INRIA

INSTITUT NATIONAL DE RECHERCHE EN INFORMATIQUE ET EN AUTOMATIQUE

Team reo

Numerical simulation of biological flows

Rocquencourt

THEME BIO

Table of contents

1. Team

REO is a joint project of the INRIA Research Unit of Rocquencourt and the Jacques-Louis Lions Laboratory (LJLL) of the Pierre et Marie Curie (Paris 6) University.

Head of project-team
Marc Thiriet [CR1 CNRS]

Co-head of project-team
Jean-Frédéric Gerbeau [CR1 INRIA]

Administrative assistant
Maryse Desnous [TR]

Staff member Inria
Miguel Ángel Fernández [CR2 INRIA]
Céline Grandmont [Maître de conférences (on secondment)]

Staff member Paris 6
Laurent Boudin [Maître de conférences]
Laurent Dumas [Maître de conférences]

Research scientist (partner)
Adel Blouza [Maître de conférences, Rouen University]
Bertrand Maury [Professor, Paris 11 University]
Yvon Maday [Professor, Paris 6 university]

Post-doctoral fellow
Paola Causin [post-doc RTN Haemodel]

Ph. D. student
Nuno Diniz Dos Santos [PhD position INRIA / RTN Haemodel]
Laurence Vial [PhD CIFRE Air Liquide]

2. Overall Objectives

REO is a joint project of the INRIA Research Unit of Rocquencourt and the Jacques-Louis Lions Laboratory (LJLL) of the Pierre and Marie Curie (Paris 6) University. Its research activities are aimed at

- modeling the flow of biological fluids, more especially blood in large vessels and air in the respiratory tracts, both in normal and pathological states;
- developing and analyzing efficient, robust and reliable numerical methods for the simulation of such flows;
- developing simulation software to guide medical decision and to design more efficient medical devices.

3. Scientific Foundations

3.1. Multiphysics modeling

Keywords: *fluid-structure interaction, spray modelling*.

In large vessel and in large bronchi, blood and air flows are generally supposed to be governed by the incompressible Navier-Stokes equations. Indeed in large arteries, blood can be supposed to be Newtonian, and at rest air can be modeled as an incompressible fluid. The cornerstone of the simulations is therefore the Navier-Stokes solver. But other physical features have also to be taken into account in simulations of biological

flows, in particular fluid-structure interaction in large vessels and transport of sprays, particules or chemical species.

3.1.1. Fluid-structure interaction

Fluid-structure coupling occurs both in the respiratory and in the circulatory systems. We focus mainly on blood flows since our work is more advanced in this field. But the methods developped for blood flows could be also applied to the respiratory system.

Here "fluid-structure interaction" means a coupling between the 3D Navier-Stokes equations and a 3D (possibly thin) structure in large displacement.

The numerical simulations of the interaction between the artery wall and the blood flows raise many issues: (1) the displacement of the wall cannot be supposed to be infinitesimal, geometrical nonlinearities are therefore present in the structure and the fluid problem have to be solved on a moving domain (2) the densities of the artery walls and the blood being close, the coupling is strong and has to be tackled very carefully (implicitly) to avoid numerical instabilities, (3) "naive" boundary conditions on the artificial boundaries induce spurious reflection phenomena.

Simulation of valves, either at the outflow of the cardiac chambers or in veins, is another example of difficult fluid-structure problems arising in blood flows. In addition to the above mentionned difficulties, here we have to deal with very large displacements and changes of topology (contact problems).

Because of the above mentioned difficulties, the interaction between the blood flow and the artery wall has often been neglected in most of the classical studies. The numerical properties of the fuid-structure coupling in blood flows are rather different from other classical fluid-structure problems. In particular, due to stability reasons it seems impossible to succesfully apply the explicit coupling schemes used in aeroelasticity.

As a result, fluid-structure interaction in biological flows raise new challenging issues in scientific computing and numerical analysis : new schemes have to be developped and analysed.

3.1.2. Aerosol

Complex two-phase fluids can be modeled in many different ways. Eulerian models describe both phases by physical quantities such as the density, velocity or energy of each phase. In the mixed fluid-kinetic models, the diphasic fluid has one dispersed phase, which is constituted by a spray of droplets, with a possibly variable size, and a continuous classical fluid.

This type of model was first introduced by Williams [41] in the frame of combustion. It was later used to develop the Kiva code [34] at the Los Alamos National Laboratory, or the Hesione code [37], for example. It has a wide range of applications, besides the nuclear setting: diesel engines, rocket engines [35], therapeutic sprays, *etc*. One of the interests of such a modeling is that various phenomena on the droplets can be taken into account with an accurate precision: collision, breakups, coagulation, vaporization, chemical reactions, *etc*., at the level of the droplets.

The model usually consists in coupling a kinetic equation, that describes the spray through a probability density function, and classical fluid equations (typically Navier-Stokes). The numerical solution of this system relies on the coupling of a method for the fluid equations (for instance, a finite volume method) with a method fitted to the spray (particle method, Monte Carlo).

We are mainly interested in modeling therapic sprays either for local or general treatments. The study of the underlying kinetic equations should lead us to a global model of the ambient fluid and the droplets, with some mathematical significance. Well-chosen numerical methods can give some tracks on the solutions behavior and help to fit the physical parameters which appear in the models.

3.2. Multiscale modeling

Multiscale modeling is a necessary step for blood and respiratory flows. In this section, we focus on blood flows. Nevertheless, preliminary investigations are currently carried out in our team on respiratory flows.

