Universidad Carlos III de Madrid

Fluids Mechanics Department

NUMERICAL ANALYSIS OF THE INFLUENCE OF GEOMETRY ON STEADY STREAMING FLOWS

Bachelor Thesis

Aerospace Engineering

Author: David Cerisola Guardiola
Tutor: Wilfried Coenen

September 2015

Acknowledgements

I would like to express my sincere gratitude to my tutor, Wilfried Coenen for guiding and supporting me through the development of this bachelor thesis. I would like to highlight the confidence he has placed in me and my possibilities, allowing me to work in a comfort and motivated way.

Also, I would like thank all people, my family, my professors and specially my friends and classmates, that have helped me to get where I am today, finishing my bachelor thesis and closing a wonderful stage of my life, to become a junior engineer.

Abstract

The current project consists on the design of a numerical model to solve the steady streaming that is produced when an oscillating flow faces solid cylinders. This model was created by using finite element modelling techniques through the use of Freefem++ software. The development of the code to solve the flow involves the implementation of the Navier-Stokes equations and the creation of a mesh, which define the geometry of the problem. Three different geometries are presented. The first one consisting on one single cylinder in the middle of the microdevice, the second geometry consists of two cylinders positioned side by side separating by a distance g_a, and the last one consist of four cylinders, two of them positioned in the x axis and the other two positioned in the y axis. In this last geometry some parameters, that define the geometry, are modified, to see how is affected the resultant flow.

Table of contents

Table of figures

Chapter 2

Chapter 3

Chapter 4

Chapter 1

Introduction

1.1 Background

The ability to trap and confine single particles for their later analysis is increasing in importance in the last decade for applied science; it is a very important process in many microfluidic lab-on-a-chip devices. There are several methods for particle trapping; the most important ones are based on: magnetic, optical, electrokinetic, acoustic and hydrodynamic fluid flow [1]. Thanks to these methods and its functionalities, major advances in physics and biology in a molecular to cellular level have been possible.

This project focuses in the hydrodynamic fluid flow method, in which trapping a particle is achieved through the use only of hydrodynamic forces. The main objective of this work is to analyze the steady streaming flow that is produced in these mentioned before methods to trap particles, and how different configurations may affect to the streamlines or to the microeddies produced.

Current hydrodynamic methods can be split in two categories: contact-based or non-contact-based methods. The firs type, contact-based, use fluid flow to immobilize a particle against a wall, whereas the other method, non-contact-based, use fluid flow to create microeddies, thanks to this vortical flow, particles are confined at the center of the microeddie, where there is a stagnation point, (point in the flow where there is no velocity).

Results of magnetic, optical, electrokinetic and acoustic methods are acceptable, but they are very expensive and they need very advanced equipment [2]. Methods to achieve particle trapping using purely hydrodynamic forces are much cheaper that those, moreover they have comparable trapping capabilities with methods in which there are special and very expensive devices. There is another advantage of using hydrodynamic methods, it is that the immobilization of the particle is achieved without the need for potentially perturbation of electric and magnetic fields [3].

The proposed numerical model done in this project to solve the flow is able to predict without the need to perform an experimental test, where are the possible locations to capture micro particles just by identifying small vortical flow regions, with enough trapping force to maintain them at a fixed position for their posterior analysis. This previous mentioned vortical flow is created as a result of a fast small-amplitude oscillating flow around geometric forms. This project has been specially done to solve the flow in the regime of high streaming Reynolds number. The reason to work in this regime is the relation between the streaming Reynolds number and the trapping force, since high Reynolds implies the trapping forces and the velocities to increase one order of magnitude [4].

Trapping forces directly depend on the streaming velocity in the microeddies. These vortices can be generated by passing fluid through sudden expansions or contractions, cavities or protrusions [4]. The main disadvantage of this method of creating microeddies is that the trapping capacity is directly coupled to the flow rate going through the channel, what could compromise the controllability of the flow if there is need a big trapping force. For the code developed, this is not an inconvenient since the analysis of the flow is numerical instead of experimental, but this is neither the only option nor the best option.

Therefore in this project another option to create those necessary vortices is studied. This option is taking advantage of the steady streaming that is developed when an oscillating flow face solid objects, what also crates vortical flow, and it is much easier to control the parameters that govern the flow. These microeddies strength does not directly depend on the flow rate passing through the channel, but on the streaming Reynolds and on the variations in the frequency and amplitude of the oscillation, which in fact, are easier to control than the flow rate, if it has to be very high.

This is a project of investigation since there have not been previous authors to guide our steps. It is true that we are not pioneers in working with hydrodynamic fluid trapping methods, and there are lots of researches done about steady streaming flow about cylinders and how particles are trapped with these types of methods. What is really innovative in this project is working in the regime where the streaming Reynolds number has a high value, as mentioned before.

The main problem is that this regime of R_s is still unexplored in the literature on particle trapping applications. One of the reasons why no authors have written about this hydrodynamic tweezers method in high streaming Reynolds number regime could be the huge difficulty at the time to control the flow, when the flow is govern by a high streaming Reynolds, just a very small variation in the required value for a control, could rapidly lead to turbulent flow or undesired flow topologies [4].

Therefore, to control properly the flow it is needed a high level of accuracy, since there is a thin line separating an inoperative device from a perfectly operative trapping device.

Because it is an innovative method of trapping particles there is no a regulatory framework where all the restrictions, rules or techniques that are applicable to these methods appear and are specified. Due to this fact reasonable values for the parameters have been taken, without any type of restriction.

Some of the possible uses for trapping particles could be from the diagnosis of diseases, DNA sequencing, to the detection of contaminants in drinking water, among others.

1.2 Objectives

The main goal of this project is to develop a numerical model for an oscillating flow around different solid surfaces, and being able to compute the solution of this flow.
As a second objective there is the need to create microvortices, modifying the geometry of the problem, to be able to trap particles in those vorticial flows.

To achieve these ends it was necessary to create:

- Numerical tools to model the streaming flow, to model the streaming Reynolds number (R_s) and the non-dimensional amplitude (ε), all this in combination with a superposed oscillating flow.
- Different geometries, to prove the numerical model, in which the flow is solved.

To reach those goal, it was necessary to understand:

- How microeddies are created around these solid surfaces.
- The steady streaming motion produced by the oscillating flow around these solid surfaces.
- How different configurations of the position of solid surfaces may affect to the flow and the microeddies.
- How FreeFem++, the software used to simulate the flow, works.

1.3 Planning

The project was divided from the very beginning into different phases, each of them well defined and different from the others; this was done for a better understanding and a better track of the evolution of the project.
These phases are explained below:

- The first phase to complete this project was the collection of information, necessary to provide the require background of trapping particle methods, steady streaming flows, potential flow…

- Once the first phase was complete, an exhaustive study on FreeFem++ was done from the basis, since this Finite Element Model and the language that this program uses, C++, was completely unknown for the author.

- The third step was to start with the mesh in FreeFem++ of the different geometries to be studied.

- After this, it was the turn to start working on the code, understanding the Navier-Stokes equations, how they could be implemented, the numerical iterative Newton Raphson method to solve the equation, how to define the mathematical problem in Freefem++…

- The fifth phase was the understanding of results, in order to provide with them a better mesh adaptation.

- Once the mesh adaptation and the code run properly, it was implemented in all the different geometries to solve the flow around them.

- Finally, the last phase of the project was the understanding of the results.

- Write the current memory.

Phases of the project	Hours
Bibliographic research	30
Learning how to use FreeFem++	45
Modeling different geometries	25
Code to simulate the oscillatory flow	100
Mesh adaptation and implementation	30
Report writing	80
Meetings	30
Total	340

Table 1.1 Hours dedicated to the project

1.4 Budget

Following the planning of the previous section, a fictitious budget is done to simulate this numerical analysis of the oscillating flow due to the oscillations of cylinders.

This is an estimation of what would have cost to carry out this project in real life, based on a fictitious salary of an engineer and on an estimation of the time consumed in all the phases excluding the time used for writing the current memory.

The proposed salary is 30 Euros per hour of work, the license to use FreeFem++ is at its own name points, free, it is assumed that the company in charge of perform this project has already the computers with the required power to perform the job, so no additional expenses are taking into account.

$$260\ h\ of\ engineering\ work \cdot 30\frac{€}{h} = 7800\ €$$

The virtual budget to carry out this project would be of 7800 €.

1.5 Outline

This project is divided in 5 chapters through which the evolution of the project and its phases of development are shown.

These 5 chapters are:

1) **Introduction**: a global view of the problem to be solve with this work is presented, why it is needed a method to trap particles, what are the methods commonly used, why in this project is chosen an hydrodynamic method… also in this first chapter are presented the objectives of the projects, the planning carried out to perform the work and the virtual budget that would have cost this project in the real life.

2) **Theory**: in this second chapter, the description of the problem to be solved is presented. In addition it is also given here a theoretical background in steady streaming flows and potential flow.

3) **Numerical method**: here the mathematical tools to solve the problem are explained. Some of these mathematical tools are: Newton-Raphson iterative method, what is Freefem++, how it is used and how equations are implemented in Freefem++…

4) **Models of study and results:** in this fourth chapter a detailed view of the proposed geometries is exposed, it is shown how they are built, and how the meshes are refined in the critical areas to produce better and more accurate results. In addition here the results of solving the oscillatory flow around the meshes are presented with some discussions of how the different positions and configurations affect the resultant flow. Here in the fourth chapter it is also explained the validation of the model.

5) **Conclusion and future projects**: the main conclusions extracted from all the work in the project are shown in this section, as well as a briefly explanation of how the author thinks that this project may continue or be improved in a possible future work.

6) **Bibliography:** Here in this last chapter the references used to understand and complete all the phases of the project are listed.

Chapter 2

Theory

In the second chapter it is given a short background as well as it is detailed the problem definition.

2.1 Steady streaming

Steady streaming is defined as the time-average of a fluctuating flow often results in a nonzero mean. Such steady streaming can be produced by three different causes [5]:

1. If there exists an oscillatory non conservative body force.
2. If the force is conservative, indirectly through the action of Reynolds stresses in the body in contact with the fluid.
3. If the force is conservative, the action of Reynolds stresses may act also in a thin boundary layer at no slip boundaries.

These are the three ways of creating a steady streaming flow in homogeneous fluids. The concept of steady streaming was first discussed by Riley in 1967, he treated with flows around solid bodies performing periodic translational oscillations in the flow.

The **Reynolds stress** is defined as "the net rate of transfer of momentum across a surface in a fluid resulting from fluctuations in the fluid" [6].

In other words, the Reynolds stress is obtained from applying Navier-Stokes equations to take into account the fluctuations in fluid momentum, and the Reynolds stress is rate of momentum transfer. The Navier-Stokes equations arise from applying Newton´s second law to fluid motion, its solution describe the motion of viscous fluid substances by a velocity field. It is a velocity field, since it is defined at every point in a region of space and an interval of time.

2.2 Problem formulation

This project treats the oscillatory flow about different geometries. A Cartesian coordinate system (x,y) is created with its origin at the midpoint between the cylinders. The flow is symmetric about x=0 and y=0 planes. The flow has a velocity magnitude $U \cdot cos(wt)$, and is directed perpendicular to the x-axes. The characteristic lengths of the problem are: the radius of the cylinders (a), the distance between the centers of the cylinders (g), the amplitude of displacement of the fluid particles in the fluctuating flow, $d = \frac{U}{\omega}$, and the measure of the Stokes layer at the surface of the cylinder, $\delta = \sqrt{\frac{\nu}{\omega}}$.

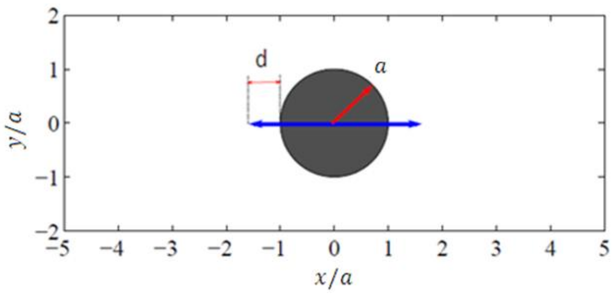


Figure 2.1 Representation of parameters d and a [1]

Here in this figure are represented some of the parameters mentioned before, it is represented the amplitude of displacement of the fluid particles, d, the radius of the cylinder, a, and the direction of the flow velocity, perpendicular to the x axis.

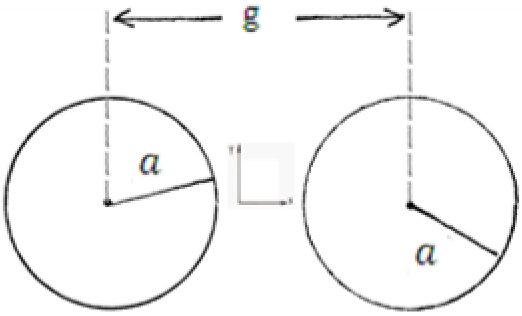


Figure 2.2 Representation of parameter g

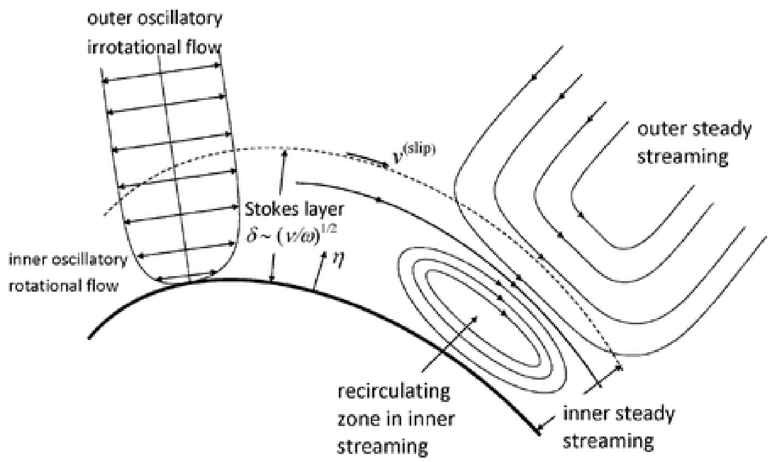


Figure 2.3 Representation of the Stokes layer [2]

The Stokes layer thickness is defined by Daniel T. Schwartz as a "natural scaling parameter for oscillating flows that describes how far viscous damping of an oscillating velocity gradient persist from a surface" $\delta = \sqrt{\frac{\nu}{\omega}}$, where ω is the oscillation frequency and v is the kinematic viscosity.

