

École polytechnique de Louvain

Penalization in a fourth order code for incompressible flows to handle fixed and moving immersed bodies

Author: **Yassine BERRADA**

Supervisors: **Grégoire WINCKELMANS, Matthieu DUPONCHEEL**

Readers: **Philippe CHATELAIN, Vincent LEGAT**

Academic year 2018–2019

Master [120] in Electro-mechanical Engineering

Acknowledgements

Je tiens à remercier toutes les personnes qui ont contribué de près ou de loin à l'élaboration de ce mémoire. Dans un premier temps, je voudrais exprimer toute ma reconnaissance et ma gratitude à l'assistant qui m'a encadré, Baptiste Hardy. Sans lui, ce mémoire ne serait pas ce qu'il est aujourd'hui. Ses lectures, ses conseils ainsi que ses permanences nombreuses m'ont été d'une grande aide dans la rédaction de ce mémoire. Un énorme merci pour sa patience, sa disponibilité ainsi que ses précieux conseils durant toute l'année académique.

Je remercie également mes promoteurs de mémoire, le Professeur Grégoire Winckelmans et le Professeur Matthieu Duponcheel, pour leurs disponibilités, leurs différentes suggestions et remarques pertinentes qui ont contribué à alimenter ma réflexion.

J'aimerais aussi remercier toutes les personnes qui étaient présentes aux différents "tours-de-table" des mercredis après-midi.

Mes remerciements s'adressent aussi au Professeur Philippe Chatelain et au Professeur Vincent Legat pour la lecture de ce mémoire. Merci d'avance. Je n'oublie pas tous les Professeurs qui m'ont fourni les outils nécessaires à la réussite de mes études universitaires. Merci à tous les intervenants qui ont accepté de me rencontrer et de répondre à mes questions afin de fructifier mon travail.

Sans oublier mes très chers parents ainsi que mes frères, qui par leurs encouragements et leur bienveillance, m'ont été d'un immense soutien.

Cela a été un travail de longue haleine et je n'y serais jamais arrivé sans la contribution de toutes ces formidables personnes. Je n'oublie pas non plus mes potes, ils se reconnaîtront.

Merci à toi minou, tu as toujours été là au quotidien pour moi. Je te dois quasiment tout.

Poser la bonne question est plus difficile que d'y répondre.

Georg Cantor

Contents

Introduction	1
1 Penalization method in incompressible flows	3
1.1 Governing equations	3
1.2 Penalization method for moving obstacles	5
1.3 Fluid-structure interactions (FSI)	7
1.3.1 Hydrodynamic forces and moments	7
1.3.2 Derivation of the equations of motion for a rigid body im- mersed in an incompressible fluid	8
2 Implementation in BigFlow	13
2.1 BigFlow: an overview	13
2.2 Adding the penalization	14
2.2.1 Implicit Euler time-marching	15
2.2.2 Explicit Euler time-marching	16
2.3 FSI solver	16
3 Numerical validation	18
3.1 Fixed case	18
3.1.1 Flow past a sphere	19
3.1.2 Flow past a cylinder	26
3.2 Moving case	31
3.2.1 Sedimentation of a sphere	32
4 Applications	37
4.1 Flow past a cube at $Re = 100$ with different orientations	37
4.1.1 Numerical setup	38
4.1.2 Evaluation of the projected surface	39
4.1.3 Results	41
4.2 Flow past a wavy wall	44
4.2.1 Numerical setup	44

4.2.2	Results	47
4.3	Sedimentation of spheroidal bodies	51
4.3.1	Numerical setup	52
4.3.2	Results	54
5	Conclusion	59
	Bibliography	62

List of Figures

1	Comparison between (a) Boddy-fitted mesh and Cartesian grid (b).	1
1.1	Domain definition. Solid obstacle Ω_s immersed in a fluid domain Ω_f .	4
1.2	Representation of a mask function for a 2D cylinder.	6
1.3	Normal profile of the mask function χ corresponding to different amounts of smoothing.	6
2.1	MAC mesh arrangement. The gray arrows are reserved for the velocity in the x component, the blue for the velocity in the y component and the red for the velocity in the z component. The green diamond node is the location for scalar fields (i.e., pressure, temperature).	13
3.1	Computational domain for the flow past a sphere.	20
3.2	Stretched grid in the Oxy plane used for flow past a sphere.	20
3.3	(a) Time evolution of the lift and drag coefficients for flow past a sphere. (b) Power spectrum based on the lift coefficient for the flow past a sphere to evaluate the shedding frequency.	22
3.4	$\Delta\tau$ -convergence analysis for flow past a sphere.	23
3.5	Snapshots of the magnitude of the velocity field $\frac{ \mathbf{u} }{U_\infty}$	24
3.6	Snapshots of the magnitude of the vorticity field $\frac{ \boldsymbol{\omega} D}{U_\infty}$	24
3.7	Vortical structures detected using the λ_2 criterion for the flow past a sphere.	25
3.8	Computational domain for the flow past a cylinder.	26
3.9	Stretched grid in the Oxy plane used for flow past a cylinder.	27
3.10	(a) Time evolution of the lift and drag coefficients for flow past a cylinder. (b) Power spectrum based on the lift coefficient for flow past a cylinder to evaluate the shedding frequency.	29
3.11	Snapshot of the magnitude of the velocity field $\frac{ \mathbf{u} }{U_\infty}$	30

3.12	Snapshot of the magnitude of the vorticity field $\frac{ \boldsymbol{\omega} D}{U_\infty}$	30
3.13	Computational domain for the sedimentation of a sphere.	32
3.14	Creation of a uniform grid around the sphere with the stretching grid method.	33
3.15	Comparison with the experimental data of the sphere fall velocity and the normalized height from the bottom wall for different Reynolds numbers (symbols: experiment [1], solid lines: present study). . . .	34
3.16	Contours of the magnitude of the velocity $ \mathbf{u} $ for each Reynolds number took when the the normalized height in each case is $\frac{z}{D} = 1.5$. . .	35
3.17	Contours of the magnitude of the vorticity $ \boldsymbol{\omega} $ for each Reynolds number took when the the normalized height in each case is $\frac{z}{D} = 1.5$. . .	36
4.1	All configurations studied. ψ going from 0° to 45° with a step of 11.25°	37
4.2	Computational domain for the flow past a cube.	38
4.3	Projected surface versus ψ , the angle around the z -axis.	40
4.4	Drag and lift coefficients against the z -axis angle ψ	41
4.5	Snapshot of the magnitude of the velocity field $\frac{ \mathbf{u} }{U_\infty}$ in the Oxy plane at $\frac{z}{L} = 3$	42
4.6	Snapshot of the magnitude of the vorticity field $\frac{ \boldsymbol{\omega} D}{U_\infty}$ in the Oxy plane at $\frac{z}{L} = 3$	43
4.7	Computational domain for the flow past a wavy wall in a half channel.	45
4.8	Top view of the domain (Oxz plane).	45
4.9	Snapshots of the magnitude of the velocity field (left) and the vorticity field (right) in the Oxy plane at $\frac{z}{2A} = 3.5$. Comparison at different Reynolds numbers.	48
4.10	Snapshots of the magnitude of the velocity field (left) and the vorticity field (right) in the Oxz plane at $\frac{y}{2A} = 0.85$. Comparison at different Reynolds numbers.	49
4.11	Vortical structures detected using the λ_2 criterion for the wavy wall. Comparison at different Reynolds number.	50
4.12	Two different spheroids: if $c < a$ oblate (left), if $c > a$ prolate (right).	51
4.13	Computational domain of the sedimentation of spheroids.	52
4.14	Initial orientations for the oblate and the prolate before falling.	53
4.15	Time-evolution of the roll angle described with the Euler angles for all cases. Dashed lines: oblate, solid lines: prolate.	54
4.16	Snapshot of the contour of $\frac{ \mathbf{u} D_{eq}}{\nu}$ in the plane Oxz at $\frac{y}{L_y} = \frac{1}{2}$	55
4.17	Snapshot of the contour of $\frac{ \boldsymbol{\omega} D_{eq}^2}{\nu}$ in the plane Oxz at $\frac{y}{L_y} = \frac{1}{2}$	56

4.18	Vortical structures detected using the λ_2 criterion for the sedimentation of the oblate. Comparison with different initial orientations. . .	57
4.19	Vortical structures detected using the λ_2 criterion for the sedimentation of the prolate. Comparison with different initial orientations. . .	58

List of Tables

3.1	Setup for the simulation of the flow past a sphere.	20
3.2	Comparison of drag and lift coefficients and Strouhal number for flow past a sphere.	22
3.3	Setup for the simulation of the flow past a cylinder.	27
3.4	Comparison of drag and lift coefficients and Strouhal number for flow past a cylinder.	29
3.5	Sedimentation of a sphere in an enclosure: numerical setup.	32
3.6	Parameters for the sedimentation of the sphere test problem.	33
4.1	Setup for the simulation of the flow past a cube.	38
4.2	Setup for the simulation of the wavy wall.	46

Introduction

Numerical simulations related to Computational Fluid Dynamics (CFD) have widely progressed since advances in computing enabled it, especially the simulation of incompressible flows around obstacles of arbitrary shapes. The advantage of these simulations is their ability to predict the physical behavior of fluid flows with sufficient of precision. In some cases, CFD can replace expensive experiments, which could constitute a real economic issue for some applications.

For the case of incompressible flows around complex geometry, two general techniques are used, namely: body-fitted grid methods [2] and Immersed Boundary Methods (IBM) [3] (see Figure 1 for the grids comparison).

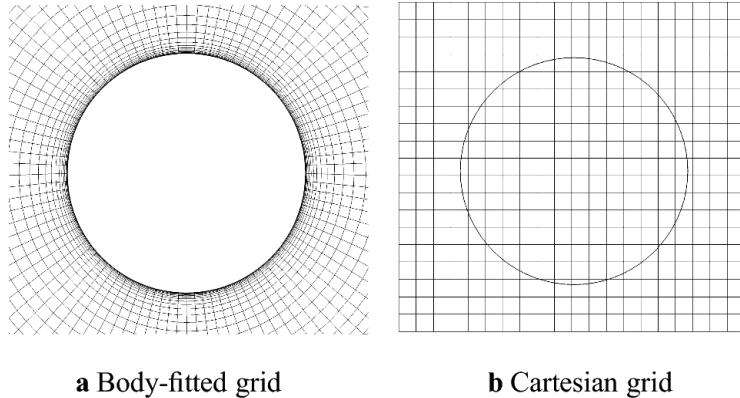


Figure 1: Comparison between (a) Boddy-fitted mesh and Cartesian grid (b).

In structured/unstructured body-fitted grid methods, the mesh is generated in a way to conform to the complex boundaries. The advantages of these techniques are the easy to impose boundary conditions and the high accuracy that can be attain by refining the mesh around the obstacles to capture the boundary layers, where fine resolutions are required for high Reynolds number. Nevertheless, the disadvantage of the grid generation process is not without consequence. Indeed, this process can be expensive even for simple geometries. But the main inconvenient

is for moving obstacles, where the grid generation process has to be done at each time step. An alternative is the immersed boundary methods, where it is carry out to use a fixed non-body conformal Cartesian grid to simulate flows around bodies. The procedure of taking into account the immersed boundary effects on the fluid depends on the family of methods used belonging to the IBM. One of the main advantage of these methods is the possibility of using fast and efficient spectral, finite differences or finite volumes approximations on Cartesian meshes.

In this thesis, the implementation and validation of a IBM, named penalization method, is presented. The method has been implemented in BigFlow, a Navier-Stokes equations numerical solver developed at UCL by different members of Thermodynamics and Fluid Mechanics team (TFL).

The thesis is organized as follows. The first chapter is devoted to the introduction of the penalization method and to the equations governing the fluid-structure interactions for rigid bodies in incompressible flows. The second chapter is dedicated to the numerical methods used and how it was implemented in BigFlow. In the third chapter, validation cases are proposed for fixed and moving bodies in order to prove that the implementation of the penalization method is accurate and reliable. Finally, three different applications are presented in chapter four, namely: flow past a cube with different orientations, flow past a wavy wall and the sedimentation of spheroid bodies, in order to visualize more complex flows than canonical validation flows.

Chapter 1

Penalization method in incompressible flows

The penalization method for incompressible flows is a modelling technique to solve the flow past obstacles of complex geometries. This method belongs to the class of continuous forcing IBM and has been widely used in recent years, which confirms the effectiveness of that method.

In this chapter, the governing equations of the Navier-Stokes equations using the penalization method will be introduced. After that, a smoothing procedure will be explained in order to deal with moving bodies. Finally, the governing equations of the fluid-structure interactions for rigid bodies in incompressible flows will be detailed.

1.1 Governing equations

The behaviour of an incompressible viscous flow past a body is governed by the Navier-Stokes (N-S) equations:

$$\rho_f \left[\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} \right] + \nabla p - \mu \nabla^2 \mathbf{u} = \mathbf{f}, \quad \text{in } \Omega_f \quad (1.1)$$

$$\nabla \cdot \mathbf{u} = 0, \quad \text{in } \Omega_f \quad (1.2)$$

$$\mathbf{u} = \mathbf{u}_{\Gamma_s}, \quad \text{on } \Gamma_s \quad (1.3)$$

where $\mathbf{u}(\mathbf{x}, t)$ is the velocity field, $\mathbf{f}(\mathbf{x}, t)$ is a body force (e.g., gravity), $p(\mathbf{x}, t)$ is the pressure field, ρ_f is the density of the fluid and μ its viscosity. The solid body occupies the domain Ω_s delimited by the boundary Γ_s , and the surrounding fluid

occupies the domain Ω_f , as illustrated in Figure 1.1.

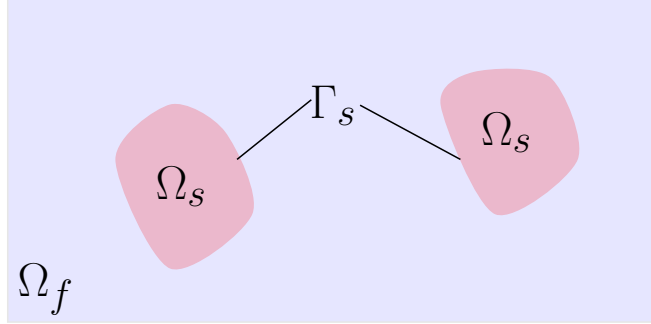


Figure 1.1: Domain definition. Solid obstacle Ω_s immersed in a fluid domain Ω_f .

The momentum conservation equation can be cast into another form dividing all terms by the density and lumping the hydrostatic pressure contribution $\rho_f g z$ and the classical pressure p together. We thus obtain:

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla P + \nu \nabla^2 \mathbf{u}. \quad (1.4)$$

where $P = \frac{p + \rho g z}{\rho}$ is the kinematic pressure and $\nu = \frac{\mu}{\rho}$ is the kinematic viscosity.

The main idea of the penalization method is to solve the N-S equations in the whole domain $\Omega = \Omega_f \cup \Omega_s$ while modifying the momentum equation to include the boundary conditions. The boundary conditions of the obstacles are taken into account by adding an extra term to Eq.(1.4). This extra term can take various forms depending of the method selected from the IBM. Here, since we work with the penalization method, the modified N-S equations take the following form:

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla P + \nu \nabla^2 \mathbf{u} - \chi \frac{(\mathbf{u} - \mathbf{u}_s)}{\Delta \tau}, \quad \text{in } \Omega \quad (1.5)$$

$$\nabla \cdot \mathbf{u} = 0, \quad \text{in } \Omega \quad (1.6)$$

$$\mathbf{u}_s = \mathbf{u}_{\Gamma_s}. \quad \text{on } \Gamma_s \quad (1.7)$$

The presence of the extra term brings a new parameter and new variables:

- $\Delta \tau$ which has the dimension of time and can be considered as the permeability of a medium. The obstacles are considered as a fluid with very low permeability $\Delta \tau \ll 1$.

- A mask function χ , whose utility is to track the obstacles. It is defined as:

$$\chi(\mathbf{x}, t) = \begin{cases} 1 & \text{if } \mathbf{x} \in \Omega_s \\ 0 & \text{if } \mathbf{x} \in \Omega_f. \end{cases}$$

- The velocity field of the obstacles $\mathbf{u}_s$, which maps the Lagrangian velocities of the obstacles from a Lagrangian grid to an Eulerian grid.

In the limit of $\Delta\tau \rightarrow 0$, it is proved in [4] that Eqs.(1.5), (1.6) and (1.7) tend to Eqs.(1.1), (1.2) and (1.3).

1.2 Penalization method for moving obstacles

As discussed so far, the penalization method uses a discontinuous mask function. If we use this mask for a non-grid aligned body, it will produce the so-called “staircase” effect. Results for stationary obstacles are acceptable, but spurious oscillations are reported in the case of moving ones [5]. Indeed, since the discontinuous mask can only be translated by integer multiples of the grid space h , this shaking motion will cause some oscillations in the computation of the hydrodynamic forces, causing some trouble during the resolution of the equations of motion of the body.

