

École polytechnique de Louvain

Implementation and validation of symmetry and no-slip boundary conditions in a 3-D vortex particle- mesh method

Authors : **Clément Lepot**
Supervisors : **Grégoire Winckelmans**
Readers : **Philippe Chatelain, Vincent Legat**
Academic year 2018–2019
Master [120] in Electro-mechanical Engineering

Abstract

This master thesis presents the implementation and validation of a symmetry and no-slip boundary conditions in the 3-D vortex particle-mesh (VPM) method.

The first part focuses on the implementation of the no-through flow using a symmetry boundary condition. The implementation allows the diffusion through the wall, which makes it possible to study the vortex-ring reconnection at low Reynolds number ($Re_\Gamma = 2000$).

The second part presents how the no-slip boundary condition is implemented in the vortex particle-mesh method, which is a vorticity-velocity formulation flow solver. It presents how the required vortex sheet at the wall is computed to nullify the slip velocity.

The third part is the validation of the method using two well-known results: the Poiseuille flow in a channel and the Blasius boundary layer development. The method provides almost an exact solution for the Poiseuille flow. The boundary layer develops in a channel with a slip and a no-slip wall. The flow above the boundary layer is thus accelerated but the numerical results closely match the analytical ones.

The last part is dedicated to test cases. Two main cases are studied

The first one is the interaction between a vortex-ring and a no-slip wall with different initial angles relative to the ring axis. When the vortex-ring approaches the no-slip wall, a boundary layer develops. After some time, the boundary layer separation occurs and the vorticity wraps the vortex ring. It generates a secondary vortex-ring that impacts the dynamic of the primary one. Omega-loops instabilities are generated, which causes the destruction of the vortex-ring core.

The second is a turbulent channel flow at low Reynolds number. The results are compared to the ones obtained by Moser and Lee. They match each other very closely. Some differences are observed at the middle of the channel, due to the shorter length of the computational domain used with VPM than the one used by Moser and Lee.

Finally, an appendix shows nice results for a turbulent boundary layer development. The results are a little bit under resolved but the thicknesses, the Reynolds numbers and some velocity profiles are shown and discussed.

Acknowledgements

First of all, nothing would have been possible without the help of the partners as the UCL, the CECI, the FNRS and the ULG. All the simulations were performed thanks to the computational resources provided by the supercomputing facilities of the Université Catholique de Louvain (UCL) and the Consortium des Equipements de Calcul Intensif en Fédération Wallonie Bruxelles (CECI) funded by the Fond de la Recherche Scientifique de Belgique (F.R.S.-FNRS). The simulations were mainly performed on Lemaitre3 from UCL and NIC4 from the Université de Liège (ULG).

I would like to thank firstly my UCL promotor, Prof. Grégoire Winckelmans for proposing me this subject for my thesis and for taking the time to advise me in the realisation. Thank you for your interest in my work. It was very rewarding to know that some of my results were shown and appreciated by some members of the scientific community. I'm lucky to have benefit from your advice, your criticism and your expertise.

I would like to thank in particular Denis-Gabriel Caprace for always being present and available for me, from the start to the end. Thank you for the reading and listening of my results.

Finally, I would like to thank my parents for reading and correcting the English wording. I thank them for reading 80 pages without understanding anything, I hope it will give them a taste for fluid mechanics.

Contents

1	Foreword	8
1.1	Methodology	8
1.1.1	Navier-Stokes equations	8
1.1.2	Vortex particle method	8
1.1.3	Vortex particle-mesh method	9
2	Vortex-ring reconnection	10
2.1	Introduction	10
2.2	The vortex-ring	10
2.3	Vortex-rings reconnection	12
2.3.1	Initial condition and numerical setup	12
2.3.2	Results and discussions	13
2.4	Conclusion	18
3	No-slip boundary condition implementation	19
3.1	Introduction	19
3.2	Expression of boundary conditions	19
3.2.1	3D Vorticity field definition	19
3.2.2	Boundary conditions	20
3.2.3	Decentered finite difference of fourth order	20
3.2.4	Numerical expression of boundary conditions	21
3.3	Implementation in VPM method	21
3.3.1	General method	21
3.3.2	First step: No-through-flow	21
3.3.3	Second step: Imposing the boundary condition on vorticity	21
3.3.4	Third step: right hand side of the Navier-Stokes equation	22
3.3.5	Fourth step: time update of particle location and strength	23
3.4	Conclusion	23
4	Validation	24
4.1	Introduction	24
4.2	One dimensional flow	24
4.2.1	Analytical solution	24
4.2.2	Numerical setup	25
4.2.3	Numerical results - Velocity	26
4.2.4	Results - Vorticity	27
4.2.5	Pressure gradient evolution	27
4.2.6	Conclusion	28
4.3	Two-dimensional flow	28
4.3.1	Introduction	28
4.3.2	Numerical setup	28
4.3.3	Numerical results - Limitations	29
4.3.4	Numerical results - Validations - Velocity	31
4.3.5	Numerical results - Validations - Vorticity	32
4.3.6	Wall vorticity flux	33
4.3.7	Numerical results - Validations - Boundary layer thicknesses	35
4.3.8	Isotropic mesh discussion	38
4.3.9	Conclusion	38

5	Test-cases	40
5.1	Interaction between a vortex-ring and a no-slip wall perpendicular to the ring axis	40
5.1.1	Numerical setup	40
5.1.2	Mesh Reynolds number	40
5.1.3	Numerical results and discussions	41
5.1.4	Conclusion	46
5.2	Interaction between a vortex-ring and a no-slip wall with an initial angle relatively to the ring axis	46
5.2.1	Numerical setup	46
5.2.2	Mesh Reynolds number	46
5.2.3	Numerical results and discussion	46
5.2.4	Origins of instabilities discussions	49
5.2.5	Conclusion	51
5.3	Turbulent channel flow at $Re_\tau = 180$	51
5.3.1	Introduction	51
5.3.2	Turbulent channel flow characterisation	51
5.3.3	Numerical setup	52
5.3.4	Numerical results and discussions	53
5.3.5	Conclusion	58
6	Conclusion	59
6.1	A global overview of the work	59
6.2	What's next ?	60
6.3	Final word	60
A	Blasius solution for boundary layer flow	61
A.1	Analytic solution for velocity	61
A.2	Analytic solution for vorticity	63
A.3	Boundary layer thicknesses	64
B	Anisotropic mesh additional results	67
B.1	Anisotropic mesh in laminar boundary layer	67
B.2	Anisotropic mesh in turbulent channel flow	68
C	Ring interaction with the no-slip wall	70
C.1	Initial angle $\alpha = 10^\circ$	70
C.2	Initial angle $\alpha = 30^\circ$	72
C.3	Initial angle $\alpha = 40^\circ$	73
D	Statistics quality diagnostic	76
E	Turbulent boundary layer development	77
E.1	Introduction	77
E.2	Numerical setup	77
E.3	Instantaneous numerical results	78
E.3.1	Resolution of the simulation : mesh Reynolds numbers	78
E.3.2	Three dimensional results for vorticity	83
E.4	Average numerical results	85
E.4.1	Boundary layer thicknesses and Reynolds number	85
E.4.2	Friction coefficient and friction velocity	87
E.4.3	Average velocity profiles	89
E.5	Conclusion	92

Introduction

Global demand for energy is becoming increasingly important. This increase is indeed exponential since 1900. Durable solutions have to be found for the future in order to continue living in the world as we know it today. Since the beginning of the industrial era, most of the energies are provided by fossil fuels. Today, it is still the source of energy that dominates the production. It would be illusive to think that the demand in energy will lower in the coming years

A solution still in progress for the energy problem was born. The past decades have seen the emergence of renewable energy based on solar and wind energy. The most common industrial application of the renewable energy is the wind turbines. But the increase in the number and size of the wind turbines needs accurate predictions of their wakes and performances. How far is the wind turbine's wake harmful for another one? How does the atmospheric turbulent boundary layer impact the efficiency and the wake of the wind turbine? How does the ground and the wake interact together? The TFL¹ group of UCL is working on wake prediction of wind turbines, under others, based on in-house numerical solvers of the fluid dynamic.

In the past decades, the exponential growth of computer performance has lead to the development of several numerical tools to solve the Navier-Stokes equations. Two main numerical tools are currently used: the solvers based on an Eulerian formulation of Navier-Stokes equation (mesh-based approach) and those based on an Lagrangian formulation (particle method). Basically, the two methods can be summarized as follows:

- The particle method based on the Lagrangian formulation of the Navier-Stokes equation uses particles to discretize the Navier-Stokes equations. It allows the use of bigger time step than those used in a mesh based-approach, which makes the simulation calculation faster. In addition, the particle method doesn't suffer from dispersion errors and accurately captures longer distance convection flows than a mesh-based approach. The Lagrangian based flow solver is pretty well suited for the simulations of incompressible, convection-dominated vortical flows but are prone to inaccuracy due to Lagrangian distortion.
- The mesh-based approach is easier to compute and imposes boundary conditions and very well suited to solve the Poisson equation in an efficient way in terms of computational costs.

This master thesis is focused on an hybrid method: the vortex particle-mesh method or VPM method. This flow solver is an in-house three dimensional Navier-Stokes flow solver. It is a good compromise between the two methods. It benefits from the advantages of the two methods at the same time. The VPM code is used to compute the wake of vertical and horizontal wind turbines, blade's flexion, helicopter wake and rotor dynamics and many other applications. The imposed boundary conditions can be unbounded or periodic. This allows a lot of physical simulations. But two boundary conditions are still missing: the no-through flow and no-slip conditions. These boundary conditions are essential in order to simulate flow interaction with a flat ground.

The VPM solver is based on the vorticity-velocity formulation of the Navier-Stokes equations. Therefore, imposing the no-through flow and no-slip boundary conditions isn't as simple as in a velocity-pressure based solver. The boundary conditions must be expressed in terms of vorticity and not in terms of velocity.

This work proposes a simple and efficient way to compute the needed vortex sheet at the wall to nullify the slip velocity. The document is organized around five chapters:

1. The first chapter is a global explanation of the principle behind the vortex particle-mesh method necessary to understand the implementation of the no-through flow and no-slip boundary conditions.

¹Thermodynamics and fluid mechanics

2. The second chapter is the presentation of the symmetry boundary condition used to impose no-through flow. By allowing the diffusion through the wall it is possible to study a fundamental problem in fluid mechanics: the reconnection of two vortex-rings.
3. The third chapter is the development of the method to impose the no-slip boundary condition. It recalls the needed vorticity at the wall to nullify the slip velocity and details how this vortex sheet is computed in the numerical code.
4. The fourth chapter is dedicated to the validation of the method presented at the previous chapter. The validation of the numerical code is done through two well known cases: the one-dimensional Poiseuille flow in a channel and the boundary layer development above a flat plate. The analytical solution is the well known Blasius solution.
5. The last chapter presents two different test-cases of the no-slip implementation. The chapter deals with two main applications: the first is the interaction between a no-slip wall and a vortex ring presented in chapter 2.1. The second is a classical problem in computational fluid dynamics: the turbulent channel flow at low Reynolds number.

The simulation of a turbulent boundary layer development is proposed as illustration at the end of this document.

The conclusion of the thesis is the implementation, validation and application of a no-through flow and a no-slip boundary condition in VPM method through analytical cases and test cases.

Chapter 1

Foreword

The vortex particle-mesh method is an in-house flow solver developed at UCL-TFL. It is based on the vorticity-velocity formulation of Navier-Stokes equation. VPM is a hybrid method. It uses particles to discretize the convection terms and mesh to solve the Poisson equation and computes the stretching and diffusion terms. It allows a high-efficient computation of three-dimensional incompressible, unsteady vortical flows on massively parallel architectures.

The following section develops the VPM method in basic and its implementation. The interested reader can find more information at [1] and [2].

1.1 Methodology

1.1.1 Navier-Stokes equations

The Navier-Stokes equations for an incompressible flow are the following:

$$\nabla \cdot \mathbf{u} = 0 \quad (1.1)$$

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

By taking the curl of equation (1.2), the vorticity-velocity formulation is obtained, in a Lagrangian formulation, it gives:

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

The vorticity is defined as the curl of the velocity, so if the velocity field is recovered from vorticity field through a Poisson-equation:

$$\boldsymbol{\omega} = \nabla \times \mathbf{u} \Rightarrow \nabla^2 \mathbf{u} = -\nabla \times \boldsymbol{\omega} \quad (1.3)$$

And finally, the vorticity field is divergence free: the divergence of the vorticity field must be zero, meaning that the vortex lines always form close curves, from the equation of mass conservation (equation (1.1)):

$$\nabla \cdot (\nabla \times \mathbf{u}) = \nabla \cdot \boldsymbol{\omega} = 0$$

1.1.2 Vortex particle method

The vortex particles method is a Lagrangian method that uses particles to discretize the Navier-Stokes equations. Each particle is characterized by a position $\mathbf{x}_p$, a volume V_p and a strength α_p :

$$\alpha_p = \int_{V_p} \omega dV = \omega V_p$$

It yields to the following vorticity field:

$$\tilde{\boldsymbol{\omega}} = \sum_p \alpha_p \zeta(\mathbf{x} - \mathbf{x}_p)$$

The particles positions and particles strengths are computed using:

$$\frac{d\mathbf{x}_p}{dt} = \mathbf{u}_p \quad (1.4)$$

$$\frac{d\alpha_p}{dt} = \int_V ((\boldsymbol{\omega} \cdot \nabla) + \nu \nabla^2 \boldsymbol{\omega}) \quad (1.5)$$

$$\simeq ((\boldsymbol{\omega} \cdot \nabla) \mathbf{u}_p + \nu \nabla^2 \boldsymbol{\omega}) V_p \quad (1.6)$$

The vortex method, and more generally the Lagrangian method, offers advantages in convection-dominate problems because the vorticity field has a more compact support than velocity. As it avoid to discretize the convection part of Navier-Stokes equations on a grid, the vortex-method allows the use of greater time-step while the Eulerian-based methods are constrained by the CFL condition.

The first drawback of the vortex method is that, after few time step, the particles are concentrated at some point of the computational domain and some zones are completely depleted of particles and thus information. The most common solution is the *remeshing* procedure. The remesh operation consists of rearranging the information carried by particles on regular position in the computational domain. This is done using a high order interpolation scheme from the actual positions of particles to the new positions.

The second drawback of vortex method is the resolution of the Poisson equation to get the velocity. It basically uses Green function solution were the Eulerian method can benefit from the underlying mesh to use fast Poisson solver.

The third drawback of vortex method is the difficulty to discretize and compute the differential operator of equation (1.6). On a mesh, the differential operator can easily be computed using classical finite differences.

And finally, the last drawback of Lagrangian method is that it does not keep the zero divergence of the vorticity field. So, it is necessary to correct the vorticity field after some time steps.

1.1.3 Vortex particle-mesh method

The vortex particle-mesh method is born by combining the advantages of both methods: particle method and mesh-based approach. Basically, the convection is computed using particles, the Poisson equation is solved on a grid, the diffusion and stretching term are also computed on a grid. The method benefits from large time step due to the computation of convection using particle method, fast resolution of Poisson-equation on a grid and easy computation of differential operators.

The VPM method always passes the information from the particles to the mesh and vice versa through the operation of *particle to mesh* and *mesh to particle*. The interpolation from particles information (or from the mesh) to the mesh (or to the particle) is done through high order interpolation. After the initiation of particle intensity, the sub-steps for one time step are basically the following:

1. Interpolation of particle information to the grid points (particle to mesh step: *P2M*).
2. Get the velocity from the vorticity on the mesh through the resolution of equation (1.3).
3. Evaluation of the right hand side (RHS) of equation 1.6 on the mesh using finite differences and velocity computed at the previous step.
4. Interpolation of the RHS back on particle (mesh to particle step: *M2P*).
5. Time integration of equation (1.4) and (1.6) using a Rung-Kutta 3 scheme.

Finally, as explained above, the convection using particle has the drawback of not keeping the divergence-free condition on vorticity. The solution is obtained by the reprojection step. At each $n_{reprojection}$ time steps, the vorticity field is corrected to re-obtain zero divergence, which is done through the resolution of a Poisson equation.

Chapter 2

Vortex-ring reconnection

2.1 Introduction

The first part of the implementation of a no-slip boundary condition is to implement a no-through flow boundary condition. This chapter is focused on how this condition is implemented in the vortex particle-mesh method. To achieve this goal, the Poisson solver used in this work imposes a symmetric domain with the no-through flow wall as symmetric plane. The constitutive equations are:

$$\begin{aligned}\nabla^2 \mathbf{u} &= -\nabla \times \boldsymbol{\omega} \\ \mathbf{u} \cdot \hat{\mathbf{n}} &= 0\end{aligned}$$

In order to impose the boundary condition, odd extension on the parallel components of vorticity has to be imposed: ω_x and ω_z , and even extension on the normal component of vorticity: ω_y :

$$\begin{aligned}\omega_x(x, -y, z) &= -\omega_x(x, y, z) \\ \omega_z(x, -y, z) &= -\omega_z(x, y, z) \\ \omega_y(x, y, z) &= \omega_y(x, y, z)\end{aligned}$$

This extends the vorticity field like this:

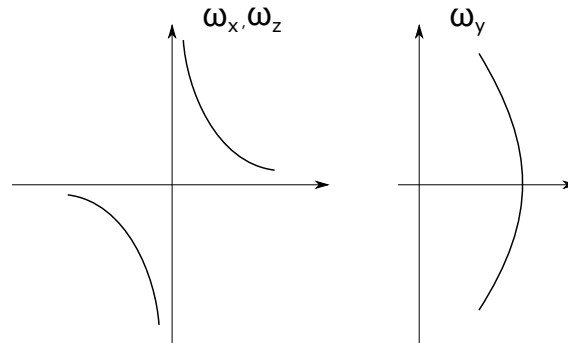


Figure 2.1: Extended vorticity field

The no-through flow boundary condition computed like this cancels the normal velocity but keeps diffusion at the wall. It is useful to study a fundamental problem in fluid mechanics: the vortex-ring reconnection.

This first part studies the vortex-ring reconnection with forced symmetry.

2.2 The vortex-ring

A vortex-ring is a vorticity tube shaped like a ring. It presents a vorticity profile axisymmetric around its central line. The vortex-ring under study is a toroidal vortex-ring because the central line is a circle. Due to the induced velocity of the ring, it is displacing perpendicular to the plane

formed by the center line.

Several configurations are possible for the axisymmetric profile: Gaussian and Gaussian compact. The previous master thesis studied the isolated vortex-ring of different axisymmetric distribution [3]. The profile used in this work is a compact Gaussian as defined in [3]. The profile is Gaussian inside the ring but the vorticity is set to zero past a certain distance of the center line, this distance being defined as the tube's radius. Mathematically it gives:

$$\frac{\pi\omega_\theta\sigma^2}{\Gamma} = \xi(\rho) = \begin{cases} C(\beta^2) \exp\left(-\frac{\rho^2}{\beta^2(1-\rho^2)}\right) & \text{if } \rho \leq 1 \\ 0 & \text{otherwise} \end{cases}$$

With $\rho = s/R_c$, $s^2 = z^2 + (r - R)^2$, r is the distance between the center line and a point in the domain, R is the center line radius and R_c is the tube radius. The other components of vorticity: ω_r and ω_z equal zero because the vorticity is only azimuthal. The parameters β is the dimensionless standard deviation of the Gaussian profile: $\beta = \sigma/R_c$. The constant $C(\beta^2)$ is chosen so that the circulation of the ring equals Γ :

$$\begin{aligned} \int_S \omega_\theta(s) dS &= \Gamma \\ \int_0^{2\pi} \int_0^{R_c} \omega_\theta(s) s ds d\theta &= \Gamma \\ 2\pi \int_0^{R_c} \omega_\theta(s) s ds &= \Gamma \end{aligned}$$

By introducing the definition of $\omega_\theta(s)$:

$$\begin{aligned} 2\pi \int_0^{R_c} \omega_\theta\left(\frac{s}{\sigma}\right) \frac{s}{\sigma} d\left(\frac{s}{\sigma}\right) ds &= \Gamma \\ 2 \int_0^1 \xi(\rho) \rho d\rho &= 1 \end{aligned}$$

Finally, it gives:

$$C(\beta^2) = \frac{1}{2 \int_0^1 \exp\left(-\frac{\rho^2}{\beta^2(1-\rho^2)}\right) \rho d\rho}$$

In this chapter $\Gamma = 1$, $R_c/R = 1/2$, $\beta = 1/4$ and it gives $C(\beta^2) = 1.11866$. The dimensionless profile of the ring is:

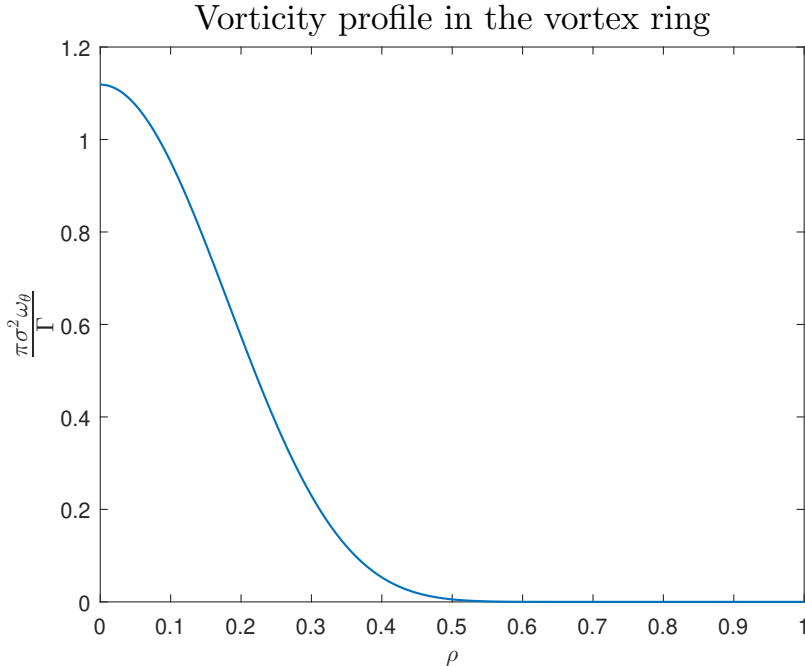


Figure 2.2: Vorticity profile around the center line

2.3 Vortex-rings reconnection

2.3.1 Initial condition and numerical setup

In this section, the no-through flow solver with diffusion through the wall is used to study the reconnection of two vortex-rings. One vortex-ring is placed at the origin of the computational domain. The normal vector of the ring is in the $\hat{e}_z$ direction but it is tilted of an angle of 15° so it approaches the no-through flow plane. By forced symmetry another ring approaches the $x - z$ plane by the other side. After few seconds, the two vortex-rings interact together. The boundary conditions are unbounded and no-through flow:

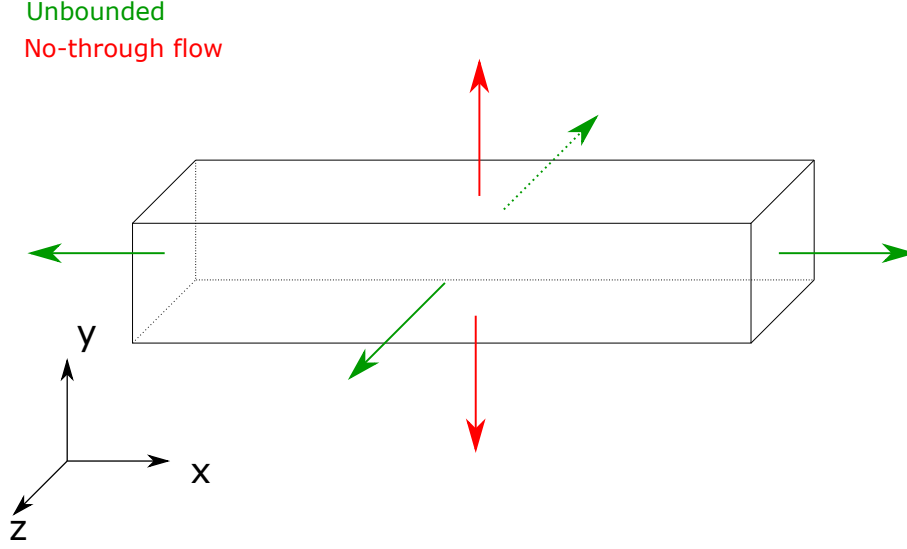


Figure 2.3: Boundary condition ring-reconnection

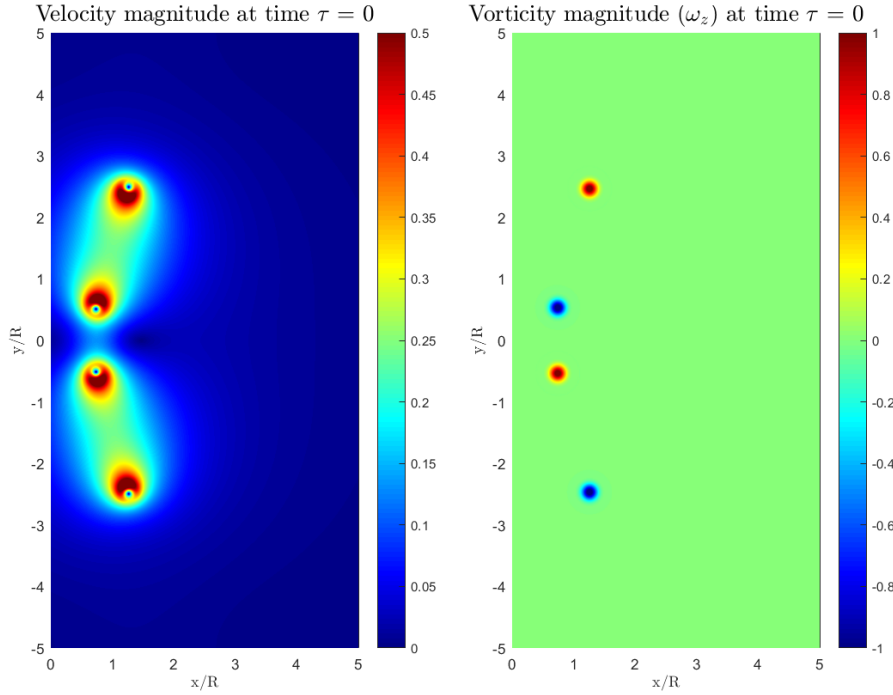


Figure 2.4: Initial condition for the vorticity and velocity field

The vortex-ring located at $y < 0$ is not in the computational domain but the Poisson solver supposes that this vortex-ring exists and interacts with the one above. The parameters of the simulation are the following:

Parameter:	Value
$N_x \times N_y \times N_z$	$512 \times 512 \times 512$
$L_x/R \times L_y/R \times L_z/R$	$5.0 \times 5.0 \times 5.0$
$t_{ref} = \frac{R^2}{\Gamma}$	1 [sec]
Time step	Adaptive and max. value fixed at $\Delta\tau = 0.01$
Dimensionless final time	10 [-]
Reynolds based on circulation: $Re_\Gamma = \frac{\Gamma}{\nu}$	2000 [-]
Remesh and reprojection	Every 5 and 20 time step

Table 2.1: Numerical setup of the ring reconnection

The simulation runs on $4 \times 4 \times 4$ processors during 10 hours of physical time.

2.3.2 Results and discussions

In order to check the resolution of the numerical solution, the mesh Reynolds number based on vorticity is computed. The definition of these quantity is:

$$Re_\omega = \frac{(|\omega|)h^2}{\nu} \quad (2.1)$$

The values are monitored at each time step, its evolution is the following:

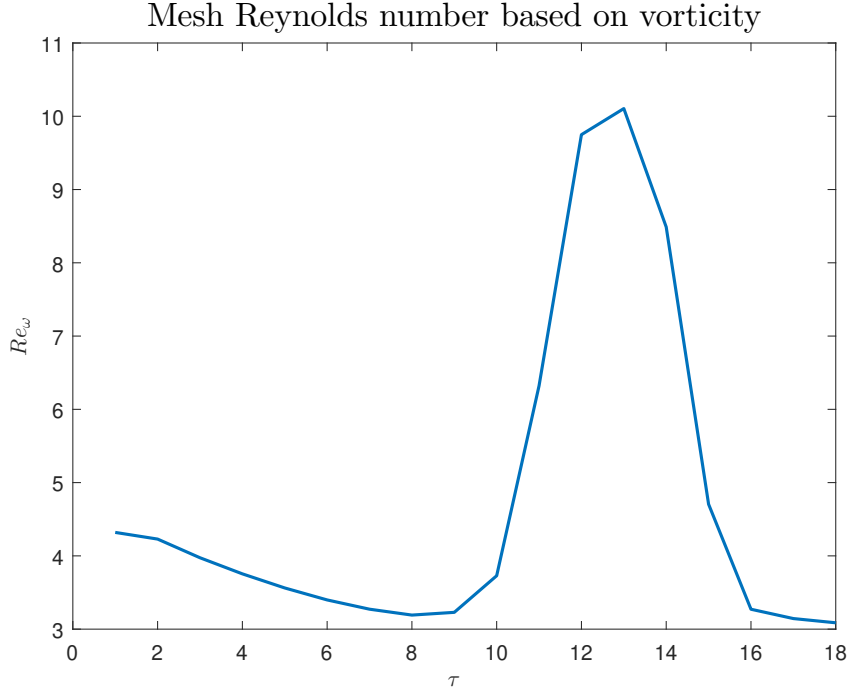


Figure 2.5: Time evolution of mesh Reynolds number based on vorticity for the simulation

The spike is when the two vortex-rings reconnect, the two vortex rings flatten one on the other so vorticity becomes locally high. These values are still low enough and all scales will be well captured by the numerical simulation.

The two dimensional slices of the reconnection are presented below at figure 2.7 and 2.8. The three dimensions results are also provided at figure 2.9.

At the beginning of the simulation, the vortex-rings approach the symmetry plane by their induced velocities. At $\tau \simeq 5$ the vortex-rings get in touch together and the tubes flatten one on the other to form an elliptical shape as seen at figure 2.7b. The diffusion through the wall allow the reconnection of the tubes, but this reconnection isn't complete, some threads connect the reconnected part together, figure 2.8d. These threads undergo successive reconnections and produce small scales of vorticity. At the end, the two vortex rings only form one vortex ring bigger then initially. If the simulation had run longer, the upper and lower part of the big vortex

ring would have moved closer and then touched to form two smaller vortex-rings. This step is called the fission step, and it is shown in the Schatzle experience [4].

The total circulations in the z -direction (plane of the figures 2.7) and in the plane of symmetry (plane of the figure 2.8) are computed in absolute value:

$$\Gamma_{tot} = \int_S |\omega| dS$$

The results are plotted at the figure 2.6. The different phases are clearly visible:

1. The step *a*) corresponds to non-viscous advection phase of the vortex-rings. It ranges from 0 to $\tau = 4.9$. The vortex-rings approach each other with negligible diffusion.
2. The step *b*) corresponds to reconnection phase (*bridging*). It ranges from $\tau = 4.9$ to $\tau = 6.5$. The vorticity diffuses through the wall, the vortex lines are reassociated from the direction $\hat{e}_z$ to the direction $\hat{e}_y$ which causes a decrease of circulation in the $x - y$ plane.
3. The step *c*) corresponds to the free evolution of the bigger vortex-ring from the reconnection of the two initial vortex rings. The difference between the curve is due to the non-complete reconnection of the ring. The threads are stretched by the reconnected part and finally disappear. If the reconnection was perfect, the circulation in the two plane would be equal.

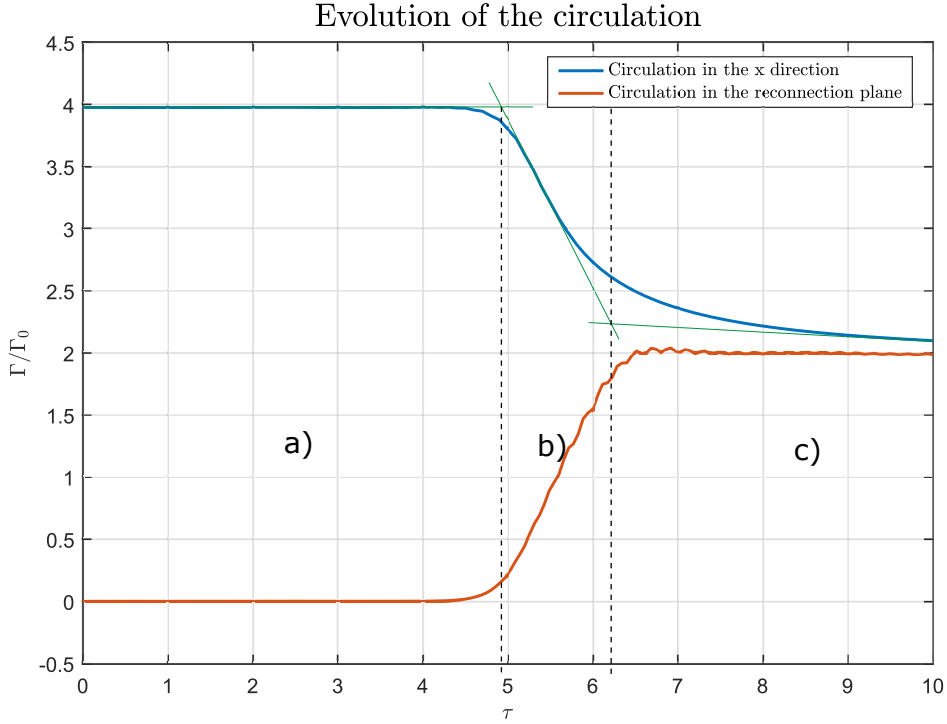


Figure 2.6: Time evolution of the total circulation

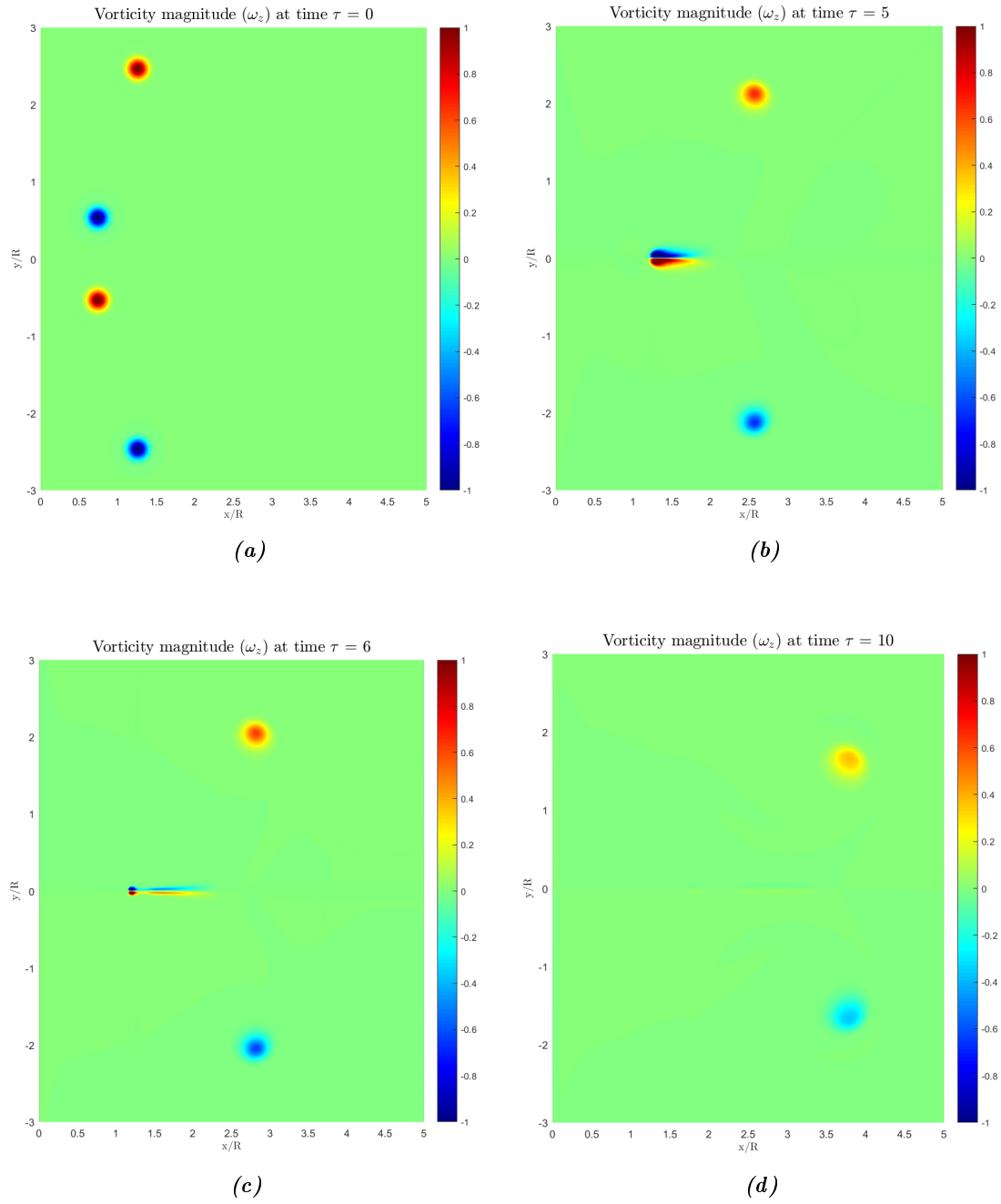


Figure 2.7: Time evolution of dimensionless $\hat{e}_z$ component of vorticity in the plane $x-y$