Here, in these two pictures are represented what is the Stokes layer thickness and, what is g, the distance between cylinder. It is also represented where is positioned the center of the reference frame.

Thanks to the dimensional analysis in our project it is concluded that there are three main parameters that mostly govern the steady streaming induced by an oscillating flow around an object; they defined by three dimension characteristic values of the problem, U as velocity, ω^{-1} as time and "a" as length:

- The non-dimensional oscillation amplitude:

$$\varepsilon = \frac{U}{\omega a} = \frac{d}{a} \qquad [1]$$

where d is the amplitude of oscillations, a is the characteristic length of the object in contact with the oscillating flow.

- The streaming Reynolds number:

$$R_s = \frac{U^2}{\omega \nu} = \frac{\varepsilon^2 \omega a^2}{\nu} = \varepsilon \cdot \frac{Ua}{\nu} = \varepsilon \cdot Re \qquad [2]$$

where ν is the kinematic viscosity of the fluid, and ω is the angular oscillation frequency.

- The non-dimensional separation between cylinders:

$$g_a = \frac{g}{a} \qquad [3]$$

being g the distance between the cylinder centers.

There is another non-dimensional parameter important to define, it is the non-dimensional time, represented by τ

Epsilon is the dimensionless oscillation amplitude, which is assumed in this project, to be very small in comparison to order unity (ε <<1).

A time dependent steady streaming motion is created due to the fluctuating Reynolds stresses in the stokes layer at the surface of the cylinders; this steady streaming motion will persist beyond the stokes layer and will be responsible for an outer streaming motion with a velocity of the order $\varepsilon \cdot U$ [7].

The second parameter governing the flow is the streaming Reynolds number $R_s$which is based on the latter velocity $\varepsilon \cdot U$, this R_s is the appropriate Reynolds number for the steady streaming.

And finally the third dimensionless parameter, g_a, is the ratio of the gap width to the cylinder radius, and is assumed to be of the order unity.

Why these parameters are important?

Oscillation Amplitude (ε): is important because this parameter may cause the flow to be inviscid or not, this implies having viscous effects at the surface of the object if the flow is not inviscid (no viscous effects). Oscillation amplitude can also cause the flow to be steady streaming. If ε is small, the flow is essentially inviscid with no viscous effects, while if it is one order of magnitude higher, the flow is a steady streaming whose velocities are on the order of εU, and U, defined before, is equal to $\omega\varepsilon a$ [8]. When ε (oscillation amplitude) is too large, flow separation can occur.

Streaming Reynolds number (R_s): On the streaming Reynolds number depends the fact that the flow behaves like a Stokes flow or not, and also it affects the velocities that are presents in this flow, hence the trapping forces [8]. This parameter is quite important because our analysis is only valid if the Stokes layer is very thin, hence the inner steady streaming is not playing an important role in determining the flow, what actually defines the flow is the outer steady streaming, what is the flow outside the stokes layer. Vorticity is confined in this layer and it is possible to treat the flow as it was an irrotational flow.

If the streaming Reynolds number R_s is much smaller than 1, the streaming flow is essentially Stokes flow, and the associated velocities are very weak, what makes trapping forces almost inexistent. At $R_s \ll 1$ the Stokes layer is quite huge, the flow is essentially the inner steady streaming instead of the outer steady streaming, what is what is analyzed in this project. Microeddies are formed around the object, and the trapping force is directly proportional to the streaming velocities in that eddies. As the Reynolds increases, eddies become thinner layers. Last but not least, in the regime where R_s is much higher than 1 is where the strongest eddies are created, and thus the highest trapping forces. As mentioned before the flow govern by a high streaming Reynolds is less controllable, just a very small variation could lead to undesired flow topologies.

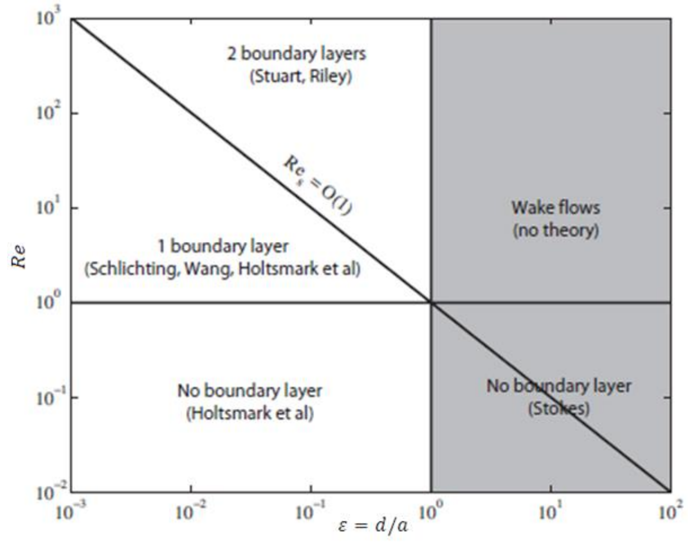


Figure 2.4 Influence on the flow of Re and ε [3]

This figure resumes what is explained above about the parameters that are important for the flow control. As the oscillation amplitude increases, flow separation is more likely to happen. If it is small the flow behaves like inviscid flow. On the other hand streaming Reynolds number is important because it determines whether or not there is a boundary layer for a fixed value of the oscillation amplitude. As R_s increases 2 boundary layer are formed, however if R_s is much smaller than one there is no boundary layer, it behaves like Stokes flow.

Non dimensional separation between cylinders (g_a): on this parameters depend the different geometries proposed to be studied, and the values of the velocity and streamlines developed in the resultant flow.

Coming back to the description of the problem; the dimensionless Navier-Stokes equation governing the oscillatory flow is written below in terms of the stream function (ψ); the stream function is related to the velocity as follows:

$v = (v_x, v_y)$, where $\quad v_x = \frac{\partial\psi}{dy} \quad$ and $\quad v_y = -\frac{\partial\psi}{dx}$

Navier-Stokes equation:

$$\frac{\partial}{\partial t}(\nabla^2\psi) - \varepsilon\,\frac{\partial(\psi, \nabla^2\psi)}{\partial(x, y)} = \varepsilon^2\nabla^4\psi \qquad [4]$$

The boundary conditions require that:

- $v = 0$ on the surface of the cylinders
- Far away from the cylinders $\psi = x \cdot \cos(\tau)$, what actually implies that $v_x = 0$, since $v_x = \frac{\partial\psi}{dy}$, there is only velocity in y component.

Integrating the Navier Stokes equation subject to the boundary conditions explained above, gives as a result the complete temporal evolution of the flow. If the time average over an oscillation is performed the steady streaming motion is obtained. As $\tau \to \infty$, the influence of the particular initial condition vanishes, and therefore the time average over the next cycles is kept constant, it does not change any longer. In this project is analyzed this long term steady streaming behavior. Solving the time dependent problem of the Navier-Stokes numerically during a sufficient long time span to obtain the asymptotic steady streaming behavior is costly [9]. This is why an approximation method is used, this approximation takes advantage of the limit $\varepsilon \ll 1$, in which the stokes layer thickness, of the order of ($\frac{\delta}{a} = \varepsilon \cdot R_s^{-1/2}$), becomes very small, and in which the steady streaming motion can be obtained by a single integration of the steady Navier-Stokes equations, imposing as a boundary condition on the surface of the cylinders the streaming velocity that persist at the edge of the stokes layer. To calculate the solution when $\varepsilon \ll 1$, the stream function is expanded as follows:

$$\psi(x,\tau) = \psi_0(x,\tau) + \varepsilon\left\{\psi_1^{(u)}(x,\tau) + \psi_1^{(s)}(x)\right\} + \mathcal{O}(\varepsilon^2) \tag{5}$$

Where the steady streaming term is indicated with the superscript (s) and the unsteady term is indicated with the superscript (u) [9].

At the leading order term, (obviating terms that have epsilon) the flow is inviscid, and is given by:

$$\psi_0(x,\tau) = \breve{\psi}_0(x)\cdot\cos(\tau) \tag{6}$$

Where $\breve{\boldsymbol{\psi}}_0$ correspond to the potential flow around the geometry. Then the new equation to solve is:

$$\nabla^2\breve{\psi}_0 = 0 \tag{7}$$

To be solved using the same boundary conditions that were explained before, in the far field they are:

$$\breve{\psi}_0 \sim x \qquad as\ \ |x| \to \infty$$

At the surface of the cylinders, the no-slip condition cannot be satisfied. Instead only the normal part of the velocity $\tilde{\boldsymbol{v}}_0 \cdot \hat{\boldsymbol{n}}$ is set to zero, what actually makes sense, in this way no fluid enters or goes out the cylinder. The tangential part is then the slip velocity, $V_0 = \tilde{v}_0 \cdot \hat{\boldsymbol{t}}$. This velocity is accommodate in the inner viscous stokes layer mentioned at the beginning. However what is studied in this project is the steady streaming part of the solution of the outer flow, the Stokes layer is assumed very small thanks to $\varepsilon \ll 1$. An analysis of leading order solution of the inner stokes layer was done by Wilfred Coenen, it is explain in detail on the equation 3.10 of Coenen and Riley (2009) [10]. This paper shows that the streaming therein persists beyond the inner Stokes layer, and it is seen in the outer flow as a streaming velocity $\varepsilon \cdot U_e^{(s)}$ at the edge of the Stokes layer, where:

$$U_e^{(s)} = -\frac{3}{4}V_0\hat{t}\cdot\nabla V_0 \tag{8}$$

An analytic expression for $U_e^{(s)}$ in a bipolar coordinate system is given by equations (3.5) and (3.15) in the previous mentioned paper: Coenen and Riley (2009) [10]. The steaming velocity $\varepsilon U_e^{(s)}$ at the edge of the stokes layer drives the outer streaming motion at $\mathcal{O}(\varepsilon)$, governed by the steady Navier Stokes equations with R_s as Reynolds number,

$$\frac{\partial\left(\psi^{(s)},\nabla^2\psi^{(s)}\right)}{\partial(x,y)} + \frac{1}{R_S}\nabla^4\psi^{(s)} = 0 \tag{9}$$

What is the same that:

$$\nabla \cdot v^{(s)} = 0 \tag{10}$$

$$v^{(s)} \cdot \nabla v^{(s)} = -\nabla p^{(s)} + \frac{1}{R_S} \nabla^2 v^{(s)} \tag{11}$$

If the primitive variables are used, $v^{(s)} = \left(u^{(s)}, v^{(s)}\right)$ and $p^{(s)}$

The outer steady streaming velocity $v^{(s)}$ can be obtained by solving the two previous equations subject to the boundary conditions

- $v^{(s)} = U_e^{(s)} \hat{t}$ on the surface of the cylinders
- $v^{(s)} \to 0$ far away from the cylinders

This is the problem that has to be solved. To calculate V_0, needed for equation [8] it is necessary to solve the potential flow, besides some of the results given to represent the flow will be the potential flow and the streamlines. So in the following paragraphs it is explained how the resultant potential flow, that later will be shown as a solution, is obtained.

External flows around objects can be treated as inviscid if they are frictionless; it is assumed that the contact between the fluid and the object is smooth; or irrotational if the fluid particles are not rotating.

$$\nabla \times \vec{\mathrm{V}} = 0 \tag{12}$$

This flow can be assumed irrotational mainly because all the viscous effects are contained in the thin Stokes layer, as explained before. This is the assumption followed in this project, which is why it can be said that the external steady streaming is irrotational, and therefore the potential function can be obtained from this flow.

Potential flow is defined as a function that is continuous and that satisfies the conservation mass, conservation of momentum laws, with the assumption of incompressible, irrotational and inviscid flow.

Potential flow is then defined by the function $\phi = \phi(x, z, t)$, whose components in Cartesian coordinates are expressed as follows:

$$u = \frac{\partial \phi}{\partial x}, \quad v = \frac{\partial \phi}{\partial y}, \quad w = \frac{\partial \phi}{\partial z}$$

As the velocity has to satisfy the conservation of mass it can be written:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = U \tag{13a}$$

In other words:

$$\frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} + \frac{\partial^2 \phi}{\partial z^2} = U \qquad [13b]$$

Which is actually the Laplace equation:

$$\nabla^2 \phi = U \qquad [13c]$$

There are lines where $\boldsymbol{\phi}$ is kept constant, that lines are called potential lines, for this project, where only 2 dimensions are considered, potential lines are given by:

$$d\phi = \frac{\partial \phi}{\partial x} dx + \frac{\partial \phi}{\partial y} dy \qquad [14]$$

$$d\phi = udx + vdy \qquad [15]$$

What finally becomes:

$$\frac{\partial y}{\partial x} = -\frac{u}{v} \qquad [16]$$

Since, as mentioned above $\boldsymbol{d\phi}$ is kept constant and equal U along a potential line [10].