Therefore, the idea in this work is to add the possibility of smoothing the mask function over a thin smoothing layer. To do so, let’s introduce the signed distance, which in the case of a 2D cylinder of radius R and center $\mathbf{x}_b$ takes the following form:

$$\delta(\mathbf{x}, t) = ||\mathbf{x} - \mathbf{x}_b|| - R. \quad (1.8)$$

Now, we can use the following smoothing mask function, where ϵ is the semi-thickness of the smoothing layer. In general, it is defined relative to the grid size h : $\epsilon = C_{smth}h$.

$$\chi(\delta) = \begin{cases} 1 & \text{if } \delta < -\epsilon \\ 0.5 \left[1 + \left(\frac{\delta}{\epsilon} \right) + \left(\frac{1}{\pi} \right) \sin \left(\frac{\pi\delta}{\epsilon} \right) \right] & \text{if } |\delta| \leq \epsilon \\ 0 & \text{if } \delta > \epsilon. \end{cases} \quad (1.9)$$

The Figure 1.2 compares the mask function of a 2D cylinder with a smoothing layer $\epsilon = 8h$ and with no smoothing. The Figure 1.3 shows the normal profile of the mask function in terms of different smoothing layer thickness. In general, a smoothing layer thickness of $4h$ or $5h$ is sufficient.

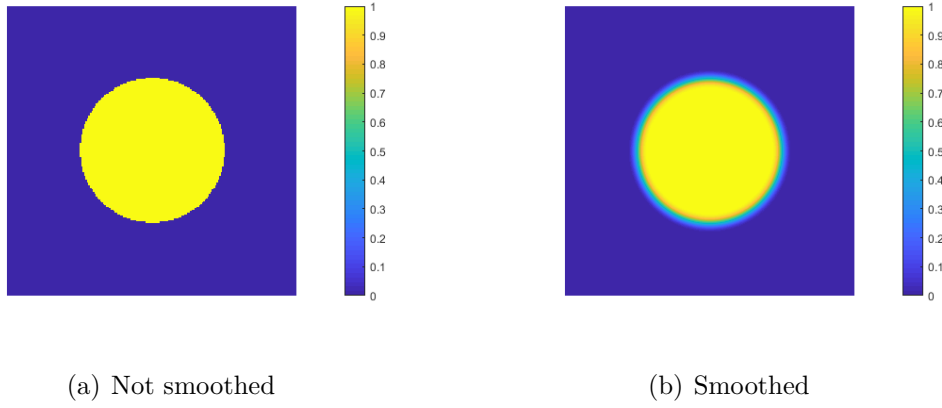


Figure 1.2: Representation of a mask function for a 2D cylinder.

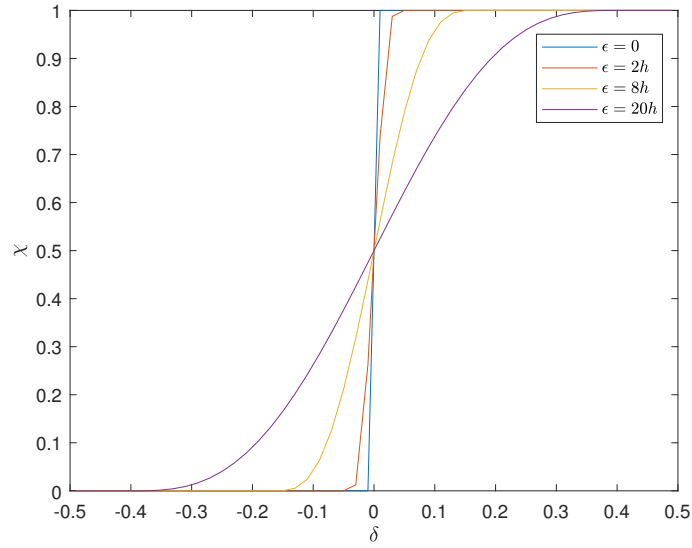


Figure 1.3: Normal profile of the mask function χ corresponding to different amounts of smoothing.

1.3 Fluid-structure interactions (FSI)

1.3.1 Hydrodynamic forces and moments

For fluid-structure interactions problems, the penalization method offers a practical way to compute the hydrodynamic forces and moments acting on the immersed bodies. Indeed, since the penalization term has dimensions of a force density, the intuition would be to integrate this term over the volume of the body. In fact, it will be shown later that this intuition is correct only for bodies at rest or evolving at constant velocity.

The hydrodynamic forces acting on a body are obtained by integrating the stress tensor over the surface of the body:

$$\mathbf{F}_{f \rightarrow b} = \rho_f \oint_{\Gamma_s} (\boldsymbol{\sigma} \cdot \hat{\mathbf{n}}) dS = \rho_f \underbrace{\int_{\Omega_s} (\nabla \cdot \boldsymbol{\sigma}) dV}_{\text{Divergence theorem}} \quad (1.10)$$

where $\boldsymbol{\sigma} = -P\mathbf{I} + \nu [\nabla \mathbf{u} + (\nabla \mathbf{u})^T]$ is the hydrodynamic stress tensor, $\hat{\mathbf{n}}$ the unit normal pointing outward the body and ρ_f the fluid density.

The N-S equations expressed in terms of the divergence of the stress tensor and with the penalization term are

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} \triangleq \frac{D\mathbf{u}}{Dt} = \nabla \cdot \boldsymbol{\sigma} - \chi \frac{(\mathbf{u} - \mathbf{u}_s)}{\Delta\tau} \quad (1.11)$$

and thus

$$\nabla \cdot \boldsymbol{\sigma} = \chi \frac{(\mathbf{u} - \mathbf{u}_s)}{\Delta\tau} + \frac{D\mathbf{u}}{Dt}. \quad (1.12)$$

Finally, combining Eq.(1.10) and Eq.(1.12) leads to the final expression of the hydrodynamic force:

$$\mathbf{F}_{f \rightarrow b} = \rho_f \int_{\Omega_s} \chi \frac{(\mathbf{u} - \mathbf{u}_s)}{\Delta\tau} dV + \rho_f \int_{\Omega_s} \frac{D\mathbf{u}}{Dt} dV. \quad (1.13)$$

The term $\rho_f \int_{\Omega_s} \frac{D\mathbf{u}}{Dt} dV$ in Eq.(1.13) is called the unsteady correction. If the body is rigid, it can be computed as $\rho_f V_b \dot{\mathbf{u}}_b$, as described in [6], where V_b is the volume of the body and $\dot{\mathbf{u}}_b$ the acceleration vector of its center of mass. Conversely, if the body is flexible, the unsteady correction has to be computed numerically with all the difficulties that entails.

Similarly, using the divergence theorem and Eq.(1.12), the computation of the hydrodynamic moment is done in the same way as the hydrodynamic force:

$$\mathbf{M}_{f \rightarrow b} = \rho_f \oint_{\Gamma_s} \mathbf{r} \times (\boldsymbol{\sigma} \cdot \hat{\mathbf{n}}) dS \quad (1.14)$$

$$= \rho_f \int_{\Omega_s} \mathbf{r} \times (\nabla \cdot \boldsymbol{\sigma}) dV \quad (1.15)$$

$$= \rho_f \int_{\Omega_s} \mathbf{r} \times \left[\chi \frac{(\mathbf{u} - \mathbf{u}_s)}{\Delta \tau} \right] dV + \rho_f \int_{\Omega_s} \mathbf{r} \times \left(\frac{D\mathbf{u}}{Dt} \right) dV, \quad (1.16)$$

where $\mathbf{r} = \mathbf{x} - \mathbf{x}_b$ is the position vector relative to the center of mass.

Here again, the unsteady correction $\rho_f \int_{\Omega_s} \mathbf{r} \times \left(\frac{D\mathbf{u}}{Dt} \right) dV$ can be computed as $\rho_f (\mathbf{J} \cdot \dot{\boldsymbol{\Omega}}_b)$ for rigid bodies, where $\mathbf{J}$ is the inertia tensor of the body and $\boldsymbol{\Omega}_b$ is the angular velocity vector.

To summarize, as this study focuses only on rigid bodies, the computation of the hydrodynamic forces and moments will be performed as

$$\mathbf{F}_{f \rightarrow b} = \rho_f \int_{\Omega_s} \chi \frac{(\mathbf{u} - \mathbf{u}_s)}{\Delta \tau} dV + \rho_f V_b \dot{\mathbf{u}}_b \quad (1.17)$$

$$\mathbf{M}_{f \rightarrow b} = \rho_f \int_{\Omega_s} \mathbf{r} \times \left[\chi \frac{(\mathbf{u} - \mathbf{u}_s)}{\Delta \tau} \right] dV + \rho_f (\mathbf{J} \cdot \dot{\boldsymbol{\Omega}}_b). \quad (1.18)$$

1.3.2 Derivation of the equations of motion for a rigid body immersed in an incompressible fluid

A body immersed in a fluid is subject to hydrodynamic and buoyancy forces in addition to gravity. On the other hand, the body exerts a force on the fluid through the force density penalization term. Therefore, the rigid body motion equations (so called Newton-Euler) have to be solved together with the fluid equations:

$$\rho_b V_b \dot{\mathbf{u}}_b^{(g)} = \mathbf{F}_{f \rightarrow b}^{(g)} + V_b (\rho_b - \rho_f) \mathbf{g}^{(g)} \quad (1.19)$$

$$\rho_b \left[\mathbf{J}^{(b)} \cdot \dot{\boldsymbol{\Omega}}_b^{(b)} + \boldsymbol{\Omega}_b^{(b)} \times (\mathbf{J}^{(b)} \cdot \boldsymbol{\Omega}_b^{(b)}) \right] = \mathbf{M}_{f \rightarrow b}^{(b)} \quad (1.20)$$

where the superscripts (g) and (b) are used to express that a vector is defined in the global or body frame, respectively. $\mathbf{F}_{f \rightarrow b}^{(g)}$ and $\mathbf{M}_{f \rightarrow b}^{(g)}$ are obtained from Eqs.(1.17) and (1.18).

To handle the rotational degrees of freedom while avoiding the so-called ‘Gimbal lock’ problem, a quaternion based formulation, similar to the one proposed in [5], is used. The rotation matrix R_b based on the normalized quaternions q_i , where $\sum_i q_i^2 = 1$, is expressed as

$$R_b = \begin{pmatrix} q_0^2 + q_1^2 - q_2^2 - q_3^2 & 2(q_1q_2 - q_3q_0) & 2(q_1q_3 + q_2q_0) \\ 2(q_1q_2 + q_3q_0) & q_0^2 - q_1^2 + q_2^2 - q_3^2 & 2(q_2q_3 - q_1q_0) \\ 2(q_1q_3 - q_2q_0) & 2(q_2q_3 + q_1q_0) & q_0^2 - q_1^2 - q_2^2 + q_3^2 \end{pmatrix}. \quad (1.21)$$

The initial values for q_i can be conveniently computed from a set of yaw, pitch and roll angles (ψ, θ, ϕ)

$$\begin{pmatrix} q_0 \\ q_1 \\ q_2 \\ q_3 \end{pmatrix} = \begin{pmatrix} \cos\left(\frac{\psi}{2}\right) \cos\left(\frac{\beta}{2}\right) \cos\left(\frac{\phi}{2}\right) + \sin\left(\frac{\psi}{2}\right) \sin\left(\frac{\beta}{2}\right) \sin\left(\frac{\phi}{2}\right) \\ \sin\left(\frac{\psi}{2}\right) \cos\left(\frac{\beta}{2}\right) \cos\left(\frac{\phi}{2}\right) - \cos\left(\frac{\psi}{2}\right) \sin\left(\frac{\beta}{2}\right) \sin\left(\frac{\phi}{2}\right) \\ \cos\left(\frac{\psi}{2}\right) \sin\left(\frac{\beta}{2}\right) \cos\left(\frac{\phi}{2}\right) + \sin\left(\frac{\psi}{2}\right) \cos\left(\frac{\beta}{2}\right) \sin\left(\frac{\phi}{2}\right) \\ \cos\left(\frac{\psi}{2}\right) \cos\left(\frac{\beta}{2}\right) \sin\left(\frac{\phi}{2}\right) - \sin\left(\frac{\psi}{2}\right) \sin\left(\frac{\beta}{2}\right) \cos\left(\frac{\phi}{2}\right) \end{pmatrix}. \quad (1.22)$$

The governing equation for the quaternions is

$$\dot{\mathbf{q}} = \frac{1}{2} \mathbf{S}^T \cdot \boldsymbol{\Omega}_b \quad (1.23)$$

with the matrix $\mathbf{S}$ defined as

$$\mathbf{S} = \begin{pmatrix} -q_1 & q_0 & -q_3 & -q_2 \\ -q_2 & -q_3 & q_0 & q_1 \\ -q_3 & q_2 & -q_1 & q_0 \end{pmatrix}. \quad (1.24)$$

The Euler angles can be obtained from the quaternions via the relations:

$$\begin{pmatrix} \psi \\ \theta \\ \phi \end{pmatrix} = \begin{pmatrix} \arctan\left(\frac{2(q_0q_3 + q_2q_1)}{1 - 2(q_1^2 + q_2^2)}\right) \\ \arcsin(2(q_0q_2 - q_3q_1)) \\ \arctan\left(\frac{2(q_0q_3 + q_1q_2)}{1 - 2(q_2^2 + q_3^2)}\right) \end{pmatrix}. \quad (1.25)$$

The solid phase velocity field $\mathbf{u}_s^{(g)}$ expressed in the global frame is obtained as

$$\mathbf{u}_s^{(g)} = \mathbf{u}_b^{(g)} + R_b^{-1} \cdot (\boldsymbol{\Omega}_b^{(b)} \times \mathbf{r}^{(b)}) \quad (1.26)$$

with

$$\mathbf{r}^{(b)} = R_b \cdot (\mathbf{x}^{(g)} - \mathbf{x}_b^{(g)}), \quad (1.27)$$

the position vector relative to the center of mass expressed in the body frame.

Assuming that $\mathbf{J}^{(b)}$ is constant in the body and that the body axes to be the principal axes of inertia (which will be always the case in this work), $\mathbf{J}^{(b)}$ is diagonal. The set of differential equations governing the body motion is then expressed as:

$$\begin{cases} \dot{\mathbf{x}}_b^{(g)} &= \mathbf{u}_b^{(g)} \\ \dot{\mathbf{q}} &= \frac{1}{2} \mathbf{S}^T \cdot \boldsymbol{\Omega}_b^{(b)} \\ \dot{\mathbf{u}}_b^{(g)} &= \frac{1}{V_b(\rho_r - 1)} \left(\int_{\Omega_s} \chi \frac{(\mathbf{u} - \mathbf{u}_s)}{\Delta\tau} dV \right)^{(g)} + \mathbf{g}^{(g)} \\ \dot{\boldsymbol{\Omega}}_b^{(b)} &= \frac{(\mathbf{J}^{(b)})^{-1}}{(\rho_r - 1)} \left(\int_{\Omega_s} \mathbf{r} \times \left[\chi \frac{(\mathbf{u} - \mathbf{u}_s)}{\Delta\tau} \right] dV \right)^{(b)} - \frac{(\mathbf{J}^{(b)})^{-1}}{(\rho_r - 1)} \rho_r \boldsymbol{\Omega}_b^{(b)} \times (\mathbf{J}^{(b)} \cdot \boldsymbol{\Omega}_b^{(b)}) \end{cases} \quad (1.28)$$

where $\rho_r = \frac{\rho_b}{\rho_f}$ is called the relative density.

By introducing the following notations (the superscripts which indicates the frame of work are removed for simplicity):

$$\begin{aligned} \mathbf{F} &= \int_{\Omega_s} \chi \frac{(\mathbf{u} - \mathbf{u}_s)}{\Delta\tau} dV, & \mathbf{M} &= \int_{\Omega_s} \mathbf{r} \times \left[\chi \frac{(\mathbf{u} - \mathbf{u}_s)}{\Delta\tau} \right] dV, \\ \mathbf{M}_c &= \rho_r \boldsymbol{\Omega}_b \times (\mathbf{J} \cdot \boldsymbol{\Omega}_b), & \mathbf{M}_{tot} &= \mathbf{M} - \mathbf{M}_c, \end{aligned}$$

$$\mathbf{Q} = \frac{1}{2} \mathbf{S}^T \cdot \boldsymbol{\Omega}_b,$$

the system of equations (1.28) can be recast in a more suitable form:

$$\begin{cases} \dot{\mathbf{x}}_b &= \mathbf{u}_b \\ \dot{\mathbf{q}} &= \mathbf{Q} \\ \dot{\mathbf{u}}_b &= \frac{1}{V_b(\rho_r - 1)} \mathbf{F} + \mathbf{g} \\ \dot{\boldsymbol{\Omega}}_b &= \frac{(\mathbf{J}^{(b)})^{-1}}{(\rho_r - 1)} \mathbf{M}_{tot}. \end{cases} \quad (1.29)$$

The system of equations (1.29) presents a singularity when $\rho_r = 1$. To overcome this problem and being able to solve the equations for a wide range of relative densities, the interpretation of the motion equations must be modified when $\rho_r \approx 1$. The singularity occurs because of the unsteady correction during the evaluation of the hydrodynamic forces and moments. Instead of treating this term implicitly as before, it will be treated explicitly. Let's illustrate the idea.

