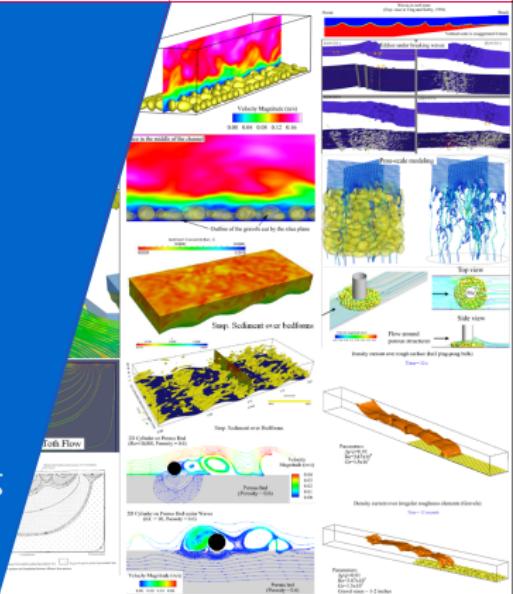




Chapter 2, Part I: Introduction to OpenFOAM®

Xiaofeng Liu, Ph.D., P.E.
Assistant Professor
Department of Civil and Environmental Engineering
Pennsylvania State University
xliu@engr.psu.edu



Level of users

An overview of OpenFOAM

A brief introduction of OpenFOAM

Sample Applications of OpenFOAM

Modeling capabilities of OpenFOAM

Pre-processing capabilities of OpenFOAM

Runtime modification

Post-processing capabilities of OpenFOAM

History of OpenFOAM

Our use of OpenFOAM

Summary

- ▶ Beginner level:
 - A brief introduction of OpenFOAM®
 - Demonstration of OpenFOAM® usages
 - Installation of OpenFOAM®
 - Hands-on exercises for the basics
 - Detailed walk-through of the code
 - ...
- ▶ Intermediate and advanced levels (*later*):
 - Demonstration of OpenFOAM® development
 - Solver
 - Boundary conditions
 - Tools
 - Hands-on exercise
 - Discussion of specific applications

Before we start:

4 / 45

- ▶ What you should know before you get your hands dirty?
- ▶ What we will do during this chapter?
- ▶ What you can do after this chapter?

- ▶ A brief introduction of OpenFOAM®
- ▶ Sample applications of OpenFOAM®
- ▶ Installation of OpenFOAM®
- ▶ Hands-on exercises for the basics
- ▶ Detailed walk-through of the code

A brief introduction of OpenFOAM®

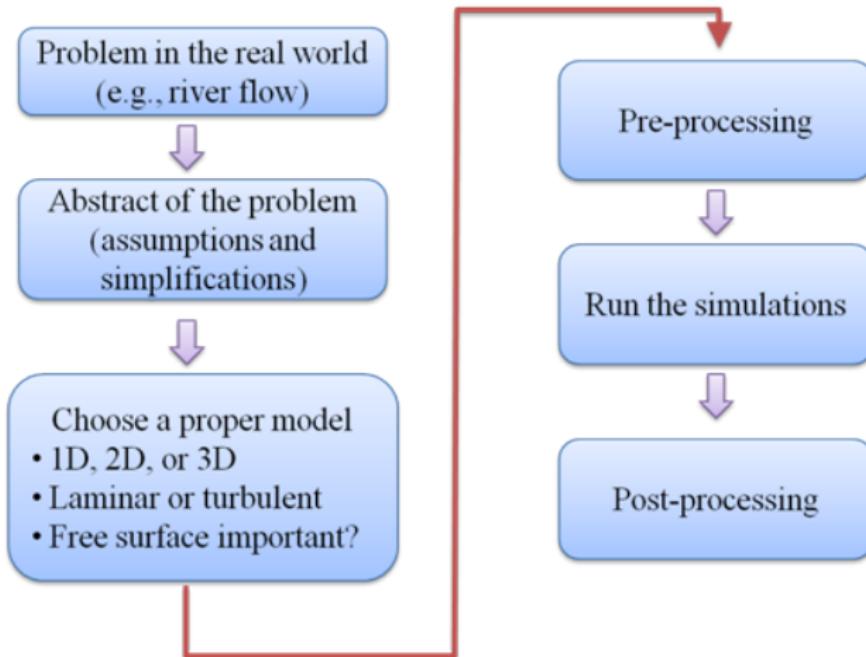
6 / 45

- ▶ OpenFOAM is an open source computational fluid dynamics platform written in C++
- ▶ FOAM stands for “Field Operation And Manipulation?
- ▶ OpenFOAM is not limited to fluid dynamics
 - It is a generic modeling platform
 - It can be used to solve (m)any equations

