River hydrodynamics - steady-state flows

Numerical Simulation

András Mátyás Biricz

Physics MSc 1. semester

December 12, 2017

Computer Simulations in Physics

Eötvös Loránd University

Contents

1 Introduction

Investigating the behaviour of the fluid flows is a topic that is close to our everyday life, because when we drink tap water, go swimming or when we just look at the weather forecast, we meet with this problem. We could ask, what is a fluid exactly, how could we formulate a theory to describe it and if we could, how would the governing equations look like? Will they be easy to solve and how realistic the results would be?

This works is aimed at to give an insight to the fluid motion through not only by explaining the theoretical background, but also showing few examples and their analitical or numerical treatment. Since the investigated problems are of easier ones, there will be a section for exploration with even more interesting thoughts.

2 Theoretical background

To get started into this huge topic it is a good start to consider the fluid motion phenomenologically. A fluid is actually a state of the matter, where the particles are not strongly bounded, but they also can not be separated far from each other. Even they are able to roll on one another. To give answers to these and other phenomena seen in our every day life, we are about to investigate flows with the help of physical quatities, like velocity, pressure, density and viscosity, but not only these. There are also some, which do not have exact physical meaning. Either they are motivated by mathematical approach to find better solutions or just to give empirical estimation, such as the Reynolds number, which describes the type of the flow (laminar or turbulent). Let's find out more about these during the following sections.

2.1 Navier Stokes equation

Physically the motion happens in both space and time, so we should consider giving physical variables with a four dimensional mathematical description. Since we are talking about fluid dynamics, we can write the Newton's equation ($F = ma$) as a relation between the material or convective derivative of the velocity field and the forces acting on the fluid particles. This is the most important thing to discuss and analyze, because it is responsible for the strange behaviours seen:

$$\frac{D\mathbf{v}(\mathbf{r},t)}{Dt} = [\mathbf{v}(\mathbf{r},t) \cdot \nabla]\, \mathbf{v}(\mathbf{r},t) + \frac{\partial \mathbf{v}(\mathbf{r},t)}{\partial t}, \tag{1}$$

where ∇ is the differencial operator for spatial derivatives, $\mathbf{v}(\mathbf{r},t)$ is the velocity field given in a coordinate system determined by $\mathbf{r}$. Here we should notice, that this term in non-linear!

To continue, we should collect all force terms to the right side, including the influence of pressure, viscosity, gravity and other body forces, and if present, friction due to the boundary effects on flows. In our case, the external body forces including gravity, the friction and other surface effects of the fluid will be neglected. For the governing equation we get:

$$\rho(\mathbf{r},t)\frac{D\mathbf{v}(\mathbf{r},t)}{Dt} = -\nabla P(\rho, T, \mathbf{r}) + \mu \nabla^2 \mathbf{v}(\mathbf{r},t), \tag{2}$$

where ρ is the fluid density, P is the pressure, that also depends on the temperature T, and μ is the dynamic viscosity for describing how strong each fluid particle is "kept" by one another, so how fast they can roll on each other.

To go further, we should consider, that if we imagine a cube inside the flow, the density of the fluid inside can only be changed if there is in- or outflow, so we get a conservation law for the mass. This equation is called the continuity equation and has the following form:

$$\frac{\partial \rho(\mathbf{r},t)}{\partial t} + \nabla \left[\rho \mathbf{v}(\mathbf{r},t)\right] = 0. \tag{3}$$

After we obtained these two important equations, we could jump into a problem in fluid dynamics and solve it. But not that quick, because it can be noticed, that the pressure needs also an equation, which is called, the state equation. If we are talking about gases, we can pick the state equation for the ideal or even for the Van der Waals gas. Not so easy is it for the flows and even if we find out a relation, if it has temperature dependence, that should be also given somehow. As a conclusion to this complex problem according to the pressure, we will proceed with a very simple solution, so let P undependent from the density and temperature.

As there is a phenomena we can observe, that fluids tend to be incompressible versus gases, the following simplification occurs: $\rho(\mathbf{r},t) = const$, because $\nabla \mathbf{v}(\mathbf{r},t) = 0$.