Making advantage of the fact that potential lines are perpendicular to the modulus of the velocity, it can be defined another function $\boldsymbol{\psi}$, called streamfunction that determine the streamlines, they are related with potential lines by a perpendicular relationship, streamlines are perpendicular to potential lines, therefore they are tangent to the velocity, what makes the analysis much more intuitive.

Streamlines are then defined by:

$$\frac{\partial y}{\partial x} = \frac{v}{u} \qquad [17]$$

Streamlines are always perpendicular to potential lines except in the case of a stagnation point, where there is no velocity.

Chapter 3

Numerical method

In this third chapter the software used is described, its characteristics and the advantages of it use. Here it is also explained briefly the method to solve the equations of the problem, the Newton-Raphson method.

3.1 Software used

FreeFem ++ as its name implies is free and is based on the finite element method.

FreeFem++ is based on C++, however it has its own language, it was developed in the Université Pierre et Marie Curie. It runs on GNU/Linux, Solaris, OS X and MS Windows systems.

FreeFem ++ is the software used to compute the flow analysis, it is a high level IDE (integrated development environment) capable of solving numerically a huge quantity of PDE (partial differential equations) in two or three dimensions, for our project only PDEs in two dimensions were required to be solved. This software is perfect to deal with finite element models due to its advanced mesh generator, which is able to adapt and improve the mesh once the computation has been performed. Besides this FreeFem has an elliptic solver interfaced with fast algorithms, for instance themulti-frontal method UMFPACK, SuperLU…

A partial differential equation is an equation where there exist a relation between a variable or several variables and its partial derivatives. Since our problem is modeled by several PDEs this software is particularly interesting. The following characteristics have been obtained from: freefem++.doc.pdf [11]

Some of the characteristics of FreeFem++ are [11]:

- Problem description (real or complex valued) by their variational formulations, with access to the internal vectors and matrices if needed.

- Multi-variables, multi-equations, bi-dimensional and three-dimensional static or time dependent, linear or nonlinear coupled systems; however the user is required to describe the iterative procedures which reduce the problem to a set of linear problems.

- Easy geometric input by analytic description of boundaries by pieces; however this part is not a CAD system; for instance when two boundaries intersect, the user must specify the intersection points.

- Automatic mesh generator, based on the Delaunay-Voronoi algorithm; the inner point density is proportional to the density of points on the boundaries.

- Metric-based anisotropic mesh adaptation. The metric can be computed automatically from the Hessian of any FreeFem++ function.

- High level user friendly typed input language with algebra of analytic and finite element functions.

- Multiple finite elements mesh within one application with automatic interpolation of data on different meshes and possible storage of the interpolation matrices.

- A large variety of triangular finite elements: linear, quadratic Lagrangian elements and more, discontinuous P1 and Raviart-Thomas elements, elements of a non-scalar type, the mini-element,…

- Tools to define discontinuous Galerkin finite element formulations P0, P1dc, P2dcand keywords: jump, mean, intalledges.

- A large variety of linear direct and iterative solvers (LU, Cholesky, Crout, CG, GMRES,UMFPACK, MUMPS, SuperLU…) and eigenvalue and eigenvector solvers(ARPARK).

- Near optimal execution speed (compared with compiled C++ implementations programmed directly).

- Online graphics, generation of, .txt,.eps,.gnu, mesh files for further manipulations of input and output data.

- Many examples and tutorials: elliptic, parabolic and hyperbolic problems, Navier-Stokes, elasticity, Fluid structure interactions, Schwarz's domain decomposition method, eigenvalue problem, residual error indicator, etc…

Another mathematical tool very useful to solve the problem of analyze the flow has been the Newton-Raphson iterative method of solving equations.

3.2 Newton method in Freefem++

Newton-Raphson method is widely used for solving equations numerically. This method is based in the idea of linear approximation.

Newton method starts with an estimation of the root, sometimes called "guess", this initial estimation has to be chosen with care, since the iterative process to find the solution could be very quickly if the guess is close enough to it, or could be horrid if it is not the case.

Once the initial estimation of the solution (x_n) has been chosen, an improved estimation (x_{n+1}) is calculated. From this new calculated estimation it is produced another one (x_{n+2}), until the estimation is close to the actual solution or the variation between the estimations is so small that is clear that we are close enough to the solution [12].

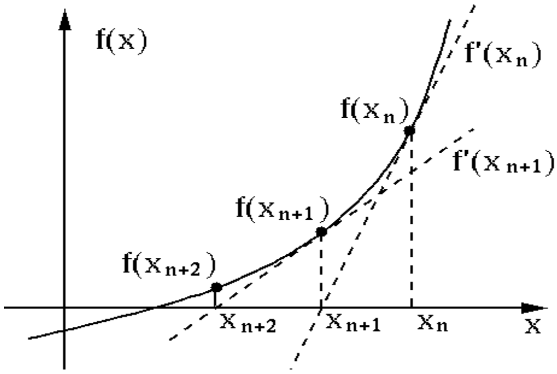


Figure 3.1Newton-Raphson iterative process [4]

This is the easy explanation when there is only one unknown, in our equations there are three of them, which is one of the reasons to use Freefem++, since it will help in this task.

The equations to be solve with Freefem++ are:

$$\nabla \cdot v^{(s)} = 0 \qquad [10]$$

$$v^{(s)} \cdot \nabla v^{(s)} = -\nabla p^{(s)} + \frac{1}{R_S} \nabla^2 v^{(s)} \qquad [11]$$

With the boundary conditions

- $v^{(s)} = U_e^{(s)} \hat{t}$ on the surface of the cylinders
- $v^{(s)} \rightarrow 0$ far away from the cylinders

To solve these equations and been able to get the three variables, the Newton iterative method is implemented in Freefem++ as follows:

$$\begin{bmatrix} u \\ v \\ p \end{bmatrix}^{n+1} = \begin{bmatrix} u \\ v \\ p \end{bmatrix}^{n} - [J^n]^{-1} \cdot F\left(\begin{bmatrix} u \\ v \\ p \end{bmatrix}^{n}\right) \qquad [18]$$

Where F is the function coming from [10] and [11] and the matrix J is the jacobian operator.

Equation [18] can be written in vectorial form as:

$$\bar{X}^{n+1} = \bar{X}^n - \bar{\bar{J}}(\bar{X}^n)^{-1} \cdot \bar{F}(\bar{X}^n) \quad [19]$$

Where vector X is the three unknowns in vectorial form. Equation [19] is the same that:

$$\bar{\bar{J}}(\bar{X}^n)(\bar{X}^{n+1} - \bar{X}^n) = -\bar{F}(\bar{X}^n) \quad [20]$$

Where $(\bar{X}^{n+1} - \bar{X}^n)$ can be expressed as the minus increment in the solution, what is defined as: $-\bar{W}^n = (\bar{X}^{n+1} - \bar{X}^n)$, so:

$$\bar{\bar{J}}(\bar{X}^n)\bar{W}^n = \bar{F}(\bar{X}^n) \quad [21]$$

In Freefem++ this can be solved introducing the operator $DF(r_i)\delta_i = F(r_i)$

Being $\delta_i = r_i - r_{i+1}$

Where DF(r) is the differential of F at point r, this is a linear application such that:

$$F(r+\delta) = F(r) + DF(r)\delta + o(\delta\,) \quad [22]$$

What means: $DF(u,v,p)(\delta u, \delta v, \delta p) = -F(u,v,p)$

$$DF(r) = \frac{F(r+\delta) - F(r)}{\delta} \quad [23]$$

$$\delta \cdot DF(r) = F(r+\delta) - F(r) \quad [24]$$

$$-F(r) = F(r+\delta) - F(r) \quad [25]$$

$$F(u+\delta u, v+\delta v, p+\delta p) = 0 \quad [26]$$

This at the end becomes in the following three equations to be solved:

$$(u+\delta u)\cdot\frac{\partial(u+\delta u)}{\partial x} + (v+\ \delta v)\cdot\frac{\partial(u+\delta u)}{\partial y} = \frac{\partial(p+\delta p)}{\partial x} + \frac{1}{R_s}\nabla^2(u+\delta u) \quad [27]$$

$$(u+\delta u)\cdot\frac{\partial(v+\delta v)}{\partial x} + (v+\ \delta v)\cdot\frac{\partial(v+\delta v)}{\partial y} = \frac{\partial(p+\delta p)}{\partial y} + \frac{1}{R_s}\nabla^2(v+\delta v) \quad [28]$$

$$\frac{\partial}{\partial x}(u+\delta u) + \frac{\partial}{\partial y}(v+\ \delta v) = 0 \quad [29]$$

Chapter 4

Models of study and results

In this section, the design of the different models created is explained. The complexity of the models is increased in order to approximate the model to the final one, with capabilities to create microeddies where particles can be trapped.

The main idea behind this project is to be able to predict whether a geometry is able to produce microeddies around solid surfaces as the flow is oscillating inside the microdevice with a net zero flow. This means no additional flow is injected in the device, and the law of conservation of mass is fulfilled, there is no a free stream inside the microdevice.

As mentioned above, solving the time dependent problem of the Navier Stokes numerically during a sufficient long time span to obtain the asymptotic steady streaming behavior is costly; that´s why an approximation method is used. Thanks to the fact that the Stokes layer thickness, of the order of ($\frac{\delta}{a} = \varepsilon \cdot R_s^{-1/2}$), becomes smaller than 1, the steady streaming motion can be obtained by a single integration of the steady Navier Stokes equations, what makes things easier. The only thing to do is impose as a boundary condition on the surface of the cylinders the streaming velocity that persists at the edge of the Stokes layer, which actually is:

$$U_e^{(s)} = -\frac{3}{4} V_0 \hat{t} \cdot \nabla V_0 \qquad [8]$$

This is the modulus of the velocity. To be able to solve the problem it is required also the direction of this velocity. So, the first thing to perform here is to define the tangent vectors of the boundary where this velocity is to be settled.

Tangent vector is divided in two components, one in each direction of the defined geometry:

The tangent vector in x component is defined to be:

$$\mathrm{tx} = -\sin\left[\pi - \tan^{-1}\left(\frac{y-0}{-x+Ccx}\right)\right] \qquad [30]$$

The tangent vector in y component is defined to be:

$$\mathrm{ty} = \cos\left[\pi - \tan^{-1}\left(\frac{y-0}{-x+Ccx}\right)\right] \qquad [31]$$

Where Ccx is the distance of the center of the cylinder to the origin of the coordinate system.

Once the tangent vector is defined, it is possible to calculate the u and v components of the velocity; the velocity in x direction, u, is calculated as it is explained below:

$$u_s = -\frac{3}{4} \cdot \left(\frac{\partial\psi}{\partial x} \cdot tx + \frac{\partial\psi}{\partial y} \cdot ty\right)$$
$$\cdot \left[tx \cdot \left(\frac{\partial^2\psi}{\partial x^2} \cdot tx + \frac{\partial\psi}{\partial x} \cdot \frac{\partial tx}{\partial x} + \frac{\partial^2\psi}{\partial x\partial y} \cdot ty + \frac{\partial\psi}{\partial y} \cdot \frac{\partial ty}{\partial x}\right) + ty\right.$$
$$\left. \cdot \left(\frac{\partial^2\psi}{\partial x\partial y} \cdot tx + \frac{\partial\psi}{\partial x} \cdot \frac{\partial tx}{\partial y} + \frac{\partial^2\psi}{\partial y^2} \cdot ty + \frac{\partial\psi}{\partial y} \cdot \frac{\partial ty}{\partial y}\right)\right] \cdot tx \qquad [32]$$

On the other hand the velocity component in y direction, v, is calculated thanks to the next formula:

$$v_s = -\frac{3}{4} \cdot \left(\frac{\partial\psi}{\partial x} \cdot tx + \frac{\partial\psi}{\partial y} \cdot ty\right)$$
$$\cdot \left[tx \cdot \left(\frac{\partial^2\psi}{\partial x^2} \cdot tx + \frac{\partial\psi}{\partial x} \cdot \frac{\partial tx}{\partial x} + \frac{\partial^2\psi}{\partial x\partial y} \cdot ty + \frac{\partial\psi}{\partial y} \cdot \frac{\partial ty}{\partial x}\right) + ty\right.$$
$$\left. \cdot \left(\frac{\partial^2\psi}{\partial x\partial y} \cdot tx + \frac{\partial\psi}{\partial x} \cdot \frac{\partial tx}{\partial y} + \frac{\partial^2\psi}{\partial y^2} \cdot ty + \frac{\partial\psi}{\partial y} \cdot \frac{\partial ty}{\partial y}\right)\right] \cdot ty \qquad [33]$$

Where the only difference between the two components of the velocity is the projection, the component in x-direction, u, is multiplied by the tangent in this same direction, while for v, in y-direction the same happens with the tangent in y-direction.

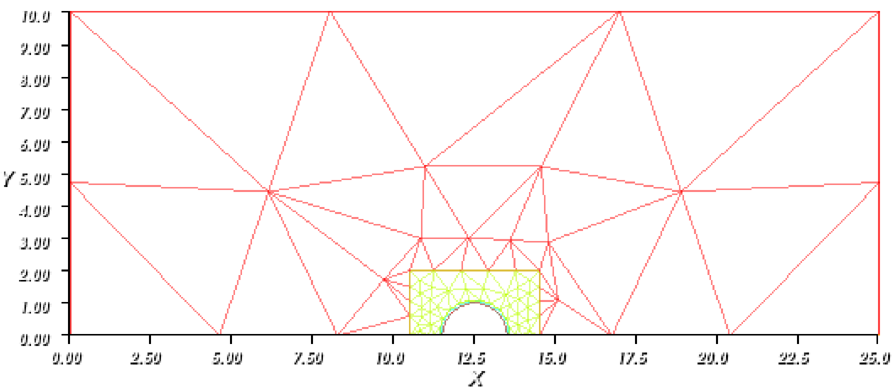
Once the velocity of slip is calculated it is fixed in the surface of the solid bodies. The next step is complete the mesh definition.

