

École polytechnique de Louvain

Efficient and accurate simulation of the flow past moving bodies

Author: **Youri MARCHAL**

Supervisors: **Philippe CHATELAIN, Matthieu DUPONCHEEL, Grégoire WINCKELMANS**

Reader: **Jean-François REMACLE**

Academic year 2021–2022

Master [120] in Mechanical Engineering

Acknowledgements

I would like to address my deepest thanks to my advisors, Professor Philippe Chatelain, Doctor Matthieu Duponcheel and Professor Grégoire Winckelmans for their availability, their criticism and implication. I really appreciated the numerous discussions we had on the results over the academic year. I enjoyed the experience that they shared to me during those discussions.

Also, I address my warmest thanks to Philippe Billuart, who shared me its time, and helped me to conduct this thesis to its end. I am deeply grateful for the many hours spent discussing and reflecting throughout this academic year, and for his implication from the beginning to the end.

I also want to thank Professor Jean-François Remacle, for accepting to be the reader of my work.

I would like to give a special thank to my parents and sisters, who always brought me support, and enabled me to overcoming the difficulties during my studies. I am grateful for everything they have done for me, and enabled me to achieve.

Besides, I need to thank Claire Chouinard for the assistance she provided me for improving my English writing style.

My last thanks are directed to my friends, and I address a special thought to Marion Dubois, Celia Brand, Antonin Engel, Romain Weydisch, Florian Lefebvre, Lorenzo Salgado for the years I have spent trying to survive by living with them.

Computational resources have been provided by the Consortium des Équipements de Calcul Intensif (CÉCI), funded by the Fonds de la Recherche Scientifique de Belgique (F.R.S.-FNRS) under Grant N° 2.5020.11 and by the Walloon Region.

Abstract

Numerical methods in fluid dynamics have been developed for many years, now, with two main categories of method: Eulerian and Lagrangian methods, which both have their respective pros and cons. In order to achieve both the accuracy and computational efficiency of the numerical simulation of external flows, a new hybrid method has been developed by Billuart et al. [1]. The latter enables to have a very fine resolution of the flow in the near-wall region and to capture the small scale details of the near wake, while convecting the wake over long distances, for the 2D simulation of incompressible external flows. This method relies on the weak coupling between a body-fitted, finite difference solver for the Navier-Stokes equations in the velocity-pressure formulation, and the Vortex Particle-Mesh method. The present work therefore consists in an adaptation of the methodology developed by Billuart et al. [1] to moving rigid bodies. The method is verified for forced oscillations of a cylinder in the transverse direction to the free stream, and for a rotating cylinder in a free stream, experiencing the *Magnus effect*. The oscillation of a cylinder in the streamwise direction and the simulation of a heaving airfoil are presented as applications. Other applications to this work concern *Fluid Structure Interaction* (FSI), with a possible further step for adapting the method to soft bodies.

A brief introduction to the different methods used throughout this thesis is provided in Chapter 1, as well as a short review of the literature in the domain of forced moving cylinders and *Fluid Structure Interaction*. Then, the detailed methodology of each component solver as well as the coupling strategy between both solvers for moving bodies is presented in Chapter 2, for moving bodies. Afterwards, the first results of the present method are presented in Chapter 3 in order to verify as far as possible the present method. Three cases of transverse oscillations of a cylinder in a free stream at Reynolds $Re = 100$, and a cylinder rotating at three angular velocities in a free stream at $Re = 200$ are presented. After that, three applications are presented. In Chapter 4, an oscillating cylinder in the streamwise direction, and a heaving airfoil in the free stream at Reynolds $Re = 500$ are presented. Finally, the achievements of this work are summarized and the further possible developments are discussed in Chapter 5.

Contents

Acknowledgements	ii
Abstract	iii
Contents	iv
List of Figures	vi
List of Tables	ix
1 Introduction	1
1.1 Foreword	1
1.2 Scope of the work and literature review	2
1.2.1 Eulerian methods	2
1.2.2 Lagrangian vortex methods	2
1.2.3 Hybrid methods	3
1.2.4 Arbitrary Lagrangian-Eulerian formulation	4
1.2.5 Study of the flow past moving bodies	6
1.3 Objectives of the thesis	9
2 Methodology	10
2.1 Near-wall solver: FDoom	11
2.2 The Vortex Particle-Mesh method	12
2.3 Coupling procedure for moving bodies	14
2.3.1 Implementation of the methodology	15
2.3.2 Boundary conditions for the near-wall solver	19
2.4 Numerical stability and accuracy	22
2.5 Discussion on the coupling parameters	22
3 Verification	24
3.1 Transverse oscillation of a cylinder at $Re = 100$	25
3.1.1 Case 1: Locked configuration $Re = 100 - A^* = 0.25 - F = 0.9$	27
3.1.2 Case 2: Unlocked configuration $Re = 100 - A^* = 0.25 - F = 0.5$	31
3.1.3 Case 3: Locked configuration $Re = 100 - A^* = 1.0 - F = 0.9$	36
3.2 Rotating cylinder at $Re = 200$	39
3.2.1 Case 1: $Re = 200 - \alpha = 0.5$	41

3.2.2	Case 2: $Re = 200$ - $\alpha = 2.5$	45
3.2.3	Case 3: $Re = 200$ - $\alpha = 3.0$	49
3.2.4	Impact of the size of the domain	53
4	Applications	55
4.1	Inline oscillations of a cylinder at $Re = 500$	55
4.1.1	Case 1: $Re = 500$ - $A^* = 0.5$ - $F = 0.9$ without flow perturbation	57
4.1.2	Case 2: $Re = 500$ - $A^* = 0.5$ - $F = 0.9$ with flow perturbation	61
4.1.3	Case 3: $Re = 500$ - $A^* = 1.0$ - $F = 0.9$ without flow perturbation	64
4.2	Heaving airfoil at $Re = 500$	67
5	Conclusions	71
5.1	Discussion	71
5.2	Perspectives	71
A	Potential flow past a rotating cylinder	73
	Bibliography	74

List of Figures

2.1	Domain decomposition into a near-wall (NW) and VPM subdomains, and boundary conditions imposed on the VPM subdomain	14
2.2	Two subsets of particles and boundary conditions of the VPM solver	15
2.3	Correction of the VPM fields by the NW fields	17
2.4	Boundary condition of the NW solver: spatial interpolation of the VPM fields at sub-time $t^{n,m}$, correction of velocity for mass conservation, ghost tangential velocity computation	20
3.1	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.9$: contours of dimensionless vorticity in FDoom and VPM solvers near the body	27
3.2	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.9$: contours of dimensionless vorticity and pressure over the domain	28
3.3	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.9$: lift and drag coefficients from the present method and from Placzec et al.	29
3.4	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.9$: phase portraits of the lift and drag coefficients. Results from the present method and from Placzec et al.	29
3.5	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.9$: combined phase portrait of the lift and drag coefficients from the present method	30
3.6	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.9$: FFT of the lift coefficient	30
3.7	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.5$: contours of dimensionless vorticity and pressure over the domain	31
3.8	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.5$: lift coefficient from the present method and from Placzec et al.	32
3.9	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.5$: beating of the lift and drag coefficients	33
3.10	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.5$: phase portrait of the lift coefficient	34
3.11	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.5$: FFT of the lift coefficient	34
3.12	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.5$: contours of dimensionless vorticity near the body for $t/T_f \simeq 12.25$ to $t/T_f \simeq 13.0$	35
3.13	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.5$: contours of dimensionless vorticity near the body for $t/T_f \simeq 25.25$ to $t/T_f \simeq 26.0$	35

3.14	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 1$, $F = 0.9$: contours of dimensionless vorticity and pressure	36
3.15	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 1$, $F = 0.9$: contours of dimensionless vorticity near the body for $t/T_f \simeq 27.25$ to $t/T_f \simeq 28.0$	37
3.16	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 1$, $F = 0.9$: lift coefficient over 6 forced oscillations.	37
3.17	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 1$, $F = 0.9$: phase portraits of the lift and drag coefficients.	38
3.18	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 1$, $F = 0.9$: combined phase portrait of the lift and drag coefficients from the present method.	38
3.19	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 1$, $F = 0.9$: FFT of the lift coefficient.	38
3.20	Rotating cylinder - $Re = 200$, $\alpha = 0.5$: comparison of the contours of dimensionless vorticity over $12D$ downstream of the cylinder in FDoom and VPM solvers for the present method and the present method with the fixed Eulerian mesh.	41
3.21	Rotating cylinder - $Re = 200$, $\alpha = 0.5$: contours of dimensionless vorticity in FDoom and VPM solvers from the present method with fixed Eulerian mesh at $t^* \simeq 219.5$	42
3.22	Rotating cylinder - $Re = 200$, $\alpha = 0.5$: lift and drag coefficients. Results from the present method and the present method with the Eulerian mesh held fixed.	43
3.23	Rotating cylinder - $Re = 200$, $\alpha = 0.5$: phase portrait of the combined lift and drag coefficients. Results from the present method and the present method with the Eulerian mesh held fixed.	43
3.24	Rotating cylinder - $Re = 200$, $\alpha = 0.5$: Top: $\omega^*(t^* \simeq 220)$. Middle: $\bar{\omega}^*$ averaged over 21 Strouhal periods.	44
3.25	Rotating cylinder - $Re = 200$, $\alpha = 2.5$: contours of dimensionless vorticity at $t^* \simeq 205$	45
3.26	Rotating cylinder - $Re = 200$: comparison between the viscous case with $\alpha = 2.5$ and the inviscid case with $\alpha_{\text{pot}} = 1.3$: streamlines and colour plot of the velocity field	46
3.27	Rotating cylinder - $Re = 200$, $\alpha = 2.5$: zoom on the two saddle points	46
3.28	Rotating cylinder - $Re = 200$, $\alpha = 2.5$: $\omega^*(t^* \simeq 205)$, $\bar{\omega}^*$ averaged over $\Delta t^* \simeq 144$, $\bar{u}_x/U_\infty$ averaged over $\Delta t^* \simeq 144$	47
3.29	Rotating cylinder - $Re = 200$, $\alpha = 2.5$: dimensionless wall vorticity ω^* at $t^* \simeq 205$. Results from the present method, the present method with fixed Eulerian mesh, and Mittal and Kumar	48
3.30	Rotating cylinder - $Re = 200$, $\alpha = 2.5$: dimensionless skin friction τ_{wall}^* at $t^* \simeq 205$. Results from the present method and the present method with fixed Eulerian mesh	48
3.31	Rotating cylinder - $Re = 200$, $\alpha = 3.0$: contours of dimensionless vorticity at $t^* \simeq 137$	49
3.32	Rotating cylinder - $Re = 200$: comparison between the viscous case with $\alpha = 3.0$ and the inviscid case with $\alpha_{\text{pot}} = 1.75$: streamlines and colour plot of the velocity field	50
3.33	Rotating cylinder - $Re = 200$, $\alpha = 3.0$: zoom on the two saddle points	50
3.34	Rotating cylinder - $Re = 200$, $\alpha = 2.5$: Top: $\omega^*(t^* \simeq 137)$. Middle: $\bar{\omega}^*$ averaged over $\Delta t^* \simeq 144$. Bottom: $\bar{u}_x/U_\infty$ averaged over $\Delta t^* \simeq 144$	51

3.35	Rotating cylinder - $Re = 200$, $\alpha = 3.0$: dimensionless wall vorticity at $t^* \simeq 137$. Results from the present method, the present method with fixed Eulerian mesh, Mittal and Kumar, and Padrino and Joseph	52
3.36	Rotating cylinder - $Re = 200$, $\alpha = 3.0$: dimensionless skin friction from the present method and the present method with fixed Eulerian mesh	52
3.37	Rotating cylinder - $Re = 200$, $\alpha = 2.5$: comparison of the time evolution of the angle of the velocity vector for two domain sizes	54
3.38	Rotating cylinder - $Re = 200$, $\alpha = 2.5$: comparison of the time evolution of the lift and drag coefficients for two domain sizes	54
4.1	Inline oscillation of a cylinder - $Re = 500$, $A^* = 0.5$, $F \approx 0.9$: dimensionless vorticity field ω^* near the body at several times between $t/T_f \simeq 16.75$ and $t/T_f \simeq 17.5$	58
4.2	Inline oscillation of a cylinder - $Re = 500$, $A^* = 0.5$, $F \approx 0.9$: dimensionless vorticity field ω^* over the domain at several times between $t/T_f \simeq 16.4$ and $t/T_f \simeq 19.4$	59
4.3	Inline oscillation of a cylinder - $Re = 500$, $A^* = 0.5$, $F \approx 0.9$: drag coefficient over 5 periods of forced oscillation	60
4.4	Inline oscillation of a cylinder - $Re = 500$, $A^* = 0.5$, $F \approx 0.9$: phase portrait of the drag coefficient	60
4.5	Inline oscillation of a cylinder with flow perturbation - $Re = 500$, $A^* = 0.5$, $F \approx 0.9$: dimensionless vorticity field ω^* over the whole domain at $t^* \simeq 195$. . .	61
4.6	Inline oscillation of a cylinder with flow perturbation - $Re = 500$, $A^* = 0.5$, $F \approx 0.9$: lift and drag coefficients	62
4.7	Inline oscillation of a cylinder with flow perturbation - $Re = 500$, $A^* = 0.5$, $F \approx 0.9$: dimensionless vorticity field ω^* near the body at several times between $t^* \simeq 165$ and $t^* \simeq 170.6$	63
4.8	Inline oscillation of a cylinder - $Re = 500$, $A^* = 1.0$, $F \approx 0.9$: dimensionless vorticity field ω^* over the domain at $t^* \simeq 122$	64
4.9	Inline oscillation of a cylinder - $Re = 500$, $A^* = 1.0$, $F \approx 0.9$: drag coefficient over 5 periods of forced oscillation	65
4.10	Inline oscillation of a cylinder - $Re = 500$, $A^* = 1.0$, $F \approx 0.9$: phase portrait of the drag coefficient	65
4.11	Inline oscillation of a cylinder - $Re = 500$, $A^* = 1.0$, $F \approx 0.9$: dimensionless vorticity field ω^* near the body at several times between $t/T_f \simeq 22.75$ and $t/T_f \simeq 23.62$	66
4.12	Heaving airfoil - $Re = 500$, $A^* = 0.268$, $F = 0.5$: dimensionless vorticity field ω^* over the domain at $t^* \simeq 51.76$, corresponding to $t/T_f \simeq 7.75$	68
4.13	Heaving airfoil - $Re = 500$, $A^* = 0.268$, $F = 0.5$: dimensionless vorticity field ω^* near the body at several times between $t/T_f \simeq 6.75$ and $t/T_f \simeq 7.75$. The plots are saturated for $\omega^* > 10$	69
4.14	Heaving airfoil - $Re = 500$, $A^* = 0.268$, $F = 0.5$: lift and drag coefficients	70
4.15	Heaving airfoil - $Re = 500$, $A^* = 0.268$, $F = 0.5$: phase portrait of the lift and drag coefficients	70

List of Tables

3.1	Mesh parameters of the NW solver and VPM solver for the validation cases . . .	25
3.2	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.9$: force coefficients computed with wall quantities of the NW solver	29
3.3	Transverse oscillation of a cylinder - $Re = 100$, $A^* = 1.0$, $F = 0.9$: force coefficients computed with wall quantities of the NW solver.	36
3.4	Rotating cylinder - $Re = 200$, $\alpha = 0.5$: Strouhal number and Strouhal period based on the FFT of the lift coefficient.	42
3.5	Rotating cylinder - $Re = 200$, $\alpha = 0.5$: force coefficients computed with wall quantities of the NW solver averaged over 17 Strouhal periods.	43
3.6	Rotating cylinder - $Re = 200$, $\alpha = 2.5$: approximate position of the saddle points from the present method with the Eulerian mesh held fixed	46
3.7	Rotating cylinder - $Re = 200$, $\alpha = 2.5$: force coefficients computed with the averaged wall quantities from the NW solver	47
3.8	Rotating cylinder - $Re = 200$, $\alpha = 2.5$: position of the attachment and separation points at the cylinder surface, and positions extremum values of ω^*	48
3.9	Rotating cylinder - $Re = 200$, $\alpha = 3.0$: approximate position of the saddle points from the present method with the Eulerian mesh held fixed	50
3.10	Rotating cylinder - $Re = 200$, $\alpha = 3.0$: force coefficients computed with the averaged wall quantities from the NW solver, and values from literature	51
3.11	Rotating cylinder - $Re = 200$, $\alpha = 3.0$: position of the attachment and separation points at the cylinder surface, and positions extremum values of ω^*	52
3.12	Mesh parameters of the NW solver and VPM solver for the two domain sizes . .	53
3.13	Rotating cylinder - $Re = 200$, $\alpha = 2.5$: averaged force coefficients computed with the wall quantities from the NW solver	54
4.1	Mesh parameters of the NW solver and VPM solver for the cylinder oscillating inline at $Re = 500$	56
4.2	Mesh parameters of the NW solver and VPM solver for the heaving airfoil: parameters are values on the cylinder mesh before the regularized conformal transform.	67

Chapter 1

Introduction

1.1 Foreword

Industry is growing at an ever-increasing rate. To maintain this growth, new technologies that must be both simple to use and economically profitable are required. Hence, the development of numerical tools, one class of new technology, faces two main challenges: (1) being efficient in terms of economic and time constraints while (2) still achieving adequate accuracy. It is therefore important to find a trade-off between the accuracy of the results that we obtain and the computational cost of the engineering simulations.

This is especially true in the numerical study of the flow past immersed bodies. The latter has many applications in various domains. It has been studied for years in civil engineering applications (ocean pipes, off-shore platforms, bridges, towers, etc.), and aerodynamics. More recently, interests are also growing in the biological field, which also enables to develop bio-inspired technologies, like birds formation flight that could be applied to commercial flights. In addition to that, the wind energy field also rely on such studies. With the developments and improvements in the wind turbines designs in the last decades, the numerical simulation of the wakes and their interaction with other wind turbines is major for the improvement of wind farms. For vertical axis wind turbines, the interaction of the wake of each blade with the others is also of great interest. Moreover, with the Net Zero Emissions objective to achieve by 2050, the development of the renewable energies is expected to increase in next few years. Therefore, it can be expected that improvements will still be needed in that field, requiring therefore efficient and accurate numerical tools that enable to assess aerodynamic performances, together with the ability to evaluate the wake behaviour and its interaction with the surrounding.

Besides, a specific interest can be addressed to moving objects immersed in a flow. The movement of the immersed bodies can lead to unsteady behaviours of the wake, which are less well-known and which can be important to learn. Especially for applications where the wake interacts with other bodies. For example, the interaction of the wakes of three blades of a vertical axis wind turbine can be crucial to enhance the efficiency of the latter. Another application can be the change in the aerodynamic forces exerted on a pitching or heaving airfoil, as long as the description of its far wake. The behaviour of the wake is of great interest for applications where the detection of the wake from another aircraft has to be perceived and/or predicted.

1.2 Scope of the work and literature review

Since the previously presented applications have been studied for many years, a lot of different methods have been developed in order to take into account the presence of an immersed body in the flow. In computational fluid dynamics, the two broad categories of methods are the so-called *Eulerian* and *Lagrangian* (or vortex) methods. Of course, there exists many different methods based on those formulations, all of them having their own advantages and disadvantages. Hybrid methods have then been developed. The latter consist in coupling a Lagrangian method with an Eulerian method, so that the resulting method combines the assets of both. In addition to that, an important challenge that all methods should face is the ability to represent solid boundaries in the flow and to obtain information about wall quantities in an accurate and efficient manner. After that, an underlying challenge that is encountered is the ability of representing moving boundaries, which - potentially - may deform (but this will not be discussed in this work).

1.2.1 Eulerian methods

Eulerian methods are grid-based methods. In this family of methods, the three most widely used are the finite volumes, finite elements and finite differences methods. They can afford to be body-fitted, which leads to a high accuracy resolution of the flow in the near-body region, and, therefore, enables to capture with high precision the boundary layer, even at high Reynolds numbers. The main issue with such methods is their computational cost that is tremendous when the domain size increases. However, the advantage they have is that the mesh can be stretched, so that, near solid boundaries, the anisotropic character of boundary layers can be captured in a more efficient way than it would be by using vortex methods with increased number of particles [2]. Another difficulty with Eulerian solvers is to define a proper boundary condition at infinity and for the outflow, while it is naturally satisfied with vortex methods, as it will be discussed in Section 1.2.2.

1.2.2 Lagrangian vortex methods

Vortex methods [3, 4] for incompressible flow are based on the vorticity-velocity formulation of the Navier-Stokes equation since they rely on the fact that vorticity is carried by vortex elements (or particles) throughout the flow. Those methods are very efficient in problems of external flows where the vorticity is compact, i.e. low diffusion problems, as the evolution of vorticity is computed in a Lagrangian way. It follows that these methods are less efficient near solid interfaces since, in the boundary layers that develop in those regions, the viscous effects are significant. Such methods would require increasing the number of vortex particles in these regions of the flow in order to capture the flow fields more accurately. A first important advantage of Lagrangian methods is that they are not constrained by a CFL condition - as for Eulerian methods - which allows them to use larger time steps, since the method is more stable. Secondly, the dispersion and diffusion errors are low. Those methods are thus efficient to simulate the convection of vortical structures over long distances, as vorticity is only present in boundary layers and in the wakes for external flows. Finally, the far-field unbounded boundary conditions are enforced naturally.

However, a few well-known drawbacks are also related to the inherent characteristics of the method already presented here above. First, the fact that the vortex particles, which are the

support of information, are convected by the flow will entail spreading and clustering of the computational elements in some regions. This is the so-called *particle depletion* drawback, which deteriorates the accuracy of the method. In order to overcome this particle depletion, a redistribution procedure can be applied by interpolating the particles' information on a fixed position of the domain. Secondly, the particles that carry information are isotropic, i.e. the information they carry is uniform over their volume. The method is thus not able to capture with high precision the anisotropic nature of boundary layers. The only way to balance that is to increase the number of particles in those regions, which drastically increases the computational cost of the method. Finally, this leads to another challenge of vortex methods: obtaining an accurate treatment of solid boundaries. Immersed interface methods, panel methods, and penalization methods have been developed in many different ways, in order to model solid boundaries. However, the main issue with those methods is the to capture with a sufficient level of accuracy and efficiently (from a computational cost point of view) the flow quantities near the solid boundaries, while it is less costly and easier in Eulerian methods.

1.2.3 Hybrid methods

In light of the two previous sections, it can be understood that methods which could benefit from advantages of both Eulerian and Lagrangian aspects are appealing. Such methods are called *hybrid methods*. They rely on a coupling between an Eulerian and a Lagrangian solvers, so that the resulting method could face the issues of taking into account solid boundaries, and efficiently compute the flow quantities in this region in vortex methods while decreasing the size of an Eulerian domain, so that it only carries the flow solution in that near-wall region.

The two challenges for using coupled solvers are, first, that information needs to be exchanged between the two solvers, and, secondly, that correct boundary conditions must be enforced. Indeed, since the two solvers work differently, the flow quantities owned by each could present inconsistencies; the boundary condition must thus enforce consistency in-between the two solvers. A subdomain must be defined for both solvers: the Eulerian solver covers the near-wall region of the computational domain, while the Lagrangian solver covers the rest of the domain. There exist two kinds of coupling approaches: strong couplings and weak couplings. The main drawback of strong couplings is that the latter need iterations (using Schwarz alternating method) to provide the boundary condition, which is computationally expensive. Since the present work is only based on a weak coupling approach, no further description of strong coupling is provided but more details about strong couplings can be found in [5, 6, 7, 8]. Regarding the weak coupling methods that have already been developed for incompressible external flows [2, 5, 9], they suffer from several shortcomings concerning a lack of accuracy and inconsistent velocity and vorticity fields at the outer boundary. Among other things, they also do not conserve the circulation. A recent work achieved by Billuart et al. [1] tackles the imprecision of the previous works with a second order rate of convergence.

Regarding the work achieved by Billuart et al. [1], the principle is to divide the computational domain into subdomains. The subdomains of each of the solvers are fully overlapped. According to the respective advantages of the Eulerian and Lagrangian methods that have been presented previously, an Eulerian solver is used for the near-wall subdomain while a Lagrangian solver is suited for the wake flow. Indeed, as Eulerian solvers can be grid-based, and body-fitted, they can solve the flow in the near-wall region, where viscous effects are important by preserving the

anisotropic character of the boundary layers thanks to the use of a stretched grid. On the other hand, Lagrangian solvers are efficient for convecting compact vortical structures in the wake flow. Two important principles are therefore settled:

1. The Lagrangian method solves the flow over the entire domain, and its flow information is used in order to provide the outer boundary condition to the Eulerian solver.
2. The Eulerian domain covers only the near-wall region and, using the boundary condition provided by the Lagrangian solver, it computes with a higher level of accuracy the flow over its domain. Thus, it can afterwards correct the flow information of the Lagrangian solver on the part of the domain where both solvers are overlapped.

1.2.4 Arbitrary Lagrangian-Eulerian formulation

Taking into account distortions of the continuum in fluid mechanics problems solved with grid-based methods might be challenging when using classical Lagrangian or Eulerian kinematical descriptions, especially when studying moving solid boundaries. The *Arbitrary Lagrangian-Eulerian* (ALE) formulation has therefore been introduced. According to Donea, Giuliani, and Halleux [10], the first ALE methods were developed by Noh [11], Trulio [12], and Hirt, Amsden, and Cook [13] in the 1960s-1970s for finite difference methods. Then, the ALE formulation has been specified for incompressible, and viscous flows by Hughes et al. [14]. Finally, a good reference giving an overview of the ALE formulation and ALE methods is Donea, Huerta, Ponthot, and Rodriguez-Ferran [15]. The ALE formulation has been introduced with the intention of combining the advantages of both the Lagrangian and Eulerian kinematical descriptions.

The ALE formulation, and the underlying ALE methods that have been developed, try to mitigate the mesh distortions, especially when dealing with interfacial problems (like fluid-solid and fluid-fluid interfaces, e.g.). Indeed, a Lagrangian description enforces the mesh nodes to follow the continuum, which can therefore lead to large distortions of the mesh. While a Lagrangian description easily enables to track the interface, the Eulerian description better captures the details of the interface. On the other hand, the ALE approach can account for moving - and deforming - meshes. It is especially convenient in the hybrid methods framework that has been introduced in the previous section. In fact, as explained by Donea, Huerta, Ponthot, and Rodriguez-Ferran [15], the ALE kinematical description relies on the definition of a new frame, which is the *referential frame*; the latter is described by the considered mesh coordinates χ , while the classical Lagrangian and Eulerian frames are described by the material coordinates $\mathbf{X}$ and the spatial coordinates $\mathbf{x}$, respectively. There are therefore three velocities that are associated to the mappings between Lagrangian, Eulerian and ALE frames given by Eqs. 1.1, 1.2, and 1.3 respectively.

$$\frac{D\mathbf{x}}{Dt} = \left. \frac{\partial \mathbf{x}(\mathbf{X}, t)}{\partial t} \right|_{\mathbf{x}} = \mathbf{u}(\mathbf{X}, t) , \quad (1.1)$$

$$\left. \frac{\partial \mathbf{x}(\chi, t)}{\partial t} \right|_{\chi} = \mathbf{U}_{\chi}(\chi, t) , \quad (1.2)$$

$$\left. \frac{\partial \chi(\mathbf{X}, t)}{\partial t} \right|_{\mathbf{X}} = \mathbf{w} , \quad (1.3)$$

where $\mathbf{u}$ is the classical material velocity, $\mathbf{U}_\chi = \mathbf{U}_{\text{mesh}}$ is the mesh velocity, and $\mathbf{w}$ is the particle velocity as seen by the referential frame of reference (i.e. the frame associated to the mesh). Finally, the actual convective velocity as seen by the laboratory frame is given in Eq. 1.4.

$$\mathbf{u} - \mathbf{U}_{\text{mesh}} = \frac{\partial \mathbf{x}}{\partial \chi} \cdot \mathbf{w} . \quad (1.4)$$

In the context of hybrid methods, the ALE kinematical description allows to keep the grid of the Eulerian method undeformed, while moving at its reference frame velocity $\mathbf{U}_\chi = \mathbf{U}_{\text{mesh}}$. It follows that, when combining an Eulerian method with a Lagrangian method, the reference frame of the ALE description is naturally determined as the mesh of the Eulerian method.

Another important aspect of the ALE kinematical description is the fact that it provides modified formulations of the conservative laws which are very simple. Indeed, the weak forms of the conservation laws are obtained from the Eulerian formulation but, in all the convective terms, the material velocity $\mathbf{u}$ has to be replaced by the actual convective velocity $\mathbf{u} - \mathbf{U}_{\text{mesh}}$, which takes into account the mesh motion. As an example, Eq. 1.5 provides the ALE formulation of the momentum conservation:

$$\rho \left. \frac{\partial \mathbf{u}}{\partial t} \right|_\chi + \rho (\mathbf{u} - \mathbf{U}_{\text{mesh}}) \cdot \nabla \mathbf{u} = \nabla \cdot \boldsymbol{\sigma} \quad (1.5)$$

where χ is the reference frame associated to the mesh, $\mathbf{u}$, and $\boldsymbol{\sigma}$ are respectively the usual notations for the material velocity, and the stress tensor. The stress tensor is given by

$$\boldsymbol{\sigma} = -p \mathbf{I} + 2\mu \mathbf{d} , \quad (1.6)$$

with p the pressure, μ the dynamic viscosity, $\mathbf{I}$ the identity tensor, and $\mathbf{d}$ the rate of strain tensor, so that, for μ constant, $\nabla \cdot \boldsymbol{\sigma} = -\nabla p + \mu \nabla^2 \mathbf{u}$ is obtained. Using Eq. 1.6 in Eq. 1.5, it can finally be obtained that

$$\rho \left. \frac{\partial \mathbf{u}}{\partial t} \right|_\chi + \rho (\mathbf{u} - \mathbf{U}_{\text{mesh}}) \cdot \nabla \mathbf{u} = -\nabla p + \mu \nabla^2 \mathbf{u} , \quad (1.7)$$

The ALE method applied to grid-based methods requires the use of a remeshing procedure for taking into account the mesh movement. According to Donea, Huerta, Ponthot, and Rodriguez-Ferran [15], two kinds of remeshing procedures exist: (1) a *mesh regularization*, or (2) a *mesh adaptation*. The former minimizes the mesh distortion, while the latter allows larger mesh distortions in order to keep the finest mesh near the regions of the computational domain where the gradients are strongest. In other words, the *mesh regularization* corresponds to a prescribed motion of the mesh, while the *mesh adaptation* optimizes the mesh movement in a manner to keep a homogeneous accuracy of the method over the domain. In this work, since the bodies are rigid, a prescribed mesh motion is thus imposed; the latter is applied to any node of the mesh, so that the mesh is not deformed over the simulation.

In summary, the ALE description selects the most suitable description of the continuum, depending on the studied case. The simple implementation of the moving mesh through the ALE kinematical description makes ALE methods appropriate for the study of the flow past moving boundaries. It is especially interesting in hybrid methods, where both the Eulerian and Lagrangian methods are used, since they allow to choose as the reference frame solid boundaries, which can either deform or remain rigid throughout the simulation.

1.2.5 Study of the flow past moving bodies

As briefly mentioned in the previous section when discussing about the advantages of the ALE description, the latter especially complies with moving bodies. Besides, the influence of moving bodies can also be of great interest. Indeed, the unsteady effects on the wake developed behind a body strongly influence the flow seen by others that are located downstream with respect to the first one. Moreover, the interaction between the body itself and its own wake can be observed.

The study of the flow behaviour past moving cylinders was of great interests in the end of the previous century, but, nowadays, few numerical studies are considering this kind of application. The interest for forced oscillating cylinder in a flow grew from the 1970s, after the first few studies were published in the 1960s. It has been discovered that the frequency of oscillation together with the amplitude of oscillation of an immersed cylinder strongly influence the vortex shedding patterns, and the frequency of the lift and drag forces. Among others, Bishop and Hassan (1964) [16], Koopman (1967) [17], and Williamson and Roshko (1988) [18] studied experimentally various cases of oscillating cylinders. These former studies depict the main results that can be learned from forcing the cylinder to oscillate in the transverse direction of a stream. However, regarding the oscillations of a cylinder in the streamwise direction, the studies are more scarce in the literature.

Namely, Bishop and Hassan [16] observed that, when the cylinder is forced to oscillate, there are two origins to the oscillations of the hydrodynamic forces: there are contributions coming (1) from the *free oscillations* of the wake, and (2) from the forced oscillations of the cylinder. These two contributions lead to a behaviour of the wake that is influenced by both phenomena. In addition to that, they claim that there exists a range of forced frequencies that lead to a synchronization of the Strouhal frequency with the forced frequency; this phenomenon is called *lock-in*. Outside of this *lock-in* range, they state that the combination of the two contributions in the hydrodynamic forces mentioned above can lead to a loss of the Strouhal frequency in the forces signal; in other words, the *free oscillations* of the wake do not correspond to the Strouhal frequency anymore, and the new frequency changes depending on whether the forced frequency is larger or lower than the Strouhal frequency. Though, in the *lock-in* range, the amplitude and phase of the forces change depending on the forced frequency. In other words, they observed that the phase between the forced motion of the cylinder and the lift force changes drastically near the Strouhal frequency of the cylinder, with a jump that occurred in that region of frequency.

Koopman [17] reported on a map the coupled values of amplitude and frequency that are the boundaries of the lock-in phenomenon, for three values of Reynolds number ($Re = 100, 200$, and 300). This map has been used by other authors, as a reference for comparison.

Later, Williamson and Roshko [18] studied various cases of transverse oscillations, by varying both the amplitude of oscillation (up to five cylinder diameters) and the frequency. They identified several vortex patterns associated to the region of the (λ, A) map that they built in [18], well-known as the *Williamson and Roshko map*. This map identifies the wake patterns corresponding to the period and the amplitude of the oscillations imposed for a range of Reynolds number $300 < Re < 1000$. The parameters mentioned above are respectively identified by the *wavelength ratio* $\lambda/D = T_f U_\infty/D$ and the *amplitude ration* A/D , where T_f is the period of oscillation, D is the diameter of the cylinder, and A is the amplitude of the movement imposed. In their study, they investigated a wide range of the map; they thus investigated movements of

large amplitude, that had not been performed before. They identified three main wake patterns near the *lock-in* region. These are defined based on the vortices shed into the wake during each cycle of oscillation, and they are referred by the following modes:

1. $2S$ mode: in each cycle, two successive vortices of opposite sign are shed. This mode is similar to the classical vortex shedding of a fixed cylinder.
2. $2P$ mode: in each cycle, two pairs of vortices of opposite sign are shed, but the latter are not convected on the line $y/D = 0$ in the wake, they are shifted above and below this center line; i.e.: one of the two pairs is convected above $y/D = 0$, and the other below.
3. $P - S$ mode: in each cycle, a pair of opposite signs vortices, followed by a single vortex are shed into the wake. The pair of vortices is always shed either above or below y/D , while the single vortex is convected in the wake on the other side of the center line. The pair of vortices or the single vortex is shed above or below $y/D = 0$ depending on the flow history.

