NUMERICAL ANALYSIS AND SCIENTIFIC COMPUTING FOR PDEs
AND THEIR CHALLENGING APPLICATIONS
J. Haataja, R. Stenberg, J. Periaux, P. Raback and P. Neittaanmaki (Eds.)

2D/3D TURBINE SIMULATIONS WITH FREEFEM

S. Del Pino
CEA / DAM Ile de france
Département des Sciences de la Simulation et de l'Information
B.P. 12, 91680 Bruyères le Châtel, France

B. Maury
Laboratoire de Mathématiques
Université Paris-Sud
91405 Orsay Cedex, France
http://www.math.u-psud.fr/∼maury

Abstract. The purpose of this document is to illustrate how a new generation of open source "General Purpose" Finite Element Solvers make it possible to solve complex, two- or three-dimensional problems. Olivier Pironneau initiated this tendancy by proposing, in the 80's, its pioneering `freefem` software. We present here how the use of such softwares, in connection with some adapted penalty strategies, can be used to handle the motion of complex bodies in a fluid, by circumventing the difficulty of dynamic mesh generation.

Key words: Finite Element, Method, penalty method, saddle-point formulation, fluid-rigid body interaction, Navier-stokes equations

1 INTRODUCTION

In 1985, Olivier Pironneau published one of the first open source finite element based package: `MacFem` and `PCfem`, written in Pascal. Then with D. Bernardi and C. Prud'homme he developed in C, the first version of `freefem` that exploited the idea of a finite element solver driven by a *user friendly* language and mesh adaptation. Then, with F. Hecht, it was recoded in C++ and distributed as `freefem+` which permitted the use of multiple meshes at once.

Finally, two new versions appeared in the early 21^{th} century: `freefem++` and `freefem3d`. `Freefem++` was developped by F. Hecht in continuation of `freefem+`, and incorporates many kinds of finite elements and new capabilities. `Freefem3d`, developed by the first author, is the first 3D version of the family. All those softwares can be downloaded from [6]. Let us also mention other open source projects dedicated to scientific computing, which have been developped independently by mathematicians, computer scientists, or physicists (see e.g. [7], [8] or [10]).

In this paper, we will show how the use of ficitious domain-like methods (namely the penalty method) enables to perform simulations of complex interactions of a fluid and a rigid body. In particular, we shall consider the problem of a turbine propelled by an imposed flow. We will first present the fluid-rigid body coupling method and then give some numerical results in 2D (`freefem++`) and 3D (`freefem3d`).

2 FLUID/BODY INTERACTION

Computing the motion of rigid or deformable bodies in a viscous fluid is one of the great challenges that numericists face today. If we restrict ourselves to Finite Element approaches, two general classes of approaches dominate the landscape :

(i) Conforming approach : an unstructured mesh covers the domain occupied by the fluid, and follows its motion. In the case of large deformations (systematic when bodies float freely in the fluid), this method calls for frequent and expensive remeshing.

(ii) Eulerian methods : the whole domain is covered by a cartesian mesh, and the presence of the inclusion is taken into account by one of the several methods which have been introduced in the last two decades (see e.g. [4, 9, 11]).

We shall present how the simplest strategy in class *(ii)*, namely the penalty method, can be implemented straightforwardly within `freefem++` in 2 dimensions, or `freefem3d` in 3 dimensions, to compute the motion of inclusions in a fluid. A first approach was proposed in [5], where the motion of a simplified two-dimensional cardiac valve was computed with `freefem++`. The rigid motion constraint was treated by penalizing the strain tensor $\nabla \mathbf{u} + {}^t\nabla \mathbf{u}$ within the inclusion. We propose here a new method, based on a L^2 penalty term. Such a strategy is commonly used (see e.g. [2]) to prescribe a given value (say $\mathbf{U}$) for the velocity within a part of the domain. It consists in adding a penalty term

$$\frac{1}{\varepsilon}\int_{\mathcal{O}} (\mathbf{u} - \mathbf{U}) \cdot \mathbf{v}$$

to the variational formulation (here $\mathbf{v}$ stands for the test-function). Yet, it seems to be restricted to the handling of prescribed velocities. We show here that it can be extended quite naturally to compute the motion of freely (or partially freely) moving bodies.

