

Multiphase (VOF) Simulation Project

Training session

József Nagy¹

¹Institute of Polymer Injection Molding and Process Automation, Johannes Kepler University Linz, Austria

Introduction

- How can you learn OpenFOAM within a couple of days?

Introduction

- How can you learn OpenFOAM within a couple of days?

You can't...

Introduction

- How can you learn OpenFOAM within a couple of days?

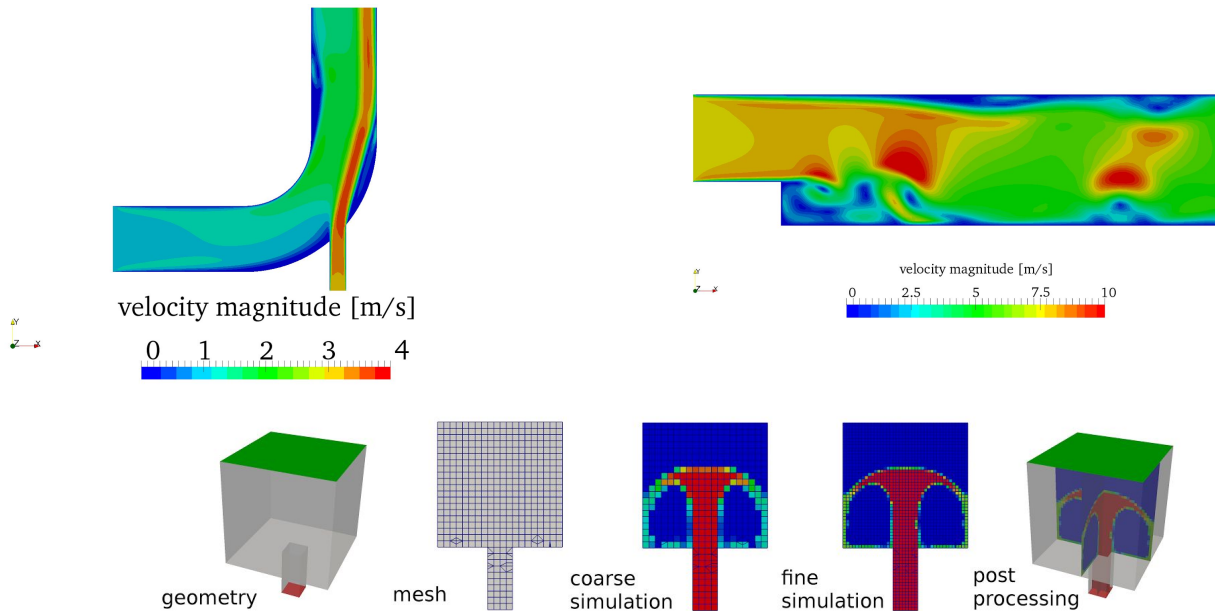
What can you learn in a couple of hours/days?

Introduction

- How can you learn OpenFOAM within a couple of days?
- What does a case setup look like?

Introduction

- How can you learn OpenFOAM within a couple of days?
- What does a case setup look like?
- How do you set up, run and evaluate OpenFOAM simulations?



Introduction

- How can you learn OpenFOAM within a couple of days?
- What does a case setup look like?
- How do you set up, run and evaluate OpenFOAM simulations?

Usually tutorials stop here... not us!

Introduction

- How can you learn OpenFOAM within a couple of days?
- What does a case setup look like?
- How do you set up, run and evaluate OpenFOAM simulations?
- How can you implement the knowledge and the experience implement in your own application?

Introduction

- How can you learn OpenFOAM within a couple of days?
- What does a case setup look like?
- How do you set up, run and evaluate OpenFOAM simulations?
- How can you implement the knowledge and the experience implement in your own application?
- How can you immediately restart after months without OpenFOAM?

Introduction

- Instead of
 - a lot of text
 - a lot of slides
 - a lot of audio/video content
- We use the most important resource...

