Project LMECA2660 : Natural convection

1 Introduction

Parler de hauteur H = 3/2 L

1.1 General equations

The system of equations to be solved is a simplified formulation of the general Navier-Stokes equations for an incompressible fluid after considering several assumptions. The simplification assumptions we made are the following:

1. The viscous dissipation is considered negligible compared to the other terms of the temperature equation, so the Eckert number of the flow is $Ec = \frac{U^2}{c_p \Delta T} << 1$
2. The Boussinesq hypothesis is applied. It is therefore considered that the density of the fluid evolves with its temperature variation while maintaining the assumption of incompressibility of the latter. The mass of a parcel of fluid is then expressed as a function of temperature: $d\rho = -\beta \Delta T$ where $\beta = \frac{d\rho}{dT}$ is the thermal expansion coefficient of the considered fluid.

Thanks to these approximations, the equation of the problem of natural convection of a fluid in a box :

$$\nabla \cdot \mathbf{v} = 0 \tag{1.1}$$

$$\frac{D\mathbf{v}}{Dt} = \frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} = -\nabla P + \nu \nabla^2 \mathbf{v} - \beta (T - T_\infty)\mathbf{g} \tag{1.2}$$

$$\frac{DT}{Dt} = \frac{\partial T}{\partial t} + \mathbf{v} \cdot \nabla T = \alpha \nabla^2 T \tag{1.3}$$

This system of equations constitutes the Navier-Stokes equations in the case of our problem in which the following parameters are involved:

- $\mathbf{v}$ $[m/s]$:the two-dimensional velocity vector of the fluid, the problem being plane.
- T $[K]$: the temperature of the fluid.
- $\nu = \frac{\mu}{\rho}$ $[m^2/s]$: the kinematic viscosity of the fluid, it is assumed to be constant considering the variations of density due to temperature negligible.
- $\alpha = \frac{\kappa}{\rho}$ $[m^2/s]$: the thermal diffusivity of the fluid, it is assumed constant for the same reasons as the kinematic viscosity.
- $\mathbf{g} = -g\hat{\mathbf{e_y}}$ $[m/s^2]$: the gravitational acceleration assumed constant.
- $P = \frac{(p - p_{ref}) + \rho g y}{\rho}$ $[m^2/s^2]$: the kinematic pressure for an incompressible fluid. It is the dynamic pressure obtained with Bernouilli's theorem divided by the density (which is still considered as constant here). We can see that the pressure is a function of the vertical displacement y, however we will neglect in this report this dependence in the pressure term. Indeed, only the pressure gradient is involved in the equations and this variation in y contributes relatively little to it.

1.2 Non-dimensionalization of the system

In order not to have many assumptions about the characteristic dimensions, it is customary in fluid mechanics to scale the problems to be solved. We have therefore scaled the problem so that our system can be described only in terms of dimensionless numbers. We have thus proceeded to the following adimensionalizations:

$$\hat{x} = \frac{x}{H} \quad \hat{y} = \frac{y}{H} \quad \hat{t} = \frac{tU}{H} \quad \hat{\nabla}\mathbf{a} = H\nabla\mathbf{a}$$

$$\hat{u} = \frac{u}{U} \quad \hat{v} = \frac{v}{U} \quad \hat{P} = \frac{P}{U^2} \quad \Theta = \frac{T - T_{inf}}{\Delta T}$$

where we dimensioned all the main variables of the system thanks to the characteristic values of the height of the fluid domain H and the reference speed of the natural convection $U = \sqrt{\beta g \Delta T H}$. Looking carefully at the units of each term and knowing that the Grashof and Prandtl numbers are defined respectively as $Gr = (\frac{UH}{\nu})^2 = \frac{\beta g \Delta T H^3}{\nu^2}$ et $Pr = \frac{\nu}{\alpha}$.

We can then write the Navier-Stokes system of our problem in non-dimensional :

$$\frac{\partial \hat{u}}{\partial \hat{x}} + \frac{\partial \hat{v}}{\partial \hat{y}} = 0 \tag{1.4}$$

$$\frac{D\hat{u}}{D\hat{t}} = -\hat{\nabla}P + \nu\hat{\nabla}^2\hat{u} \tag{1.5}$$

$$\frac{D\hat{v}}{D\hat{t}} = -\hat{\nabla}\hat{P} + \frac{1}{\sqrt{Gr}}\hat{\nabla}^2\hat{v} - \Theta \tag{1.6}$$

$$\frac{D\Theta}{D\hat{t}} = \frac{1}{\sqrt{Gr}Pr}\hat{\nabla}^2\Theta \tag{1.7}$$

We see that the only parameters to select in this system are the Grashof number Gr, which characterizes the natural convection, and the Prandtl number Pr, which is only dependent on the fluid and not on the flow and which characterizes the impact of viscous diffusion compared to thermal diffusion. In the case of our problem, we will consider that $Gr = 10^{10}$ and $Pr = 4$. If the Grashof number is of such high order, this is because the kinematic viscosity involved in the denominator of the Grashof expression is of very low order. In the rest of this report, for the sake of clarity and simplicity in the notations we will dispense with the notation $\hat{}$ to deal with adimensional variables. Thus, unless otherwise specified, we will consider the variables as being all dimensionless.

1.3 Boundary conditions

In order for our problem to have a unique solution and to be completely determined, it must have two boundary conditions on the temperature and the velocities as well as an initial condition for each of the variables, which vary with time. In our case, we consider that the plate starts to heat up at time $t = 0$, all variables being zero before this time. The fluid in $t = 0$ is thus static, at constant temperature and the variation of the hydrostatic pressure with altitude is neglected in our case.

Concerning the boundary conditions of our problem, we know that a no-slip condition is applied for the three walls of the box, the two velocity components are imposed zero by a Dirichlet condition. On the other hand, at the free surface, the horizontal velocity u is not necessarily zero, and we simply impose a Neumann condition so that the fluid remains flat at the interface. The vertical velocity then remains zero at this interface. As far as the temperature is concerned, the walls being considered as adiabatic, the heat transfer is null. Moreover, the heat flow on the bottom wall being constant, 3 Neumann conditions are to be imposed on each wall of the box. The flux on the bottom wall being $q_w = \frac{k\Delta T}{H}$, the dimensionalization gives :

$$q_w = \frac{k\Delta T}{H} = -k\frac{\partial T}{\partial y} = -\frac{k\Delta T}{H}\frac{\partial \Theta}{\partial \hat{y}} \iff \frac{d\Theta}{dy} = -1$$

Finally, at the free surface, we consider that the convective heat flux (Newton's law) must be identical to the internal conduction in the fluid (Fourier's law). Since Newton's law involves the temperature and Fourier's law the derivative of the temperature, we have a Robin condition at the free surface. We define for this condition the number $l_0 = \frac{k}{h_c}$ with k the thermal conduction coefficient of the fluid and h_c the convection coefficient at the free surface. In our problem, we will impose the value of l_0, after adimensionalization (the coefficient with the dimension of a length) to $l_0 = 10^{-3}$.