3.2.1. Arterial tree modelling

Problems arising in the numerical modeling of the human cardiovascular system often require an accurate description of the flow in a specific sensible subregion (carotid bifurcation, stented artery, *etc.*). The description of such local phenomena is better addressed by means of three-dimensional (3D) simulations, based on the numerical approximation of the incompressible Navier-Stokes equations, possibly accounting for compliant (moving) boundaries. These simulations require the specification of boundary data on artificial boundaries that have to be introduced to delimit the vascular district under study. The definition of such boundary conditions is critical and, in fact, influenced by the global systemic dynamics. Whenever the boundary data is not available from accurate measurements, a proper boundary condition requires a mathematical description of the action of the reminder of the circulatory system on the local district. From the computational point of view, it is not affordable to describe the whole circulatory system keeping the same level of detail. Therefore, this mathematical description relies on simpler models, leading to the concept of *geometrical multiscale* modeling of the circulation [38]. The underlying idea consists in coupling different models (3D, 1D or 0D) with a decreasing level of accuracy, which is compensated by their decreasing level of computational complexity.

The research on this topic aims at providing a correct methodology and a mathematical and numerical framework for the simulation of blood flow in the whole cardiovascular system by means of a geometric multiscale approach. In particular, one of the main issues will be the definition of stable coupling strategies between 3D and 1D models that generalizes the work reported in [36] to general geometries coming from medical imaging.

When modeling the arterial tree, a standard way consists in imposing a pressure or a flow rate at the inlet of the aorta, *i.e.* at the network entry. This strategy does not allow to describe important features as the overload in the heart caused by backward travelling waves. Indeed imposing a boundary condition at the begining of the aorta artificially disturbs physiological pressure waves going from the arterial tree to the heart. The only way to catch this physiological behavior is to couple the arteries with a model of heart, or at least a model of left ventricle.

A constitutive law for the myocardium, controlled by an electrical command, has been recently developped in the ICEMA project [40]. One of our objectives is to couple artery models with this heart model.

A long term goal is to achieve 3D simulations of a system including heart and arteries. One of the difficulties of this very challenging task is to simulate the aortic valve. To this purpose, we plan to mix arbitrary Lagrangian Eulerian and immersed boundary or fictitious domain approaches.

3.2.2. Respiratory tract modelling

Work is in progress to develop a multiscale modelling of the respiratory tract. Intraprenchymal airways distal from generation 7 of the tracheabronchial tree (TBT), which cannot be visualized by common medical imaging techniques, are modelled either by a single simple model or by a model set according to their order in TBT. The single model is based on straight pipe fully developed flow (Poiseuille flow in steady regimes) with given alveolar pressure at the end of each compertment. It will provide boundary conditions at the bronchial ends of 3D TBT reconstructed from imaging data. The model set includes three serial models. The generation down to the pulmonary lobule will be modelled by reduced basis elements. The lobular airways will be represented by a fractal homogenization approach. The alveoli, which are the gas exchange loci between blood and inhaled air, inflating during inspiration and deflating during expiration, will be described by multiphysics homogenization.

4. Application Domains

4.1. Blood flows

Keywords: *blood flows.*

Cardiovascular diseases like atherosclerosis or aneurisms are a major cause of mortality. It is generally admitted that a better knowledge of local flow patterns could improve the treatment of these pathologies (although many other biophysical phenomena obviously take place in the developpement of such diseases). In particular, it has been known for years that the association of low wall shear stress and high oscillatory shear index give relevant indications to localize possible zones of atherosclerosis. It is also known that medical devices (graft or stent) perturbate blood flows and may create local stresses favourable with atherogenesis. Numerical simulations of blood flows can give access to this local quantities and may therefore help to design new medical devices with less negative impacts. In the case of aneurisms, numerical simulations may help to predict possible zones of rupture and could therefore give a guide for treatment planing.

In clinical routine, many indices are used for diagnosis. For example, the size of a stenosis is estimated by a few measures of flow rate around the stenosis and by application of simple fluid mechanics rules. In some situations, for example in the case a sub-valvular stenosis, it is known that such indices often give false estimations. Numerical simulations may give indications to define new indices, simple enough to be used in clinical exams, but more precise than those currently used.

It is well-known that the arterial circulation and the heart (or more specifically the left ventricle) are strongly coupled. Modifications of arterial walls or blood flows may indeed affect the mechanical properties of the left ventricle. Numerical simulations of the arterial tree coupled to the heart model could shed light on this complex relationship.

One of the goals of the REO team is to provide various models and simulation tools of the cardiovascular system. The scaling of these models will be adapted to the application in mind: low resolution for modeling the global circulation, high resolution for modeling a small portion of vessel.

4.2. Respiratory tracts

Keywords: *lungs modelling, respiration.*

Breathing, or “external” respiration (“internal” respiration corresponds to cellular respiration) involves gas transport though the respiratory tract with its visible ends, nose and mouth. Air streams then from the pharynx down to the trachea. Food and drink entry into the trachea is usually prevented by the larynx structure (epiglottis). The trachea extends from the neck into the thorax, where it divides into right and left main bronchi, which enter the corresponding lungs (the left being smaller to accommodate the heart). Inhaled air is then convected in the bronchus tree wich ends in alveoli, where gaseous exchange occurs. Surfactant reduces the surface tension on the alveolus wall, allowing them to expand. Gaseous exchange relies on simple diffusion on a large surface area over a short path between the alveolus and the blood capillary under concentration gradients between alveolar air and blood. The lungs are divided into lobes (three on the right, two on the left) supplied by lobar bronchi. Each lobe of the lung is further divided into segments (ten segments of the right lung and eight of the left). Inhaled air contains dust and debris, which must be filtered, if possible, before they reach the alveoli. The tracheobronchial tree is lined by a layer of sticky mucus, secreted by the epithelium. Particles which hit the side wall of the tract are trapped in this mucus. Cilia on the epithelial cells move the mucous continually towards the nose and mouth.