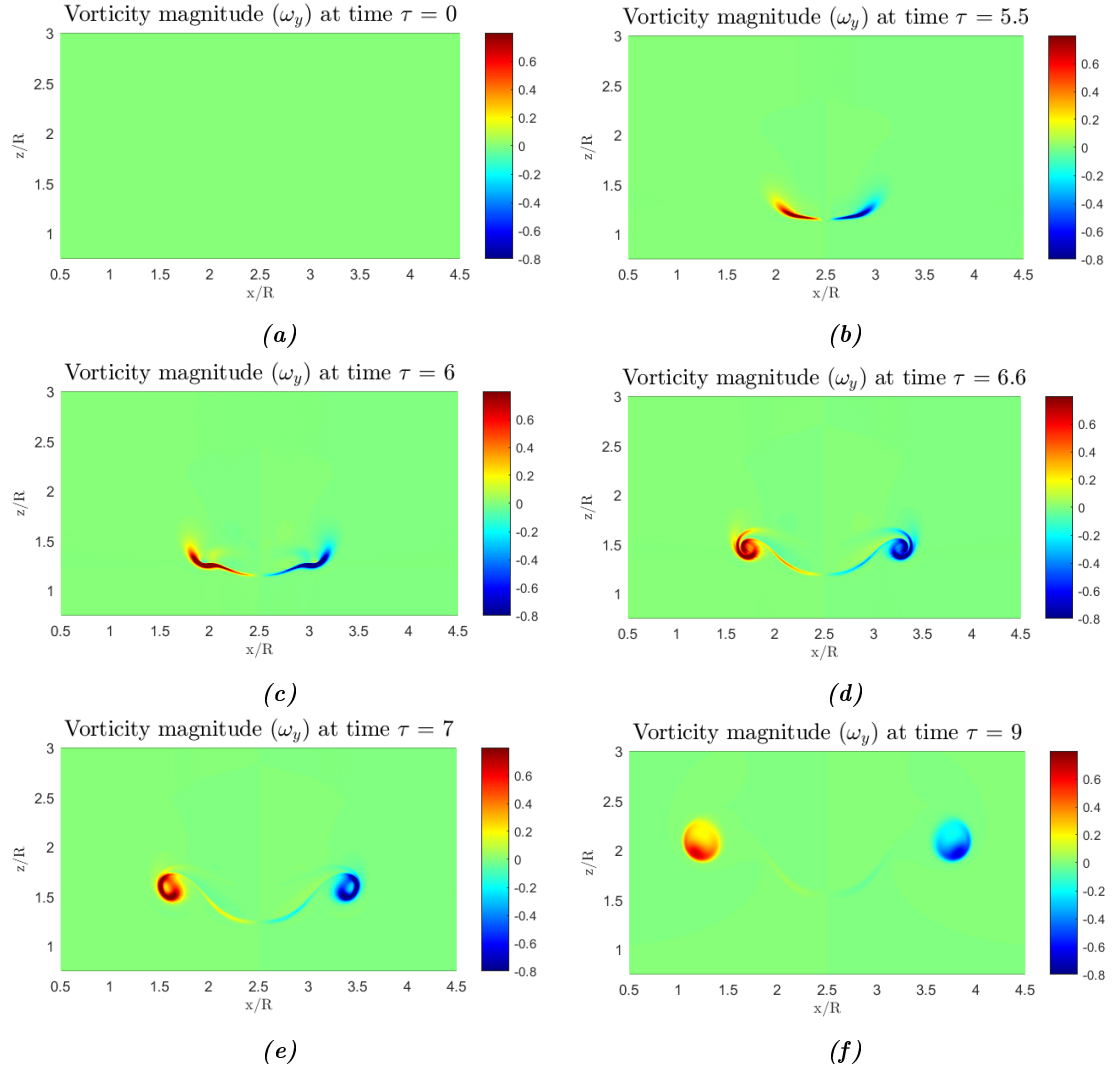


Figure 2.8: Time evolution of dimensionless $\hat{e}_z$ component of vorticity in the plane $y-z$

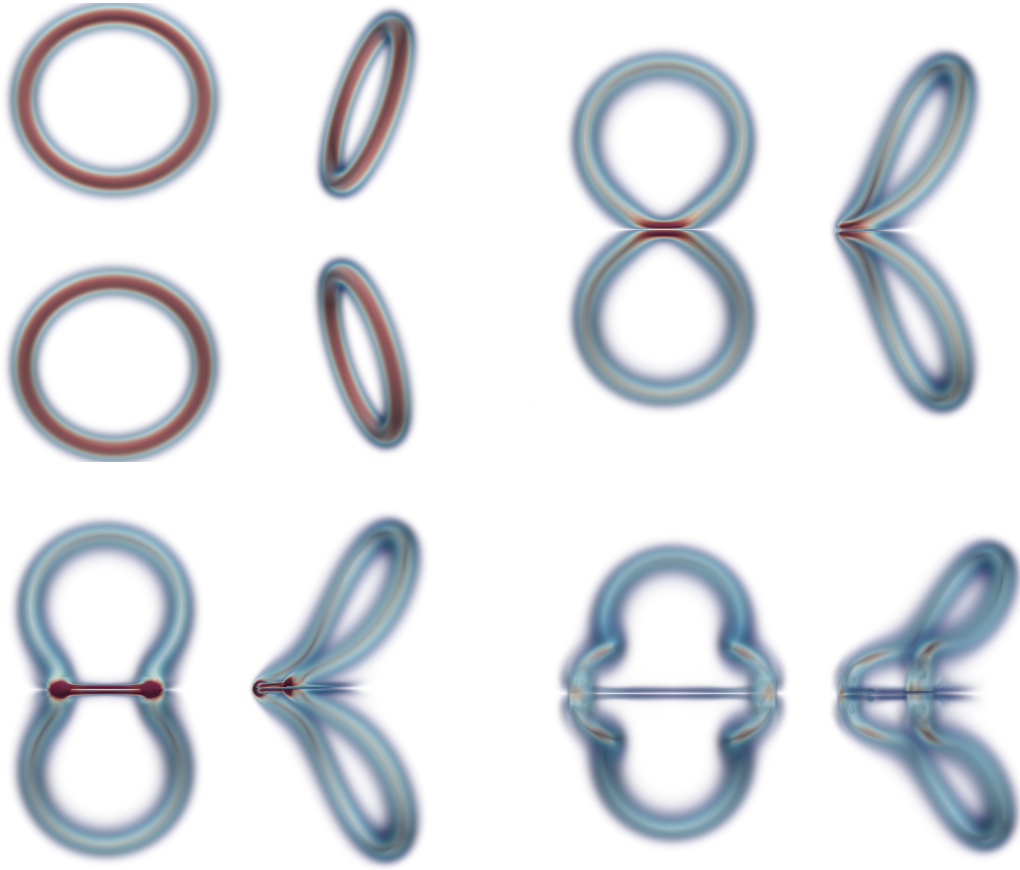


Figure 2.9: Volume rendering of vortex-rings reconnection

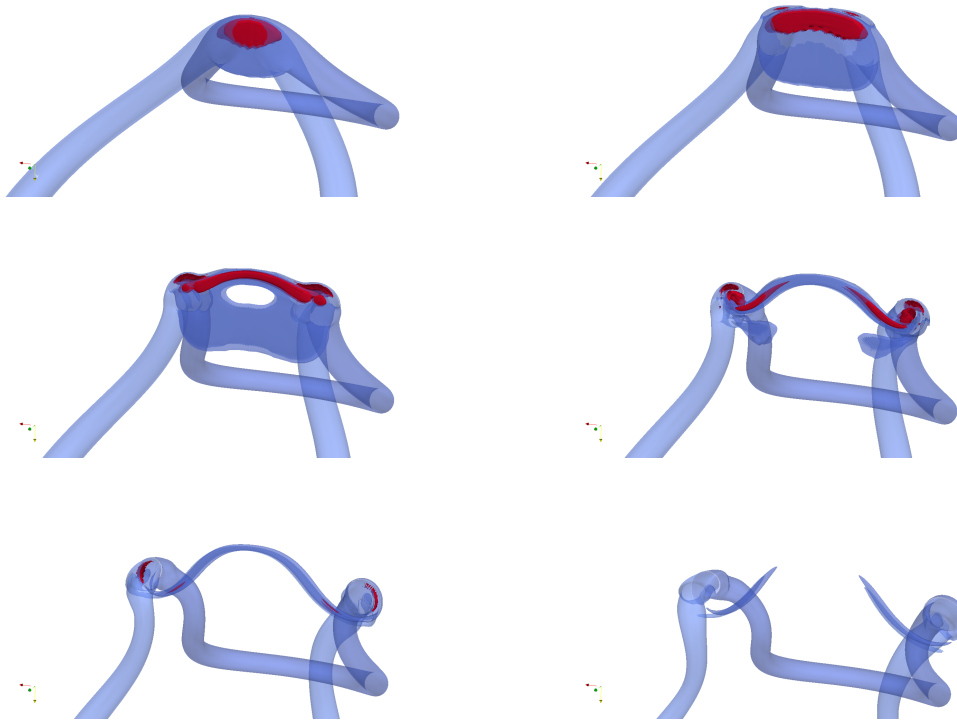


Figure 2.10: Iso-surface of vortex ring reconnection at $\omega/\omega_{ref} = 0.3$ and $\omega/\omega_{ref} = 1$

2.4 Conclusion

The no-through flow solver is implemented and tested with diffusion at the wall to simulate the vortex-rings reconnection. The reconnection is simulated for only one vortex-ring but the solver supposes that another vortex-ring exists by imposing symmetry. The simulation is correctly resolved and all scales of vorticity are captured. The next step of simulation could be to capture the fission of the final big vortex-ring into two smaller vortex-rings as shown by Schatzle in [4]. The next step of this work is to present how the no-slip boundary condition is implemented in the vortex particle-mesh method using the no-through flow solver presented here above.

Chapter 3

No-slip boundary condition implementation

3.1 Introduction

The goal of this chapter is to show how the no-slip boundary condition can be implemented in a vortex particle-mesh method. The VPM method is based on the vorticity-velocity formulation of Navier-Stokes equation. Consequently, the no-slip boundary cannot be simply implemented just by imposing $u = 0$ as done in a velocity-pressure formulation code (Big-flow is an example of such a code). To impose a no-slip velocity at the wall, a vortex-sheet is computed at each time step. The strength of this vortex sheet is computed using the velocity above the no-slip wall. It will be shown how the vortex sheet is computed and how it is numerically implemented in the code.

3.2 Expression of boundary conditions

3.2.1 3D Vorticity field definition

In this document, the no-through flow is always in the $\hat{e}_y$ direction and the surface with no-slip condition is located at $y = 0$.

The vorticity has three components:

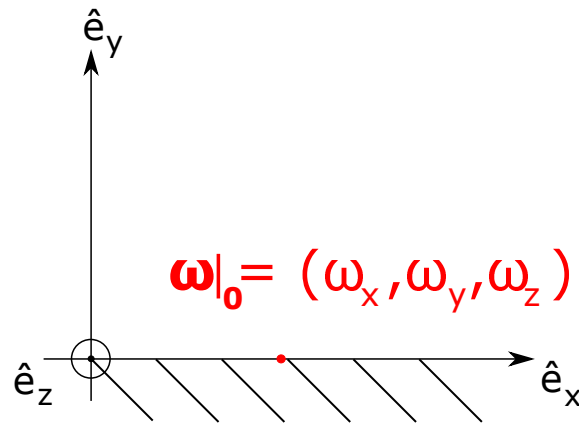


Figure 3.1: Vorticity at the wall

It is defined as :

$$\omega = \nabla \times u$$

The three components of the vorticity are:

$$\begin{aligned}\omega_x &= \frac{\partial u_z}{\partial y} - \frac{\partial u_y}{\partial z} \\ \omega_y &= \frac{\partial u_x}{\partial z} - \frac{\partial u_z}{\partial x} \\ \omega_z &= \frac{\partial u_y}{\partial x} - \frac{\partial u_x}{\partial y}\end{aligned}$$

3.2.2 Boundary conditions

At the wall, three boundary conditions must be imposed on the three components of velocity $\mathbf{u} = (u_x, u_y, u_z)$:

1. No through flow: $u_y = 0$
2. No-slip velocity: $u_x = u_z = 0$

The above conditions give the following constraints on the first spatial derivative of the velocity components:

$$\begin{aligned}\frac{\partial u_y}{\partial x}|_{y=0} &= \frac{\partial u_y}{\partial z}|_{y=0} = 0 \\ \frac{\partial u_x}{\partial x}|_{y=0} &= \frac{\partial u_x}{\partial z}|_{y=0} = \frac{\partial u_z}{\partial x}|_{y=0} = \frac{\partial u_z}{\partial z}|_{y=0} = 0\end{aligned}$$

So, at the wall, the three components of the vorticity become:

$$\begin{aligned}\omega_x &= \frac{\partial u_z}{\partial y} \\ \omega_y &= 0 \\ \omega_z &= -\frac{\partial u_x}{\partial y}\end{aligned}$$

The vorticity needed at the wall is computed based on the spatial derivative of the velocity using finite difference.

3.2.3 Decentered finite difference of fourth order

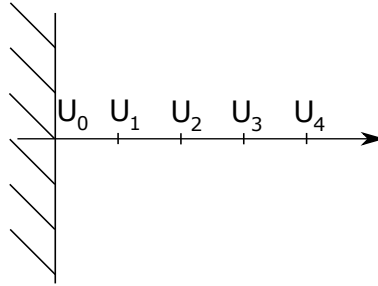


Figure 3.2: Vorticity at the wall

Using Taylor series, the decentered finite difference using 5 points (including the point at the wall) is found:

$$\begin{aligned}U_1 &= U_0 + h \frac{du}{dx}|_0 + \frac{h^2}{2} \frac{d^2u}{dx^2}|_0 + \frac{h^3}{6} \frac{d^3u}{dx^3}|_0 + \frac{h^4}{24} \frac{d^4u}{dx^4}|_0 + \frac{h^5}{120} \frac{d^5u}{dx^5}|_0 + O(h^6) \\ U_2 &= U_0 + 2h \frac{du}{dx}|_0 + 2^2 \frac{h^2}{2} \frac{d^2u}{dx^2}|_0 + 2^3 \frac{h^3}{6} \frac{d^3u}{dx^3}|_0 + 2^4 \frac{h^4}{24} \frac{d^4u}{dx^4}|_0 + 2^5 \frac{h^5}{120} \frac{d^5u}{dx^5}|_0 + O(h^6) \\ U_3 &= U_0 + 3h \frac{du}{dx}|_0 + 3^2 \frac{h^2}{2} \frac{d^2u}{dx^2}|_0 + 3^3 \frac{h^3}{6} \frac{d^3u}{dx^3}|_0 + 3^4 \frac{h^4}{24} \frac{d^4u}{dx^4}|_0 + 3^5 \frac{h^5}{120} \frac{d^5u}{dx^5}|_0 + O(h^6) \\ U_4 &= U_0 + 4h \frac{du}{dx}|_0 + 4^2 \frac{h^2}{2} \frac{d^2u}{dx^2}|_0 + 4^3 \frac{h^3}{6} \frac{d^3u}{dx^3}|_0 + 4^4 \frac{h^4}{24} \frac{d^4u}{dx^4}|_0 + 4^5 \frac{h^5}{120} \frac{d^5u}{dx^5}|_0 + O(h^6)\end{aligned}$$

The part including h^2 , h^3 and h^4 are cancelled to find the decentered finite difference of 4^{th} order:

$$\frac{du}{dx}\bigg|_0 = \frac{-25U_0 + 48U_1 - 36U_2 + 16U_3 - 3U_4}{12h} - \frac{h^4}{5} \frac{d^5u}{dx^5}$$

3.2.4 Numerical expression of boundary conditions

With the no slip boundary condition on velocity: $u_x = 0$ and $u_z = 0$, the numerical approximations of vorticity needed to enforce the no-slip boundary condition are the following:

$$\begin{aligned}\omega_x(x, 0, z) &= \frac{48u_z(x, \Delta y, z) - 36u_z(x, 2\Delta y, z) + 16u_z(x, 3\Delta y, z) - 3u_z(x, 4\Delta y, z)}{12\Delta y} \\ \omega_z(y, 0, z) &= -\frac{48u_x(x, \Delta y, z) - 36u_x(x, 2\Delta y, z) + 16u_x(x, 3\Delta y, z) - 3u_x(x, 4\Delta y, z)}{12\Delta y}\end{aligned}$$

3.3 Implementation in VPM method

3.3.1 General method

In order to impose no-slip and no-through flow boundary conditions in the VPM method, the following steps are done at each time-step:

1. Solve the Poisson-equation with no-through flow boundary condition only
2. Impose the boundary condition on the vorticity to nullify the slip velocity (section 3.2.4).
3. Evaluate the right hand side of the Navier-Stokes equation
4. Update the location and vorticity of particles

3.3.2 First step: No-through-flow

The velocity based on the vorticity is computed on a mesh. The first step consists of the interpolation of particle value on a mesh (particle to mesh). The Poisson equation is solved in the same way as explained in chapter 2.1 except that the diffusion is no-more allowed through the wall, the normal vorticity being set to zero. This extends the vorticity field like this:

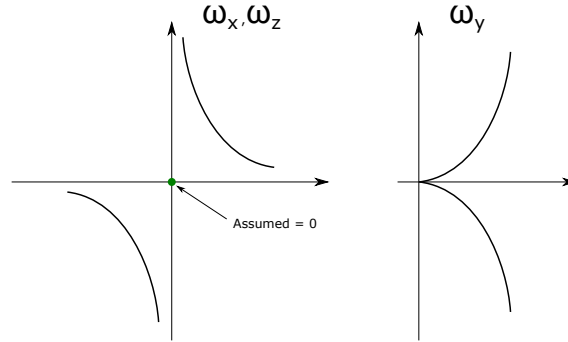


Figure 3.3: Vorticity at the wall

At the end, the velocity and vorticity field both satisfy the no-through flow condition on the mesh but a slip velocity exists at the wall.

3.3.3 Second step: Imposing the boundary condition on vorticity

The Poisson solver imposes that the parallel components of vorticity are equal to zero. In order to enforce the no-slip condition, the values of wall-vorticity must be changed. Recall the formula

from section 3.2.4. The vortex sheet strength needed to nullify the slip velocity is computed using:

$$\begin{aligned}\omega_x(x, 0, z) &= \frac{48u_z(x, \Delta y, z) - 36u_z(x, 2\Delta y, z) + 16u_z(x, 3\Delta y, z) - 3u_z(x, 4\Delta y, z)}{12\Delta y} \\ \omega_z(y, 0, z) &= -\frac{48u_x(x, \Delta y, z) - 36u_x(x, 2\Delta y, z) + 16u_x(x, 3\Delta y, z) - 3u_x(x, 4\Delta y, z)}{12\Delta y}\end{aligned}$$

At the wall, the values of the component of velocity computed by the Poisson solver are overwritten and set to zero, to be sure that these values are null everywhere on the no-slip surface:

$$\begin{aligned}u_x(x, 0, y) &= 0 \\ u_y(x, 0, y) &= 0 \\ u_z(x, 0, y) &= 0\end{aligned}$$

At the end of this step, the velocity and vorticity fields on the mesh are consistent with the no-slip and no-through flow conditions.

3.3.4 Third step: right hand side of the Navier-Stokes equation

In order to update the location and the vorticity carried by the particles, the right-hand side of the Navier-Stokes equation (3.1) has to be evaluated. The vorticity-velocity formulation of momentum conservation in the case of an incompressible flow in a Lagrangian description is the following:

$$\frac{D\boldsymbol{\omega}}{Dt} = (\boldsymbol{\omega} \cdot \nabla)\mathbf{u} + \nu \nabla^2 \boldsymbol{\omega} \quad (3.1)$$

Three cases are distinguished:

1. Where $y \geq \Delta y$
2. Where $y = 0$
3. Where $y \leq 0$

3.3.4.1 $y \geq \Delta y$

The right-hand side is evaluated as usual: the stretching and the diffusion are computed in the entire domain using fourth order schemes.

3.3.4.2 $y = 0$

At the wall, the no-through flow and no slip conditions fix the velocity to: $\mathbf{u} = 0$. The vortex stretching $(\boldsymbol{\omega} \cdot \nabla)\mathbf{u}$ simply becomes zero, so the previous equation simplifies to:

$$\frac{D\boldsymbol{\omega}}{Dt} = \nu \nabla^2 \boldsymbol{\omega}$$

It shows that, at the wall, the vorticity of the particle only diffuses in the bulk; there are no stretching and no convection. The Laplacian is computed using a centered finite difference of second order. In order to compute the Laplacian, a fourth order ghost point at $y = -\Delta y$ is computed to ensure the second order of Laplacian scheme:

$$\omega(x, -\Delta y, z) = 4\omega(x, 0, z) - 6\omega(x, \Delta y, z) + 4\omega(x, 2\Delta y, z) - \omega(x, 3\Delta y, z)$$

Note: a similar way to compute the Laplacian at the wall is to use decentered finite differences of second order. This exactly provides the same result.

3.3.4.3 $y \leq 0$

In the vortex particle-mesh method, the right hand side is computed on a mesh and the convection on particles. At the beginning of the evaluation of the right-hand side, the values of the particles are interpolated on a mesh (particle to mesh step), and, at the end of the evaluation of the right-hand side, it is interpolated back on the particle (mesh to particle step) using a high order scheme (M'_4) in order to do the time update. The interpolation scheme needs a value beyond the boundary, so the value of right hand side must be extrapolated beyond the boundary. This is done using a fourth order ghost point on the right hand side.

3.3.5 Fourth step: time update of particle location and strength

At the end of these steps, the right-hand side based on velocity and vorticity field is consistent with the no-through flow and no-slip conditions. The last step is to interpolate the right-hand side back on particles using M'_4 and update the location and the vorticity carried by the particles using a Rung-Kutta 3 scheme.

3.4 Conclusion

The method to implement the no-slip condition for a flat wall is presented in this chapter. The diffusion at the wall is no more allowed, the Poisson-solver configured in this way is a purely no-through flow. The vortex sheet needed to nullify the slip velocity is computed using decentered finite differences of fourth order on velocity. At each point on the wall, the velocity is exactly zero, so the vortex sheet at the wall is only diffused in the bulk. For that reason, no stretching and no convection occurs to the particles on the wall. In the next sections, the no-slip boundary condition is firstly validated using a one-dimensional flow of Poiseuille, secondly validated using a two-dimensional flow of boundary layer development of Blasius.

Chapter 4

Validation

4.1 Introduction

This section proposes to validate the numerical method presented above using two simple cases:

- The time evolution of a one dimensional flow in a channel with constant mass flow, the analytic solution being provided at the steady-state
- The time evolution of a two-dimensional flow: the development of a boundary layer in a channel, the steady-state being compared to the analytic solution of Blasius flow.

4.2 One dimensional flow

4.2.1 Analytical solution

The flow inside a thin channel is driven by a constant mass flow rate (per unit length) that is fixed by the condition at the inlet:

$$Q = \rho u_{\infty} d$$

The complete velocity profile is the following:

$$\mathbf{u} = u_x(x, y, z, t)\hat{e}_x + u_y(x, y, z, t)\hat{e}_y + u_z(x, y, z, t)\hat{e}_z$$

The channel configuration is: infinitely long in the $\hat{e}_z$ and $\hat{e}_x$ direction, thin in the $\hat{e}_y$ direction, and the incoming flow is in the $\hat{e}_x$. Based on this configurations, several hypothesis are made on the flow:

- The channel is supposed to be infinitely long in the $\hat{e}_z$ direction, so the velocity component in the $\hat{e}_z$ direction is zero and the flow doesn't vary in this direction
- The channel is supposed to be thin, so the velocity component in the $\hat{e}_y$ direction is zero
- The flow doesn't vary with the $\hat{e}_z$ direction because it is infinitely long
- The channel is supposed to be infinitely long in the $\hat{e}_x$ direction so the flow doesn't vary in this direction
- The flow has reached its steady-state

Based on these hypothesis the flow is simply:

$$\mathbf{u} = u_x(y)\hat{e}_x$$

The Navier-Stokes equations becomes:

$$0 = -\frac{1}{\rho} \frac{dp}{dx} + \nu \frac{\partial^2 u}{\partial y^2} \quad (4.1)$$

The dimensionless equations are obtained by choosing three references values: $U = U_\infty$, $Y = L_y$ and ρ :

$$\begin{aligned} u' &= \frac{u}{U_\infty} \\ y' &= \frac{y}{L_y} \\ x' &= \frac{x}{L_y} \\ p' &= \frac{p}{\rho U_\infty^2} \end{aligned}$$

The dimensionless equation is:

$$0 = -\frac{dp'}{dx'} + \frac{1}{Re} \frac{\partial^2 u'}{\partial y'^2}$$

With the Reynolds number of the flow: $Re = \frac{U_\infty L_y}{\nu}$.

Conjugated to the no-slip boundary condition and the imposed mass flow, the velocity profile and the pressure gradient is found at the steady-state, the final profile is a Poiseuille flow and is shown at the figures 4.2:

$$\begin{aligned} u'(y') &= -6(y'^2 - y') \\ \frac{dp'}{dx'} &= -\frac{12}{Re} \end{aligned}$$

The analytical vorticity profile is easily obtained from the velocity profile:

$$\begin{aligned} w'_z &= -\frac{du'}{dy'} \\ &= 6(2y' - 1) \end{aligned}$$

4.2.2 Numerical setup

In order to validate the numerical method presented above, the same flow is solved using the VPM code. The VPM code is a three dimensions flow solver. To emulate the one dimensional flow behaviour, two periodic boundary conditions are set in the $\hat{e}_x$ and $\hat{e}_z$ directions, the $\hat{e}_y$ is the no-through flow direction:

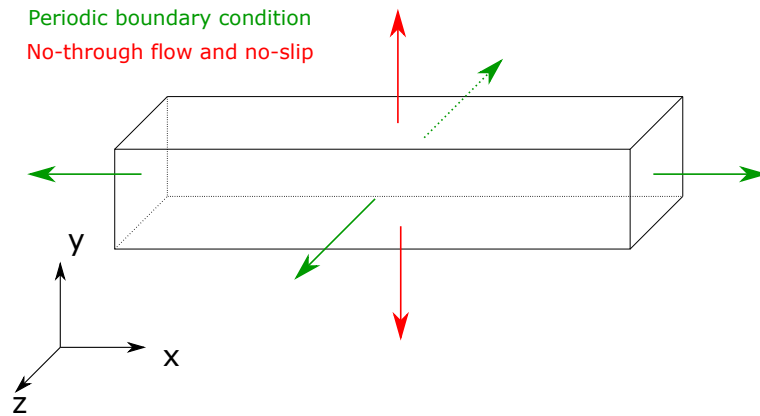


Figure 4.1: Boundary conditions in the numerical simulation

The half height of the channel is δ and the numerical parameters are the following:

Parameter:	Value
$N_x \times N_y \times N_z$	$16 \times 64 \times 16$
$L_x/\delta \times L_y/\delta \times L_z/\delta$	$0.25 \times 2 \times 0.25$
Re (based on U_∞ and δ)	100 [-]
$t_{ref} = \frac{\delta}{U_\infty}$	0.5 [sec]
Dimensionless final time	50 [-]
Remesh and reprojection	Every 5 and 20 time step

Table 4.1

4.2.3 Numerical results - Velocity

The numerical result starts from an initial condition of $u'(y') = 1$ everywhere except at the boundary, and converges until it reaches the steady-state. The evolution of the velocity in the channel is sketched at the following figure:

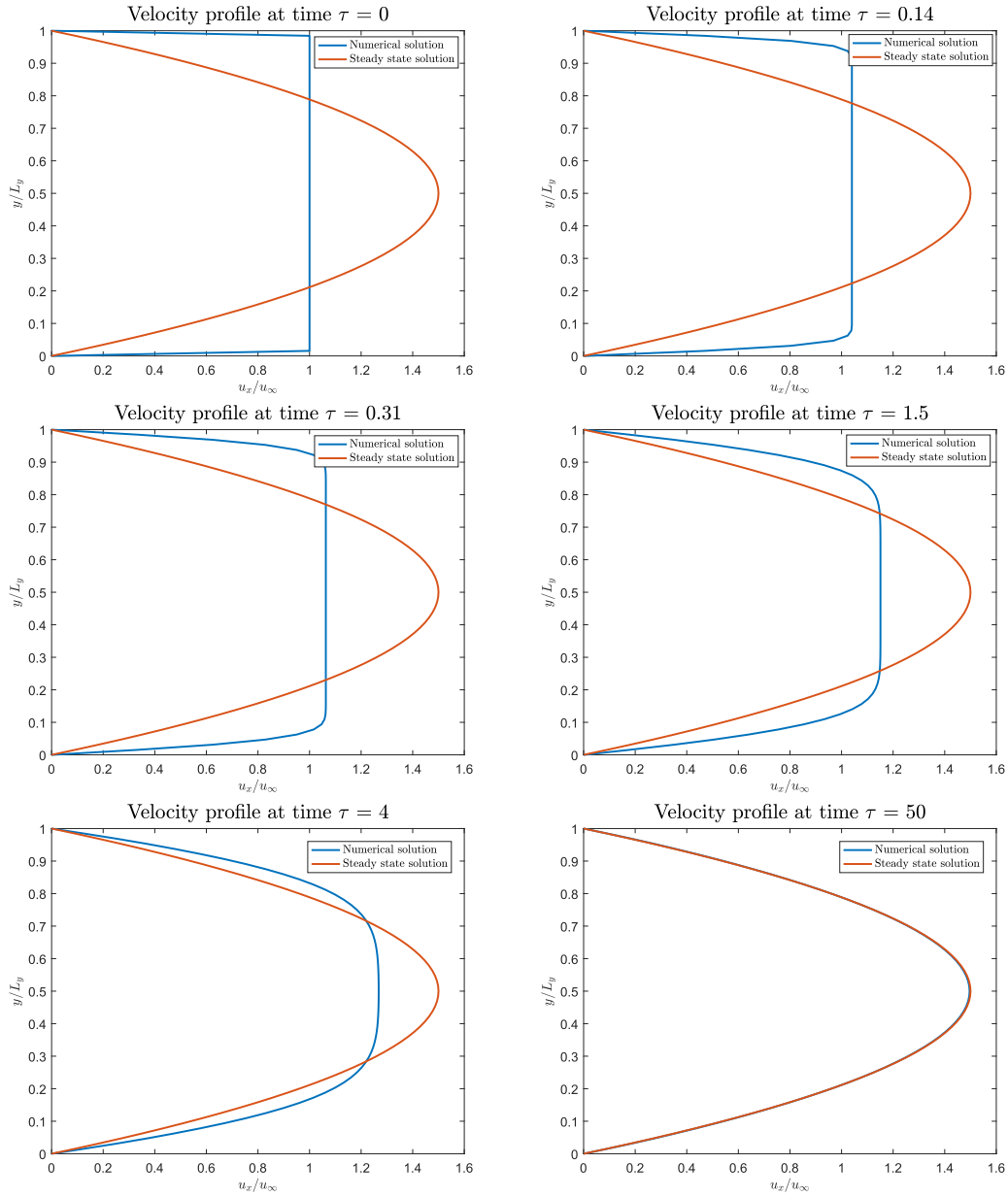


Figure 4.2: Time development of the velocity profile

At the end of the simulation, the relative error L^2 between the numerical solution (u_n) and

the exact solution (u_e) is computed :

$$Err_{L^2} = \frac{\int_0^{L_y} (u_e - u_n)^2 dy}{\int_0^{L_y} u_e^2 dy} = 1.32 \times 10^{-4}$$

4.2.4 Results - Vorticity

The same analysis can be performed on the vorticity field:

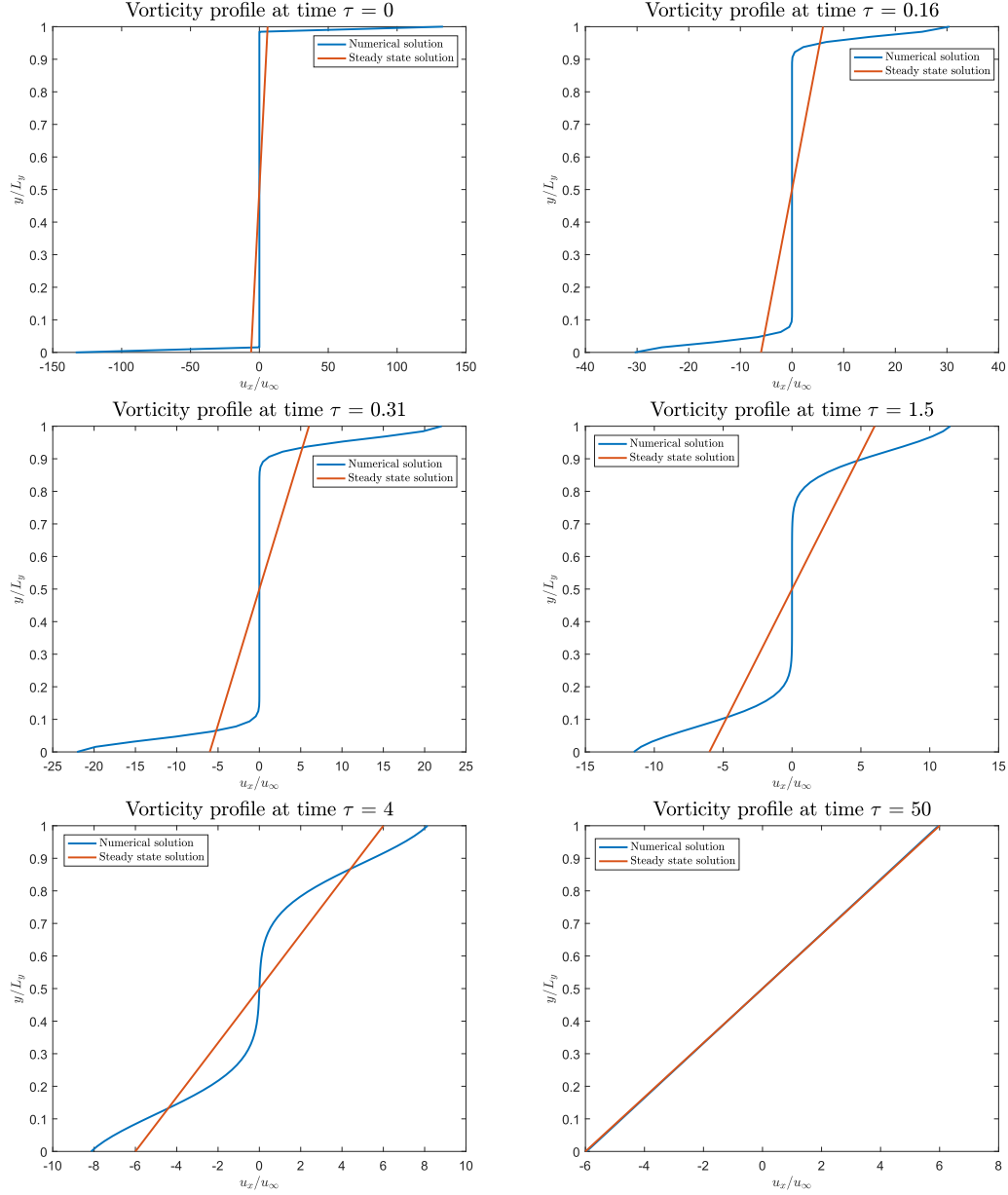


Figure 4.3: Time development of the vorticity profile

4.2.5 Pressure gradient evolution

In the simulation, the flow is driven by a constant mass flow rate. In order to keep this mass flow rate constant, the pressure gradient must be adapted at each time step. The pressure gradient required to impose the mass-flow is computed at each time step using finite differences on the

Navier-Stokes equation of the flow. The evolution is sketched at the figure 4.4.

$$\frac{\partial p'}{\partial x'} = -\frac{\partial u'}{\partial t'} + \frac{1}{Re} \frac{\partial^2 u'}{\partial y'^2}$$

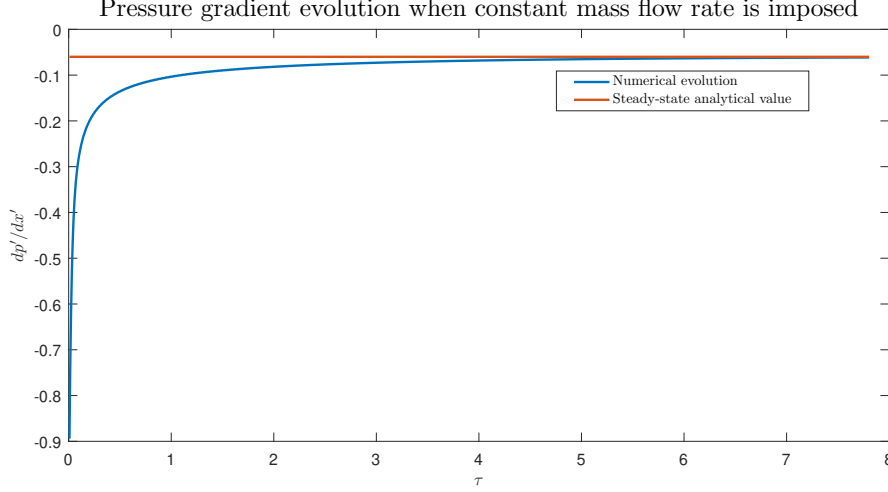


Figure 4.4: Pressure gradient evolution to impose constant mass-flow rate

At the beginning of the simulation, the pressure gradient must be infinite. As the simulation progresses, the pressure gradient is lower and lower and finally reaches the steady-state analytical value.

