free alternatives to RVM are the Core-Spreading Method, Vorticity-Redistribution Method, Particle-Strength-Exchange Method, and Diffusion-Velocity Method. A comparison between these deterministic methods and RVM is presented in [10].

In the Core-Spreading Method, the vorticity diffusion process is accurately modeled by having the core size σ of a Gaussian vortex blob grow with time. More specifically, the core size grows with $\sqrt{2\nu t}$, exactly satisfying the two-dimensional diffusion equation [11]. However, it has been shown that this method is physically incorrect because of the inappropriate treatment of the advection process [12]. In DVM, the vortex blobs are moved according to the flow velocity at the center of each vortex blob. This is an acceptable approximation only if the sizes of the blobs are small and bounded. With the Core-Spreading Method, however, the sizes of the vortex blobs grow continuously with time, and the error in advection causes the Core-Spreading Method to converge to a wrong set of equations even when an infinite number of vortex blobs are used. This issue with advection is solved by the Corrected Core-Spreading Method [13]. In this method, the core sizes of the vortex blobs are allowed to grow with time until a preset limit is reached. When the size of a vortex blob exceeds this limit, the vortex blob is broken up into several smaller vortex blobs while conserving the circulation, linear and angular momentum. However, the vortex splitting leads to an exponential increase in the number of vortex blobs, resulting in high computation costs especially at low Reynolds numbers [10].

In the Vorticity-Redistribution Method, the diffusion process is modeled by distributing a fraction of the circulation of each vortex blob to its close neighbors [14]. A pair of vortex blobs are considered to be close neighbors if the distance between the two is on the order of the diffusion length $\sqrt{\nu\Delta t}$. The fractions of circulation to be distributed to the neighbors are determined by solving an underdetermined system of equations that approximates the diffusion equation as a linear programming problem. For stability, the fractions of circulation to be distributed must always be positive. However, a solution for the fractions that satisfies this constraint might not exist. In this case, new vortex blobs are inserted in the neighborhood at prescribed locations until a solution that satisfies the constraint is found. This process is constantly occurring at the edge of the region occupied by the vortex blobs to allow the region with non-zero vorticity to grow through diffusion. The Vorticity-Redistribution Method can have an arbitrarily high order of accuracy in time; however, the operator-splitting error from the fractional step formulation is still present [14]. One of the drawbacks of this method is the heavy computational load that stems from the need to solve a large number of linear programming problems. A similar method called the Particle-Strength-Exchange (PSE) method has also been widely used [15]. In the PSE method, the diffusion process is also modeled by exchanging the circulation between nearby vortex blobs. The Laplacian of vorticity in the diffusion equation is replaced by an integral operator with a Gaussian kernel. In an infinite fluid domain, the diffusion steps can be carried out simultaneously with the advection step, removing the operator-splitting error. However, if a body is present, fractional step formulation is still used [15]. Unlike the vorticity-redistribution

method, the PSE method requires a highly regular distribution of vortex blobs. Therefore, it is necessary to frequently re-initialize the positions of vortex blobs back onto a uniform grid by interpolating the vorticity at the grid points. This process may introduce some numerical diffusion. Nevertheless, high-quality numerical results of flow past bodies have been produced with the PSE method [16, 17]. A rather unique aspect of the PSE method is that new vortex blobs are not created on the body surface. Instead, the no-slip boundary condition is satisfied by modifying the circulation of vortex blobs near the surface [15]. Although this method generates smooth solutions near the body surface, the PSE method lacks an intuitive procedure for introducing new vortex elements into the flow.

The Diffusion-Velocity Method represents the vorticity diffusion process by moving the vortex blobs. The circulation and core size of each vortex blob remain unchanged. In this sense, the Diffusion-Velocity Method is the closest in spirit to the original RVM formulation. Different from RVM, the movements of the vortex blobs are fully deterministic. The two-dimensional vorticity equation can be manipulated to show that the combined effect of advection and diffusion can be represented by advecting the vorticity according to the actual flow velocity combined with a diffusion velocity [8]. The diffusion velocity is a pseudo velocity that is proportional to the vorticity gradient and inversely proportional to the local vorticity. In the original formulation of [8], the vorticity and vorticity gradient is directly computed from the superposition of a large number of Gaussian vortex blobs. This method is shown to generate smoother solutions compared to RVM. Also, the convection step and the diffusion step can be carried out simultaneously, removing the operator-splitting error. However, the Diffusion-Velocity Method breaks down when the overlap between vortex blobs is lost; the lack of overlap will cause the viscous diffusion process to cease. To overcome this issue, a hybrid approach that combines the Diffusion-Velocity Method and RVM can be used. For instance, the Diffusion-Velocity Method is used near the body where a large number of overlapping vortex blobs exist and a smooth solution is important. In the far wake where the overlap between vortex blobs is lost because of diffusion, RVM is used instead so that viscous diffusion can continue. A smooth transition between the Diffusion-Velocity Method and RVM is adopted in the region in-between [18].

1.2 Current DVM implementation

For the current DVM implementation, the Diffusion-Velocity Method is chosen for its efficiency and relative ease in implementation. However, we need to overcome the difficulty associated with the lost of overlap between vortex blobs. The hybrid approach of [18] works but introduces additional complexity to the simulation, especially when multiple bodies are present. Instead, a relatively recent approach called the Viscous-Vortex-Domains (VVD) method is adopted [19, 20]. In the VVD method, the local vorticity and vorticity gradient, which are needed for evaluating the diffusion velocity, are computed based on nearby vortex blobs using an exponential smoothing kernel. The radius of the smoothing kernel is chosen adaptively based on the local spacing of vortex blobs, allowing the diffusion process to con-

tinue even when the vortex blobs are relatively far apart. The details of this method are given in Sec. 2.3.2.

1.3 Applications of DVM simulations

Using the current implementation of DVM, we have studied two novel flow problems of practical interest. First, the lift generation of a hovering and flapping foil under both full and partial ground effects is investigated. This problem is pertinent to the design and control of Micro-Aerial Vehicles (MAVs). Second, the flow past a micro-scale Bach-type flow turbine has been studied. This work aims at evaluating whether Bach-type turbines can be miniaturized for use as micro-scale energy harvesters to power devices such as wireless sensor nodes. The findings have been published in [21, 22].

1.3.1 Flapping foil with ground effects

Unlike most large man-made aerial vehicles that fly on fixed wings, insects, avian species, and aquatic animals often rely on flapping wings or fins for lift or thrust generation. Flapping wings provide improved maneuverability at low speeds and may be more efficient than fixed wings at very small scales (low Reynolds number) with optimized wing motion [23]. There are mainly three distinct mechanisms that are responsible for the high lift force generated by the flapping motion: delayed stall, rotational circulation, and wake capture [24]. The delayed stall phenomenon occurs when the wing moves through air at large angles of attack. A leading edge vortex forms which creates a suction effect on the low pressure side of the wing. The rotational circulation has a strong effect on the lift force when the wing rotates to reverse its direction. The wake capture effect accounts for the interaction between the wing and its wake shed during the previous stroke. Depending on the timing of the rotation relative to stroke reversal, wing-wake interaction may increase lift generation, allowing the wing to effectively recapture energy from the shed wake.

Numerical or experimental investigations are needed to study the lift or thrust generating mechanisms of a flapping wing because the flow around the wing is highly dynamic with strong boundary-layer separation. The complicated motion of the body boundary presents unique challenges for computational investigation especially in terms of mesh generation. As a result, mesh-free techniques such as DVM [25] and SPH [26] or methods based on non-body conformal grids such as the Immersed Boundary-Lattice Boltzmann Method (IB-LBM) are often preferred [27–29]. However, conventional methods based on body-fitted grids in a non-inertial frame have also been successfully applied [30]. Meanwhile, experimental investigations have been carried out by several researchers [30–34].

In recent years, the research on flapping wings is further motivated by the interests in developing MAVs which have a wide range of practical applications. One direction of research is focused on efficient lift/thrust generation which is of great importance to MAVs because of their limited power capacity. Another important direction is on the navigation

of MAVs in a confined space. More specifically, it is important to understand the variation in forces experienced by flapping wings in the presence of boundaries. For example, the flow near a flapping wing in the presence of ground effect has been studied using numerical methods by several authors in both two dimensions [35–39] and three dimensions [40–42]. Experimental investigations have also been carried out [43,44]. For wings flapping in normal hovering mode, three regimes of ground effect have been observed: force enhancement, force reduction, and force recovery [36]. When the clearance between the flapping wing and the ground is sufficiently small, the wing will be in the force-enhancement regime, and the lift force is increased by the presence of the ground. As the ground clearance increases, the flow enters the force-reduction regime where the lift force is drastically reduced. If the clearance continues to increase, the flow will eventually enter the force-recovery regime where the lift force gradually recovers to the level without ground effect. The existence of these three regimes is later confirmed experimentally [44]. Even though the three force regimes are observed with a rigid ground surface, we expect to see similar changes to the lift and drag forces when a light-weight MAV with rapid wing motion is hovering above a free surface of water because we can approximate the free surface as a rigid ground plane in this case.

As a natural extension to the research on ground effect, we have investigated the forces on a flapping foil in normal hovering mode near a finite-sized platform in two dimensions. Special attention is given to partial ground effects when the wing is hovering near the edge of the platform. This study has significance in the control of an MAV when it needs to slowly approach and land on a small landing platform. In addition, this work is also pertinent to the control of biomimetic underwater robots with flapping-fin-based locomotion. These robots are expected to navigate complex environments and are likely to experience partial ground effects during operation. The Reynolds numbers we are using are much smaller than what is typical for modern MAVs and biomimetic underwater robots; however, we do expect to see qualitatively similar flow behaviors around the flapping wing or fin for both applications.

1.3.2 Micro-scale Bach-type turbine

Currently, on-shore utility-scale wind power generation is dominated by Horizontal-Axis Wind Turbines (HAWT). Large commercial HAWTs incorporate active orientation (yaw) control [45] and collective or independent blade pitch control [45,46] to ensure the turbine is constantly facing the wind and operating at near-optimal load. As an alternative to HAWTs, Vertical-Axis Wind Turbines (VAWT) are also being developed and improved. Although less popular for on-shore applications, VAWTs have a number of advantages over HAWTs. First of all, VAWTs can work with any wind direction, and a yaw-control mechanism is not needed, leading to simpler and cheaper constructions. Further, VAWTs have a smaller footprint and can be arranged in close-packed arrays. In fact, it has been found that it is possible to increase power extraction efficiency of VAWTs through constructive aerodynamic interactions between a pair of counter-rotating VAWTs [47].

VAWTs can be divided into two different types: lift-based Darrieus VAWTs and drag-based Savonius-type [48] or Bach-type [49] VAWTs. Compared to drag-based VAWTs, Dar-

rius VAWTs operate at a higher tip-speed ratio (λ) and are generally more efficient [50]; thus, Darrieus VAWTs are usually preferred for large-scale wind energy applications. However, Darrieus VAWTs have a low starting torque and are sometimes used in conjunction with a Savonius-type VAWT to improve their self-starting capability [51]. On the other hand, drag-based VAWTs operate at a much lower λ and have high starting and operational torque. Darrieus rotors and Savonius or Bach rotors can also be utilized as water current or tidal turbines [52, 53].

Although less efficient than their lift-based counterparts, drag-based turbines are believed to be particularly suitable for use as micro-scale power harvesters. In particular, Savonius-type and Bach-type turbines can be used as power sources for Wireless Sensor Networks (WSN) [54]. A WSN may consist of thousands of low-cost sensor nodes spread out in a large geographic area. Each sensor node hosts a suite of Micro-Electro-Mechanical Systems (MEMS) based sensors to monitor temperature, pressure, motion, illumination, etc. A sensor node also contains a microcontroller, wireless transmitters and receivers, data storage, and a power supply, typically a battery. Each node can communicate with each other, forming a network which provides continuous monitoring of a large region. Such a network can be deployed in various terrestrial and marine environments and has a wide range of civil and military applications, from pollution monitoring to motion tracking. Even though the sensor nodes are low-power devices, the usefulness of a battery-powered node is hampered by its finite power supply. Therefore, it is ideal to incorporate an universal micro-scale power harvester in each sensor node. Such a harvester needs to be low-cost, reasonably efficient, and able to operate in an uncontrolled orientation. A drag-based turbine is a potential candidate for this role because of its simple construction, self-starting capability, and insensitivity to the wind or current direction. Finally, a drag-based turbine will need to be able to operate at very low Reynolds numbers because of its small size and low wind or current speeds. For example, a small sensor node placed on the ground will experience very little wind because of the ground boundary layer. To evaluate the potential of drag-based turbines for use as universal micro-scale energy harvesters, we have simulated the flow past a Bach-type turbine at a low Reynolds number ($Re = 1,500$) using DVM and calculated its power extraction efficiency. Detailed numerical investigation on drag-based turbines at this micro scale has not been found in the existing literature. However, several researchers have already started to show interest in small-scale wind turbines [55, 56]. In particular, a very recent study investigated experimentally the performances of 3D printed miniature drag-based turbines of a slightly larger scale (Reynolds number based on turbine diameter around 5×10^4) [56]. The miniature turbines were used to power 4 light-emitting diodes, demonstrating the potential for this type of small-scale wind-energy devices.

A sizable number of numerical simulations of flow past drag-based turbines can be found [57]. Typically, these numerical analyses are carried out in the turbulent regime using mesh-based methods and various turbulence models (see e.g., [58–60]). Few studies, however, have been done on the performance of drag-based turbines at smaller scales with low wind or current speeds. In this case, the flow can be modeled as laminar and two dimensional. The viscous DVM is particularly attractive for solving this type of flow. There have been several

previous DVM analyses of drag-based turbines [61–67]. These previous DVM computations are focused on the wake of the turbine and do not directly simulate vorticity diffusion; boundary-layer separation and vortex shedding are limited to the edge of the turbine blade. Computational approaches in a similar spirit have also been used to efficiently predict the blade forces and wake of a lift-based turbine [68–71]. Free vortex filaments released near the blade trailing edge are used to construct the unsteady wake, and a dynamic stall model may be incorporated to accommodate large blade angle of attack [70–72]. In this work, the fully viscous DVM is used to simulate the flow. Viscosity and the effect of Reynolds number are explicitly accounted for through vorticity diffusion, and the occurrence of boundary-layer separation is no longer limited to blade edges.

1.4 Research objectives

This dissertation mainly consists of three parts. The first part involves the formulation, implementation, and validation of the discrete-vortex algorithm (Chapters 2-3). In this part, a major objective is to design, for various components of the algorithm, a set of new efficient implementation strategies that utilizes both CPU and GPU parallel computing. The new strategies should significantly improve the computational performance of the DVM.

In the second part of this dissertation, the present DVM algorithm is applied to investigate the lift-generating mechanism of a flapping foil under full and partial ground effects (Chapter 4). This work represents a first investigation of the partial ground effect caused by the presence of an effectively semi-infinite ground platform. The main objective is to discover new lift-modifying flow mechanisms unique to partial ground effect.

In the last part of this dissertation, viscous DVM simulations are carried out to investigate the fluid-dynamic characteristics and power-generating capacity of a micro-scale Bach-type flow turbine (Chapter 5). The main objective is to evaluate whether Bach-type turbines can be effectively miniaturized for use as micro-scale energy harvesters. Therefore, a very low Reynolds number of 1,500 that has not been considered in the existing literature is used. Turbine-wake interaction at this low Reynolds number is also investigated.

Chapter 2

Description of numerical method

This chapter aims at providing a complete description of the theoretical formulation which the current DVM implementation is based on. Because significant efforts were made to improve the computational performance of the software we have developed from the ground up, various aspects of the numerical implementation are also discussed.

2.1 Problem description and general setup

The DVM software, in its current stage, is capable of solving the two-dimensional flow around an arbitrary number N_B of rigid bodies submerged in an infinite and viscous fluid of constant and uniform density ρ and kinematic viscosity ν . The general setup of the problem is shown in Fig. 2.1. The bodies can perform a prescribed motion (translation and rotation), free motion, or a combination of both. The position and orientation of the i -th body are fully described by the position of a reference center $\mathbf{r}_{c,i} = (x_{c,i}, y_{c,i})$ fixed on the body and the angle of rotation about this center θ_i (positive in the counterclockwise direction). Similarly, the motion of the i -th body is given by the velocity of the reference center $\mathbf{u}_{c,i} = (u_{c,i}, v_{c,i}) = (\dot{x}_{c,i}, \dot{y}_{c,i})$ and the angular velocity $\Omega_i = \dot{\theta}_i$.

Each two-dimensional body can be arbitrarily shaped with the i -th body occupying the region B_i . Numerically, the contour of the i -th body, ∂B_i , consists of $N_{p,i}$ linear panels and the same number of nodes between the panels. The unit normal and tangential vector of ∂B_i is given by $\mathbf{n}_i$ and $\mathbf{s}_i$. Since we use linear panels, $\mathbf{n}_i$ and $\mathbf{s}_i$ are constant and equal to $\mathbf{n}_{i,k}$ and $\mathbf{s}_{i,k}$, respectively, on the k -th panel of i -th body.

In the fluid domain D , there are a total of N vortex blobs that, together, numerically represent the vorticity field. Each vortex blob has three basic attributes: circulation Γ_j , center position $\mathbf{r}_j = (\xi_j, \eta_j)$, and core radius σ_j where $1 \leq j \leq N$ is the index of the vortex blob. As time t progresses, nascent vortex blobs are being created next to the body contours after each time step to enforce the no-slip and no-penetration boundary conditions at the fluid-body interface. This is achieved by creating a vortex sheet along each body contour to enforce the no-penetration boundary condition. The vortex sheet is immediately converted into nascent vortex blobs by conserving circulation. Existing vortex blobs move with time to satisfy the two-dimensional vorticity equation given in Eq. (1.2). In the current formulation, both advection and diffusion processes are simulated by the movement of the vortex blobs. The shape and size of each vortex blob remain constant over time except when merging of vortex blobs occurs.

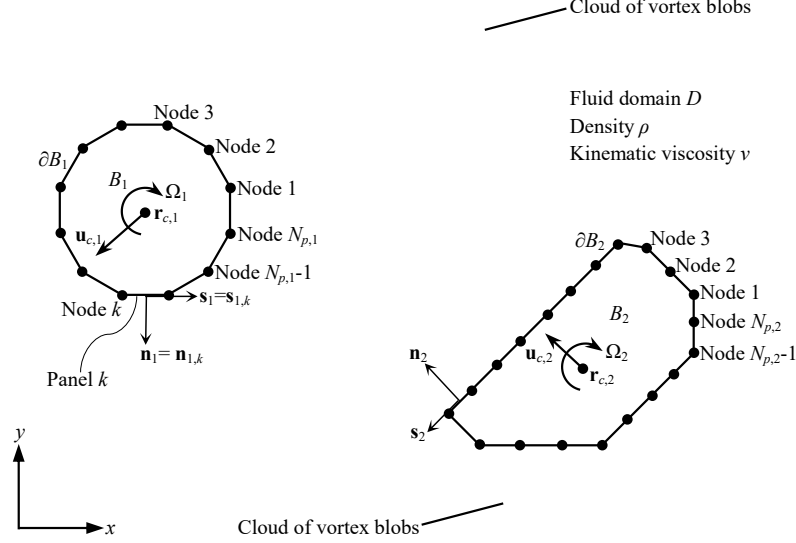


Figure 2.1: Schematic of the viscous-fluid flow problem with submerged bodies.

2.2 Choice of vortex blob structure

The structure of the vortex blobs, *i.e.* the vorticity distribution associated with each blob, needs to be decided first. Since the vortex blobs are merely numerical constructs, the exact form of the vorticity distribution is rather arbitrary apart from a few constraints. First, the vorticity field associated with a vortex blob should be finite, smooth, and have finite support; the vorticity decays to zero if the distance to the center of the blob is much larger than the core radius σ . Also, as $\sigma \rightarrow 0$, the vortex blob should become a point vortex. It is, however, problematic to directly use point vortices in DVM because the induced velocity is singular at the center of a point vortex. A converged solution cannot be achieved by increasing the number and decreasing the strength of vortices [11]. Various types of vortex blobs with desingularized induced velocity have been introduced. In SS-FSRVM [4], the original vortex blob introduced by Chorin is used [73]. The stream function of a Chorin vortex blob centered at $\mathbf{r}_0 = \xi\mathbf{i} + \eta\mathbf{j}$ is given by

$$\psi(\mathbf{r}) = \begin{cases} \Gamma/(2\pi) \log |\mathbf{r} - \mathbf{r}_0| & \text{if } |\mathbf{r} - \mathbf{r}_0| \geq \sigma \\ \Gamma/(2\pi)(|\mathbf{r} - \mathbf{r}_0|/\sigma) + \text{constant} & \text{if } |\mathbf{r} - \mathbf{r}_0| < \sigma \end{cases} \quad (2.1)$$

where Γ is the circulation of the vortex blob, σ is the core radius of the blob, and $\mathbf{r} = x\mathbf{i} + y\mathbf{j}$ is the position vector where the stream function is evaluated. The constant in Eq. (2.1) is chosen such that the stream function is continuous at $|\mathbf{r} - \mathbf{r}_0| = \sigma$. Outside the core, the induced velocity is the same as that of a point vortex. Inside the core, the tangential velocity

is finite and constant. However, the vorticity is still singular at the center of the blob. This presents challenges when computing the vorticity field.

A common alternative to the Chorin blob is the Gaussian blob of second order [11]. A Gaussian blob of radius σ and circulation Γ located at $\mathbf{r}_0$ creates a vorticity field:

$$\omega(\mathbf{r}) = \frac{\Gamma}{\pi\sigma^2} \exp\left(-\frac{|\mathbf{r} - \mathbf{r}_0|^2}{\sigma^2}\right) \quad (2.2)$$

The induced velocity at $\mathbf{r}$ is

$$u(\mathbf{r}) = \frac{\Gamma(\eta - y)}{2\pi|\mathbf{r} - \mathbf{r}_0|^2} \left(1 - \exp\left(-\frac{|\mathbf{r} - \mathbf{r}_0|^2}{\sigma^2}\right)\right) \quad (2.3)$$

$$v(\mathbf{r}) = -\frac{\Gamma(\xi - x)}{2\pi|\mathbf{r} - \mathbf{r}_0|^2} \left(1 - \exp\left(-\frac{|\mathbf{r} - \mathbf{r}_0|^2}{\sigma^2}\right)\right) \quad (2.4)$$

The advantage of the second-order Gaussian blob is that it has a smooth vorticity and velocity distribution. The tangential velocity inside the core approximates rigid-body rotation, whereas outside the core, the tangential velocity quickly approaches that of a point vortex. The transition between the two region is also smooth.

The stream function of the second-order Gaussian blob is shown to be:

$$\psi(\mathbf{r}) = -\frac{\Gamma}{2\pi} \left(\ln|\mathbf{r} - \mathbf{r}_0| - \frac{1}{2} \text{Ei}\left(-\frac{|\mathbf{r} - \mathbf{r}_0|^2}{\sigma^2}\right) \right) + \text{constant} \quad (2.5)$$

The presence of the exponential integral Ei means that generating streamlines can be expensive computationally when a large number of vortices are present in the flow field. However, this issue can be partially resolved by approximating the stream function of the Gaussian blob with that of a point vortex when the center of the blob is sufficiently far away (*e.g.* $|\mathbf{r} - \mathbf{r}_0| > 4\sigma$). In the present implementation of DVM, the second-order Gaussian blob is adopted.

2.3 Motion of vortex blobs

In RVM, the positions of the vortex blobs are updated using a fractional step formulation where diffusion is simulated using random walk separate from advection. A major disadvantage of this approach is that conventional high-order time-integration schemes can not be applied directly because of the presence of a stochastic process [6, 7]. Chang proposed a second-order-accurate time-integration scheme by interlacing random walk with advection [6]. To further increase the order of accuracy, it becomes clear that a formulation that merges advection and diffusion into a single step is needed. This can be achieved through the use of diffusion velocity proposed by Ogami and Akamatsu [8].

The two-dimensional vorticity equation, Eq. (1.2), can be rewritten as

$$\frac{\partial \omega}{\partial t} + \frac{\partial}{\partial x}(u\omega) + \frac{\partial}{\partial y}(v\omega) = \frac{\partial}{\partial x} \left[\frac{\nu}{\omega} \frac{\partial \omega}{\partial x} \omega \right] + \frac{\partial}{\partial y} \left[\frac{\nu}{\omega} \frac{\partial \omega}{\partial y} \omega \right] \quad (2.6)$$

This form allows us to combine the advection terms and the diffusion terms:

$$\frac{\partial \omega}{\partial t} + \frac{\partial}{\partial x} \left[\left(u - \frac{\nu}{\omega} \frac{\partial \omega}{\partial x} \right) \omega \right] + \frac{\partial}{\partial y} \left[\left(v - \frac{\nu}{\omega} \frac{\partial \omega}{\partial y} \right) \omega \right] = 0 \quad (2.7)$$

The combined effect of advection and diffusion results in vorticity being transported by $u + u_d$ in the x -direction and $v + v_d$ in the y -direction where the diffusion velocity u_d and v_d are defined as:

$$u_d = -\frac{\nu}{\omega} \frac{\partial \omega}{\partial x}, \quad v_d = -\frac{\nu}{\omega} \frac{\partial \omega}{\partial y} \quad (2.8)$$

With the above formulation, higher-order time-integration schemes (*e.g.* fourth-order Runge-Kutta method) can be directly applied to update the vortex blob positions. In addition, the diffusion is modeled deterministically, significantly reducing noise in the solution and improving convergence [8]. The numerical implementation of advection and diffusion is presented in this chapter.

2.3.1 Treatment of advection process

The vortex blobs advect according to the local flow velocity at the center of each blob. The flow velocity $\mathbf{u}$ is determined by summing the velocity induced by all vortex blobs present [11] and the velocity induced by rotating bodies:

$$\begin{aligned} \mathbf{u}(\mathbf{r}, t) = (u, v) = & -\frac{1}{2\pi} \sum_{j=1}^N \frac{(\mathbf{r} - \mathbf{r}_j) \times \mathbf{k} \Gamma_j g(|\mathbf{r} - \mathbf{r}_j|/\sigma_j)}{|\mathbf{r} - \mathbf{r}_j|^2} \\ & + \sum_{i=1}^{N_B} \frac{\Omega_i}{\pi} \iint_{B_i} \frac{\mathbf{k} \times (\mathbf{r} - \mathbf{r}_0)}{|\mathbf{r} - \mathbf{r}_0|^2} d\mathbf{r}_0 \end{aligned} \quad (2.9)$$

where $\mathbf{k}$ is the unit vector in the out-of-plane direction. The form of the function g depends on the type of vortex blob used. For the Gaussian vortex blob used here, the function g is given as

$$g(z) = 1 - \exp(-z^2) \quad (2.10)$$

The first term on the RHS of Eq. (2.9) is the sum of the induced velocities of all N vortex blobs in the flow. This term represents the mutual interaction between vortex blobs. The position, strength, and core radius of the j -th vortex blob are $\mathbf{r}_j$, Γ_j , σ_j , respectively. The second term is the velocity induced by N_B rotating bodies in the flow with Ω_i and B_i being

the angular velocity of and the area occupied by the i -th rotating body. Mathematically, the spaces inside the bodies are filled with flow under rigid-body rotation and have uniform vorticity $2\Omega_i$.

At each time step, Eq. (2.9) needs to be evaluated for every vortex blob. This task occupies a significant portion of the computing time; therefore, a careful implementation is required for efficient computation. The numerical implementations of the mutual interaction term, first term on the RHS of Eq. (2.9), and body induced term, second term on the RHS Eq. (2.9), are separately discussed in Sec. 2.3.1.1 and Sec. 2.3.1.2.

2.3.1.1 Mutual interaction between vortex blobs

A direct summation of the mutual interaction term for each vortex blob would require an operation count of $O(N^2)$ where N is the total number of vortex blobs. The computing time required would be impractical because N is of $O(10^5)$ to $O(10^6)$ for most cases of interest. A more efficient algorithm for N -body interaction must be adopted. The current implementation of DVM utilizes the Fast Multipole Method (FMM) developed by Greengard which reduces the operation count to $O(N)$ [74].

At each time step, a quadtree is constructed to spatially partition the domain. The highest level (root level) of the quadtree contains a single node which is a single square box encompassing all the vortex blobs. Nodes on the next level of the quadtree are constructed by subdividing the nodes (boxes) on the current level into their four quadrants. This process continues recursively until a node contains less than N_{max} vortex blobs at which point the node is marked as childless. A childless node will not be divided further. All empty nodes (containing no vortex blobs) are discarded. The hierarchical structure of the quadtree is illustrated in Fig. 2.4. An example quadtree constructed for the boundary layer and wake of a translating cylinder can be seen in Fig. 2.2. Only childless nodes are shown for clarity.

In FMM, only Near-Field Interactions (NFI) are computed in a pairwise manner. NFI refers to the interaction between vortex blobs belonging to the same or neighboring childless nodes. Interactions between vortex blobs belonging to non-neighboring childless nodes, Far-Field Interactions (FFI), are accounted for efficiently by combining the effects of vortex blobs in the same node. This allows FFI to be evaluated in a node-to-node fashion through the use of multipole expansions and local expansions, thus significantly reducing operation counts. Further more, FMM fully utilizes the hierarchical nature of a quadtree by always evaluating node-to-node interactions at the highest level (largest boxes) possible. The exact procedure of FMM, given in [74], is somewhat complicated and will not be repeated here for the sake of brevity.

The computational load of FMM can be separated into two major parts: pairwise NFI and node-level operations (including quadtree construction). NFI is relatively simple but needs to be carried out a large number of times. In addition, NFI of each vortex blob can be summed independently of each other. This character makes NFI highly parallel in nature. Therefore, to further improve performance, we have adopted a heterogeneous computing framework in which NFI is processed by a Graphics Processing Unit (GPU) using the Compute Unified

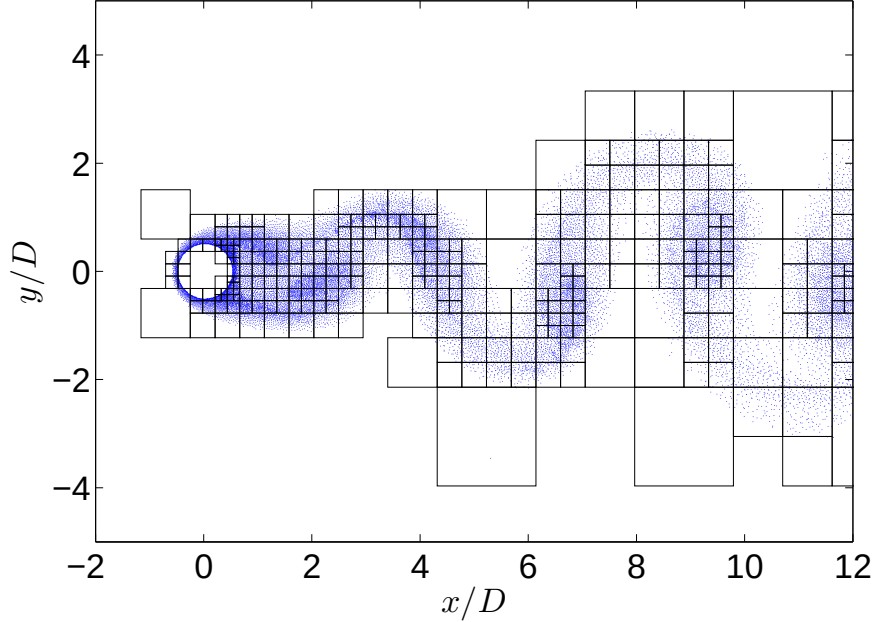


Figure 2.2: Example of quadtree construction for the boundary layer and wake of a translating cylinder. The blue dots represent vortex blobs, and the black boxes are childless nodes (boxes) of the quadtree.

Device Architecture (CUDA) and node-level operations are done by the Central Processing Unit (CPU).

The unique architecture of a GPU makes it more suitable to parallel computing than a CPU. For example, a single NVIDIA GeForce GTX TITAN Black GPU contains 2880 single-precision CUDA cores and 960 double-precision units [75]. This is vastly superior to a multi-core CPU even considering the fact that a GPU cannot run double-precision floating-point operations at full speed. Common commercial GPUs are designed and optimized for single-precision operations.

In GPU computing, a large number of threads are executed simultaneously. The threads are grouped into blocks, and multiple blocks form a grid. All threads in a grid must run the same set of instructions called a kernel. The NFI computation is implemented in the most straightforward manner here. Each block corresponds to a childless node, and each thread in the block is responsible for a single vortex blob in the node. Each thread sums the induced velocities of all vortex blobs in the current and neighboring childless nodes at the center of the vortex blob it is responsible for.

Memory access is an important factor that determines the performance of GPU computing. Different from CPU programming, efficient memory access on a GPU is largely the responsibility of the programmer. There are multiple memory types on a GPU: global memory, shared memory, register, and other specialized memory types, such as texture memory.

The largest is the global memory which normally has a size of several gigabytes. Before any computing can be done, data must first be copied from the system memory to the global memory on the GPU. The GPU can directly access any data stored in the global memory. However, global memory is relatively slow, and efficient access to global memory requires memory access coalescing. One of the requirements of memory coalescing is that threads in the same warp (32 consecutive threads) must access global memory in an aligned and sequential fashion. Shared memory, on the other hand, is allocated per block and is visible to all the threads within the same block. Shared memory is considerably faster than global memory but is very limited in size, normally between 16 to 48 kilobytes per block. Register is the fastest on chip memory. Like shared memory, register size is also very limited. Variables stored in the register are only available to each individual thread, not across threads.

Under the current implementation of NFI, data containing the circulations, positions, and sizes of all vortex blobs are first copied from the system memory into the global memory of the GPU. Because the vortex blobs are not stored in the memory in any specific order, global memory coalescing is not possible. To improve performance, each block uses all its threads to copy the data on vortex blobs located inside its corresponding childless node and neighboring childless nodes into its shared memory. This allows the threads to efficiently access all the relevant information for NFI computation. Because of the limited size of shared memory, each neighboring childless node is loaded sequentially. Once the velocity from NFI are calculated for each vortex blob, the results are copied back to the global memory and then back to the system memory. The use of shared memory reduces the frequency of non-coalesced global memory access.

Because GPU can operate independently from the CPU, NFI on GPU and node-level operation on CPU can run in an asynchronous fashion. The CPU first constructs the quadtree based on vortex blob distribution. Next, vortex blob and quadtree data are copied to the GPU. From this point on, the GPU and CPU work on different tasks simultaneously. The results from GPU and CPU are combined at the end. The CPU code is also parallelized using the Open Multi-Processing (OpenMP) Application Programming Interface to fully utilize multi-core CPUs. Load balancing between CPU and GPU is important. Ideally, both GPU and CPU should complete their individual tasks at roughly the same time. This can be achieved through adjusting the parameter N_{max} . A higher N_{max} will lead to fewer levels and nodes in the quadtree, thus reducing CPU workload. At the same time, each childless node will contain more vortex blobs, resulting in heavier NFI computation. Advanced methods for dynamically balancing the load between multiple CPUs and GPUs have been developed (*e.g.* [76]). However, here we decided to simply set N_{max} to 1024. The reason is that the GPU currently being used has the Kepler GK110 architecture which supports a maximum of 1024 threads per block. By setting N_{max} to 1024, we ensure that there are enough threads and shared memory in each block to process all the vortex blobs in the node in one sweep. The high value of N_{max} is a consequence of the high computing power of the GPU. For a FMM subroutine implemented solely on the CPU, N_{max} is typically around 200 for better performance.

Our GPU-accelerated FMM subroutine is evaluated by computing the mutually induced

2.3. Motion of vortex blobs

Table 2.1: Performance and accuracy of the Fast Multipole Method (FMM) subroutine: single-thread versus GPU-accelerated

N	Single-thread FMM			GPU-accelerated FMM			Factor of Acceleration
	t_{comp} (s)	$\bar{E}_u$	$\bar{E}_v$	t_{comp} (s)	$\bar{E}_u$	$\bar{E}_v$	
10^4	1.00	2.2×10^{-11}	1.8×10^{-11}	0.0129	1.1×10^{-11}	6.6×10^{-12}	$\times 77.5$
10^5	7.20	3.7×10^{-11}	6.9×10^{-11}	0.120	2.8×10^{-11}	5.8×10^{-11}	$\times 60.0$
10^6	52.1	1.6×10^{-10}	9.7×10^{-11}	1.69	1.3×10^{-10}	8.4×10^{-11}	$\times 30.8$
2×10^6	190	9.2×10^{-11}	6.6×10^{-11}	2.67	7.3×10^{-11}	5.9×10^{-11}	$\times 71.3$

velocities of N randomly generated vortex blobs. The position (ξ_j, η_j) and circulation Γ_j of the j -th blob are randomly chosen such that ξ_j and η_j are in the range $[0, 10]$ and $\Gamma_j \in [-1, 1]$. The core radii of all vortex blobs are set to 0.001. Four values of N between 10^4 to 2×10^6 are used. The performance of the GPU accelerated FMM subroutine is compared against an equivalent subroutine that only run on a single CPU thread with $N_{max} = 200$. The computing time t_{comp} is measured on a computer with two Intel Xeon X5690 CPUs with 6 physical cores each (12 total) and a GeForce GTX TITAN Black GPU operating in double precision mode. As a reference, the mutually induced velocities are also computed using the much less efficient direct summation method with $O(N^2)$ work load. The relative errors in the x - and y -components of the induced velocity at the j -th vortex blob, $E_{u,j}$ and $E_{v,j}$, are defined as

$$E_{u,j} = \frac{u_{j,FMM} - u_{j,direct}}{u_{j,direct}} \quad E_{v,j} = \frac{v_{j,FMM} - v_{j,direct}}{v_{j,direct}} \quad (2.11)$$

where $u_{j,FMM}$ and $v_{j,FMM}$ are the x - and y -components of the induced velocity at the center of the j -th blob computed using the FMM subroutines. The velocities computed from direct summation are given by $u_{j,direct}$ and $v_{j,direct}$. The averaged relative errors are simply defined as

$$\bar{E}_u = \frac{1}{N} \sum_{j=1}^N E_{u,j} \quad \bar{E}_v = \frac{1}{N} \sum_{j=1}^N E_{v,j} \quad (2.12)$$

The averaged relative errors are computed for both the single-thread and the GPU-accelerated FMM subroutines. The results are compiled in Table. 2.1.

From Table. 2.1, we can see that the factor of acceleration is typically around 60 for the GPU-accelerated FMM subroutine, except for the case of $N = 10^6$ where the factor of acceleration drops to around 30. This indicates that the performance improvement is sensitive to the density and distribution of the vortex blobs. Nevertheless, the performance improvement is significant when GPU is used. For all four cases, the FMM subroutines show adequate accuracy with averaged relative errors in the induced velocity typically on the order of 10^{-11} .

2.3.1.2 Body rotation induced flow velocity

The rotating-body-induced velocity term in Eq. (2.9) involves a double integral over the area occupied by each body. However, utilizing the fact that the vorticity inside each body is uniform, the area integral can be replaced by a more efficient contour integral along the body boundary. It can be easily shown that

$$\frac{\mathbf{r} - \mathbf{r}_0}{|\mathbf{r} - \mathbf{r}_0|^2} = -\nabla_{\mathbf{r}_0} \ln |\mathbf{r} - \mathbf{r}_0| \quad (2.13)$$

where $\nabla_{\mathbf{r}_0}$ represents the gradient with respect to $\mathbf{r}_0$. Using the divergence theorem, we arrive at the following alternative expression for body-rotation-induced flow velocity [77]:

$$\sum_{i=1}^{N_B} \frac{\Omega_i}{\pi} \iint_{B_i} \frac{\mathbf{k} \times (\mathbf{r} - \mathbf{r}_0)}{|\mathbf{r} - \mathbf{r}_0|^2} d\mathbf{r}_0 = - \sum_{i=1}^{N_B} \frac{\Omega_i}{\pi} \int_{\partial B_i} \mathbf{s}_i(\mathbf{r}_0) \ln |\mathbf{r} - \mathbf{r}_0| ds \quad (2.14)$$

where $\mathbf{s}_i(\mathbf{r}_0)$ is the unit tangent vector of the body contour ∂B_i at $\mathbf{r}_0$ in the counterclockwise direction.

If $\mathbf{r}$ is not too close to the body boundary panels, the contour integral can be easily integrated numerically using the trapezoidal rule:

$$\int_{\partial B_i} \mathbf{s}_i(\mathbf{r}_0) \ln |\mathbf{r} - \mathbf{r}_0| ds \approx \sum_{k=1}^{N_{p,i}} \mathbf{s}_{i,k} \frac{\ln |\mathbf{r} - \mathbf{r}_{i,k}| + \ln |\mathbf{r} - \mathbf{r}_{i,k+1}|}{2} \ell_k \quad (2.15)$$

where $N_{p,i}$ is the number of linear panels in the body contour ∂B_i . Also, ℓ_k , $\mathbf{s}_{i,k}$, $\mathbf{r}_{i,k}$, and $\mathbf{r}_{i,k+1}$ are the length, constant unit tangent vector, and the two endpoints of the k -th panel of the i -th body, respectively. The simplicity of the trapezoidal rule means that the integration can be implemented on the GPU in a relatively straightforward manner for better performance. However, if a body panel is too close to the location $\mathbf{r}$, the integrand becomes almost singular, and the trapezoidal rule is not adequate. In this case, an adaptive quadrature with three-point Simpson's rule and five-point double Simpson's rule is used on that panel. Unfortunately, the adaptive quadrature is recursive in nature and is more easily implemented on the CPU. A simple solution is to use trapezoidal integration on the GPU whenever the body panel is reasonably far away from $\mathbf{r}$, which is true for the vast majority of the time. Adaptive quadrature on the CPU is only used when any part of a body panel falls within a certain distance to $\mathbf{r}$. The threshold distance that triggers adaptive quadrature is chosen to be three times the maximum panel size. Again, since the CPU and GPU can operate independently, the adaptive-quadrature integration and trapezoidal-rule integration can be carried out simultaneously.

2.3.2 Treatment of diffusion process

The evaluation of the diffusion velocities requires the local vorticity and vorticity gradient at the center of each vortex blob. Because of the lack of a mesh, computing derivatives presents

some challenge in DVM as well as in other Lagrangian methods. In Ogami and Akamatsu's original formulation, the vorticity field is directly constructed by superposing the vorticity of many overlapping Gaussian vortex blobs of fixed radius [8]. As simulation progresses, the separation between vortex blobs increases as a result of diffusion, and the overlap between blobs is lost in the far wake. The loss of overlap leads to zero vorticity gradient at the center of vortex blobs, preventing further diffusion. This issue is documented by Clarke and Tutty [18] who proposed a hybrid method as a solution. In the hybrid method, diffusion velocity is used for vortex blobs near the body where overlap between blobs is expected. As blobs move away from the body, diffusion velocity is gradually graded to random walk. This method allows diffusion to continue in the far wake.

In the current implementation, the local vorticity and vorticity gradient are determined using the Viscous-Vortex-Domain (VVD) method [19] which circumvents the issue with the loss of blob overlap altogether. In VVD, the vorticity ω at some point $\mathbf{r}$ in the flow field is approximated using an exponential smoothing kernel of an adaptive radius ϵ that depends on the local vortex blob spacing. When the gradient of the vorticity is needed, the gradient of the exponential kernel is used instead. This approach is mathematically similar to that being used in SPH, another popular Lagrangian particle method [78].

At a point $\mathbf{r}$ in the flow field, the vorticity can be given by

$$\begin{aligned}\omega(\mathbf{r}) &= \lim_{\epsilon \rightarrow 0} \frac{1}{I(\mathbf{r})} \iint_D \omega(\mathbf{r}_0) e^{-r'/\epsilon} d\mathbf{r}_0 \\ I(\mathbf{r}) &= \iint_D e^{-r'/\epsilon} d\mathbf{r}_0\end{aligned}\tag{2.16}$$

where $d\mathbf{r}_0 = dx dy$ is a differential area of the flow field. Also, we define $\mathbf{r}' \equiv \mathbf{r}_0 - \mathbf{r}$, and $r' = |\mathbf{r}'|$. By taking a sufficiently small value for ϵ , we can take the following approximation:

$$\omega(\mathbf{r}) \approx \frac{1}{I(\mathbf{r})} \iint_D \omega(\mathbf{r}_0) e^{-r'/\epsilon} d\mathbf{r}_0\tag{2.17}$$

It is of interest to note that we have $I(\mathbf{r}) = 2\pi\epsilon^2$ for any $\mathbf{r}$ in an infinite fluid. However, when solid bodies are present in the fluid, a correction is needed because the areas occupied by the bodies must be excluded from the integration:

$$I(\mathbf{r}) = 2\pi\epsilon^2 - \sum_{i=1}^{N_B} \iint_{B_i} e^{-r'/\epsilon} d\mathbf{r}_0\tag{2.18}$$

Using the divergence theorem, we can replace the area integral in Eq. (2.18) with contour integrals along body boundaries:

$$I(\mathbf{r}) = 2\pi\epsilon^2 - \epsilon \sum_{i=1}^{N_B} \int_{\partial B_i} -(\mathbf{n}_i \cdot \mathbf{r}') \frac{r' + \epsilon}{r'^2} e^{-r'/\epsilon} ds\tag{2.19}$$

The following identity has been invoked [19]:

$$-\frac{e^{-r'/\epsilon}}{\epsilon} = \nabla_{\mathbf{r}_0} \cdot \left(\frac{\mathbf{r}'(r' + \epsilon)}{r'^2} e^{-r'/\epsilon} \right) \quad (2.20)$$

The correction term associated with the presence of the body decays exponentially as $\mathbf{r}$ moves away from the body boundaries.

As discussed earlier, an integral representation of the vorticity gradient can be derived from Eq. (2.17). It can be easily shown that the gradient of the exponential kernel is given by

$$\nabla_{\mathbf{r}} e^{-r'/\epsilon} = e^{-r'/\epsilon} \frac{\mathbf{r}'}{\epsilon r'} \quad (2.21)$$

Therefore, with the application of chain rule, the integral representation of the vorticity gradient at $\mathbf{r}$ is simply given by

$$\begin{aligned} \nabla_{\mathbf{r}} \omega(\mathbf{r}) &= \frac{1}{I(\mathbf{r})} \iint_D \omega(\mathbf{r}_0) \nabla_{\mathbf{r}}(e^{-r'/\epsilon}) d\mathbf{r}_0 - \frac{\nabla_{\mathbf{r}} I(\mathbf{r})}{I^2(\mathbf{r})} \iint_D \omega(\mathbf{r}_0) e^{-r'/\epsilon} d\mathbf{r}_0 \\ &= \frac{1}{I(\mathbf{r})} \iint_D \omega(\mathbf{r}_0) e^{-r'/\epsilon} \frac{\mathbf{r}'}{\epsilon r'} d\mathbf{r}_0 - \frac{1}{I^2(\mathbf{r})} \left[\iint_D \omega(\mathbf{r}_0) e^{-r'/\epsilon} d\mathbf{r}_0 \right] \left[\iint_D e^{-r'/\epsilon} \frac{\mathbf{r}'}{\epsilon r'} d\mathbf{r}_0 \right] \end{aligned} \quad (2.22)$$

Using the definition given in Eq. (2.8), we arrive at the following equation that expresses the diffusion velocity in terms of the vorticity in the fluid domain D :

$$\mathbf{u}_d = -\nu \iint_D \omega(\mathbf{r}_0) e^{-r'/\epsilon} \frac{\mathbf{r}'}{\epsilon r'} d\mathbf{r}_0 \left(\iint_D \omega(\mathbf{r}_0) e^{-r'/\epsilon} d\mathbf{r}_0 \right)^{-1} + \frac{\nu}{I(\mathbf{r})} \iint_D e^{-r'/\epsilon} \frac{\mathbf{r}'}{\epsilon r'} d\mathbf{r}_0 \quad (2.23)$$

where $\mathbf{u}_d = (u_d, v_d)$. As pointed out in [19], the second term on the RHS of Eq. (2.23) is only present when the point $\mathbf{r}$ is close to the body boundary and decays exponentially when $\mathbf{r}$ moves away from the body. The term can be more easily understood and computed in its alternative form. Using divergence theorem again, we have

$$\frac{\nu}{I(\mathbf{r})} \iint_D e^{-r'/\epsilon} \frac{\mathbf{r}'}{\epsilon r'} d\mathbf{r}_0 = -\frac{\nu}{I(\mathbf{r})} \sum_{i=1}^{N_B} \iint_{B_i} e^{-r'/\epsilon} \frac{\mathbf{r}'}{\epsilon r'} d\mathbf{r}_0 = \frac{\nu}{I(\mathbf{r})} \sum_{i=1}^{N_B} \int_{\partial B_i} \mathbf{n}_i e^{-r'/\epsilon} ds \quad (2.24)$$

where $\mathbf{n}_i$ is the unit normal vector of body boundary ∂B_i pointing away from the body and into the fluid. From Eq. (2.24), it can be easily observed that the second term on the RHS of Eq. (2.23) is essentially a repulsion velocity that pushes the vortex blobs away from the body boundaries [19]. The direction and magnitude of this repulsion velocity only depend on the geometric arrangement and the kernel width ϵ . Also, this repulsion velocity is only

present when the distance from $\mathbf{r}$ to the body boundaries is comparable to ϵ . It appears that this repulsion effect is only a consequence of the numerical formulation being used and is not directly related to any physical flow mechanism.