The approach is based on the following considerations, which we present first in the context of the stationary heat equation. Let us consider a domain Ω, a subdomain $\mathcal{O} \subset\subset \Omega$, and the Poisson problem in $\Omega \setminus \overline{\mathcal{O}}$

$$-\triangle u = f,$$

with homogeneous Dirichlet boundary condition on both $\Gamma = \partial\Omega$ and $\gamma = \partial\mathcal{O}$. We shall denote by $u \in H^1_0(\Omega)$ the solution to that problem extended by 0 in $\mathcal{O}$. Note that u is not expected to belong to $H^2(\Omega)$. The penalized problem reads (the right-hand side f is extended by 0 within $\mathcal{O}$)

$$-\triangle u^\varepsilon + \frac{1}{\varepsilon}\mathbb{1}_{\mathcal{O}} u^\varepsilon = f \quad \text{in } \Omega,$$

where $\mathbb{1}_{\mathcal{O}}$ is the characteristic function of $\mathcal{O}$. The solution u^ε is known to converge to u in $H^1(\Omega)$. Now the exact solution verifies

$$-\triangle u = f - \delta_\gamma \frac{\partial u}{\partial n}$$

in the sense of distribution (or in $H^{-1}(\Omega)$), where n is the inward normal to $\mathcal{O}$, and $\xi = \delta_\gamma \partial u/\partial n$ stands for

$$v \in H_0^1(\Omega) \longmapsto \left\langle \delta_\gamma \frac{\partial u}{\partial n}\,, v \right\rangle = \int_\gamma v \frac{\partial u}{\partial n}.$$

As a direct consequence, $\mathbb{1}_\mathcal{O} u/\varepsilon$ converges (strongly) to ξ in $H^{-1}(\Omega)$. Note that $-\xi$ represents the amount of heat one has to provide to achieve the vanishing of u over $\mathcal{O}$, so that

$$\frac{1}{\varepsilon}\int_\mathcal{O} u^\varepsilon$$

approximates the global heat rate necessary to achieve the objective $u = 0$ in $\mathcal{O}$.

Similarly, consider a viscous fluid in $\Omega \setminus \overline{\mathcal{O}} \subset \mathbb{R}^d$, and assume that the flow follows the incompressible Stokes equations

$$\begin{cases} -\mu\triangle\mathbf{u} + \nabla \mathrm{p} &= \mathbf{f} \quad \text{in } \Omega\setminus\overline{\mathcal{O}} \\ \nabla\cdot\mathbf{u} &= 0 \quad \text{in } \Omega\setminus\overline{\mathcal{O}} \end{cases}$$

with boundary conditions $\mathbf{u} = 0$ on $\Gamma = \partial\Omega$ and $\mathbf{u} = \mathbf{U}$ on $\gamma = \partial\mathcal{O}$. If we suppose that $\mathbf{U}$ is defined all over $\mathcal{O}$, the penalty formulation is

$$-\mu\triangle\mathbf{u}^\varepsilon + \frac{1}{\varepsilon}\mathbb{1}_\mathcal{O}(\mathbf{u}^\varepsilon - \mathbf{U}) + \nabla\mathrm{p}^\varepsilon = \mathbf{f} \text{ in } \Omega,$$

whereas the exact velocity $\mathbf{u}$ verifies (in the sense of distributions)

$$-\mu\triangle\mathbf{u} + \nabla\mathrm{p} = \mathbf{f} - \boldsymbol{\xi} \text{ in } \Omega,$$