4.2.6 Conclusion

The implementation of the boundary condition is validated for a one dimensional flow using an analytical solution of steady-state flow in a channel with constant mass flow. The numerical solution tends to the analytic one for large time. The pressure gradient required to impose the mass flow rate is also computed at each time step.

4.3 Two-dimensional flow

4.3.1 Introduction

This section studies the numerical resolution of the flow above a flat plate with the no-slip boundary condition. The analytical solution is well known as Blasius solution for boundary layer development. This solution is given for the velocity and the vorticity at appendices A.1 and A.2.

The numerical solution is computed in a channel with a no-slip and a slip wall. The velocity encounter an acceleration and the velocity above the boundary is not so simple as in the Blasius case, several velocity above the boundary layer are proposed and compared for the spatial development of boundary layer in a channel.

4.3.2 Numerical setup

To validate the implementation of the method for the no-slip boundary condition, the VPM code is run using periodic boundary condition in the $\hat{e}_z$ direction in order to emulate the two-dimensional flow (VPM code is a three dimensions solver), with an inflow/outflow condition in the $\hat{e}_x$ direction, a no-through flow and no slip conditions in the $\hat{e}_y$ direction:

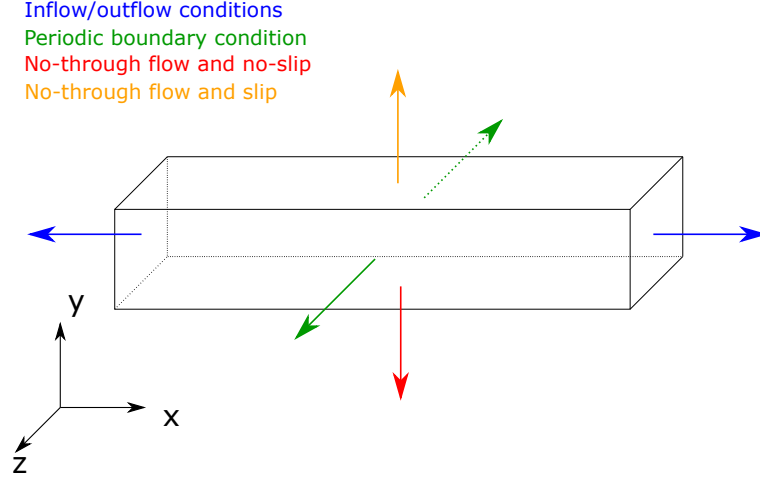


Figure 4.5: Boundary conditions in the numerical simulation

The numerical parameters are the following:

Parameter:	Value
$N_x \times N_y \times N_z$	$256 \times 256 \times 16$
$L_x / L_x \times L_y / L_x \times L_z / L_x$	$1.0 \times 1.0 \times 0.0625$ [m]
Re (based on U_∞ and L_x)	2400 [-]
$t_{ref} = L_x / U_\infty$	15 [sec]
Dimensionless final time	7 [-]
Remesh and reprojection	Every 5 and 20 time step

Table 4.2: Numerical parameters for the boundary layer simulation

4.3.3 Numerical results - Limitations

This first part of numerical results deals with the limitations of the numerical simulation of boundary layer development in a channel.

The simulation clearly doesn't reproduce the exact Blasius boundary layer solution because of the no-through flow at $y = L_y$.

In the simulation, the vertical velocity has to become zero when $y \rightarrow L_y$. The flow outside the boundary layer will encounter an acceleration because of the restriction of the passing area due to the boundary layer. But if L_y is large enough, the velocity and vorticity profile in the boundary layer will be well captured by the numerical simulation.

Moreover, the outflow boundary condition imposes that the streamlines are normal to the outflow surface. This outflow condition influences the boundary layer near the outlet.

4.3.3.1 Velocity

The horizontal component outside the boundary layer is accelerated due to the restriction of the passing area caused by the presence of the boundary layer. The relative acceleration above the boundary layer is shown at the following figure:

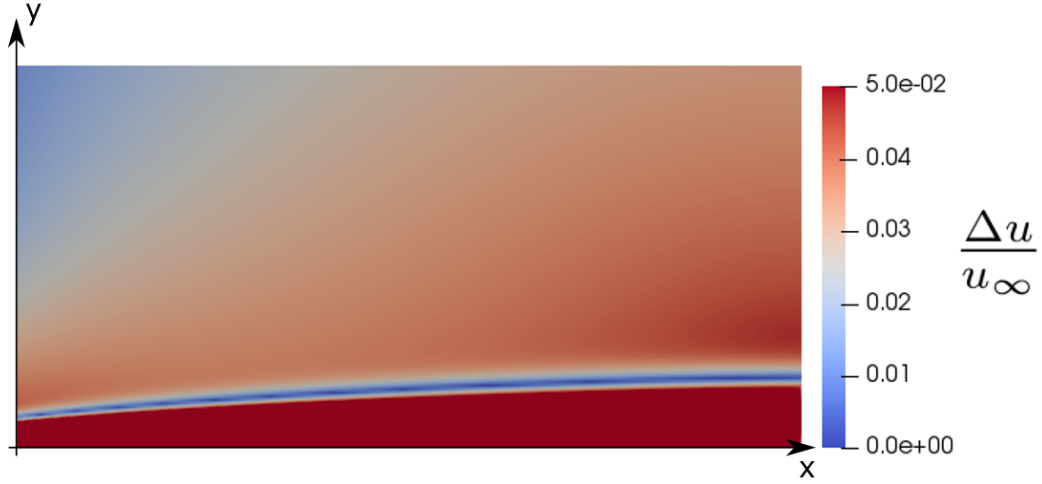


Figure 4.6: Relative acceleration of the flow above the boundary layer

As the boundary layer grows, the flow is accelerated. The velocity profile presents an overshoot just above the boundary layer. The local profile is shown at the next figure. The acceleration of the flow is maximum close to the outlet, but it doesn't exceed 5% of the U_∞ .

The velocity above the boundary is not as simple as in the Blasius case, it can be the maximum velocity at the overshoot, the velocity at the slip wall or other definitions. This is clearly discussed in the next sections. However, it can be already observed at the figure 4.7 and 4.6, that the maximum velocity just above the boundary layer is decreasing then increasing. The velocity at the slip wall is a monotonously increasing function.

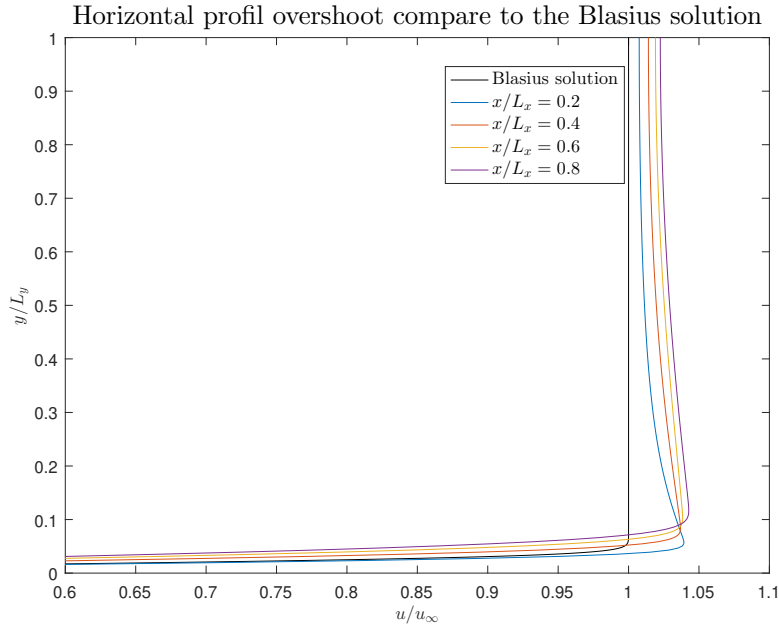


Figure 4.7: Local speed overshoot of the flow above the boundary layer

In addition, the vertical velocity has to become zero, the analytical solution of the Blasius boundary layer supposes that the flow is unbounded above the plate. This leads to an additional difference between the numerical profile computed with VPM and the analytical one.

4.3.3.2 Streamlines

As briefly mentioned above, the outlet boundary condition affects the flow near the outlet. In order to quantify *how far the influence of the outlet is experienced by the flow*, the streamlines near the outlet and near the wall are computed and shown at the figure 4.8.

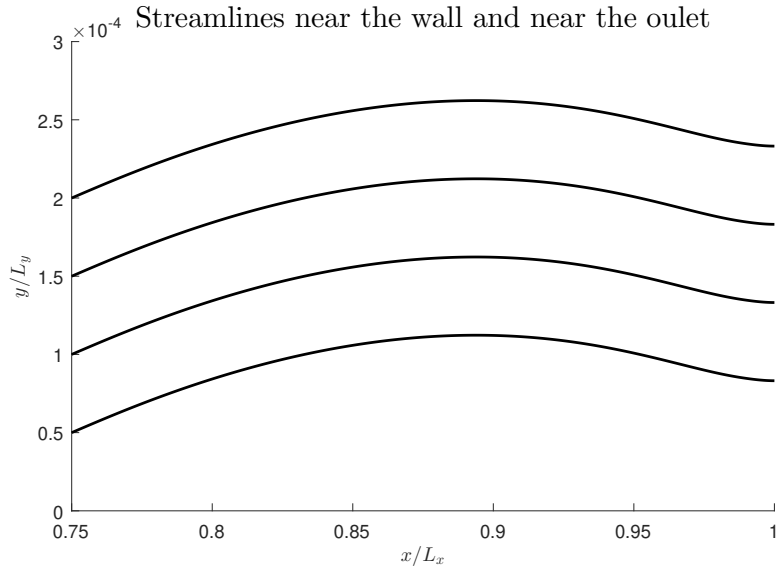


Figure 4.8: Streamlines near the outlet

The streamlines are normal at the outlet surface due to the outlet condition and the influence on the streamlines is a negative slope around $\frac{x}{L_x} \simeq 0.9$. Physically, for a boundary layer, this maximum is a no-sense. So, for the accuracy of the comparison, no result beyond $\frac{x}{L_x} = 0.8$ will be considered.

4.3.4 Numerical results - Validations - Velocity

The velocity profile in the boundary layer develops proportionally to the square root of the distance, at a time $\tau = \frac{t}{t_{ref}} = 5$ the solution is completely steady and the comparison with Blasius can be performed.

The numerical two-dimensional solution is sketched in the following figures from the beginning till the end:



Figure 4.9: Velocity magnitude at time $\tau = 0$ and $\tau = 0.33$

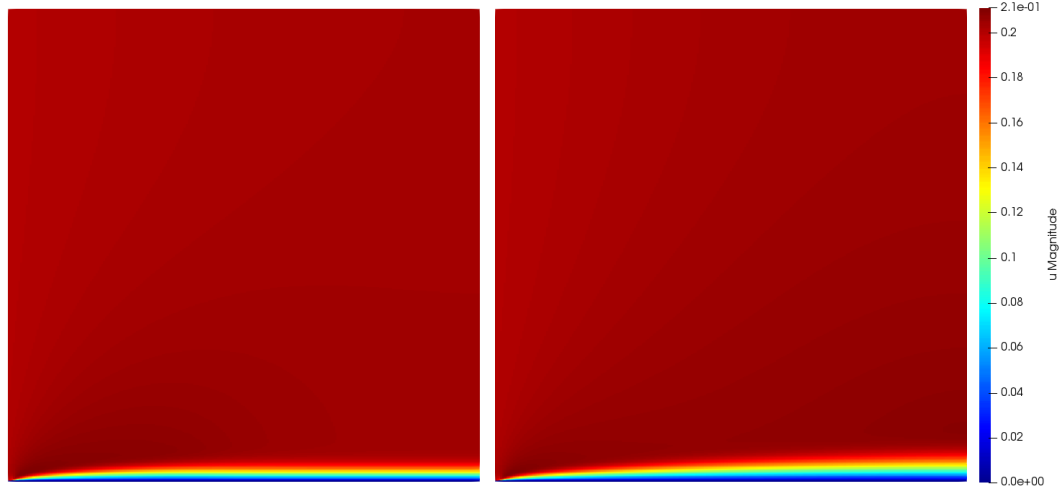


Figure 4.10: Velocity magnitude at time $\tau = 1$ and $\tau = 5$

In order to validate the method and compare the numerical and analytical solutions, the flow at different sections in the streamwise direction is shown at the next figures.

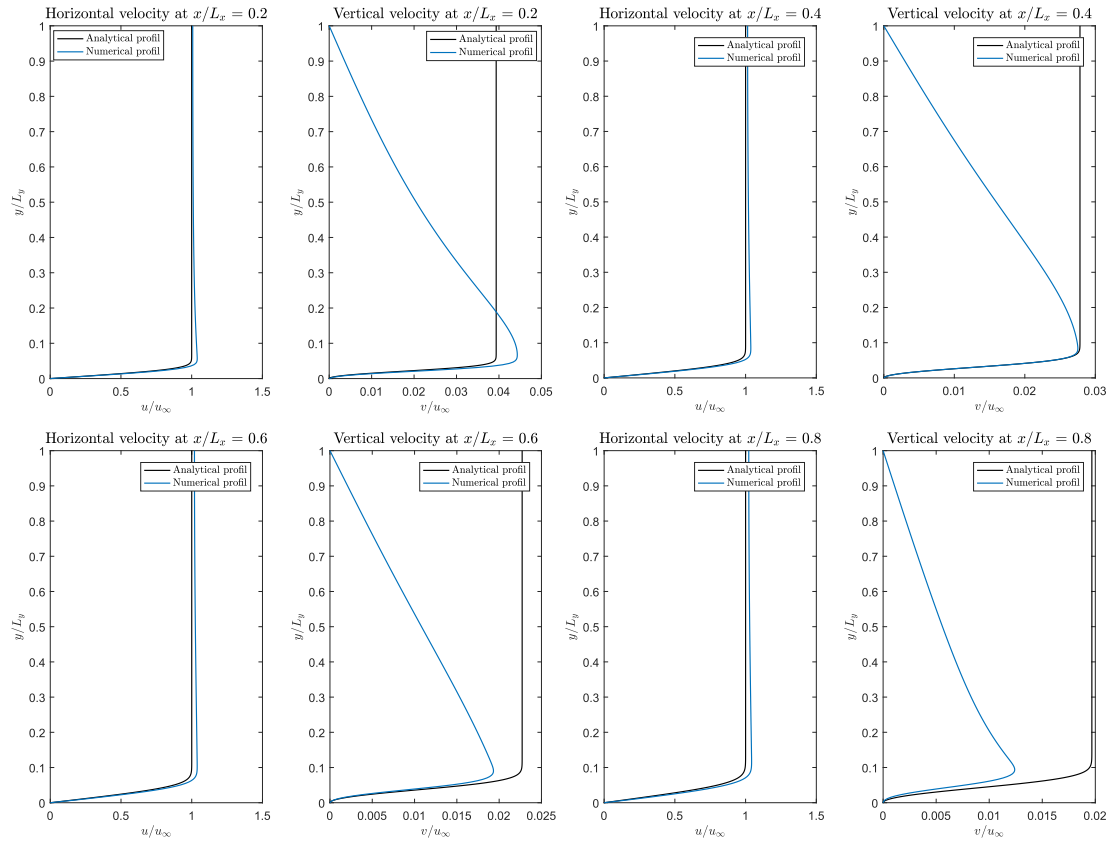


Figure 4.11: Velocity profiles after convergence for different sections: $x/L_x = 0.2, 0.4, 0.6, 0.8$,

The numerical solution for the horizontal velocity very closely follows the analytical one. The vertical velocity follows quite well the analytic solution for low value of y/L_y (e.g. in the boundary layer). The profile goes far from the analytic solution as we move the boundary layer away because it has to go back to zero. There is no such a condition for Blasius flow. It supposes that there is no upper boundary.

4.3.5 Numerical results - Validations - Vorticity

The numerical vorticity profile is also compared to the analytical solution. The vorticity is sketched at different sections in the streamwise direction:

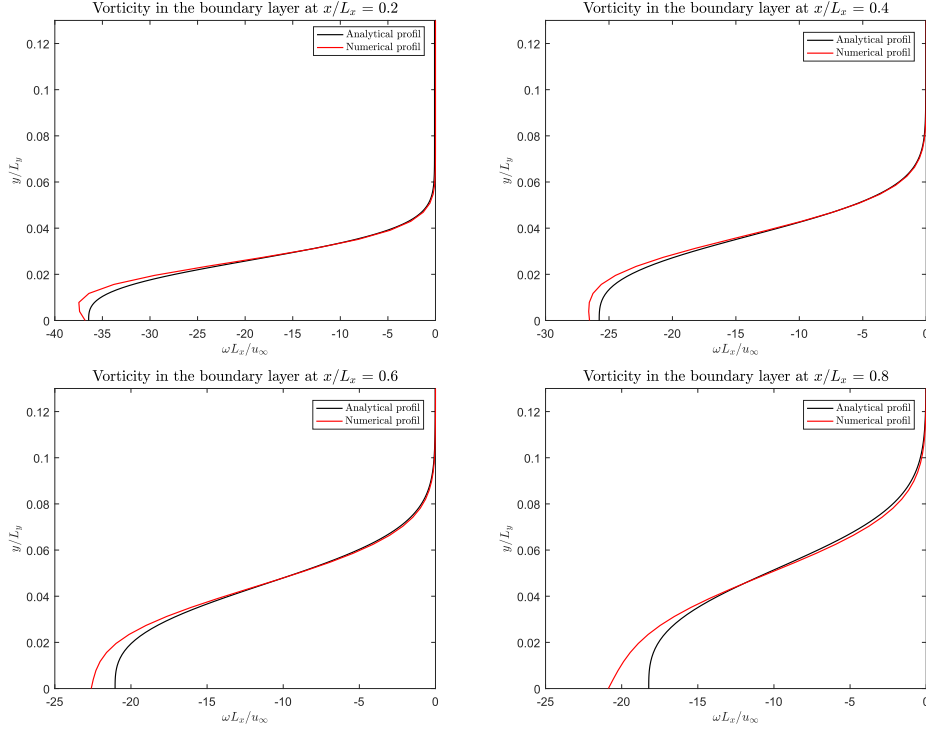


Figure 4.12: Vorticity profiles after convergence for different sections: $x/L_x = 0.2, 0.4, 0.6, 0.8$,

4.3.6 Wall vorticity flux

The wall-vorticity flux $q_\omega = -\nu \frac{\partial \omega}{\partial y}$ is computed based on the steady-state Navier-Stokes equation:

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = u_p \frac{du_p}{dx} + \nu \frac{\partial^2 u}{\partial y^2}$$

At the wall, it becomes:

$$\Leftrightarrow u_p \frac{du_p}{dx} = -\nu \frac{\partial^2 u}{\partial y^2} = \nu \frac{\partial \omega}{\partial y} \quad (4.2)$$

It gives for the vorticity flux :

$$q_\omega = -u_p \frac{du_p}{dx} \quad (4.3)$$

In the Blasius solution case, the velocity above the boundary layer is constant, so the vorticity flux at the wall is zero, and it is visible at the figure 4.12:

$$\begin{aligned} \frac{du_p}{dx} &= 0 \\ \Leftrightarrow q_\omega &= \frac{\partial \omega}{\partial y} = 0 \end{aligned}$$

The numerical vorticity profile is very close to the analytic one but, due to the acceleration of the flow above the boundary layer, the wall vorticity flux is not zero. To compute the vorticity flux, two approaches are used : the first one directly computes the spatial derivative of the vorticity at the wall and the second one computes the right hand side of equation (4.3).

The point is to know the possible definitions of the u_p velocity. This answer is not that easy. Three solutions for u_p are proposed and compared :

1. u_p is the velocity reached at the slip wall : at $y = L_y$
2. u_p is the velocity at the speed overshoot (figure 4.7)

3. u_p is an equivalent velocity computed using the vorticity flux at the wall

The last velocity computation is based on the equation (4.3). The vorticity flux is known and the following differential equation is solved for this velocity :

$$\frac{du_{eq}}{dx} = -\frac{q_\omega}{u_{eq}}$$

The three velocity definitions $u_p(x)$ are shown at the next figure :

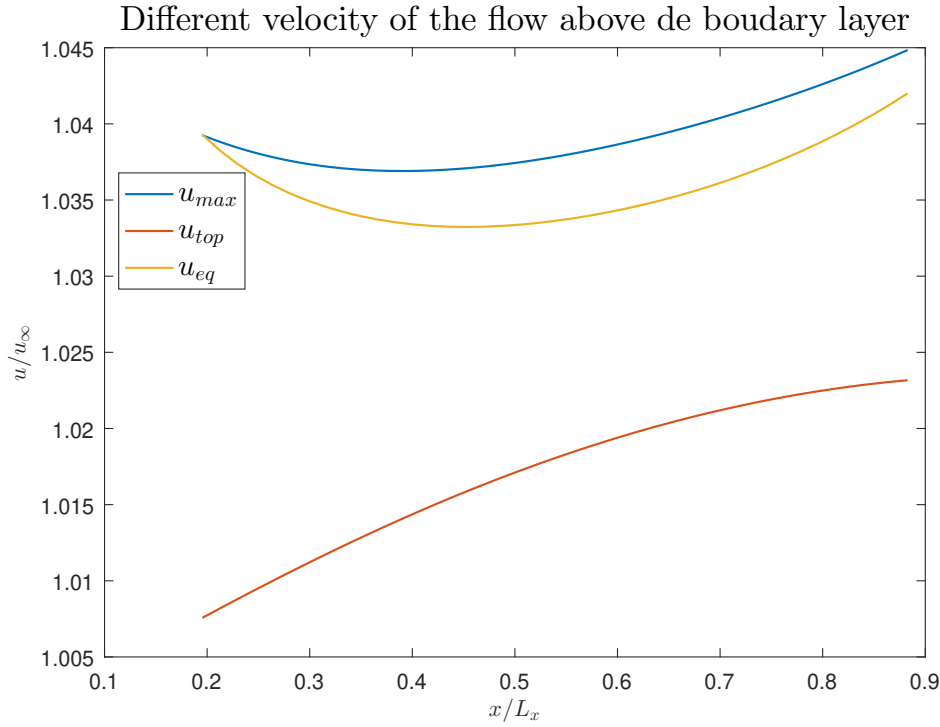


Figure 4.13: Different velocity profiles above the boundary layer

In reality the velocity above the boundary layer is not only a function of the streamwise direction it depends also on the wall-normal direction as shown at the figure 4.10. It is of the form of : $u_p(x, y)$. The Prandl't equation of boundary layer are no-more valid, but the influence is very minimal as the height of the channel is sufficiently high.

The results of the RHS of equation (4.3) are shown at the next figure. Obviously, the flux based on the third definition gives the exact result as the velocity used is based on the vorticity flux :

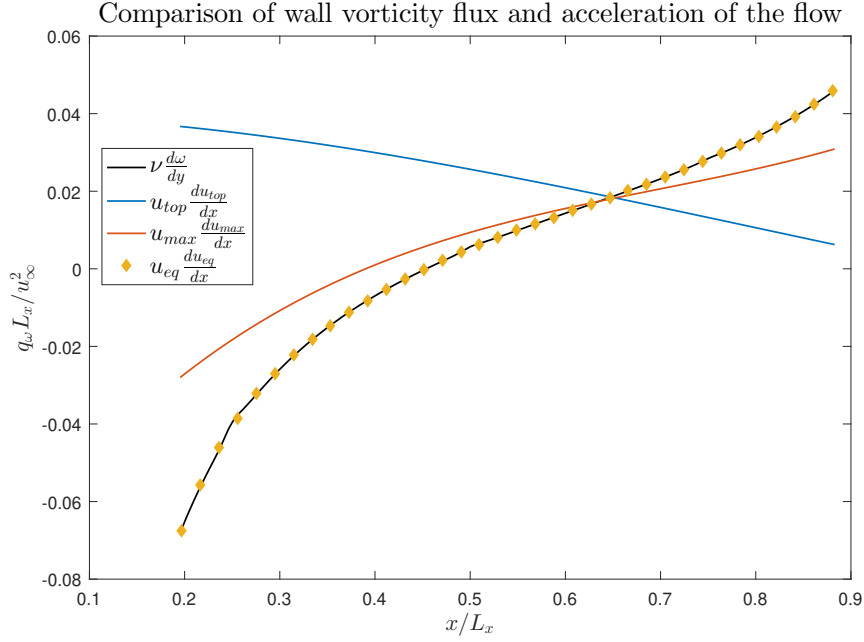


Figure 4.14: Wall vorticity flux and acceleration of the flow

Finally, the vorticity at the wall to enforce the no-slip boundary condition is computed and compared to the analytical one:

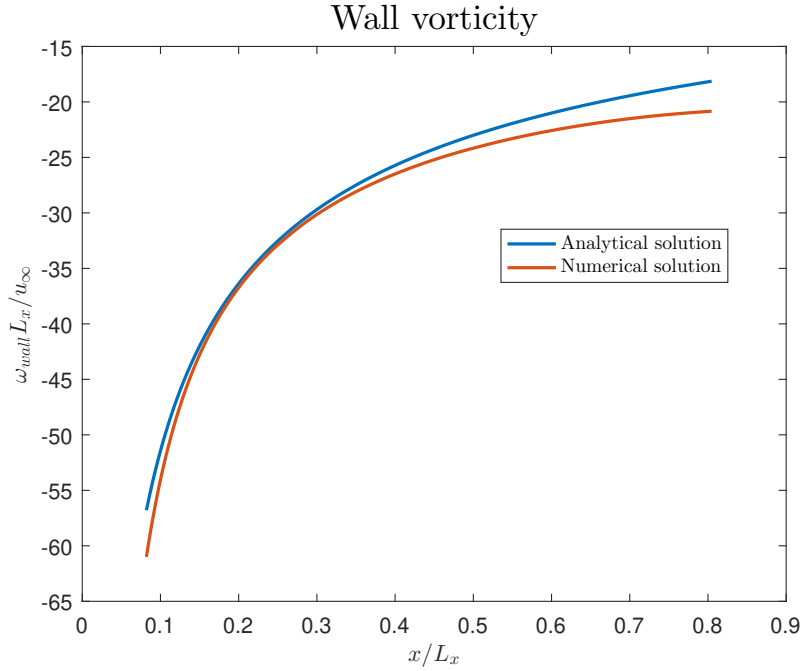


Figure 4.15: Vorticity at the wall

4.3.7 Numerical results - Validations - Boundary layer thicknesses

In order to close the chapter of boundary layer development and two-dimensional flow validation, the value of the boundary layer thicknesses are numerically computed and the results are compared to the analytical solution. The theory of boundary layer thicknesses is developed in appendix A.3.

For the numerical evaluation of the boundary layer thicknesses, the definition is slightly different because the boundary layer develops in a channel. The flow is accelerated and presents an

overshoot just above the boundary layer. Two definitions for the boundary layer thicknesses are proposed:

- The first one is based on the velocity reached by the flow at the upper wall (the slip wall !) the $\delta_{0.95}$ and $\delta_{0.99}$ is the height where the flow reaches 95% and 99% of the velocity at $y = L_y$. For δ^* and θ , the integration of the velocity profile are performed from the no-slip wall to the upper wall. The mathematical formulations are given at equations (4.4) and (4.5).
- The second one is based on the maximum value reached by the flow at the horizontal speed overshoot just above the boundary layer. The height where this maximum is achieved is called δ and its evolution is depicted at figure 4.16. The $\delta_{0.95}$ and $\delta_{0.99}$ is the height where the flow reaches 95% and 99% of this maximum velocity. For δ^* and θ , the integration of the velocity profile is performed from the no-slip wall to the height where the horizontal velocity is maximal (δ). The mathematical formulations are given at equations (4.6) and (4.7).

For the first definition, let $u_{top}(x) := u(x, y = L_y)$, the δ^* and θ are computed using:

$$\delta^*(x) = \int_0^{L_y} \left(1 - \frac{u}{u_{top}(x)} \right) dy \quad (4.4)$$

$$\theta(x) = \int_0^{L_y} \left(\frac{u}{u_{top}(x)} \left(1 - \frac{u}{u_{top}(x)} \right) \right) dy \quad (4.5)$$

For the second definition, let $u_{max}(x) := \max\{u(x, y)\}$, the δ^* and θ are computed using:

$$\delta^*(x) = \int_0^{\delta} \left(1 - \frac{u}{u_{max}(x)} \right) dy \quad (4.6)$$

$$\theta(x) = \int_0^{\delta} \left(\frac{u}{u_{max}(x)} \left(1 - \frac{u}{u_{max}(x)} \right) \right) dy \quad (4.7)$$

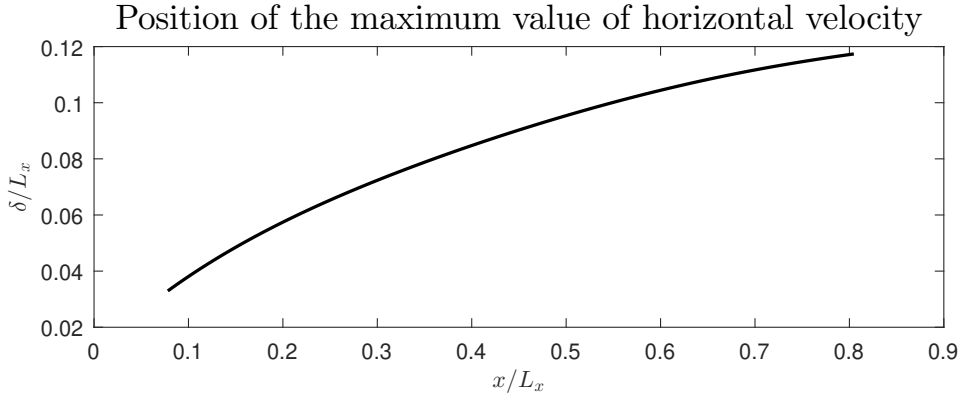


Figure 4.16: Height of the maximum value of the velocity

The solution for the boundary layer thicknesses are compared to the method proposed by Thwaites for laminar boundary layer. The method is based on the integral equation of Von Karman :

$$\frac{d\theta}{dx} + (2 + H) \frac{\theta}{u_p} \frac{du_p}{dx} = \frac{C_F}{2}$$

The following factors are define :

$$\begin{aligned} Re_\theta &= \frac{u_p \theta}{\nu} \\ K &= \frac{\theta^2}{\nu} \frac{du_p}{dx} \\ S &= \frac{C_F}{2} Re_\theta \\ F &= 2(S - (2 + H)K) \end{aligned}$$

Thwaites introduces a relation between the factor F and K using experimental data :

$$F(K) = a - bK$$

It allow the resolution of Von Karman equation and gives the following equation for the momentum thickness :

$$\frac{u_p^6(x)\theta(x)^2}{\nu} - \frac{u_p^6(x_0)\theta(x_0)^2}{\nu} = 0.45 \int_{x_0}^x u_p^5(x) dx$$

As explained above the u_p can be the maximum value of the horizontal velocity or the value reaches at the slip wall.

The numerical results for the two definitions for the thicknesses and the results of the method of Thwaites are shown at the following figures.

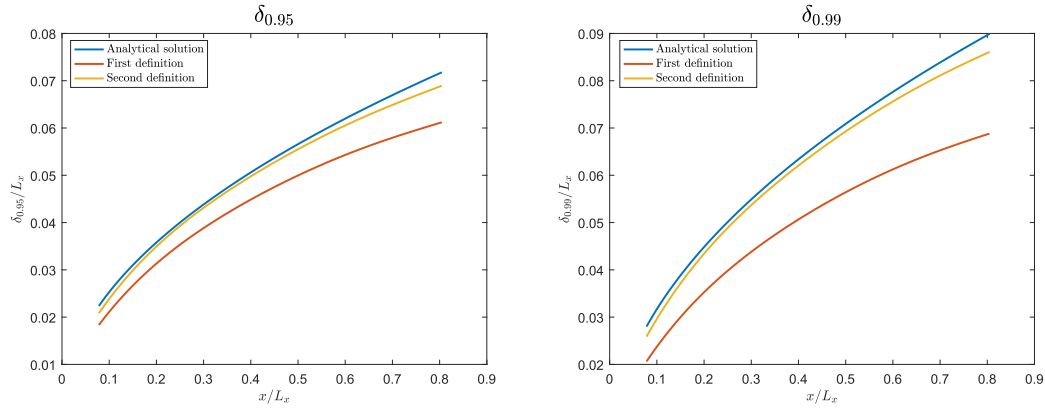


Figure 4.17: Boundary layer thickness at 95% and 99%

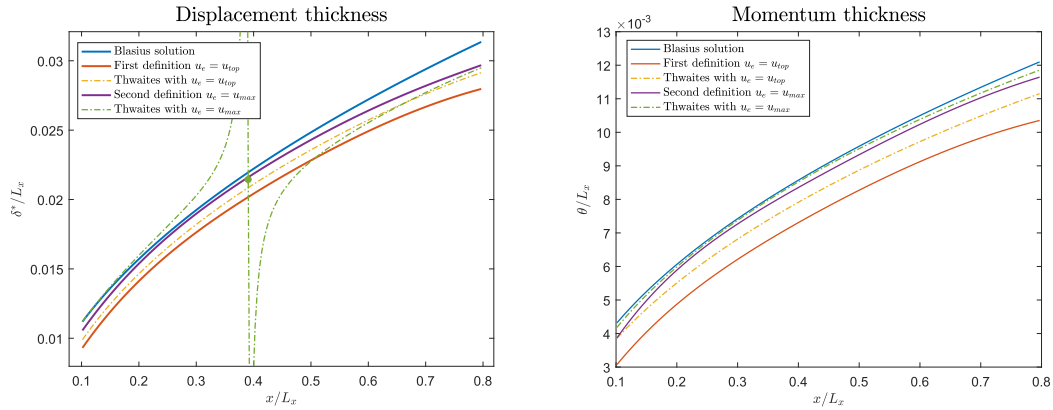


Figure 4.18: Displacement and momentum thickness

The values of the thicknesses are computed at $x/L_x = 0.8$ and compared to the analytical one:

Quantity	Analytic Value	First definition	Second definition
$\delta_{0.95}/L_x$	0.072	0.061	0.069
$\delta_{0.99}/L_x$	0.090	0.080	0.086
δ^*/L_x	0.032	0.030	0.029
θ/L_x	0.012	0.010	0.012

Table 4.3: Comparison of analytical and numerical thicknesses at $x/L_x = 0.8$

The numerical and analytical values are quite close. In the case of boundary layer development in a channel, the second definition seems to match better. The method of Thwaites

approximates pretty well the boundary layer thicknesses, the δ^* predicted by Thwaites in the case $u_p(x) = u_{max}$ tend to infinity because U_∞ possesses a minimum so $K \rightarrow 0$. The value at the minimum is defined and is represented by a dot.

In general, the values for the thicknesses are lower then the analytical one. This is easily understandable by the fact that the mass flow rate is more important in the numerical boundary layer because of the no-through flow at $y = L_y$ and the restriction of passing area caused by the boundary layer.

4.3.8 Isotropic mesh discussion

In all the previous simulations, an isotropic mesh has been used: $\Delta x = \Delta y = \Delta z$. The question still remains if an anisotropic mesh can be used in the VPM method. Due to the particle to mesh and mesh to particle step, the answer is not obvious. Two simulations have been computed in addition to the ones presented above. They are using two anisotropic meshes. The spatial step for the two simulations are: $\Delta x = 2\Delta y$ and $\Delta x = 4\Delta y$. The horizontal and vertical dimensionless velocity profiles are plotted on the same figure to study the effect of an non-isotropic mesh. The results are provided at $x/L_x = 0.4$. Other section in the streamwise direction can be found in the appendix B.

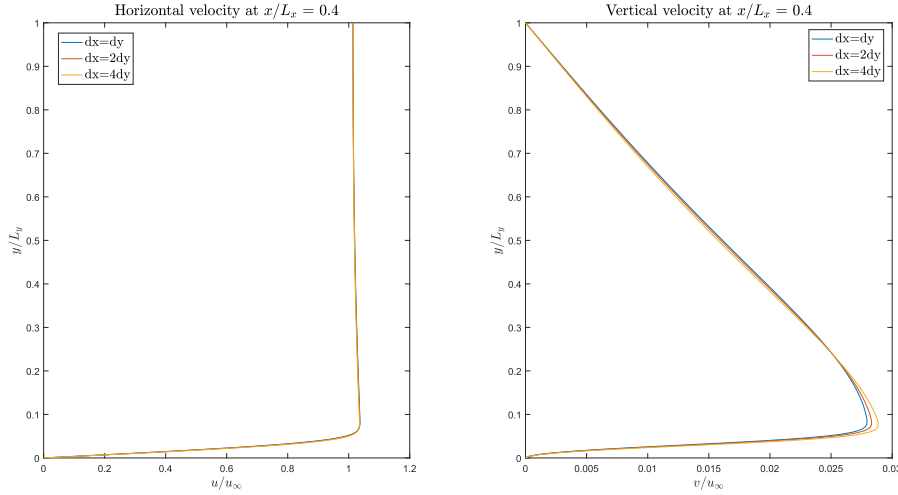


Figure 4.19: Isotropic and non-isotropic meshes, velocity profile comparison

Using a spatial step in the streamwise direction four times bigger still conserves the good resolution of the simulation: the variation on the results doesn't come from the reduction of resolution.