Before going on to the next section, the meaning of one empirical parameter should be clarified, that is, the Reynolds number:

$$R = \frac{UL}{\nu}. \tag{4}$$

where U is the velocity, L is the lenght scale of the flow, while ν is the kinematic viscosity (that is the dynamic viscosity divided by the density). If the viscosity is getting larger, the Reynolds number will decrease and vica versa. Only from this we are able to consider, that if this number is small, we expect a laminar flow, but if large, vortices may develop and change the type of the fluid motion.

2.2 Stream function and vorticity

In the previous section, we talked about the incompressible case of flows, when $\nabla \mathbf{v}(\mathbf{r},t)$ equals zero. This lets us find a different mathematical approach, because it is known, that for any vector field:

$$\nabla \left[\nabla \times \mathbf{u}(\mathbf{r},t)\right] = 0. \tag{5}$$

As a result we can introduce the stream function (or a vector potential for the velocity):

$$\mathbf{v}(\mathbf{r}, t) = \nabla \times \mathbf{u}(\mathbf{r}, t), \tag{6}$$

where $\mathbf{u}(\mathbf{r}, t)$ is the stream function, that is also a vector. There is another variable closely related to $\mathbf{u}(\mathbf{r}, t)$, which is given by the following equation:

$$\omega(\mathbf{r}, t) = \nabla \times \mathbf{v}(\mathbf{r}, t), \tag{7}$$

where $\omega(\mathbf{r}, t)$ is a vector quantity describing the local angular velocity of the fluid, called vorticity. After some mathematical steps the relation can be revealed:

$$\nabla^2 \mathbf{u}(\mathbf{r}, t) = \omega(\mathbf{r}, t), \tag{8}$$

which is analogous to the Poisson's equation of electrostatics.

As a concluding note it should be mentioned, that these formulations satisfies the continuity equation by definition.

3 Problems and their solutions

The motivation behind this work is to get involved in computational fluid dynamics; to "try out" the theoretical models through the solution of few exercises while practicing the programming. It is not even a trivial problem to solve a partial differential equation numerically, because lots of challanges awaits along the way until satisfying results are obtained.

I chose to investigate two simpler problems. Both are related to the steady-state flows formulated in rivers due to placed obstacles: the first when two thin parallel plates are given horizontally, the second when a rectangular shaped beam is floating in between the bottom and the surface.

Since only steady-state solutions are looked for, an iteration algorithm is chosen, which starts from an initial guessed field and converges to the solution. To achieve this, appropriate initial velocity and pressure or vorticity and stream function should be given and also the boundary conditions should be applied carefully.

3.1 Analitical solution for plates

Since this problem, when two thin parallel plates are placed horizontally in a flow - where only the horizontal component of the velocity is non-zero - can be solved analitically, we will do it. The boundary conditions are: $v_y \equiv 0$ and $v_x(0) = v_x(H) = 0$, where H is the distance between the plates. The equations to solve are the continuity and momentum equations, the latter for x and y components:

$$\frac{\partial v_x}{\partial x} = 0, \tag{9}$$

$$\frac{\partial P}{\partial x} = \rho\nu \frac{\partial^2 v_x}{\partial y^2}, \tag{10}$$

$$\frac{\partial P}{\partial y} = 0. \tag{11}$$

The equation (10) can be solved with separating both sides, since the RHS[1] describes a derivative according to x, while the LHS[2] to y. Therefore both sides should equal to a constant

[1] Right-hand side,
[2] left-hand side.

(C). After integrating the equation we got from the LHS and applied the boundary conditions, the following result is obtained:

$$\rho\nu v_x(y) = \frac{1}{2}\frac{\partial P}{\partial x}(y^2 - yH), \tag{12}$$

which means, $v_x(y)$ has a parabolic profile in y, so between the two plates.

After further calculations written in [1], the pressure variation can be also determined:

$$\frac{\partial P}{\partial x} = -12\frac{\rho\nu V_0}{H^2}. \tag{13}$$

After all of this work, we got the reference values we are expecting from the first simulation.

3.2 Successive overrelaxation method

This method is used to reach the steady-state values of the variables describing the flowfield, when those are initiated with a guess at the start of the iteration process. At each step, the old variable values will be updated according to the governing equations and the boundary conditions.