where $\boldsymbol{\xi}$ is the force exerted by the fluid onto the body. As previously, the field $\boldsymbol{\xi}^\varepsilon = \frac{1}{\varepsilon}\mathbb{1}_\mathcal{O}(\mathbf{u}^\varepsilon - \mathbf{U})$, which can be identified to a linear functional over $H_0^1(\Omega)^d$ by L^2 duality, converges strongly toward $\boldsymbol{\xi}$ in the dual space $H^{-1}(\Omega)^d$. Note that $\boldsymbol{\xi}^\varepsilon$ is in $L^2(\Omega)^d$, and its restriction to $\mathcal{O}$ is smooth. In particular, the moment of the force about a point $\mathbf{x}_0$ can be approximated by

$$\frac{1}{\varepsilon}\int_\mathcal{O}(\mathbf{x} - \mathbf{x}_0)\times(\mathbf{u}^\varepsilon - \mathbf{U}). \tag{1}$$

3 TWO-DIMENSIONAL TURBINE PROBLEM WITH FREEFEM++

We present here the 2d model. We consider a rectangular domain $\Omega \subset \mathbb{R}^2$ and we denote by $\mathcal{O} \subset\subset \Omega$ the turbine (see Fig. 1). We suppose that $\Omega \setminus \overline{\mathcal{O}}$ is filled with a Newtonian fluid of density ρ and viscosity μ, governed by the Navier-Stokes equations and that $\mathcal{O}$ is occupied by a rigid body which is allowed to rotate without friction around $\mathbf{x}_0$.

The fluid obeys Navier-Stokes equations in $\Omega \setminus \overline{\mathcal{O}} = \Omega \setminus \overline{\mathcal{O}}(t)$ at every time t, and the body motion follows the Newton law, which reduces here to an equation on the angular velocity around $\mathbf{x}_0$. Those equations are coupled by hydrodynamic forces exerted by the fluid on the solid. Finally, viscosity imposes no-slip conditions on the boundary of $\mathcal{O}$: at each point of γ, the velocity on the fluid side is equal to the velocity on the rigid side.

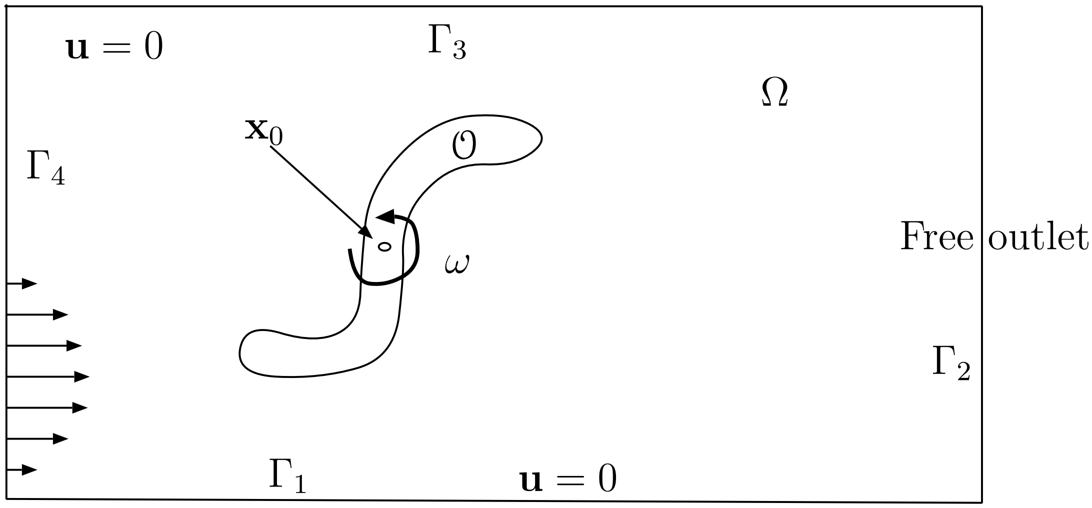

Figure 1: Geometry

Denoting by $\omega = \dot{\theta}$ the angular velocity of $\mathcal{O}$, the problem reads