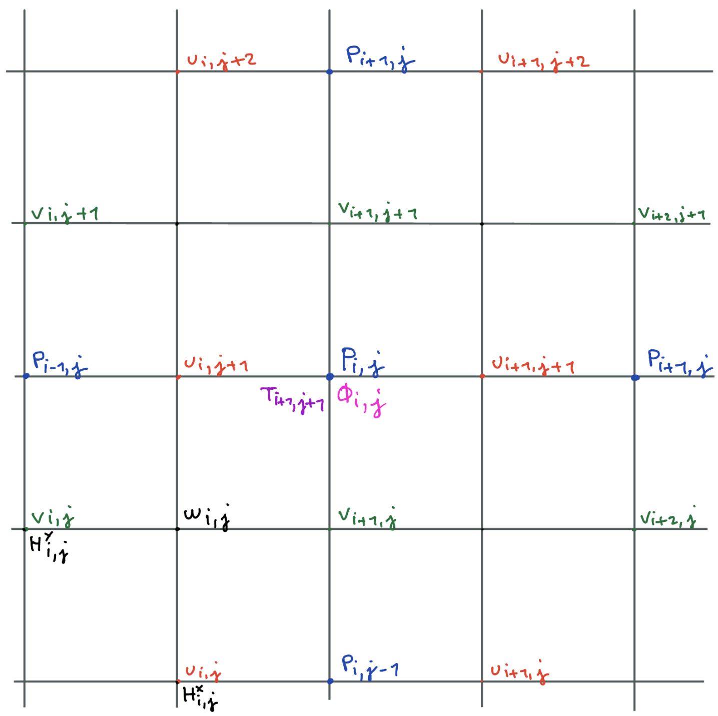
The table 1 groups all the boundary conditions of our problem:

	Left wall	Right wall	Bottom wall	Free surface
u	$u = 0$	$u = 0$	$u = 0$	$\frac{du}{dy} = 0$
v	$v = 0$	$v = 0$	$v = 0$	$v = 0$
Θ	$\frac{d\Theta}{dx} = 0$	$\frac{d\Theta}{dx} = 0$	$\frac{d\Theta}{dy} = -1$	$l_0 \frac{\partial \Theta}{\partial y} + \Theta = 0$

Table 1: Summary of the boundary conditions

2 Numerical implementation

2.1 Marker and Cell (MAC) Stagered grid

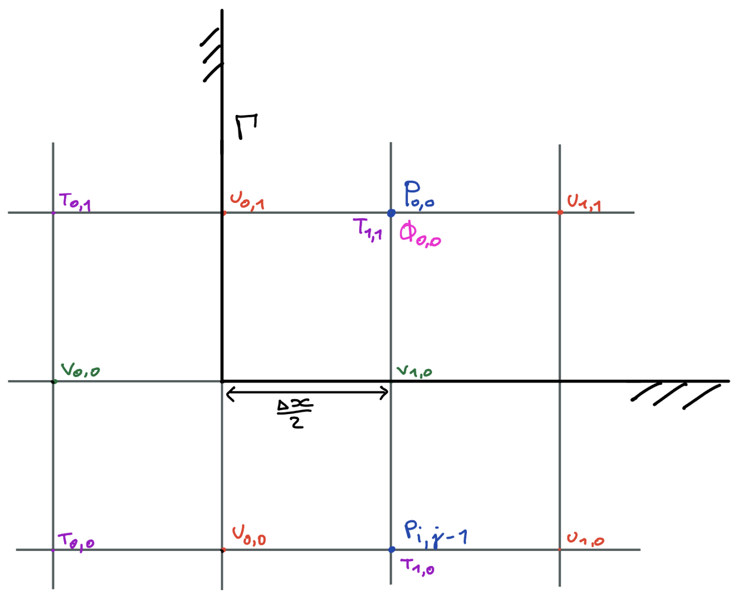


Pros
The MAC Staggered grid enables to avoid odd-even decoupling between the pressure and velocity. Let's color a grid in even and odd cells like a chess board where all elements are stored at the same position. When we make our centered differences the terms on the odd (black) cells will only be matched with the terms on the even (white) cells. The MAC mesh avoids this discretization error.
Cons
The different locations of the different variables make the discretization of the different terms of the Navier-Stokes equations more computationally heavy (see 2.2).

2.1.1 Ghost points

The discretized horizontal velocity u of the mesh has no values on the lower and upper edges of the domain. The vertical velocity v discretized from the mesh has no values on the lateral edges of the domain. The temperature never has values discretized on the edges of the domain. The use of ghost points allows to impose the boundary conditions shown in the table 1. The previous formula was found using a smart combination of the Taylor developments of v (or every other quantitiy needing ghost points) to minimize the error order (to impose the value of the boundary condition at the wall).

$$v_0 = -\frac{1}{5}(v_3 - 5v_2 + 15v_1 - 16v_\Gamma)$$



These ghost points explain why there is a mismatch between the pressure indices and those of temperature and velocity.

For the Dirichlet conditions as the no-slip conditions at the wall, we used the formula above with $V_\Gamma = 0$.

For the Neumann condition on u ($\frac{du}{dy} = 0$) and on T ($\frac{d\Theta}{dy} = 0$ or -1), in these cases we simply linked the value of our ghost point to its nearest value in the box. As an example, for the temperature flux on the lower wall, our ghost point value was simply : $T_{Ghost} = T_0 = T_1 + \delta y$ because we discretized the vertical gradient on temperature with the Explicit Euler formula.)

We did the same for our Robin condition on the temperature ($l_0 \frac{\partial \Theta}{\partial y} + \Theta = 0$) but this time we used another value for V_γ. Indeed, the temperature gradient is not computed is not calculated at the same point as the temperatures. We therefore used the average of the temperature value between the ghost point and the first point in the domain. The formula for T_0 becomes: $T_{m+1} = \frac{2l_0 - \Delta y}{2l_0 + \Delta y} T_m$ also with a Euler formula for the discretization of the gradient.

2.2 Spatial discretization of the terms of the Navier-Stokes equation

Divergence form

$$\begin{aligned}
(\mathbf{v} \cdot \nabla \mathbf{v})_x =& \frac{\partial}{\partial x}(uu) + \frac{\partial}{\partial y}(uv) \\
H^x_{i,j} =& \frac{1}{2}\Big(\frac{u_{i+1,j} + u_{i,j}}{2} \frac{u_{i+1,j} + u_{i,j}}{\Delta x} - \frac{u_{i,j} + u_{i-1,j}}{2} \frac{u_{i,j} - u_{i-1,j}}{\Delta x} \Big) \\
&+ \frac{1}{2}\Big(\frac{u_{i,j+1} + u_{i,j}}{2} \frac{v_{i+1,j} + v_{i,j}}{\Delta y} - \frac{u_{i,j} + u_{i-1,j}}{2} \frac{u_{i,j} - u_{i-1,j}}{\Delta y} \Big)
\end{aligned}$$

$$
\begin{aligned}
(\mathbf{v}\cdot\nabla\mathbf{v})_y =& \frac{\partial}{\partial x}(vu) + \frac{\partial}{\partial y}(vv) \\
H^y_{i,j} =& \frac{1}{2}\Big(\frac{u_{i,j+1}+u_{i,j}}{2}\frac{v_{i+1,j}+v_{i,j}}{\Delta x} - \frac{u_{i-1,j+1}+u_{i-1,j}}{2}\frac{v_{i,j}+v_{i,j}}{\Delta x}\Big) \\
&+ \frac{1}{2}\Big(\frac{v_{i,j+1}+v_{i,j}}{2}\frac{v_{i,j+1}+v_{i,j}}{\Delta y} - \frac{v_{i,j}+v_{i,j-1}}{2}\frac{v_{i,j}+v_{i,j-1}}{\Delta y}\Big)
\end{aligned}
$$

