CSDMS 2013 Meeting

Modeling of Earth Surface Dynamics and Related Problems Using OpenFOAM®

Xiaofeng Liu, Ph.D., P.E.
Assistant Professor
Department of Civil and Environmental Engineering
University of Texas at San Antonio, Texas
Email: xiaofeng.liu@utsa.edu
http://engineering.utsa.edu/~xiaofengliu

Outline

- A brief introduction of OpenFOAM
- Sample applications of OpenFOAM
- Demonstration (Time permitting)

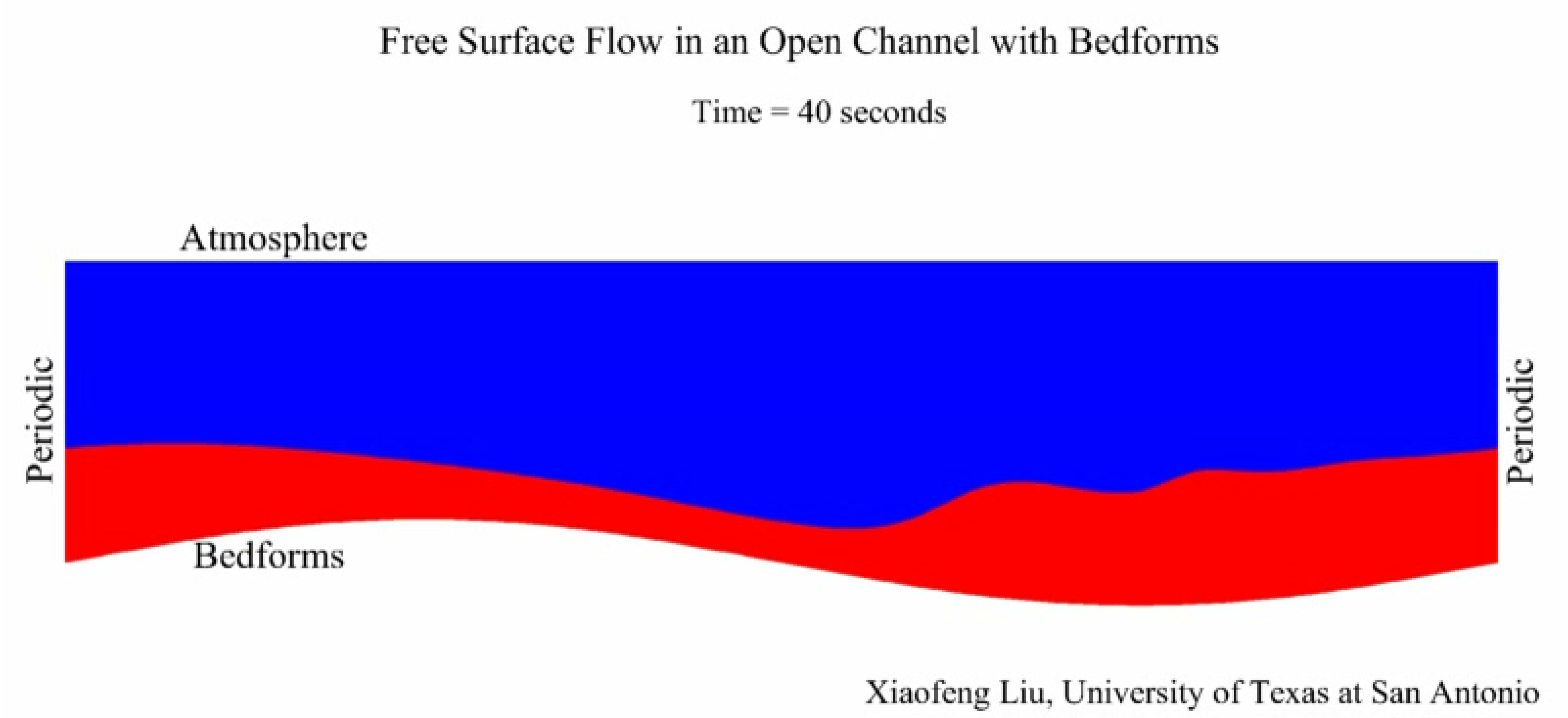
A brief introduction of OpenFOAM

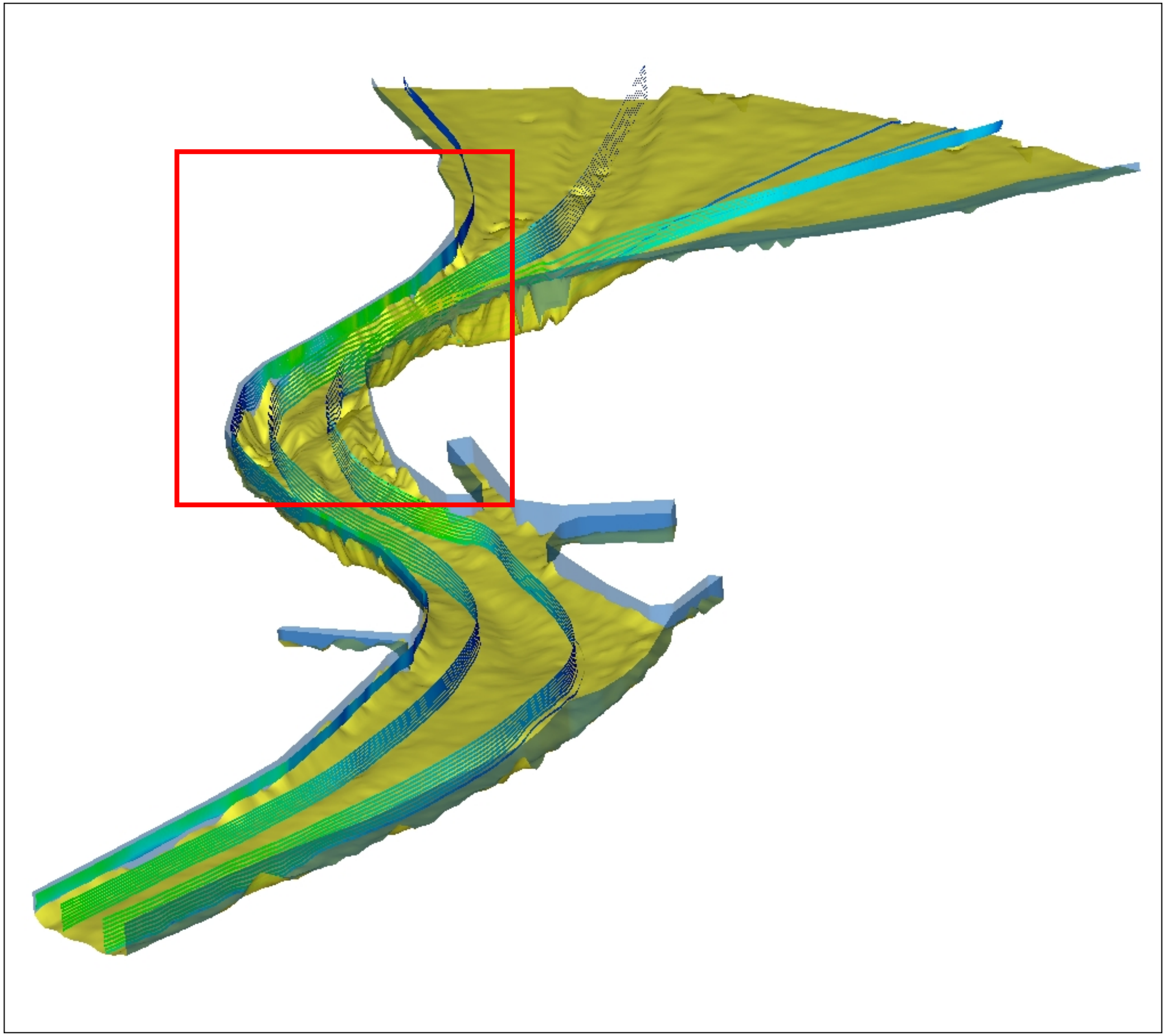
- OpenFOAM is an open source multi-physics modeling platform written in C++
- FOAM stands for "**F**ield **O**peration **A**nd **M**anipulation"
- OpenFOAM is not limited to fluid dynamics
  - It is a generic modeling platform
  - It can be used to solve (m)any differential equation(s)

Example applications of OpenFOAM

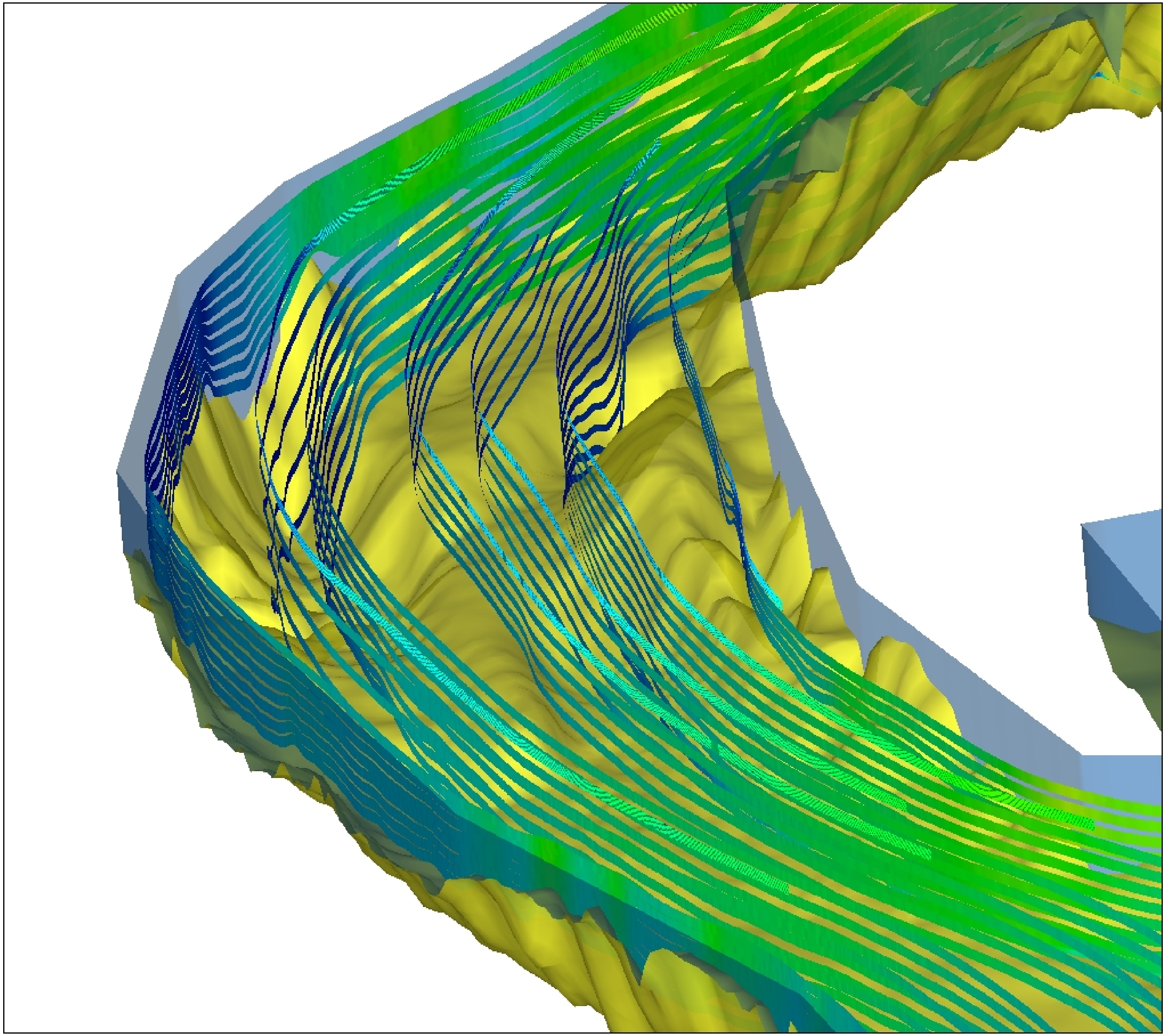
- River flow, coastal flow, waves, and sediment transport
- Hydraulic structures
- Porous media flow and solute/particle transport
- Buoyant flows and multiphase flows
- Some new developments in our group