It is shown that only the vertical velocity component is affected by the use of anisotropic mesh. The horizontal velocity profile is the same for the three different meshes.

4.3.9 Conclusion

The implementation of the no-slip boundary condition is valid using a two-dimensional flow: the Blasius solution of the Prandtl's equation in a channel with a slip and a no-slip wall. The effects of the simulation in a channel are multiple:

- The outlet condition forces the streamlines to be normal to the outlet surface, this impacts the flow near the outlet; the flow is impacted till $x/L_x \simeq 0.8$ and no solution after can be used to validate the implementation
- The no-through flow above causes the acceleration of the flow due to restriction of the passing area due to the boundary layer. This causes an overshoot in the speed compared to the free stream velocity. The overshoot reaches 5% at the outlet.
- The vertical velocity has to go back to zero far from the boundary layer. This affects the vertical profile above the boundary layer. See section 4.3.4

Apart from these limitations, the boundary layer is pretty well captured by the method in terms of velocity and vorticity. The velocity above the boundary layer definition is not as simple as in the Blasius case because the boundary layer develops in a channel, different velocity is proposed and compared. For the boundary layer thicknesses, two definitions are proposed and compared.

The next section shows some results that can be obtained with a no-slip wall in VPM code.

Chapter 5

Test-cases

This chapter presents some results with the no-slip implementation in the VPM code. The flows presented in this section are three-dimensional flows and no analytical solution exists for them. The goal of this section is to show some results that can be obtained in a real interaction between the flow and a no-slip wall using the vortex particle-mesh (VPM) method.

The chapter 5 is organized around three main topics:

- The simulation of a vortex-ring interacting with a flat wall when the axis of the vortex-ring is initially perpendicular to the wall
- The simulation of a vortex-ring interacting with a flat wall when the axis is not perpendicular to the wall. Different initial angles are tested.
- The simulation of a turbulent channel flow at low Reynolds number: $Re_\tau = 180$.

5.1 Interaction between a vortex-ring and a no-slip wall perpendicular to the ring axis

5.1.1 Numerical setup

Initially, the vortex-ring is in the middle of the box, the no-slip walls are above and below the ring, as the channel of section 4. The boundary conditions are taken periodic in the other direction to limit the computational cost of each simulation, and hence allow to investigate various configurations within this work.

The numerical parameters are the following:

Parameter:	Value
Radius of the ring R	0.5 [m]
Radius of the tube R_c	0.25 [m]
Initial circulation of the ring Γ	1 [$\frac{m^2}{s}$]
$t_{ref} = \frac{R^2}{\Gamma}$	0.25 [sec]
Reference vorticity $\omega_0 = \frac{\Gamma}{\pi\sigma^2}$	81.5 [s^{-1}]
$N_x \times N_y \times N_z$	$512 \times 256 \times 512$
$L_x/R \times L_y/R \times L_z/R$	$6 \times 3 \times 6$
$Re_\Gamma = \frac{\Gamma}{\nu}$	2000 [-]
Dimensionless final time $\frac{t}{t_{ref}}$	14 [-]
Remesh and reprojection frequency	Every 5 and 20 time step

Table 5.1: Numerical parameters of the simulation

The simulation ran on $4 \times 4 \times 4$ processors during 9 hours (physical).

5.1.2 Mesh Reynolds number

In order to check the resolution of the numerical solution, the mesh Reynolds number based on vorticity is computed as defined by equation (2.1). The values are monitored at each time step;

their evolution is the following:

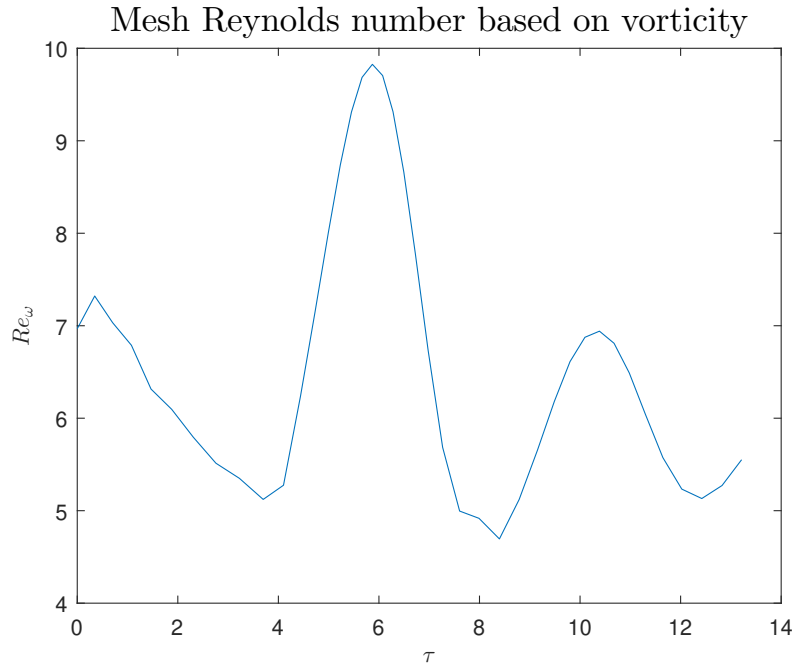


Figure 5.1: Time evolution of mesh Reynolds number based on vorticity for the simulation

The two spikes correspond to the emergence of a secondary and tertiary vortex-ring as explained later. The value is very small, the simulation is highly resolved.

The nested sections present the time evolution of the interaction between the vortex-ring and the no-slip wall.

5.1.3 Numerical results and discussions

The first slice is in the $x - y$ plane so only the $\hat{e}_z$ component of the vorticity is important.

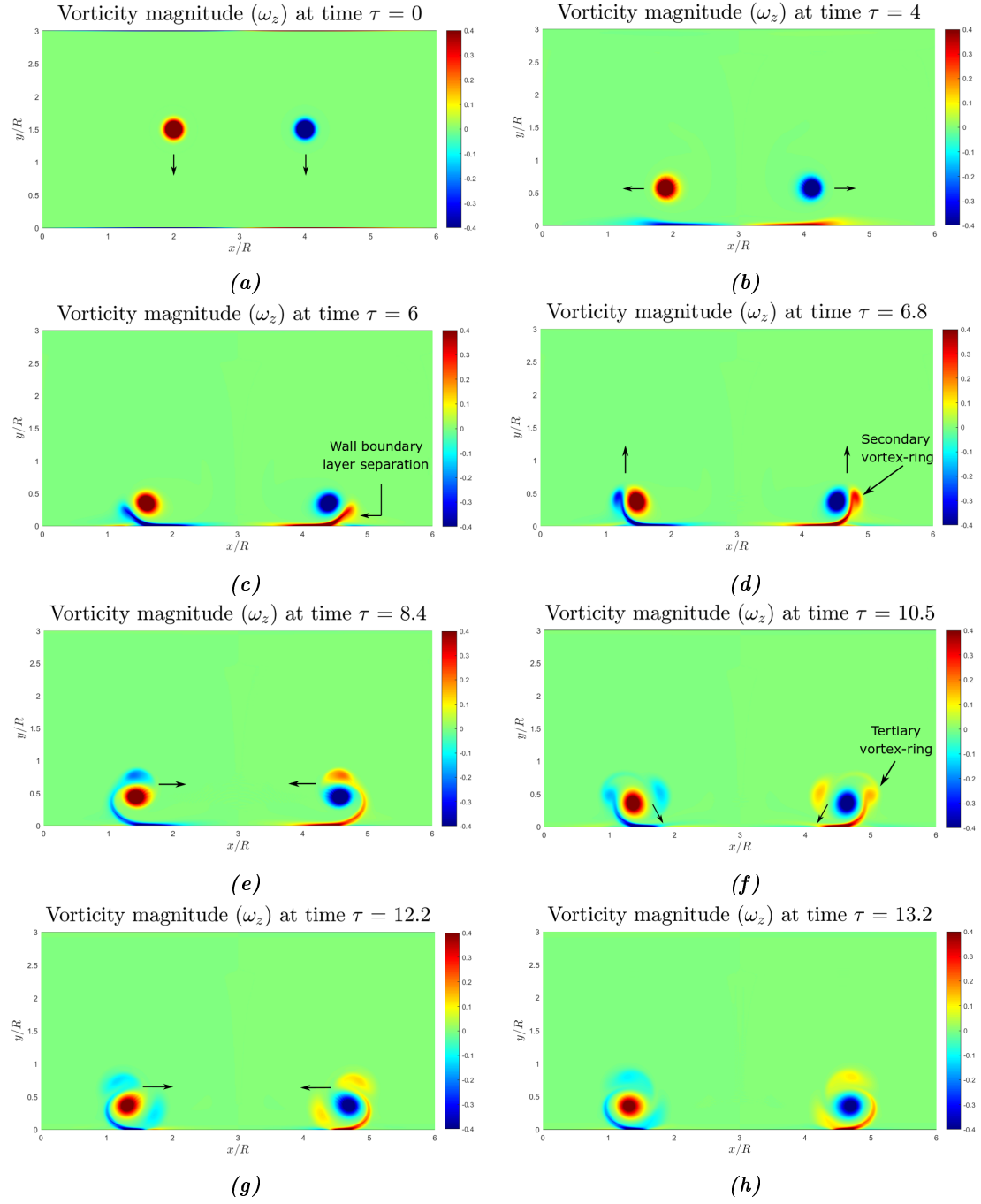


Figure 5.2: Dimensionless vorticity evolution $\frac{\omega}{\omega_0}$

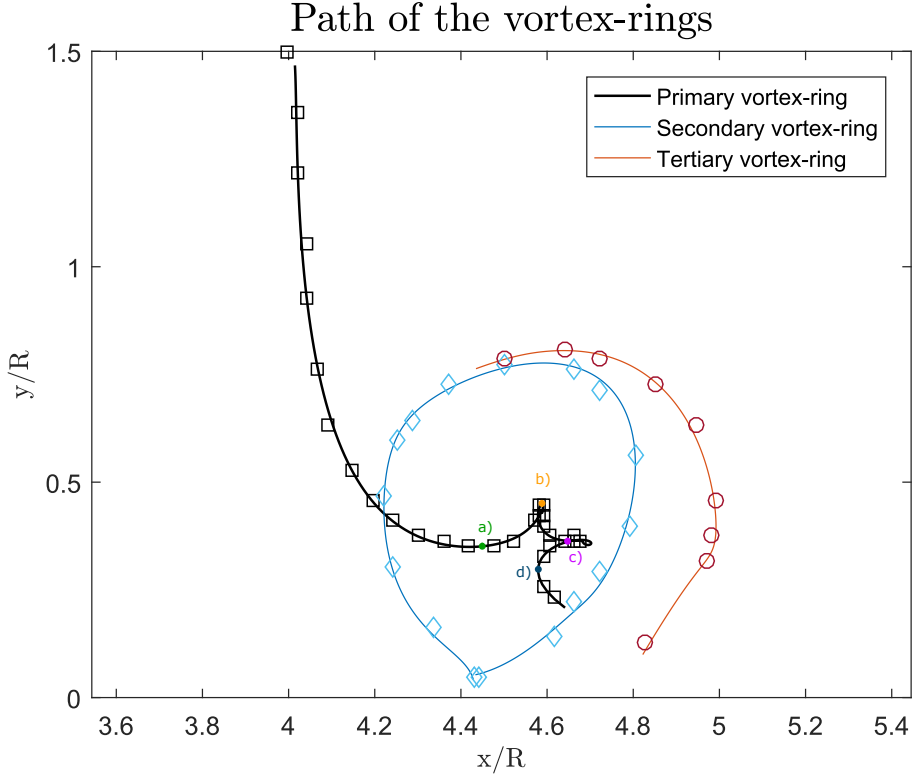


Figure 5.3: Path of the vortex-rings the continuous lines are curves-fitting of measured data

The paths followed by the different vortex-rings constitute a complex phenomenon between the separation of the boundary layer and the induced velocity of vortex-rings. The explanations of the vortex-rings dynamics are presented using the figures 5.2 and figure 5.3.

Initially, the primary vortex-ring starts its motion to the wall because of its induced velocity. Figure 5.2b shows how the vortex-ring's radius increases as it spreads radially outwards when it approaches the wall. During the ring's approach, a boundary layer has developed at the wall because of the no-slip condition. Before $\tau \simeq 5$, the boundary layer remain attached, but, after that, an adverse pressure gradient slows down the flow and the boundary layer separation occurs (figure 5.2c and 5.5). The separation of the boundary layer occurs at the point *a*) on figure 5.3; this generates a new vortex-ring (blue curve on same figure). The new pair of counter-rotating vortices forces the primary vortex-ring to get away from the wall. The path of the primary vortex-ring seems to bounce on the wall. It happens from point *a*) to point *b*) (visible on figure 5.2d). At point *b*), the secondary vortex-ring is above the primary one. It causes the primary vortex-ring to remain globally stationary because the interaction with the wall tends to spread radially outwards the ring, but the pair of counter vortices tends to move towards the collision axis as shown at figure 5.2e. The secondary vortex-ring is less intense than the primary one, so it turns around the primary vortex-ring core, that causes the descent to the wall and the radially outwards spreading of primary vortex, figure 5.2f. At the figure 5.2f, a tertiary vortex-ring appears still because of the boundary layer detachment, this causes the primary vortex-ring to move towards the collision axis, it can be observed on the path of the primary vortex at point *c*). Finally, at point *d*), the tertiary vortex-ring wraps the primary one and it spreads radially outwards at the same time as it approaches the wall.

To conclude the study of the dynamics of vortex-rings, the trajectory of a vortex-ring interacting with a slip flat ground is computed and compared to the trajectory of the primary vortex-ring mentioned above. Without the no-slip condition, no boundary layer exists, so no separation occurs and no secondary and tertiary vortex-ring appear. The vortex-ring just spreads radially outwards. The figure 5.4 shows the comparison between the trajectories of a vortex-ring interacting with a wall with and without the no-slip boundary condition. At the beginning, the trajectories are the same, but at $y/R \simeq 0.5$, the primary vortex-ring with the no-slip wall interacts with the boundary layer and the two trajectories split.

Finally it is important to note that periodic boundary conditions were used in this simulation. It is highly likely that this affect the vortex dynamics.

The total circulation in the $x - y$ plane is computed and the evolution is shown at the figure 5.7.

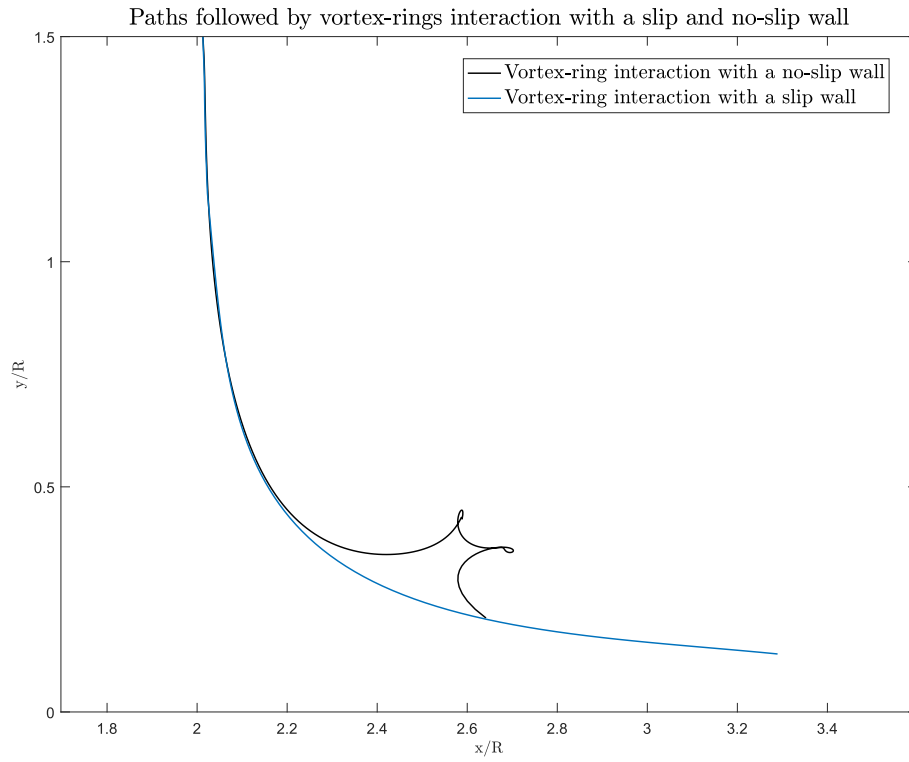


Figure 5.4: Path comparison of a wall vortex-ring interaction with and without no-slip boundary condition

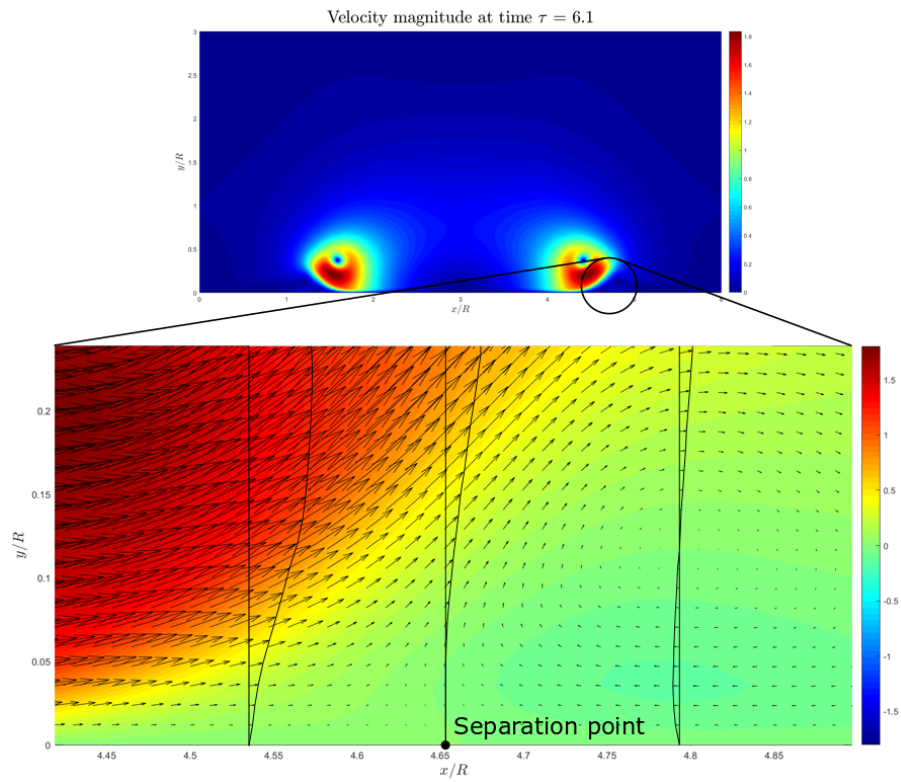


Figure 5.5: Boundary layer separation at $\tau \simeq 5$

The three dimensional results are shown in the following figures. The primary, secondary and

tertiary vortex-rings are also depicted.

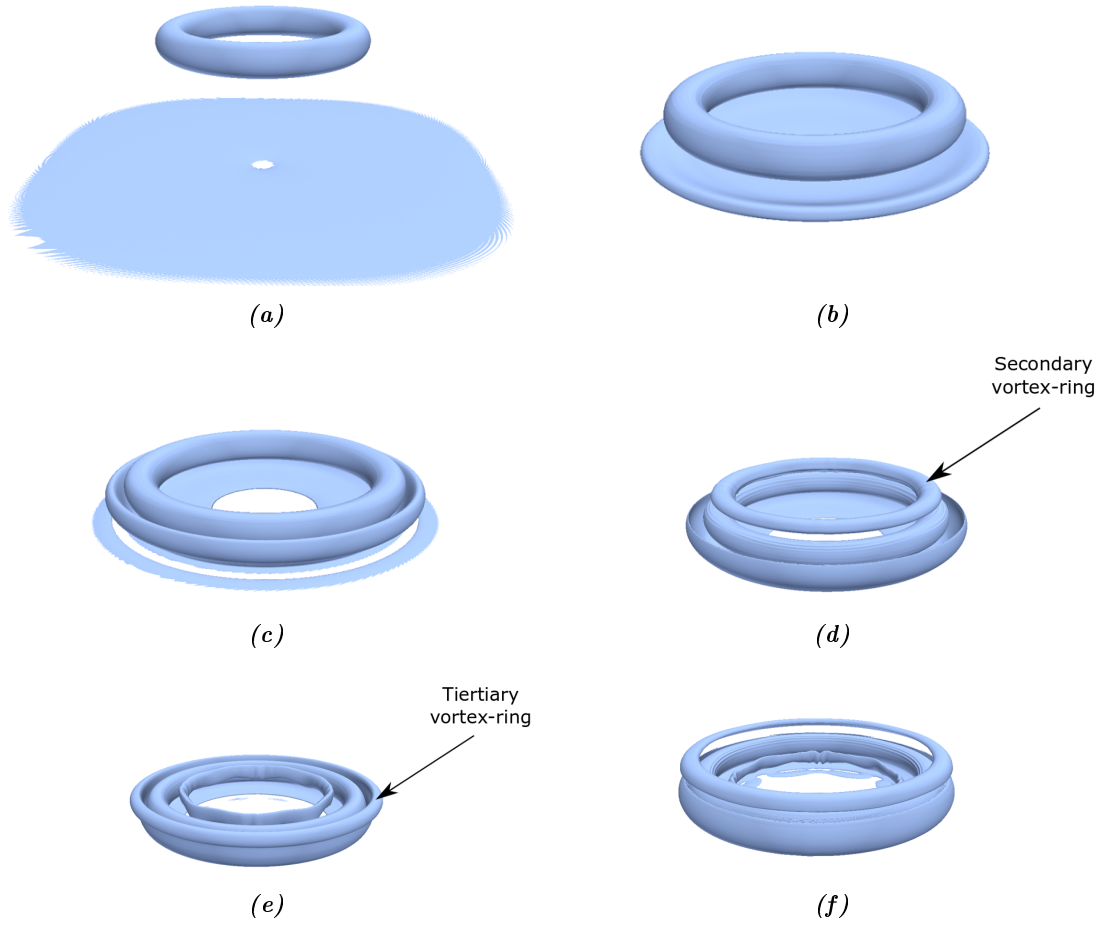


Figure 5.6: Iso-surfaces of dimensionless vorticity magnitude

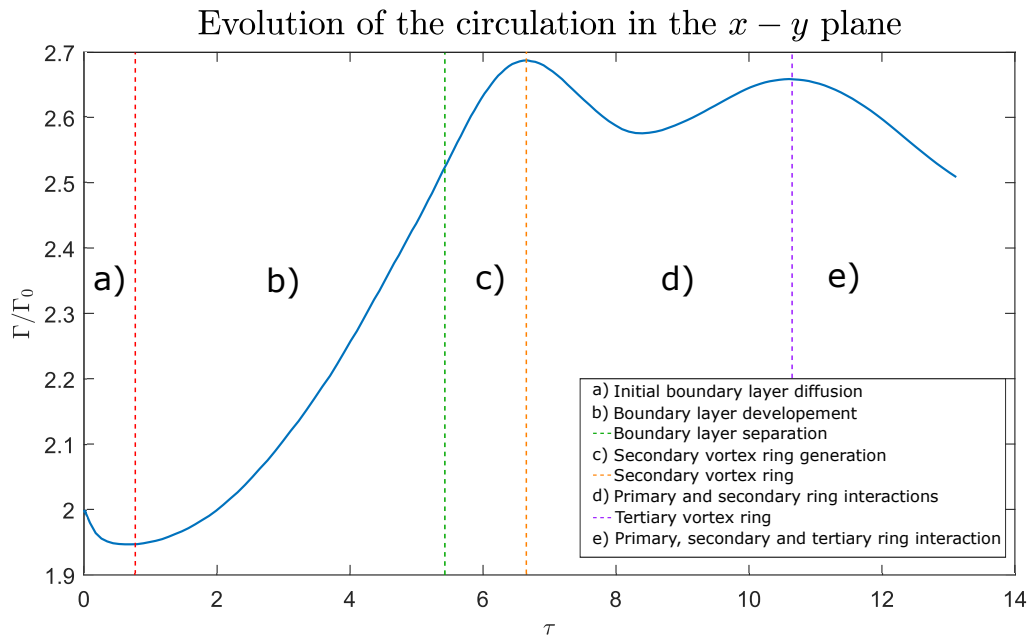


Figure 5.7: Evolution of the absolute value of circulation

5.1.4 Conclusion

The interaction between a vortex ring and a no-slip wall was studied in this section. The trajectory of the vortex-ring is the result of complex interactions between the ring and the boundary layer generated by the no-slip condition. It is shown that secondary and tertiary vortex-rings appear caused by the separation of the wall boundary layer. The second and tertiary vortex-rings have a huge impact on the path followed by the initial vortex-ring. The main impact is to stop the outwards radial spreading of the initial vortex-ring. To conclude the study, the path followed by a vortex ring with a slip flat ground is computed and compared to the path followed by the initial vortex-ring. It is shown that the first path only spreads radially outwards while the second stops due to the presence of secondary and tertiary vortex-rings.

5.2 Interaction between a vortex-ring and a no-slip wall with an initial angle relatively to the ring axis

5.2.1 Numerical setup

The boundary conditions, numerical setup and ring's parameters are the same as before.

5.2.2 Mesh Reynolds number

The mesh Reynolds number is also computed and monitored to ensure the good resolution of the simulation:

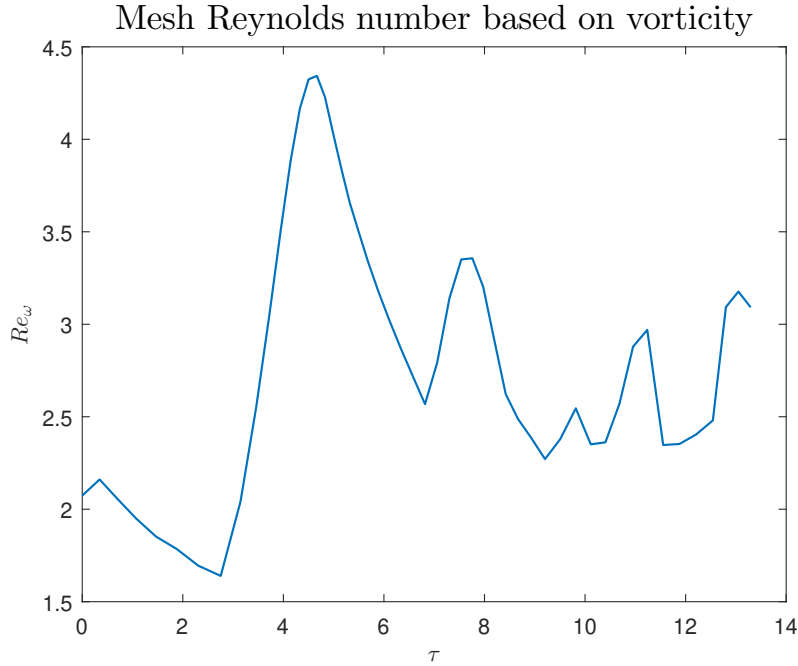


Figure 5.8: Time evolution of mesh Reynolds number based on vorticity for the simulation at an angle of 20°

The value is very small, the simulation is very highly resolved.

5.2.3 Numerical results and discussion

The interaction between a vortex-ring and a no-slip wall with an initial angle is mostly a three dimensional problem while the previous present axisymmetry. The case examined in this section¹ is the one having an initial angle of 20° . The initial condition is the following:

¹Other initial angles were simulated, the three dimensional results are presented in appendix C

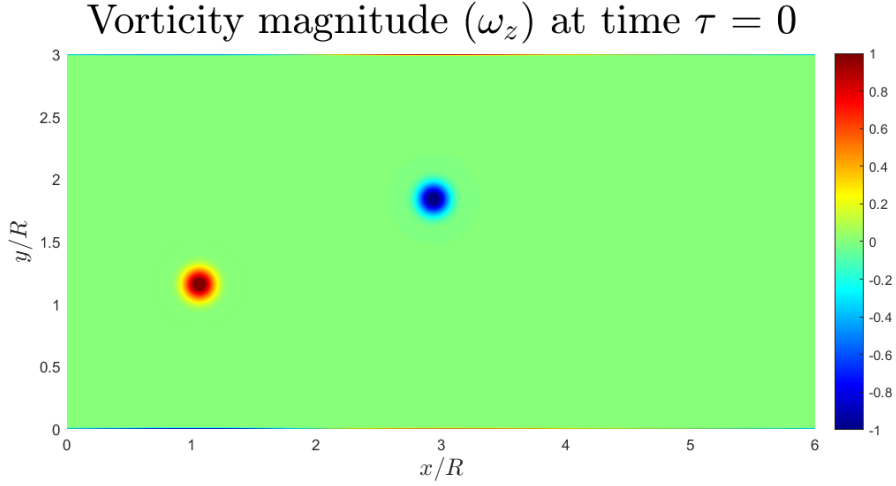


Figure 5.9: Initial condition for the wall ring interaction

The ring starts its motion because of its induced velocity and approaches the no-slip wall. An initial vortex sheet is present because of the vortex-ring induced velocity in the entire computational domain. After some time, the lower part of the vortex-ring reaches the wall so the boundary layer split occurs and a secondary vortex-ring appears. The secondary vortex-ring starts at the lower part of the primary vortex-ring while the boundary layer is still attached at the upper part. A second boundary layer separation occurs leading to the creation of a tertiary vortex ring. At the same time, the boundary layer at the upper part of the primary vortex-ring splits and the secondary vortex-ring is now complete. The secondary vortex wraps the primary one and starts generating Ω – loops instabilities at the lower part. It is shown at the figure 5.11. These Ω – loops are caused by the retraction of the secondary vortex-ring when it wraps the primary one and goes inside it. The center line is compressed, and finally it bends. This creates the Ω shape of the instabilities. These generate other instabilities that go up along the vortex-ring center line and destroy the vortex-ring structure.

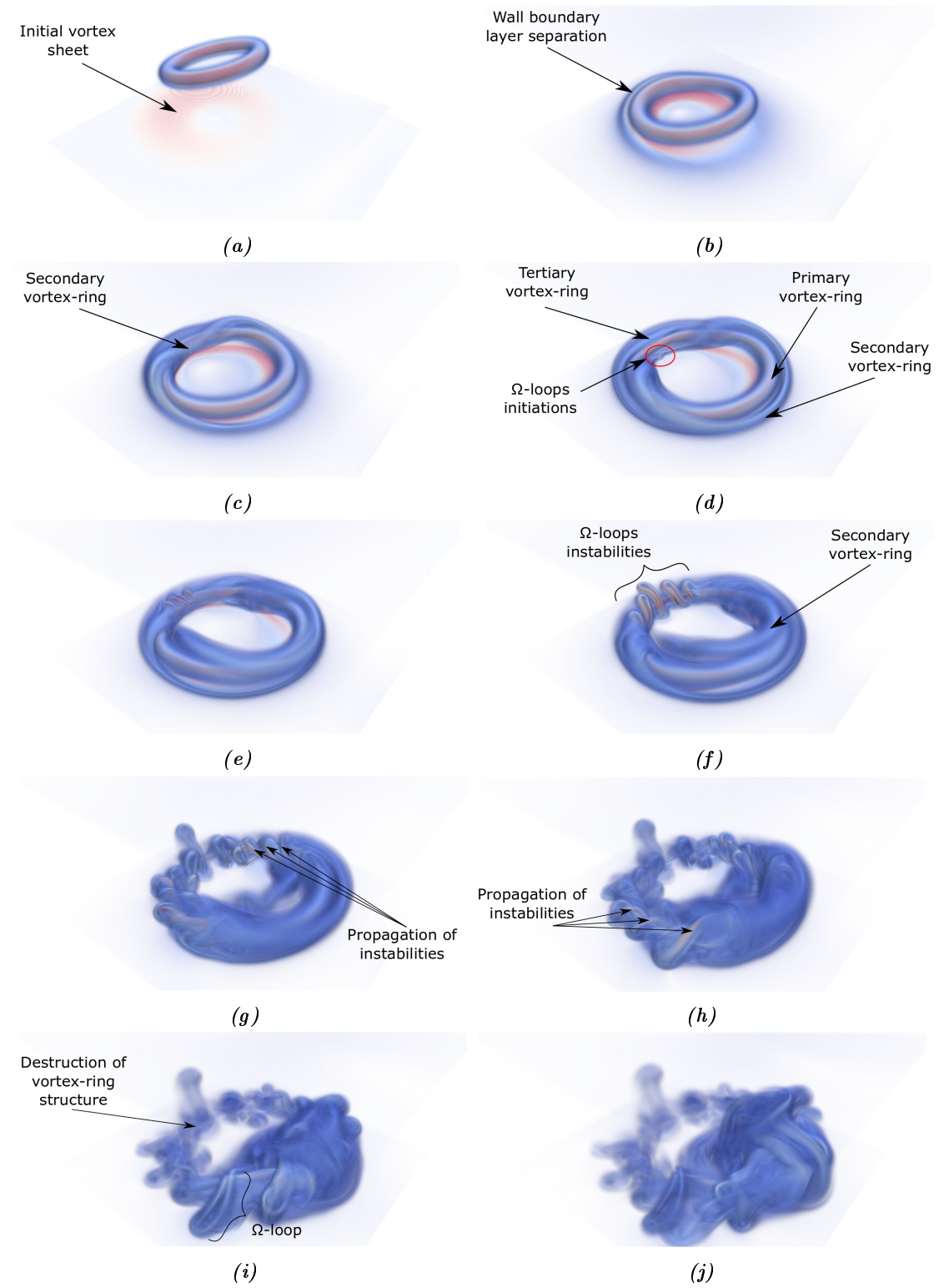


Figure 5.10: Volume rendering of vorticity magnitude

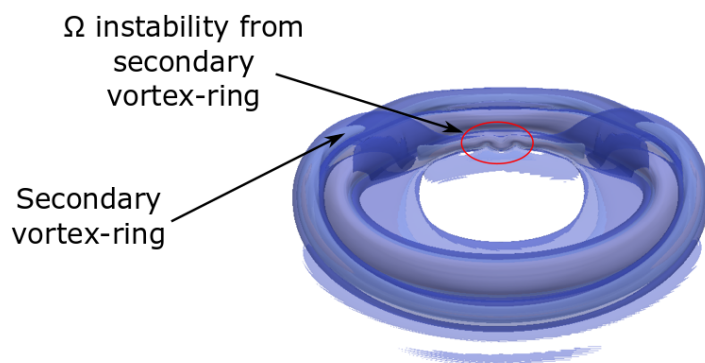


Figure 5.11: Ω – loops initiation by the secondary vortex-ring

Depending on the initial angle of attack, the Ω –loops instabilities appear at different moments and the numbers are different. Several initial angles were simulated and the comparison of the Ω –loops is shown at the figure 5.12. The higher the angle of attack, the sooner the instabilities appear, but in smaller numbers. The vortex ring-core collapses faster at high initial angle. At $\alpha = 40^\circ$, two main Ω –loops are dominant while at $\alpha = 10^\circ$, 12 loops appear before the destruction of the vortex ring core.