2.2.1 Convective term of the temperature

2.2.2 Pressure gradient

$$
(\nabla_h P^n) = \begin{pmatrix} (\nabla_h P)^x_{i,j} = \frac{P_{i,j-1}-P_{i-1,j-1}}{\Delta x} \\ (\nabla_h P)^y_{i,j} = \frac{P_{i-1,j}-P_{i-1,j-1}}{\Delta y} \end{pmatrix}
$$

2.2.3 Laplacian of the velocity

$$
(\nabla^2_h \mathbf{v}) = \begin{pmatrix} (\nabla_h u)_{i,j} = \frac{u_{i+1,j}-2u_{i,j}+y_{i-1,j}}{\Delta x^2} + \frac{u_{i,j+1}-2u_{i,j}+y_{i,j-1}}{\Delta x^2} \\ (\nabla_h v)_{i,j} = \frac{v_{i-1,j}-2v_{i-1,j-1}+v_{i,j}}{\Delta y^2} + \frac{v_{i,j+1}-2v_{i,j}+v_{i,j-1}}{\Delta y^2} \end{pmatrix}
$$

2.2.4 Laplacian of the temperature

$$
(\nabla^2_h \mathbf{T}_{i,j}) = \frac{T_{i-1,j}-2T_{i,j}+T_{i+1,j}}{\Delta x^2} + \frac{T_{i,j-1}-2T_{i,j}+T_{i,j+1}}{\Delta y^2}
$$

2.3 Navier-Stokes solver

We have now spatially discretized all the terms of the Navier-Stokes equations of an incompressible fluid.

2.3.1 ADI method

At each time iteration, we solve the following numerical equations.

$$
\mathbf{v}^* = \mathbf{v}^n + \Delta t\Big(-\frac{1}{2}(3\mathbf{H}^n_h - \mathbf{H}^{n-1}_h) - \nabla_h P^n + \nu\nabla^2_h\mathbf{v}^n - \beta(T^n - T_\infty)\mathbf{g}\Big) \tag{2.1}
$$

$$
\nabla^2_h\phi = \nabla_h\cdot\mathbf{v}^* \tag{2.2}
$$

$$
\mathbf{v}^{n+1} = \mathbf{v}^* - \Delta t\nabla_h\phi \tag{2.3}
$$

$$
P^{n+1} = P^n + \phi \tag{2.4}
$$

$$
T^{n+1} = T^n + \Delta t\Big(-\frac{1}{2}(3\mathbf{H}^n_{T,h} - \mathbf{T}^{n-1}_{T,h}) + \alpha\nabla^2_h\mathbf{T}^n\Big) \tag{2.5}
$$

$$
\tag{2.6}
$$

All equations are explicit except the poisson equation (2.2)which is implicit.

The ADI method first computes an estimate of the velocity by integrating the discretized equation of conservation of momentum. This velocity is only an estimate of the velocity because we do not respect the assumption of incompressibility which is in the conservation of mass equation. The resolution of the poisson equation (2.2) allows to obtain ϕ which is a corrective term. This one imposes the incompressibility of the fluid. Its Laplacian is equal to the divergence of the estimated velocity. In an incompressible fluid, the conservation of mass equation imposes zero mass. The corrected velocity is calculated with the ϕ term by the equation (2.3) and the pressure by (2.4). The principle of the method is to obtain an initial predictor v^* and to project it on the $\mathbf{v}$ space so that the incompressibility condition on the velocity is respected.

The temperature is not calculated with this reprojection system and is simply integrated at each time step by Adams-Bashfort of order two.
The convective terms are integrated using the second-order Adams-Bashforth method. The other terms are all computed with explicit euler of order one.

2.3.2 Poisson solver

We used a Poisson solver that was given to us in the course and in which we used the PETSc library. This solver was useful for the reprojection of $\mathbf{v}^*$ so that the velocity respects the continuity equation at all points. The operation of this Poisson solver is detailed in the document "PETSc installation notes". The Poisson equation being an elliptic equation, the solutions of all the points of the domain are all interdependent. This allows to force the continuity equation in our system.

2.4 Penalization

The second part of our problem consisted in using a penalization method to involve the rotation of a helix inside our fluid.

We applied a Brinkman penalization method. This consists in adding a term to the equation in the points of the domain in contact with the helix. The aim of this method is to force the fluid to adopt the same speed as the propeller thanks to this new term. The dimensionless Navier-Stokes equations thus become :

$$\frac{D\mathbf{v}}{Dt} = -\nabla P + \frac{1}{\sqrt{Gr}}\nabla^2\mathbf{v} - \Theta - \chi\frac{(\mathbf{v} - \mathbf{v_s})}{\Delta\tau} \tag{2.7}$$

$$\frac{D\Theta}{Dt} == \frac{1}{\sqrt{Gr}Pr}\nabla^2\Theta - \chi\frac{(\Theta - \Theta_s)}{\Delta\tau} \tag{2.8}$$

In the above equations, the parameter χ is a mask function, which is 0 outside the propeller and one at the common abcissa. The parameter $\Delta\tau$ characterizes the impact of the helix precense in the fluid. When this parameter is small, the term added by the penalty in the equations will take a greater importance in the system. When this coefficient tends to 0, it can be shown that the penalization that the real equations of Navier Stokes. In our case, we considered a $\Delta\tau = 10^{-3}\Delta t$. The parameters v_s and T_s are respectively the velocity of the points on the helix and the average temperature of the points on the disk of radius D delimited by the blade's tip. One of the points of interest during the implementation of the penalty was that we had to be careful with the MAC mesh that the different quantities (the vertical velocity v, the horizontal velocity u and the temperature T) are not defined at the same place and so we had to take this into account when defining the mask functions for each of the equations.

2.5 Stability

To ensure the stability of our solution, the fineness of our spatial mesh must respect certain relations with our temporal discretization. The two relations that must be respected between the two are the CFL (Current-Friedrich-Current) condition and a condition on the Fourier number, specific to heat transfer problems.

The CFL condition imposes that, for a numerical method in CFD to be stable, it must respect the relation $\frac{|mathbfv|_{max}\Delta t}{h} < C_1$ where C_1 is a parameter depending on the numerical method used. $|v|$ is the temporal and spatial maximum of the velocity of our flow. For a time step that respects this relation, the method is indeed able to capture all the higher frequency variations of the system. We have read in the literature that typical values of CFL for our type of problem were of the kind of $C_1 = 0.75$. We therefore used this value for our selection criterion.

The Fourier coefficient is defined as the ratio between the characteristic diffusion time and the square of the mesh fineness $r = \frac{\nu\Delta t}{h^2}$. In the case where only dimensionless parameters are used, this coefficient is $\frac{\nu\Delta t}{Re_H h^2}$, where H is the height of the box. In the literature, we have seen cases where this number must be less than 0.3 for an acceptable simulation. This is the value we used.

We have therefore a double condition for the value of our time step on the CFL and the Fourier coefficient to guarantee the stability of our method. In our case, we imposed a dimensionless h of 1/300 and a dimensionless time step of 1/300.

3 Results

3.1 Part I : Empty box

This part will discuss the results we have obtained and the physical meaning of the evolution of the solution in the case where the box is completely filled with our incompressible liquid.

