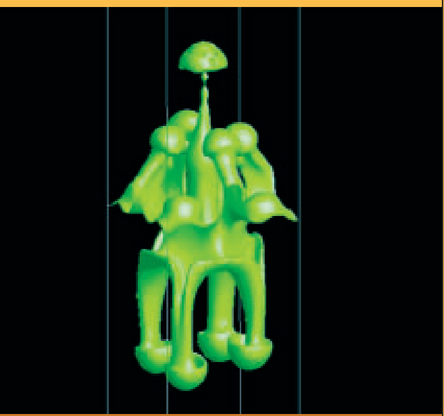
Fluid mechanics is everywhere!

>>> Jacques MAGNAUDET, CNRS Senior Scientist, Head of IMFT (joint laboratory UPS/INP/CNRS) and Henri-Claude BOISSON, CNRS Senior Scientist, Deputy Head of IMFT

Bursting with mathematical equations and physics theory, fluid mechanics finds itself in domains as diverse as energy production to the circulation of cerebral fluids.

Moving fluids exist on microscopic to astrophysical scales, and from fractions of seconds to light years. These huge scale disparities often make it difficult to understand fluid flows and predict how they will behave. However such an understanding is imperative for finding solutions to a great number of problems, related to the environment, transportation, energy, industrial process and health, where fluids are involved as transport agents and/or in chemical reactions.
The most common way to study fluid flows is based on a multi-scale approach. This consists of understanding and modelling the complexity of a process or a phenomenon in successive stages, starting from elementary, detailed local mechanisms and integrating these step-by-step into larger scale models. The approach generally combines mathematical modelling, laboratory experiments and calculations. From a fundamental point of view, the goal is to improve our knowledge of media in which complexity arises from specific properties.



>>> A heavy fluid is superimposed on a lighter one in a vertical square duct. Under the effect of gravity both fluids mix together: the heavy fluid flows down (preferentially near the corners), while the lighter one flows up (preferentially in the core of the duct). The image shows the boundary of the mixed zone at a given instant, represented by the surface on which both fluid phases are present in the same concentration.

Scientists in the Toulouse region are studying the following topics:

Hydrodynamic instabilities:

Researchers look at how the flow of a fluid changes when one of its characteristic parameters is varied. Starting from a stable state, the flow becomes unstable because its governing equations are nonlinear - problems that are often studied using mathematical methods and numerical simulation. Detailed laboratory experiments are then used to complete these studies to observe specific aspects of flow instabilities in real life. The goal is to understand how instability starts and to predict how the unstable flow evolves. Such studies might eventually allow fluid instabilities to be controlled, an area in which the Fluid Mechanics Institute of Toulouse (IMFT) has direct collaborations with the Toulouse Mathematics Institute (IMT), the Laboratory for Plasma and Energy Conversion (Laplace), the Institute of Aeronautics and Space (ISAE) and the French Aerospace Laboratory ONERA. The ultimate objectives of this research are to improve the performance of airplanes, ships, cars and trains.

Two-phase Flows:

Flows in two-phase media are intrinsically complex owing to the relative motion of the two distinct phases. They exist in a wide variety of topological configurations (drops, bubbles or films, for instance) in which mobile or deformable interfaces deeply affect the flow structure within each phase. Research in this area is concerned with the specific dynamics of this type of flow, which dramatically modifies heat and mass transfer. A good understanding of these flows is the key to controlling and optimising many industrial and environmental processes as well as energy management, especially in space applications.

Reactive Flows:

Reactive flows involve heterogeneous fluids under the influence of internal effects produced by chemical reactions. These reactions create energy transfers, modify the structure of the corresponding flows and increase the complexity of the system. The phenomena encountered in such flows may be turbulent or unstable -- for example flame instabilities. While some situations are similar to those encountered in the study of hydrodynamic instabilities, reactive flows have certain fundamental characteristics. Direct applications as a result of this research include improving car engines, combustion chambers, rockets and furnaces.

