PPS 2024
30 Sept.-03 Oct. | Ferrol · Spain

Introduction to OpenFOAM® Computational Library and Viscoelastic Fluid Flow Simulation

P3 - Case studies: Single- and Two-Phase Flow Solvers

This Presentation was Adapted from József Nagy's Presentation

Mohammadreza Aali & J. Miguel Nórbega

mohammadreza.aali@jku.at & mnobrega@dep.uminho.pt

University of Minho
School of Engineering

Computational Rheology @IPC

iPC INSTITUTE FOR POLYMERS AND COMPOSITES

JKU IPPD Institute of Polymer Processing and Digital Transformation

JKU JOHANNES KEPLER UNIVERSITY LINZ

Outline

9:00 – 10:30	**Introduction to OpenFOAM (P1)**
10:30 – 12:00	**Mesh generation and post-processing (P2)**
12:00 – 13:00	Lunch break
13:00 – 14:30	**Case studies: Single- and two-phase flow solvers (P3)**
14:30 – 16:00	**Case studies: Viscoelastic fluid flow solvers (P4)**

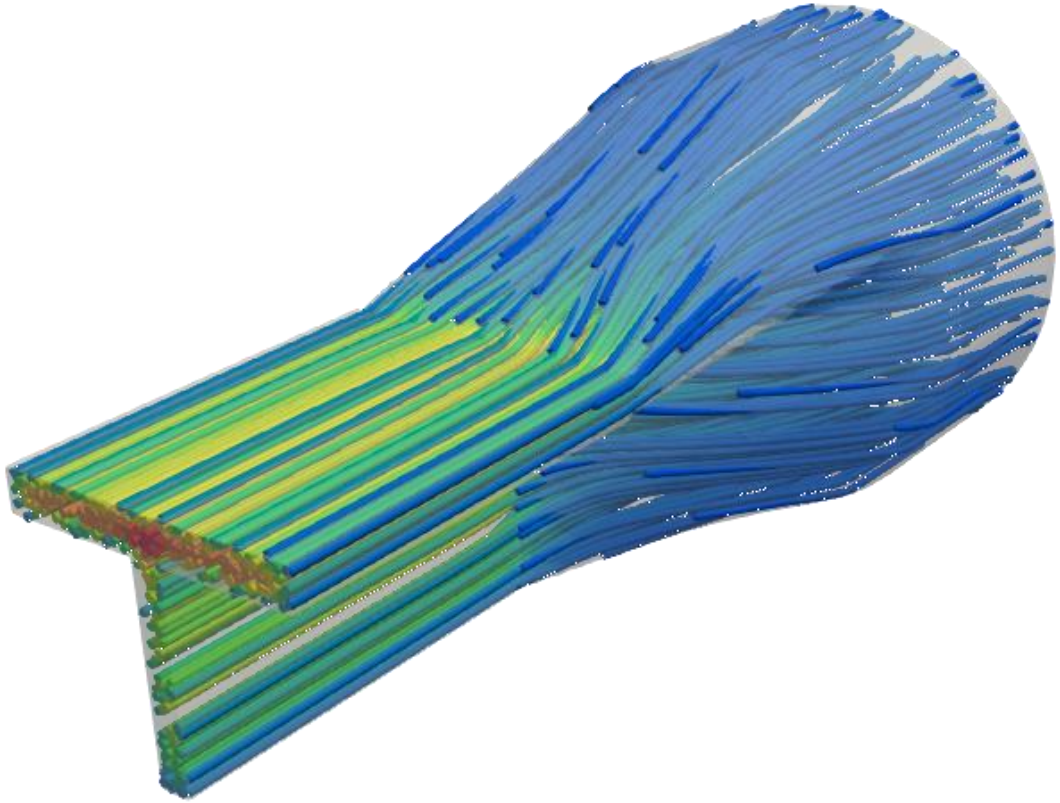
CRheo@IPC

iPC

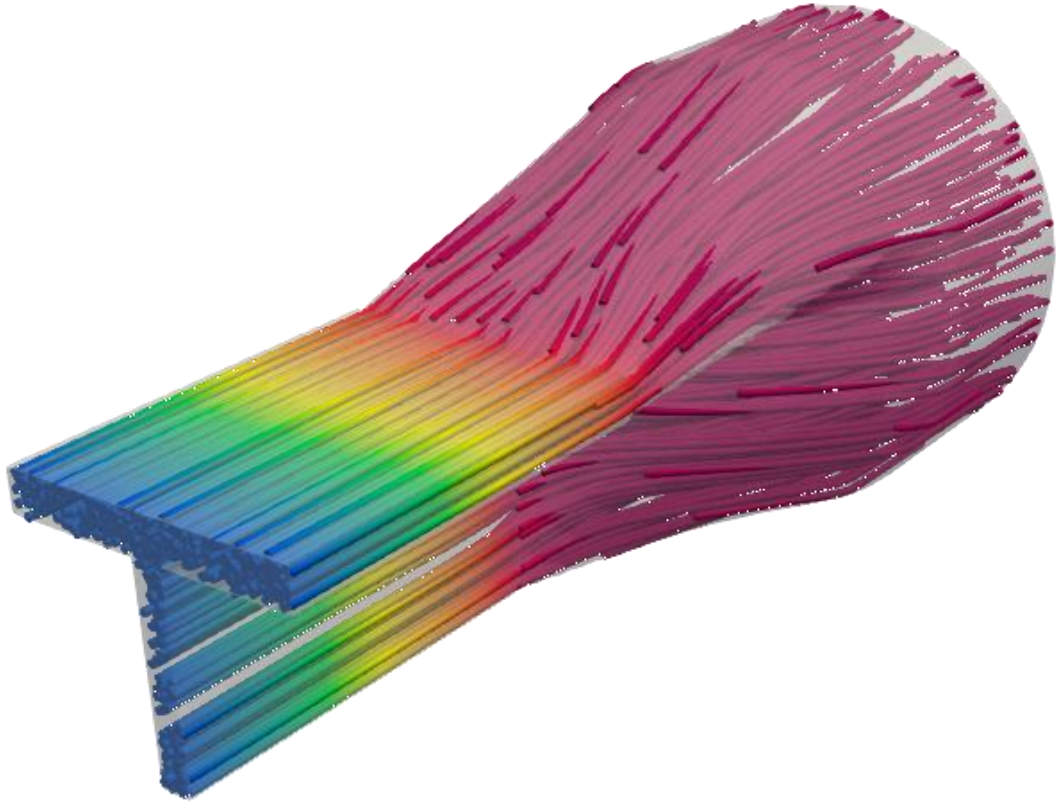
PPS 2024
30 Sept.-03 Oct. | Ferrol · Spain