Each lung is enclosed in a space bounded below by the diaphragm and laterally by the chest wall and the mediastinum. The air movement is achieved by alternately increasing and decreasing the chest pressure (and volume). When the airspace transmural pressure rises, air is sucked in. When it decreases, airspaces collapse and air is expelled. Each lung is surrounded by a pleural cavity, except at its hilum where the inner pleura give birth to the outer pleura. The pleural layers slide over each other. The tidal volume is nearly equal to 500 ml.

The lungs may fail to maintain an adequate supply of air. In premature infants surfactant is not yet active. Accidental inhalation of liquid or solid and airway infection may occur. Chronic obstructive lung diseases and lung cancers are frequent pathologies and among the three first death causes in France.

One of the goals of REO team in the ventilation field, in the framework of "R-MOD" (RNTS 2001) and of "le-poumon-vous-dis-je" (ACI Nouvelles Interfaces des Mathématiques, 2003), is to visualize the airways (virtual endoscopy) and simulate flow in image-based 3D models of the upper airways (nose, pharynx, larynx) and the first generations of the tracheobronchial tree (trachea is generation 0), whereas simple models of the small bronchi and alveoli are used (reduced-basis element method, fractal homogenization, multiphysics homogenization, lumped parameter models), in order to provide the flow distribution within the lung segments.

5. Software

5.1. LiFE-V library

Keywords: *Finite element library*.

Participants: Jean-Frédéric Gerbeau [correspondant], Miguel Ángel Fernández.

LiFE-V[1] is a finite element library providing implementations of state of the art mathematical and numerical methods. It serves both as a research and production library. It has been used already in medical and industrial context to simulate fluid structure interaction and mass transport. LiFE-V is the joint collaboration between three institutions: Ecole Polytechnique Fédérale de Lausanne (CMCS) in Switzerland, Politecnico di Milano (MOX) in Italy and INRIA (REO) in France. It is a free software under LGPL license.

6. New Results

6.1. Mathematical modelling and numerical methods in fluid dynamics

Participants: Paola Causin, Nuno Diniz dos Santos, Miguel Ángel Fernández, Jean-Frédéric Gerbeau, Céline Grandmont, Yvon Maday.

6.1.1. Existence results in fluid-structure interaction

In [13] and [31] we study $2D$ or $3D$ viscous incompressible fluid governed by the Navier–Stokes equations, interacting with elastic plate or beam located on one part of the fluid boundary. These models can be viewed as a first attempt to described the evolution of the blood flow past arteries. In [13] existence of weak solutions is proven for a three dimensional fluid interacting with a "viscous" plate in flexion. From the mechanical point of view, adding a viscous term is a way to introduce dissipation in the beam model and from the mathematical point of view, this is a way to regularize the structure velocity. In [31] we show that for a $2D$–fluid coupled with a beam one can pass to the limit as the coefficient modeling the viscoelasticity (resp. the rotatory inertia) of the beam tends to zero. In this case the dissipation coming from the fluid enables us to control the high frequencies of the structure velocity. As a consequence, we obtain the existence of at least one weak solution for the limit problem (Navier–Stokes equation coupled with Euler–Bernoulli equation) as long as the beam does not touch the bottom of the fluid cavity. In [32], we present and analyse a dynamical geometrically nonlinear formulation that models the motion of two–dimensional and three–dimensional elastic structures in large displacements–small strains. In a first part we derive the equations describing the motion of the body. In a second part, existence of a weak solution is proven using a Galerkin method. We also prove that the solution is unique. Those type of models can be used for instance to described the motion of red blood cells.

6.1.2. Numerical methods in fluid-structure interaction

This activity is done in close collaboration with the MACS project, in particular with Marina Vidrascu.

[1] http://www.lifev.org/

We have proposed in [6] a numerical method to solve the coupling between the Navier-Stokes equations on moving domains (ALE formulation) and a shell model in large displacements. The core of the algorithm is a Newton-Krylov method based on a reduced model which offers a significative gain in robustness and efficiency compared to standard methods commonly used in this field. In [29], we propose a nested preconditionner of GMRES and acceleration techniques which improve the efficiency of Newton-Krylov methods. The basic idea is to use previously computed Krylov basis in order to build a cheap and efficient preconditionner for subsequent problems.

In [11], we give, on a simplified model, a theoretical explanation of several empirical facts observed in the simulation of blood flows in compliant vessels. In particular, we show that under certain choices of the physical parameters, typically when the densities of the fluid and of the structure are close or when the domain is a slender geometry, loosely coupled schemes are unstable irrespectively of the time step.

6.1.2.1. Work in progress

We are currently investigating numerical methods, in particular based on ALE formulation and fictitious domain method, to simulate the behaviour of biological valves.

6.1.3. Stabilized finite element methods in fluid mechanics

We first addressed the problem from the classical point of view, namely, by stabilizing both the velocities and the pressure using a residual based stabilization (SUPG/SDEFM). Hence our method consists in subtracting a mesh dependent term (including the equation residual) from the formulation without compromising consistency. Our contribution relies on the fact that the design of mesh dependent term, as well as the stabilization parameter involved, are suggested by bubble condensation. As a result, no free constants have to be set. Stability was proved for any combination of velocity and pressure spaces, under the hypotheses of continuity for the pressure space. Optimal order error estimates were derived for the velocity and the pressure. Numerical experiments in 2D confirmed these theoretical results [25]. This work was carried out in collaboration with G. Barrenechea and C. Vidal.