In order to express the diffusion velocity in terms of the strengths and positions of vortex blobs, the discrete form of Eq. (2.23) can be used. The diffusion velocity $\mathbf{u}_{d,m} = \mathbf{u}_d(\mathbf{r}_m)$ at the center of the m -th vortex blob located at $\mathbf{r}_m$ is given by

$$\mathbf{u}_{d,m} = -\nu \left(\sum_{j=1, j \neq m}^N \Gamma_j e^{-r_{jm}/\epsilon_m} \frac{\mathbf{r}_{jm}}{\epsilon_m r_{jm}} \right) \left(\sum_{j=1}^N \Gamma_j e^{-r_{jm}/\epsilon_m} \right)^{-1} + \frac{\nu}{I_m} \sum_{i=1}^{N_B} \sum_{k=1}^{N_{p,i}} \mathbf{n}_{i,k} \int_{\partial B_{i,k}} e^{-r'_m/\epsilon_m} ds \quad (2.25)$$

$$I_m = I(\mathbf{r}_m) = 2\pi\epsilon_m^2 - \epsilon_m \sum_{i=1}^{N_B} \sum_{k=1}^{N_{p,i}} \mathbf{n}_{i,k} \cdot \left(\int_{\partial B_{i,k}} \mathbf{r}'_m \frac{r'_m + \epsilon_m}{r_m'^2} e^{-r'_m/\epsilon_m} ds \right) \quad (2.26)$$

where $\mathbf{r}_{jm} = \mathbf{r}_j - \mathbf{r}_m$ with $r_{jm} = |\mathbf{r}_{jm}|$, $\mathbf{r}'_m = \mathbf{r}_0(s) - \mathbf{r}_m$ with $r'_m = |\mathbf{r}'_m|$, and $\partial B_{i,k}$ denotes the k -th panel of the i -th body. The value of ϵ_m is chosen to be the distance from $\mathbf{r}_m$ to the nearest vortex blob multiplied by a safety factor that is greater than one.

The VVD formulation for vorticity diffusion described here is superior to the original RVM. The current approach is deterministic; thus, convergence is improved, and the noise in the results caused by the random vortex motion in RVM is removed. Also, the diffusion-velocity formulation allows the merging of the advection and diffusion steps, removing the error from the use of fractional steps in RVM and allowing the use of high-order time-integration schemes. However, the current approach has one numerical issue: the diffusion velocity can become excessively large in regions with near-zero vorticity but non-zero vorticity gradient, leading to very chaotic vortex blob distributions. This situation is usually a consequence of local “cancellation” between vortex blobs with positive and negative circulations. One simple solution to this issue, proposed and adopted by Ogami [79], is to compute the diffusion velocities for positive and negative vortex blobs separately, thus avoiding “cancellation.” This is justified by the fact that the diffusion process is linear [79]. Good quality solutions were obtained with this treatment as shown in Chapter 3.

2.3.2.1 Computation of diffusion velocity without body boundaries

Although the mathematical development of VVD is outlined in [19], an efficient strategy for evaluating the diffusion velocity is not provided. An efficient implementation is very important because, unlike the random-walk step which requires almost no computing effort compared to the advection step, the diffusion velocity requires considerable work to compute. With a primitive implementation, the computing time spent on evaluating the diffusion velocity can easily dominate over the time taken up by our GPU-accelerated FMM subroutine for advection.

The RHS of Eq. (2.25) shows that the diffusion velocity is the sum of two contributions: one from vortex blobs in the flow (first term of RHS) and one from body panels (second term of RHS). The two parts can be computed separately. The computation of the first term, which is essentially the diffusion velocity without body boundaries, is considered first.

A verbatim implementation of this term will, again, require a workload of $O(N)$ to compute the diffusion velocity for each vortex blob, resulting in a total workload of $O(N^2)$. Fortunately, the kernel e^{-r_{jm}/ϵ_m} decays to zero rapidly with increasing distance to r_m ; therefore, only the vortex blobs in the neighborhood of the m -th vortex blob are needed in the summation when computing $\mathbf{u}_{d,m}$, potentially reducing the total workload to $O(N)$. This is different from the advection velocity where the interaction between vortex blobs is only inversely proportional to the distance. In this sense, the key to rapidly evaluating the first term on the RHS of Eq. (2.25) lies in an efficient neighbor search algorithm (also needs to be $O(N \log N)$ workload).

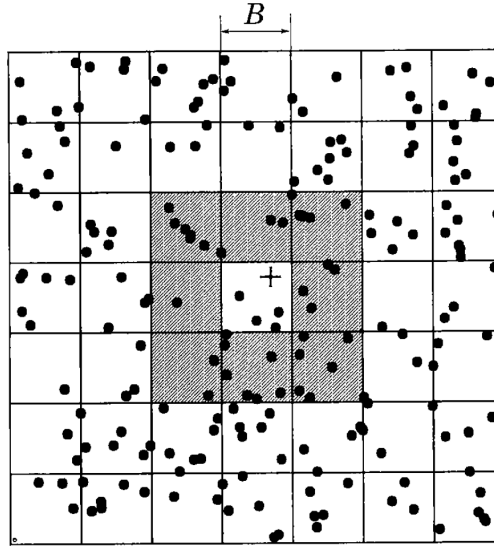


Figure 2.3: Block for diffusion velocity (Dots represent vortex blobs and the “+” sign marks the calculation point) [80].

If the same kernel radius is used for all vortex blobs, $\epsilon_m = \epsilon_0$ for $m = 1, \dots, N$, the relatively straight forward method given in [80] can be used. As shown in Fig. 2.3, the domain is partitioned into square regions of size B -by- B . When evaluating the first term on the RHS of Eq. (2.25) for the calculation point labeled by a “+” sign, only the vortex blobs belonging to the center block, which contains the calculation point, and the eight neighboring blocks (shaded) are included in the summations. This procedure guarantees that all vortex blobs within a distance B to the calculation point are retained. The error of the calculation is controlled by the ratio B/ϵ_0 . This simple method is also relatively trivial to implement on the GPU; however, it cannot be efficiently applied to the current formulation where the value of ϵ_m varies considerably from vortex to vortex depending on the local vortex blob

spacing. To maintain the level of accuracy everywhere in the flow, the value of B needs to be chosen based on the largest ϵ_m , leading to very large blocks and long computing time.

To accommodate nonuniform ϵ_m , a more robust and adaptive neighbor-search algorithm based on a k -d tree or a quadtree (see Sec. 2.3.1.1) is preferred [81]. Both tree data structures spatially partition the entire domain into blocks. Each block is called a node, and each node is recursively subdivided into two (k -d tree) or four (quadtree) child nodes until each node contains less than N_{max} vortex blobs, at which point the node is designated as a childless node (also called a leaf node) and is not subdivided any further. This process creates a data structure resembling an upside-down tree as shown in Fig. 2.4. As an example, the childless nodes (blocks) of a typical flow instant is illustrated in Fig. 2.2. For the diffusion step, the value of N_{max} is chosen to be 128. Once the tree data structure is constructed, a Nearest Neighbor Search (NNS) is performed first to identify the nearest neighbor of each vortex blob in order to determine the value of ϵ_m . Each NNS query requires one tree traversal with a workload of $O(\log N)$. A tree traversal starts from the root node (*i.e.* the entire domain) and moves down the tree level-by-level until the childless node containing the query point is reached. The distance, r_{best} , from the query point to the nearest vortex blob inside this childless node is computed and recorded as the nearest distance so far. Next, the tree data structure is traversed from the root node containing the query point upwards level-by-level, checking all other branches of the tree along the way until the root node is reached again. In this phase of the tree traversal, only neighboring nodes within a distance of r_{best} to the query point are qualified and need to be checked; the value of r_{best} is updated if the qualified neighboring node contains a vortex blob whose distance to the query point is less than the current value of r_{best} . If the distance of a node to the query point is larger than r_{best} , this node, as well as any children of this node, can be skipped. As a result, we only need to check the vortex blobs belonging to a small number of childless nodes. The total workload for the NNS phase to determine ϵ_m is $O(N \log N)$. In practice, the search for the nearest neighbor takes negligible time even for large number of vortex blobs.

Once ϵ_m , $m = 1, \dots, N$ are determined, a fixed range search is performed for each vortex blob. For the m -th vortex blob, any other vortex blob that is within distance $c \cdot \epsilon_m$ to $\mathbf{r}_m$ is identified. These vortex blobs are used to evaluate the summations in the numerator and denominator of the first term on the RHS of Eq. (2.25). To perform the range search, a tree traversal similar to that of NNS is conducted. Also, the same tree data structure constructed for NNS can be reused. The only difference is that, instead of a varying r_{best} , we use a fixed range $c \cdot \epsilon_m$ to determine whether a node needs to be checked. The constant c is chosen to be 10. The exponential kernel decays to $e^{-10} = 4.5 \times 10^{-5}$ at the distance $10\epsilon_m$, and neglecting vortex blobs beyond this distance yields acceptable error. The computational workload of this step is also of $O(N \log N)$; however, it is significantly more computationally intensive than the NNS step. Once the two summations over vortex blobs are performed, the first term of the RHS of Eq. (2.25) can be easily computed.

The program for tree traversal used in the search algorithm described above is recursive (a function calling itself) in nature and relies on complicated memory access patterns. This makes an efficient GPU implementation challenging. In fact, tree traversal on the GPU is

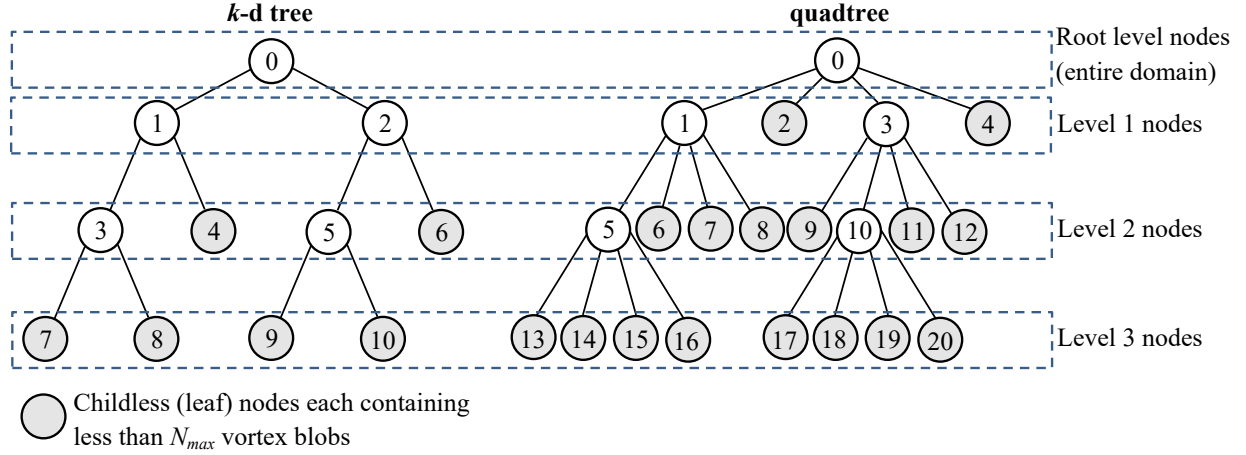


Figure 2.4: k -d tree and quadtree data structures for neighbor search.

still an active area of research. In order to utilize the computing power of the GPU without having to contend with this difficulty, we have adopted a hybrid approach that combines both a simple search method similar to [80] and the more sophisticated search algorithm based on tree data structures. In this approach, a quadtree data structure is constructed first on the CPU. Next, a parallelized subroutine that runs on multiple CPU cores is executed to perform NNS and determine the value of ϵ_m for all vortex blobs. Both steps consume negligible time. Subsequently, the evaluation of the two summations in the RHS of Eq. (2.25), the most computationally intensive step in the evaluation of diffusion velocity, is carried out on both the CPU and the GPU. A critical kernel radius of $\epsilon_{crit} = 4\sigma$ is chosen. If $\epsilon_m > \epsilon_{crit}$, the diffusion velocity of the m -th vortex blob is computed on the CPU by performing a tree traversal on the quadtree. If $\epsilon_m \leq \epsilon_{crit}$, the diffusion velocity is computed on the GPU. The GPU only utilizes the childless nodes of the quadtree. The vortex blobs belonging to the same childless node that satisfy the requirement $\epsilon_m \leq \epsilon_{crit}$ are processed in parallel using the large number of threads available on the GPU (also, multiple childless nodes are being processed concurrently). The vortex blobs inside the current childless node as well as nearby childless nodes within a distance of $10\epsilon_{crit}$ are used to compute the diffusion velocity. Similar to the method of [80], this procedure guarantees that all neighboring vortex blobs within a distance of $10\epsilon_{crit}$, which is greater than $10\epsilon_m$, are included in the summations.

It is observed that the majority of vortex blobs have relatively small $\epsilon_m \leq \epsilon_{crit}$; thus, the GPU is shouldering most of the computation, and the CPU is only used to process the more “difficult” cases with larger ϵ_m . Of course, the value of ϵ_{crit} can also be fine tuned to balance the workload between the GPU and the CPU for better performance.

2.3.2.2 Computation of the boundary repulsion velocity

The second contribution to the diffusion velocity is the boundary repulsion velocity given by the second term on the RHS of Eq. (2.25). The two integrals over panel k of body i ,

$\partial B_{i,k}$ in both Eq. (2.25) and Eq. (2.26) need to be evaluated. In particular, the integrand of the boundary integral in Eq. (2.26) can become almost singular if the computation point $\mathbf{r}_m$ is very close to the panel $\partial B_{i,k}$. To ensure the accuracy of the integration, the adaptive quadrature with three-point Simpson's rule and five-point double Simpson's rule mentioned in Sec. 2.3.1.2 is used here again.

The computation of the boundary repulsion velocity can be sped up significantly by exploiting the fact that the integrands in both boundary integrals decay exponentially with increasing distance between the computation point $\mathbf{r}_m$ and the panel $\partial B_{i,k}$. Therefore, in the current implementation, the integrals over panel $\partial B_{i,k}$ are computed only if any part of this panel falls within a distance of $15\epsilon_m$ to the point $\mathbf{r}_m$. Otherwise, the contributions from panel $\partial B_{i,k}$ are neglected from the summations.

To achieve this strategy efficiently, a tight bounding box is computed for each body in the flow first. If the point $\mathbf{r}_m$ is not inside any bounding box and the distance from $\mathbf{r}_m$ to any bounding box is more than $15\epsilon_m$, it is safe to approximate the boundary repulsion velocity at $\mathbf{r}_m$ as zero. With this simple check that costs negligible computing time, the vast majority of the wake vortex blobs far from any body surface can be excluded from the computation of body repulsion velocity. On the other hand, if $\mathbf{r}_m$ is inside a bounding box or within a distance of $15\epsilon_m$ from any bounding box, we will have to check for any body panel that might at least partially fall within a distance of $15\epsilon_m$ to the point $\mathbf{r}_m$. This check can be done efficiently using the range search algorithm with a k -d tree or quadtree described in Sec. 2.3.2.1. More specifically, instead of search for neighboring vortex blobs as done in Sec. 2.3.2.1, the range search algorithm is used to identify any panel endpoint whose distance to $\mathbf{r}_m$ is less than $15\epsilon_m$ plus the maximum panel length. If this distance is exceeded, it can be guaranteed that no part of the panel associated with this endpoint is within a distance of $15\epsilon_m$ to $\mathbf{r}_m$, and the contributions from this panel can be safely neglected. Otherwise, the boundary integrals in Eq. (2.25) and Eq. (2.26) are evaluated over this panel using adaptive quadrature, and the contributions from this panel need to be included in the summations. The procedure described above significantly reduces the computational workload and allows for very efficient and accurate evaluation of the boundary repulsion velocity.

2.4 Creation of vortex blobs and enforcement of boundary conditions

At the end of each time step, new vortex blobs are created near the body surface to simulate the vorticity creation process. First, a vortex sheet is constructed along the contour of each body, and the strengths of the sheets are determined by satisfying the no-penetration velocity boundary condition on each body contour simultaneously. For clarity, the case with a single rigid body is considered first, and the subscript associated with body number is dropped from the variable names. The complex conjugate velocity $w_\gamma = u_\gamma - iv_\gamma$ induced by the vortex sheet $\gamma(s)$ along the body contour ∂B is given by

$$w_\gamma(z) = -\frac{i}{2\pi} \int_{\partial B} \frac{\gamma(s)}{z - z_0(s)} ds \quad (2.27)$$

where $z = x + iy$ and s is the arclength coordinate along ∂B . Numerically, the body contour is separated into N_p linear panels. We assume $\gamma(s)$ is continuous and varies linearly over each panel. In this case, we can express the vortex sheet strength γ as

$$\gamma(s) = \sum_{k=1}^{N_p} \Lambda_k \gamma_k \quad (2.28)$$

where

$$\Lambda_k(z_0) = \begin{cases} \frac{z_0 - z_{k-1}}{z_k - z_{k-1}} & \text{if } z_0 \text{ is on panel } k-1 \\ \frac{z_0 - z_{k+1}}{z_k - z_{k+1}} & \text{if } z_0 \text{ is on panel } k \\ 0 & \text{otherwise} \end{cases} \quad (2.29)$$

In Eq. (2.29), z_k is the complex coordinate of the k -th node of ∂B . Following a simple substitution, we have

$$w_\gamma(z) = -\frac{i}{2\pi} \sum_{k=1}^{N_p} \gamma_k \int_{\partial B} \frac{\Lambda_k(z_0)}{z - z_0} ds \quad (2.30)$$

where z_0 is a function of s only. Assuming the body nodes are ordered in the counterclockwise direction, we have the following relation on the k -th panel:

$$\left. \frac{dz_0}{ds} \right|_k = \frac{z_{k+1} - z_k}{|z_{k+1} - z_k|} = -(\mathbf{n}_k)_y + i(\mathbf{n}_k)_x \quad (2.31)$$

The unit normal vector on the k -th panel is $\mathbf{n}_k$. By changing the variable from s to z_0 , the integral in Eq. (2.30) can be evaluated analytically, leading to the following expression for w_γ :

$$w_\gamma(z) = \frac{i}{2\pi} \sum_{k=1}^{N_p} \gamma_k \left[\left(\frac{z - z_{k-1}}{z_k - z_{k-1}} \ln \frac{z_k - z}{z_{k-1} - z} + 1 \right) \frac{ds}{dz_0} \Big|_{k-1} + \left(\frac{z - z_{k+1}}{z_k - z_{k+1}} \ln \frac{z - z_{k+1}}{z - z_k} - 1 \right) \frac{ds}{dz_0} \Big|_k \right] \quad (2.32)$$

The unknowns γ_k , $k = 1, 2, \dots, N_p$ can be solved for by enforcing the no-penetration boundary condition at a set of N_p control points on the body boundaries. Here, we shall use the body panel midpoints $\mathbf{r}_m$, $m = 1, 2, \dots, N_p$ as the control points. The corresponding

2.4. Creation of vortex blobs and enforcement of boundary conditions

complex coordinate is z_m . The flow velocities induced by the vortex sheets at these points are given by

$$w_\gamma(z_m) = \sum_{k=1}^{N_p} \mathbf{A}_{mk} \gamma_k \quad (2.33)$$

where

$$\begin{aligned} \mathbf{A}_{mk} = \frac{i}{2\pi} & \left[\left(\frac{z_m - z_{k-1}}{z_k - z_{k-1}} \ln \frac{z_k - z_m}{z_{k-1} - z_m} + 1 \right) \frac{ds}{dz_0} \Big|_{k-1} \right. \\ & \left. + \left(\frac{z_m - z_{k+1}}{z_k - z_{k+1}} \ln \frac{z_m - z_{k+1}}{z_m - z_k} - 1 \right) \frac{ds}{dz_0} \Big|_k \right] \end{aligned} \quad (2.34)$$

Now, we can state the no-penetration boundary condition on the body surfaces as:

$$\Im \left(\sum_{k=1}^{N_p} \mathbf{A}_{mk} \gamma_k \frac{dz}{ds} \Big|_m \right) + \mathbf{u}(\mathbf{r}_m) \cdot \mathbf{n}_m = (\mathbf{u}_c + \boldsymbol{\Omega} \times (\mathbf{r}_m - \mathbf{r}_c)) \cdot \mathbf{n}_m \quad m = 1, \dots, N_p \quad (2.35)$$

where $\boldsymbol{\Omega} = \Omega \mathbf{k}$ is the angular velocity of the body and the velocity $\mathbf{u}(r_m)$ is induced by all existing vortex blobs and rotating bodies as defined in Eq. (2.9). The symbol $\Im$ denotes the imaginary part.

Equation (2.35) forms an N_p -by- N_p linear system of equations. However, it is well known that the rank of the system is only $N_p - 1$. This is because the vortex sheet automatically satisfies conservation of mass (Laplace's equation). If the mass fluxes through the first $N_p - 1$ body panels satisfy the no-penetration boundary condition, the flux through the last body panel N_p will automatically conform with the no-penetration boundary condition by virtue of conservation of mass (there is no source or sink inside the body). As a result, the last equation obtained from Eq. (2.35) with $m = N_p$ is linearly dependent on the other $N_p - 1$ equations. In order to acquire a unique solution for γ_k , an additional constraint that specifies the net rate of creation of circulation is required [77]:

$$2 \frac{d\Omega}{dt} A - \oint_{\partial B} \nu \frac{\partial \omega}{\partial n} ds = 0 \quad (2.36)$$

where A is the area occupied by the body bounded by ∂B . The first term represents the change in the circulation of the imaginary fluid inside the body because of body angular acceleration. The second term on the left-hand side (LHS) of Eq. (2.36) is the net rate of creation of circulation from the body surface, with $-\nu(\partial \omega / \partial n)$ being the vorticity flux away from the surface.

Equation (2.36) is similar to Kelvin's circulation theorem. It requires the total circulation in the flow, including the circulation $2\Omega A$ of the imaginary fluid inside the body, to remain

2.4. Creation of vortex blobs and enforcement of boundary conditions

constant over time. A proof of this relation can be obtained from the fact that pressure needs to be single-valued on the body surface (see Sec. 2.5.1).

In order to derive the time-discretized form of Eq. (2.36), we should first recognize that the circulation created over one time step Δt per unit length of the body contour is given by

$$-\nu \frac{\partial \omega}{\partial n} \Delta t = \gamma^q - \gamma_L^q \quad \text{on } \partial B \quad (2.37)$$

where γ^q is the strength of the vortex sheet created along the body contour at step q . Owing to the inaccuracies in the numerical time integration of vortex blob positions, a few vortex blobs will inevitably cross the body boundary ∂B and enter the interior of the body. These vortex blobs are deleted from the computation. This loss is accounted for through γ_L^q . The time-discretized version of Eq. (2.36) is now in the form

$$\oint_{\partial B} \gamma^q ds - \Gamma_L^q + 2(\Omega^q - \Omega^{q-1})A = 0 \quad (2.38)$$

where Ω^q is the angular velocity of the body at step q . The sum of the circulation of the vortex blobs lost from crossing the body contour during step $q-1$ to q is given by Γ_L^q :

$$\Gamma_L^q = \oint_{\partial B} \gamma_L^q ds \quad (2.39)$$

Combining Eq. (2.35) and Eq. (2.38), we have $N_p + 1$ equations (only N_p linearly independent equations) and N_p unknowns which can be solved using the least-square method. Once γ^q is known, the vortex sheet can be discretized into N_p vortex blobs. The vortex sheet on each panel is replaced by a nascent Gaussian vortex blob of equal circulation. The core radius of the nascent vortex blobs is chosen to be the same as the panel size. Also, the center positions of the nascent blobs are offset by a small distance δ in the normal direction from the panel midpoints. We choose $\delta = 1.12\sigma$ based on the argument of Clarke and Tutty [18]. The choice of blob radius ensures that the vortex blobs start out with a high degree of overlap near the body which is needed to represent a smooth vorticity field [11]. In subsequent time steps, these nascent vortex blobs participate in the advection and diffusion processes the same way as the rest of the vortex blobs, gradually moving away from the body surface and becoming part of the wake.

The introduction of nascent vortices at each time step also helps satisfy the no-slip boundary condition on the body surfaces, even though only the no-penetration condition is used when solving for γ^q . This is because, in the current DVM formulation, the no-slip and no-penetration boundary conditions are equivalent and consistent in their role of determining the vorticity creation on body surfaces. The no-slip condition is automatically satisfied if the no-penetration condition is enforced, and vice versa [82]. In fact, it is equally valid to solve for γ^q using the no-slip boundary condition and Eq. (2.38) [18]. However, it can be argued that the no-penetration condition, which is related to the conservation of mass, is more critical and should be the one that is enforced directly [82].

2.4. Creation of vortex blobs and enforcement of boundary conditions

In the procedure for vortex blob creation described above, we have used an intermediate construct of vortex sheets before discretizing them into vortex blobs. Of course, one may skip this intermediate step and directly determine the circulation of the nascent vortex blobs by enforcing the no-penetration condition. In this alternative approach, however, the circulation of the nascent blobs will have dependencies on the core radius, core structure, and the offset distance δ of the nascent blobs. The use of the intermediate vortex sheet has the benefit that the amount of circulation carried by the nascent blobs into the boundary layer is independent of the numerical parameters being used.

When multiple bodies are present in the flow, a similar procedure for vortex blob creation can be used. However, the strength of the vortex sheet bounding each body needs to be solved together to ensure that the no-penetration condition is simultaneously satisfied on all body boundaries. The flow velocity induced by the vortex sheets bounding other bodies also needs to be taken into account. More specifically, the system of equations in Eq. (2.35) needs to be modified into:

$$\Im \left(\sum_{j=1}^{N_B} \sum_{k=1}^{N_{p,j}} \mathbf{A}_{imjk} \gamma_{jk} \frac{dz}{ds} \Big|_{im} \right) + \mathbf{u}(\mathbf{r}_{im}) \cdot \mathbf{n}_{i,m} = (\mathbf{u}_{c,i} + \boldsymbol{\Omega}_i \times (\mathbf{r}_{im} - \mathbf{r}_{c,i})) \cdot \mathbf{n}_{i,m} \quad (2.40)$$

$i, j = 1, \dots, N_B, \quad m = 1, \dots, N_{p,i}, \quad \text{and} \quad k = 1, \dots, N_{p,j}$

where

$$\mathbf{A}_{imjk} = \frac{i}{2\pi} \left[\left(\frac{z_{im} - z_{jk-1}}{z_{jk} - z_{jk-1}} \ln \frac{z_{jk} - z_{im}}{z_{jk-1} - z_{im}} + 1 \right) \frac{ds}{dz_0} \Big|_{jk-1} + \left(\frac{z_{im} - z_{jk+1}}{z_{jk} - z_{jk+1}} \ln \frac{z_{im} - z_{jk+1}}{z_{im} - z_{jk}} - 1 \right) \frac{ds}{dz_0} \Big|_{jk} \right] \quad (2.41)$$

The subscript i and j represent the i -th and the j -th body, respectively. The double summation in Eq. (2.40) can be collapsed into one to form a single system of equations by replacing the subscript im with $\bar{m}$ and jk with $\bar{k}$ where

$$\bar{m} = \begin{cases} m, & \text{for } i = 1 \\ \sum_{i'=1}^{i-1} N_{p,i'} + m, & 2 \leq i \leq N_B \end{cases}, \quad \bar{k} = \begin{cases} k, & \text{for } j = 1 \\ \sum_{j'=1}^{j-1} N_{p,j'} + k, & 2 \leq j \leq N_B \end{cases} \quad (2.42)$$

Lastly, we need to constraint the net creation of circulation be enforcing Eq. (2.38) for each body individually. Again, this constraint stems from the requirement for pressure to be single-valued on each body.

2.5 Hydrodynamic force and moment

The hydrodynamic forces and moments exerted on the submerged bodies are an important part of the computational results. In two dimensions, the stress on a submerged body can be expressed using index notation as

$$\mathbf{f} = \sigma_{ij} n_j \mathbf{e}_i, \quad i, j = 1, 2 \quad (2.43)$$

where σ_{ij} is the stress tensor and $\mathbf{n} = (n_1, n_2) = (n_x, n_y)$ is the unit normal vector of the body surface pointing into the fluid. Also, $\mathbf{e}_1$ and $\mathbf{e}_2$ are the unit basis vectors $\mathbf{i}$ and $\mathbf{j}$, respectively.

For an incompressible Newtonian fluid, the stress tensor is given by

$$\sigma_{ij} = -p\delta_{ij} + \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right), \quad i, j = 1, 2 \quad (2.44)$$

where p is the pressure, δ_{ij} is Kronecker Delta, and μ is the dynamic viscosity. The flow velocity is given by $\mathbf{u} = (u_1, u_2) = (u, v)$. The stress can be split into a pressure component and a viscous component:

$$\mathbf{f} = \mathbf{f}_p + \mathbf{f}_v = -p\mathbf{n} + \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) n_j \mathbf{e}_i \quad (2.45)$$

Therefore, the determination of the total hydrodynamic force and moment on a submerged body can be done in two separated steps: the determination of pressure force/moment and the determination of viscous force/moment. As Reynolds number increases, the contribution from viscous stress decreases rapidly compared to the contribution from pressure. Therefore, the viscous component may be neglected for sufficiently high Reynolds numbers as done in SS-FSRVM [4]. Here, however, the viscous component is retained to accommodate low-Reynolds number flow.

Since the procedure for computing the force and moment is simply repeated for each body in the flow, the subscript associated with body number in the variable names is again dropped for clarity.

2.5.1 Pressure force and moment

With reference to [77], the pressure variation along the surface of a body submerged in a viscous fluid is closely related to the surface vorticity flux. The relation can be derived from the Navier-Stokes equation:

$$\frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho}\nabla p + \nu\nabla^2\mathbf{u} \quad (2.46)$$

By taking the tangential component of the Navier-Stokes equation on the body surface, we get

$$\frac{1}{\rho} \frac{\partial p}{\partial s} = -\mathbf{s} \cdot \frac{D\mathbf{u}}{Dt} + \nu(\mathbf{s} \cdot \nabla^2 \mathbf{u}) \quad \text{on } \partial B \quad (2.47)$$

To proceed, we need to invoke the following vector calculus identity:

$$\nabla \times (\nabla \times \mathbf{u}) = \nabla(\nabla \cdot \mathbf{u}) - \nabla^2 \mathbf{u} \quad (2.48)$$

Since we have a two-dimensional isochoric flow, $\nabla \cdot \mathbf{u} = 0$, and the following relation is obtained:

$$\nabla^2 \mathbf{u} = -\nabla \times (\nabla \times \mathbf{u}) = -\nabla \times (\omega \mathbf{k}) \quad (2.49)$$

If the unit tangent vector $\mathbf{s} = (s_x, s_y)$ is pointing in the counterclockwise direction, the unit normal vector $\mathbf{n}$ is related to $\mathbf{s}$ by the relations $n_x = s_y$ and $n_y = -s_x$. The tangential component of Eq. (2.49) becomes

$$\mathbf{s} \cdot \nabla^2 \mathbf{u} = \mathbf{n} \cdot \nabla \omega = \frac{\partial \omega}{\partial n} \quad (2.50)$$

Substitution of Eq. (2.50) into Eq. (2.47) leads to

$$\frac{1}{\rho} \frac{\partial p}{\partial s} = -\mathbf{s} \cdot \frac{D\mathbf{u}}{Dt} + \nu \frac{\partial \omega}{\partial n} \quad \text{on } \partial B \quad (2.51)$$

On the body surface ∂B , fluid velocity is the same as the body surface velocity because of the no-slip and no-penetration boundary conditions. From rigid-body translation and rotation, the velocity of a point $\mathbf{r}$ on the body surface (fixed relative to the body) is given by

$$\mathbf{u} = \mathbf{u}_c + \boldsymbol{\Omega} \times (\mathbf{r} - \mathbf{r}_c), \quad \mathbf{r} \text{ is on } \partial B \quad (2.52)$$

Using chain rule and the fact that $d\mathbf{r}/dt = \mathbf{u}$, we can now compute the tangential component of the pressure gradient on the body surface from the body motion and surface vorticity flux

$$\frac{1}{\rho} \frac{\partial p}{\partial s} = -\mathbf{s} \cdot \frac{d\mathbf{u}_c}{dt} - \frac{d\Omega}{dt} (\mathbf{r} - \mathbf{r}_c) \cdot \mathbf{n} + \Omega^2 (\mathbf{r} - \mathbf{r}_c) \cdot \mathbf{s} + \nu \frac{\partial \omega}{\partial n} \quad (2.53)$$

For prescribed motion, the first three terms on the RHS of Eq. (2.53) are known. The last term of Eq. (2.53), the surface vorticity flux, can be computed from the net creation of circulation using Eq. (2.37) as discussed in Sec. 2.4.

Now, the pressure along the body surface can be determined by integrating Eq. (2.53) along the arclength s using the trapezoidal rule:

$$p_{k+1} \approx p_k + \frac{1}{2} \left(\left. \frac{\partial p}{\partial s} \right|_k + \left. \frac{\partial p}{\partial s} \right|_{k+1} \right) \ell_k \quad (2.54)$$

where p_k is the pressure at the k -th body node and ℓ_k is the length of the k -th panel. This method can only determine the hydrodynamic pressure up to a constant. However, this

2.5. Hydrodynamic force and moment

unknown constant has no effect on the total hydrodynamic pressure force and moment on the body which can now be expressed as

$$\mathbf{F}_p \approx \sum_{k=1}^N -\mathbf{n}_k(p_k + p_{k+1})(\ell_k/2) \quad (2.55)$$

$$\mathbf{M}_p \approx \sum_{k=1}^N -(\mathbf{r}_k - \mathbf{r}_c) \times \mathbf{n}_k(p_k + p_{k+1})(\ell_k/2) \quad (2.56)$$

where $\mathbf{r}_k$ is the position of the midpoint of the k -th panel.

Because the tangential component of the local pressure gradient on the body surface is related to the surface vorticity flux, the single-valuedness of the surface pressure provides an additional constraint on the net rate of creation of circulation from the body surface. Using the divergence theorem and Eq. (2.53), we acquire the following equation:

$$\frac{1}{\rho} \oint_{\partial B} \frac{\partial p}{\partial s} ds = -2 \frac{d\Omega}{dt} A + \nu \oint_{\partial B} \frac{\partial \omega}{\partial n} ds \quad (2.57)$$

The single-valuedness of p requires Eq. (2.57) to evaluate to zero. This is exactly the same constraint given in Eq. (2.36) used to obtain a unique solution for the circulation of nascent vortex blobs. If multiple bodies are present, pressure must be single-valued on each body; therefore, it is justified to apply Eq. (2.36) to each body individually, leading to a stronger constraint than what global conservation of angular momentum implies [82].

2.5.2 Viscous force and moment

Following the development given in [20], the viscous stress can be decomposed into three parts:

$$\mathbf{f}_v = 2\mu(\nabla \cdot \mathbf{v})\mathbf{n} + 2\mu \left(-\frac{\partial v}{\partial s} \mathbf{i} + \frac{\partial u}{\partial s} \mathbf{j} \right) + \mu\omega \mathbf{s} \quad (2.58)$$

The first term on the RHS of Eq. (2.58) is zero for isochoric flow. The second term contains only derivatives of velocity along the body boundary. The no-slip and no-penetration boundary conditions require the velocity on body boundary to be given by Eq. (2.52). By differentiating Eq. (2.52), we acquire the final expression for viscous stress:

$$\mathbf{f}_v = -2\mu\Omega \mathbf{s} + \mu\omega \mathbf{s} \quad (2.59)$$

Integration of Eq. (2.59) along the body boundary yields the total viscous force $\mathbf{F}_v$:

$$\mathbf{F}_v = \oint_{\partial B} \mathbf{f}_v ds = \mu \oint_{\partial B} \omega \mathbf{s} ds \approx \mu \sum_{k=1}^N \omega_k \ell_k \mathbf{s}_k \quad (2.60)$$

2.6. Free motion of rigid bodies

The first term on the RHS of Eq. (2.59) has no net contribution to the total viscous force. Similarly, the viscous moment is given by

$$\begin{aligned}\mathbf{M}_v &= \oint_{\partial B} (\mathbf{r} - \mathbf{r}_c) \times \mathbf{f}_v \, ds = -4\mu\Omega A \mathbf{k} + \mu \oint_{\partial B} (\mathbf{r} - \mathbf{r}_c) \times \boldsymbol{\omega} \, ds \\ &\approx -4\mu\Omega A \mathbf{k} + \mu \sum_{k=1}^N (\mathbf{r}_k - \mathbf{r}_c) \times \omega_k \ell_k \mathbf{s}_k\end{aligned}\tag{2.61}$$

where Ω and A are the angular velocity and area of the body. The body surface vorticity on the k -th panel, ω_k , can be approximated using Eq. (2.16) at the panel midpoint.

With $\mathbf{F}_v$ and $\mathbf{M}_v$ determined, the total hydrodynamic force and moment on the body are simply given by

$$\mathbf{F} = \mathbf{F}_p + \mathbf{F}_v \tag{2.62}$$

$$\mathbf{M} = \mathbf{M}_p + \mathbf{M}_v \tag{2.63}$$

2.6 Free motion of rigid bodies

When free motions are considered, the time histories of the body velocities and positions are part of the output of the simulation. The external hydrodynamic force and moment influence the body motion through the Equations of Motion (EOM). For convenience, the case where only one body is present in the flow is considered first. The subscript associated with body number is dropped from the variable names.

The position of the body at time step q is determined using the two-step Adams-Bashforth method:

$$\mathbf{r}_c^q = \mathbf{r}_c^{q-1} + \left(\frac{3}{2} \mathbf{u}_c^{q-1} - \frac{1}{2} \mathbf{u}_c^{q-2} \right) \Delta t \tag{2.64}$$

$$\theta^q = \theta^{q-1} + \left(\frac{3}{2} \Omega^{q-1} - \frac{1}{2} \Omega^{q-2} \right) \Delta t \tag{2.65}$$

where θ^q is the angular position of the body at step q . The body contour at step q , ∂B^q , is determined by $\mathbf{r}_c^q$ and θ^q . In order to repeat the above procedure and advance to the next time step, the body velocities $\mathbf{u}_c^q$ and $\boldsymbol{\Omega}^q$ at time step q need to be determined. We shall assume the body reference center coincides with the body center of mass. In this case, the two-dimensional rigid-body EOM attain the simplest form:

$$\mathbf{F} + \mathbf{F}_{ex} = m \frac{d\mathbf{u}_c}{dt} \tag{2.66}$$

$$\mathbf{M} + \mathbf{M}_{ex} = I \frac{d\boldsymbol{\Omega}}{dt} \tag{2.67}$$

2.6. Free motion of rigid bodies

where m and I are the mass of the body and mass moment of inertia about the center of mass, respectively. Arbitrary external force and moment, such as a spring force or damping force, can be incorporated through $\mathbf{F}_{ex}$ and $\mathbf{M}_{ex}$. Of course, if the body reference center is not located at the center of mass, a more general set of EOM can be used instead. The time-discretized form of the EOM can be obtained using the backward Euler method:

$$\mathbf{u}_c^q - \mathbf{u}_c^{q-1} = \frac{\Delta t}{m}(\mathbf{F}^q + \mathbf{F}_{ex}^q) \quad (2.68)$$

$$\boldsymbol{\Omega}^q - \boldsymbol{\Omega}^{q-1} = \frac{\Delta t}{I}(\mathbf{M}^q + \mathbf{M}_{ex}^q) \quad (2.69)$$

The equations above allow us to determine the unknown velocities at step q , $\mathbf{u}_c^q$ and $\boldsymbol{\Omega}^q$, from the known velocities at step $q - 1$. However, as indicated by Eq. (2.53), the pressure components of the hydrodynamic force and moment $\mathbf{F}^q$ and $\mathbf{M}^q$ will also depend on the body translational and angular accelerations which are unknown. This is the added-mass effect. Further, the evaluation of the last term on the RHS of Eq. (2.53), which represents the body surface vorticity flux, requires the knowledge of γ^q as shown in Eq. (2.37). The vortex sheet strength γ^q is, again, determined from the unknown body velocity $\mathbf{u}_c^q$ and $\boldsymbol{\Omega}^q$. Therefore, the unknown vortex sheet strength γ^q and the unknown body velocities at step q need to be solved together.

The body accelerations can be approximated using the first-order backwards differencing:

$$\begin{aligned} \left(\frac{d\mathbf{u}_c}{dt}\right)^q &= \frac{\mathbf{u}_c^q - \mathbf{u}_c^{q-1}}{\Delta t} \\ \left(\frac{d\boldsymbol{\Omega}}{dt}\right)^q &= \frac{\boldsymbol{\Omega}^q - \boldsymbol{\Omega}^{q-1}}{\Delta t} \end{aligned} \quad (2.70)$$

The above approximation allows Eq. (2.53) to be rewritten as

$$\begin{aligned} \frac{1}{\rho} \left(\frac{\partial p}{\partial s}\right)^q &= -\mathbf{s} \cdot \frac{\mathbf{u}_c^q - \mathbf{u}_c^{q-1}}{\Delta t} - (\mathbf{r} - \mathbf{r}_c) \cdot \mathbf{n} \frac{\boldsymbol{\Omega}^q - \boldsymbol{\Omega}^{q-1}}{\Delta t} \\ &\quad + (\mathbf{r} - \mathbf{r}_c) \cdot \mathbf{s} [2\boldsymbol{\Omega}^{q-1}\boldsymbol{\Omega}^q - (\boldsymbol{\Omega}^{q-1})^2] + \frac{\gamma_L^q - \gamma^q}{\Delta t} \quad \mathbf{r} \in \partial B^q \end{aligned} \quad (2.71)$$

Here, we have approximated $(\boldsymbol{\Omega}^q)^2$ as $2\boldsymbol{\Omega}^{q-1}\boldsymbol{\Omega}^q - (\boldsymbol{\Omega}^{q-1})^2$ in order to obtain an expression that depends linearly on the unknowns $\mathbf{u}_c^q = (u_c^q, v_c^q)$, $\boldsymbol{\Omega}^q$, and γ^q . Following the procedure given in Sec. 2.5, Eq. (2.71) can be integrated with respect to s twice along ∂B^q to obtain expressions for $\mathbf{F}^q$ and $\mathbf{M}^q$ that are linear in the unknowns:

$$\mathbf{F}^q = \mathbf{F}(\gamma^q, u_c^q, v_c^q, \boldsymbol{\Omega}^q) \quad (2.72)$$

$$\mathbf{M}^q = \mathbf{M}(\gamma^q, u_c^q, v_c^q, \boldsymbol{\Omega}^q) \quad (2.73)$$

Next, the expressions for $\mathbf{F}^q$ and $\mathbf{M}^q$ thus obtained can be substituted into Eq. (2.68) and Eq. (2.69) to form three additional linear equations. These additional equations are to be solved simultaneously with Eq. (2.35) and Eq. (2.38) for the unknowns $\mathbf{u}_c^q$, Ω^q , and γ^q . Once the body velocities at step q are determined, the position of the body can be updated for the next time step $q + 1$.

If more than one body is present in the flow, we may repeat the above procedure and end up with a total of $3N_B$ additional equations and the same number of additional unknowns $u_{c,i}^q$, $v_{c,i}^q$, and Ω_i^q , $i = 1, \dots, N_B$. Similar to the case with a single body, the $3N_B$ additional equations need to be solved together with Eq. (2.40) and another N_B equations, one for each body, in the form of Eq. (2.38). Of course, it is not necessary for a body to have all three degrees of freedom. For instance, we can have the body i oscillate in the y -direction only by prescribing the values of $u_{c,i}^q$ and Ω_i^q and replacing the corresponding EOM.

2.7 Merging and decay of vortex blobs

In the current DVM formulation, the number of computational elements (vortex blobs) increases linearly with time. Therefore, the required computing time for longer simulations with a total number of time steps on the order of 10^4 or more can become prohibitively long. It is necessary to utilize techniques that slow down or even stop the increase in the number of vortex blobs with time. To this end, we have implemented merging and decay of vortex blobs.

2.7.1 Merging of vortex blobs

One approach to reduce the number of vortex blobs in the flow field is to merge vortex blobs that are sufficiently close to each other. For instance, if we have n neighboring vortex blobs labeled $j = 1, \dots, n$ located at $\mathbf{r}_j$ that are within a distance of R_m to vortex blob i at $\mathbf{r}_i$:

$$|\mathbf{r}_j - \mathbf{r}_i| \leq R_m \quad j = 1, \dots, n \quad (2.74)$$

and the circulations of vortex blobs i and j have the same sign, we can then merge the vortex blobs $j = 1, \dots, n$ with vortex blob i using the following relations:

$$\Gamma_0 = \Gamma_i + \sum_{j=1}^n \Gamma_j \quad (2.75)$$

$$\Gamma_0 \mathbf{r}_0 = \Gamma_i \mathbf{r}_i + \sum_{j=1}^n \Gamma_j \mathbf{r}_j \quad (2.76)$$

$$\Gamma_0 \sigma_0^2 = \Gamma_i (\sigma_i^2 + |\mathbf{r}_i - \mathbf{r}_0|^2) + \sum_{j=1}^n \Gamma_j (\sigma_j^2 + |\mathbf{r}_j - \mathbf{r}_0|^2) \quad (2.77)$$

where Γ_0 , $\mathbf{r}_0$, and σ_0 are the circulation, position, and core radius of the new merged vortex blob that replaces vortex blobs i and $j = 1, \dots, n$. The zeroth, first, and second moment of vorticity are conserved after the merging [83].

In terms of implementation, the merging of vortex blobs is done by performing a single sweep through all vortex blobs ($i = 1, \dots, N$) starting from the first vortex blob ($i = 1$). All vortex blobs with circulations in the same direction as vortex blob i and fall within a distance of R_m to vortex blob i are identified efficiently using a k -d tree or quadtree search algorithm. These vortex blobs are then merged with vortex blob i using Eqs. (2.75)-(2.77), and vortex blob i is replaced by the new merged vortex blob. Neighboring vortex blobs $j = 1, \dots, n$ are all removed. If no neighboring vortex blob that satisfies the criteria for merging is found, vortex blob i is left unchanged. After vortex blob i is processed, the sweep continues on to the next vortex blob $i + 1$.

The above procedure is relatively straightforward. However, care must be taken to prevent a single vortex blob from participating in multiple merging events. This is done by adding a label to each vortex blob indicating whether it is eligible for merging. Initially, all vortex blobs are labeled as eligible. Once vortex blob i is processed in the sweep, the label for vortex blob i is changed from eligible for merging to ineligible irrespective of whether vortex blob i has participated in a merging event. If, however, vortex blob i has indeed merged with other vortex blobs, all blobs that merged with blob i also need to be relabeled from eligible for merging to ineligible. Therefore, if the sweep reaches a vortex blob that has been previously labeled as ineligible, the sweep can skip this vortex blob because it has already participated in a prior merging event. Also, each vortex blob is only allowed to merge with other vortex blobs that are still labeled as eligible for merging. The above procedure guarantees that each vortex blob participates in at most one merging event. The merging of vortex blobs is performed once every five time steps to prevent excessive loss of computational elements.

The error caused by vortex blob merging is primarily controlled by the parameter R_m . In general, a smaller value of R_m leads to reduced error. At the same time, a smaller R_m means merging is less likely to occur, resulting in more vortex blobs and longer computing time. Typically, the value of R_m is chosen to be $0.05 \min_{i=1, \dots, N} (\sigma_i)$. This is a relatively stringent criterion for merging. If more aggressive merging is desired, the flow domain can be partitioned into different regions depending on the distance to the bodies, and a larger value of R_m can be used in regions further away from the bodies. This approach leads to drastically reduced number of vortex blobs in the far wake while maintaining the computational resolution near the bodies. If a strict error bound for vortex merging is required, additional merging constraints on vortex core sizes and core strengths can be implemented. Details on these additional constraints for Gaussian vortex elements can be found in [83].