$$\begin{cases} \rho \dfrac{\mathrm{D}\mathbf{u}}{\mathrm{D}t} - \mu \triangle \mathbf{u} + \nabla \mathrm{p} &= \mathbf{f} \quad \text{in } \Omega \setminus \overline{\mathcal{O}} \\ \nabla \cdot \mathbf{u} &= 0 \quad \text{in } \Omega \setminus \overline{\mathcal{O}} \end{cases} \tag{2}$$

with boundary conditions on Γ

$$\begin{cases} \mathbf{u} &= \mathbf{u}_\Gamma \quad \text{on } \Gamma_i \text{ for } i = 1\,,\ 3 \text{ and } 4, \\ \boldsymbol{\sigma} \cdot \mathbf{n} &= 0 \quad\ \ \text{on } \Gamma_2 \end{cases} \tag{3}$$

and coupling conditions

$$\begin{cases} \mathbf{u}(\mathbf{x}) &= \omega \times (\mathbf{x} - \mathbf{x}_0) \qquad\qquad \text{on } \gamma \\ J \dfrac{d\omega}{dt} &= -\displaystyle\int_\gamma (\mathbf{x} - \mathbf{x}_0) \times \boldsymbol{\sigma} \cdot \mathbf{n} \end{cases} \tag{4}$$

We propose the following time-discretization scheme, which relies on the method of characteristics (see [12]) :

$$\rho \frac{\mathbf{u}^{m+1} - \mathbf{u}^m \circ X}{\delta t} - \mu \triangle \mathbf{u}^{m+1} + \nabla \mathrm{p}^{m+1} = \mathbf{f}\,,\ \nabla \cdot \mathbf{u}^{m+1} = 0 \tag{5}$$

with

$$\mathbf{u}(\mathbf{x}) = \omega^{m+1} \times (\mathbf{x} - \mathbf{x}_0) \text{ on } \gamma \tag{6}$$

$$J \frac{\omega^{m+1} - \omega^m}{\delta t} = -\int_\gamma (\mathbf{x} - \mathbf{x}_0) \times \boldsymbol{\sigma} \cdot \mathbf{n} \tag{7}$$

The approach we propose is based on the two points

1. Given some $\omega \in \mathbb{R}$, assume that we are able to compute the solution $\mathbf{u}_\omega$ to (5) with the Dirichlet boundary condition (6) : $\mathbf{u}(\mathbf{x}) = \omega \times (\mathbf{x} - \mathbf{x}_0)$ on γ, and