Consider the translational equation of motion

$$\rho_b V_b \dot{\mathbf{u}}_b = \rho_f \int_{\Omega_s} \chi \frac{(\mathbf{u} - \mathbf{u}_s)}{\Delta\tau} dV + \underbrace{\rho_f V_b \dot{\mathbf{u}}_b}_{\text{Unst. corr.}} + V_b(\rho_b - \rho_f) \mathbf{g}, \quad (1.30)$$

so that

$$\dot{\mathbf{u}}_b = \frac{1}{\rho_r V_b} \int_{\Omega_s} \chi \frac{(\mathbf{u} - \mathbf{u}_s)}{\Delta\tau} dV + \frac{1}{\rho_r} \dot{\mathbf{u}}_b + \frac{(\rho_r - 1)}{\rho_r} \mathbf{g}. \quad (1.31)$$

Therefore, by calling the acceleration term of the R.H.S. $\mathbf{a}_b$, the update of a light body ($\rho_r \approx 1$) velocity is performed as

$$\dot{\mathbf{u}}_b = \frac{1}{\rho_r V_b} \int_{\Omega_s} \chi \frac{(\mathbf{u} - \mathbf{u}_s)}{\Delta\tau} dV + \frac{1}{\rho_r} \mathbf{a}_b + \frac{(\rho_r - 1)}{\rho_r} \mathbf{g}. \quad (1.32)$$

The same procedure can be applied for the rotational equation:

$$\dot{\boldsymbol{\Omega}}_b = \frac{(\mathbf{J})^{-1}}{\rho_r} \left(\int_{\Omega_s} \mathbf{r} \times \left[\chi \frac{(\mathbf{u} - \mathbf{u}_s)}{\Delta\tau} \right] dV - \rho_r \boldsymbol{\Omega}_b \times (\mathbf{J} \cdot \boldsymbol{\Omega}_b) \right) + \frac{1}{\rho_r} \boldsymbol{\alpha}_b \quad (1.33)$$

with $\boldsymbol{\alpha}_b$ is the angular acceleration due to the unsteady correction term.

By writing the system of equations as follows, we can deal with light and heavy bodies:

$$\left\{ \begin{array}{lcl} \dot{\mathbf{x}}_b & = & \mathbf{u}_b \\ \dot{\mathbf{q}} & = & \mathbf{Q} \\ \dot{\mathbf{u}}_b & = & c_F \mathbf{F} + c_a \mathbf{a}_b + c_g \mathbf{g} \\ \dot{\boldsymbol{\Omega}}_b & = & \mathbf{c}_M \mathbf{M}_{tot} + c_\alpha \boldsymbol{\alpha}_b \end{array} \right. \quad (1.34)$$

where the coefficients c_F, c_a, c_g, c_α and the tensor $\mathbf{c}_M$ are defined depending of the relative density of the body:

$$\textbf{Light body : } c_F = \frac{1}{V_b \rho_r} \quad c_a = \frac{1}{\rho_r} \quad c_g = \frac{(\rho_r - 1)}{\rho_r} \quad c_M = \frac{(\mathbf{J})^{-1}}{\rho_r} \quad c_\alpha = \frac{1}{\rho_r}.$$

$$\textbf{Heavy body : } c_F = \frac{1}{V_b(\rho_r - 1)} \quad c_a = 0 \quad c_g = 1 \quad c_M = \frac{(\mathbf{J})^{-1}}{(\rho_r - 1)} \quad c_\alpha = 0.$$

Chapter 2

Implementation in BigFlow

2.1 BigFlow: an overview

BigFlow is a Navier-Stokes equations solver developed at UCL by different members of Thermodynamics and Fluid Mechanics group (TFL). The space discretization used is based on the fourth order finite difference schemes developed by Vasilyev [7]. As shown in Figure 2.1, the structured Cartesian mesh is staggered, following Harlow and Welsh's [8] Marker and Cell (MAC) arrangement.

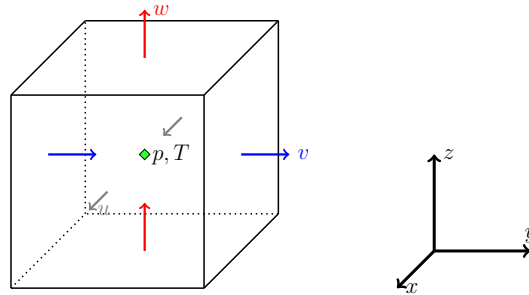


Figure 2.1: MAC mesh arrangement. The gray arrows are reserved for the velocity in the x component, the blue for the velocity in the y component and the red for the velocity in the z component. The green diamond node is the location for scalar fields (i.e., pressure, temperature).

The grid can be stretched using one-dimensional mappings

$$\frac{x_i}{L_i} = f(\xi_i)$$

where $x_i \in [0, L_i]$ is the coordinate in the physical space and $\xi_i \in [0, 1]$ the coordinate in the computational domain.

The time-stepping scheme used in BigFlow to solve the unsteady Navier-Stokes equations for incompressible flows is the fractional step method of Lee and Kim [9]. The intermediate velocity $\mathbf{u}^*$ is obtained using an Adams-Bashforth 2 time-stepping for the convective term and an implicit Crank-Nicolson scheme for the diffusive term:

$$\frac{\mathbf{u}^* - \mathbf{u}^n}{\Delta t} = -\nabla P^n - \frac{1}{2} \left(3\mathcal{H}^n - \mathcal{H}^{n-1} \right) + \frac{\nu}{2} (\nabla^2 \mathbf{u}^* + \nabla^2 \mathbf{u}^n) \quad (2.1)$$

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

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

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

where $\mathcal{H}$ is the convective term, and with the following boundary conditions

$$\mathbf{u}_\Gamma^* = \mathbf{u}_\Gamma^{n+1} \quad (2.5)$$

$$\left. \frac{\partial \phi^{n+1}}{\partial n} \right|_\Gamma = 0. \quad (2.6)$$

In fact, the diffusive term is treated implicitly only if the user requests it. For example, the turbulent flow in a channel must have a very thin resolution near the wall for the direction normal to it (e.g. y -direction), leading to a strong constraint on the time-step. Indeed, the stability condition for the diffusive part depends on the Fourier number

$$r = \frac{\nu \Delta t}{\Delta y^2}, \quad (2.7)$$

which has to be sufficiently small.

In the present work, the diffusive term is integrated explicitly together with the convective term using an Adams-Bashforth 2 scheme. Therefore, Eq.(2.1) can be rewritten as:

$$\frac{\mathbf{u}^* - \mathbf{u}^n}{\Delta t} = -\nabla P^n - \frac{1}{2} \left(3\mathcal{H}^n - \mathcal{H}^{n-1} \right) + \frac{\nu}{2} \left(3\nabla^2 \mathbf{u}^n - \nabla^2 \mathbf{u}^{n-1} \right). \quad (2.8)$$

For more details about the space discretization and how the BigFlow solver works, please refer to Duponcheel [10].

2.2 Adding the penalization

The integration of the penalization term will be performed with an explicit or an implicit Euler scheme. In the general case,

$$\begin{aligned} \frac{\mathbf{u}^* - \mathbf{u}^n}{\Delta t} = & -\nabla P^n - \frac{1}{2} \left(3\mathcal{H}^n - \mathcal{H}^{n-1} \right) + \frac{\nu}{2} \left(3\nabla^2 \mathbf{u}^n - \nabla^2 \mathbf{u}^{n-1} \right) \\ & - \chi \left(\frac{\mathbf{u}^k - \mathbf{u}_s^{n+1}}{\Delta \tau} \right), \end{aligned} \quad (2.9)$$

where the superscript k in the penalization term depends on the scheme used (* for implicit and n for explicit). The superscript $n + 1$ of the velocity field $\mathbf{u}_s$ assumes that the body motion has been solved.

2.2.1 Implicit Euler time-marching

The intermediate velocity field $\mathbf{u}^*$ is updated as follows when using an implicit Euler scheme for the penalization term:

$$\begin{aligned} \frac{\mathbf{u}^* - \mathbf{u}^n}{\Delta t} &= \underbrace{-\nabla P^n + \frac{1}{2} \left(3\mathcal{F}^n - \mathcal{F}^{n-1} \right)}_{\text{RHS}} - \chi \left(\frac{\mathbf{u}^* - \mathbf{u}_s^{n+1}}{\Delta \tau} \right) \\ \mathbf{u}^* &= \mathbf{u}^n + \Delta t (\text{RHS}) - \chi \frac{\Delta t}{\Delta \tau} \mathbf{u}^* + \chi \frac{\Delta t}{\Delta \tau} \mathbf{u}_s^{n+1} \\ \mathbf{u}^* &= \underbrace{\frac{1}{\left(1 + \chi \frac{\Delta t}{\Delta \tau} \right)}}_{\text{Amplification factor}} \mathbf{u}^n + \frac{\Delta t}{\left(1 + \chi \frac{\Delta t}{\Delta \tau} \right)} (\text{RHS}) + \frac{\chi \frac{\Delta t}{\Delta \tau}}{\left(1 + \chi \frac{\Delta t}{\Delta \tau} \right)} \mathbf{u}_s^{n+1} \end{aligned}$$

Inside the body ($\chi = 1$), we have that for a large $\frac{\Delta t}{\Delta \tau}$ ($\frac{\Delta t}{\Delta \tau} \gg 1$) :

$$\frac{1}{\left(1 + \frac{\Delta t}{\Delta \tau} \right)} \rightarrow 0, \quad \frac{\Delta t}{\left(1 + \frac{\Delta t}{\Delta \tau} \right)} \rightarrow 0, \quad \frac{\frac{\Delta t}{\Delta \tau}}{\left(1 + \frac{\Delta t}{\Delta \tau} \right)} \rightarrow 1,$$

which leads to

$$\mathbf{u}^* \rightarrow \mathbf{u}_s^{n+1}$$

in the body. This is the best situation possible as the accuracy of the method can be controlled through the value given to $\left(\frac{\Delta t}{\Delta \tau} \right)$, without touching at the time step Δt , which is a desirable feature.

In terms of stability, this scheme is unconditionally stable since the amplification factor

$$\rho = \left| \frac{1}{\left(1 + \frac{\Delta t}{\Delta \tau} \right)} \right|$$

is always lower than one, whatever the choice of $\frac{\Delta t}{\Delta \tau}$.

2.2.2 Explicit Euler time-marching

When using an explicit Euler scheme for the penalization term, the intermediate velocity field $\mathbf{u}^*$ is updated as follows:

$$\frac{\mathbf{u}^* - \mathbf{u}^n}{\Delta t} = \underbrace{-\nabla P^n + \frac{1}{2}(3\mathcal{F}^n - \mathcal{F}^{n-1})}_{\text{RHS}} - \chi \left(\frac{\mathbf{u}^n - \mathbf{u}_s^{n+1}}{\Delta \tau} \right) \quad (2.10)$$

$$\mathbf{u}^* = \underbrace{\left(1 - \chi \frac{\Delta t}{\Delta \tau} \right)}_{\text{Amplification factor}} \mathbf{u}^n + \Delta t(\text{RHS}) + \chi \frac{\Delta t}{\Delta \tau} \mathbf{u}_s^{n+1}. \quad (2.11)$$

Inside the body ($\chi = 1$), we have that for $\frac{\Delta t}{\Delta \tau} = 1$,

$$\mathbf{u}^* = \Delta t(\text{RHS}) + \mathbf{u}_s^{n+1}.$$

By putting $\frac{\Delta t}{\Delta \tau} = 1$, the amplification factor term cancels, thus defining this choice as optimal for the explicit Euler scheme.

Compared to the implicit Euler scheme, the residual in the body is $\mathcal{O}(\Delta t)$. Therefore, the accuracy of the method can no longer be controlled with the value given to $\Delta \tau$, independently of Δt .

In terms of stability, this scheme is conditionally stable since

$$\rho = \left| 1 - \frac{\Delta t}{\Delta \tau} \right| \leq 1 \iff \frac{\Delta t}{\Delta \tau} \leq 2.$$

2.3 FSI solver

The numerical integration of the body equations is done with an Adam-Bashford 2 scheme, in order to keep the second order in time from the flow solver:

$$\begin{cases} \mathbf{x}_b^{n+1} &= \mathbf{x}_b^n + \frac{\Delta t}{2} (3\mathbf{u}_b^n - \mathbf{u}_b^{n-1}) \\ \mathbf{u}_b^{n+1} &= \mathbf{u}_b^n + \frac{c_F \Delta t}{2} (3\mathbf{F}^n - \mathbf{F}^{n-1}) + c_a \mathbf{a}_b^n + c_g \mathbf{g} \\ \mathbf{q}^{n+1} &= \mathbf{q}^n + \frac{\Delta t}{2} (3\mathbf{Q}^n - \mathbf{Q}^{n-1}) \\ \mathbf{\Omega}_b^{n+1} &= \mathbf{\Omega}_b^n + \frac{c_M \Delta t}{2} (3\mathbf{M}_{tot}^n - \mathbf{M}_{tot}^{n-1}) + c_\alpha \mathbf{\alpha}_b^n \end{cases} \quad (2.12)$$

where the acceleration terms due to the unsteady corrections are computed with a backward finite difference scheme:

$$\mathbf{a}_b^n = \frac{\mathbf{u}_b^n - \mathbf{u}_b^{n-1}}{\Delta t} \quad \quad \quad \boldsymbol{\alpha}_b^n = \frac{\boldsymbol{\Omega}_b^n - \boldsymbol{\Omega}_b^{n-1}}{\Delta t}.$$

From the system of equations (2.12), we can extract two numbers which are important to control the stability of the equations:

$$\lambda_{\mathbf{u}_b} = c_F \left(\frac{\Delta t}{\Delta \tau} \right) \quad (2.13)$$

$$\lambda_{\boldsymbol{\Omega}_b} = \max(c_{M_{i,j}}) \left(\frac{\Delta t}{\Delta \tau} \right). \quad (2.14)$$

Indeed, rewriting the translational and rotational equations of the system (1.34) gives

$$\begin{cases} \dot{\mathbf{u}}_b &= \frac{c_F}{\Delta \tau} \left(\int_{\Omega_s} \chi(\mathbf{u} - \mathbf{u}_s) dV \right) + c_a \mathbf{a}_b + c_g \mathbf{g} \\ \dot{\boldsymbol{\Omega}}_b &= \frac{\mathbf{c}_M}{\Delta \tau} \cdot \left(\int_{\Omega_s} \mathbf{r} \times (\chi(\mathbf{u} - \mathbf{u}_s)) dV \right) - \mathbf{c}_M \cdot \mathbf{M}_c + \frac{1}{\rho_r} \boldsymbol{\alpha}_b + c_\alpha \boldsymbol{\alpha}_b. \end{cases} \quad (2.15)$$

Since $\mathbf{u}_s = \mathbf{u}_b + \boldsymbol{\Omega}_b \times \mathbf{r}$, the sources of instability will mainly come from the integral terms. This is why the Euler explicit scheme for the penalization term is used when we are dealing with moving bodies. Indeed, as highlighted in the previous section, for an implicit Euler scheme, the ratio $\frac{\Delta t}{\Delta \tau}$ has to be as big as possible. In contrast, for the explicit Euler scheme, the optimal choice is $\frac{\Delta t}{\Delta \tau} = 1$, which guarantee a more stable behaviour of the FSI solver.

Chapter 3

Numerical validation

This chapter is devoted to the validation of the penalization method for fixed and moving bodies. The fixed case is dedicated to the study of the flow past a sphere at $Re = 300$ and flow past a cylinder at $Re = 200$. The choice of these benchmarks is motivated by their large documentation in the literature. To validate the fluid-structure interaction part of the code, the sedimentation experiment of Ten Cate *et al.* [1] is reproduced.

3.1 Fixed case

For this study, the flow past a sphere at $Re_D = 300$ and the flow past a 3D cylinder at $Re_D = 200$ will be investigated, where

$$Re_D = \frac{U_\infty D}{\nu}$$

is the Reynolds number based on their diameter D and the upstream velocity U_∞ .

The validation of these two cases is based on the comparison of some important figures with these reported in the literature. The drag and lift coefficients and the Strouhal number are respectively defined as:

$$C_D = \frac{2F_D}{\rho_f S_p U_\infty^2} \quad C_L = \frac{2F_L}{\rho_f S_p U_\infty^2} \quad St = \frac{f_{sh} D}{U_\infty} \quad (3.1)$$

where f_{sh} is the shedding frequency based on the the lift coefficient, ρ_f is the fluid density and S_p is the projected surface, the surface ‘seen’ by the inflow. For the sphere and for the cylinder, this surface is expressed as

$$S_{p,sph} = \pi \left(\frac{D^2}{4} \right) \quad S_{p,cyl} = DL$$

whit L the length of the cylinder.