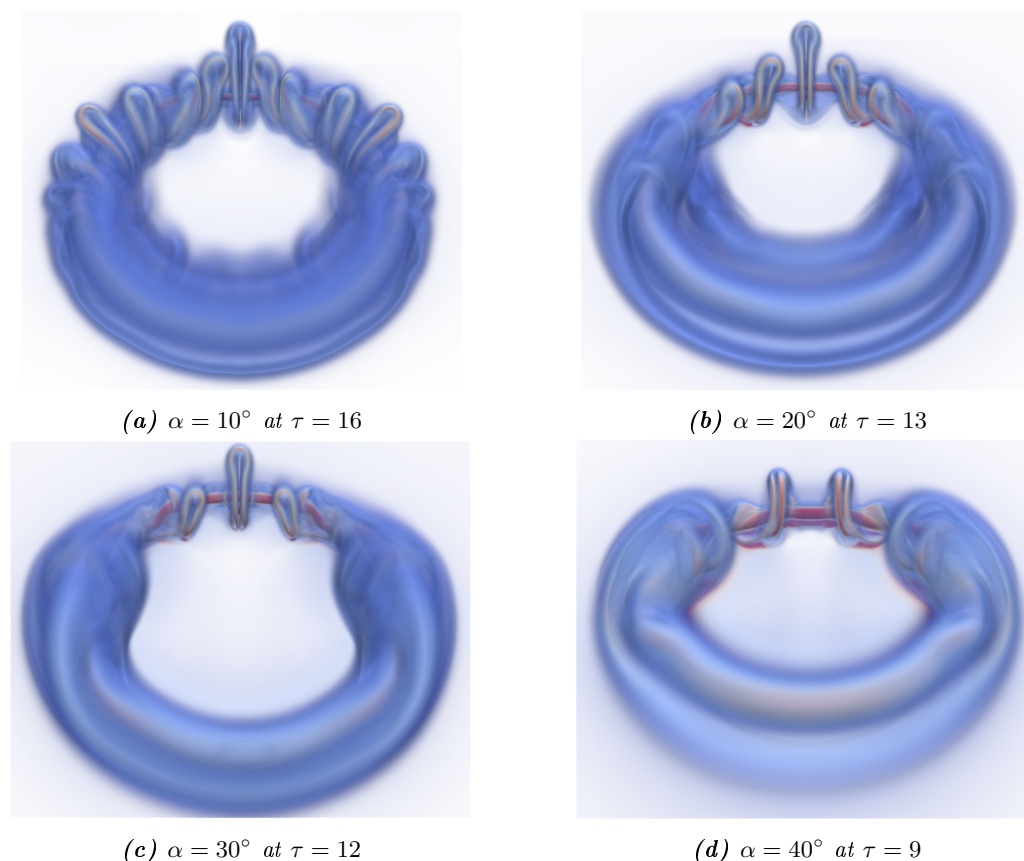


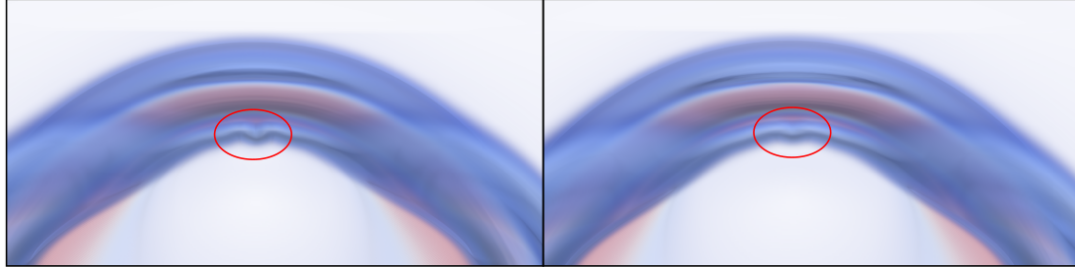
Figure 5.12: Ω – loops for different initial angles of attack

5.2.4 Origins of instabilities discussions

The interaction between a vortex ring and a no-slip wall generate Ω –loops instabilities. These are physical instabilities but the point is to know if these Ω –loops are not triggered by the numerical code. Indeed, the vortex-ring is circular structure and the mesh used is square, thus numerical instabilities could be at the origin of the Ω –loops. The simulation is run again with a finer mesh of a factor two in the three directions. The simulation is performed till the instabilities

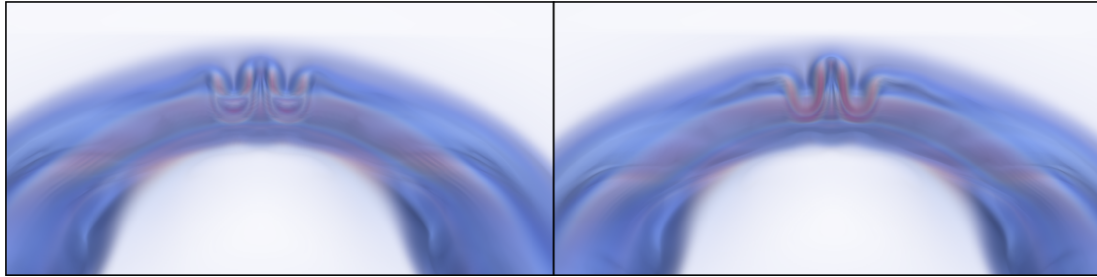
appeared and the results are compared to the one obtained above. If the Ω – loops begins at the same time, same place and have the same aspect, their apparitions are due to the physics and not the numerical code.

Different results are compared together at exactly the same time. The results with the mesh used at the previous section are on the left and the results with the mesh twice finer are on the right.



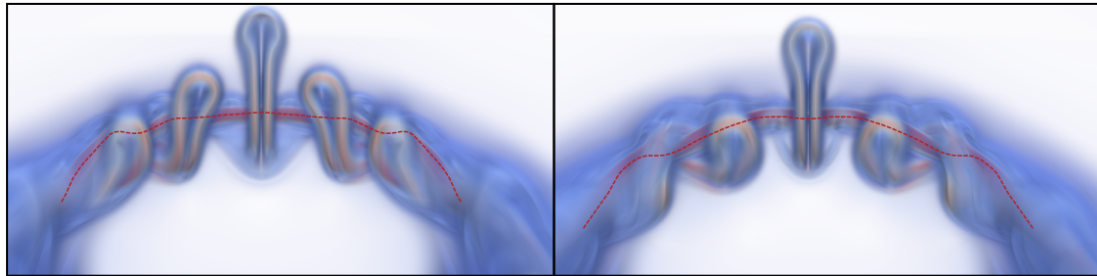
(a)

Figure 5.13: $\tau = 8$



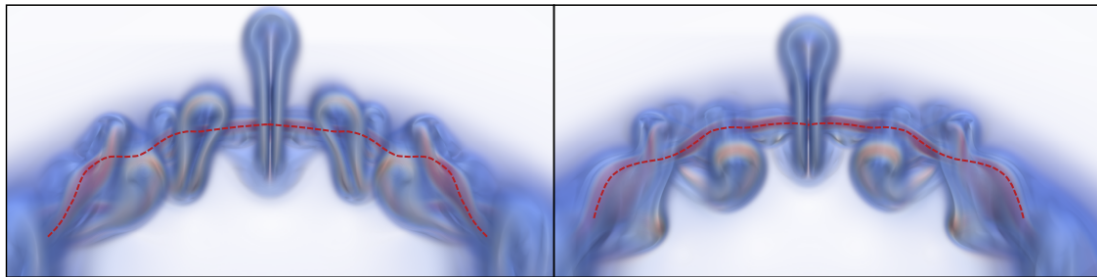
(a)

Figure 5.14: $\tau = 10$



(a)

Figure 5.15: $\tau = 13$



(a)

Figure 5.16: $\tau = 14$

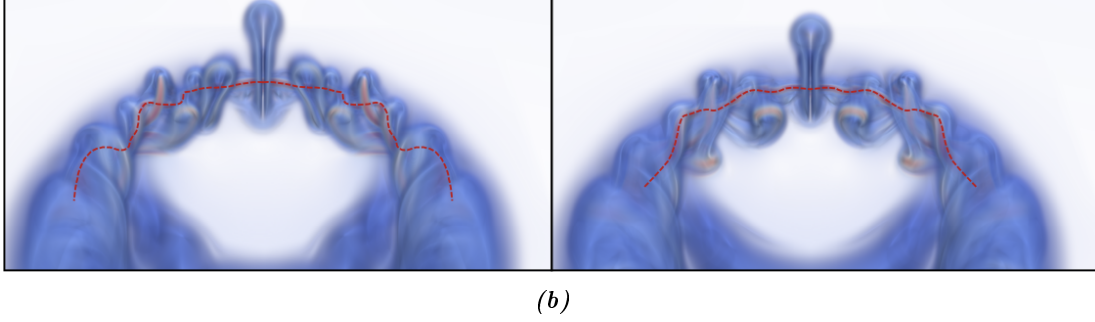


Figure 5.16: $\tau = 15$

The Ω – loops appear at the same time and the same place in both simulation, the results are not far from each other, it can be concluded that these are physical and not numerical instabilities. Some differences can be however observed between the two simulations. The instabilities appear to be bigger with the coarser mesh and the vortex-ring core is perturbed sooner.

5.2.5 Conclusion

The interaction between a vortex-ring and a no-slip wall is shown at this section. The vortex ring interaction generates Ω – loops instabilities and generate other instabilities that go up along the vortex-ring center line and destroy the vortex ring structure.

The instabilities origin can be from physical or numerical origin. The simulation is performed again on a thinner mesh and the results are compared at the same time. The two results were very close to each other, it can be concluded that the instabilities are well physical and not numerical.

5.3 Turbulent channel flow at $Re_\tau = 180$

5.3.1 Introduction

In order to close this work, the simulation of the well known turbulent channel flow at moderate Reynolds number is studied. The Reynolds number based on the half channel height and mean flow velocity is $Re_\delta = 2800$ and the Reynolds based on the friction velocity is $Re_\tau = 180$. The results are compared to the DNS results of Myoungkyu Lee and Robert D. Moser [5]. The implementation of the no-slip boundary condition in the VPM method is strongly validated at the end of this chapter.

5.3.2 Turbulent channel flow characterisation

A fully developed turbulent flow can be described by two components: a mean velocity profile independent of the time and a fluctuation that depends on the moment the flow is observed:

$$u(t) = \bar{u} + u'(t) \quad (5.1)$$

$$v(t) = \bar{v} + v'(t) \quad (5.2)$$

$$w(t) = \bar{w} + w'(t) \quad (5.3)$$

For the three components of the velocity, the following relations are valid:

$$\begin{aligned} \bar{u} &= \frac{1}{T} \int_0^T u(y, t) dt \\ \bar{v} &= \frac{1}{T} \int_0^T v(y, t) dt = 0 \\ \bar{w} &= \frac{1}{T} \int_0^T w(y, t) dt = 0 \end{aligned}$$

So, the fluctuations have a zero time average. The spanwise and transverse velocity have just a fluctuation term.

In the turbulent channel flow of this section, the mass flow is imposed. It is computed using the using the mean flow velocity :

$$u_m = \frac{1}{\delta} \int_0^\delta \bar{u}(y) dy$$

$$Q = 2\rho\delta u_m$$

The Reynolds number based on the half channel width and on the mean velocity is defined as:

$$Re_\delta = \frac{u_m \delta}{\nu} = \frac{Q}{2\nu}$$

The value is taken to $Re_\delta = 2800$. Another way to characterise the turbulent channel flow is to compute the Reynolds based on the friction velocity. The friction velocity is a form by which the wall shear stress is expressed as a velocity. The friction velocity is computed after the simulation, its expression is based on the wall shear stress:

$$u_\tau = \sqrt{\frac{\tau_w}{\rho}}$$

$$\tau_w = \mu \frac{\partial \bar{u}}{\partial y} \Big|_{y=0}$$

In the simulation, its value is $\frac{u_\tau}{u_m} = 0.064$. Finally, the Reynolds number based on this velocity is mostly used to characterise a turbulent channel flow, its definition is the following:

$$Re_\tau = \frac{u_\tau \delta}{\nu}$$

The dimensionless variables such as velocity and length in the channel are computed using the friction velocity, the half height of the channel δ and the dynamic viscosity ν . It gives the dimensionless velocity and dimensionless wall distance called $\bar{u}^+$ and y^+ . The results expressed in this dimensionless form only depend on the Re_τ . This is very useful in order to compare the result provided by the VPM method with other results already done and validated.

$$y^+ = \frac{u_\tau y}{\nu}$$

$$\bar{u}^+ = \frac{\bar{u}}{u_\tau}$$

5.3.3 Numerical setup

The channel flow is a complex three-dimensions flow. Two directions are set with periodic boundary conditions and one direction is set with no-slip boundary condition:

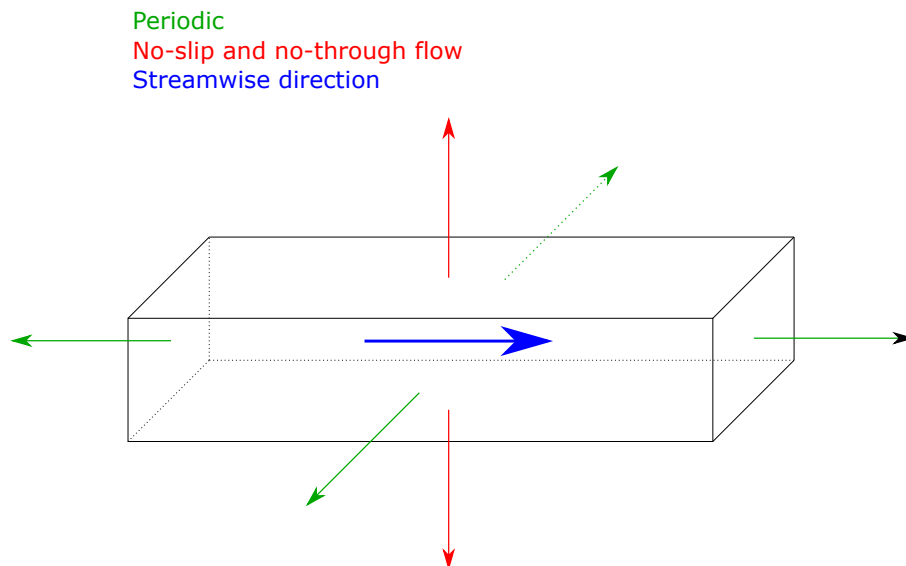


Figure 5.17: Boundary conditions on the channel

The dimensions of the channel used to compute the turbulent channel flow is shorter in the streamwise and spanwise direction than the one simulated by Moser and Lee. The reason of that is the fact that the mesh in VPM isn't stretched. To accurately capture the laminar region, many grid points have to be placed in the entire computational domain. Another reason, briefly mentioned at the end, is the fact that VPM seems to become inaccurate if the mesh isn't isotropic. As shown further down, the results of the simulation remained quite good, but some differences appear when statistics on the flow are computed.

The spatial discretisation is given in dimensionless wall distance and the mesh is isotropic.

$$h^+ = \frac{hu_\tau}{\nu}$$

Re_τ	L_x/δ	L_y/δ	L_z/δ	N_x	N_y	N_z	h^+
180	6	2	4	768	256	512	1.4

Table 5.2: Numerical parameters of the turbulent channel flow simulation

There is 5-6 mesh points in the laminar region.

The transitional Reynolds number from laminar to turbulent channel flow is about $Re_\delta \simeq 7600$. So, to start the turbulent flow at this Reynolds number, the initial condition must be a very perturbed Poiseuille flow. The perturbation is in the streamwise and wall-normal direction:

$$\begin{aligned}\omega_x &= 0 \\ \omega_y &= 0 \\ \omega_z &= -\frac{3U_\infty}{\delta^2}(\delta - y) + A\left(\omega_1 \cos(\omega_1 y) + \omega_2 \cos(\omega_2 y)\right) \sin(\omega_3 x)\end{aligned}$$

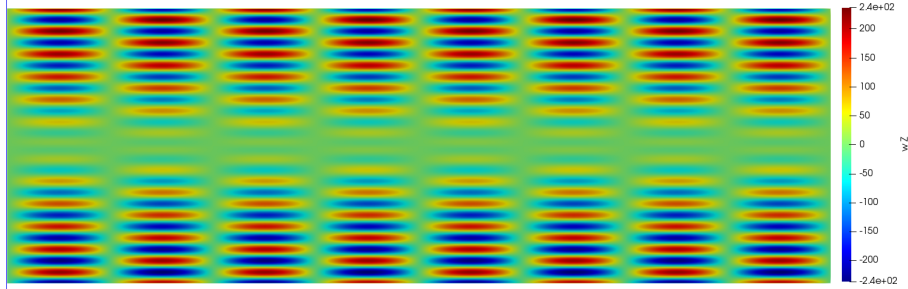


Figure 5.18: Initial perturbation on vorticity field

It leads to the following velocity field:

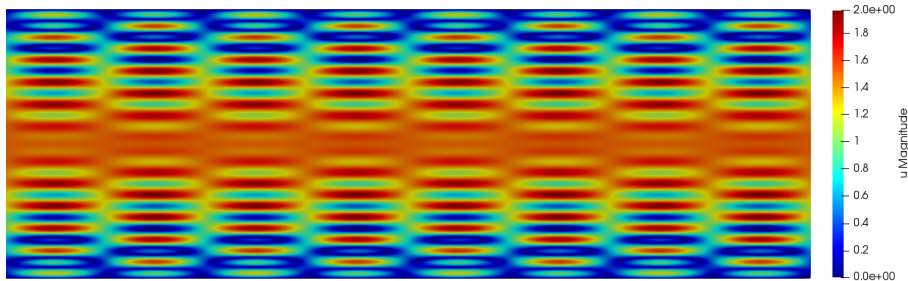


Figure 5.19: Initial perturbation on velocity field

5.3.4 Numerical results and discussions

The flow undergoes a transitional phase during $\tau \simeq \frac{t}{t_{ref}} = 100$, with $t_{ref} = \delta/u_\tau$. After that, the flow is fully turbulent and the statistics can be performed.

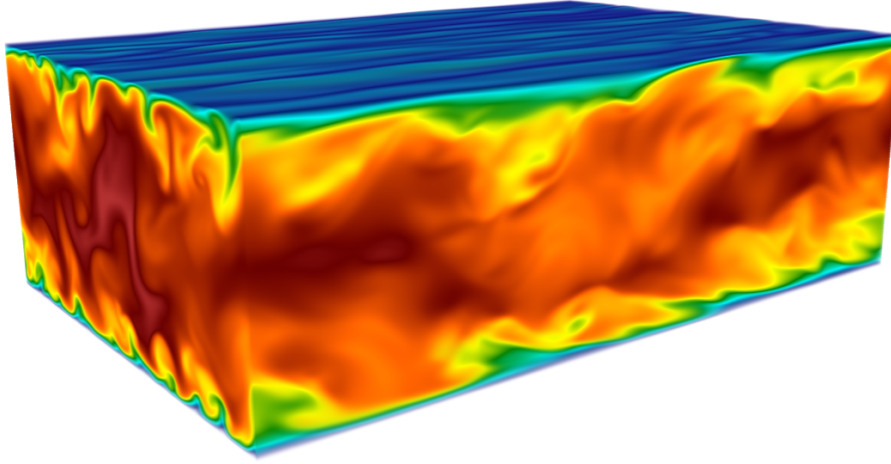


Figure 5.20: Fully developed turbulent flow

As the flow is three dimensional and periodic in two directions, the mean velocity profile is computed using spatial averages in the streamwise and spanwise directions and temporal average on the time interval of $\tau = 100 \rightarrow 200$. It provides $N_x \times N_z \times N_t$ velocity profile along the wall-normal direction and so very stable mean velocity profile. To validate the turbulent channel flow, the mean velocity profile is compared to the one computed by Lee and Moser [5]

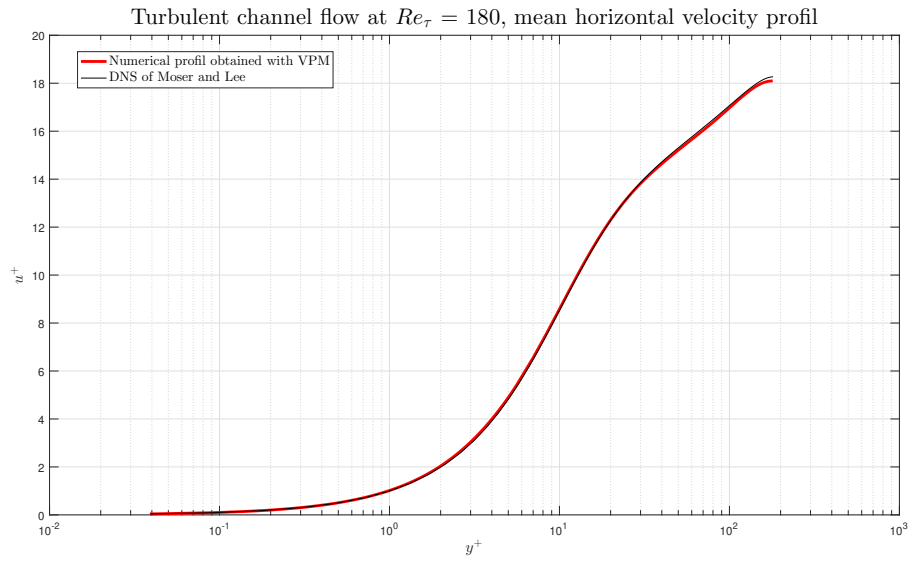


Figure 5.21: Dimensionless velocity profile compared to the one computed by Moser and Lee

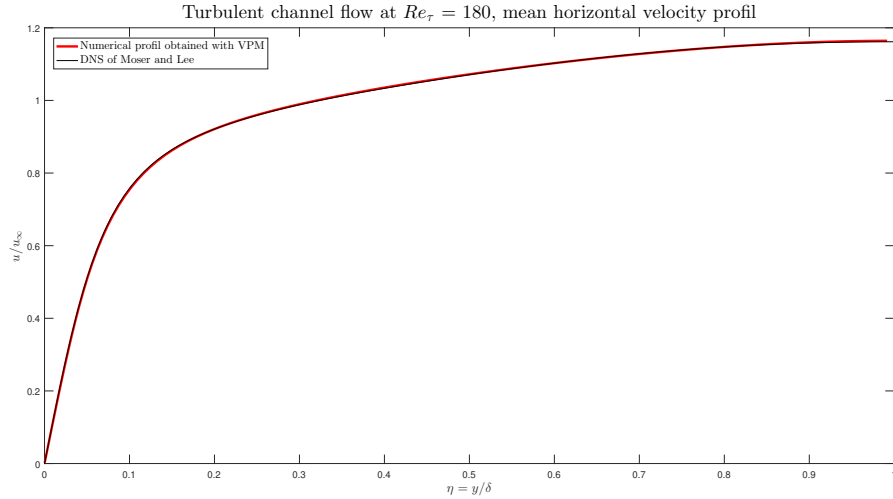


Figure 5.22: Velocity profile compared to the one computed by Moser and Lee

The mean velocity profile is very close to the one obtained by Moser and Lee, a difference can be observed at the center of the channel but it doesn't exceed 1.1% of the u_m .

Other quantities such as root mean square values of the velocity profiles can be computed. The root mean square (RMS) value is computed based on the variance of the fluctuation term :

$$\begin{aligned}\langle u'u' \rangle &= \frac{1}{T} \int_0^T (u(t) - \bar{u})^2 dt \\ &= \frac{1}{T} \int_0^T (u'(t))^2 dt \\ u_{rms} &= \sqrt{\langle u'u' \rangle}\end{aligned}$$

The RMS values are computed for the three components of velocity and compared to the ones obtained by Lee and Moser. Based on theses results, the turbulent kinetic energy is computed:

$$k = \frac{1}{2} (\langle u'u' \rangle + \langle v'v' \rangle + \langle w'w' \rangle)$$

The RMS values are quite good for spanwise and transverse velocity. For the streamwise velocity, the value is good for low value of y^+ but deviates at the center of the channel of about 30%. The differences may come from the fact that the channel is shorter in the streamwise direction than the one simulated by Moser and Lee. It leads to some constraints of the turbulent flow and lowers the accuracy of the simulation.

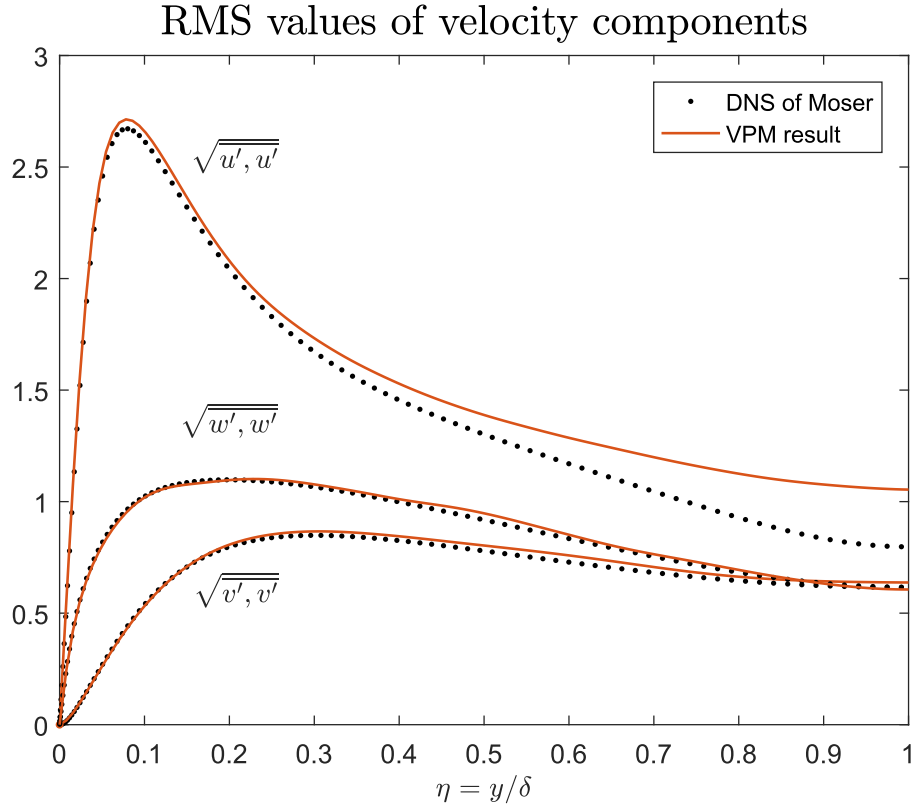


Figure 5.23: RMS values of velocity components

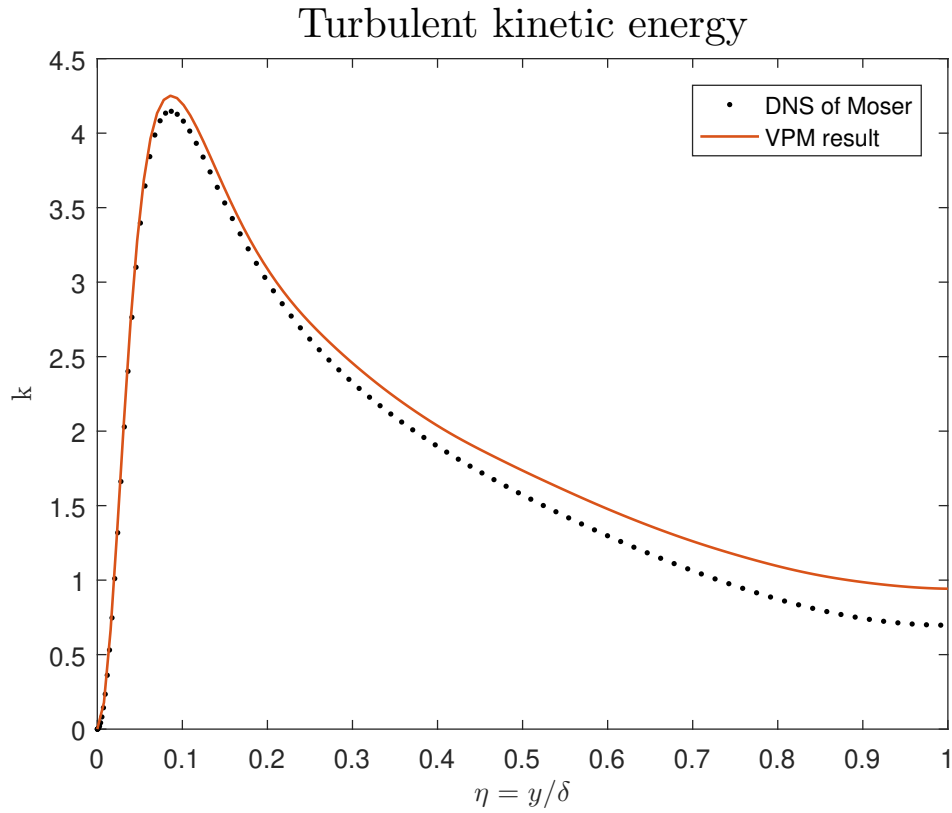


Figure 5.24: Turbulent kinetic energy

Finally the effective turbulent stress profile and molecular stress profile are computed and are compared to those obtained by Moser and Lee. The mathematical definitions of the two

quantities are respectively :

$$\text{Turbulent shear stress : } -\frac{\overline{u'v'}}{u_\tau^2}$$

$$\text{Molecular stress : } \nu \frac{d\bar{u}/dy}{u_\tau^2}$$

The sum of the two is the total shear stress in the turbulent flow and should be a strain line.

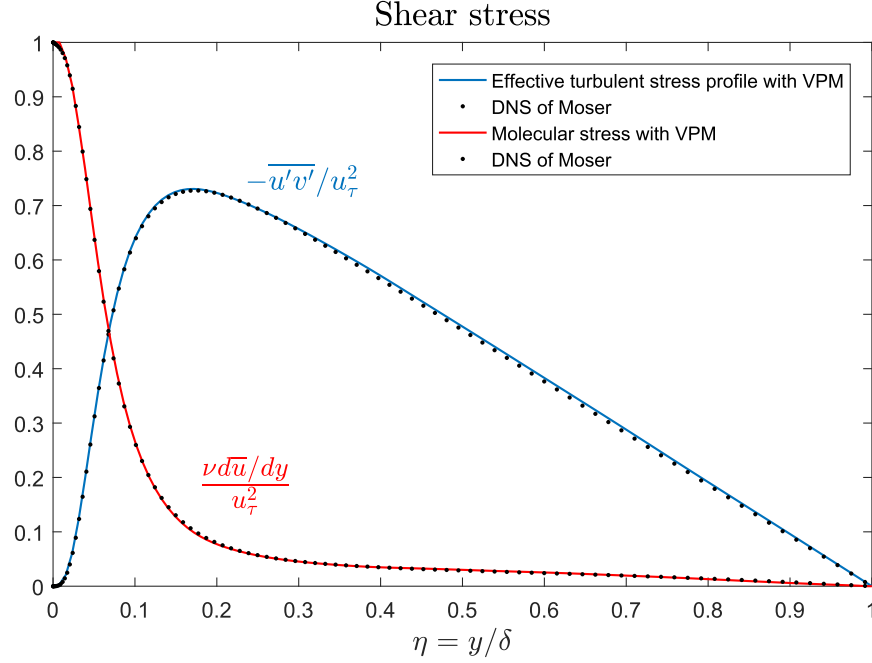


Figure 5.25

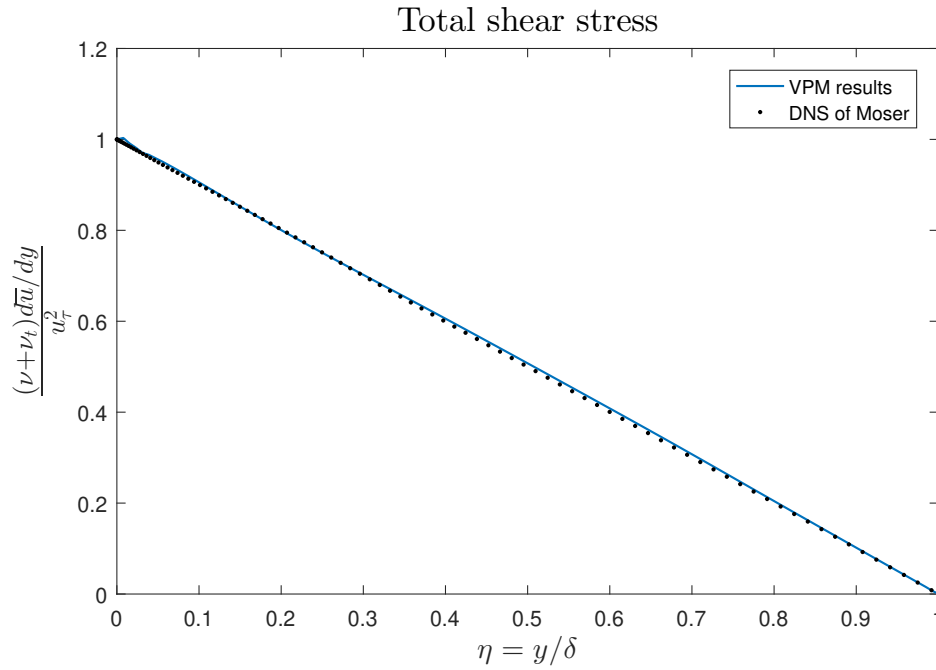


Figure 5.26

To summarize the analysis of the results, different values on the turbulent channel flow are shown in the following table:

Value	VPM results	Moser results
Streamwise domain size	6δ	25δ
Spanwise domain size	4δ	9δ
Re_τ	179.72	182.09
u_τ [m/s]	0.0642	0.0637
u_c^+	18.27	18.10
$u_{rms}^+ _{peak}$	2.71	2.67
$v_{rms}^+ _{peak}$	0.85	0.85
$w_{rms}^+ _{peak}$	1.09	1.10
$k^+ _{peak}$	4.26	4.15

Table 5.3: Numerical parameters of the turbulent channel flow simulation

The value for the mean spanwise, transverse velocity and covariance between velocity component ($\langle u'w' \rangle$, $\langle v'w' \rangle$) should be zero due to statistical symmetries in the flow. These are plotted at appendix D nevertheless as an indicator of the statistics quality.

5.3.5 Conclusion

The low Reynolds turbulent channel flow is a well known and classical flow in computational fluid dynamics. The section presents the results that can be obtained for such a flow in the VPM code with the no-slip boundary conditions. All the results are compared to the DNS of Moser and Lee at the same Reynolds number.

The turbulent channel flow using the no-slip boundary condition in the VPM method very accurately captures the mean horizontal velocity profile. The RMS values are also computed but, due to the shorter size of the channel than the one used by Moser and Lee, these values are different at the center of the channel. Finally the covariance between the velocity components are computed and shown in the appendix as an indicator of the statistics quality.

The next step of this section would be to run the simulation using a channel twice as large and to recompute the RMS values.

Chapter 6

Conclusion

As a result of this thesis, improvements have been implemented in the vortex particle-mesh method to enhance the processing of no-through flow and a no-slip boundary conditions. These modifications were implemented with the ambition to model the interaction between a real flow and a no-slip wall, and to maybe one day study the impact of a flat ground with a wake of a wind-turbine or a helicopter when it is taxiing. The boundary condition is expressed in terms of a vortex-sheet computed at each time step to nullify the slip velocity.

6.1 A global overview of the work

The first step consisted in implementing a no-through flow solver by using a symmetry boundary condition with the no-through flow plane as symmetry plane. The solver configured in this way allowed the diffusion through the wall which makes it possible to study the vortex-ring reconnection at low Reynolds number ($Re_\Gamma = 2000$). One main advantage was to only simulate one vortex-ring, the other one being supposed to be present by the Poisson solver, that reduced by two the computational costs.

The second step was to implement the no-slip boundary condition. The diffusion through the wall was firstly nullified, providing a real no-through flow solver. In addition, a vortex-sheet was computed and placed at the wall so that the slip velocity computed by the Poisson-solver was canceled. The required vortex-sheet at the wall was based on the velocity above and was computed using decentered finite differences of fourth order.

A validation was performed using two well-known cases, for which analytical solutions are available: the Poiseuille flow in a channel and a boundary layer development. The Poiseuille is a one-dimensional flow and the analytical solution was computed at the steady state. The numerical simulation was driven by a constant mass flow rate and the pressure gradient needed to achieve this goal was also computed at each time step. The boundary layer development was done in a channel with a slip and a no-slip wall, which caused an acceleration of the flow u_p . This was due to the restriction of passing area above the boundary layer. It was the main difference with the Blasius analytical solution that assumes no upper boundary exists. As the velocity above the boundary layer was no longer constant, it generated a velocity overshoot just above the boundary layer. A detailed discussion about velocity above the boundary layer was made. Three definitions were proposed and compared: the first was simply to choose u_p as the velocity reached at the upper wall : at $y = L_y$, the second definition was to choose u_p as the velocity at the overshoot just above the boundary layer, and a final definition was based on the vorticity flux at the wall. The thicknesses were computed using the first and the second definitions. By choosing a height for the channel sufficiently large relatively to the length of the channel, the numerical and analytical results were very close. The section finished by pointing an interesting problem in VPM: is it possible to use an anisotropic mesh with the method?

The last section was dedicated to test-cases of the no-slip wall. Several simulations were made and presented. The first simulation was the study of the interaction between a vortex-ring with a no-slip wall. The paths followed by the different vortex-rings constituted a complex phenomenon between the separation of the boundary layer and the induced velocity of vortex-rings. The no-slip boundary condition generated a boundary layer that detached to wrap the primary vortex-ring. The boundary layer that has detached generated a secondary vortex-ring. A second boundary layer separation occurred, and a tertiary vortex-ring appeared. After that,

the numerical instabilities generated Ω – loops that perturbed the vortex-ring core and finally the ring was destroyed. In conclusion, the path of the primary vortex-ring interacting with a no-slip wall was compared with the path followed by the same vortex-ring but with a slip wall.

The second test-case was still using a vortex-ring but with an initial angle towards the ring axis. A boundary layer also developed at the wall and finally detached. It generated a secondary vortex ring that caused the emergence of Ω – loops. They went up along the vortex ring core and finally destroyed the vortex-ring structure. A discussion was made about the origin of these instabilities. Were they triggered by the numerical code? A simulation with a mesh twice finer in every direction was made. The Ω – loops arrived at the same time and almost the same place in both simulations. The Ω – loops seemed to grow less in the fine resolution but they were the same in numbers and places. Simulations were made for different initial angles: $\alpha = 10^\circ$, $\alpha = 20^\circ$, $\alpha = 30^\circ$ and $\alpha = 40^\circ$. The larger the angle, the faster the vortex ring core collapsed, but fewer Ω – loops appeared.

The third test-case was focused on a classical fluid dynamics problem: a turbulent channel flow at low Reynolds. The Reynolds number based on the friction velocity was about $Re_\tau = 180$. The simulation's results were compared to the one obtained by Moser and Lee. The mean velocity profile closely matched the Moser profile. The RMS value matched the result for low value of y^+ quite good, but differed at the middle of the channel. It could be due to the fact that the channel used in VPM was shorter than the one simulated by Moser and Lee. The turbulent flow was more constrained, and it lowered the accuracy of the simulation.