Fields results

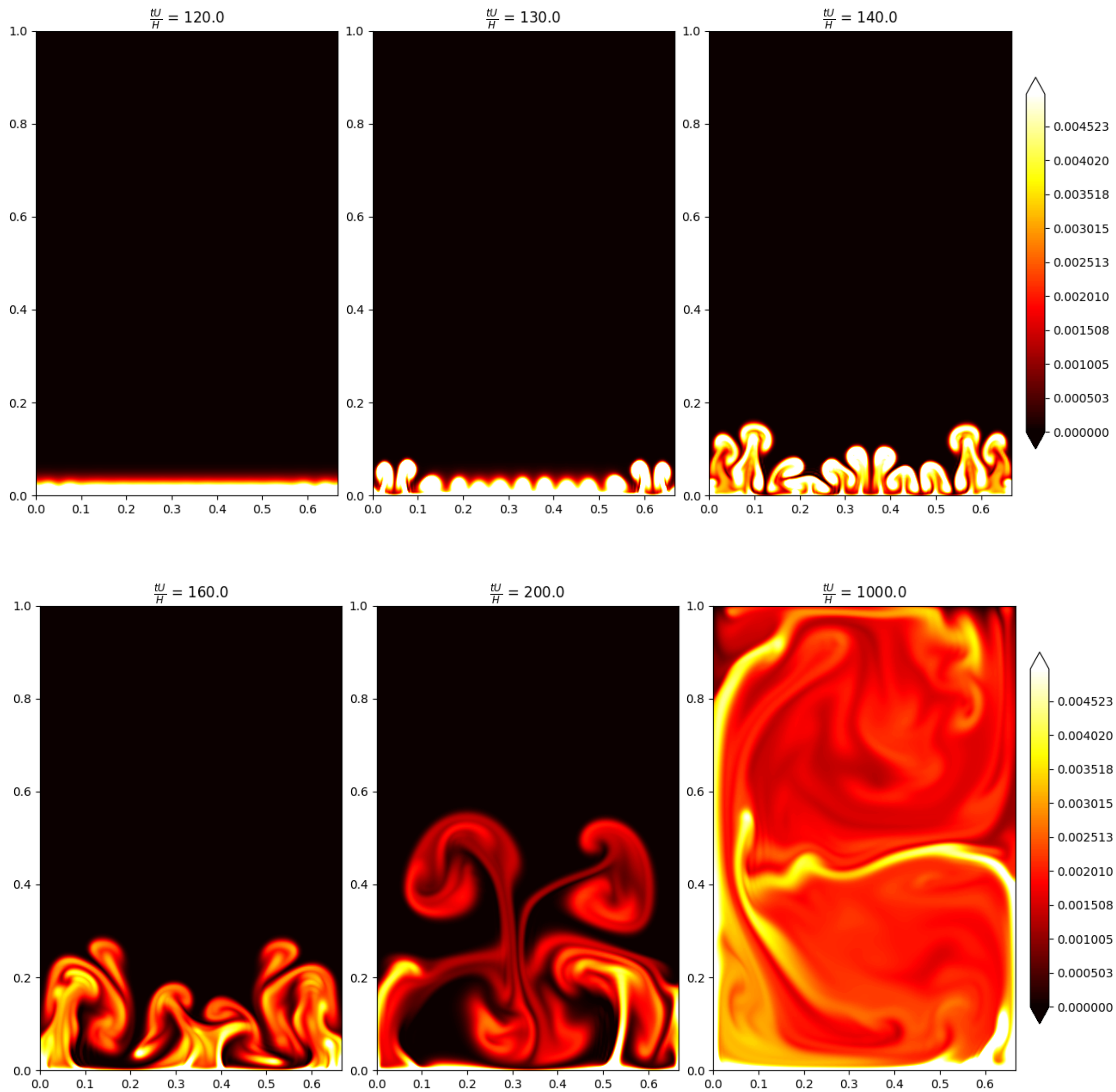
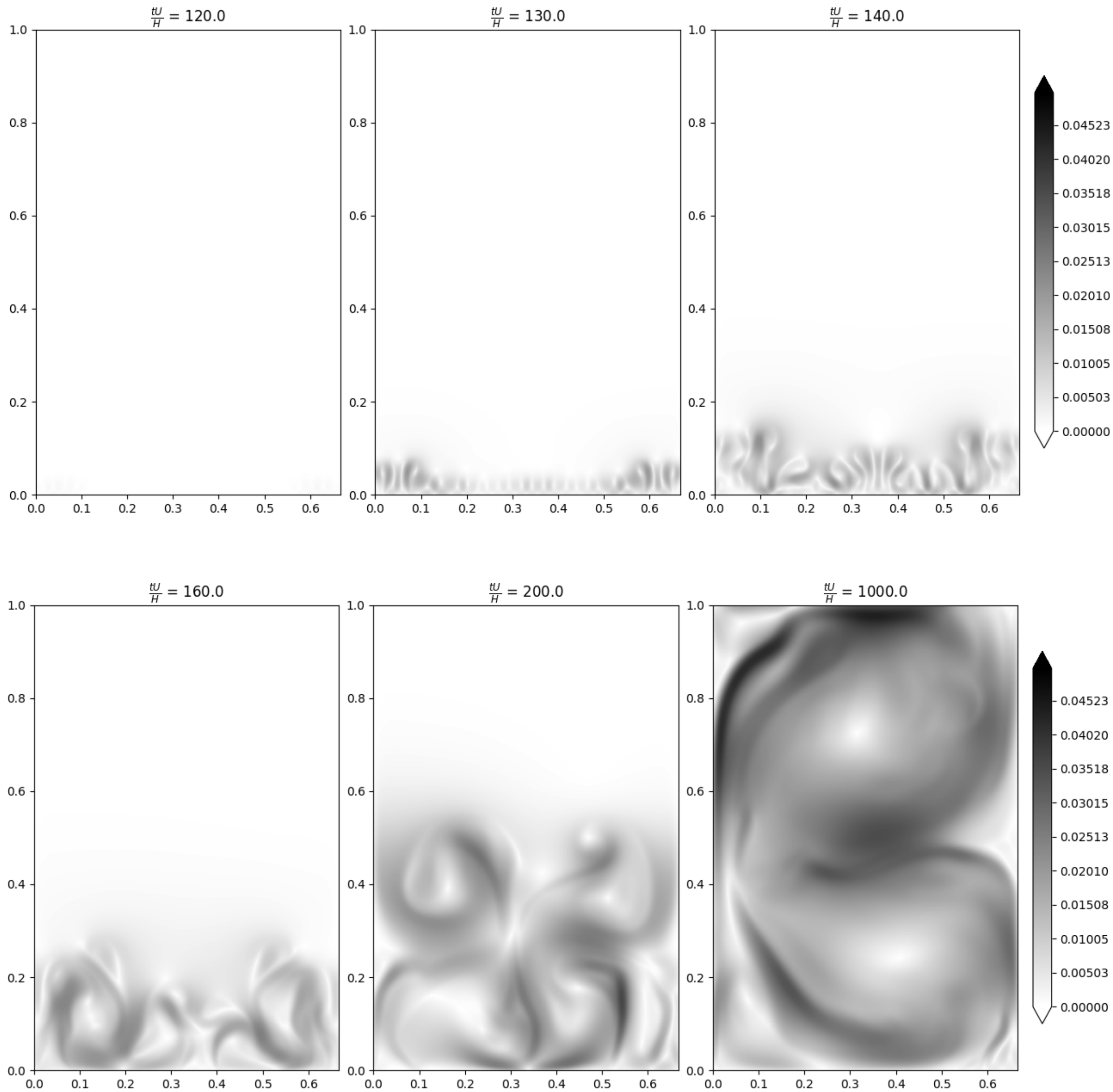