In this work a second-order accurate finite-difference scheme is applied to carry out the SOR[3] algorithm. The equation to use will look like the following:

$$v_{i,j} = v_{i,j} + \omega \cdot r_{i,j}, \quad r_{i,j} = v_{i,j}^{(new)} - v_{i,j}^{(old)}, \tag{14}$$

where $\{i, j\}$ are indices running along the numerical grid in $\{x, y\}$ directions, and $r_{i,j}$ is the residual, that updates the value. The control parameter for the convergence is ω: at the condition $\omega < 1$ underrelaxation, while at $\omega > 1$ overrelaxation occurs.

3.2.1 Thin plates

From the equations (9)-(11) using the finite-difference scheme, the following equations can be derived:

$$v_{i,j}^x = v_{i,j}^x + \omega \cdot r_{i,j}, \tag{15}$$

[3] Successive overrelaxation method.

$$r = \frac{1}{4}\left\{v^x_{i+1,j} + v^x_{i-1,j} + v^x_{i,j+1} + v^x_{i,j-1} - \frac{h}{2}v^x_{i,j}\left[v^x_{i+1,j} - v^x_{i-1,j}\right] - \frac{h}{2}\left[P_{i+1,j} - P_{i-1,j}\right]\right\} - v^x_{i,j}, \quad (16)$$

where h is the spatial step size in the simulation.

Related back to the previous section, where the theory of the fluid motion was introduced briefly, the pressure field should be also given, not only the velocity. Because of this problem, that the pressure is not determined, and even the book [1] is not saying a thing at all about its value, this problem cannot be carried out numerically.

Few attempts were made to find a workaround to this, but unfortunately it did not provide useful data. A different numerical scheme, with different code was also written (can be found in the appendix files). For more information about it, please read section 3.3.

3.2.2 Beam as obstacle

In order to solve the problem of the rectangular shaped beam placed in the flowfield, I used the stream function and the vorticity as variables. With the help of this formulation, the continuity equation is satisfied by definition, but the greatest advantage lies behind that the pressure field is eliminated. I iterated the two variables (for SOR equations see [1]) and found out the results, that I will present in Section 4.

The most important aspect of getting the satisfying results were to apply the boundary conditions properly. The beamsides: the front, top and back and the outer boundary of the computational domain. Since the problem is symmetric along the centerline of the rectangular shaped beam, it is enough to analyze only the upper half of it.

I determined the boundary conditions for the velocity and then I transformed it to give information for the stream function. For the calculation of the vorticity, I considered its meaning: it is zero everywhere at my initial guess, because I assumed, that at the beginning the flow is laminar.

Inside the domain of the beam, the stream function and the vorticity stays zero. The case on the beam's surface is the same, but with different consideration, u should be zero, since no-slip boundary conditions are applied, that means, both components of the velocity should vanish. The trick comes, when the vorticity around the beam's surface is calculated: a Taylor series is used to get the result of u at a numerical step h away from the surface. With the relation between the stream function and the vorticity, an equation for the vorticity at the boundaries can be obtained.

It is also important to mention, that this problem is only in two dimensions, where the stream function and the vorticity will have only z component, so it becomes a pseudo-scalar.

For more information about the description of this problem, please see [1].

3.3 Explicit MaxCormack Scheme

After the failure to apply the SOR algorithm for the problem of the parallel plates, I tried to use this numerical scheme to gain satisfying results.

The motivation to use this algorithm and not the other one is because it solves the compressible Navier-Stokes equation with mass conservation, which determines exact relation between the density and the pressure. So I could get an equation for the pressure field.

Not only the pressure field is a modification compared to the SOR algoritm, this one is a time dependent scheme, so it also gives extra information about the time development of the variables. If a steady-state flow is about to formulate (which we look for), then it should be able to develop after sufficient iterations in time.

I read the related chapter in [2][4] and tried to write my own code, (which I handle in also), but unfortunately after more than a day full of work on it, I could not develop a stable simulation.

[4] Read chapter 11 for more information.

4 Numerical analysis

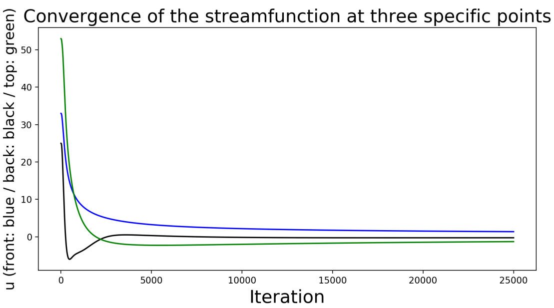
After getting to know the theoretical background and the application of the simulation, it is time to analyze the output data. In this section the problem of the rectangular shaped beam will be presented.