They also state that the vortex shedding patterns that can be observed for the $2S$ and $2P$ modes is due to the roll-up of four region of vorticity in the near-wall region over one cycle of oscillation. The rolling-up of these vorticity regions is due to the acceleration of the cylinder in the cycle, according to Williamson and Roshko [18]; the latter can be identified for large values of λ , which leads to a $2P$ mode of vortex shedding. Otherwise, two regions of vorticity (of the same sign) are merged, and, therefore, the mode that can be observed is a $2S$ mode. Those four vortices play an essential role in the forces applied on the cylinder, and they strongly depend on the wavelength of the cylinder movement λ . Finally, they also attribute the jump of phase between the cylinder movement and the lift force - observed by Bishop and Hassan [16] - to a transition between two modes of vortex shedding patterns. Indeed, Bishop and Hassan [16] did only force measurements; they did not visualize the wake. On the other hand, Williamson and Roshko [18] did not investigate the hydrodynamic forces in their experiments, they only examined the wake patterns.

In addition to the famous experimental works mentioned above, numerical studies have been performed by, among many others, Meneghini and Bearman (1995) [19], Blackburn and Henderson (1999) [20], Mittal and Kumar (1999) [21], Anagnostopoulos (2000) [22], Leontini (2006) [23], Nobari and Naredan (2006) [24], Placzec (2008) [25], Kumar, Navrose and Mittal (2016) [26]. Although these numerical works are more recent in comparison with the previous experimental studies mentioned, there were also older numerical studies published in the 1970s. However, the number of numerical investigations sharply increased in the 1990s. Most of the numerical techniques used were finite elements methods (e.g.: Anagnostopoulos [22], Nobari and Naredan [24], Kumar, Navrose and Mittal [26]) and spectral elements methods (e.g.: Blackburn and Henderson [20], Leontini [23]); only a few used a finite difference or finite volume methods (e.g.: Placzec [25]). In most of the studies conducted, the authors try to evaluate the lock-in for some frequencies and amplitudes, and their results are compared with the observations reported by Williamson and Roshko [18]; more precisely, they also identify the synchronization of the wake oscillations with the forced frequency of the cylinder, and identify the wake patterns according to the *Williamson and Roshko map*. In fact, the *Williamson and Roshko map* has become a cornerstone for the identification of the wake patterns for a transversely oscillating cylinder. However, the very recent numerical studies of transversely oscillating cylinders are scarce; there is not a lot of research conducted on this subject anymore. In addition to that,

the results provided by the mentioned studies are often difficult to use as a reference, since many of them chose their own pairs of parameters (i.e.: amplitude and frequency of oscillation), and Reynolds number, in such a way that the same case is nearly never investigated by several references. This kind of movement will be investigated with the present work in Chapter 3, Section 3.1.

Studies concerning oscillating cylinders in the streamwise direction, are even more scarce (even old) in the literature. Hurlbut et al. (1982) [27] performed a few numerical simulations of an oscillating cylinder in the streamwise direction, using a finite difference method at $Re = 1, 10$, and 100 . They also performed three simulations of transverse oscillations at $Re = 80$. Nobari and Naredan (2006) [24], and Leontini (2011) [28]. Due to the lack of references available, it has been decided to present one case of inline oscillations of a cylinder at $Re = 500$ as an application. The results are discussed in Chapter 4, Section 4.1.

Another kind of movement of cylinder has been investigated by, namely, Mittal and Kumar (2003) [29]; they numerically simulated the rotation of a cylinder for various dimensionless rates of rotation. Padrino and Joseph (2006) [30] also performed a few cases and compared their results to those of Mittal and Kumar [29]. This kind of movement will be investigated with the present work in Chapter 3, Section 3.2.

Interest also grew for the oscillations undergone by immersed bodies due their own vortex shedding, or to an oscillating flow while experiencing external forces such as gravity, friction, spring and damper forces. This corresponds to the well-known *Fluid Structure Interaction* (FSI) and *Vortex Induced Vibrations* (VIV) applications. Even though vortex induced vibrations were investigated in the early 1970s, they still remain a subject of many studies nowadays, although the exact subjects of studies published has evolved. In fact, the first studies were conducted on the effects on elastically mounted cylinder with one or two degree(s) of freedom, as explained in the review made by Williamson and Govardhan (2004) [31]; specifically, the vortex shedding patterns and the corresponding vibration mode, and the energy transfer between the fluid and the body are investigated. The coupling between the body movement and the vortex shedding and forces frequency is the basis of the VIV investigations. This subject is closely related to the forced oscillations observations that are made from the previously mentioned studies. Notably, the results from Williamson and Roshko [18] are often used to characterize the wake. However, the number of studies conducted on elastically mounted cylinders is significantly more important in comparison to the investigations on forced oscillations. The analyses of such elastically mounted cylinders focused, among other objectives, on the amplitude that can be reached by varying the spring and damper values, the existing modes corresponding to the induced vibrations undergone by the cylinder, the transition between those modes, and, more practically, the consistency to scale up those results to real applications. As it was the case for numerical simulations of forced oscillations, in the VIV problems, ALE methods are also widely used, especially in Finite Elements methods, as they enable to mitigate the mesh distortions. For instance, Singh and Mittal (2005) [32], and Prasanth and Mittal (2008) [33] used a finite element method. Even though such simulations could be performed with the method presented in this work, no further details are provided about that part of the VIV studies since such cases have not been investigated. Other subjects have been investigated in the FSI field. Indeed, nowadays, the interest of FSI focuses more on real applications in civil engineering structures which are subjected to oscillatory fluid flows like (e.g.: bridges, wind turbines towers, oceanic risers, ocean pipes, etc.), bio-propulsion, or any other field which would benefit from the energy harvesting

from the surrounding fluid. This domain is therefore very broad and has many applications in the real life. The studies linked to VIV and FSI applications is abundant nowadays, and the applications are as wide as they are interesting.

1.3 Objectives of the thesis

Since the present work is based on the method developed by Billuart et al. [1], some challenges concerning the adaptation of the coupling methodology were not expected initially. Indeed, as it will be further explained in Chapter 2, Section 2.3.2, a part of the coupling procedure has required an unexpected adaptations, which therefore required several trials in the implementation of the concerned step of the methodology. The latter is related to pressure oscillations that are generated by the boundary condition in the finite difference (FD) method, which solves the flow in the near-wall (NW) region. These pressure oscillations were also observed by Billuart et al. [1], but with a lower impact on the aerodynamic forces. Investigations have been made in order to determine whether the origin of the pressure oscillations observed in the present work are also related to the redistribution frequency of the particles of the VPM method, as Billuart et al. [1] observed. These analyses will be more deeply examined in Chapter 2, Section 2.5. The main objective of this Master thesis is therefore to further develop and validate the coupling methodology developed by Billuart et al. [1] in the case of moving bodies.

A verification of the methodology is presented in Chapter 3, in order to assess the accuracy of the methodology developed in this work. The verification is done for two kinds of movements of a cylinder: (1) a transverse oscillation of the cylinder in a free stream, and (2) a rotating cylinder in a free stream. After that, two kinds of application are presented in Chapter 4: (1) a cylinder oscillating in the streamwise direction, and (2) a heaving airfoil. Finally, the achievements and discussions on the present work are summarized in Chapter 5.

Chapter 2

Methodology: coupling between an Eulerian and Lagrangian methods

The coupling procedure for handling moving bodies is based on the work performed by Billuart et al. [1]. In what follows, the methodology presented only relies on one moving body with general prescribed movements, even though the latter consists in adapting and modifying the coupling method for one body at rest developed and more deeply described by Billuart et al. [1]. The implementation of moving bodies in the coupling method can be divided into two main parts.

First, the handling of a moving mesh is performed in the Finite Difference on orthogonal mesh (FDoom) solver - which is an in-house Finite Difference (FD) solver for the velocity-pressure formulation of the 2D incompressible Navier-Stokes equations - through the implementation of an *Arbitrarily Lagrangian-Eulerian* (ALE) formulation, as described in [15]. Let us recall that the goal of ALE methods is to combine the advantages of both the Lagrangian and Eulerian kinematical descriptions, and that a specific aspect of the latter is that the conservative laws are expressed in the Eulerian formulation, but with the material velocity $\mathbf{u}$ that has to be replaced by the actual convective velocity $\mathbf{u} - \mathbf{U}_{\text{mesh}}$ in all the convective terms. We recall the weak form of the momentum conservation that was previously presented in Eq. 1.7, Section 1.2.4:

$$\rho \left. \frac{\partial \mathbf{u}}{\partial t} \right|_{\chi} + \rho (\mathbf{u} - \mathbf{U}_{\text{mesh}}) \cdot \nabla \mathbf{u} = -\nabla p + \mu \nabla^2 \mathbf{u} ,$$

In this work, the method that is used is not an ALE method, since there are several grids and methods employed, and, thus, that the whole body-fitted grid of the Eulerian method remains undeformed. However, the formulation of the convective term in the Navier-Stokes equations relies on the ALE formulation, allowing us to take into account the motion of the body. In addition to that, without the coupling with the VPM solver, the formulation allows easily to take into account a movement of the body, without deforming the mesh (even though, in the coupling procedure, the mesh of the near-wall solver is actually moving on the VPM background mesh, see later in Section 2.3.1).

Secondly, the coupling between FDoom and the Vortex-particle mesh (VPM) solver must also be adapted to the moving mesh. Indeed, in this case, the Eulerian domain (i.e.: the domain on which the method used to solve the flow is an Eulerian method) moves in a Lagrangian way

in the inertial frame. On the other hand, the Lagrangian domain (i.e. the domain on which the VPM method is used to solve part of the flow) remains fixed. As already discussed, this coupling procedure presents two main challenges: an accurate transfer of information between the two solvers and the enforcement of a correct boundary condition, so that the flow fields remain consistent at the interface between the two solvers.

In what follows, the Eulerian and Lagrangian solvers are briefly presented individually, in Sections 2.1 and 2.2 respectively. Then, the coupling procedure between the two solvers is presented in Section 2.3.1.

2.1 Near-wall solver: FDoom

For the near-wall region, a body-fitted Finite Difference on orthogonal mesh (FDoom) solver is used in order to capture accurately the boundary layer and small scale details of the near wake by benefiting from the anisotropic character of a stretched grid. In such an orthogonal mesh, the velocity components are expressed in a local coordinates system, which is defined at each considered grid position. The two velocity components are therefore orthogonal and oriented in the normal and tangential directions of the local grid frame.

The solver is based on a classical fractional step method [34, 35, 36] with computation of a prediction of the velocity which is reprojected. The spatial discretization ensures a second order accuracy and is based on a staggered Marker and Cell (MAC) mesh [37] (the schemes are adapted to the local frames linked to the stretched orthogonal mesh). The solver being based on the velocity-pressure formulation of the Navier-Stokes equations, the temporal integration scheme used for the convective term is an explicit Adams-Bashforth scheme of order 2, while, for the diffusive term, an implicit Crank-Nicholson scheme is used. Therefore, the fractional time steps are given by:

$$\frac{(\mathbf{u}^* - \mathbf{u}^n)}{\Delta t^+} = \left(1 + \frac{1}{2} \frac{\Delta t^+}{\Delta t^-}\right) \mathbf{H}_h^n - \frac{1}{2} \frac{\Delta t^+}{\Delta t^-} \mathbf{H}_h^{n-1} - \nabla_h \phi^n + \frac{1}{2} \nu (\nabla_h^2 \mathbf{u}^* + \nabla_h^2 \mathbf{u}^n) \quad (2.1)$$

$$\nabla_h^2 g = \frac{1}{\Delta t^+} \nabla_h \cdot \mathbf{u}^* \quad (2.2)$$

$$\frac{(\mathbf{u}^{n+1} - \mathbf{u}^*)}{\Delta t^+} = -\nabla_h g \quad (2.3)$$

$$\phi^{n+1} = \phi^n + g \quad (2.4)$$

$$P^{n+1} = \phi^{n+1} - \frac{\nu \Delta t^+}{2} \nabla_h^2 g \quad (2.5)$$

where $\mathbf{H}_h \triangleq ((\mathbf{u} - \mathbf{U}_{\text{mesh}}) \cdot \nabla \mathbf{u})_h$ is the convective term that takes into account the ALE kinematical description of the mesh ($\mathbf{U}_{\text{mesh}}$ is the mesh velocity expressed in the inertial frame of reference), $\nu \triangleq \mu/\rho$ is the kinematic viscosity, $P \triangleq (p/\rho)$ is the kinematic pressure, g is a scalar, and Δt^+ and Δt^- are respectively the current and previous time steps.

According to the fractional time step method [34, 35, 36], Eq. (2.1) first enables to obtain a prediction $\mathbf{u}^*$ of $\mathbf{u}^{n+1}$. Then, Eq. (2.2) - obtained from Eq. (2.3) together with the divergence-free constraint on $\mathbf{u}^{n+1}$ - is a Poisson equation to solve in order to compute the scalar g . In order to solve the Poisson equation, a homogeneous Neumann condition must be imposed on

the boundaries of the domain, so that the normal components of the velocity predictor and the exact velocity at the next time step are equal: $u_n^* = u_n^{n+1}$. This Neumann condition is given by: $\frac{\partial g}{\partial n} = 0$. After that, Eq. (2.3) corresponds to the reprojection of the predictor $\mathbf{u}^*$ into a solenoidal field. Finally, Eqs. (2.4) and (2.5) enable to obtain the pressure field at the next time step. It is important to mention that, due to the homogeneous condition that is used to solve the Poisson equation Eq. (2.2), the normal component at the next time step must be beforehand known. That information will come from the second component of the hybrid solver; that is, the VPM solver. The way the normal component of the velocity field is computed on the external boundary of the FDoom domain will be explained in details in Section 2.3.

Concerning the wall conditions of the FDoom solver, Dirichelet boundary conditions are used. The latter enforce the value of the normal and tangential components of the velocity field to the mesh velocity projected onto the local orthogonal frame (even though the tangential velocity at the wall is actually obtained by attributing the correct value to ghost points):

$$u_n(\mathbf{x}_{\text{wall}}) = \mathbf{U}_{\text{mesh}}(\mathbf{x}_{\text{wall}}) \cdot \hat{\mathbf{e}}_n(\mathbf{x}_{\text{wall}}) , \quad (2.6)$$

$$u_s(\mathbf{x}_{\text{wall}}) = \mathbf{U}_{\text{mesh}}(\mathbf{x}_{\text{wall}}) \cdot \hat{\mathbf{e}}_s(\mathbf{x}_{\text{wall}}) , \quad (2.7)$$

where $\mathbf{U}_{\text{mesh}}(\mathbf{x}) = U_{\text{mesh}}(\mathbf{x}) \hat{\mathbf{e}}_x + V_{\text{mesh}}(\mathbf{x}) \hat{\mathbf{e}}_y$ is the mesh velocity field in the inertial frame of reference, $\hat{\mathbf{e}}_n(\mathbf{x})$ and $\hat{\mathbf{e}}_s(\mathbf{x})$ are, respectively, the local normal and tangential directions of the grid. Obviously, when the mesh velocity is zero, those boundary conditions end up being no-through flow and no-slip boundary conditions.

Finally, the conservation of mass must be enforced over the Eulerian domain. When the body is rigid, there is no deformation of the wall. Thus there is no flux at the wall by symmetry. The conservation of mass is enforced by correcting uniformly the normal component of the velocity field on the outer boundary, so that the total flux through the boundary is zero, as translated in Eq. (2.8).

$$\oint_{\partial\Omega_{\text{NW}}^{\text{outer}}} u_n dS = 0 . \quad (2.8)$$

This is performed by removing homogeneously the excess of velocity on each grid point of the external boundary. However, when the body is moving, Eq. (2.6) imposes a flux at the wall. Therefore, the correction of the flow on the entire domain must now take into account that additional flux¹. Since the flux at the wall must not be changed, in order to verify Eq. (2.6), the total flux on the inner and outer boundaries are computed and the correction is, again, only applied on the outer boundary.

2.2 The Vortex Particle-Mesh method

The Vortex Particle-Mesh (VPM) method is an in-house solver that can already be considered as a *hybrid vortex method* since there are both Lagrangian and Eulerian components used in the solver. It is based on the vorticity-velocity formulation of the Navier-Stokes equations for incompressible flows in 2D, given in Eq. (2.9).

$$\frac{D\omega}{Dt} = \frac{\partial\omega}{\partial t} + \mathbf{u} \cdot \nabla\omega = \nu \nabla^2\omega . \quad (2.9)$$

¹For rigid bodies, the surface integral would be zero, but, for deforming bodies, it could be non-zero.

As already mentioned, there are two components in the VPM solver. First, the Lagrangian component comes from the particles used in order to discretize the convective term of Eq. (2.9). As described in [3, 4] (and reduced to 2D instead of 3D), the 2D particles carry integral information over their surface S_p about a certain physical field, which - in this case - is the vorticity field. The so-called *particle strength* α_p is defined by:

$$\alpha_p = \int_{S_p} \omega dS . \quad (2.10)$$

Here, we have that $S_p = h_{\text{VPM}}^2$, as the VPM grid size is uniform (i.e.: $\Delta x = \Delta y = h_{\text{VPM}}$). Since the particles' position is formulated in a Lagrangian fashion, the position $\mathbf{x}_p$ of each particle p over time can therefore be computed since the particles are convected by the flow. Eventually, the Lagrangian discretization of the particles entails the three following differential equations (two for the position and one for the particle's strength):

$$\frac{d\mathbf{x}_p}{dt} = \mathbf{u}(\mathbf{x}_p) \quad (2.11)$$

$$\frac{d\alpha_p}{dt} = \nu \int_{S_p} \nabla^2 \omega dS . \quad (2.12)$$

Secondly, the Eulerian component comes from the *mesh* that is used in order to recover the velocity field from the vorticity field by solving a Poisson equation on a grid. Indeed, when considering the Helmholtz decomposition of the velocity field: $\mathbf{u} = \mathbf{U}_\infty + \mathbf{u}_\omega$, and knowing that the relation between velocity and vorticity (in 2D) is given by: $\omega \hat{\mathbf{e}}_z = \nabla \times \mathbf{u}$, the velocity field can be recovered from the vorticity field (on the grid) through the resolution of the following Poisson equation:

$$\nabla \times (\omega \hat{\mathbf{e}}_z) = -\nabla^2 \mathbf{u}_\omega . \quad (2.13)$$

The use of a grid in order to be able to compute the velocity field has two main advantages: it is easier to compute spatial differential operators on a grid, and it allows to use fast and efficient solvers for solving the Poisson equation Eq. (2.13).

Concerning the boundary conditions implemented in the VPM solver, it uses the *FLUPS* library [38] in order to implement unbounded and natural outflow boundary conditions.

As a summary of the VPM method, the differential equations given in Eqs. (2.11) and (2.12) are temporally integrated on the particles using a Runge-Kutta of order 3 scheme (RK3), while the spatial differential operators are evaluated on a grid; the velocity field is computed on that grid as well through the resolution of Eq. (2.13).

Underlying the previous assets and working principles of having a coupled method for the VPM solver, it follows that exchanges of information are required. Therefore, two interpolation procedures are defined: (1) interpolation of information from the grid points (or mesh) to the particles (*M2P*) or (2) from particles to grid points (*P2M*). Both *M2P* and *P2M* interpolation procedures are performed using the high order M_4' scheme [39]. Finally, another important feature of the VPM method is its ability to redistribute the vorticity field from the particles at fixed grid locations. The vorticity carried by particles is thus interpolated from particles to grid points, which can be seen as a reset of the particles. This enables to avoid the *particle depletion* drawback that was already discussed in Section 1.2.2.

2.3 Coupling procedure for moving bodies

The coupling strategy that is presented in this section is a slight adaptation of the approach presented by Billuart et al. [1], which was inspired by the approach suggested by Daeninck et al. [5]. In this approach, two domains are used: the Eulerian domain, which only covers the near-wall part of the whole domain, and the Lagrangian domain, which covers the whole domain. The latter are depicted in Fig. 2.1. On the Eulerian domain, the FDoom (or near-wall)

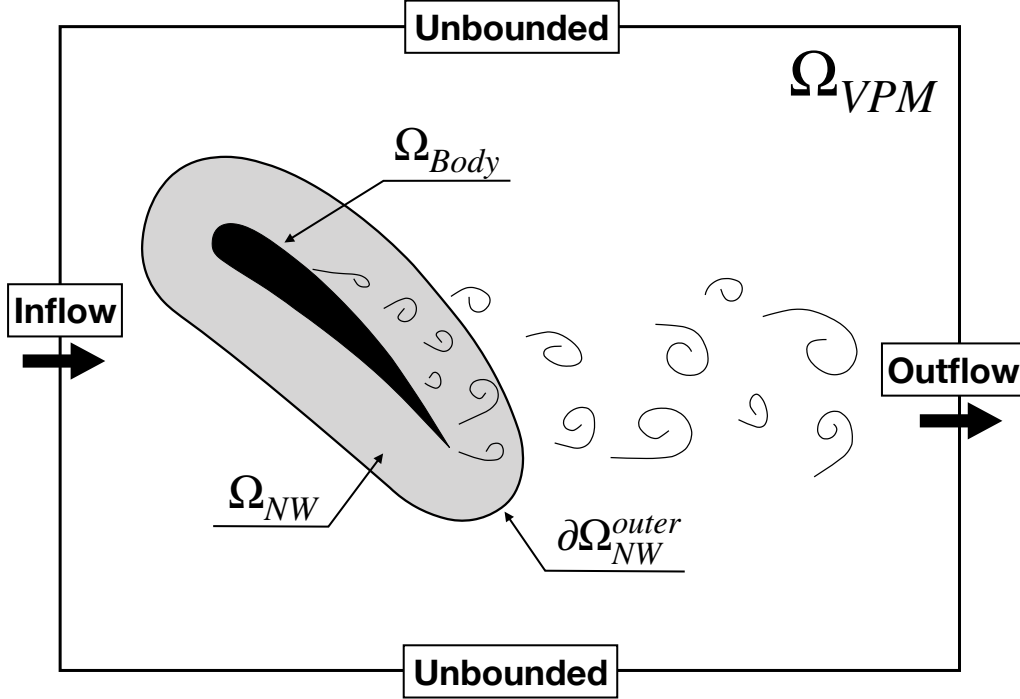


Fig. 2.1: Decomposition into a near-wall (NW) and VPM subdomains, and boundary conditions imposed on the VPM subdomain (sketch from Billuart et al. [1]).

solver is used to solve the flow, while on the Lagrangian domain, the VPM solver is used. These designations lead us to the following notations: Ω_{VPM} stands for the Lagrangian subdomain (where the VPM method is used) and Ω_{NW} stands for the Eulerian subdomain (where the FDoom solver is used). The interaction between the two solvers and the exchange of information between the computational subdomains is the basis of the coupling methodology. On the one hand, the near-wall solver uses a mixed boundary condition on its external boundary $\partial\Omega_{NW}^{outer}$ that is obtained by interpolating fields from the Lagrangian domain (more details on the boundary condition and interpolation procedures are provided in Section 2.3.1). The solver is therefore able to compute the flow fields with a correct boundary condition and with high accuracy. On the other hand, in the Lagrangian domain, the flow fields quantities are also known over the whole domain but, in the near-wall region, this information is highly under-resolved. The flow fields in the near-wall region of the Lagrangian domain are thus corrected by the near-wall solver, which owns well resolved flow fields in that region.

Another particularity of the methodology presented by Billuart et al. [1] is the use of two subsets of particles in the VPM solver. The purpose is to decrease the computational cost of

the method by relying on a redistribution frequency that is different for the two sets. The first subset is used in order to compute the flow quantities in the vicinity of the NW subdomain Ω_{NW} and extends further away to a part of the VPM domain. The part of the domain on which the first subset of particles is distributed is denoted Ω_1 , as depicted in Fig. 2.2. The particles

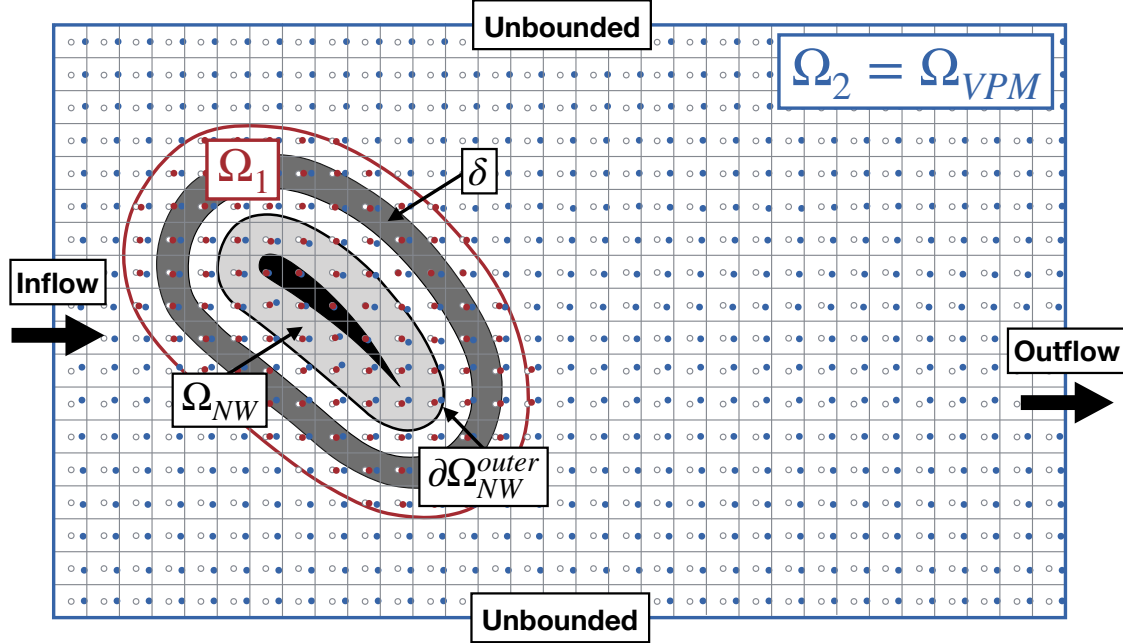


Fig. 2.2: Two subsets of particles and boundary conditions of the VPM solver: red and blue bullets depict the particles of subsets 1 and 2, respectively (Sketch from Billuart et al. [1])

of this subset are thus redistributed on the VPM grid at every time step in order to obtain the most accurate flow information available. The second subset of particles covers the whole domain (i.e.: $\Omega_2 = \Omega_{VPM}$), and is redistributed at a lower frequency in order to decrease the computational cost, since the latter is used for the part of the domain that is further away from the body. The second set of particles obviously suffers more from the particles depletion, due to the lower rate of redistribution. Still, it enables to decrease the computational cost of the method by keeping accurate flow fields in the region of the domain where the vorticity gradients are more intense. Since the two sets of particles are overlapped, it is therefore required to define a transition between the two sets for the $P2M$ interpolation procedure. Either information come from the first subset, or it is a combination, or, finally, it only comes from the second subset. The selection of the particles that are used for that interpolation is performed through the use of a transition function $1_{\Omega_1}^\delta$, that is smoothly achieved over a length δ , at a certain distance from the outer boundary of the Eulerian subdomain $\partial\Omega_{NW}^{outer}$, as advised by Billuart et al.[1]. The two subsets of particles and the transition length are sketched in Fig. 2.2.

2.3.1 Implementation of the methodology

At the beginning of each VPM time step, both solvers are at the same physical time t^n . However, the near-wall solver owns more accurate fields quantities than the VPM solver. The full time step procedure is therefore described in what follows.

1. In the near-wall region, the velocity field $\mathbf{u}_{\text{NW}}(\mathbf{x}, t^n)$ of the Eulerian grid is interpolated onto the VPM grid, but using a staggered grid². This interpolation is performed through a third order Lagrange interpolation of $\mathbf{u}_{\text{NW}}(\mathbf{x}, t^n)$, and the resulting velocity field is denoted $\mathbf{u}_{\text{VPM, corr}}(\mathbf{x}, t^n)$. Since the body is moving, the interpolation coefficients together with the location of the grid points on which the velocity field will be interpolated must be determined at every time step. A sketch of the interpolation procedure, and of the underlying grids is depicted in Fig. 2.3.
2. The velocity field $\mathbf{u}_{\text{VPM, corr}}$ inside the body is set to the body velocity:

$$\mathbf{u}_{\text{VPM, corr}}(\mathbf{x}_{\text{body}}, t^n) = \mathbf{U}_{\text{body}}(\mathbf{x}_{\text{body}}, t^n) . \quad (2.14)$$

This velocity field must take into account any prescribed³ movement (i.e.: translations and rotation around an arbitrary point of the domain), and is used for computing the underlying vorticity. Again, the VPM grid points where the velocity has to be set to the body velocity must be determined at each time step.

3. A corrected vorticity field can be computed on the VPM mesh, since the corrected velocity field $\mathbf{u}_{\text{VPM, corr}}$ is known on the staggered grid from steps (1) and (2). The vorticity must therefore be computed on the VPM collocated grid using

$$\boldsymbol{\omega}_{\text{VPM, corr}} = \nabla_h \times \mathbf{u}_{\text{VPM, corr}} . \quad (2.15)$$

Nevertheless, the computation of the vorticity cannot be performed over all VPM grid points that are located inside the intersection of the VPM and NW subdomains. In fact, near the outer boundary of the Eulerian domain, some VPM grid points cannot benefit from that correction, since the curl operator might not have the required information about $\mathbf{u}_{\text{VPM, corr}}(\mathbf{x}, t^n)$ in the vicinity of the VPM grid point on which the vorticity must be computed. A sketch of the staggered grid on which the corrected velocity field is interpolated is depicted in Fig. 2.3, as well as the stencil of the curl operator for the computation of the corrected vorticity field.

4. The corrected vorticity field $\omega_{\text{VPM, corr}}$ is integrated to the VPM vorticity field which is not corrected, as discussed in step (3), with

$$\omega_{\text{VPM}}(\mathbf{x}_{ij}, t^n) = 1_{\Omega_{\text{NW}}^{\text{corr}}}(\mathbf{x}_{ij}) \omega_{\text{VPM, corr}}(\mathbf{x}_{ij}, t^n) + (1 - 1_{\Omega_{\text{NW}}^{\text{corr}}}(\mathbf{x}_{ij})) \omega_{\text{VPM}}(\mathbf{x}_{ij}, t^n) , \quad (2.16)$$

where $1_{\Omega_{\text{NW}}^{\text{corr}}}$ is the indicator function for the overlapped region of the Eulerian and Lagrangian subdomains where the vorticity can be corrected. The latter ensures a smooth transition between the vorticity that was corrected by the NW solver and the uncorrected vorticity on the VPM grid. This indicator function has to be recomputed at every VPM step for the case of a moving body.

5. The vorticity field $\omega_{\text{VPM}}(\mathbf{x}_{ij})$ on the VPM grid is interpolated and transferred to the two subsets of particles through the *M2P* interpolation procedure. Since the particles' subset 1 is redistributed at every VPM time step, it directly benefits from that correction, while the subset 2 does not necessarily, since it is redistributed every N_r time steps.

²The staggered grid is only used for the interpolation of the velocity from the NW solver.

³For FSI applications, an adaptation of the methodology has to be done, since the movement is not known a priori.

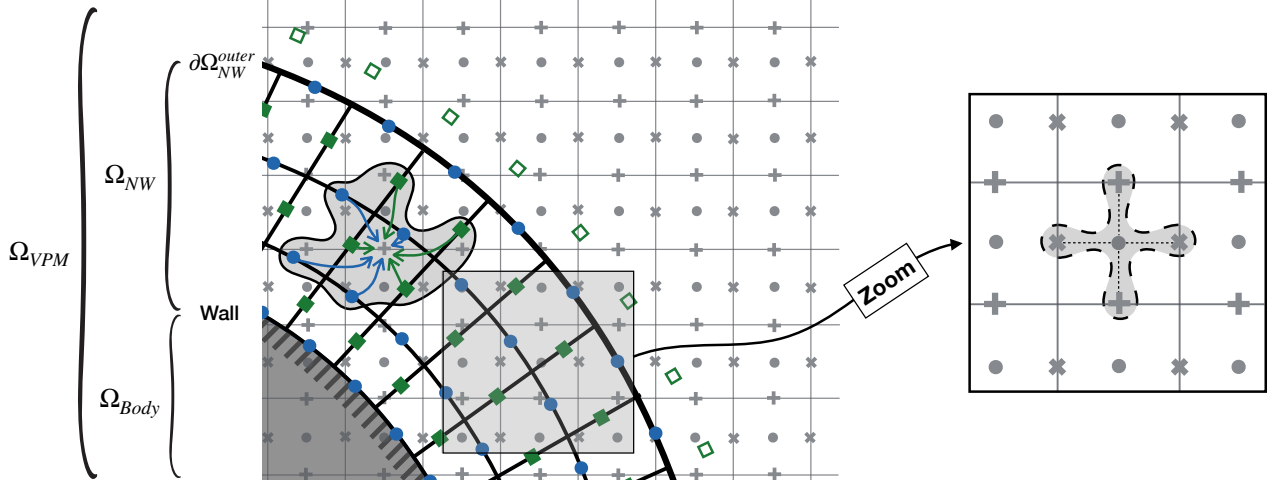


Fig. 2.3: Correction of the VPM fields by the NW fields: the orthogonal mesh of the NW solver is in bold in foreground with its flow fields u_n (●), u_s (■), and u_s^{ghost} (□); the VPM collocated grid is in background, with ω_{VPM} and $\mathbf{u}_\omega$ (●); the staggered grid for the interpolation of the velocity from the NW solver is superimposed on the VPM (collocated) grid, with $u_{VPM, \text{corr}}$ (⊕) and $v_{VPM, \text{corr}}$ (⊗); a zoom on the staggered grid used for the computation of the corrected vorticity field $\omega_{VPM, \text{corr}}$ is also depicted (sketch from Billuart et al. [1])

6. The Poisson equation Eq. (2.13) is solved in order to recover the velocity field $\mathbf{u}_\omega$ on the collocated grid of the VPM solver. Since, during step (4), the vorticity field has been corrected in a part of the overlapped region of the two subdomains, the underlying velocity field has changed as well and is therefore no longer the same as the velocity field obtained at the end of the previous RK3 time step.
7. RK3 time integration from t^n to $t^{n+1} = t^n + \Delta t_{VPM}$ starts, and, at each RK3 sub-time step, steps (7.1) to (7.3) are repeated:
 - 7.1 The vorticity evolution on the particles is computed using Eq. (2.12) and the advection of the particles is computed using Eq. (2.11).
 - 7.2 The $P2M$ interpolation procedure using the M'_4 scheme [39] is performed over the two sets of particles. Therefore, we retrieve on the VPM grid $\omega_{1,VPM}$ and $\omega_{2,VPM}$, the vorticity fields interpolated from the respective subsets of particles with

$$\omega_{1,VPM}(\mathbf{x}_{ij}) = P2M(\mathbf{x}_{p,1}, \alpha_{p,1}) \quad (2.17)$$

$$\omega_{2,VPM}(\mathbf{x}_{ij}) = P2M(\mathbf{x}_{p,2}, \alpha_{p,2}) . \quad (2.18)$$

- 7.3 The transition function $1_{\Omega_1}^\delta$, that computes the smooth transition between the two subsets of particles, is recomputed at every RK3 sub-time step. In this way, the body movement in-between each sub-time step can be taken into account and the information about the vorticity field on the VPM grid can be interpolated using the most precise location of the subdomain Ω_1 that is available. The vorticity field that is interpolated from the particles onto the VPM grid therefore uses information from the two subsets of particles in the transition length δ with:

$$\omega_{VPM}(\mathbf{x}_{ij}) = 1_{\Omega_1}^\delta(\mathbf{x}_{ij}) \omega_{1,VPM}(\mathbf{x}_{ij}) + (1 - 1_{\Omega_1}^\delta(\mathbf{x}_{ij})) \omega_{2,VPM}(\mathbf{x}_{ij}) . \quad (2.19)$$

- 7.4. Since, during step (7.3), the vorticity field has been corrected on the VPM grid from the interpolation of the particles' strength around the considered grid point, the Poisson equation Eq. (2.13) is solved again in order to recover the corresponding velocity field on the collocated VPM grid.