2.7.2 Decay of vortex blobs

In addition to the merging of vortex blobs, the strengths (circulation) of the vortex blobs are also allowed to decay once they have moved sufficiently far away from the bodies. More specifically, a threshold distance of R_d from a body center can be chosen, and the circulation of a vortex blob will start to decay linearly with time once it moves past this threshold. After a fixed time T_{dc} , the circulation of the vortex blob reaches zero, and the vortex blob is removed from the computation. It should be noted that the decay of vortex blobs is a rather artificial numerical treatment aimed at reducing the number of vortex blobs. It is not intended to be a quantitative model for any physical vorticity dissipation process. The value of R_d needs to be large enough so that the effect of the wake on the body forces is adequately captured. It is typically chosen to be 20 or more body characteristic lengths for bodies submerged in a stream.

2.8 General numerical procedure

The individual components of the current DVM algorithm have been discussed in detail in Sec. 2.1-2.7. The overall numerical procedure is briefly presented here. The flowchart summarizing the algorithm for prescribed body motions is given in Fig. 2.5.

At the start of the simulation, the contours of the bodies are constructed according to the inputs. Based on the initial velocities of the bodies, the vortex sheets along body contours are solved for to satisfy the no-penetration boundary condition. The initial vortex sheets essentially represent an inviscid solution to the flow problem; the vorticity is concentrated in infinitely thin boundary layers on the bodies. The vortex sheets are subsequently discretized into nascent vortex blobs following the procedure described in Sec. 2.4.

To advance the solution in time, the positions of the vortex blobs are integrated in time using the 4th order Runge-Kutta method to simulate the advection and diffusion of vorticity as described in Sec. 2.3. For prescribed motions, the positions of the bodies are also updated using the same Runge-Kutta method based on prescribed body velocities. The rotation of the bodies also induces a flow velocity that needs to be accounted for when updating the positions of the vortex blobs. In addition, a few vortex blobs may enter the bodies during the *intermediate* steps of the Runge-Kutta scheme because of time-integration error. These vortex blobs cannot be deleted for consistency. Therefore, we simply “bounce” these vortex blobs by truncating their displacement vectors and placing them back on the body surface (in practice, a small distance away from the surface). The vortex blobs that have entered the interiors of the bodies at the end of the Runge-Kutta integration are deleted [18]. New vortex sheets are then applied along the body contours to help enforce the no-penetration and no-slip boundary conditions. The new vortex sheets are discretized into vortex blobs, introducing new computational elements into the flow. At the end of the time step, the hydrodynamic pressure, forces, and moments on the bodies are determined using the procedure given in Sec. 2.5. This process is repeated until the end of the computation is reached.

The numerical procedure for free motions is summarized in the flowchart in Fig. 2.6. Overall, the procedure is very similar to that for prescribed motions. There are two major differences: the positions of the bodies are updated using the two-step Adams-Bashforth method. In addition, the body velocities at the end of each time step are also unknown and needed to be solved simultaneously with the strengths of the new vortex sheets as discussed in Sec. 2.6.

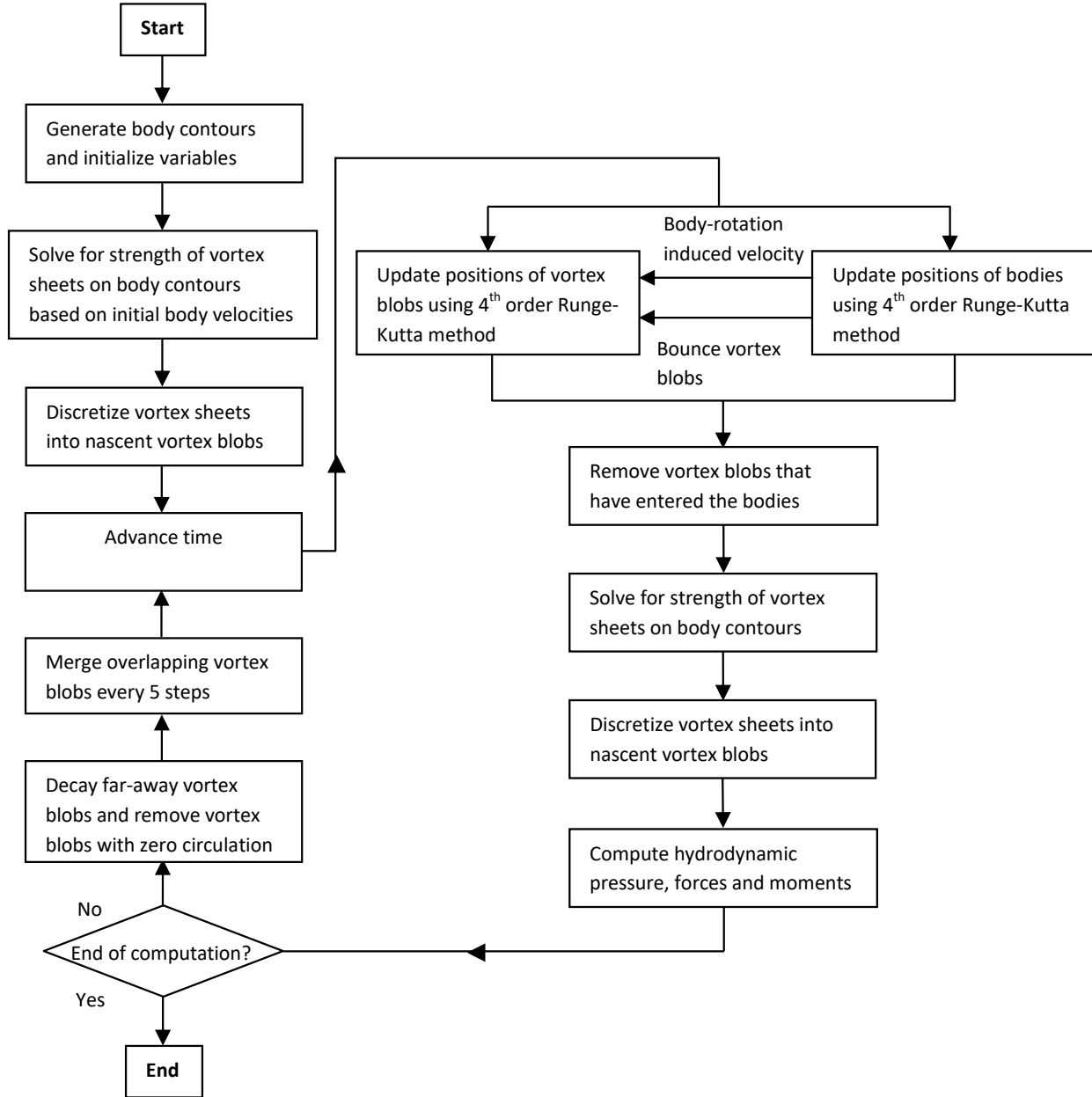


Figure 2.5: Flowchart of the DVM simulation with prescribed body motions.

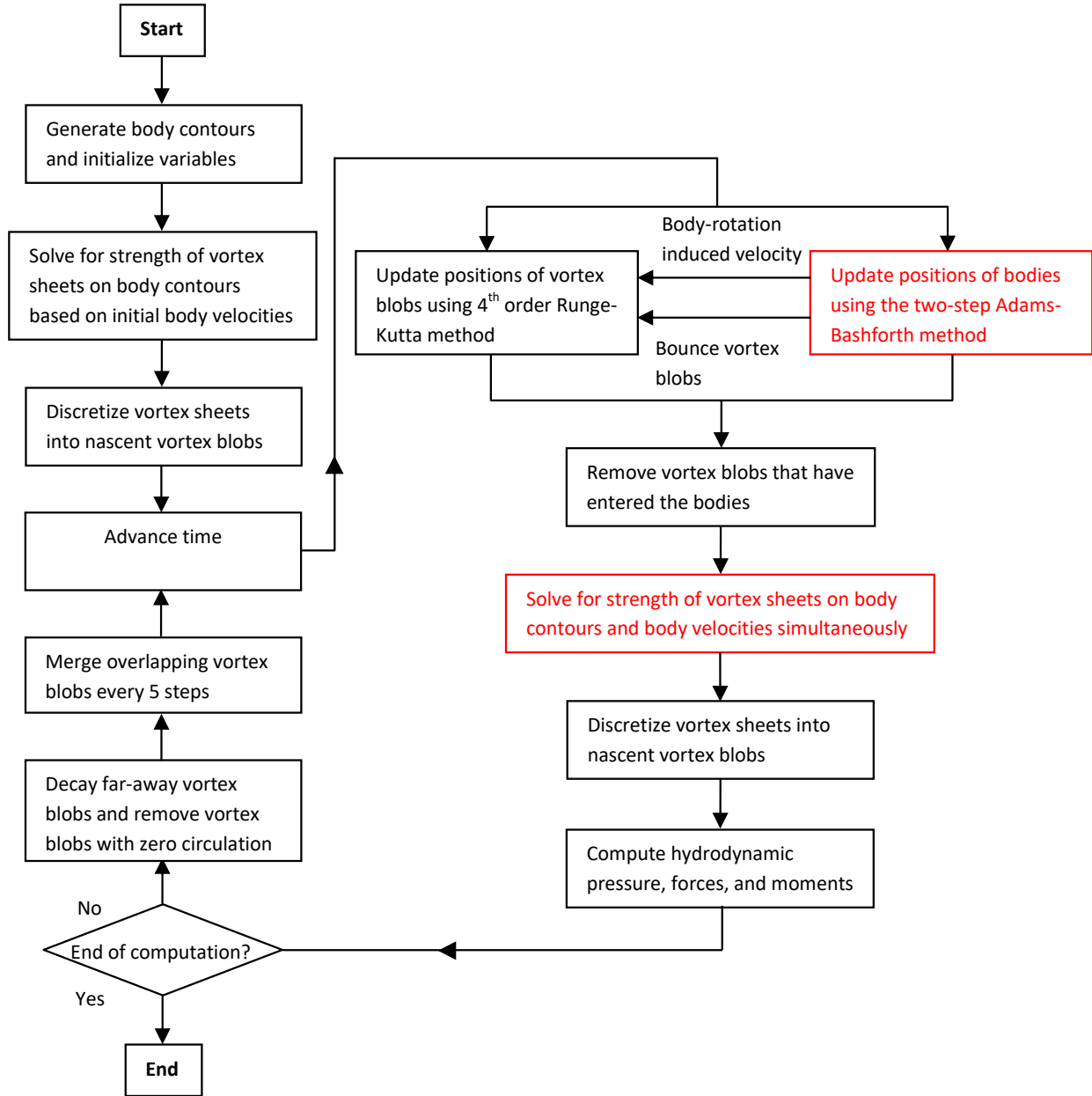


Figure 2.6: Flowchart of the DVM simulation with free body motions. Differences from the procedure for prescribed motions are highlighted in red.

Chapter 3

Validation of the DVM algorithm

Several validation cases are presented in this chapter to test the various capabilities of the DVM algorithm. The cases are selected to demonstrate that the DVM algorithm can be applied to a reasonably wide range of problems without the need of case specific special treatments apart from appropriate choices of body panel size and time step size.

3.1 Flow past a translating cylinder

For the purpose of validating our DVM software, we shall first consider the classical problem of flow around a translating circular cylinder of diameter D . A wealth of numerical and experimental results of this problem exist in the literature which can serve as useful references. At time $t = 0$, the cylinder is at rest. Subsequently, instead of an impulsive start, the cylinder rapidly accelerates in the positive x -direction from rest and reaches a constant speed U_0 at time $t = T_0$ with $U_0 T_0 / D = 0.05$. The speed of the cylinder $U(t)$ in the x -direction is given by the following simple function of time:

$$U(t)/U_0 = \begin{cases} \frac{1}{2} \sin(\pi(t/T_0 - 0.5)) + \frac{1}{2}, & \text{if } 0 \leq t \leq T_0 \\ 1, & \text{if } t > T_0 \end{cases} \quad (3.1)$$

The schematic of the problem is given in Fig. 3.1. The only other adjustable parameter for this problem is the Reynolds number defined as $Re = U_0 D / \nu$. Three cases with $Re = 100$, 1,000, and 9,500 are considered.

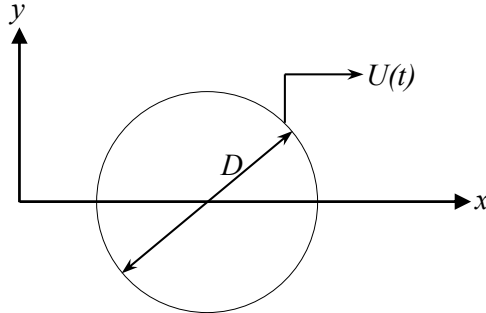


Figure 3.1: A translating circular cylinder.

For $Re = 100$ and 1,000, the cylinder contour consists of 400 equal-sized panels. A constant time step size of $U_0 \Delta t / D = 0.01$ is used. This configuration more than adequately

3.1. Flow past a translating cylinder

resolves the low Reynolds number laminar flow around the cylinder. For the case of $Re = 9,500$, a much higher resolution is needed to resolve the fine vortical structures of the flow. We have used two different configurations for this Reynolds number. The first configuration uses a total of 800 panels to construct the cylinder contour, and the time step is reduced to $U_0\Delta t/D = 0.005$. With this configuration, the simulation is run up to $U_0t/D = 45$. To better resolve the vorticity field right after the initial startup, a second finer configuration with 1,200 panels and the same time step size is used. Because of the significantly increased computational load of the second configuration, the simulation is only run up to $U_0t/D = 5$. For all three Reynolds numbers, the vortex blobs are allowed to decay once they are 24 diameters away from the center of the cylinder.

The hydrodynamic forces on the cylinder are given in terms of the lift and drag coefficients, C_L and C_D . The usual definitions of the force coefficients are used:

$$C_L = \frac{F_y}{(1/2)\rho U_0^2 D}, \quad C_D = -\frac{F_x}{(1/2)\rho U_0^2 D} \quad (3.2)$$

where F_x and F_y are the x - and y -components of the hydrodynamic force on the cylinder.

For the case of $Re = 100$, the flow is simulated up to $U_0t/D = 115$. The time histories of the lift and drag coefficients are shown in Fig. 3.2 and Fig. 3.3. The pressure and friction components of the lift and drag coefficients are also shown separately. At this low Reynolds number, the friction force contributes about 22% of the total drag force. For the lift force, however, the contribution from friction is fairly small compared to the pressure component. From Fig. 3.3, we can see that the final periodic steady state is reached for $U_0t/D > 70$. The time-averaged total drag coefficient, computed from the periodic steady state, is $\overline{C}_D = 1.33$. This value is in excellent agreement with the drag coefficient of $\overline{C}_D = 1.33$ given by the mesh-based Eulerian methods of Park *et al.* [84] and Sharman *et al.* [85]. The oscillating lift force is associated with alternate shedding of vortices in the wake. The shedding frequency is given by the Strouhal number $St = f_s D/U_0$ where f_s is the vortex shedding frequency. The Strouhal number given by the current solution is $St = 0.168$ which is fairly close to the values of $St = 0.165$ and $St = 0.164$ given in [84] and [85], respectively. The lift-coefficient amplitude given by our simulation is $\tilde{C}_L = 0.36$. This value is slightly higher than $\tilde{C}_L = 0.33$ given by Park *et al.* [84] and Sharman *et al.* [85], but lower than $\tilde{C}_L = 0.39$ given by Rossinelli *et al.* [86]. Overall, the lift and drag forces computed using our DVM software are consistent with the values from the literature.

Both lift and drag are global forces. To validate the details of the solution, the pressure coefficient $C_p = p/(1/2\rho U_0^2)$ on the cylinder surface is also compared. Using the final periodic steady-state solution, the time-averaged pressure coefficient $\overline{C}_p$ as a function of angular position on the cylinder surface is computed. The results are shown in Fig. 3.4. Again, good agreement between the current solution and those of [84, 85] is obtained, indicating correct prediction of the boundary-layer separation point by the DVM software. It should be noted that the current DVM formulation can only determine the surface pressure p up to a constant, and the matching shown in Fig. 3.4 is achieved by adding a constant pressure uniformly to our pressure solution.

3.1. Flow past a translating cylinder

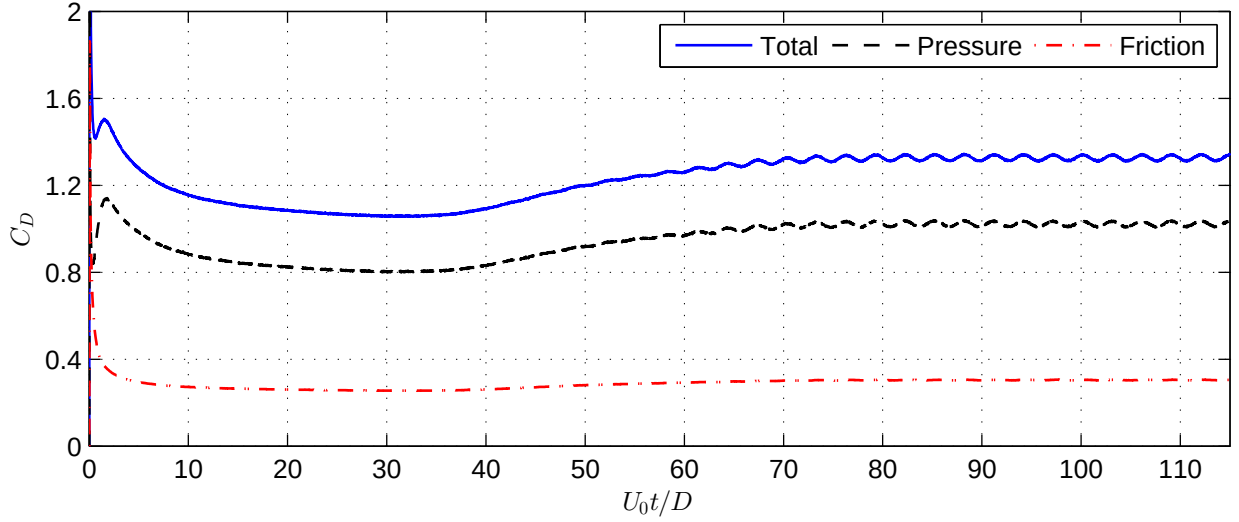


Figure 3.2: Time history of the drag coefficient of a rapidly started circular cylinder at $Re = 100$.

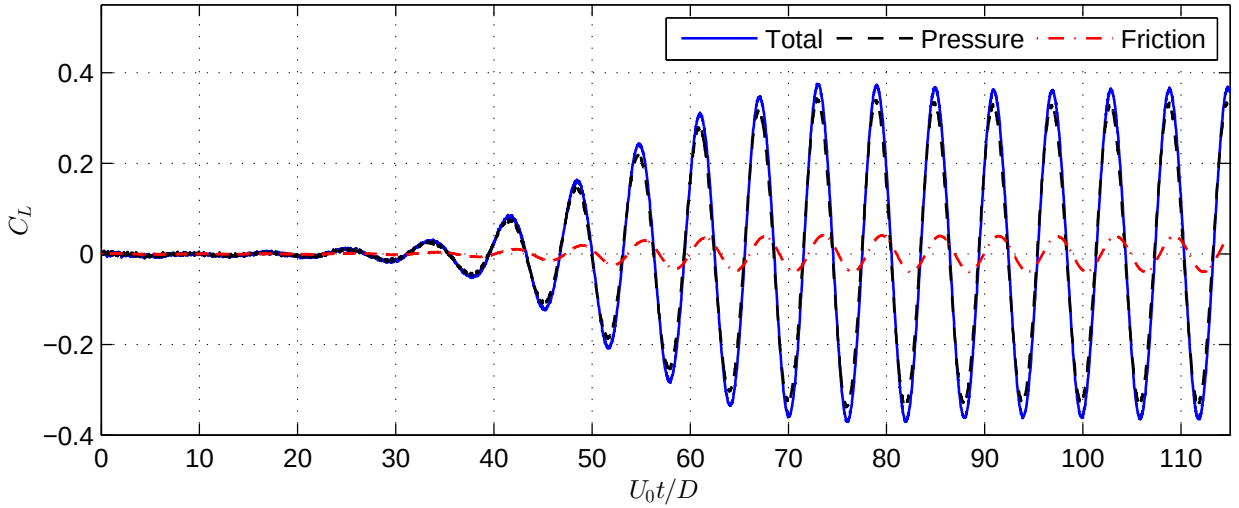


Figure 3.3: Time history of the lift coefficient of a rapidly started circular cylinder at $Re = 100$.

The time histories of the lift and drag coefficients for $Re = 1,000$ are shown in Fig. 3.5. It is well known that at this Reynolds number, the flow near the cylinder becomes three dimensional, and a two-dimensional simulation will slightly over predict the drag and Strouhal number compared to experiments; however, many authors still performed two-dimensional simulations at this Reynolds number to study the important qualitative features of the flow (see *e.g.* [87,88]).

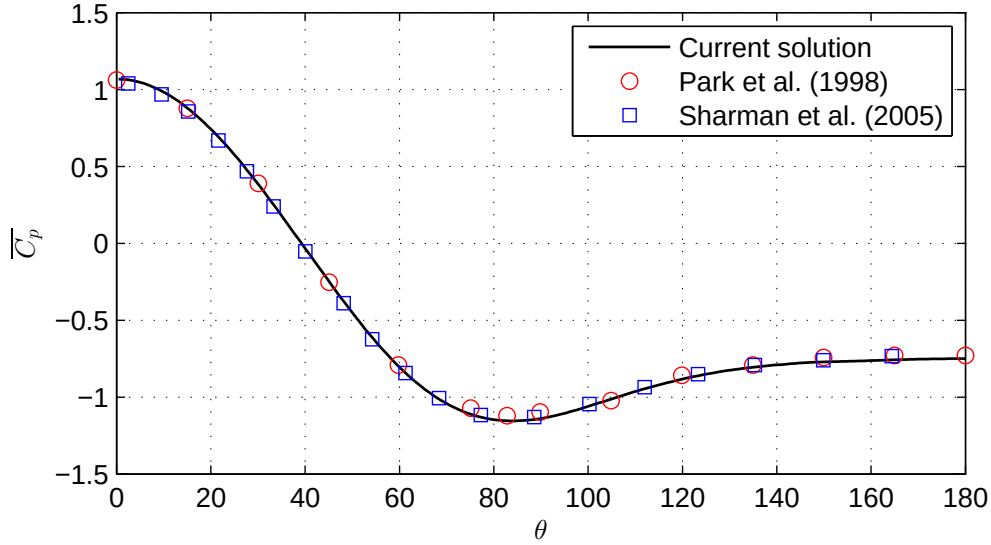


Figure 3.4: Time-averaged pressure coefficient as a function of angular position of a steadily translating circular cylinder at $Re = 100$. The forward stagnation point corresponds to $\theta = 0$.

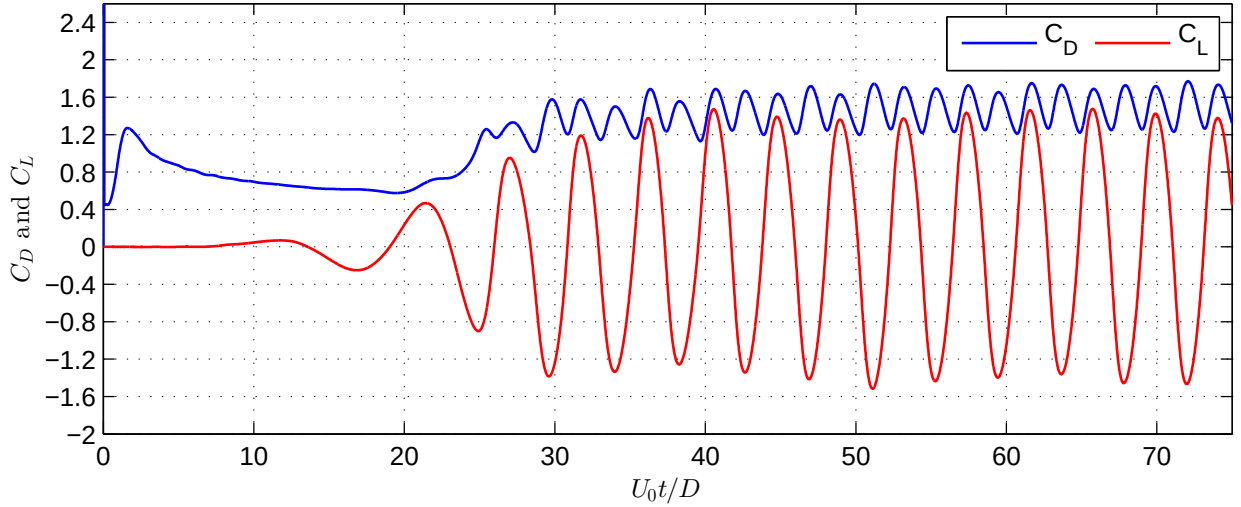


Figure 3.5: Time histories of the lift and drag coefficients of a rapidly started circular cylinder at $Re = 1,000$.

As shown in Fig. 3.5, a periodic steady state is reached after approximately $U_0 t / D = 40$. The period of lift oscillation is twice that of drag. The time-averaged drag coefficient and lift-coefficient amplitude of the periodic steady state are $\overline{C_D} = 1.49$ and $\tilde{C}_L = 1.41$, respectively. The Strouhal number is $St = 0.24$. These results agree well with the two-dimensional finite-element solution of Jester and Kallinderis [88] which gives $\overline{C_D} = 1.51$, $\tilde{C}_L = 1.40$, and

3.1. Flow past a translating cylinder

$St = 0.25$.

The force time histories for $Re = 9,500$ are shown in Fig. 3.6. Of course, three-dimensional effects are very significant in this case, and the flow is becoming turbulent. Nevertheless, several authors have performed two-dimensional simulations at this Reynolds number and obtained useful qualitative results (see *e.g.* [17, 86, 89, 90]).

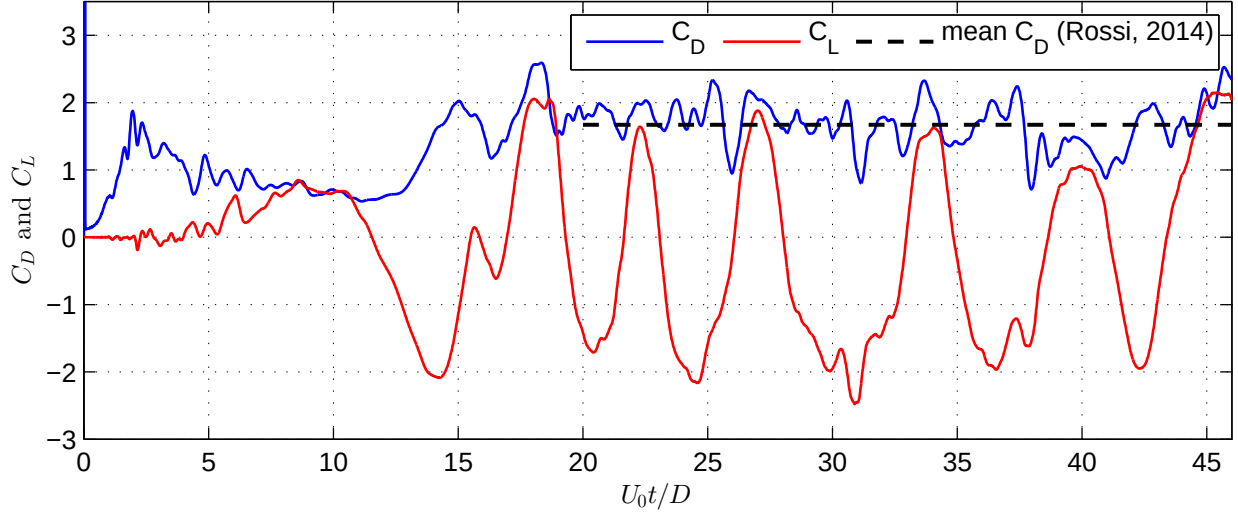


Figure 3.6: Time histories of the lift and drag coefficients of a rapidly started circular cylinder at $Re = 9,500$. Cylinder contour consists of 800 panels. The numerically computed mean drag coefficient from [89] is included for comparison.

The time histories of C_D and C_L shown in Fig. 3.6 are markedly more irregular than those in Fig. 3.5 for $Re = 1,000$. The oscillations in lift and drag become less periodic, and the shedding frequency is not constant at $Re = 9,500$. This observation is qualitatively consistent with the results of Rossi *et al.* [90]. As noted in [90], a pointwise comparison in time is not possible at this Reynolds number. This is especially true for large time solutions. The time history of the flow is extremely sensitive to the various numerical disturbances, and the solution can only be interpreted in a statistical sense. For comparison, we have included, in Fig. 3.6, the mean drag coefficient of a translating circular cylinder at $Re = 9,500$ obtained by Rossi [89] using two-dimensional computation. Both the current DVM solution and the result of [89] give a mean drag coefficient of 1.7.

The non-dimensional vorticity fields ($\omega D/U_0$) at various time instants immediately after the rapid start of the cylinder are shown in Figs. 3.7-3.9. Strong Reynolds number dependence is observed. When $Re = 100$, the flow is dominated by two large and symmetric primary vortices formed from the separation of the thick boundary layer on the cylinder surface. The continuous separation maintains the strength of the primary vortices. The flow features are heavily smoothed out by viscosity. When $Re = 1,000$, the overall vortical structure looks somewhat similar to that of $Re = 100$; however, the vortical regions of the wake

3.1. Flow past a translating cylinder

become more clearly defined because of the reduced viscous effect. The wake of the cylinder when $Re = 9,500$ demonstrates significant differences from that of the two low-Reynolds number cases. Instead of the two large primary vortices, the wake is characterized by small, fragmented vortices with concentrated vorticity. Slight top-and-bottom asymmetry of the wake is observed when $U_0 t/D = 3.5$.

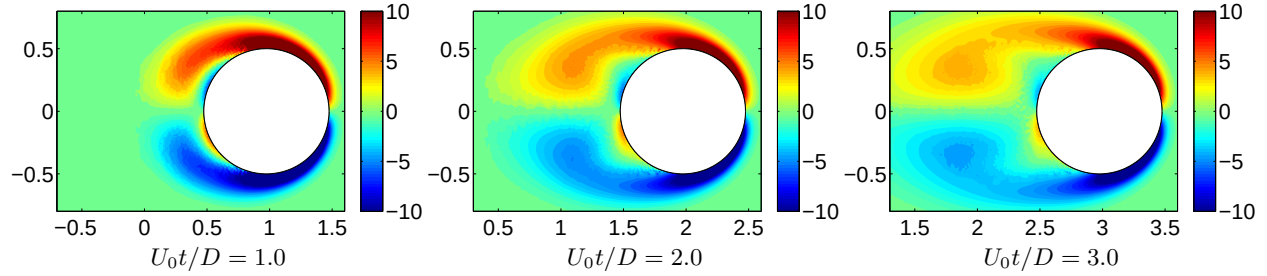


Figure 3.7: Non-dimensional vorticity fields ($\omega D/U_0$) at various time instants after the rapid start of the cylinder. $Re = 100$.

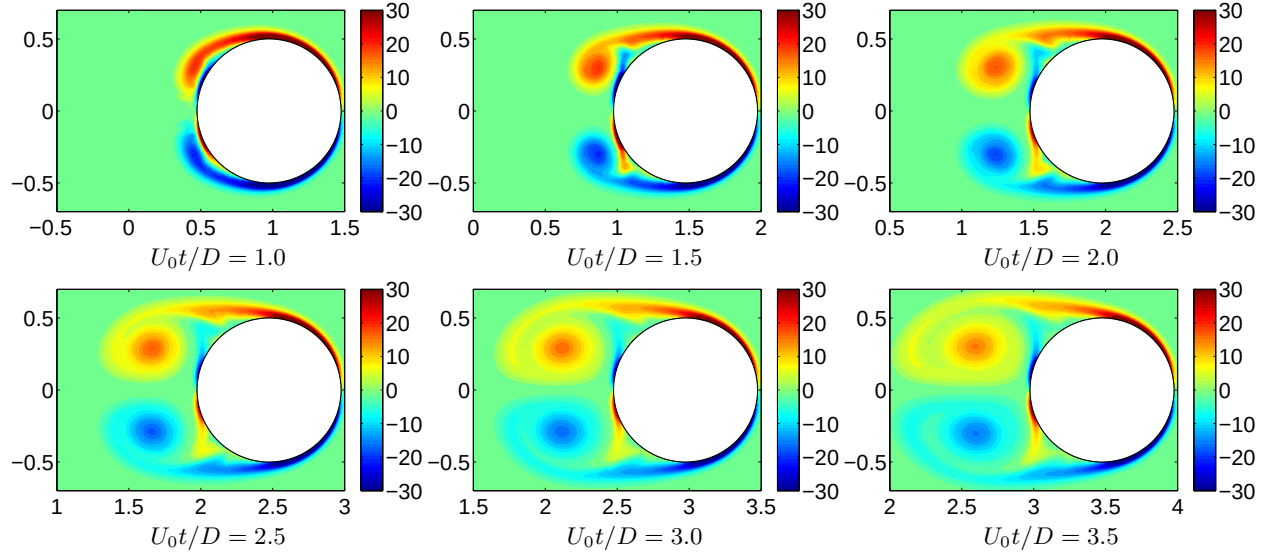


Figure 3.8: Non-dimensional vorticity fields ($\omega D/U_0$) at various time instants after the rapid start of the cylinder. $Re = 1,000$.

The wakes of the cylinder at longer times are shown in Fig. 3.10 for all three Reynolds numbers. A regular von Kármán vortex street is observed for both $Re = 100$ and $Re = 1,000$. Compared to the case of $Re = 100$, significantly less viscous diffusion is observed for $Re = 1,000$, and the distance between consecutive vortices in the vortex street is reduced, consistent with the increased Strouhal number. When $Re = 9,500$, the regular vortex street pattern disappears, and strong vortex pairs with concentrated vorticity are being shed by the cylinder. The lack of a regular wake structure is related to the irregular shedding frequency.

3.2. Flow past two translating cylinders in tandem

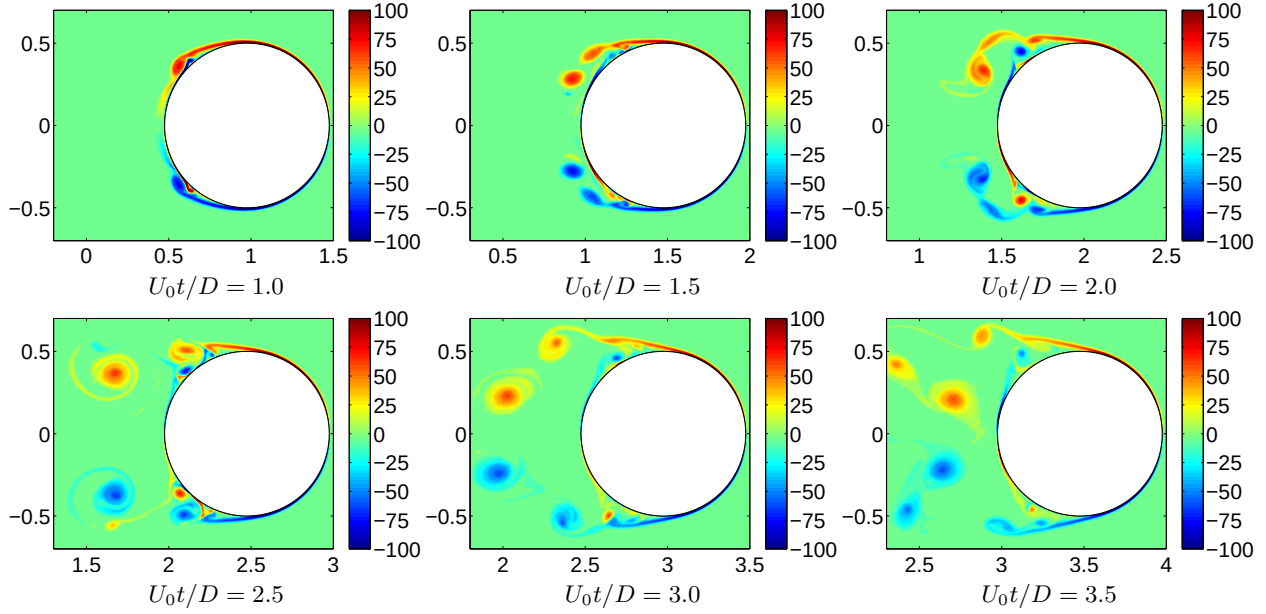


Figure 3.9: Non-dimensional vorticity fields ($\omega D/U_0$) at various time instants after the rapid start of the cylinder. $Re = 9,500$. Cylinder contour consists of 1,200 panels.

3.2 Flow past two translating cylinders in tandem

In this section, the flow past two translating cylinders in tandem is simulated using DVM in order to validate the software's ability to model multiple bodies. As shown in Fig. 3.11, the two identical cylinders of diameter D , both centered on the x -axis, are separated by a distance s (center-to-center) in the x -direction. For the purpose of validation, only the case with $s/D = 2$ and $Re = U_0 D/\nu = 100$ is considered. Same as in Sec. 3.1, both cylinders are at rest at $t = 0$ and rapidly accelerate to a constant speed U_0 in the positive x -direction. The speed of the two cylinders as a function of time is, again, given by Eq. (3.1). Each cylinder is constructed from 400 equal-sized panels, and a time step of $U_0 \Delta t/D = 0.01$ is used.

The vorticity field of the flow past the two tandem cylinders at $U_0 t/D = 175$ is shown in Fig. 3.12. Because of the close proximity, the separated boundary layer of the front cylinder reattaches on the sides of the rear cylinder, and the two cylinders behave like a single and slightly more streamlined body. The time-dependent drag and lift coefficients, defined in Eq. (3.2), of the two cylinders are shown in Fig. 3.13. At steady state, we have $\overline{C}_D = 1.14$ and $\overline{C}_D = -0.07$ for the front and rear cylinders, respectively. This result is in reasonably good agreement with that of Sharman *et al.* [85] which gives $\overline{C}_D = 1.16$ for the front cylinder and $\overline{C}_D = -0.08$ for the rear one. Compared to the case of a single cylinder, the drag on the front cylinder is slightly reduced by the presence of the rear one. Also, because of the low pressure in the region between the two cylinders, the rear cylinder is being pulled forward by the flow, resulting in a slightly negative drag.

Finally, the time-averaged pressure distributions on the front and rear cylinders at steady

3.2. Flow past two translating cylinders in tandem

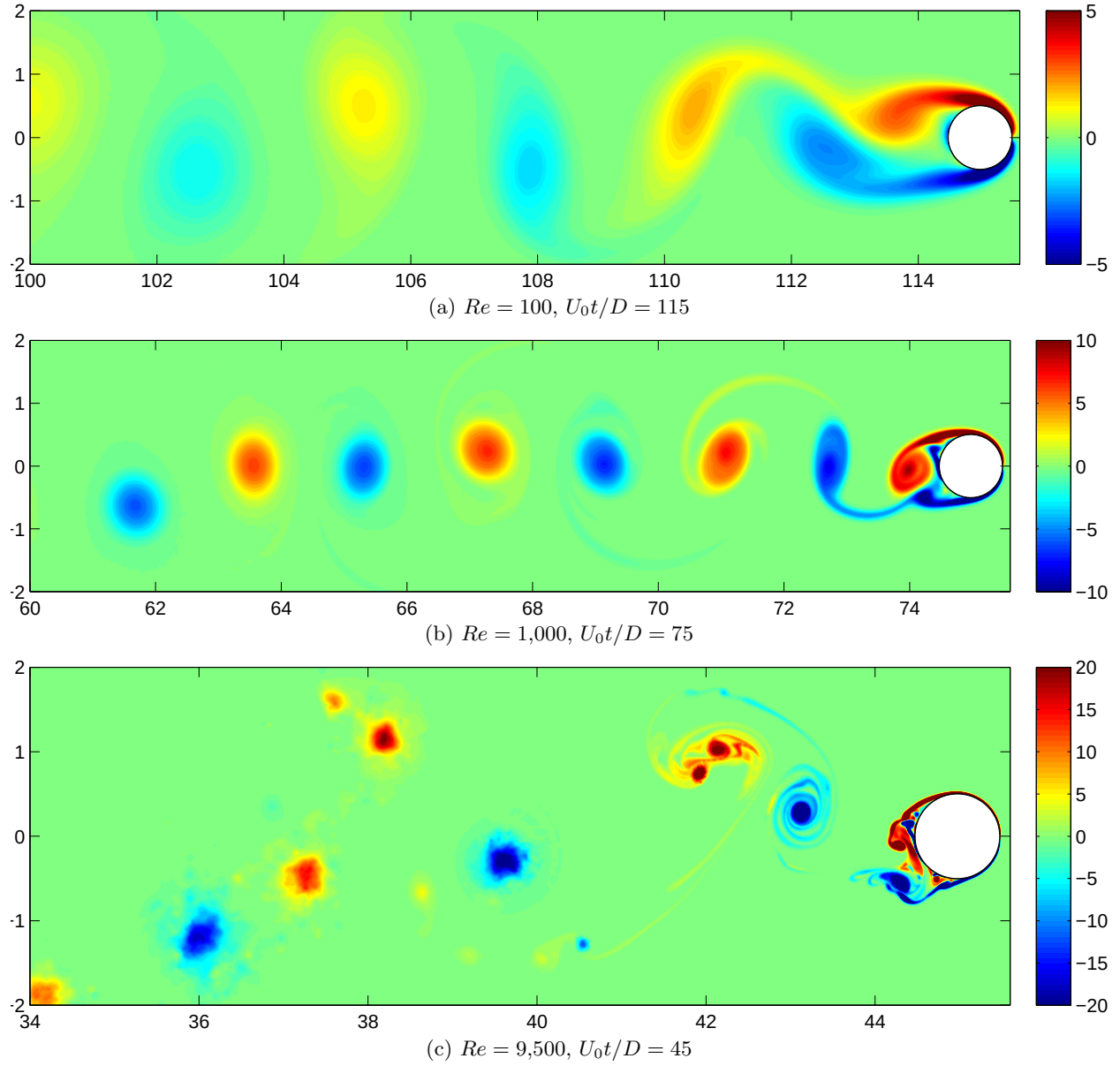


Figure 3.10: Non-dimensional vorticity fields ($\omega D/U_0$) of the wake of a translating cylinder at (a) $Re = 100$, (b) $Re = 1,000$, and (c) $Re = 9,500$. For $Re = 9,500$, 800 panels are used.

state are shown in Fig. 3.14. Reasonably good agreement with the results from [85] is observed. For the rear cylinder, the slight discrepancy between the pressure distribution is partially caused by the inaccuracies in the reference results extracted from [85]. The original figure of [85] is scaled in such a way that accurate digitization is difficult.

3.2. Flow past two translating cylinders in tandem

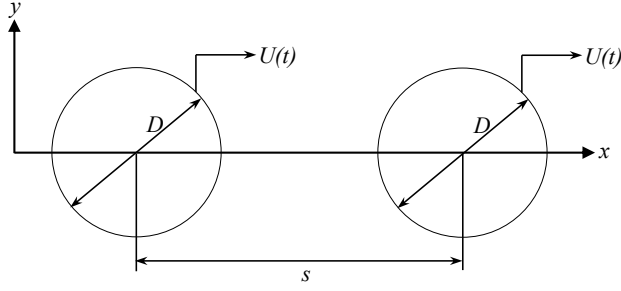


Figure 3.11: Two circular cylinders translating in tandem.

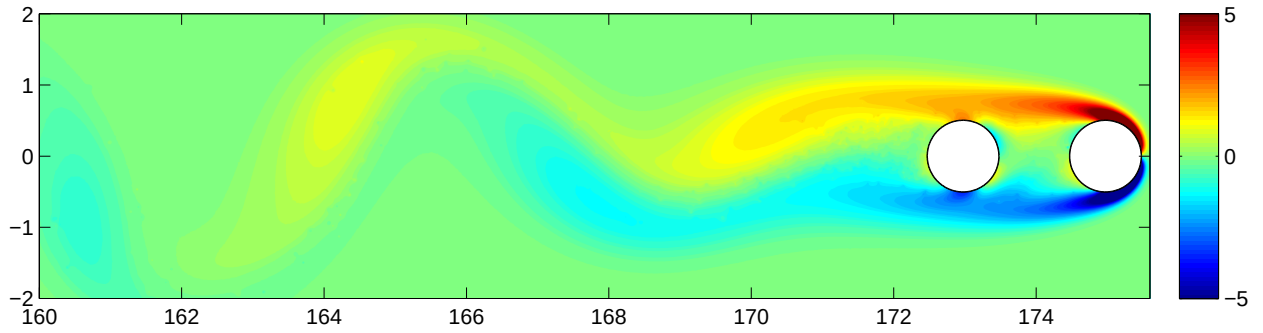


Figure 3.12: Non-dimensional vorticity field ($\omega D/U_0$) of the wake of two translating cylinders in tandem at $Re = 100$ and $U_0 t/D = 175$. Center-to-center separation between the two cylinders is $s/D = 2$.

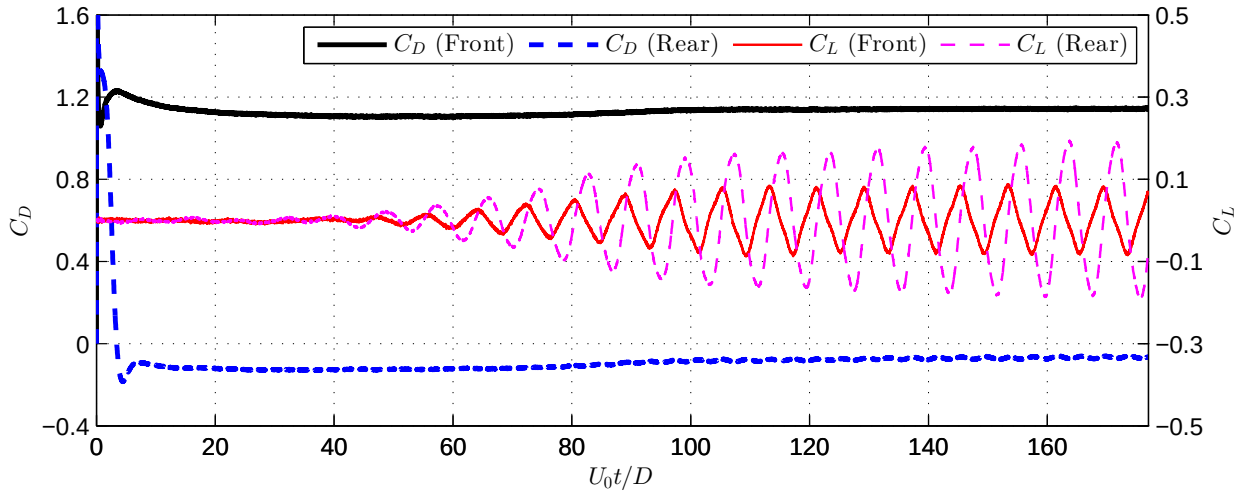


Figure 3.13: Time histories of the lift and drag coefficients of two rapidly started circular cylinders in tandem at $Re = 100$. Center-to-center separation between the two cylinders is $s/D = 2$.

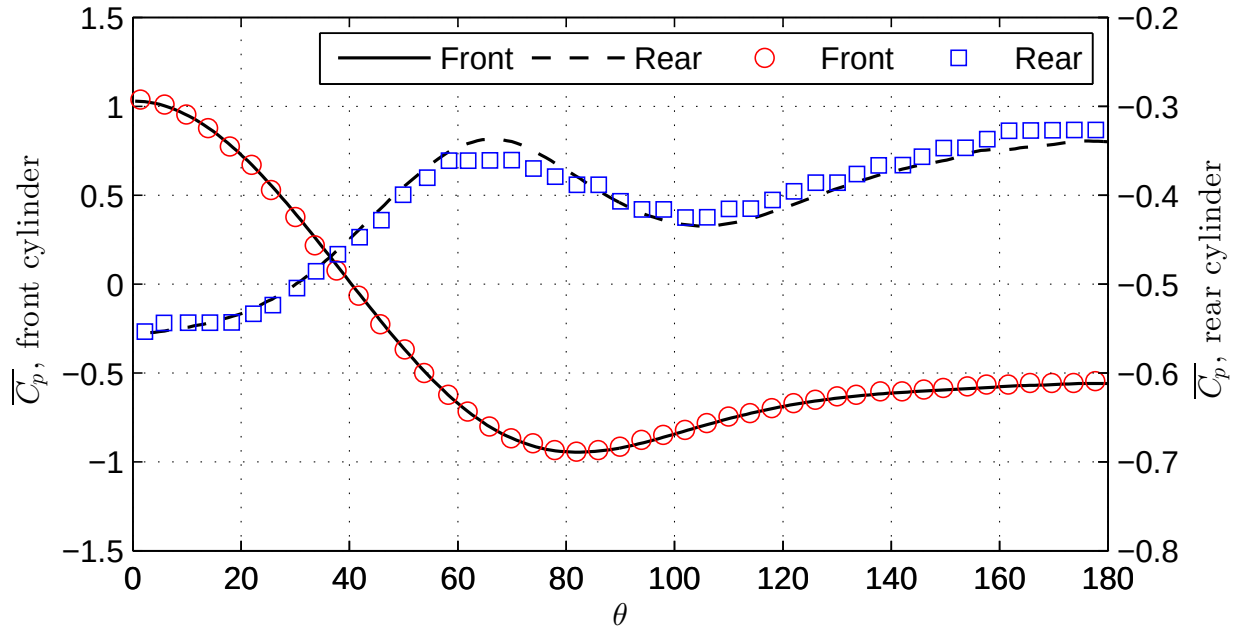


Figure 3.14: Time-averaged pressure coefficients as functions of angular position of two steadily translating circular cylinders in tandem at $Re = 100$. Center-to-center separation between the two cylinders is $s/D = 2$. Curves are current DVM results, and symbols are results from Sharman *et al.* [85].

