# Porous medium with OpenFOAM

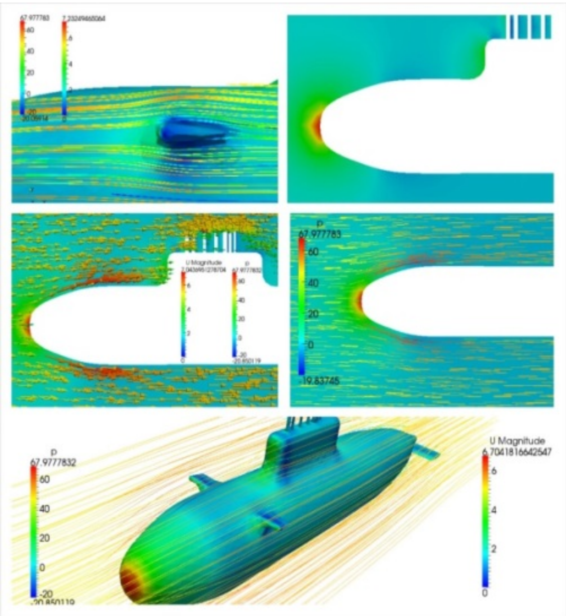
## Open Source Toolbox for Fluid Mechanisms

Romain Guibert – *Research Engineer* – IMFT/INP
Pierre Horgue – *Research Engineer* – MFEED platform
Gérald Debenest – *Professor* – IMFT/INP

# What is OpenFOAM?

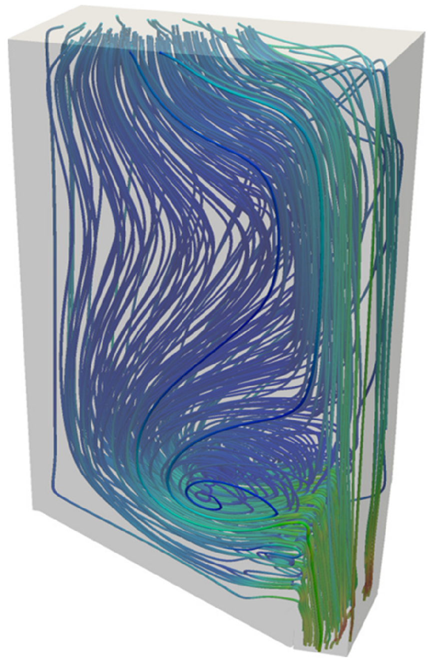
The Open Source CFD Toolbox

**OpenFOAM** (for "Open source Field Operation And Manipulation") is a C++ toolbox for the development of customized numerical solvers, and pre-/post-processing utilities for the solution of continuum mechanics problems, including computational fluid dynamics (CFD). The code is released as **free and open source software** under the GNU General Public License. It is managed, maintained and distributed by The OpenFOAM Foundation which is supported by **voluntary contributors**. The OpenFOAM name is a registered trademark of OpenCFD Ltd and licensed to the OpenFOAM Foundation Ltd.



$$\nabla \cdot \left( \left( M_a + M_b \right) \nabla p_a \right) = -\nabla \cdot \left( \left( L_a + L_b \right) \mathbf{g} - M_b \frac{\partial p_c}{\partial S_b} \nabla S_b \right) + q_a + q_b$$

```
{
    fvScalarMatrix pEqn
        (
            fvm::laplacian(-Mf, p) + fvc::div(phiG)
            // capillary term
            + fvc::div(phiPc)*activateCapillarity
            // wellbore terms
            - (-SrcExt*Wext+SrcInj*Winj)*activateWellbores
        );

    pEqn.solve();

    phiP = pEqn.flux();

    phi = phiP+phiG+phiPc*activateCapillarity;

    phib == Fbf*phiP + (Lbf/Lf)*phiG + phiPc*activateCapillarity;
    phia == phi - phib;

    U = fvc::reconstruct(phi);
    U.correctBoundaryConditions();

    Ub = fvc::reconstruct(phib);
    Ua = U-Ub;

    Ub.correctBoundaryConditions();
    Ua.correctBoundaryConditions();

}
```



# Multi-scales: different models and numerical tools

**Microscale** (local scale)

Usual models:

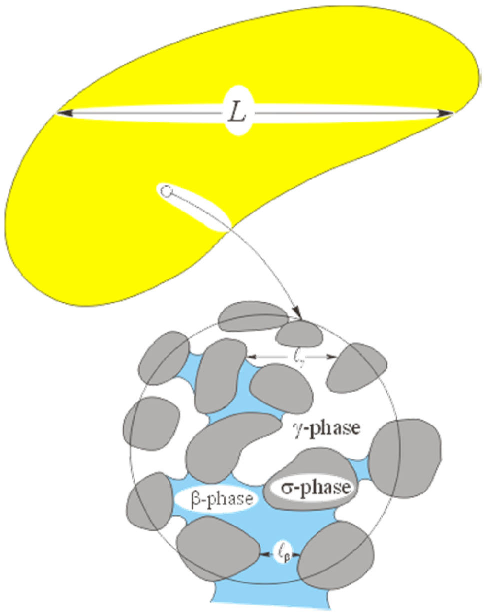
- Stokes/Navier-Stokes
- VOF for multiphase
- Lagrangian Particle Tracking

Issues:

- Complex geometries
- CPU time expensive

In OpenFOAM:

- Most of the tools already available




**Macroscale** (large scale)

Specific models:

- Darcy-Forchheimer
- Generalized Darcy's law

Issues:

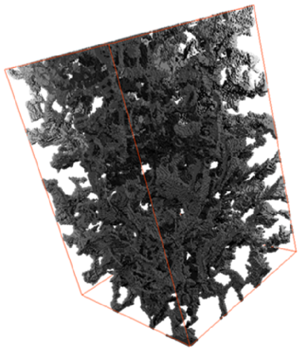
- Effective medium properties
- Upscaling local physics

In OpenFOAM:

- No dedicated tools
- Simple "penalization" approach

# Micro-scale : effective properties determination

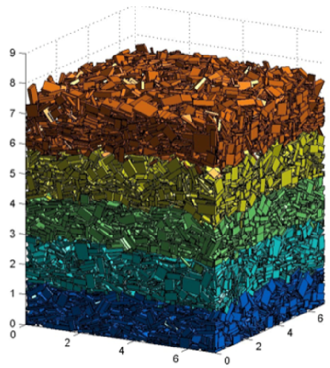
- "*No*" homemade solvers => `simpleFoam` / `laplacianFoam`
- Most of the numerical work concerns bash scripts / few improvements.



Permeability determination

Mesh sensitivity
Method/BC comparison
Parallel efficiency

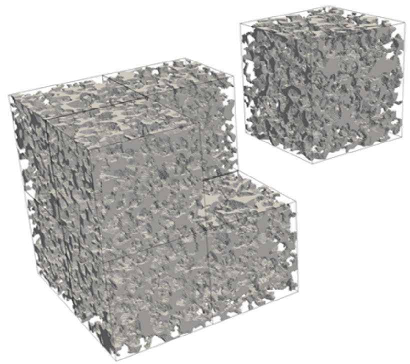
(Transport in Porous Media 2015)
(Mathematical Geosciences 2016)



Inertial and thermal flows

Forchheimer coefficient
Effective thermal conductivity

(Oil Shale Symposium 2015)