4.1 First geometry model

The first model consists on a simulated microdevice where the flow encounters a single cylinder.

In this section how the mesh, for the first model, was developed is explained. The first thing to keep in mind is that there are regions with more importance than others; therefore a more detailed mesh is done in the region near the cylinders. It is in this region where there are produced the biggest and remarkable variations in velocity. This is mainly among other causes, because of the condition imposed in the boundary of the cylinders. All that happens in the inner Stokes layer is simplified in such a way that all the influence that this layer has in the outer region, is modulated as the slip velocity $v^{(s)} = U_e^{(s)}\hat{t}$, and it is assumed in the surface of the cylinder. A very tight region above the cylinders is meshed with a quite huge density of nodes to have an accurate and a great amount of data points to analyze better what it is really happening there. Another mesh with lower density of nodes is done to cover the outer region of the cylinder. Then another mesh is done for the region of lowest interest, where there is the external flow.

In all the borders laying in the x=0 line it is fixed a condition of symmetry, the reason for this is to save time, being able to avoid a computation that is known due to symmetry.



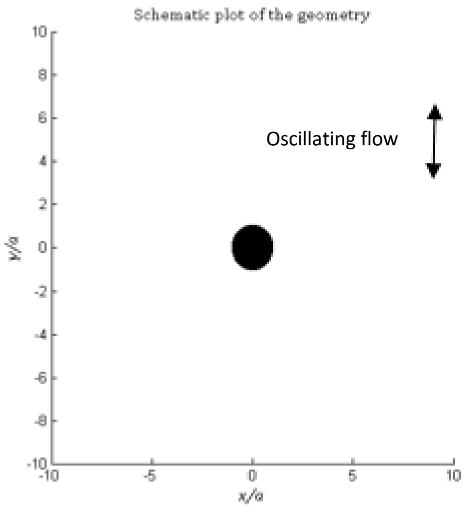


Figure 4.1 First mesh of the first geometry

This first mesh has a total number of triangles of 172 and a total number of vertices of 113

Once the code is run with this geometry, it gives the potential flow, the tangent vector through the surface of the cylinder, the velocity, the streamlines…

With this primitive mesh the results obtained were not accurate enough. The reason for them to not be smooth is the lack of data in some points; this is because the primitive aspect of the mesh, more point and an adaptation of the mesh is required. Since this mesh is not proper for obtaining accurate results, a new code was developed to solve this problem. Once the results have been obtained from running the code with the primitive mesh, the code search for cells where the variations in velocity are bigger to divide these cells into a proportional number of cells according to the variations found. This turns into a high number of cells, each one providing a different value in a region where before there was only a single value. This fact implies an accurate result and a better and smooth representation of the resultant flow. The adaptation of the mesh is also based on a criterion to assign the density of nodes. This criterion is based in several coefficients, different one per each one of the three main regions. The value of the coefficients is increasing as they are close to the cylinder surface. Every region is composed by the joint of borders, which are defined by mathematical functions in C++, the language of Freefem++. The number of vertices of the triangular cells that are going to appear in each border for the new mesh is then calculated by multiplying the coefficient of the region times the length of the border. In this way a better and uniform mesh is generated taking into account the first approximation solution.

The refined mesh is:

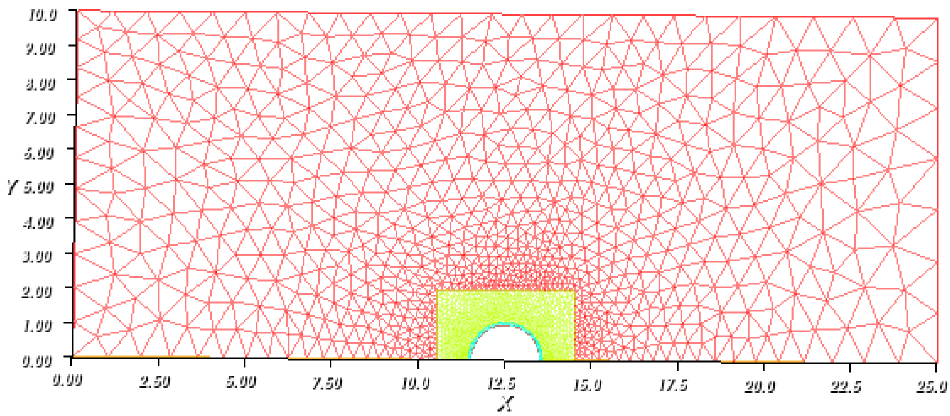


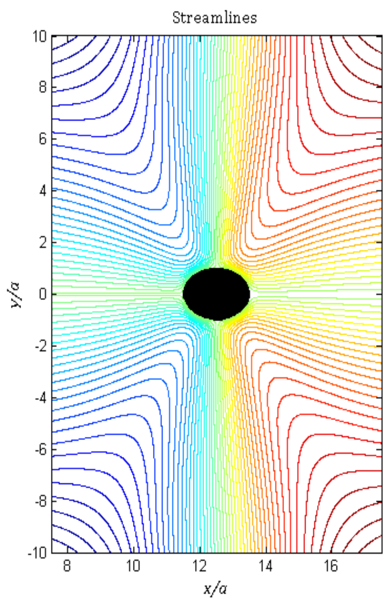
Figure 4.2 Refined mesh of the first geometry

This refined mesh has a total number of triangles of 4428 and a total number of vertices of 2345. Most important than the increment in the number of triangles is the fact that those triangles are positioned in critical regions where the variations in velocity are bigger.

If all these points' data are exported into a matrix to Matlab, it is possible to represent the complete geometry of the problem.

This is possible due to the condition of symmetry imposed in the lower border of the mesh.

The final result for the representation of the streamlines and potential flow are the following ones:



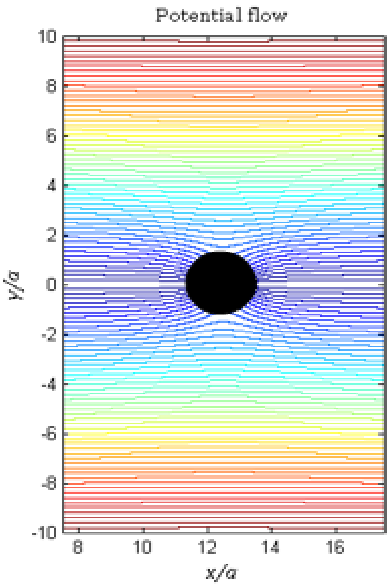


Figure 4.3 Streamlines one single cylinder

Figure 4.4 Potential flow one single cylinder

Figure 4.1 represent the result for solving an oscillatory flow around a cylinder with the before explained values for the main parameters that define the flow. As can be see here there is no region where microeddies appear, that is mainly due, among other causes, to the lack of influence of our oscillations in the flow, what is caused by too high velocity in comparison with our frequency. The result for this configuration is that two jets are ejected from the upper and lower part of the cylinder.

Seeing the result it is concluded that one single cylinder has not enough strength to influence the flow as it is desired, so this geometry is not a possible solution and a new one has to be developed, one with two cylinders oscillating at the same frequency, together, side by side.

4.2 Second geometry model

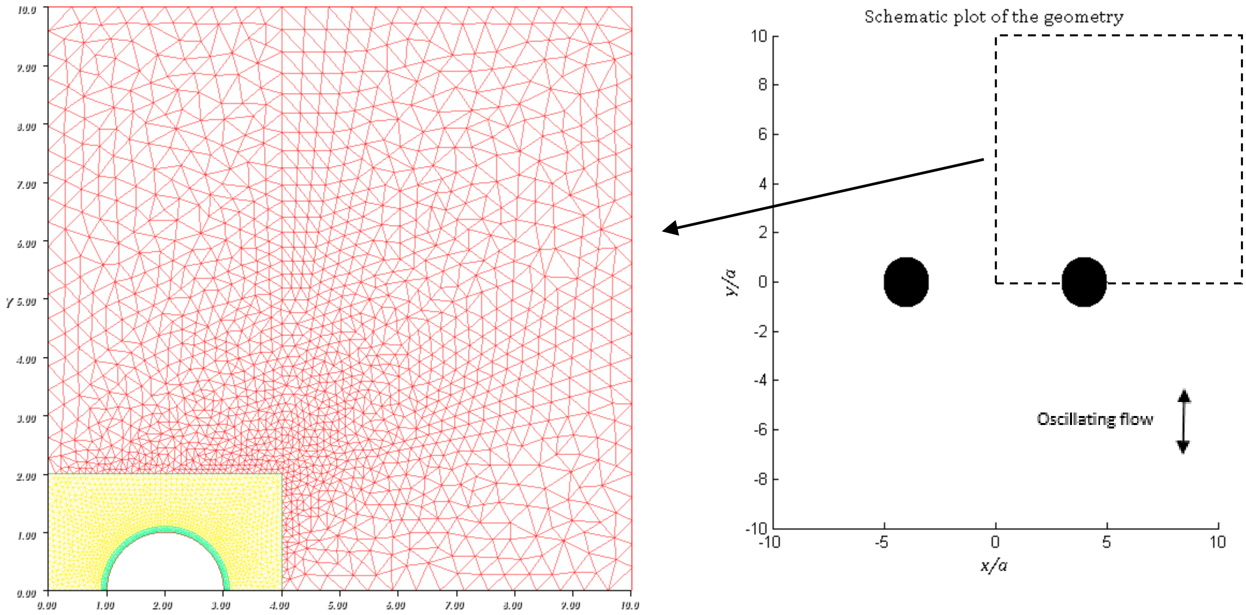


Figure 4.5 Refined mesh of the second geometry

Then the new geometry is now developed. As before the mesh is created with the three regions of study, very close to the cylinder, to be accurate with the values of velocities, another one in the surroundings of the cylinder and the third one to the external flow. In this model where there are two cylinders it is possible to take a double advantage of the symmetric characteristics given in the geometry. This model has, as before, symmetry about the x=0 line, but also about the y=0 line. This mesh is created with the hope that this time the model would be able to produce the required microeddies to trap particles.

4.2.1 Results for the second geometry model

The results to be study in this section are the resultant potential flow, and the obtained streamlines that comes from solving the steady streaming flow on this model of geometry. The following results have been obtained for fixed values of streaming Reynolds number ($R_s = 20$) and the non-dimensional oscillatory amplitude ($\varepsilon = 0.15$), unless other values are specified.

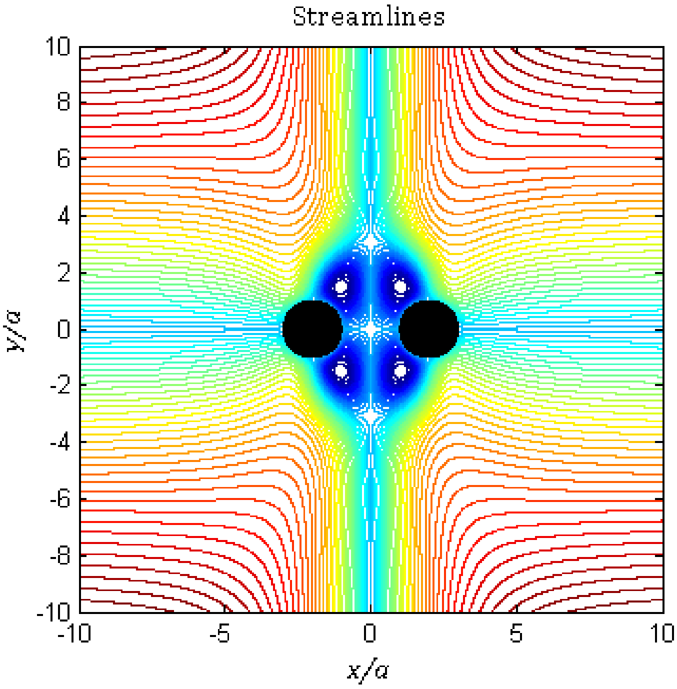


Figure 4.6 Streamlines second model

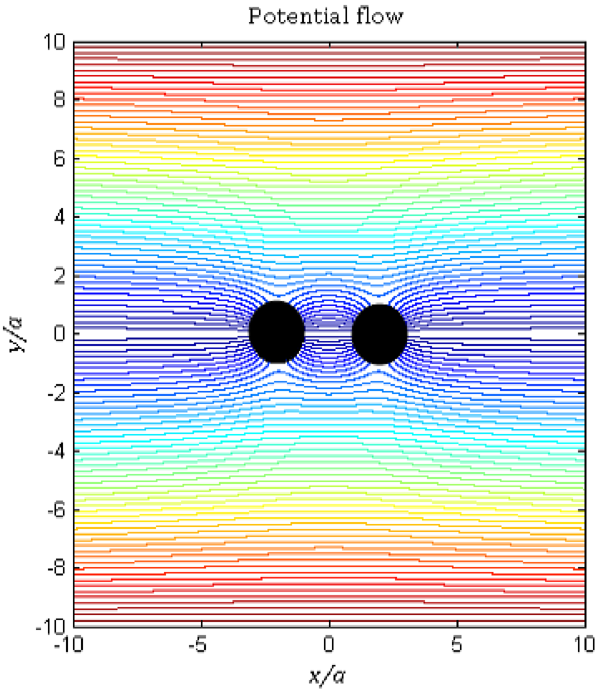
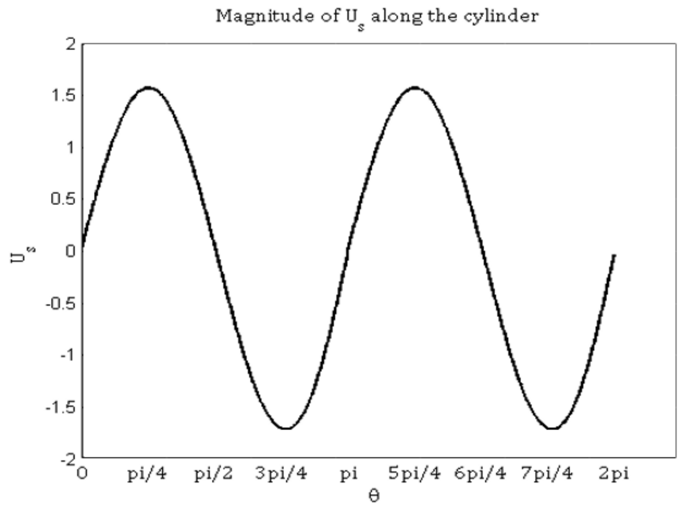


Figure 4.7 Potential flow second model

These figures represent the potential flow and the resultant streamlines when the flow has reached the state of steady streaming.