Free surface flows over bedforms

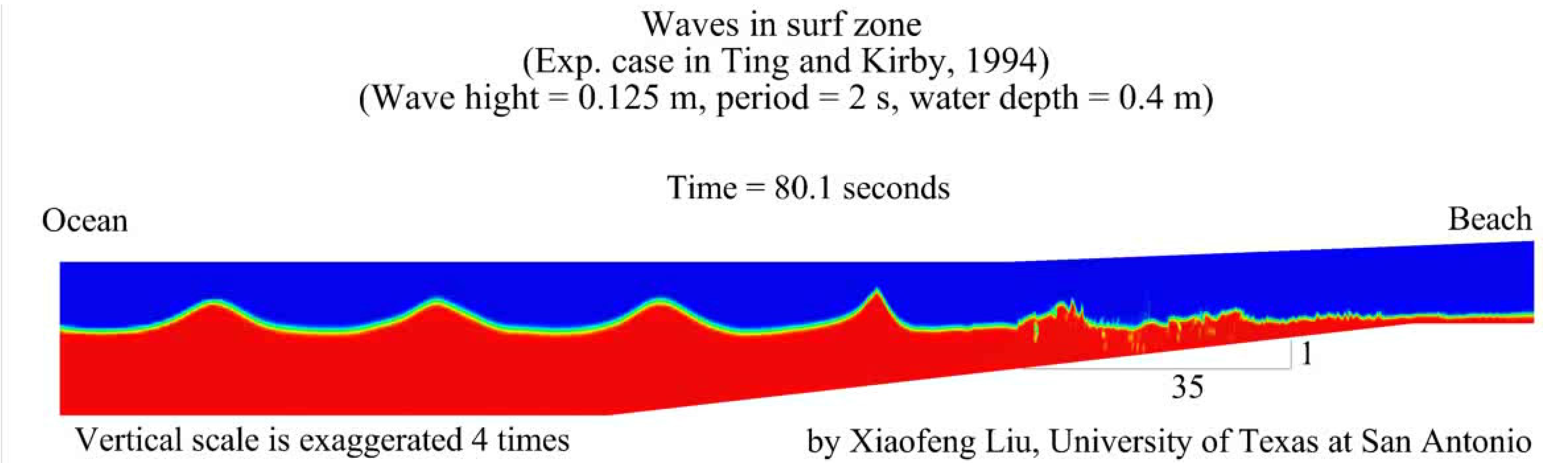




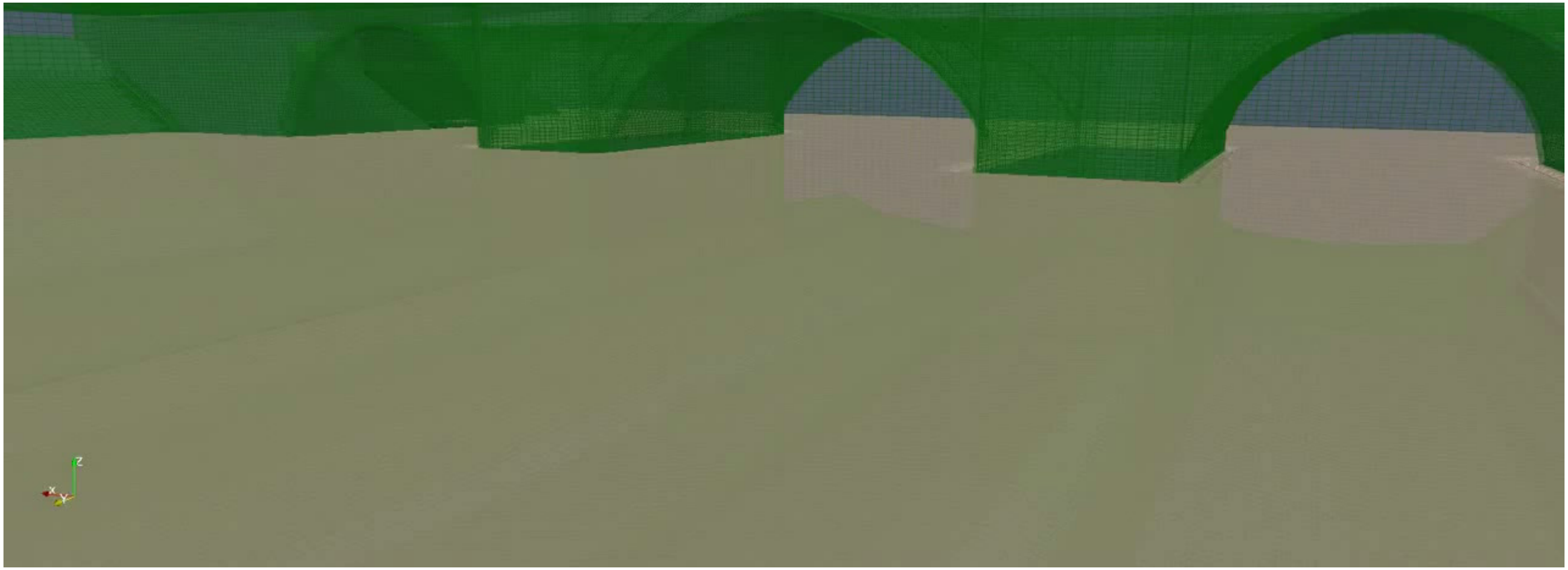
Liu et al., 2011



Liu et al., 2011


Waves in surf zone
(Exp. case in Ting and Kirby, 1994)
(Wave hight = 0.125 m, period = 2 s, water depth = 0.4 m)
Time = 80.1 seconds
Ocean
Beach
1
35
Vertical scale is exaggerated 4 times
by Xiaofeng Liu, University of Texas at San Antonio