Basic steps for a modeling task

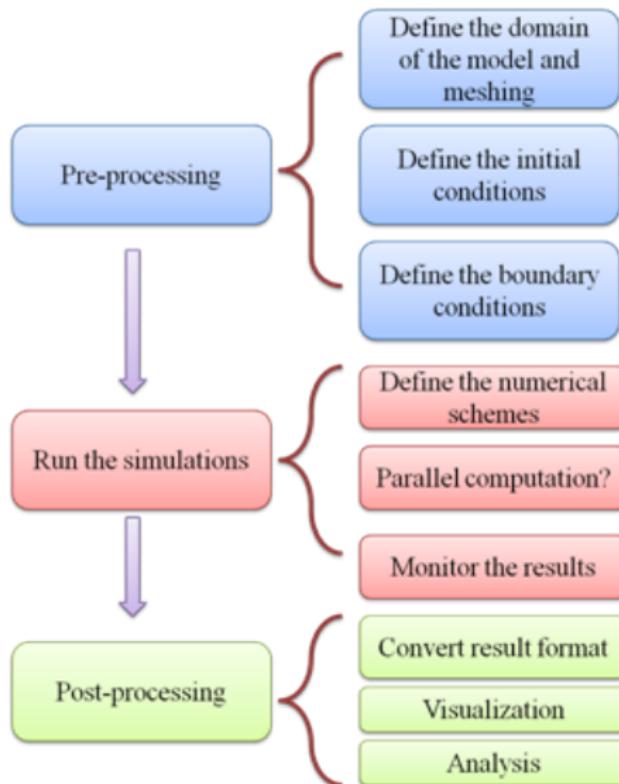
7 / 45

► Basic steps for a modeling task



Basic steps for a modeling task

► Basic steps for a modeling task

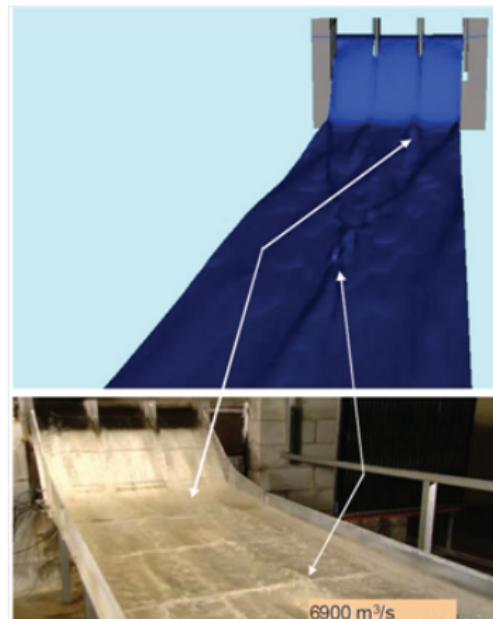


Sample Applications of OpenFOAM®

9 / 45

Sample Applications of OpenFOAM

- ▶ Free surface flows: Spillway discharge

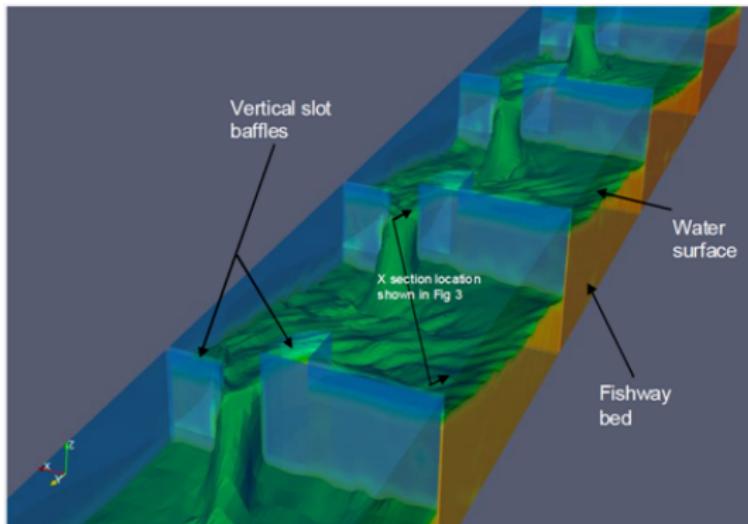


Franz Jacobsen, Application of OpenFOAM® for designing hydraulic water structures

Sample Applications of OpenFOAM®

10 / 45

- ▶ Free surface flows: Fish passages

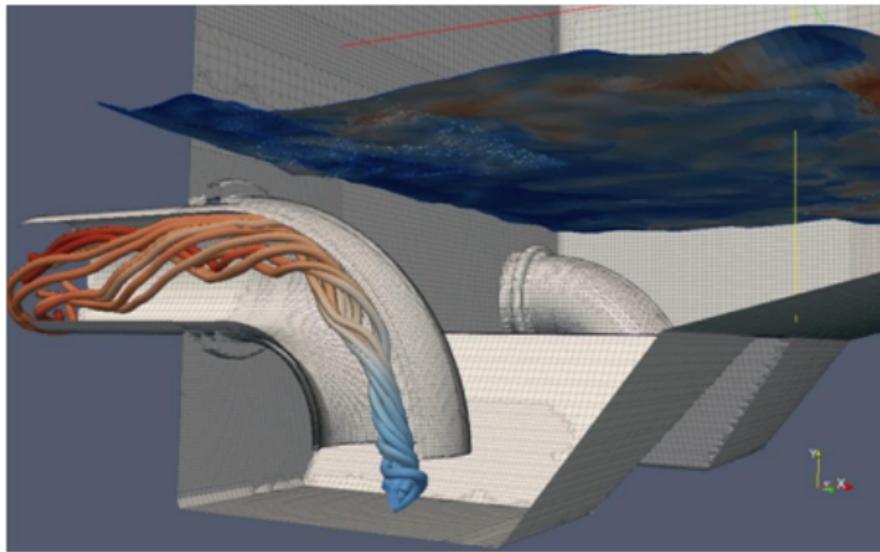


Franz Jacobsen, Application of OpenFOAM® for designing hydraulic water structures