F_D and F_L are the drag and lift forces. They are computed by projecting the hydrodynamic forces $\mathbf{F}_{f \rightarrow b}$ as follows:

$$\begin{pmatrix} F_D \\ F_L \end{pmatrix} = \begin{pmatrix} \mathbf{F}_{f \rightarrow b} \cdot \hat{\mathbf{e}}_x \\ \mathbf{F}_{f \rightarrow b} \cdot \hat{\mathbf{e}}_y \end{pmatrix} \quad (3.2)$$

where $\mathbf{F}_{f \rightarrow b}$ is evaluated from Eq.(1.17) setting $\mathbf{u}_s = \mathbf{0}$:

$$\mathbf{F}_{f \rightarrow b} = \rho_f \int_{\Omega_s} \chi \frac{\mathbf{u}}{\Delta \tau} dV. \quad (3.3)$$

Here, since the body is fixed, the penalization term will be treated implicitly to give a big value to the ratio $\frac{\Delta t}{\Delta \tau}$. For the 2 cases, we will have

$$\frac{\Delta t}{\Delta \tau} = 10^5.$$

3.1.1 Flow past a sphere

Numerical setup

The computational domain is shown in Figure 3.1 and the parameters of the simulation are resumed in Table 3.1. The grid is stretched in all directions, yielding gives a resolution $(D/h) \approx 83$. The Figure 3.2 shows the stretched grid in the Oxy plane.

To trigger the unsteady part of the flow, a perturbation is added to the uniform upstream flow between the non-dimensional time $\tilde{t} = t(U_\infty/D) = 3$ and $\tilde{t} = 4$ in the y direction. This perturbation has the form

$$v_{in} = \begin{cases} 0.1U_\infty \sin(\pi(\tilde{t} - 3)) & \text{if } 3 \leq \tilde{t} \leq 4 \\ 0 & \text{otherwise .} \end{cases}$$

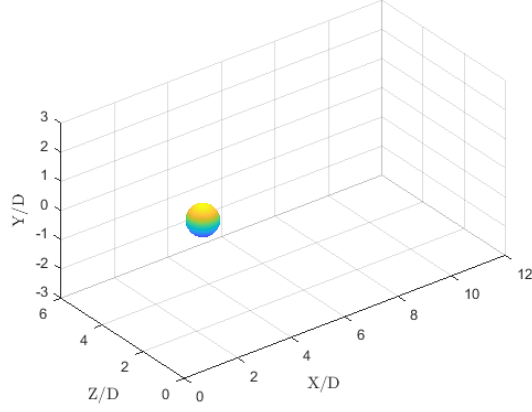


Figure 3.1: Computational domain for the flow past a sphere.

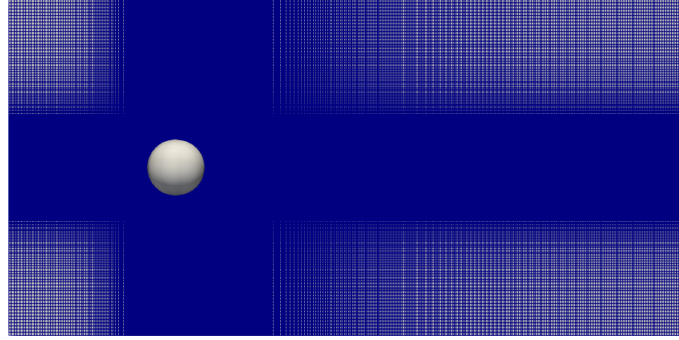


Figure 3.2: Stretched grid in the Oxy plane used for flow past a sphere.

Parameter	Value	Unity
Physical domain $L_x \times L_y \times L_z$	$12 \times 6 \times 6$	$[m^3]$
Box size $N_x \times N_y \times N_z$	$512 \times 256 \times 256$	$[-]$
Time step Δt	10^{-3}	$[s]$
Penalization parameter $\Delta\tau$ (implicit)	10^{-8}	$[s]$
Viscosity ν	$1/300$	$[m^2/s]$
Upstream velocity U_∞	1	$[m/s]$
Reynolds number Re_D	300	$[-]$

Table 3.1: Setup for the simulation of the flow past a sphere.

The conditions imposed on the boundaries of the domain are:

- X-direction

- A uniform free stream velocity

$$\mathbf{u}_\infty = (U_\infty, 0, 0)$$

is imposed at the inlet.

- A convective outflow condition

$$\frac{\partial \mathbf{u}}{\partial t} + U_m \frac{\partial \mathbf{u}}{\partial \mathbf{n}} = 0$$

is imposed at the outlet, where U_m is taken from the average value of the normal velocity at the outflow boundary.

- Y-direction

- A free slip condition is imposed on both sides:

$$\frac{\partial u}{\partial y} = 0 \qquad v = 0 \qquad \frac{\partial w}{\partial y} = 0$$

- Z-direction

- A free slip condition is imposed on both sides:

$$\frac{\partial u}{\partial z} = 0 \qquad \frac{\partial v}{\partial z} = 0 \qquad w = 0$$

Flow analysis and validation

The time-evolution of the drag and lift coefficients are presented in Figure 3.3(a). One way to compute the shedding frequency is to identify the frequency with the highest amplitude in the spectrum of the lift coefficient, truncated from the starting of the vortex shedding to the end, as shown in Figure 3.3(b).

The Table 3.2 show the comparison with the literature, and we can conclude that the lift and drag coefficients and the Strouhal number fit quite well.

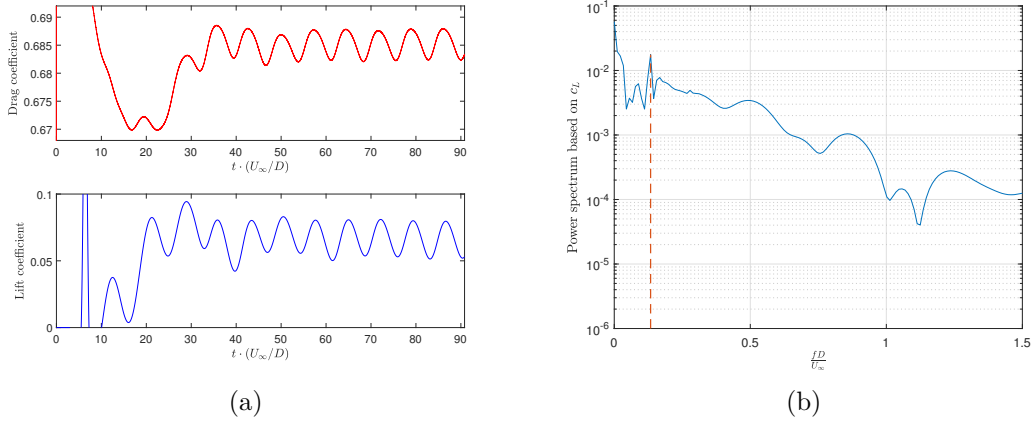


Figure 3.3: (a) Time evolution of the lift and drag coefficients for flow past a sphere. (b) Power spectrum based on the lift coefficient for the flow past a sphere to evaluate the shedding frequency.

Authors	$\bar{c}_D$	$\bar{c}_L$	S_t
Kim & Choi (2002)	0.657	0.067	0.134
Ploumhans et al. (2002)	0.683	0.061	0.135
Mimeau (2015)	0.673	0.066	0.133
Present	0.685	0.066	0.134

Table 3.2: Comparison of drag and lift coefficients and Strouhal number for flow past a sphere.

Convergence analysis

Here, a convergence analysis of the $\Delta\tau$ parameter is carried out in order to verify the rate of convergence. The value of $\Delta\tau$ ranges from 10^{-1} to 10^{-6} , and the reference case is $\Delta\tau = 10^{-8}$. The L_1 , L_2 and L_∞ norms of the error $e(y)$ reported in Figure 3.4 are defined as

$$e_{L_1}(y) = \left(\frac{1}{U_\infty D} \right) \int_{-\frac{D}{2}}^{\frac{D}{2}} |u(y) - u_{ref}(y)| dy$$

$$e_{L_2}(y) = \left(\frac{1}{U_\infty \sqrt{D}} \right) \sqrt{\int_{-\frac{D}{2}}^{\frac{D}{2}} (u(y) - u_{ref}(y))^2 dy}$$

$$e_{L_\infty}(y) = \left(\frac{1}{U_\infty} \right) \sup_{y \in [-D/2; D/2]} |u(y) - u_{ref}(y)|.$$

The procedure is to evaluate the x -component of the velocity field inside the solid in the range $y \in [-D/2; D/2]$ for every $\Delta\tau$ in the range $[10^{-6}; 10^{-1}]$ and compare it with u_{ref} , which is the x -component of the velocity field inside the solid when $\Delta\tau = 10^{-8}$.

The results are plotted in Figure 3.4. We can notice that the error decreases as $\mathcal{O}(\Delta\tau)$ for $\Delta\tau \geq 10^{-4}$ and as $\mathcal{O}(\Delta\tau^{1/2})$ when $\Delta\tau \leq 10^{-4}$. This convergence behaviour is in agreement with the results of Angot *et al.* [4], which state that the smaller $\Delta\tau$, the more the resolution must be increased to keep a first order convergence. Also, the theoretical results from [4] predict that the upper bound on L_∞ for the penalization is $\|\mathbf{u}_{\Delta\tau} - \mathbf{u}\| \leq C\Delta\tau^{1/2}$.

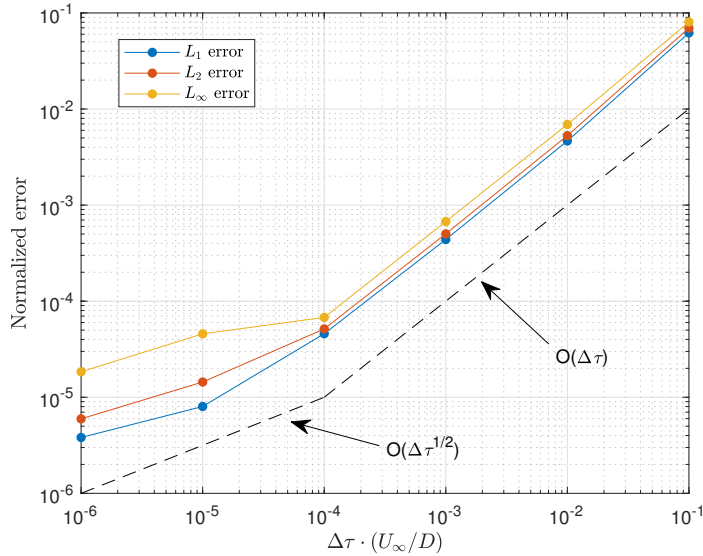


Figure 3.4: $\Delta\tau$ -convergence analysis for flow past a sphere.

Flow visualization

The Figures 3.5 and 3.6 show snapshots of the velocity and vorticity fields in the Oxz plane (top view) and in the Oxy plane (side view). As can be observed, a true periodic regime is reached. Moreover, the wake exhibits symmetric patterns in the Oxz plane. These observations confirm the description of the flow regime corresponding to Reynolds 300, as explained in [11].

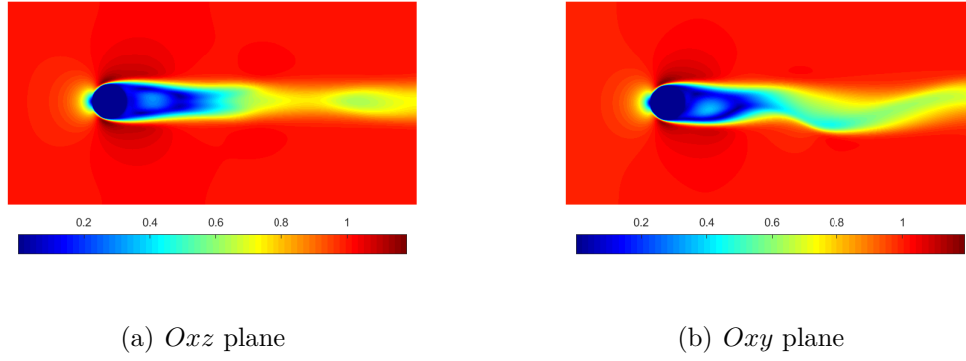


Figure 3.5: Snapshots of the magnitude of the velocity field $\frac{||\mathbf{u}||}{U_\infty}$.

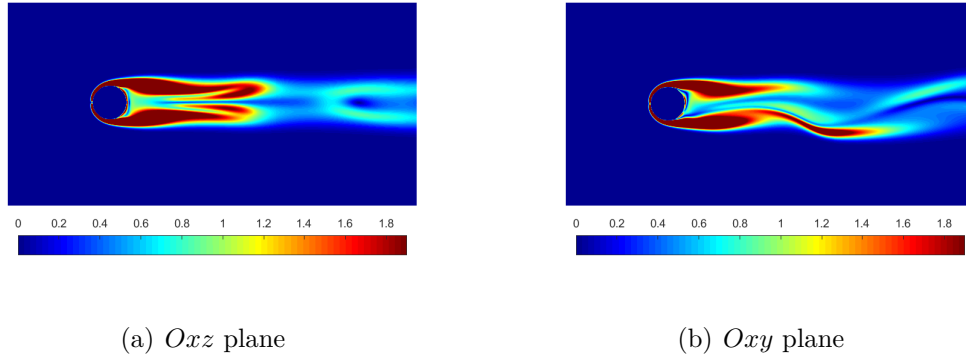
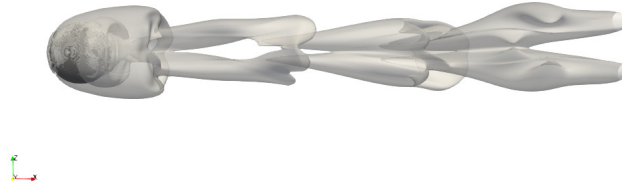


Figure 3.6: Snapshots of the magnitude of the vorticity field $\frac{||\boldsymbol{\omega}||D}{U_\infty}$.

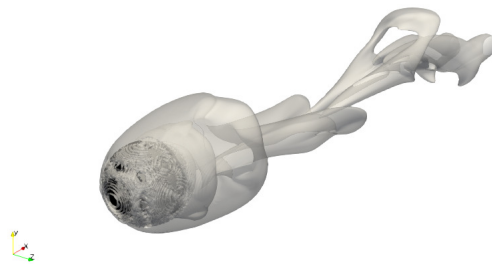
The Figure 3.7 shows the so-called hairpin vortices, which preserves the plane of symmetry. Moreover, one can observe that hairpin vortices are all shed with the same orientation.



(a) Top view



(b) Side view



(c) Perspective view

Figure 3.7: Vortical structures detected using the λ_2 criterion for the flow past a sphere.

3.1.2 Flow past a cylinder

In the scientific literature, the flow past a cylinder is well known in the 2D case, but less in 3D. Since BigFlow is a 3D solver, the setup of the flow will be done in a way to emulate the 2D flow by imposing periodic boundary conditions in the spanwise direction and by imposing that the length of the cylinder is the length of the spanwise direction.

Numerical setup

The computational domain of the simulation is shown in Figure 3.8. We can see the difference with the sphere computed just before. The spanwise direction has been reduced to get closer to a 2D domain, which permits to increase the domain in the x and y directions. Concerning the grid, since the spanwise is a periodic direction, the stretching is applied in the two other directions, i.e. the x and y directions, as shown in Figure 3.9. The resolution inside the cylinder obtained is $D/h \approx 80$. The Table 3.3 gives the different parameters used for the simulation. Here again, the same perturbation has been applied to trig the vortex shedding quickly in a way to save time.

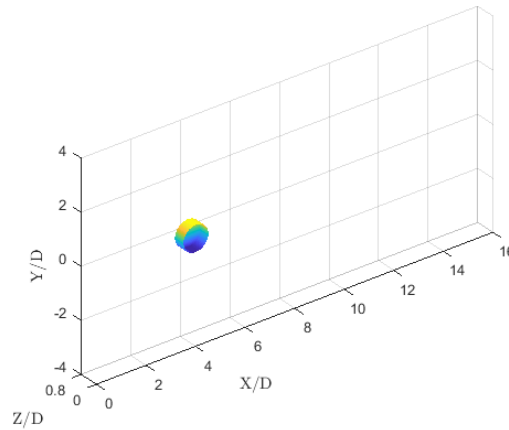


Figure 3.8: Computational domain for the flow past a cylinder.

Parameter	Value	Unity
Physical domain $L_x \times L_y \times L_z$	$16 \times 8 \times 0.8$	$[m^3]$
Box size $N_x \times N_y \times N_z$	$512 \times 256 \times 64$	$[-]$
Time step Δt	10^{-3}	$[s]$
Penalization parameter $\Delta\tau$ (implicit)	10^{-8}	$[s]$
Viscosity ν	$5 \cdot 10^{-2}$	$[m^2/s]$
Upstream velocity U_∞	1	$[m/s]$
Reynolds number Re_D	200	$[-]$

Table 3.3: Setup for the simulation of the flow past a cylinder.