Figure 1

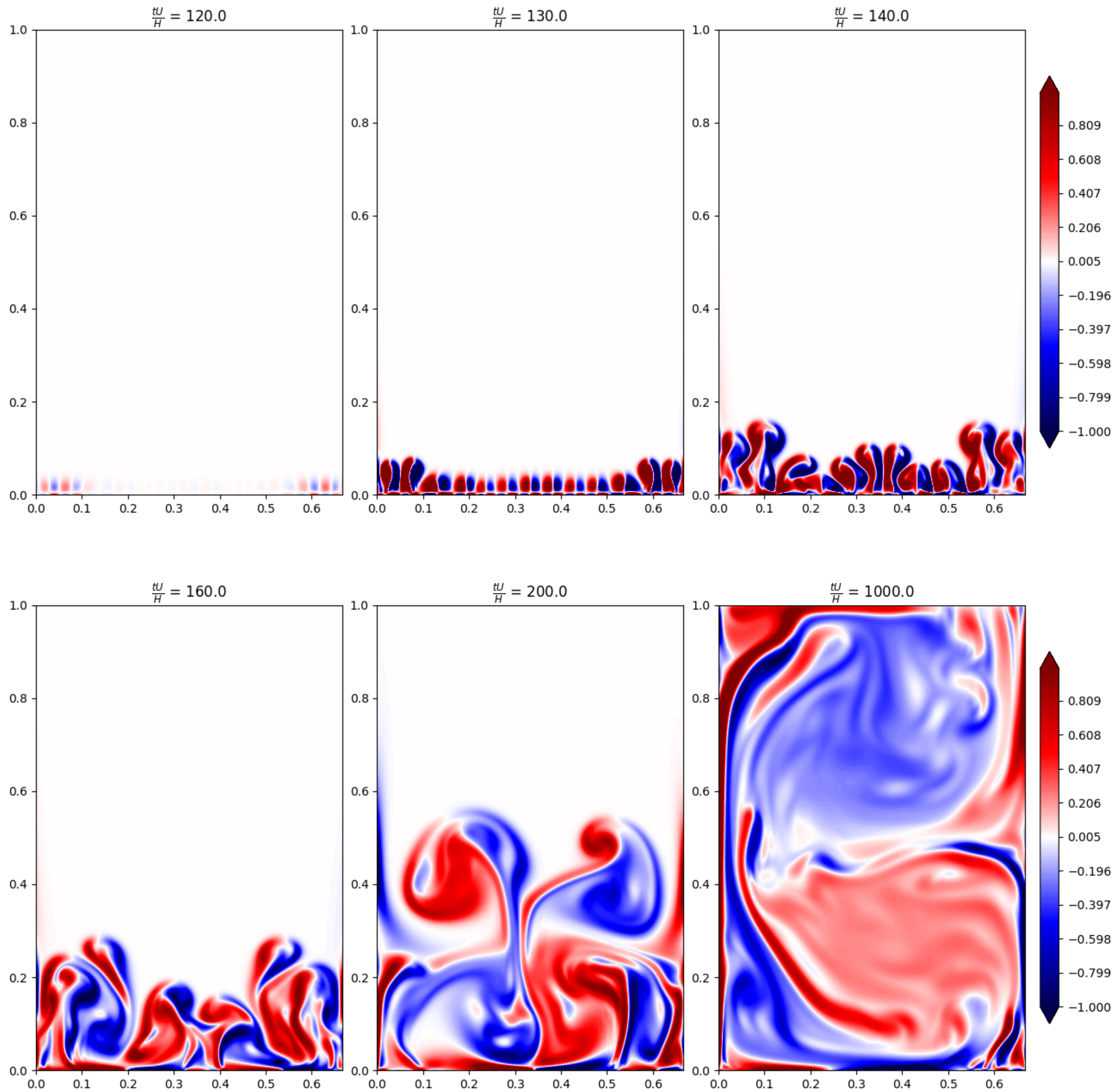
The figure 1 represents the evolution of the temperature as a function of time. We notice that up to a dimensionless time of about $t = 120$, the temperature variation occurs only in the very near wall area where the heat flow is involved. At this time, the diffusion term is the most impacting in the temperature expression. Around the time 130 appears the transition zone, in which the convective motion of the fluid will start to appear. This transition is organized during a certain time until the variation of temperature intervenes in all the box. This is when we will start to see the appearance of connvection cells in the box, the boundary conditions blocking the continuous expansion of the temperature. The condition on the outgoing flow prevents that all the incoming flow is evacuated from the box, which will generate a general heating of the box, which is well visible on the figure 1 at time $t = 1000$.



7

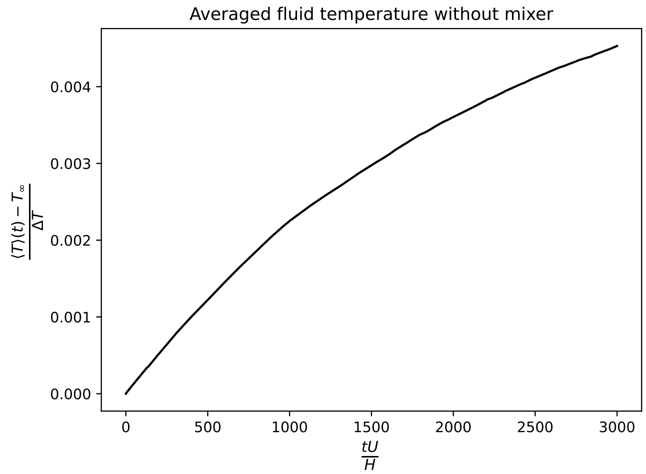
Figure 2

The behavior of the norm of our velocity as a function of time is visible in figure 2, this behavior is globally the same as that of the temperature. Indeed, logically, the speed increases from the movement where the natural convection really takes place. We can see that where the temperature seemed very homogeneous at time $t = 1000$, here the norm of the velocity is very variable according to the position in the box, a cell of convection in the shape of 8 having appeared. This convection cell is due to the fact that the fluid having cooled down near the top of the box will go down in this one while the hot one will go up. The horizontal component u of the velocity is due to the impact of the convective terms in its equation.

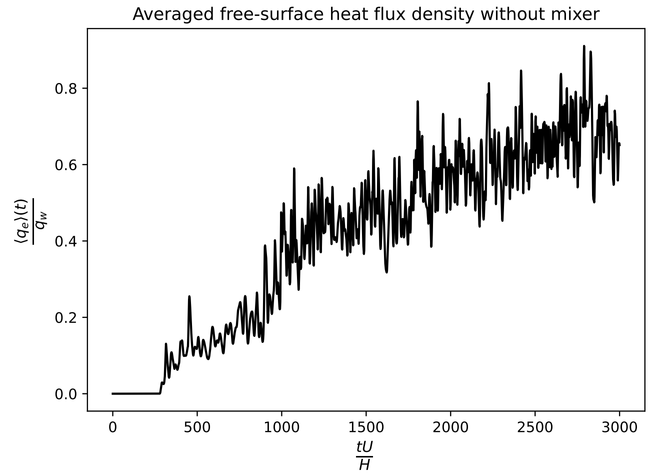


$\frac{tU}{H}$ = 120.0
$\frac{tU}{H}$ = 130.0
$\frac{tU}{H}$ = 140.0
$\frac{tU}{H}$ = 160.0
$\frac{tU}{H}$ = 200.0
$\frac{tU}{H}$ = 1000.0
0.0
0.2
0.4
0.6
0.8
1.0
0.1
0.3
0.5
0.6
0.809
0.608
0.407
0.206
0.005
−0.196
−0.397
−0.598
−0.799
−1.000