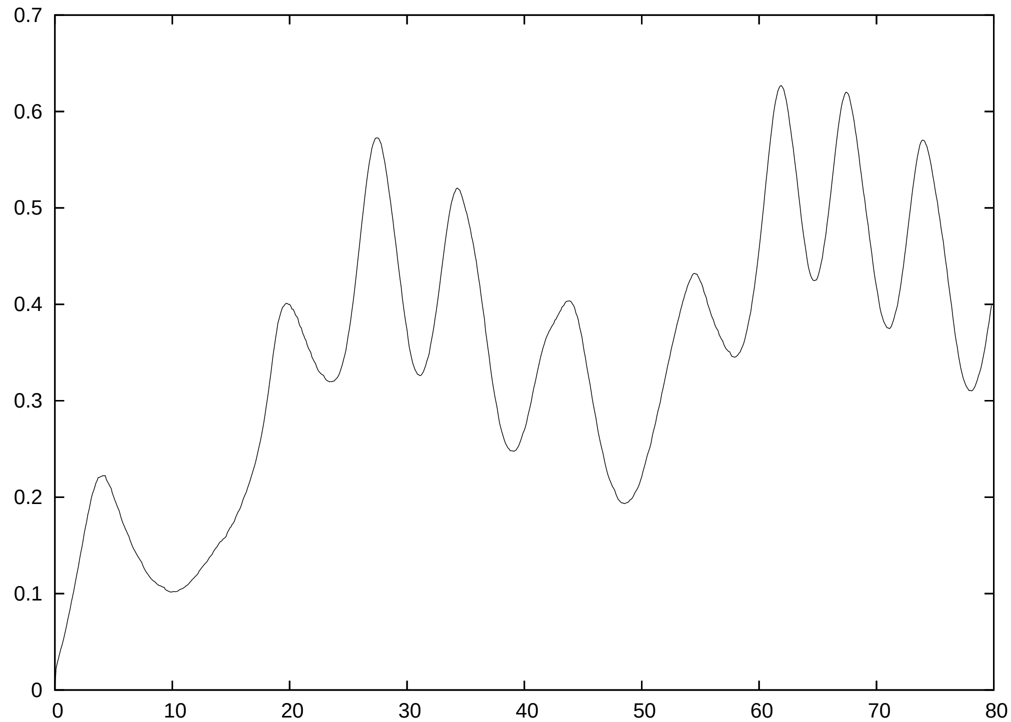

Figure 2: $t \longmapsto \omega(t)$

assume furthermore that the computation provides the associated momemt exerted by the fluid on the turbine :

$$M_\omega = -\int_\gamma (\mathbf{x} - \mathbf{x}_0) \times \boldsymbol{\sigma}_\omega \cdot \mathbf{n}.$$

Then ω^{m+1} is uniquely defined as the only ω such that $J(\omega - \omega^m)/\delta t = M_\omega$. As the mapping $\omega \mapsto M_\omega$ is affine, it amounts to compute M_0 and M_1, which leads to the solution

$$\omega^{m+1} = \left(\omega^m + \frac{\delta t}{J} M_0\right) / \left(1 + \frac{\delta t}{J}(M_0 - M_1)\right)$$

2. The solution $\mathbf{u}_\omega$ (for $\omega = 0$ and $\omega = 1$) is approximated by penalty. Denoting by $\mathbf{U}_\omega$ the rigid velocity on $\mathcal{O}$ associated to angular velocity ω, we compute $(\mathbf{u}^\varepsilon_\omega, \mathrm{p}^\varepsilon_\omega)$ as the solution of

$$(1 - \mathbb{1}_\mathcal{O})\rho \frac{\mathbf{u}^\varepsilon_\omega - \mathbf{u}^m \circ X}{\delta t} - \mu \triangle \mathbf{u}^\varepsilon_\omega + \frac{1}{\varepsilon} \mathbb{1}_\mathcal{O} \left(\mathbf{u}^\varepsilon_\omega - \mathbf{U}_\omega\right) + \nabla \mathrm{p}^\varepsilon_\omega = 0, \qquad (8)$$

with the divergence-free condition on $\mathbf{u}^\varepsilon_\omega$. The moment M_ω is then computed by means of (1).

The space-discretization is based on the so-called mini-element (also called bubble element), and it lies on a fixed cartesian mesh (the penalty approach makes it possible to use non-conforming meshes).

The rectangular domain is $(-1.4, 2.6) \times (-1, 1)$, and the turbine rotates about the origin. The cartesian mesh is 100×50. The physical and numerical parameters are

$$\rho = 1\,,\ \rho_s = 10\,,\ \mu = 2.10^{-2}\,,\ \delta t = 0.1\,,$$

The inlet velocity is a parabolic profile supported by the lower half of the left-hand boundary, with maximal velocity set to 1. Fig. 2 represents the computed angular velocity with respect to the time (the helix starts from rest). Note that the motion is not periodic, even after 9 revolutions. Fig. 3 represents the streamlines at consecutive times. The eight figures cover about half a revolution.