3.3 Flow past a translating and rotating ellipse

Another interesting problem involves a rotating ellipse submerged in an uniform free stream. Several authors have used this problem to validate their numerical method. For example, Maruoka [91] and Zhang *et al.* [92] have solved this problem using a Chimera mesh. More recently, Zhang and Liang used the rotating ellipse problem to validate their high-order spectral-difference method with sliding-mesh interface [93]. The sliding-mesh interface is used to couple the rotating inner domain containing the rotating ellipse and the fixed outer domain. The results of [91–93] show very good agreement with each other.

Here, we solve the equivalent problem of an ellipse of major-axis length D rotating and translating at a constant speed in a calm fluid shown in Fig. 3.15. The fluid is initially at rest. At $t = 0$, the ellipse, with its major axis aligned with the x -axis, undergoes an impulsive start and translates at a constant speed U_0 in the positive x -direction and rotates at a constant angular velocity Ω_0 when $t > 0$. The ellipse rotates in the clockwise direction, and, by convention, we have $\Omega_0 < 0$. The time for one revolution is given by $T = -2\pi/\Omega_0$. In order to compare with [93], the same Reynolds number $Re = U_0 D/\nu = 200$ and tip-speed ratio $\lambda = -(\Omega_0 D/2U_0) = \pi/4$ are used. The thickness ratio of the ellipse is 0.5. A total of 400 panels are used to construct the ellipse. The nodes are equally spaced in terms of their angular positions about the center of the ellipse. This leads to smaller panels being used

3.3. Flow past a translating and rotating ellipse

near the two ends of the ellipse with high curvature. The time step is again $U_0 \Delta t / D = 0.01$.

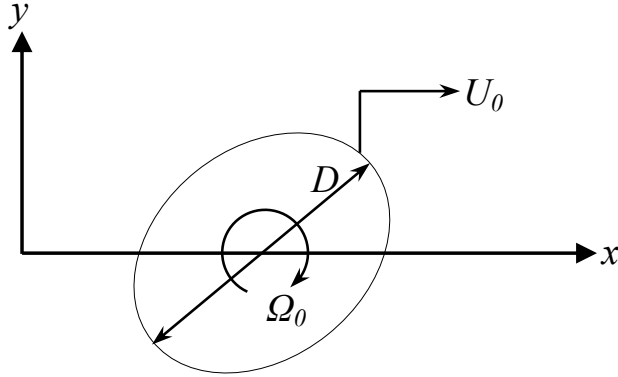


Figure 3.15: A translating and rotating elliptic cylinder.

The time-dependent lift and drag coefficients, defined in Eq. (3.2), after the impulsive start are shown in Fig. 3.16(a). The forces become approximately periodic after only one revolution. In Fig. 3.16(b), the time histories of C_D and C_L during the tenth revolution are shown. Our current DVM solutions are in very good agreement with those of [93].

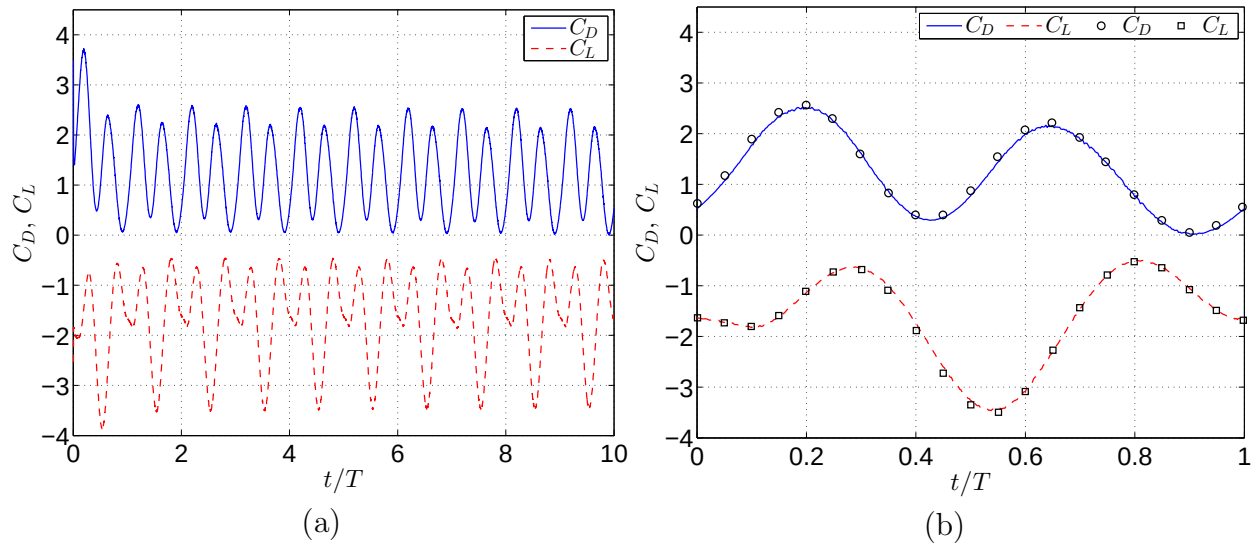


Figure 3.16: Time histories of the lift and drag coefficients of a translating and rotating elliptic cylinder. (a) Time histories from impulsive start to the end of the tenth rotation cycle. (b) The variations of drag and lift coefficients over the tenth period. Curves are current solutions, and symbols are results of Zhang & Liang [93]. $Re = 200$ and $\lambda = \pi/4$.

The vorticity fields near the ellipse at various time instants are shown in Fig. 3.17. The flows near the ellipse during the first and second half of a revolution are not identical. A

highly regular vortical pattern is observed in the wake. Also, the wake vortices move upwards slowly with time (positive y -direction), indicating a weak upwards current. This is, of course, expected because the rotating ellipse is experiencing a negative lift force at all times.

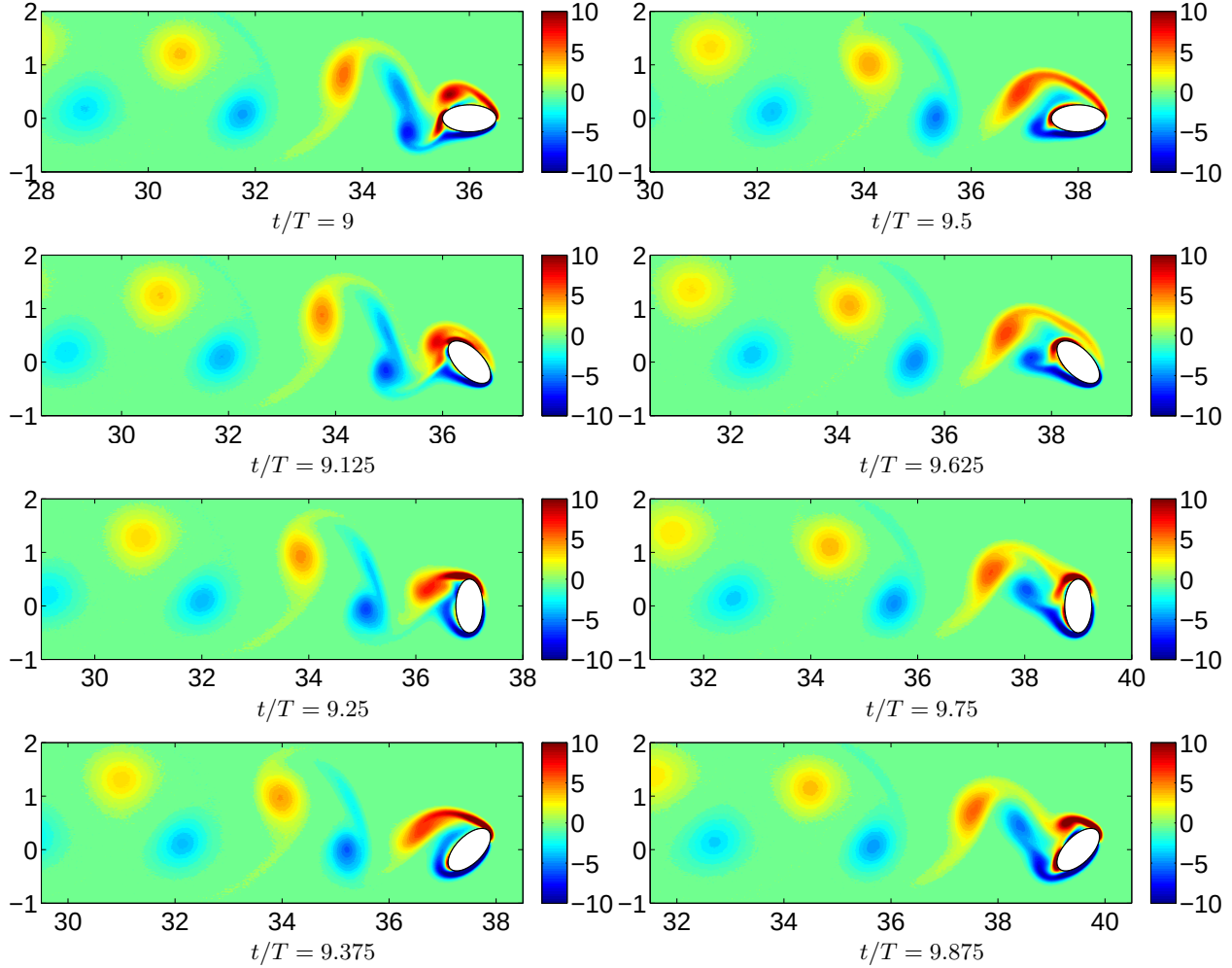


Figure 3.17: Non-dimensional vorticity fields ($\omega D/U_0$) during the tenth revolution of the translating and rotating ellipse. $Re = 200$ and $\lambda = \pi/4$.

3.4 Flow near a flapping foil in normal hovering mode

To further validate our DVM software, bodies performing unsteady translational and rotational motions should be considered. We shall first compute the forces on a single foil with an elliptic cross section performing a flapping motion. This problem is closely related to the lift-generating ability of insects and avian species.

3.4. Flow near a flapping foil in normal hovering mode

The flapping of the elliptic foil is a combination of translational and rotational motions. During one period T , the translational motion of the foil can be separated into two parts: upstroke and downstroke (see Fig. 3.18). The switch in the direction of translation between upstrokes and downstrokes is referred to as stroke reversals. The foil also undergoes two rotational motions, supination and pronation, about the mid-chord point which allow the foil to re-orient in conjunction with stroke reversals. In the simplest form, the motions are described by two sinusoidal functions in time:

$$X(t) = A \cos(2\pi ft), \quad Y(t) = 0, \quad (3.3)$$

$$\alpha(t) = \alpha_0 + \beta \sin(2\pi ft + \phi) \quad (3.4)$$

where X and Y are the coordinates of the mid-chord point of the foil. The angle between the chord and the x -axis is given by α . The mean angle of the airfoil is given by α_0 . The translational and rotational amplitudes are A and β , respectively. The frequency of the motion is $f = 1/T$, and ϕ is the phase shift between rotation and translation.

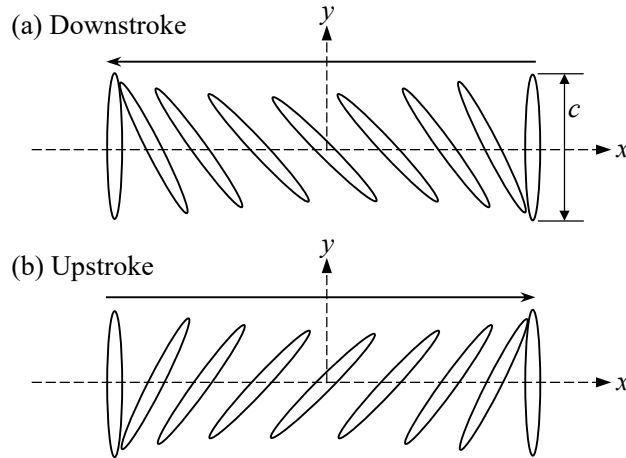


Figure 3.18: An elliptic foil flapping in normal hovering mode. The x -axis coincides with the stroke plane. (a) Downstroke. (b) Upstroke. The foil orientation shown corresponds to symmetric rotation with $\phi = 0$.

The flapping motion of the hovering foil generally falls into one of three different modes. The “water treading mode” is associated with $\alpha_0 = 0$; the mean orientation of the foil is horizontal [94]. The “normal hovering mode” is associated with $\alpha_0 = \pi/2$, and the foil rotates about the upright position [94]. Lastly, we have the “dragonfly mode” that uses an oblique mean pitch angle α_0 [32]. The value of ϕ determines the timing of the rotation. We have advanced, symmetric, and delayed rotations for $\phi > 0$, $\phi = 0$, and $\phi < 0$, respectively.

In order to compare with the results of Eldredge [25], we use the same parameters: $A = 1.4c$, $\alpha_0 = \pi/2$ (normal hovering mode), and $\beta = \pi/4$ where c is the chord of the foil. The Reynolds number based on maximum translational velocity $U_{max} = 2\pi f A$ and chord c

3.5. Clap-and-fling motion

is 75. The elliptic foil has an aspect ratio of 10. Both advanced and symmetric rotations are simulated with $\phi = \pi/4$ and $\phi = 0$, respectively. For both cases, the elliptic foil consists of 200 panels, and the time step is set to $\Delta t/T = 2.5 \times 10^{-3}$. To account for the unsteady foil translation, the lift and drag coefficients are defined as:

$$C_L = \frac{F_y}{(1/2)\rho U_{max}^2 c}, \quad C_D = -\frac{F_x(U/|U|)}{(1/2)\rho U_{max}^2 c} \quad (3.5)$$

where F_x and F_y are the x - and y -components of the aerodynamic force on the airfoil, respectively, and $U = \dot{X}$ is the instantaneous translational velocity of the foil.

The time histories of the lift and drag coefficients during the first four periods after the impulsive start are shown in Figs. 3.19 and 3.20. The results from [25] are also included as a reference. The simulated lift coefficients are in good agreement with each other. Note that the original lift coefficients from Eldredge [25] are normalized by the maximum quasi-steady lift coefficient. These results have been re-normalized using the definition of C_L used here to facilitate the comparison.

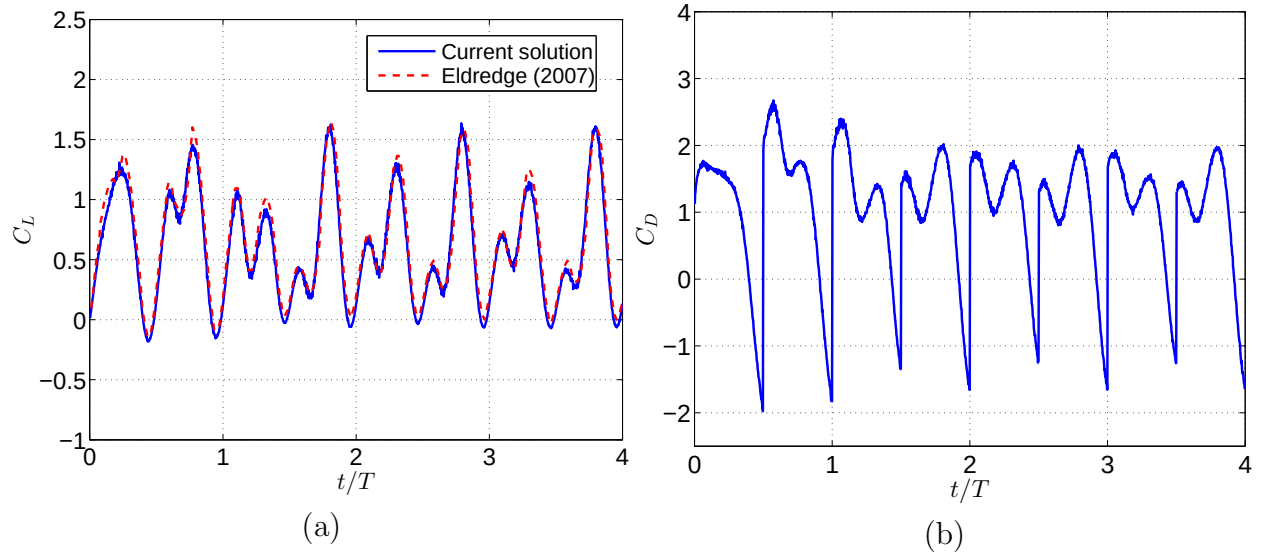


Figure 3.19: Time histories of the hydrodynamic force coefficients on the flapping foil in hovering. (a) Lift coefficient. (b) Drag coefficient. $Re = 75$, $\phi = 0$.

The non-dimensional vorticity fields at various time instants during the fourth period are shown in Figs. 3.21 and 3.22 for advanced $\phi = \pi/4$ and symmetric $\phi = 0$ rotations.

3.5 Clap-and-fling motion

Another problem closely related to insect flight is the clap-and-fling motion described in [28]. We simulated a single clap-and-fling cycle as illustrated in Fig. 3.23. At $t = 0$, two elliptic

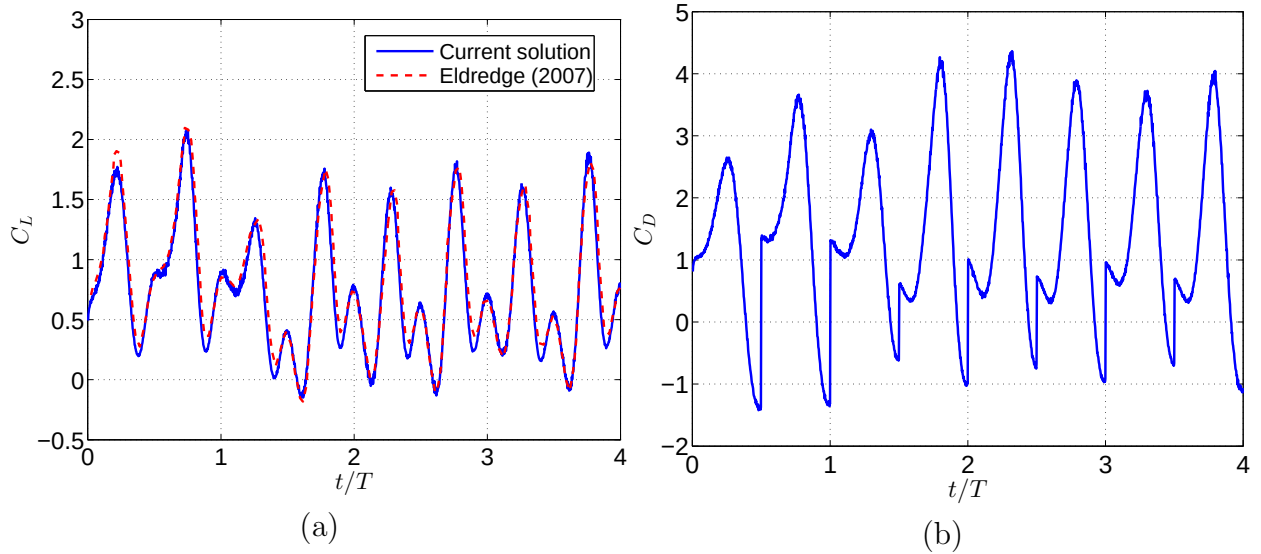


Figure 3.20: Time histories of the hydrodynamic force coefficients on the flapping foil in hovering. (a) Lift coefficient. (b) Drag coefficient. $Re = 75$, $\phi = \pi/4$.

foils of chord c and aspect ratio 10 are in the upright position with a small separation d in between. During the fling phase, the two foils rotate to 45° angle of attack (AOA) about their respective trailing edges (TE) and translate away from each other. Towards the end of the fling phase, the foils rotate back to the upright position about their TE while the translational velocity decreases to zero. During the clap phase, the foils rotate about their leading edges (LE) to 45° AOA and translate towards each other. At the end of the clap phase, the foils rotate back to the upright position about their LE and stop with distance d between the two foils.

A detailed description of the foil kinematics during the clap-and-fling cycle can be found in [28] and will not be repeated here for the sake of brevity. For our validation case, we use a Reynolds number of 128 based on c and the maximum translational velocity U_{max} . The initial separation between the foils is $d/c = 0.1$. The non-dimensional period of the clap-and-fling cycle is $TU_{max}/c = 15.48$. The velocity of the right foil is shown in Fig. 3.24 as a function of time. The motion of the left foil is a mirror image of that of the right. Because of the symmetry, the lift forces on the two foils are identical. The time history of the lift coefficient of the right foil, again following the definition in Eq. (3.5), is shown in Fig. 3.25. Good agreement is observed between the current DVM solution and the IB-LBM solution from [28]. In our simulation, each foil consists of 100 panels. The time step size is set to $\Delta t U_{max}/c = 0.01$. At the end of the clap-and-fling cycle, approximately 2.4×10^4 computational elements (vortex blobs) are used.

The instantaneous vorticity fields near the foil at several time instants are shown in Figs. 3.26 and 3.27. The vorticity fields from [28] are also included for comparison. It should be noted that the color scale and spatial scales of the vorticity field plots from [28] are not

3.5. Clap-and-fling motion

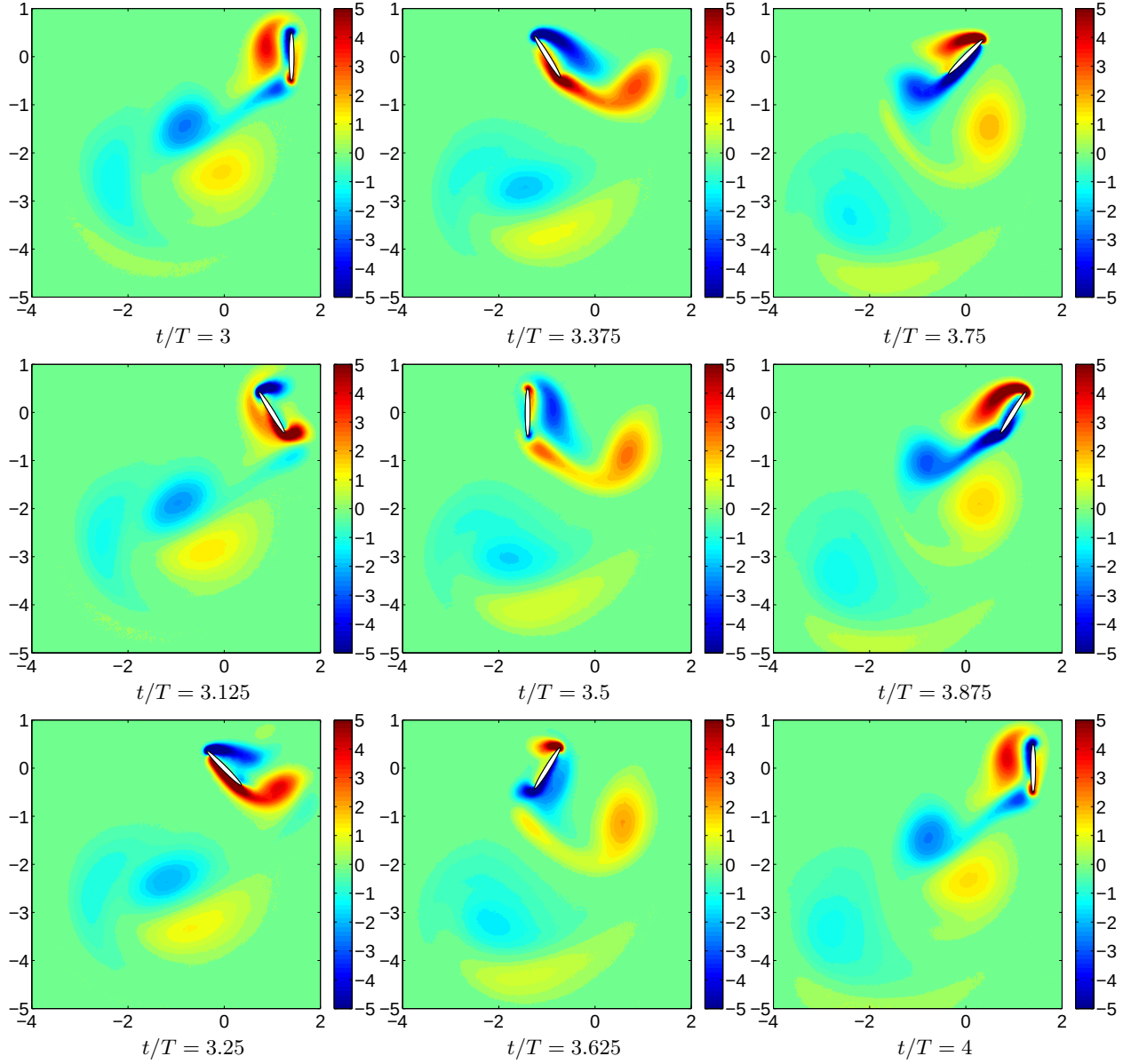


Figure 3.21: Non-dimensional vorticity fields ($\omega c/U_{max}$) during the fourth flapping cycle. $Re = 75$ and $\phi = 0$.

provided. Therefore, it is not possible to perfectly match the colormap and the geometric proportions of the plots. Nevertheless, good qualitative agreement is observed between the current solutions and those of [28].

3.6. Two-dimensional Vortex-Induced Vibration of an elastically mounted circular cylinder

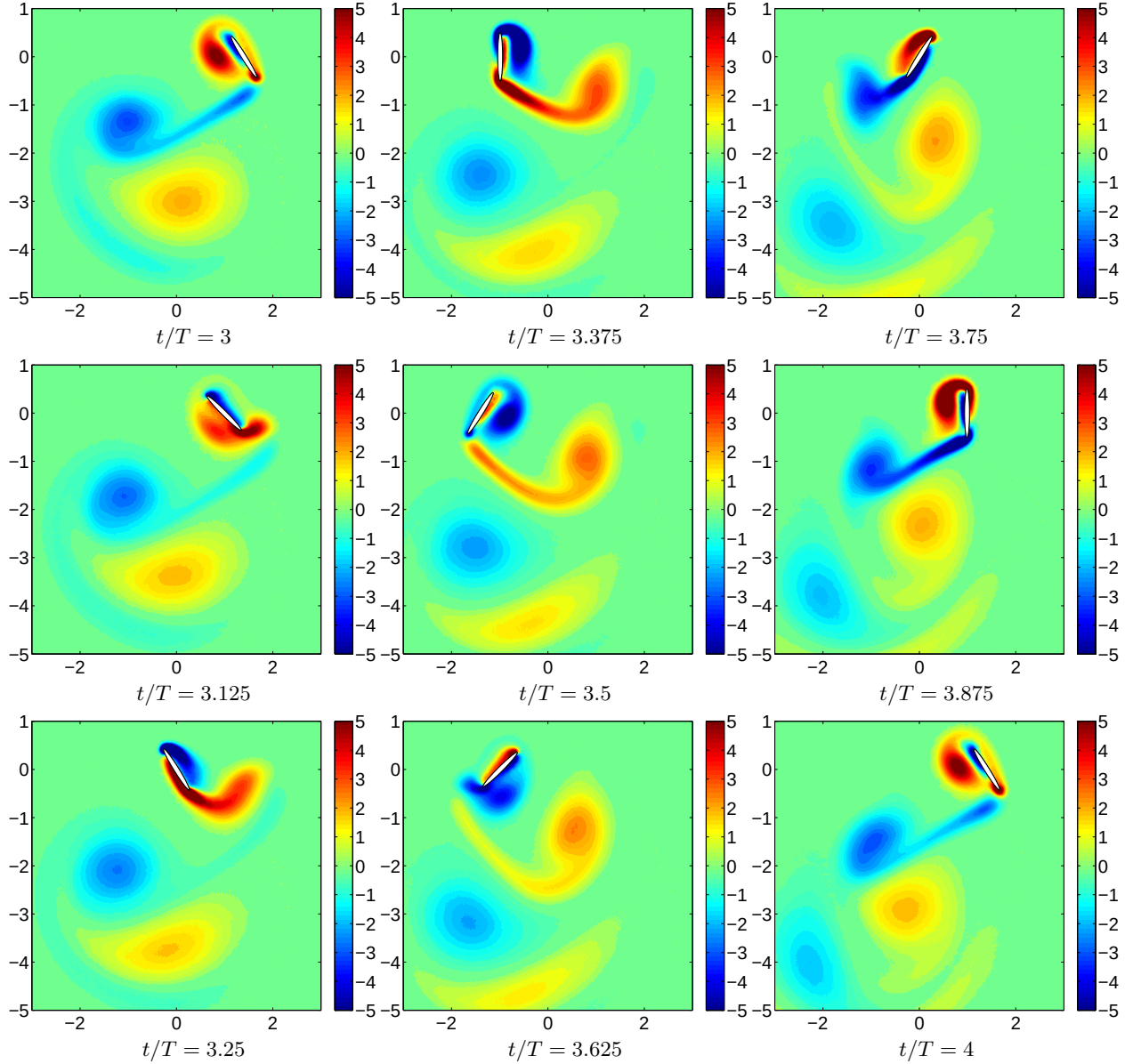


Figure 3.22: Non-dimensional vorticity fields ($\omega c/U_{max}$) during the fourth flapping cycle. $Re = 75$ and $\phi = \pi/4$.

3.6 Two-dimensional Vortex-Induced Vibration of an elastically mounted circular cylinder

In the last validation study, the ability of the software to simulate free motion is validated by considering the Vortex-Induced Vibration (VIV) of an elastically mounted circular cylinder. Initially at $t = 0$, the cylinder, centered at $x_c = 0$ and $y_c = 0$, is at rest. For $t > 0$, the

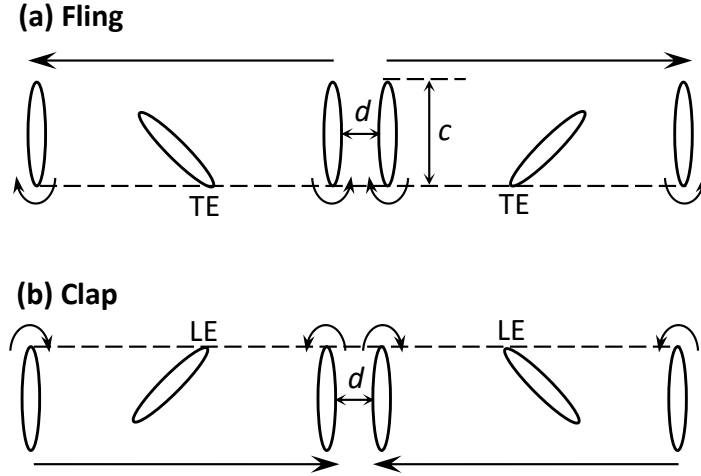


Figure 3.23: Two elliptic foils performing the clap-and-fling motion. (a) Fling. (b) Clap.

x -component of the velocity of the cylinder is, again, prescribed by Eq. (3.1). When $t \geq T_0$ where $U_0 T_0 / D = 0.05$, the x -component of the velocity is constant and equal to U_0 . The cylinder is free to oscillate in the y -direction (one degree of freedom) under the influence of the flow. The cylinder has mass m (per unit distance into the paper) and diameter D . The y -position of the center of the cylinder, y_c , is, therefore, determined from the following equation of motion:

$$m \frac{d^2 y_c}{dt^2} + c \frac{dy_c}{dt} + k y_c = F_y \quad (3.6)$$

where k is the restoring spring stiffness and c is the external damping coefficient. A schematic of the problem is illustrated in Fig. 3.28. The parameters used are Reynolds number $Re = U_0 D / \nu = 150$, reduced mass $M_{red} = m / (\rho D^2) = 2$, and damping ratio $\zeta = c / \sqrt{4km} = 0$ (no external damping). The DVM software is used to predict the motion history of the cylinder and the maximum non-dimensional excursion $(y_c / D)_{max}$ as a function of the reduced velocity $U_{red} = U / (f_n D)$ where the natural frequency f_n is given by $2\pi f_n = \sqrt{k/m}$.

Apart from the initial acceleration during $0 \leq t \leq T_0$ given by Eq. (3.1), the problem described above is equivalent to the more common VIV setup where a cylinder is fixed in the x -direction and free to oscillate in the y -direction under the influence of an uniform current U_0 in the negative x -direction.

Six different cases are simulated with $U_{red} \in [3, 8]$. The maximum excursion in the y -direction is shown as a function of U_{red} in Fig. 3.29. The numerical results of Ahn & Kallinderis based on a finite-volume Arbitrary Lagrangian-Eulerian (ALE) scheme [95] and the immersed-boundary-method solutions of Borazjani & Sotiropoulos [96] are used for comparison. The current numerical solution agrees well with those of [95, 96]. The large body

3.6. Two-dimensional Vortex-Induced Vibration of an elastically mounted circular cylinder

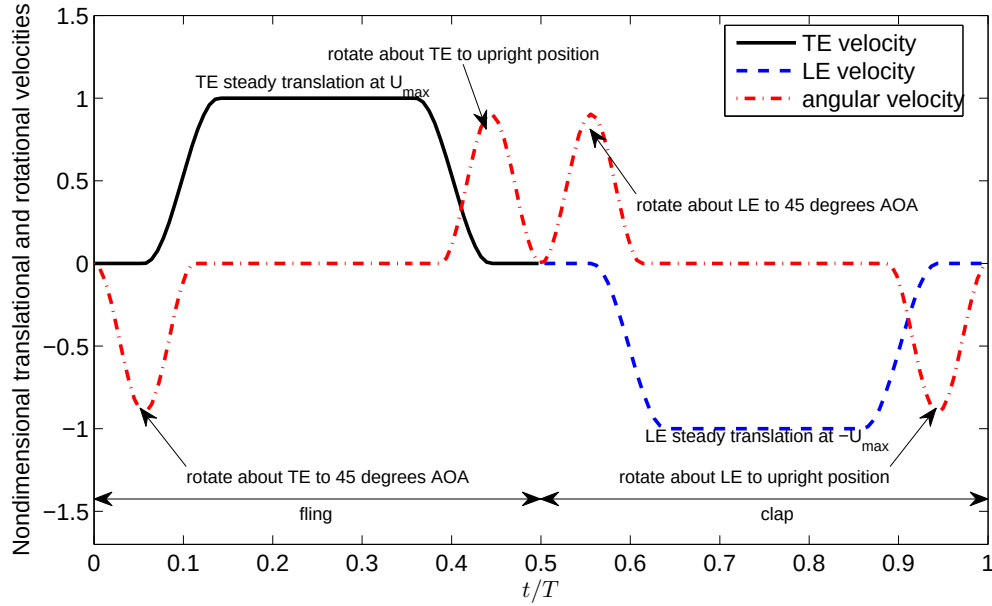


Figure 3.24: The non-dimensional translational and rotational velocities of the right foil in clap-and-fling motion. The translational velocity and rotational velocity are normalized by U_{max} and U_{max}/c , respectively. During the fling phase, the foil pivots about the trailing edge (TE), and the translational velocity of the trailing edge is given. Similarly, during the clap phase, the foil rotates about the leading edge (LE), and the translation velocity of the leading edge is shown instead. The foil motion shown here is identical to the case with rotational overlap $\xi = 50\%$ in [28].

excursion of the locked-in region ($4 \leq U_{red} \leq 7$) is successfully captured by the DVM simulation.

The phase relation between the lift coefficient C_L , defined in Eq. (3.2), and the y -position of the cylinder y_c is illustrated in Fig. 3.30 for all six values of U_{red} . Consistent with the observation made by [95], the lift force and the oscillation of the cylinder are in-phase for lower reduced velocity $U_{red} \leq 5$ (more rigid structure for a given cylinder mass). As U_{red} increases (less rigid structure), the lift force and the cylinder oscillation gradually become out of phase (see $U_{red} = 8$).

The frequency of the VIV normalized by the natural frequency is shown as a function of U_{red} in Fig. 3.31. The numerically determined VIV frequencies of a slightly difference case with $Re = 150$, $M_{red} = 1.57$, and $\zeta = 0.007$ from [97] are included as a reference. The range of U_{red} in which the vortex shedding frequency locks into the natural frequency of the structure ($f/f_n \approx 1$) is successfully predicted by the current DVM simulation. Overall, the normalized frequencies given by the current solution agree well with that of [97] despite the slightly different parameters used.

3.6. Two-dimensional Vortex-Induced Vibration of an elastically mounted circular cylinder

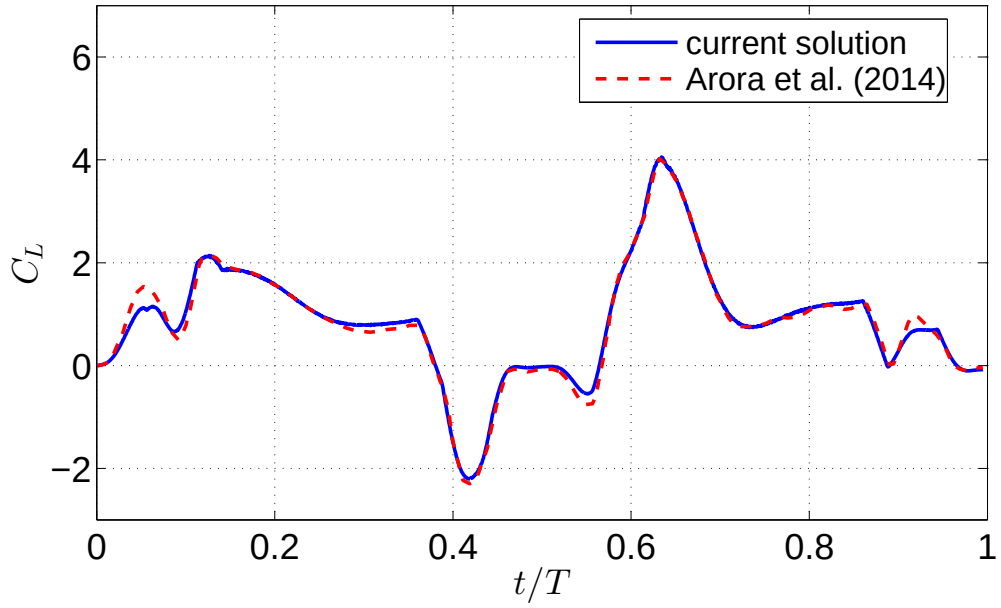


Figure 3.25: Time history of the lift coefficient of the right foil over a single clap-and-fling cycle at $Re = 128$. The lift coefficient of the left wing is identical to that of the right. The result from [28] is used for comparison.

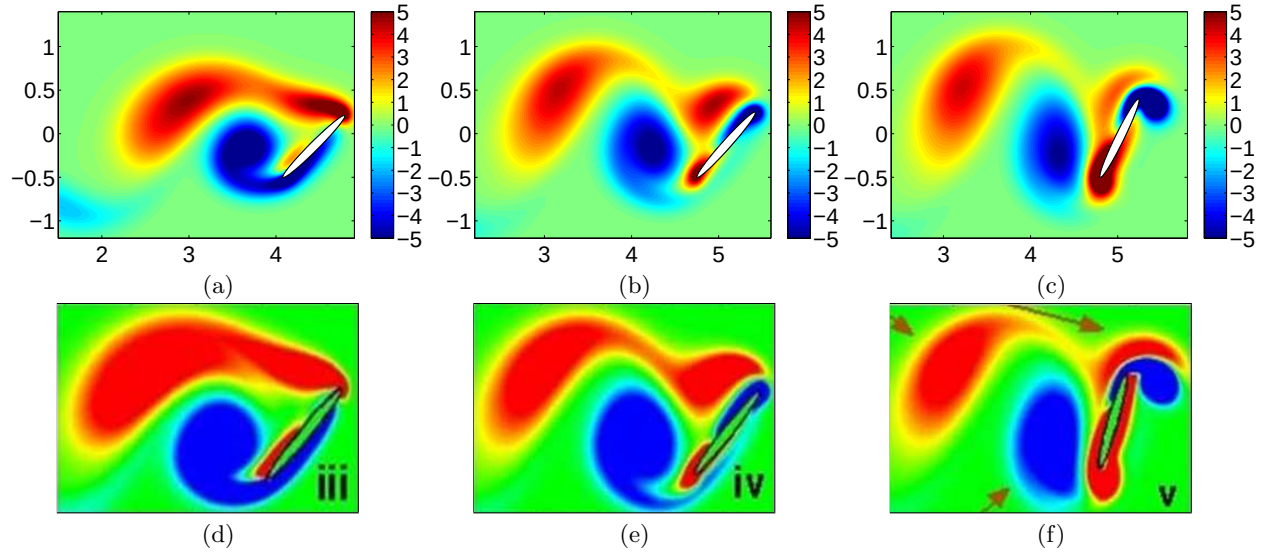


Figure 3.26: Instantaneous vorticity fields near the foil at several time instants during the fling motion: $t/T =$ (a) 0.35, (b) 0.41, and (c) 0.44. Panels (d)-(f) show the vorticity fields at approximately the same time instants extracted from [28].

In Fig. 3.32, a contour plot of the non-dimensional vorticity field ($\omega D/U_0$) at the instant of maximum excursion for $U_{red} = 4$ is shown.

3.6. Two-dimensional Vortex-Induced Vibration of an elastically mounted circular cylinder

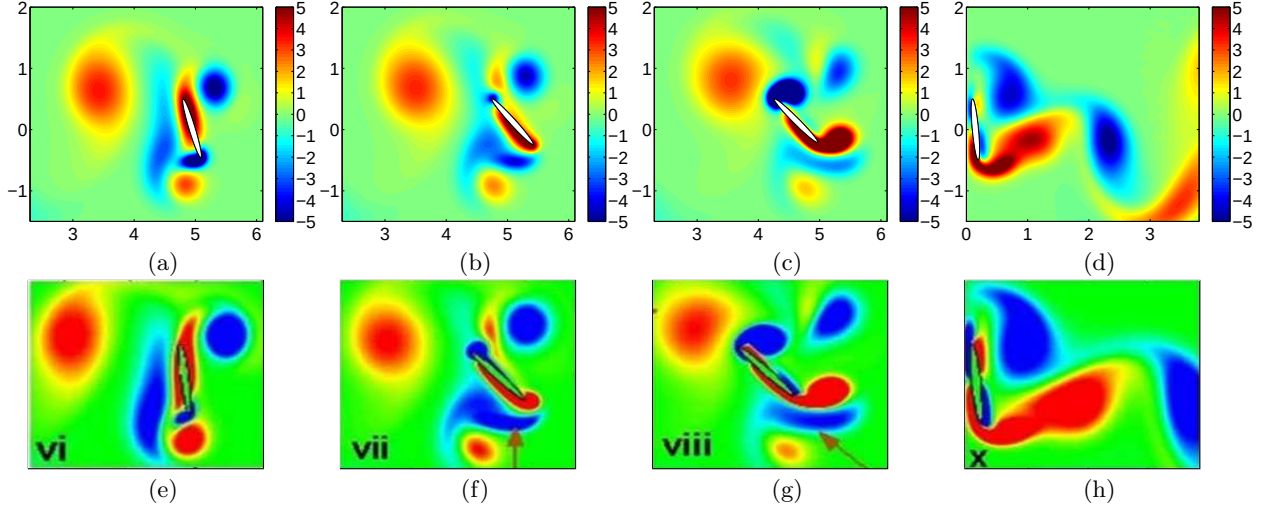


Figure 3.27: Instantaneous vorticity fields near the foil at several time instants during the clap motion: $t/T =$ (a) 0.55, (b) 0.59, (c) 0.63, and (d) 0.97. Panels (e)-(h) show the vorticity fields at approximately the same time instants extracted from [28].

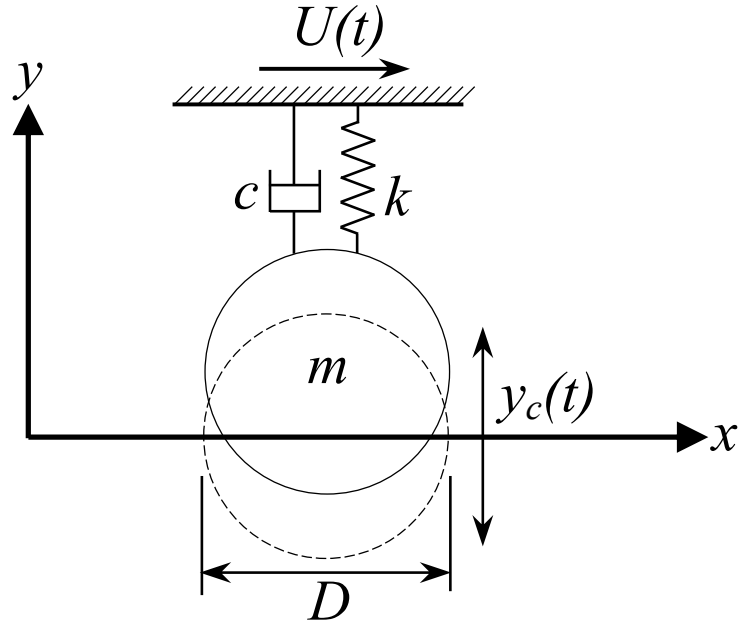


Figure 3.28: A circular cylinder translating in the x -direction undergoing VIV in the y -direction.

3.6. Two-dimensional Vortex-Induced Vibration of an elastically mounted circular cylinder

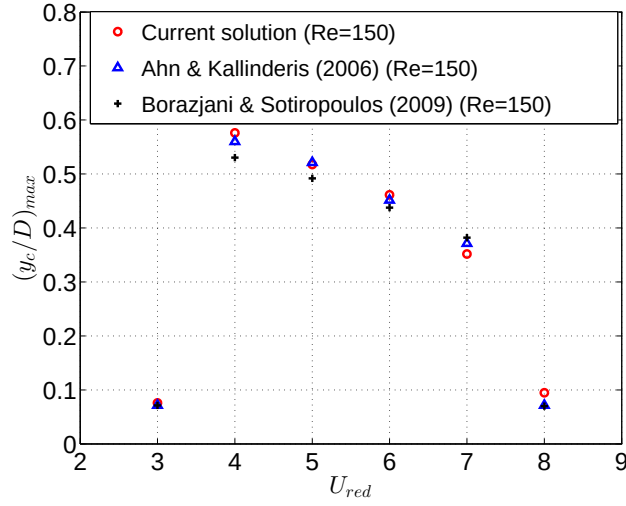


Figure 3.29: Maximum excursion of a two-dimensional spring-mounted circular cylinder undergoing VIV. $Re = 150$, $M_{red} = 2$, and $\zeta = 0$. The results from [95, 96] are used for comparison.