Two-step Upscaling

Method efficiency study

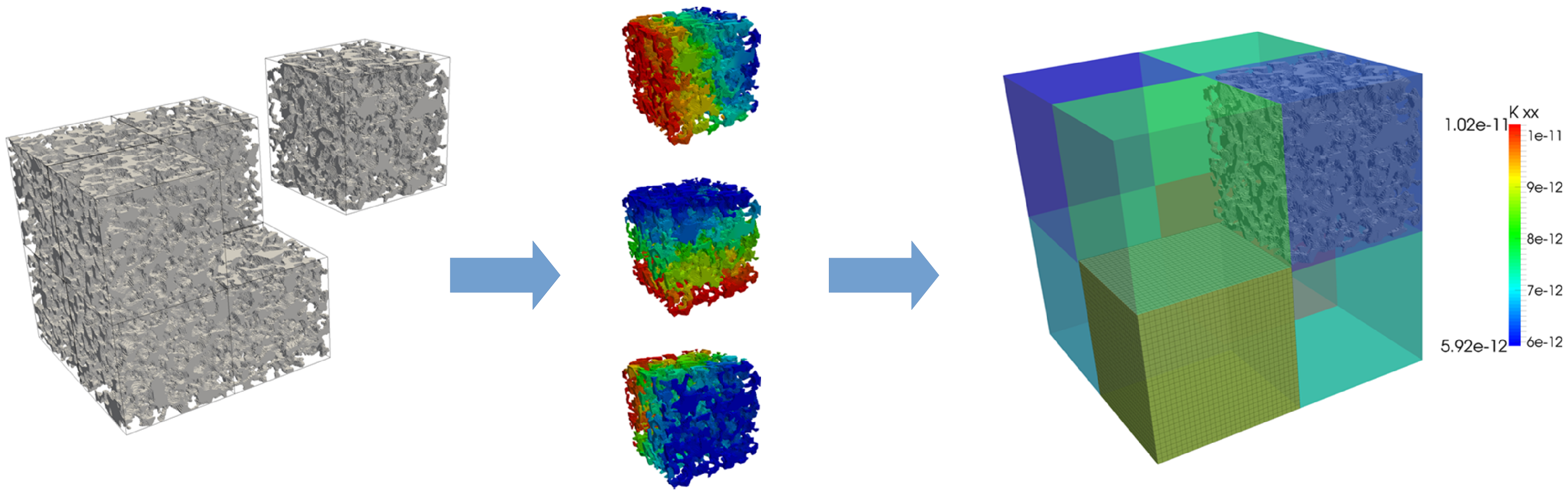
(Computational Geosciences 2015)

# Two-step upscaling method for permeability evaluation

1) Divide in sub-domains

2) First upscaling

3) Second upscaling




(Computational Geosciences 2015)

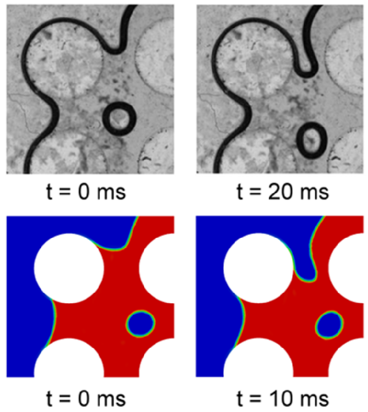
Results
- Drastic reduction of computational resources required (CPU, RAM)
- Allows mesh refinement and large sample volumes.

# Micro-scale : transport and two-phase flows

Dispersion coefficient
`SimpleParticleFoam`

Passive LPT algorithm
Statistical tools to determine dispersion coefficient

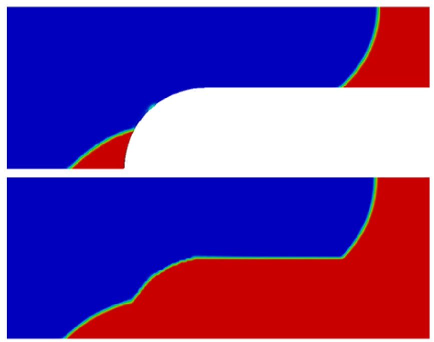
(EAGE, Wien, 2016)




Tube bundle experiments
`heleShawInterFoam`

Penalization for wall effects
Good agreement between numerical and experimental

(Chemical Engineering Science 2013)



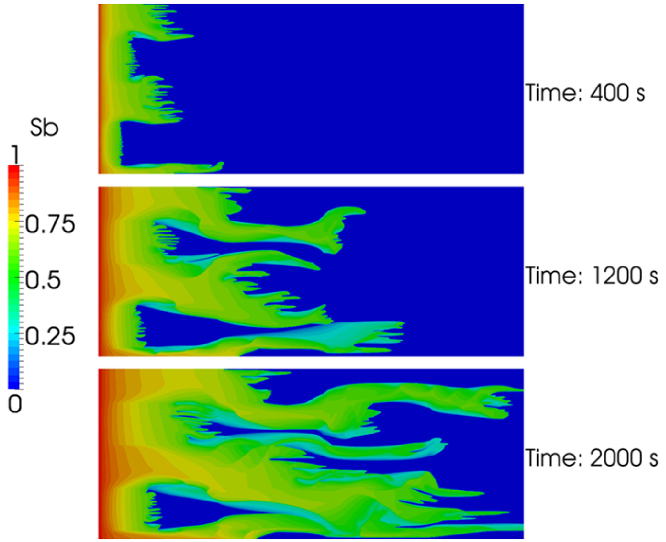
Immersed boundary
`penalizedInterFoam`

Solid domain penalization
Wetting effect on immersed boundaries

(Computer & Fluids 2014)