Once the full RK3 time step is achieved, the VPM solver is at t^{n+1} , while the FDoom solver is at t^n . The latter therefore needs to make up with the VPM solver. In order to reconcile them in time, the VPM solver must provide the outer boundary condition to the FDoom solver. This is the most tricky and sensitive part of the coupling procedure. Indeed, determining the conditions for the outer boundary of the near-wall solver is the most difficult part, as slight variations in the velocity field on the boundary leads to high pressure oscillations in the velocity-pressure solver, which deteriorate drastically the accuracy of the aerodynamic forces obtained with the wall quantities. The interpolation procedure of the VPM fields for determining the boundary condition is therefore very important. The latter is presented in Section 2.3.2, where the various interpolation procedures that have been tested are presented and the observations on the results are discussed.

8. Once the interpolation procedure is performed, the NW solver performs N_c sub-time steps, during which equations Eqs. 2.1 to 2.5 are used. At each sub-time step m , the outer boundary condition must be obtained through an interpolation of the VPM fields. Therefore, at the sub-time $t^{n,m} = t^n + \sum_{l=1}^m \Delta t_{NW}^l$ of the NW solver, on each $\mathbf{x}$ of $\partial\Omega_{NW}^{\text{outer}}$, the velocity components in the normal and tangential⁴ directions of the local coordinates system of the orthogonal grid, and the vorticity can be determined. It is important to stress that the boundary condition is not a pure Dirichlet condition as it could be thought. Indeed, it consists in a mixed boundary condition, as explained by Billuart et al. [1]. Eq. (2.20) is a Dirichlet condition on u_n , and Eq. (2.21) is a Neumann condition on u_s .

$$u_n(\mathbf{x}) = \mathbf{u}_{\text{VPM, interp}}(\mathbf{x}) \cdot \hat{\mathbf{e}}_n(\mathbf{x}) \quad (2.20)$$

$$\frac{\partial}{\partial n} [h_s(\mathbf{x}) u_s(\mathbf{x})] = [h_n(\mathbf{x}) h_s(\mathbf{x})] \omega_{\text{VPM, interp}}(\mathbf{x}) + \frac{\partial}{\partial s} [h_n(\mathbf{x}) u_n(\mathbf{x})] , \quad (2.21)$$

where $\mathbf{u}_{\text{VPM, interp}}$ and $\omega_{\text{VPM, interp}}$ are the velocity and vorticity fields interpolated from the respective VPM fields, h_n and h_s are the metrics in, respectively, the normal and tangential direction of the Eulerian orthogonal mesh. In practice, Eqs. (2.20) and (2.21) are applied with the following points:

- 8.1 u_n is imposed on each position $\mathbf{x}$ of the outer boundary $\partial\Omega_{NW}^{\text{outer}}$. It has then to be corrected to ensure the mass conservation in the Eulerian subdomain, as depicted in Fig. 2.4. The resulting normal component u_n^c thus satisfies:

$$\oint_{\partial\Omega_{NW}^{\text{outer}}} u_n^c dS = 0 . \quad (2.22)$$

- 8.2 From $\omega_{\text{VPM, interp}}$ and $u_{\text{VPM, interp}}$, the tangential component of the velocity u_s that corresponds to $\omega_{\text{VPM, interp}}$ can be computed and, since there is no grid point that corresponds to u_s^{outer} , the values at the ghost points u_s^{ghost} are set in such a way that they correspond to the desired value of the tangential velocity u_s^{outer} , as shown in Fig. 2.4.

⁴for the tangential component, there is no grid point on the boundary, thus the corresponding ghost value is used.

Now that the NW solver has performed its N_c sub-time steps, the full time step of the coupling method is completed.

The methodology presented here above is actually the most straightforward extension of the methodology from a stationary body presented by Billuart et al. [1] to a moving body. This extension mainly concerns the recomputing of the positions where the fields have to be interpolated from the VPM grid to the Eulerian grid, recomputing the weights, and the transitions and indicator functions. Nevertheless, the non-trivial part of the extension of the coupling procedure is the interpolation procedure of the VPM fields for the boundary condition of the NW solver presented in step (8). The latter is discussed in greater detail in Section 2.3.2.

2.3.2 Boundary conditions for the near-wall solver

The boundary condition for the NW solver has been quite challenging to adapt in the coupling procedure, since, as already mentioned before, it is the key component in order to obtain accurate aerodynamic forces computed using the wall quantities. Slight oscillations in the boundary conditions can lead to prohibitive oscillations in the pressure field, which is very sensitive to the fluctuations of velocity on the outer boundary. The main implementations that have been tested for the boundary condition are presented in this section in order to show that, even if some conditions are close in the working principles, slight changes can lead to important deterioration of the method. Depending on the way the VPM fields are interpolated in time and in space, the effect on the accuracy of the aerodynamic forces obtained using the wall quantities is drastic, with pressure oscillations that vary in magnitude.

Present implementation

For the present work the following interpolation procedure is applied. As the VPM fields are known on the VPM grid at every VPM step, for obtaining the fields on the outer boundary at each sub-time $t^{n,m}$, the fields $\mathbf{u}_{\text{VPM}}(\mathbf{x}_{ij})$ and $\omega_{\text{VPM}}(\mathbf{x}_{ij})$ obtained on the whole VPM grid are stored at times t^n and t^{n+1} . After that, at every sub-time step m , the following procedure is applied:

- (i) The VPM fields are linearly interpolated in time from the values at t^n and t^{n+1} on every VPM grid point $\mathbf{x}_{ij}$ with

$$\mathbf{u}_{\text{VPM, interp}}(\mathbf{x}_{ij}, t^{n,m}) = \alpha^m \mathbf{u}_{\text{VPM}}(\mathbf{x}_{ij}, t^{n+1}) + (1 - \alpha^m) \mathbf{u}_{\text{VPM}}(\mathbf{x}_{ij}, t^n) \quad (2.23)$$

$$\omega_{\text{VPM, interp}}(\mathbf{x}_{ij}, t^{n,m}) = \alpha^m \omega_{\text{VPM}}(\mathbf{x}_{ij}, t^{n+1}) + (1 - \alpha^m) \omega_{\text{VPM}}(\mathbf{x}_{ij}, t^n), \quad (2.24)$$

$$\text{where } \alpha^m = \frac{t^{n,m} - t^n}{t^{n+1} - t^n}.$$

- (ii) The positions $\mathbf{x}_{u_n}$ and $\mathbf{x}_\omega$ of the grid points for the normal velocity component u_n (in the local frame of the Eulerian mesh) and the vorticity ω on the outer boundary of the Eulerian mesh $\partial\Omega_{\text{NW}}^{\text{outer}}$ at time $t^{n,m}$ are updated. Then, the spatial interpolation of the VPM fields $\mathbf{u}_{\text{VPM, interp}}(\mathbf{x}_{ij}, t^{n,m})$ and $\omega_{\text{VPM, interp}}(\mathbf{x}_{ij}, t^{n,m})$ onto the Eulerian grid points $\mathbf{x}_{u_n}$ and $\mathbf{x}_\omega$ located on the outer boundary $\partial\Omega_{\text{NW}}^{\text{outer}}$ is performed using the M'_4 [39] scheme, as shown in Fig. 2.4.
- (iii) The mixed boundary condition can be applied using Eqs. (2.20) and (2.21) of step (8). It is important to mention that the velocity field interpolated from the VPM grid at $\mathbf{x}_{u_n}$ does

not yet take into account the local orientation of the Eulerian mesh in order to compute the local normal component u_n .

This interpolation procedure is the most successful, because it enables to mitigate the oscillations generated by the boundary condition on the NW pressure field, leading to oscillations in the pressure component of the aerodynamic forces that have the lowest amplitude, compared to the other interpolation procedures presented in what follows. However, it was neither the only approach tested, nor the first tested.

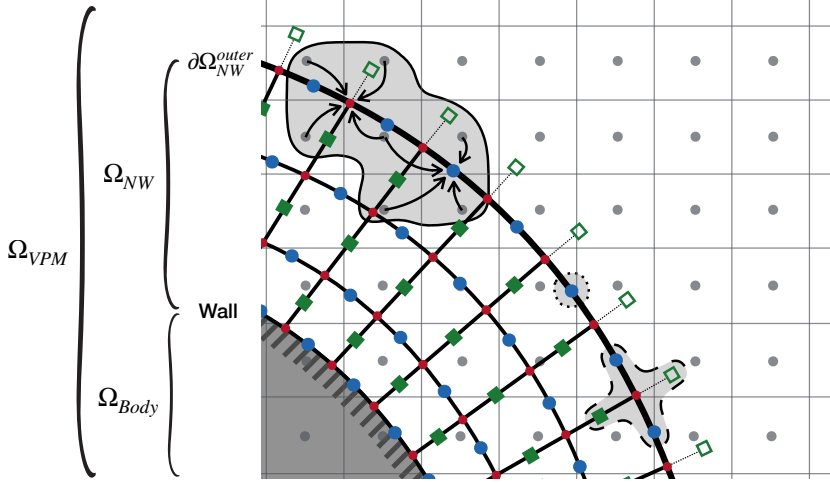


Fig. 2.4: Boundary condition of the NW solver: spatial interpolation of the VPM fields at sub-time $t^{n,m}$ (step (ii) - $\square$), correction of u_n for mass conservation (step (8.1) - $\square$), ghost tangential velocity computation (step (8.2) - $\square$). Eulerian mesh is in bold in foreground with ω (•), u_s (■), u_n (●), u_s^{ghost} (□); the VPM collocated grid is in background with ω and u_ω (•) (sketch from Billuart et al. [1])

Most straightforward interpolation procedure

The approach presented in this section was initially implemented. Even though it was quite straightforward with respect to the work performed by Billuart et al.[1], it was discarded due to the important pressure oscillations it generated. The procedure consists in the following steps:

- (i) The positions $\mathbf{x}_{u_n}$ and $\mathbf{x}_\omega$ of the grid points on the outer boundary of the Eulerian mesh $\partial\Omega_{\text{NW}}^{\text{outer}}$ at time t^{n+1} are updated. Thus, the spatial interpolation of the VPM fields at time t^{n+1} onto the Eulerian grid points located on the outer boundary $\partial\Omega_{\text{NW}}^{\text{outer}}$ can be performed using M'_4 [39], as illustrated in Fig. 2.4.
- (ii) At each sub-time step m of the NW solver, the outer boundary condition is obtained through a linear interpolation in time from the NW fields at time t^n (which are known from previous time step on the boundary locations at that time $\mathbf{x}_{u_n}^{\text{outer}}(t^n)$ and $\mathbf{x}_\omega^{\text{outer}}(t^n)$) and the VPM fields at time t^{n+1} , interpolated on $\mathbf{x}_{u_n}^{\text{outer}}(t^{n+1})$ and $\mathbf{x}_\omega^{\text{outer}}(t^{n+1})$. Therefore, at the time $t^{n,m} = t^n + \sum_{l=1}^m \Delta t_{\text{NW}}^l$, on each $\mathbf{x}$ of $\partial\Omega_{\text{NW}}^{\text{outer}}$:

$$\mathbf{u}_{\text{VPM, interp}}^{n,m}(\mathbf{x}(t^{n,m})) = \alpha^m \mathbf{u}_{\text{VPM, interp}}^{n+1}(\mathbf{x}(t^{n+1})) + (1 - \alpha^m) \mathbf{u}_{\text{VPM, interp}}^n(\mathbf{x}(t^n)) \quad (2.25)$$

$$\omega_{\text{VPM, interp}}^{n,m}(\mathbf{x}(t^{n,m})) = \alpha^m \omega_{\text{VPM, interp}}^{n+1}(\mathbf{x}(t^{n+1})) + (1 - \alpha^m) \omega_{\text{VPM, interp}}^n(\mathbf{x}(t^n)) \quad , \quad (2.26)$$

where $\mathbf{u}_{\text{VPM, interp}}^n(\mathbf{x}(t^n))$ stands for $\mathbf{u}_{\text{VPM, interp}}(\mathbf{x}(t^n), t^n)$ for the sake of brevity.

(iii) The mixed boundary condition can be applied with Eqs. (2.20) and (2.21) of step (8).

The only difference with the interpolation procedure that is finally used in this work is the fact that the temporal and spatial interpolations have been reversed. In other words, the temporal interpolation is not applied on the VPM fields, as it is the case for the final method.

For the approach presented here above, the linear interpolation in time of the fields on the boundary is performed on $u_{\text{VPM, interp}}$, $v_{\text{VPM, interp}}$, and $\omega_{\text{VPM, interp}}$, at each NW sub-time step, and, after that, the normal component u_n can be determined based on the local orientation of the FDoom mesh. However, the resulting velocity component u_n does not benefit from the correction such that the mass is conserved on the NW subdomain. This correction will be applied in a second time, and for every sub-time step, as described in step (8) and illustrated in Fig. 2.4. Another approach could therefore try to mitigate the error by applying the temporal interpolation directly on the normal component u_n instead of the velocities expressed in the inertial frame.

In effect, after the spatial interpolation of the VPM fields at time t^{n+1} , the resulting normal component u_n can already be computed, since the position of the Eulerian grid points on the boundary are known as well. Moreover, since the flux at the wall at time t^{n+1} is also known, the mass conservation can already be enforced. Thus, instead of interpolating u_{VPM} and v_{VPM} in time, computing the resulting u_n , and then correcting the latter to obtain u_n^c , we could directly compute u_n , correct it, and interpolate u_n^c between t^n and t^{n+1} . By doing so, we do not need to apply a correction at each NW sub-time step, since the normal components are obtained by linear interpolation of fields that already take into account the mass conservation. This procedure has been tested as well, and does not present any significant reduction of the amplitude of the oscillations. Therefore, it seems equivalent to the straightforward interpolation.

Temporal interpolation at higher order of the VPM fields

Since the VPM fields are known at each sub-time step of the RK3 scheme (i.e. at the physical times t^n , $t^{n+1/3}$, $t^{n+2/3}$, and t^{n+1}) - as they are retrieved on the VPM grid from the particles' strength and position evolution through a $P2M$ procedure - the fields information at those times can be used. As a reminder, at each sub-time step, a $P2M$ interpolation procedure is applied in order to get back the vorticity field on the VPM grid. Then a Poisson equation Eq. (2.13) is solved in order to determine the underlying velocity field u_ω on those collocated grid points, as it is described above in step (7.3). Therefore, information about those VPM fields can be used for increasing the order of the interpolation procedure of the VPM fields for the boundary condition of the near-wall solver. Indeed, regarding the previous procedures presented, they only involve linear interpolations in time.

However, this procedure has been tested in the same way the present procedure is introduced above (see steps (i) to (iii) of the present implementation, but with a close difference: the order of the temporal interpolation is larger here), and it has been observed that it increased even more the magnitude of the oscillations in the aerodynamic forces as compared to the other approaches. This is probably due to the fact that the third order interpolation captures second derivatives that are false; in other words, it captures a spurious temporal behaviour, which can be considered as an overfitting of the temporal evolution of the fields on the boundary. In addition to that, this method also increases even more the computational cost, as it requires to store the VPM fields at more times.

2.4 Numerical stability and accuracy

The stability and CFL-like constraints are basically the same as for the case of a static body, as also recalled in Billuart et al. [1]. Concerning the FDoom solver, there is a slight change in the classical CFL formula, in order to take into account the ALE formulation in the convective term.

First, the classical CFL condition has to take into account the mesh movement by replacing the material velocity $\mathbf{u}$ by the convective velocity $(\mathbf{u} - \mathbf{U}_{\text{mesh}})$ in the relative frame attached to the mesh, as it was done for the convective term in the Navier-Stokes equations presented in Section 2.1. The CFL number in the relative frame is thus given in Eq. 2.27.

$$CFL_{\text{rel}} = \max \left(\frac{\|\mathbf{u} - \mathbf{U}_{\text{mesh}}\|}{h} \right) \Delta t . \quad (2.27)$$

On the other hand, the VPM method does not need the same kind of CFL-like condition for the stability of the method. Still, the two following Lagrangian criteria are used in order to ensure the accuracy of the method and limit the particles depletion

$$\max \|\mathbf{d}\| \Delta t \leq 0.2 ; \max |\omega| \Delta t \leq 0.2 , \quad (2.28)$$

where the norm of the rate of strain tensor is, here, defined as

$$\|\mathbf{d}\| \triangleq \left| \frac{\partial u}{\partial x} \right| + \frac{1}{2} \left| \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right| . \quad (2.29)$$

In addition to the classical criteria presented above, the coupling method of the moving Eulerian mesh on the VPM grid, requires a new CFL-like stability condition. The latter is provided in Eq. 2.30 and translates the fact the grid points of the Eulerian mesh should not cover a distance larger the distance between two VPM grid points over one VPM time step.

$$CFL_{\text{mesh}} = \frac{\max \|\mathbf{U}_{\text{mesh}}\| \Delta t}{h_{\text{VPM}}} < 1 . \quad (2.30)$$

2.5 Discussion on the coupling parameters

In order to figure out more precisely what improvements in the method should be performed in order to decrease the amplitude of oscillations mentioned in Section 2.3.2, several parametric tests have been performed. The parameters that have been investigated are the following.

- The ratio of FD time steps for each VPM time step: $N_{\text{FD/VPM}}$.
- The remeshing frequency of the second subset of particles N_r .
- The width of the transition zone for the two sets of particles, and the length of the extent of the first set of particles beyond the outer boundary $\partial\Omega_{\text{NW}}^{\text{outer}}$ of the NW solver.
- The values of the Lagrangian criteria presented in Section 2.4 for the accuracy of the method.

Several tests⁵ have been performed and the following conclusions have been retrieved:

1. The smaller the time step, the better the results.
2. The remeshing frequency and the Lagrangian criteria for stability are not responsible for such a high amplitude of oscillation.
3. Expanding the boundaries considered for the indicator function for the transition zone between the two sets of particles does not decrease the amplitude of oscillations.

⁵The results are not presented in this work.

Chapter 3

Verification: flow past a moving cylinder

In this chapter, the validation of several single movements is presented. The study of the flow past moving cylinders has been studied for many years, but, nowadays, it does not present much attention anymore, as it was presented in the introduction Chapter 1, Section 1.2.5. This explains the fact that the references used in this chapter are sometimes old research studies and that the Reynolds numbers used for those studies were quite low. Two kinds of movements are presented:

1. A cylinder oscillating in the transverse direction of the freestream velocity.
2. A rotating cylinder in a freestream.

The following common dimensionless numbers are used in the following chapters:

$$t^* \triangleq \frac{t U_\infty}{D}, \quad \omega^* \triangleq \frac{\omega D}{U_\infty}, \quad P^* \triangleq \frac{P}{\frac{1}{2} U_\infty^2}, \quad h^+ \triangleq \frac{h_{\text{NW}}^{n, \text{wall}} u_\tau}{\nu}, \quad C_d \triangleq \frac{F_x}{\frac{1}{2} \rho U_\infty^2 D}, \quad C_l \triangleq \frac{F_y}{\frac{1}{2} \rho U_\infty^2 D}, \quad (3.1)$$

with u_τ , the shear velocity defined by $u_\tau^2 \triangleq |\tau_{\text{wall}}|/\rho$ (τ_{wall} being the wall shear stress), and where $h_{\text{NW}}^{n, \text{wall}}$ is the size of the first cell of the Eulerian orthogonal mesh in the normal direction.

For all simulations, at time $t^* = 0$, the flow solution is set to the potential flow solution. The coupling parameters between the VPM solver and the NW solver are chosen as advised by Billuart et al. [1]. As a reminder, still, the second subset of particles that covers the whole VPM domain is redistributed every $N_r = 5$ time steps, while the other is redistributed every time step. Concerning the transition length between the two subsets of particles, $\delta = 8 h_{\text{VPM}}$, and the starting and ending of the indicator function $1_{\Omega_1}^\delta$ with respect to the end of the Eulerian subdomain are set according to the recommendations from Billuart et al. [1]. The mesh parameters used for the three investigated cases are the same and provided in Table 3.1. They have been chosen in such a way that there is a consistency between the cell size at the outer boundary of the NW solver and the VPM cell size, and that the flow is sufficiently resolved. The FD grid is fine in order to present accurate results. Namely, the first cell enables a resolution such that $h^+ < 1$ for $t^* \geq 0.025$ in the simulations of the transverse motion (Section 3.1), and $t^* \geq 0.82$ in the cases of the rotating cylinder motion (Section 3.2).

Table 3.1: Mesh parameters of the NW solver and VPM solver for the validation cases

w/R	R/h_{VPM}	$R/h_{\text{NW}}^{n,\text{wall}}$	$h_{\text{NW}}^{n,\text{wall}}/h_{\text{NW}}^{\theta,\text{wall}}$
2	12.8	50	1.3

3.1 Transverse oscillation of a cylinder at $Re = 100$

The dimensionless parameters that characterize the body movement are:

1. The dimensionless amplitude of oscillation: $A^* = A/D$.
2. The dimensionless frequency of the forced oscillation: $F = f_f/f_{st}$, where f_{st} is the Strouhal frequency that corresponds to the flow past the fixed cylinder at the same Reynolds number, and f_f is the forced frequency at which the cylinder oscillates.

The validation will be made by comparing similar simulations and results with the ones presented by Placzec et al. [25]. The method used by Placzec et al. [25] is a finite volume method with 28,800 cells over the whole computational domain. The latter is $32.5 D$ long, and $20 D$ high, with the cylinder positioned at $x/D = 12.5$. Regarding the top and bottom boundaries, a slip wall and $\partial u/\partial y = 0$ are imposed as boundary condition, and a Neumann condition on the velocity field with $\partial u/\partial x = 0$ and $\partial v/\partial x = 0$ for the outflow boundary condition. In the three cases that are presented, the Reynolds number was $Re = 100$. They assessed the Strouhal number as $St \simeq 0.169$ for a shedding regime at the same Reynolds number for a fixed cylinder, while the present method assesses the Strouhal number at $St \simeq 0.1608$. The first two cases presented have an amplitude of $A^* = 0.25$ but have a forced frequency corresponding, respectively to a so-called *locked* and *unlocked* configurations. In Section 3.1.1, a *locked* configuration is presented. *Locked* configurations correspond to cases where the forced oscillations of the cylinder drive the oscillations of the aerodynamic forces, and of the shedding. Results from Placzec et al. [25] should be compared with caution, since they did not validate their results by comparing them with other references for the prescribed movements they investigated. This thus reflects the lack of studies of the flow past forced moving cylinder, and it can explain why some results obtained with present method can differ from those of Placzec et al. [25]. In Section 3.1.1, an *locked* configuration is presented. *Unlocked* configurations mean that the cylinder motion does not exactly enforce the oscillations in the aerodynamic forces. In other words, other frequencies can be implied in the behavior of the aerodynamic forces. Finally, in Section 3.1.3, a third case is discussed for the purpose of presenting a different vortex shedding pattern, due to the high amplitude oscillation of $A^* = 1$.

Placzec et al. [25] assess the locked/unlocked regime with several results: the Power Spectral Density (PSD) of the aerodynamic forces, which should present a peak at $f^* = f/f_f \approx 1.0$ for a locked regime, and through what they call a *phase portrait*, which presents the aerodynamic force as a function of the cylinder position, for example $C_l(y)$, $C_d(y)$ or $C_l(C_d)$. In the case where there is a temporal synchronization of the oscillation in the aerodynamic forces with the cylinder motion, the phase portrait obtained should be a close curve. What is interesting with the phase portrait is that they provide an image of the energy transfer between the moving

body and the fluid. The vertical motion of the cylinder is imposed with Eq. 3.2.

$$Y_{mesh}(t)/D = A^* \sin(2\pi f_f t) = A^* \sin\left(\frac{2\pi}{T_f} t\right), \quad (3.2)$$

where $T_f \triangleq 1/f_f$ is the period of the forced oscillation.

3.1.1 Case 1: Locked configuration $Re = 100$ - $A^* = 0.25$ - $F = 0.9$

As already mentioned above, the shedding regime of a cylinder in a locked configuration is driven by the forced frequency of oscillation of the cylinder. Instantaneous plots of the dimensionless vorticity close to the cylinder at three times of one oscillation period are shown in Fig. 3.1. These times correspond to the extreme positions of the cylinder in its forced motion, i.e. when $y/D \simeq \pm 1$ and the last correspond to a moment when the cylinder is at the beginning of a new cycle $y/D \simeq 0$. The iso-contours of vorticity show that there is a very good agreement between the vorticity from the near-wall and VPM solvers, which reflects that the mixed boundary condition is well adapted to the moving Eulerian mesh on the VPM background. When looking at the whole wake developed behind the cylinder in Fig. 3.2, one can realize that the transverse oscillation creates a wake that looks like to the one behind a static cylinder, but here, the consecutive vortices of opposite sign that are shed are not aligned. Indeed, the vortices of same sign are mutually aligned, but to around $y/D = 0$. There is an offset, with the negative vortices that are shed above $y/D = 0$ and the positive ones below.

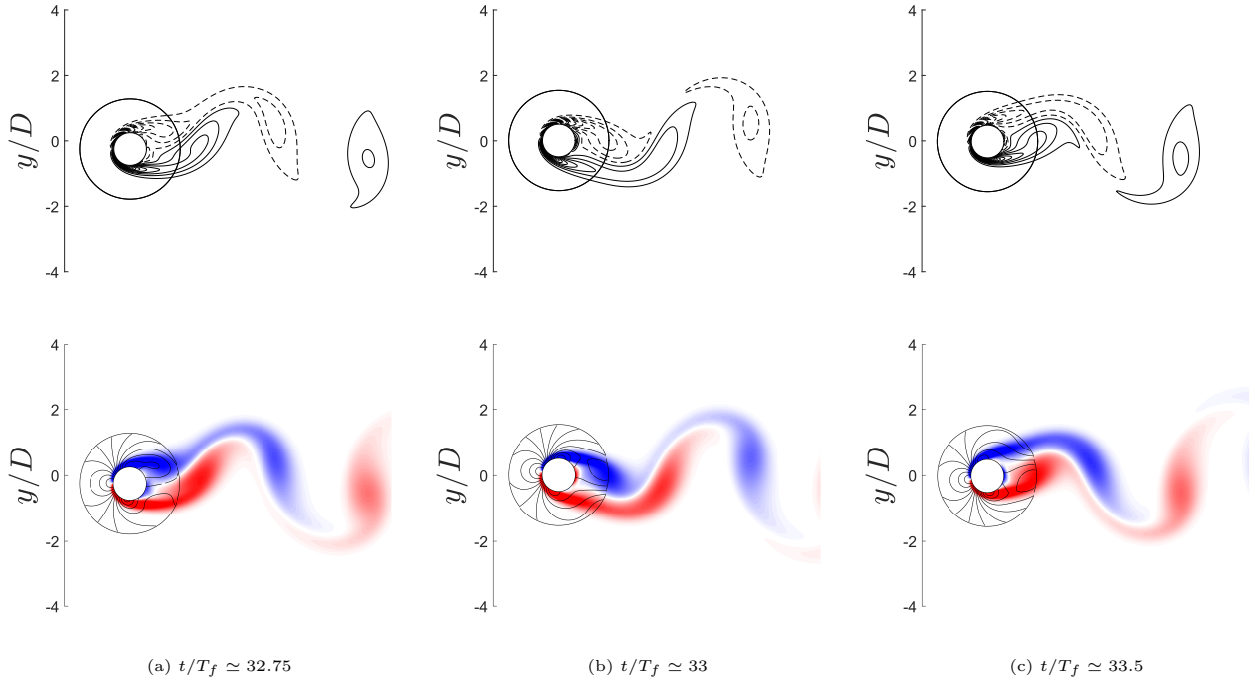


Fig. 3.1: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.9$: contours of dimensionless vorticity in FDoom and VPM solvers. Levels are by steps of 1 (starting from $|\omega^*| = 0.5$) - levels for P^* are by steps of 0.2 (starting from P_{max}^*). The colour plot is saturated for $|\omega^*| > 3$. The limit between the FDoom and VPM results is represented by a thin black line.

Placzec et al. [25] do not provide plots of vorticity field, it is therefore not possible to compare this result from the present study. In addition to that, the simulation they performed was only 18 periods of oscillation long, and they also include the transient starting which is less relevant since the interest is greater for the statistically converged times. Since their simulation is quite short, they cannot show many oscillations for the statistically converged times. Thus, the results from the present method are taken once the simulation is statistically converged, and compared with the results from Placzec et al. [25] at smaller times, that is, the results from Placzec et al. [25] have been shifted in time, so that they are in phase with the results from the present study, and they therefore do not correspond to the times indicated on the plot. When comparing

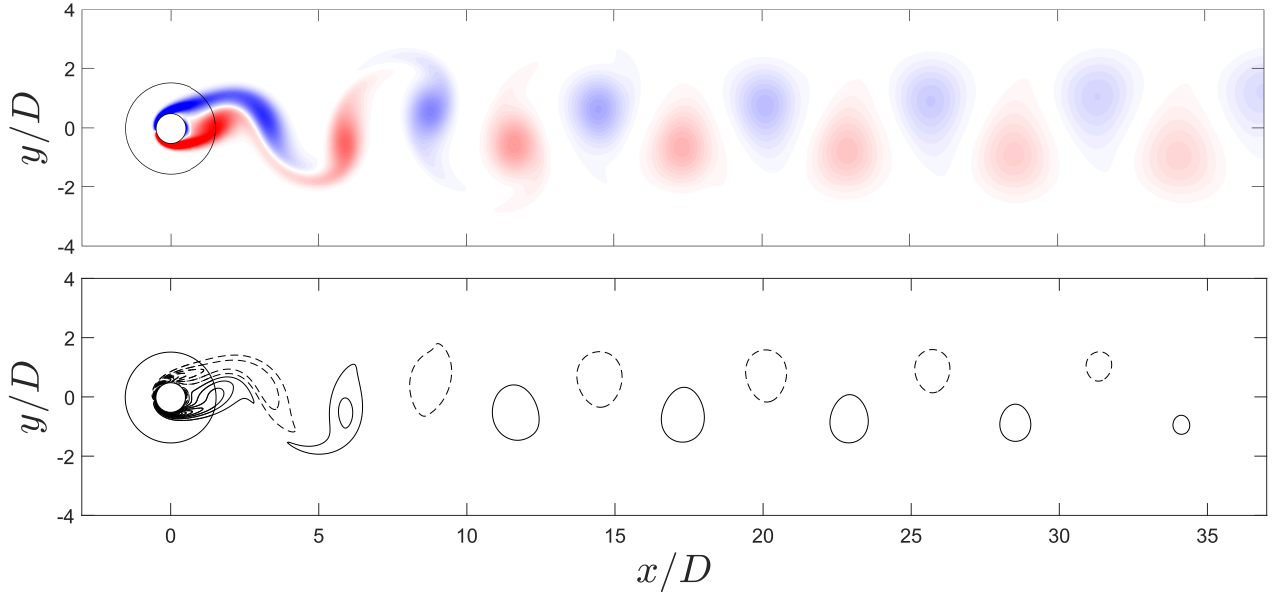


Fig. 3.2: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.9$: contours of dimensionless vorticity and pressure (bottom) at $t/T_f \simeq 33.5$ in the FDoom and VPM solvers. Levels for ω^* are by steps of 1 (starting from $|\omega^*| = 0.5$) - levels for P^* are by steps of 0.2 (starting from P_{max}^*). The colour plot is saturated for $|\omega^*| > 3$. The limit between the FDoom and VPM results is represented by a thin black line.

both curves in Fig. 3.3, it is obvious that there is an important difference in the results. As the other simulations will show, the results from the present method are in better agreement, which lead us to suspect that the results that are compared do not correspond to the same case. Regarding the results from the present method, the oscillation in the aerodynamic forces can be seen in Fig. 3.3. What is also important to notice is that the drag coefficient presents more oscillations than the lift coefficients. The fact that the peaks of maximum (resp. minimum) lift coefficient do not happen exactly at the moment when the cylinder reaches its minimum (resp. maximum) position y/D comes from the fact that the vortices are not shed at that exact moment. Table 3.2 provides the values of lift and drag coefficients averaged over the last 16 oscillation periods compared to the values that can be extracted from the results of Placzec et al. [25].

In addition to the time history of the lift and drag coefficients, the phase portraits are depicted in Figs. 3.4 and 3.5. The difference in amplitude of the lift coefficient between can be observed again, and the high frequency oscillations in the drag coefficient are even more evident. Physically, the phase portrait of the lift coefficient can represent the energy transferred by the cylinder to the flow (or vice versa).

Finally, the spectral analysis of the lift coefficient is shown in Fig. 3.6, where the *locked* shedding regime can clearly be assessed, since the main peak appears at $f/f_f = 1$, exactly. This therefore means that the lift coefficient oscillates exactly at the forced frequency of the cylinder. Another peak at $f/f_f = 3$ appears in the spectrum, but this can be discarded with respect to the main one.

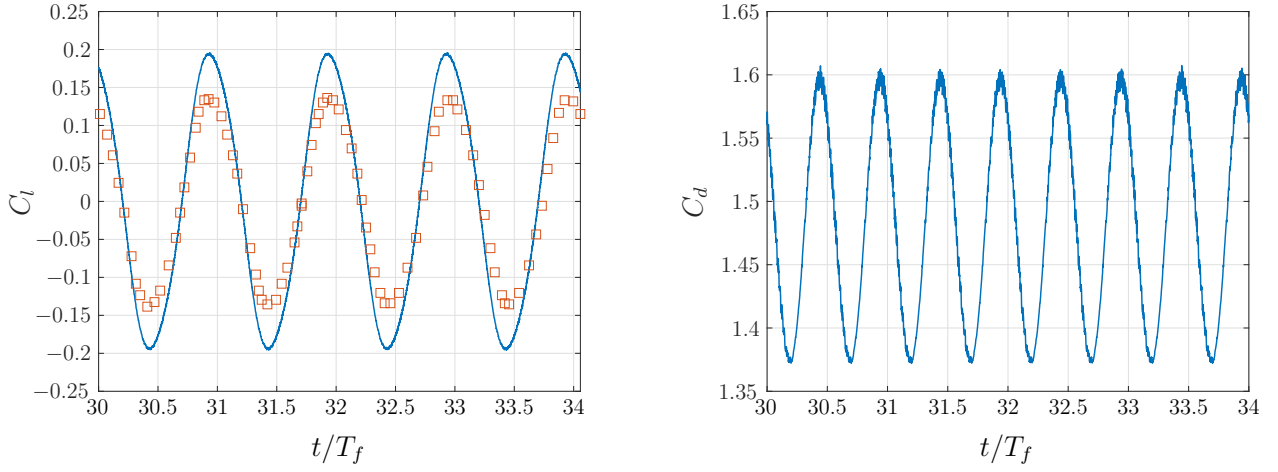


Fig. 3.3: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.9$: lift and drag coefficients over 4 forced oscillations once the flow is statistically steady. Results from the present method (—) and from Placzec et al. [25] ($\square$). Results from Placzec et al. [25] have been shifted in time so that they can be compared with the present results.