Although the previous approach gives good results in practice, it has several undesirable features. Among others, artificial boundary conditions and artificial pressure-velocity couplings are introduced, which (in particular) makes time stepping awkward. To overcome these disadvantages we have recently introduced a new method based on the addition of gradient jumps to the discrete formulation. Stability is obtained in a unified fashion without introducing pressure velocity couplings or additional unknowns. The method has been analyzed and tested with very promising results in 3D [27]. This work was carried out in collaboration with E. Burman and P. Hansbo.

6.2. Respiration tree modelling

Participants: Laurent Boudin, Céline Grandmont, Yvon Maday, Bertrand Maury, Marc Thiriet.

6.2.1. Modelling

In [24], our interest is, starting from a simple, rather naive, model of the acini, to show how an upstream model of the respiration tree can be hooked up and result in a well posed coupled system that will allow for simulations.

6.2.1.1. Work in progress

In colaboration with N. Meunier, we are looking at a simplified model of the respiration tree. This work has been initiated at CEMRACS 2004. In a first part we assume that we have a viscous fluid which flows through a tree connected pipes, each of which being characterized by its resistance. We establish the relation between pressures and fluxes at the outlets and investigate the convergence of this operator as the height of the tree goes to infinity. In a second part we couple the tree model with a mass–spring chain and study the convergence of it as the number of outlets (i. e. the height of the tree) tends to infinity. We obtain a wave equation with an additional non local dissipative term This term represents the influence of the tree. In collaboration with Yves Capdeboscq we are studying, in order to obtain a simplified model of the acini, the homogenization of

an incompressible elastic structure with holes. These holes are filled with an incompressible fluid that can be evacuated through possibly connected pipes. In the same spirit, we are looking at the homogenization of an incompressible elastic structure containing small compressible gaz inclusions.

6.2.2. Modeling of biosprays for upper airways

A work initiated during the CEMRACS 2004 [30] is the first step of the study of sprays in the upper airways. In the following kinetic equation, which is similar to the one in [39]

$$\partial_t f + v \cdot \nabla_x f + \nabla \cdot (f(u - v)) + \partial_r(\chi f) = \Delta_v f,$$

the probability density function of droplets f depends on time t, position x, velocity v, radius r. The quantity χ describes the evolution of the radius of the droplets, and u denotes the velocity field of the ambient air. This equation is coupled with standard Navier-Stokes equations.

Our work consists in investigating the behaviour of the 2D numerical solution of this fluid-kinetic problem, which is solved with a finite volume method for the fluid part and a particle method for the spray. For the moment, the throat is roughly modelled as a set of two rectangles (horizontal and vertical), the injected droplets have high velocities, the inhalated sprays are almost motionless. The first numerical tests will be presented in [30]. In the future, the geometrical description of the throat will be improved.

6.2.3. Airway flow

The flow in the proximal airways has been studied numerically on image-derived computational domains. Ventilation distribution has been studied at different phases of the respiratory cycle (mid decelerating phase of inspiration, mid accelerating phase of expiration, peak expiration, mid decelerating phase of expiration, mid accelerating phase of inspiration, and peak inspiration). But the results of ventilation distribution are questionable because the boundary conditions remains nowadays non-suitable and modelling is in progress to take into account adjoining parts of the pipe network. An example of fluid particle trajectories are plotted at selected phases of the inspiration in Fig. 1.

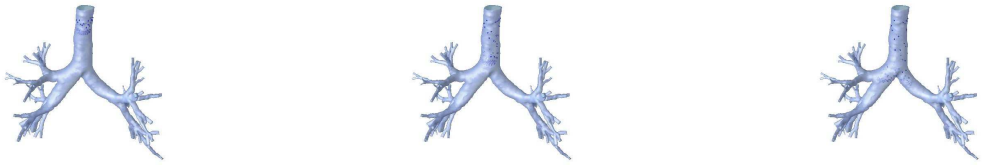

Figure 1. Particle paths during inspiration.

Surprisingly, between mesh comparisons, using meshes of 173788, 521468 and 1340723 elements, did not exhibit large variations in the flow fields. On an image-based model of upper airways, mass conservation and cycle reproducibility were checked. Our initialisation procedure by solving a Stokes problem did not disturb too much the flow results in the tracheobronchial tree. However, less good reproducibility is observed at flow inversion phases.

Results have been compared using either different meshes and fluid solvers (GAMBIT and FIDAP or INRIA softwares YAMS, GHS3D and NSI3IFS). We did not observed great discrepencies between the tests.

6.3. Blood flows

Participants: Adel Blouza, Laurent Dumas, Jean-Frédéric Gerbeau, Marc Thiriet.

6.3.1. Fluid-structure interaction

In [18], we have proposed a practical method to apply our new fluid-structure algorithms on geometries coming from medical imaging, in particular cerebral aneurisms and carotid bifurcation. Figure 2 shows for example the propagation of a pressure wave in an aneurism.

Figure 2. Propagation of a pressure wave in a cerebral aneurism (fluid-structure interaction).