4 THREE-DIMENSIONAL TURBINE PROBLEM WITH `FREEFEM3D`

The formulation of the three dimensional problem is quite similar. The only difference lies in the fact that the moment of the hydrodynamic forces is no longer about a point, but about an axis. We shall not go deep into details concerning the technical problems raised by the three-dimensional computations. Let us simply say here that, in order to alleviate the computational cost of those computations, we integrate the penalty strategy onto a projection algorithm (see [1]) to solve the non-stationary Navier-Stokes equations. The mesh is cartesian (31 nodes in each direction). The space-discretization is based on the so-called Q^1iso-Q^2/Q^1. The helix rotates about the x-axis, its half size is 1.3, and the fluid domain is $(-2,2)^3$. The inlet velocity is uniform (speed 1). The physical and numerical parameters are

$$\rho = 1\,,\ \rho_s = 1\,,\ \mu = 0.1\,,\ \delta t = 0.1.$$

Fig. 4 represents the velocity field in a cross section and some streamlines of the flow at time $t = 20$. Note that, in the top part of the figure, the velocity comes from the back, and the turbine rotates clockwise.

5 Conclusion

We presented a new strategy to compute the motion of rigid bodies in a fluid, based on simple principles. As it involves some numerical parameters which have to be tuned up properly, it calls for proper validations and comparaison with more sophisticated methods. Yet, it illustrates how complex phenomena can be simulated straightforwardly with the use of the new generation of General Purpose Finite Element Solvers which Olivier Pironneau initiated.