Introduction

- Instead of
 - a lot of text
 - a lot of slides
 - a lot of audio/video content
- We use the most important resource...
- The case itself
 - short and informative comments
 - foam.foam file for information

Case files

```
/*-----*- C++ -*------*/
=====
\ \ \ \ \ F i e l d      | OpenFOAM: The Open Source CFD Toolbox
\ \ \ \ \ O p e r a t i o n | Version:  4.0
\ \ \ \ \ A n d              | Web:      www.OpenFOAM.org
\ \ \ \ \ M a n i p u l a t i o n |
/*-----*/

FoamFile
{
    version      2.0;
    format       ascii;
    class        volVectorField;
    location     "0";
    object       U;
}
// ***** //

dimensions      [0 1 -1 0 0 0 0]; //kg m s K mol A cd

internalField   uniform (0 0 0); //Initially the velocity is (0 0 0) m/s

boundaryField
{
    inlet
    {
        type      fixedValue;
        value      uniform (0 0 3.5); //fixed inlet velocity
    }

    outlet
    {
        // This velocity inlet/outlet boundary condition is applied to pressure
        // boundaries where the pressure is specified. A zero-gradient condition is
        // applied for outflow (as defined by the flux); for inflow, the velocity is
        // obtained from the patch-face normal component of the internal-cell value.
        // The tangential patch velocity can be optionally specified.
        type      pressureInletOutletVelocity;
        value      uniform (0 0 0);
    }

    pipe
    {
        type      noSlip; //no slip along the walls - replaces fixedValue with uniform (0 0 0)
    }

    tank
    {
        type      noSlip; //no slip along the walls - replaces fixedValue with uniform (0 0 0)
    }
}

// ***** //
```

Case files

```
/*-----*- C++ -*------*/
=====
\\      F ield      | OpenFOAM: The Open Source CFD Toolbox
\\      O peration   | Version: 4.0
\\      A nd         | Web:      www.OpenFOAM.org
\\      M anipulation|
/*-----*/
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    location     "system";
    object       controlDict;
}
// ***** //

application      interFoam;

startFrom        latestTime; //If we restart simulation continues from the last time step

startTime        0;

stopAt           endTime;

endTime          5; //we stop after 5 s - adjustableRunTime!

deltaT           0.005; //initial time step size

writeControl      adjustableRunTime; //simulation can automatically adjust the time step according to the Courant number

writeInterval     0.05; //we write out every 0.05 s

purgeWrite        0;

writeFormat       ascii;

writePrecision    6;

writeCompression  uncompressed;

timeFormat        general;

timePrecision     6;

runTimeModifiable yes; //adjustableRunTime can modify the time step size

adjustTimeStep    yes; //adjustableRunTime can modify the time step size
```

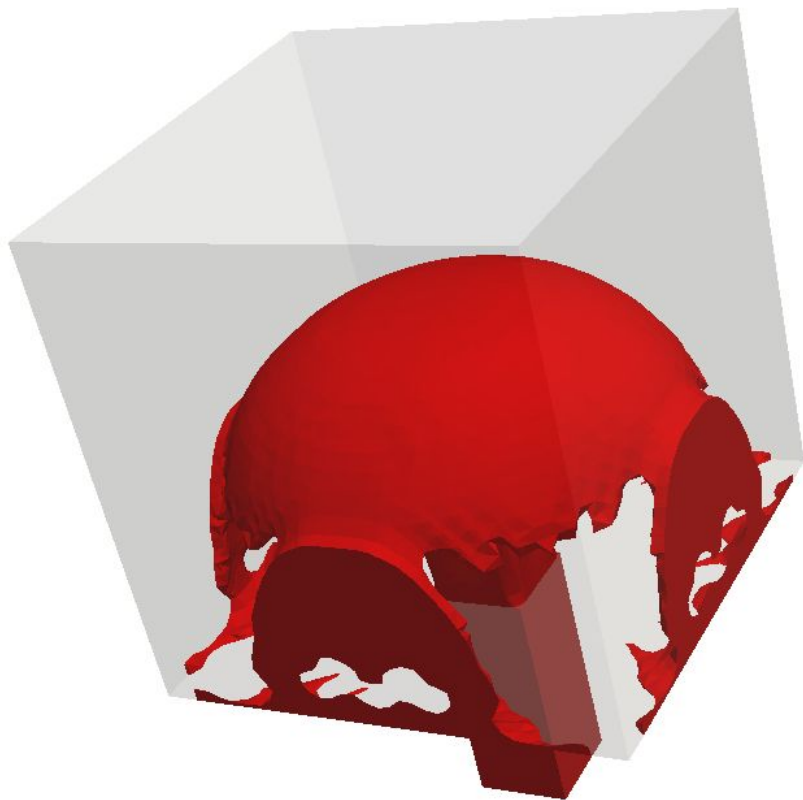
Introduction

- Instead of
 - a lot of text
 - a lot of slides
 - a lot of audio/video content
- We use the most important resource...
- The case itself
 - short and informative comments
 - foam.foam file for information
- 'One command' case setup



```
> sh setUpCase.sh &
```