Streamlines represent the tangential velocity at every point in the flow, therefore there cannot be flow across them. In the Figure 4.6 it can be seen how four vortices are created between the two cylinders. It is in those vortices where particles can be trapped, just in the middle of them, where there are stagnation points, one per vortex. In that figure it is also plotted the variation of the velocity and pressure. If the separation between the streamlines is studied, it can be concluded that it gives the pressure in the zone, if the streamlines are very close to each other the pressure in that region will be low, what implies that the velocity is higher. Following this thought it can be seen how the pressure is decreasing from the right upper corner of the figure towards the direction of the cylinder. The opposite happens with the velocity, it is increased from the right upper corner, where there is almost no velocity, towards the closest region of the cylinder. Analyzing the separation the increment can be determined, if the separation is big the increment in velocity will be big, while if there is no a remarkable separation between streamlines, the difference between those two values will not be a big difference.



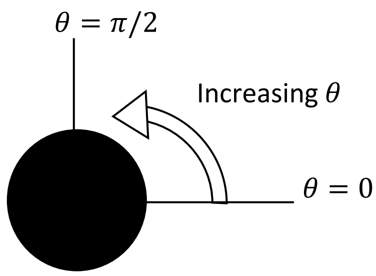


Figure 4.8 Slip velocity assumed in the surface of the cylinders

This is the adimensional magnitude of the slip velocity assumed in the surface of the cylinders, it can be seen that there are four points where the velocity is zero, one in the right of the cylinder at $\theta = 0$, another in the upper part of the cylinder where $\theta = \pi/2$, the third point with no velocity is located in the left of the cylinder at $\theta = \pi$, and the last point is in the lower part of the cylinder, where $\theta = 3\pi/2$. These points results in stagnation points, what can be checked in the streamlines distribution.

4.2.2 Validation of the model

How is it possible to be confident with these results? It is necessary to validate the code and the obtained results somehow, to be sure that the project is able to predict the actual resultant flow produced by passing oscillating fluid through a microchannel where the flow faces solid cylinders. This validation process is required to produce meaningful data, otherwise no one will trust the results obtained with this code.

Validation process is defined as: "the confirmation by examination and the provision of objective evidence that the particular requirements for a specific intended use are fulfilled" [8]

Validation of the model is also important to be respectful with ethical, commercial and regulatory commitments

What is done to validate the code, and therefore to be sure that the results are the real ones, is to compare the resultant flow providing by the code in Freefem++, with the result of an experiment done in the laboratory.

The experiment fixes the value of the parameters, since it is more difficult to control the real flow than the flow in the code.

Two experiments with this same geometry model were performed, each one with different parameters:

$g_a = 3,\ \ \varepsilon = 0.14,\ \ R_s = 31$

The flow was solved with this parameters and the obtained streamlines are:

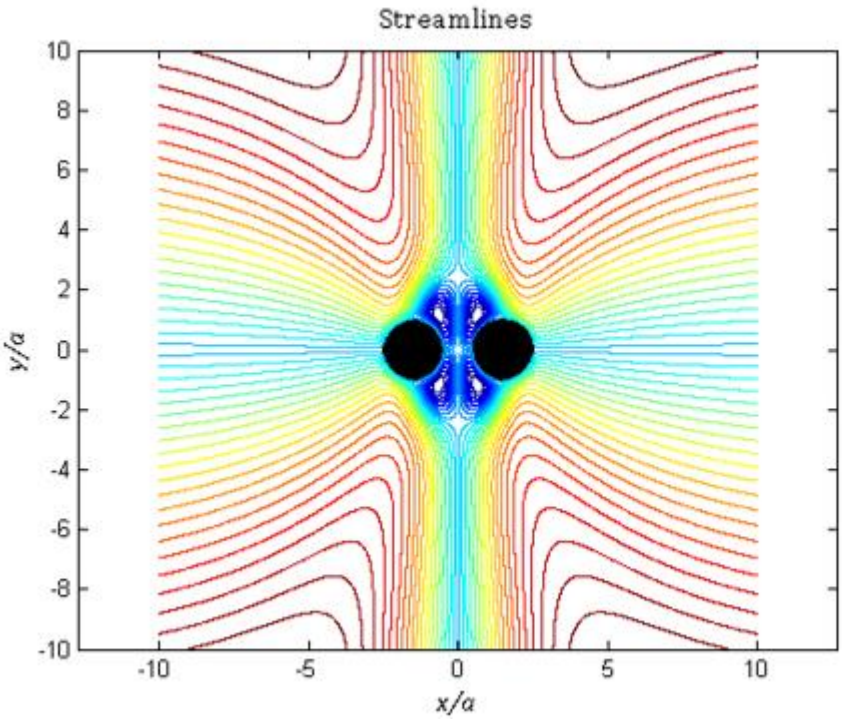


Figure 4.9 Streamlines for $g_a = 3,\ \ \varepsilon = 0.14,\ \ R_s = 31$

As before, the streamlines for this configuration are obtained. Little variations are found between them. The most remarkable ones are found in the position and length of the eddies and the velocity variation due to the difference streaming Reynolds number.

Making zoom in the interest region:

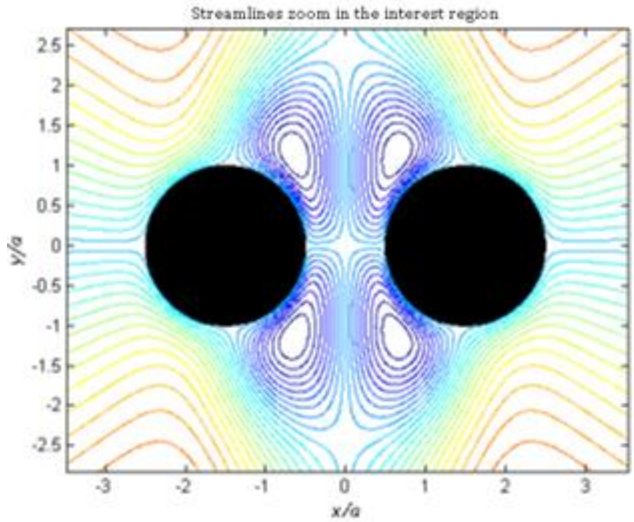


Figure 4.10 Zoom streamlines obtained by the code

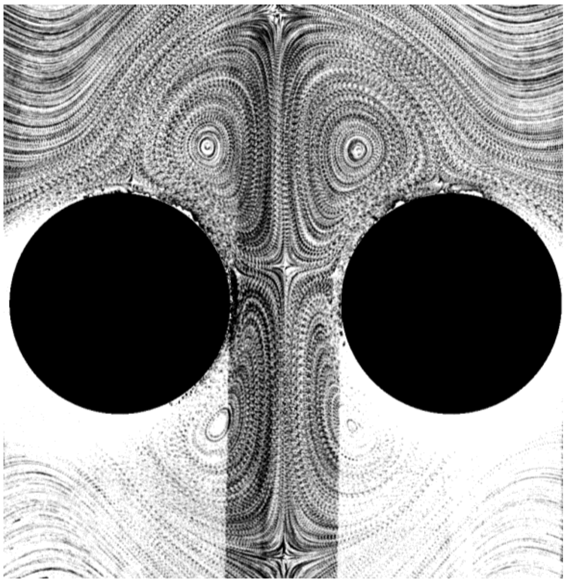


Figure 4.11 Streamlines obtained experimentally

The picture in the right was provided by the tutor of this end of degree project, the experiment shown was performed in the laboratory of the university by him, Wilfried Coenen.

$g_a = 5,\ \ \varepsilon = 0.14,\ R_s = 27$

The flow was solved with this parameters and the obtained streamlines are:

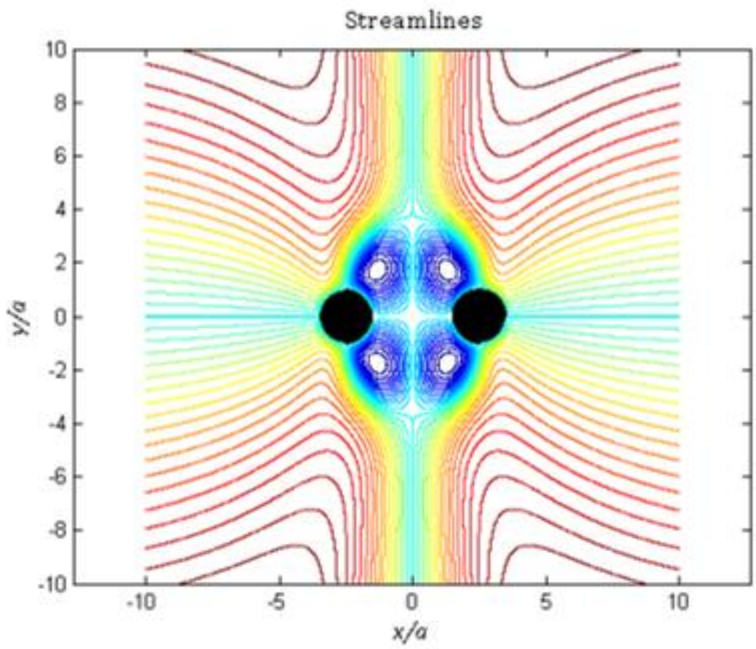


Figure 4.12 Streamlines for $g_a = 5,\ \ \varepsilon = 0.14,\ \ R_s = 27$

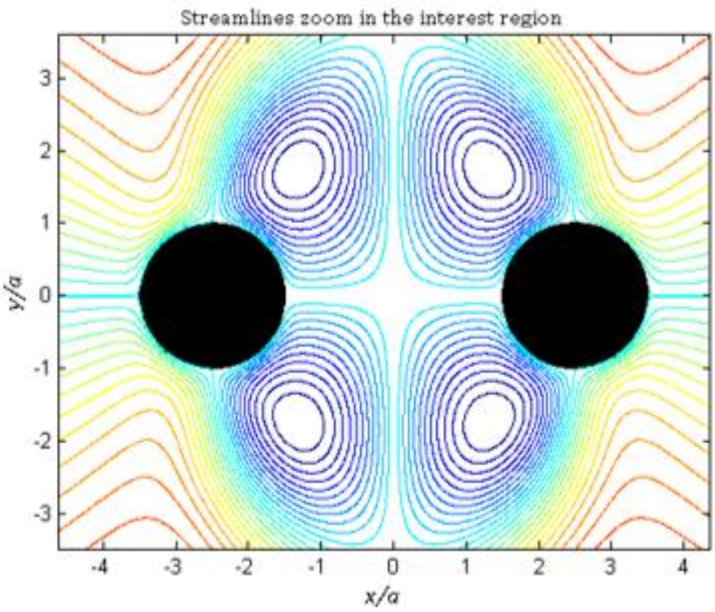


Figure 4.13 Zoom streamlines obtained by the code

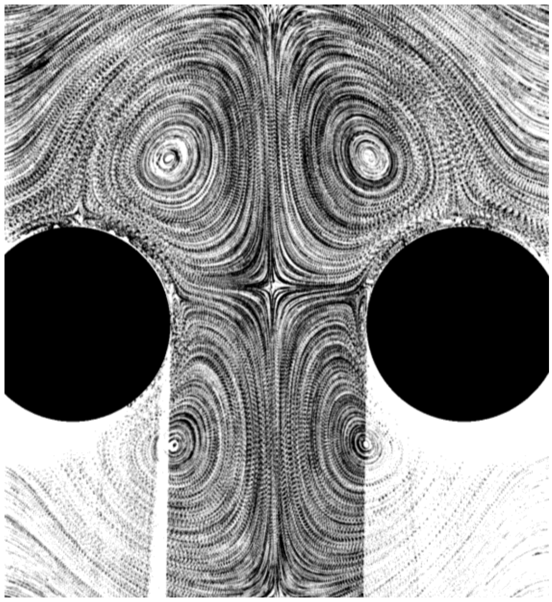


Figure 4.14 Streamlines obtained experimentally

It can be concluded, comparing the experimental results and the numerical analysis of the flow, that the code is able to predict accurately the flow produced by an oscillating streaming flow.

4.3 Third geometry model

This third model consist of four cylinders, grouped in two pairs of them, one pair in each axis. To perform this mesh, the same procedure explained before has been followed. However in this mesh there are 4 regions to divide the domain. As before the most important zones has a high density of nodes, and it is decreased as the zone lose interest. There are now, two regions where the density has to be high enough to reach an acceptable level of accuracy. These zones are the distributed through the surface of the two semi cylinders represented in this mesh thanks to the advantage of symmetry.