4.1 Convergence of the algorithm

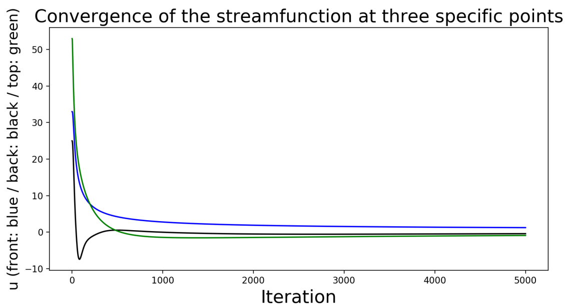
To investigate the convergence of the algorithm it is worth to check the "three-place-convergence", that means, to plot the changes in the stream function u at three different places of the computational domain after each iteration step. These places will be: near the front, top and back of the beam's surface. We know, that ω is the control parameter for the convergence (see equation:(14)), so if it is smaller, we obtain slower convergence, but more stable simulation.

To check only the convergence of the algorithm, all parameters except ω should be fixed. Analyzing the following figures, we can conclude, that the convergence is obtained at every values of ω, where the simulation stays stable, but the only difference is, that we need to iterate for less or more steps. We should also consider, that it is fine, if we get only approximately the same at the end of the iteration steps, because we applied only a second order finite difference scheme. After thousands of steps, the numerical errors will begin to rise.

The two limits at varying ω were 0.05 and 0.29. Figure for these:



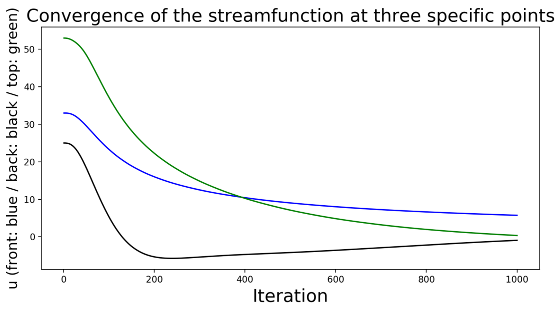
(a)



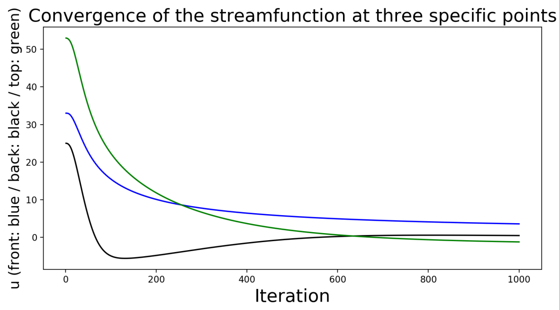
(b)

Figure 1: Three-place convergence plot for $\omega = 0.05$ over 25000 iteration steps (a) and for $\omega = 0.29$ over 5000 iteration steps (b).

Checking the convergence for other ω values:



(a)



(b)

Figure 2: Three-place convergence plot over 1000 iteration steps for $\omega = 0.1$ (a) and for $\omega = 0.2$ (b).

To sum up, at least a thousand of iteration steps is required, when $\omega = 0.2$. If we change that to one-half of that value, twice as many iteration steps would be needed.