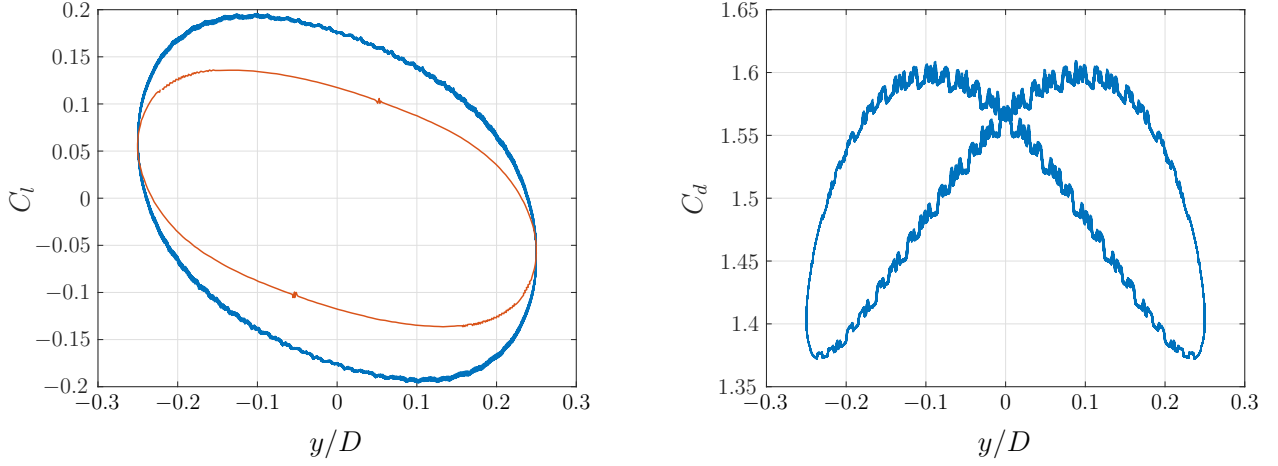


Fig. 3.4: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.9$: phase portraits of the lift and drag coefficients. Results from the present method (—) and from Placzec et al. [25] (—).

Table 3.2: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.9$: force coefficients computed with wall quantities of the NW solver averaged over 16 periods of forced oscillation (T_f).

	$\overline{C_d}$	$\overline{C_d^{min}}$	$\overline{C_d^{max}}$	$\overline{C_l}$	$\overline{C_l^{min}}$	$\overline{C_l^{max}}$
Present method	1.485	1.372	1.606	-1.5×10^{-5}	-0.194	0.194
Placzec et al. [25]	1.5	/	/	/	-0.135	0.135

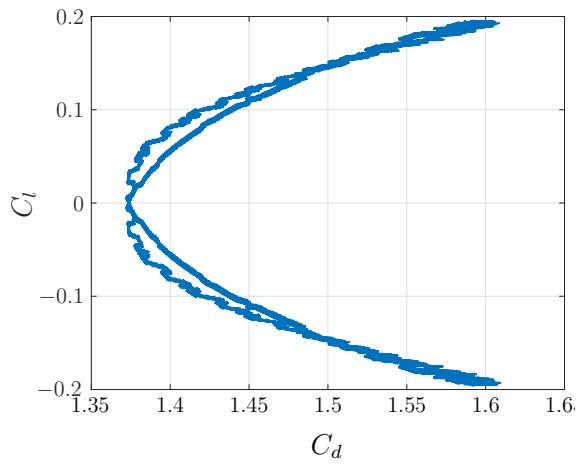


Fig. 3.5: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.9$: phase portraits of the lift and drag coefficients from the present method.

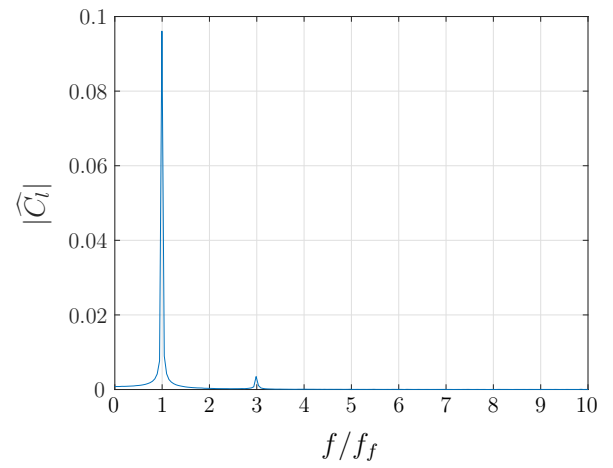


Fig. 3.6: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.9$: FFT of the lift coefficient: main peak at $f^* = f/f_f = 1$, showing the synchronization of the vortex shedding with the forced oscillation f_f of the cylinder. Lower peaks are at $f^* = 2$ and 3.

3.1.2 Case 2: Unlocked configuration $Re = 100$ - $A^* = 0.25$ - $F = 0.5$

The foremost aspect to underline in this case, is that, since the dimensionless frequency of oscillation of the cylinder $F = f_f/f_{St}$ is exactly half the Strouhal frequency, and when looking beforehand at the results from Placzec et al. [25], it has been decided to figure out the Strouhal frequency of the cylinder at $Re = 100$ with the present method. This enables to then perform the simulation at a different forced frequency f_f from Placzec et al. [25] but still at the same dimensionless frequency $F = 0.5$. Indeed, it could be thought *a priori* that the physical behaviour of the vortex shedding could present specific features due to the presence of two driving frequencies with the forced frequency that is exactly half of the natural frequency. This intuition will be verified with the following results obtained with the aerodynamic forces. The Strouhal number was obtained by both probing the velocity in the wake and performing the spectral analysis of the lift force. The Strouhal number from the present method is therefore $St \simeq 0.1608$, while the value from Placzec et al. [25] is $St \simeq 0.169$.

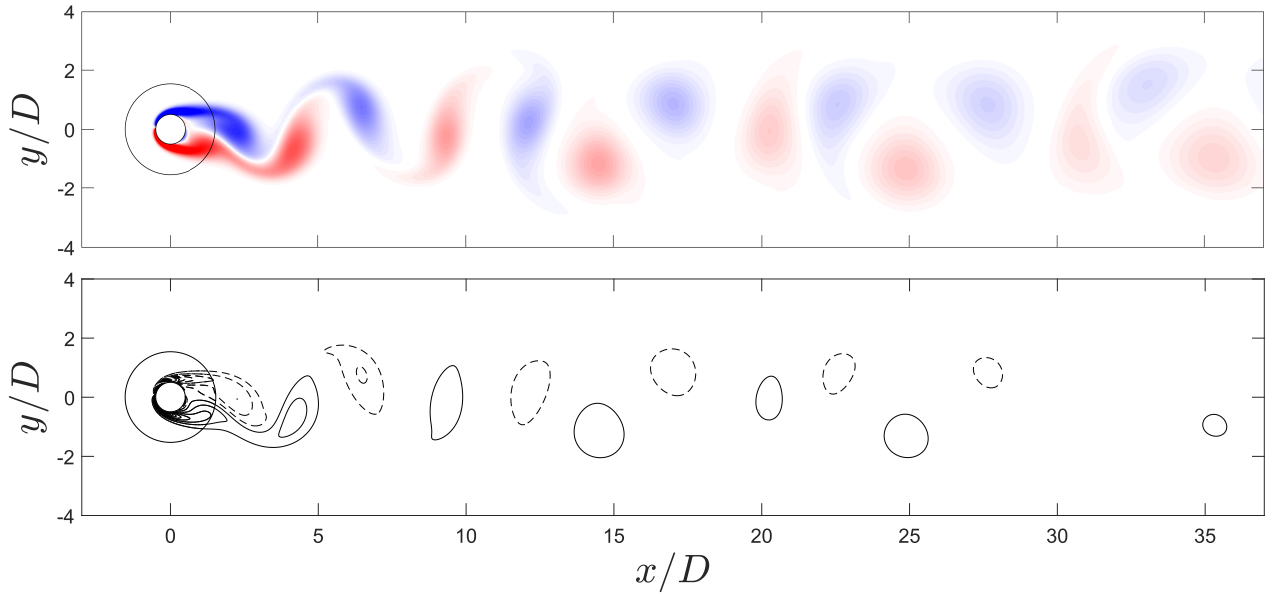


Fig. 3.7: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.5$: contours of dimensionless vorticity and pressure (bottom) at $t/T_f \simeq 26$ in the FDoom and VPM solvers. Levels for ω^* are by steps of 1 (starting from $|\omega^*| = 0.5$) - levels for P^* are by steps of 0.2 (starting from P_{max}^*). The colour plot is saturated for $|\omega^*| > 3$. The limit between the FDoom and VPM results is represented by a thin black line.

In the case where the forced frequency is not close to the natural shedding frequency of the cylinder, i.e. its Strouhal frequency, the shedding regime presents a behaviour that is completely different. In fact, the shedding regime is not only driven by the forced frequency as it is the case for a locked configuration. Instead, two main frequencies drive the shedding regime: the forced oscillation and the Strouhal frequency. In Fig. 3.7, the vorticity field over the whole domain is shown, once the flow has reached a statistically converged pattern. It can clearly be seen that both the relative position of the vortices in the wake and the shape of the vortices is different from the case of a locked configuration, presented above. The shape of the vortices is not regular. Some vortices are more stretched in the vertical direction, and have thus a less compact support of circulation. This is even more obvious in the iso-contours plot than in the colour plot. Near the end of the domain, two vortices are so stretched that their support of vorticity is very diffuse compared to the previous case. In order to highlight that the forced

frequency does not drive alone the shedding regime, Figs. 3.12 and 3.13 show two series of plots where the cylinder is at the same place y/D in its periodic motion, spaced by 13 periods of oscillation T_f . When comparing respectively each plot of the two series, it can be seen that they do not present the same behaviour, as it is the case in locked shedding regimes. In addition to that, the zoom on the cylinder body enables the agreement of the vorticity field across the outer boundary of the near-wall $\partial\Omega_{NW}^{\text{outer}}$.

Considering now the aerodynamic forces exerted on the cylinder, Fig. 3.8 compares the results from Placzec et al. [25] to those from the present method. They do not correspond to the same times of the simulation, due to the diverging behaviour of the starting. In addition to that, since the results from the reference are not provided for a long simulation, the whole behaviour cannot be compared. Indeed, for a longer simulation, the beating behaviour could be observed, as it has been done with the present results shown in Fig. 3.9. Therefore, only 6 periods are compared in Fig. 3.8. Even though the periods that are compared do not correspond to the same moments, the beating behaviour in the results from Placzec et al. [25] can be guessed. Moreover, both the high and low amplitudes of oscillations in the lift coefficient are in good agreement. That is to say, the global tendency of the beating frequency is present, since the maxima and minima from both amplitudes change. This is even more obvious in Fig. 3.9, where the results from the present study show the beating behaviour.

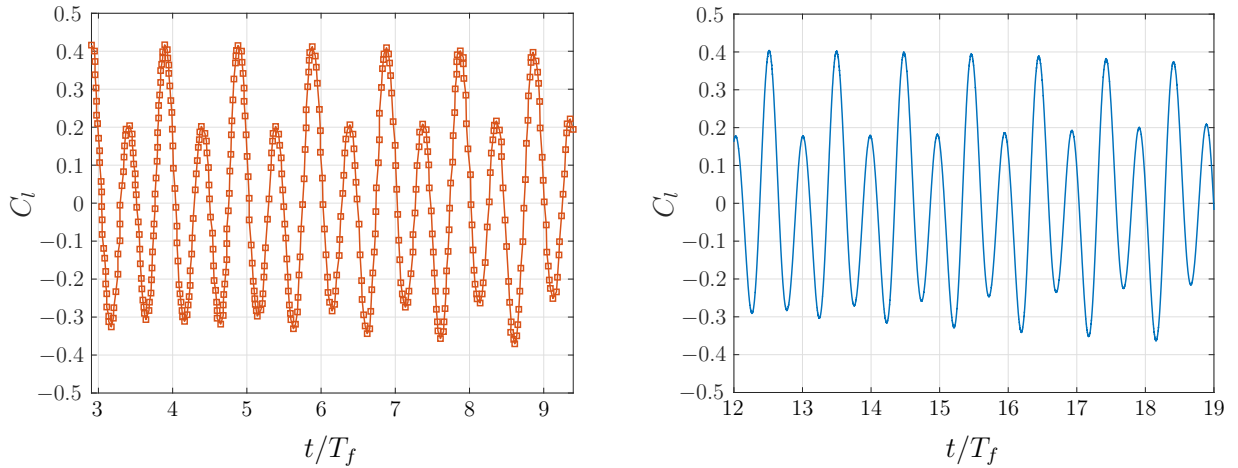


Fig. 3.8: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.5$: lift coefficient over 6 forced oscillation periods. Results from the present method (—) are run with $St = 0.1608$. Results from Placzec et al. [25] (—□—) are run with $St = 0.1669$.

The drag coefficient is also presented in Fig. 3.9, and it presents a beating behaviour likewise. However, it seems that the beating frequency for the drag and lift forces is not the same, which is related to the fact that the drag coefficients oscillates at a frequency close to twice the oscillation frequency of the lift coefficient, as it is the case for the shedding past a static cylinder, or even in the case where lock-in occurs. Finally, it should be pointed out that, as it was the case for simulation performed at $F = 0.9$, that the oscillations in the drag coefficients are more important in the drag curve than in the lift curve. Nevertheless, it seems that those high frequency oscillations have a lower amplitude than in the case $F = 0.9$, which would therefore mean that the higher the velocity, the higher the amplitude of oscillations in the force measured at the wall.

The phase portrait $C_l(y/D)$ shown in Fig. 3.10 can also depict the beating behaviour. Indeed,

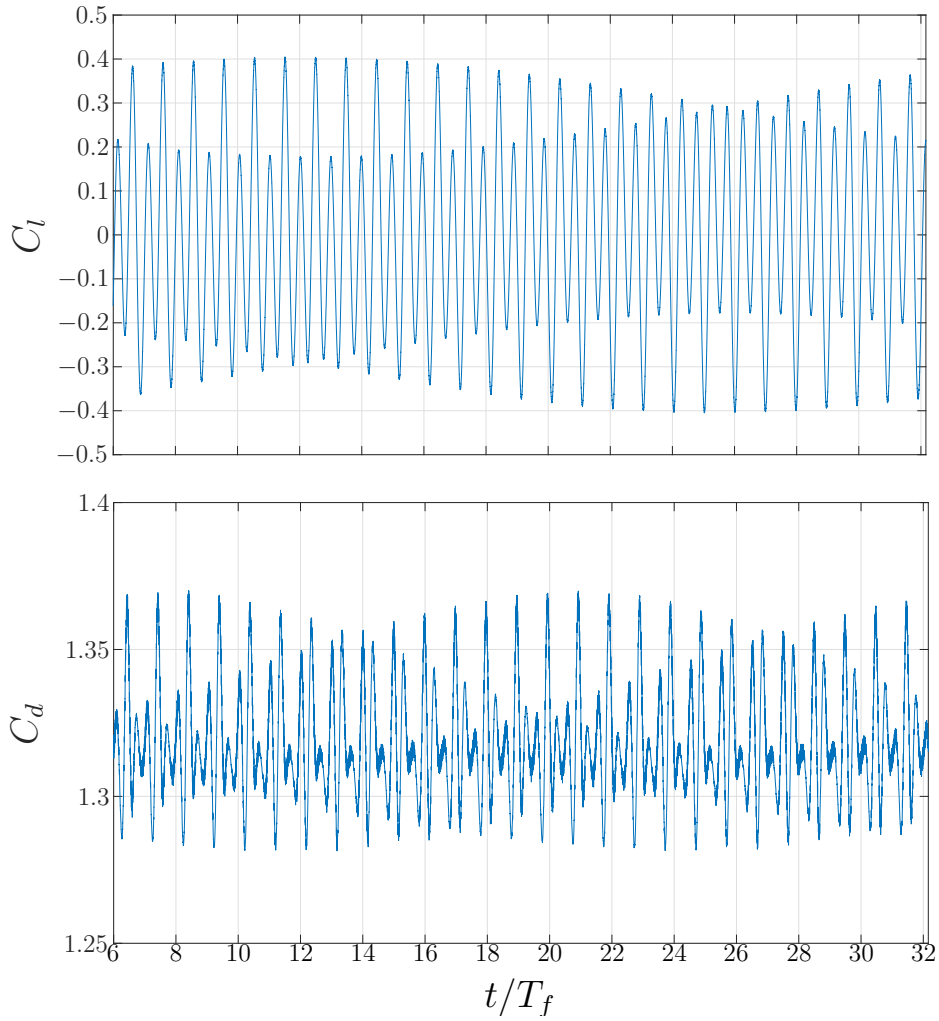


Fig. 3.9: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.5$: lift and drag coefficients over the whole simulation (discarding the transient times of the beginning of the simulation): beating frequency can be observed for 2 frequencies in the force oscillations.

the maxima of the lift coefficient shape is an ellipsoid. The curve $C_l(y/D)$ seems to have a symmetry of axis $C_l = 0$. Indeed, the ellipsoid drawn by the maxima of the lift coefficient seems to be the mirror image of the ellipsoid drawn by the minima. The two ellipsoids are not horizontal, they have a slight inclination with respect to the axis of motion y/D .

Finally, the Fourier transform of the lift coefficient is shown in Fig. 3.11, where it can clearly be seen that the two frequencies that drive the shedding regime are, first, the Strouhal frequency¹, that is, at $f/f_f = 2$, and secondly, the forced frequency. Thus, in unlocked configurations, the Strouhal frequency mainly dictates the behaviour of the vortex shedding, even though there is also the forced frequency that plays a role as well.

¹Actually, the main peak is at a frequency slightly higher than the Strouhal frequency. This might be due to a Strouhal number that is slightly underestimated.

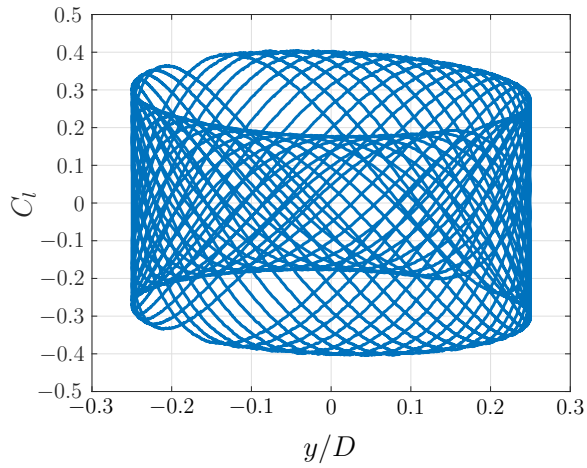


Fig. 3.10: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.5$: phase portrait of the lift coefficient: periodic behavior at 2 frequencies $f^* = 1$ and 2 and beating frequency can be observed by the ellipsoid shape of the curves that connect the maxima and minima (two ellipsoids are visible).

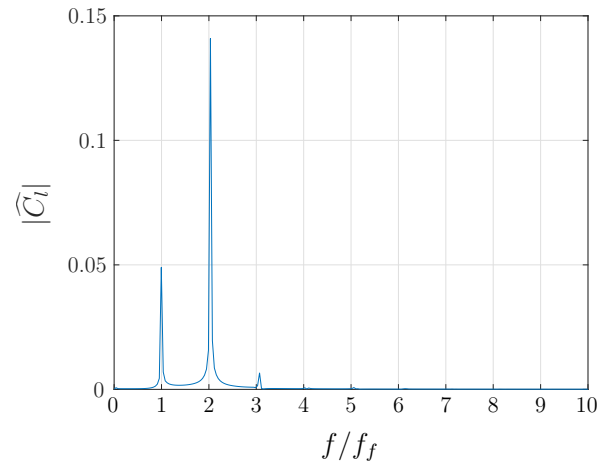


Fig. 3.11: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.5$: FFT of the lift coefficient: main peak at $f^* = f/f_f \simeq 2$ (actually, f^* is slightly above 2), second important peak at $f^* = 1$ corresponding to the forced oscillation frequency. The natural Strouhal frequency drives the behavior of the vortex shedding in competition with the forced oscillation.

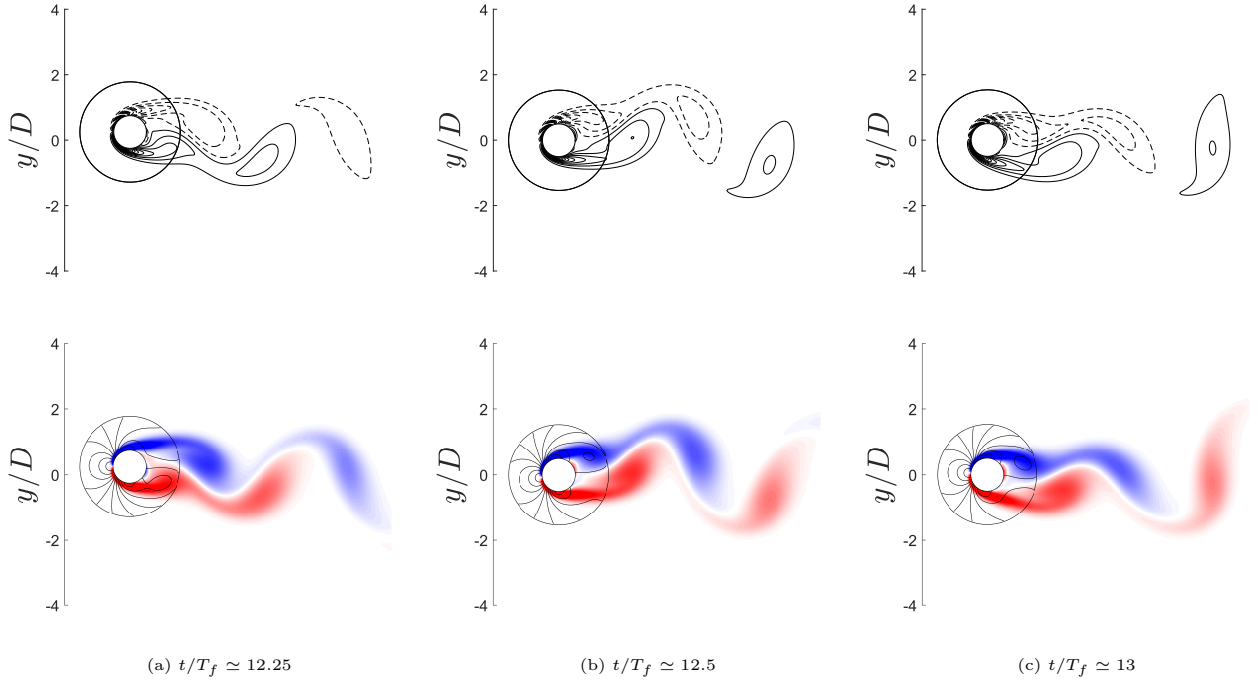


Fig. 3.12: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.5$: contours of dimensionless vorticity in FDoom and VPM solvers. Levels are by steps of 1 (starting from $|\omega^*| = 0.5$) - levels for P^* are by steps of 0.2 (starting from P_{max}^*). The colour plot is saturated for $|\omega^*| > 3$. The limit between the FDoom and VPM results is represented by a thin black line.

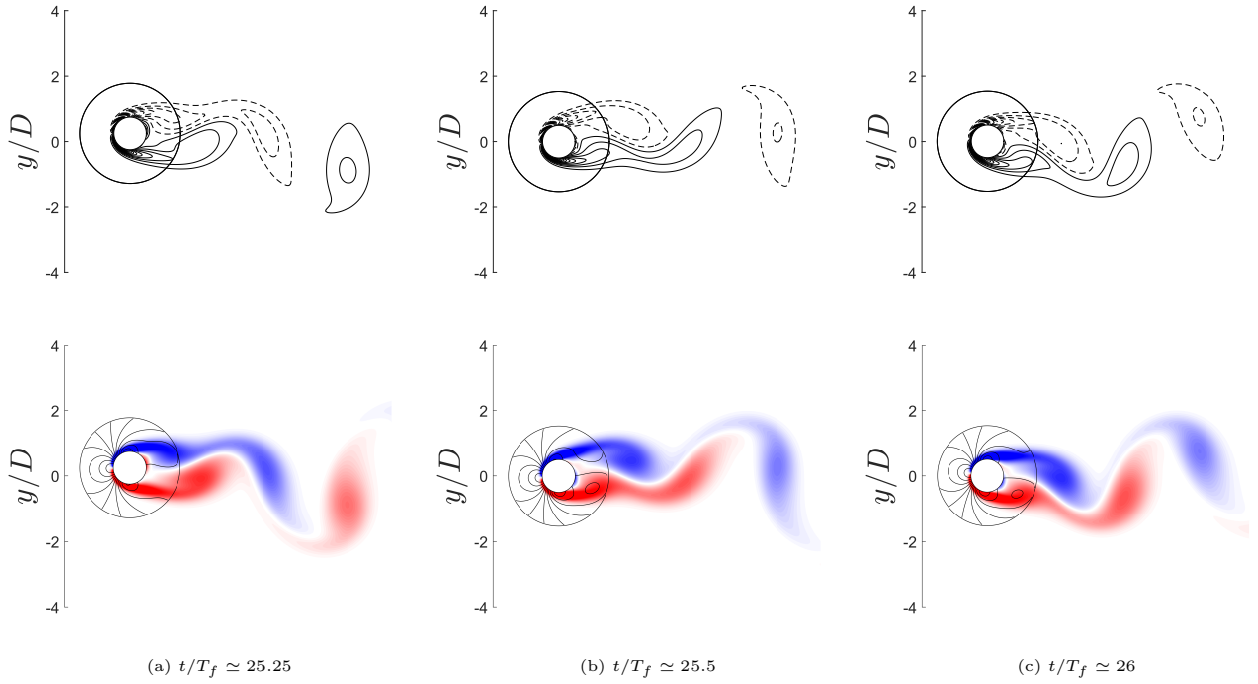


Fig. 3.13: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 0.25$, $F = 0.5$: contours of dimensionless vorticity in FDoom and VPM solvers. Levels are by steps of 1 (starting from $|\omega^*| = 0.5$) - levels for P^* are by steps of 0.2 (starting from P_{max}^*). The colour plot is saturated for $|\omega^*| > 3$. The limit between the FDoom and VPM results is represented by a thin black line.

3.1.3 Case 3: Locked configuration $Re = 100$ - $A^* = 1.0$ - $F = 0.9$

For cases where the amplitude of oscillations increases, the vortex shedding mode will be strongly impacted as well. Indeed, according to Williamson and Roshko [18], the shedding regime that corresponds to such a movement is called a $2S$ mode, meaning that there is actually two single vortices of opposite sign that are shed consecutively. However, Placzec, Sigrist and Hamdouni [25] took the values (A^*). This is probably due to the fact that the Reynolds number is very low ($Re = 100$), while in the results of Williamson and Roshko, the range of Reynolds number was higher: $300 < Re < 1000$.

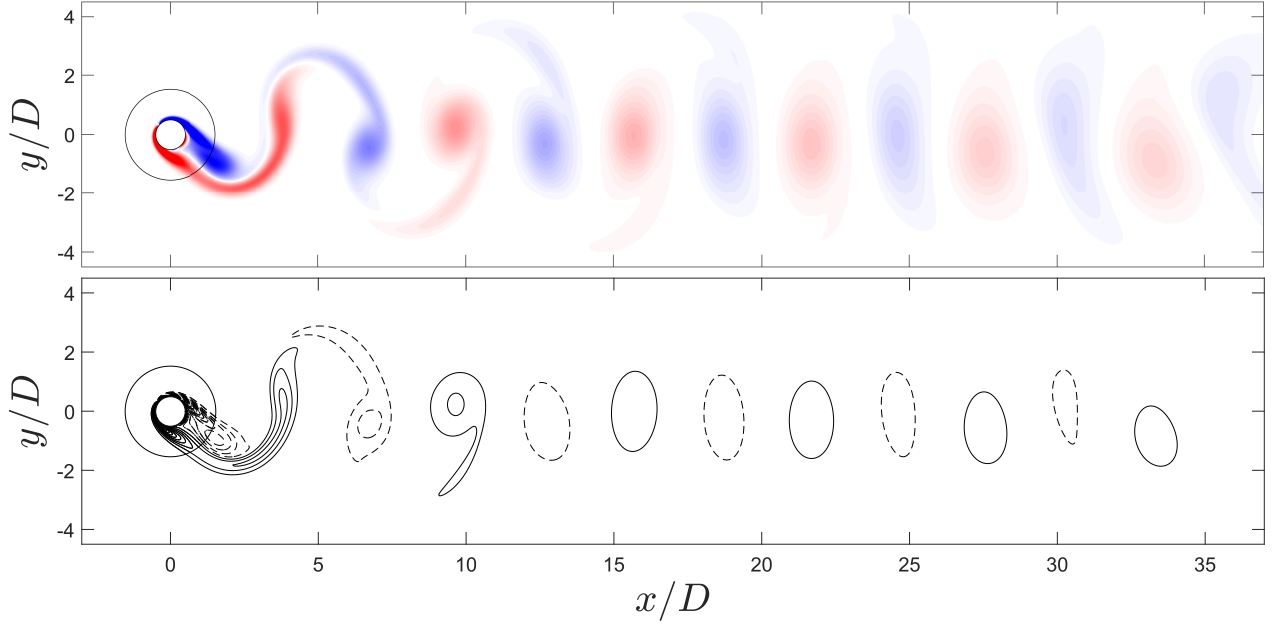


Fig. 3.14: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 1$, $F = 0.9$: contours of dimensionless vorticity and pressure (bottom) at $t/T_f \simeq 28$ in the FDoom and VPM solvers. Levels for ω^* are by steps of 1 (starting from $|\omega^*| = 0.5$) - levels for P^* are by steps of 0.2 (starting from P_{max}^*). The colour plot is saturated for $|\omega^*| > 4$. The limit between the FDoom and VPM results is represented by a thin black line.

In Fig. 3.15, a zoom on the cylinder at three times of the oscillation period shows the consistency in between the VPM and FDoom fields through the outer boundary of the FDoom solver. In addition to that, it can be seen that, due to the important amplitude of the forced movement, the position of the stagnation point on the cylinder significantly moves throughout the period of oscillation. When looking at the plots at $t/T_f \simeq 27.5$ and $t/T_f \simeq 28$, it seems that the vortex shedding is symmetric up/down over one motion period. This can also be observed by looking at the phase portrait of the lift coefficient presented in Fig. 3.17. However, due to the high frequency oscillations of the forces computed with the wall quantities, the averaged values given in Table 3.3 depict a non-symmetric behaviour of the lift coefficient.

Table 3.3: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 1.0$, $F = 0.9$: force coefficients computed with wall quantities of the NW solver averaged over 16 periods of forced oscillation (T_f).

$\overline{C_d}$	$\overline{C_d^{min}}$	$\overline{C_d^{max}}$	$\overline{C_l}$	$\overline{C_l^{min}}$	$\overline{C_l^{max}}$
2.107	1.300	3.044	-0.007	-1.356	1.304

In Fig. 3.16, the lift coefficient curve from the present results is in good agreement with the

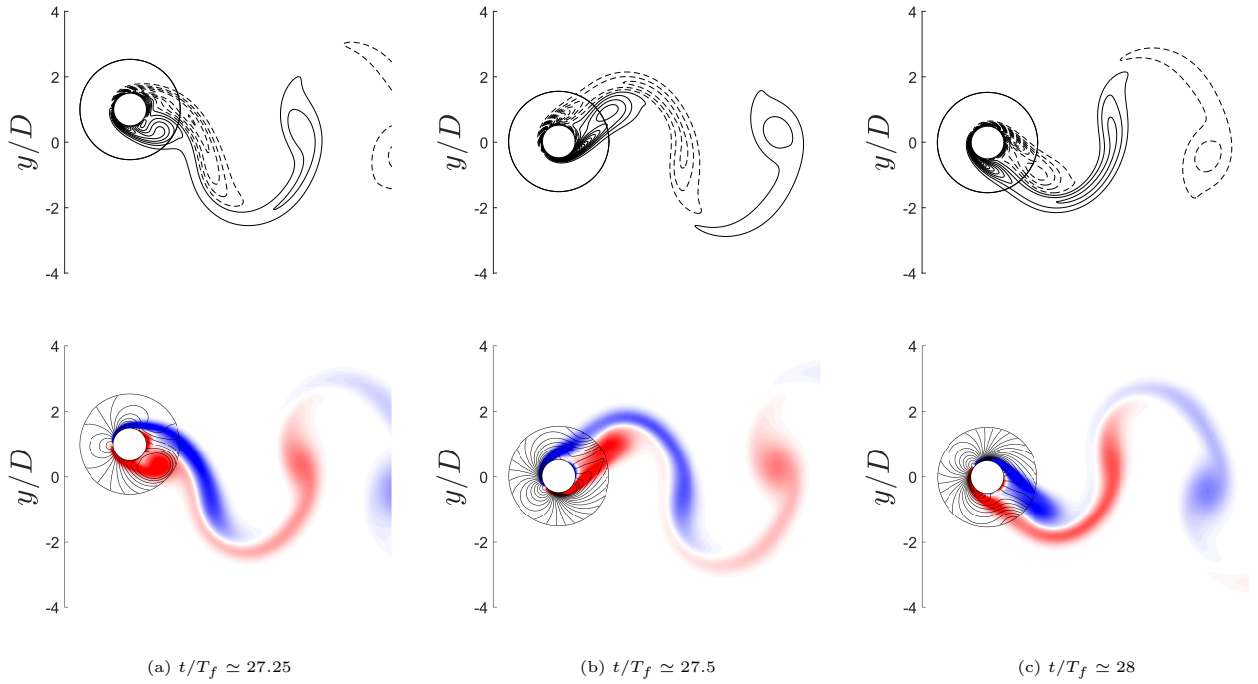


Fig. 3.15: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 1$, $F = 0.9$: contours of dimensionless vorticity in FDoom and VPM solvers. Levels are by steps of 1 (starting from $|\omega^*| = 0.5$) - levels for P^* are by steps of 0.2 (starting from P_{max}^*). The colour plot is saturated for $|\omega^*| > 4$. The limit between the FDoom and VPM results is represented by a thin black line.

results from Placzec et al. [25] that have been shifted in time in order to properly compare them with results from the present method. The curve of the drag coefficient presents, again, more important oscillations than the lift coefficient. The phase portraits of the lift and drag coefficient are provided in Fig. 3.17. The drag presents a shape of phase portrait similar to that of the case where $A^* = 0.25$ (and at the same forced frequency). On the other hand, the phase portrait of the lift coefficient is different from that of the case where $A^* = 0.25$. In effect, due to the large oscillation amplitude and, therefore, the formation of a tail to the shedded vortices, the lift coefficient does not follow a purely sinusoidal evolution (see Fig 3.16), as it is the case for the lower amplitude of oscillation (see Fig. 3.5).