Sample Applications of OpenFOAM®

11 / 45

- ▶ Pumping station



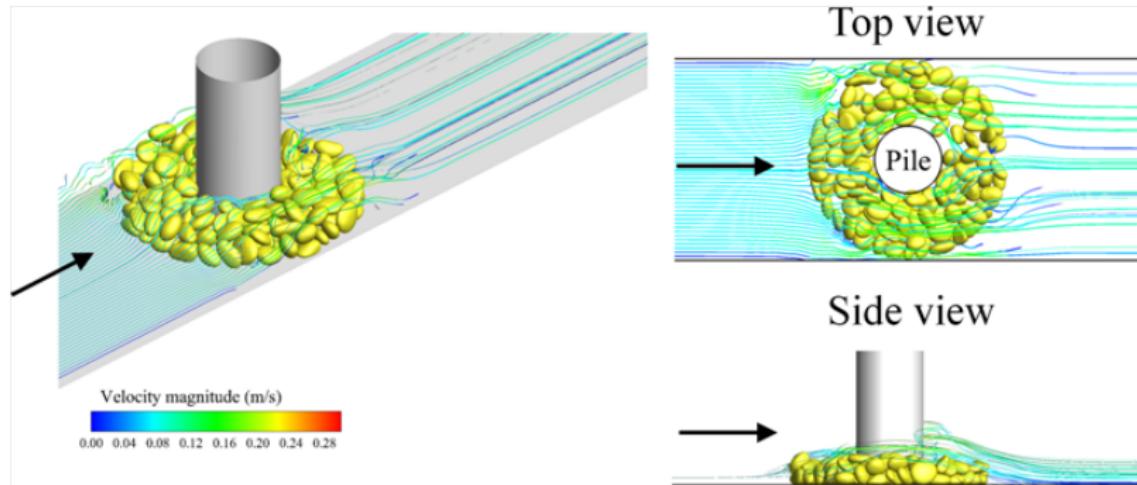
Franz Jacobsen, Application of OpenFOAM® for designing hydraulic water structures

- ▶ River flow under a bridge
- ▶ <http://www.youtube.com/watch?v=BqKN5QwGPB4>
- ▶ <http://www.edenvaleyoung.com>

- ▶ River flow under a bridge
- ▶ <http://www.youtube.com/watch?v=BqKN5QwGPB4>
- ▶ <http://www.edenvaleyoung.com>