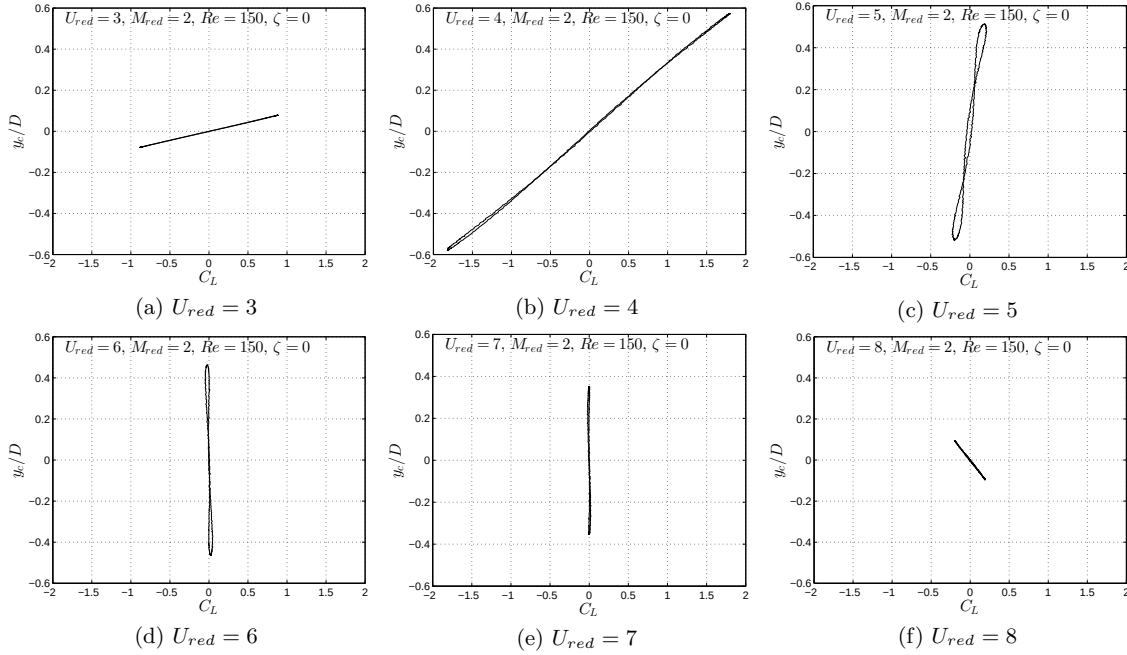


Figure 3.30: Phase relation between lift and excursion at periodic steady state of a two-dimensional spring-mounted circular cylinder undergoing VIV. $Re = 150$, $M_{red} = 2$, and $\zeta = 0$.

3.6. Two-dimensional Vortex-Induced Vibration of an elastically mounted circular cylinder

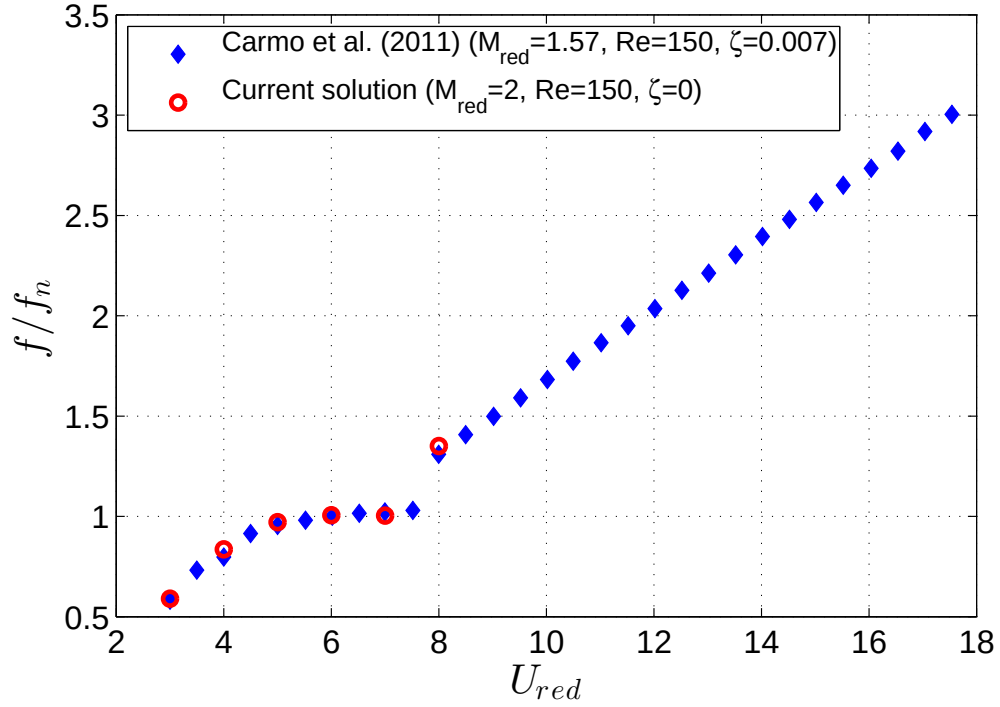


Figure 3.31: Normalized VIV frequency as a function of the reduced velocity, U_{red} . For current DVM solution, $M_{red} = 2$, $Re = 150$, and $\zeta = 0$. The results for $M_{red} = 1.57$, $Re = 150$, and $\zeta = 0.007$ from [97] are included for comparison.

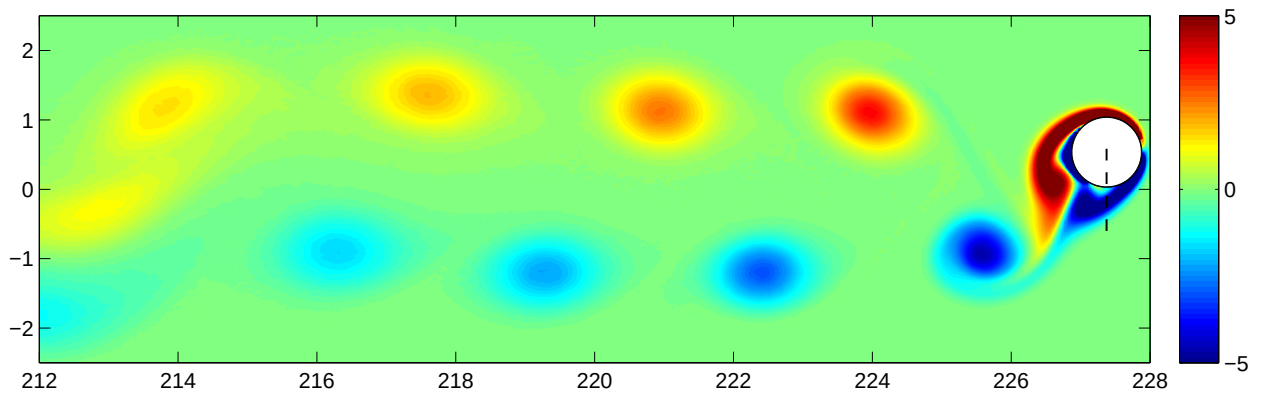


Figure 3.32: Non-dimensional vorticity field ($\omega D/U_0$) of the wake of a circular cylinder undergoing VIV. The cylinder is near maximum excursion. The dashed line shows the range of cylinder center excursion. $M_{red} = 2$, $U_{red} = 4$, $\zeta = 0$, and $Re = 150$. The horizontal and vertical scales are x/D and y/D , respectively.

Chapter 4

Investigation of full and partial ground effects on a flapping foil hovering above a finite-sized platform

4.1 Problem description

The flapping motion of an elliptic airfoil of chord c near a finite-sized platform in two dimensions is considered. As pointed out in Sec. 1.3.1, this problem is pertinent to the control of an MAV slowly approaching and landing on a small platform. Similar to Sec. 3.4, the translational motion of the airfoil during one period T can be separated into two parts: upstroke and downstroke (see Fig. 4.1). The switch in the direction of translation between upstrokes and downstrokes is referred to as stroke reversals. In conjunction with the stroke reversals, the airfoil rotates about the mid-chord point to re-orient. The translation and rotation of the airfoil are, again, described by simple sinusoidal functions in time:

$$X(t) = X_0 + A \cos(2\pi ft), \quad Y(t) = Y_0, \quad (4.1)$$

$$\alpha(t) = \alpha_0 + \beta \sin(2\pi ft + \phi) \quad (4.2)$$

where X and Y are the coordinates of the mid-chord point of the airfoil. The angle between the chord and the x -axis is given by α . The mean position and angle of the airfoil are given by (X_0, Y_0) and α_0 . The stroke and rotation amplitudes are A and β , respectively. The frequency of the motion is $f = 1/T$, and ϕ is the phase shift between rotation and translation.

Here, we shall only focus on the “normal hovering mode” with $\alpha_0 = \pi/2$. The value of ϕ determines the timing of the rotation. We have advanced, symmetric, and delayed rotations for $\phi > 0$, $\phi = 0$, and $\phi < 0$, respectively. The value of ϕ strongly influences the wake-capture mechanism and, therefore, affects the mean lift force [24]. However, ground effects show similar trends for all three types of rotation timing [36]. Based on this observation, we decided to limit our analysis to only symmetric rotation $\phi = 0$.

As shown in Fig. 4.1, a platform of finite length L_p and thickness t_p is placed below the airfoil parallel to the stroke plane. The top-right edge of the platform is located at the origin. The vertical distance D between the mid-chord point to the platform surface is simply equal to Y_0 , and the lateral distance S between the mean airfoil position and the platform edge is given by X_0 .

The fluid is viscous and incompressible. Both no-penetration and no-slip boundary conditions are enforced on the airfoil surface and the platform surface. In the far field, the flow is undisturbed.

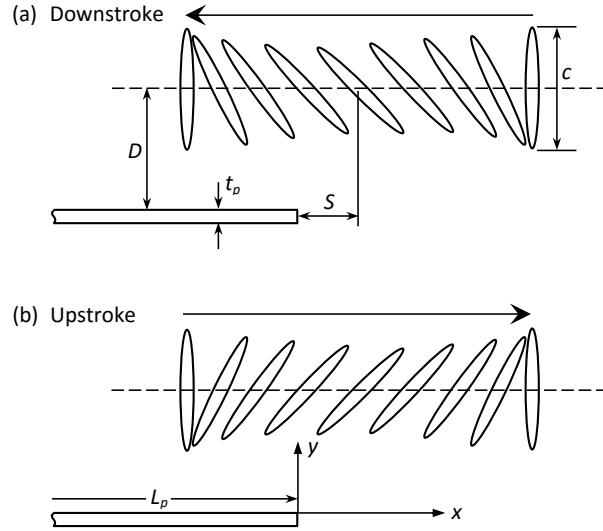


Figure 4.1: An elliptic airfoil flapping in normal hovering mode above the edge of a flat platform. The dashed line represents the stroke plane. (a) Downstroke. (b) Upstroke.

4.2 Numerical results

In this study, we mainly focus on the effect of the placement of the platform, which is determined by the choices of S/c and D/c . All other parameters are fixed in order to reduce the number of cases we need to simulate to a manageable level. For all subsequent results presented, we have used $L_p/c = 10$ and $L_p/t_p = 80$ for the platform length and thickness. The elliptic airfoil has aspect ratio 8. The stroke and rotation amplitudes are set to $A/c = 1.25$ and $\beta = \pi/4$, respectively. The Reynolds number is set to $Re = U_{max}c/\nu = 100$ with $U_{max} = 2\pi fA$.

We have used the case with $S/c = 0$ and $D/c = 1$ to check the convergence of the numerical solution. Three different settings, coarse, medium, and fine, are used. The configuration of each setting is listed in Table 4.1. The time-dependent lift and drag coefficients, as defined in Eq. (3.5), of the first seven periods are shown in Fig. 4.2.

Table 4.1: Configurations for convergence tests

Case	Time step ($\Delta t/T$)	No. of panels (airfoil)	No. of panels (platform)
Coarse	12.5×10^{-3}	60	503
Medium	5×10^{-3}	100	807
Fine	2.5×10^{-3}	200	1615

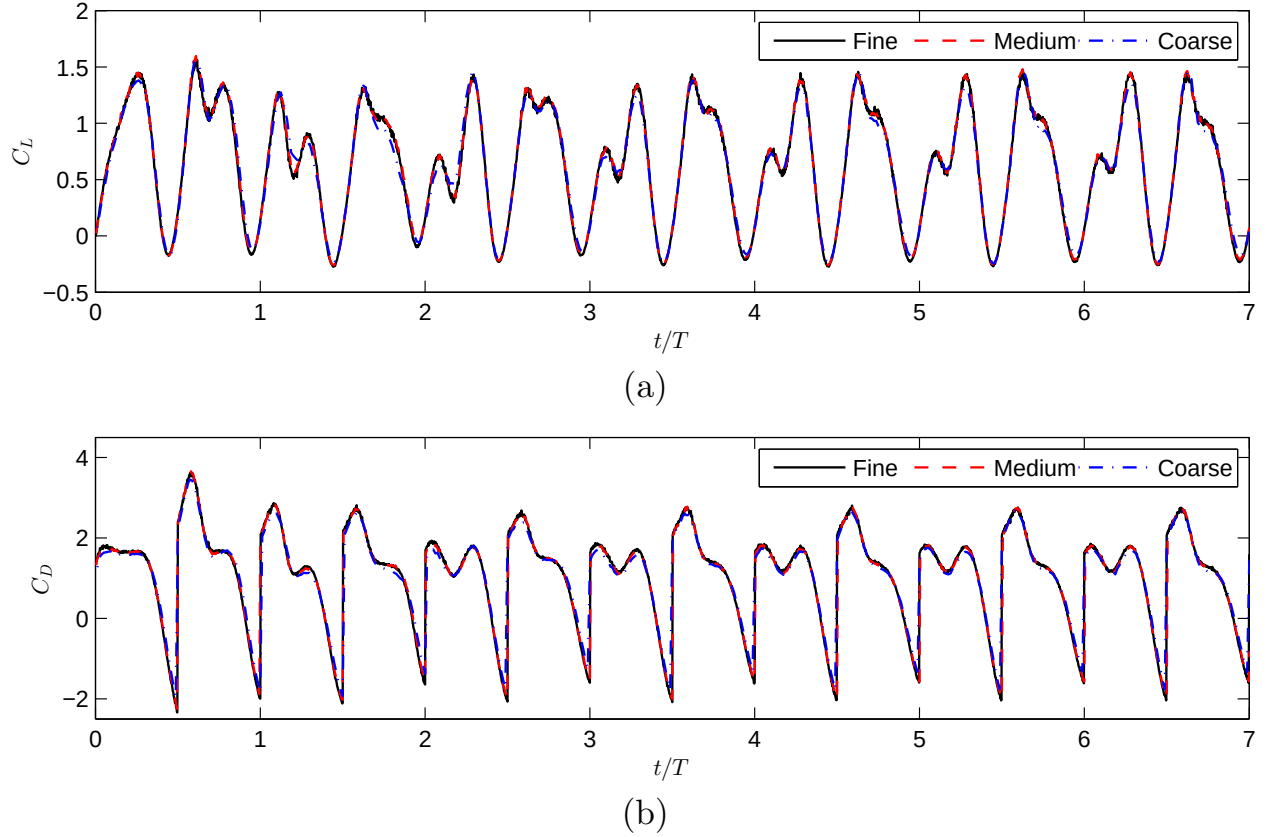


Figure 4.2: Time-dependent (a) lift and (b) drag coefficients over the first seven periods for $S/c = 0$ and $D/c = 1$. The configurations of the coarse, medium, and fine cases are listed in Table 4.1.

Visually, the results from the fine and medium cases are indistinguishable from each other, and the difference in mean lift coefficient of the seventh period is approximately 1% between the two cases. We believe the medium configuration shows adequate convergence for our analysis. The time step of $\Delta t/T = 5 \times 10^{-3}$ of the medium configuration significantly reduces computation time, allowing us to simulate more periods of motion for each case. The airfoil and the platform consist of 100 and 807 panels, respectively. Smaller panels are used near the leading and trailing edges of the airfoil.

In subsequent sections, each case is run up to 15 periods of flapping motion. As an example, the time histories of the lift and drag coefficients for the case $S/c = 0.25$ and $D/c = 1$ are shown in Fig. 4.3. The startup effect is very prominent during the first three periods and becomes negligible after approximately eight periods. The solution of the last five periods is used to calculate time-averaged lift and drag coefficients, $\overline{C_L}$ and $\overline{C_D}$. The time-dependent lift and drag coefficients presented, C_L and C_D , over one cycle ($t/T \in [0, 1)$) are also obtained by averaging the instantaneous lift and drag coefficients at the same time instant within each of the last five periods.

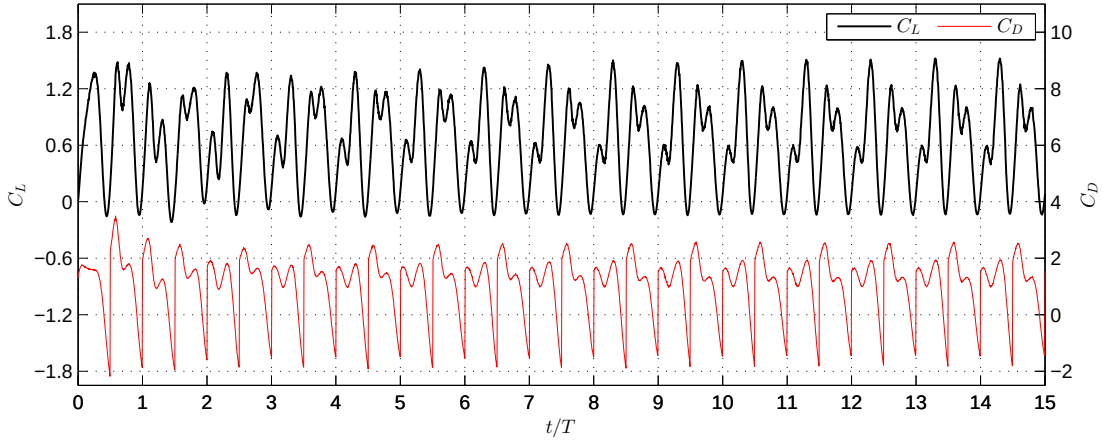


Figure 4.3: Time histories of the lift and drag coefficients for $S/c = 0.25$ and $D/c = 1$. The top curve is the lift coefficient.

4.2.1 Full ground effect

We shall first consider cases with $S/c = -5$. The mean position of the flapping airfoil is right above the center of the platform. Because the airfoil remains far away from the edges of the platform on both sides, the airfoil should experience close to full ground effect. As shown in Fig. 4.4, the time-averaged lift coefficient over one period is well above the infinite-fluid averaged lift coefficient, $\overline{C_{L\infty}}$, when $D/c = 1$. As D/c increases to 3, the averaged lift coefficient rapidly drops below $\overline{C_{L\infty}}$. A gradual recovery of $\overline{C_L}$ towards $\overline{C_{L\infty}}$ is observed when $D/c = 4$ and 5. As in [36], we can identify three force regimes: force-enhancement, force-reduction, and force-recovery regimes. The time-averaged drag coefficient shows the same dependence on D/c as the lift coefficient.

The time-dependent lift and drag coefficients of the airfoil over one period are shown in Fig. 4.5. Over one period, four peaks in lift coefficient are typically observed with two peaks occurring during the downstroke around $t/T = 0.1$ and 0.3 and another two peaks occurring during the upstroke around $t/T = 0.6$ and 0.8 (see e.g. Fig. 4.5(a), $D/c = 5$).

The instantaneous non-dimensional vorticity fields, $\omega c/U_{max}$, during the downstroke and upstroke with full ground effects are shown in Figs. 4.6 and 4.7, respectively. The three ground clearances shown, $D/c = 1, 3$, and 5 , correspond to the three force regimes. For all vorticity plots in this chapter, solid contour lines represent positive (counterclockwise) vorticity, and dashed contour lines represent negative (clockwise) vorticity. Unless stated otherwise, increment of contour level is 1. The horizontal and vertical coordinates in all vorticity and velocity field plots in this chapter are x/c and y/c , respectively.

In the force-enhancement regime ($D/c = 1$), the first and third peaks are greatly enhanced and overshadow the second and fourth peaks (Fig. 4.5(a)). Timewise, the first and third peaks occur right after stroke reversals. Because of the reduced ground clearance in the force-enhancement regime, the leading edge vortex (LEV) from the previous stroke cannot

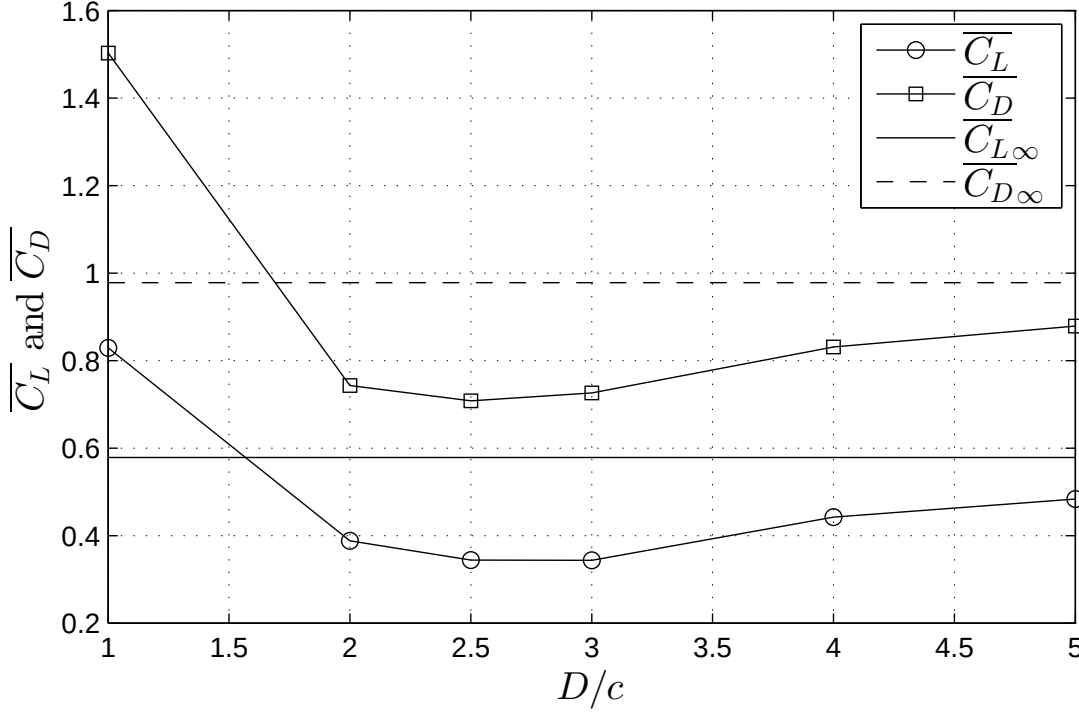


Figure 4.4: Time-averaged lift and drag coefficients, $\overline{C_L}$ and $\overline{C_D}$, of the flapping elliptic airfoil in normal hovering mode above the center of the platform ($S/c = -5$). The time-averaged lift and drag coefficients of the same airfoil in infinite fluid are $\overline{C_{L\infty}}$ and $\overline{C_{D\infty}}$, respectively.

convect downwards and remains near the leading edge during stroke reversal as shown in Fig. 4.6(a) $t/T = 0$ and Fig. 4.7(a) $t/T = 0.5$. This strong vortex forms a vortex pair with the trailing edge vortex (TEV) from the previous stroke, inducing a strong jet impinging on the wind side of the foil. This jet is responsible for the high lift and drag experienced by the foil after stroke reversal [44]. In addition, the LEV from the previous stroke is partially forced over the leading edge of the airfoil right after stroke reversal, resulting in the formation of a stronger new LEV of the opposite sign (see Fig. 4.6(a) $t/T = 0.1$ and Fig. 4.7(a) $t/T = 0.6$). As pointed out in [36], this stronger new LEV on the lee side of the airfoil may also contribute to the high lift and drag on the airfoil right after stroke reversal. However, the strong LEV also detaches from the airfoil sooner, diminishing the second and fourth peaks of the lift curve (see Fig. 4.6(a) $t/T = 0.3$ and Fig. 4.7(a) $t/T = 0.8$). For convenience, the above flow behaviors in the force-enhancement regime shall be collectively referred to as the ground-enhanced wake-capture mechanism.

4.2. Numerical results

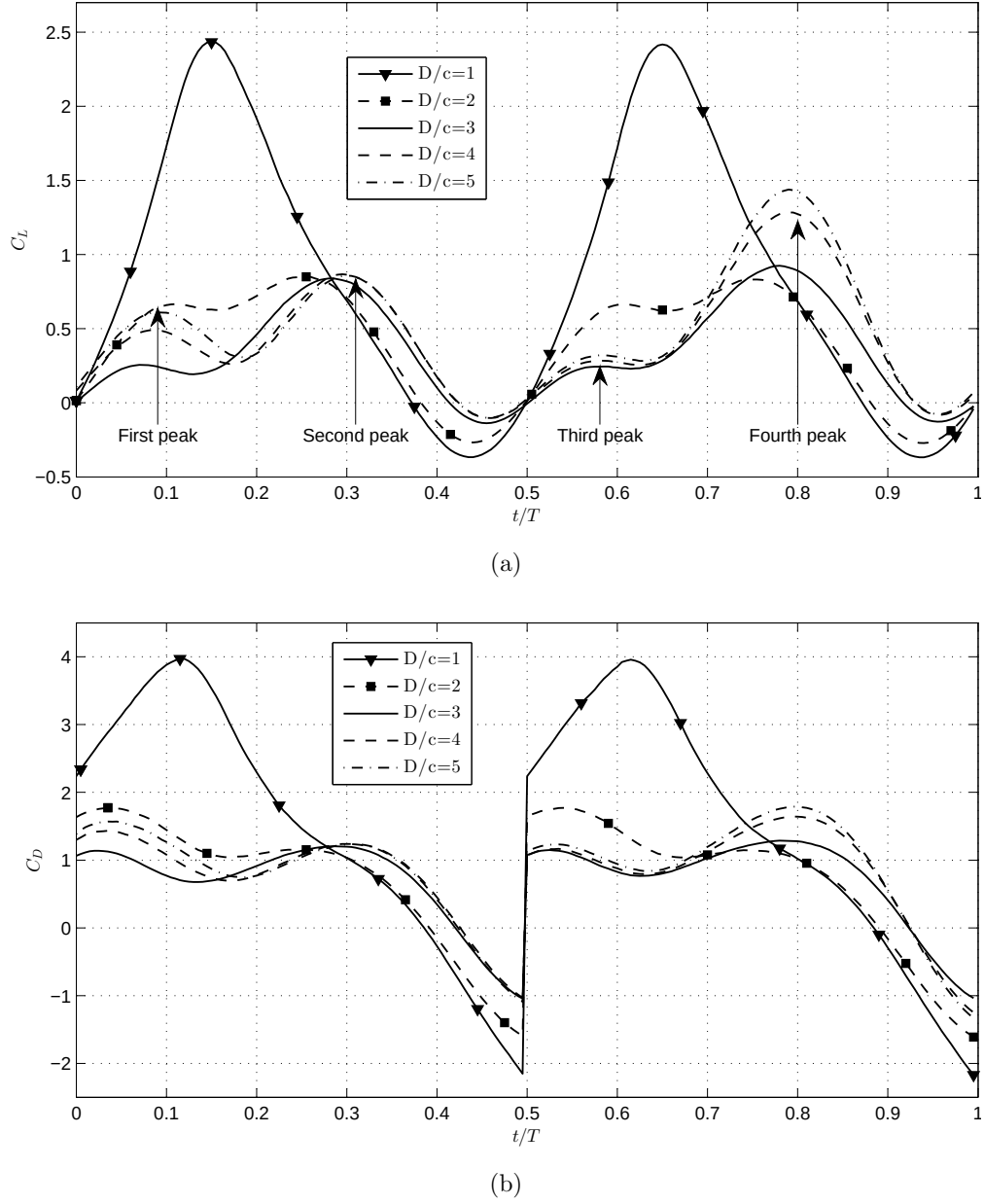


Figure 4.5: Time-dependent (a) lift coefficient and (b) drag coefficient of the flapping elliptic airfoil over one period of flapping motion under full ground effect ($S/c = -5$). Downstroke occurs over $t/T = 0$ to 0.5 , and upstroke occurs over $t/T = 0.5$ to 1 . Stroke reversals occur at $t/T = 0(1)$ and 0.5 .

4.2. Numerical results

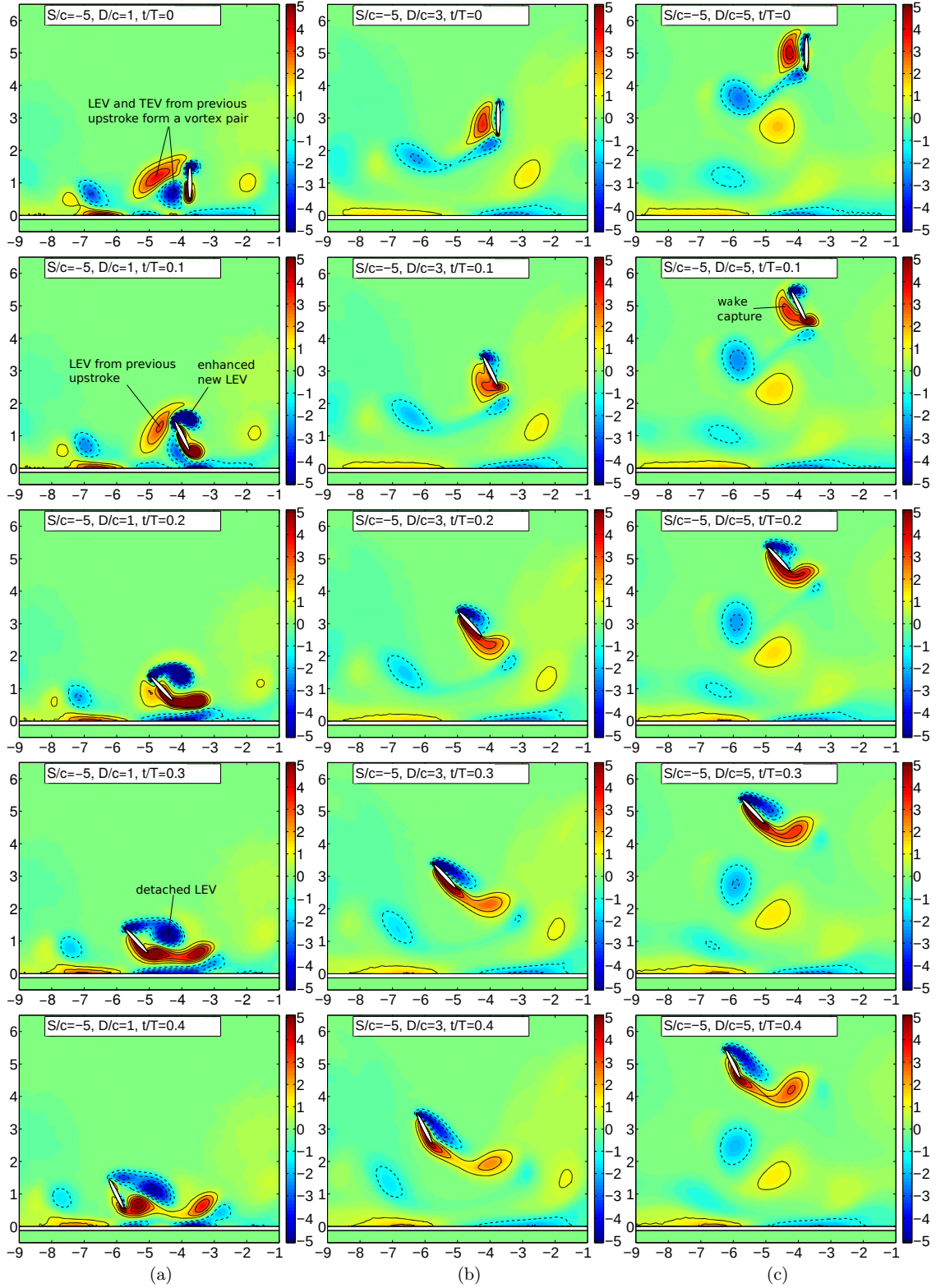


Figure 4.6: Non-dimensional vorticity fields ($\omega c/U_{max}$) during downstroke for $S/c = -5$ and
(a) $D/c = 1$, (b) $D/c = 3$, and (c) $D/c = 5$.

4.2. Numerical results

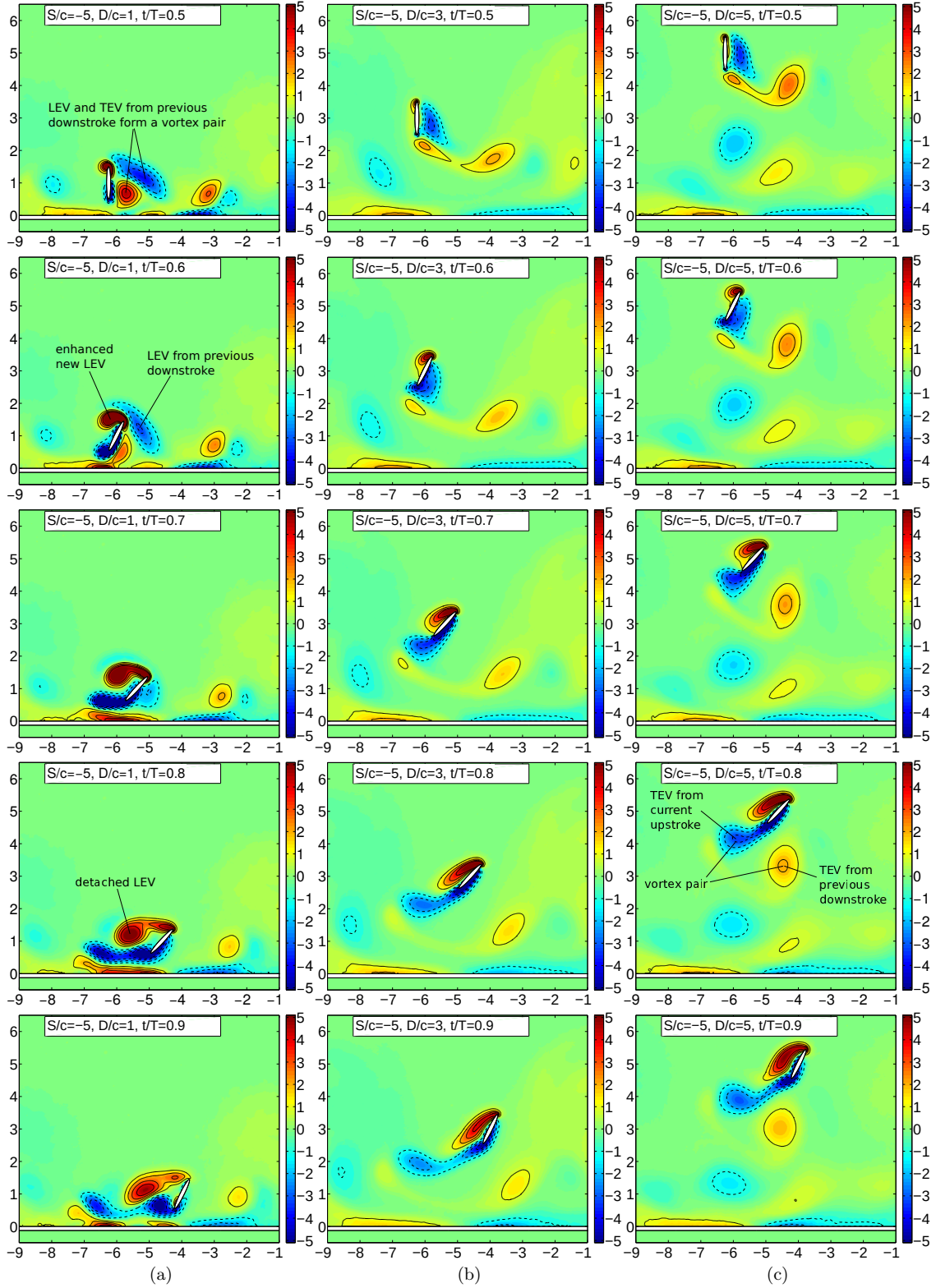


Figure 4.7: Non-dimensional vorticity fields ($\omega c/U_{max}$) during upstroke for $S/c = -5$ and
(a) $D/c = 1$, (b) $D/c = 3$, and (c) $D/c = 5$.

In the force-recovery regime ($D/c = 4$ and 5), the increase in lift force is in large part contributed by the enhancement of the fourth peak (see Fig. 4.5(a)). During the second half of the upstroke (the fourth peak), the negative (clockwise) TEV formed during the upstroke combines with the positive (counterclockwise) TEV from the previous downstroke into a strong vortex pair (see Fig. 4.7(c) $t/T = 0.8$). This vortex pair induces a downward jet which leads to increased lift. In order for this mechanism to take place, a sufficiently large ground clearance is needed to accommodate the vortex pair. In addition to the increased fourth peak, the first peak at $t/T = 0.1$ is also slightly enhanced possibly because of a wake-capture effect. Comparing Figs. 4.6(b) and (c) for $t/T = 0.1$, we can see that the positive LEV from previous upstroke is placed closer to the leading edge when $D/c = 5$. This wake-capture effect is, however, much weaker than what is observed in the force-enhancement regime when $D/c = 1$.

In the force-reduction regime ($D/c = 2$ and 3), ground clearance is large enough so that the LEV from the previous stroke tends to slide down along the wind side of the airfoil after stroke reversal and merge with the new TEV instead of being forced over the leading edge (Fig. 4.6(b) $t/T = 0.1$ and Fig. 4.7(b) $t/T = 0.6$). As a consequence, the ground-enhanced wake-capture mechanism does not take place. At the same time, the ground clearance is not sufficient to accommodate the formation of the vortex pair during the second half of the upstroke (Fig. 4.7(b) $t/T = 0.8$), and the fourth peak is also diminished.

4.2.2 Partial ground effect

To investigate the force on the flapping airfoil under partial ground effect, DVM simulations of the flapping airfoil have been carried out for $S/c \in [-3, 2]$ and $D/c \in [1, 5]$. The results are summarized in Fig. 4.8.

Figure 4.8(a) shows the time-averaged lift coefficients. In the force-enhancement regime ($D/c = 1$), the general trend shows the lift coefficient increasing as S/c decreases. When $S/c = 2$, the mean lift force experienced by the airfoil is approximately equal to that in infinite fluid. When $S/c = -3$, the lift approaches its full-ground-effect strength. In the force-reduction regime ($D/c = 2$ and 3), the lift coefficient decreases below its infinite-fluid level as S/c decreases and approaches that of full ground effect when $S/c = -3$. The force-recovery regime is reached when $D/c = 4$ and 5 . In this regime, the dependence of lift on S/c weakens, and the lift coefficient is close to that in an infinite fluid. The drag coefficient shown in Fig. 4.8(b) demonstrates similar trends.

Considering the lift and drag coefficients, both time-averaged and time-dependent, generally show similar dependence on S/c and D/c , we will mainly focus on the lift force and its relation with the vorticity field in the following sections.

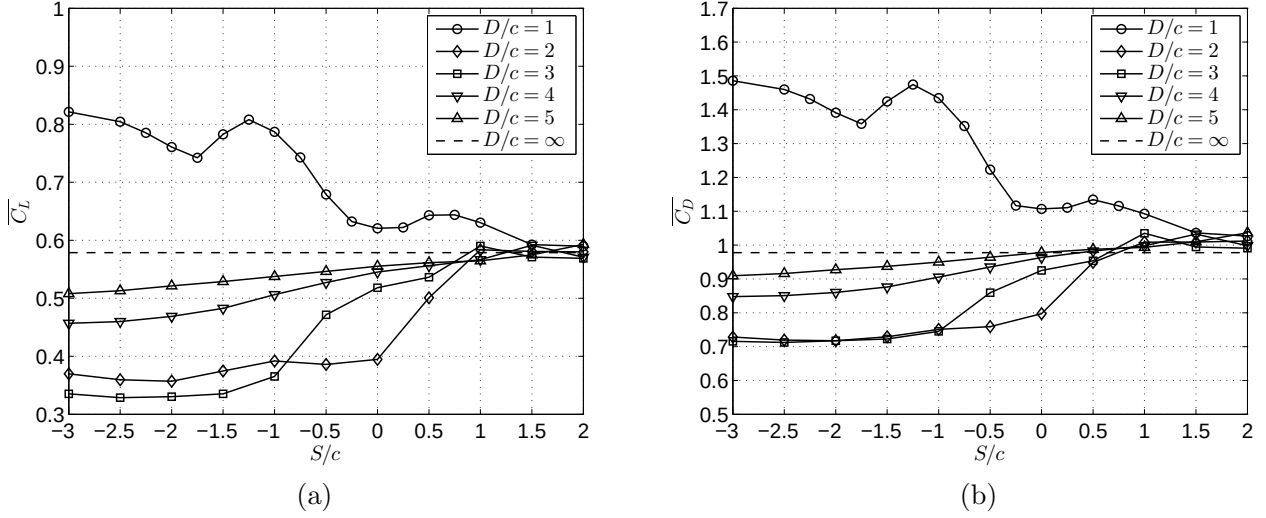


Figure 4.8: Time-averaged (a) lift coefficients and (b) drag coefficients of a flapping elliptic airfoil in normal hovering mode near the edge of a platform.

4.2.2.1 Force enhancement under partial ground effect

The time-dependent lift coefficient over one period of flapping motion is presented in Fig. 4.9 for several S/c values of interest in the force-enhancement regime ($D/c = 1$).

According to the time history, the initial increase in mean lift observed as S/c decreases from 2 towards 0 can be attributed to three different mechanisms. Comparing the time-dependent lift coefficient of $S/c = 0.75$ against that of $S/c = 2$, we see that the second and third peaks of the lift curve are enhanced while the fourth peak is diminished. For $S/c = 0.75$, the second peak occurs during the second half of the downstroke as the airfoil sweeps past the edge of the platform. As shown in Fig. 4.10(a), the positive TEV of the airfoil forms a vortex pair with the negative vortex shed from the edge of the platform, inducing a downward jet and enhancing the lift force. In Fig. 4.11, the velocity fields near the foil are shown for both $S/c = 0.75$ and $S/c = 2$ at the time instant corresponding to the second peak in lift ($t/T = 0.35$). In Fig. 4.11(a), the fluid is being squeezed through the gap between the trailing edge of the foil and the platform edge, forming a strong downward jet. A similar downward stream is observed in Fig. 4.11(b) below the foil when $S/c = 2$. However, the stream is considerably weaker. Using the velocity magnitude at the point $0.5c$ vertically below the trailing edge of the foil as a reference, the jet of $S/c = 0.75$ is close to 40% faster compared to that of $S/c = 2$. The significantly stronger downward jet is closely related to the increase in lift. This lift-enhancing mechanism is not previously observed and only occurs when the platform edge is present. Subsequently, the stroke reversal at $t/T = 0.5$ occurs above the edge of the platform when $S/c = 0.75$, and the LEV from the previous downstroke is obstructed from convecting downwards after stroke reversal (see Fig. 4.10(b)). The ground-enhanced wake-capture mechanism now partially takes effect, leading to the enhancement of the third peak. Lastly, the edge of the platform partially obstructs the

4.2. Numerical results

formation of the vortex pair during the second half of the upstroke, diminishing the fourth peak of the lift coefficient curve (see Fig. 4.10(c)). The net effect of the three mechanisms generates over 10% increase in time-averaged lift coefficient. Also, a local minimum in averaged lift coefficient is observed when $S/c = 0$ because of the rapidly diminishing fourth peak.

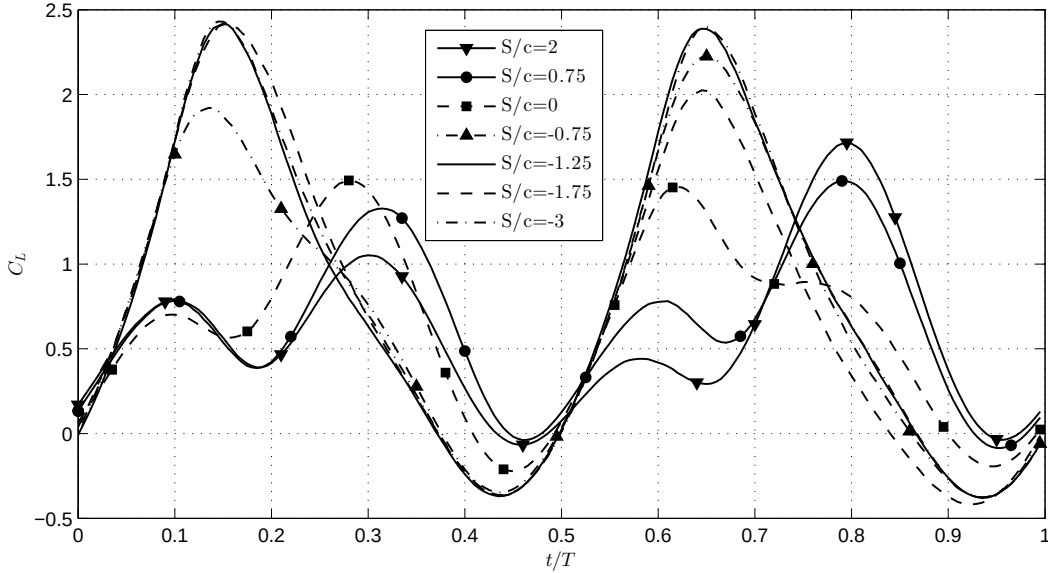


Figure 4.9: Time-dependent lift coefficient of the flapping elliptic airfoil over one period of flapping motion in the force-enhancement regime ($D/c = 1$).

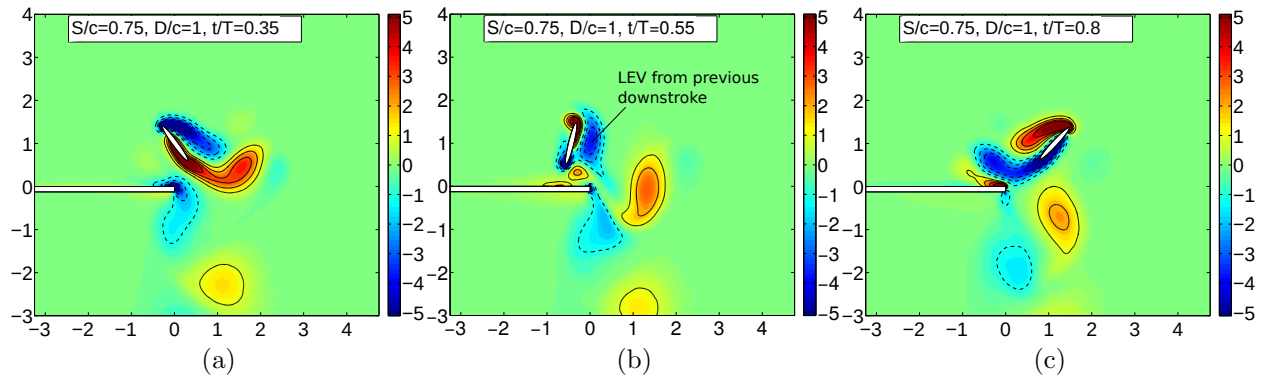


Figure 4.10: Non-dimensional vorticity fields ($\omega c/U_{max}$) for $S/c = 0.75$ and $D/c = 1$ at $t/T =$ (a) 0.35, (b) 0.55, and (c) 0.80.

When S/c further decreases to -0.75 , the stroke reversal at $t/T = 0$ also occurs very close to the platform edge. The positive LEV from the previous upstroke partially moves over

4.2. Numerical results

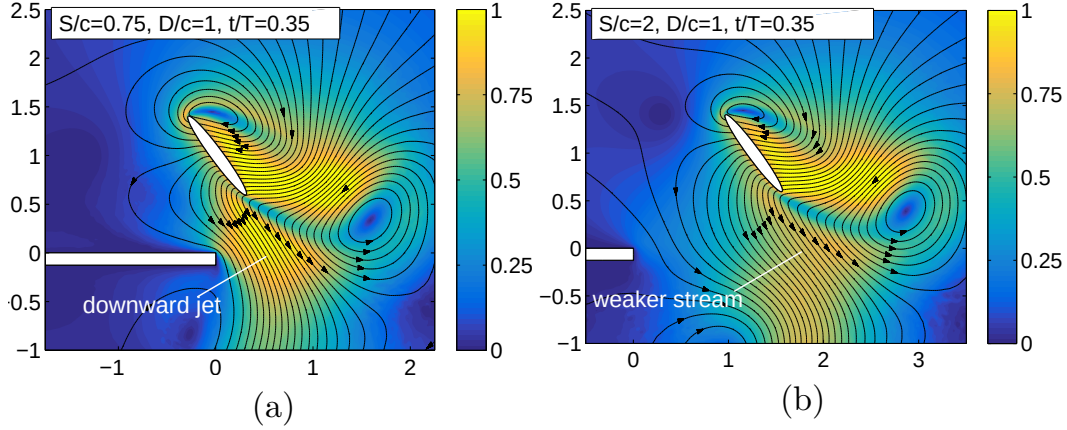


Figure 4.11: Non-dimensional velocity magnitude normalized by U_{max} (color) and streamlines (dark curves) for $D/c = 1$ with (a) $S/c = 0.75$ and (b) $S/c = 2$ at $t/T = 0.35$.