And then, following the same criteria than in the previous geometry, the other two regions with different density nodes are fixed. As occurred in the previous model symmetry is used about the x=0 line and y=0 line.

For this geometry, several types of the mesh are shown, for a better understanding of the influence of some parameters.

The different types of the geometry consist on modify the radius and the parameter g_a, to see how is affected the flow.

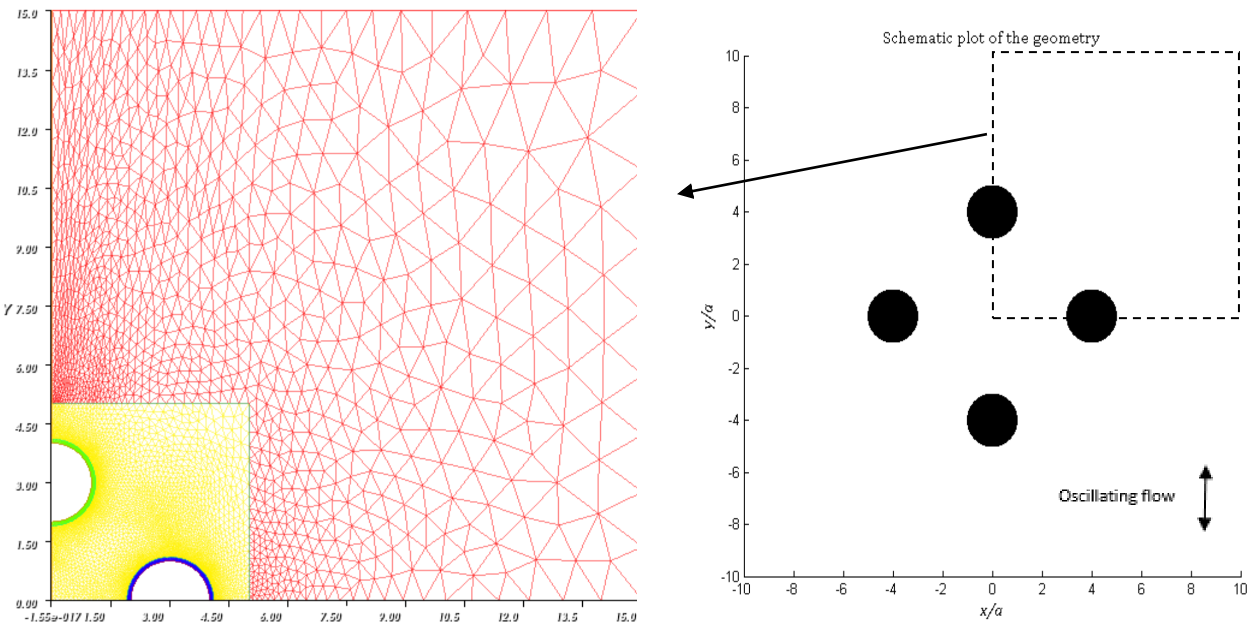


Figure 4.15 Refined mesh for the third model

This picture represent the third model to be study, with the two cylinders and the four regions of interest. Each one of these regions has its cells to provide later the numerical results.

The parameter that fix this geometry is still g_a , but in this case, as there are two pair of cylinders, it is needed to define another parameter g_a, one per pair of cylinders. From now on, and for this geometry g_{ax} will define the distance between centers of the pair of cylinders positioned in the x axis, while g_{ay} will do so with the pair positioned in the y axis

4.3.1 Results for the third geometry model

These following figures represent the resultant flow for the third proposed geometry. As expected four vortices have been created between the cylinders. Just in the middle point of each one of the vortices there is a stagnation point.

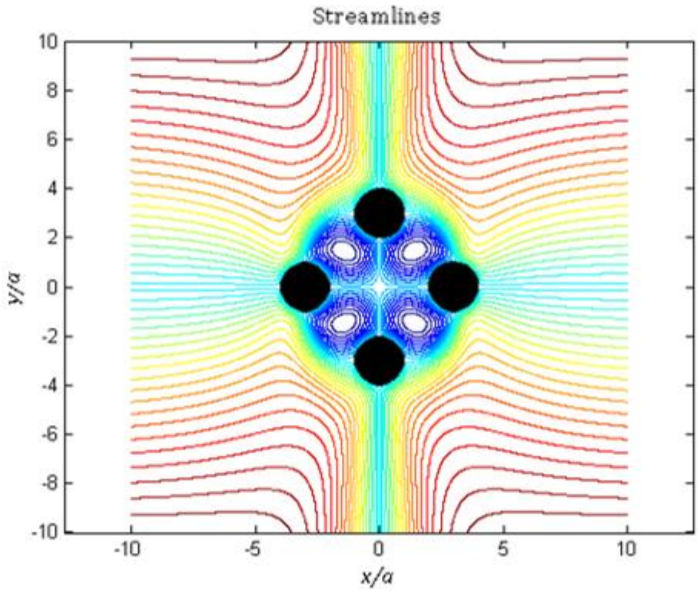


Figure 4.16 Potential flow obtained for the third model

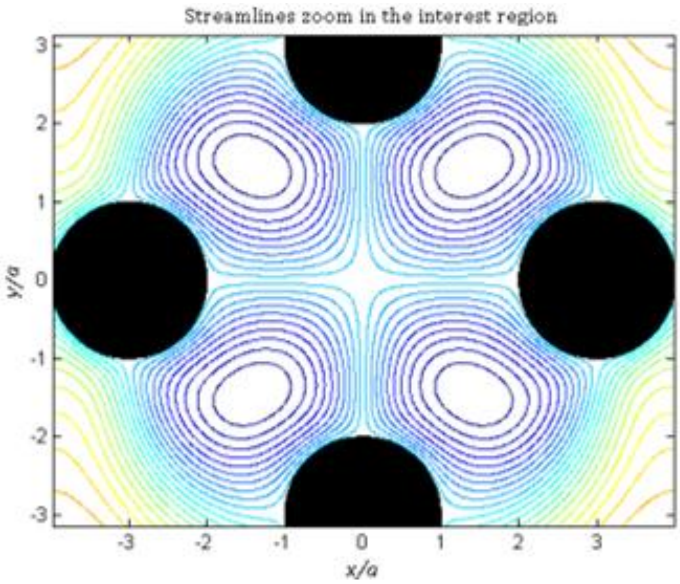


Figure 4.17 Streamlines zoom third model

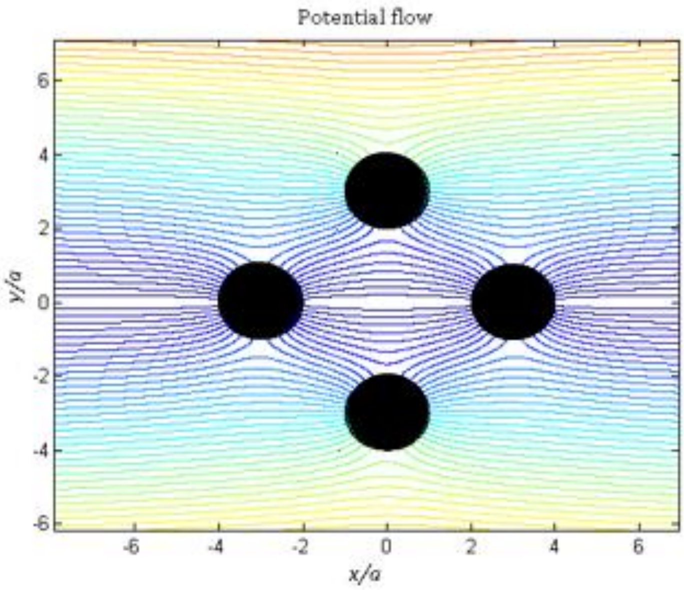


Figure 4.18 Potential flow third model

The flow is oscillating until it becomes steady streaming, as mention in the first chapter of this project the velocity of the flow has a magnitude of $U \cdot cos(\omega t)$, therefore it is oscillating with time. At the initial time the velocity is U directed upwards, and there is a time where this velocity changes its sense.

A velocity field can be represented, this velocity field implies a distribution of velocity in the region. It is a function of the spatial coordinates and time. The velocity of the flow in three different non-dimensional time instants are plotted below, to represent the oscillatory movement of the flow. This non-dimensional time is calculated as time divided by period of oscillation. These three instant are when time is zero, the initial moment, which correspond to the first graph, when time is $\frac{\pi}{2 \cdot \omega}$ that represent the instant where the velocity is about to change its sense from going upwards to downwards, and then the last instant represented is when time is $\frac{\pi}{\omega}$ in which the flow is going downwards.

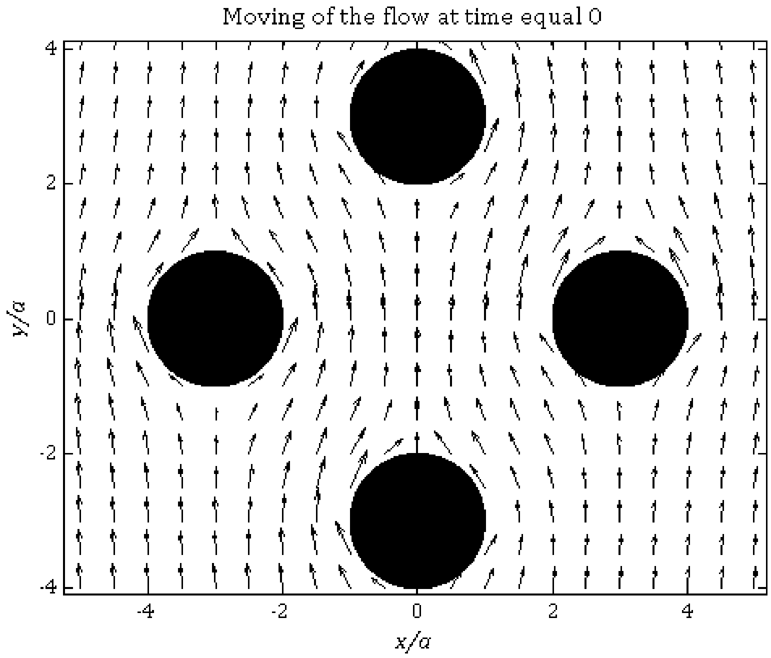


Figure 4.19 Oscillating flow at time equal 0

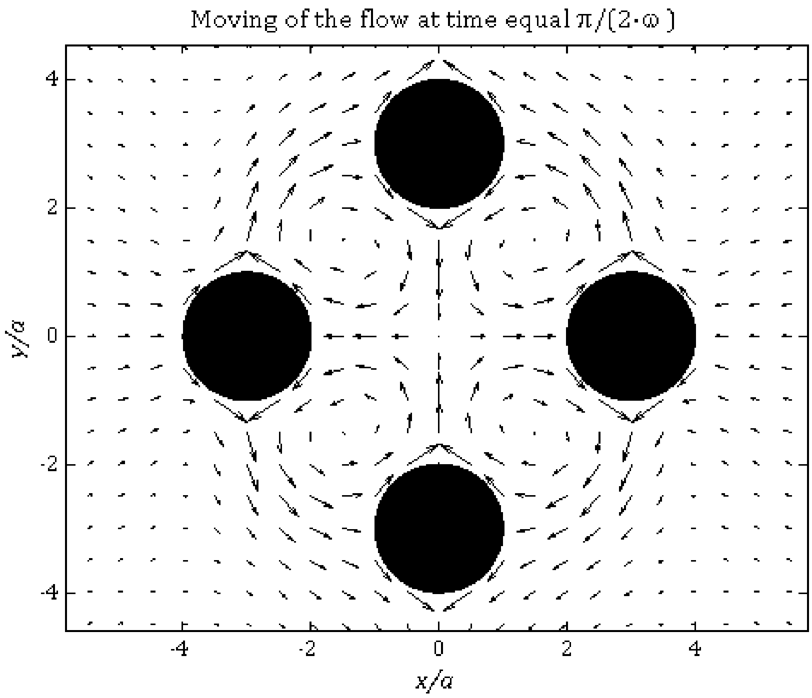


Figure 4.20 Oscillating flow at time equal $\pi/(2 \cdot \omega)$

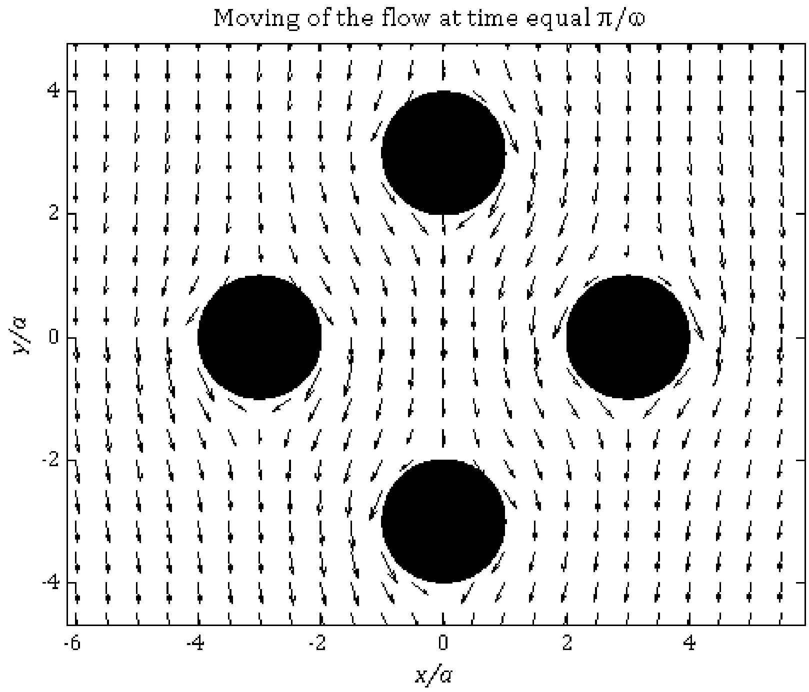


Figure 4.21 Oscillating flow at time equal π/ω

In these pictures where the oscillating flow is represented, a reduction of points to be represented was necessary to be done. Arrows points in the direction of the flow and has a length proportional to its modulus. This fact is remarkable in the second figure, where the flow has stopped to oscillate, and the velocity is zero.