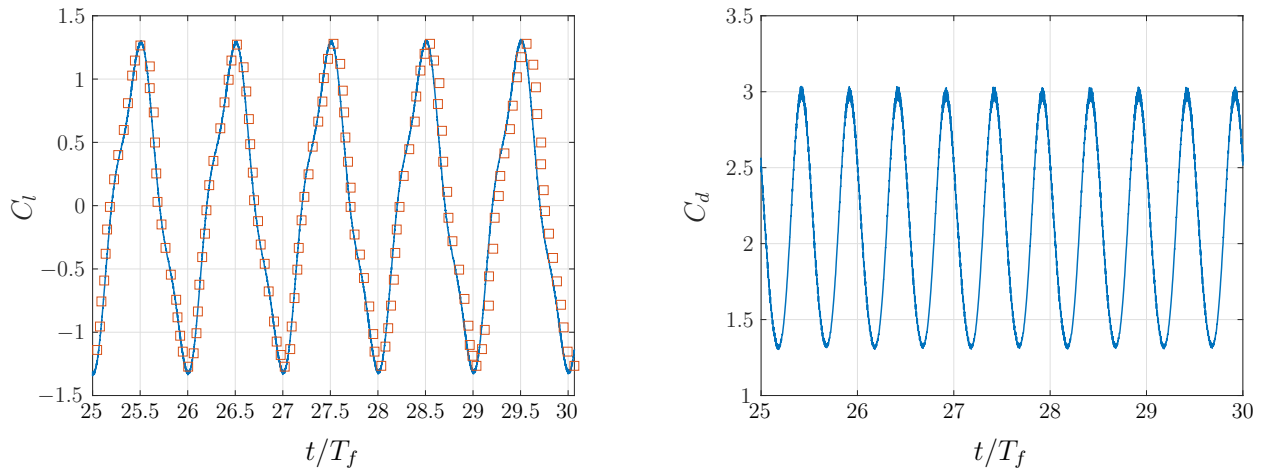


Fig. 3.16: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 1$, $F = 0.9$: lift coefficient over 6 forced oscillations. Results from the present method (—) and from Placzec et al. [25] ($\square$).

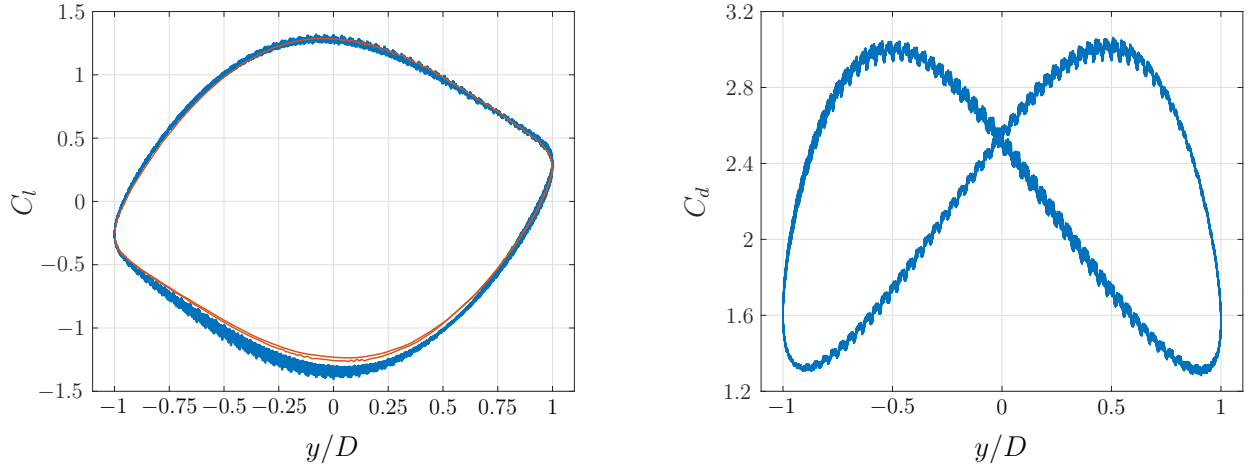


Fig. 3.17: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 1$, $F = 0.9$: phase portraits of the lift and drag coefficients. Results from the present method (—) and from Placzec et al. [25] (—).

The combined phase portrait of the lift and drag coefficients is depicted in Fig. 3.18. Even though, there are high frequency oscillations in the curve, it also seems to present a symmetric curve with $C_l = 0$ as an axis of symmetry.

Finally, the Fourier transform of the lift coefficient shows that three frequencies influence the behaviour of the vortex shedding, with the main one that obviously is the forced frequency, as the vortex shedding is synchronized with the forced frequency, which means that lock-in occurs. Smaller peaks appear at higher frequencies $f/f_f \simeq 2$ and 3. They probably depict the high frequency oscillations that have a periodic behaviour, as it can be seen in the phase portrait of the lift coefficient in Fig. 3.17.

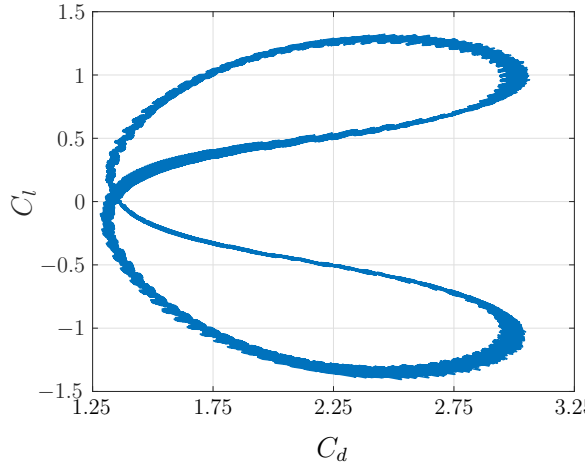


Fig. 3.18: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 1$, $F = 0.9$: combined phase portrait of the lift and drag coefficients from the present method.

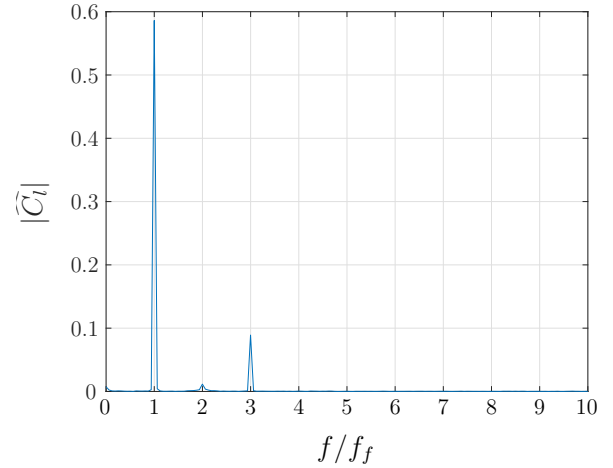


Fig. 3.19: Transverse oscillation of a cylinder - $Re = 100$, $A^* = 1$, $F = 0.9$: FFT of the lift coefficient: main peak at $f^* = f/f_f = 1$, showing the synchronization of the vortex shedding with the forced oscillation f_f of the cylinder. Lower peaks are at $f^* = 2$ and 3.

3.2 Rotating cylinder at $Re = 200$

The dimensionless parameter that characterizes the rotating motion of the cylinder is

$$\alpha = \frac{\Omega R}{U_\infty},$$

where Ω is the angular speed of the cylinder, R is the radius of the cylinder and U_∞ is the freestream velocity. This parameter represents the ratio of the azimuthal velocity at the surface of the cylinder with respect to the freestream velocity. In the present simulations, the initial condition is set to the potential flow solution of a static cylinder, and after that, the rotation rate is directly set to Ω .

Three cases are presented in this section. The case of a rotating cylinder at different rotation speeds. The first case is a simulation at $\alpha = 0.5$. At this rate of rotation, vortex shedding is still observable. For the two other runs, the rates of rotation are given by $\alpha = 2.5$ and $\alpha = 3.0$. For these cases, the vortex shedding vanishes, and lobes of vorticity of opposite sign are wrapped around the cylinder. What is interesting in these cases is that the flow gets closer to the potential flow, but at a smaller rotation speed. In fact, the streamlines of the potential flow solutions are like the streamlines of the viscous flows investigated in this section. However, the difference is that, for similar streamlines, the potential flow needs a lower rate of rotation. The results are compared with those from Mittal and Kumar [29]. They used a finite element method for solving the Navier-Stokes in primitive variables. The computational domain is made of 31,800 elements (and 32,190 nodes), with a length of $100 D$. Concerning the top and bottom boundaries, a slip wall ($v = 0$) and Neumann ($\partial u / \partial y = 0$) conditions are imposed, while for the outflow boundary a Neumann condition on the velocity is imposed with $\partial u / \partial x = 0$ and $\partial v / \partial x = 0$. In their paper, Mittal and Kumar [29] also provide a mesh independence study, in which they assess that the position of the outflow boundary does not influence the value of the lift coefficient, but that it influences the values of the drag coefficient for a domain size $L/D < 75$. The effect of the relative position of the outflow boundary with respect to the cylinder position has also been observed in the present study; it is discussed in Section 3.2.4. For the case $\alpha = 3.0$, the results from Padrino and Joseph [30] are also used.

It is also important to stress the fact that, for the case of a rotating cylinder, an artefact can be used in the present method so as to keep the Eulerian mesh fixed. Therefore, in the three cases that are presented, the results from the present study are always presented with and without this artefact. The use of the artefact removes the errors that could come from the interpolation procedure, as this step is very similar (but still different²) to the interpolation procedure used by Billuart et al. [1].

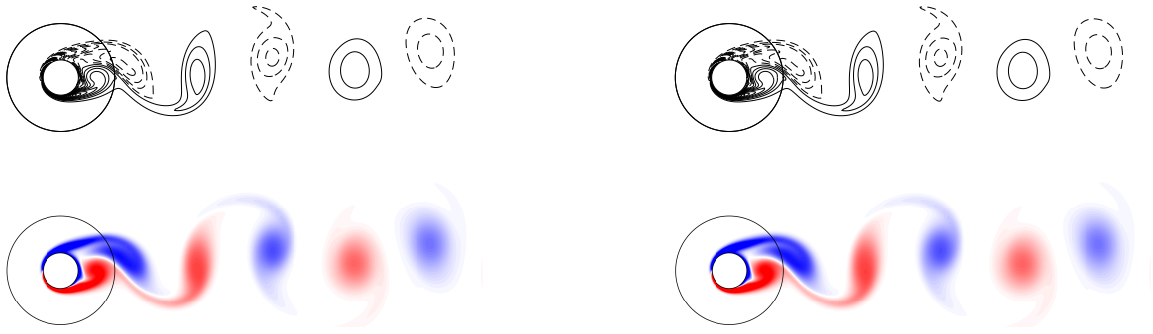
Three results can thus be compared: the results from the present *classical* method (with the Eulerian mesh that rotates), the present method with the use of the artefact, and the results from Mittal and Kumar [29]. Globally, the results that are obtained are in good agreement with those of Mittal and Kumar, except the drag coefficient. Indeed, it has been observed that the drag coefficient was strongly impacted by the outflow. In fact, as soon as the starting vortex reaches the right boundary of the VPM solver, there is a deflection in the freestream velocity

²The difference comes from the fact that spatial and temporal interpolations have been reversed, as explained in Section 2.3.2.

direction at the inflow, which impacts the drag. While the lift force is only influenced during a transient period, the drag coefficient is impacted over the remaining of the simulation.

3.2.1 Case 1: $Re = 200$ - $\alpha = 0.5$

For small values of α , vortex shedding is observed, while, for larger values, a steady state is reached. Mittal and Kumar [29] claim that vortex shedding is observed for $\alpha < 1.91$. As already mentioned before, for the case of a rotating cylinder, an artefact can be used in order to keep the Eulerian mesh fixed with respect to the inertial frame of reference. The vorticity field obtained with both present method and present method with the fixed mesh are compared in Fig. 3.20. They are in good agreement and, moreover, the vorticity field is consistent through the outer boundary of the FDoom solver. The vortex shedding that is observed is close to the vortex shedding past a static cylinder, but the direction of the wake is not horizontal as it is the case for a static cylinder. Indeed, there is a slight upward deflection of the wake, due to the circulation generated by the rotating cylinder.



(a) Present coupling with rotating Eulerian mesh - $t U_\infty / D \simeq 219.53$

(b) Present coupling with fixed Eulerian mesh - $t U_\infty / D \simeq 219.47$

Fig. 3.20: Rotating cylinder - $Re = 200$, $\alpha = 0.5$: contours of dimensionless vorticity in FDoom and VPM solvers shown over $12 D$ downstream of the cylinder. Levels are by steps of 1 (starting from $|\omega^*| = 0.5$). The colour plot is saturated for $|\omega^*| > 4$. The limit between the FDoom and VPM results is represented by a thin black line. Good agreement between the vorticity contours of both methods.

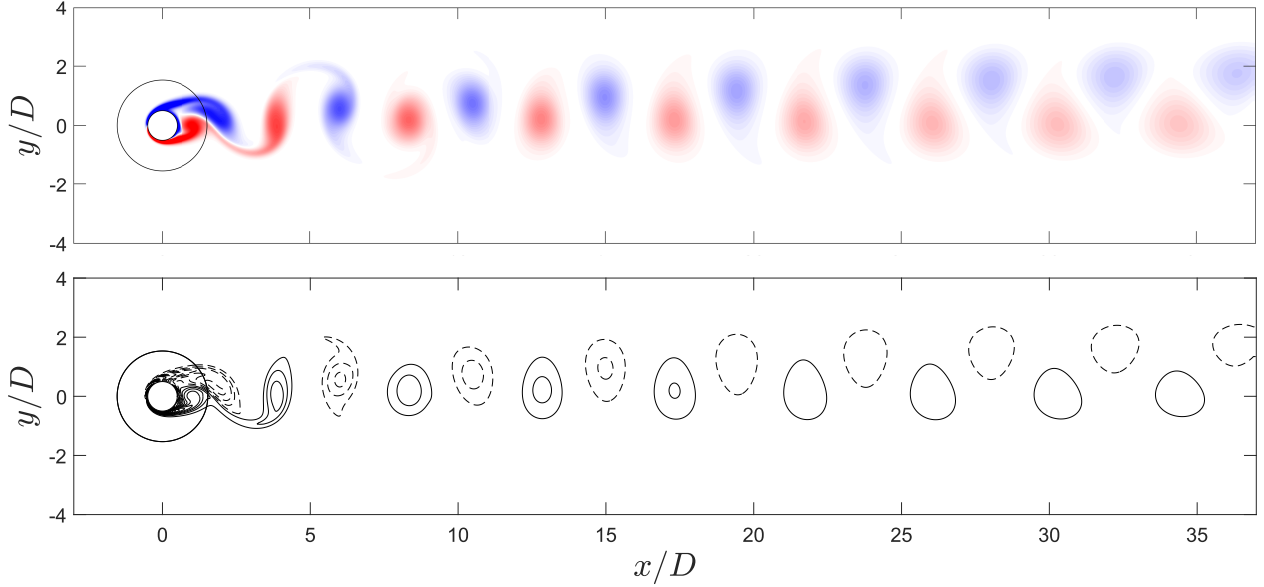
In Fig. 3.21, the vorticity field over the whole length of the computational domain is depicted, demonstrating the deflection of the long wake (that extends up to 37 diameters downstream of the cylinder) due to the rotation of the cylinder. This plot uses the results from the present method with the Eulerian mesh kept fixed, as it is considered as the reference compared to the case where the Eulerian mesh rotates in the inertial frame of reference. Indeed, as the results that are presented in what follows show, the present method with a fixed mesh obtains better results, i.e. the forces obtained with the wall quantities provide less noisy curves due to the fact that the interpolation of the boundary condition for the NW solver provides very accurate results when the Eulerian mesh remains fixed. In addition to that, since there is a vortex shedding, a Strouhal number can be computed. Based on the Fourier transform of the lift force³, the main frequency in the spectral signal of the lift curve can be retrieved, and, thus, the pseudo-Strouhal numbers can be computed. The values obtained with both present methods are provided in Table 3.4.

The curves of the lift and drag coefficients for a few dimensionless times are shown in Fig. 3.22, and the time-averaged, maximum and minimum values are provided in Table 3.5, compared with the values from Mittal and Kumar [29]. The lift coefficient fluctuates as in the case of the

³The values are taken from the moment when the simulation is statistically converged until the end of the simulation

Table 3.4: Rotating cylinder - $Re = 200$, $\alpha = 0.5$: Strouhal number and Strouhal period based on the FFT of the lift coefficient.

	St	T_{St}
Present method with rotating Eulerian mesh	0.1908	0.131
Present method with fixed Eulerian mesh	0.1921	0.130


 Fig. 3.21: Rotating cylinder - $Re = 200$, $\alpha = 0.5$: contours of dimensionless vorticity in FDoom and VPM solvers from the present method with fixed Eulerian mesh at $t^* \simeq 219.5$. Levels are by steps of 1 (starting from $|\omega^*| = 0.5$). The colour plot is saturated for $|\omega^*| > 4$. The limit between the FDoom and VPM results is represented by a thin black line.

vortex shedding of a fixed cylinder, but the values are not symmetric around $C_l = 0$, due to the *Magnus effect*, as shown in Fig. 3.22. Indeed, for a rotating cylinder, a negative lift force is created, due to the circulation created by the rotating body. The acceleration of the flow on the lower side of the cylinder leads to a decrease of pressure, while on the upper side of the cylinder, the azimuthal velocity on the edge of the cylinder is opposite to the freestream direction, which therefore leads to an increase of pressure. Therefore, a downward resultant force is applied on the cylinder. After that, it can be observed that there is a slight offset between the lift curves from the present method and present method with the Eulerian mesh fixed. This means that, for the case where the Eulerian mesh rotates, the impulsive starting of the cylinder does not present the same transient behaviour, which thus leads to a slight time offset. Regarding the drag coefficient, the behaviour is different from the purely sinusoidal shape of that for a fixed cylinder. In fact, when the maximum peak of drag is reached, the drag does not decrease monotonically, it has a second peak of lower amplitude due to the rotating movement, which therefore differentiates the drag curve of a static cylinder from a rotating cylinder, both experimenting vortex shedding. What can also be clearly seen in Fig. 3.22, is that for the case where the Eulerian mesh is fixed, there is no high frequency oscillations as it is the case when the Eulerian mesh is rotating. This is especially obvious in the drag coefficient curve. The combined phase portrait of the lift and drag coefficient is depicted in Fig. 3.23.

In Fig. 3.24, the long wake of the vorticity field is again depicted, but, in addition to that, the

time-averaged vorticity and horizontal velocity fields over $\Delta t^* \simeq 109$ are shown. The latter depict the asymmetry of the vorticity field with respect to $x/D = 0$. Indeed, the region where the time-averaged vorticity field is zero is not horizontal but follows a line that is deflected upward.

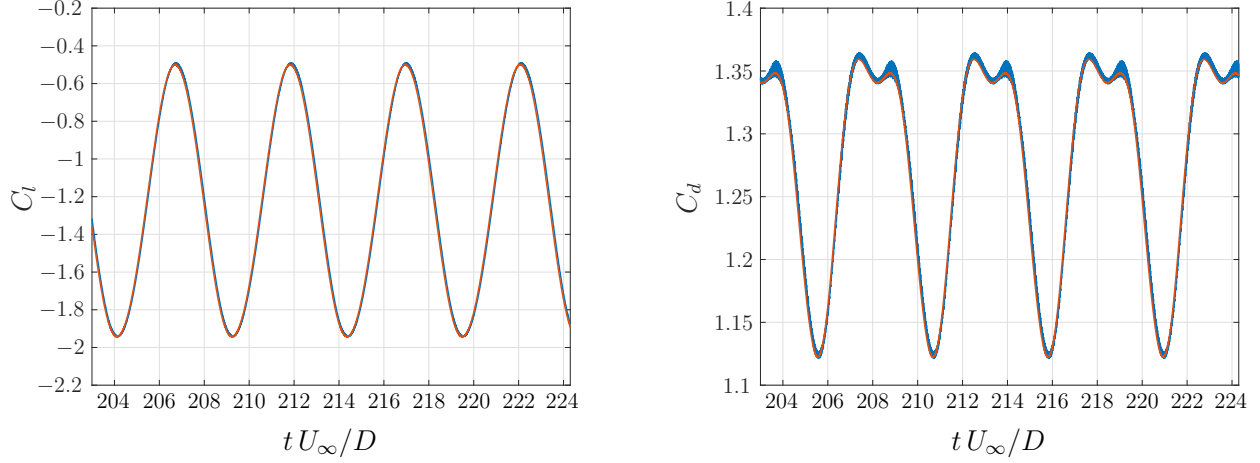


Fig. 3.22: Rotating cylinder - $Re = 200$, $\alpha = 0.5$: lift and drag coefficients. Results from the present method (—) and the present method with the Eulerian mesh held fixed (—).

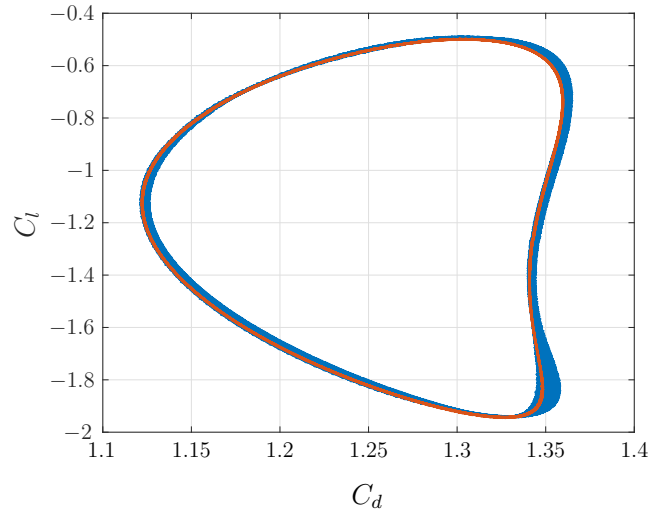


Fig. 3.23: Rotating cylinder - $Re = 200$, $\alpha = 0.5$: phase portrait of the combined lift and drag coefficients. Results from the present method (—) and the present method with the Eulerian mesh held fixed (—).

Table 3.5: Rotating cylinder - $Re = 200$, $\alpha = 0.5$: force coefficients computed with wall quantities of the NW solver averaged over 17 Strouhal periods¹.

	$\overline{C_d}$	$\overline{C_d^{min}}$	$\overline{C_d^{max}}$	$\overline{C_l}$	$\overline{C_l^{min}}$	$\overline{C_l^{max}}$
Present method	1.278	1.121	1.365	-1.225	-1.944	-0.489
Present method (fixed mesh)	1.275	1.122	1.360	-1.229	-1.945	-0.498
Mittal and Kumar [29]	/	1.123	1.347	/	-1.990	-0.599

¹For the present methods, the respective Strouhal periods provided in Table 3.4 are used.

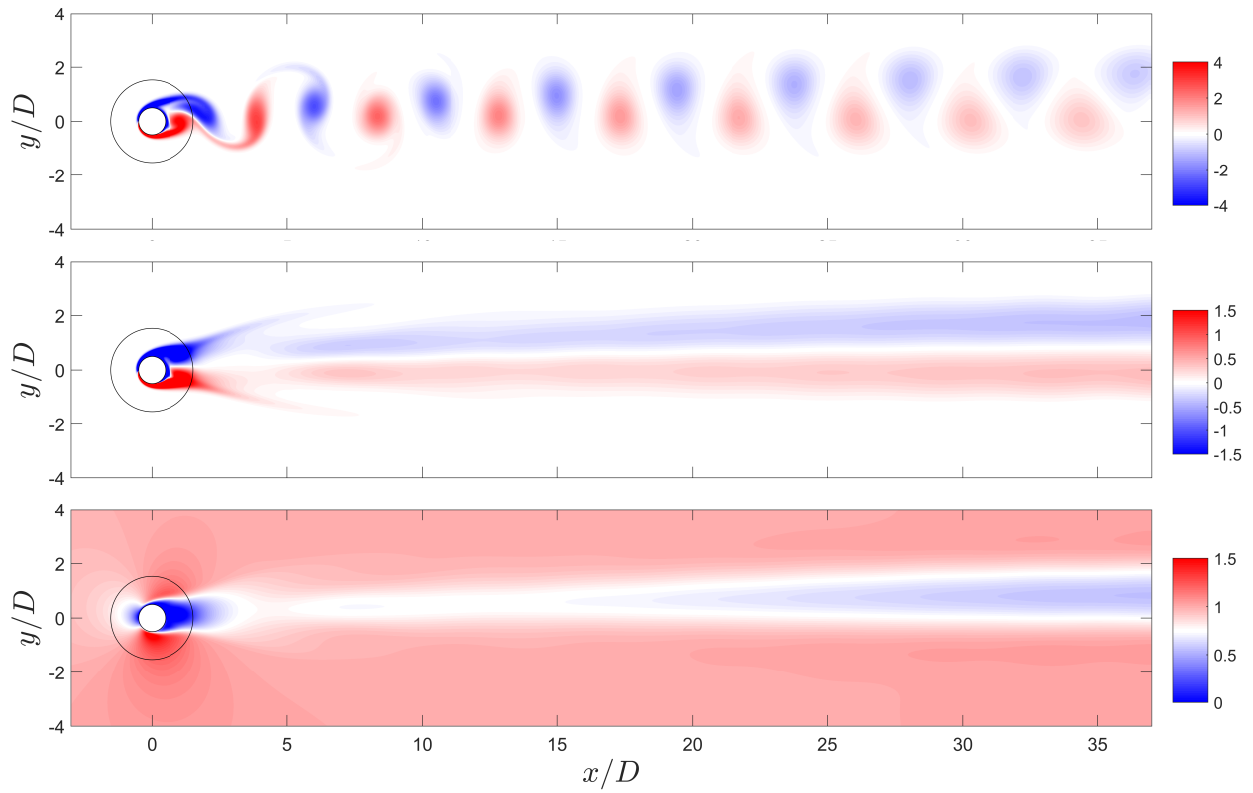


Fig. 3.24: Rotating cylinder - $Re = 200$, $\alpha = 0.5$: Top: $\omega^*(t^* \simeq 220)$. Middle: $\overline{\omega^*}$ averaged over $\Delta t^* \simeq 109$, i.e.: 21 Strouhal periods. Bottom: $\overline{u_x}/U_\infty$ averaged over $\Delta t^* \simeq 109$. The limit between the FDoom and VPM results is represented by a thin black line.

3.2.2 Case 2: $Re = 200 - \alpha = 2.5$

As previously mentioned, for values of α larger than 1.91, vortex shedding is no longer observed, but a steady state flow is established. This can be observed in Fig. 3.25, where a zoom on the vorticity field and the underlying iso-contours is provided. It shows the comparison of the steady state vorticity fields in-between both the present method and present method with the Eulerian mesh held fixed in the inertial frame of reference. The iso-contours show that there is a slight "kink" near the outer boundary of the near-wall solver. Indeed, when comparing with high precision the iso-contours near the outer boundary $\partial\Omega_{NW}^{outer}$, it can be seen that the iso-contours are not smooth through the boundary, while it is the case for the results from the present method with the static Eulerian mesh. Furthermore, it is even more obvious that there is a difference since some of the iso-contours do not extend as far in the case where the Eulerian mesh is rotating. This can be especially obvious for the positive iso-contour that extends the farthest in the wake and for the positive iso-contour that closes up near the outer boundary $\partial\Omega_{NW}^{outer}$.

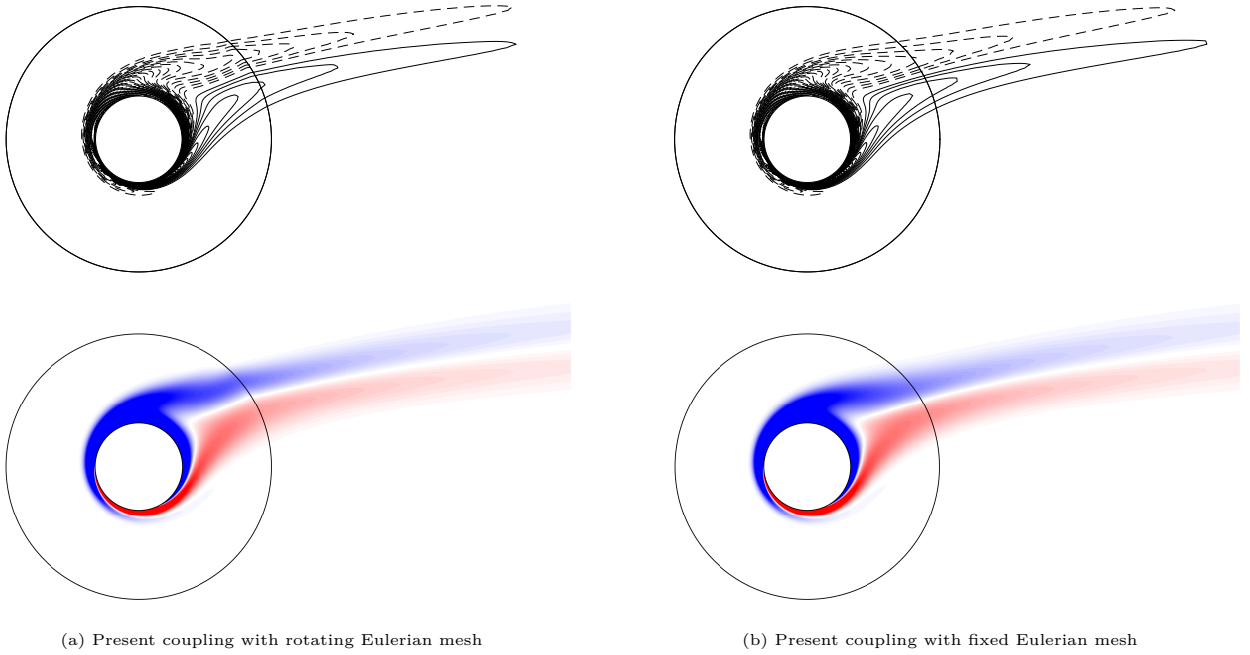


Fig. 3.25: Rotating cylinder - $Re = 200$, $\alpha = 2.5$: contours of dimensionless vorticity at $t^* \simeq 205$ in FDoom and VPM solvers shown over $8D$ downstream of the cylinder. Levels are by steps of 0.5 (starting from $|\omega^*| = 0.5$). The colour plot is saturated for $|\omega^*| > 4$. Inconsistency can be observed in the vorticity field of both methods.

In Fig. 3.26, a zoom near the cylinder depicts the norm of the velocity field and the corresponding streamlines for the time-averaged solution from the present method with the Eulerian mesh kept fixed. Even though the velocity field has been time-averaged, one should have observed the same fields for an instantaneous plot, since a steady state is reached. After that, the position of the saddle points near the surface of the cylinder are shown in Fig. 3.27. From those plots, the approximate positions of the saddle point can be figured out. The latter are provided in Table 3.6. On the left of the second saddle point, a large recirculation area is present.

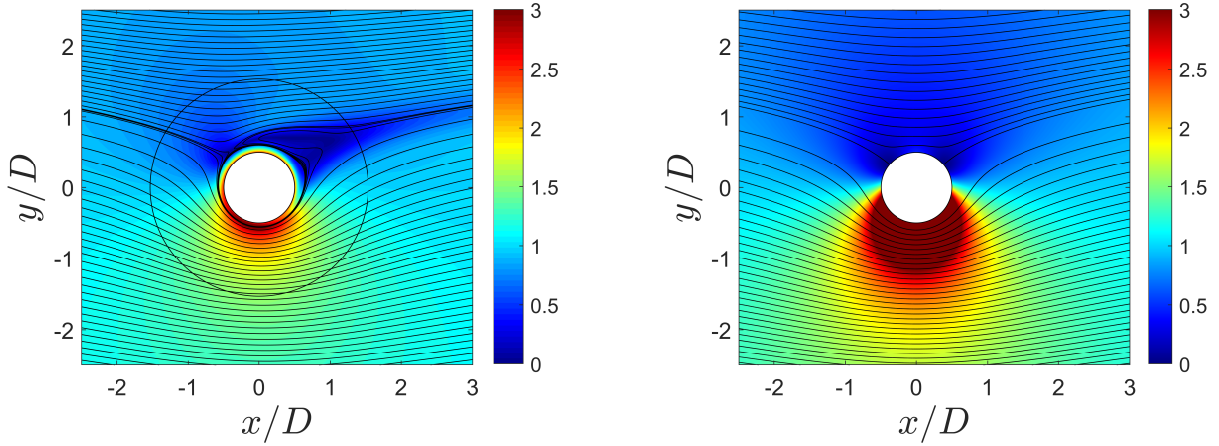


Fig. 3.26: Rotating cylinder - $Re = 200$: comparison between the viscous case from the present study $\alpha = 2.5$ (left) and the inviscid case $\alpha_{\text{pot}} = 1.3$ (right): streamlines and colour plot corresponding to the dimensionless norm of the velocity $\|\bar{\mathbf{u}}\|/U_\infty$. Results from the present work are averaged over $\Delta t^* \simeq 144$.

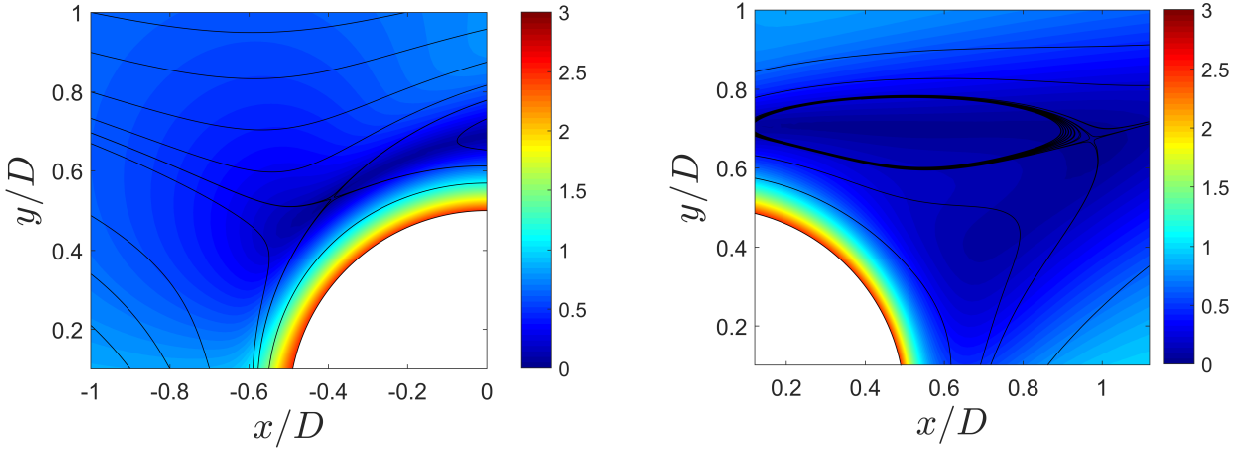


Fig. 3.27: Rotating cylinder - $Re = 200$, $\alpha = 2.5$: zoom on the two saddle points located about $(x/D \simeq -0.390; y/D \simeq 0.531)$ and $(x/D \simeq 0.972; y/D \simeq 0.672)$, respectively. The colour plot corresponds to the dimensionless norm of the velocity $\|\bar{\mathbf{u}}\|/U_\infty$.