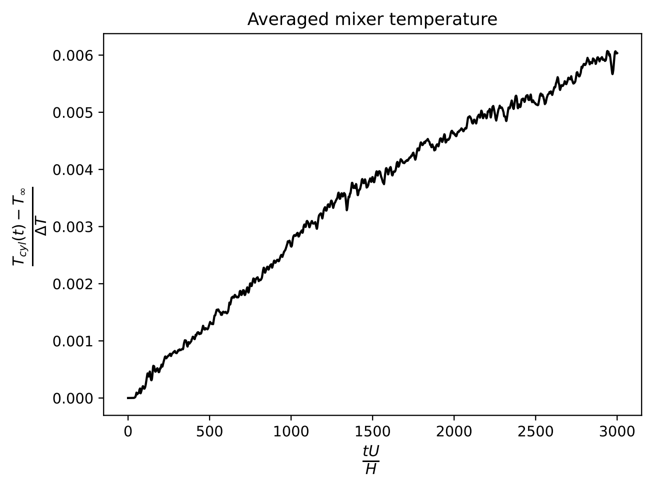
Diagnostics



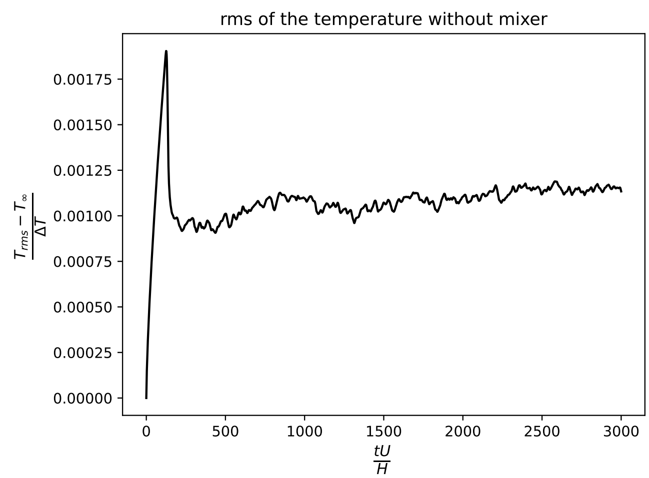
(a) Averaged fluid temperature with the mixer



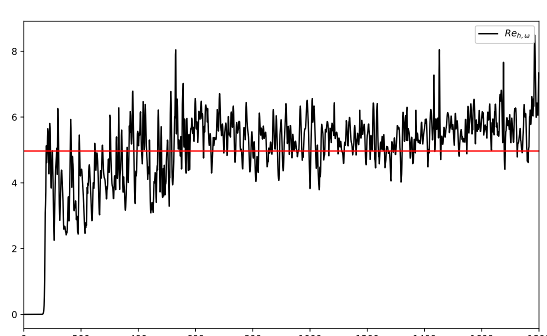
(b) Averaged free-surface heat flux density with the mixer