6.3.2. Stent optimization

In order to reduce the effects of atherosceloris, a vascular disesase that partially obturate an artery, the deployment of a device called stent by a catheter technique is now commonly used. Unfortunately, a re-stenosis is observed, in some cases, few mounths after the operation. In order to reduce this failure, the optimization of the shape of the stent is investigated on two aspects: structural and fluid. On the first aspect, the minimization can deal with the mass, the mechanical stresses and/or the dissipation energy with respect to the geometry of the structure namely its chart and its thickness. On the second aspect, the minimization can deal with the flow recirculation length appearing or the regions of low wall shear stress with respect to the geometry of the stent.

Work in progress: the optimization of a simplified 2D stent is under investigation, first from an hemodynamic viewpoint. The total vorticity of a pulsatile incompressible viscous flow has been minimized with respect to three parameters defining the stent: the strut height, the strut pitch and the strut width. The optimization method that has been chosen is the genetic algorithm method, in order to obtain a global optimum and to avoid gradient computations.

Future work: the influence of the stent structure will be incorporated in the actual fluid simulation. Then a 3D fluid optimization will be investigated. In this case, due to the higher cost of each computation, a hybrid optimization method that has already been described in [1] will be used: the general idea is to couple a genetic algorithm with a deterministic descent method which will explore more rapidly the local minima of the functional. In order to maintain the stochastic aspect of the GA and for computational time reasons, the descent method is only and eventually applied at the best current element of the population after each generation.

6.3.3. Collapsed tube flow

The laminar steady flow of incompressible Newtonian fluid has been studied in rigid pipes with cross configuration of a collapsed tube to determine both the entry length and the wall shear stress (WSS). Entry length, axial and cross variations in WSS are indeed computed to design flow chambers in order to explore the mechanotransduction function of the endothelial cells. The cross section shapes have been defined from the collapse of an infinitely long elastic tube subjected to an uniform transmural pressure. Cross variation in wall curvature, which are also observed in collapsed tubes due to the transverse bending of the wall, induces transverse gradient of the WSS axial component. The cross-section configuration is not aimed at mimicing actual collapsed veins but creates at the cell scale a shear stress which generates two forces applied at the cell inertia center: (i) a shear force which stretches the cell in the streamwize direction and (ii) a shear torque which twists the cell perpendicularly to the cell plane, induced by the WSS transverse gradient. Five

characteristic collapsed configurations, from the unstressed down to the point-contact states, with a finite and infinite curvature radius at the contact point, are investigated. The numerical tests are performed with the same value of the volume flow rate whatever the tube configuration. The entry length was estimated by introducing three indices: (i) the first is defined by using the axial fluid velocity, (ii) the second by using the wall shear stress and (iii) the third by the pressure field, the pressure decease being non-linear in the entrance region. The results are analyzed in order to exhibit the mechanical environment of cultured endothelial cells in the flow chamber for which the test conditions will be well-defined.

6.3.4. Carotid bifurcation simulations

Blood flow simulations have been performed in a carotid artery network (Fig. 3). Mesh influence on the numerical results have been shown, using the same YAMS tool but different control parmeters, especially influences induced by the mesh of the carotid stem. Such a network will be very useful to compar full 3D detailed models with coupled models of shorter 3D segments and simplified 1D-0D model of vessel parts at distance form branching sites.

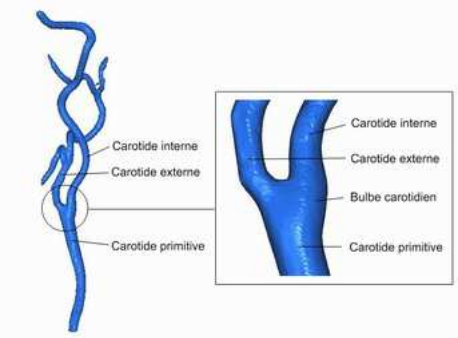

Figure 3. 3D reconstruction of the carotid artery network.

7. Contracts and Grants with Industry

7.1. Air Liquide

Participants: Marc Thiriet, Laurence Vial.

The purpose of this industrial grant (CIFRE PhD thesis) is to develop a numerical platform dedicated to the air flow simulation. This work is performed in the framework of the RTNS “R-MOD”.

7.2. Pechiney

Participant: Jean-Frédéric Gerbeau.

Industrial contracts with Ecole Nationale des Ponts et Chaussées in the framework of a collaboration with Aluminium Pechiney on the mathematical modelling of aluminium electrolysis cells (magnetohydrodynamics in presence of free interfaces). The work is done in collaboration with Claude Le Bris, Tony Lelièvre and Antonin Orriols (CERMICS & MicMac project).

8. Other Grants and Activities

8.1. National research program

8.1.1. RNTS “R-MOD”

Participants: Yvon Maday, Marc Thiriet, Laurence Vial.

RMOD study is aimed at developing a morpho-functional simulator of the respiratory conduits for computer-aided diagnosis (virtual bronchoscopy) and interventional pneumology. Upper and proximal airways are reconstructed from MSHCT performed at Pitié-Salpétrière hospital in Paris in collaboration with Artemis team at INT. Numerical results are compared with MRI of tracheal flow in human volunteers after hyperpolarized He inhalation.

8.1.2. ACI "le-poumon-vous-dis-je"

Participants: Jean-Frédéric Gerbeau, Céline Grandmont, Yvon Maday, Bertrand Maury, Marc Thiriet.

This project[2] aims at studying mathematical and numerical issues raised by the modelling of the lungs.

8.2. European research program

8.2.1. Research Training Network "Haemodel"

Participants: Paola Causin, Nuno Diniz dos Santos, Miguel Ángel Fernández, Jean-Frédéric Gerbeau [coordinator], Céline Grandmont, Yvon Maday, Marc Thiriet.