Sample Applications of OpenFOAM®

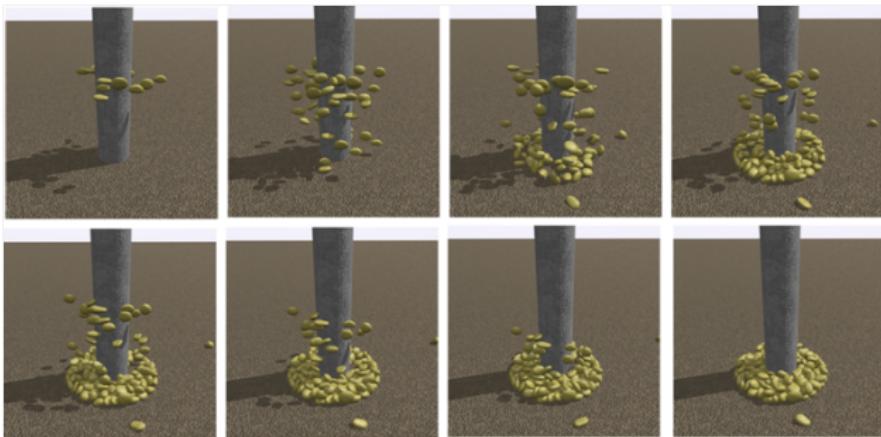
14 / 45



Liu, 2012

Sample Applications of OpenFOAM®

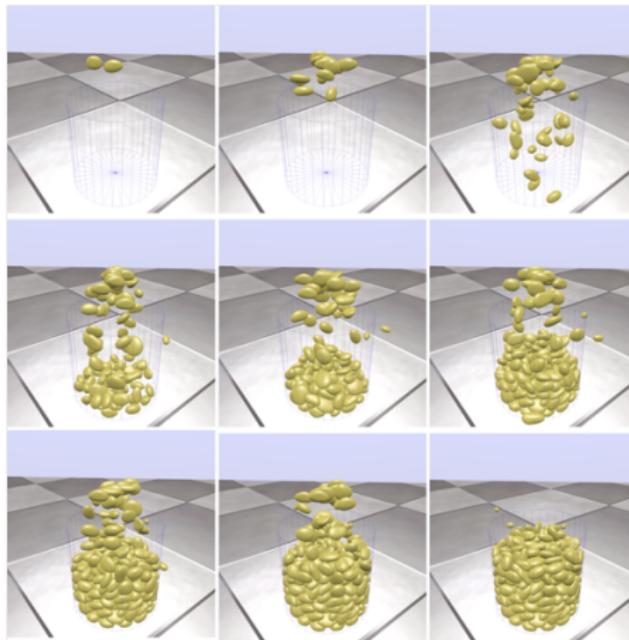
15 / 45



Liu, 2012

Sample Applications of OpenFOAM®

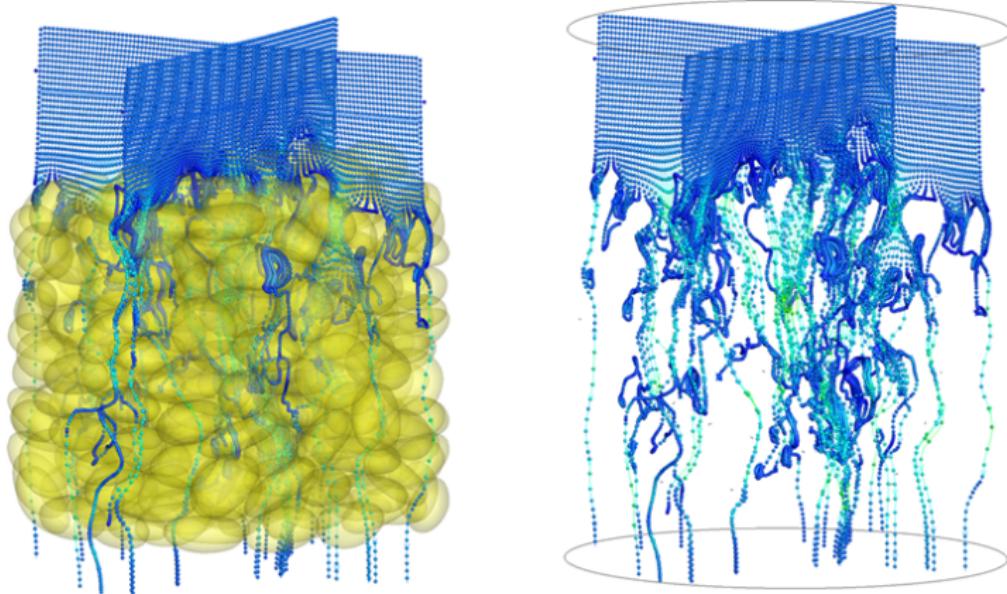
16 / 45



Sinir and Liu, 2012

Sample Applications of OpenFOAM®

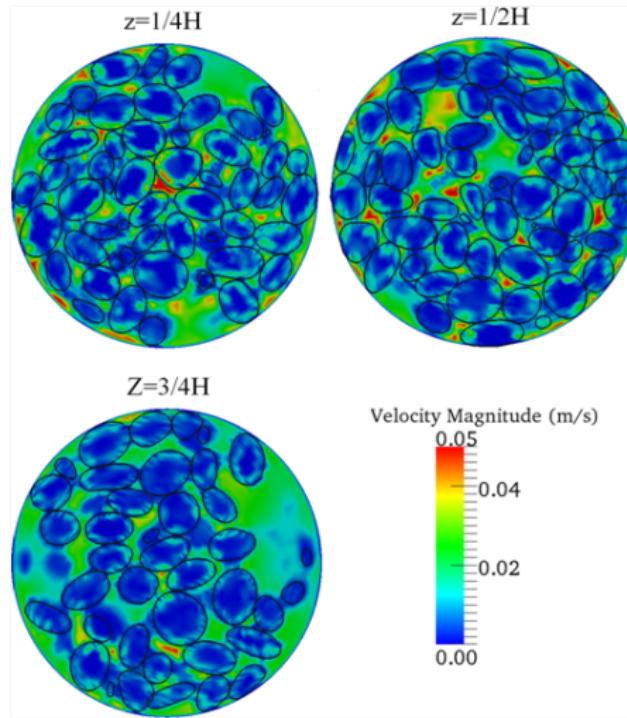
17 / 45



Sinir and Liu, 2012

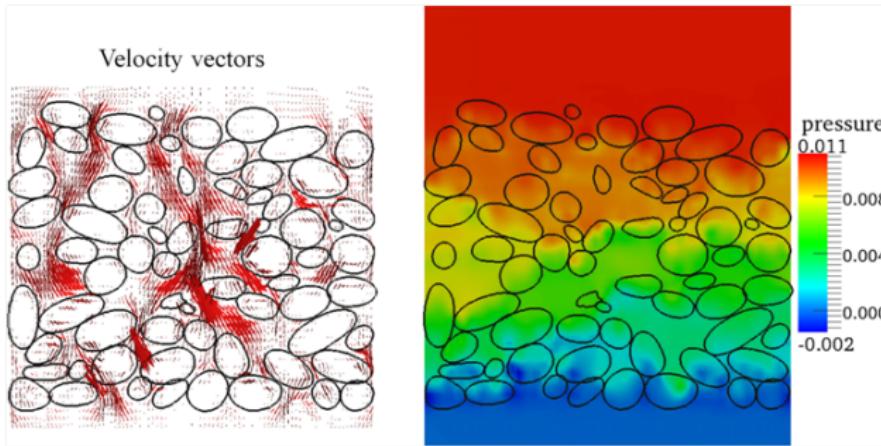
Sample Applications of OpenFOAM®

18 / 45



Sinir and Liu, 2012

Sample Applications of OpenFOAM®



Sinir and Liu, 2012

Numerical features of OpenFOAM®

20 / 45

- ▶ 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®

21 / 45

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

- ▶ Discretizations schemes chosen at run-time
 - Spatial: upwind, central, TVD, NVD, etc.
 - Temporal: Euler, backward, CN, etc.
- ▶ Linear system solvers
 - PBiCG (asymmetric matrix)
 - PCG (symmetric matrix)
 - GAMG (multi-grid method)
 - Smooth solver and diagonal solver
 - ...

Modeling capabilities of OpenFOAM®

23 / 45

- ▶ 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®

24 / 45

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

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
 - ...
- ▶ ...