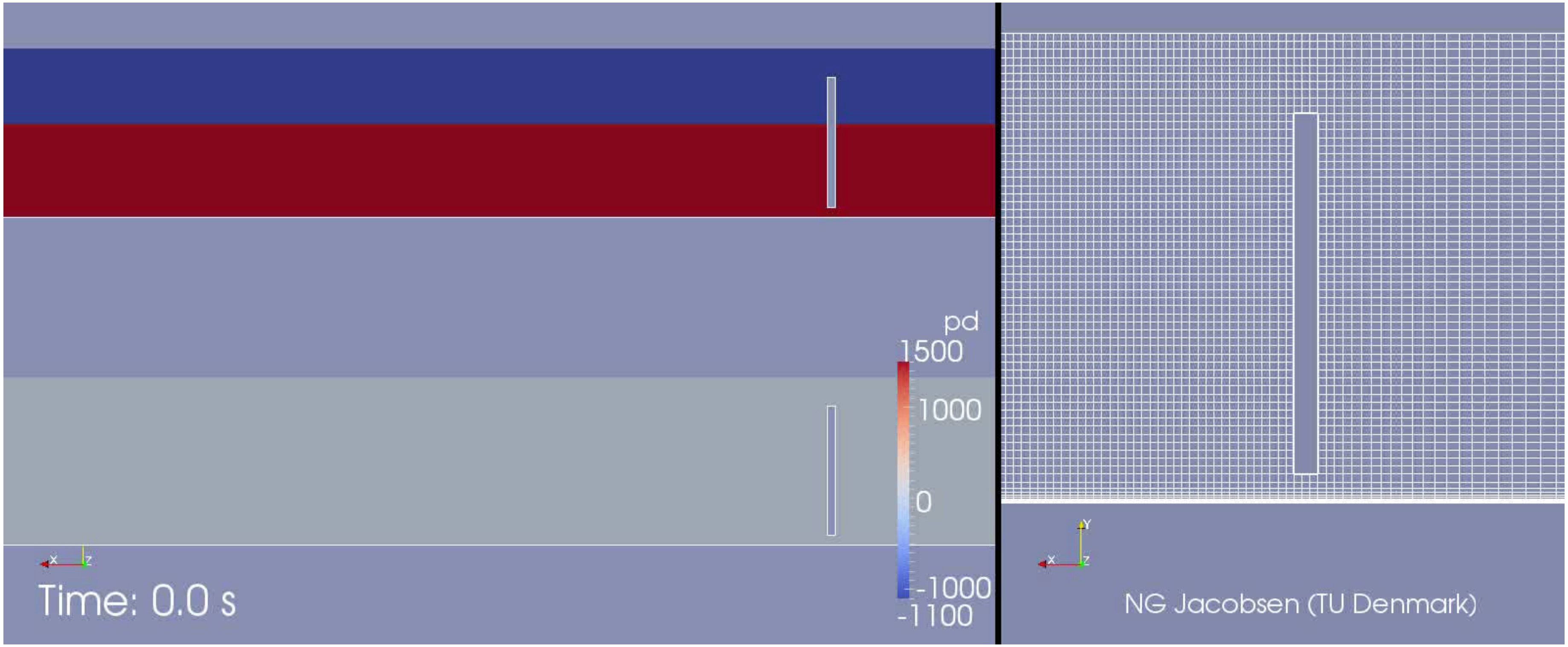
River flow under a bridge



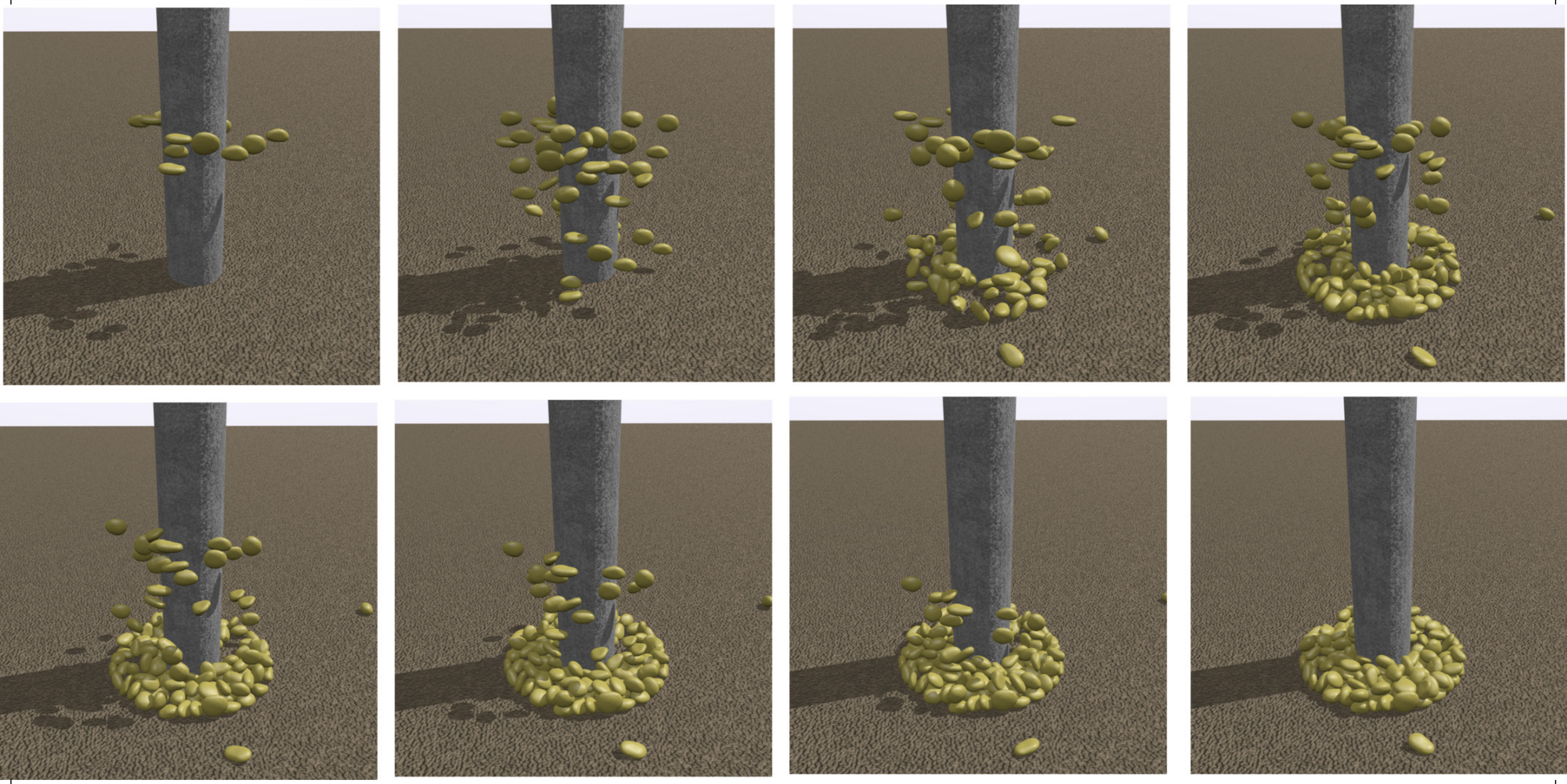
http://www.edenvaleyoung.com

http://www.youtube.com/watch?v=BqKN5QwGPB4

Scour under an elevated wall due to waves

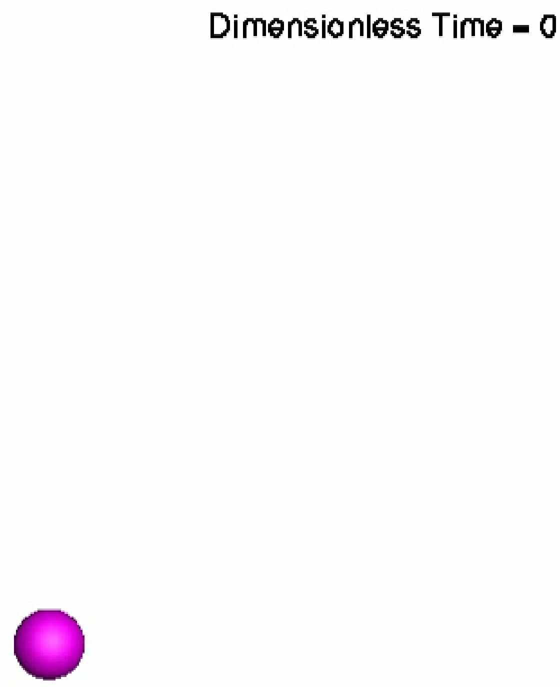


Particle resolving scour protection simulations

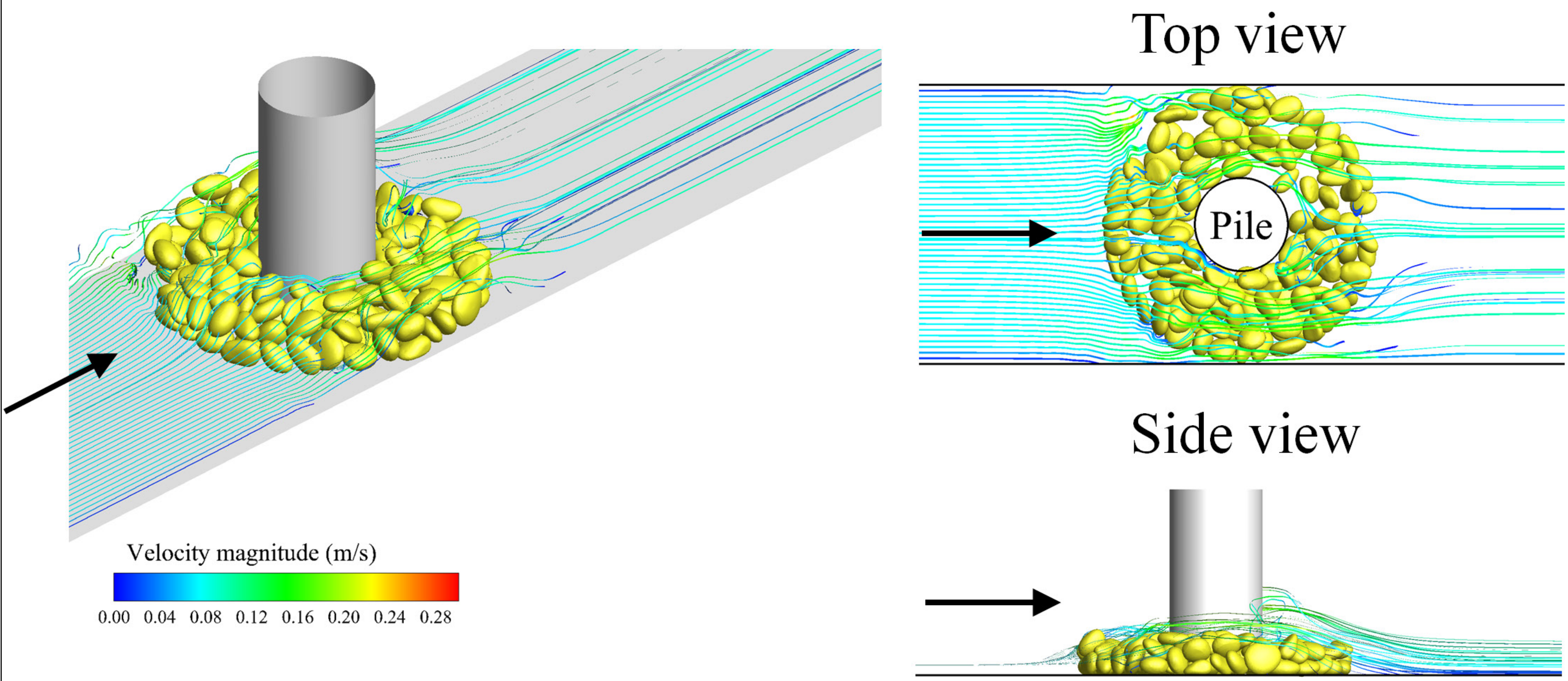


Liu et al, 2012

Flow past a Sphere using Immersed Boundary Method
with Unstructured Mesh in OpenFOAM (Re=350)
Vortical Structure using λ_2 Criteria

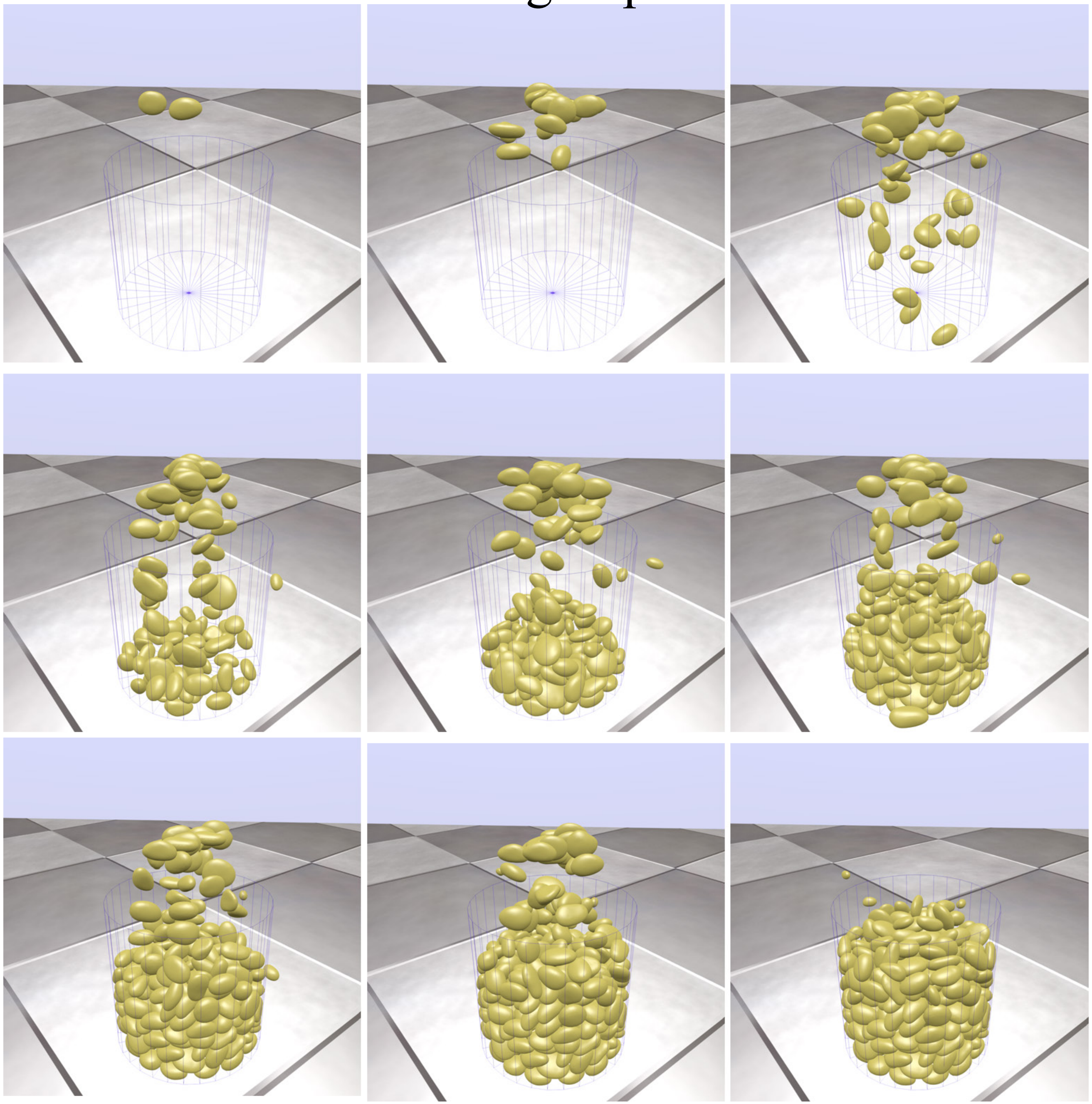


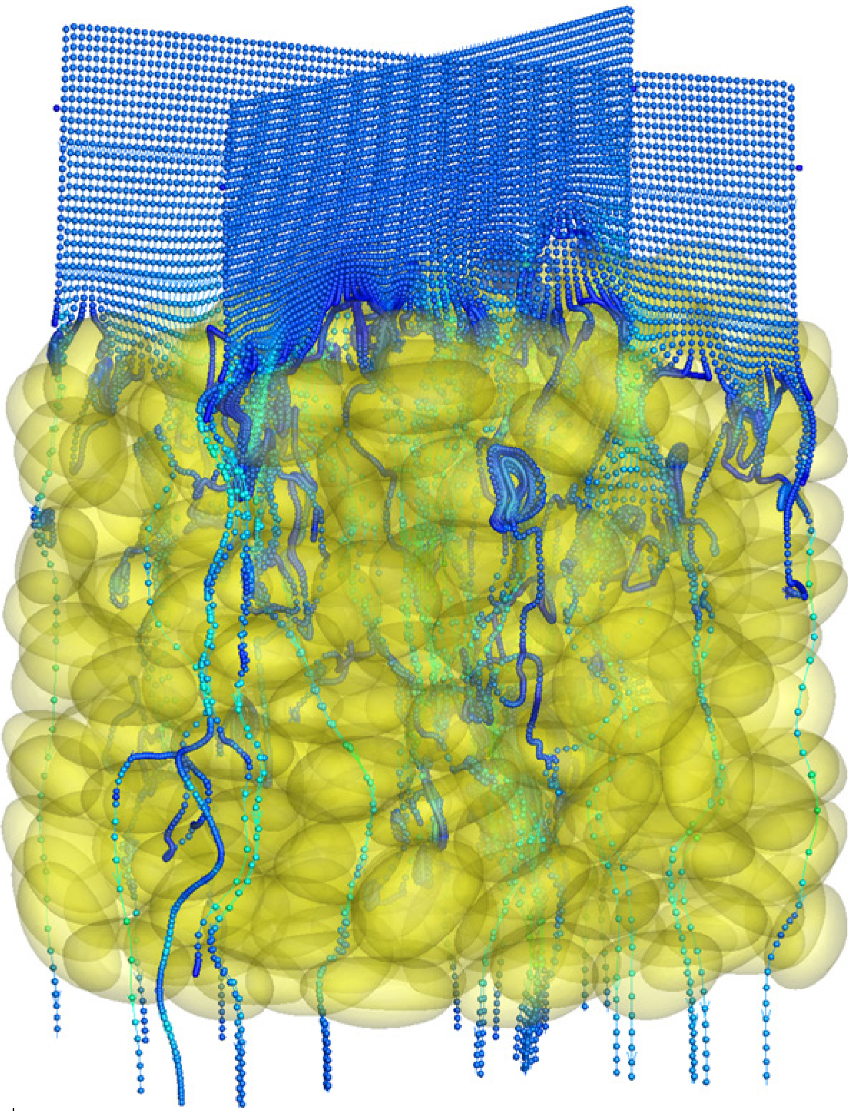
Xiaofeng Liu, University of Texas at San Antonio, USA
http://engineering.utsa.edu/~xiaofengliu