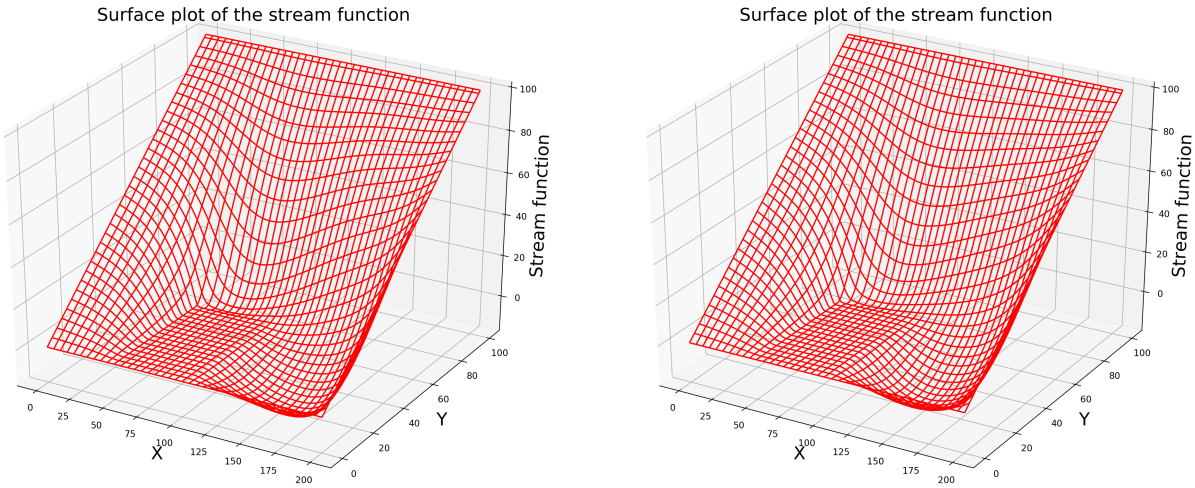
4.2 Properties of the flow

After the initial guessed field has converged to the solution, we can investigate the properties of the developed steady-state flow. To do so, the surface plots of the stream function and vorticity will help to analyze the velocity field.

Let's use the results for the plots obtained after the convergence is reached, that was showed in figure 1. It can be seen, that the two subplots of the figure 3 and 4 are showing almost the same (within numerical accuracy), that means, the convergence is reached at both cases.

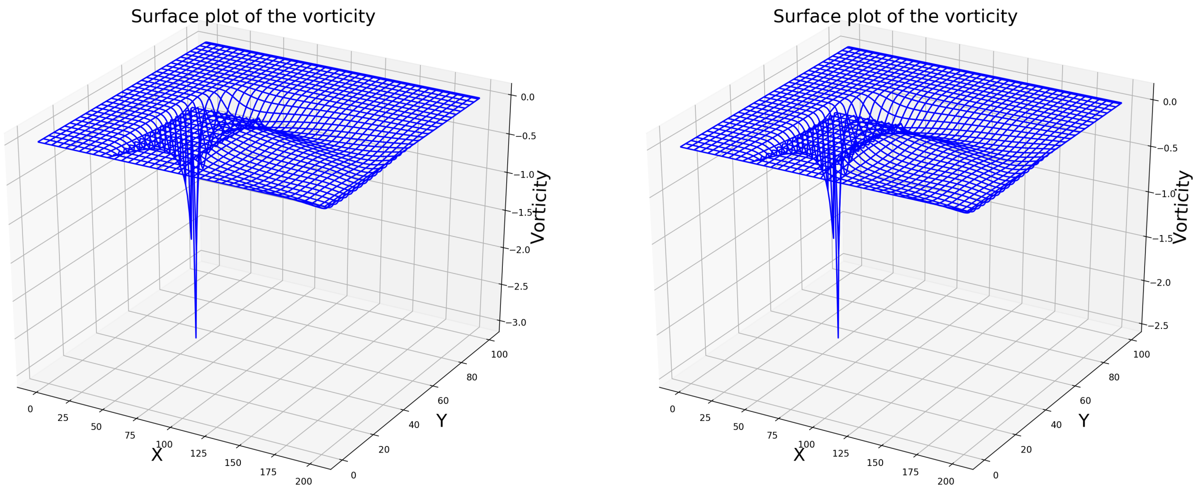
To give explanation to the surface plots, the equations (6) and (7) should be used in two dimensions. Considering the case of the stream function, it is growing linearly in y direction, if the flow has only a uniform horizontal velocity component. The small and the big "smooth valley" top and behind the beam's surfaces shows the disturbance in the flowfield, that it might rotate, because both v_x and v_y has nonzero values. To investigate the rotational motion, the surface plot of the vorticity will be used. As expected, its values are "jumping" or smoothly changing at the places of those "valleys". Bigger changes means smaller vortices and vica versa.

After all of these, finally let's have a look at the velocity field, which is just showing, what we expected (figure 5). Two vortices are developed due to the presence of the beam.