To calculate the mean velocity (average of the different velocities in one cycle) what have to be done is to calculate the velocity for several times in a period and then an average of those velocities.

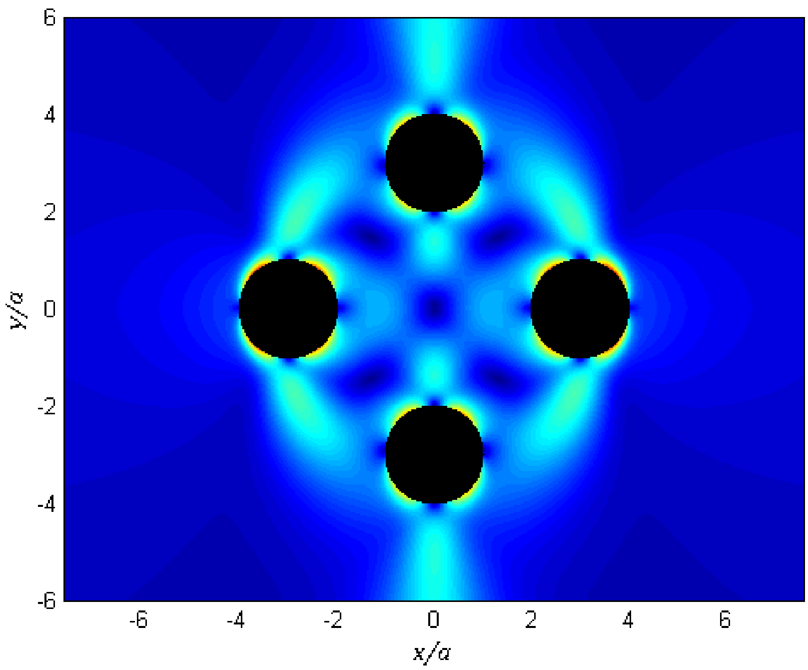


Figure 4.22 Mean velocities in the third model

This figure represent the mean velocities for the configuration $g_a = 6,\ \varepsilon = 0.15,\ R_s = 20$. Values represented in blue mean small values of velocity, the darkest colors means the smallest values. The mean of the oscillatory flow is zero, so what is plotted here is the streaming velocity. As can be seen there are four stagnation points in the cylinders in the positioned mentioned in Figure 4.8, there are also another five stagnation points, which four of them correspond to the stagnation points in the vortex formed in the steady streaming flow, the fifth one correspond to the collision of jets coming out from each cylinder.

4.3.2 First modification (g_{ax}) and its results

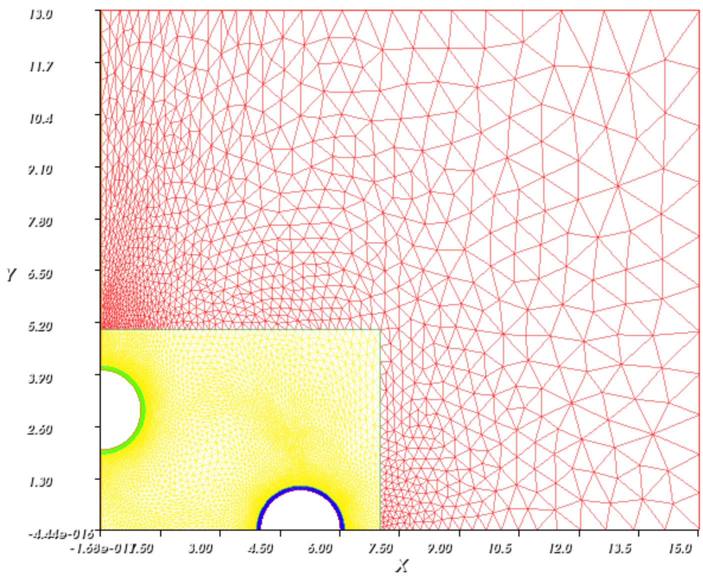


Figure 4.23 Mesh first modification for the third model

In this mesh g_{ax} is increased up to 10, while g_{ay} is 6, this modification is done to be able to see how the distance between the cylinders affect the microeddies formed.

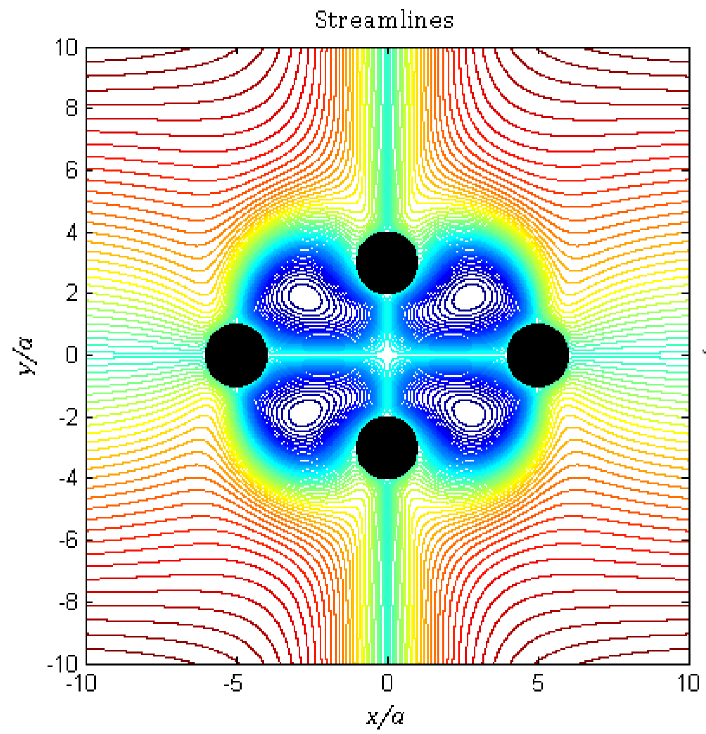


Figure 4.24 Streamlines first modification

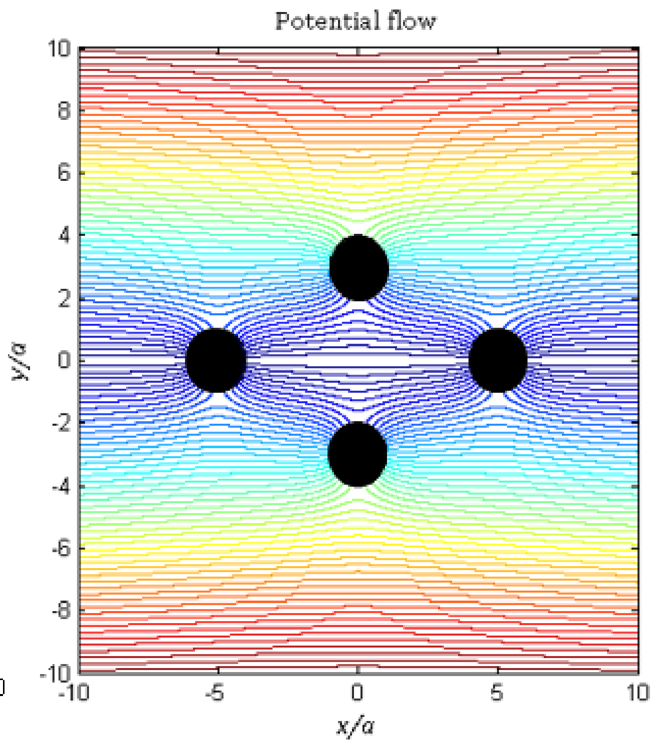


Figure 4.25 Potential flow first modification

The same general results as before are obtained with this modification of increasing g_{ax} are obtained. The potential flow is pretty much the same, potential lines comes out from the surface of the cylinders, what implies that velocity is tangential to the surface of the cylinders, perpendicular to potential lines. The main difference between previous results and these is the length and the position of the vortices formed between the cylinders.

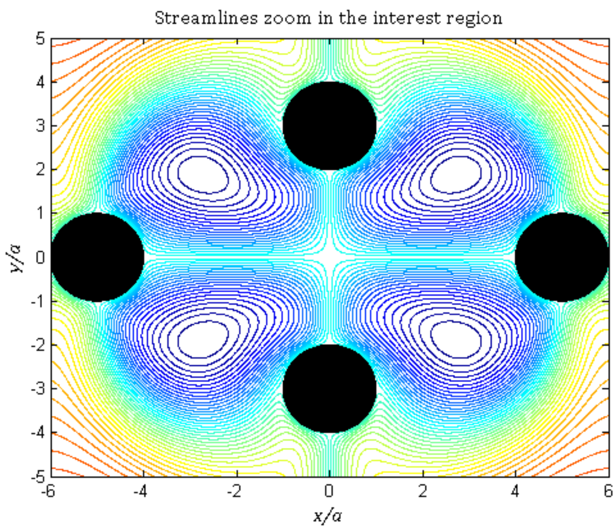


Figure 4.26 Streamlines zoom first modification

4.3.3 Second modification (g_{ax} & g_{ay}) and results

Another mesh modifying g_{ax} and g_{ay} is done, in this case these parameters are 12 both of them.

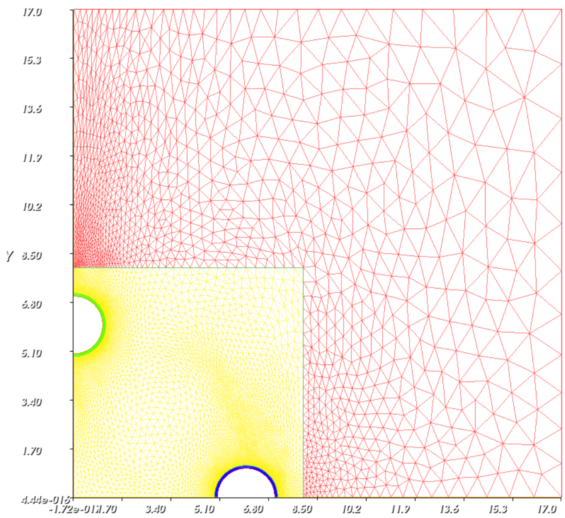
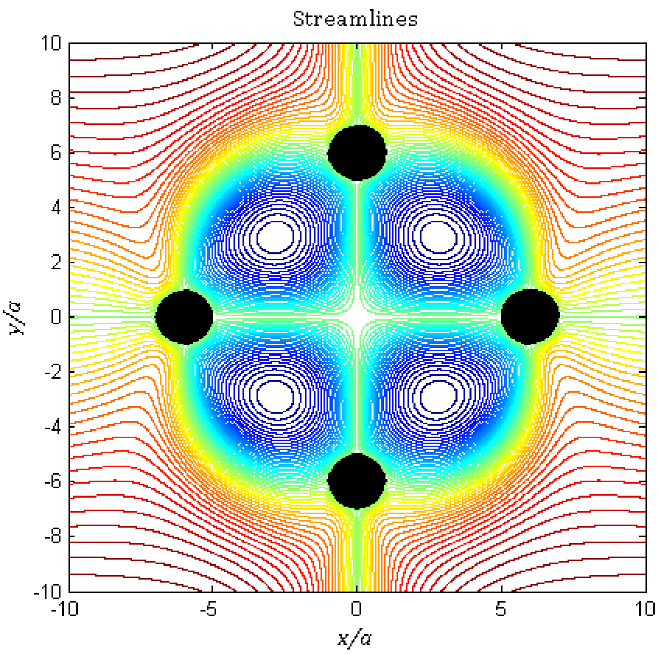


Figure 4.27 Mesh second modification for the third model

As before, there are no significant changes on the resultant flow besides the change in position and the increment in size of the eddies.