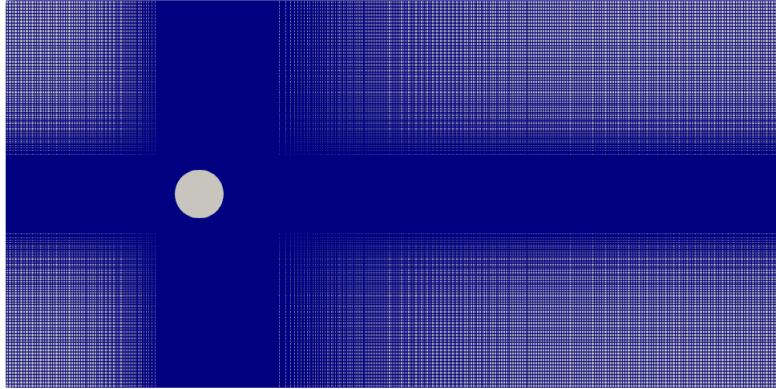


Figure 3.9: Stretched grid in the Oxy plane used for flow past a cylinder.

The following boundary conditions were applied to each face of the domain:

- X-direction

- A uniform free stream velocity

$$\mathbf{u}_\infty = (U_\infty, 0, 0)$$

is imposed at the inlet.

- A convective outflow condition

$$\frac{\partial \mathbf{u}}{\partial t} + U_m \frac{\partial \mathbf{u}}{\partial \mathbf{n}} = 0$$

is imposed at the outlet, where U_m is taken from the average value of the normal velocity at the outflow boundary.

- Y-direction

- A free slip condition is imposed on both sides:

$$\frac{\partial u}{\partial y} = 0 \qquad v = 0 \qquad \frac{\partial w}{\partial y} = 0$$

- Z-direction

- A periodic boundary condition is imposed on both sides.

Flow analysis and validation

The drag and lift coefficients are reported as function of time in Figure 3.10(a). The methodology to compute the Strouhal number is identical to the case of the sphere, as illustrated in Figure 3.10(b). Table 3.4 shows the comparison with literature results from 2D simulations. The match is less accurate than the 3D case. A reason is the stronger confinement effect of 2D flows with respect to 3D flows. To improve the accuracy, one has to increase the domain to let the flow to develop.

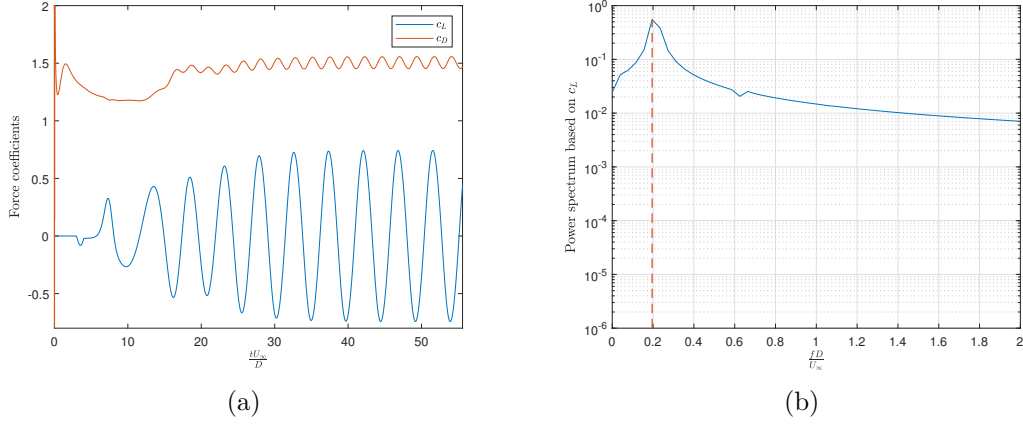


Figure 3.10: (a) Time evolution of the lift and drag coefficients for flow past a cylinder. (b) Power spectrum based on the lift coefficient for flow past a cylinder to evaluate the shedding frequency.

Authors	$\bar{c}_D$	$\bar{c}_L$	S_t
Braza et al. (1986)	1.40 ± 0.050	0 ± 0.75	0.190
Russel and Wang (2003)	1.45 ± 0.036	0 ± 0.63	0.201
Mimeau (2015)	1.44 ± 0.050	0 ± 0.75	0.200
Present	1.50 ± 0.052	0 ± 0.74	0.195

Table 3.4: Comparison of drag and lift coefficients and Strouhal number for flow past a cylinder.

Flow visualization

The Figures 3.11 and 3.12 represent snapshots of the velocity and vorticity fields in the Oxy plane. We can notice the presence of the so-called Von-Karman street in the vorticity field. Also, as explained in the previous section, the contours of the velocity field are cut at the boundaries, showing that the size of the domain is too small.

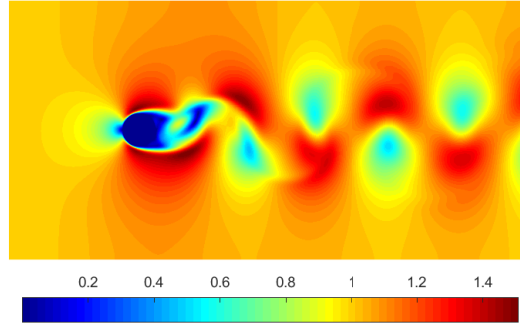


Figure 3.11: Snapshot of the magnitude of the velocity field $\frac{\|\mathbf{u}\|}{U_\infty}$.

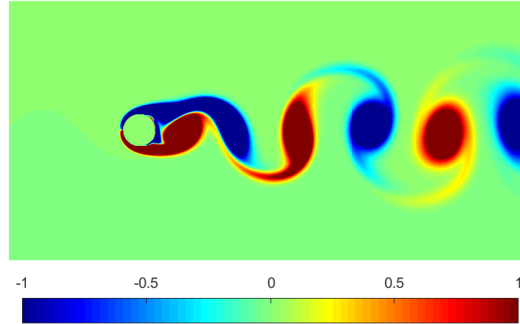


Figure 3.12: Snapshot of the magnitude of the vorticity field $\frac{\|\boldsymbol{\omega}\|D}{U_\infty}$.

3.2 Moving case

A next step in the validation procedure is to reproduce benchmarks for moving bodies. The sedimentation experiment of Ten Cate *et al.* [1] has been selected for its simplicity. Indeed, during the sedimentation of a sphere, a terminal velocity is reached when the net gravitational force (gravity minus buoyancy) equals the drag force. Therefore, a validation can be established in terms of the terminal velocity.

A way to compute the terminal velocity is to write the motion equation in the vertical direction and to cancel the acceleration:

$$0 = F_D - V_b(\rho_b - \rho_f)g. \quad (3.4)$$

Expressing F_D with the definition of the drag coefficient from Eq.(3.1) gives:

$$0 = \frac{1}{2}\rho_f S_p C_D(Re_D) u_{b,z}^2 - V_b(\rho_b - \rho_f)g. \quad (3.5)$$

For low Reynolds number, the drag coefficient C_D can be expressed as a function of Re_D , using the Abraham's model [12]:

$$C_D(Re_D) = \frac{24}{9.06^2} \left(\frac{9.06}{\sqrt{Re_D}} + 1 \right)^2, \quad (3.6)$$

with $Re_D = \frac{\rho_f D u_{b,z}}{\mu}$. At terminal velocity u_∞ , it follows that

$$0 = \frac{1}{2}\rho_f S_p C_D(u_\infty) u_\infty^2 - V_b(\rho_b - \rho_f)g. \quad (3.7)$$

The non-linear equation (3.7) can be solved with a Newton-Raphson scheme, for example.

To compare with the experiment of [1], a sphere is released in 4 different fluids and the terminal velocity reached by the sphere can be compared with the solution of Eq.(3.7).

3.2.1 Sedimentation of a sphere

Numerical setup

The computational domain is represented in Figure 3.13. The sphere is released from rest at a height of 12 [cm]. The parameters of the simulation are listed in Table 3.5. The properties of the four fluids are summarized in Table 3.6. As detailed in Chapter 2, the treatment of moving bodies requires to use an explicit Euler scheme to integrate the penalization term.

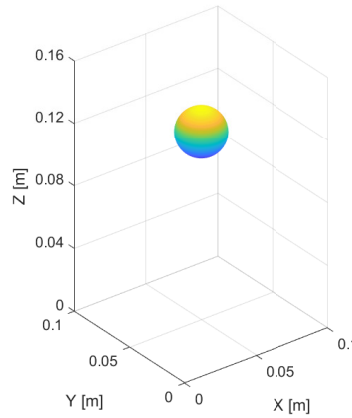


Figure 3.13: Computational domain for the sedimentation of a sphere.

Parameter	Value	Unity
Physical domain $L_x \times L_y \times L_z$	$10 \times 10 \times 16$	$[cm^3]$
Box size $N_x \times N_y \times N_z$	$128 \times 128 \times 192$	$[-]$
Time step Δt	$0.5 \cdot 10^{-4}$	$[s]$
Penalization parameter $\Delta \tau$ (explicit)	$0.5 \cdot 10^{-4}$	$[s]$
Diameter D	1.5	$[cm]$
Sphere density ρ_b	1120	$[kg/m^3]$

Table 3.5: Sedimentation of a sphere in an enclosure: numerical setup.

Remark :

As evidenced in Table 3.5, the grid is not isotropic with that combination of grid points. The ratio of $L_z/L_x = L_z/L_y = 1.6$ and the ratio of $N_z/N_x = N_z/N_y = 1.5$. A stretching was applied in the x and y directions to help the convergence of the Poisson solver. It has been decided to keep the ratio of 1.5 between the number

Case	ρ_f (kg/m^3)	μ_f ($10^{-3}Ns/m^2$)	u_∞ (m/s)	Re_D
C1	970	373	0.038	1.5
C2	965	212	0.06	4.1
C3	962	113	0.091	11.6
C4	960	58	0.128	31.9

Table 3.6: Parameters for the sedimentation of the sphere test problem.

grid points and to create a zone around the sphere where the grid is uniform, as shown in Figure 3.14. It must be noticed that the stretching has not been used to increase the resolution around the sphere but only to create a uniform mesh around it, which is required for the penalization method. By the way, the resolution inside the sphere $D/h \approx 18$, which is enough for these Reynolds number.

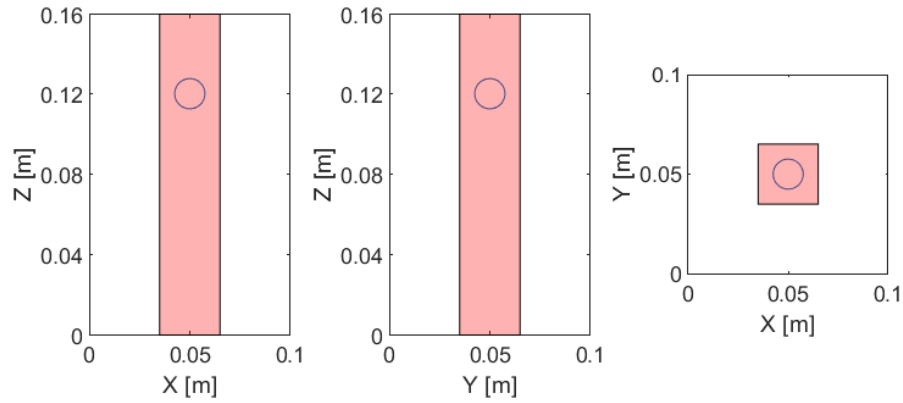


Figure 3.14: Creation of a uniform grid around the sphere with the stretching grid method.

Flow analysis and validation

Figure 3.15 compares the vertical velocity and the vertical position of the sphere with the experimental results from [1]. The terminal velocity is added in dashed line for the different cases. The agreement between our numerical results and the experiments is satisfying. Since the implementation of a collision model was not the purpose of this work, we did not consider the data after the sphere has touched the ground. Therefore, the data in solid lines (present study) that are after the symbols (data experiment) cannot be interpreted.

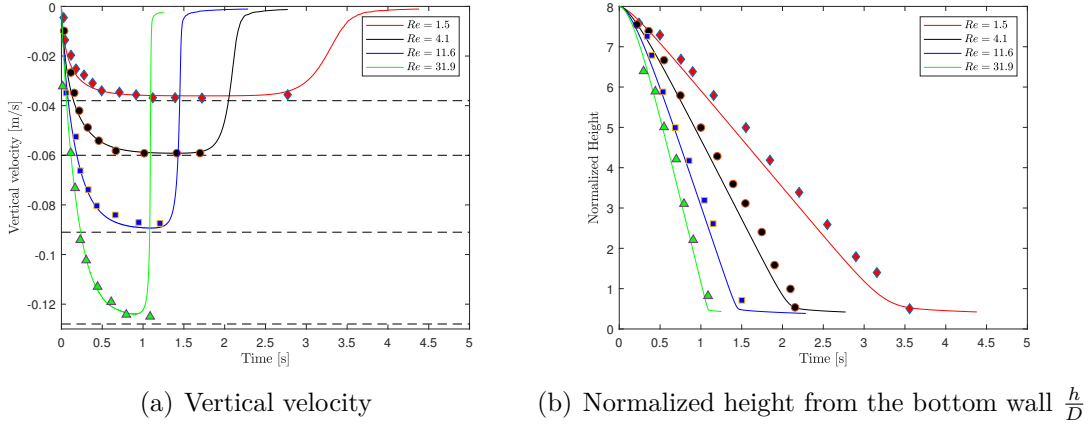
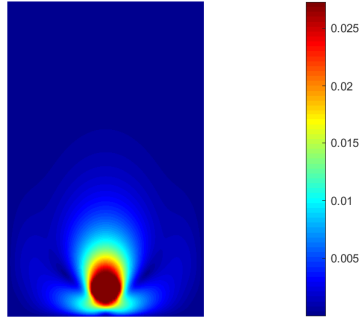


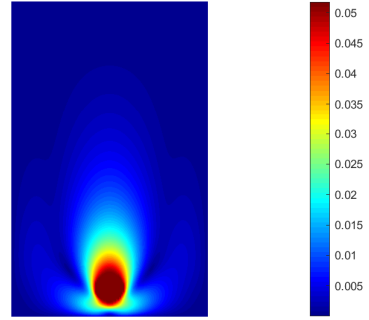
Figure 3.15: Comparison with the experimental data of the sphere fall velocity and the normalized height from the bottom wall for different Reynolds numbers (symbols: experiment [1], solid lines: present study).

Flow visualization

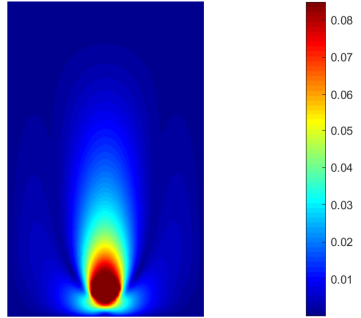
The Figures 3.16 and 3.17 are the contours of the magnitude of the velocity and vorticity, respectively. They have been taken at the time where $\frac{z}{D} = 1.5$. This normalized height has been selected to see the maximum developed wake just before the collision with the ground. Also, at this height, the ground effects are clearly observable in both fields, and more the Reynolds number grows, more this effect is important.



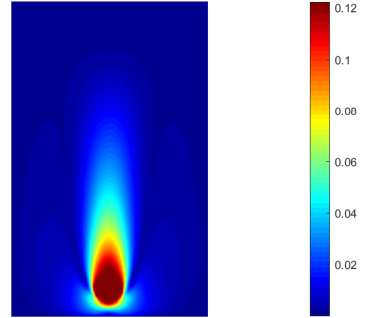
(a) Case 1



(b) Case 2



(c) Case 3



(d) Case 4

Figure 3.16: Contours of the magnitude of the velocity $||\mathbf{u}||$ for each Reynolds number took when the the normalized height in each case is $\frac{z}{D} = 1.5$.

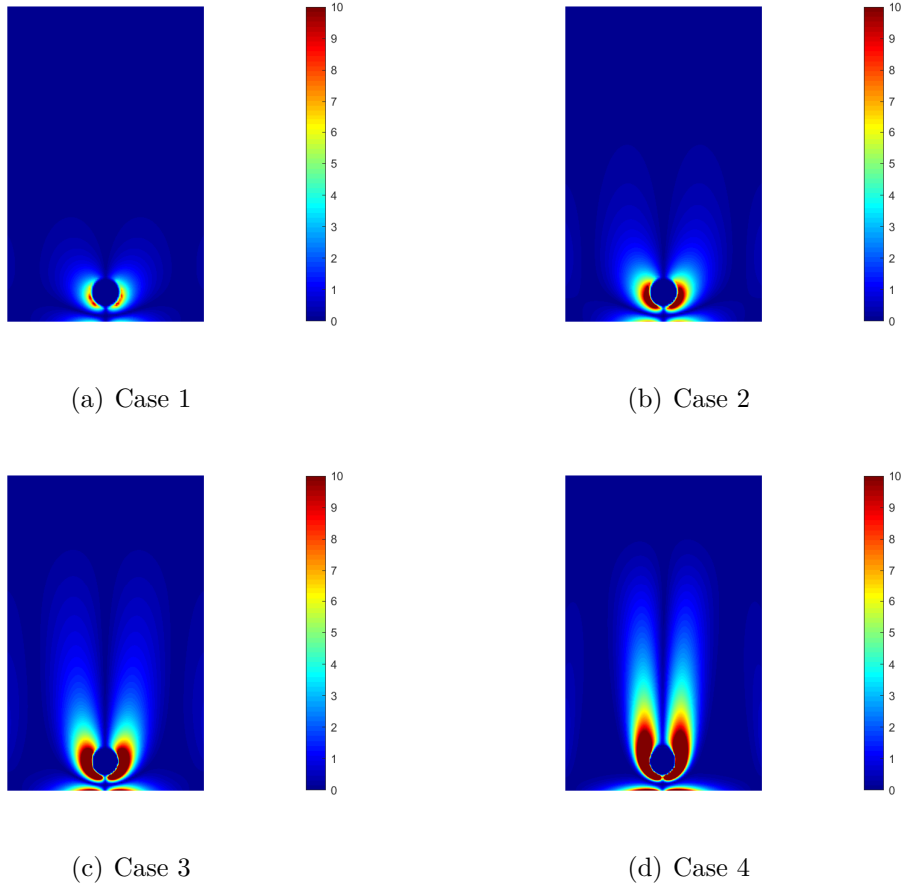


Figure 3.17: Contours of the magnitude of the vorticity $||\boldsymbol{\omega}||$ for each Reynolds number took when the the normalized height in each case is $\frac{z}{D} = 1.5$.