The aim of this project[3] is to investigate the main mathematical and numerical problems related to the simulation of the human cardiovascular system. Participants: INRIA, Université Paris 6, Politecnico di Milano (Italy), Imperial College (UK), Ecole Polytechnique Fédéral de Lausanne (Switzerland), Instituto Superior Técnico de Lisboa (Portugal), Technische Universität Graz (Austria).

8.2.2. ERCIM working group "IM2IM"

Participant: Marc Thiriet [main coordinator].

The ERCIM Working Group "IM2IM"[4] has been initiated in june 2003 in the context of minimally invasive treatment in medicine and surgery.

9. Dissemination

9.1. Scientific community animation

9.1.1. Various academic responsibilities

- L. Boudin is scientific coordinator in Mathematics Departement (DSPT 1) of the *Mission Scientifique, Technique et Pédagogique* (MSTP), of the *Ministère de l'Education Nationale, de l'Enseignement Superieur et de la Recherche*.
- J.F Gerbeau was reviewer of the PhD thesis of Simone Deparis (EPFL, march 2004), and examinator of the PhD thesis of Emmanuel Audusse (Paris 6, september 2004)
- J.F. Gerbeau was a member of the *Comité des bourses* of Rocquencourt RU (2003-2004)
- J.F. Gerbeau is a member of the *Commission de spécialistes* of the Montpellier II University.

[2] http://www.insa-rennes.fr/ACINIMpoumon/index.html
[3] http://mox.polimi.it/it/progetti/haemodel/
[4] http://www-rocq1.inria.fr/Marc.Thiriet/Im2im/

9.1.2. Summer schools organization

9.1.2.1. CEMRACS summer school

J.F. Gerbeau has organized, with E. Cancès (MicMac project), the 6 week summer school CEMRACS on "Mathematics and applications in biology and medicine".
The CEMRACS is an initiative promoted by SMAI (French Society of Applied and Industrial Mathematics), whose goal is to create and reinforce interdisciplinary collaborations between applied mathematicians and scientists of other fields from academia and industry, focusing on selected topics.

CEMRACS 2004 took place from the 26th of July to the 3rd of September 2004 at CIRM (Centre International des Rencontres Mathematiques, located at Luminy on the campus of the University of Marseille). The first week was devoted to lectures, attended by about 80 persons. During the five other weeks, the participants worked in teams on projects proposed by an industrial or academic partner. Each team was composed of young researchers assisted by one or more senior researchers. Moreover, a one hour daily seminar has been organized, given either by a participant of the CEMRACS or a visiting scientist.

CEMRACS 2004 received 130 researchers (15 different citizenships). This is the strongest participation since the first edition in 1996. From 50 to 80 participants were simultaneously present during the 6 weeks.

On the industrial side, projects have been proposed by ELA Medical (optimization of stimulation by pacemakers), Air Liquide (breath modelling) and CEA (mesh generation from medical imaging). On the academic side, several projects have been supervised by pluridisciplinary teams (mathematicians, biologists, bioengineers and medical doctors). The most significant contributions will be published in the journal *ESAIM Proceedings*.

9.1.2.2. CEA-EDF-INRIA summer school "Models and Solvers coupling"

P. Causin and J.F. Gerbeau have been assistant for the course of Charbel Farhat (Stanford University) during the CEA-EDF-INRIA summer school "Models and Solvers coupling". The purpose was to allow the participants of the school to pratice, through original test cases, various coupling strategies in fluid-structure interaction. This gave us the opportunity to experiment the REO fluid-structure solvers in the framework of SALOME (a CORBA platform developped in particular by EDF and CEA). This work has been done in collaboration with Marc Tajchman (CEA).

9.2. Teaching

- Laurent Boudin
  - Linear algebra and numerical methods, Licence, Paris 6 University.
- Jean-Frédéric Gerbeau
  - Analysis and scientific computing courses, Ecole Nationale des Ponts et Chaussées.
  - Fluid-structure interaction in biological flows, Master of numerical analysis, Paris 6 University (with Y. Maday and M. Thiriet)
  - Master in mathematical engineering, Ecole Polytechnique de Tunisie.
- Marc Thiriet
  - Fluid-structure interaction in biological flows, Master of numerical analysis, Paris 6 University (with Y. Maday and M. Thiriet)

9.3. Participation in conferences, workshops and seminars

- Paola Causin
  - Sixth World Congress on Computational Mechanics, Bejing (China) invited talks at two minisymposia organised by Z. Tian and by C. Taylor
  - VII Congresso SIMAI, (september, Venice, Italy)
  - Invited seminar: CEMRACS (Marseille, august)
- Laurent Dumas
  - Indo-French workshop (IIT Kanpur, India, october).
  - Workshop “Fluides et Structures” (Mulhouse, november).
  - Invited seminars: CERFACS (Toulouse, february), LAMA (Chambéry, may), CEMRACS (Marseille, august).
- Miguel Ángel Fernández
  - Fourth European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2004), Jyväskylä, Finland.
  - Workshop “Fluides et Structures” (Mulhouse, november).
  - Invited seminar: Franche-Comté University (Besançon, october).
- Jean-Frédéric Gerbeau
  - Congrès National d’Analyse Numérique (Obernay, june). Organizer of a minisymposium “Mathematics and applications in biology and medicine”
  - Invited seminars: Metz university (january), Bordeaux university (april), Versailles university (november)
  - CEA-EDF-INRIA school “Heart” (april). Lecturer.
  - CEA-EDF-INRIA summer school “Models and Solvers Coupling” (june). Lecturer.
  - Workshop “Numerical simulation for Aluminium industry” (Lausanne, october). Invited speaker.
- Céline Grandmont
  - Journées de Metz (april)
  - Invited seminars: CERMICS (Champs-sur-Marne, january), MIP (Toulouse, march), Rennes I university (may).
- Marc Thiriet
  - Journée Thématique de la Société de Biomécanique “Imagerie Médicale et Simulation Cardiovasculaire”, Université des Sciences de Montpellier, décembre 2003 (invited speaker).
  - 1st French-Taiwanese conference in IT, Ecole Polytechnique, Palaiseau, 14-16 avril 2004 (invited speaker).
- Laurence Vial
  - 14th European Society of Biomechanics (ESB) conference, Eindhoven, 2004, July 4-7.