# Macro-scale open-source porous multiphase toolbox

Water invasion in
heavy oil reservoir




(Computer Physics Communications, 2015)

Context :

Usual approach in OpenFOAM :
=> penalization term in momentum equation

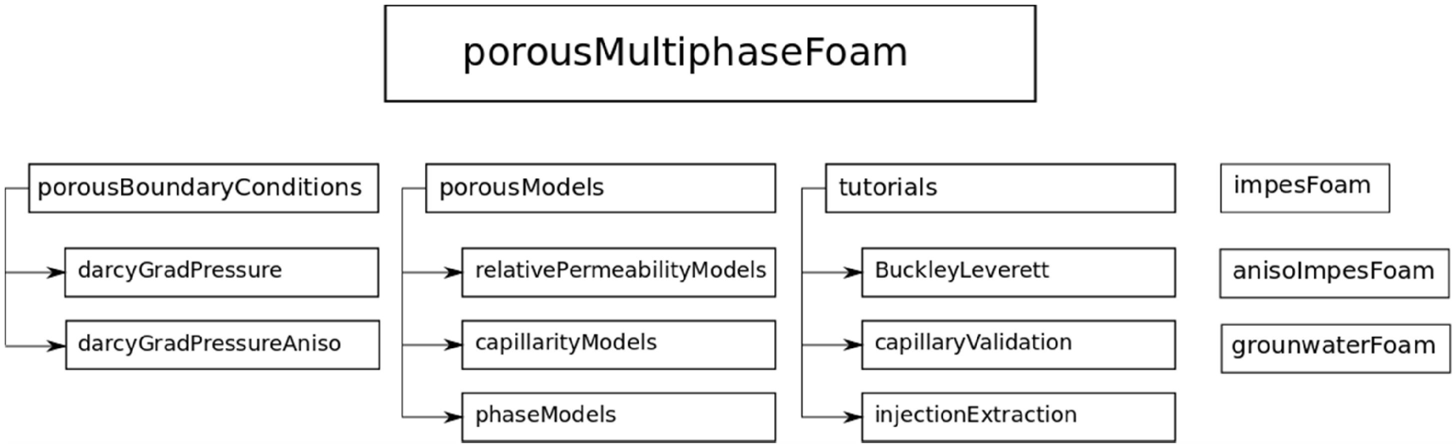
Missing essential features:

- phase saturations
- relative permeabilities
- capillary pressure laws
- boundary conditions for fixed fluxes

Objective :

- develop solver dedicated to porous multiphase flow
- make it as an evolutive toolbox (libraries, BC,...)

# Toolbox content




**Boundary conditions**
Pressure boundary condition
for fixed velocity

**Porous models**
Brooks and Corey (1964)
Van Genuchten (1980)
Linear model

**Tutorials**
Validation cases
(analytical solutions)
Practical test-cases

# Toolbox extensions: Richards' equation

Model :

- pressure gradient in the non-wetting phase neglected.

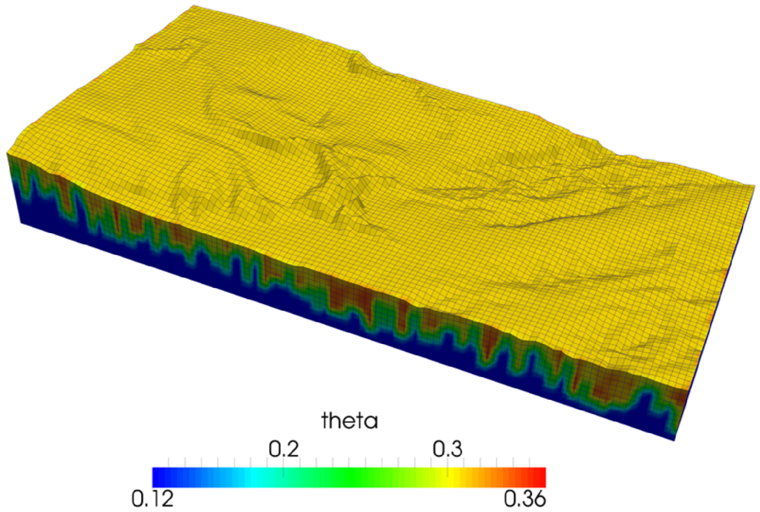
Improvements:

- Picard's algorithm
- Improvement of the existing C++ classes in the dynamic library libporousModels.so

Technical report :

- Model description
- Validation cases
- Efficiency study

(Technical report, HAL/arXiv, 2015)




Water infiltration on unstructured mesh
(topographic dataset)

# Recent developments: Reactive flows

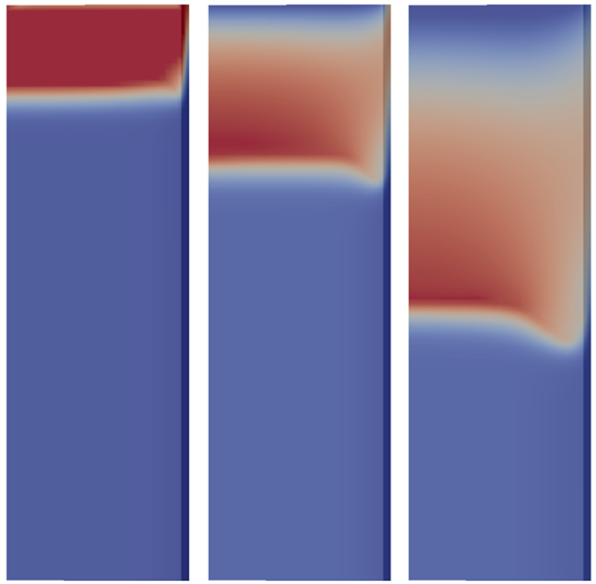
`drymethaFoam` for methanation modeling :

- Incompressible two-phase flow in waste
- Double porosity approach (static and mobile water)
- Solute transport and transfer in water
- Biodegradation model
- Mixed tank boundary condition

`combustionPorousFoam` for decarbonation :

- Compressible flow
- Temperature equation
- Gas component transport
- Solid component degradation

(Bioresource Technology, 2015)



Combustion front in
oil shale semi-coke

# On-going work and collaborations

**Around the toolbox**

- Specific stability criteria for time-step management

- Compressible version of the IMPES solver.

- Upscaling method (Multiphase Darcy to Reservoir scale)
ECMOR 2016, Amsterdam

- Many others depending on our possibilities

**Collaborations**

- Phd
- Master internships

MFEED

- Numerical studies for industrials
- Support for students or Engineers
- Dedicated developments
- Open-source tools training

# Contact

**Team**

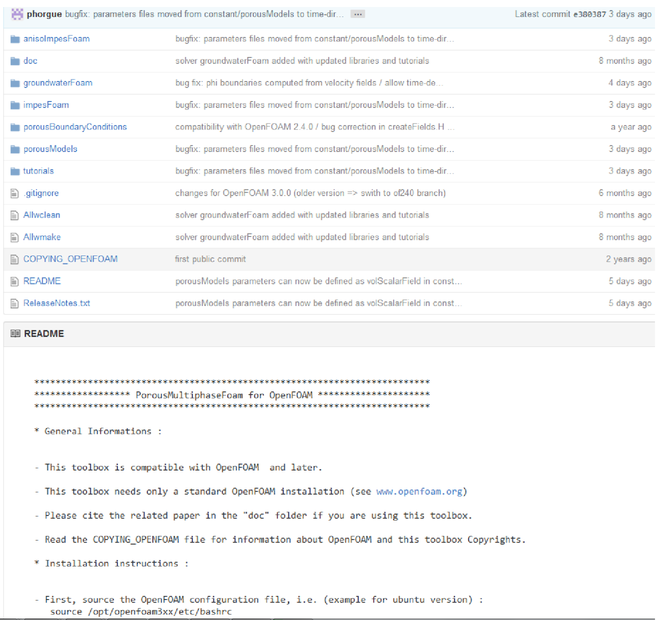
G. Debenest : debenest@imft.fr / debenest@enseeiht.fr
P. Horgue : pierre.horgue@mfeed.fr / phorgue@imft.fr

