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?

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.
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

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

(Computational Geosciences 2015)
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

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

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)
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
- 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

Turbulent flow in alumine filters

Air flow around Orléans’ Stadium