10. Bibliography

Major publications by the team in recent years

[1] L. BOUDIN, L. DESVILLETTES, R. MOTTE. *A modeling of compressible droplets in a fluid*, in "Commun. Math. Sci.", vol. 1, n° 4, 2003, p. 657–669.

[2] L. DUMAS, F. GOLSE. *Homogenization of transport equations*, in "SIAM J. Appl. Math.", vol. 60, n° 4, 2000, p. 1447–1470.

[3] M. A. FERNÁNDEZ, P. LE TALLEC. *Linear stability analysis in fluid-structure interaction with transpiration. II. Numerical analysis and applications*, in "Comput. Methods Appl. Mech. Engrg.", vol. 192, n° 43, 2003, p. 4837–4873.

[4] M. A. FERNÁNDEZ, P. LE TALLEC. *Linear stability analysis in fluid-structure interaction with transpiration. I. Formulation and mathematical analysis*, in "Comput. Methods Appl. Mech. Engrg.", vol. 192, n° 43, 2003, p. 4805–4835.

[5] L. FORMAGGIA, J.-F. GERBEAU, F. NOBILE, A. QUARTERONI. *On the Coupling of 3D and 1D Navier-Stokes equations for Flow Problems in Compliant Vessels*, in "Comp. Meth. Appl. Mech. Engrg.", vol. 191, n° 6-7, 2001, p. 561-582.

[6] J. GERBEAU, M. VIDRASCU. *A quasi-Newton algorithm based on a reduced model for fluid-structure interactions problems in blood flows*, in "Math. Model. Num. Anal.", vol. 37, n° 4, 2003, p. 631-648.

[7] S. SALMON, M. THIRIET, J. F. GERBEAU. *Medical image-based computational model of pulsatile flow in saccular aneurisms*, in "M2AN", vol. 37, n° 4, 2003, p. 663-679.

[8] M. THIRIET, R. ISSA, J. M. R. GRAHAM. *A pulsatile developing flow in a bend*, in "J. Phys. III", vol. 2, 1992, p. 995-1013.

Articles in referred journals and book chapters

[9] J.-D. BOISSONNAT, R. CHAINE, P. FREY, G. MALANDAIN, S. SALMON, E. SALTEL, M. THIRIET. *From arteriographies to computational flow in saccular aneurisms: the INRIA experience*, to appear in Medical Images Analysis (MedIA).

[10] C. CARSTENSEN, P. CAUSIN, R. SACCO. *A Posteriori Dual-Mixed (Hybrid) Adaptive Finite Element Error Control for Lamé and Stokes Equations*, to appear in Numer. Math..

[11] P. CAUSIN, J.-F. GERBEAU, F. NOBILE. *Added-mass effect in the design of partitioned algorithms for fluid-structure problems*, INRIA RR-5084 to appear in Comp. Meth. Appl. Mech. Engng., 2004.

[12] P. CAUSIN, R. SACCO. *A Discontinuous Petrov–Galerkin method with Lagrangian multipliers for second order elliptic problems*, to appear in SIAM Jour. Numer. Anal..

[13] A. CHAMBOLLE, B. DESJARDINS, M. ESTEBAN, C. GRANDMONT. *Existence of weak solutions for an unsteady fluid–plate interaction problem*, to appear in J. Math. Fluid Mech..

[14] L. DUMAS, V. HERBERT, F. MUYL. *Hybrid method for aerodynamic shape optimization in automotive industry*, in "Comp. & Fluids", vol. 33, 2004, p. 849-858.

[15] M. FERNÁNDEZ, V. MILISIC, A. QUARTERONI. *Analysis of a geometrical multiscale blood flow model based on the coupling of ODE's and hyperbolic PDE's*, to appear in SIAM J. on Multiscale Model. Simul..

[16] M. FERNÁNDEZ, M. MOUBACHIR. *A Newton method using exact jacobians for solving fluid-structure coupling*, to appear in Comp. & Struct..

[17] J.-F. GERBEAU, T. LELIÈVRE, C. LE BRIS. *Modeling and simulation of the industrial production of aluminium: the nonlinear approach*, in "Comp. & Fluids", vol. 33, 2004, p. 801-814.

[18] J.-F. GERBEAU, M. VIDRASCU, P. FREY. *Fluid-Structure Interaction in Blood Flows on Geometries coming from Medical Imaging*, INRIA RR-5052 (2003) to appear in Comp. & Struct., http://www.inria.fr/rrrt/rr-5052.html.

[19] S. NAILI, M. THIRIET. *Pressure field in flow through uniform straight pipes with varying wall cross curvature*, to appear in Computers in Biology and Medicine, 2004.