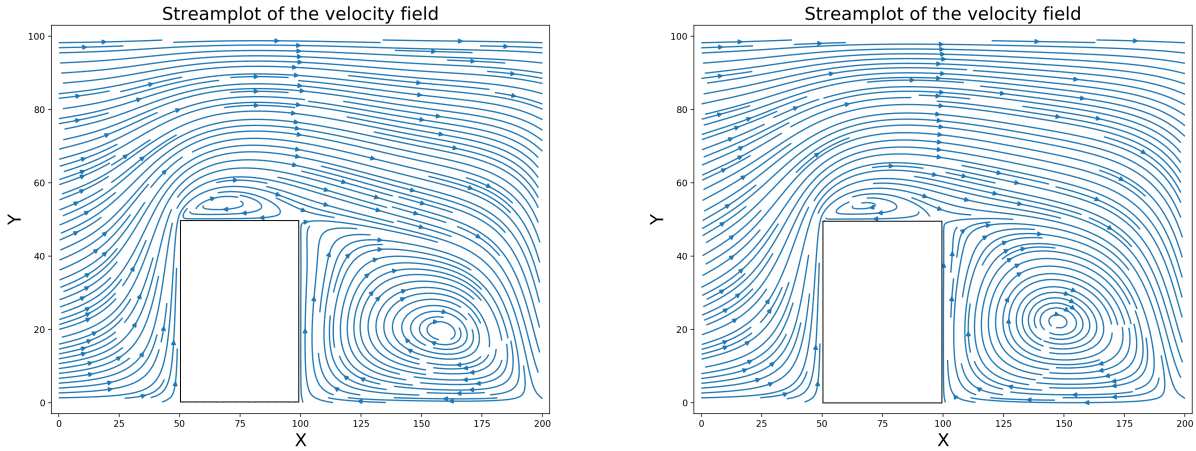
(a) (b)

Figure 3: The plot of the stream function for $\omega = 0.05$ after 25000 iteration steps (a) and for $\omega = 0.29$ after 5000 iteration steps (b).



(a) (b)

Figure 4: The plot of the vorticity for $\omega = 0.05$ after 25000 iteration steps (a) and for $\omega = 0.29$ after 5000 iteration steps (b).



(a)

(b)

Figure 5: The plot of the velocity field for $\omega = 0.05$ after 25000 iteration steps (a) and for $\omega = 0.29$ after 5000 iteration steps (b).

To give answer, where a big fish could rest, let's think of the following: where is the place, where the velocity of the flow is almost zero? As seen on figure 5, it is at the center of the vortex developed behind the beam.

4.3 Flows at different Reynolds number

Now let's analyze an other phenomena, the type of the flow. It was said, that the Reynold number is an empirical parameter for this. Let's investigate, if it is true, what we wrote at the theoretical background.

According to figure 6, it is confirmed, that the theoretical approach gives satisfying explanation to the type of the flowfield. The one thing that should be considered is the following: when is the Reynolds number "big" or "small". Within numerical stability numbers between zero and four could be set. At $Re = 1$ (seen in figure 5) the small vortex at the top is formed, so the change between laminar and turbulent flow type should be around $Re = 0.5$, because at $Re = 0.2$ in figure 6 we are not able to see it. Finally, it is interesting to note, that at $Re = 4$ many more vortices forms.