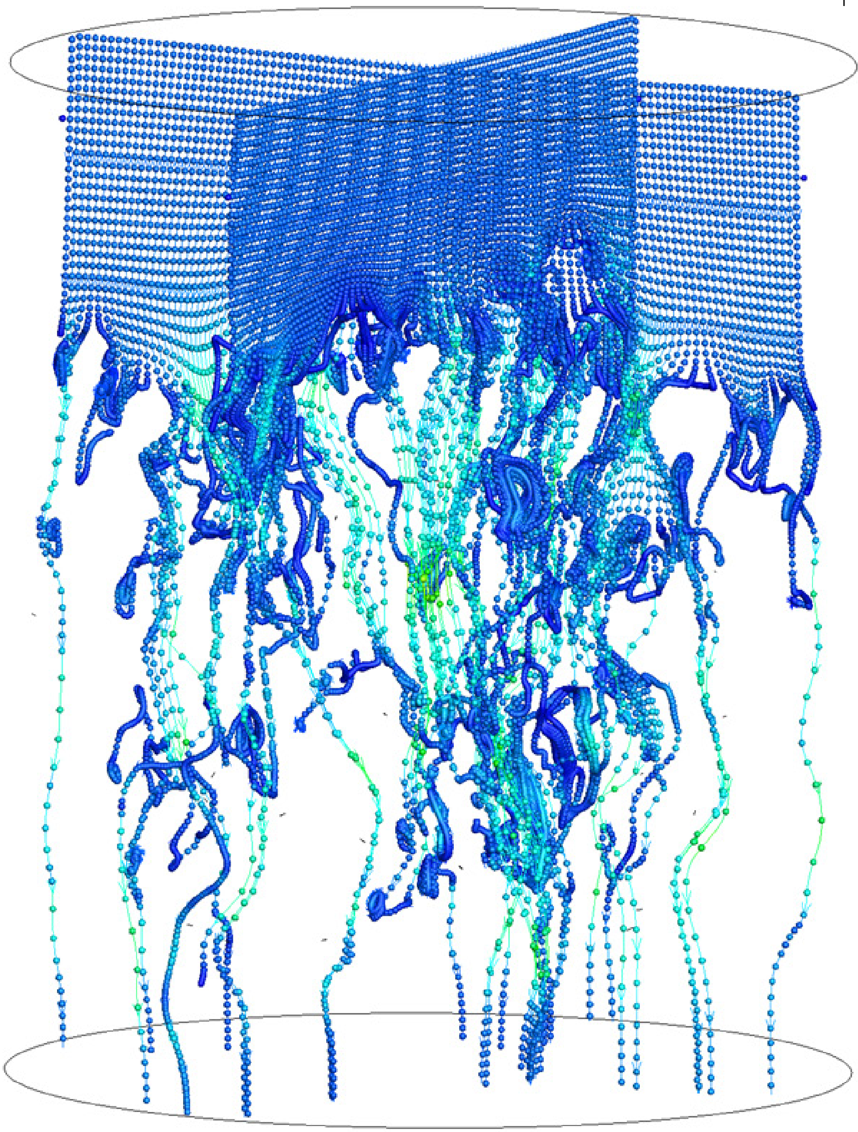


Liu et al, 2012

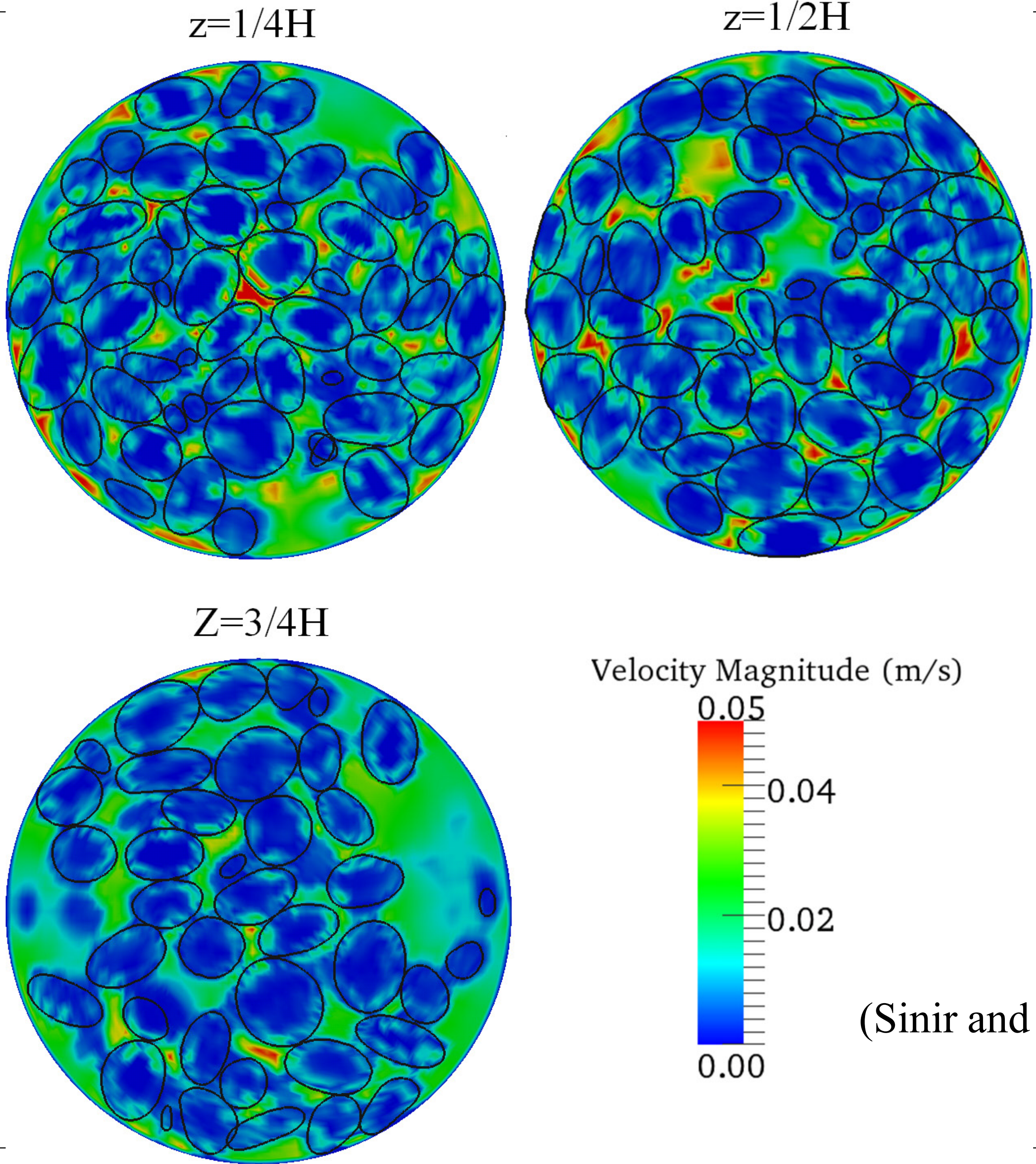
Pore-scale modeling of porous media flow





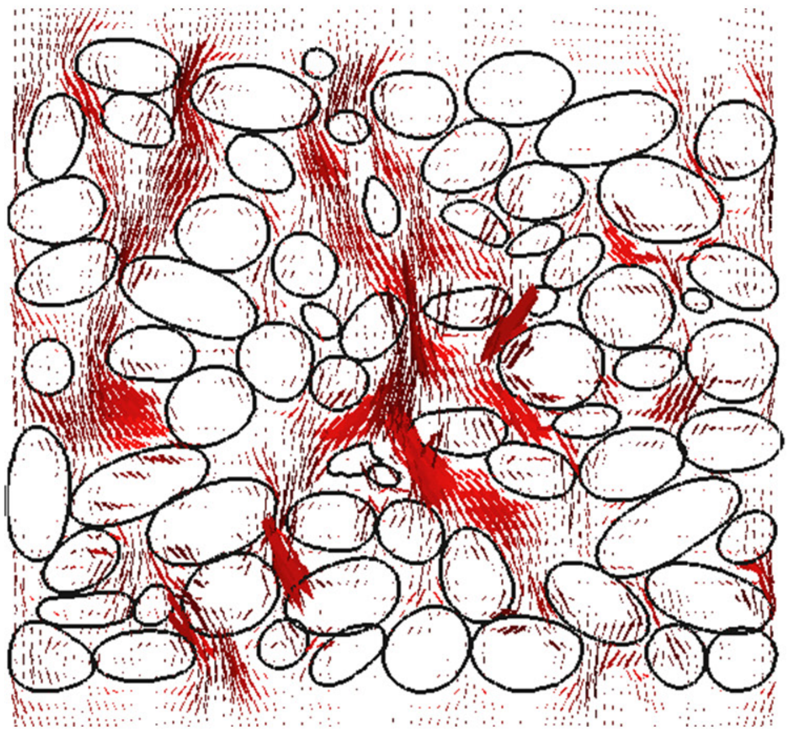


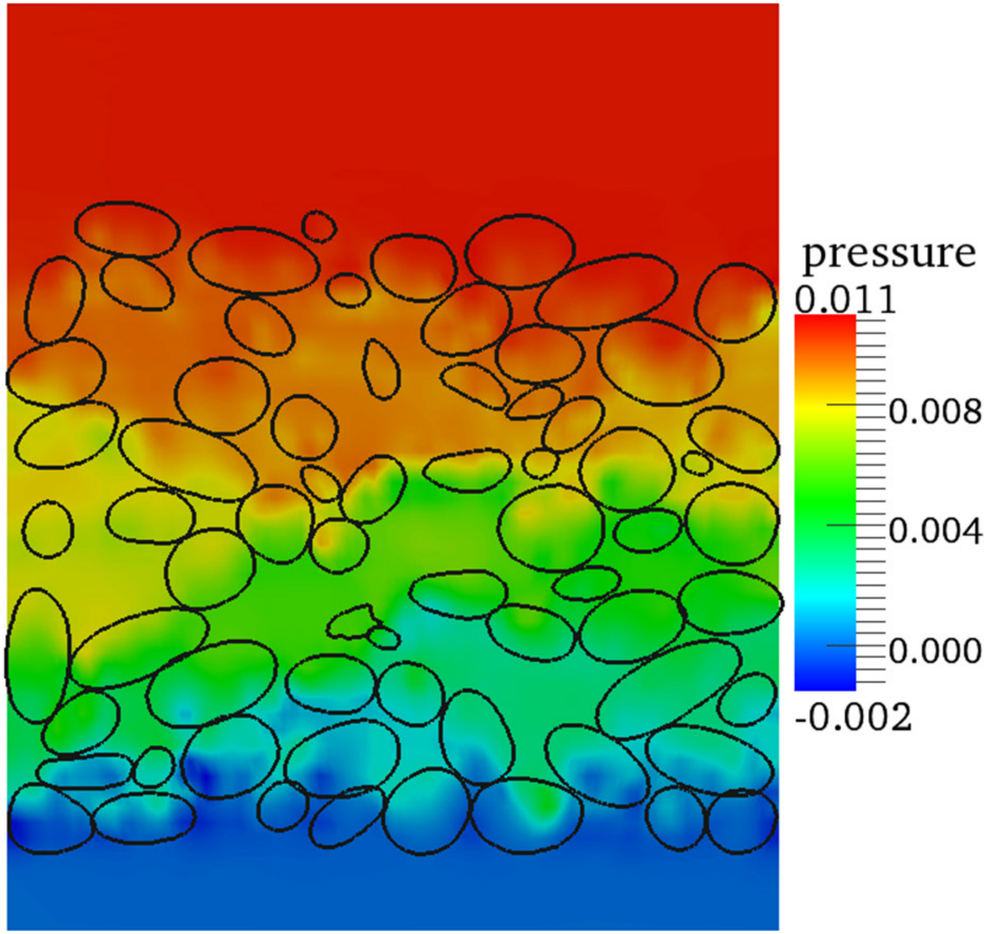
Sinir and Liu, 2012



(Sinir and Liu, 2012)

Velocity vectors





(Sinir and Liu, 2012)

Porous media flow and solute/particle transport

- Solves simple groundwater flow equation

$$S_s \frac{\partial h}{\partial t} = K \nabla^2 h + Q$$

where h is the pressure head, S_s is the specific storage coefficient, K is hydraulic conductivity, Q is source/sink

This governing equation is a simple heat equation. The solution of which is very easily implemented in OpenFOAM using tensor notations.

- Also solves advection-diffusion-reaction (ADR) equation

$$\frac{\partial C}{\partial t} + \nabla \cdot (\mathbf{U} C) = D \nabla^2 C - \frac{\rho_b}{n} \frac{\partial S}{\partial t}$$

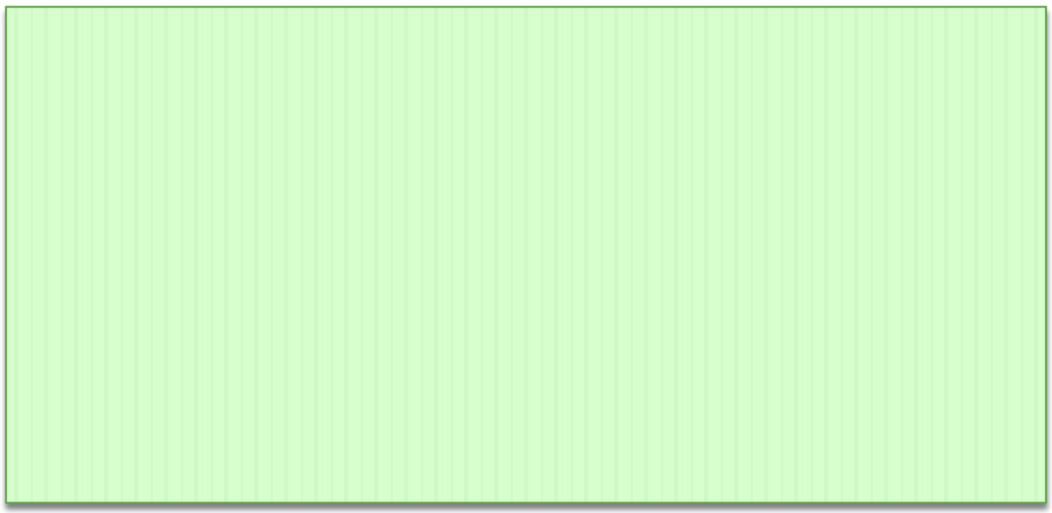
$$\frac{\partial S}{\partial t} = \frac{n}{\rho_b} K_a C - K_d S$$

All these with less than 50 lines of code!