- ▶ Lagrangian particle tracking

- Particles are modelled in Lagrangian framework: each one is tracked for its motion in the fluid as well as inter-particle collisions
- Equation of motion for Lagrangian particles

$$m_p \frac{d\mathbf{v}_p}{dt} = \frac{1}{2} \rho |\mathbf{v} - \mathbf{v}_p| (\mathbf{v} - \mathbf{v}_p) \frac{d_p^2 \pi}{4} C_D + m_p \mathbf{g} \frac{\rho_I - \rho}{\rho_I} + \mathbf{F}_{\nabla p} \quad (1)$$

where

m_p is the particle mass

\mathbf{v}_p is the particle velocity

\mathbf{v} is the fluid velocity

d_p is particle diameter

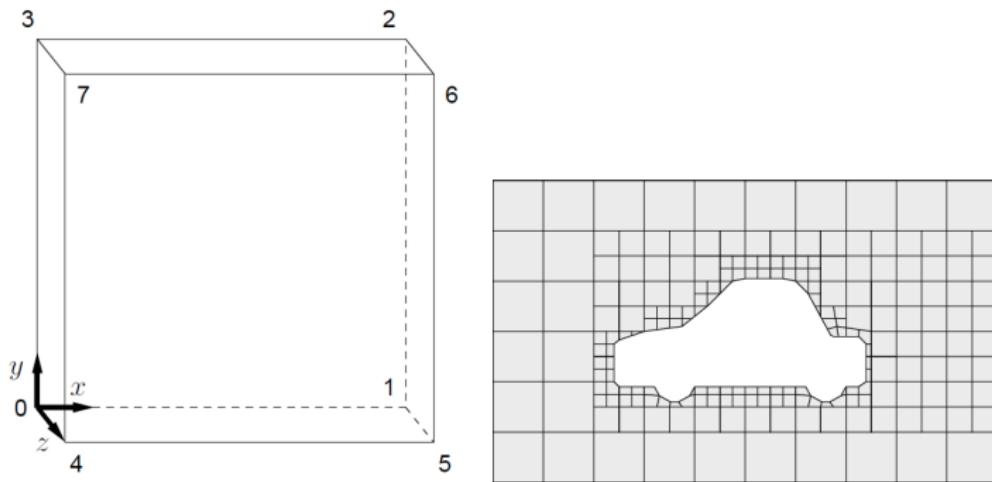
C_D is the drag coefficient

$\mathbf{F}_{\nabla p}$ is the force on the particle due to the fluid pressure gradient

Pre-processing capabilities of OpenFOAM®

27 / 45

- ▶ 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.



- ▶ 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

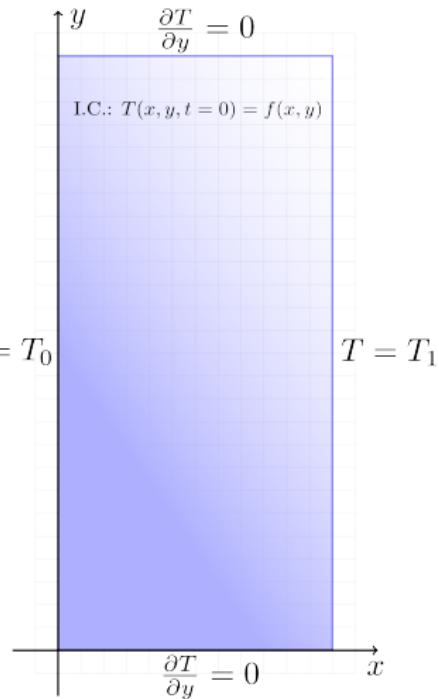


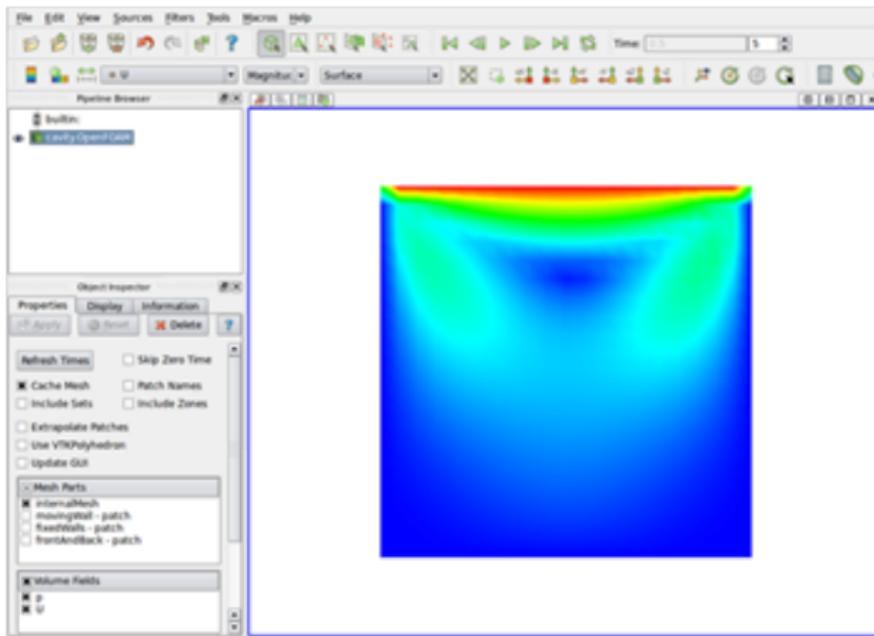
Figure: IC and BC for a problem

Runtime modification

- ▶ Some of the features are runtime modifiable
 - Such as:
 - Discretization schemes
 - Time step sizes
 - Result output frequency
 - Modification done through file updating
 - OpenFOAM® will check the file during simulation
- ▶ Runtime modification switch is in file case/system/controlDict

```
...  
writeCompression uncompressed;  
  
timeFormat      general;  
  
timePrecision   6;  
  
runTimeModifiable yes;  
...
```