To conclude the implementation of a symmetry and no-slip boundary conditions in the 3-D vortex particle-mesh method, a study of a turbulent boundary layer development was proposed in appendix. The simulation was a bit under-resolved in the laminar sub-layer. Making a wall-resolved direct numerical simulation of a boundary layer without anisotropic and without stretched mesh is very expensive. But the results remained quite good. A lot of two- and three-dimensional views were provided to appreciate the method. Statistics were performed over a long time, so boundary layer thicknesses, Reynolds number and velocity profiles were computed. The discussion and the analysis were not performed in depth.

6.2 What's next ?

All the simulations were performed in direct numerical simulation (DNS), that is very expensive in terms of computational resources. To make a wall resolved simulation, a lot of grid points need to be used at the wall, but the VPM method doesn't accept a stretched mesh or an anisotropic mesh. The simulation of bigger problems, such as atmospheric boundary layer, wind turbine wakes or helicopter wakes is not possible in DNS with current technological capabilities.

The next step of this work could be the implementation of a wall-model to make Large Eddy Simulation (LES) with wall interaction which would make it possible to simulate bigger structures as mentioned above.

It would also be interesting to investigate more in detail the reasons why an anisotropic mesh doesn't work with the current implementation of VPM.

6.3 Final word

I hope that my work in the VPM method will be appreciated, improved and used to make additional simulations. It was very interesting to work in such an environment and to use super-computing facilities of UCL. I would like to thank UCL to give me the opportunity to make this master thesis in such conditions.

Appendix A

Blasius solution for boundary layer flow

The flow is a two-dimensional flow of the velocity profile in the boundary layer. Some hypothesis are made on the flow:

- The flow is purely a two-dimensional flow in the plane $x - y$
- The flow is laminar and steady
- The diffusion in the $\hat{e}_x$ direction is negligible compared to the diffusion in the $\hat{e}_y$ direction
- The wall at $y = 0$ respects the no-through flow and no-slip boundary conditions

A.1 Analytic solution for velocity

The solution is based on the Prandl't equations of the laminar boundary layer solved by Blasius. In the case of Blasius solution the $u_\infty(x) = U_\infty = cst$, one can demonstrate that under these conditions, the equations of Prandl't are:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (\text{A.1})$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = \nu \frac{\partial^2 u}{\partial y^2} \quad (\text{A.2})$$

With the boundary conditions: $u(x, 0) = v(x, 0) = 0$ and $u(x, y \rightarrow \infty) = U_\infty$
The flow is an incompressible flow, so the velocity comes from a scalar potential ψ :

$$u = \frac{\partial \psi}{\partial y}$$
$$v = -\frac{\partial \psi}{\partial x}$$

Balsius introduces the following variable:

$$\delta(x) = \sqrt{\frac{2}{Re(x)}} x = \sqrt{\frac{2\nu x}{U_\infty}}$$
$$\eta = \frac{y}{\delta(x)}$$
$$\psi = U_\infty \delta(x) f(\eta)$$

With $\delta(x)$ the boundary layer thickness, η the dimensionless variable in the $\hat{e}_y$ direction and ψ the stream function. The velocities in the two directions are given by the derivative of the stream

function as mentioned above:

$$\begin{aligned}
 u &= U_\infty \delta(x) \frac{\partial f(\eta)}{\partial y} = U_\infty \delta(x) \frac{df(\eta)}{d\eta} = U_\infty \delta(x) f'(\eta) \frac{\partial \eta}{\partial y} \\
 &= U_\infty f'(\eta) \\
 v &= -U_\infty (\delta'(x) f(\eta) + \delta(x) \frac{\partial f(\eta)}{\partial x}) = -U_\infty (\delta'(x) f(\eta) + \delta(x) f'(\eta) \frac{\partial \eta}{\partial x}) \\
 &= U_\infty \delta'(x) (\eta f'(\eta) - f)
 \end{aligned}$$

In the same way, we compute the derivative of momentum equation A.2:

$$\begin{aligned}
 u \frac{\partial u}{\partial x} &= -U_\infty^2 f'(\eta) f''(\eta) \eta \frac{\delta'(x)}{\delta(x)} \\
 v \frac{\partial u}{\partial y} &= U_\infty^2 f''(\eta) (\eta f'(\eta)) f(\eta) \frac{\delta'(x)}{\delta(x)} \\
 \frac{\partial^2 u}{\partial y^2} &= U_\infty f'''(\eta) \frac{1}{\delta(x)^2}
 \end{aligned}$$

If we put the result above back in the momentum equation (A.2), we obtain a non linear ordinary differential equation (ODE) of third order:

$$f'''(\eta) + f(\eta) f''(\eta) = 0 \quad (\text{A.3})$$

Applying the boundary condition on the velocity to the function $f(\eta)$ we have:

$$f(0) = 0, \quad f'(0) = 0, \quad f'(\eta \gg 1) = 1$$

The non linear ordinary differential equation is numerically solved as a system of three first order differential equations and using the shooting method to satisfy the third boundary condition, we find $f'''(0) = 0.46960$. The curves obtained by numerical integration of equation (A.3) are the following:

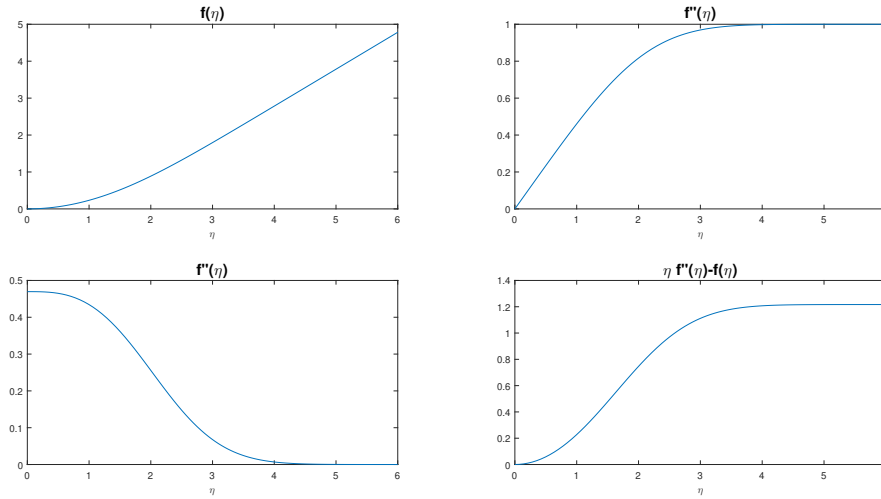


Figure A.1: Solution of Blasius for boundary layer equations

The velocity profiles are shown at the next figure at different sections in the direction of U_∞ :

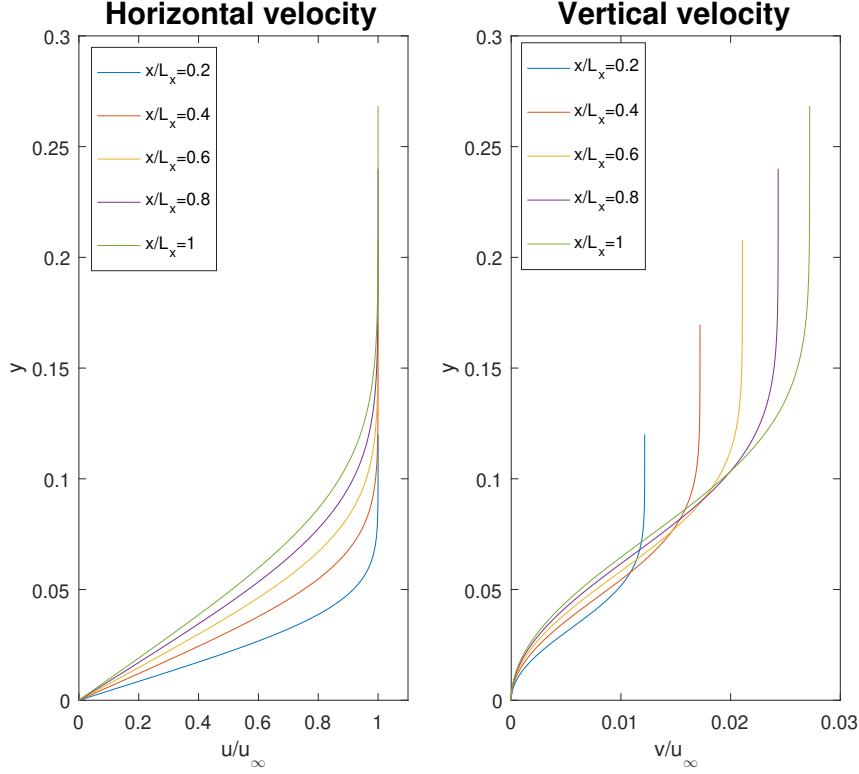


Figure A.2: Velocity profiles at different section

A.2 Analytic solution for vorticity

The vorticity field in the boundary layer can be found on the basis of the velocity field. The vorticity of a two-dimensional flow has only one component, and it is directed in the $\hat{e}_z$ direction, recall subsection 3.2.1. We have:

$$\omega_z = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}$$

The challenge is to find the partial derivatives of the two components of the velocity. Using the previous section, we have:

$$\begin{aligned} u &= U_\infty f'(\eta) \\ v &= U_\infty \delta'(x)(\eta f'(\eta) - f) \\ \eta &= \frac{y}{\delta(x)} \\ \delta(x) &= \sqrt{\frac{2\nu x}{U_\infty}} \end{aligned}$$

Let's define some useful derivatives:

$$\begin{aligned} \frac{\partial \eta}{\partial x} &= -\frac{y}{\delta(x)^2} \delta'(x) = -\eta \frac{\delta'(x)}{\delta(x)} \\ \frac{\partial \eta}{\partial y} &= \frac{1}{\delta(x)} \end{aligned}$$

For the derivative of the horizontal component with respect to the $\hat{e}_y$ direction we have:

$$\frac{\partial u}{\partial y} = U_\infty \frac{\partial f'}{\partial y} = U_\infty f''(\eta) \frac{1}{\delta(x)}$$

and for the derivative of the vertical component with respect to the $\hat{e}_x$ direction we have:

$$\begin{aligned}\frac{\partial v}{\partial x} &= U_\infty \left(\delta''(x)(\eta f' - f) + \delta'(x) \left(\frac{\partial \eta}{\partial x} f + \eta \frac{\partial f'}{\partial x} - \frac{\partial f}{\partial x} \right) \right) \\ &= U_\infty \left(\delta''(x)(\eta f' - f) + \delta'(x) \left(-\eta \frac{\delta'(x)}{\delta(x)} f - \eta^2 f'' \frac{\delta'(x)}{\delta(x)} + \eta \frac{\delta'(x)}{\delta(x)} f' \right) \right)\end{aligned}$$

The vorticity profiles are shown in the next figure at different sections in the direction of U_∞ :

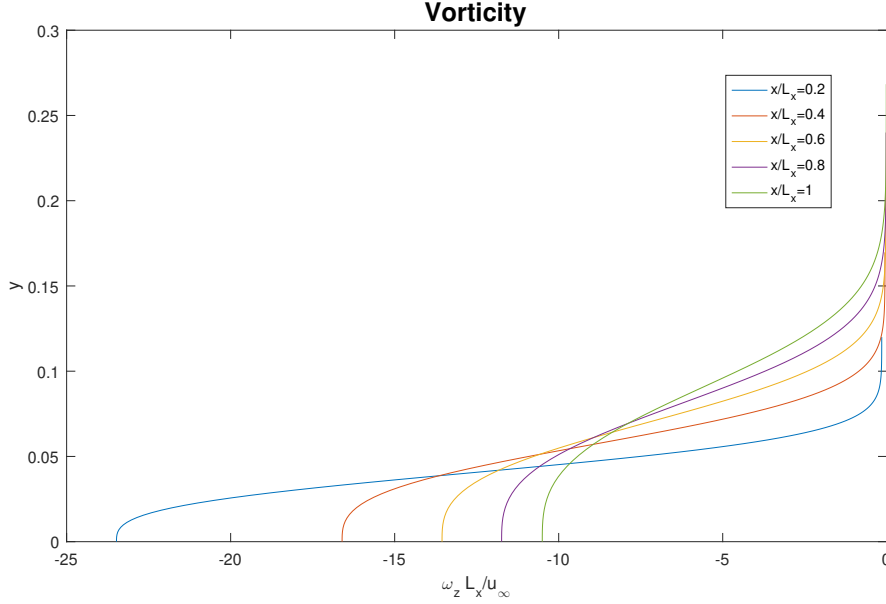


Figure A.3: Vorticity profiles at different section

A.3 Boundary layer thicknesses

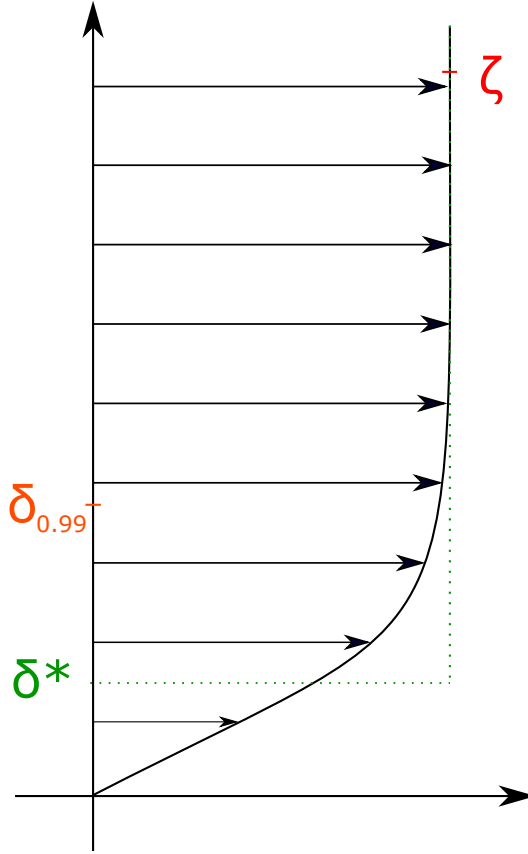
The interesting thing about boundary layer is the boundary layer thickness. The first way to define a thickness is to find where the horizontal velocity reaches 99% of the free stream velocity U_∞ , called the $\delta_{0.99}$. It corresponds to $\eta_{0.99} = 3.4719$ and so, the boundary layer thickness at 99% of U_∞ is:

$$\frac{\delta_{0.99}(x)}{x} = \frac{\eta_{0.99}}{\frac{U_\infty x}{2\nu}} = \frac{4.91}{Re^{1/2}}$$

But we can also define the boundary layer thickness at 95% of U_∞ , it gives:

$$\frac{\delta_{0.95}(x)}{x} = \frac{\eta_{0.95}}{\frac{U_\infty x}{2\nu}} = \frac{3.92}{Re^{1/2}}$$

The boundary layer thickness directly based on velocity is not a specific concept, a more specific boundary layer thickness is the displacement thickness. The concept is based on the mass flow of the flow with the boundary layer effect and the mass flow without the boundary layer (e.g. the free stream U_∞). *If you consider a flow without boundary layer, how far should you move the wall to have the same mass flow as a flow with boundary layer ?*:



With ζ defined as the height where the flow reaches the freestream condition. Mathematically, it gives:

$$\int_0^\zeta u(y)dy = U_\infty(\zeta - \delta^*) = U_\infty \int_0^\zeta dy - U_\infty \delta^*$$

$$\delta^* = \int_0^\zeta 1 - \frac{u}{U_\infty} dy$$

With the Blasius solution, it gives:

$$\frac{\delta^*}{\delta} = \int_0^{\eta \gg 1} 1 - f'(\eta) d\eta = 1.127$$

$$\frac{\delta^*(x)}{x} = \frac{1.127}{\frac{U_\infty x}{2\nu}} = \frac{1.721}{Re^{1/2}}$$

Finally, we define the momentum thickness as:

$$\theta = \int_0^\zeta \frac{u}{U_\infty} \left(1 - \frac{u}{U_\infty}\right) dy$$

For the Blasius boundary layer solution, the momentum thickness is:

$$\frac{\theta}{\delta} = \int_0^{\eta \gg 1} f'(\eta)(1 - f'(\eta)) d\eta = 0.47$$

$$\frac{\theta}{x} = \frac{0.47}{\frac{U_\infty x}{2\nu}} = \frac{0.664}{Re^{1/2}}$$

Physically, the momentum thickness is linked to the friction coefficient of the plate as follows:

$$C_{f,m}(L) = 2 \frac{\theta(L)}{L} \quad (\text{A.4})$$

The figure hereafter shows the different boundary layer thicknesses.

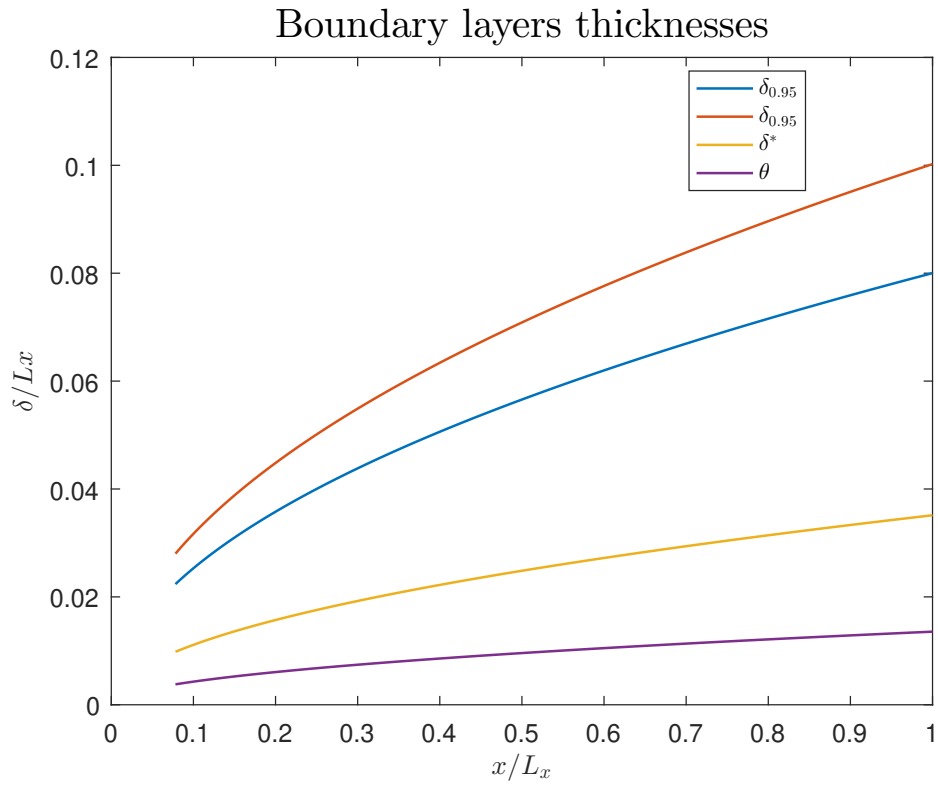


Figure A.4: The different boundary layer thicknesses, $\delta_{0.95}$, $\delta_{0.99}$, δ^* , θ

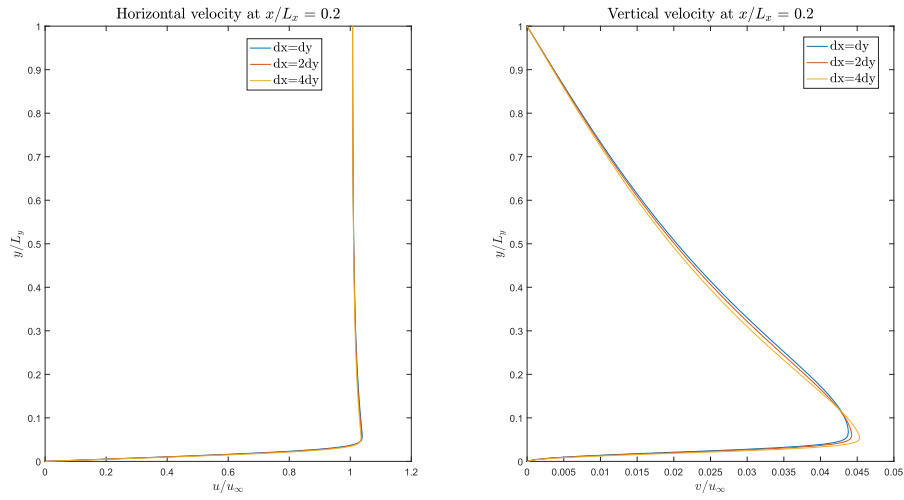
Appendix B

Anisotropic mesh additional results

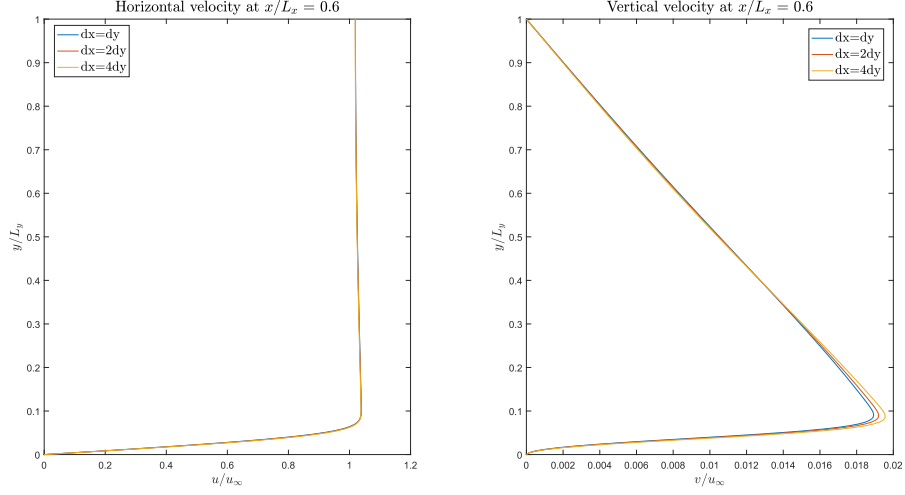
Non-isotropic mesh is used in the VPM method, additional results and figures are presented in this appendix.

B.1 Anisotropic mesh in laminar boundary layer

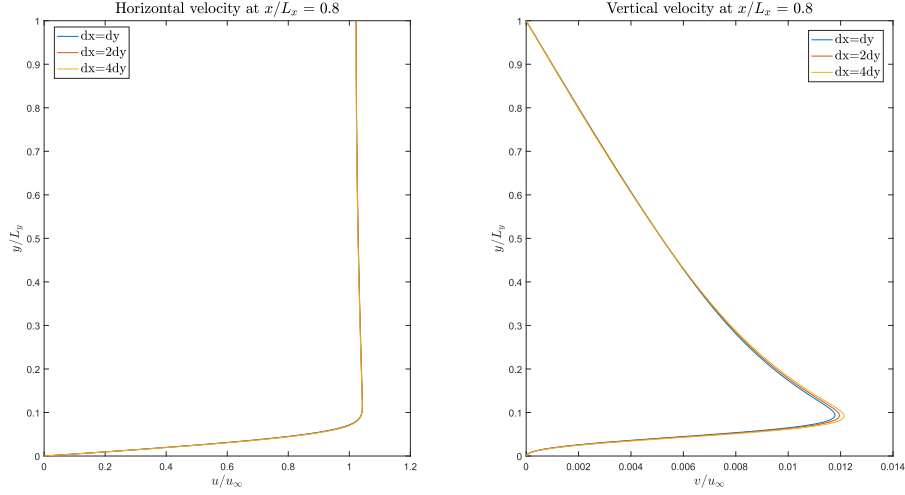
In the case of Blasius boundary layer, several velocity profiles with anisotropic mesh are compared together :



(a)



(b)



(c)

Figure B.1: Isotropic and non-isotropic meshes, velocity profiles comparison

B.2 Anisotropic mesh in turbulent channel flow

To reduce the computational cost of the turbulent channel flow two techniques can be used, the first is the simplest one : the flow vary much in the wall normal direction and so it is natural to use a thinner mesh in that direction. The second is to use more point near the wall then in the middle of the channel because the variation of the flow quantities are more important at the wall than in the middle of the channel, it is called the stretching of the mesh.

An anisotropic mesh where tested for the turbulent channel flow explained in section 5.3. The results for the mean velocity profile is compared to the one obtained by Moser and Lee [5]. The spatial step in the streamwise and spanwise direction is twice bigger that the spatial step in the wall-normal direction.

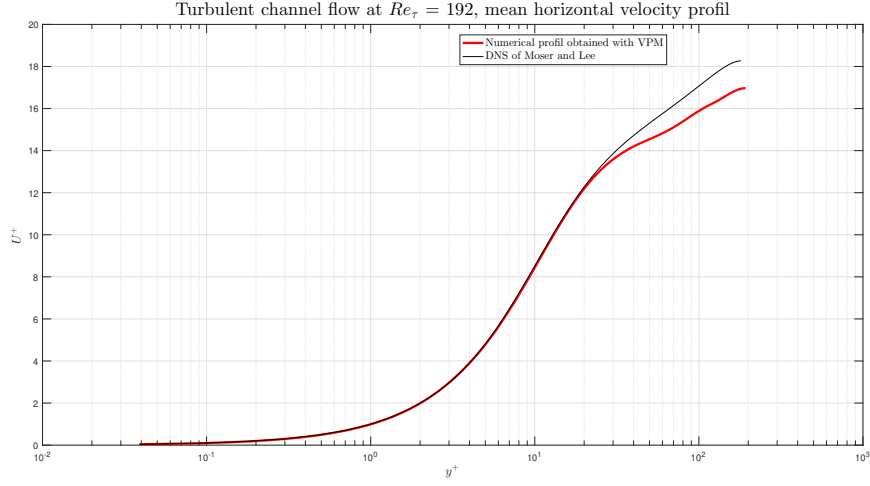


Figure B.2: Velocity profile in dimensionless wall distance with anisotropic mesh

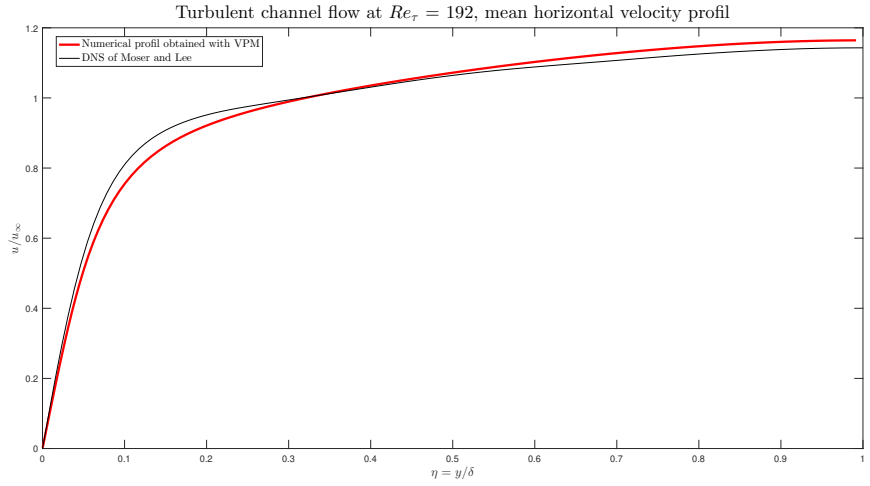


Figure B.3: Velocity profile with anisotropic mesh

Once again the result are not very satisfying. The skin friction are different that the one obtained by Moser and the profile is not the same. The question of why VPM seems to dislike anisotropic mesh is still pending. The answer could be found in the particle to mesh and mesh to particle step.

Appendix C

Ring interaction with the no-slip wall

Several angles of attack have been simulated. The results of the simulation are provided in the following figures.

C.1 Initial angle $\alpha = 10^\circ$

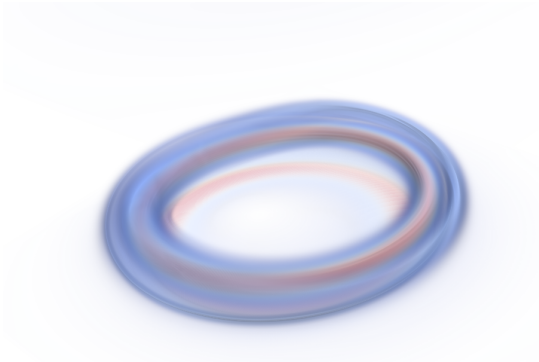
The time evolution of the interaction between a vortex-ring and a no-slip is shown at the next figures.



(a)



(b)



(c)



(d)

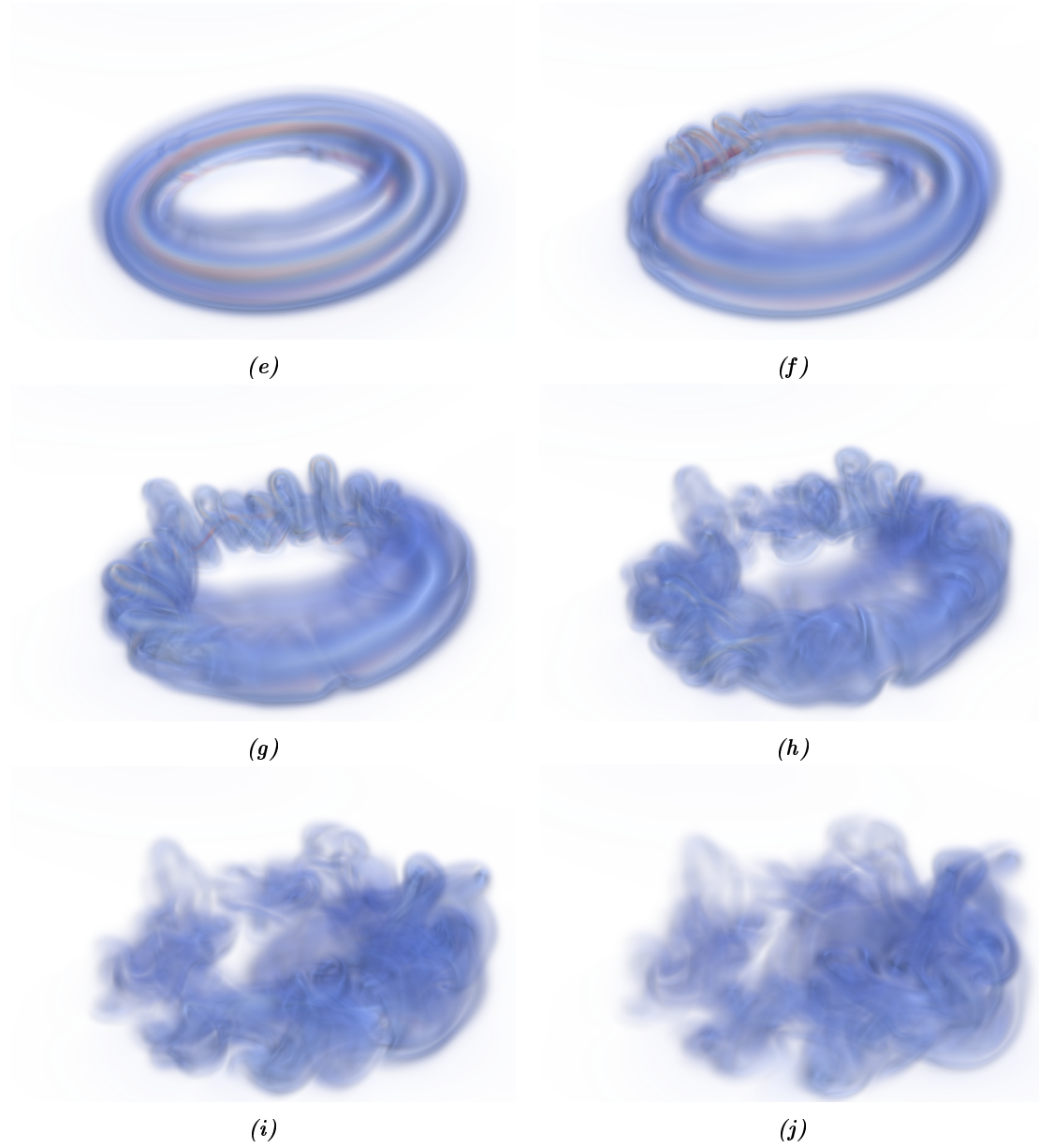
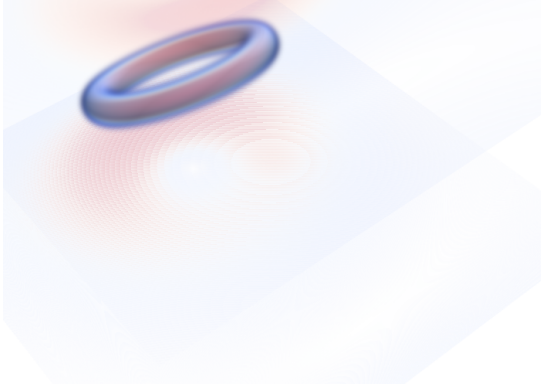


Figure C.1: Volume rendering of vorticity magnitude at $\alpha = 10^\circ$

C.2 Initial angle $\alpha = 30^\circ$

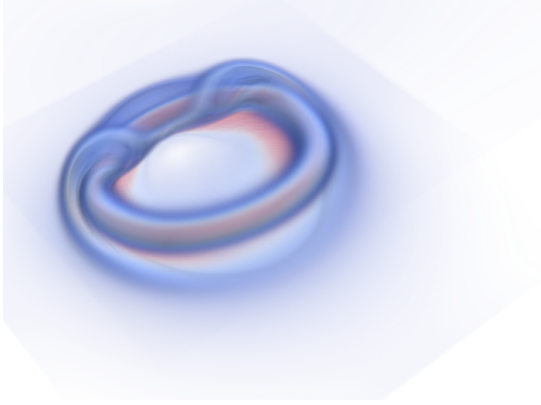
The time evolution of the interaction between a vortex-ring and a no-slip wall is shown at the next figures.