**Toolbox**

https://github.com/phorgue/porousMultiphaseFoam

**MFEED informations**

www.mfeed.fr

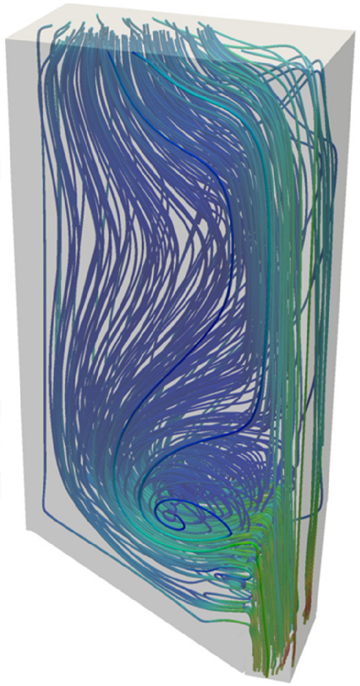


phorgue bugfix: parameters files moved from constant/porousModels to time-dir… Latest commit e380387 3 days ago

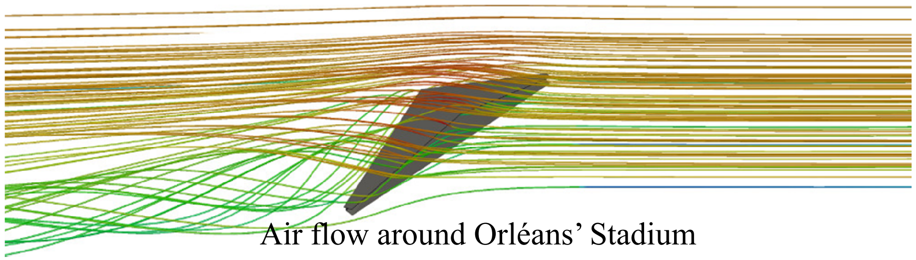
| | | |
|---|---|---|
| anisoImpesFoam | bugfix: parameters files moved from constant/porousModels to time-dir… | 3 days ago |
| doc | solver groundwaterFoam added with updated libraries and tutorials | 8 months ago |
| groundwaterFoam | bug fix: phi boundaries computed from velocity fields / allow time-de… | 4 days ago |
| impesFoam | bugfix: parameters files moved from constant/porousModels to time-dir… | 3 days ago |
| porousBoundaryConditions | compatibility with OpenFOAM 2.4.0 / bug correction in createFields.H … | a year ago |
| porousModels | bugfix: parameters files moved from constant/porousModels to time-dir… | 3 days ago |
| tutorials | bugfix: parameters files moved from constant/porousModels to time-dir… | 3 days ago |
| .gitignore | changes for OpenFOAM 3.0.0 (older version => swith to of240 branch) | 6 months ago |
| Allwclean | solver groundwaterFoam added with updated libraries and tutorials | 8 months ago |
| Allwmake | solver groundwaterFoam added with updated libraries and tutorials | 8 months ago |
| COPYING_OPENFOAM | first public commit | 2 years ago |
| README | porousModels parameters can now be defined as volScalarField in const… | 5 days ago |
| ReleaseNotes.txt | porousModels parameters can now be defined as volScalarField in const… | 5 days ago |

README

```
************************************************************************
***************** PorousMultiphaseFoam for OpenFOAM ********************
************************************************************************

* General Informations :

- This toolbox is compatible with OpenFOAM  and later.

- This toolbox needs only a standard OpenFOAM installation (see www.openfoam.org)

- Please cite the related paper in the "doc" folder if you are using this toolbox.

- Read the COPYING_OPENFOAM file for information about OpenFOAM and this toolbox Copyrights.

* Installation instructions :

- First, source the OpenFOAM configuration file, i.e. (example for ubuntu version) :
  source /opt/openfoam3xx/etc/bashrc
```



Turbulent flow in
alumine filters



Air flow around Orléans' Stadium