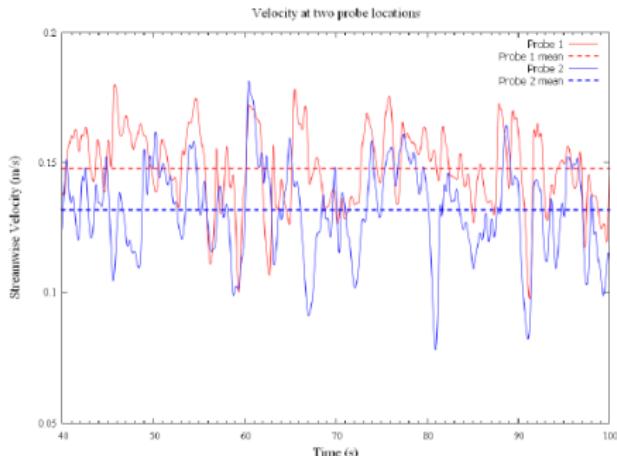
- ▶ Directly load into ParaView
 - ParaView is open source and free
 - Based on the Visualization Toolkit (VTK)



- ▶ Convert OpenFOAM® results to other formats
 - Generic tools: foamToFluent, foamToFieldView, foamToVTK, foamToTecplot360
- ▶ User programming: e.g., I have written oneFieldToTecplot



- ▶ Run-time post-processing through function objects (more on this later)
 - probe at certain location
 - averaging, min, max
 - surface interpolation, iso-surface, streamlines
 - calculate forces
 - system call
 - etc.



History of OpenFOAM®



From David A. Boger, 6th OpenFOAM Workshop, PennState, June, 2011

what is
OpenCFD Ltd.?

- Established in 2004 by Henry Weller, coinciding with the release of OpenFOAM under general public license
- From www.openfoam.com/about: “OpenCFD® produces the OpenFOAM® open source CFD toolbox and documentation and distributes it through this web site. OpenCFD provides contracted developments, support and training for users of OpenFOAM”

From David A. Boger, 6th OpenFOAM Workshop, PennState, June, 2011

what is the Extend Project?

- Originated by Wikki, Ltd. (Hrv Jasak et al.) in 2004
- Core group
 - Hrv Jasak, Hakan Nilsson, Bernhard Gschaider, Henrik Rusche, Holger Marschall, Martin Beaudoin
- From www.extend-project.de: "The goal of the Extend-Project is to open the OpenFOAM® CFD toolbox to **community contributed extensions** in the spirit of the OpenSource development model."

From David A. Boger, 6th OpenFOAM Workshop, PennState, June, 2011

what's the difference?

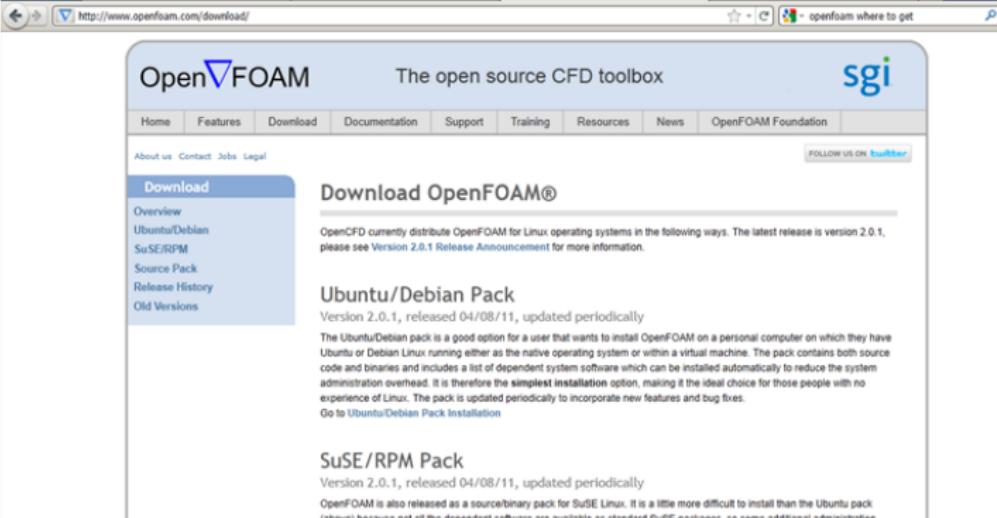
1.6-ext adds extra functionality relative to 1.6, but lags behind core functionality added to 1.7. Some notable extensions:

- **generalized grid interface (GGI)**
 - Martin Beaudoin, Hrv Jasak
 - turbomachinery stage interface
- **radial basis functions (RBF)**
 - Frank Bos, Hrv Jasak
 - mesh morphing, shape optimization
- **block matrix solvers / coupled solvers**
 - Ivor Clifford, Hrv Jasak
- additional turbulence models, convection schemes, linear solvers, ...
- topological changes (sliding, layering), finite area method, ...

Where to get OpenFOAM®

37 / 45

- ▶ <http://www.openfoam.com/download/>



The screenshot shows the 'Download' section of the OpenFOAM website. The main navigation bar includes Home, Features, Download, Documentation, Support, Training, Resources, News, and OpenFOAM Foundation. A sidebar on the left lists options like Overview, Ubuntu/Debian, SuSE/RPM, Source Pack, Release History, and Old Versions. The main content area features a large heading 'Download OpenFOAM®' and a sub-section titled 'Ubuntu/Debian Pack' which describes the pack as a good option for personal computers running Ubuntu or Debian Linux. It mentions the version is 2.0.1, released on 04/08/11, and provides a link to the installation guide. Another section for 'SuSE/RPM Pack' is also visible.

OpenFOAM currently distribute OpenFOAM for Linux operating systems in the following ways. The latest release is version 2.0.1, please see [Version 2.0.1 Release Announcement](#) for more information.