JKU IPPD Institute of Polymer Processing and Digital Transformation

P3 - Case 31



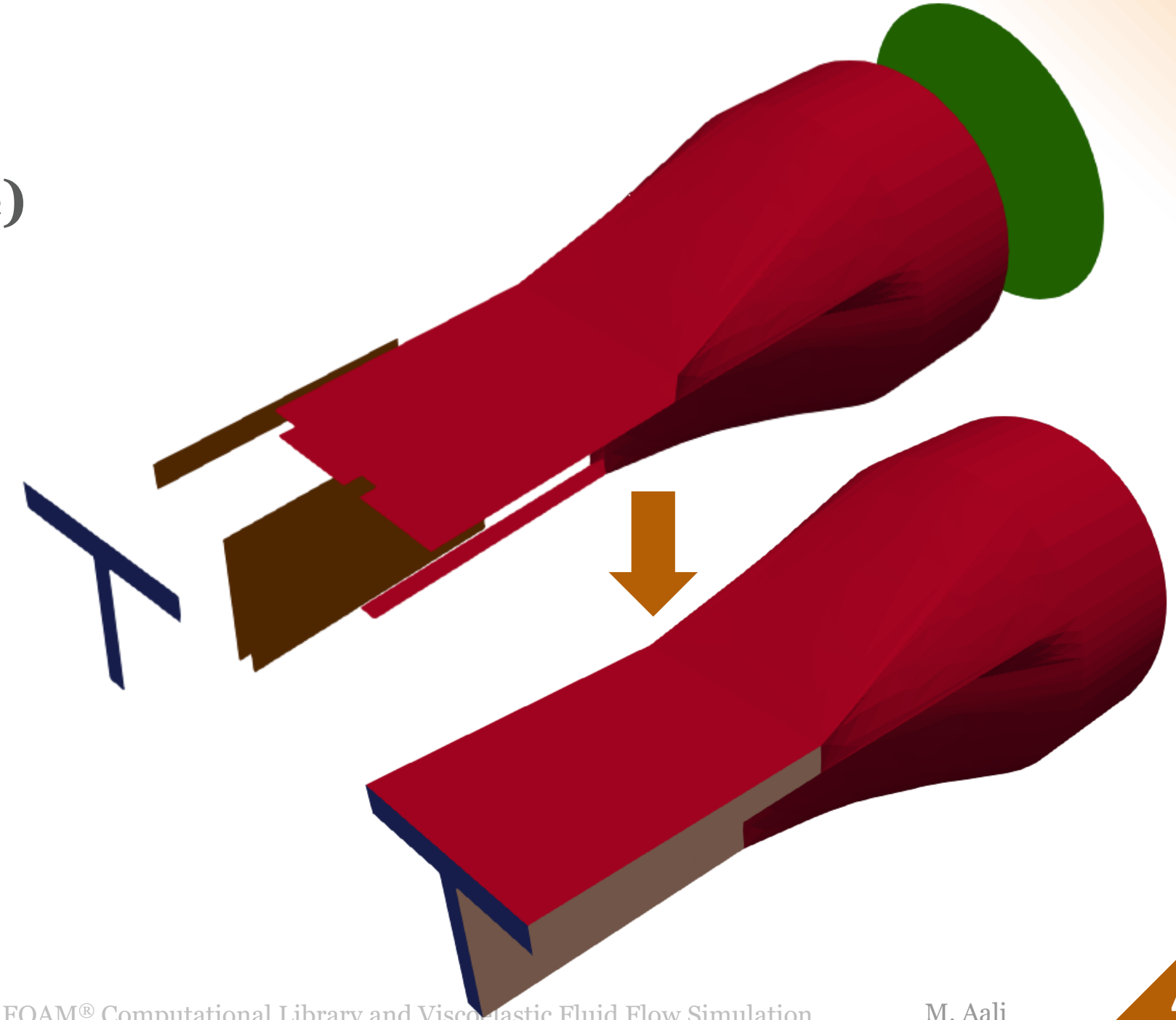
Velocity Field



Pressure Field

Monophase Flow

P3 - Case 31
Geometry
(Profile extrusion die)



