Multiphase (VOF) Simulation Project

Training session

József Nagy[1]

[1]Institute 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

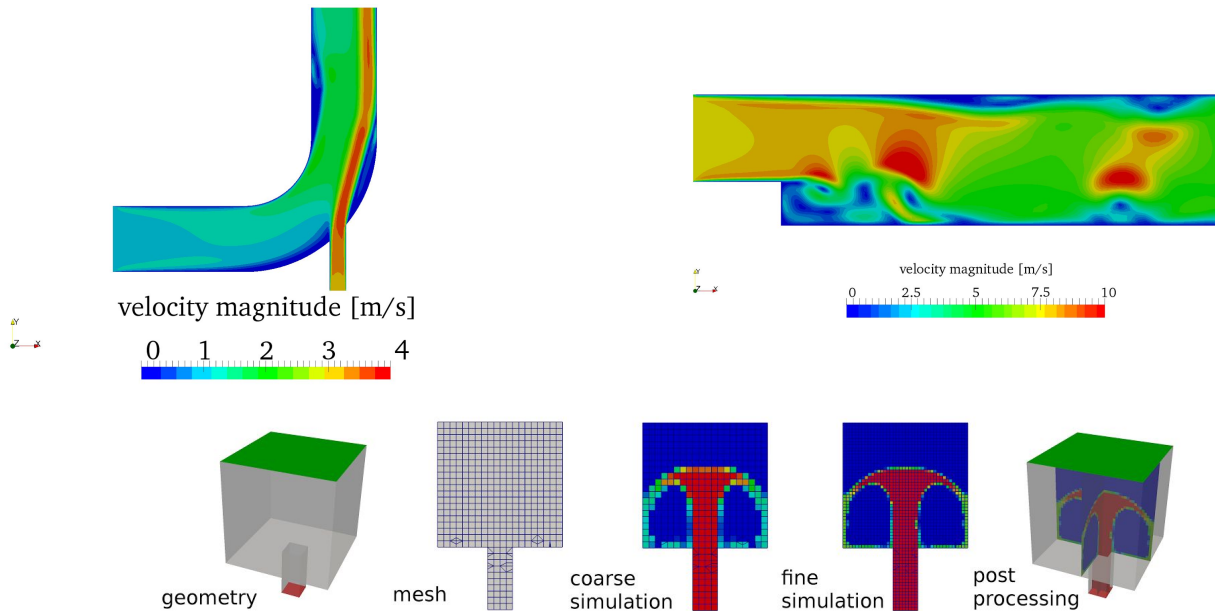- How can you learn OpenFOAM within a couple of days?

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?

velocity magnitude [m/s]

velocity magnitude [m/s]

geometry   mesh   coarse simulation   fine simulation   post processing

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 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       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 modifiy the time step size

adjustTimeStep  yes;//adjustableRunTime can modifiy the time step size
```
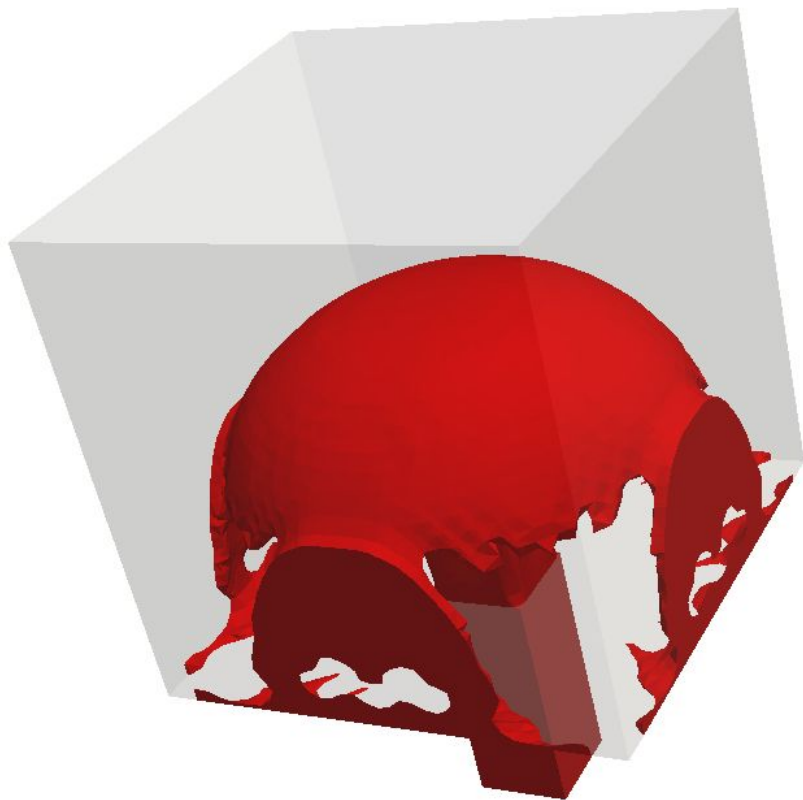
Introduction

- 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
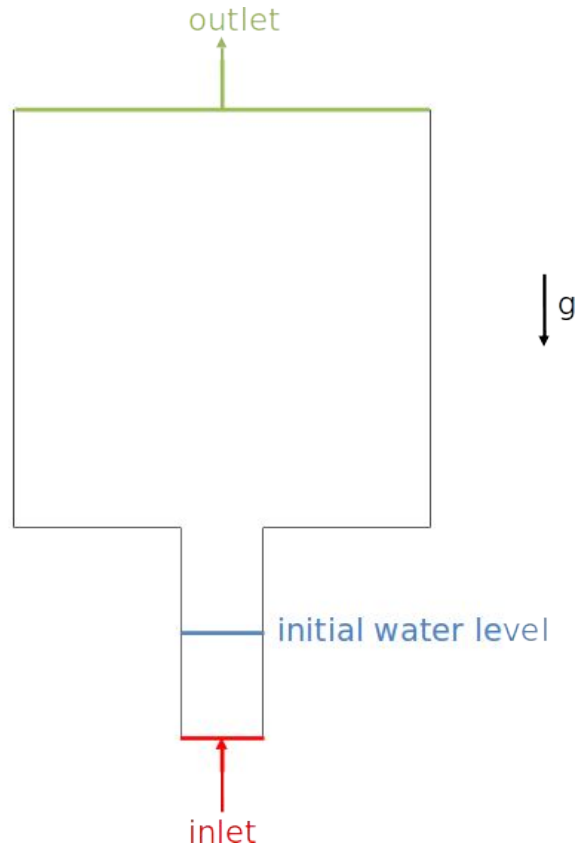- 'One command' case setup

> **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:

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

Momentum equations:

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

Volume of Fluid:

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

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

First step

Download files from

wiki.openfoam.com

Have fun!