[20] M. THIRIET, S. NAILI, C. RIBREAU. *Entry length and wall shear in uniformly collapsed veins Key Engineering Materials*, in "Trans Tech Publications", n° 243-244, 2003, p. 243-248.

Publications in Conferences and Workshops

[21] G. BARRENECHEA, M. FERNÁNDEZ, C. VIDAL. *A Stabilized Finite Element Method for the Navier-Stokes Equation with Dominating Reaction*, in "Proceedings of the Fourth European Congress on Computational Methods in Applied Sciences and Engineering, 24-28 July 2004. Jyväskylä, Finland.", 2004.

[22] E. BURMAN, M. FERNÁNDEZ. *Stabilized finite element schemes for incompressible flow using velocity/pressure spaces satisfying the LBB-condition*, in "Proceedings of the Sixth World Congress in Computational Mechanics, 5-10 September 2004. Beijin, China", 2004.

[23] E. BURMAN, M. FERNÁNDEZ, P. HANSBO. *Edge Stabilization: An Interior Penalty Method for the Incompressible Navier-Stokes Equation*, in "Proceedings of the Fourth European Congress on Computational Methods in Applied Sciences and Engineering, 24-28 July 2004. Jyväskylä, Finland", 2004.

[24] B. MAURY, C. GRANDMONT, Y. MADAY. *A multiscale/multimodel approach of the respiration tree*, 2004.

Internal Reports

[25] G. BARRENECHEA, M. FERNÁNDEZ, C. VIDAL. *A Stabilized Finite Element Method for the Oseen Equation with Dominating Reaction*, Technical report, INRIA RR-5213, 2004, http://www.inria.fr/rrrt/rr-5213.html.

[26] M. BELHADJ, E. CANCÈS, J. GERBEAU, A. MIKELIC. *Homogenization approach to filtration through a fibrous medium*, submitted, Technical report, INRIA RR-5277, 2004, http://www.inria.fr/rrrt/rr-5277.html.

[27] E. BURMAN, M. A. FERNÁNDEZ, P. HANSBO. *Edge Stabilization for the Incompressible Navier-Stokes Equations: a Continuous Interior Penalty Finite Element Method*, Technical report, INRIA RR-5349, 2004, http://www.inria.fr/rrrt/rr-5349.html.

[28] P. CAUSIN, R. SACCO, C. BOTTASSO. *Flux-Upwind Stabilization of the Discontinuous Petrov-Galerkin Formulation with Lagrangian Multipliers for Advection-Diffusion Problems*, Technical report, INRIA RR-5157, 2004, http://www.inria.fr/rrrt/rr-5157.html.

[29] S. DEPARIS, J.-F. GERBEAU, X. VASSEUR. *A Dynamic Preconditioner for Newton-Krylov Algorithms. Application to Fluid-Structure Interaction*, submitted, Technical report, INRIA RR-5352, 2004, http://www.inria.fr/rrrt/rr-5352.html.

Miscellaneous

[30] C. BARANGER, L. BOUDIN, P. JABIN, S. MANCINI. *A modeling of biospray for the upper airways*, submitted to the Proceedings of CEMRACS 2004, 2004.

[31] C. GRANDMONT. *Existence of weak solutions for an unsteady fluid-beam interaction problem*, submitted.

[32] C. GRANDMONT, Y. MADAY, P. MÉTIER. *Modeling and analysis of an elastic problem in large displacement and small strains:* $2D$ *and* $3D$*–cases*, submitted.

[33] P. LE TALLEC, P. HAURET, J. GERBEAU, M. VIDRASCU. *Fluid Structures Interaction Problems in Large Deformations.*, submitted.

Bibliography in notes

[34] A. AMSDEN, P. O'ROURKE, T. BUTLER. *A computer program for chemically reactive flows with sprays*, Technical report, n^o LA-11560-MS, Los Alamos National Laboratory, 1989.

[35] J. DUPAYS, Y. FABIGNON, P. VILLEDIEU, G. LAVERGNE, G. ESTIVALEZES. *Some aspects of two phase flows in solid propellant rocket motors, in Solid propellant chemistry, combustion and interior ballistics*, in "Progress in Astronautics and Aeronautics", V. YANG, T. BRILL, W. PEN (editors)., vol. 185, 2000.

[36] L. FORMAGGIA, J. F. GERBEAU, F. NOBILE, A. QUARTERONI. *On the coupling of 3D and 1D Navier-Stokes equations for flow problems in compliant vessels*, in "Comput. Methods Appl. Mech. Engrg.", vol. 191, n^o 6-7, 2001, p. 561–582.

[37] R. MOTTE. *A numerical method for solving particle-fluid equations*, in "Trends in numerical and physical modeling for industrial multiphase flows, Cargèse, France", 2000.

[38] A. QUARTERONI, S. RAGNI, A. VENEZIANI. *Coupling between lumped and distributed models for blood flow problems*, in "Comput. Visual Sci.", vol. 4, 2001, p. 111–124.

[39] G. P. R. CAFLISCH. *Dynamic theory of suspensions with Brownian effects*, in "SIAM J. Appl. Math", vol. 43, nº 4, 1983, p. 885–906.

[40] M. SERMESANT, Y. COUDIÈRE, H. DELINGETTE, N. AYACHE, J. SAINTE-MARIE, D. CHAPELLE, F. CLÉMENT, M. SORINE. *Progress towards model-based estimation of the cardiac electromechanical activity from ECG signals and 4D images*, in "ESAIM-Proceedings", vol. 12, 2003, p. 153-161.

[41] F. WILLIAMS. *Combustion theory*, 2nd, Benjamin Cummings, 1985.