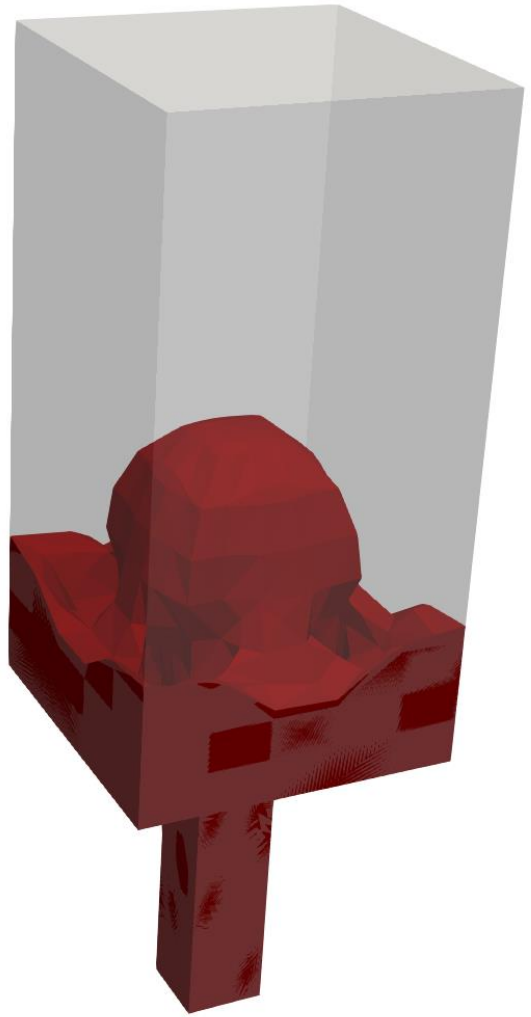
P3 - Case 31

1. Open Ubuntu terminal
2. of2206 //Load OpenFOAM variables
3. >> run
4. >> explorer.exe .
5. Copy (by dragging) case31 folder from *caseFiles* to the run folder
6. >> cd c1Geo
7. >> code .
8. Visualize *in.stl, wall01.stl, wall02.stl* and *out.stl* in paraview
9. *Check files myList*
10. *>> chmod +x uniqueSTL.X*
11. *>> ./uniqueSTL.X*
12. Visualize *total.stl* in paraview
13. Copy *total.fms* to folder *c1*

P3 - Case 31

1. >> cartesianMesh
2. Check the mesh in Paraview
3. Check the case files folders 0, constant, and system
4. >> transformPoints -scale 1e-3
5. Check the mesh in Paraview
6. Run simpleFoam in background with output for log.simpleFoam
7. Check results in paraview
8. >> ./run_parallel.sh /Run the case in Parallel
9. After convergence check results in paraview (decomposed case)
10. >> reconstructPar –latestTime
11. Check results in paraview
12. Plot the streamlines

P3 - Case 32



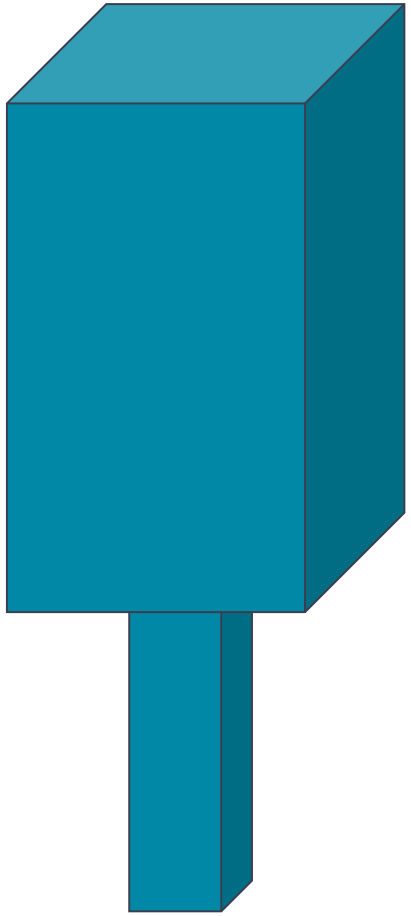
Multiphase Flow

CRheo@IPC

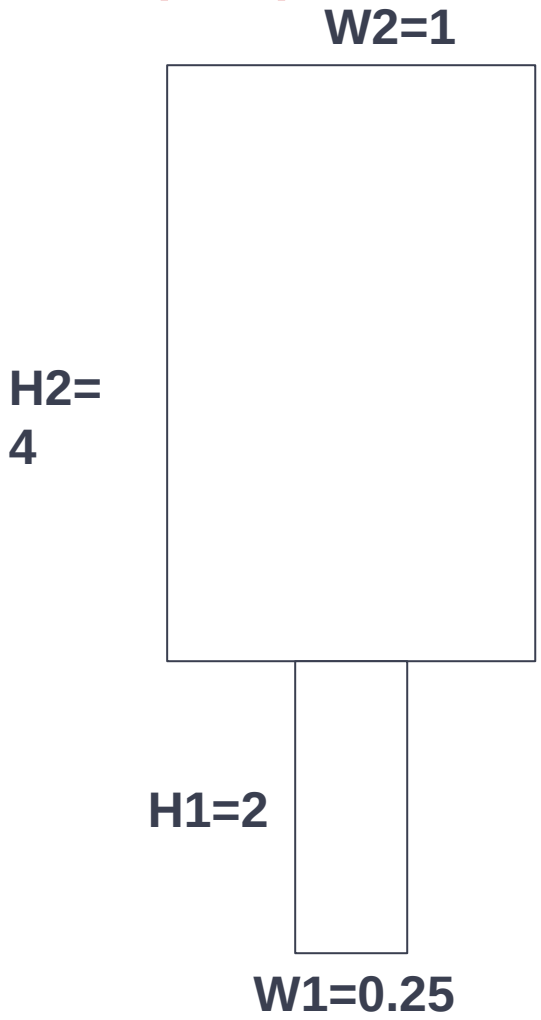
PPS 2024
30 Sept.-03 Oct. | Ferrol · Spain