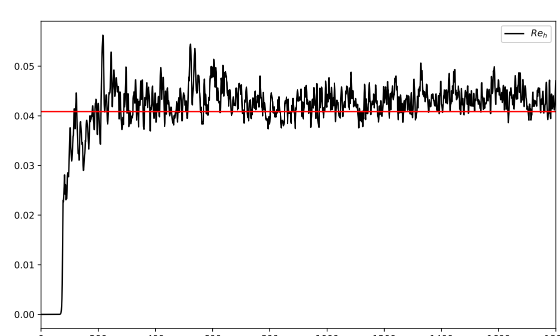
(a) Space averaged mixer temperature



(b) rms of the temperature with the mixer



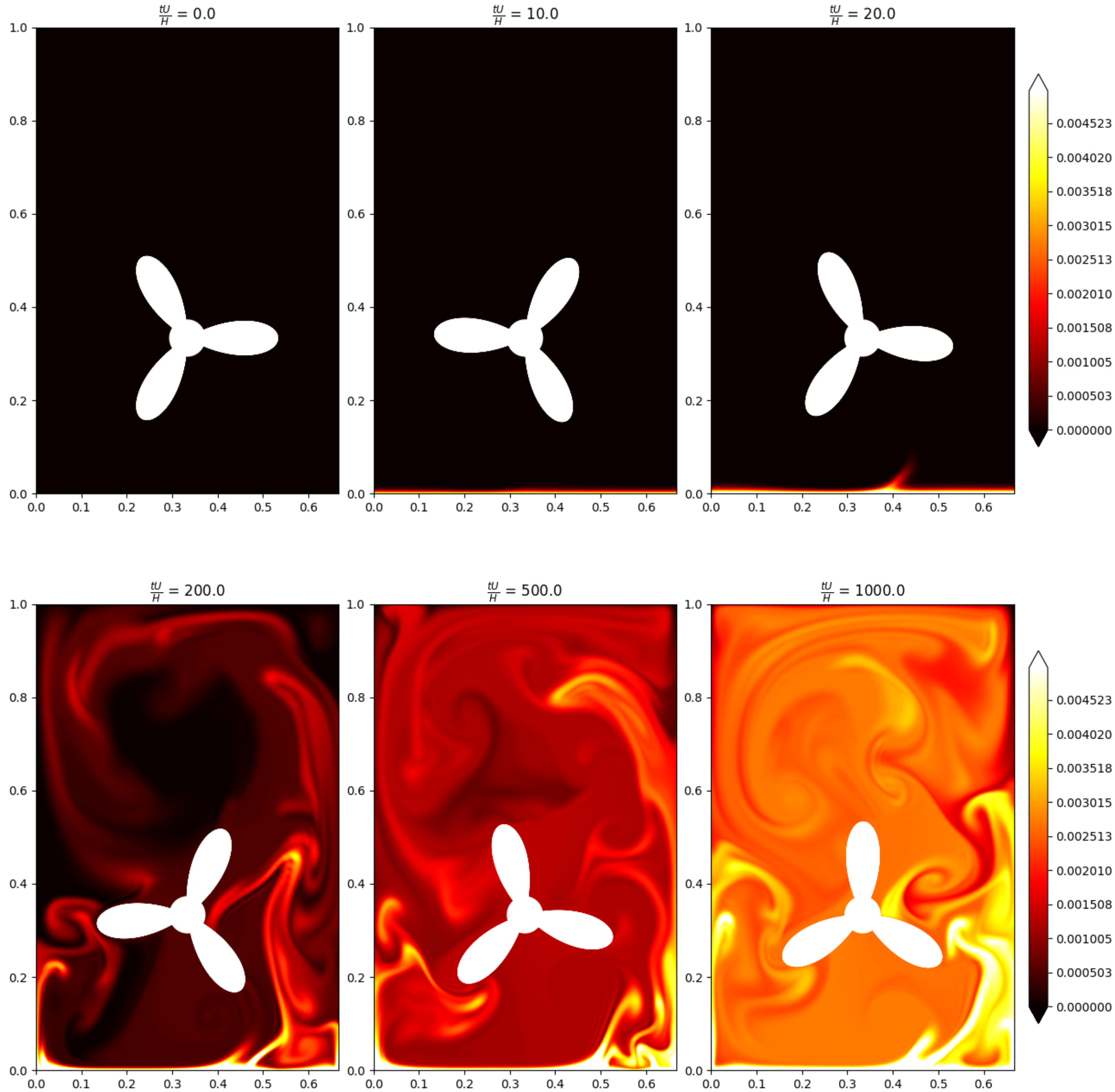
(a) $Re_{\omega,h}$ without mixer

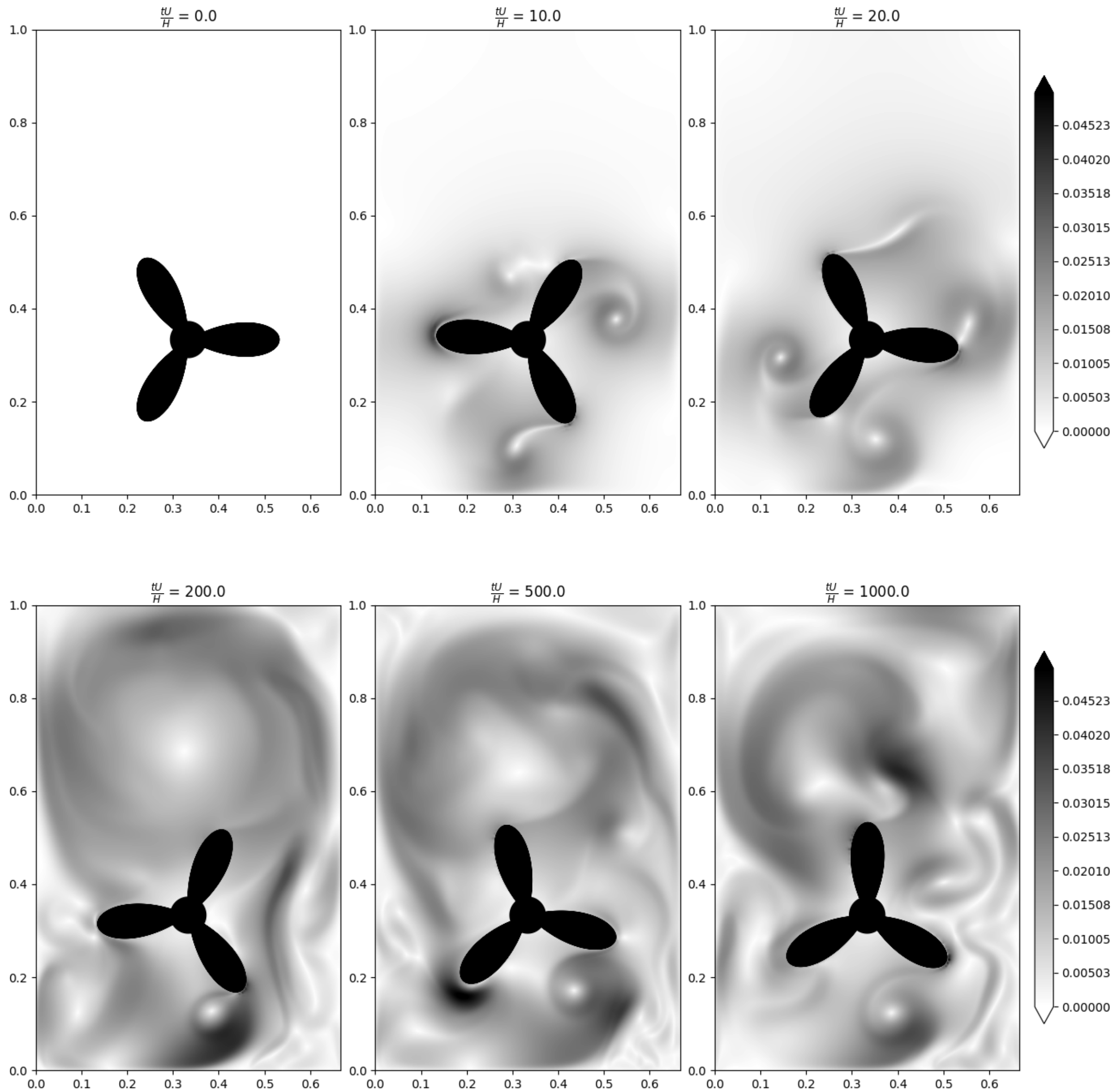


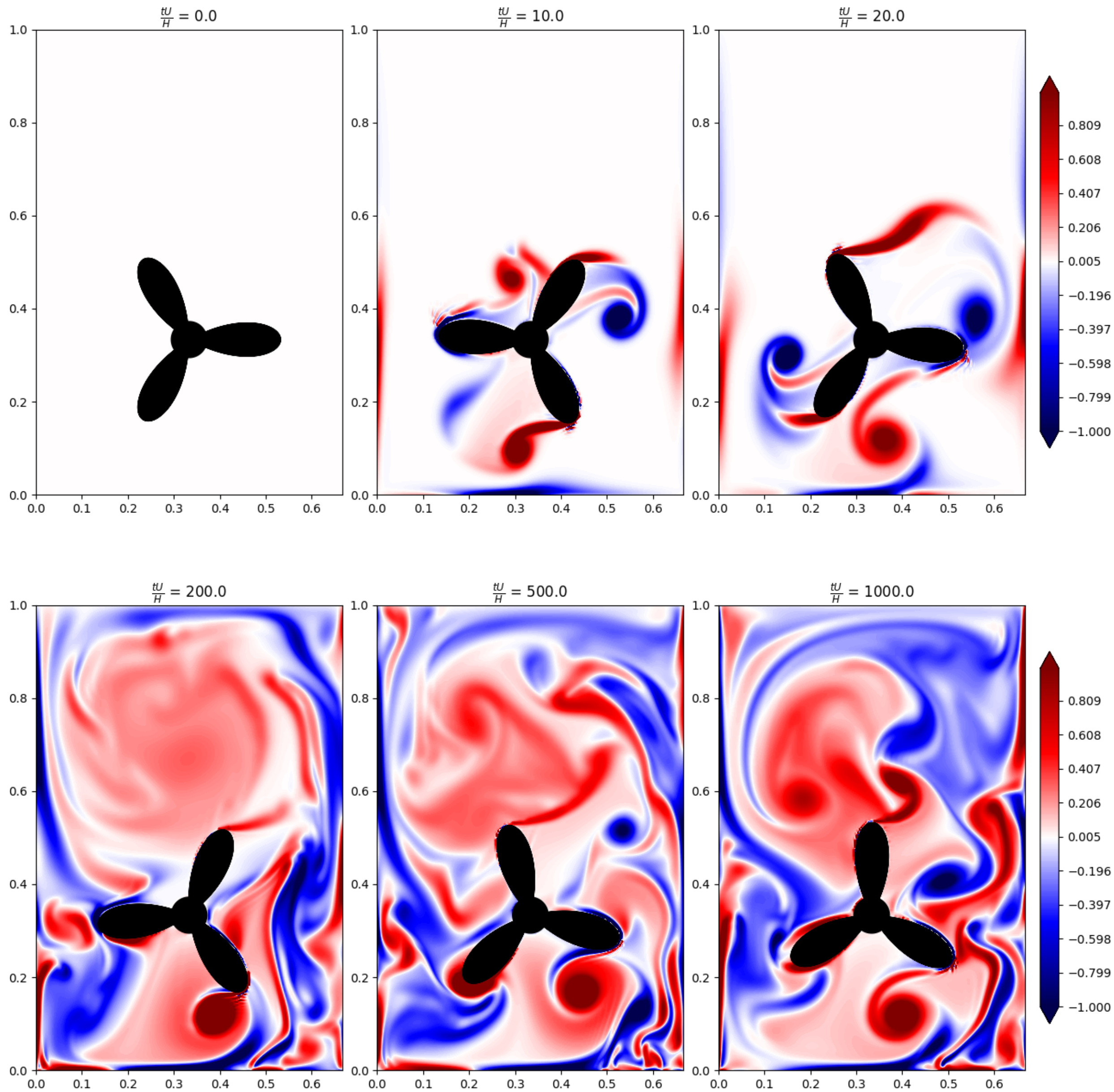
(b) Re_h without mixer

3.2 Part II : Mixer

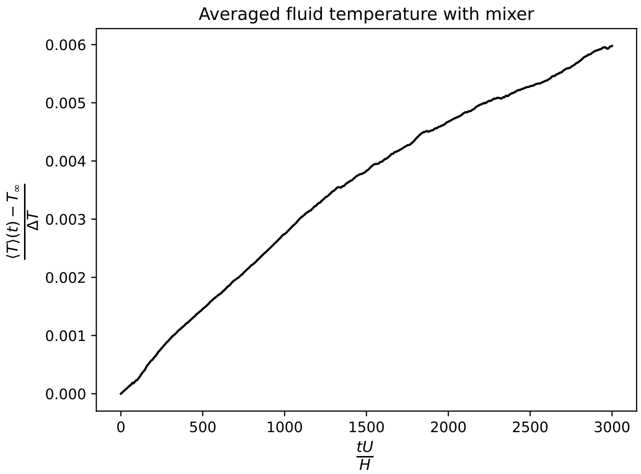
Fields results



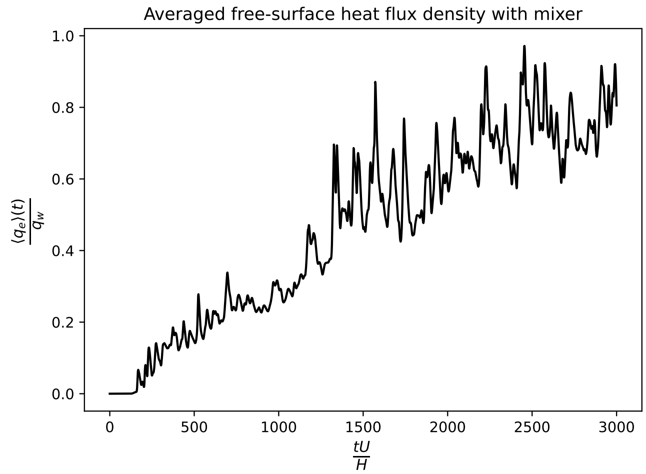

$\frac{tU}{H}$ = 0.0
$\frac{tU}{H}$ = 10.0
$\frac{tU}{H}$ = 20.0
$\frac{tU}{H}$ = 200.0
$\frac{tU}{H}$ = 500.0
$\frac{tU}{H}$ = 1000.0
0.0
0.1
0.2
0.3
0.4
0.5
0.6
0.8
1.0
0.04523
0.04020
0.03518
0.03015
0.02513
0.02010
0.01508
0.01005
0.00503
0.00000


$\frac{tU}{H}$ = 0.0
$\frac{tU}{H}$ = 10.0