Flow interactions in the living world

Novel mechanisms, such as natural selection, genetic evolution and metabolism, come into play in fluids that contain or transport living

>>>

Headline

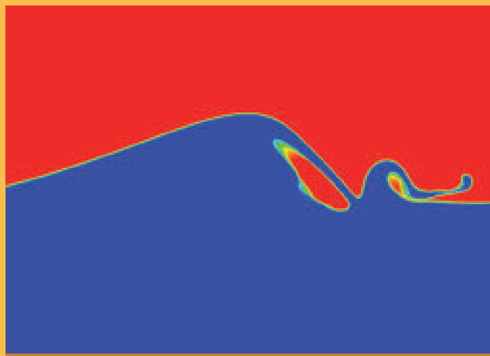
A clash of two sciences: When biology meets fluid mechanics...

Both these sciences have grown in parallel. Biologists and biotechnologists are concerned in isolating genes or making them react together without taking evolutionary modifications into account. For their part, fluid mechanists did not believe they could integrate living behaviour into mathematical models. Only over the last few years has dialogue been established between these two domains. Multi – disciplinary approaches still need to progress though in order to face the huge challenges in this field.

>>>

organisms. In environmental flows, researchers also encounter problems related to bio-film formation, which plays an important role in the growth of phytoplankton in rivers. Such interactions also have an important impact on soil pollution. Finally, fluid interactions are crucial for internal exchanges inside the human body. A typical situation is brain microcirculation, where the fluid flows through an incredibly complex network. Research in this field requires close collaboration between fluid mechanists, biologists, neurologists and many other scientists.

The Fluid Mechanics Institute of Toulouse (IMFT) is entirely devoted to fluid mechanics and promotes close links with many other laboratories in the Toulouse area that are also working in the domain of fluid mechanics, applied mathematics and computer sciences. These include labs at Paul Sabatier University, such as Laplace, IMT, the Informatics Research Institute of Toulouse (IRIT), labs from the Midi-Pyrénées Observatory, the National Centre of Meteorological Research (CNRM), the Biosystems and Process Engineering Laboratory (LISBP), the Chemical Engineering Laboratory (LGC), the Mechanical Engineering Laboratory (LMGT) and centres like the European Centre for Research and Advanced Training in Scientific Computation (CERFACS) and the French Aerospace Laboratory (ONERA).



>>> Simulation of a Stokes wave plunging in deep water performed with JADIM (in-house code)

Contacts: *Jacques.Magnaudet@imft.fr and Henri.Boisson@imft.fr*

Teaching and research in fluid mechanics at Paul Sabatier University

The Mécanique Energétique Procédés (Mechanics, Energetics and Process) masters degree offers many specialised courses in fluid mechanics. The second year of this course is split into two: Dynamique des Fluides Energétique et Transferts (Fluid Dynamics, Energetics and Transfers), which is geared more towards research, and Modélisation et Simulation en Mécanique et Energétique (Modelling and simulation in Mechanics and Energetics), which is oriented towards industry.
See: http://www.ups-tlse.fr/82364038/0/fiche___pagelibre/&RH=rub02.

Fluid mechanics is also taught in the Sciences de la Terre et de l'Environnement (Earth Science and the Environment) masters degree, specifically in the Hydrologie Hydrochimie Sol et Environnement (Hydrology, Hydrochemistry, Soil and Environment) course or in the Génie de l'Environnement (Environmental engineering) course.
See : http://www.ups-tlse.fr/3AHHS0_71/0/fiche___formation/&RH=rub02

PhDs are undertaken within the framework of doctoral schools :
Mécanique Energétique Génie Civil et Procédés (MEGeP) (http://www.imft.fr/MEGeP/); Sciences de l'Univers de l'Environnement et de l'Espace (SDU2E) (http://sdu2e.omp.obs-mip.fr/sdu2e/); and Aéronautique et Astronautique (AA) (http://www.isae.fr/ed-aa/)

Controlling flow to reduce pollution

Fewer harmful gases and lower noise emissions: flow control could significantly reduce the impact of transport on the environment. The economic implications are huge.

>>> Christophe AIRIAU, assistant professor at Paul Sabatier University, moderator of the “Instability and Control” team at IMFT (joint laboratory UPS/INPT/CNRS)

Enhancing the lift force and reducing drag are the main goals in flight transport today. Large lift allows a civil aircraft to fly with a maximal load while a small drag decreases the power and propulsion needed and reduces energy consumption in the engine (be it electrical, solar or fossil fuel).