- Porous media flow test case: Toth (1963):

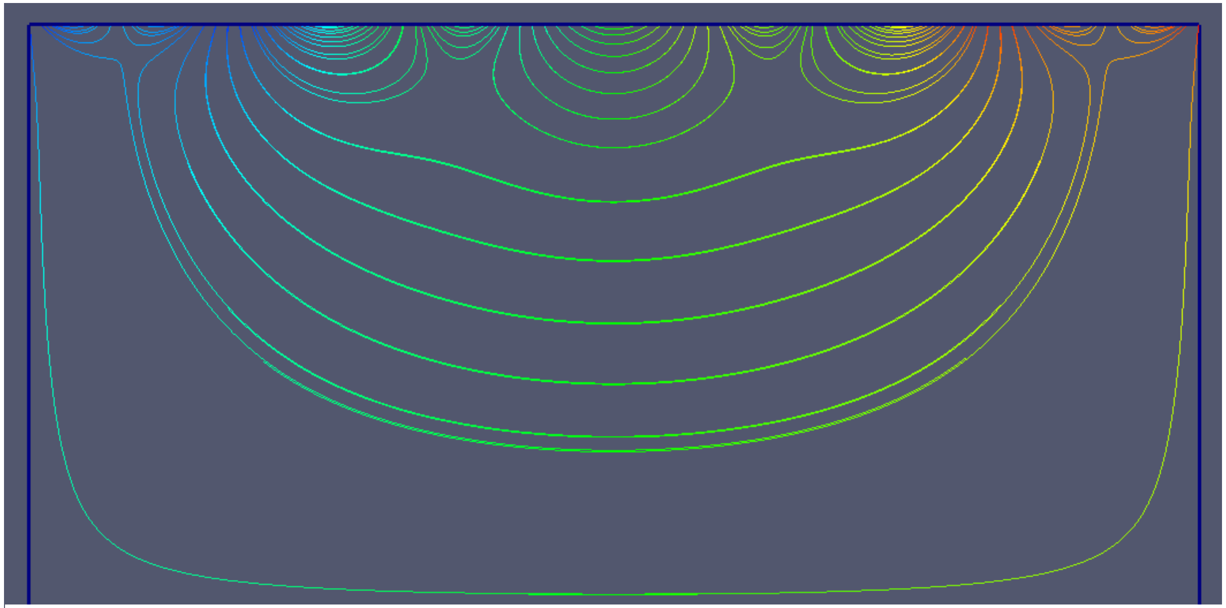
$$h(x, z_0) = z_0 + c' x + a' \sin(b' x)$$

$$\frac{\partial h}{\partial x} = 0$$



$$\frac{\partial h}{\partial x} = 0$$

$$\frac{\partial h}{\partial z} = 0$$



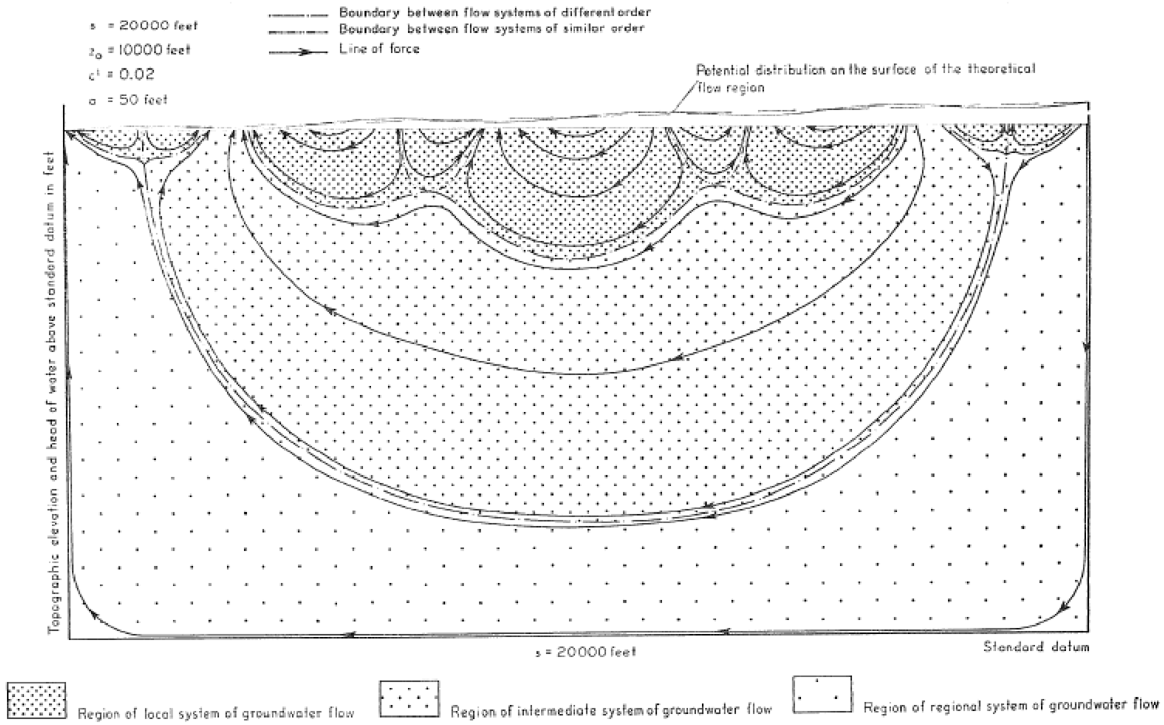
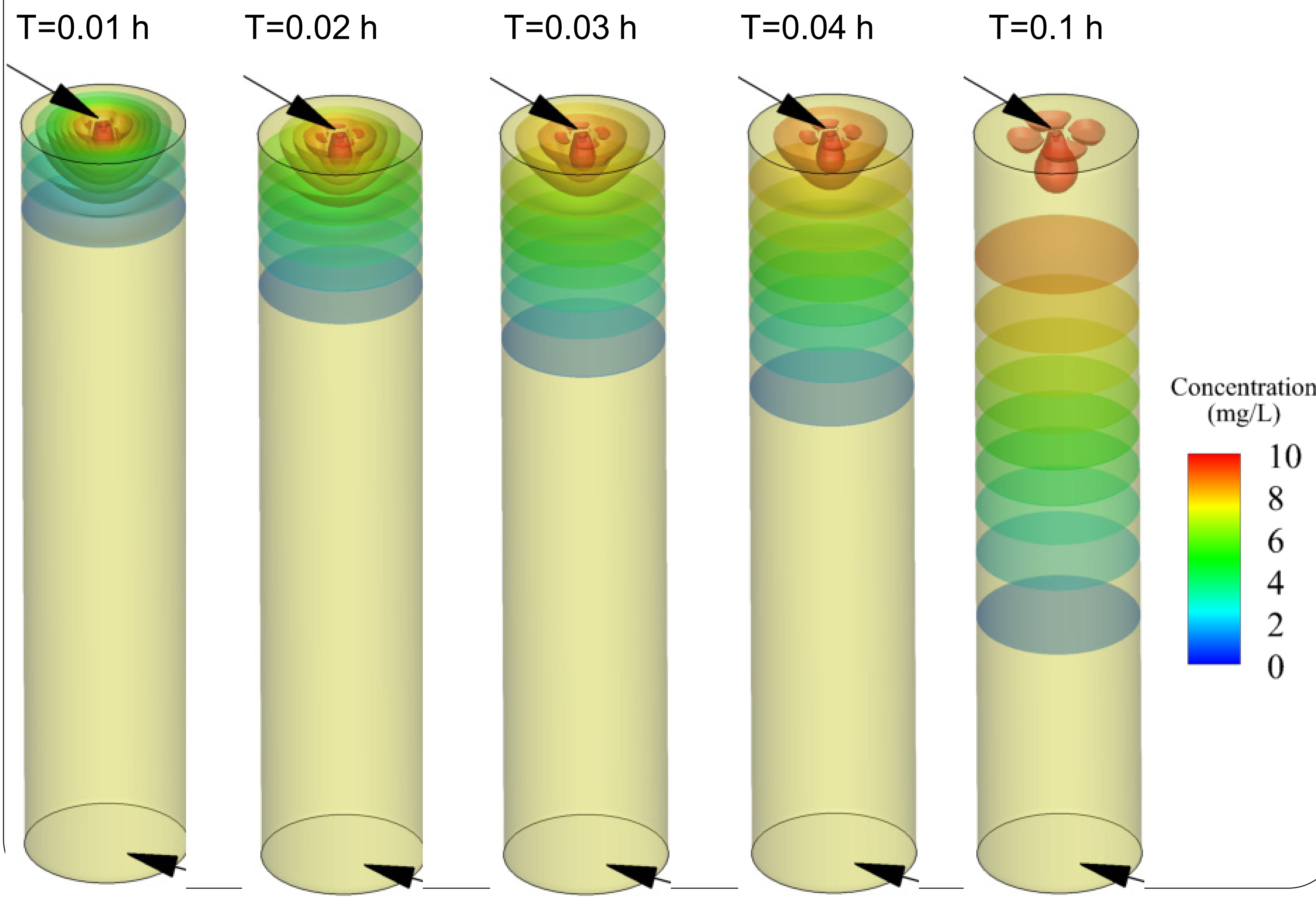
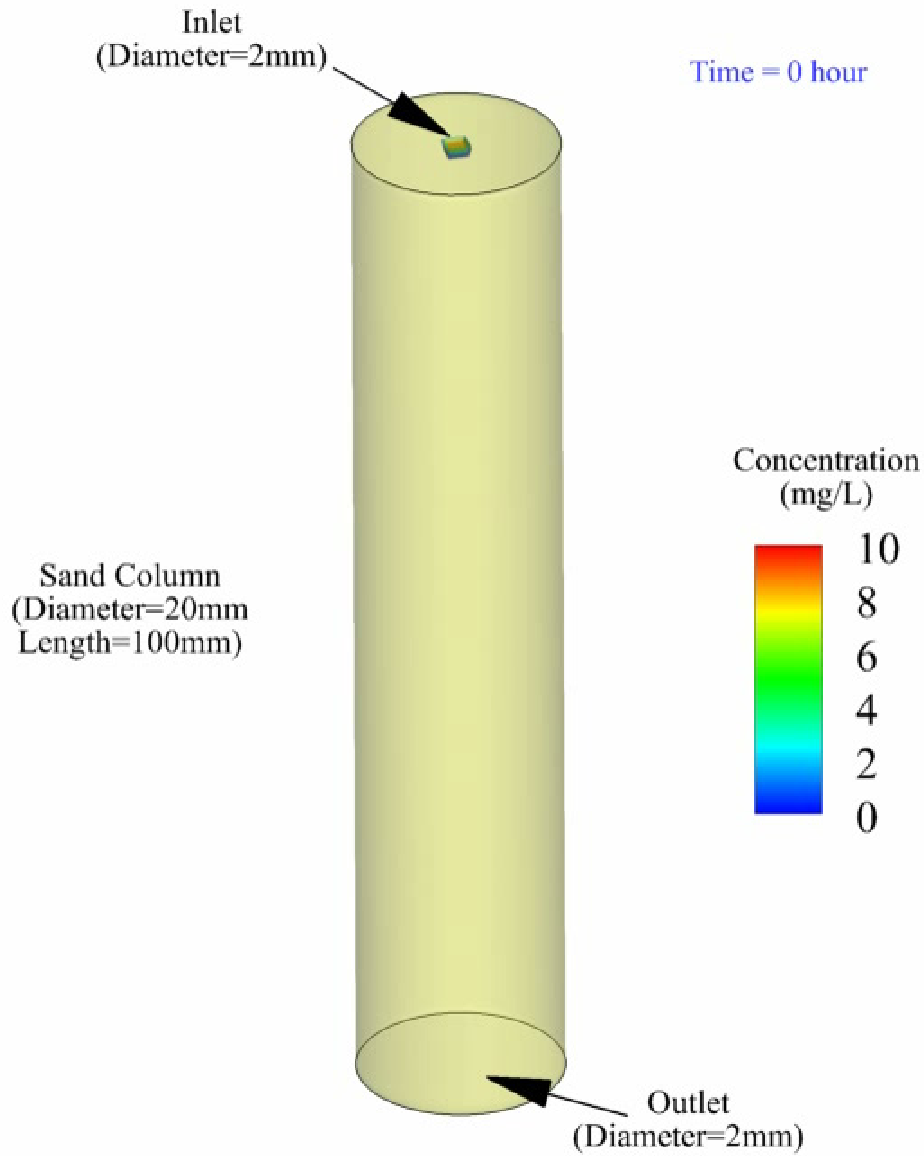


Fig. 3. Theoretical flow pattern and boundaries between different flow systems.

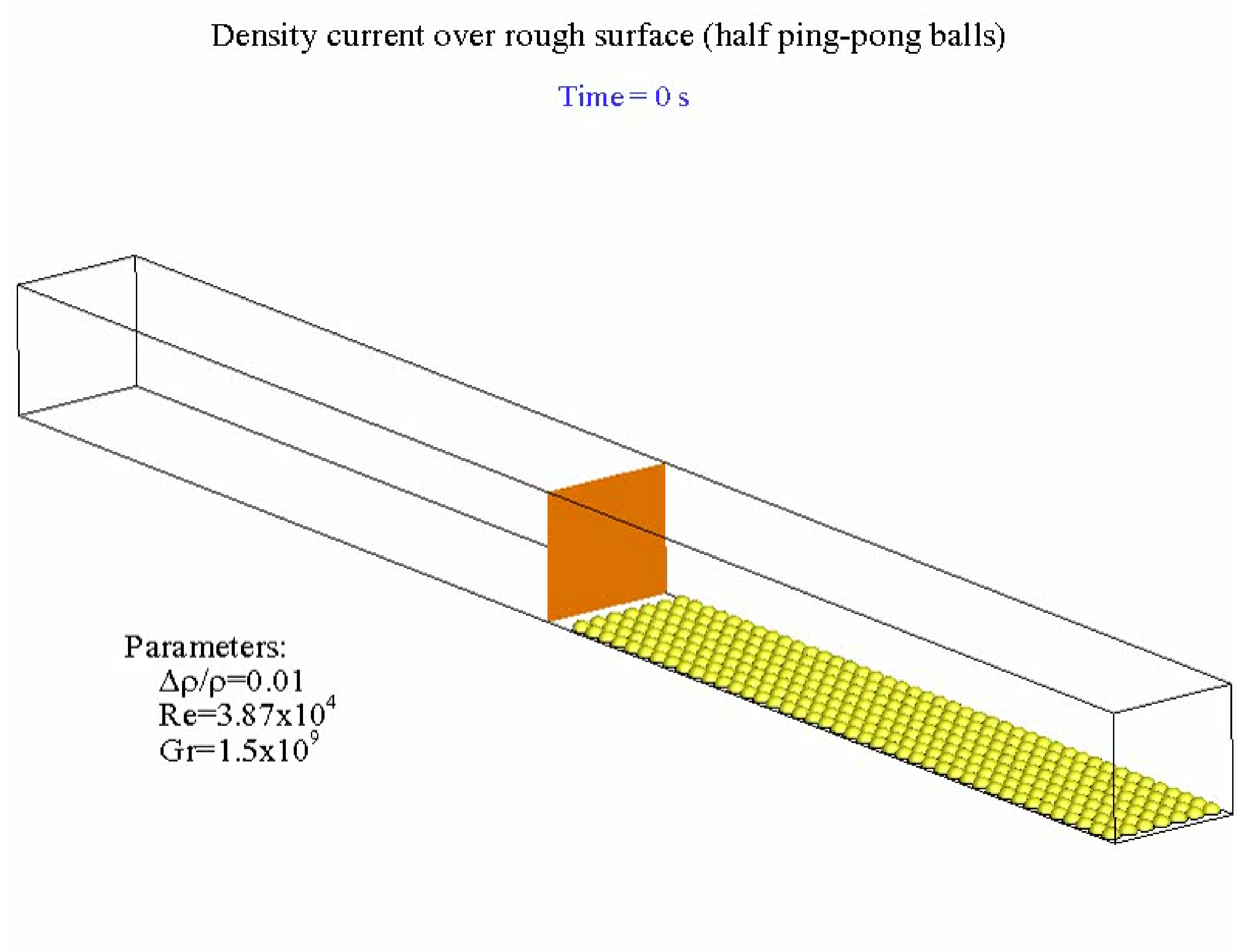
Nanoparticle Concentration in Sand Column Test