IPPD Institute of Polymer Processing and Digital Transformation

P3 - Case 32 Geometry

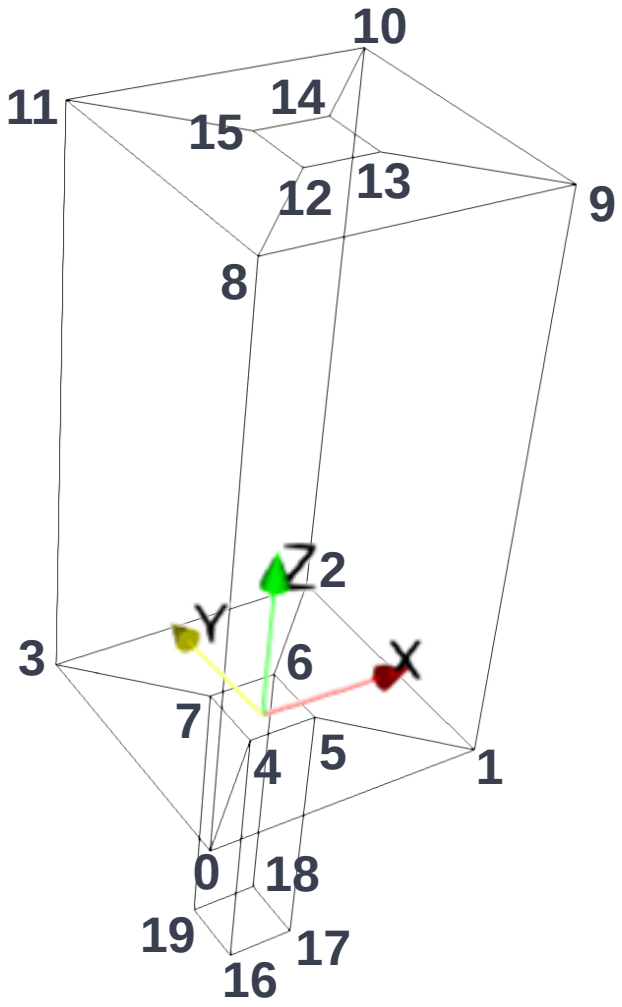


Dimensions (in m)