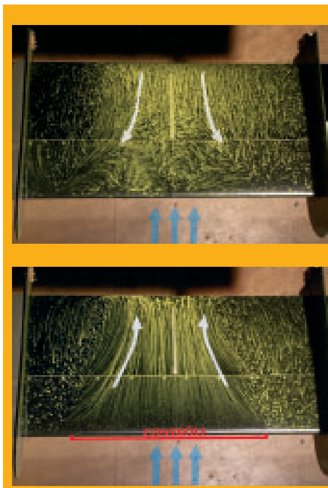
A predictable flow (also known as laminar flow) induces a weaker drag than a turbulent flow, where predictability is hindered by the generation and dissipation of random fluctuations and coherent structures. However, a laminar flow is very sensitive to any perturbations and quickly becomes unstable, which explains why natural flows are mainly turbulent. Moreover, turbulence generates strong pressure fluctuations that propagate in air, inducing unwanted noise. The goals of flow control are to reduce drag and noise emission while increasing lift by acting on turbulence or stabilising any perturbations.

Active control

Passively controlling the shape of an airfoil or of any vehicle has always been important for optimising aerodynamics. Many new experimental, theoretical and numerical approaches to active control have been developed since the 1990s, where actuators transfer energy from the wall into the fluid to improve performance (for example, from the wing section to the entire vehicle itself).

Recently, at IMFT and some other labs around the world, theories usually employed in industry to automatically control flow were used to render flows more stable or even make them stationary. However, this new work mainly concerned laminar or weakly turbulent flows. Moreover, the approach rapidly reached its limits since fluid mechanics deals with from 1000 to several million mathematical variables, figures that cannot be treated automatically in classic fluid control theory.

Today, two new methods are employed. The first one is open loop flow control where the control law, once activated, is no longer modified with respect to how efficient it is. With the technique of adjoining operators and direct numerical simulations using 0.1 million variables, the noise emission in a simple compressible flow (shear layer) has been successfully decreased by several decibels. Similar applications have eliminated instabilities in some other laminar flows.



>>> Micro-jet control on a wing to decrease drag and enhance lift close to the aerodynamic stall. Before (top) and after (bottom) control: lift increased by 20 %. Photo: A. Kourta, IMFT + LEA (Poitiers)

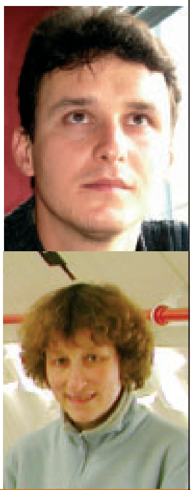
Feedback

At the Toulouse Mathematics Institute (IMT) Jean-Pierre Raymond's research team is developing other new flow-control techniques. These consist of computing control feedback laws of reduced order (the model has just 1000 to 10000 unknowns). Such feedback control is more efficient than an open-loop control since it depends on the real flow state measured by sensors. IMT and IMFT are working together to apply these new methods to more and more complex flows.
Developing and testing more efficient actuators is currently being carried out inside these labs and in collaboration with many academic partners and industry (for example, Dassault and Renault enterprises) by the Control of Flow Separation research group.

Contact: Christophe.Airiau@imft.fr

Headline

The science of sprays and bubbles



>>> Sébastien TANGUY, assistant professor at Paul Sabatier University and Catherine COLIN, professor at INPT (researchers at l'IMFT, joint laboratory UPS/INP/CNRS)

How does bubbling fuel behave inside a spacecraft's propulsion engine? The answer lies in the study of two-phase flows.

Two-phase flows are very important in fluid mechanics, both for their wide range of applications and their fundamental scientific interest. There are many two-phase flows applications where a liquid phase and a gas phase interact, for example in spray formation (with applications in internal combustion for propulsion) or bubbly flows commonly encountered in process engineering.