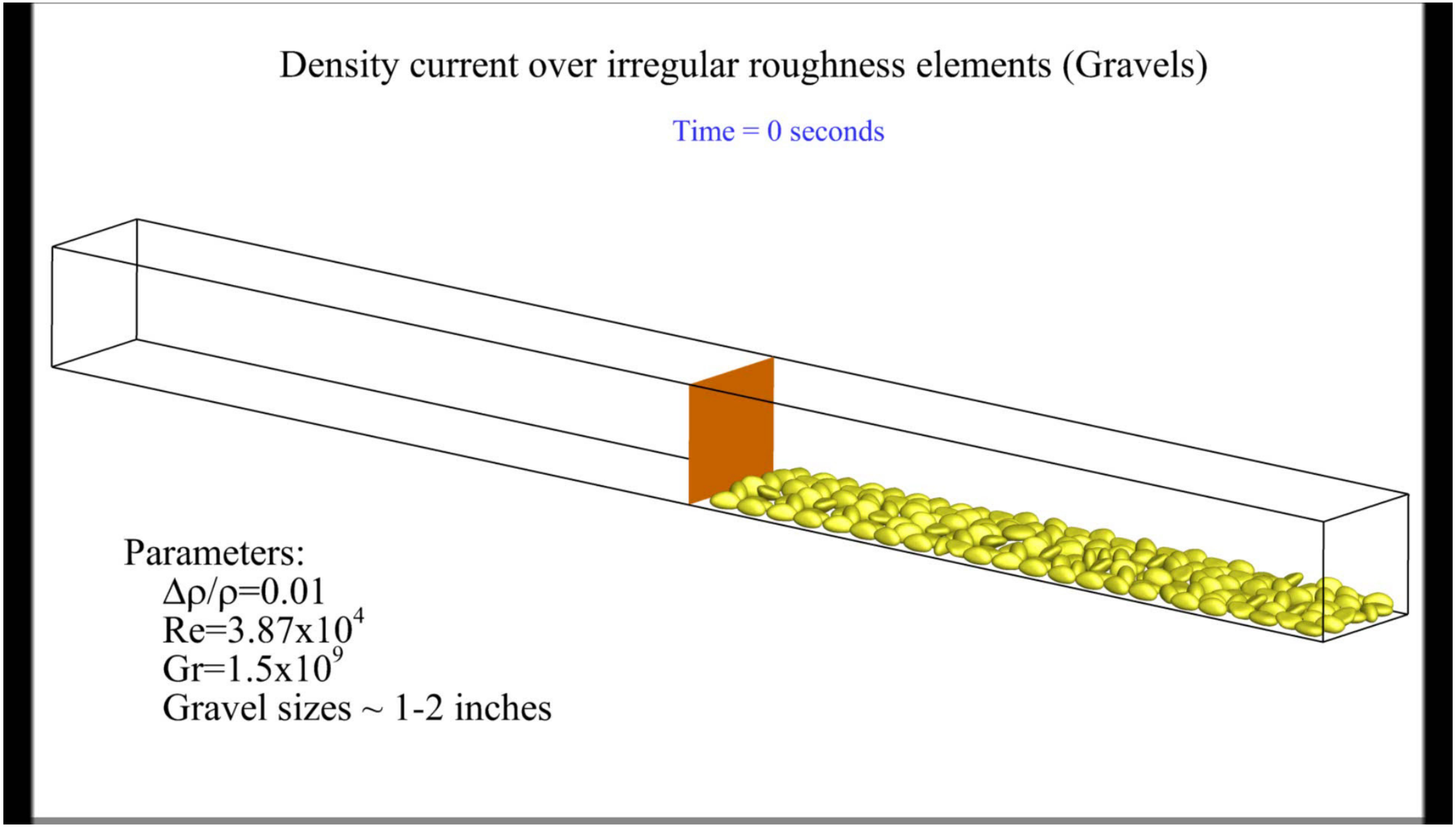
Nanoparticle Advection-Diffusion-Reaction in Porous Media



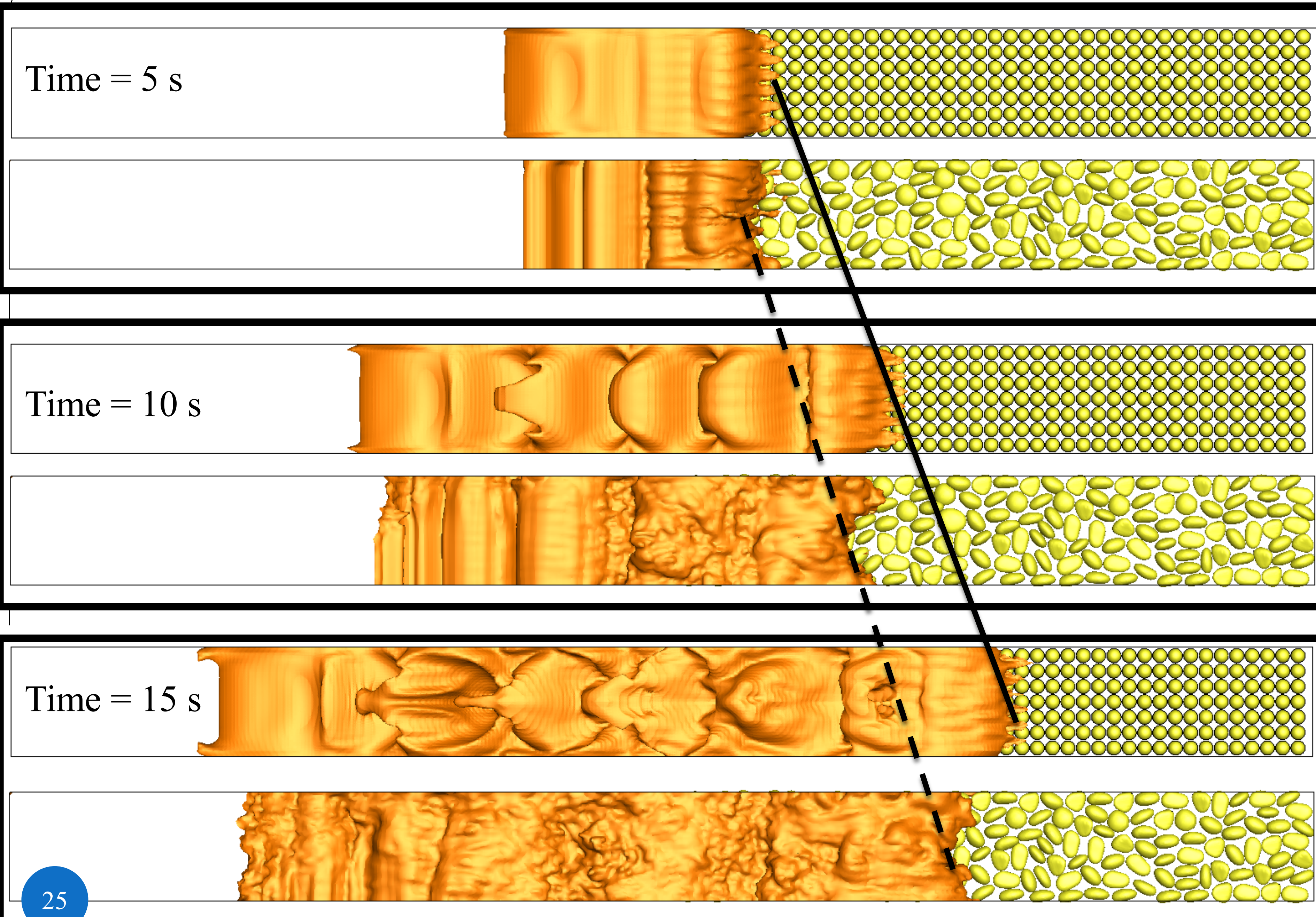
Buoyancy affected flows (gravity current over roughness and bedforms)