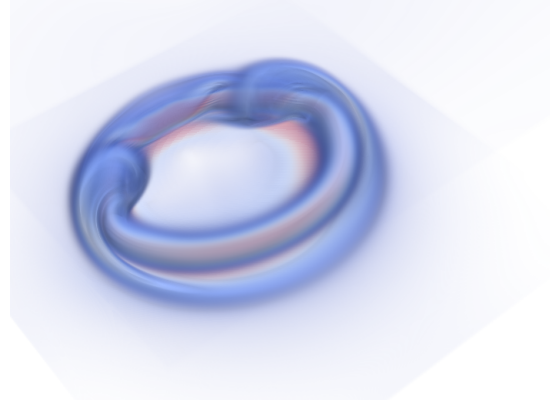
(a)



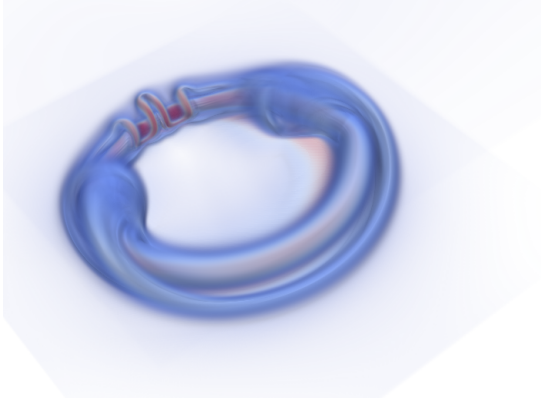
(b)



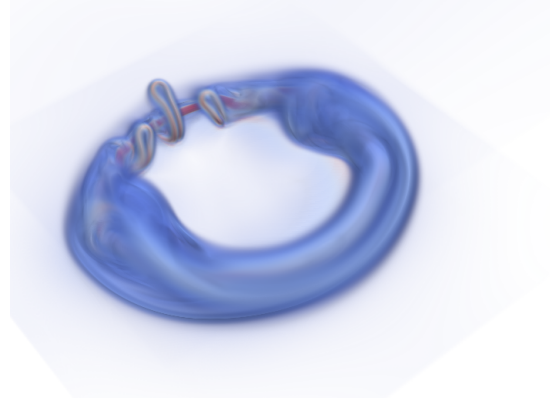
(c)



(d)



(e)



(f)

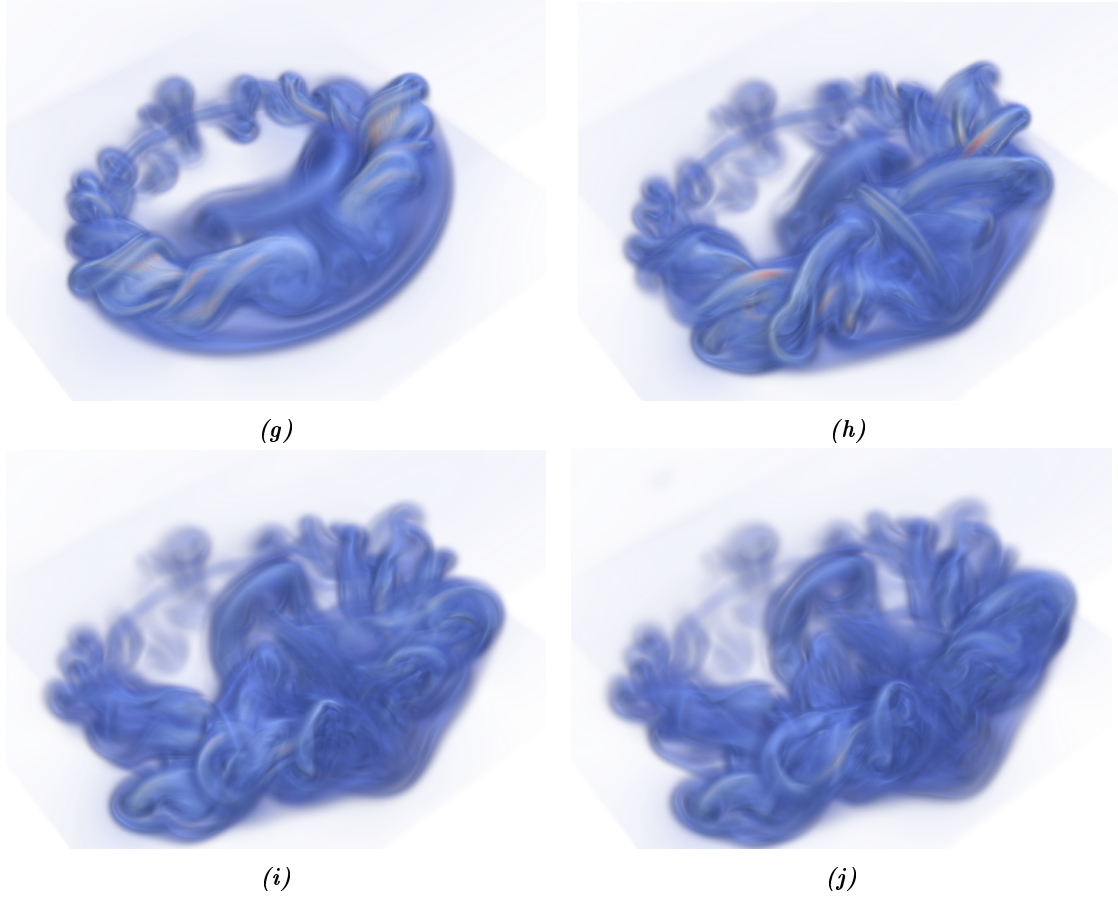
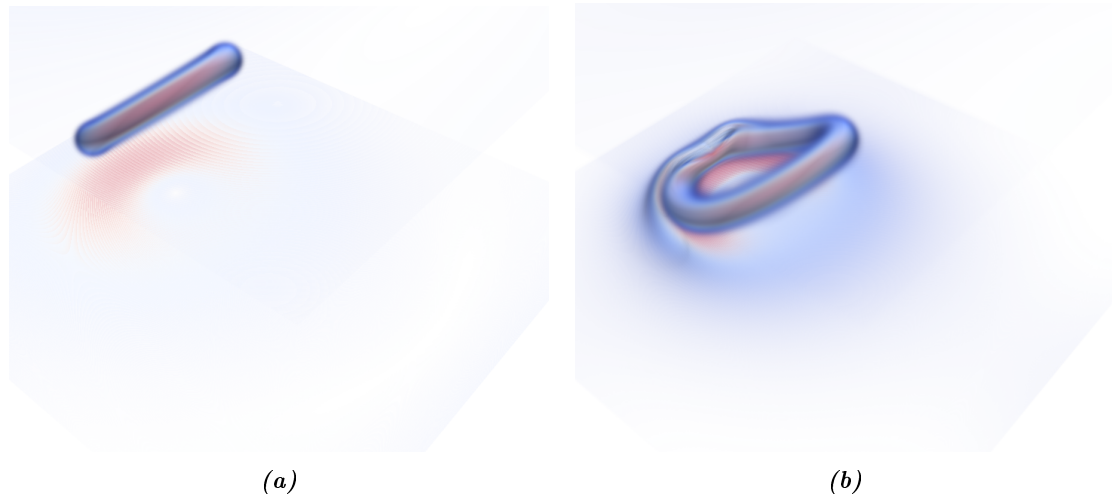
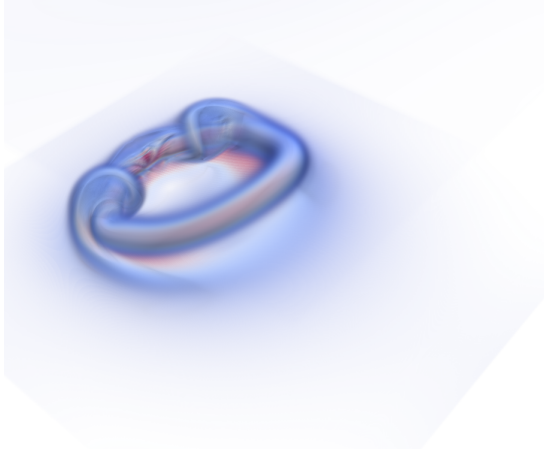


Figure C.2: Volume rendering of vorticity magnitude at $\alpha = 30^\circ$

C.3 Initial angle $\alpha = 40^\circ$

The time evolution of the interaction between a vortex-ring and a no-slip is shown at the next figures.

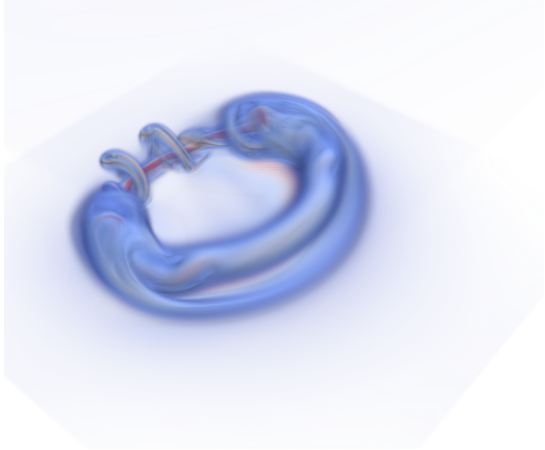




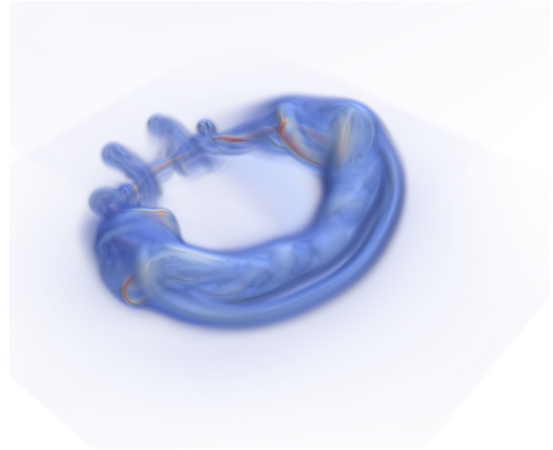
(c)



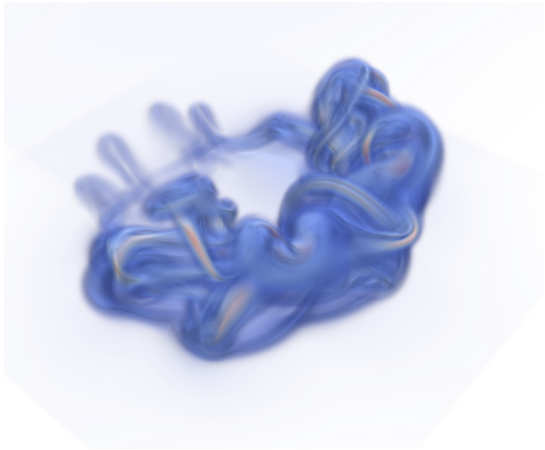
(d)



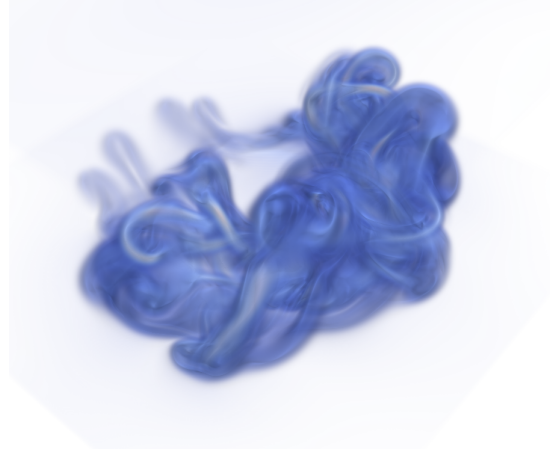
(e)



(f)



(g)



(h)

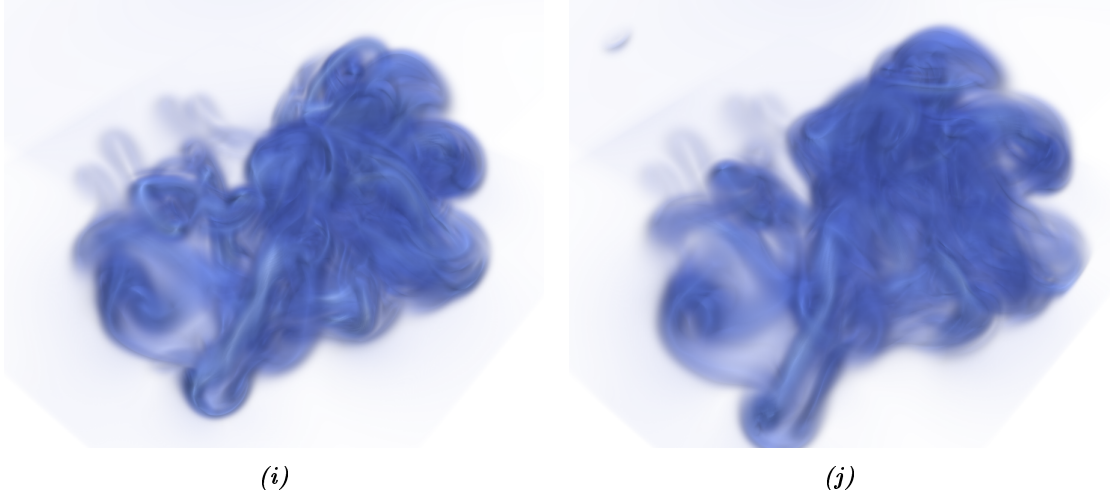


Figure C.3: Volume rendering of vorticity magnitude at $\alpha = 40^\circ$

Appendix D

Statistics quality diagnostic

The values of the mean wall-normal and spanwise velocity components and covariance between the velocity components ($\langle u'w' \rangle$, $\langle v'w' \rangle$) should be zero due to the statistical symmetries in the flow. They are computed as an indicator of the statistics quality.

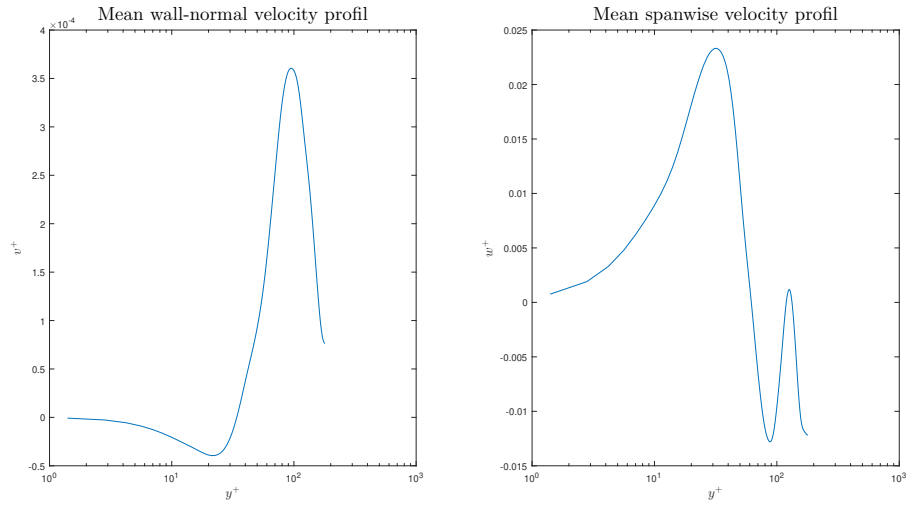


Figure D.1: Mean velocity profile of wall-normal and spanwise velocity components

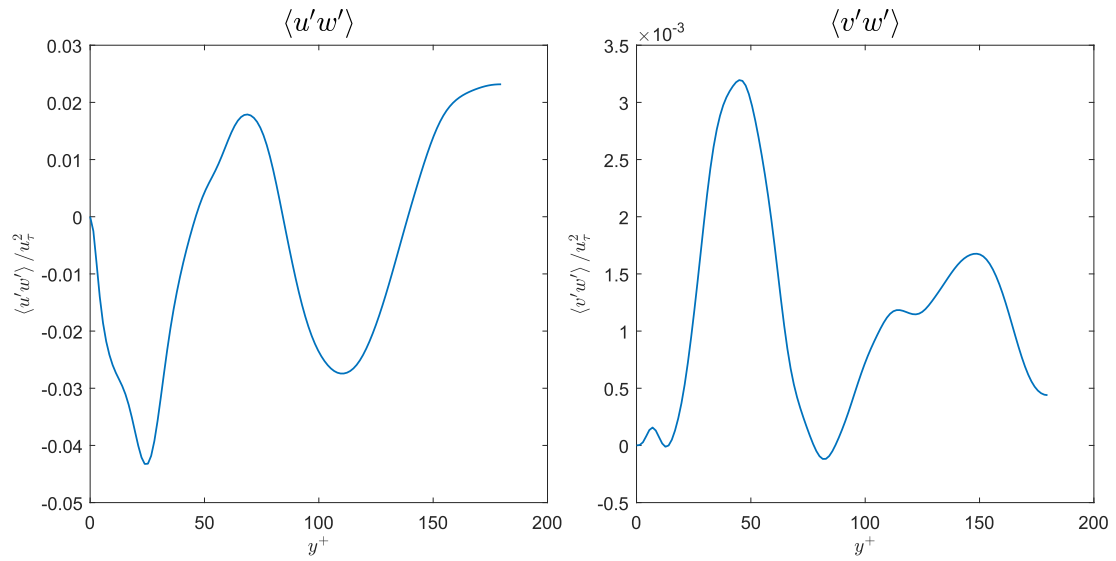


Figure D.2: Covariance between spanwise and streamwise and spanwise and wall-normal velocity components

Appendix E

Turbulent boundary layer development

E.1 Introduction

This appendix presents a result obtained with the no-slip boundary condition in the VPM method. The laminar boundary layer at low Reynolds number were presented in a previous section. This appendix proposes the direct numerical simulation of a boundary layer development at a higher Reynolds number. At this Reynolds number, the boundary layer become quickly unstable and turn to turbulent. The results are presented in appendix because there are a little bit under-resolved and the analysis is not performed in depth. Indeed, the isotropic and non-stretched mesh needed for VPM make it necessary to use a lot of grid points in the entire computational domain to ensure a good resolution at the wall, making the simulation very expensive.

Nevertheless, the results are visually quite good and turbulent boundary layer constitutes a good topic to be thoroughly studied. The goal of this appendix is to show some *nice animations* that can be obtained with a no-slip wall in the vortex particle-mesh method. The analysis of the results will not be achieved in-depth as the result is a little bit under-resolved. A lot of pictures of the flow will be presented.

The boundary layer thicknesses, friction velocity, friction coefficient, Reynolds numbers and mean velocity profiles are also provided and a bit discussed.

E.2 Numerical setup

The numerical setup is the trickiest part for the turbulent boundary layer simulation. It's a compromise between numerical resolution, accuracy of the results, computational costs and initiation of instabilities in the boundary layer. Indeed, a lot of grid points insure the good resolution of the simulation, but the transient from laminar to turbulent is very long and the computational costs are high. On the contrary, less grid points generate numerical instabilities that are favorable for the transition from laminar to turbulent, but the resolution might be too low. The numerical setup presented in this section is the result of (a lot of) tests. It is a compromise between the numerical costs and the numerical accuracy. The boundary layer develops in a channel. The reference length is the length of the channel L_x and the reference time t_0 is based on the time necessary to cross the channel in the streamwise direction $t_0 = \frac{L_x}{U_\infty}$:

L_y/L_x	L_z/L_x	Re_L	N_x	N_y	N_z	h/L	$\delta t/t_0$
0.1	0.1	5.9×10^4	1280	128	128	7.8×10^{-4}	0.001

Table E.1: Numerical parameters of the turbulent boundary layer simulation

The spatial step in dimensionless wall distance is about $h^+ = 3.3$, so there are not enough grid points in the laminar sub-layer. A mesh twice finer would provide more accurate results.

The simulation runs on $4 \times 4 \times 16$ processors during 40 hours (physical time), the simulation computed 24.000 time steps.

E.3 Instantaneous numerical results

E.3.1 Resolution of the simulation : mesh Reynolds numbers

The mesh Reynolds number based on velocity and on vorticity is computed and monitored at each time step to check the resolution of the simulation. The mesh Reynolds based on velocity is quite good but the Reynolds number based on vorticity is too high. It shows once again that a mesh twice finer would make the simulation enough resolved (it would divide the mesh Reynolds by four!).

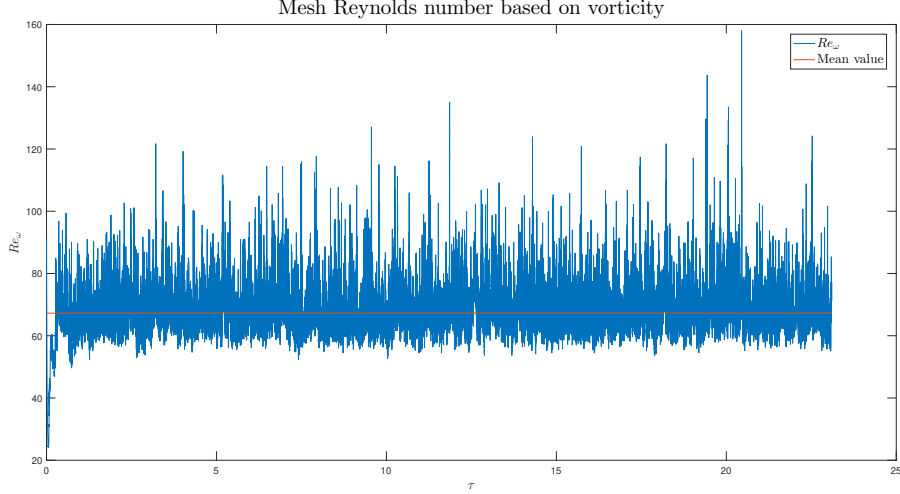


Figure E.1: Mesh Reynolds number based on vorticity, mean value = 67.3

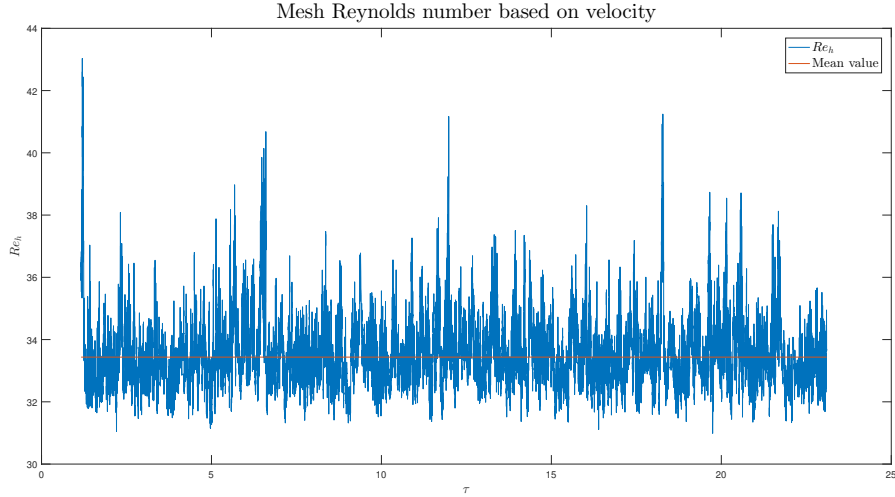
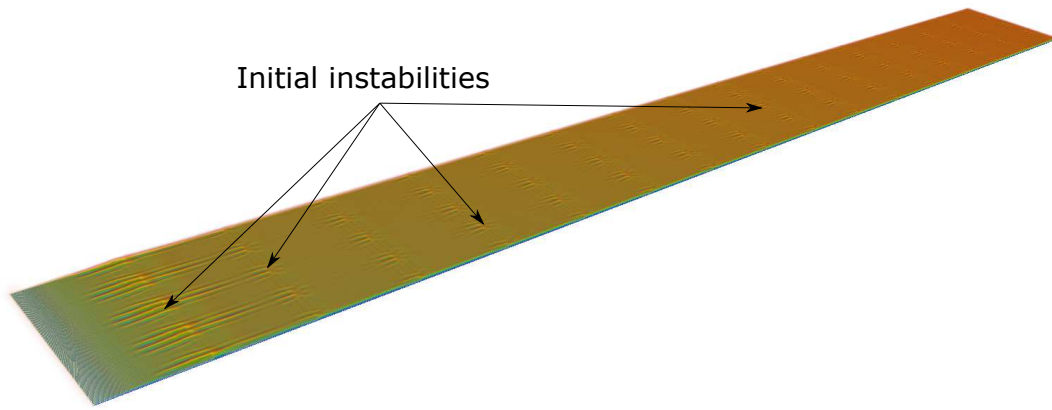


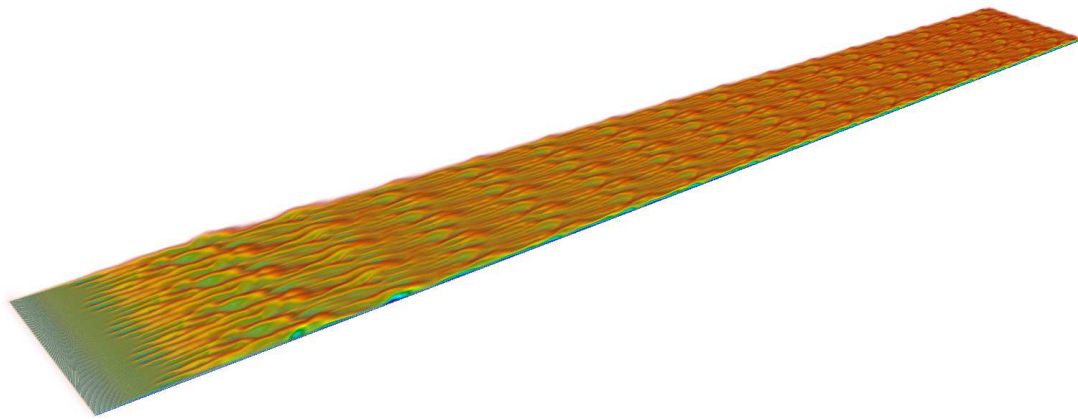
Figure E.2: Mesh Reynolds number based on velocity, mean value = 33.4

E.3.1.1 Boundary layer transition to turbulent

Initially, no boundary layer exists. The boundary layer grows up at the same time that instabilities. These are clearly from numerical origin due to the periodicity of these perturbations. They grow up quickly and generate turbulences but the flow presents some periodicity due to theses numerical instabilities. It is needed to wait $\tau \simeq 1.5$ to get a fully turbulent flow without any periodicity. The evolution of the boundary layer since the beginning are shown at the next figures.



(a) $\tau = 0.05$



(b) $\tau = 0.1$

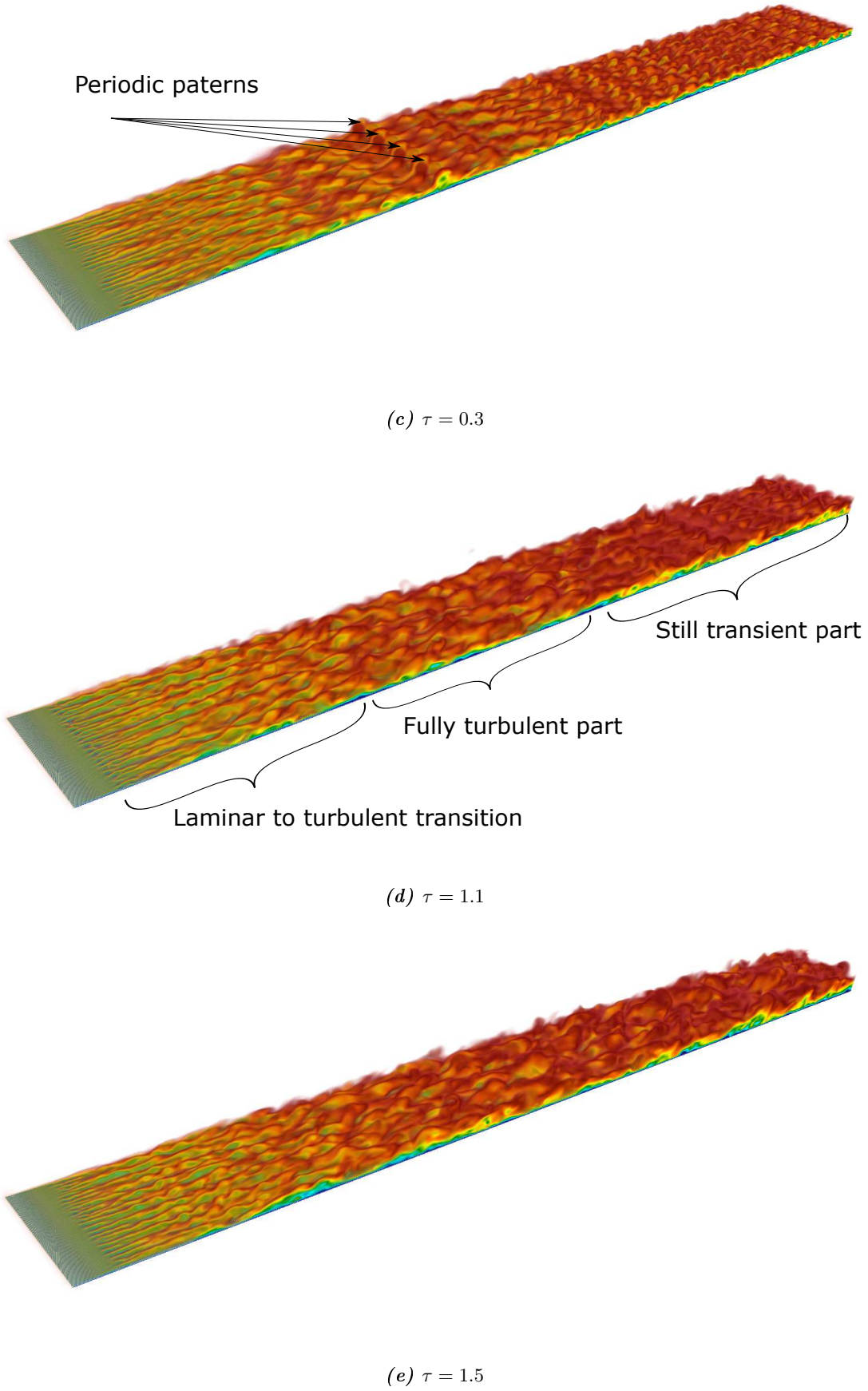
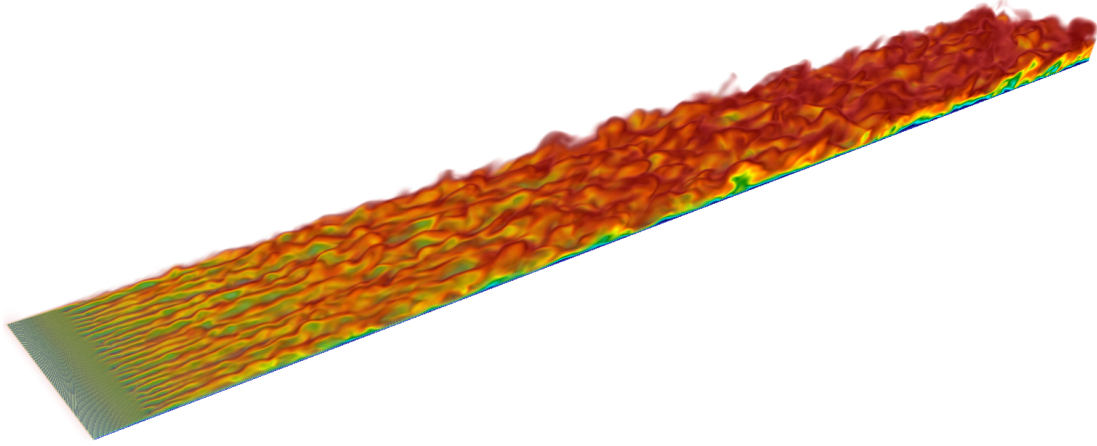


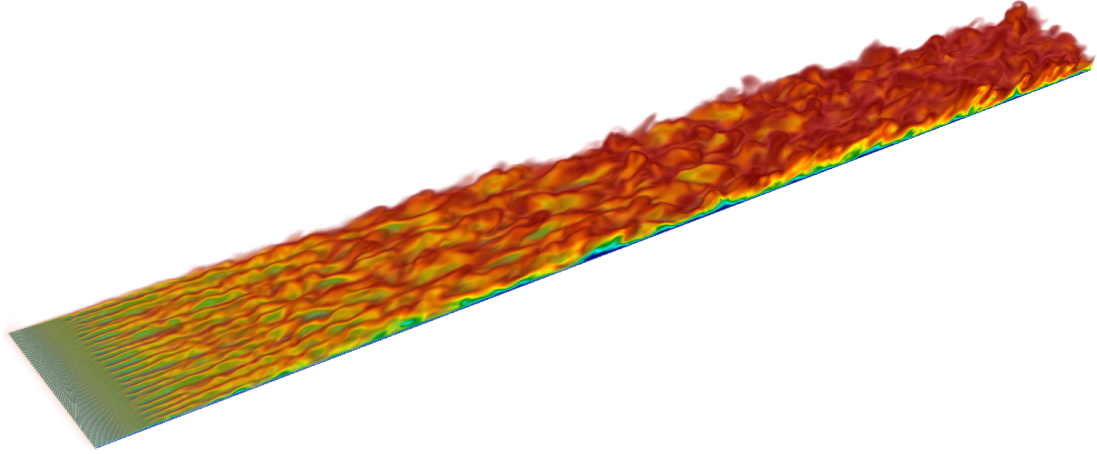
Figure E.3: Transition from laminar to turbulent due to numerical instabilities

E.3.1.2 Three dimensional results for velocity

Additional results for velocity are provided at the next figures for different dimensionless time. The flow is fully turbulent, a transition from laminar to turbulent boundary layer exists, no solution is considered in this region ($x/L \leq 0.2$).



(a) $\tau = 10$

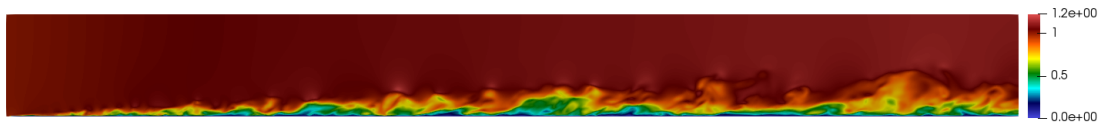


(b) $\tau = 20$

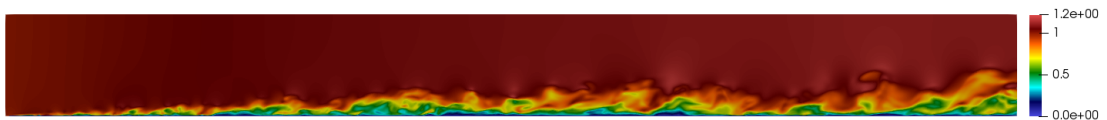
Figure E.4: Fully developed turbulent boundary layer (instantaneous view)

E.3.1.3 Two dimensional results for velocity

The turbulent boundary layer can also be observed in two dimensional slices :



(a) $\tau = 10$



(b) $\tau = 15$

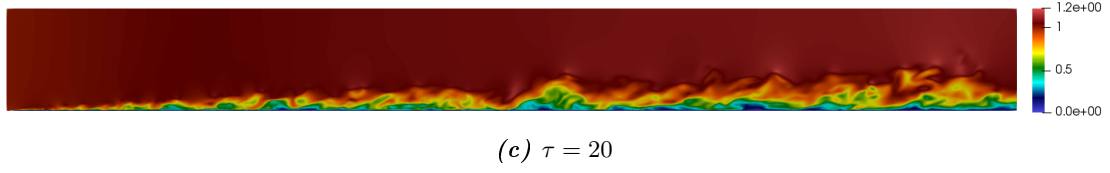
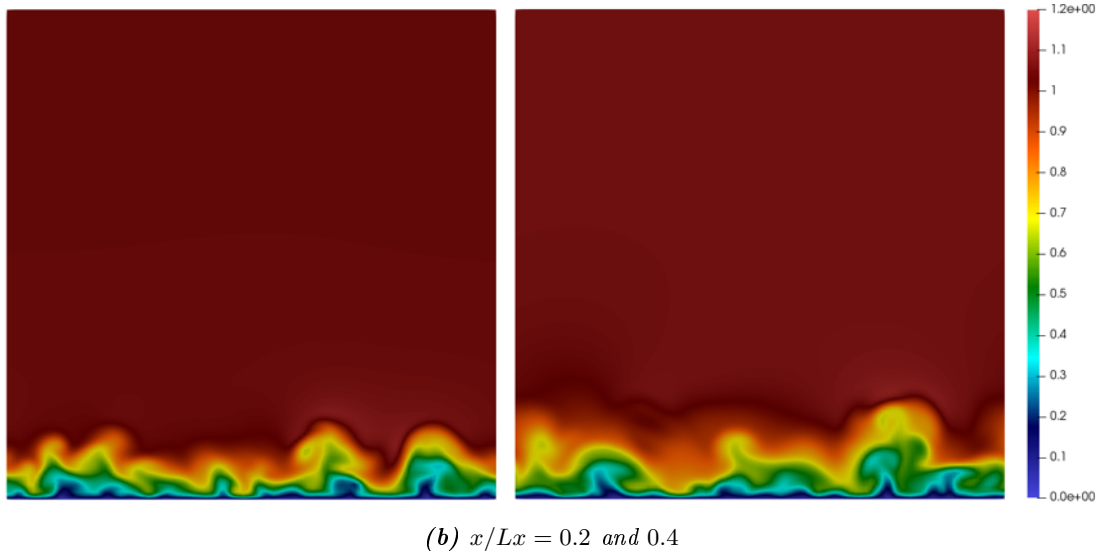
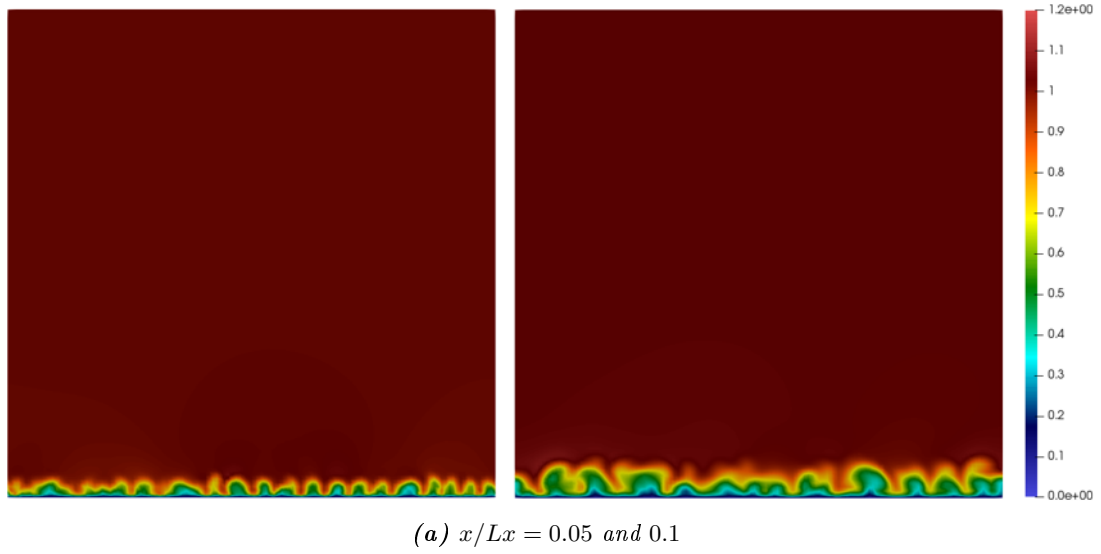


Figure E.5: *Instantaneous slices of the fully turbulent boundary layer in the spanwise direction*

The following figure presents slices in the streamwise direction (the normal vector of the slices is the streamwise direction) for different x/L_x at the same time. It clearly shows the space development of the boundary layer in the streamwise direction. One can also observe the relative acceleration of the flow above the boundary layer due to the restriction of passing area (see section 4.3 for more details).