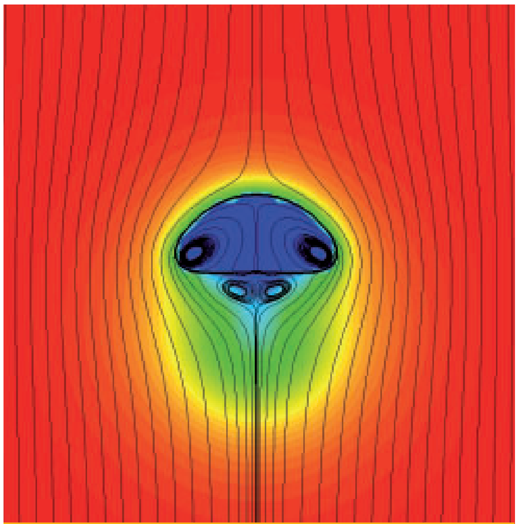
For the last two decades, the INTERFACE team at IMFT has been studying phase changes in gas-liquid flows. Phenomena such as boiling flows, droplet vaporisation or bubble condensation need to take into account thermal effects that affect hydrodynamic flow behaviour, which leads to more complex flows.

Nucleate boiling (hot wall bubble formation) is a very efficient thermal mechanism. Indeed, experiments have shown that wall convective thermal transfer coefficients can be improved if vapour bubbles are formed. However, if the wall temperature is too high, a film boiling transition (where the hot wall is completely covered by a continuous vapour film isolating the wall from the liquid) can occur. This is known as the so-called "boiling crisis".

This change in the two-phase flow configuration leads to a fast transition in the thermal transfer regime, which leads to a large decrease in the convective thermal transfer coefficient. This phenomenon is still poorly understood, and is particularly important in the cooling systems of nuclear plants.

The space industry is also interested in efficient cooling systems -- current thermal stresses for rocket propulsion systems are huge, both for burnt gas nozzles and combustion chambers. It is therefore crucial to use integrated and efficient cooling systems to alleviate the thermal and mechanical stresses that spacecraft materials experience. In this context, boiling flow studies are important for developing optimal systems for heat transfer.

Our work consists in studying different kinds of heat transfer -- latent heat, conduction and convection in simplified configurations. We have carried out nucleate boiling experiments for single bubbles to determine bubble growth rates, detachment diameters and how frequently bubbles are formed.



>>> Simulation that allows the temperature field and the current lines around an ascending, boiling bubble to be visualised

These experiments were performed under both standard gravity and micro-gravity conditions during airplane flights or rocket probes. Flat plate-free boiling was studied under micro-gravity conditions during the CNES "COMPortement des Ergols dans les Reservoirs" programme for the ARIANE V spacecraft.

The effect of forced convection effects on bubble detachment is the goal of the European Space Agency's project "Convective Boiling and Condensation". Thanks to these different experiments, new expressions for the bubble detachment diameter as a function of wall overheating and flow shear rate have been proposed.

Nucleate boiling heat transfer models are essentially based on empirical data. Direct numerical simulation of boiling flows could therefore be performed in parallel, for a more astute understanding of physical processes that occur on small scales during boiling.

Contacts: sebastien.tanguy@imft.fr and catherine.colin@imft.fr

Particles, droplets and engines

How can particles and droplets with a diameter of just a few microns influence oil refining and combustion in car engines? Numerical tools and supercomputers can give us some answers.

>>> Pascal FEDE, assistant professor Paul Sabatier University, Department of Mechanical Engineering and Benoît BÉDAT, assistant professor at Paul Sabatier University, Department of Mechanics at IMFT (joint laboratory UPS/CNRS/INP).

Flows containing several billion particles or droplets are present in fluidised beds and internal combustion engines. In such systems, several exchanges occur on the micron scale. These are: transfer of momentum, mass and heat between the carrier fluids and inclusions, inter-particle collisions and collisions with walls, and droplet coalescence. These transfers are tiny on the scale of individual particles and droplets but they start to influence the macroscopic flow via collective effects as the numbers of particles become huge.