Chapter 4

Applications

4.1 Flow past a cube at $Re = 100$ with different orientations

In this application, 5 simulations of a flow past a cube at $Re = 100$ have been done with 5 different orientations around the spanwise axis (z -axis). The angle ψ starts from $\psi = 0^\circ$ to $\psi = 45^\circ$ with a step of $\psi = 11.25^\circ$. The Figure 4.1 illustrates the 5 situations.



Figure 4.1: All configurations studied. ψ going from 0° to 45° with a step of 11.25° .

The idea behind this application is to see how the drag and lift coefficients are influenced by the different orientations of the cube.

4.1.1 Numerical setup

The computational domain is shown on Figure 4.2 and the parameters of the simulation are resumed on Table 4.1. The Reynolds number is based on the length of the cube L and on the free stream velocity U_∞ :

$$Re_L = \frac{U_\infty L}{\nu}$$

Since the body is fixed, the Euler implicit scheme has been preferred for the penalized term. The resolution inside the cube is $L/h \approx 43$, which is enough for a Reynolds number of 100. The grid is not stretched and the mesh is uniform everywhere.

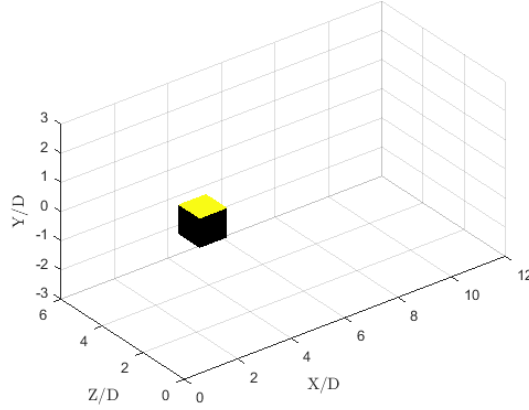


Figure 4.2: Computational domain for the flow past a cube.

Parameter	Value	Unity
Physical domain	$12 \times 6 \times 6$	$[m^3]$
Box size	$512 \times 256 \times 256$	$[-]$
Time step Δt	10^{-3}	$[s]$
Penalization parameter $\Delta\tau$ (implicit)	10^{-6}	$[s]$
Viscosity ν	0.01	$[m^2/s]$
Upstream velocity U_∞	1	$[m/s]$
Reynolds number Re_L	100	$[-]$

Table 4.1: Setup for the simulation of the flow past a cube.

The boundary conditions used are the following :

- X-direction

- An uniform free stream velocity

$$\mathbf{u}_\infty = (U_\infty, 0, 0)$$

is imposed at the inlet.

- A convective outflow condition

$$\frac{\partial \mathbf{u}}{\partial t} + U_m \frac{\partial \mathbf{u}}{\partial \mathbf{n}} = 0$$

is imposed at the outlet, where U_m is taken from the average value of the normal velocity at the outflow boundary.

- Y-direction

- A free slip condition is imposed on both sides:

$$\frac{\partial u}{\partial y} = 0 \qquad v = 0 \qquad \frac{\partial w}{\partial y} = 0.$$

- Z-direction

- A free slip condition is imposed on both sides:

$$\frac{\partial u}{\partial z} = 0 \qquad \frac{\partial v}{\partial z} = 0 \qquad w = 0.$$

4.1.2 Evaluation of the projected surface

Before computing the lift and drag coefficients, we need to evaluate the projected surface in function of the cube orientation, that is to say, the surface that the flow ‘seen’ when passing around the cube. The expressions of the drag and lift coefficients are

$$C_D = \frac{2F_D}{\rho_f S_P(\psi, \theta, \phi) U_\infty^2}, \qquad C_L = \frac{2F_L}{\rho_f S_P(\psi, \theta, \phi) U_\infty^2} \quad (4.1)$$

where (ψ, θ, ϕ) is the attitude of the cube expressed with the Euler angles. The projected surface S_p can be computed by taking the sum of the 3 projected surfaces of the cube that face the flow:

$$S_p(\psi, \theta, \phi) = L^2 (|\hat{n}_u \cdot \hat{n}_1| + |\hat{n}_u \cdot \hat{n}_2| + |\hat{n}_u \cdot \hat{n}_3|) \quad (4.2)$$

where

$$\hat{n}_{\mathbf{u}} = \frac{\mathbf{u}}{\|\mathbf{u}\|} \quad (4.3)$$

$$\hat{n}_1 = R(\psi, \theta, \phi) \cdot \hat{n}_x \quad (4.4)$$

$$\hat{n}_2 = R(\psi, \theta, \phi) \cdot \hat{n}_y \quad (4.5)$$

$$\hat{n}_3 = R(\psi, \theta, \phi) \cdot \hat{n}_z, \quad (4.6)$$

with $R(\psi, \theta, \phi)$ the rotation matrix of the Euler angles

$$R(\psi, \theta, \phi) = \begin{pmatrix} \cos \psi \cos \phi - \sin \psi \cos \theta \sin \phi & -\cos \psi \sin \phi - \sin \psi \cos \theta \cos \phi & \sin \psi \sin \theta \\ \sin \psi \cos \phi + \cos \psi \cos \theta \sin \phi & -\sin \psi \sin \phi + \cos \psi \cos \theta \cos \phi & -\cos \psi \sin \theta \\ \sin \theta \sin \phi & \sin \theta \cos \phi & \cos \theta \end{pmatrix}.$$

Here, since we only deal with rotations around the z -axis, the projected surface takes the following form:

$$S_p(\psi) = L^2(|\cos \psi| + |\sin \psi|). \quad (4.7)$$

The Figure 4.3 shows the projected surface versus ψ . As expected, the angle ψ where the surface is maximum is at $\psi = 45^\circ$.

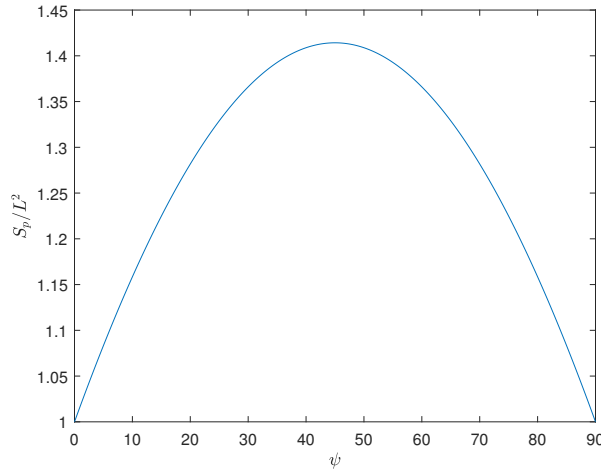


Figure 4.3: Projected surface versus ψ , the angle around the z -axis.

4.1.3 Results

The definition of a reference surface to evaluate the lift and drag coefficients can be different depending on the application. Here, we compare the force coefficients with two surfaces of reference:

- C_D and C_L , with the reference surface $S_p(\psi)$ calculated in the previous section, and thus depending on the orientation.
- $\tilde{C}_D$ and $\tilde{C}_L$, with the reference surface $S_p = L^2$, which is constant in all situations.

The Figure 4.4 shows the results obtained for the force coefficients. Depending on the definition of S_p , the curves don't exhibit the same shape. For the drag coefficients, C_D decreases to reach a minimum at $\psi = 33.75^\circ$ while $\tilde{C}_D$ keeps growing. In fact, $\tilde{C}_D$ is a direct image of the drag force F_D because $\tilde{C}_D$ scales F_D with the constant surface area $S_p = L^2$. Therefore, it is evident that the drag force increases with ψ in the range $[0^\circ; 45^\circ]$ since the projected surface S_p increases. The lift coefficients C_L and $\tilde{C}_L$ have the same shape and are both null when the orientation of the cube is symmetric, i.e. at $\psi = 0^\circ$ and $\psi = 45^\circ$.

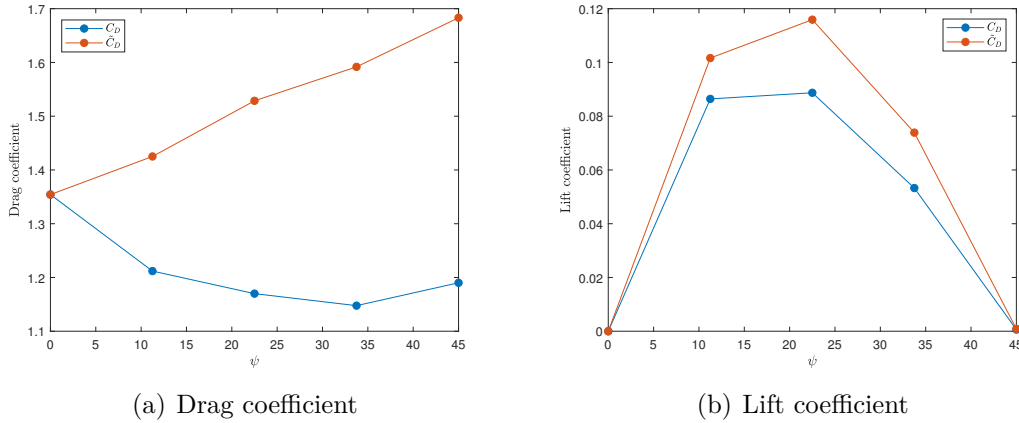
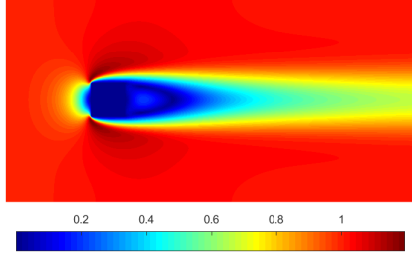
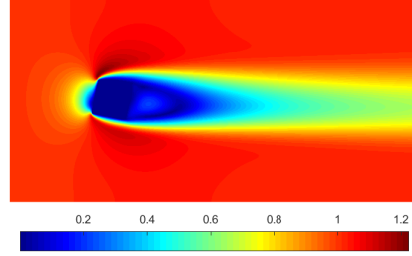


Figure 4.4: Drag and lift coefficients against the z -axis angle ψ .

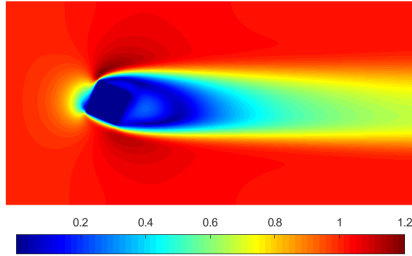
The Figure 4.5 and 4.6 are snapshots of the normalized velocity field and vorticity field respectively, for all configurations. We can notice that the flow is steady for all configurations. Compared to a square cylinder, a cube produces vortex sheddings at higher Reynolds number.



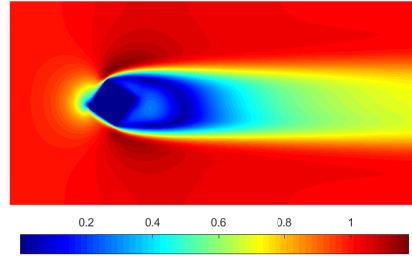
(a) $\psi = 0^\circ$



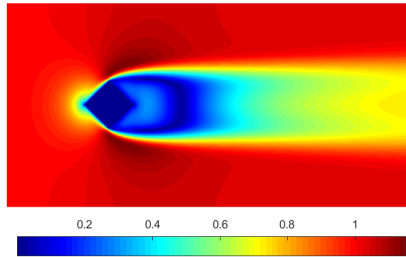
(b) $\psi = 11.25^\circ$



(c) $\psi = 22.5^\circ$

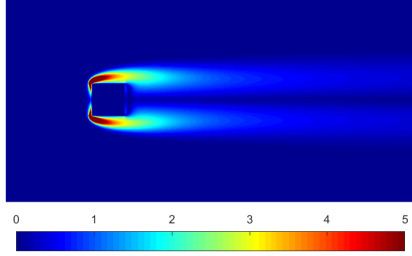


(d) $\psi = 33.75^\circ$

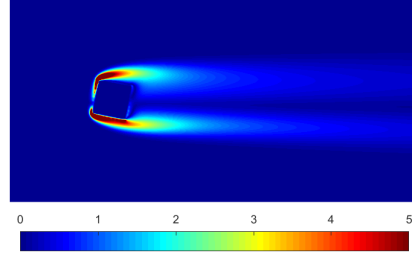


(e) $\psi = 45^\circ$

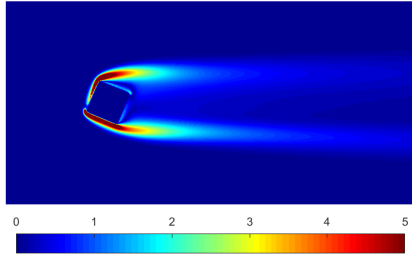
Figure 4.5: Snapshot of the magnitude of the velocity field $\frac{||\mathbf{u}||}{U_\infty}$ in the Oxy plane at $\frac{z}{L} = 3$.



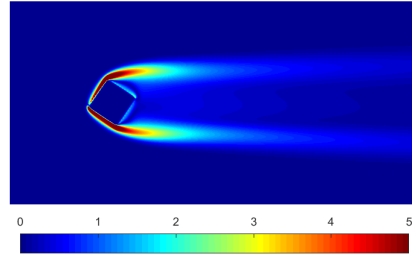
(a) $\psi = 0^\circ$



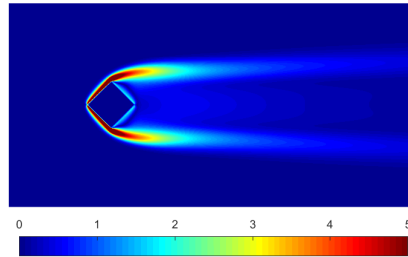
(b) $\psi = 11.25^\circ$



(c) $\psi = 22.5^\circ$



(d) $\psi = 33.75^\circ$



(e) $\psi = 45^\circ$

Figure 4.6: Snapshot of the magnitude of the vorticity field $\frac{||\omega||D}{U_\infty}$ in the Oxy plane at $\frac{z}{L} = 3$.

4.2 Flow past a wavy wall

This application consists of studying the flow past a wavy wall. The waves are present in the streamwise and spanwise directions. This kind of waves can be used to model for examples the roughness of a wall, mountains chain, etc... Here, we only focus on a simple case at low Reynolds numbers.

4.2.1 Numerical setup

The mathematical function describing the shape of the wall is composed of three sub-functions, as illustrated in Figure 4.7:

1. The first is a S-shape function, whose utility is to elevate the wall smoothly until the height $y = A$, where A is the amplitude of the waves.
2. The second is the wavy function, composed of a product of sine functions in the streamwise and spanwise directions.
3. The third is again a S-shape function, whose utility is the contrary of the first one, that is, to connect the height $y = A$ to 0 smoothly.

Therefore, the elevation of the wall H_{wall} is described as follows:

$$H_{wall}(x, z) = \begin{cases} A \sin^2\left(\frac{\pi}{2}\eta_1\right) & \text{if } x \leq L_{in} \\ A (1 + \sin(k_x x + \phi_x) \sin(k_z z)) & \text{if } L_{in} < x \leq (L_{in} + L_w) \\ A \sin^2\left(\frac{\pi}{2}\eta_2\right) & \text{if } (L_{in} + L_w) < x \leq L_x \end{cases}$$

where $\eta_1 = \frac{x}{L_{in}}$ and $\eta_2 = \frac{L_x - x}{L_{out}}$. The wavenumbers λ_x and λ_z are defined as follows:

$$\lambda_x = \frac{L_w}{4}, \quad \lambda_z = \frac{L_z}{4}.$$

The parameters L_{in} , L_w and L_{out} represent the lengths of each function in the x direction, as illustrated in Figure 4.8. The general parameters of the simulation are resumed in Table 4.2.