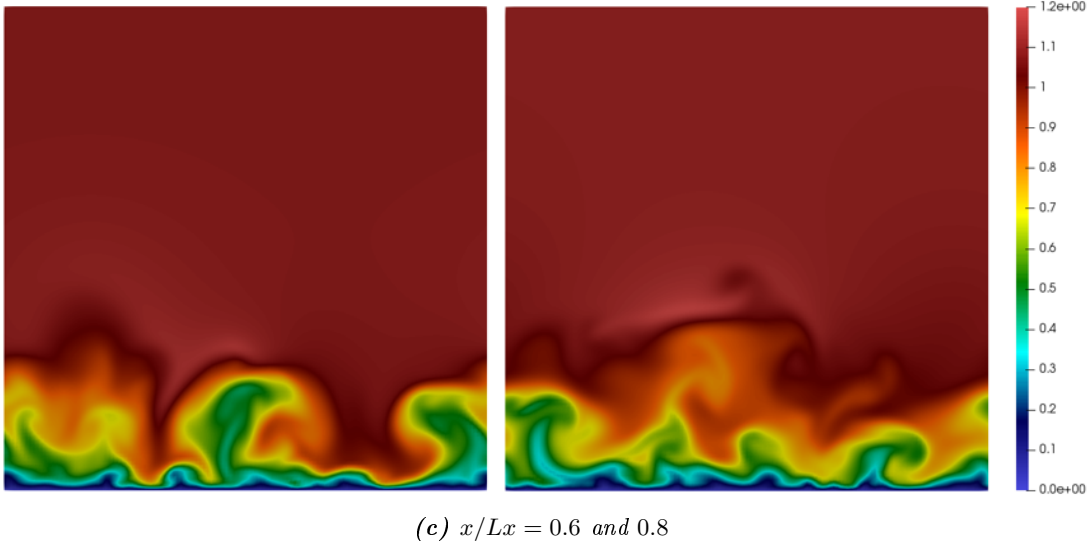


Figure E.6: Instantaneous slices of the fully turbulent boundary layer in the streamwise direction at $\tau = 20$

And finally some slices in the wall normal direction at $y/L_x = 0.01$:

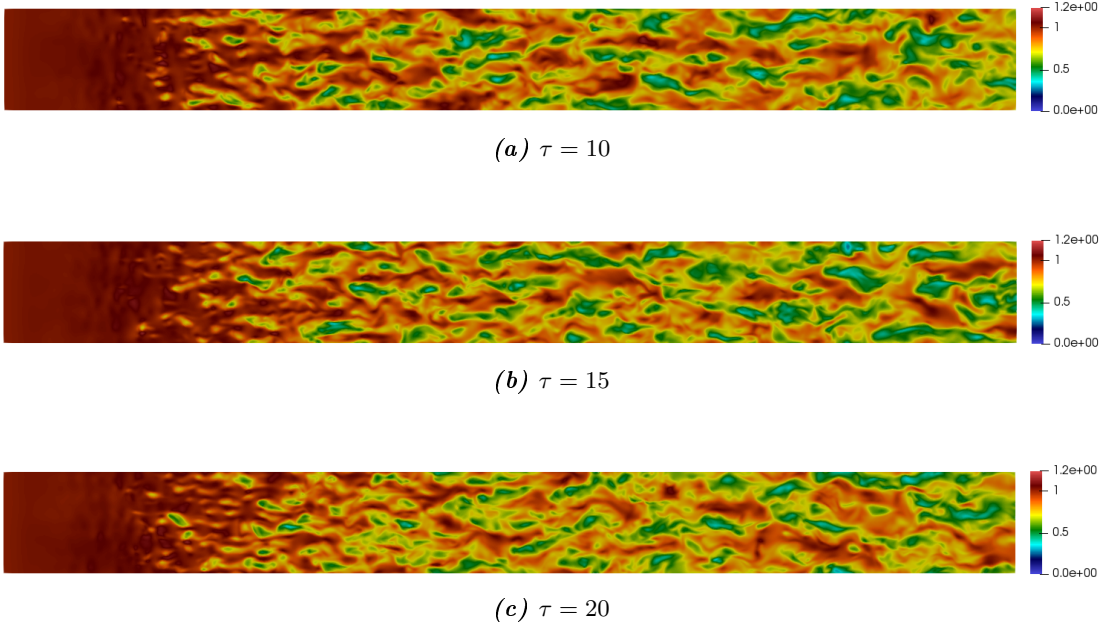
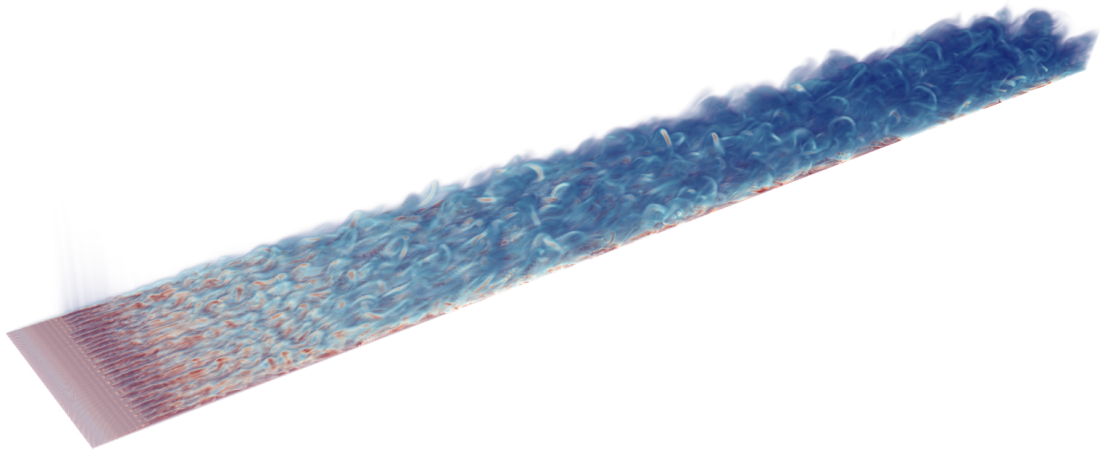


Figure E.7: Instantaneous slices of the fully turbulent boundary layer in the wall-normal direction

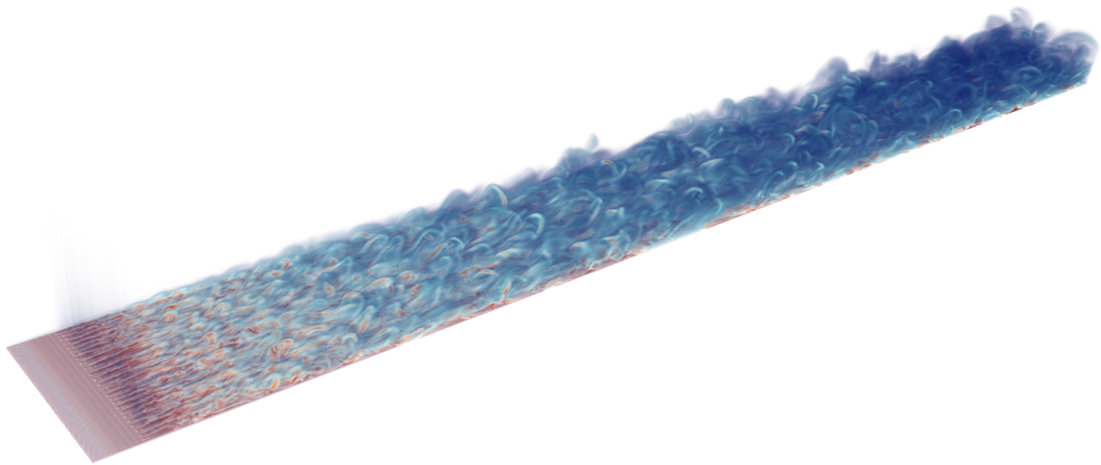
E.3.2 Three dimensional results for vorticity

The turbulent boundary layer presents a complex vorticity field as vorticity is generated at each time step by the no-slip wall. At the start of the boundary layer ($x/L_x \simeq 0.05$), there is a generation of hairpin vortices. The generated vortex tubes then interact together, and are also diffused and convected by the flow. This leads to high turbulences in the boundary layer. Some three dimensional results of the vorticity field are shown at this section.

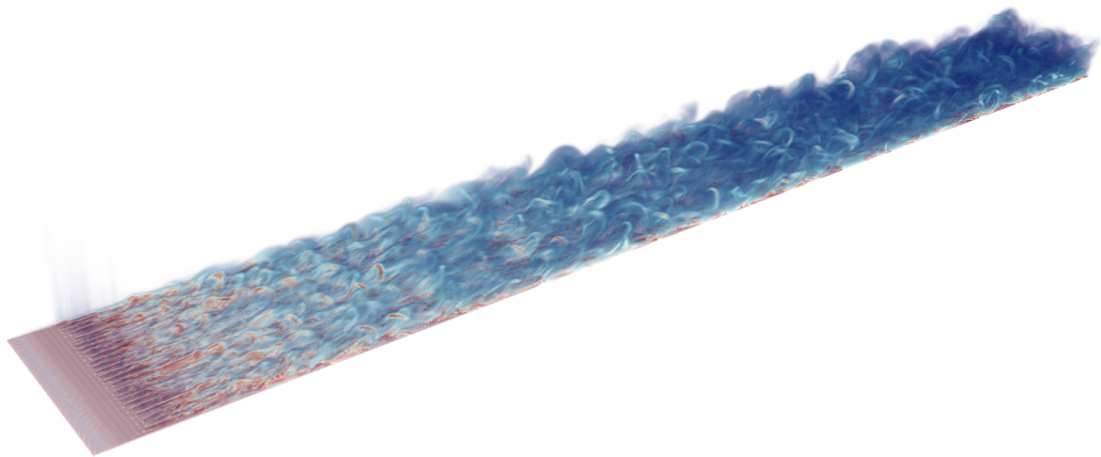
The transition from laminar to turbulent, the generation of hairpin vortices, the diffusion of the vortex tubes and the turbulence are clearly observable.



(a) $\tau = 10$



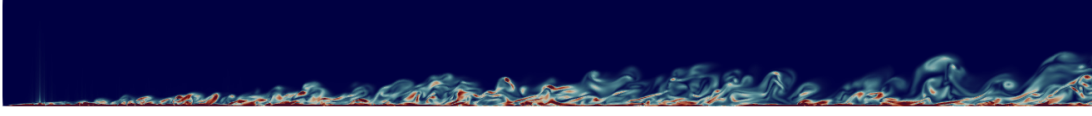
(b) $\tau = 15$



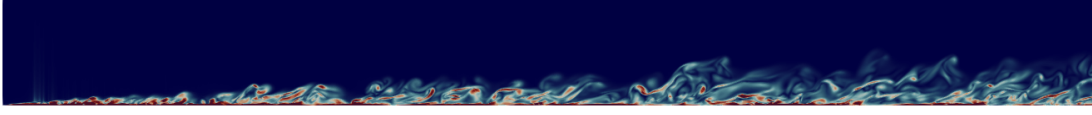
(c) $\tau = 20$

Figure E.8: Fully developed turbulent boundary layer (instantaneous view)

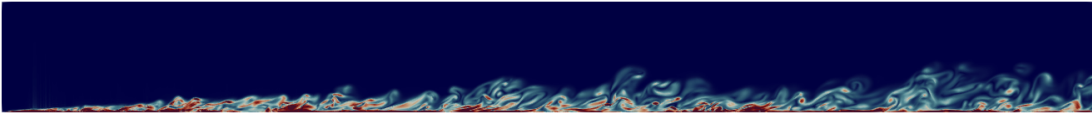
E.3.2.1 Two dimensional results for vorticity



(a) $\tau = 10$



(b) $\tau = 15$



(c) $\tau = 20$

Figure E.9: Instantaneous slice of the fully turbulent boundary layer in the spanwise direction

E.4 Average numerical results

The boundary layer becomes very quickly turbulent. The statistics are performed after 1500 time steps ($\tau = 1.5$), when the boundary layer is fully developed and no symmetries are visible in the spanwise and streamwise direction.

As the flow is three dimensional and periodic in one direction, the two dimensional mean velocity profile is computed using spatial average in the spanwise direction and temporal average on the time interval of $\tau = 1.5 \rightarrow 24$. It provides $N_z \times N_t$ velocity profile for the average.

E.4.1 Boundary layer thicknesses and Reynolds number

The first important quantity is the Reynolds number based on the streamwise direction. The definition is provided at equation (E.1).

As done in the laminar case, the boundary layer thicknesses are computed. Due to the restriction of passing area above the boundary layer, the velocity profiles also present an overshoot, but this overshoot is very insignificant and the value is almost the same than the one obtained at the slip wall. The integrations for the displacement thickness and momentum thickness are performed from the no-slip wall to the slip wall. The thicknesses at 95% and 99% are computed compared to the velocity reached at the slip wall.

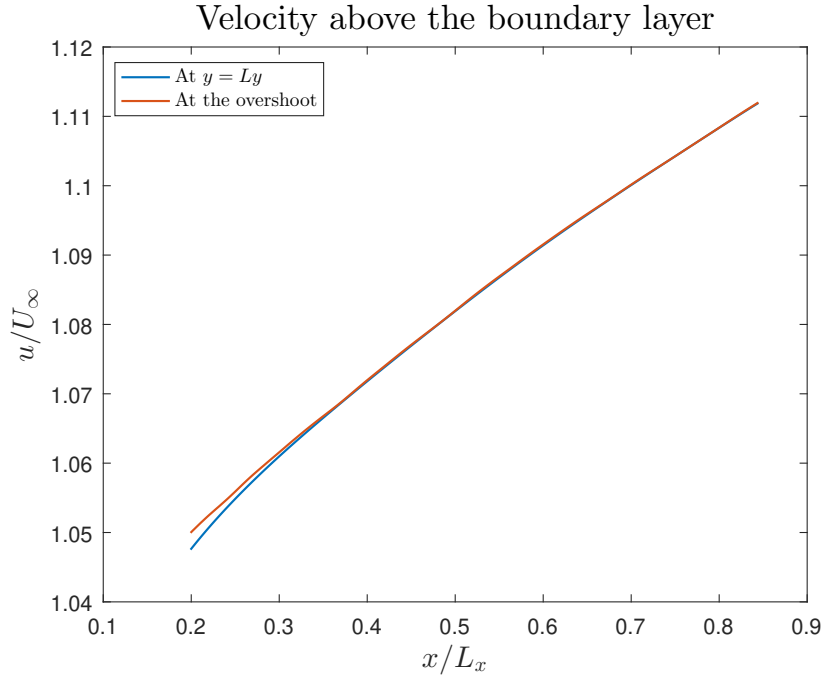


Figure E.10: Velocity above the boundary layer

The values are compared to the those obtained by Prandl't in 1927 with the simplified profile by exponent (see [8] for more details). The formulas obtained are the following :

$$\begin{aligned}\frac{\delta}{x} &= \frac{0.382}{Re_x} \\ \frac{\delta^*}{x} &= \frac{0.0477}{Re_x} \\ \frac{\theta}{x} &= \frac{0.0371}{Re_x}\end{aligned}$$

Reynolds numbers based on the boundary layer thicknesses can be computed, let's define some of them :

$$\begin{aligned}Re_\delta &= \frac{u_p(x)\delta_{99}}{\nu} \\ Re_\theta &= \frac{u_p(x)\theta}{\nu}\end{aligned}$$

The evolution of the thicknesses are shown at the next figures :

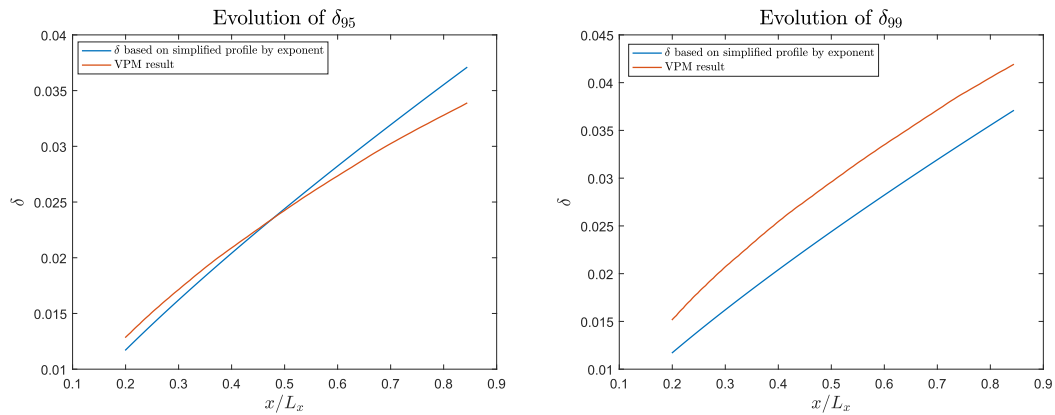


Figure E.11: Boundary layer thicknesses at 95% and 99%

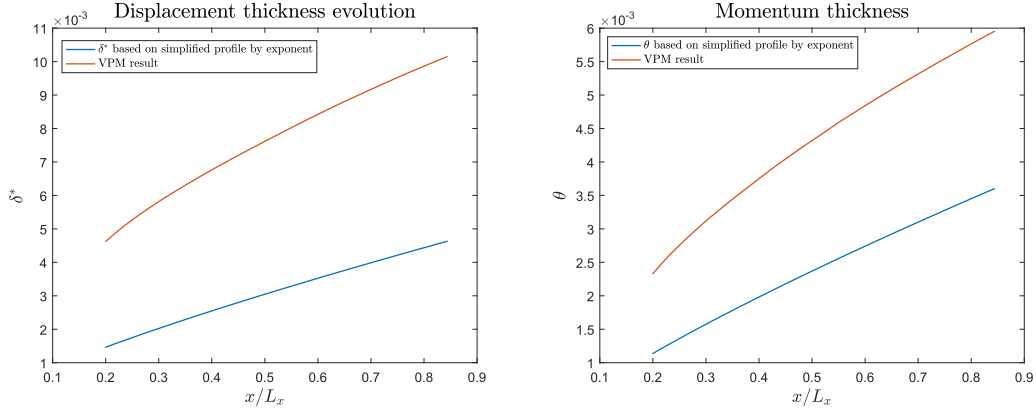


Figure E.12: Displacement and momentum thickness

The main difference comes from the fact that the boundary layer development computed in this section is in a channel. The no-through flow boundary condition above causes a restriction of passing area and the flow accelerates. The velocity above the boundary layer is no-more constant.

The evolution of the Reynolds numbers based on the boundary layer thicknesses are also computed and sketched at the following figures :

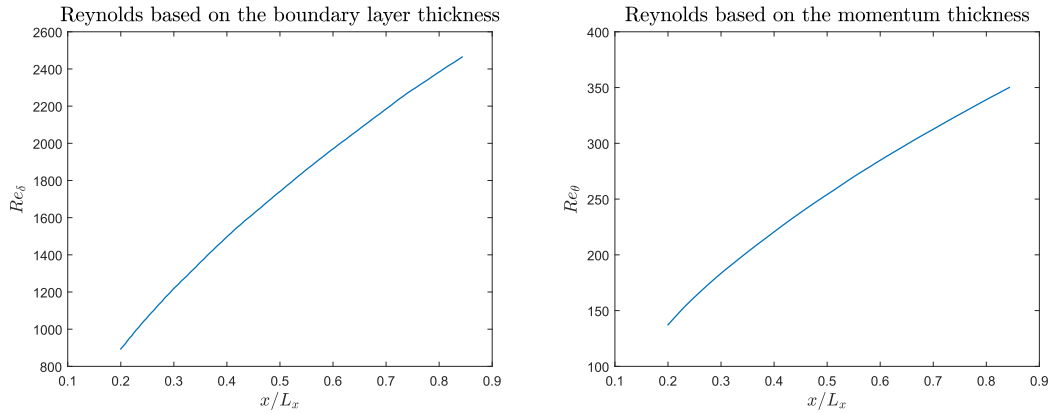


Figure E.13: Reynolds number based on the boundary layer thicknesses

E.4.2 Friction coefficient and friction velocity

The friction velocity is computed in the same way as done in section 5.3. The Reynolds number based on this velocity and on the streamwise location is defined as :

$$Re_x = \frac{U_{\infty} x}{\nu} \quad (\text{E.1})$$

$$u_{\tau}(x) = \sqrt{\tau_w} = \sqrt{\mu \frac{d\bar{u}(x)}{dy}} \quad (\text{E.2})$$

$$Re_{\tau}(x) = \frac{u_{\tau} x}{\nu} \quad (\text{E.3})$$

The evolution of the skin-friction coefficient is directly linked to the friction velocity as they are both based on the skin friction :

$$C_f = \frac{\tau_w}{\frac{1}{2} \rho U_{\infty}^2}$$

The value of Re_{τ} and C_f are sketched in function of the streamwise location and in function of the Re_{θ} :

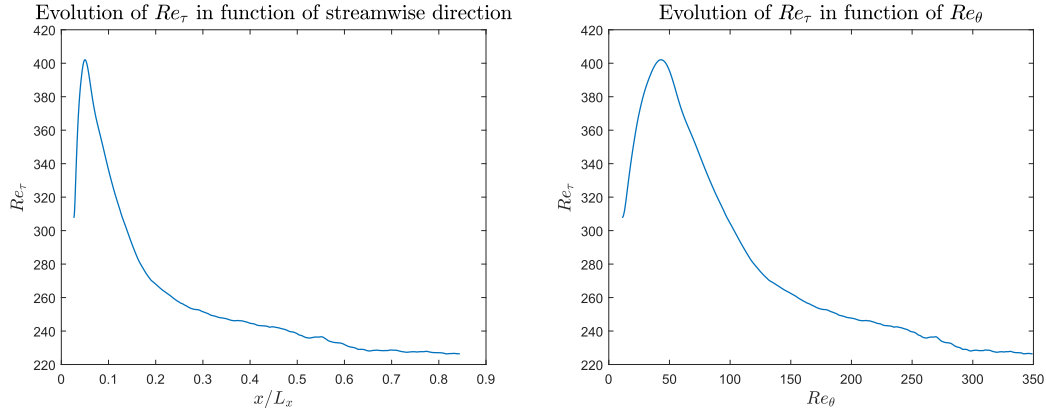


Figure E.14: Evolution of Reynolds number based on friction velocity and streamwise direction in function of the streamwise direction and in function of the Reynolds based on momentum thickness

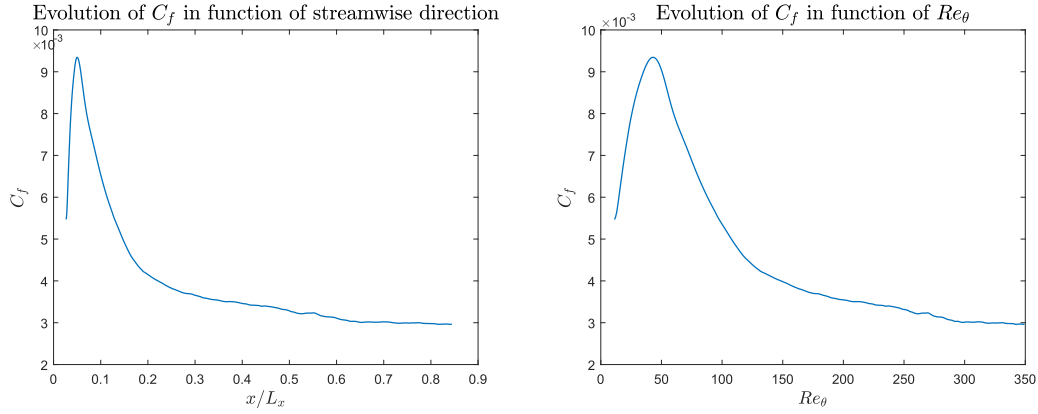


Figure E.15: Evolution of skin friction coefficient in function of the streamwise direction and in function of the Reynolds based on momentum thickness

The transition region at the beginning of the domain is clearly visible. The flow seems to reach a fully turbulent state around $x/L_x \simeq 0.1$, which corresponds to $Re_\theta \simeq 50$.

Finally, the Reynolds number in function of the friction velocity and boundary layer thickness is computed :

$$Re_{\tau,\delta} = \frac{u_\tau \delta}{\nu}$$

The value is plotted in function of the Reynolds based on momentum thickness :

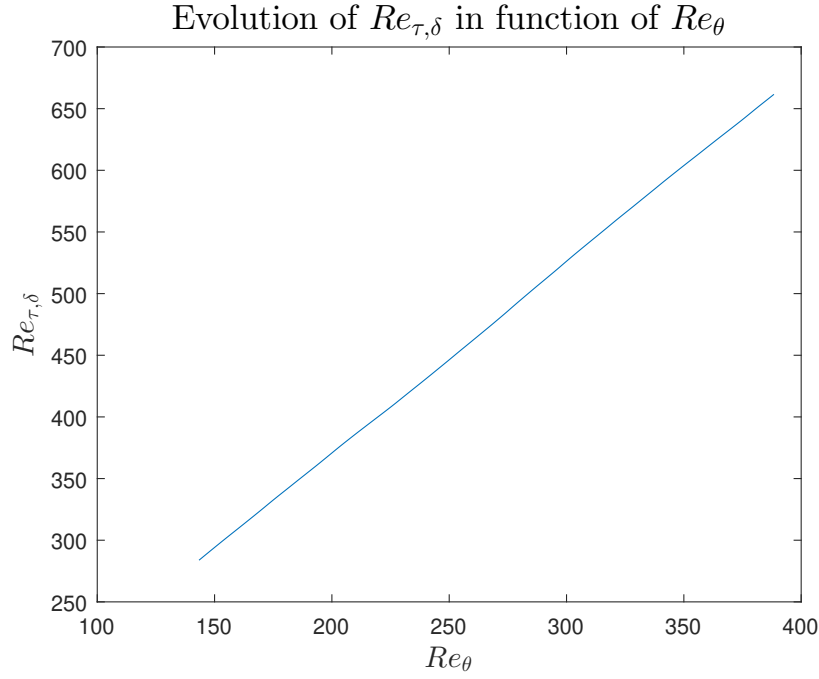


Figure E.16: Evolution of Reynolds number based on friction velocity and boundary layer thickness in function of the Reynolds based on momentum thickness

E.4.3 Average velocity profiles

The statistical spatial development of the boundary layer is presented in a two dimensional slice :

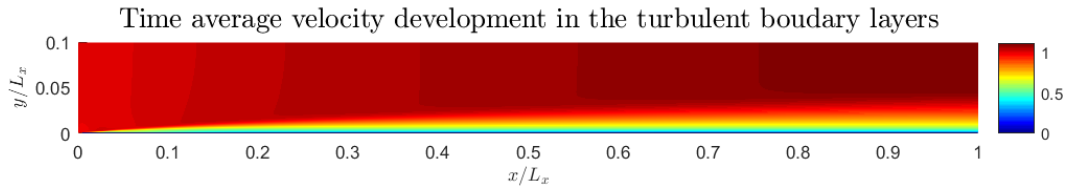


Figure E.17: Statistical velocity evolution in the boundary layer

The statistics totally converge after the 21500 time step and the velocity profiles are stable. One can observe that the turbulent boundary layer grows faster than the laminar one. The turbulent boundary layer reaches $\simeq 4\%$ of the length of the channel at $x/L_x = 0.8$, while a laminar one only reaches 1%.

Different velocity profiles in the streamwise direction are computed. One can observe the acceleration of the flow above the boundary layer. Due to the important development of the boundary layer compared to the height of the channel, the relative acceleration reaches 10% of the U_{∞} .

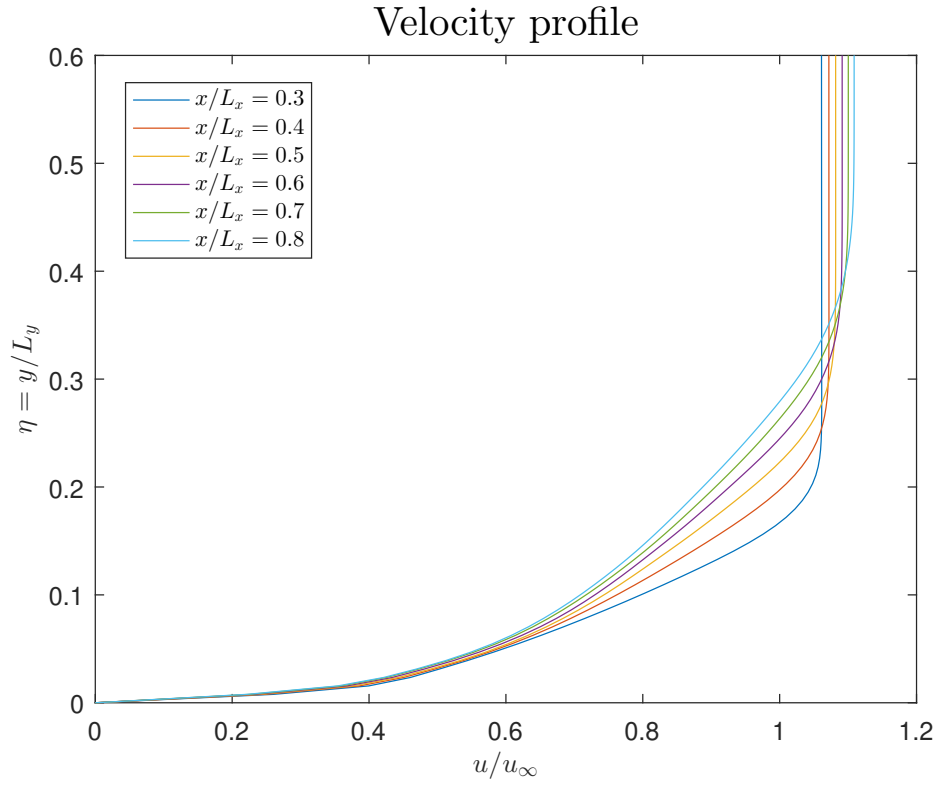


Figure E.18: Velocity profiles at different location in the streamwise direction

The profiles can also be expressed in dimensionless wall distance :

$$y^+ = \frac{yu_\tau}{\nu}$$
$$u_\tau = \frac{u}{u_\tau}$$

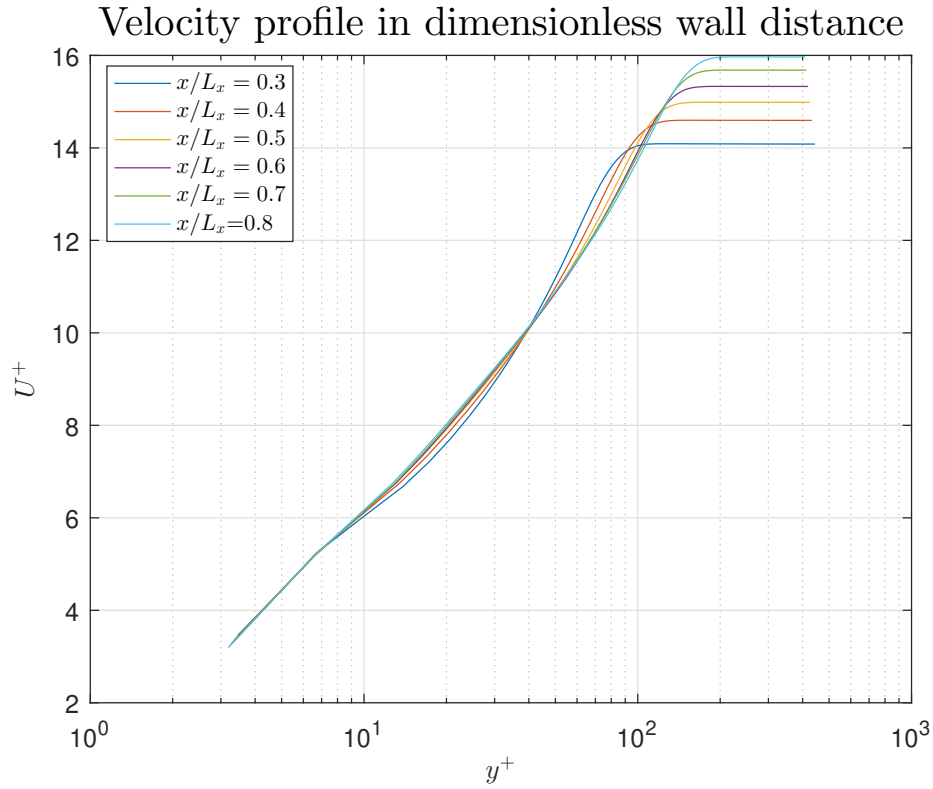


Figure E.19: Dimensionless wall distance velocity profiles

One can clearly observe that the flow in the laminar sub-layer $y^+ \leq 6$ doesn't have enough grid points. The concavity of the profiles seem to be in the opposite direction to what it should be. This is one of the main argument to say that the simulation is a little bit under resolved.

Finally the velocity profile in the wall normal direction are sketched at the next figure :

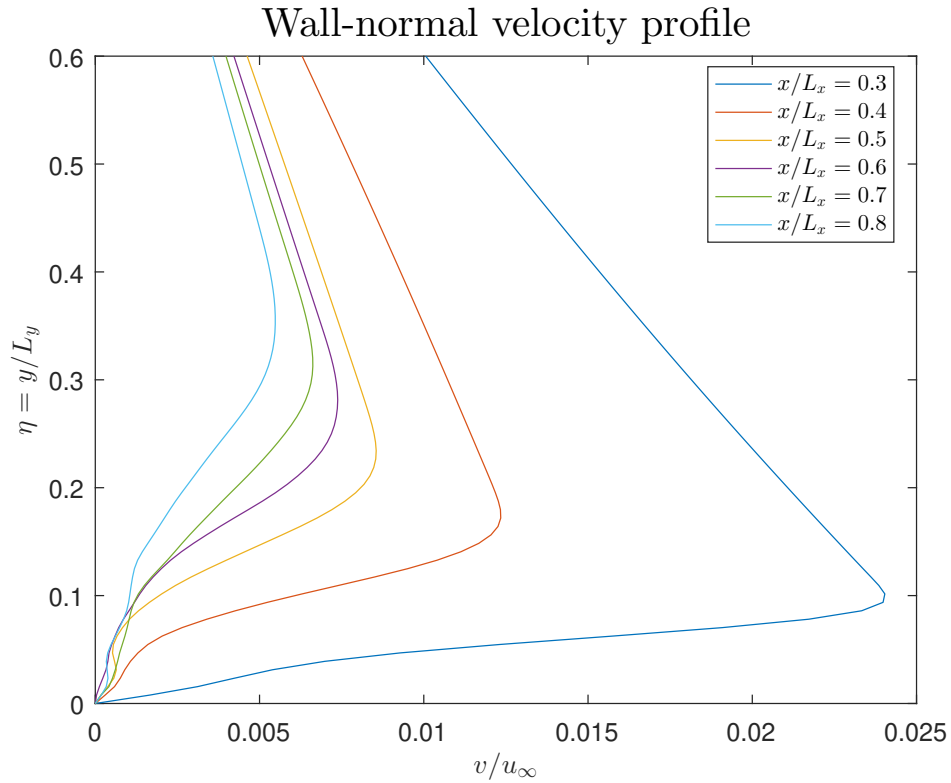


Figure E.20: Wall-normal velocity profile

E.5 Conclusion

This appendix has shown the direct numerical simulation of a turbulent boundary layer development in a channel. The classical limitations of the simulation in a channel are still in application (see 4.3 for more detail).

It is pretty hard to find a good numerical setup. It is a trade-off between accuracy and computational costs. A setup is proposed: not too expensive, but a little bit under resolved in the laminar sub-layer.

A lot of two and three dimensional instantaneous views for the velocity and vorticity fields are shown to appreciate the quality of the method.

Statistics are also made on the flow, the boundary layer thicknesses and Reynolds numbers are computed and sketched. The appendix also presents some of velocity profiles at different locations in the streamwise direction.

The results are quite satisfactory despite the under-resolution in the laminar sub-layer. A next step would be to run the simulation again with a mesh twice finer and perform an in-depth analysis of the turbulent boundary layer.

Bibliography

- [1] S. Backaert., *Development of a vortex particle-mesh method for the large eddy simulation of wind turbine aerodynamics and wakes. Master's thesis, UCL, 2011*
- [2] P. Chatelain, A. Curioni, M. Bergdorf, D. Rossinelli, W. Andreoni, and P. Koumoustakos. *Billion vortex particle direct numerical simulations of aircraft wakes. Computer methods in applied mechanics and engineering, 2008.*
- [3] Jacquemin, Gaëtan. *Interaction d'anneaux vortex de circulations différentes* Ecole polytechnique de Louvain, Université catholique de Louvain, 2017. Prom.: Winckelmans, Grégoire ; Chatelain, Philippe
- [4] P. R. SCHATZLE. *An Experimental Study of Fusion of Vortex Rings. Thèse de doct. Pasadena, California: California Institute of Technology, mai 1987.*
- [5] Myoungkyu Lee and Robert D. Moser *Direct numerical simulation of turbulent channel flow up to $Re_\tau = 5200$, 2015, Journal of Fluid Mechanics, vol. 774, pp. 395-415*
- [6] Grégoire Winckelmans, *Encyclopedia of Computational Mechanics, Part 1. Fluids, Vortex Methods, pp. 129-153*
- [7] *Georges-Henri Cottet, Petros D. Koumoutsakos, Vortex Methods Theory and Applications (2000)*
- [8] *Vincent Legat, Grégoire Winckelmans, MECANIQUE DES FLUIDES ET TRANSFERTS I, 2017-2018*
- [9] *Philippe Chatelain, Matthieu Duponcheel, Denis-Gabriel Caprace, Yves Marichal1, and Grégoire Winckelmans, Vortex particle-mesh simulations of vertical axis wind turbine flows: from the airfoil performance to the very far wake, June 2017*
- [10] *Philippe Chatelaina, Yves Marichal, Matthieu Duponcheel, Maud Moensa, Grégoire Winckelmans, Vortex methods applied to the large eddy simulation of wind turbine flows*
- [11] *Buckingham, Sophia., Prandtl number effects in abruptly separated flows : LES and experiments on an unconfined backward facing step flow. Prom. : Winckelmans, Grégoire ; Bartosiewicz, Yann*
- [12] *T. H. New and B. Zang, Head-on collisions of vortex rings upon round cylinders.*

UNIVERSITÉ CATHOLIQUE DE LOUVAIN
École polytechnique de Louvain

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