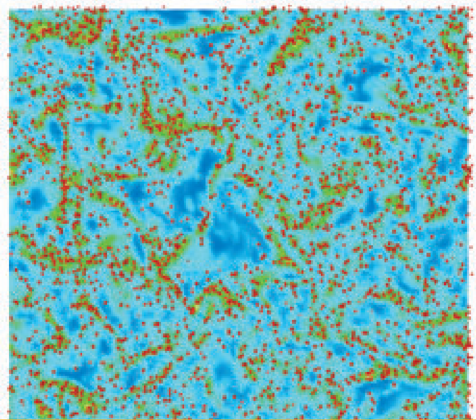
Million droplet trajectories

One of the objectives of the Flow and Combustion group at IMFT is to model such phenomena to improve predictive numerical tools. The researchers in the team are developing a mesoscopic method, called discrete particle simulation, in which numerical simulation of gaseous turbulent flow is coupled with computing the individual trajectories of several million particles and droplets. The transfers around each particle and droplet are not explicitly resolved but are taken into account by integral laws instead -- for example, via the drag force or vaporisation laws. These computations are numerical experiments that allow scientists to understand the local structure of the droplet or particle cloud and to validate existing theories. This strategy has allowed the team at IMFT to propose theoretical models that better take particle collisions and interface transfers into account. Moreover, a new approach to analyse turbulent-laden particle flows has been developed.

Droplet clusters

When particles or droplets begin to move at the same speed as large vortices in a fluid, they can segregate into clusters in a specific area of the flow where the vorticity is low. This phenomenon, called preferential concentration, can be important for the practical applications mentioned earlier because evaporation or chemical reactions are then concentrated in some areas of the turbulent flow.

For example, we can observe fuel droplets forming in clusters, which is characteristic of the preferential concentration phenomenon in the figure below. The



>>> Snapshot extracted from a 3D simulation of a droplet cloud evaporating in hot turbulent air. (The coloured portion corresponds to the fuel vapour field).

droplets evaporate in their own vapour, leading to the formation of rich or lean areas of fuel vapour. These heterogeneities at large scales can remain in the turbulent flow and produce poor combustion with unburned gas and soot formation.

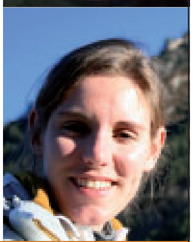
Contacts: fede@imft.fr and bedat@imft.fr

Fluidised bed

A fluidised bed is an industrial process used, for example, in the production of hydrocarbon fuel. In such a bed, a cloud of particles is transported by air flow injected into the bottom of the reactor. The behaviour of the dispersed phase is assimilated in the fluid. This process improves the exchange surface of the reaction between the two phases and assures good mixing.

When life and fluid mechanics come together...

Modelling river and coastal flows is far from trivial when chemical exchanges need to be taken into account, when there are living species present or when seashells make the bottom of the flow rougher...



>>> Frédéric MOULIN, assistant professor at Paul Sabatier University at IMFT (joint laboratory INPT/UPS/CNRS) and Stéphanie BOULÊTREAU, CNRS scientist at the ECOLAB (joint laboratory UPS/CNRS).

A crude description is often sufficient when modelling the purely dynamical aspects of a river or coastal flow on short time scales. To this end, researchers use the "hydraulic roughness", which measures how the drag exerted by a coastal or river bed slows the flow down.

However, river and coastal beds are complicated places and house living organisms that adapt themselves to the flow above but modify it by changing the conditions below. The organisms also modify biogeochemical fluxes in the flow because they consume, fix and produce various chemical species (like nitrates for instance).

Impact of human activity

Human activities greatly complicate models of hydrosystems on time scales large enough to take into account biological evolution at the bottom of a flow. Predicting the impact of human activity in the long term is based on models that include how living populations evolve as they are exposed to various environmental factors. For example, hydrodynamic forcing is one of the dominant factors in river and coastal ecosystems.

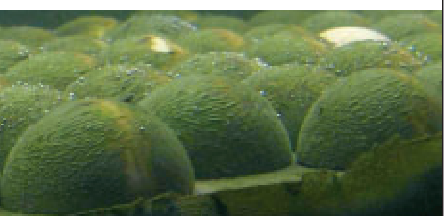
From a purely biological point of view, laboratory studies or in situ measurements produce a correct description of the development of organisms as a function of the physico-chemical conditions present. For instance, nutrient availability, light and the concentration of pollutants, can all be described. In the case of river biofilms, the taxonomic composition will depend on the development conditions but this also evolves with time, making it even more difficult to model this major biological parameter in some river ecosystems.