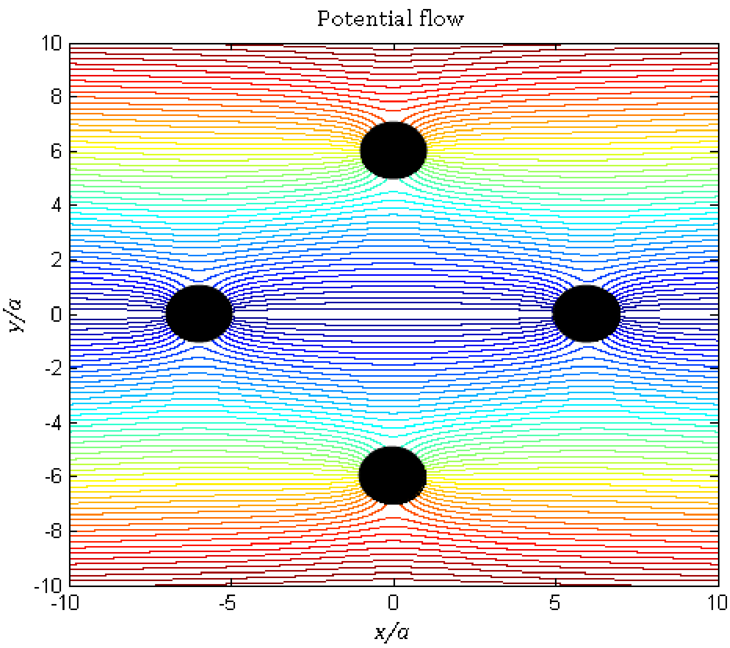


Figure 4.28 Streamlines second modification

Figure 4.29 Potential flow second modification

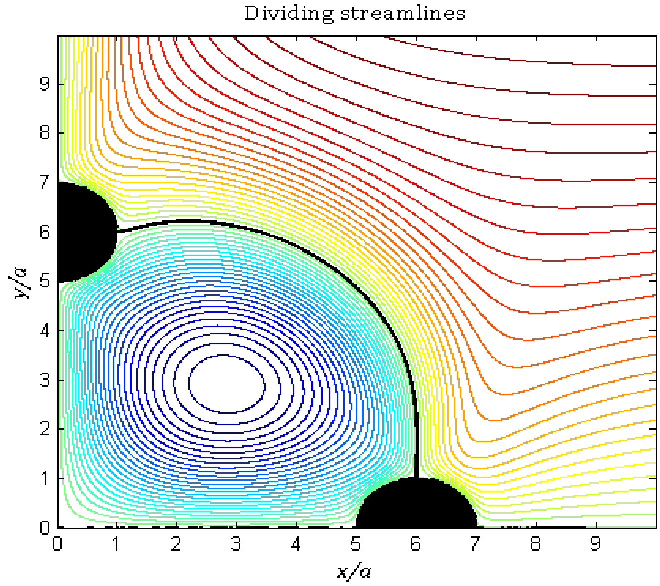


Figure 4.30 Dividing streamlines second modification

Figure 4.30 shows the dividing streamline which separates the fluid coming from the free stream and the fluid that is oscillating in the vortex. If a particle is left in the region below the dividing streamline it will end in the stagnation point in the middle of the vortex. The streamlines represent the movement of a massless particle if forces like inertia or basset are not taking into account. Therefore they do not represent the actual movement of a particle. The real movement of a particle is a spiral trajectory towards the center of the eddie. On the other hand if the particle is left in the region above the dividing streamline, it will not be affected by the vortex and will not be trapped.

4.3.4 Third modification (rx) and results

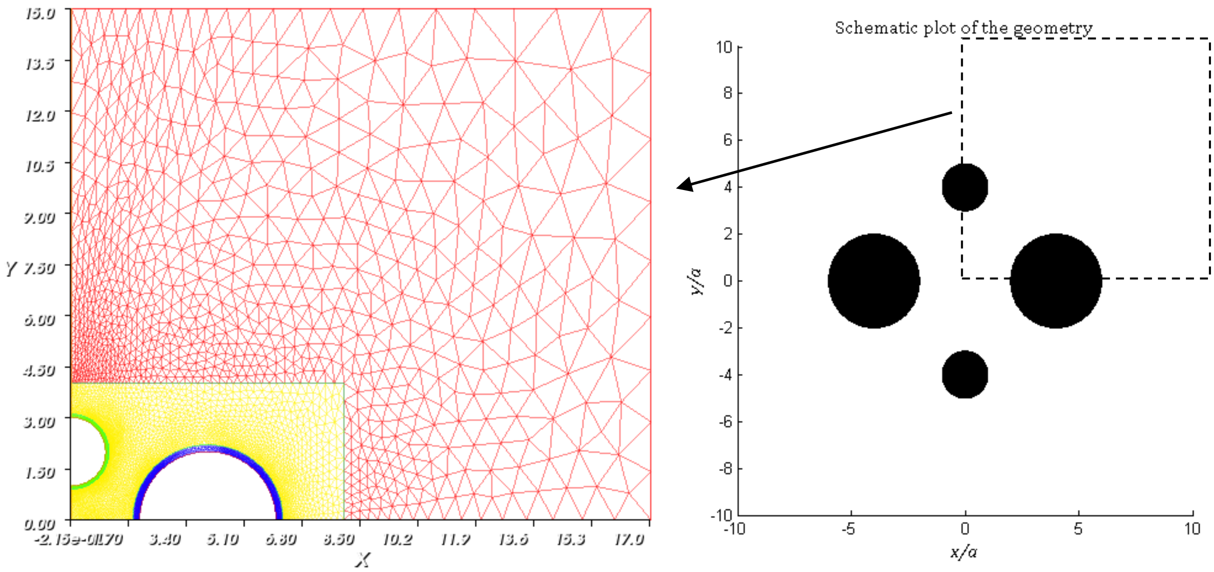


Figure 4.31 Mesh third modification of the third model

In this modified model one of the radius of a cylinder is doubled, the values of the parameters are: $g_{ax} = 8,\ r_x = 2,\ g_{ay} = 4,\ r_y = 1$, where sub index x refers to the cylinder that lays in the x axis and y refers to the cylinder in y axis.

Thanks to this modification the effect that has having one radius bigger than the other can be studied.

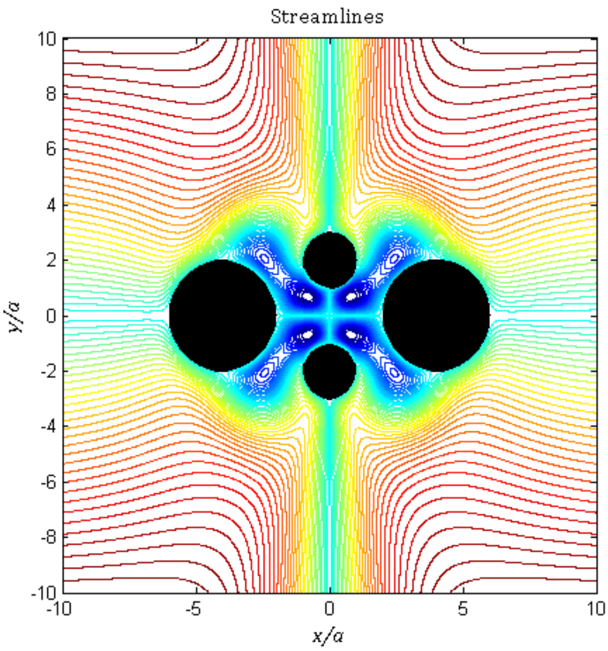


Figure 4.32 Streamlines third modification

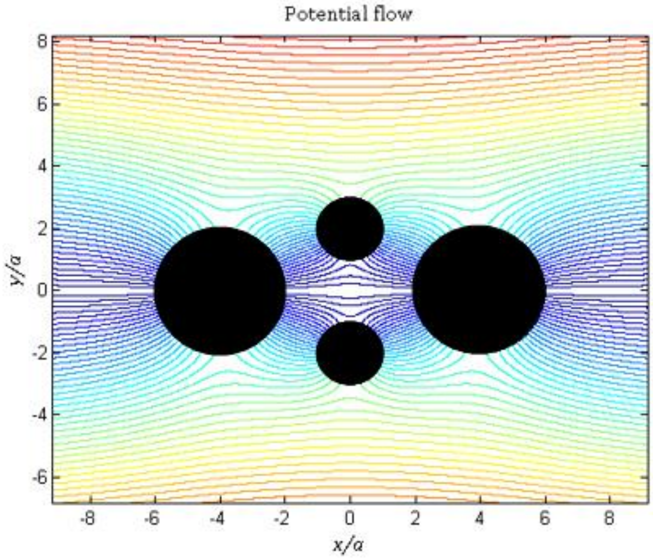


Figure 4.33 Potential flow third modification

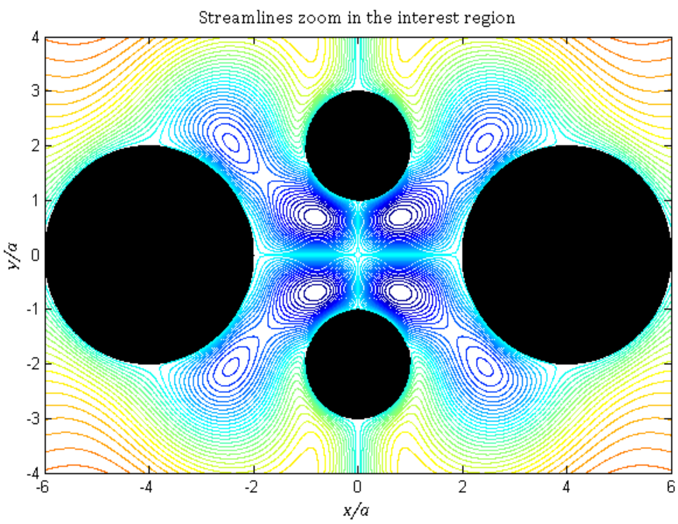


Figure 4.34 Streamlines zoom third modification

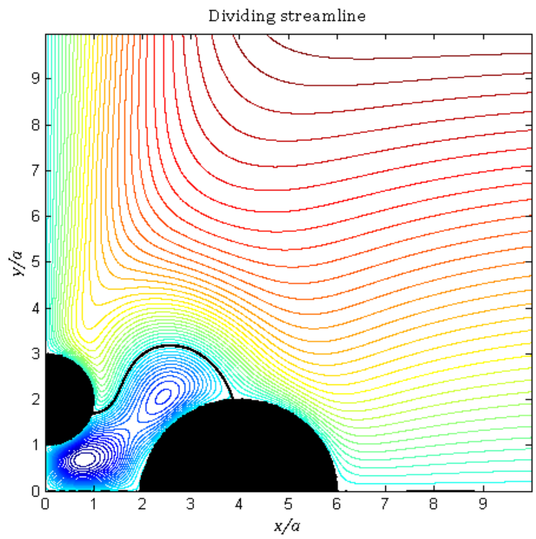


Figure 4.35 Dividing streamline third modification

An interesting thing happens here when the radius of a cylinder is increased, there are formed eight micro eddies. The number of microeddies has been doubled and the size has been reduced due to the increment in the radius in two out of the four cylinders.

This is mainly produced, among other causes, because of the increment in the zone affected by the cylinder positioned in the x-axis. Its influence zone has increased and it is able to affect more flow. It can be seen how the inner vortex is affected by the two cylinders, while the outer vortex is only affected by the bigger cylinder.

Chapter 5

Conclusions and future projects

The realization of the current project through the use of Freefem++ software has demonstrated the validity of the application of finite element method in the analysis of a steady streaming flow. The prediction of the streamlines are accurately obtained by introducing a geometry in the code, and assigning the proper conditions to the borders of this geometry. This method can save time for future experiments in which the resultant streamlines distribution could be unknown for a given steady streaming flow. Results obtained in this numerical procedure can be extrapolated to a real model, since comparing the results obtained in the finite element software with respect to the results obtained from two different experimental tests, both results are considered similar. The time saving is done by the reduction of the need of construction real models and the need to perform tests.

This mathematical model is capable of solving the steady streaming flow produced by an oscillatory flow around any geometry and therefore it could be used as a design tool in this type of hydrodynamic non-contact methods for trapping particles. Design a geometry to trap particles in a determined position or to know before-hand the size of the eddies produced by the steady streaming.

There are two parameters that mainly determine the conditions of the flow, they are the oscillation amplitude and the streaming Reynolds number, in this project the regime where R_s is higher than one is studied in combination with the oscillation amplitude much lower than one, the reasons for this are:

- Trapping forces are directly proportional to R_s.
- Avoid the inner Stokes layer to govern the flow, making this layer to be very thin and concentrating all the vorticity.
- Be able to expand the stream function as follows:

$$\psi(x,\tau) = \psi_0(x,\tau) + \varepsilon\left\{\psi_1^{(u)}(x,\tau) + \psi_1^{(s)}(x)\right\} + \mathcal{O}(\varepsilon^2)$$

  Solving for the leading order term, terms with epsilon are neglected because epsilon is small, at the end, in the leading order term, turns into the Laplace equation: $\nabla^2\breve{\psi}_0 = 0$ thanks to the fact that the unsteady term can be obviated when τ is sufficiently large.

It is how in this way this code solve the flow in an intelligent way, since it solves the most interesting regime, the regime where the steady streaming is produced, thanks to the previous study of the parameters ε and R_s.

As future work for this project would be interesting study this same steady streaming flow with a superposed free stream. It could also be studied the influence that other geometries forms, like diamonds or squares, have in this flow and how are the resultant streamlines. In addition it could be studied the strength of the microeddies formed. It could also be done an improvement in the code to get a higher computation efficiency, or study another regime for the streaming Reynolds number.

Chapter 6

References

[1] Review of cell and particle trapping in microfluidic systems by: J. Nilsson, M. Evander, B. Hammarström and T. Laurell.

[2] Electrokinetic motion of particles and cells in microchannels by Yuejun Kang and Dongqing Li

[3] Optical trapping and manipulation of microscopic particles and biological cells by laser beams by S. Sato and H. Inaba.

[4] Journal of Fluid Mechanics / Volume 10 / Issue 03 / Copyright © 1961 Cambridge University Press

[5] Steady streaming by N. Riley. Annu. Rev. Fluid Mech. 33 (2001)

[6] http://www.cfd-online.com/Wiki/Reynolds_stress_model_(RSM)

[7] Oscillatory flow about a cylinder pair by W. Coenen and N. Riley (2009)

[8] Particle manipulation in viscous streaming by Chong (2013)

[9] Steady streaming around a cylinder by E. Galán-Vicente and W. Coenen (2013)

[10] http://web.mit.edu/2.016/www/handouts/2005Reading4.pdf

[11] freefem++.doc.pdf

[12] http://www.uv.es/~diaz/mn/node20.html

References for pictures

[1] Particle manipulation in viscous streaming by Chong (2013)

[2] Acoustofluidics 13: Analysis of acoustic streaming by perturbation methods

[3] Steady streaming around a cylinder by E. Galán-Vicente and W. Coenen (2013)