Table 3.6: Rotating cylinder - $Re = 200$, $\alpha = 2.5$: approximate position of the saddle points from the present method with the Eulerian mesh held fixed.

	x/D	y/D
Saddle point 1	-0.390	0.531
Saddle point 2	0.972	0.672

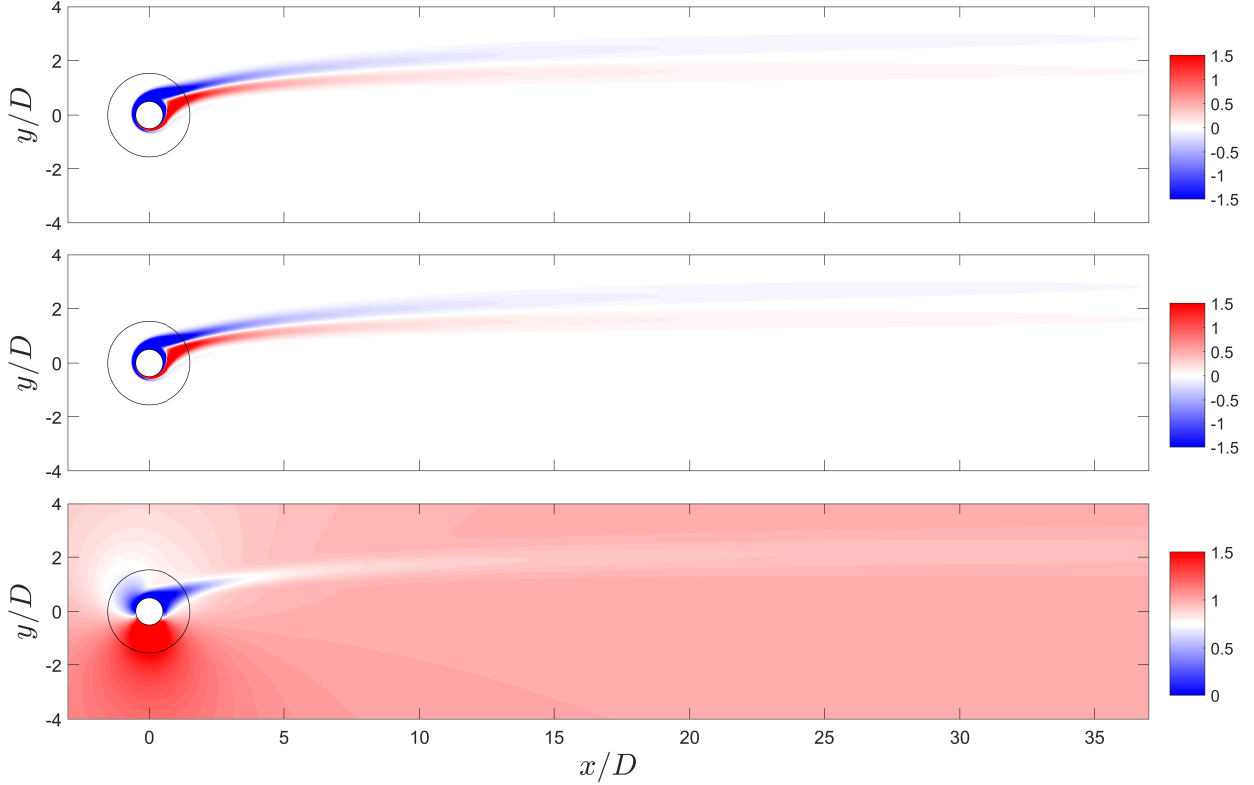
The comparison of the instantaneous vorticity field regarding the time-averaged vorticity field is provided in Fig. 3.28. As already mentioned, both plots are equivalent, since the steady state is reached for such a value of α . In addition to that, the time-averaged horizontal velocity is also provided. The mean values⁴ of the lift and drag coefficients are compared for both the present methods, with the results from Mittal and Kumar [29] in Table 3.7.

Finally, the dimensionless wall vorticity from both present methods and from Mittal and

⁴The mean is computed once the flow reaches the steady state.

Table 3.7: Rotating cylinder - $Re = 200$, $\alpha = 2.5$: force coefficients computed with the averaged wall quantities from the NW solver.

	$\overline{C_d}$	$\overline{C_l}$
Present method	0.092	-7.650
Present method (fixed mesh)	0.056	-7.648
Mittal and Kumar [29]	0.146	-7.614


 Fig. 3.28: Rotating cylinder - $Re = 200$, $\alpha = 2.5$: Top: $\omega^*(t^* \simeq 205)$. Middle: $\overline{\omega^*}$ averaged over $\Delta t^* \simeq 144$. Bottom: $\overline{u_x}/U_\infty$ averaged over $\Delta t^* \simeq 144$. The limit between the FDoom and VPM results is represented by a thin black line.

Kumar [29] is provided in Fig. 3.29. The values from the present methods are the values corresponding⁵ to $t^* \simeq 205$. There is a slight offset of the values from the present method with respect to the values from Mittal and Kumar [29]. In addition to that, the maximum and minimum values of the vorticity at the wall are slightly larger with present methods than those of the reference. The skin friction, for a rotating cylinder, can be computed with

$$\begin{aligned} \frac{\tau_{\text{wall}}}{\mu} &= \hat{\mathbf{e}}_\theta \cdot (\hat{\mathbf{n}} \times (\boldsymbol{\omega}_{\text{wall}} - 2\boldsymbol{\Omega})) \\ &= -(\omega_{\text{wall}} - 2\Omega) , \end{aligned} \quad (3.3)$$

where $\hat{\mathbf{e}}_\theta$ is the azimuthal direction, and $\hat{\mathbf{n}}$ is the normal at the surface of the cylinder. The values where the skin friction cancels out correspond to the separation and reattachment points

⁵Actually, the values from both present methods are not exactly at the same dimensionless time t^* , but it should not matter, since the steady state is already reached.

on the surface of the cylinder. The skin friction, shown in Fig. 3.30, is made dimensionless as the wall vorticity is, i.e. with

$$\tau_{\text{wall}}^* = \frac{\tau_{\text{wall}}}{\mu} \frac{D}{U_{\infty}}. \quad (3.4)$$

The values of maximal and minimal wall vorticity, as well as their location, and the location of the attachment and separation points on the cylinder are provided in Table 3.8 for both present methods, and Mittal and Kumar [29]. It is important to stress that the exact values for the results from Mittal and Kumar [29] are not known. In fact, the values provided in Table 3.8 have been taken from digitized results, together with the wall vorticity results. Therefore, one should take an uncertainty interval⁶ on both the position and the values of ω^* .

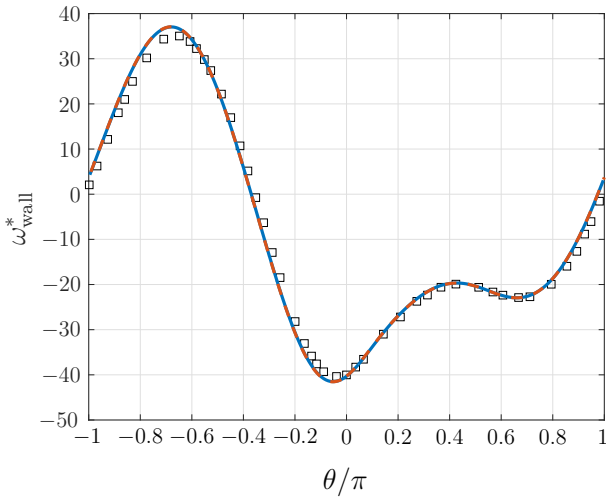


Fig. 3.29: Rotating cylinder - $Re = 200$, $\alpha = 2.5$: dimensionless wall vorticity ω^* at $t^* \simeq 205$. Results from the present method (—), the present method with fixed Eulerian mesh (---), and Mittal and Kumar [29] ($\square$).

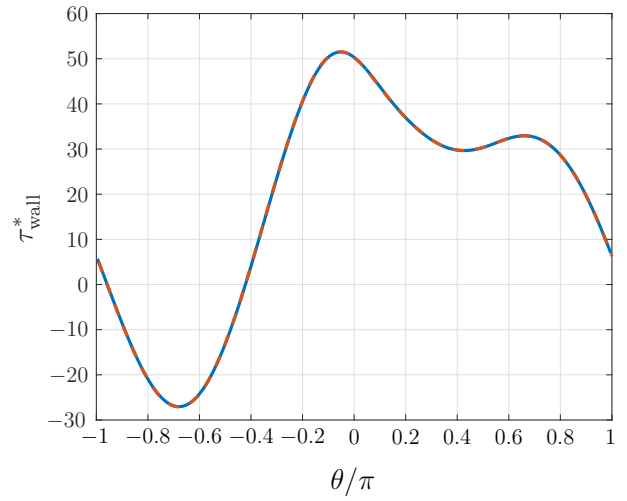


Fig. 3.30: Rotating cylinder - $Re = 200$, $\alpha = 2.5$: dimensionless skin friction τ_{wall}^* at $t^* \simeq 205$. Results from the present method (—), the present method with fixed Eulerian mesh (---). Attachment and separation points are located around $\theta \simeq -75.9^\circ$, and $\theta \simeq -172.5^\circ$, respectively.

Table 3.8: Rotating cylinder - $Re = 200$, $\alpha = 2.5$: position of the attachment (θ_a) and separation (θ_s) points at the cylinder surface, and positions and values of the maximum and minimum values of ω^* .

	θ_a	θ_s	$\theta(\omega_{\text{max}}^*)$	ω_{max}^*	$\theta(\omega_{\text{min}}^*)$	ω_{min}^*
Present method	-172.5°	-75.9°	-122.2°	37.0	-9.7°	-41.5
Present method (fixed mesh)	-172.5	-76.0	-123.0°	37.1	-9.0°	-41.5
Mittal and Kumar [29]	/	/	-119	35.3	-7	-40.2

⁶It can be considered that the values of position lie $\pm 2^\circ$ around the provided values, while the values of ω^* lie ± 0.2 the values given in Table 3.8.

3.2.3 Case 3: $Re = 200$ - $\alpha = 3.0$

The results presented in the present section are the same as in Section 3.2.2, where the dimensionless rotation rate α was 2.5. In this case, again, a steady-state is reached. The results from both Mittal and Kumar [29], and Padrino and Joseph [30] are used as references.

First, let us compare the vorticity field obtained by the present method and the present method with the Eulerian mesh held fixed. In Fig. 3.31, it can be seen that the difference between both present methods is even more exacerbated with respect to the case where $\alpha = 2.5$. In effect, the colour plot depicts low intensity vorticity slightly above the long lobe of negative vorticity. In addition to that, the "kink" near the boundary $\partial\Omega_{NW}^{\text{outer}}$ is more important as well than in the case $\alpha = 2.5$. When comparing iso-contours plots of both methods, it is also clear that the corresponding contours do not have the same length, especially the positive iso-contour of lowest intensity (i.e. the continuous iso-contour which extends the furthest in the wake in Fig. 3.31). This shows, once more, that the present method with the Eulerian mesh kept fixed must be considered as the reference from the present method. Finally, it should be noticed that in the case where the Eulerian mesh is rotating, the point where the contours cross the boundary $\partial\Omega_{NW}^{\text{outer}}$ is shifted by a positive⁷ angle.

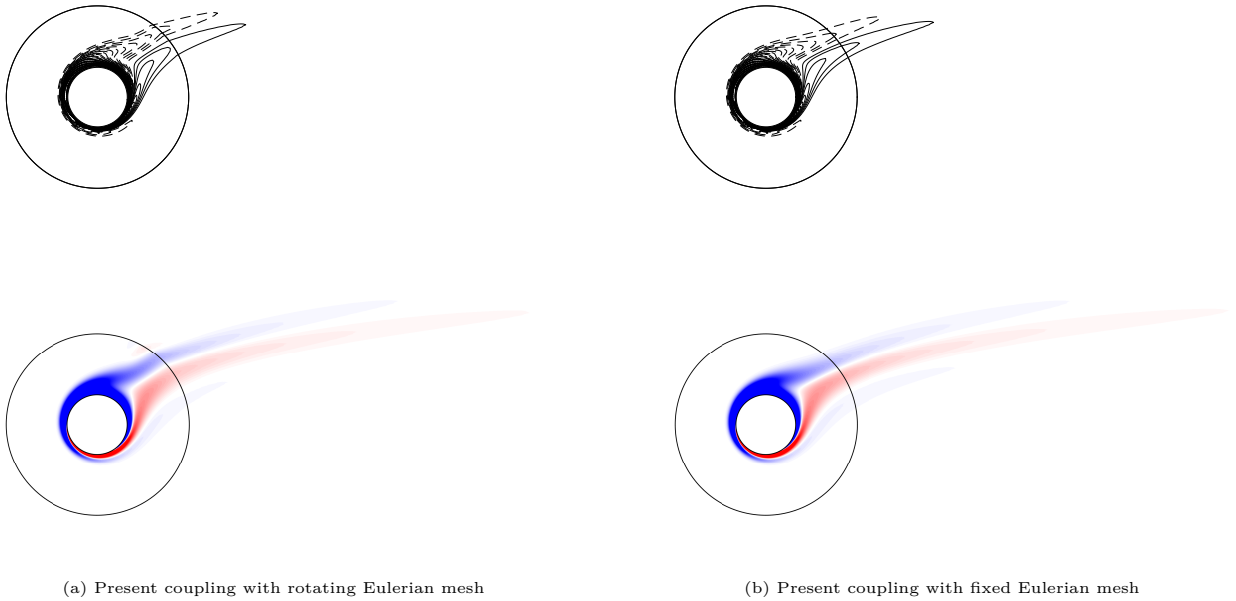


Fig. 3.31: Rotating cylinder - $Re = 200$, $\alpha = 3.0$: contours of dimensionless vorticity at $t^* \simeq 137$ in FDoom and VPM solvers shown over $8D$ downstream of the cylinder. Levels are by steps of 0.5 (starting from $|\omega^*| = 0.5$). The colour plot is saturated for $|\omega^*| > 4$. The limit between the FDoom and VPM results is represented by a thin black line.

The velocity magnitude near the cylinder, and the corresponding streamlines are also depicted in Fig. 3.32. The plot looks very similar to the case $\alpha = 2.5$. Still, the position of the saddle points and the angle formed by the streamlines upstream are different, as depicted in Fig. 3.9 and quantified in Table 3.9. In Fig. 3.32, the streamlines can also be compared with the potential flow at a dimensionless rate of rotation $\alpha_{\text{pot}} = 1.75$. The streamlines of the viscous flow past the cylinder rotating at $\alpha = 3.0$ are similar to those of the inviscid flow past the rotating cylinder

⁷Angle is counted positive in the trigonometric direction, or counter clockwise.

at $\alpha = 1.75$. The value for the rate of rotation in the potential flow case has been found by comparing the angle formed by the velocity vector around the positions $(x/D; y/D) = (-3; 0)$ and $(x/D; y/D) = (3; 0)$ with the angles obtained with the present study. The parameter α has to be smaller than 2 in the case of the potential flow, in order to have two stagnation points on the cylinder (see Appendix A for more details about the corresponding potential flow). For higher rates, there is one unique stagnation point, located above the cylinder, along the axis $x = 0$. Another difference is that the viscous flow presents a recirculation zone and has two saddle points that are not on the surface of the cylinder, unlike the potential flow case.

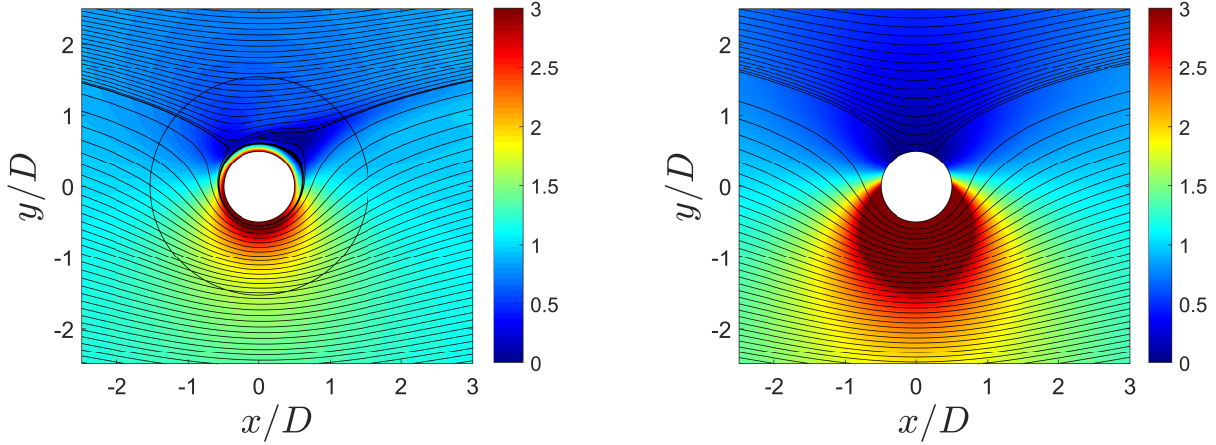


Fig. 3.32: Rotating cylinder - $Re = 200$, viscous case from the present study $\alpha = 3.0$ (left) and inviscid case $\alpha_{\text{pot}}=1.75$ (right): streamlines and colour plot corresponding to the dimensionless norm of the velocity $\|\bar{\mathbf{u}}\|/U_\infty$. Results from the present work are averaged over $\Delta t^* \simeq 109$.

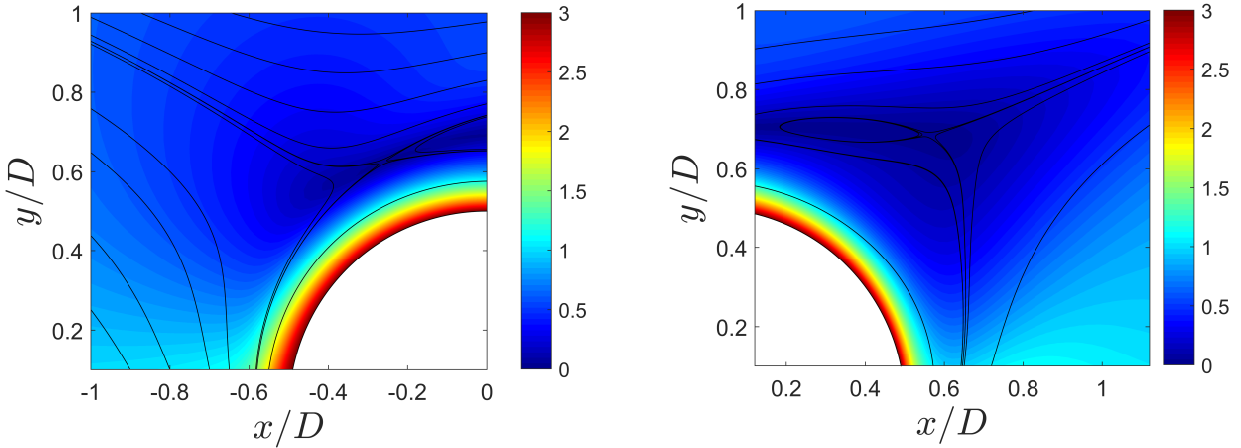


Fig. 3.33: Rotating cylinder - $Re = 200$, $\alpha = 3.0$: zoom on the two saddle points located about $(x/D \simeq -0.259; y/D \simeq 0.625)$ and $(x/D \simeq 0.553; y/D \simeq 0.684)$, respectively. The colour plot corresponds to the dimensionless norm of the velocity $\|\bar{\mathbf{u}}\|/U_\infty$.

Table 3.9: Rotating cylinder - $Re = 200$, $\alpha = 3.0$: approximate position of the saddle points from the present method with the Eulerian mesh held fixed.

	x/D	y/D
Saddle point 1	-0.259	0.625
Saddle point 2	0.553	0.684

The comparison of the lift and drag coefficients reached at the steady-state between the cases with the rotating Eulerian mesh, the fixed Eulerian mesh, and the results from Mittal and Kumar [29] and Padrino and Joseph [30], are provided in Table 3.10. Again, it can be observed that the values for the lift coefficient are in good agreement, while the value for the drag coefficient is lower with present methods than with the results from the references.

In Fig. 3.34, the instantaneous plot of vorticity again shows that the instantaneous vorticity field at steady-state is the same as the time-averaged values over the simulation times where the steady-state seems to be settled. The time-averaged dimensionless horizontal velocity is also depicted. In this Figure, as in the case where $\alpha = 2.5$, the results used are those from the present method with the Eulerian mesh held fixed in the inertial frame, as it is considered to be the present method reference.

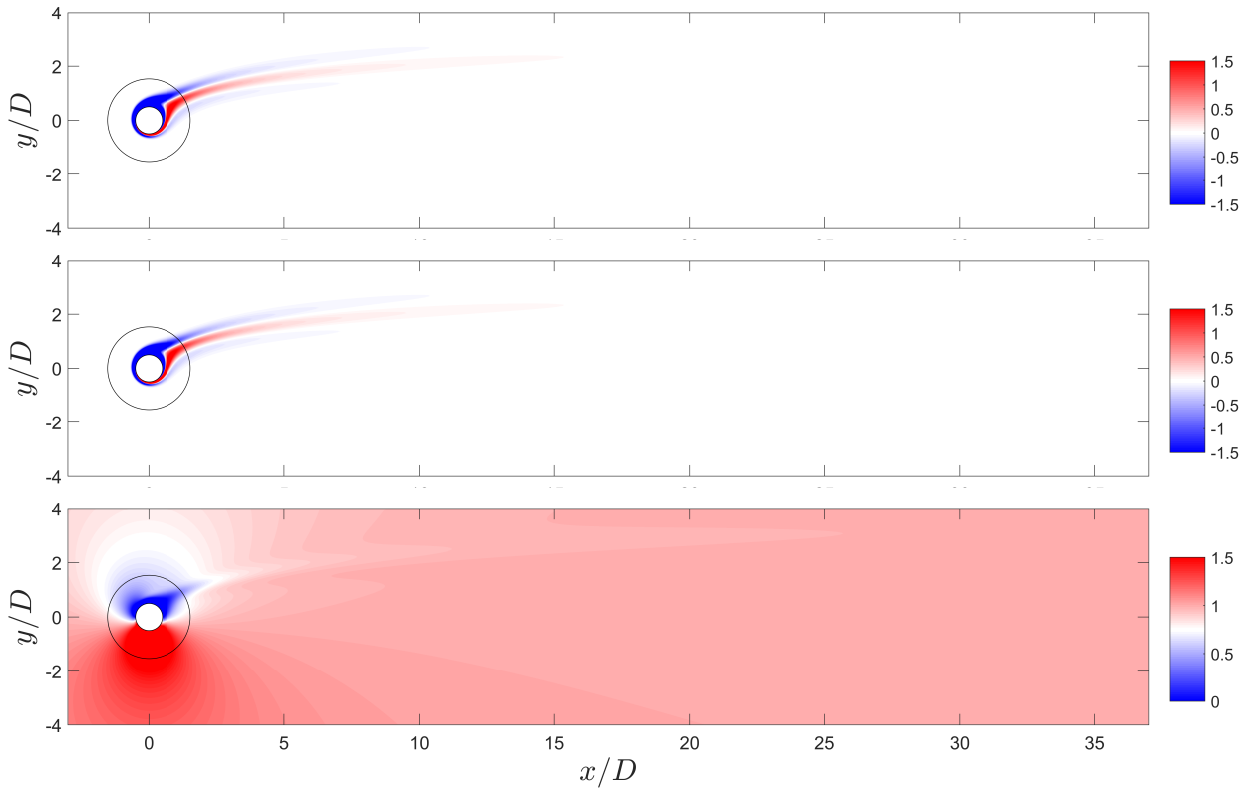


Fig. 3.34: Rotating cylinder - $Re = 200$, $\alpha = 2.5$: Top: $\omega^*(t^* \simeq 137)$. Middle: $\overline{\omega^*}$ averaged over $\Delta t^* \simeq 144$. Bottom: $\overline{u_x}/U_\infty$ averaged over $\Delta t^* \simeq 144$. The limit between the FDoom and VPM results is represented by a thin black line.

Table 3.10: Rotating cylinder - $Re = 200$, $\alpha = 3.0$: averaged force coefficients computed with wall quantities from the NW solver, and values from literature.

	$\overline{C_d}$	$\overline{C_l}$
Present method	-0.084	-10.383
Present method (fixed mesh)	-0.127	-10.375
Mittal and Kumar [29]	0.035	-10.366
Padrino and Joseph [30]	0.012	-10.340

The dimensionless vorticity at the cylinder wall and skin friction are also depicted in Figs. 3.35 and 3.36, where the results from both present methods are compared with those from Mittal and Kumar [29] and those from Padrino and Joseph [30]. Again, the results are in good agreement and the results from the present methods have peaks of maximum (resp. minimum) vorticity that are slightly higher (resp. lower) than those from the two references, as specified in Table 3.11. The separation and attachment points on the cylinder surface for present methods are also provided. Compared to the case $\alpha = 2.5$, the peak of negative wall vorticity is close to 0° while it is slightly shifted in the lower part of the cylinder surface in the case $\alpha = 2.5$. On the other hand, the peak of positive vorticity is more or less at the same location, as are the attachment and separation points. As mentioned in Section 3.2.2 for the case $\alpha = 2.5$, an uncertainty interval on the values given in Table 3.11 should be considered, since values do not come directly from the references. This interval can be considered as the same as previously, i.e. $\pm 2^\circ$ for the positions and ± 0.2 for ω^* .

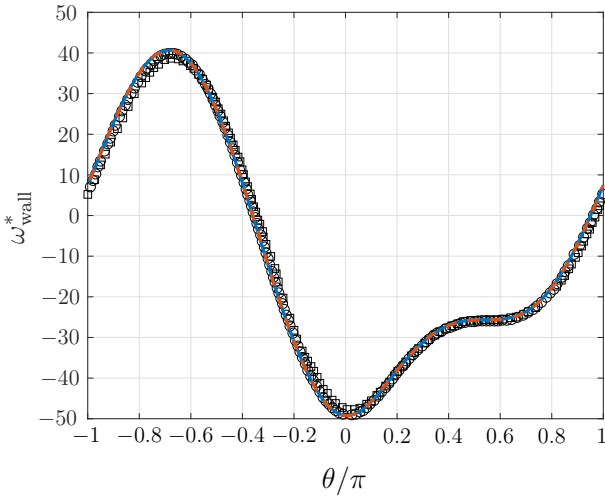


Fig. 3.35: Rotating cylinder - $Re = 200$, $\alpha = 3.0$: dimensionless wall vorticity ω^* at $t^* \simeq 137$. Results from the present method (—), the present method with fixed Eulerian mesh (---), Mittal and Kumar [29] ($\square$), and Padrino and Joseph [30] ($\circ$).

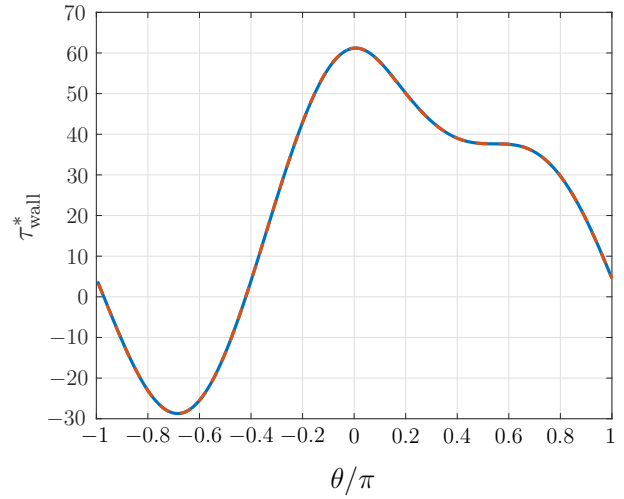


Fig. 3.36: Rotating cylinder - $Re = 200$, $\alpha = 3.0$: dimensionless skin friction τ_{wall}^* at $t^* \simeq 205$. Results from the present method (—), the present method with fixed Eulerian mesh (---). Attachment and separation points are located around $\theta \simeq -75.6^\circ$, and $\theta \simeq -174.8^\circ$, respectively.

Table 3.11: Rotating cylinder - $Re = 200$, $\alpha = 3.0$: position of the attachment (θ_a) and separation (θ_s) points at the cylinder surface, and positions and values of the maximum and minimum values of ω^* .

	θ_a	θ_s	$\theta(\omega_{\max}^*)$	$\omega_{\max}^*$	$\theta(\omega_{\min}^*)$	$\omega_{\min}^*$
Present method	-174.8°	-75.6°	-123.7°	40.7	0.8°	-49.2
Present method (fixed mesh)	-174.8°	-75.7°	-123°	40.7	1.5°	-49.2
Mittal and Kumar [29]	/	/	-120°	38.8	4°	-47.8
Padrino and Joseph [30]	/	/	-120°	39.9	1°	-49.2

3.2.4 Impact of the size of the domain

From the previous simulations of a cylinder rotating at various angular speeds, it has been observed that there is a sudden change in the lift and drag coefficient when the starting vortex reaches the outflow boundary. More precisely, the lift coefficient seems to be influenced in a transient way, while the drag coefficient seems to be influenced for the remaining of the simulation. In order to see whether the position of the boundary has an influence on those sudden changes in the aerodynamic forces, a run for $Re = 200$, $\alpha = 2.5$ has been performed with the same dimensionless positions of the cylinder in the domain, but the cylinder diameter has been decreased by a factor 2, so that the length of the domain is actually twice as long as the initial case.

We are interested in the angle formed by the velocity vector at the inlet of the domain, and how it is influenced by the outflow of the starting vortex. We compare the results from the initial simulation, presented in Section 3.2.2, and the similar simulation, with a domain twice as long. The dimensionless mesh parameters for the two simulations are identical, and they are the same as for the results presented in the previous sections. The only difference for the case of the longer domain is that the VPM grid is twice as long, and that the diameter of the cylinder is twice as small as in the other cases. The dimensionless mesh parameters are thus kept the same as for the shorter domain, whose results have been discussed in a previous section. However, the parameters are still listed in Table 3.12

Table 3.12: Mesh parameters of the NW solver and VPM solver for the two domain sizes

Domain size	w/R	R/h_{VPM}	$R/h_{\text{NW}}^{n,\text{wall}}$	$h_{\text{NW}}^{n,\text{wall}}/h_{\text{NW}}^{\theta,\text{wall}}$
[40D; 20D]	2	12.8	50	1.3
[80D; 20D]	2	12.8	50	1.3

As it can be seen in Fig. 3.37, the direction of the velocity vector at the inflow (and at positions located a few diameters downstream of the cylinder) is strongly influenced by the outflow of the starting vortex, since an abrupt change in the orientation of the velocity vector happens when the starting vortex reaches the boundary. Moreover, it can be seen that this transition is less abrupt when the domain is twice as long, and that the variation of orientation $\Delta\theta$ achieved is less strong when the domain is longer as well. In addition to that, the impact on the aerodynamic forces - which is consistent with the impact on the orientation of the velocity vector - can also be observed in Fig. 3.38. It can be seen that the impact on the lift is less severe than the effect on the drag. The lift undergoes a transient perturbation, but its value is roughly the same before and after the starting vortex goes out of the domain, while the value of the drag coefficient changes in the asymptotic value it reaches for the steady state. Eventually, the transient perturbation for the aerodynamic forces is more severe in the case where the domain is shorter, as the zooms on the times where the starting vortex reaches the outflow boundary show in Fig. 3.38.

When comparing the asymptotic values reached by the aerodynamic coefficients with the results already presented before, it can be concluded that there is no impact on the lift coefficient for a longer domain, but that the steady state value of the drag coefficient is closer to the results from Mittal and Kumar [29] when the domain is twice as long.

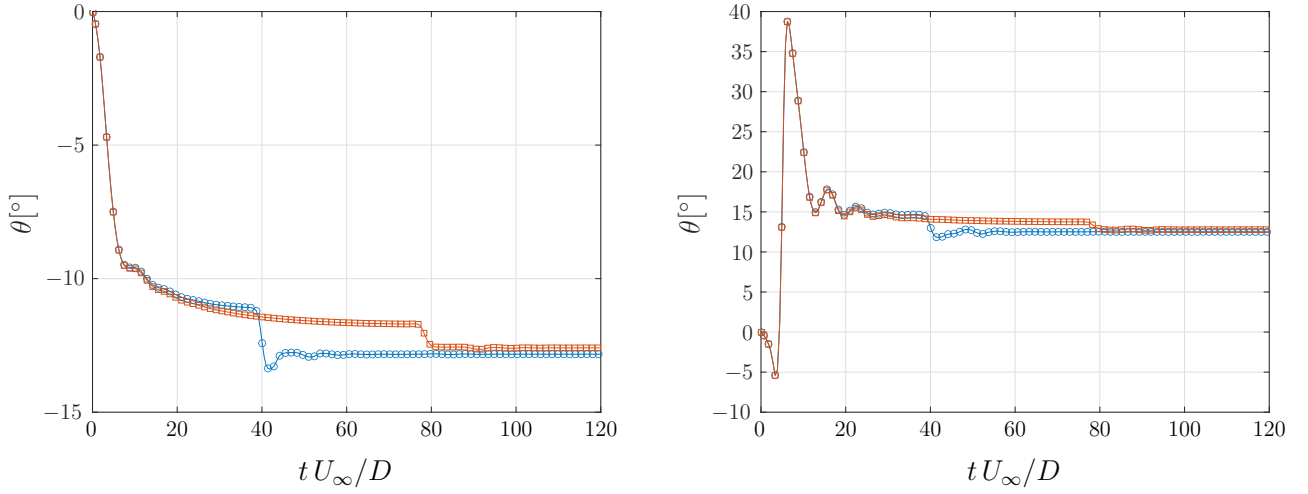


Fig. 3.37: Rotating cylinder - $Re = 200$, $\alpha = 2.5$: time evolution of the angle of the velocity vector at $(x/D \simeq -3; y/D \simeq 0)$ (left), and $(x/D \simeq 3; y/D \simeq 0)$ (right). Results from the present method with fixed Eulerian mesh for the domain length of $40D$ ($\circ$) and $80D$ ($\square$).