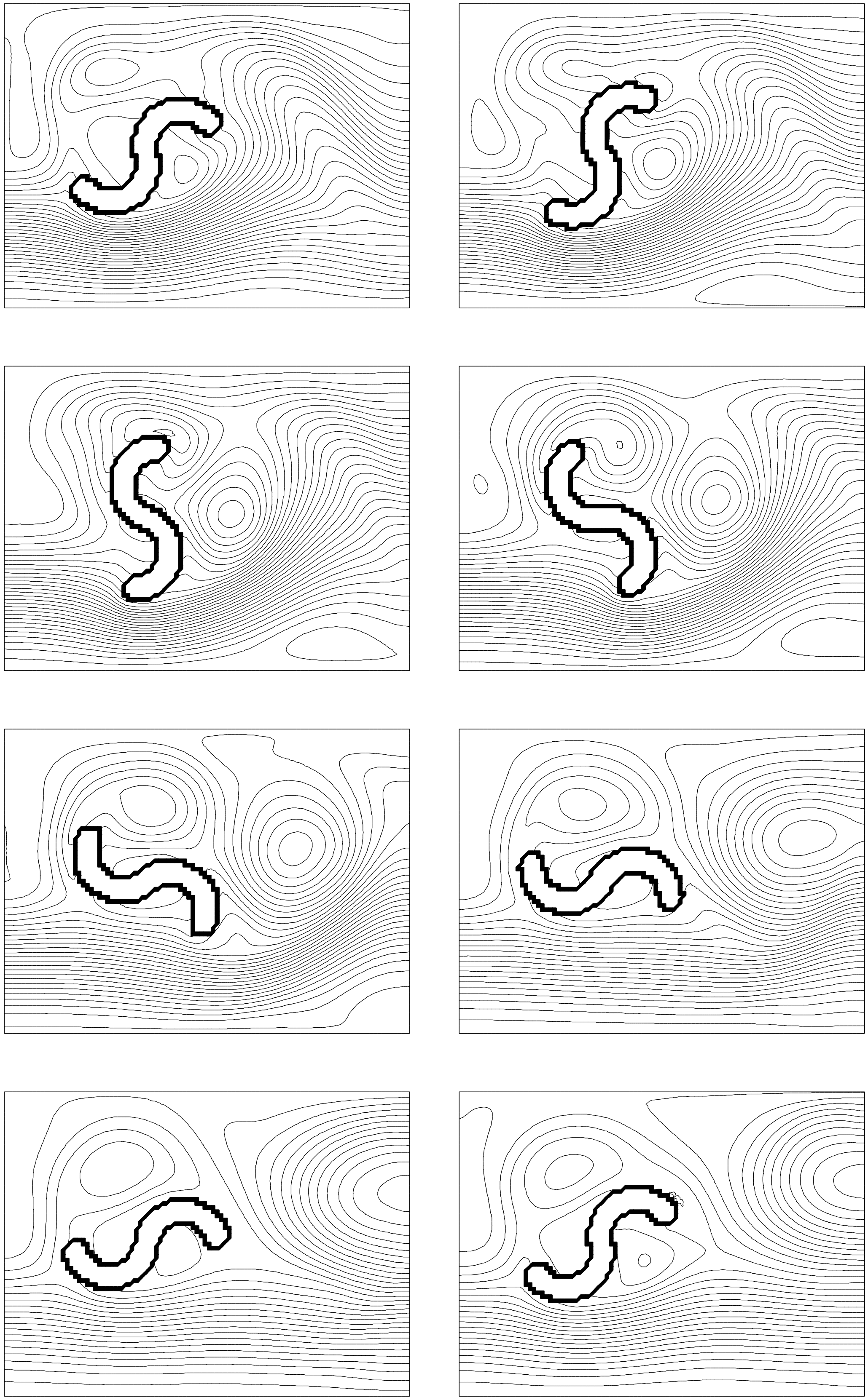

Figure 3: Streamlines of the velocity fields at times 25, 26, 27, 28, 29, 30, 31, and 32.