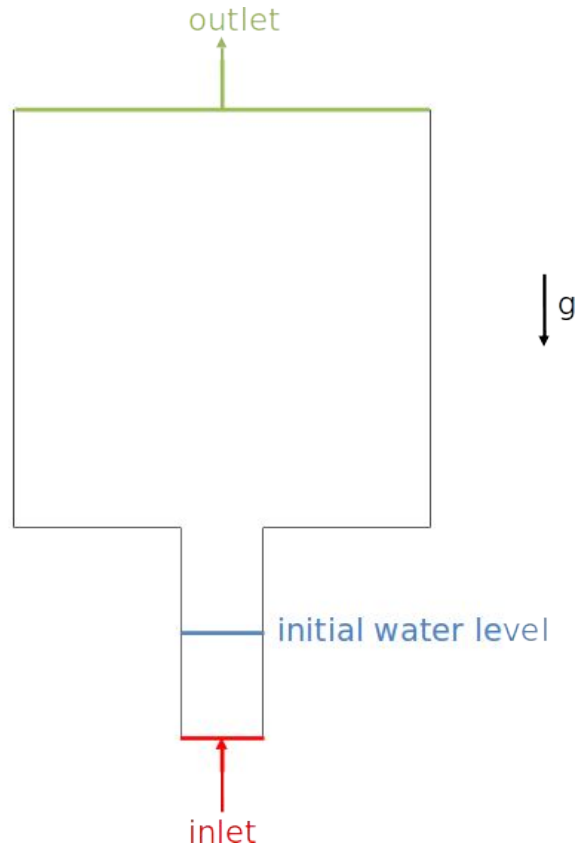
The flow



Goals

- Gain knowledge
 - snappyHexMesh
 - multiple phases (gas-liquid)
 - Volume of Fluid
- Case setup
- Initial values (BC)
- Simulate flow
 - coarse
 - refined
 - dynamic mesh
- Postprocessing

Geometry



Solver

interFoam:

'Solver for 2 incompressible, isothermal immiscible fluids using a VOF (volume of fluid) phase-fraction based interface capturing approach'

- incompressible
- transient
- laminar and turbulent
- multi phase
- immiscible
- VOF
- isothermal

Theory

- incompressible
- transient
- laminar and turbulent
- multi phase
- immiscible
- VOF
- isothermal

Continuity equation:

$$\nabla \cdot \mathbf{u} = 0$$

Momentum equations:

$$\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \cdot \rho \mathbf{v}[2S] + F$$

Volume of Fluid:

$$\rho = \alpha \rho_l + (1 - \alpha) \rho_g$$

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{u}) + \nabla \cdot (\alpha(1 - \alpha) \mathbf{u}_r) = 0$$

First step

Download files from

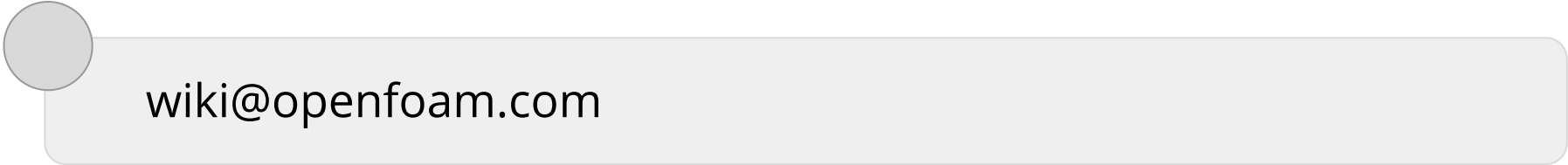


wiki.openfoam.com

Have fun!

Feedback

Please send us feedback around Christmas (in 5-6 months):



wiki@openfoam.com

Have fun!