$\frac{tU}{H}$ = 20.0
$\frac{tU}{H}$ = 200.0
$\frac{tU}{H}$ = 500.0
$\frac{tU}{H}$ = 1000.0
0.809
0.608
0.407
0.206
0.005
−0.196
−0.397
−0.598
−0.799
−1.000

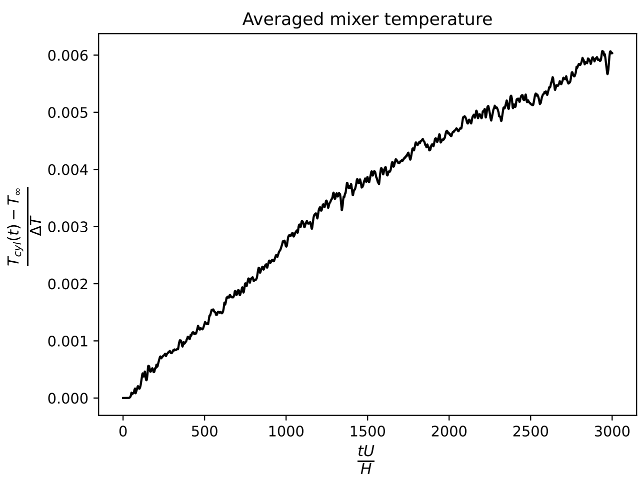
Diagnostics



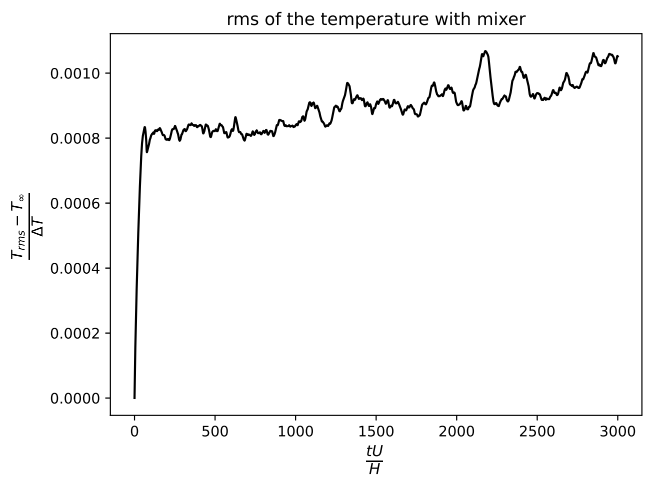
(a) Averaged fluid temperature with the mixer



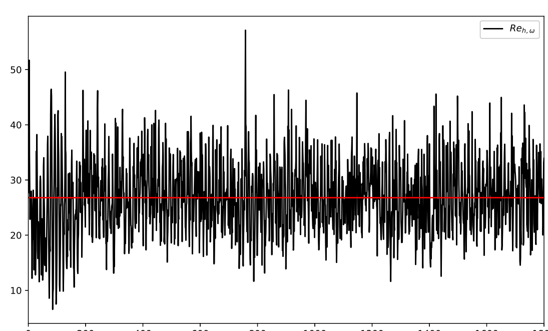
(b) Averaged free-surface heat flux density with the mixer



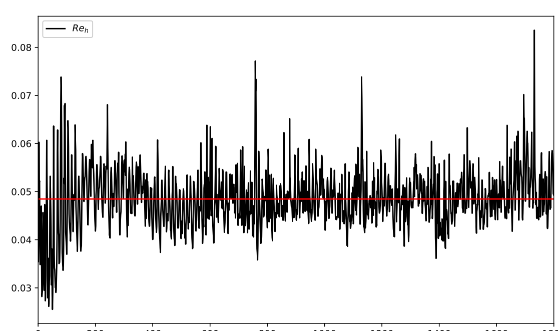
(a) Space averaged mixer temperature



(b) rms of the temperature with the mixer



(a) $Re_{\omega,h}$ with mixer



(b) Re_h with mixer

4 Conclusion

5 Bibliography

- https://en.wikipedia.org/wiki/Projectionmethod(fluid dynamics)
- https://www.cfd-online.com/Wiki/Staggeredgrid