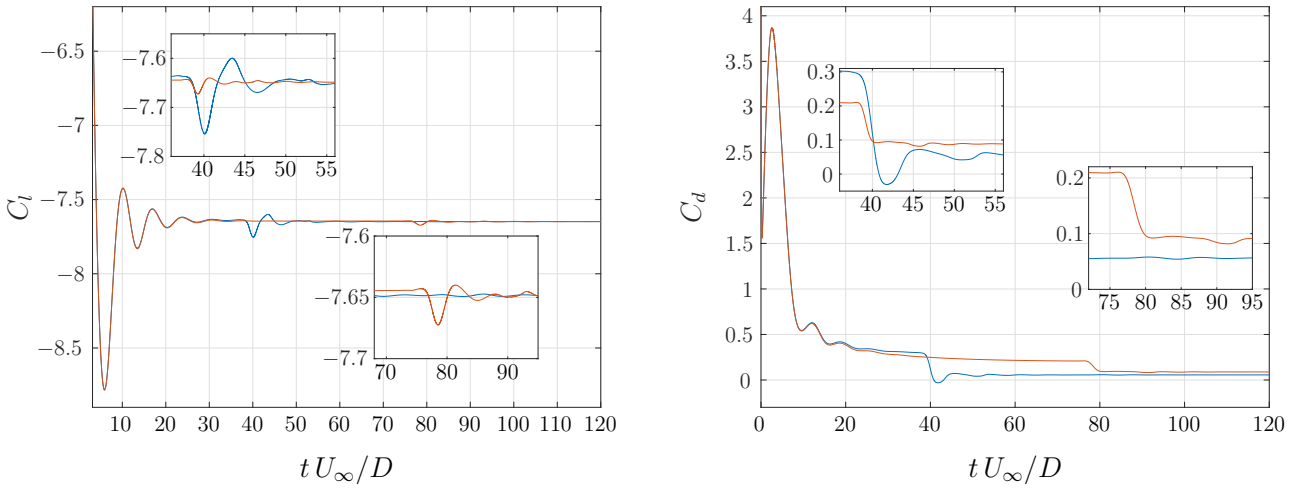


Fig. 3.38: Rotating cylinder - $Re = 200$, $\alpha = 2.5$: comparison of the time evolution of the lift (left) and drag (right) coefficients for two domain sizes. Results from the present method with fixed Eulerian mesh for the domain length of $40D$ (—) and $80D$ (—): the starting vortex reaches the outflow boundary around $t^* \simeq 40$ and $t^* \simeq 80$, respectively.

Table 3.13: Rotating cylinder - $Re = 200$, $\alpha = 2.5$: averaged force coefficients computed with the wall quantities from the NW solver.

	$\overline{C_d}$	$\overline{C_l}$
Present method	0.092	-7.650
Present method (fixed mesh)	0.056	-7.648
Present method with longer domain (fixed mesh)	0.089	-7.648
Mittal and Kumar [29]	0.146	-7.614

Chapter 4

Applications

4.1 Inline oscillations of a cylinder at $Re = 500$

The third movement that can be investigated in order to assess the method for moving bodies is the movement of an oscillatory cylinder in the freestream velocity direction. With the previously presented methodology for moving bodies, and with the prior knowledge of the results of transverse oscillations of a cylinder in a freestream, interesting combinations of forced frequency and amplitude of motion can be found, leading to different wake patterns. In this section, it has been decided to present cases at a higher Reynolds number with respect to the results that can be found in the literature. In fact, as it has been mentioned in Chapter 1, the results for inline oscillations of a cylinder are scarce, and the few references that can be found do not investigate Reynolds numbers such as $Re = 500$. Therefore, in light of the analyses that have been presented in Chapter 3, Sections 3.1.1, 3.1.2, and 3.1.3, it has been decided to choose a period of oscillation corresponding to the theoretical Strouhal number of a shedding cylinder at $Re = 500$, i.e. $St \simeq 0.2$. A posteriori, an additional run for the case of a fixed cylinder has been performed in order to determine the Strouhal number of the flow past the cylinder at the considered Reynolds number. The latter enables a consistency in the analysis of the frequency content of the aerodynamic forces, and in order to assess whether there is *lock-in* or not in the present case investigated, even though, with the chosen oscillation period, it is expected to observe *lock-in* since the theoretical value of the Strouhal number should be close to the Strouhal found with the present method. The Strouhal number is about $St \simeq 0.2237$., which gives a dimensionless forced frequency $F \simeq 0.894 \approx 0.9$.

The dimensionless parameters that characterize the body movement are, as it was the case for the transverse oscillations:

1. The dimensionless amplitude of oscillation: $A^* = A/D$.
2. The dimensionless frequency of the forced oscillation: $F = f_f/f_{St}$, where f_{St} is the Strouhal frequency that corresponds to the flow past the fixed cylinder at the same Reynolds number.

Eq. 4.1 provides the kinematical description of the cylinder movement over time.

$$X_{mesh}(t)/D = A^* \sin\left(\frac{2\pi}{T_f} t\right). \quad (4.1)$$

The VPM and FDoom grids are fine in order to keep $h^+ < 1$ for $t^* \geq 0.005$, and to keep a consistency is cell size between the two component solvers, i.e.: at the outer boundary of the NW solver $\partial\Omega_{NW}^{outer}$, the VPM and FDoom cell sizes are close. The mesh parameters of the NW and VPM solvers are provided in Table 4.1.

Table 4.1: Mesh parameters of the NW solver and VPM solver for the cylinder oscillating inline at $Re = 500$.

w/R	R/h_{VPM}	$R/h_{NW}^{n,wall}$	$h_{NW}^{n,wall}/h_{NW}^{\theta,wall}$
2	25.6	312.5	0.24

In what follows, the cases investigated correspond to the $A^* = 0.5$, and $A^* = 1$. In Section 4.1.1, the flow is studied for $A^* = 0.5$, with a dimensionless oscillation frequency $F \approx 0.9$, this flow is symmetric. Then, Section 4.1.2 investigates the same parameters of motion of the cylinder, but with a perturbation in the flow that is set, so that a different shedding pattern can be triggered and described. Finally, Section 4.1.3 focuses on the symmetric flow for a dimensionless amplitude of oscillation $A^* = 1$, with the same frequency, i.e.: $F \approx 0.9$.

4.1.1 Case 1: $Re = 500$ - $A^* = 0.5$ - $F = 0.9$ without flow perturbation

In this section, the results are presented for the impulsive start of a cylinder oscillating in the freestream direction, for an amplitude of oscillation corresponding to half the cylinder diameter. The flow should remain $2D$, since there is no perturbation and that the oscillation is in the freestream direction. However, due to the accumulation of errors, after approximately 20 periods of oscillations, the errors triggers an asymmetry in the domain, which leads to a different pattern. The results of this section therefore focus only on the flow that is symmetric, i.e. for the times $t/T_f < 20$.

The shedding process is much complex than in the cases investigated in the previous chapters. Indeed, in this case, the backward movement of the cylinder triggers the shedding and makes the vortices that are shed spinning in the near wake. These vortices are interacting with the previously shed vortices. Fig. 4.1 provides the details of the vorticity fields in the near wall and near wake regions over one period of oscillation. Both figures are further discussed below. Then, Fig. 4.2 provides the colour plots of vorticity fields over the domain for a few cycles. It gives a rough idea of the way the far wake evolves over time.

As shown in Fig. 4.1, every cycle, two pairs of opposite sign vortices are shed around 1 diameter above and below the centerline $y/D = 0$. The latter are then convected into the far wake. However, before being convected into the far wake, some complex interactions appear. First, the twin vortices (or primary vortices) are growing as the cylinder moves in the opposite direction from the freestream velocity, with two secondary vortices of opposite sign at the back of the cylinder ($t/T_f \simeq 16.75$). This is actually very similar to the impulsive start of a cylinder at a higher Reynolds number, since the effective Reynolds number is higher than $Re = 500$ due to the velocity of the body. After that, the backward motion of the cylinder triggers the shedding of the twin vortices. However, as the cylinder moves backward, the secondary vortices at the back of the cylinder are growing in the transverse direction, until they reach the external flow, and that the primary vortices detach from the body. From the moment when the primary vortices separate and that the secondary vortices reach the outer flow, the latter are wrapping around the primary vortices ($t/T_f \simeq 17$). The secondary vortices are thus stretched around the primary vortices ($t/T_f \simeq 17.24$), but, while being shed into the near wake, the pairs of vortices are spinning, due to the rolling up of the secondary vortices and the interaction of the two vortices in each respective pair. Once the cylinder has reached its extreme position (from $t/T_f \simeq 17.24$ to $t/T_f \simeq 17.37$), the stretched secondary vortices are then shrunk again; their support of vorticity is thus more compact again. After that, since the pairs of vortices are still spinning, the secondary vortices start to interact with the primary vortices of the previous cycle (i.e.: the vortices downstream), and are again stretched, as they interact with both the primary vortices shed from the same cycle and from the previous cycle. Finally, those secondary vortices have a sigmoid shape ($t/T_f \simeq 17.5$) due to their interaction with the primary vortices of two successive cycles.

The far wake evolution is better depicted in Fig. 4.2. The converged regime is not reached on these instantaneous plots. Nevertheless, the complex interaction between the primary vortices of two successive cycles can clearly be observed; they are rolling up. Moreover, it can be seen that the region in the wake where this rolling up happens is moving downstream over time. In fact, the second primary vortex that rolls up around the previous one is creating a specific wake pattern, i.e.: a Kelvin-Helmholtz-like instability. It can also be observed that the vortex, when

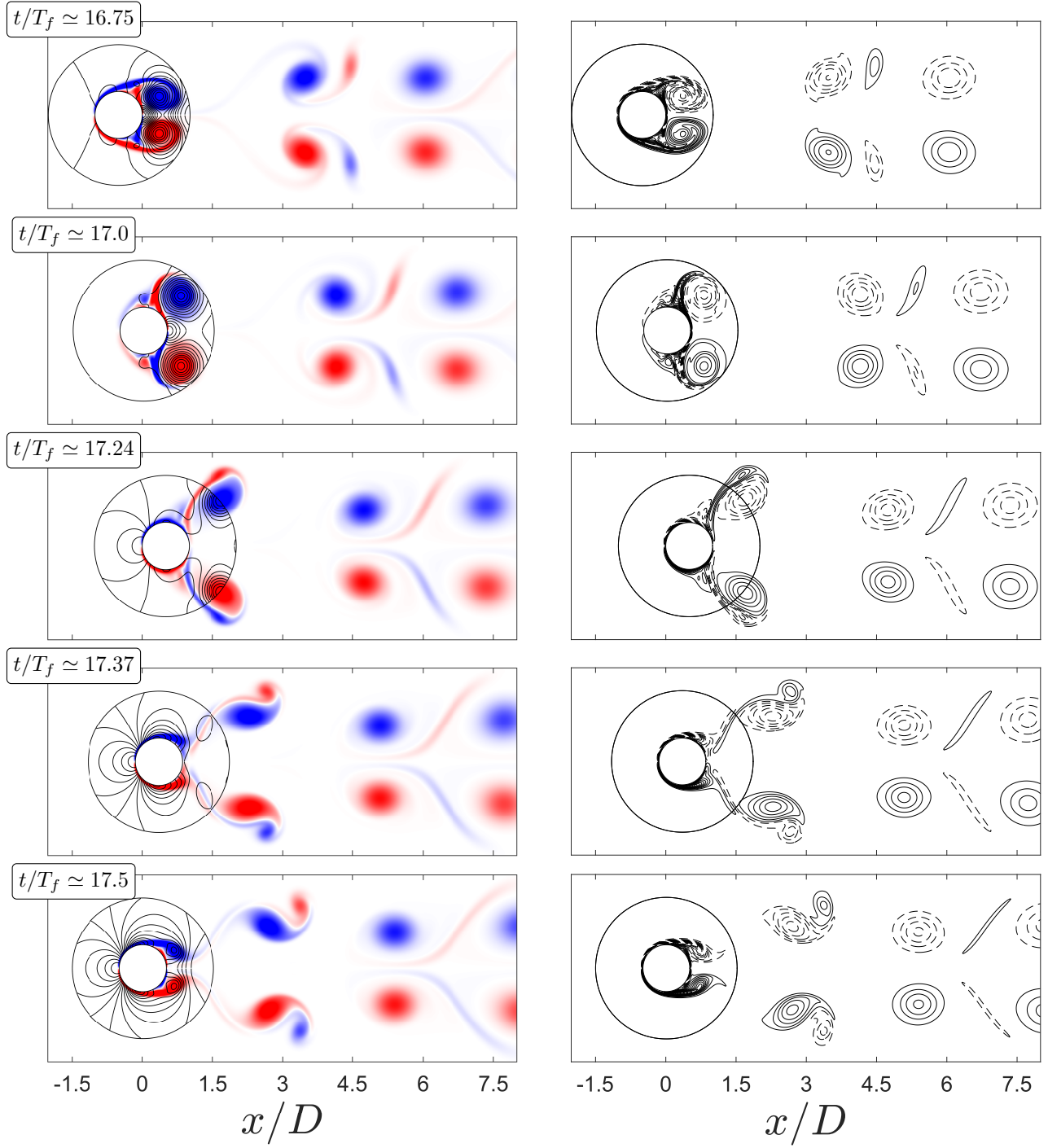


Fig. 4.1: Inline oscillation of a cylinder - $Re = 500$, $A^* = 0.5$, $F \approx 0.9$: colour plot and contours of dimensionless vorticity in FDoom and VPM solvers of the dimensionless vorticity field ω^* , and pressure field P^* (left) at several times. Levels are by steps of 2 (starting from $|\omega^*| = 1$) - levels for P^* are by steps of 1 (starting from P_{max}^*). The colour plot is saturated for $|\omega^*| > 8$. The limit between the FDoom and VPM results is represented by a thin black line.

wrapping up around the previous vortex does not thicken the wake downstream. Indeed, the far wake remains thin for all the times that are shown in Fig. 4.2. Regarding the secondary vortices, which have a very low strength compared to the primary vortices, they are convected faster downstream than the primary vortices. In fact, they are accelerated by the spin of the primary vortices. Therefore, the secondary vortices are moving downstream by following a sinusoidal

track, passing over the primary vortices of the previous cycles.

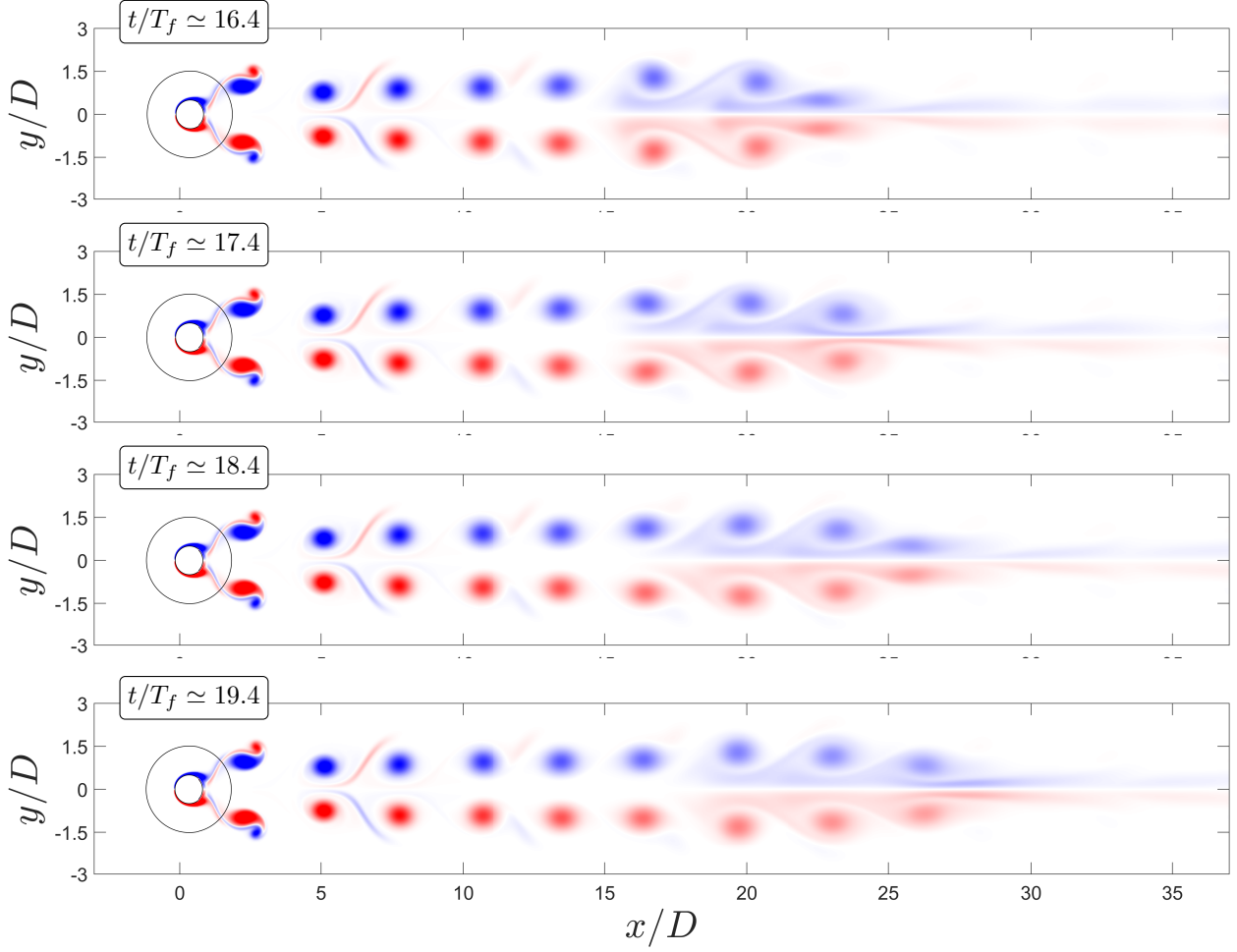


Fig. 4.2: Inline oscillation of a cylinder - $Re = 500$, $A^* = 0.5$, $F \approx 0.9$: dimensionless vorticity field ω^* at several times. The plots are saturated for $\omega^* > 6$. The limit between the FDoom and VPM results is represented by a thin black line.

Finally, the drag curve over 5 periods of oscillation is provided in Fig. 4.3, and the phase portrait of the drag coefficient is provided in Fig. 4.4. The oscillations of the drag coefficient are obviously driven by the oscillation frequency, as the previous description of the vortex shedding pattern already showed. The shape of the oscillation of the drag curve is different from a perfectly sinusoidal pattern. This can especially be seen around $T_f \simeq 0.25$, which corresponds to the moment when the vortices are shed. It can actually be seen, in Fig. 4.1, that the boundary layers are quite thin, right after the separation of the vortices occurred. The phase portrait also confirms that the change in the sinusoidal pattern happens when the cylinder reaches $x/D \simeq 0.5$.

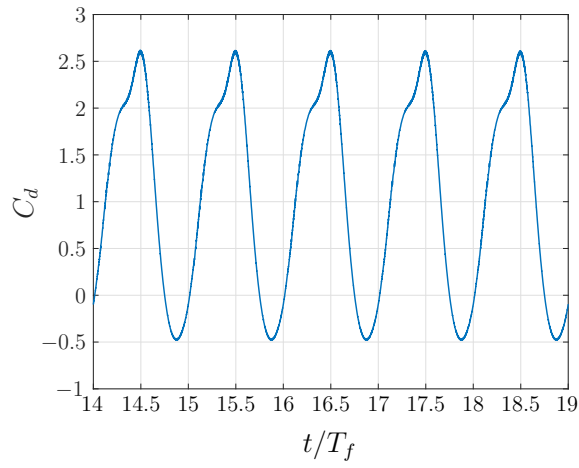


Fig. 4.3: Inline oscillation of a cylinder - $Re = 500$, $A^* = 0.5$, $F \approx 0.9$: drag coefficient over 5 periods of forced oscillation.

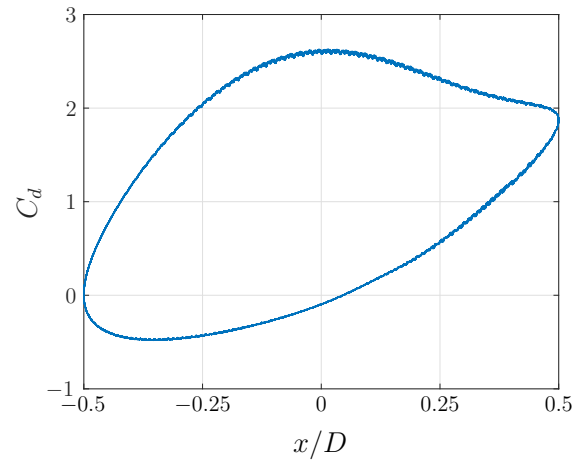


Fig. 4.4: Inline oscillation of a cylinder - $Re = 500$, $A^* = 0.5$, $F \approx 0.9$: phase portrait of the drag coefficient.

4.1.2 Case 2: $Re = 500$ - $A^* = 0.5$ - $F = 0.9$ with flow perturbation

In this section, the same parameters of oscillation are taken as in the previous section, but a perturbation in the flow is set in order to trigger a different shedding pattern. For this purpose, the flow experiences a vertical perturbation over one period of forced oscillation during $t/T_f = 10$ and $t/T_f = 11$. The vertical perturbation on the freestream velocity is described by $V_\infty/U_\infty = 1/2 \sin(2\pi t/T_f)$, where V_∞ is the vertical perturbation in the freestream velocity, and its value is always zero otherwise. The vorticity field over the whole computational domain at a late time of the run is depicted in Fig. 4.5, where complex interactions between the successive vortices that are shed can be observed. The curves of lift and drag coefficients from the moment of the simulation the perturbation in the flow is set are shown in Fig. 4.6.

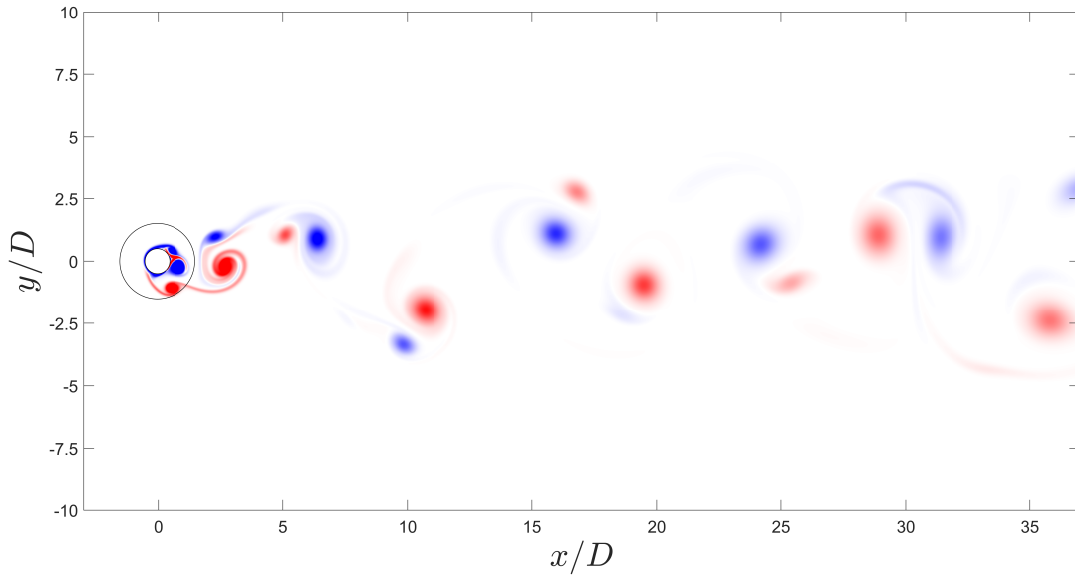


Fig. 4.5: Inline oscillation of a cylinder with flow perturbation - $Re = 500$, $A^* = 0.5$, $F \approx 0.9$: dimensionless vorticity field ω^* over the whole domain at $t^* \simeq 195$. The plot is saturated for $\omega^* > 6$. The limit between the FDoom and VPM results is represented by a thin black line.

The curves of the aerodynamic coefficients are provided in Fig. 4.6. In this figure, the chaotic behaviour of the flow can be observed, with very high amplitude oscillations in the forces. Although, after $t/T_f = 30$, it seems that a more or less periodic pattern establishes, it cannot be asserted since the simulation is too short to confirm this trend. The details of the vortex shedding near the body are depicted in Fig. 4.7, where the complex vortex patterns are shown over approximately one oscillation cycle.

Due to the high frequency oscillation and the chaotic behaviour of the flow, the separation of the vortices over the oscillation cycle is difficult to describe accurately. The various vorticity plots provided in Fig. 4.7 show that there are different scales of vortices that are shed. It seems that a tail of negative¹ circulation extends when the cylinder is moving forward between $t^* \simeq 167.4$ and $t^* \simeq 168.6$. This tail-like layer - emanating from the top of the cylinder - gives rise to a

¹The sign of the latter depends on the cycle; for the next cycle, the sign is positive.

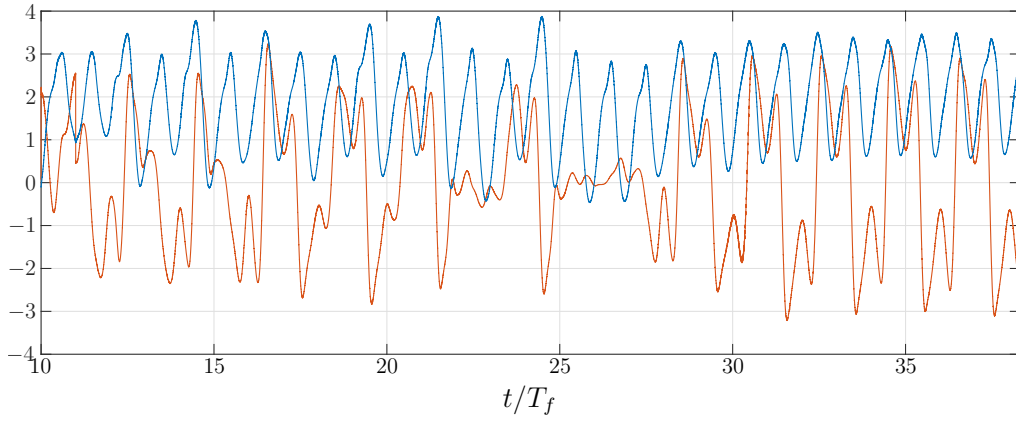


Fig. 4.6: Inline oscillation of a cylinder with flow perturbation - $Re = 500$, $A^* = 0.5$, $F \approx 0.9$: drag coefficient C_d (—) and lift coefficient C_l (—) coefficients over the whole simulation (discarding the times before the perturbation is set).

large scale vortex that is rolling up between $t^* \simeq 168.6$ and $t^* \simeq 170.6$. Moreover, it can be observed that the tail-like layer changes direction while the large scale vortex is rolling up and the cylinder is moving backward. This therefore creates a smaller scale vortex of the same sign which remains weakly linked to the core of the large scale vortex (see $t^* \simeq 169.9$ and $t^* \simeq 170.6$). Eventually, the two vortices separate from the body (a few time steps after $t^* \simeq 170.6$), and, while they are shed into the wake, they interact with other vortices, as shown in the global view in Fig. 4.5. The large scale vortex separates from the small scale vortex shortly after they both separate from the cylinder. This is due to the interaction of the large scale vortex with a counter-rotating vortex emanating from the bottom of the cylinder while the tail is growing (see the vortex of positive circulation about $t^* \simeq 168.6$). Indeed, this vortex of opposite sign also separates from the cylinder about $t^* \simeq 170.6$, and will brake the weak link between the two vortices of same sign. Eventually, it is important to stress that the tail-like structure that develops from $t^* \simeq 167.4$ to $t^* \simeq 168.6$ experiences a change in orientation, which is the result of the common effects of its own interaction with the near wake and the body motion. The first times shown in Fig. 4.7 between $t^* \simeq 165$ and $t^* \simeq 166.9$ also show that the discussion here above is also for the previous cycle, since the same phenomena happens on the lower side of the cylinder, and, thus, with the vortices of reverted sign.

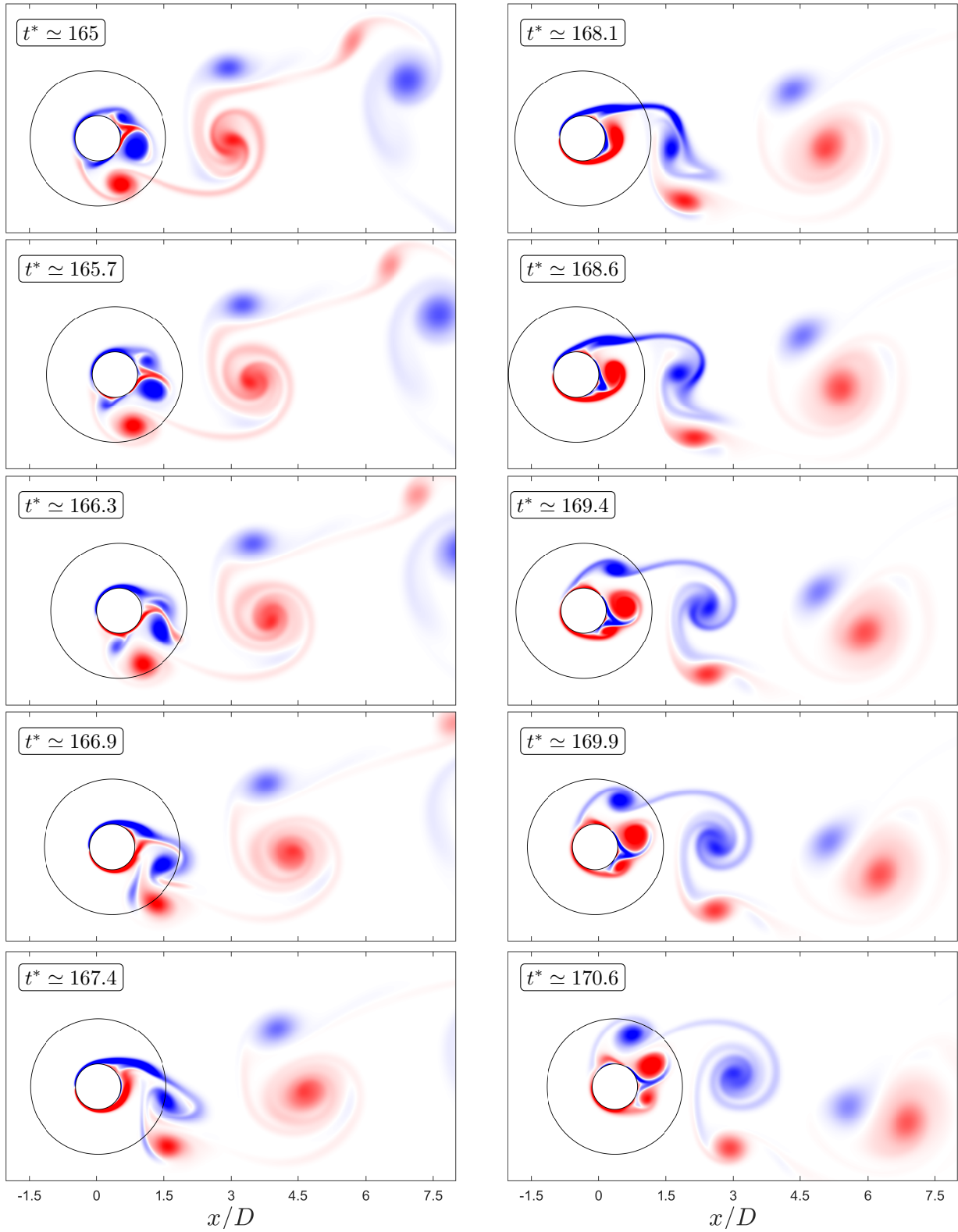


Fig. 4.7: Inline oscillation of a cylinder with flow perturbation - $Re = 500$, $A^* = 0.5$, $F \approx 0.9$: dimensionless vorticity field ω^* at several times between $t/T_f \simeq 33$ and $t/T_f \simeq 34.13$. The plots are saturated for $\omega^* > 10$. The limit between the FDoom and VPM results is represented by a thin black line.

4.1.3 Case 3: $Re = 500$ - $A^* = 1.0$ - $F = 0.9$ without flow perturbation

The last case investigated for the inline oscillations of a cylinder is a high oscillation amplitude case, with $A^* = 1$. The frequency of oscillation remains the same as for the two previous cases, i.e. the dimensionless frequency is $F \approx 0.9$. The vortex shedding pattern is similar to the case investigated in Section 4.1.1; but in the present case, the vortex shedding pattern is much more simple, since the vortices of two successive cycles do not interact, and that the vortices shed above and below the centerline weakly interact with each other.

The long wake is shown in Fig. 4.8, providing a global view of the flow in the statistically converged regime. The vortices which are shed into the wake are spinning while being convected into the far wake. In Fig. 4.11 a zoom on the near wall and near wake enables to better observe the formation of the vortex shedding pattern. As it was the case for $A^* = 0.5$ (see Section 4.1.1), when the cylinder is moving forward (i.e. in the opposite direction from the freestream velocity), a pair of primary vortices are formed, and another pair of secondary vortices is formed as well, which is similar to the flow structure of an impulsively started cylinder. As it can be observed in Fig. 4.8, the pattern of the vortex shedding is a $2P - 2S$ shedding mode, i.e. two pairs ($2P$) and two single ($2S$) vortices are shed every cycle. The details of the formation of this pattern

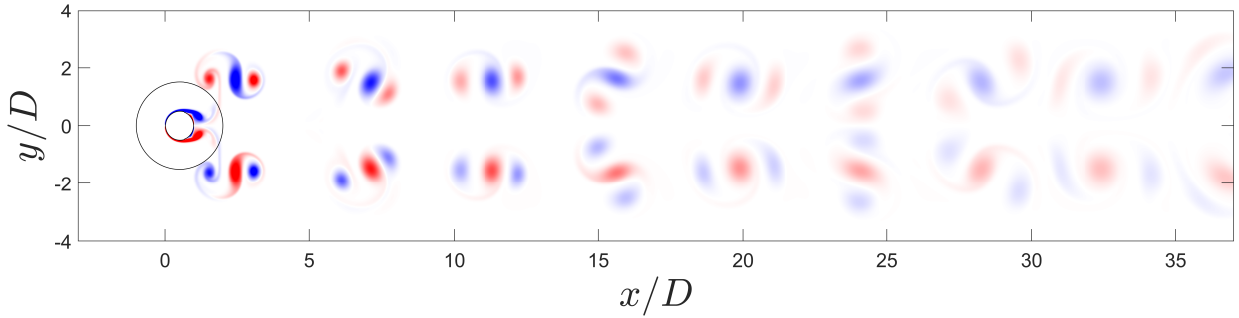


Fig. 4.8: Inline oscillation of a cylinder - $Re = 500$, $A^* = 1.0$, $F \approx 0.9$: dimensionless vorticity field ω^* over the domain at $t^* \simeq 122$. The plot is saturated for $\omega^* > 12$. The limit between the FDoom and VPM results is represented by a thin black line.

are depicted in Fig. 4.11.

The most significant part in the oscillation cycle corresponds to the times of the cycle when the cylinder is moving backward. Indeed, similarly to the case $A^* = 0.5$, the backward movement of the cylinder pushes the secondary vortex, which is strained by both the wall and the primary vortex. Eventually, the secondary vortex penetrates the primary vortex while being stretched in the transverse direction, until it reaches the external flow about $t/T_f \simeq 22.75$. In fact, on the one hand, the cylinder movement enforces the secondary vortex to flatten, and, on the other hand, it also enforces the primary vortex to separate from the cylinder. However, the difference with the case of a lower amplitude of motion ($A^* = 0.5$) is that, due to the higher acceleration of the cylinder, when the secondary vortex is wrapping up around the primary vortex between $t/T_f \simeq 22.87$ and $t/T_f \simeq 22.99$, the high acceleration of the cylinder causes the secondary vortex to split into two parts, as it can be seen between $t/T_f \simeq 23.16$ and $t/T_f \simeq 23.37$, in Fig. 4.11. This phenomenon happens because the size of the secondary vortex is initially strained by the primary vortex behind the cylinder, and that it is abruptly stretched in the transverse direction. Therefore, for each cycle, two pairs of equal sign vortices and two single vortices, whose sign is opposite to their corresponding pair, are shed into the wake. Since the amplitude of oscillation

of the cylinder is higher, the distance between the shed vortices and the centerline is larger, which therefore leads to a very weak interaction between the vortices from each side of the centerline. Indeed, while the vortices are convected, the two pairs of lower strength vortices are diffusing, and a weak interaction can be observed near the centerline in the far wake. It can also be seen that each set of vortices shed over one cycle is convected by following a nearly straight path in the wake, and that the sets of vortices, while being convected, are spinning as well.