Another major difficulty arises when results obtained in laboratory (obtained with very slow or even no flow at all) need to be applied to natural conditions. For example, where flows are energetic and strongly turbulent (like in river and coastal flows). Here, local conditions surrounding living species at the bottom of the flow (such as the concentration of nutrients and oxygen) must be known before the biological models can be applied. Yet, these local conditions will depend on the characteristics of the turbulent flow higher up. Moreover, besides this indirect effect of the flow driving matter exchange between the bottom and the free stream above, another effect may be present -- mechanical and direct -- that controls the growth of organisms. In the case of river biofilms, for instance, studies in hydraulic channels at the IMFT show that the formation of filaments is limited by the intensity of flow turbulence, which can produce energetic vortices strong enough to tear off longer filaments. This leads to the development of structurally different biofilms, depending on the hydrodynamic conditions they were exposed to during their growth. This is a typical example of flow-structure interaction which is one of the research specialities at IMFT.

Flow around shells

Using fluid mechanics to describe river and coastal hydro-systems has motivated experimental studies in hydraulic channels of river biofilm growth or of turbulent flows around shells. The hydrodynamic conditions can then be described quite accurately and input to state-of-the-art measurement techniques available at IMFT (for example, particle imaging, velocimetry and laser doppler velocimetry), allowing precise studies of the interplay between living organisms and hydrodynamics. Of course, this work is fundamentally multidisciplinary, and based on close collaboration with other research teams, including the "Hydro-bio-géochimie des bassins versants" team at the Ecolab laboratory (UMR UPS/CNRS) for research activities on river biofilms.

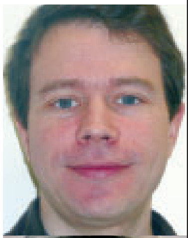
Contact: moulin@imft.fr and stephanie.bouletreau@cict.fr

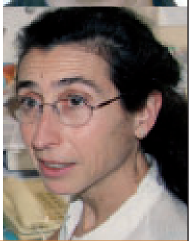


>>> artificial substrates in a hydraulic channel turbulent flow at the IMFT, covered by a growing river biofilm.

Micro-vascular networks in the brain

These networks supply healthy and cancerous cells with nutrients but they are still little understood. New cerebral imaging techniques could now solve this problem.





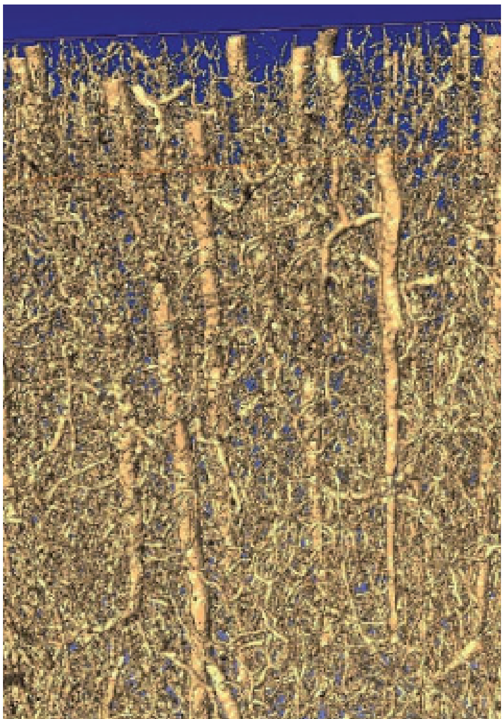
>>> Franck PLOURABOUÉ, CNRS scientist at IMFT (joint laboratory UPS /INP/CNRS) and Caroline FONTA, CNRS scientist at the Center for Brain Research (CERCO, joint laboratory UPS/CNRS).

Cortical micro-vascular organisation plays a central role in a large number of fundamental and clinical contexts, such as functional cerebral imaging, micro-vascular haemodynamic modelling, tumour growth and normal angiogenesis, and anti-tumour therapeutic strategies. Although potential applications are important, the organisation of micro-vascular networks is still poorly understood and little studied. Firstly, this is because these structures are very complex and are distributed over length scales of microns and millimetres and secondly, because of a “community effect”. Most studies have concentrated on identifying cellular or molecular mechanisms that could help us better understand the interactions between elementary biological components.

New imaging techniques are revolutionising this domain of research offering the possibility of observing vascular and cellular components in 3D, using either two-photon microscopy, ultra-fast fluorescent lasers and synchrotron tomography. Such state-of-the-art techniques require scientists from many different backgrounds to work closely together.