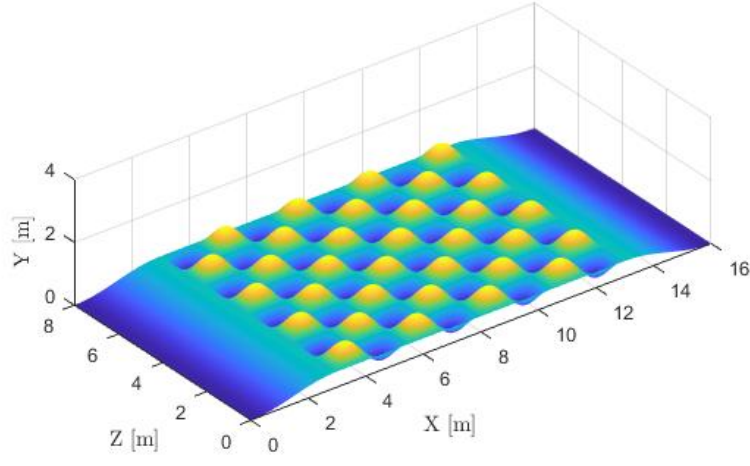


Figure 4.7: Computational domain for the flow past a wavy wall in a half channel.

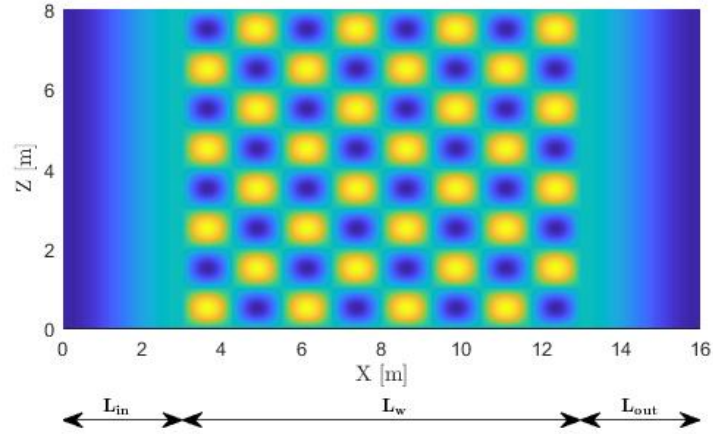


Figure 4.8: Top view of the domain (Oxz plane).

The Reynolds number is based on the free stream velocity U_∞ and twice the amplitude of the waves $2A$:

$$Re_A = \frac{U_\infty(2A)}{\nu}.$$

The simulations have been done at three different viscosity:

$$\nu = 0.01 \text{ [m}^2/\text{s]}, \quad \nu = 0.005 \text{ [m}^2/\text{s]}, \quad \nu = 0.0025 \text{ [m}^2/\text{s]}.$$

Therefore, the three Reynolds numbers of the simulations are:

$$Re_A = 100, \quad Re_A = 200, \quad Re_A = 400.$$

Parameter	Value	Unity
Physical domain $L_x \times L_y \times L_z$	$16 \times 4 \times 8$	$[m^3]$
Box size $N_x \times N_y \times N_z$	$512 \times 128 \times 256$	$[-]$
Time step Δt	10^{-3}	$[s]$
Penalization parameter $\Delta\tau$ (implicit)	10^{-6}	$[s]$
Input S-shape function length L_{in}	3	$[m]$
Wavy function length L_w	10	$[m]$
Output S-shape function length L_{out}	3	$[m]$
Waves amplitude A	0.5	$[m]$
Streamwise wavelength $\lambda_x = \frac{L_w}{4}$	2.5	$[m]$
Spanwise wavelength $\lambda_z = \frac{L_z}{4}$	2	$[m]$
Free stream velocity U_∞	1	$[m/s]$

Table 4.2: Setup for the simulation of the wavy wall.

For the boundary conditions, the following constraints have been imposed on the boudaries:

- X-direction
 - An uniform free stream velocity

$$\mathbf{u}_\infty = (U_\infty, 0, 0)$$

is imposed at the inlet.

- A convective outflow condition

$$\frac{\partial \mathbf{u}}{\partial t} + U_m \frac{\partial \mathbf{u}}{\partial \mathbf{n}} = 0$$

is imposed at the outlet, where U_m is taken from the average value of the normal velocity at the outflow boundary.

- Y-direction

- A no-slip condition is imposed at the bottom:

$$u = 0 \quad v = 0 \quad w = 0.$$

- A free slip condition is imposed on the top:

$$\frac{\partial u}{\partial y} = 0 \quad v = 0 \quad \frac{\partial w}{\partial y} = 0.$$

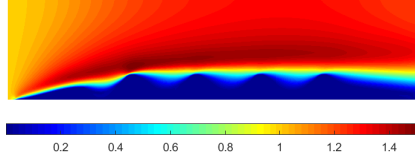
- Z-direction

- A free slip condition is imposed on both sides:

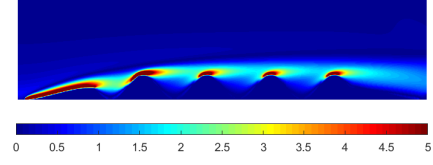
$$\frac{\partial u}{\partial z} = 0 \quad \frac{\partial v}{\partial z} = 0 \quad w = 0.$$

4.2.2 Results

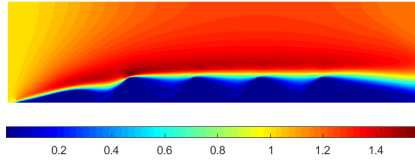
The Figures 4.9 and 4.10 are snapshots of the normalized velocity and vorticity fields taken at different planes and the Figure 4.11 represent the vortical structures detected using the λ_2 criterion. From $Re_A = 100$ to $Re_A = 200$, no particular differences can be notified. On the other hand, from $Re_A = 200$ to $Re_A = 400$, the flow becomes more unsteady. Also, when analyzing the snapshots of Figures 4.9 (a), (d) and (f) (vorticity field in the Oxy plane), we can notice that the boundary layers over the bulbs merge in a longer boundary layer for $Re_A = 400$.



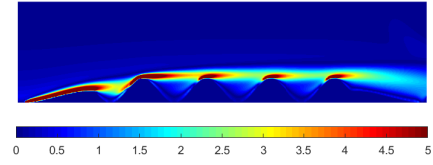
(a) $\frac{\|\mathbf{u}\|}{U_\infty}$ at $Re_A = 100$



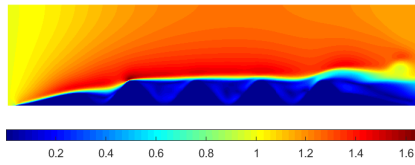
(b) $\frac{\|\boldsymbol{\omega}\|^{(2A)}}{U_\infty}$ at $Re_A = 100$



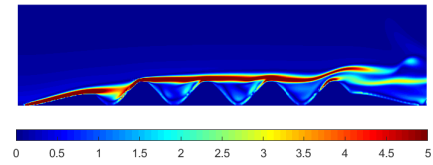
(c) $\frac{\|\mathbf{u}\|}{U_\infty}$ at $Re_A = 200$



(d) $\frac{\|\boldsymbol{\omega}\|^{(2A)}}{U_\infty}$ at $Re_A = 200$

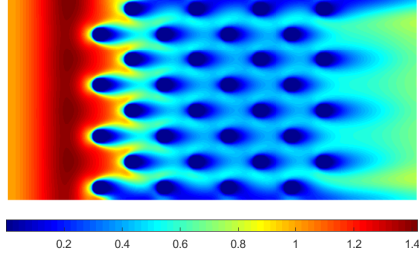


(e) $\frac{\|\mathbf{u}\|}{U_\infty}$ at $Re_A = 400$

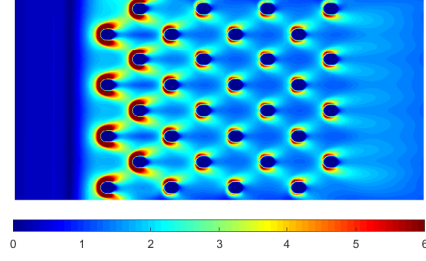


(f) $\frac{\|\boldsymbol{\omega}\|^{(2A)}}{U_\infty}$ at $Re_A = 400$

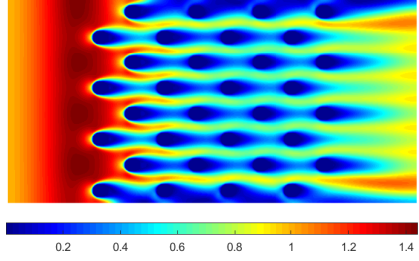
Figure 4.9: Snapshots of the magnitude of the velocity field (left) and the vorticity field (right) in the Oxy plane at $\frac{z}{2A} = 3.5$. Comparison at different Reynolds numbers.



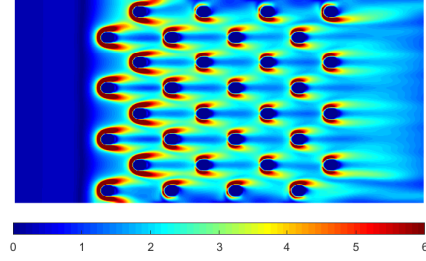
(a) $\frac{\|\mathbf{u}\|}{U_\infty}$ at $Re_A = 100$



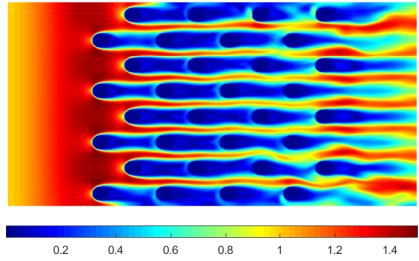
(b) $\frac{\|\boldsymbol{\omega}\|(2A)}{U_\infty}$ at $Re_A = 100$



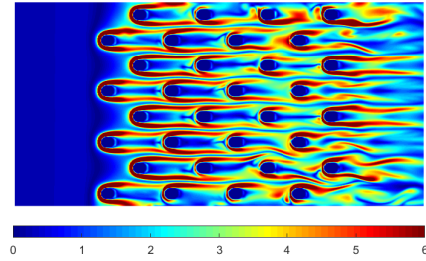
(c) $\frac{\|\mathbf{u}\|}{U_\infty}$ at $Re_A = 200$



(d) $\frac{\|\boldsymbol{\omega}\|(2A)}{U_\infty}$ at $Re_A = 200$

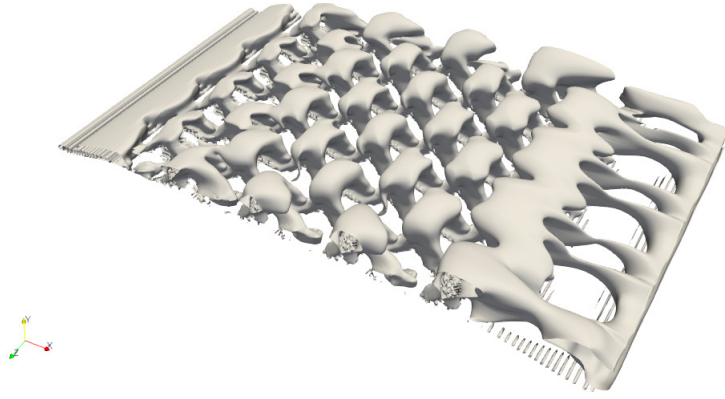


(e) $\frac{\|\mathbf{u}\|}{U_\infty}$ at $Re_A = 400$

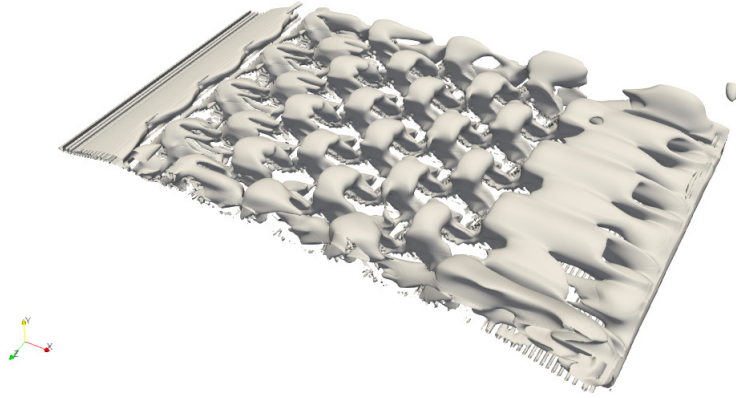


(f) $\frac{\|\boldsymbol{\omega}\|(2A)}{U_\infty}$ at $Re_A = 400$

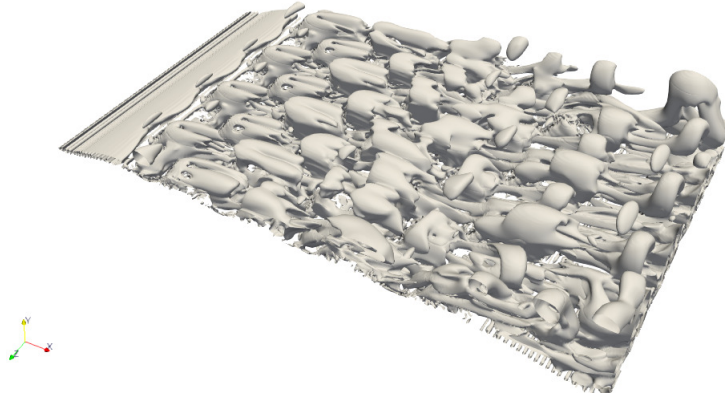
Figure 4.10: Snapshots of the magnitude of the velocity field (left) and the vorticity field (right) in the Oxz plane at $\frac{y}{2A} = 0.85$. Comparison at different Reynolds numbers.



(a) $Re_A = 100$



(b) $Re_A = 200$



(c) $Re_A = 400$

Figure 4.11: Vortical structures detected using the λ_2 criterion for the wavy wall. Comparison at different Reynolds number.

4.3 Sedimentation of spheroidal bodies

In this study, we will investigate the sedimentation of spheroidal bodies. A spheroid is a particular case of an ellipsoid for which 2 semi axis are equals. An example of two different spheroids are shown in Figure 4.12. By defining the aspect ratio to be $AR = c/a$, two different types of spheroid are defined:

- when $AR < 1$, the spheroid is called an *oblate* (left in Figure 4.12),
- when $AR > 1$, the spheroid is called a *prolate* (right in Figure 4.12).

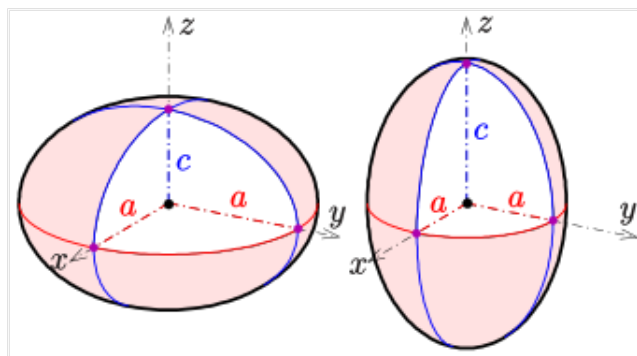


Figure 4.12: Two different spheroids: if $c < a$ oblate (left), if $c > a$ prolate (right).

The spheroids studied will have aspect ratios of $1/3$ and 3 . For this problem, three non-dimensional parameters are important to take into account: the Galileo number Ga , the aspect ratio AR and the relative density ρ_r . The Galileo number Ga is the ratio between gravitational and viscous forces, and is defined as

$$Ga = \sqrt{\frac{|\rho_r - 1|gD_{eq}}{\nu^2}} \quad (4.8)$$

where D_{eq} is the equivalent diameter of a sphere with the same volume as the ellipsoidal body and is computed as follows:

$$D_{eq} = 2a(AR)^{\frac{1}{3}}. \quad (4.9)$$

4.3.1 Numerical setup

The computational domain for the simulations is shown in Figure 4.13. A domain size of $2.3 \times 2.3 \times 9.2$ [m³] has been chosen with a resolution of $192 \times 192 \times 798$. The Galileo number will be equal to 180 for both spheroids, with a relative density $\rho_r = 10$. The dimensions of the oblate and the prolate are:

$$\textbf{Oblate} : \begin{pmatrix} a & b & c \end{pmatrix} = \begin{pmatrix} 0.3 & 0.3 & 0.1 \end{pmatrix} \text{ [m]}.$$

$$\textbf{Prolate} : \begin{pmatrix} a & b & c \end{pmatrix} = \begin{pmatrix} 0.1 & 0.1 & 0.3 \end{pmatrix} \text{ [m]}.$$

The size of the domain and the dimensions of the spheroids have been chosen to have a minimum of 8 grid points per semi-minor axis. In terms of grid points per equivalent diameter D_{eq} , we have for the oblate $D_{eq}/h \approx 35$ and for the prolate $D_{eq}/h \approx 24$. The resolution is higher than what is typically used for spherical bodies to keep an adequate number of grid points per semi-minor axis of the spheroid, which decreases with the aspect ratio [13].