Jiang and Liu, 2012

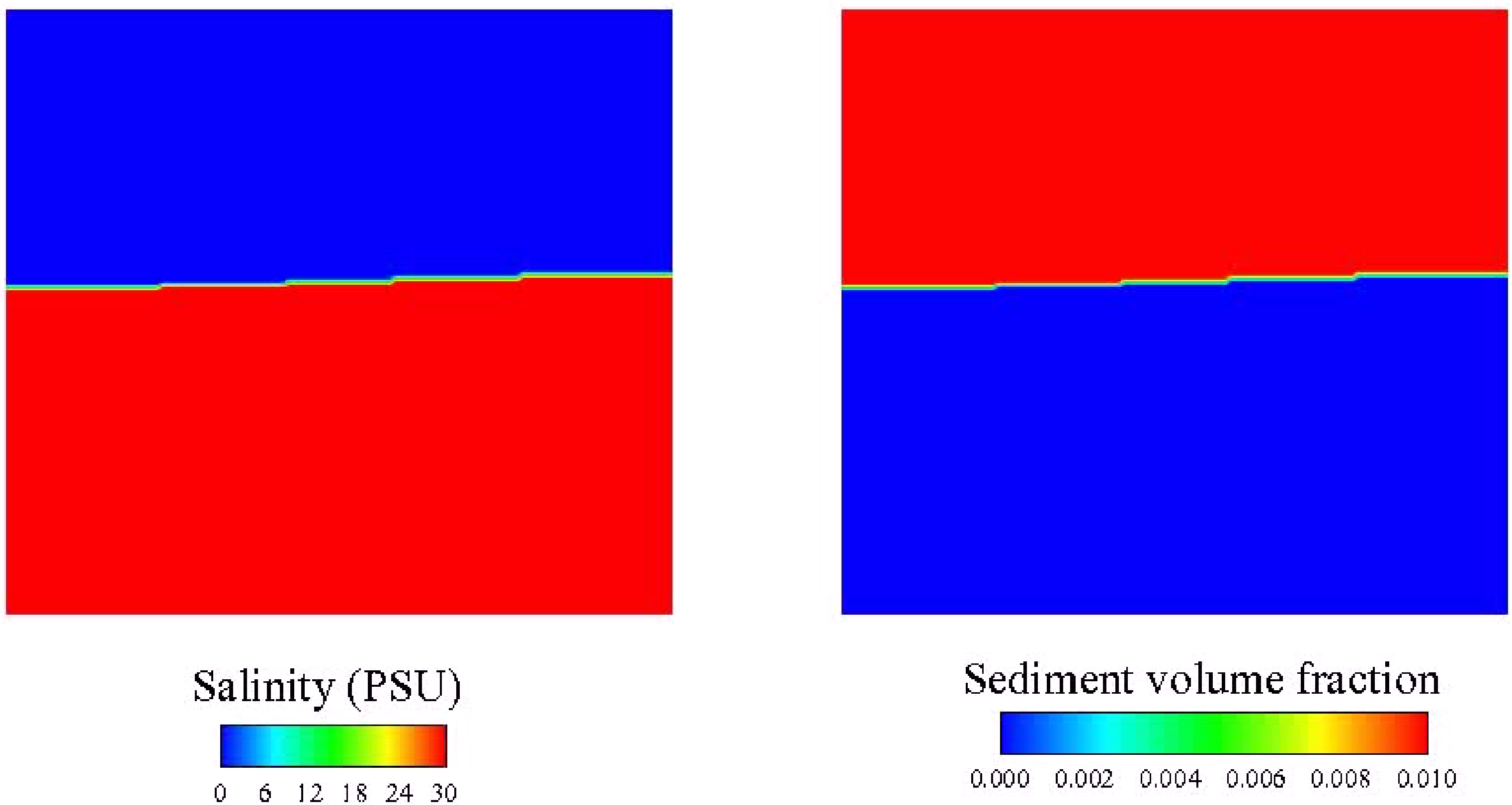


Jiang and Liu, 2012

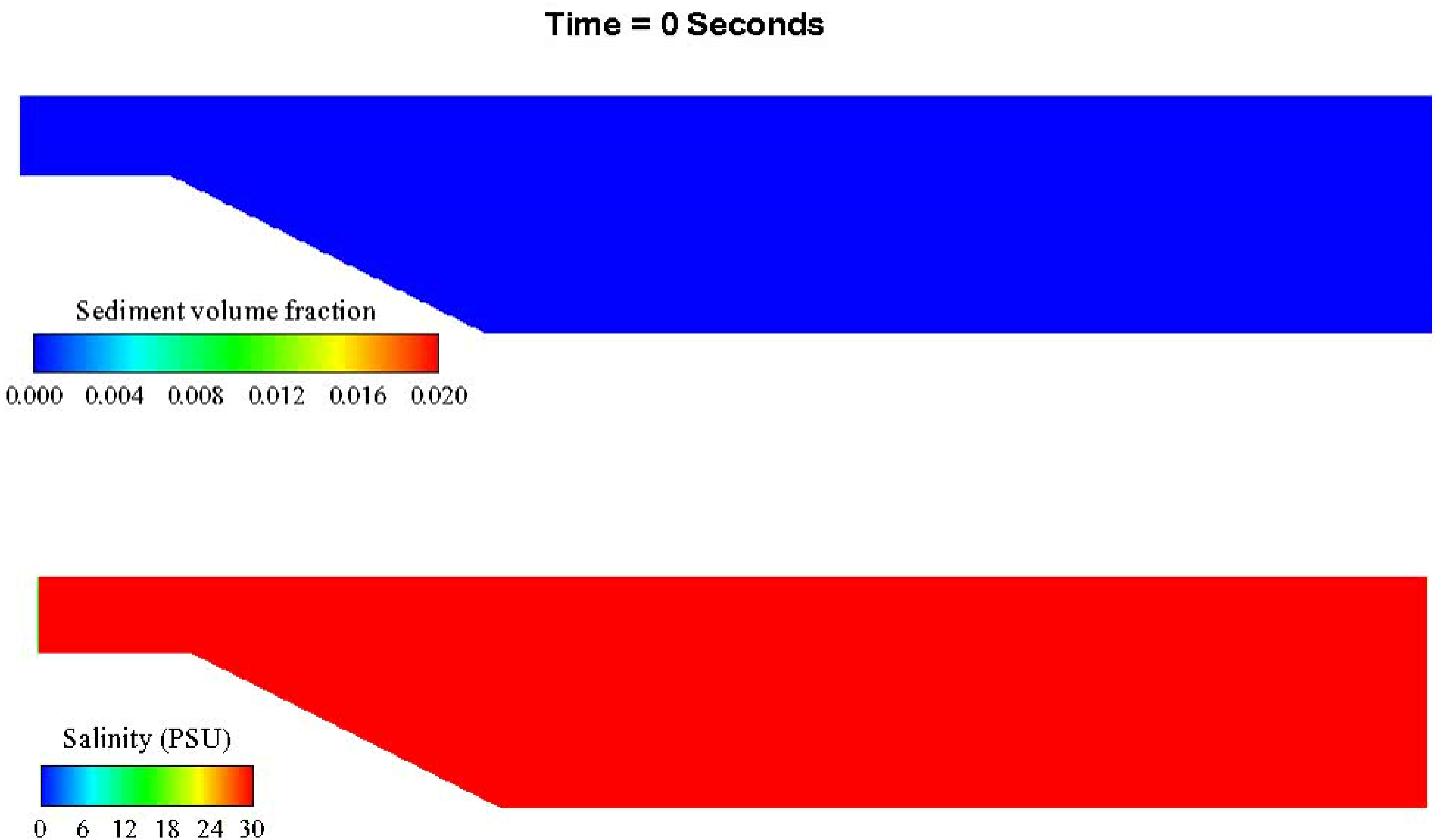

Time = 5 s
Time = 10 s
Time = 15 s

Suspended Sediment Layer over Salty Water

Time = 0 Seconds



Sediment laden plume discharging into a flume
(Case B2: Salty water in the flume. Both overflow and underflow appear.)



Particle Velocity (m/s)

2.200
1.650
1.100
0.5500
0.000
Max: 1.720
Min: 0.7353

Particle density=2.5 g/cm^3
Particle diameter=0.5 mm
Injection velocity=1.72 m/s
Orifice diameter=5.08 mm
Initial volume fraction=30%

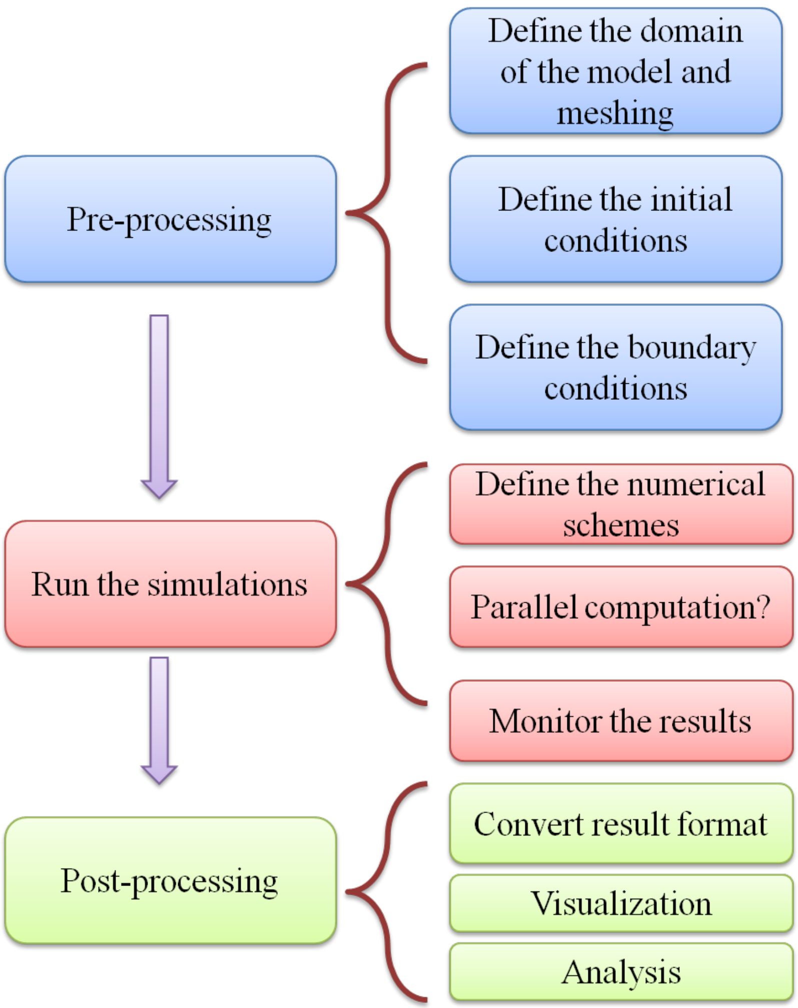
Vorticity Isosurface (1/s)

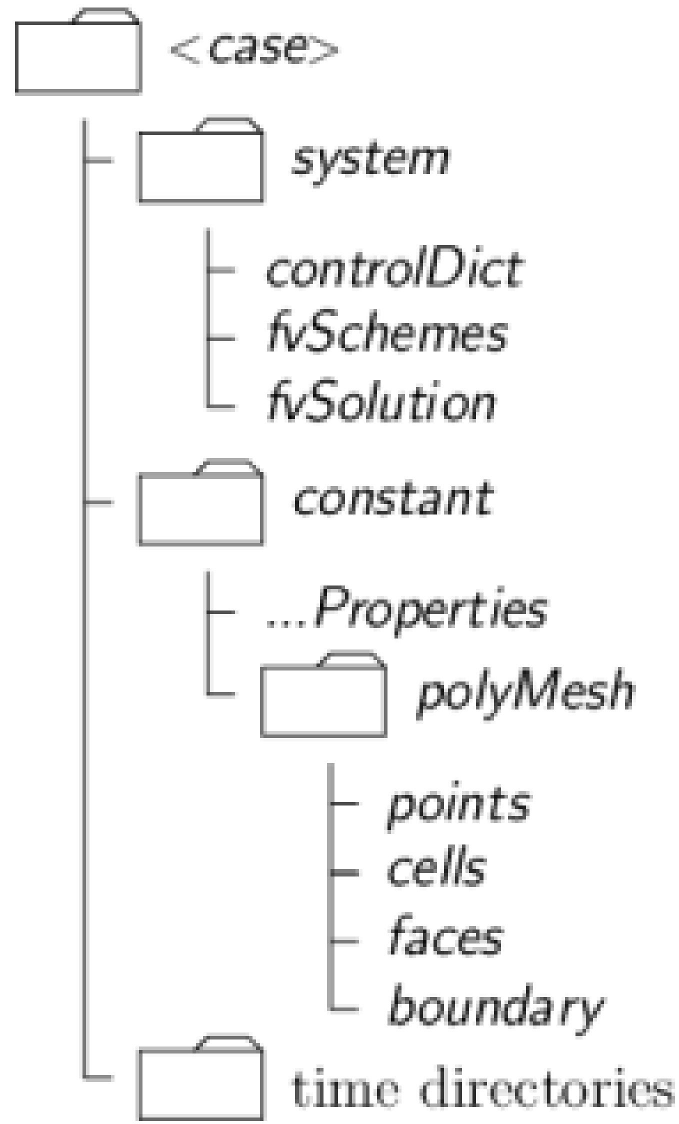
300
Max: 3133.
Min: 4.933e-09

Time=0.000 s

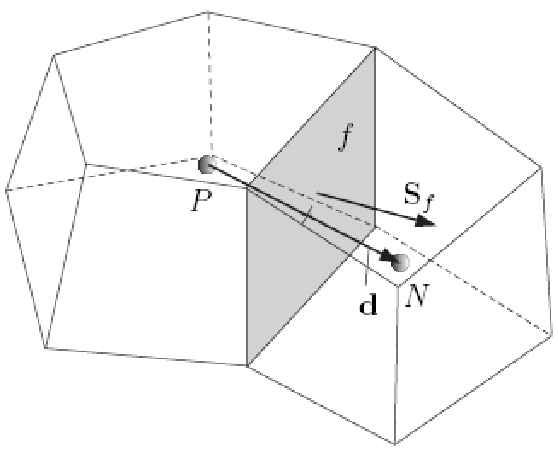
Ruo-Qian Wang
Dept. of Civil and Environ. Eng., MIT
rqwang@mit.edu

Basic steps for a modeling task





- Numerical features of OpenFOAM
  - Finite volume method
  - Also has Lagrangian particle tracking, finite element method, finite area method, etc.
  - Unstructured meshes (both fixed and deforming)



- Numerical features of OpenFOAM
  - Solve fluid dynamics equations using the segregated pressure methods (e.g., PISO, SIMPLE, SIMPLEC, etc.)
  - Can be 1D, 2D, and 3D based on the mesh and boundary conditions
  - Automatic parallel computation based on domain decomposition and MPI
  - Automatic descretizations of the equations

So how are equations solved in OpenFOAM?

Equations are essentially the group of operations on fields

Mathematical language:
Partial differential equation (PDE)

$$\frac{\partial k}{\partial t} + \nabla\bullet(\mathbf{u}k) - \nabla\bullet[(\nu+\nu_t)\nabla k] = \nu_t\left[\frac{1}{2}(\nabla\mathbf{u}+\nabla\mathbf{u}^T)\right]^2 - \frac{\epsilon_o}{k_o}k$$

Pseudo-natural language in OF

```
solve
(
   fvm::ddt(k)
 + fvm::div(phi, k)
 - fvm::laplacian(nu() + nut, k)
== nut*magSqr(symm(fvc::grad(U)))
 - fvm::Sp(epsilon/k, k)
);
```

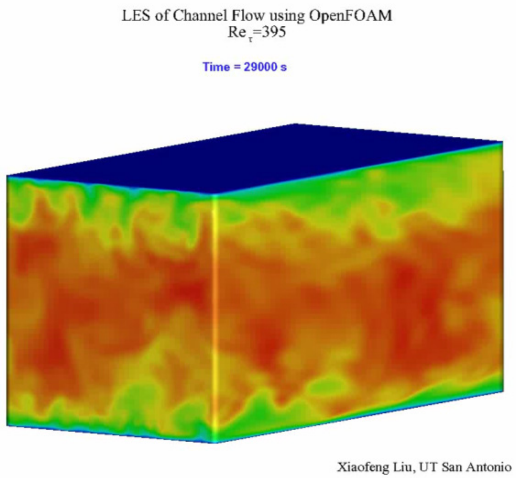
Linear system after discretization

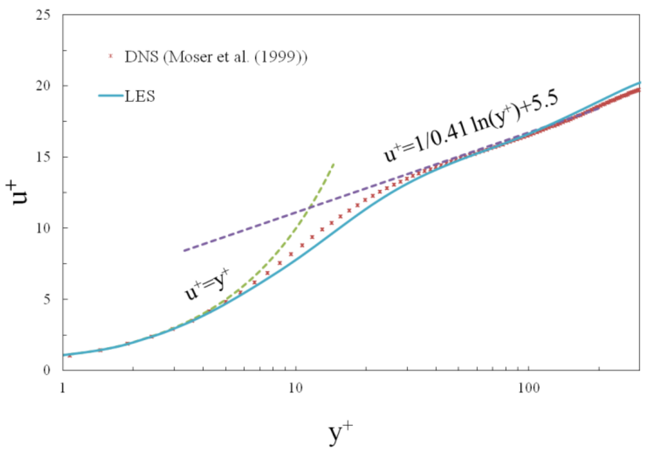
$$[A]\,[x] = [b]$$

- Numerical features of OpenFOAM
  - Descretizations schemes (*system/fvSchemes*)
    - Spatial: upwind, central, TVD, NVD, etc.
    - Temporal: Euler, backward, CN, etc.
  - Linear system solvers (*system/fvSolution*)
    - PBiCG (asymmetric matrix)
    - PCG (symmetric matrix)
    - GAMG (multi-grid method)
    - Smooth solver and diagonal solver
    - …