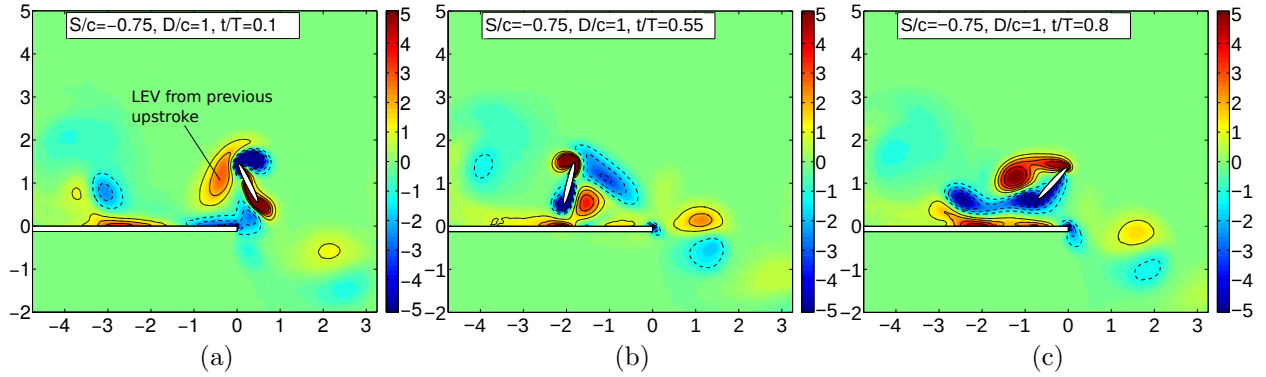


Figure 4.12: Non-dimensional vorticity fields ($\omega c/U_{max}$) for $S/c = -0.75$ and $D/c = 1$ at $t/T =$ (a) 0.1, (b) 0.55, and (c) 0.80.

the leading edge of the airfoil immediately after stroke reversal (see Fig. 4.12(a)), enhancing the first peak in lift force at $t/T = 0.15$. The third peak at $t/T = 0.65$ is further enhanced and is close to its full-ground-effect level. The ground-enhanced wake-capture mechanism is in full effect here (see Fig. 4.12(b)), similar to the situation shown in Fig. 4.7(a) for $t/T = 0.6$. During the second halves of the downstroke and upstroke, the LEV detaches from the airfoil, and the formation of the vortex pair is completely obstructed by the platform (see Fig. 4.12(c)). Thus, the second and fourth peaks are completely diminished.

As shown in Fig. 4.8(a), the averaged lift force approximately reaches its full-ground-effect level when $S/c = -1.25$. The time-dependent lift coefficient is almost identical for $S/c = -1.25$ and $S/c = -3$ as shown in Fig. 4.9. However, a peculiar local minimum in averaged lift coefficient is observed at $S/c = -1.75$ which corresponds to an approximately 10% drop in averaged lift compared to that of $S/c = -1.25$. In this case, the TEV from the

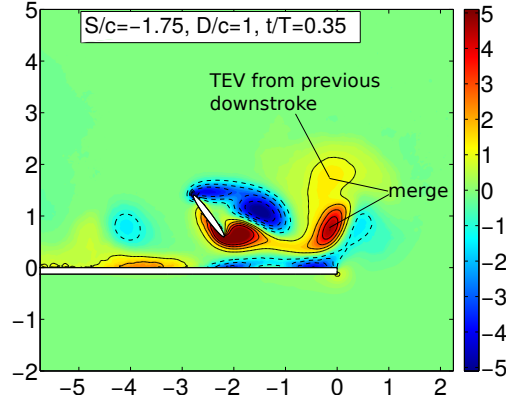


Figure 4.13: Non-dimensional vorticity field ($\omega c/U_{max}$) for $S/c = -1.75$ and $D/c = 1$ at $t/T = 0.35$. The positive TEV shed from the current downstroke is merging with the TEV shed from the previous downstroke.

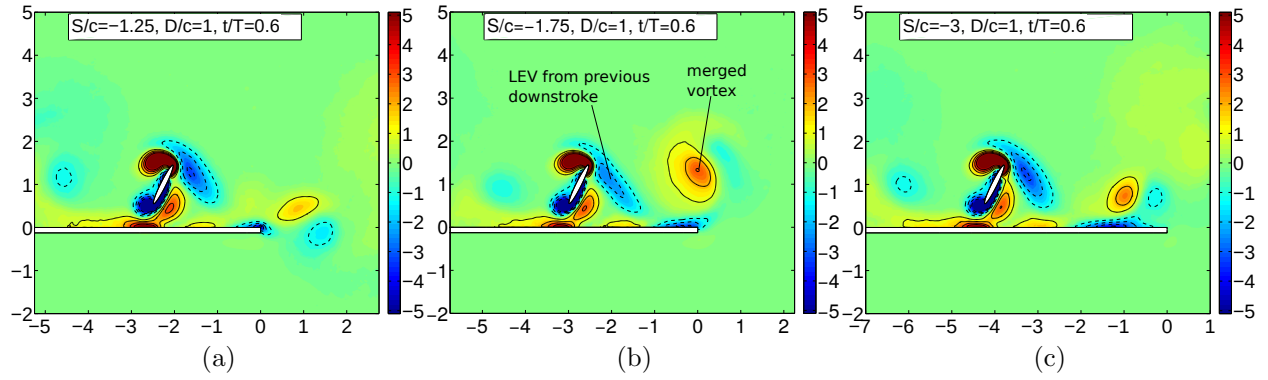


Figure 4.14: Non-dimensional vorticity fields ($\omega c/U_{max}$) at $t/T = 0.6$ for $D/c = 1$ and $S/c =$ (a) -1.25 , (b) -1.75 , and (c) -3 .

downstroke is shed right above the edge of the platform and has decreased mobility. The TEV remains near the edge of the platform during the entire period of flapping motion. This leads to the merging between the TEVs shed from the downstroke of the current period and the downstroke from the previous period as illustrated in Fig. 4.13. From Fig. 4.14(b), we can see that the TEVs merge into a strong positive vortex which remains near the edge of the platform. The velocity field induced by this strong vortex pulls the LEV formed during the downstroke towards the platform and away from the leading edge of the airfoil after the stroke reversal at $t/T = 0.5$, thus reducing the effect of the ground-enhanced wake-capture mechanism. Compared with Figs. 4.14(a) and (c), the negative LEV from downstroke in Fig. 4.14(b) is visibly being stretched and pulled towards the platform by the strong positive vortex near the edge of the platform. The velocity fields corresponding to Figs. 4.14(a) and (b) are shown in Fig. 4.15. In both Figs. 4.15(a) and (b), we see an upward jet induced by the LEV from the previous downstroke impinging on the wind side of the foil. However, the

impinging jet is weakened when $S/c = -1.75$ because of the influence of the strong positive vortex at the platform edge. The weakened jet leads to a slightly diminished third peak in lift compared with that of full ground effect (Fig. 4.9) and a local minimum in averaged lift coefficient in Fig. 4.8(a).

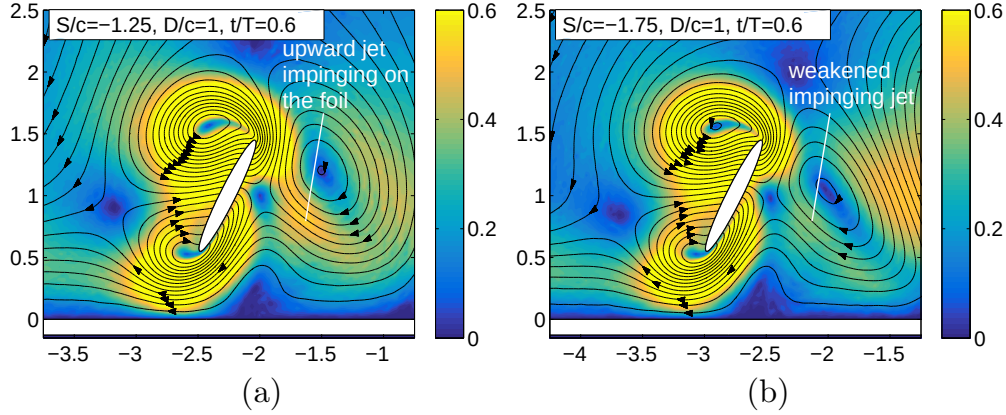


Figure 4.15: Non-dimensional velocity magnitude normalized by U_{max} (color) and streamlines (dark curves) for $D/c = 1$ with (a) $S/c = -1.25$ and (b) $S/c = -1.75$ at $t/T = 0.6$.

4.2.2.2 Force reduction under partial ground effect

In the force-reduction regime ($D/c = 2$ and 3), we observed a smooth decrease in average lift force as S/c decreases from 2 to -3 in Fig. 4.8(a). However, the change in the time history of the lift coefficient is quite complicated.

When $D/c = 2$, the mean lift force decreases rapidly from infinite-fluid level to full-ground-effect level when S/c decreases from 1 to 0 . The time-dependent lift coefficient over one period is shown in Fig. 4.16 for several S/c values of interest. In fact, for $S/c = 1$, we see that the second peak in the lift curve is slightly enhanced. This is likely caused by the edge of the platform reducing the downward mobility of the negative TEV shed during the upstroke. As shown in Fig. 4.17(a), this negative TEV and the new positive TEV shed during the subsequent downstroke form a closer vortex pair, enhancing the second peak in lift. In comparison, when $S/c = 2$, the negative TEV convects downwards further at $t/T = 0.3$, and the resulting vortex pair is not as strong (see Fig. 4.17(b)).

When S/c decreases to 0.5 , the negative TEV from the upstroke is shed on top of the platform and has very low mobility (Fig. 4.18(b)). This negative vortex remains in approximately the same position and merges with the negative TEV shed from the next upstroke after one full period as shown in Fig. 4.18(c). This strong negative vortex diminishes the first peak in the lift curve in a manner similar to what we have observed for the third peak in the case $S/c = -1.75$ and $D/c = 1$ discussed previously. By comparing Fig. 4.18(a) with Fig. 4.14(b), we see that the situation is approximately left-and-right mirrored. In contrast

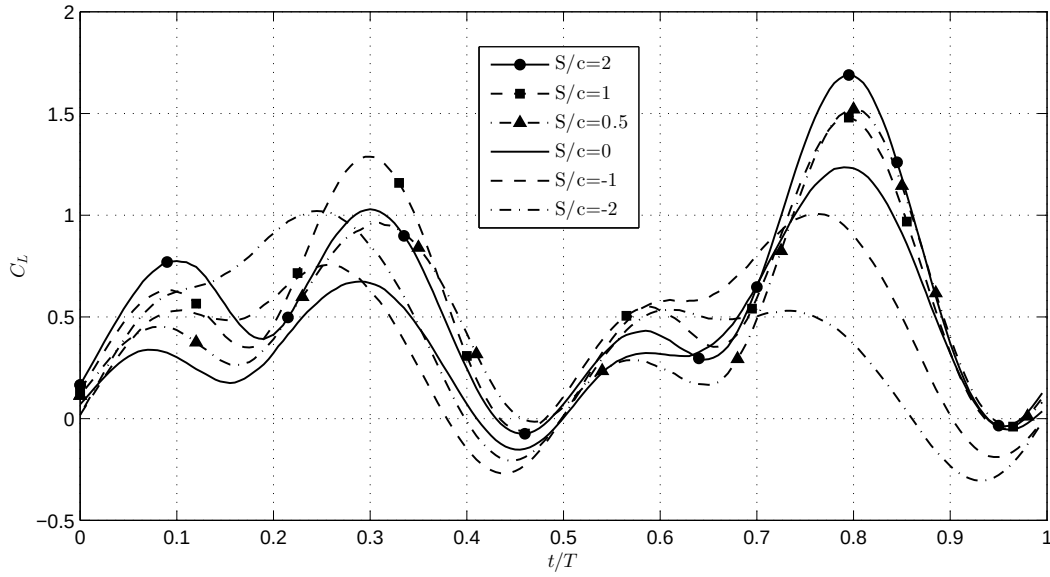


Figure 4.16: Time-dependent lift coefficient of the flapping elliptic airfoil over one period of flapping motion in the force-reduction regime ($D/c = 2$).

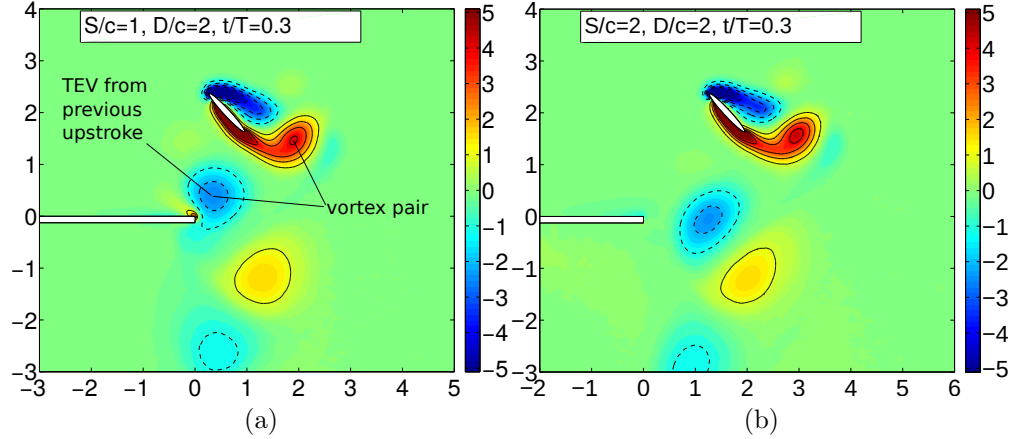


Figure 4.17: Non-dimensional vorticity fields ($\omega c/U_{max}$) at $t/T = 0.3$ for $D/c = 2$ and $S/c =$ (a) 1 and (b) 2.

to the negative TEV from upstroke, the positive TEV from downstroke convects downward, passing the edge of the platform (see Fig. 4.18(b)). As a result, a vortex pair cannot be formed during the second half of the downstroke ($t/T = 0.3$) in this case, and the second peak in Fig. 4.16 is no longer enhanced.

When S/c decreases from 0.5 to 0, the first and second peaks of the lift curve are further diminished. At the same time, the fourth peak starts to drop because the platform edge now

4.2. Numerical results

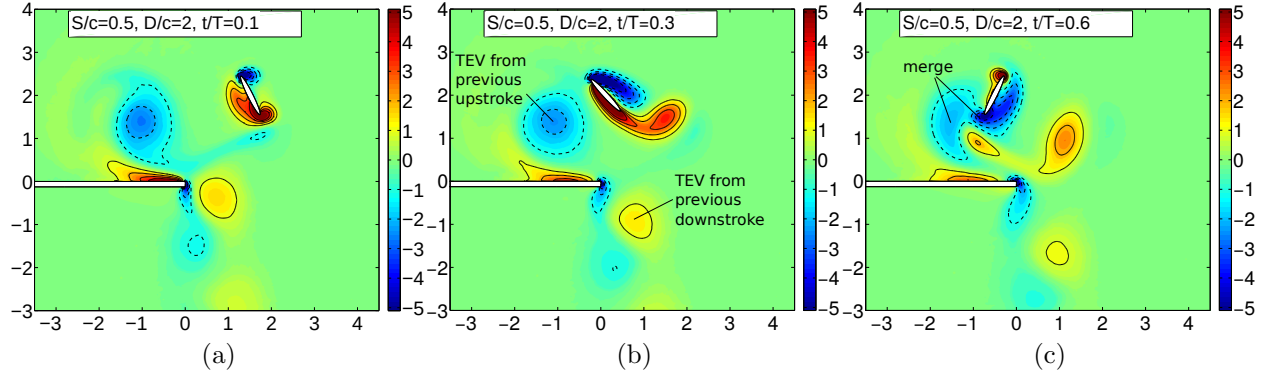


Figure 4.18: Non-dimensional vorticity fields ($\omega c/U_{max}$) for $S/c = 0.5$ and $D/c = 2$ at $t/T =$ (a) 0.1, (b) 0.3, and (c) 0.6.

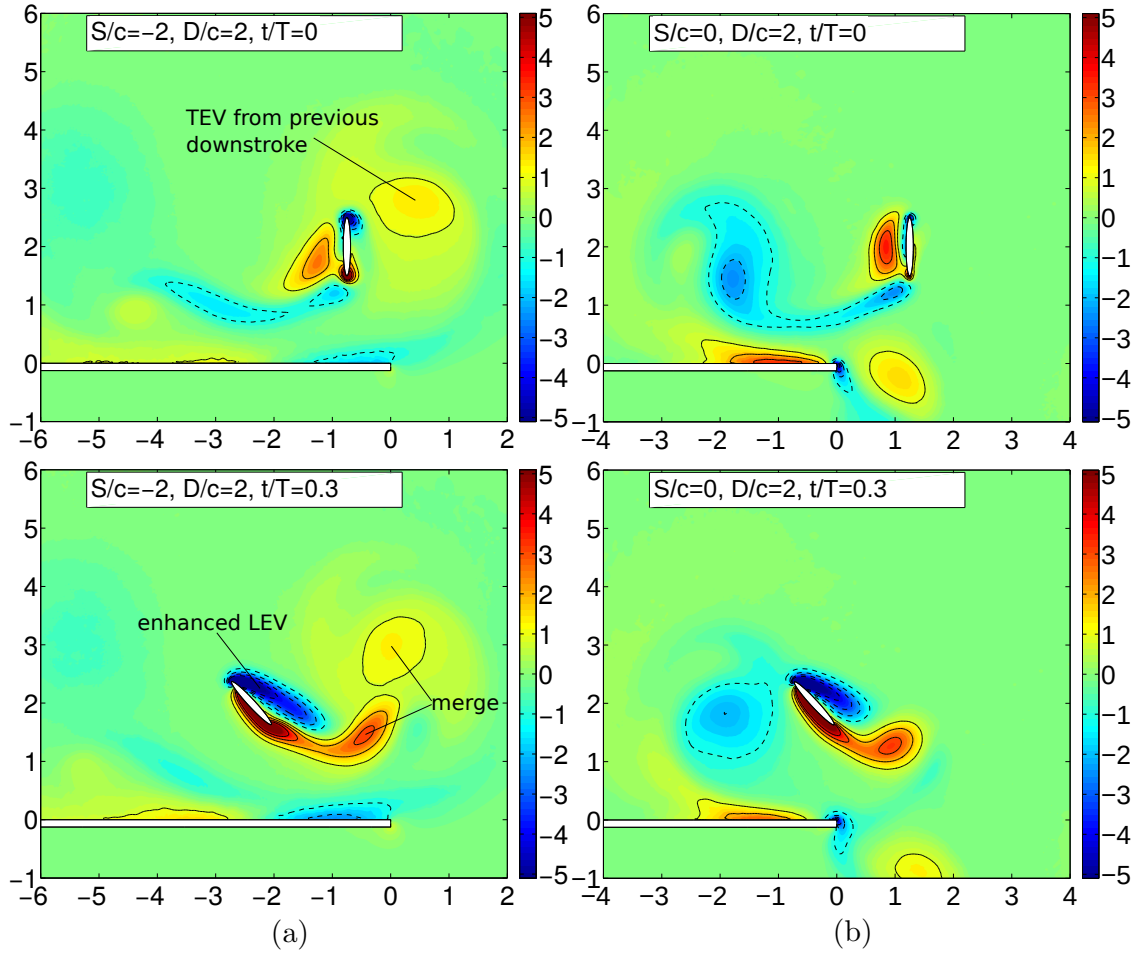


Figure 4.19: Non-dimensional vorticity fields ($\omega c/U_{max}$) for $S/c =$ (a) -2 and (b) 0 and $D/c = 2$ at $t/T = 0$ and 0.3 .

obstructs the formation of a vortex pair during the second half of the upstroke.

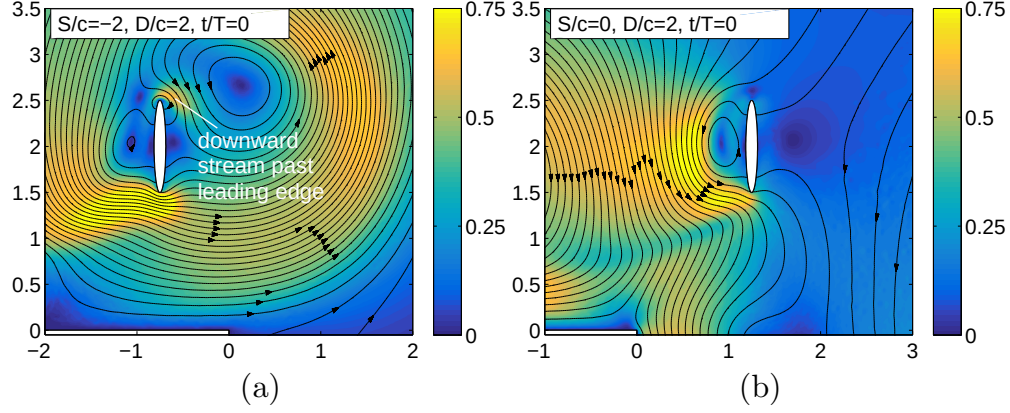


Figure 4.20: Non-dimensional velocity magnitude normalized by U_{max} (color) and streamlines (dark curves) for $D/c = 2$ with (a) $S/c = -2$ and (b) $S/c = 0$ at $t/T = 0$.

Interestingly, when S/c decreases further to -2 , we see a recovery of the first and second peaks of the lift curve. The positive TEV shed from the previous downstroke is positioned very close to the leading edge of the airfoil during stroke reversal at $t/T = 0$ (see Fig. 4.19(a)) and appears to enhance the newly formed negative LEV. The velocity fields at $t/T = 0$ of the two cases shown in Fig. 4.19 are illustrated in Fig. 4.20. When $S/c = -2$ (Fig. 4.20(a)), a downward stream induced by the positive TEV shed from the previous downstroke moves past the leading edge of the foil towards the lee side. This stream contributes to the formation of a stronger and better attached new negative LEV. This downward stream is not present when $S/c = 0$ (Fig. 4.20(b)). Comparing with the case of $S/c = 0$ shown in Fig. 4.19(b), we see that a larger LEV is formed when $S/c = -2$ which remains attached to the airfoil at $t/T = 0.3$ in Fig. 4.19(a). As a consequence, the first and second peaks of the lift curve are enhanced. In addition, the newly formed TEV eventually merges with the strong TEV from the previous downstroke, maintaining the strength of the positive vortex. The gain in lift from the first and second peaks, however, is countered by a rapid lost in lift from the fourth peak, resulting in almost constant averaged lift force for $S/c \leq 0$. When $S/c = -2$, the lift curve becomes almost identical to that under full ground effect.

When $D/c = 3$, we see that the first peak of the lift coefficient curve is slightly enhanced while the second peak is diminished when S/c decreases from 2 to 0 (see Fig. 4.21). As S/c further decreases from 0 to -2 , the first peak becomes greatly diminished while the height of the second peak gradually recovers to its infinite-fluid level.

From the vorticity field shown in Fig. 4.22(a), we see that the enhanced first peak and the diminished second peak observed at $S/c = 0$ are most likely caused by a weak wake-capture effect. The positive LEV from upstroke partially wraps around the leading edge of the airfoil at $t/T = 0.1$ after the stroke reversal, inducing a jet impinging on the wind side of the airfoil and enhancing the new negative LEV formed after stroke reversal. The new LEV has a stronger tendency to detach from the lee side of the airfoil at $t/T = 0.3$, leading

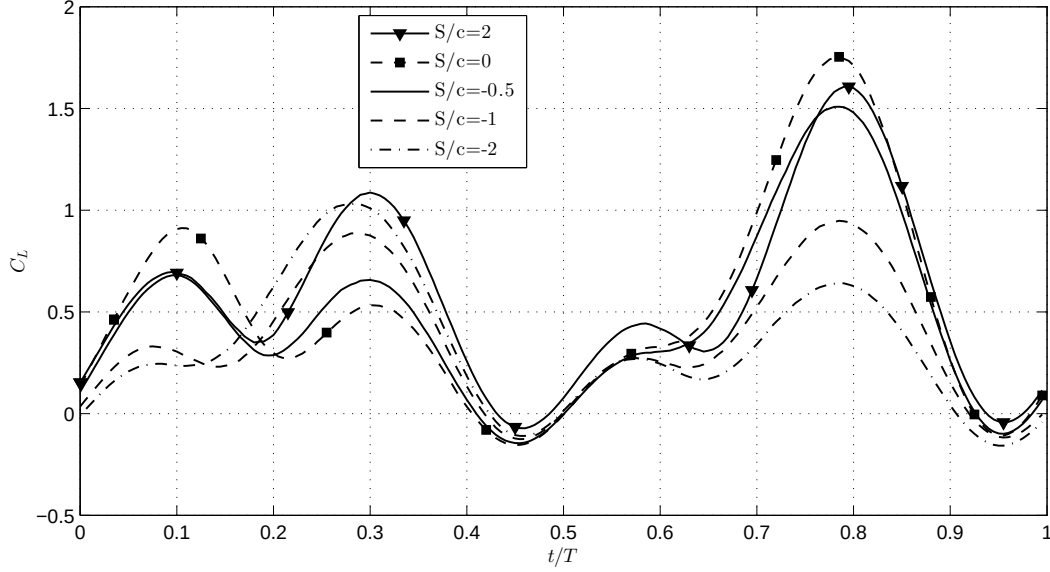


Figure 4.21: Time-dependent lift coefficient of the flapping elliptic airfoil over one period of flapping motion in the force-reduction regime ($D/c = 3$).

to a diminished second peak. When S/c decreases to -2 , the positive LEV from upstroke tends to slide down the wind side after stroke reversal, and the negative downstroke LEV remains attached to the airfoil (see Fig. 4.22(b)). Therefore, the first peak diminishes while the second peak recovers.

In addition to the change in the first and second peaks, a dramatic drop in the fourth peak is observed when S/c decreases from 0 to -2 , again, because the platform prevents the formation of a vortex pair during the second half of the upstroke (compare Figs. 4.22(a) and (b) at $t/T = 0.8$).

When D/c increases to 4 or 5, the flow enters the force-recovery regime. The mean lift and drag forces are close to their infinite-fluid levels. Compared to when $D/c = 3$, the time-dependent lift coefficient in the force-recovery regime shows very similar dependence on S/c , although the dependence is much weaker because of the increased vertical distance to the platform.

4.2.2.3 Efficiency of lift generation

We shall express the efficiency of lift generation in terms of the ratio $\overline{C_L}/\overline{C_P}$. The time-averaged power coefficient $\overline{C_P}$ is defined as

$$\overline{C_P} = \frac{\int_0^T [-F_x(t)U(t) - \tau(t)\Omega(t)] dt}{(1/2)T\rho U_{max}^3 c} \quad (4.3)$$

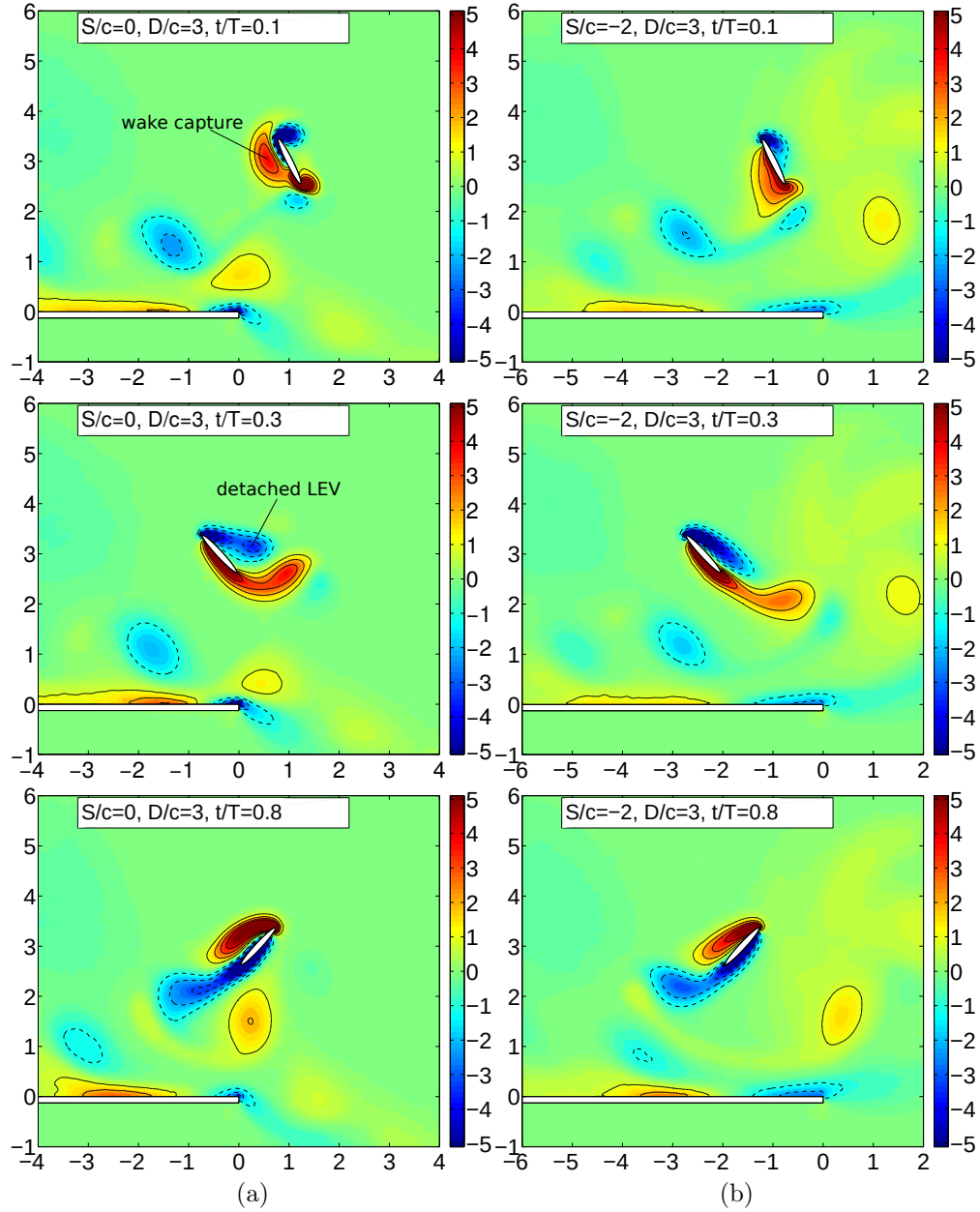


Figure 4.22: Non-dimensional vorticity fields ($\omega c/U_{max}$) for $S/c =$ (a) 0 and (b) -2 and $D/c = 3$ at $t/T = 0.1, 0.3$, and 0.8 .

where $\tau(t)$ and $\Omega(t) = \dot{\alpha}(t)$ are the instantaneous aerodynamic torque on the foil and the angular velocity of the foil, respectively. It should be noted that the power contribution from foil rotation is much smaller than that from the translational motion. The efficiencies of lift generation for various cases are shown in Fig. 4.23.

Despite the significant increase in lift in the force-enhancement regime ($D/c = 1$), the

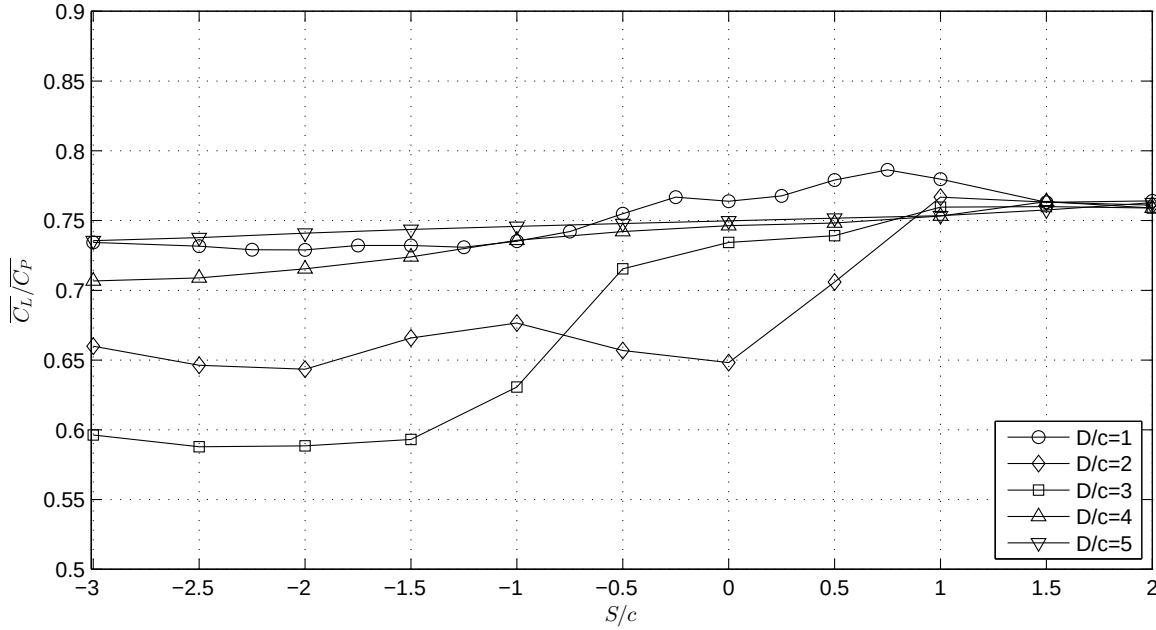


Figure 4.23: Efficiencies of lift generation in terms of the ratio $\overline{C_L}/\overline{C_P}$ for various values of S/c and D/c .

efficiency remains approximately constant for the whole range of $S/c \in [-3, 2]$ and stays within $\pm 5\%$ of the infinite-fluid efficiency of $\overline{C_L}/\overline{C_P} = 0.75$. This is because the increase in lift is accompanied by a roughly proportional increase in drag in the force-enhancement regime (see Fig. 4.8). In the force-reduction regime ($D/c = 2, 3$), however, a rather significant drop in efficiency is observed. When $D/c = 3$, the lowest efficiency of approximately 0.59 is computed for $S/c \leq -1.5$, representing a 21% drop from the infinite-fluid efficiency. In the force-recovery regime, the influence of the platform is weakened, and the lift generation efficiency gradually recovers to the infinite-fluid level.

4.2.2.4 Net lateral force under partial ground effect

When the foil is near the platform edge, it may experience a net cycle-average lateral force because of the asymmetric ground effect. To represent the lateral force, we introduce the lateral-force coefficient C_x :

$$C_x = \frac{F_x}{(1/2)\rho U_{max}^2 c} \quad (4.4)$$

Similar to the lift and drag coefficients, the time-averaged lateral-force coefficient $\overline{C_x}$ is also computed from the last five of the 15 periods of flapping motion simulated. The values of $\overline{C_x}$ for $D/c = 1$ to 4 are shown in Fig. 4.24.

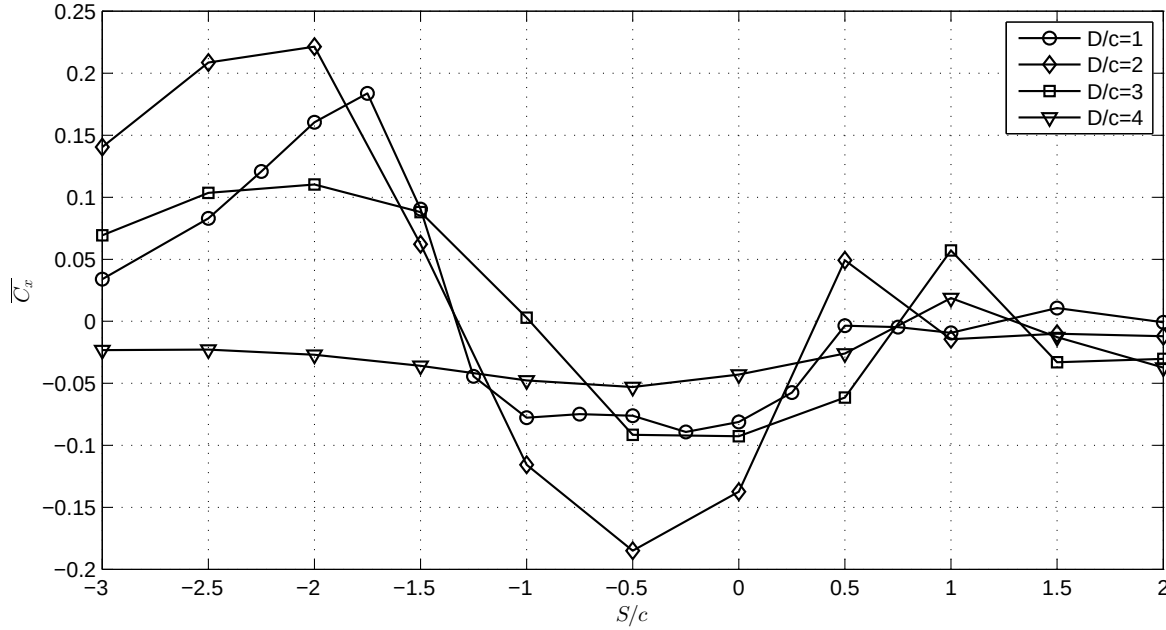


Figure 4.24: Time-averaged lateral-force coefficient $\overline{C_x}$ of a flapping elliptic airfoil in normal hovering mode near the edge of a platform.

When $\overline{C_x}$ is positive, the net lateral force, in the positive x -direction, is pushing the foil away from the platform. When $\overline{C_x}$ is negative, the net lateral force is pulling the foil towards the platform center. For all the cases shown in Fig. 4.24, the magnitude of $\overline{C_x}$ remains much smaller than the time-averaged drag coefficient $\overline{C_D}$ shown in Fig. 4.8(b), indicating cancellation between the lateral force during the upstroke and the downstroke. However, the net lateral force is still not negligible especially in the force-enhancement ($D/c = 1$) and force-reduction ($D/c = 2, 3$) regimes. Interestingly, the value of $\overline{C_x}$ shows the same overall trend in both regimes. When $S/c \geq 1.5$, the net lateral force is very close to zero, as the edge of the platform is still relatively far away. A net pull force is observed for S/c in the range of -1 to 0.25 , and a net push force is present for $S/c \leq -1.5$. When S/c decreases below -2 , the push force weakens as the foil starts to experience full ground effects. The strongest net pull and push forces are observed in the force-reduction regime when $D/c = 2$. In the force-recovery regime ($D/c \geq 4$), the effect of the ground weakens, and the net lateral force becomes negligible.

4.2.2.5 Effect of platform edge and the formation of reverse von Kármán vortex street

Under suitable conditions, the presence of the platform edge may cause the wake of the flapping airfoil to form a reverse von Kármán vortex street that extends well below the

stroke plane. In the range of S/c and D/c values we have covered, the formation of a vortex street is typically observed when $S/c \geq 0$. The vortex street is more prominent when $D/c \leq 3$ but still exists for $D/c = 4$ and 5. The formation of the vortex street is shown in Fig. 4.25 for several cases with $D/c = 1$. When $0 \leq S/c \leq 0.5$, the reverse von Kármán vortex street tends to curl towards the right and eventually roll into a positive vortex, whereas, if $S/c \geq 1$, the vortex street extends straight downward and eventually rolls into a negative vortex under the platform.

The vortex street is formed from the alternate shedding of positive downstroke TEVs and negative upstroke TEVs. In an infinite fluid where the platform edge is absent, the two TEVs tend to form closely packed vortex pairs. The large distance between consecutive vortex pairs prevents the formation of a reverse von Kármán vortex street (see Fig. 4.26). A properly placed platform edge, on the other hand, slightly impedes the downward motion of the shed vortices, reducing the distance between consecutive vortex pairs. As a result, we have a continuous train of approximately equally spaced vortices which form the vortex street (see Fig. 4.25).

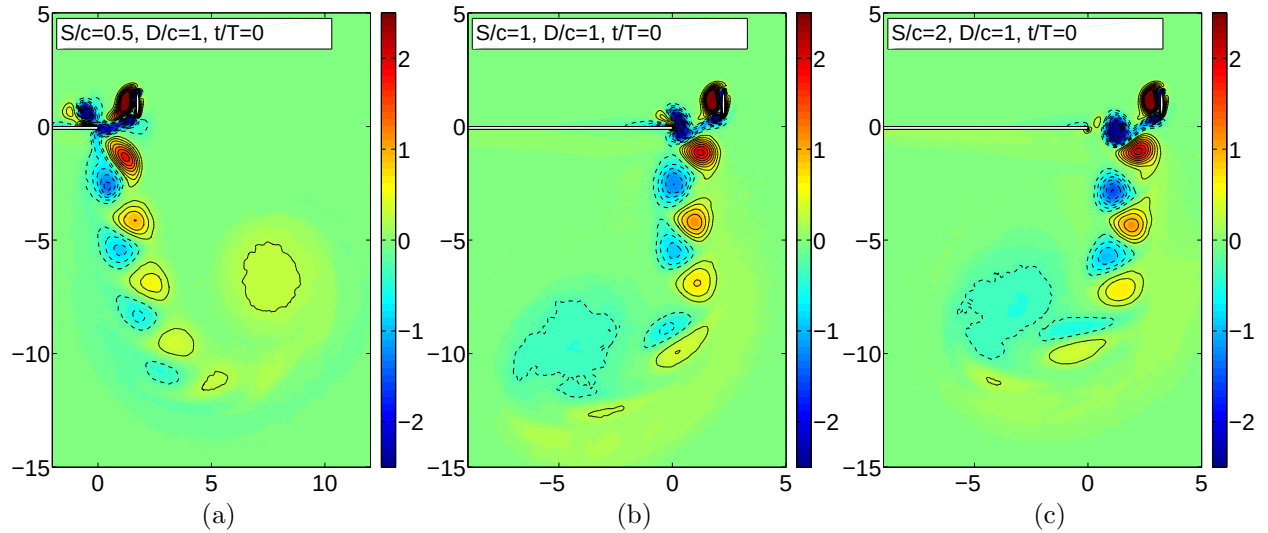


Figure 4.25: Non-dimensional vorticity fields ($\omega c/U_{max}$) of the wake of the flapping airfoil in normal hovering mode for $S/c =$ (a) 0.5, (b) 1, and (c) 2 and $D/c = 1$ at $t/T = 0$. Increment of contour level is 0.25 for the solid and dashed contour lines. The influence of the platform edge leads to the formation of a reverse von Kármán vortex street.

The reverse von Kármán vortex street induces a strong and focused downward jet that extends well below the stroke plane of the foil. This jet may present a navigational hazard for a second MAV flying under the platform edge.

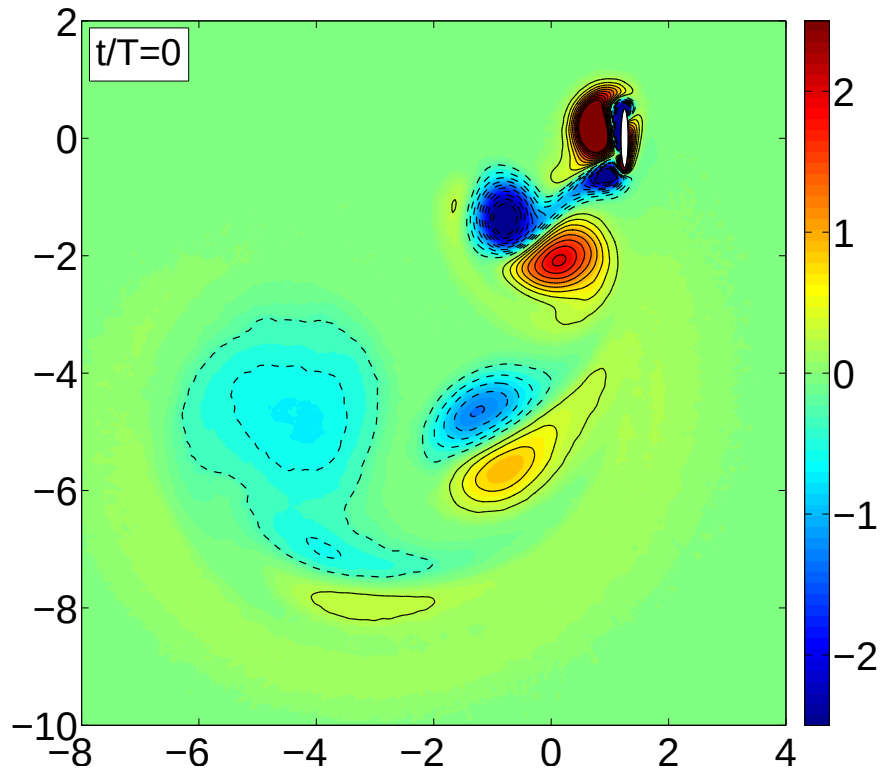


Figure 4.26: Non-dimensional vorticity field ($\omega c/U_{max}$) of the wake of the flapping airfoil in normal hovering mode in an infinite fluid at $t/T = 0$. Increment of contour level is 0.25 for the solid and dashed contour lines. The wake is dominated by close vortex pairs.

Chapter 5

Performance of a micro-scale Bach-type turbine

5.1 Problem description

A turbine with a Bach-type rotor of diameter D and radius R is subjected to a uniform current U as shown in Fig. 5.1. The fluid is viscous and incompressible with constant and uniform density ρ and kinematic viscosity ν . The no-slip and no-penetration boundary conditions are enforced on the turbine blade surface, and the uniform stream is undisturbed in the far field. The two-dimensional flow past the turbine is solved using DVM. The turbine rotates in the clockwise (CW) direction with angular velocity $\Omega < 0$. The rotor consists of two circular-arc sections of radius q subtending an angle φ on each end connected by a straight-line segment of length $2p$ in the center. The rotor has finite thickness d , and each tip of the rotor blade is closed by a semi-circular cap of diameter d . The part of the turbine moving in the direction of the current U is commonly referred to as the advancing blade while the part moving against the current is the returning blade. The rotor angular position θ , measured in the clockwise direction, is defined to be zero when the straight segment of the rotor aligns with the y -axis ($\Omega = -\dot{\theta}$). Over one period T , the turbine makes one full rotation (θ increases from 0 to 2π). In this study, the parameters used are $\varphi = 124^\circ$ and $p/q = 0.2$ [98]. The blade thickness is chosen to be $d/D = 1/80$. These parameters are summarized in Table 5.1. For convenience, we will actually be solving the equivalent problem where the turbine shaft is translating in the negative x -direction at a constant speed U in a calm fluid.

In the present work, both prescribed rotation and free rotation are considered. With prescribed rotation, the turbine turns at a constant and predefined angular velocity Ω . In this case, the flow depends on only two parameters: tip-speed ratio (λ) and the Reynolds number (Re). The conventional definition of the tip-speed ratio and the Reynolds number are used: $\lambda = -R\Omega/U$ and $Re = UD/\nu$. The Reynolds number $Re = 1,500$ is used for all the results presented in this chapter. This choice of Reynolds number is significantly lower than what is typically used in experiments and mesh-based numerical computations that

Table 5.1: Turbine blade geometric parameters

Parameters	Values
φ	124°
p/q	0.2
d/D	1/80

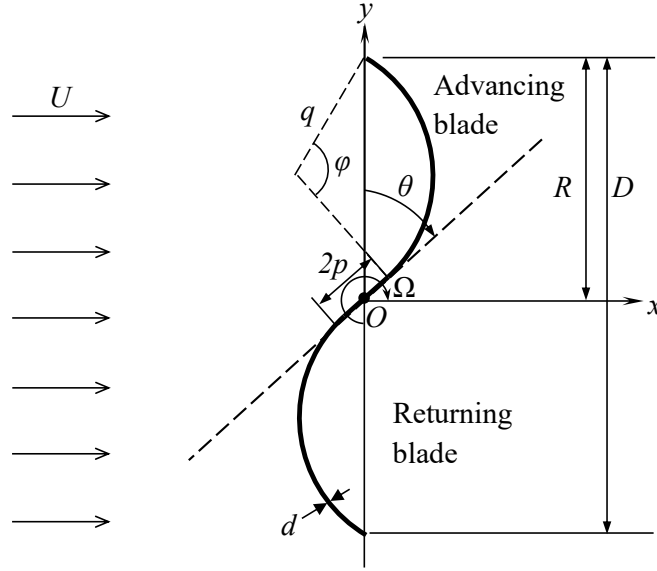


Figure 5.1: Flow past a rotating Bach-type vertical-axis turbine. The turbine rotates in the clockwise direction about point O . The cross-sectional area of the rotor is $A \approx 2d(p + \varphi q)$.

rely on some form of turbulence model. However, scientific significance aside, computation at this low Reynolds number is still of practical interests because it allows one to evaluate whether Bach-type turbines can be effectively used in micro-scale power generation.