Ubuntu/Debian Pack

Version 2.0.1, released 04/08/11, updated periodically

The Ubuntu/Debian pack is a good option for a user that wants to install OpenFOAM on a personal computer on which they have Ubuntu or Debian Linux running either as the native operating system or within a virtual machine. The pack contains both source code and binaries and includes a list of dependent system software which can be installed automatically to reduce the system administration overhead. It is therefore the simplest installation option, making it the ideal choice for those people with no experience of Linux. The pack is updated periodically to incorporate new features and bug fixes.

[Go to Ubuntu/Debian Pack Installation](#)

SuSE/RPM Pack

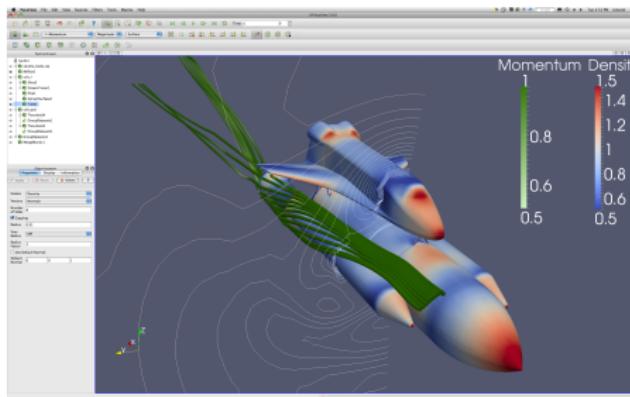
Version 2.0.1, released 04/08/11, updated periodically

OpenFOAM is also released as a source/binary pack for SuSE Linux. It is a little more difficult to install than the Ubuntu pack (above) because not all the dependent software are available as standard SuSE packages, so some additional administration

Platform and support

38 / 45

- ▶ OpenFOAM® is developed mainly on Linux/Unix: Windows and Mac OS X ports also exist but are not widely used. This needs to be improved: binary distribution for Mac OS X and native Microsoft Windows port
- ▶ Data input/output is in file form: there is no global graphical user interface that can be tailored to sufficient level of usability (work in progress)
- ▶ No built-in 3-D graphics; 2-D, graphing and sampling tools present
- ▶ External post-processing with integrated readers: paraFoam (open source)



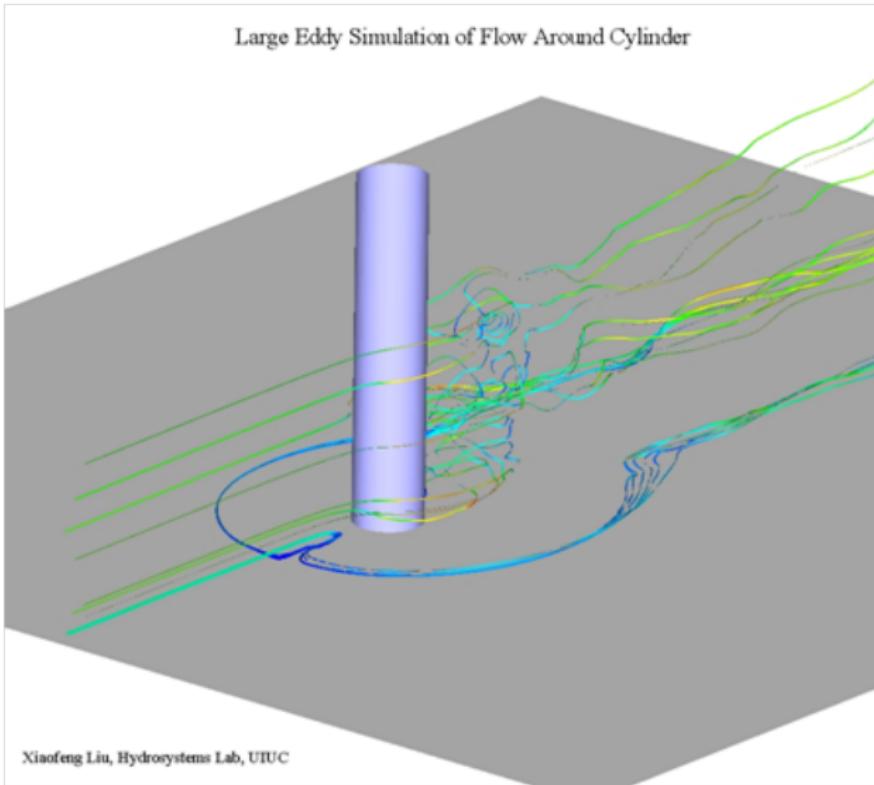
source: <http://2013.hpcs.ca>

- ▶ Project Examples

- Sediment scour and erosion
- Modeling of large rivers
- Settling tanks modeling
- Saturated and unsaturated groundwater flow
- Density/turbidity current