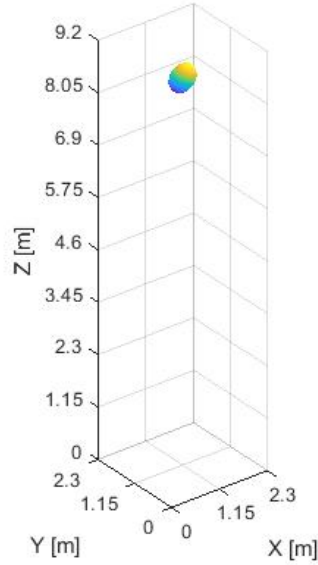


Figure 4.13: Computational domain of the sedimentation of spheroids.

To keep the same Galileo number of 180 for each spheroid, since the relative density and the equivalent diameter are fixed, the viscosity has to be adapted for both cases according to Eq.(4.8):

$$\nu_{obl} \approx 0.03367 \text{ [m}^2/\text{s]}, \quad \nu_{pro} \approx 0.02804 \text{ [m}^2/\text{s]}.$$

Periodic boundary conditions are imposed in the horizontal directions whereas a free slip and a rigid wall are used at the upper and bottom boundary:

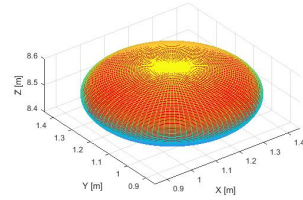
- X-direction
 - A periodic boundary condition is imposed on both sides.
- Y-direction
 - A periodic boundary condition is imposed on both sides.
- Z-direction
 - At the bottom, a solid wall boundary condition is imposed:

$$u = 0 \qquad v = 0 \qquad w = 0.$$

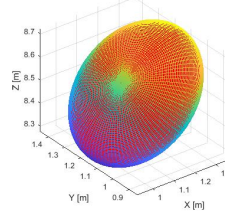
- At the upper, a free slip boundary condition is imposed:

$$\frac{\partial u}{\partial z} = 0 \qquad \frac{\partial v}{\partial z} = 0 \qquad w = 0.$$

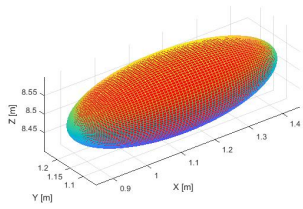
A number of 4 simulations have been done, 2 simulations per spheroid with different orientations around the x -axis, as illustrated in Figure 4.14.



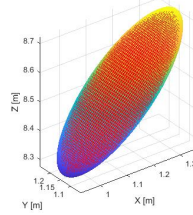
(a) Oblate 0°



(b) Oblate 45°



(c) Prolate 0°



(d) Prolate 45°

Figure 4.14: Initial orientations for the oblate and the prolate before falling.

4.3.2 Results

An interesting phenomenon measured for all simulations was the tendency of the spheroids to have their semi-major axis parallel to the direction of the gravity. This is illustrated in Figure 4.15, where the angle ϕ represents the roll angle of the Euler angles. Indeed, the initial configurations of all cases represented in Figure 4.14 are not on their configuration which minimize the energy. The configuration which minimize the energy is the configuration where the hydrodynamic torque applied to the spheroid is minimal, and this apply when the direction of the semi-minor axis is perpendicular to the flow.

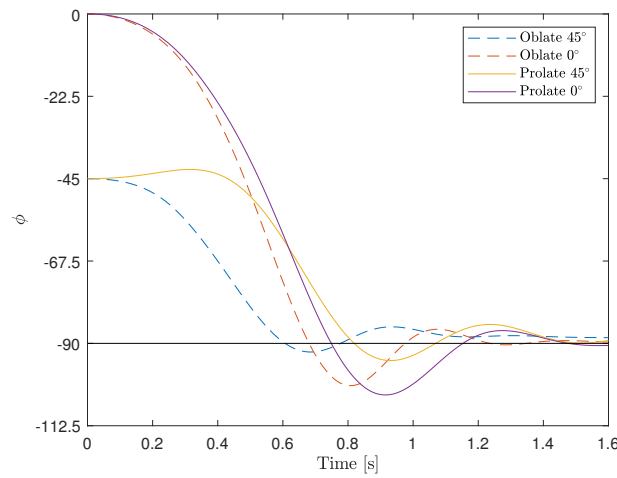
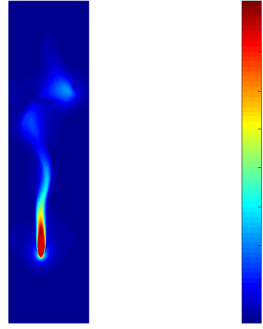


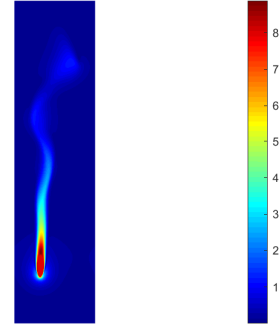
Figure 4.15: Time-evolution of the roll angle described with the Euler angles for all cases. Dashed lines: oblate, solid lines: prolate.

The Figures 4.16 and 4.17 are snapshots of $\frac{\|\mathbf{u}\|D_{eq}}{\nu}$ and $\frac{\|\boldsymbol{\omega}\|D_{eq}^2}{\nu}$ in the plane Oxz at $\frac{y}{L_y} = \frac{1}{2}$. More vortex shedding are present for the oblate. This can be explained by the aerodynamics of the shapes, which are different. The prolate has a more streamlined profile than the oblate for these aspect ratios.

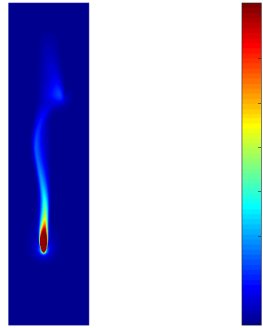
In Figure 4.18 and 4.19, the vorticity structures for the 4 cases are represented with the λ_2 criterion at 4 different times. Here again, the more aerodynamic nature of the prolate is highlighted. Also, the oblate vorticity structures have a more symmetrical behaviour than the prolate.



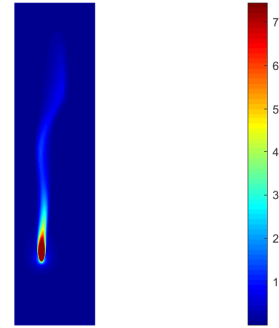
(a) Oblate at initial angle $\phi = 0^\circ$



(b) Oblate at initial angle $\phi = 45^\circ$

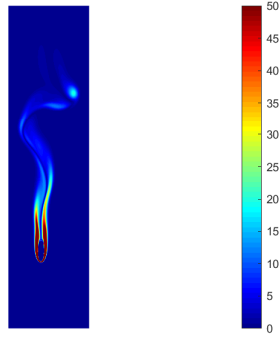


(c) Prolate at initial angle $\phi = 0^\circ$

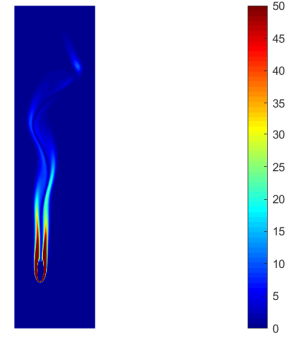


(d) Prolate at initial angle $\phi = 45^\circ$

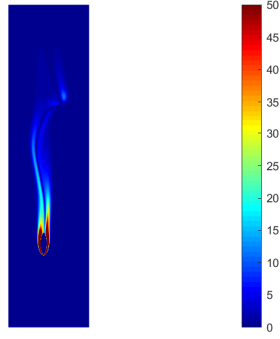
Figure 4.16: Snapshot of the contour of $\frac{||u||D_{eq}}{\nu}$ in the plane Oxz at $\frac{y}{L_y} = \frac{1}{2}$.



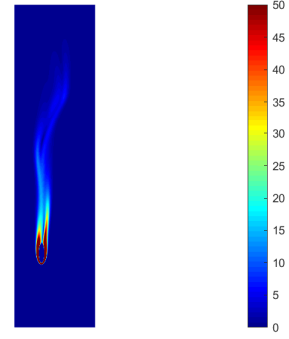
(a) Oblate at initial angle $\phi = 0^\circ$



(b) Oblate at initial angle $\phi = 45^\circ$

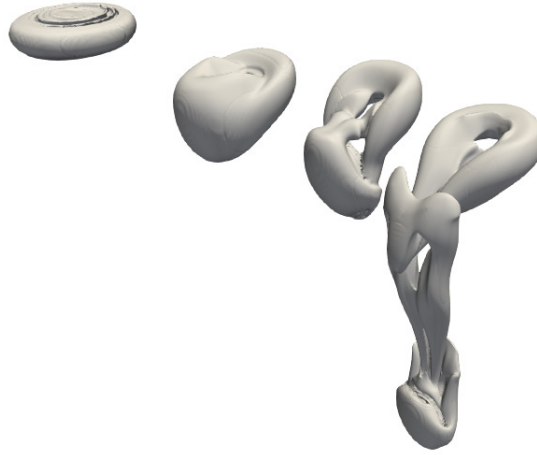


(c) Prolate at initial angle $\phi = 0^\circ$

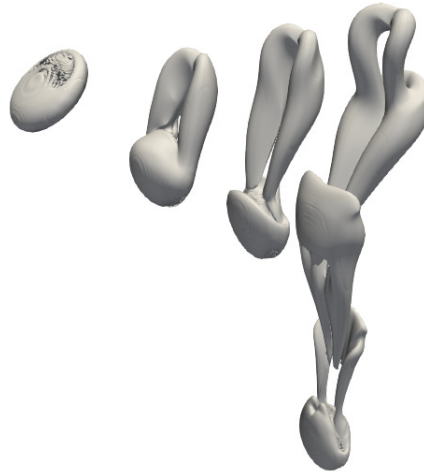


(d) Prolate at initial angle $\phi = 45^\circ$

Figure 4.17: Snapshot of the contour of $\frac{||\omega|| D_{eq}^2}{\nu}$ in the plane Oxz at $\frac{y}{L_y} = \frac{1}{2}$.

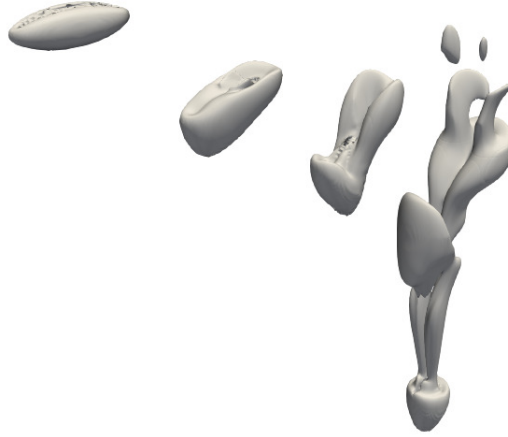


(a) Initial orientation of $\phi = 0^\circ$

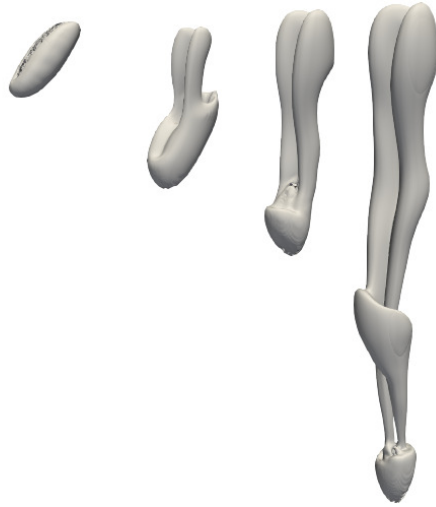


(b) Initial orientation of $\phi = 45^\circ$

Figure 4.18: Vortical structures detected using the λ_2 criterion for the sedimentation of the oblate. Comparison with different initial orientations.



(a) Initial orientation of $\phi = 0^\circ$



(b) Initial orientation of $\phi = 45^\circ$

Figure 4.19: Vortical structures detected using the λ_2 criterion for the sedimentation of the prolate. Comparison with different initial orientations.

Chapter 5

Conclusion

As a result of this thesis, the implementation of a penalization method in incompressible flows has been successfully done in the context of Direct Numerical Simulation (DNS) for both fixed and moving bodies. These contributions have been done with the ambition to extend the features of the solver BigFlow by adding obstacles in the flow.

In a first step, the different governing equations needed by the solver BigFlow to model fluid-structure interactions have been detailed in chapter one. Their temporal and spatial discretizations have been coupled in chapter two.

The second step was devoted to validate the method with classical benchmarks used in literature.

- For the fixed case, the flow past a sphere at $Re = 300$ and the flow past a cylinder at $Re = 200$ have been realized. To produce accurate results, a stretched mesh has been implemented for both cases. Results showed that the flow past a sphere was more accurate than the flow past a cylinder. The conclusion made was that shrinking a 3D domain to a 2D domain for the cylinder (with periodic boundary conditions in the spanwise direction in order to compare with 2D results in literature) creates confinement of the flow, which leads to less accurate results.
- For the moving case, the sedimentation experiment of [1] has been reproduced. The comparison of the terminal velocity in each fluid with the experiments has been done with a lot of success.

The final step was reserved to application cases. The choices of these applications have been done in a way to see beautiful flows and vorticity structures. For that, the three following topics were treated:

1. Flow past a cube at $Re = 100$ with different orientations. This application showed that the force coefficients are strongly sensible compared to the cube orientation. Also, a definition of the projected surface has to be clearly defined in order to use correctly the lift and drag coefficients. For the velocity and vorticity fields, the contours show a steady behaviour of the flow whatever the orientation.
2. Flow past a wavy wall. In this application, three simulations have been performed at different Reynolds number. The observations showed a transition regime between $Re_A = 200$ and $Re_A = 400$. The goal of this application is, maybe for a future thesis, to add a wavy wall in a wind farm or a single wind turbine. This application was a first investigation.
3. Sedimentation of spheroidal bodies. After have implemented the sedimentation of a sphere with success in the validation analysis, testing the solver for other aspect ratios than the sphere was important. Results showed that for all situations simulated, the spheroids were falling with their semi-major axis parallel to the direction of the flow.

For the future, extend the penalization method implemented in BigFlow to handle deformable bodies can be a great challenging topic. Also, since all the simulations have been done in DNS, the implementation of the method in the context of Large Eddy Simulations (LES) can be another challenge to explore, e.g. add the wavy wall in a turbulent channel flow.

Bibliography

- [1] A. Ten Cate, C. Nieuwstad, J. J. Derksen, and H. Van den Akker, “Particle imaging velocimetry experiments and lattice-Boltzmann simulations on a single sphere settling under gravity,” *Physics of Fluids*, 2002.
- [2] J. F. Thompson, Z. U. A. Warsi, and C. W. Mastin, “Boundary-fitted coordinate systems for numerical solution of partial differential equations—a review,” *Journal of Computational Physics*, vol. 47, pp. 1–108, 1982.
- [3] H. Bandringa, *Immersed Boundary Method*. PhD thesis, University of Groningen, 2010.
- [4] P. Angot, C.-H. Bruneau, and P. Fabrie, “A penalization method to take into account obstacles in incompressible viscous flows,” *Numer. Math*, 1999.
- [5] T. Engels, *Numerical Modeling of Fluid-Structure Interaction in Bio-Inspired Propulsion*. PhD thesis, Université d’Aix-Marseille, 2015.
- [6] M. Uhlmann, “An immersed boundary method with direct forcing for the simulation of particulate flows,” *Journal of Computational Physics*, 2005.
- [7] O. V. Vasilyev, “High order finite difference schemes on non-uniform meshes with good conservation properties,” *Journal of Computational Physics*, vol. 157, p. 746–761, 1999.
- [8] F. H. Harlow and J. E. Wech, “Numerical calculation of time-dependant viscous incompressible flow of fluid with free surface,” *Physics of Fluids*, vol. 8, p. 2182–2189, 1965.
- [9] M. J. Lee, B. Oh, and Y. B. Kim, “Canonical fractional-step methods and consistent boundary conditions for the incompressible navier–stokes equations,” *Journal of Computational Physics*, vol. 168, p. 73–100, 2001.
- [10] M. Duponcheel, *Direct and large-eddy simulation of turbulent wall-bounded flows : further development of a parallel solver, improvement of multiscale*

subgrid models and investigation of vortex pairs in ground effect. PhD thesis, Université Catholique de Louvain, October 2009.

- [11] C. Mimeau, *Conception and implementation of a hybrid vortex penalization method for solid-fluid-porous media : application to the passive control of incompressible flows*. Theses, Université Grenoble Alpes, July 2015.
- [12] F. Abraham, “Functional Dependence of Drag Coefficient of a Sphere on Reynolds Number,” *Physics of Fluids*, 1970.
- [13] M. N. Ardekani, P. Costa, W. P. Breugem, and L. Brandt, “Numerical study of the sedimentation of spheroidal particles,” *International Journal of Multiphase Flow*, vol. 87, pp. 16–34, 2016.

UNIVERSITÉ CATHOLIQUE DE LOUVAIN
École polytechnique de Louvain

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