In the case of free rotation, $\Omega(t)$ is time-dependent and is solved by integrating the equation of motion of the turbine:

$$\tau - B_g \Omega = I \dot{\Omega} \quad (5.1)$$

where τ , B_g , and I are the torque exerted on the turbine by the fluid, the generator damping, and the mass moment of inertia of the blade about the axis of rotation, respectively. The initial turbine position is simply $\theta(t = 0) = 0$. A non-zero initial angular velocity $\Omega(t = 0)$ is used for each run in order to reach the final periodic steady state faster. Since the problem is two-dimensional, τ , B_g , and I are all given in terms of per unit height (out of the plane) of the turbine.

For prescribed rotation, the power P per unit height of the turbine is given by

$$P = \Omega \tau \quad (5.2)$$

For free rotation, the power can be computed from the generator damping B_g :

$$P = B_g \Omega^2 \quad (5.3)$$

The torque and power coefficients, C_T and C_P , are defined as:

$$C_T = \frac{-\tau}{\frac{1}{2}\rho U^2 D R} \quad C_P = \frac{P}{\frac{1}{2}\rho U^3 D} \quad (5.4)$$

With the negative sign, C_T is positive when the torque is in the direction of rotation (CW) and negative when the torque opposes rotation. Also, the horizontal and vertical force coefficients, C_x and C_y , are defined as

$$C_x = \frac{F_x}{\frac{1}{2}\rho U^2 D} \quad C_y = \frac{F_y}{\frac{1}{2}\rho U^2 D} \quad (5.5)$$

where F_x and F_y are the horizontal and vertical components of the hydrodynamic force per unit height of the turbine.

5.2 Numerical results

The contour of the Bach-type turbine is discretized into 512 similar-sized linear panels, and a time step of $U\Delta t/D = 0.01$ is used. The vortex blobs in the wake within 20 turbine diameters to the turbine's axis of rotation are kept. After a vortex blob moves past this threshold, its circulation starts to decay linearly with time to zero over a time duration of T_{dc} . The vortex blob is removed once its circulation reaches zero. The decay time is chosen to be $UT_{dc}/D = 1$. This configuration, which caps the maximum number of vortex blobs in the flow to approximately 1.3 million, is chosen to achieve the best balance between accuracy and computation time.

Compared to previous DVM simulations of a Savonius or Bach-type turbine (see e.g. [66]), a much higher number of body panels are used in this work. This is because the current simulation is fully viscous, and a large number of overlapping vortex blobs with small core radius are needed to fully resolve the viscous and vortical flow near the turbine blade. Since each body panel only generates one vortex blob per time step in the current DVM formulation, more body panels are needed to generate sufficient number of vortex blobs.

The time-step convergence check using prescribed rotation with $\lambda = 1$ is shown in Fig. 5.2. The time-dependent torque coefficient during the first six revolutions is presented. Visually, the results obtained with $U\Delta t/D = 0.01$ and $U\Delta t/D = 0.005$ are very consistent. During the sixth period $t/T \in [5, 6]$, the mean difference in torque coefficient calculated from the two time-step sizes is about 1% of the maximum torque coefficient. This level of convergence is adequate for the ensuing discussions.

Each run of computation is carried out until the torque coefficient and the angular velocity, if free rotation is considered, become approximately periodic for several turbine revolutions. The simulations typically end at $Ut/D = 60$ to 100. For prescribed rotation, the solution from the last three revolutions of each run is used to calculate the time-averaged torque and power coefficients, $\overline{C_T}$ and $\overline{C_P}$. The time-dependent torque and power coefficients

C_T and C_P over one period ($t/T \in [0, 1]$) are also obtained by averaging the instantaneous torque and power coefficients at the same time instant within each of the last three revolutions. For free rotation, the time-averaged and time-dependent torque, power, and force coefficients and the turbine speed in terms of λ are calculated in the same way except that the solution from the last five revolutions is used.

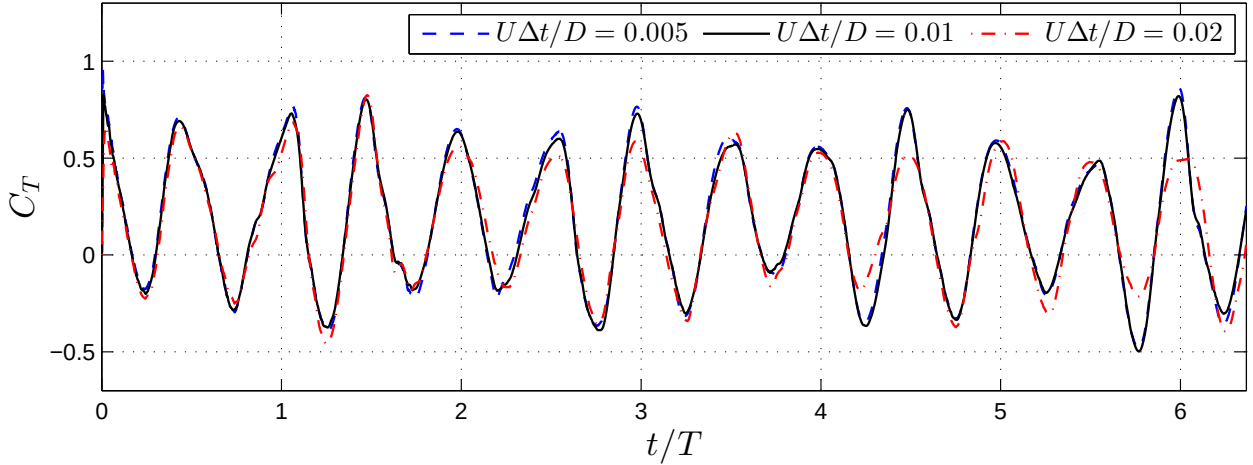


Figure 5.2: Time-step convergence check using prescribed rotation with $\lambda = 1$.

5.2.1 Prescribed rotation

5.2.1.1 Time-averaged power and torque characteristics with prescribed rotation

The performance of the micro-scale turbine under prescribed rotation with constant tip-speed ratio is investigated first. In Fig. 5.3, the time-averaged torque and power coefficients, $\overline{C_T}$ and $\overline{C_P}$, for prescribed rotation are shown for λ in the range of 0.3 to 1.2.

Overall, $\overline{C_T}$ increases with decreasing λ . A rather sharp increase in mean torque is observed when λ decreases from 0.75 to 0.725. At this low Reynolds number of 1,500, the maximum mean power coefficient of approximately 0.18 is achieved at two distinct λ values, 0.725 and 0.9. The high power coefficient at $\lambda = 0.725$ is related to the sharp increase in $\overline{C_T}$ observed at the same tip-speed ratio. The mechanism behind the high power coefficient at $\lambda = 0.725$ is discussed later on in Sec. 5.2.1.4. Rapid decrease in power coefficient is observed when λ decreases below 0.5 or increases beyond 1.

It is of interest to note that the DVM results from [66] show the power and torque coefficients of a Savonius turbine increasing monotonically and almost linearly with the tip-speed ratio. This is very different from the current DVM results. The difference is, however, expected because the DVM simulation of [66] is based on inviscid potential flow theory for very high Reynolds numbers with boundary-layer separation limited to the tips of the turbine blade. Therefore, unlike the fully viscous DVM simulation for lower Reynolds numbers used

5.2. Numerical results

in this study, the viscous dissipation at high λ is not captured in the method of [66], resulting in continued increase of $\overline{C_P}$ and $\overline{C_T}$ with λ .

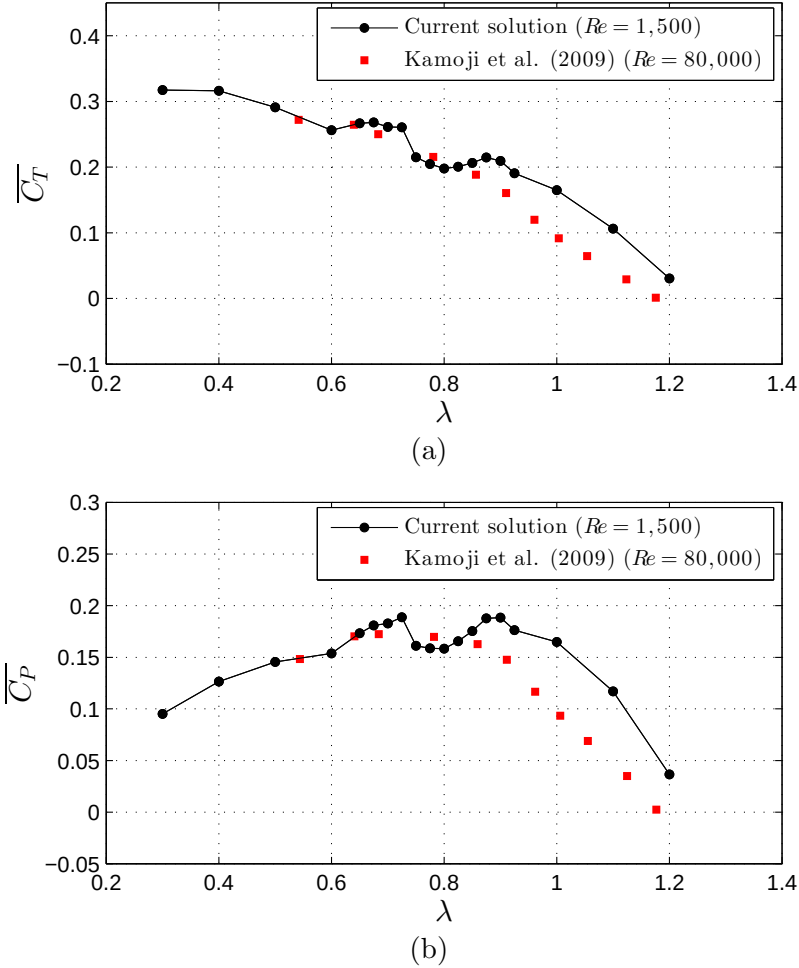


Figure 5.3: Time-averaged (a) torque and (b) power coefficients, $\overline{C_T}$ and $\overline{C_P}$, of the Bach-type turbine with $\varphi = 124^\circ$ and $p/q = 0.2$. The current numerical DVM solutions at $Re = 1,500$ is compared to the experimental results at $Re = 80,000$ [98].

The numerically determined mean torque and power coefficients at $Re = 1,500$ are also compared to the experimental results of a similar Bach-type turbine with the same p/q and φ values at $Re = 8 \times 10^4$ from [98]. The experiments were carried out in an open jet wind tunnel, and the turbine had end plates and a relatively low aspect ratio of 0.7. Despite the order of magnitude difference in Re , the current numerical results bear strong resemblance to the experimental results. The maximum $\overline{C_P}$ of approximately 0.18 at $Re = 1,500$ predicted by the DVM simulation is comparable to the optimal power coefficient reported in [98] for the same geometry. This result indicates that drag-based turbines are suitable for micro-

scale power generation because the adverse effect of miniaturization (low Re) is minimal. It is, however, difficult to identify the exact cause of the quantitative differences between the current numerical results and the experimental results of [98]. The discrepancy may be attributed to the difference in Reynolds number and/or the difference in the methods of investigation used in the two works. For instance, the DVM numerical results predict increased torque and power for $\lambda > 0.9$ at $Re = 1,500$. This may simply be a consequence of the two-dimensional flow assumption used in the DVM simulation as opposed to the difference in Reynolds numbers.

The observation that the performances of drag-based turbines are relatively insensitive to Reynolds number is also supported by several experimental studies performed at higher values of Re [55, 99]. In [99], a Savonius wind rotor intended to power a street lamp was tested in a wind tunnel at Re in the range of 2×10^5 and 3.3×10^5 ; the power and torque coefficients $\overline{C_P}$ and $\overline{C_T}$ showed no dependence on Reynolds number in this range of Re . In [55], the variation of maximum $\overline{C_P}$ as a function of Re in the lower range of 6.0×10^4 to 1.5×10^5 is shown to be around 10% for several different designs of drag-based turbines. Also, the power coefficient does not increase monotonically with Reynolds number; the optimal power coefficient is, in fact, observed at some intermediate values of Re . The numerical computations presented in the current work indicate that the range of Re insensitivity extends down to micro-scale turbines with Re on the order of 10^3 .

5.2.1.2 Instantaneous torque and forces on the turbine under prescribed rotation

The flow past a Bach-type flow turbine is inherently unsteady. The fluid forces and torque on the turbine are strongly dependent on time. As an example, the instantaneous torque coefficient C_T over one period of rotation for $\lambda = 0.9$ is shown in Fig. 5.4. Minimum torque coefficient is observed around $t/T = 0.23$ and 0.71 . At these points, C_T is negative, and the torque impedes blade rotation. Maximum torque coefficient is observed around $t/T = 0.52$ and 0.96 . The horizontal and vertical force coefficients, C_x and C_y , also show large fluctuations over one period. In fact, C_y is always positive, indicating that the turbine is experiencing a lateral force in the positive y -direction at all time.

To identify the cause of the high and low instantaneous torques during one revolution, the non-dimensional vorticity field and the non-dimensional relative pressure $p^* = p/(0.5\rho U^2)$ on the blade-surface are shown in Fig. 5.5 for two time instants: $t/T = 0.233$ and $t/T = 0.519$, corresponding to the minimum and the maximum torque coefficients, respectively. The pressure at the edge of the blade marked by a “●” in the insets of Fig. 5.5(b) and (d) is used as the reference pressure (zero relative pressure). At $t/T = 0.233$, even though the bucket (circular arc) of the advancing blade is fully facing the incoming current, the pressure difference between the wind side and the lee side of the blade is fairly small (see Fig. 5.5(b)). As a result, very little torque is being generated by the advancing blade. Meanwhile, the returning blade is experiencing a strong stagnation pressure on the center of the wind side. The net torque on the turbine opposes its rotation at this time instant.

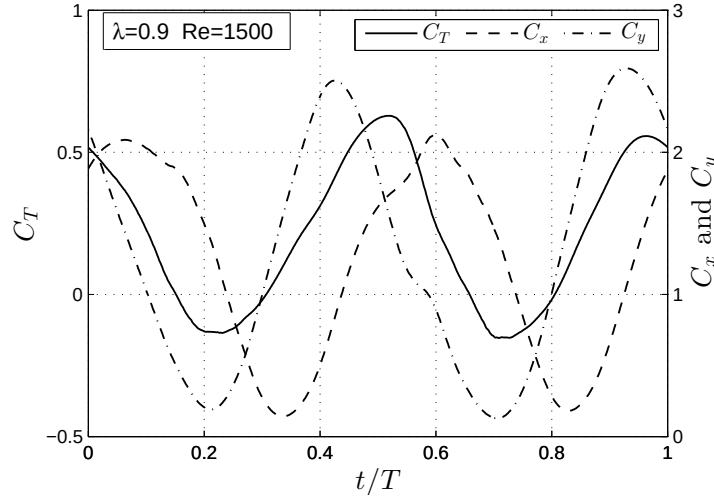


Figure 5.4: Instantaneous torque coefficient C_T and the horizontal and vertical force coefficients C_x and C_y over one period of rotation for $\lambda = 0.9$ at $Re = 1,500$.

In comparison, at $t/T = 0.519$, the lee side of the advancing blade is experiencing a much lower pressure than the wind side (see Fig. 5.5(d)), leading to a strong “lift” force in the positive y -direction on the advancing blade and a high torque on the turbine. In this orientation, the advancing blade is acting like a thin lift-generating foil with a camber. The boundary layer on the suction side remains attached over roughly two thirds of the advancing blade, contributing to the lift force (see Fig. 5.5(c)). In this respect, the Bach-type turbine is also partly lift driven even though it is customarily considered to be a drag-based turbine. This observation is consistent with the remark of [65] for a static turbine at very high Reynolds numbers, $Re \rightarrow \infty$, and indicates that the same torque generating mechanism is present at the two extremes of low and high Reynolds numbers.

5.2.1.3 Evolution of vortical flow with prescribed rotation

At low Reynolds numbers, the flow is dominated by large and coherent vortices that represent areas of concentrated flow energy. These vortices affect the performance of the turbine. Therefore, a detailed understanding of the flow field and the interaction between the turbine blade and the vortical structures is highly pertinent to the design, optimization, and operation of micro-scale turbines. Using the current viscous DVM approach, the unsteady vortical flow structures are directly resolved to provide a detailed description of the flow evolution.

Description of major flow features The non-dimensional vorticity field of the flow around the Bach-type turbine over one revolution is shown in Fig. 5.6. At $t/T = 0$, the angular position of the turbine is $\theta = 0$. The tip-speed ratio is 0.9 which corresponds to maximum power extraction. At $t/T = 0.006$, the boundary layer on the lee side of

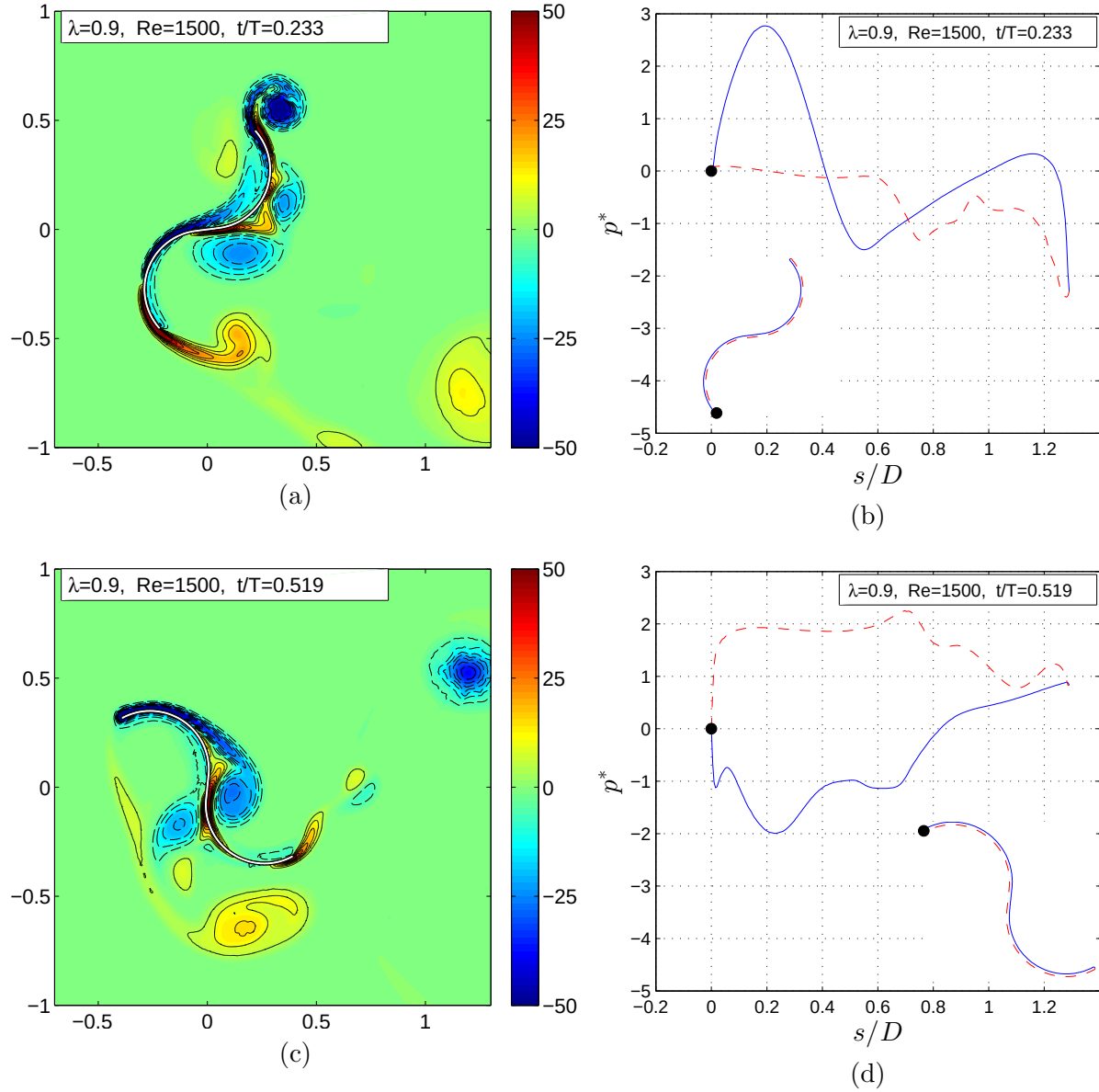


Figure 5.5: Non-dimensional vorticity ($\omega D/U$) contour plots of the flow and non-dimensional relative pressure p^* on the blade surface at $t/T = 0.233$ and $t/T = 0.519$ for $\lambda = 0.9$. Vorticity contour level increment is 5. Minimum torque is observed around $t/T = 0.233$ shown in (a) and (b). Maximum torque is observed around $t/T = 0.519$ shown in (c) and (d). The non-dimensional relative pressure p^* is shown as a function of the non-dimensional arclength s/D along the blade measured from the edge marked by a “•” in the insets (zero relative pressure at this point).

the advancing blade separates (labeled A1), and a negative clockwise (CW) vortex forms on the lee side of the returning blade (labeled B1). When $t/T = 0.121$, flow separation starts to occur at the edge of the advancing blade, leading to the formation of another CW vortex (labeled C1). A positive counterclockwise (CCW) vortex is shed from the edge of the returning blade at $t/T = 0.235$ (labeled D1), and a vortex sheet connects the blade edge and vortex D1. At the same time, the separated shear layer A1 rolls into a CW vortex while vortices B1 and C1 continue to grow in size. At this time, the stagnation point is located at the center of the returning blade. Subsequently at $t/T = 0.350$, vortex C1 detaches from the edge of the advancing blade, and vortex B1 forms a vortex pair with the CCW vortex generated on the blade surface. After approximately one half rotation ($t/T = 0.465$), vortex A1 is shed from the tip of the now returning blade, and vortex B1 is moved to the wind side of the turbine. The CW vortex on the wind side of the turbine at $t/T = 0.006$ is formed in the same manner during the previous cycle. Also at $t/T = 0.465$, the boundary layer again separates from the lee side of the advancing blade (labeled A2), and a CW vortex B2 forms behind the returning blade, repeating the same creation process of A1 and B1 observed previously at $t/T = 0.006$. The vortex sheet connecting vortex D1 and the blade edge is broken up by the returning blade at $t/T = 0.579$, and a new separation vortex C2 is forming on the tip of the advancing blade. The same processes can also be observed at $t/T = 0.121$. At $t/T = 0.694$, vortex B1 merges with the boundary layer on the wind side of the returning blade, and flow separation occurs again at the tip of the returning blade, leading to the formation of the D2 vortex. For this particular case ($\lambda = 0.9$), vortex D1 is positioned very close to the edge of the returning blade at $t/T = 0.694$. As a result, vortices D1 and D2 tend to merge together (see $t/T = 0.923$).

Based on the above observation, the main vortical structures are created from two different occasions. First, strong vortex shedding from the tips of the turbine blade happens when the turbine is fully facing the incident stream (see e.g. $t/T = 0.235$ and 0.694 of Fig. 5.6). Second, vortices are created from the boundary-layer separation from the lee side of the advancing blade (see e.g. $t/T = 0.006$ and 0.465 of Fig. 5.6). The inviscid DVM can only capture the former (see e.g. [61,64]), and a fully viscous DVM is needed to capture both vortex creation processes.

Effect of tip-speed ratio on flow evolution In order to demonstrate the effect of λ on the evolution of the vortical structures in the flow, the vorticity field near the turbine are shown for $\lambda = 0.5$ and 1.2 in Fig. 5.7 and Fig. 5.8, respectively.

The evolution of the vortices A1, A2, B1 and B2 are strongly affected by λ . When $\lambda = 0.5$, the relatively slow angular speed causes vortex B1 to be convected further downstream relative to the turbine blade at $t/T = 0.305$ (see Fig. 5.7). At this time instant, vortex A1 already disappeared because of both dissipation and merging with vortex B1. Subsequently, vortex B1 is split into two parts, B1a and B1b, by the turbine blade at $t/T = 0.417$. Vortex B1b is immediately shed from the edge of the blade, while B1a is shed from the edge of the returning blade at a later time $t/T = 0.528$. Different from the case with $\lambda = 0.9$ shown

5.2. Numerical results

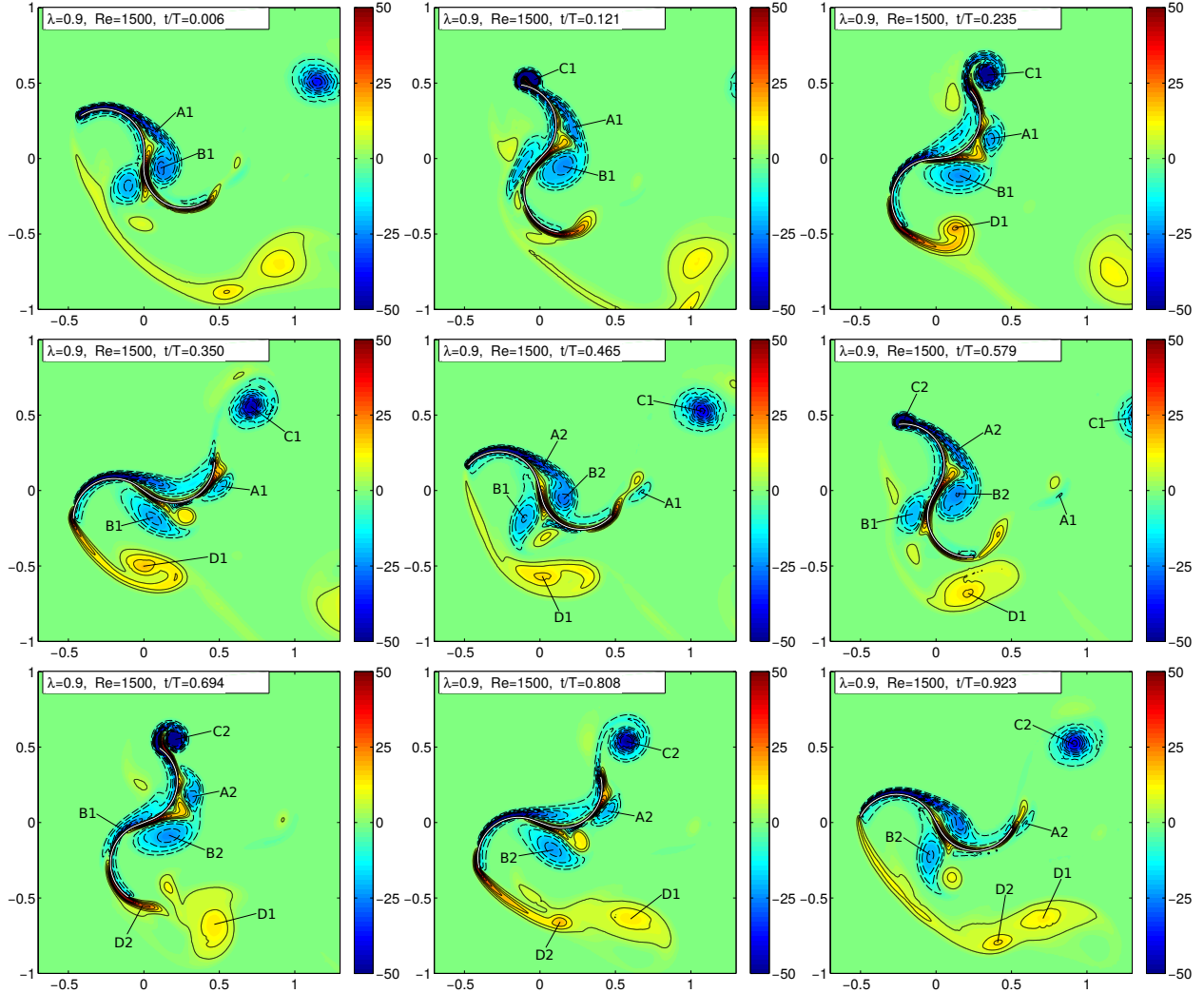


Figure 5.6: Non-dimensional vorticity ($\omega D/U$) contour plots of the flow around the Bach-type turbine under prescribed rotation with $\lambda = 0.9$ and $Re = 1,500$. Horizontal scale is x/D , and vertical scale is y/D . Solid contour lines represent positive vorticity; dashed contour lines represent negative vorticity. Contour level increment is 5.

in Fig. 5.6, vortex B1 is never moved to the wind side of the turbine. The evolution of the B2 vortex is very similar to that of B1 during the second half of the rotation. For the case $\lambda = 1.2$ shown in Fig. 5.8, both vortices A1 and B1 are moved to the wind side at $t/T = 0.529$ because of the fast turbine rotation. Subsequently at $t/T = 0.720$, vortex A1 merges with the boundary layer on the wind side of the returning blade while B1 is moved further upward to the wind side of the advancing blade. Vortex B1 is eventually shed from the edge of the advancing blade. Again, vortices A2 and B2 evolve in a fashion very similar

5.2. Numerical results

to A1 and B1.

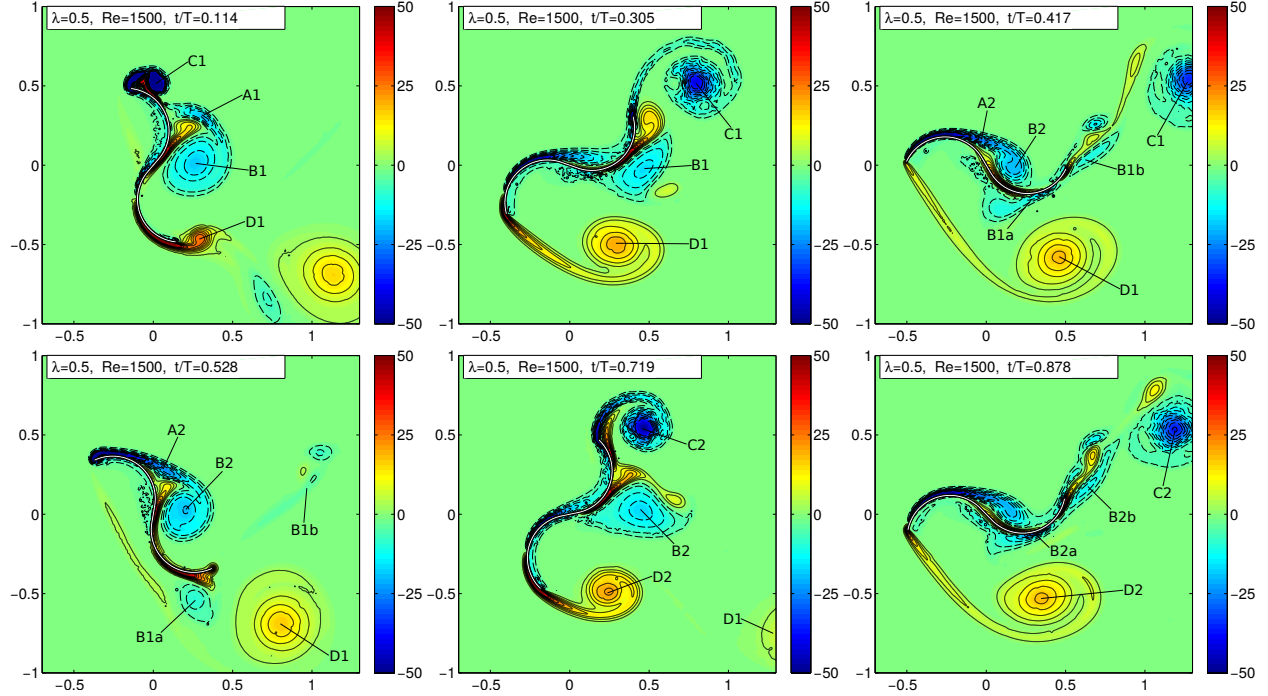


Figure 5.7: Non-dimensional vorticity ($\omega D/U$) contour plots of the flow around the Bach-type turbine under prescribed rotation operating at $\lambda = 0.5$ with $Re = 1,500$. Horizontal scale is x/D , and vertical scale is y/D . Solid contour lines represent positive vorticity; dashed contour lines represent negative vorticity. Contour level increment is 3.5.

The value of λ also has significant effects on the positions and sizes of the C1, C2, D1 and D2 vortices. For low tip-speed ratios such as $\lambda = 0.5$ shown in Fig. 5.7, the turbine rotates much slower for a given free-stream velocity; therefore, the vorticity shed from the tips of the turbine blade is able to accumulate over a longer period of time, leading to the shedding of larger C1, C2, D1, and D2 vortices compared with the case of high λ shown in Fig. 5.8. These four vortices are also convected further downstream after one half turbine revolution when the tip-speed ratio is low. For instance, vortex D1 is some distance away from the tip of the returning blade at $t/T = 0.719$ in Fig. 5.7 and does not influence the formation of the D2 vortex. This downstream convection leads to very similar vorticity fields during the first and second half of a turbine revolution. In comparison, the D1 vortex is very close to the tip of the returning blade at $t/T = 0.72$ when $\lambda = 1.2$ as shown in Fig. 5.8. The presence of D1 leads to the fracturing of vortex D2 and partial merging between D1 and D2 at $t/T = 0.872$. Because of the presence of D1, it can be seen that the vorticity field is not identical during the first and second half of one turbine revolution when $\lambda = 1.2$.

Overall, the vortices B1 and B2 tend to be split and shed from the tip of the returning

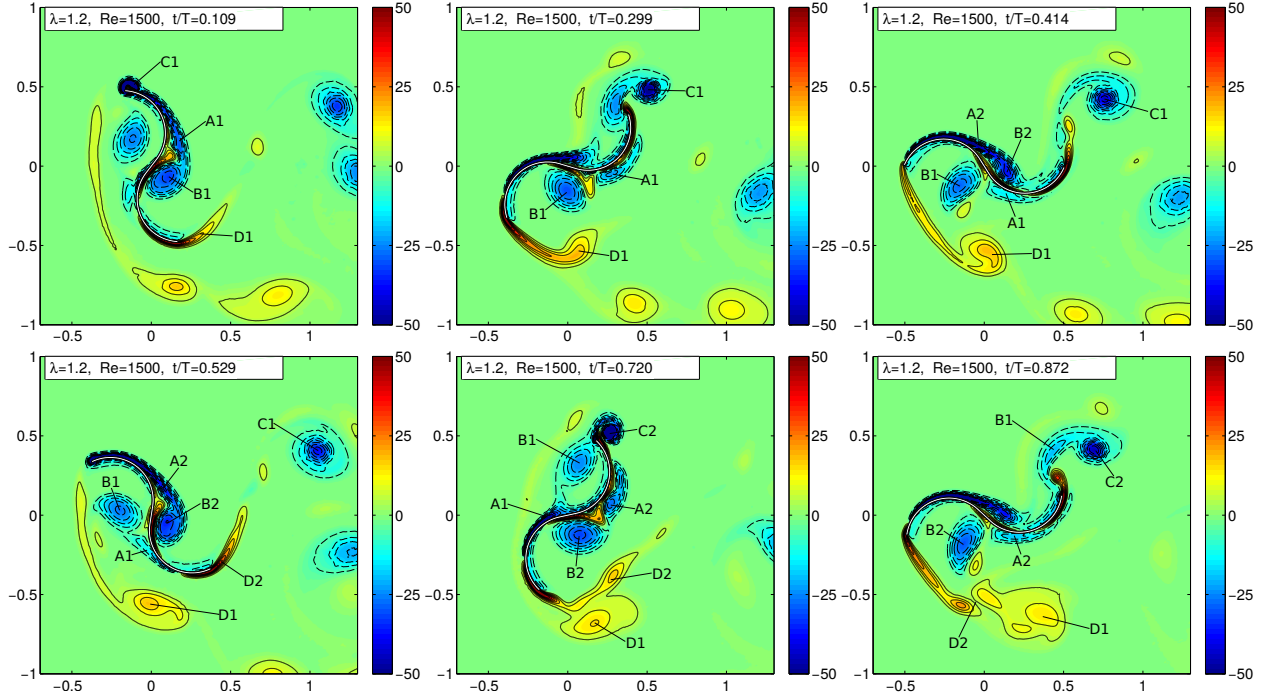


Figure 5.8: Non-dimensional vorticity ($\omega D/U$) contour plots of the flow around the Bach-type turbine under prescribed rotation operating at $\lambda = 1.2$ with $Re = 1,500$. Horizontal scale is x/D , and vertical scale is y/D . Solid contour lines represent positive vorticity; dashed contour lines represent negative vorticity. Contour level increment is 5.

blade when the tip-speed ratio is low. When tip-speed ratio is high, A1, A2, B1, and B2 are transported to the wind side of the turbine by the rapid rotation of the blade with B1 and B2 shed from the tip of the advancing blade. Lastly, as the tip-speed ratio decreases, stronger C1, C2, D1, and D2 vortices tend to form and are convected further downstream. As shown in Sec. 5.2.1.4, a certain range of λ will lead to an appropriate size and position of vortex D2 such that beneficial turbine-wake interaction can occur.

5.2.1.4 Discovery of the wake-capture mechanism

In Sec. 5.2.1.1, a second peak in $\overline{C_P}$ at $\lambda = 0.725$ is observed. To investigate the cause of this peak, the two cases of $\lambda = 0.725$ and $\lambda = 0.75$ are compared. As shown in Fig. 5.9, the peak in $\overline{C_P}$ at $\lambda = 0.725$ is caused by a drastic increase in torque at $t/T = 0.98$.

The non-dimensional relative pressure p^* on the turbine blade at $t/T = 0.98$ is shown in Fig. 5.10 for both $\lambda = 0.725$ and $\lambda = 0.75$. For the two tip-speed ratios, the values of p^* are almost equal everywhere except on the wind side of the returning blade where p^* for $\lambda = 0.725$ is drastically reduced. In fact, for $\lambda = 0.725$, the non-dimensional pressure on the wind side is lower than that on the lee side, meaning the returning blade is also generating

torque in the direction of rotation.

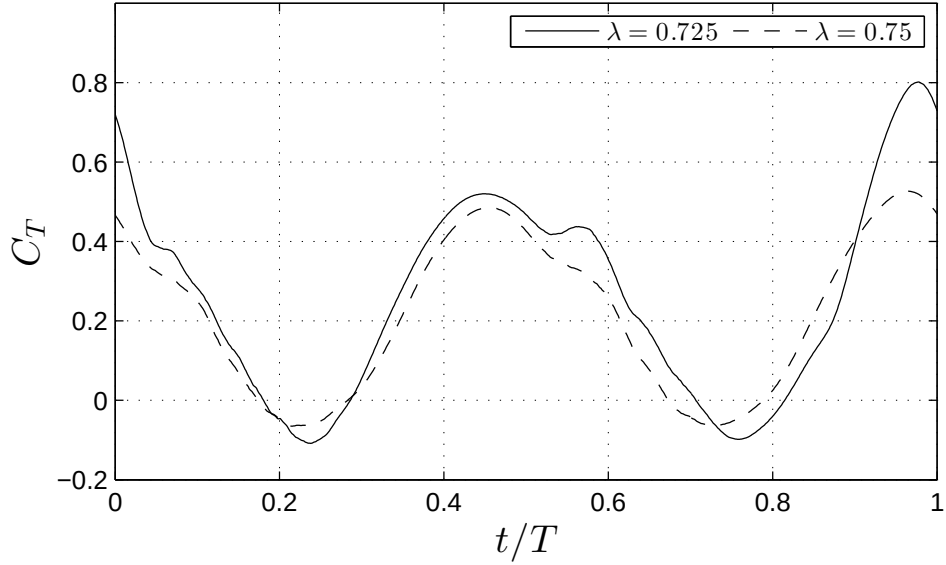


Figure 5.9: Torque coefficient over one period of rotation of the Bach-type turbine operating at $\lambda = 0.725$ and 0.75 .

The vorticity fields at $t/T = 0.98$ for $\lambda = 0.725$ and 0.75 are shown in Fig. 5.11. In Fig. 5.11(a) ($\lambda = 0.725$), the wind side of the returning blade hits the strong CCW vortex D2 shed from the edge of the now advancing blade. As a result, the returning blade experiences drastically reduced pressure where it contacts the CCW vortex. The region of contact corresponds well with the low-pressure region shown in Fig. 5.10. In contrast, the D2 vortex shed by the advancing blade misses the returning blade when $\lambda = 0.75$ (see Fig. 5.11(b)), and Fig. 5.10 shows no reduction in pressure on the wind side of the returning blade. This torque-enhancing phenomenon observed when $\lambda = 0.725$ is essentially a wake-capture effect. The turbine is able to recover energy previously lost to the wake under this special condition. This wake-capture mechanism is also observed for $\lambda = 0.675$ to 0.725 , leading to increased mean torque and power coefficients for this range of λ .

To the best of our knowledge, this beneficial wake-capture mechanism has not been observed previously for a Savonius or Bach-type turbine. The discovery of this mechanism demonstrates the important effects of the evolution of coherent vortical flow structures on the performance of low Re micro-scale turbines. Therefore, many of the performance-improving techniques designed for higher Reynolds numbers, such as rotating deflectable blades [100] and shielding plates [101], need to be adjusted or redesigned to accommodate or even utilize the concentrated flow energy carried by the large vortical structures in low Re flow. This is an area for potential future research.

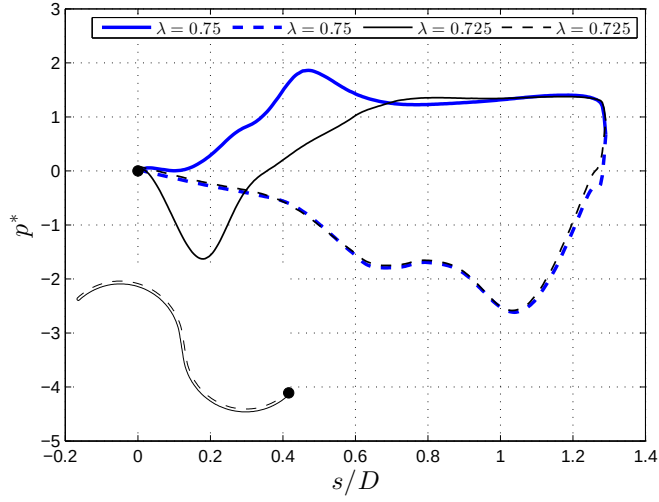


Figure 5.10: Non-dimensional relative pressure p^* on the turbine blade at $t/T = 0.98$ for $\lambda = 0.725$ and 0.75 . The value of p^* is plotted against the non-dimensional arclength s/D along the blade measured from the edge marked by a “•” in the inset (zero relative pressure at this point).

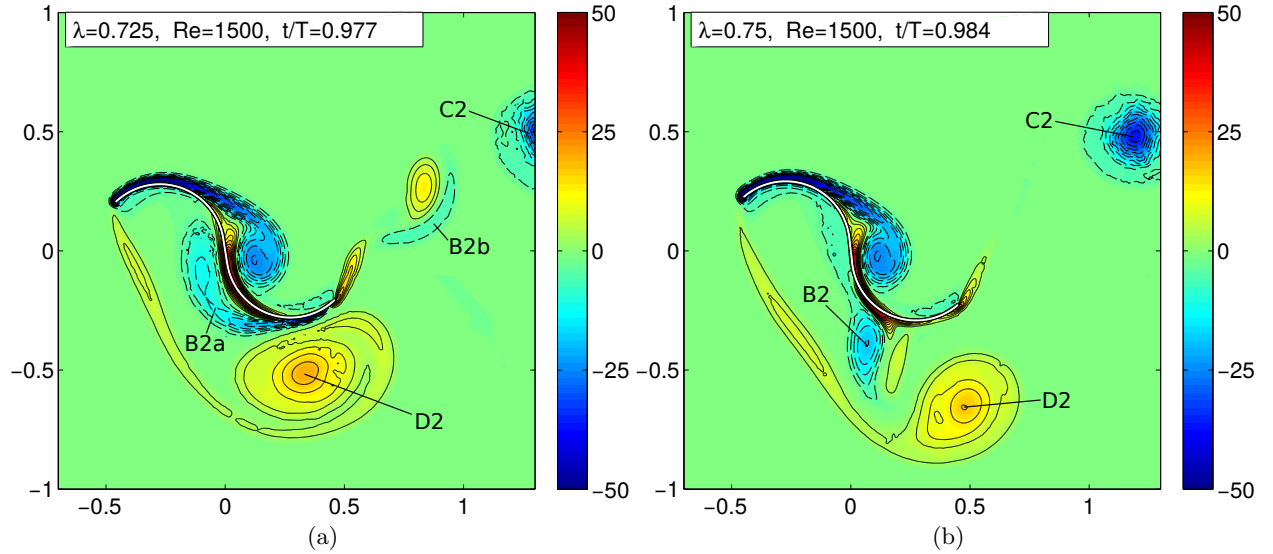


Figure 5.11: Non-dimensional vorticity ($\omega D/U$) contour plots of the flow around the Bach-type turbine in prescribed rotation at approximately $t/T = 0.98$ for $\lambda =$ (a) 0.725 and (b) 0.75 . Horizontal scale is x/D , and vertical scale is y/D . Solid contour lines represent positive vorticity; dashed contour lines represent negative vorticity. Contour level increment is 3.5 .

5.2.2 Free rotation

5.2.2.1 Power and torque characteristics with free rotation

In this section, the turbine is allowed to rotate freely according to Eq. (5.1). When the turbine is operating in air, the angular velocity of the turbine is approximately constant because of the large density difference between the fluid and the turbine [102]. The time history of the turbine torque in free rotation (variable λ) can be very similar to that in prescribed rotation (constant λ) [103]. In water, however, the relatively high density of the fluid can lead to significant fluctuation in angular speed over one period of rotation. This is especially true for micro-scale flow turbines because of the low inertia of the generator. Therefore, it is of great importance to investigate quantitatively the degradation of performance caused by large angular velocity fluctuation.

For the following analysis, the non-dimensional mass moment of inertia of the Bach-type turbine about its axis of rotation is $I^* = I/(\rho D^4) = 0.01323$. The inertia is calculated from the shape and thickness of the turbine blade section assuming a density ratio of $\rho_t/\rho = 8.05$ where ρ_t is the density of the turbine. This density ratio, which corresponds to a solid steel rotor in water, leads to highly unsteady rotation and represents a worst case scenario in terms of power generation. With free rotation, the speed of the turbine is controlled by the generator damping B_g . In subsequent results, the non-dimensional generator damping $B_g^* = B_g/(0.5\rho U D R^2)$ is used.

Figure 5.12 shows $\overline{C_T}$ and $\overline{C_P}$ of the turbine in free rotation as a function of mean tip-speed ratio $\bar{\lambda}$. Different from prescribed rotation, $\overline{C_T}$ for free rotation does not always increase with decreasing $\bar{\lambda}$. The value of $\overline{C_T}$ reaches a maximum of approximately 0.18 for $\bar{\lambda} \approx 0.88$. The free-rotation $\overline{C_P}$ is overall reduced compared to that of prescribed rotation especially for $\bar{\lambda}$ less than 0.95. When B_g^* is approximately 0.176, the maximum free-rotation $\overline{C_P}$ of 0.16 is achieved, representing an 11% decrease from that of prescribed rotation. To better illustrate the effect of generator damping, $\bar{\lambda}$, $\overline{C_T}$, and $\overline{C_P}$ are shown as functions of B_g^* in Fig. 5.13. Overall, increasing B_g^* leads to decreasing $\bar{\lambda}$ as expected. Both $\overline{C_T}$ and $\overline{C_P}$ reach their respective maximum values when B_g^* is between 0.17 and 0.2.

As an example, the time-dependent λ and C_T are shown in Fig. 5.14(a) for $B_g^* = 0.176$, the case with maximum $\overline{C_P}$. The instantaneous tip-speed ratio λ varies from 0.75 to 1.2. The amplitude of the fluctuation in λ is about 24% of $\bar{\lambda}$ in this case. However, because the angular speed is allowed to vary freely, the fluctuation in torque over one period of rotation is significantly reduced compared to prescribed rotation. For this case, the torque coefficient is between 0.08 and 0.28. In comparison, Fig. 5.4 shows that C_T varies from -0.13 to 0.63 when a prescribed $\lambda = 0.9$ is used. Similar to prescribed rotation, maximum torque coefficient is observed when θ is near 0 or π . The position θ of maximum λ lags behind that of maximum C_T by approximately 0.17π . In Fig. 5.14(b), strong fluctuations in C_x and C_y similar to that of prescribed rotation shown in Fig. 5.4 are observed.