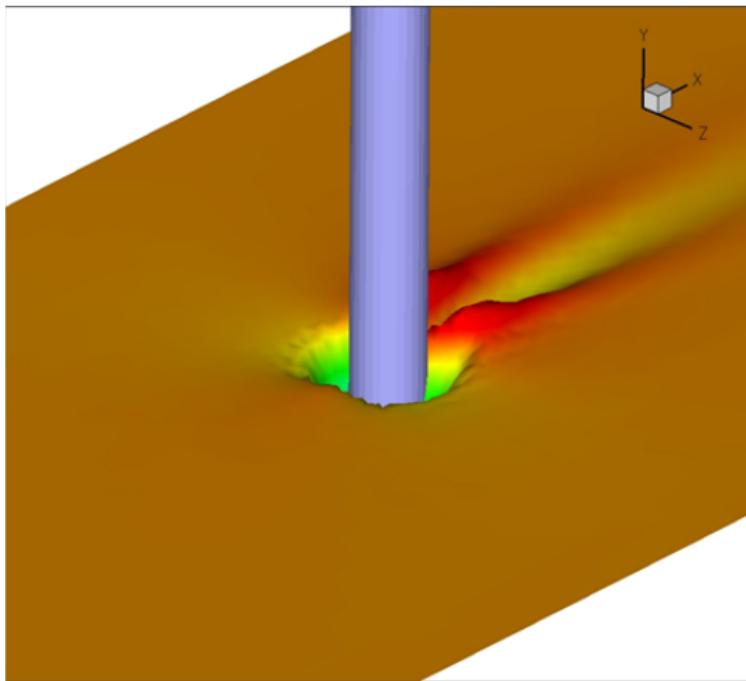
Our use of OpenFOAM®

40 / 45



Our use of OpenFOAM®

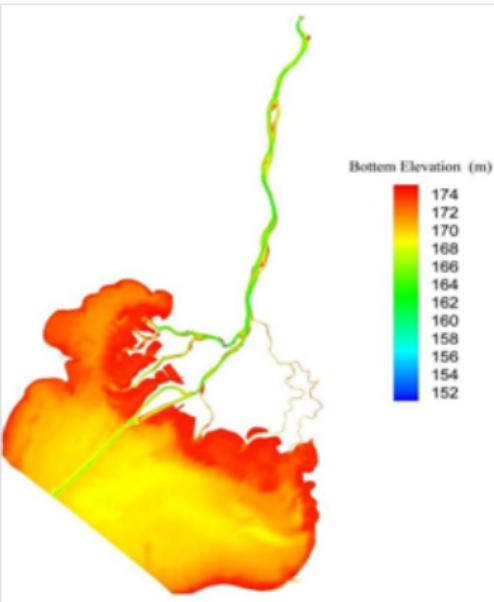
41 / 45



Our use of OpenFOAM®

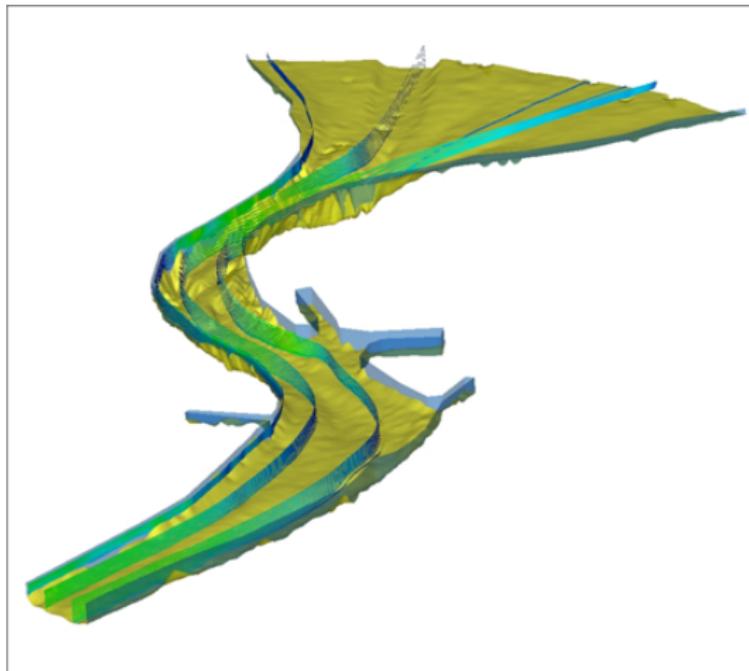
42/45

- ▶ Area map of Great Lakes and the Bathymetry (The river is about 39 miles long)



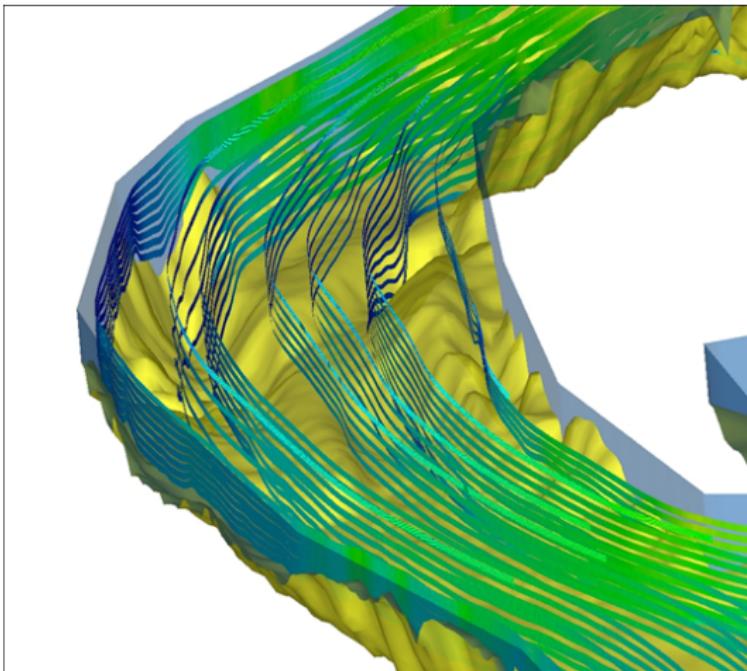
Our use of OpenFOAM®

43 / 45



Our use of OpenFOAM®

44 / 45



Summary:

- ▶ *A brief introduction of OpenFOAM®*
- ▶ *Sample applications of OpenFOAM®*
- ▶ Installation of OpenFOAM®
- ▶ Hands-on exercises for the basics
- ▶ Detailed walk-through of the code