The drag curve and its phase portrait are provided in Figs. 4.9 and 4.10, it is similar to the case $A^* = 0.5$, even though the amplitude of the drag is more than twice the amplitude of the previous case, and that the sinusoidal shape depicts the presence of a secondary frequency in the spectral content; but it is the case as well for $A^* = 0.5$. Since the behaviour of the vortex shedding is very similar to the case $A^* = 0.5$, the similar shapes of the drag curve and phase portrait is consistent as well.

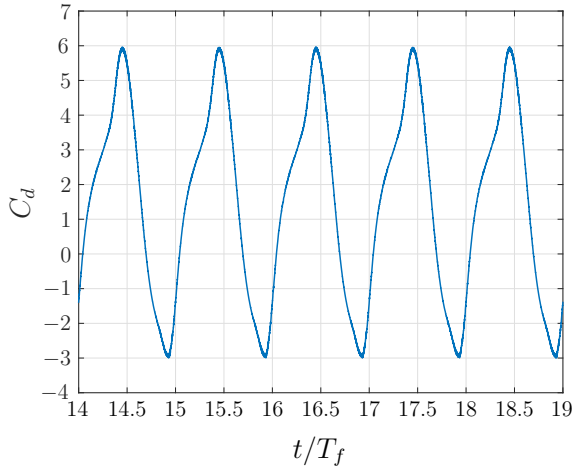


Fig. 4.9: Inline oscillation of a cylinder - $Re = 500$, $A^* = 1.0$, $F \approx 0.9$: drag coefficient over 5 periods of forced oscillation.

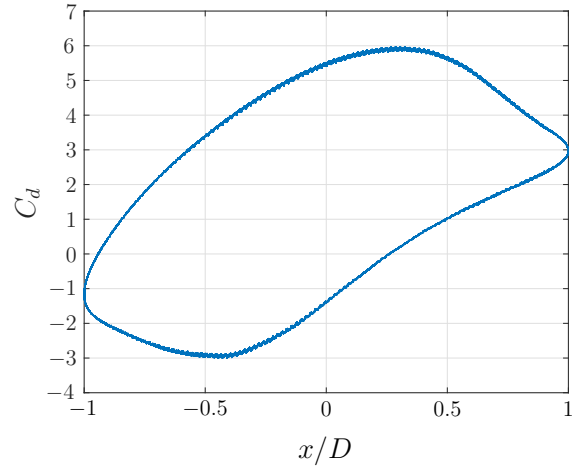


Fig. 4.10: Inline oscillation of a cylinder - $Re = 500$, $A^* = 1.0$, $F \approx 0.9$: phase portrait of the drag coefficient.

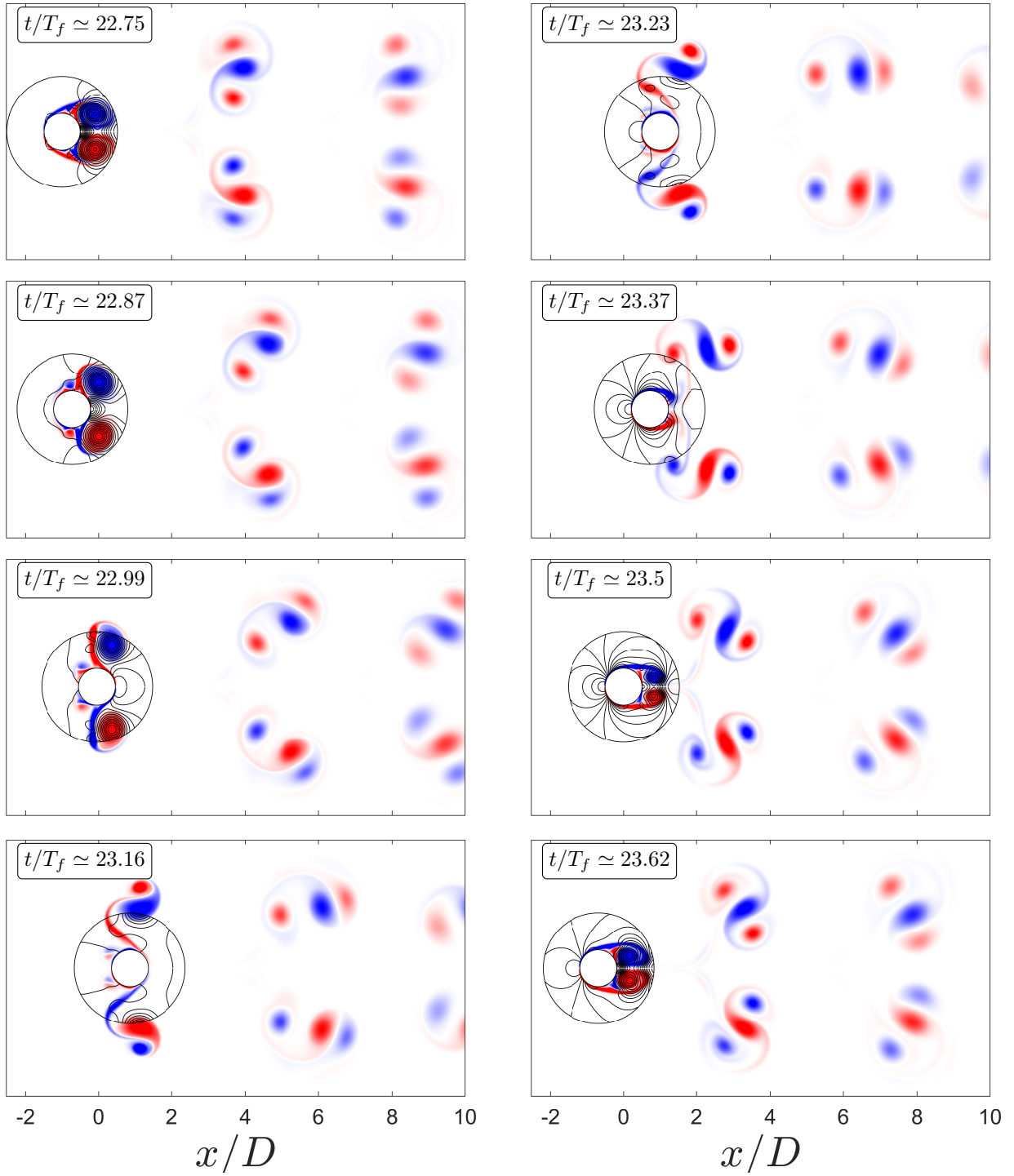


Fig. 4.11: Inline oscillation of a cylinder - $Re = 500$, $A^* = 1.0$, $F \approx 0.9$: dimensionless vorticity field ω^* at several times, and iso-contours of dimensionless pressure P^* . The plots are saturated for $\omega^* > 12$ and the levels for P^* are by step of 1, starting from P_{max}^* . The limit between the FDoom and VPM results is represented by a thin black line.

4.2 Heaving airfoil at $Re = 500$

The methodology of moving bodies can be applied to a heaving airfoil at an angle of attack (AoA) of 10° , in a free stream at $Re = cU_\infty/\nu = 500$, where c is the chord of the airfoil. In this section, the forced frequency that has been used is half the frequency corresponding to the Strouhal number of the shedding of a cylinder at $Re = 100$. In other words, the simulation performed has a forced frequency $f_f = 1/2 f_{St,100}$, where $f_{St,100}$ is the Strouhal frequency that corresponds to the flow past the static cylinder at a Reynolds number $Re = DU_\infty/\nu = 100$. This value was provided in Chapter 3, Section 3.1 (as a reminder, $St_{100} \simeq 0.1608$). The latter has been chosen in order to avoid high frequency of oscillation which would considerably decrease the possible duration of the simulation² due to computational time restrictions. This simulation has been run until $t^* = tU_\infty/c \simeq 52$. The mesh parameters that have been chosen the run ensure that $h^+ < 1$ for $t^* \geq 0.001$; they are listed in Table 4.2. It is important to stress that the final mesh obtained for the case of the heaving airfoil is the result of a Joukowski transformation of the cylinder mesh, with a regularization that allows a finite thickness at the trailing edge. In other words, the values in Table 4.2 correspond to the settings of the mesh before the conformal transformation.

Table 4.2: Mesh parameters of the NW solver and VPM solver for the heaving airfoil: parameters are values on the cylinder mesh before the regularized conformal transform.

w/R	R/h_{VPM}	$R/h_{\text{NW}}^{n,\text{wall}}$	$h_{\text{NW}}^{n,\text{wall}}/h_{\text{NW}}^{\theta,\text{wall}}$
2	25.6	312.5	0.24

The dimensionless parameters of the body movement are specified and defined, similarly to the previous cases, as follows:

1. The dimensionless amplitude of oscillation: $A^* = A/c = 0.268$. This value is actually obtained because the amplitude was initially defined based on the diameter of the cylinder before the conformal transformation, which increases the characteristic length, since the chord is larger than the cylinder diameter.
2. The dimensionless frequency of the forced oscillation: $F = f_f/f_{St,100} = 0.5$.

Similarly to the case of the cylinder, Eq. 4.2 provides the kinematical description of the airfoil movement over time.

$$X_{\text{mesh}}(t)/c = A^* \sin\left(\frac{2\pi}{T_f} t\right). \quad (4.2)$$

Fig. 4.12 depicts a global view on the wake of the regularized airfoil that is heaving. The regime is nearly statistically converged³. The instant shown in Fig 4.12 corresponds to a moment when the airfoil reaches the bottom of its trajectory. This explains why, on the suction side of the airfoil, the flow separates and the boundary layers does not remain attached over a long portion of the suction side. Besides, the wake is deflected upward with respect to the airfoil,

²A more consistent frequency of oscillation would be linked to the frequency corresponding to the Strouhal of the static airfoil at $Re = 500$.

³A longer run is actually necessary in order to confirm that the statistically converged regime is reached.



Fig. 4.12: Heaving airfoil - $Re = 500$, $A^* = 0.268$, $F = 0.5$: dimensionless vorticity field ω^* at $t^* \simeq 51.76$, corresponding to $t/T_f \simeq 7.75$. The plot is saturated for $\omega^* > 6$. The limit between the FDoom and VPM results is represented by a thin black line.

due to the downward trajectory of the previous few times. The separation of the flow when the airfoil reaches $y/c \simeq -0.5$ gives rise to a large scale vortex of negative circulation, that is shed when the airfoil moves upward again. The details of the vortex dynamics during one cycle are described here below.

In Fig. 4.13, a closer look on the airfoil and its wake is provided for five times of an oscillation cycle. The iso-contours are also provided in parallel, in order to illustrate the accurate continuity across the boundary $\partial\Omega_{NW}^{outer}$, and thus, the consistency between the the NW and VPM solvers. About $t/T_f \simeq 6.75$, the airfoil is stalling, which results in the formation of a large lobe of vorticity of negative circulation. Even though the flow is separating, that lobe still remains linked to the body. This lobe of vorticity is then shed into the wake while the airfoil is moving upwards, as it can be seen about $t/T_f \simeq 7$; indeed, the lobe of vorticity is stretched by the vortex of positive strength emanating from the pressure side. The positive vortex starts interacting with the large lobe of negative strength due to the upward motion of the airfoil about $t/T_f \simeq 7$. Finally, the lobe of negative vorticity splits and two resultant vortices are shed into the wake about $t/T_f \simeq 7.25$.

At $t/T_f \simeq 7.5$, the airfoil is already moving downwards and the formation of a new lobe of negative vorticity can also be seen, due to the downward motion of the body. About that time, another feature of the vortex pattern can be observed. In fact, since the airfoil is close to $y/c = 0$ its velocity low, which therefore gives rise to the formation of a thick wake. Then, while the airfoil is moving downwards, this thick wake becomes unstable due to the motion of the airfoil and the influence of the lobe of negative vorticity.

In Fig. 4.14, the aerodynamic coefficients are shown over the whole simulation, while, in Fig. 4.15, the phase portraits of the latter show that the vortex shedding is periodic, since the curves are closed. The phase portraits are drawn with the values of the five last oscillations, starting from the final time of the simulation, and therefore discarding the first three cycles, in order to neglect the transient starting, as the wake has not reached the outflow boundary yet.

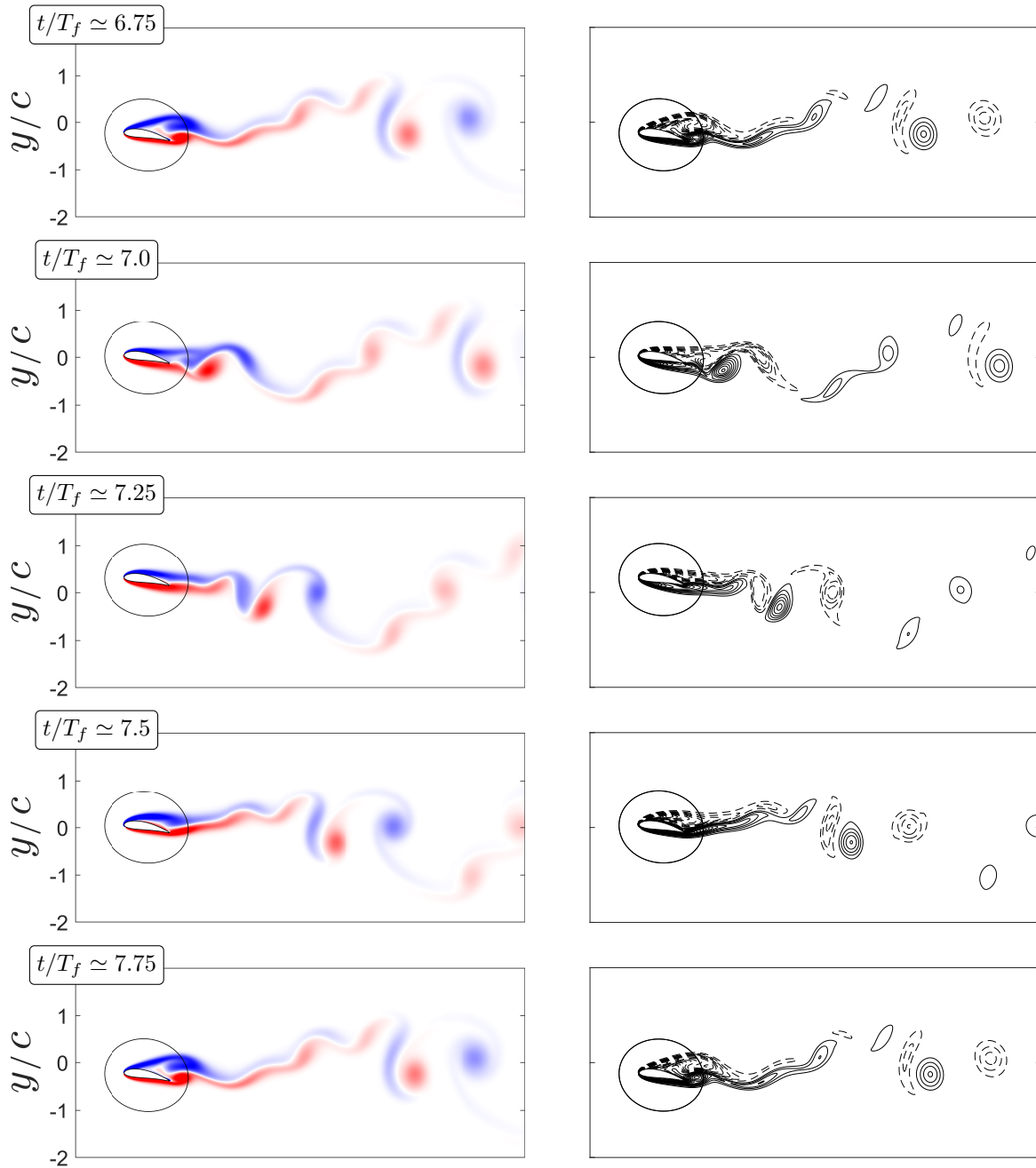


Fig. 4.13: Heaving airfoil - $Re = 500$, $A^* = 0.268$, $F = 0.5$: dimensionless vorticity field ω^* at several times. The colour plots are saturated for $\omega^* > 8$. The iso-contours are by step of 1. The limit between the FDoom and VPM results is represented by a thin black line.

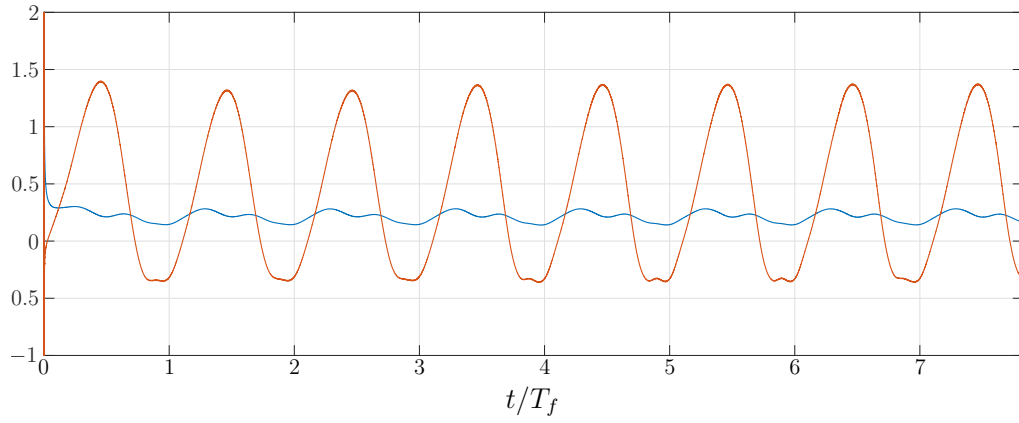


Fig. 4.14: Heaving airfoil - $Re = 500$, $A^* = 0.268$, $F = 0.5$: drag coefficient C_d (—) and lift coefficient C_l (—) coefficients over the whole simulation.

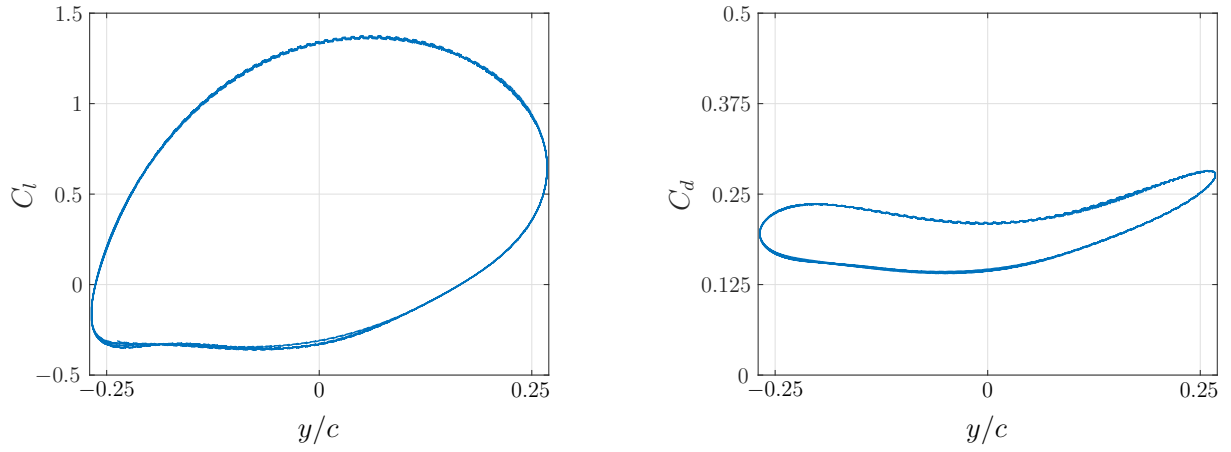


Fig. 4.15: Heaving airfoil - $Re = 500$, $A^* = 0.268$, $F = 0.5$: phase portrait of the lift and drag coefficients.

Chapter 5

Conclusions

5.1 Discussion

The main objective of this work was to extend the coupling methodology developed by Billuart et al. [1] for the 2D simulations of external flows to moving bodies. The new methodology was presented in Chapter 2, and a verification was therefore required. The latter was presented in Chapter 3. The test cases for the verification have been chosen from a literature review presented in Chapter 1. Chapter 1 also discussed the principles of the existing numerical methods used in external flow aerodynamics, and introduced the basic principles of the new coupling methodology of Billuart et al. [1]. Finally, interesting applications could be presented in Chapter 4, where the cases investigated have - namely - been chosen based on the literature review and the previous results.

The simulation of translating motions of rigid bodies have been successfully achieved in this work, even though some improvements could still be examined for decreasing the pressure oscillations in the near-wall solver. On the other hand, when considering the rotational motions, the present coupling methodology reveals some limitations, as it has been discussed in Chapter 3, for the case of a rotating cylinder.

Furthermore, this work also enabled to present again test cases that were investigated approximately 50 years ago and which do not rise much interest in the literature nowadays, although there are many studies in vortex dynamics that could still be investigated. For example, the shedding patterns of a cylinder oscillating in the streamwise direction could be more deeply studied.

Finally, the ability of the methodology developed in this work to be applied to other bodies (as an airfoil) is promising for more specific studies, as the application of the heaving airfoil showed results that give confidence in the validity of the methodology.

5.2 Perspectives

Still, the most important perspective of the present work is to investigate the origin of the spurious interpolation of the boundary condition in the case of a rotating body, as it was discussed in Chapter 3, Section 3.2.

Then, many applications can rely on the work that has been achieved in this Master thesis. A straightforward application that could be implemented is a *Fluid Structure Interaction* case. Either implementing elastically mounted bodies, as it was introduced in Chapter 1, or another example of application could be a falling body. Many other applications related to FSI could be investigated using this work as a basis. Only a slight adaptation of the methodology presented should be implemented, since the body movement is not prescribed in such applications.

Another interesting direction, which would not require many developments, is the case of multiple bodies whose wakes would interact. As an example, three airfoils rotating could be of great interest in order to study the wake interaction in vertical axis wind turbines. After that, the methodology could even be further adapted to a case where the Eulerian subdomains could overlap when dealing with several bodies. Eventually, collisions of two bodies could be implemented, as it was already investigated by Bampalas and Graham [40], and Gazzola, Mimeau, Tchieu and Koumoutsakos [41].

Finally, deforming bodies could be investigated. The body, with soft boundaries, would not be rigid anymore and there would be a non-zero divergence inside the body, which would require further developments in the present methodology.

Appendix A

Potential flow past a rotating cylinder

The theoretical developments for the potential flow past a rotating cylinder are recalled in the present Appendix. The flow past a rotating cylinder with circulation can be computed with the superposition of a dipole, a uniform flow, and a point vortex whose total circulation is the same as the circulation corresponding to a patch of uniform vorticity $\omega = \omega_0$ enclosed in the closed contour $\mathcal{C}$ drawn by the cylinder, such that, for any radial position r (in polar coordinates), we have that:

$$\omega = \begin{cases} 0 & \text{if } r > R, \\ \omega_0 & \text{otherwise.} \end{cases} \quad (\text{A.1})$$

The circulation of the patch of uniform vorticity can be computed using:

$$\begin{aligned} \Gamma &= \oint_{\mathcal{C}} \mathbf{v} \cdot d\mathbf{l} = \int_0^{2\pi} u_\theta(R) R d\theta = u_\theta(R) 2\pi R \\ \Leftrightarrow \Gamma &= \int_{\mathcal{S}} \boldsymbol{\omega} dS = \int_0^R \int_0^{2\pi} \omega_0 r d\theta dr = \omega_0 \pi R^2, \end{aligned} \quad (\text{A.2})$$

where u_θ is the azimuthal velocity component, R is the radius of the cylinder, and $\mathcal{S}$ is the surface of the vorticity patch, i.e.: the surface of the cylinder. By recalling that, for a rigid body rotation, the link between the rotational speed and the rate of rotation of a fluid element is given by $\Omega = 1/2 \omega_0$. It can finally be obtained that the circulation of the point vortex corresponding to a rigid cylinder rotation is given by: $\Gamma = 2\pi R^2 \Omega$.

Eventually, the complex potential of the complex number z , and the corresponding radial and azimuthal velocity components of the potential flow past a rotating cylinder at a rotational speed Ω are respectively given by:

$$f(z) = U_\infty \left(z + \frac{R^2}{z} \right) + \frac{\Gamma}{2\pi i} \log \left(\frac{z}{R} \right), \quad (\text{A.3})$$

$$u_r(r, \theta) = U_\infty \left(1 - \frac{R^2}{r^2} \right) \cos(\theta), \quad (\text{A.4})$$

$$u_\theta(r, \theta) = -U_\infty \left(1 + \frac{R^2}{r^2} \right) \sin(\theta) + \frac{\Gamma}{2\pi r}. \quad (\text{A.5})$$

Bibliography

- [1] P. Billuart et al. A weak coupling between a near-wall Eulerian solver and a Vortex Particle-Mesh method for the efficient simulation of 2D external flows. Revised version submitted to *Journal of Computational Physics*, in July 2022.
- [2] Y. Marichal. *An immersed interface vortex particle-mesh method*. PhD thesis, Université Catholique de Louvain, 2014.
- [3] G. S. Winckelmans. Vortex Methods. In *Encyclopedia of Computational Mechanics*, volume 3, pages 129–153. John Wiley & Sons, Ltd, 2nd edition, 2004. doi: 10.1002/0470091355.ecm055.
- [4] G.-H. Cottet, P. D. Koumoutsakos, et al. *Vortex Methods: Theory and Practice*, volume 8. Cambridge University Press, 2000.
- [5] G. Daeninck. *Developments in hybrid approaches. Vortex method with known separation location. Vortex method with near-wall Eulerian solver. RANS-LES coupling*. PhD thesis, Université catholique de Louvain, 2006.
- [6] G.-H. Cottet. Particle-grid domain decomposition methods for the Navier-Stokes equations in exterior domains. *Lectures in Applied Mathematics*, 28:103–117, 1991.
- [7] M. L. Ould-Salihi, G.-H. Cottet, and M. El Hamraoui. Blending finite-difference and vortex methods for incompressible flow computations. *SIAM Journal on Scientific Computing*, 22(5):1655–1674, 2001.
- [8] J.-L. Guermond and H.Z. Lu. A domain decomposition method for simulating advection dominated, external incompressible viscous flows. *Computers & Fluids*, 29(5):525–546, 2000. doi: 10.1016/S0045-7930(99)00017-1.
- [9] M. Stock, A. Gharakhani, and C. Stone. *Modeling Rotor Wakes with a Hybrid OVERFLOW-Vortex Method on a GPU Cluster*. American Institute of Aeronautics and Astronautics, 2015/08/11 2010. doi: 10.2514/6.2010-4553.
- [10] J. Donea, S. Giuliani, and J.P. Halleux. An Arbitrary Lagrangian-Eulerian finite element method for transient dynamic fluid-structure interaction. *Computer Methods in Applied Mechanics and Engineering*, 33:689 – 723, 1982.
- [11] W. F. Noh. Cel: A time-dependent, two-space-dimensional, coupled Eulerian-Lagrange code. In B. Alder et al., editors, *Methods in Computational Physics*. Academic Press, 1964. doi: 10.2172/4621975.

- [12] J. G. Trulio. Theory and structure of the AFTON codes. Technical report no. afwl-tr-66-19, Air Force Weapons Laboratory, Kirtland Air Force Base, 1966.
- [13] C.W. Hirt, A.A. Amsden, and J.L. Cook. An arbitrary Lagrangian-Eulerian computing method for all flow speeds. *Journal of Computational Physics*, 14(3):227–253, 1974. doi: 10.1016/0021-9991(74)90051-5.
- [14] T. J. R. Hughes, W. K. Liu, and T. Zimmermann. Lagrangian-Eulerian finite element formulation for incompressible viscous flows. *Computer Methods in Applied Mechanics and Engineering*, 29(3):329–349, 1981. doi: 10.1016/0045-7825(81)90049-9.
- [15] J. Donea, A. Huerta, J.-Ph. Ponthot, and A. Rodríguez-Ferran. Arbitrary Lagrangian–Eulerian Methods. In *Encyclopedia of Computational Mechanics*, volume 1, pages 413–437. John Wiley & Sons, Ltd, 2nd edition, 2004. doi: 10.1002/0470091355.ecm009.
- [16] R. E. D. Bishop and A. Y. Hassan. The lift and drag forces on a circular cylinder oscillating in a flowing fluid. *Proceedings of the Royal Society of London. Series A. Mathematical and Physical Sciences*, 277(1368):51–75, 1964. doi: 10.1098/rspa.1964.0005.
- [17] G. H. Koopmann. The vortex wakes of vibrating cylinders at low reynolds numbers. *Journal of Fluid Mechanics*, 28(3):501–512, 1967. doi: 10.1017/S0022112067002253.
- [18] C.H.K. Williamson and A. Roshko. Vortex formation in the wake of an oscillating cylinder. *Journal of Fluids and Structures*, 2(4):355–381, 1988. doi: 10.1016/S0889-9746(88)90058-8.
- [19] J.R. Meneghini and P.W. Bearman. Numerical simulation of high amplitude oscillatory flow about a circular cylinder. *Journal of Fluids and Structures*, 9(4):435–455, 1995. doi: 10.1006/jfls.1995.1025.
- [20] H. M. Blackburn and R. D. Henderson. A study of two-dimensional flow past an oscillating cylinder. *Journal of Fluid Mechanics*, 385:255–286, 1999. doi: 10.1017/S0022112099004309.
- [21] S. Mittal and V. Kumar. Finite element study of vortex-induced cross-flow and in-line oscillations of a circular cylinder at low reynolds numbers. *International Journal for Numerical Methods in Fluids*, 31(7):1087–1120, 1999. doi: 10.1002/(SICI)1097-0363(19991215)31:7<1087::AID-FLD911>3.0.CO;2-C.
- [22] P. Anagnostopoulos. Numerical study of the flow past a cylinder excited transversely to the incident stream. part 1: Lock-in zone, hydrodynamic forces and wake geometry. *Journal of Fluids and Structures*, 14(6):819–851, 2000. doi: 10.1006/jfls.2000.0302.
- [23] J. S. Leontini, B. E. Stewart, M. C. Thompson, and K. Hourigan. Wake state and energy transitions of an oscillating cylinder at low Reynolds number. *Physics of Fluids*, 18(6):067101, 2006. doi: 10.1063/1.2204632.
- [24] M.R.H. Nobari and H. Naderan. A numerical study of flow past a cylinder with cross flow and inline oscillation. *Computers & Fluids*, 35(4):393–415, 2006. doi: 10.1016/j.compfluid.2005.02.004.
- [25] A. Placzek, J.-F. Sigrist, and A. Hamdouni. Numerical simulation of an oscillating cylinder in a cross-flow at low Reynolds number: Forced and free oscillations. *Computers & Fluids*, 38(1):80–100, 2009. doi: 10.1016/j.compfluid.2008.01.007.

- [26] S. Kumar, Navrose, and S. Mittal. Lock-in in forced vibration of a circular cylinder. *Physics of Fluids*, 28(11):113605, 2016. doi: 10.1063/1.4967729.
- [27] S.E. Hurlbut, M.L. Spaulding, and F.M. White. Numerical Solution for Laminar Two Dimensional Flow About a Cylinder Oscillating in a Uniform Stream. *Journal of Fluids Engineering*, 104(2):214–220, 1982. doi: 10.1115/1.3241810.
- [28] J. S. Leontini, D. Lo Jacono, and M. C. Thompson. A numerical study of an inline oscillating cylinder in a free stream. *Journal of Fluid Mechanics*, 688:551–568, 2011. doi: 10.1017/jfm.2011.403.
- [29] S. Mittal and B. Kumar. Flow past a rotating cylinder. *Journal of Fluid Mechanics*, 476:303–334, 2003.
- [30] J. C. Padrino and D. D. Joseph. Numerical study of the steady-state uniform flow past a rotating cylinder. *Journal of Fluid Mechanics*, 557:191–223, 2006. doi: 10.1017/S0022112006009682.
- [31] C.H.K. Williamson and R. Govardhan. Vortex-induced vibrations. *Annual Review of Fluid Mechanics*, 36(1):413–455, 2004. doi: 10.1146/annurev.fluid.36.050802.122128.
- [32] S.P. Singh and S. Mittal. Vortex-induced oscillations at low Reynolds numbers: Hysteresis and vortex-shedding modes. *Journal of Fluids and Structures*, 20(8):1085–1104, 2005. doi: 10.1016/j.jfluidstructs.2005.05.011.
- [33] T. K. PRASANTH and S. MITTAL. Vortex-induced vibrations of a circular cylinder at low reynolds numbers. *Journal of Fluid Mechanics*, 594:463–491, 2008. doi: 10.1017/S0022112007009202.
- [34] A. J. Chorin. Numerical solution of the Navier-Stokes equations. *Mathematics of Computation*, 22(104):745–762, 1968.
- [35] J. Kim and P. Moin. Application of a fractional-step method to incompressible Navier-Stokes equations. *Journal of Computational Physics*, 59(2):308–323, 1985. doi: 10.1016/0021-9991(85)90148-2.
- [36] M. J. Lee, B. D. Oh, and Y. B. Kim. Canonical Fractional-Step Methods and Consistent Boundary Conditions for the Incompressible Navier–Stokes Equations. *Journal of Computational Physics*, 168(1):73–100, 2001. doi: 10.1006/jcph.2000.6682.
- [37] F. H. Harlow and J. E. Welch. Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free surface. *The Physics of Fluids*, 8(12):2182–2189, 1965. doi: 10.1063/1.1761178.
- [38] D.-G. Caprace, T. Gillis, and P. Chatelain. FLUPS: A Fourier-Based Library of Unbounded Poisson Solvers. *SIAM Journal on Scientific Computing*, 43(1):C31–C60, 2021. doi: 10.1137/19M1303848.
- [39] J.J. Monaghan. Extrapolating B splines for interpolation. *Journal of Computational Physics*, 60(2):253 – 262, 1985. doi: 10.1016/0021-9991(85)90006-3.
- [40] N. BAMPALAS and J. M. R. GRAHAM. Flow-induced forces arising during the im-

- pact of two circular cylinders. *Journal of Fluid Mechanics*, 616:205–234, 2008. doi: 10.1017/S0022112008003856.
- [41] M. Gazzola, C. Mimeau, A. A. Tchieu, and P. Koumoutsakos. Flow mediated interactions between two cylinders at finite re numbers. *Physics of Fluids*, 24(4):043103, 2012. doi: 10.1063/1.4704195.

UNIVERSITÉ CATHOLIQUE DE LOUVAIN
École polytechnique de Louvain

Rue Archimède, 1 bte L6.11.01, 1348 Louvain-la-Neuve, Belgique | www.uclouvain.be/epl