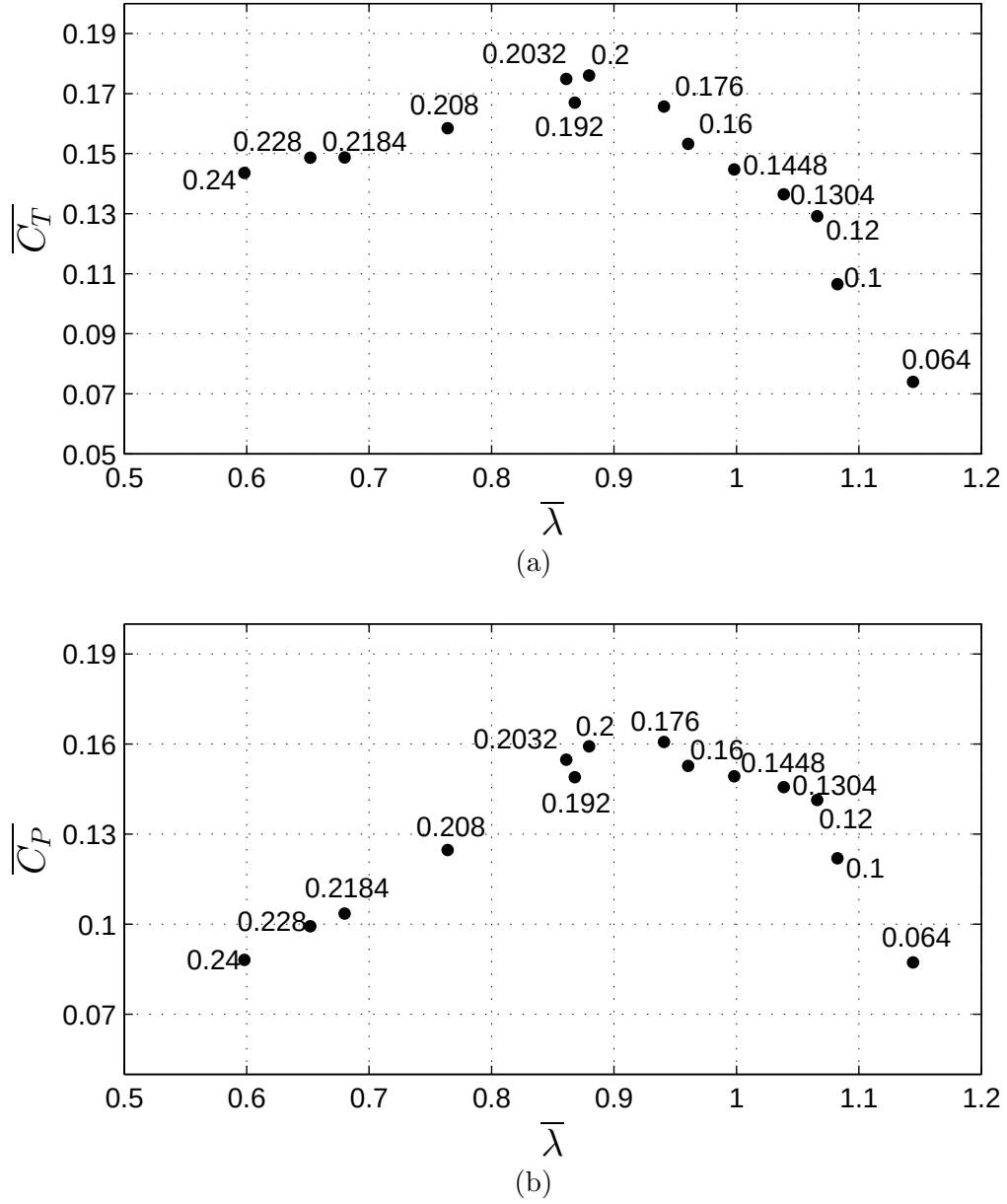


Figure 5.12: Time-averaged (a) torque and (b) power coefficients, $\overline{C_T}$ and $\overline{C_P}$, as a function of the mean tip-speed ratio $\bar{\lambda}$ of the freely rotating Bach-type turbine with $\varphi = 124^\circ$ and $p/q = 0.2$ at $Re = 1,500$. The numerical value next to each data point is the non-dimensional generator damping B_g^* .

5.2. Numerical results

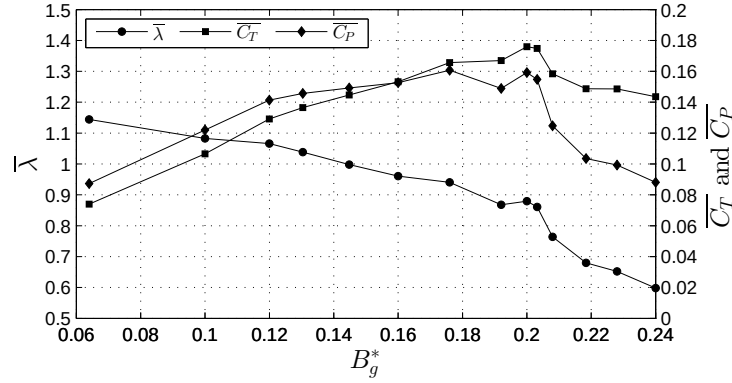


Figure 5.13: Dependence of $\overline{\lambda}$, $\overline{C_T}$, and $\overline{C_P}$ on the non-dimensional generator damping B_g^* of the freely rotating Bach-type turbine with $\varphi = 124^\circ$ and $p/q = 0.2$ at $Re = 1,500$.

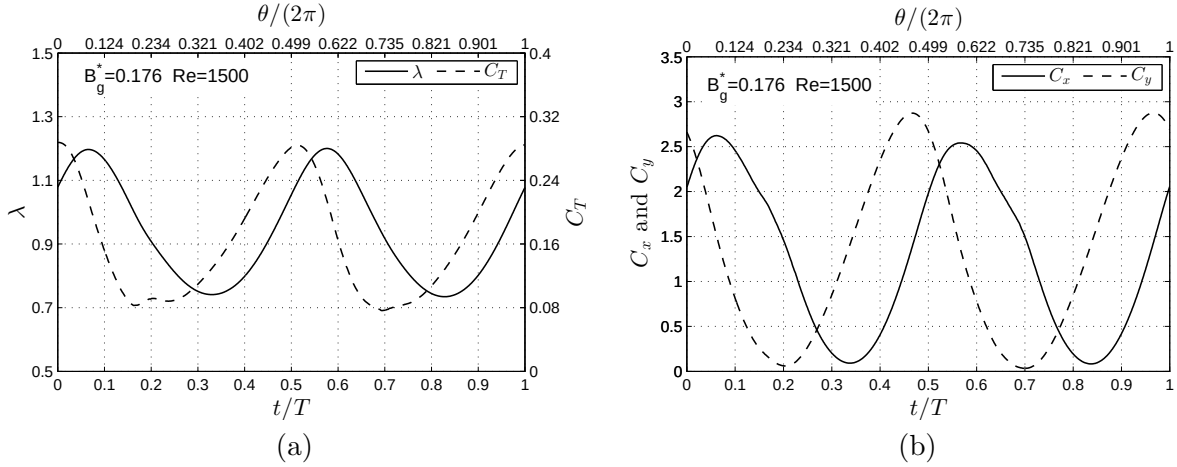


Figure 5.14: The instantaneous (a) tip-speed ratio λ , torque coefficient C_T , and (b) the force coefficients C_x and C_y over one period of rotation for $B_g^* = 0.176$. The mean tip-speed ratio and torque coefficient are $\overline{\lambda} = 0.941$ and $\overline{C_T} = 0.166$, respectively. Because of the unsteady rotation, the turbine angular position θ is also shown for reference.

Based on the above observation, the micro-scale flow turbine can, in fact, be used in water, albeit at reduced performance because of strong fluctuation in angular velocity. The efficiency of the turbine remains acceptable overall.

5.2.2.2 Evolution of vortical flow with free rotation

Despite the strong fluctuation in angular speed, the vorticity field near the freely rotating turbine is qualitatively very similar to that of prescribed rotation except for some minor

differences. The vorticity field near the turbine is shown in Fig. 5.15 for $B_g^* = 0.2$ with $\bar{\lambda} = 0.88$. Compared to prescribed rotation at $\lambda = 0.9$ (Fig. 5.6), a different evolution of vortices B1 and B2 is observed in Fig. 5.15. The turbine accelerates to its maximum angular velocity at $t/T = 0.547$. Owing to the high speed of the returning blade, vortex B1 is stretched and covers almost the entire wind side of the returning blade at $t/T = 0.604$. Eventually, vortex B1 is split into two parts B1a and B1b by the returning blade with B1a merging into the boundary layer of the returning blade and B1b shed from the edge of the returning blade. A similar evolution of B2 is observed during the next rotation not shown here.

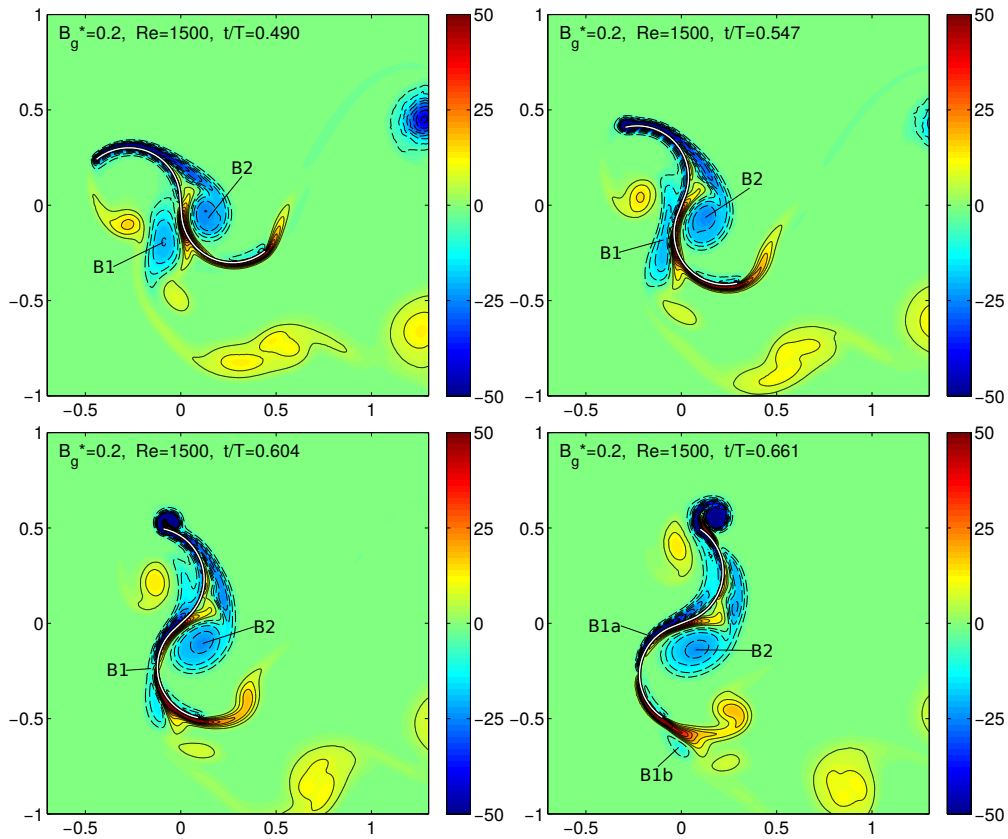


Figure 5.15: Non-dimensional vorticity ($\omega D/U$) contour plots of the flow around the freely rotating Bach-type turbine with $B_g^* = 0.2$ and $Re = 1,500$. For this case, $\bar{\lambda} = 0.88$. Horizontal scale is x/D , and vertical scale is y/D . Solid contour lines represent positive vorticity; dashed contour lines represent negative vorticity. Contour level increment is 5.

5.3. Potential applications

Table 5.2: Shaft power estimation of a micro-scale flow turbine in air and water

Fluid	Air (20°C)	Water (5°C)
Turbine diameter (cm)	1.0	1.0
Turbine height (cm)	2.0	2.0
Density (kg/m ³)	1.2	1000
Kinematic viscosity (m ² /s) × 10 ⁻⁶	15	1.5
Flow speed (m/s)	3.0-6.0	0.3-0.6
Reynolds number, Re	2000-4000	2000-4000
Power coefficient, $\overline{C_P}$	0.18	0.16
Shaft power (mW)	0.6-4.7	0.4-3.5

5.3 Potential applications

As alluded to earlier in Sec. 1.3.2, this type of micro-scale flow turbines can potentially be used as alternative or backup power sources for sensor nodes in a Wireless Sensor Network. As a feasibility study, the power generation potential of a micro flow turbine of 1.0 cm diameter and 2.0 cm height is estimated. End plates can be installed to improve the two-dimensionality of the flow. The shaft power of this micro-scale turbine is calculated using the numerically computed power coefficients $\overline{C_P}$ in both air and water currents. In air, the maximum time-averaged power coefficient of 0.18 for prescribed rotation is used; a metal turbine rotates at a roughly constant angular velocity in air because of the low fluid density. On the other hand, the maximum time-averaged power coefficient of 0.16 for free rotation is used for water current to account for the large fluctuation of angular velocity in water.

For air, the range of wind speed considered is from 3 m/s to 6 m/s. This range of wind speed corresponds to level 3 (gentle breeze) and the lower end of level 4 (moderate breeze) on the Beaufort scale. For water, the range of current speed considered is between 0.3 m/s to 0.6 m/s. The resulting range of Reynolds number in both air and water turns out to be 2,000-4,000. This range is slightly larger than the Reynolds number used in the computation but still of the same order of magnitude. Therefore, it is reasonable to approximate the turbine performance using the values of $\overline{C_P}$ computed at $Re = 1,500$ considering the insensitivity of turbine performance to Re . The flow conditions and the estimated turbine shaft power are summarized in Table 5.2. For both air and water, the estimated shaft power is roughly between 0.5-4 mW.

The power consumption of sensor nodes strongly depends on the specific design of the device. As an example, the Mica2 sensor node, originally developed at the University of California, Berkeley, is considered [104]. This sensor node contains a microprocessor, memory modules, and a radio with a maximum outdoor transmission range of 300 m. The entire device measures 5.7 cm × 3.1 cm × 1.8 cm. Therefore, it is reasonable to fit one or multiple micro flow turbines of 1.0 cm diameter onto this device. The Mica2 sensor node uses a power

source of 2.7-3.3 V (typically two AA-batteries). The current draw of Mica2 varies widely from 30 μ A to 30 mA depending on its mode of operation. In [104], a 3V power supply was used to run a Mica2 sensor node, and the current consumption profile of the sensor node for transmitting a single radio message at maximum transmit power was measured. The sensor node was initially in a low-power state consuming less than 100 μ A of current. Subsequently, the CPU woke up, and the radio started listening for roughly 0.06 s. During this phase, approximately 11 mA of current was being used. The next phase of the operation, lasting around 0.02s, involved radio transmission at maximum power, consuming 25 mA of current. After the radio transmission, the sensor node returned to the low-power state. In the low-power state, the device consumes only 0.3 mW of power. During the radio transmission, total energy consumption is approximately 3.5 mJ. Depending on the wind (current) speed, it is possible to satisfy this energy requirement using the micro flow turbine with a few seconds of charging.

The power estimation indicates that micro flow turbines can potentially be used as alternative or backup power sources for wireless sensor nodes. At the very least, the addition of a micro-turbine can significantly extend the life expectancy of a sensor node or indefinitely maintain the sensor node in a low-power sleep state to be woken up at a future time. In addition, multiple micro flow turbines can be installed to power heavier loads with potential beneficial interactions between turbines (see e.g. [105]).

Chapter 6

Conclusions

In this work, a high-performance discrete-vortex algorithm is developed and applied to the analyses of two different physical problems.

6.1 Implementation and validation of the Discrete-Vortex Method

A high-performance discrete-vortex algorithm is successfully developed and implemented. To avoid the various drawbacks caused by the stochastic nature of the Random-Vortex Method [73], the Viscous-Vortex-Domain (VVD) formulation is adopted instead [19]. The VVD method is a deterministic method for simulating diffusion based on the diffusion-velocity concept [8]. The computational performance of the discrete-vortex simulation is greatly improved with the application of the Fast-Multipole Method (FMM) for vortex interactions and a quadtree-based search algorithm for efficient neighbor search. The computation of flow velocity and diffusion velocity has also been accelerated through the use of collaborative CPU and Graphics-Processing-Unit (GPU) parallel computing (heterogeneous computing). The final implementation of the FMM that runs on just two CPUs and a single GPU achieved a speed about 70 times faster than a simple serial implementation of FMM on a single CPU of the same model.

The current implementation of Discrete-Vortex Method (DVM) has been validated by comparing the numerical results with published ones for a variety of different test cases, including the flows around a translating cylinder at various Reynolds numbers, a translating and rotating ellipse, a flapping elliptic foil, and two elliptic foils undergoing a clap-and-fling motion. The Vortex-Induced Vibration (VIV) of an elastically mounted cylinder is also successfully predicted by DVM simulation, with the computed excursion amplitudes and vibration frequencies shown to be in good agreement with published computational results.

The discrete-vortex algorithm developed in this work is a fairly versatile tool that can simulate two-dimensional low to medium-Reynolds number flows near multiple arbitrarily shaped bodies undergoing either prescribed or free motions. Two specific applications with their intricate flow physics are investigated and reported.

6.2 Flapping wing with ground effects

Using the discrete-vortex algorithm, we have first investigated the full and partial ground effects on the lift generation of a two-dimensional flapping elliptic foil in normal hovering mode. The foil, of chord c , is hovering above a long ($10c$) but finite-sized platform. When the foil is hovering above the center of the platform under full ground effect, we have observed from our numerical solution the same force-enhancement, force-reduction, and force-recovery regimes previously reported [36] and confirmed [44]. The force-enhancement regime shows up when the ground clearance is small. The leading-edge vortex (LEV) from the previous stroke is forced over the foil leading (top) edge during stroke reversal. The LEV, coupled with the trailing-edge vortex (TEV) from the previous stroke, induces a strong jet impinging on the wind side of the foil, leading to enhanced lift and drag. The LEV from previous stroke also enhances the new LEV formed during the subsequent stroke, further contributing to lift generation. We refer to this flow behavior as the ground-enhanced wake-capture mechanism. As the ground clearance increases, the LEV slides down along the wind side of the foil after stroke reversal. The ground-enhanced wake-capture mechanism no longer takes place, and the flow is in the force-reduction regime. When ground clearance further increases, the flow enters the force-recovery regime because of another lift enhancing mechanism: a strong vortex pair is formed during the second half of the upstroke, inducing a downward jet and increasing the lift force.

The lift generation of the flapping foil under partial ground effect (i.e. the foil is hovering near the edge of the platform) is investigated through conducting a large number of simulations with different combinations of vertical distance D and horizontal distance S between the foil mean position and the edge of the platform. As expected, the time-averaged and time-dependent lift coefficients show complicated dependencies on D and S . The relation between the lift force under partial ground effect and the flow vorticity field as discovered in this work is discussed. We have discovered new lift-modifying flow mechanisms that are previously not observed with full ground effect. Specifically, we have discovered that TEVs shed from the foil near the edge of the platform tend to remain relatively stationary. As a result, TEVs of the same sign shed from consecutive periods will merge together, forming a strong vortex. The formation and effect of this strong vortex depend on the values of S and D . It may reduce the lift force after stroke reversal by weakening the ground-enhanced wake-capture mechanism in the force-enhancement regime. This is observed, for example, when $S/c = -1.75$ and $D/c = 1$. Alternatively, the strong vortex may serve to recover lift force during downstroke in the force-reduction regime by enhancing LEV formation as observed when $S/c = -2$ and $D/c = 2$. The edge of the platform may also increase the lift force during the second half of the downstroke by inducing the formation of a vortex pair. For instance, when $S/c = 0.75$ and $D/c = 1$, a vortex pair is formed during the second half of the downstroke from the counterclockwise TEV and the clockwise vortex shed from the edge of the platform. An alternative way the platform edge can help with the formation of a vortex pair during downstroke is observed when $S/c = 1$ and $D/c = 2$. In this case, the edge of the platform slightly impedes the downward motion of the clockwise TEV shed

from the preceding upstroke. This clockwise TEV forms a vortex pair with the counter-clockwise TEV from the current downstroke. The asymmetric ground effect also leads to a net cycle-average lateral force that either pushes the foil away from the platform or pulls it towards the platform center depending on the values of S and D . Lastly, we have observed that, under suitable conditions, the wake of the flapping foil can form a reverse von Kármán vortex street because of the presence of the platform edge.

As future Micro Aerial Vehicles and Autonomous Underwater Vehicles are expected to navigate a large variety of complex environments, it is of scientific and engineering importance to further expand the present study. For example, more research can be conducted on the lift and thrust generations of a flapping foil near a cliff-like edge, instead of a thin platform. Also, it is of great interest to study the effects of thin flexible platforms and walls that deform and oscillate in response to the wake of the flapping foil.

6.3 Micro-scale Bach-type turbine

Next, the discrete-vortex algorithm is applied to investigate the flow past a micro-scale Bach-type vertical-axis wind or current turbine at a Reynolds number of 1,500. Different from previous DVM simulations of drag-based vertical-axis turbines, full viscous effect is included with simulation of the boundary layers. Also, boundary-layer separation is no longer restricted to the ends of the blade. Further, both the prescribed rotation (tip-speed ratio specified) and free rotation (generator damping specified) of the turbine are studied. The maximum time-averaged power coefficient $\overline{C_P}$, defined as the ratio of shaft power to incident current power, is found to be 0.18 with prescribed rotation. This value is approximately equal to the maximum $\overline{C_P}$ experimentally measured with the same turbine geometry at $Re = 80,000$ [98]. This result shows that drag-based vertical-axis turbines can be *miniaturized* into micro-scale power harvesters which have a wide range of practical applications such as driving the electronics of a low-power Wireless Sensor Network.

A detailed analysis of the creation and evolution of vortical flow structures near the turbine blade is presented, and the effect of tip-speed ratio on the flow evolution is investigated. In particular, an interesting wake-capture mechanism that boosts the performance of the turbine at low Reynolds numbers is discovered. For a certain range of prescribed tip-speed ratios, the vortex shed from the end of the advancing blade impacts the wind side of the returning blade. The low pressure vortex creates a suction effect on the returning blade, pulling it in the direction of rotation. The discovery of this mechanism demonstrates the important effects of the evolution of coherent vortical flow structures on the performance of micro-scale turbines. Therefore, many of the performance-improving techniques, such as shielding plates, designed for turbines operating at higher Reynolds numbers need to be adjusted or redesigned to accommodate or even utilize the concentrated flow energy carried by the large vortical structures in low Re flow.

The free rotation of a steel micro-scale Bach-type rotor driven by water current is also simulated. The high density of water coupled with the low inertia of the rotor leads to

surprisingly significant fluctuations in turbine angular velocity over one period of rotation. This unsteady rotation is detrimental to power extraction. Overall, the mean power coefficient for free rotation is lower than that of prescribed rotation. The maximum mean power coefficient is reduced from 0.18 (prescribed rotation) to 0.16 (free rotation). Interestingly, the amplitude of the fluctuation of the torque coefficient is significantly reduced with free rotation.

Finally, the power-generation capability of a micro-scale flow turbine is estimated based on the results of the present numerical flow simulations. The power estimation indicates that adding micro flow turbines to wireless sensor nodes as alternative or as backup-power supplies may significantly extend the life expectancy of the nodes or even maintain the nodes in a low-power state indefinitely.

6.4 Summary of major contributions

New efficient implementation strategies for various components of the discrete-vortex algorithm have been devised. In particular, the new strategies utilize highly effective hybrid CPU and GPU parallel computing to achieve significant speedup of the Fast-Multipole Method for computing vortex-vortex interaction and the Viscous-Vortex-Domain method for evaluating diffusion velocity. The speedup enables the discrete-vortex algorithm to simulate much larger problems within a realistic amount of time on a GPU-equipped computer.

The present discrete-vortex algorithm is applied to investigate the lift-generating mechanism of a flapping foil under full and partial ground effects. The partial ground effect caused by the presence of an effectively semi-infinite ground platform has not been studied before despite its relevance to the design and control of MAVs. New lift-modifying flow mechanisms unique to partial ground effect have been discovered as a result of the present investigation.

The discrete-vortex algorithm has also been used to predict the fluid-dynamic characteristics and power-generating capacity of a micro-scale Bach-type flow turbine. The performance of such small-scale turbines considered here has not been found in the existing literature. The new results from the present work indicate that Bach-type turbines do not suffer significant performance penalty when miniaturized, making this type of turbine a prime candidate for micro-scale energy harvesting. In addition, a new wake-capturing mechanism beneficial to power extraction has been discovered for micro-scale turbines.

Bibliography

- [1] P. Ploumhans, G. S. Winckelmans, J. K. Salmon, A. Leonard, and M. S. Warren, “Vortex Methods for Direct Numerical Simulation of Three-Dimensional Bluff Body Flows: Application to the Sphere at $Re=300$, 500, and 1000,” *Journal of Computational Physics*, vol. 178, pp. 427–463, 2002.
- [2] A. J. Chorin, “Numerical Study of Slightly Viscous Flow,” *Journal of Fluid Mechanics*, vol. 57, pp. 785–796, 1973.
- [3] L. A. Barba, *Vortex Method for Computing High-Reynolds Number Flows: Increased Accuracy with a Fully Mesh-less Formulation*. PhD thesis, California Institute of Technology, 2004.
- [4] Y. Jiang, *Computational Modeling in Three Dimensions of Multi-DOF Ship Motion in a Viscous Fluid*. PhD thesis, University of California, Berkeley, 2014.
- [5] R. W. Yeung, Y. Jiang, and L. Wang, “Validation of a Discrete-Vortex Method (SS-FSRVM) for Modeling Nonlinear Coupled Vertical-Plane and Lateral Ship Motions in Head Waves,” in *Proceedings of the 31st Symposium on Naval Hydrodynamics*, (Monterey, California, USA), September 2016.
- [6] C. C. Chang, “Random Vortex Methods for the Navier-Stokes Equations,” *Journal of Computational Physics*, vol. 76, pp. 281–300, 1988.
- [7] A. L. Fogelson, “Particle-Method Solution of Two-Dimensional Convection-Diffusion Equations,” *Journal of Computational Physics*, vol. 100, pp. 1–16, 1992.
- [8] Y. Ogami and T. Akamatsu, “Viscous Flow Simulation Using the Discrete Vortex Model - The Diffusion Velocity Method,” *Computers & Fluids*, vol. 19, pp. 433–441, 1991.
- [9] L. A. Barba, A. Leonard, and C. B. Allen, “Advances in Viscous Vortex Methods - Meshless Spatial Adaption Based on Radial Basis Function Interpolation,” *International Journal for Numerical Methods in Fluids*, vol. 47, pp. 387–421, 2005.
- [10] K. Takeda, O. R. Tutty, and A. D. Fitt, “A Comparison of Four Viscous Models for the Discrete Vortex Method,” in *Proceedings of the 13th AIAA CFD Conference*, (Snowmass, Colorado, USA), June-July 1997.
- [11] A. Leonard, “Vortex Methods for Flow Simulation,” *Journal of Computational Physics*, vol. 37, pp. 289–335, 1980.

- [12] C. Greengard, “The Core Spreading Vortex Method Approximates the Wrong Equation,” *Journal of Computational Physics*, vol. 61, pp. 345–348, 1985.
- [13] L. F. Rossi, “Resurrecting Core Spreading Vortex Methods: A New Scheme That Is Both Deterministic and Convergent,” *SIAM Journal on Scientific Computing*, vol. 17, pp. 370–397, 1996.
- [14] S. Shankar and L. van Dommelen, “A New Diffusion Procedure for Vortex Methods,” *Journal of Computational Physics*, vol. 127, pp. 88–109, 1996.
- [15] P. Koumoutsakos, *Direct Numerical Simulations of Unsteady Separated Flows Using Vortex Methods*. PhD thesis, California Institute of Technology, 1993.
- [16] P. Koumoutsakos, A. Leonard, and F. Pépin, “Boundary Conditions for Viscous Vortex Methods,” *Journal of Computational Physics*, vol. 113, pp. 52–61, 1994.
- [17] P. Koumoutsakos and A. Leonard, “High-Resolution Simulations of the Flow around an Impulsively Started Cylinder Using Vortex Methods,” *Journal of Fluids Mechanics*, vol. 296, pp. 1–38, 1995.
- [18] N. R. Clarke and O. R. Tutty, “Construction and Validation of a Discrete Vortex Method for the Two-Dimensional Incompressible Navier-Stokes Equation,” *Computers & Fluids*, vol. 23, pp. 751–783, 1994.
- [19] G. Y. Dynnikova, “The Viscous Vortex Domains (VVD) Method for Non-Stationary Viscous Incompressible Flow Simulation,” in *4th European Conference on Computational Mechanics*, (Paris, France), May 2010.
- [20] S. V. Guvernyuk and G. Y. Dynnikova, “Modeling the Flow Past an Oscillating Airfoil by the Method of Viscous Vortex Domains,” *Fluid Dynamics*, vol. 42, pp. 1–11, 2007.
- [21] L. Wang and R. W. Yeung, “Investigation of Full and Partial Ground Effects on a Flapping Foil Hovering above a Finite-Sized Platform,” *Physics of Fluids*, vol. 28, p. 071902, 2016.
- [22] L. Wang and R. W. Yeung, “On the Performance of a Micro-Scale Bach-Type Turbine as Predicted by Discrete-Vortex Simulations,” *Applied Energy*, vol. 183, pp. 823–836, 2016.
- [23] U. Pesavento and Z. J. Wang, “Flapping Wing Flight Can Save Aerodynamic Power Compared to Steady Flight,” *Phys. Rev. Lett.*, vol. 103, p. 118102, 2009.
- [24] M. H. Dickinson, F. O. Lehmann, and S. P. Sane, “Wing Rotation and the Aerodynamic Basis of Insect Flight,” *Science*, vol. 284, pp. 1954–1960, 1999.

- [25] J. D. Eldredge, “Numerical Simulation of the Fluid Dynamics of 2D Rigid Body Motion with the Vortex Particle Method,” *Journal of Computational Physics*, vol. 221, pp. 626–648, 2007.
- [26] L. Kajtar and J. J. Monaghan, “SPH Simulations of Swimming Linked Bodies,” *Journal of Computational Physics*, vol. 227, pp. 8568–8587, 2008.
- [27] K. Ota, K. Suzuki, and T. Inamuro, “Lift Generation by a Two-Dimensional Symmetric Flapping Wing: Immersed Boundary-Lattice Boltzmann Simulations,” *Fluid Dynamics Research*, vol. 44, p. 045504, 2012.
- [28] N. Arora, A. Gupta, S. Sanghi, H. Aono, and W. Shyy, “Lift-Drag and Flow Structures Associated with the “Clap and Fling” Motion,” *Physics of Fluids*, vol. 26, p. 071906, 2014.
- [29] K. Suzuki, K. Minami, and T. Inamuro, “Lift and Thrust Generation by a Butterfly-like Flapping Wing-Body Model: Immersed Boundary-Lattice Boltzmann Simulations,” *Journal of Fluid Mechanics*, vol. 767, pp. 659–695, 2015.
- [30] Z. J. Wang, J. M. Birch, and M. H. Dickinson, “Unsteady Forces and Flows in Low Reynolds Number Hovering Flight: Two-Dimensional Computations vs Robotic Wing Experiments,” *Journal of Experimental Biology*, vol. 207, pp. 449–460, 2004.
- [31] J. M. Birch and M. H. Dickinson, “The Influence of Wing-Wake Interactions on the Production of Aerodynamic Forces in Flapping Flight,” *Journal of Experimental Biology*, vol. 206, pp. 2257–2272, 2003.
- [32] P. Freymuth, “Thrust Generation by an Airfoil in Hover Modes,” *Experiments in Fluids*, vol. 9, pp. 17–24, 1990.
- [33] B. Singh and I. Chopra, “Insect-Based Hover-Capable Flapping Wings for Micro Air Vehicles: Experiments and Analysis,” *AIAA Journal*, vol. 46, pp. 2115–2135, 2008.
- [34] L. Schouveiler, F. S. Hover, and M. S. Triantafyllou, “Performance of Flapping Foil Propulsion,” *Journal of Fluids and Structures*, vol. 20, pp. 949–959, 2005.
- [35] T. Gao, N. S. Liu, and X. Y. Lu, “Numerical Analysis of the Ground Effect on Insect Hovering,” *Journal of Hydrodynamics, Ser. B*, vol. 20, pp. 17–22, 2008.
- [36] T. Gao and X. Y. Lu, “Insect Normal Hovering Flight in Ground Effect,” *Physics of Fluids*, vol. 20, p. 087101, 2008.
- [37] J. Wu and N. Zhao, “Ground Effect on Flapping Wing,” *Procedia Engineering*, vol. 67, pp. 295–302, 2013.
- [38] J. Wu, C. Shu, N. Zhao, and W. Yan, “Fluid Dynamics of Flapping Insect Wing in Ground Effect,” *Journal of Bionic Engineering*, vol. 11, pp. 52–60, 2014.

- [39] J. Wu, Y. L. Qiu, C. Shu, N. Zhao, and X. Wang, “An Adaptive Immersed Boundary-Lattice Boltzmann Method for Simulating a Flapping Foil in Ground Effect,” *Computers & Fluids*, vol. 106, pp. 171–184, 2015.
- [40] J. Y. Su, J. H. Tang, C. H. Wang, and J. T. Yang, “A Numerical Investigation on the Ground Effect of a Flapping-Flying Bird,” *Physics of Fluids*, vol. 25, p. 093101, 2013.
- [41] M. Ghommam, D. Garcia, and V. M. Calo, “Enclosure Enhancement of Flight Performance,” *Theoretical and Applied Mechanics Letters*, vol. 4, p. 062003, 2014.
- [42] D. Kolomenskiy, M. Maeda, T. Engels, H. Liu, K. Schneider, and J. C. Nave, “Aerodynamic Ground Effect in Fruitfly Sized Insect Takeoff,” *PLoS ONE*, vol. 11, p. e0152072, 2016.
- [43] T. V. Truong, D. Byunn, M. J. Kim, K. J. Yoon, and H. C. Park, “Aerodynamic Forces and Flow Structures of the Leading Edge Vortex on a Flapping Wing Considering Ground Effect,” *Bioinspiration & Biomimetics*, vol. 8, p. 036007, 2013.
- [44] H. Lu, K. B. Lua, T. T. Lim, and K. S. Yeo, “Ground Effect on the Aerodynamics of a Two-Dimensional Oscillating Airfoil,” *Experiments in Fluids*, vol. 55, p. 1787, 2014.
- [45] L. Y. Pao and K. E. Johnson, “Control of Wind Turbines,” *IEEE Control Systems*, vol. 31, no. 2, pp. 44–62, 2011.
- [46] E. A. Bossanyi, “The Design of Closed Loop Controllers for Wind Turbines,” *Wind Energy*, vol. 3, no. 3, pp. 149–163, 2000.
- [47] J. D. Dabiri, “Potential Order-of-Magnitude Enhancement of Wind Farm Power Density via Counter-Rotating Vertical-Axis Wind Turbine Arrays,” *Journal of Renewable and Sustainable Energy*, vol. 3, p. 043104, 2011.
- [48] S. J. Savonius, “The S-Rotor and Its Applications,” *Mechanical Engineering*, vol. 53, pp. 333–337, 1931.
- [49] G. Bach, “Untersuchungen über Savonius-Rotoren und verwandte Strömungsmaschinen,” *Forschung auf dem Gebiet des Ingenieurwesens A*, vol. 2, pp. 218–231, 1931.
- [50] F. R. Eldridge, *Wind Machines*. New York, USA: Van Nostrand Reinhold Company, 2 ed., 1980.
- [51] R. Gupta, A. Biswas, and K. K. Sharma, “Comparative Study of a Three-Bucket Savonius Rotor with a Combined Three-Bucket Savonius-Three-Bladed Darrieus Rotor,” *Renewable Energy*, vol. 33, no. 9, pp. 1974–1981, 2008.

- [52] M. J. Khan, G. Bhuyan, M. T. Iqbal, and J. E. Quaicoe, “Hydrokinetic Energy Conversion Systems and Assessment of Horizontal and Vertical Axis Turbines for River and Tidal Applications: A Technology Status Review,” *Applied Energy*, vol. 86, no. 10, pp. 1823–1835, 2009.
- [53] N. K. Sarma, A. Biswas, and R. D. Misra, “Experimental and Computational Evaluation of Savonius Hydrokinetic Turbine for Low Velocity Condition with Comparison to Savonius Wind Turbine at the Same Input Power,” *Energy Conversion and Management*, vol. 83, pp. 88–98, 2014.
- [54] J. Yick, B. Mukherjee, and D. Ghosal, “Wireless Sensor Network Survey,” *Computer Networks*, vol. 52, no. 12, pp. 2292–2330, 2008.
- [55] S. Roy and U. K. Saha, “Wind Tunnel Experiments of a Newly Developed Two-Bladed Savonius-Style Wind Turbine,” *Applied Energy*, vol. 137, pp. 117–125, 2015.
- [56] N. Han, D. Zhao, J. U. Schluter, E. S. Goh, H. Zhao, and X. Jin, “Performance Evaluation of 3D Printed Miniature Electromagnetic Energy Harvesters Driven by Air Flow,” *Applied Energy*, vol. 178, pp. 672–680, 2016.
- [57] S. Roy and U. K. Saha, “Review on the Numerical Investigations into the Design and Development of Savonius Wind Rotors,” *Renewable & Sustainable Energy Reviews*, vol. 24, pp. 73–83, 2013.
- [58] J. V. Akwa, G. A. da Silva Júnior, and A. P. Petry, “Discussion on the Verification of the Overlap Ratio Influence on Performance Coefficients of a Savonius Wind Rotor Using Computational Fluid Dynamics,” *Renewable Energy*, vol. 38, no. 1, pp. 141–149, 2012.
- [59] T. Zhou and D. Rempfer, “Numerical Study of Detailed Flow Field and Performance of Savonius Wind Turbines,” *Renewable Energy*, vol. 51, pp. 373–381, 2013.
- [60] W. Tian, B. Song, J. H. VanZwieten, and P. Pyakurel, “Computational Fluid Dynamics Prediction of a Modified Savonius Wind Turbine with Novel Blade Shapes,” *Energies*, vol. 8, no. 8, pp. 7915–7929, 2015.
- [61] T. Ogawa, “Theoretical Study on the Flow about Savonius Rotor,” *Journal of Fluids Engineering*, vol. 106, no. 1, pp. 85–91, 1984.
- [62] M. A. Kotb and T. K. Aldoss, “Flowfield around a Partially-Blocked Savonius Rotor,” *Applied Energy*, vol. 38, no. 2, pp. 117–132, 1991.
- [63] N. Fujisawa, “Velocity Measurements and Numerical Calculations of Flow Fields in and around Savonius Rotors,” *Journal of Wind Engineering & Industrial Aerodynamics*, vol. 59, no. 1, pp. 39–50, 1996.

- [64] M. S. U. K. Fernando and V. J. Modi, "A Numerical Analysis of the Unsteady Flow Past a Savonius Wind Turbine," *Journal of Wind Engineering & Industrial Aerodynamics*, vol. 32, no. 3, pp. 303–327, 1989.
- [65] V. J. Modi and M. S. U. K. Fernando, "Unsteady Aerodynamics and Wake of the Savonius Wind Turbine: A Numerical Study," *Journal of Wind Engineering & Industrial Aerodynamics*, vol. 46 & 47, pp. 811–816, 1993.
- [66] D. Afungchui, B. Kamoun, A. Helali, and A. B. Djemaa, "The Unsteady Pressure Field and the Aerodynamic Performances of a Savonius Rotor Based on the Discrete Vortex Method," *Renewable Energy*, vol. 35, no. 1, pp. 307–313, 2010.
- [67] D. Afungchui, B. Kamoun, and A. Helali, "Vortical Structures in the Wake of the Savonius Wind Turbine by the Discrete Vortex Method," *Renewable Energy*, vol. 69, pp. 174–179, 2014.
- [68] J. H. Strickland, B. T. Webster, and T. Nguyen, "A Vortex Model of the Darrieus Turbine: An Analytical and Experimental Study," *Journal of Fluids Engineering*, vol. 101, no. 4, pp. 500–505, 1979.
- [69] Y. Li and S. M. Calisal, "Three-Dimensional Effects and Arm Effects on Modeling a Vertical Axis Tidal Current Turbine," *Renewable Energy*, vol. 35, no. 10, pp. 2325–2334, 2010.
- [70] R. Urbina, M. L. Peterson, R. W. Kimball, G. S. deBree, and M. P. Cameron, "Modeling and Validation of a Cross Flow Turbine Using Free Vortex Model and a Modified Dynamic Stall Model," *Renewable Energy*, vol. 50, pp. 662–669, 2013.
- [71] E. Dyachuk and A. Goude, "Numerical Validation of a Vortex Model against Experimental Data on a Straight-Bladed Vertical Axis Wind Turbine," *Energies*, vol. 8, no. 10, pp. 11800–11820, 2015.
- [72] E. Dyachuk, A. Goude, and H. Bernhoff, "Simulating Pitching Blade with Free Vortex Model Coupled with Dynamic Stall Model for Conditions of Straight Bladed Vertical Axis Turbines," *Journal of Solar Energy Engineering*, vol. 137, no. 4, p. 041008, 2015.
- [73] A. J. Chorin and P. S. Bernard, "Discretization of a Vortex Sheet, with an Example of Roll-Up," *Journal of Computational Physics*, vol. 13, pp. 423–429, 1973.
- [74] L. F. Greengard, "The Rapid Evaluation of Potential Fields in Particle Systems," tech. rep., Yale University, 1987.
- [75] NVIDIA, "GeForce GTX TITAN Black Specifications." <http://www.geforce.com/hardware/desktop-gpus/geforce-gtx-titan-black/specifications>. Accessed: 10/11/2016.

- [76] R. E. Overman, J. F. Prins, L. Miller, and M. L. Minion, “Dynamic Load Balancing of the Adaptive Fast Multipole Method in Heterogeneous Systems,” in *Parallel and Distributed Processing Symposium Workshops & PhD Forum (IPDPSW), 2013 IEEE 27th International* (pp. 1126-1135). IEEE., 2013.
- [77] H. Lin, M. Vezza, and R. A. M. Galbraith, “Discrete Vortex Method for Simulating Unsteady Flow around Pitching Aerofoils,” *AIAA Journal*, vol. 35, pp. 494–499, 1997.
- [78] J. J. Monaghan, “Smoothed Particle Hydrodynamics and Its Diverse Applications,” *Annual Review of Fluid Mechanics*, vol. 44, pp. 323–346, 2012.
- [79] Y. Ogami, “Simulation of Heat-Vortex Interaction by the Diffusion Velocity Method,” in *ESAIM: Proceedings*, vol. 7, pp. 314–324, EDP Sciences, 1999.
- [80] Y. Ogami, “Fast Summation Algorithms for the Particle Simulation of Far-Field Potential and Rapidly-Decaying Potential,” *JSME International Journal Series B*, vol. 45, pp. 174–182, 2002.
- [81] J. L. Bentley, “Multidimensional Binary Search Trees Used for Associative Searching,” *Communications of the ACM*, vol. 18, pp. 509–517, 1975.
- [82] P. R. Spalart, “Vortex Methods for Separated Flows,” Technical Memorandum, No 100068, N88-26342, NASA, 1988.
- [83] L. F. Rossi, “Merging Computational Elements in Vortex Simulations,” *SIAM Journal on Scientific Computing*, vol. 18, pp. 1014–1027, 1997.
- [84] J. Park, K. Kwon, and H. Choi, “Numerical Solutions of Flow Past a Circular Cylinder at Reynolds Numbers up to 160,” *KSME International Journal*, vol. 12, pp. 1200–1205, 1998.
- [85] B. Sharman, F. S. Lien, L. Davidson, and C. Norberg, “Numerical Predictions of Low Reynolds Number Flows over Two Tandem Circular Cylinders,” *International Journal for Numerical Methods in Fluids*, vol. 47, pp. 423–447, 2005.
- [86] D. Rossinelli, M. Bergdorf, G. H. Cottet, and P. Koumoutsakos, “GPU Accelerated Simulations of Bluff Body Flows Using Vortex Particle Methods,” *Journal of Computational Physics*, vol. 229, pp. 3316–3333, 2010.
- [87] S. Mittal, V. Kumar, and A. Raghuvanshi, “Unsteady Incompressible Flows Past Two Cylinders in Tandem and Staggered Arrangements,” *International Journal for Numerical Methods in Fluids*, vol. 25, pp. 1315–1344, 1997.
- [88] W. Jester and Y. Kallinderis, “Numerical Study of Incompressible Flow about Fixed Cylinder Pairs,” *Journal of Fluids and Structures*, vol. 17, pp. 561–577, 2003.

- [89] E. Rossi, *2D-Vorticity Genesis and Dynamics Studied through Particle Methods*. PhD thesis, Sapienza University of Rome, 2014.
- [90] E. Rossi, A. Colagrossi, D. Durants, and G. Graziani, “Simulating 2D Viscous Flow around Geometries with Vertices through the Diffused Vortex Hydrodynamics Method,” *Computer methods in applied mechanics and engineering*, vol. 302, pp. 147–169, 2016.
- [91] A. Maruoka, “Finite Element Analysis for Flow around a Rotating Body Using Chimera Method,” *International Journal of Computational Fluid Dynamics*, vol. 17, pp. 289–297, 2003.
- [92] X. Zhang, S. Ni, and G. He, “A Pressure-Correction Method and Its Applications on an Unstructured Chimera Grid,” *Computers & Fluids*, vol. 37, pp. 993–1010, 2008.
- [93] B. Zhang and C. Liang, “A Simple, Efficient, and High-Order Accurate Curved Sliding-Mesh Interface Approach to Spectral Difference Method on Coupled Rotating and Stationary Domains,” *Journal of Computational Physics*, vol. 295, pp. 147–160, 2015.
- [94] J. Tang, D. Viieru, and W. Shyy, “Effects of Reynolds Number and Flapping Kinematics on Hovering Aerodynamics,” *AIAA Journal*, vol. 46, pp. 967–976, 2008.
- [95] H. T. Ahn and Y. Kallinderis, “Strongly Coupled Flow/Structure Interactions with a Geometrically Conservative ALE Scheme on General Hybrid Meshes,” *Journal of Computational Physics*, vol. 219, pp. 671–696, 2006.
- [96] I. Borazjani and F. Sotiropoulos, “Vortex-Induced Vibrations of Two Cylinders in Tandem Arrangement in the Proximity-Wake Interference Region,” *Journal of Fluid Mechanics*, vol. 621, pp. 321–364, 2009.
- [97] B. S. Carmo, S. J. Sherwin, P. W. Bearman, and R. H. J. Willden, “Flow-Induced Vibration of a Circular Cylinder Subjected to Wake Interference at Low Reynolds Number,” *Journal of Fluids and Structures*, vol. 27, pp. 503–522, 2011.
- [98] M. A. Kamoji, S. B. Kedare, and S. V. Prabhu, “Experimental Investigations on Single Stage Modified Savonius Rotor,” *Applied Energy*, vol. 86, no. 7-8, pp. 1064–1073, 2009.
- [99] R. Ricci, R. Romagnoli, S. Montelpare, and D. Vitali, “Experimental Study on a Savonius Wind Rotor for Street Lighting Systems,” *Applied Energy*, vol. 161, pp. 143–152, 2016.
- [100] M. H. Yang, G. M. Huang, and R. H. Yeh, “Performance Investigation of an Innovative Vertical Axis Turbine Consisting of Deflectable Blades,” *Applied Energy*, vol. 179, pp. 875–887, 2016.

- [101] K. Golecha, T. I. Eldho, and S. Prabhu, “Influence of the Deflector Plate on the Performance of Modified Savonius Water Turbine,” *Applied Energy*, vol. 88, pp. 3207–3217, 2011.
- [102] P. Jaohindy, H. Ennamiri, F. Garde, and A. Bastide, “Numerical Investigation of Airflow through a Savonius Rotor,” *Wind Energy*, vol. 17, no. 6, pp. 853–868, 2014.
- [103] V. D’Alessandro, S. Montelpare, R. Ricci, and A. Secchiaroli, “Unsteady Aerodynamics of a Savonius Wind Rotor: A New Computational Approach for the Simulation of Energy Performance,” *Energy*, vol. 35, pp. 3349–3363, 2010.
- [104] V. Shnayder, M. Hempstead, B. Chen, G. W. Allen, and M. Welsh, “Simulating the Power Consumption of Large-Scale Sensor Network Applications,” in *Proceedings of the 2nd International Conference on Embedded Networked Sensor Systems*, (Baltimore, Maryland, USA), pp. 188–200, ACM, 2004.
- [105] M. Shaheen and S. Abdallah, “Development of Efficient Vertical Axis Wind Turbine Clustered Farms,” *Renewable & Sustainable Energy Reviews*, vol. 63, pp. 237–244, 2016.