We have contributed to these new projects using synchrotron tomography for analysing how micro-vascular networks are organised in space. This unique tool has allowed us to analyse large volumes of cerebral cortex -- on the order of several tens of cubic millimetres with micron-scale resolution. We have shown that vascular density and tissue-vessel distance are fractal at small scales and have discovered that the vascular density becomes homogenous at distances of about 50 to 80 microns. Significant differences between normal and tumour-laden vascular networks have also been found since tumours have a higher metabolism than healthy tissue. We now plan to study the cortex in more detail, how micro-vascular networks evolve in time during normal and pathological angiogenesis and model blood flow in these networks.

Contacts: plourab@imft.fr and caroline.fonta@cerco.ups-tlse.fr.



>>> Synchrotron tomography of primate cortex volume injected with a contrast agent, from L. Risser, et al., J. Cereb. Blood flow and Metabolism, 27, 293-303, 2007. Scale bars are 100 microns in each direction.

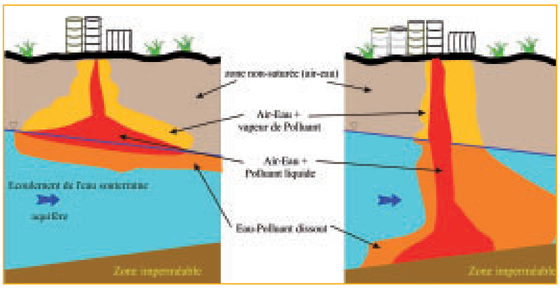
Headline

Soil and groundwater pollution

Depending on its chemical composition and the type of soil on which it is discharged, pollution can be broken down or stopped in its tracks before it reaches the aquifer layer of a river.

>>> Michel QUINTARD, CNRS senior scientist and Gerald DEBENEST, assistant professor at INPT, researchers at IMFT (joint laboratory UPS / CNRS / INP).

The study of pollution in soils is a multidisciplinary field where fluid mechanics in porous media plays an important role. When chemical contamination occurs, the amount of pollutant transferred to the aquifer is mainly determined by gravity, and/or driven by the load of rainwater filtering down to the water table. For example, the mechanical properties of fluids and porous media lead to very different contamination scenarios in cases of pollution caused by a fluid that is partially miscible with water. This is a very common scenario (see the database on contaminated land, http://basol.environnement.gouv.fr), as illustrated in the figure below.



>>> Behaviour of a partially miscible liquid pollutant during infiltration in an aquifer.

Contamination scenarios

The left hand side of the figure represents pollution by a LNAPL (light non-aqueous phase liquid), that is, a hydrocarbon liquid less dense than the water phase. The liquid enters the soil and stops flowing when it meets the water table. The fluid exchanges contaminants in the porous medium with the gas phase and with the liquid phase, where the contaminants may dissolve. The aquifer water may even end up carrying the dissolved pollutant to a domestic or irrigation pumping well.
In the second case (right hand side), the pollutant is a DNAPL (dense non-aqueous phase liquid), that is, a liquid more dense than water (such as a chlorinated solvent, for instance). Therefore, it can cross the water table barrier and eventually reach the impermeable zone at depths often much larger than the water table depth itself.
To perform risk analyses and to protect against contaminants, we need to know how the contaminants travel through the soil and the aquifer, while taking into account a possible natural decrease in the concentration of these pollutants before they arrive at a pumping well. Apart from purely mechanical mechanisms, natural attenuation may involve biochemical degradation, thanks to the presence of biofilms (for example, bacteria clusters) in the soil.

Detoxification

To model these coupled mechanisms, which is a complex task, we have developed a set of mathematical and numerical techniques that work at the various scales encountered in the natural environment.
Our methods are particularly useful in characterising a polluted site and especially when considering remedial procedures to return the site to its original state, if possible.
Understanding these processes requires a multidisciplinary approach, which has led us to develop scientific links with different laboratories, including ECOLAB (Joint laboratory UPS / CNRS) and Gérino Magali's team at Paul Sabatier University. This collaboration has allowed us to begin studying detoxification processes at the interfaces of river hydrosystems. The main objective of our research is to improve our understanding of pollution attenuation thanks to the presence of biofilms in a porous medium representative of a river bottom.

Contacts: quintard@imft.fr and debenest@imft.fr