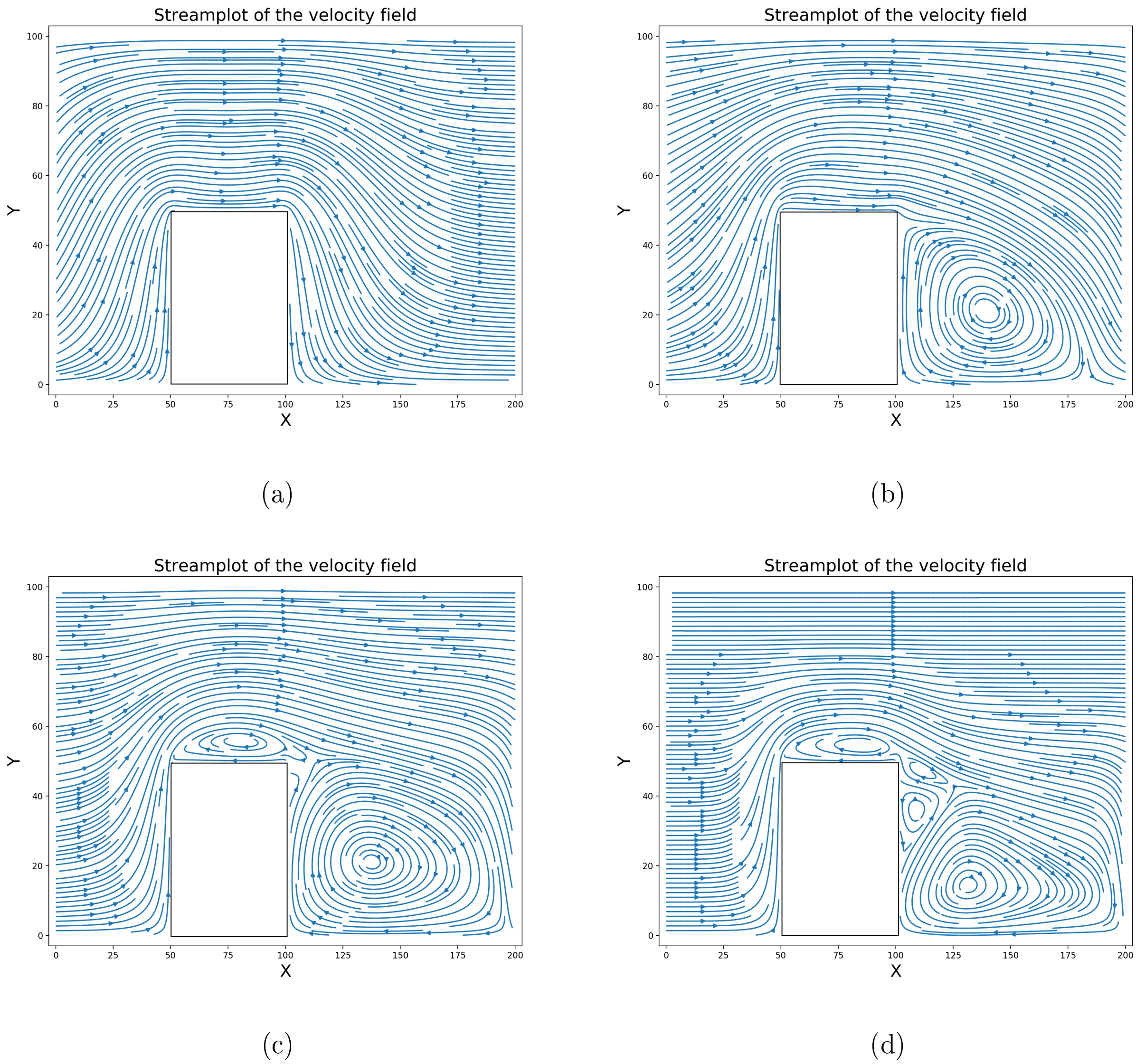


Figure 6: The plot of the velocity field after convergence is reached for $Re = 0.001$ (a), for $Re = 0.2$ (b), for $Re = 2$ (c) and for $Re = 4$.

5 Exploration

As said in the introduction, many more interesting problems awaits if we would like to investigate the fluid motion. If the problem has different geometry, like a circular rock is placed in the flowfield, then instead of cartesian coordinates, polar coordinates should be used, but the same conditions could be applied as in this work.

It would be also interesting to see what influence the second order finite difference scheme has on the results. Would it be better for a fourth order accurate scheme? Would it converge faster and would it be more stable numerically?

To go even further, finite element method could also be used, which is a much harder algorithm to implement, but the results would be equal if we would use infinite order accurate finite difference scheme.

6 Conclusion and discussion

The numerical simulations with the post-processing are done in python and the results are shared with this work. For the problem of the rectangular shaped beam I could get satisfying results, but for the thin plates unfortunately not. The cause of this problem (lack of expression for the pressure field) was described at section 3 in details. I even asked for consultation with prof. István Csabai and he told me, if I could not go ahead of solving it, I can leave it out and choose something else to investigate. I tried to use the MacCormack method of solving the PDE, but that did not work after long working hours and lot of debugging. To overcome this issue, finite element method could be applied, but that goes beyond the aim of this work.

References

[1] Rubin H Landau, Manuel J Paez, Cristian Bordeianu: A Survey of Computational Physics - Introductory Computational Science, Princeton University Press, 2008.

[2] Pijush K. Kundu, Ira M. Cohen, Fluid Mechanics, Third Edition, Elsevier Academic Press, 2004, (Chapter 11: Computational Fluid Dynamics by Howard H. Hu)

[3] T. J. Chung, Computational Fluid Dynamics, Cambridge University Press, 2002