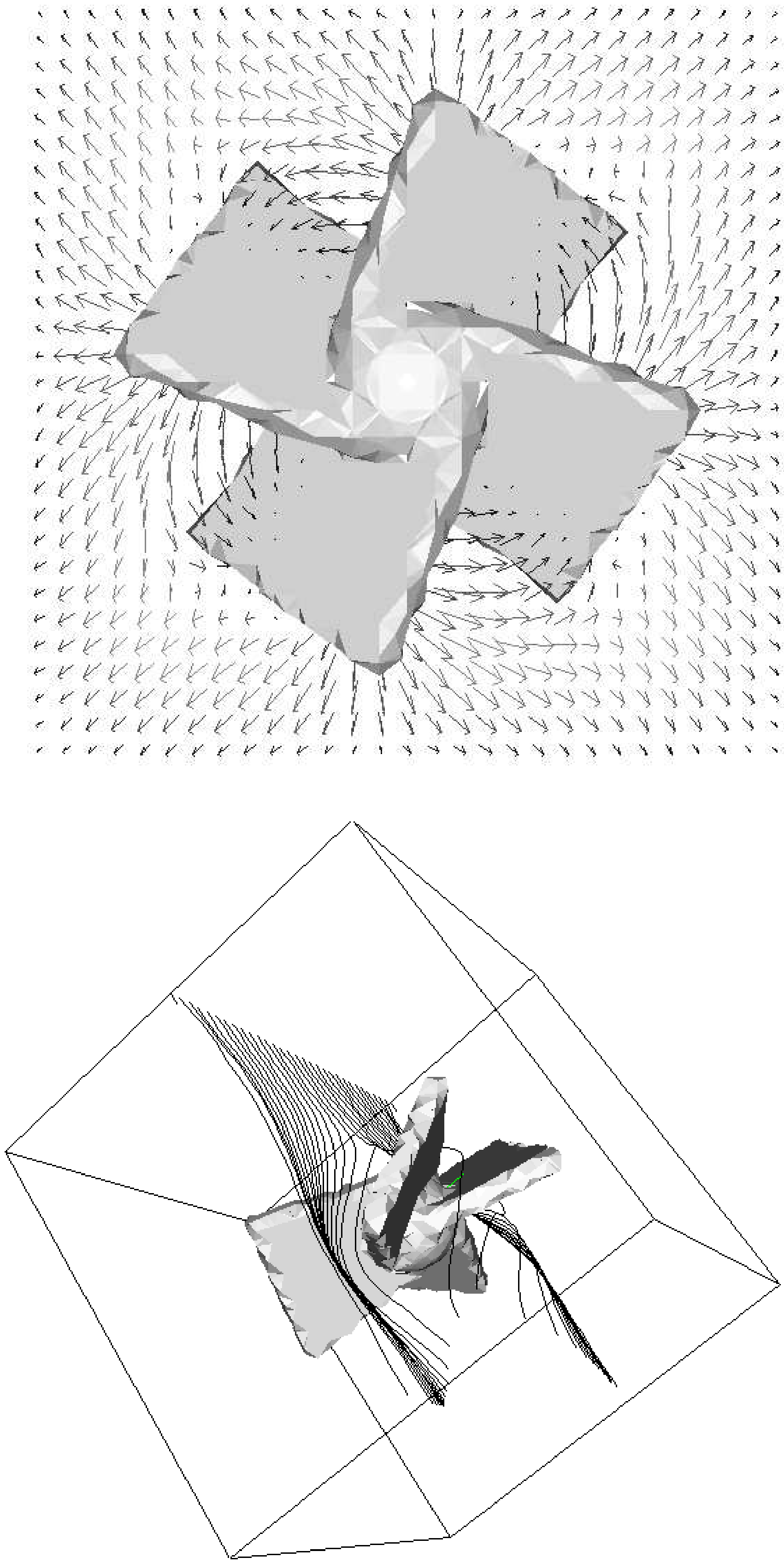

Figure 4: velocity field and streamlines for the three dimensional turbine

REFERENCES

[1] Y. Achdou, J.L. Guermond, *Convergence analysis of a finite element projection/Lagrange-Galerkin method for the incompressible Navier-Stokes equations*, SIAM J. Numer. Anal. 37 (2000), no. 3, 799–826.

[2] Angot, I. Ramière, P. Angot, M. Belliard *A fictitious domain approach with spread interface for elliptic problems with general boundary conditions.* Comput. Methods Appl. Mech. Engrg. 196 (2007), no. 4-6, 766–781.

[3] Ivo Babuška, *The finite element method with penalty*, Mathematics of Computation **27**, 1973, p.p. 221–228.

[4] R. Glowinski, T.-W. Pan, T.I. Hesla, and D.D. Joseph, *A distributed Lagrange Multiplier/fictitious domain method for particulate flows,* Int. J. of Multiphase Flow, 25 (1999), pp. 755.

[5] J. Janela, A. Lefebvre, B. Maury, A penalty method for the simulation of fluid-rigid body interaction, ESAIM Proceedings, september 2005, vol. 14, 201-212, Eric Cancès & Jean-Frédéric Gerbeau, Editors.

[6] `http://www.freefem.org`

[7] `http://www.geuz.org/getdp/`

[8] S. Labbé, J. Laminie, and V. Louvet, *Méthodologie et environnement de développement orientés objets : de l'analyse mathématique à la programmation*, Matapli No. 70 (2003).

[9] B. Maury, *A Fat Boundary Method for the Poisson Equation in a Domain with Holes*, J. of Sci. Computing, 16 (2001), no. 3, pp. 319-339.

[10] P. Saramito, `http://ljk.imag.fr/membres/Pierre.Saramito/rheolef`

[11] C. S Peskin and D. M. McQueen, *A three-dimensional computational method for blood flow in the heart I. Immersed elastic fibers in a viscous incompressible fluid,* J. of Comp. Physics, 81 (1989), pp. 372–405.

[12] O. Pironneau, J. Liou, T. Tezduyar, *Characteristic–Galerkin and Galerkin Least Squares Space–Time Formulations for the Advection–Diffusion Equation with Time–Dependent Domains*, Computer Method in Applied Mechanics and Engineering **100**, 117–141, 1992.