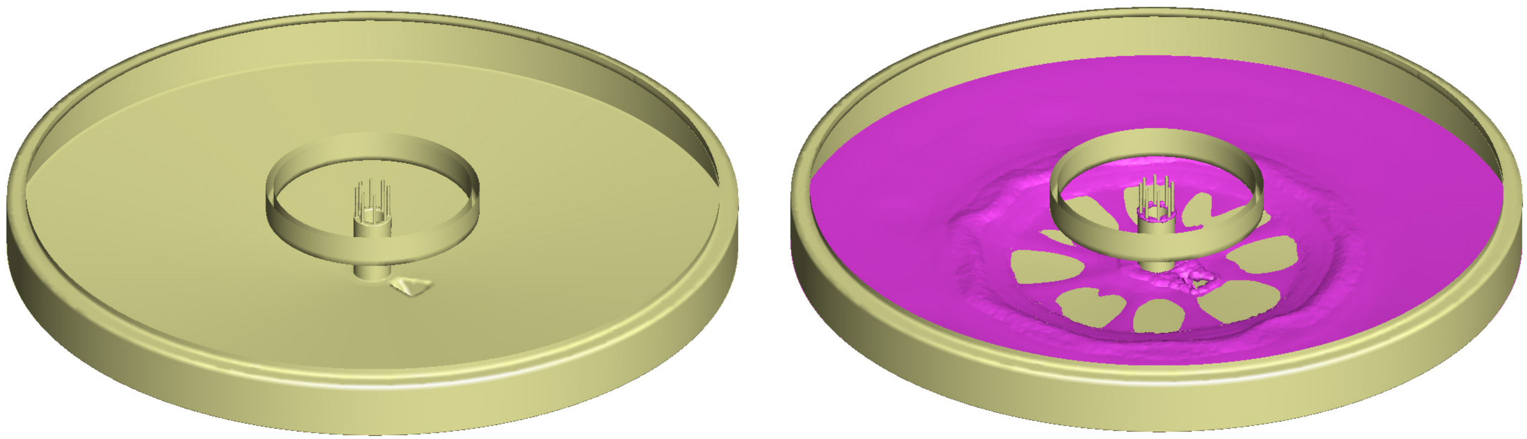
Modeling capabilities of OpenFOAM

- Incompressible and compressible flows
- Turbulence models
  - Laminar
  - RANS: Reynolds Averaged Navier-Stokes
  - LES: Large Eddy Simulations
  - DES: Detached Eddy Simulations
  - DNS: Direct Numerical Simulations



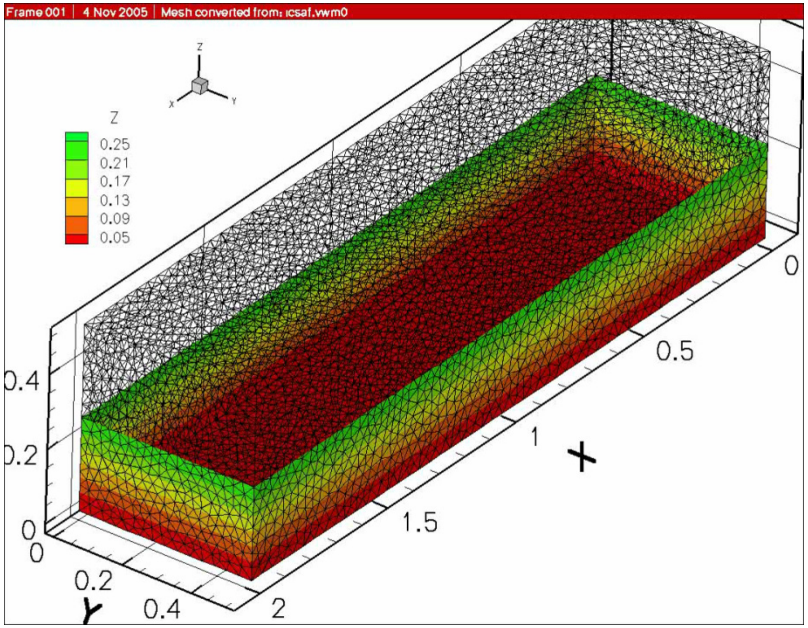


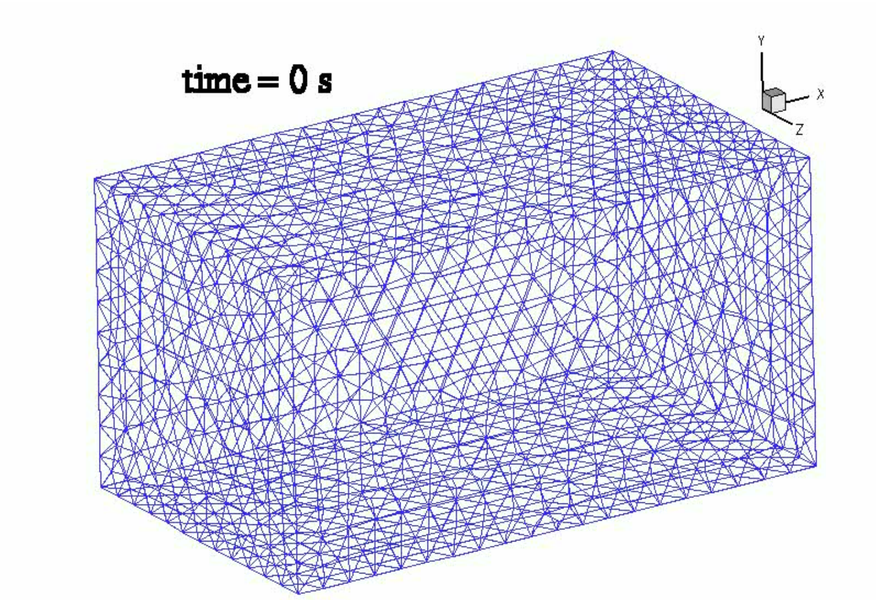
- Modeling capabilities of OpenFOAM
  - Multiphase flows
    - Free surface flows
    - Buoyant flows: due to sediment, temperature, salinity, etc.
  - Transport and rheological models
    - Newtonian
    - Non-Newtonian



(Liu and García, 2010)

- Modeling capabilities of OpenFOAM
  - Dynamic mesh
    - To model motion of the domain or object
    - Various method to deform the mesh
    - Can be used to generate a mesh
  - Immersed boundary method

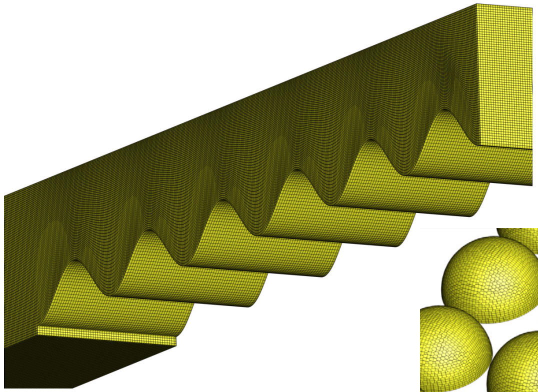


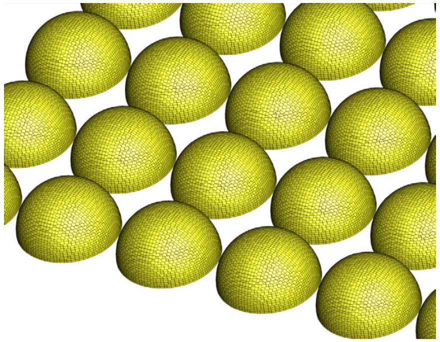


Credit: Jorge D. Abad, U. Pitt.

Pre-processing capabilities of OpenFOAM

- Mesh generation
  - Generic tools:
    - *blockMesh*
    - *snappyHexMesh*





- Mesh conversion
  - Convert meshes from/to other formats
    - e.g., Anysis, Fluent, GMESH, Gambit
- Mesh manipulation
  - Rotation, translation, extrusion, split, join, etc.

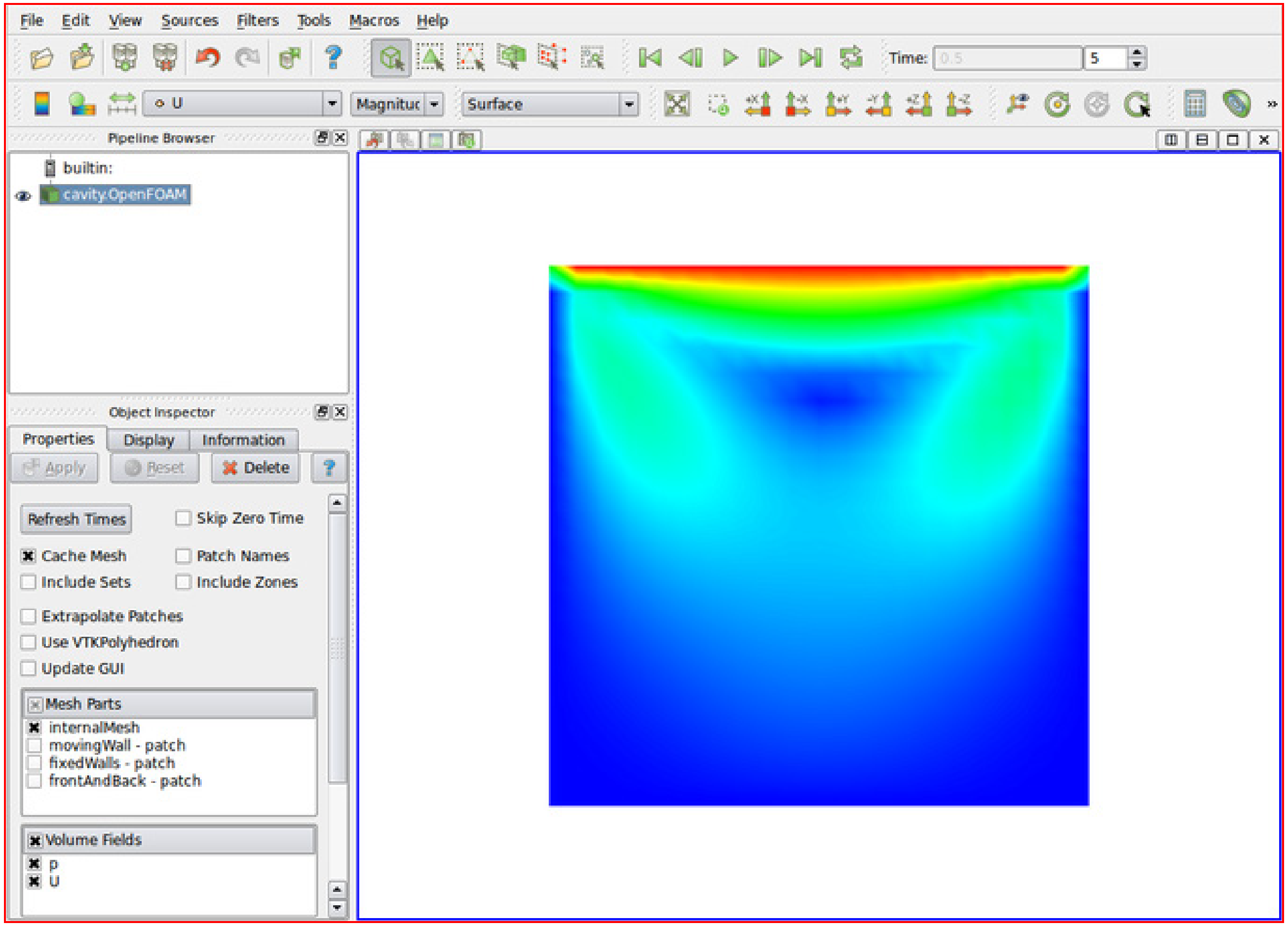
Pre-processing capabilities of OpenFOAM

- Set up initial conditions
  - Modify the files directly, or
  - Generic tool: *setFields*

- Set up boundary conditions
  - Modify the files directly, or
  - Use tools, or
  - Programming by yourself

```
dimensions      [0 1 -1 0 0 0 0];

internalField   uniform (0 0 0);

boundaryField
{
    movingWall
    {
        type            fixedValue;
        value           uniform (1 0 0);
    }

    fixedWalls
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }

    frontAndBack
    {
        type            empty;
    }
}
```

- Post-processing capabilities of OpenFOAM
  - Directly load into ParaView
    - ParaView is open source and free



- Post-processing capabilities of OpenFOAM
  - Convert OpenFOAM results to other formats
    - Generic tools:
      - *foamToFluent*
      - *foamToFieldView*
      - *foamToVTK*
      - *foamToTecplot360*

Demonstrations

CSDMS 2013 Meeting

Modeling of Earth Surface Dynamics and Related Problems Using OpenFOAM®

Xiaofeng Liu, Ph.D., P.E.
Assistant Professor
Department of Civil and Environmental Engineering
University of Texas at San Antonio, Texas
http://engineering.utsa.edu/~xiaofengliu