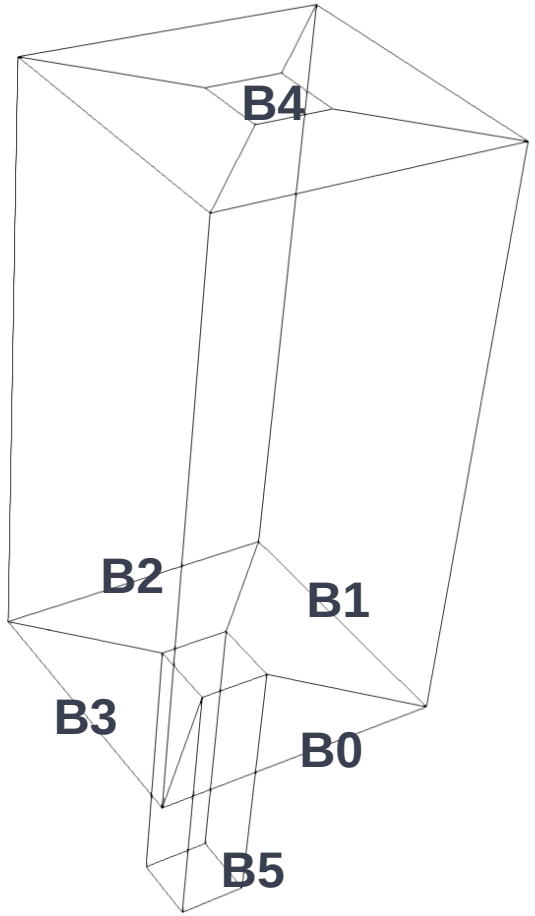


P3 - Case 32
Geometry

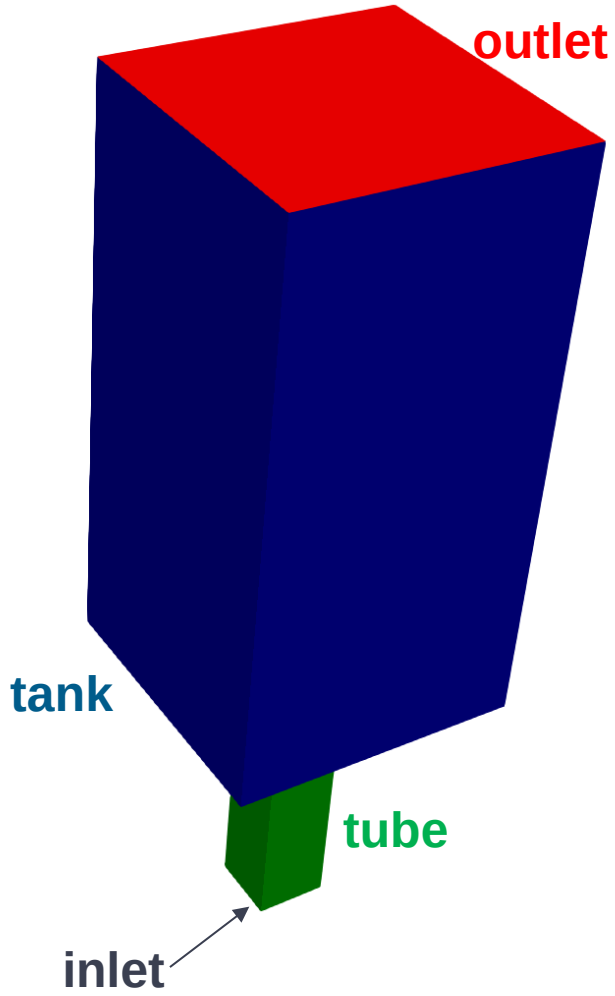
Vertices



Blocks



Patches

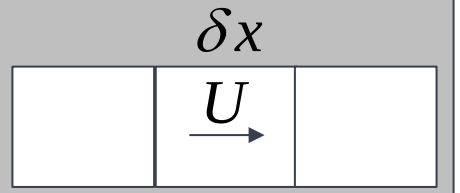


P3 - Case 32
Governing Equations

Multiphase Flow (Volume of Fluid - VOF)

Courant Number

$$C_o = \frac{U \Delta t}{\delta x}$$



Stability $\Rightarrow$ $C_o < 1$

- **Continuity**

$$\frac{\partial \rho}{\partial t} + \nabla \bullet \left(\rho \mathbf{U} \right) = 0$$

- **Momentum**

$$\frac{\partial \left(\rho \mathbf{U} \right)}{\partial t} + \nabla \bullet \left(\rho \mathbf{U} \mathbf{U} \right) + \nabla \bullet \left(p \mathbf{I} \right) + \nabla \bullet \boldsymbol{\tau} + F_g = 0$$

- **Phase**

$$\frac{\partial \alpha}{\partial t} + \nabla \bullet \left(\alpha \mathbf{U} \right) + \nabla \bullet \left(\alpha \left(1 - \alpha \right) \mathbf{U}_r \right) = 0$$

$$\begin{cases} \alpha = 1 \text{ - water} \\ \alpha = 0 \text{ - air} \end{cases}$$

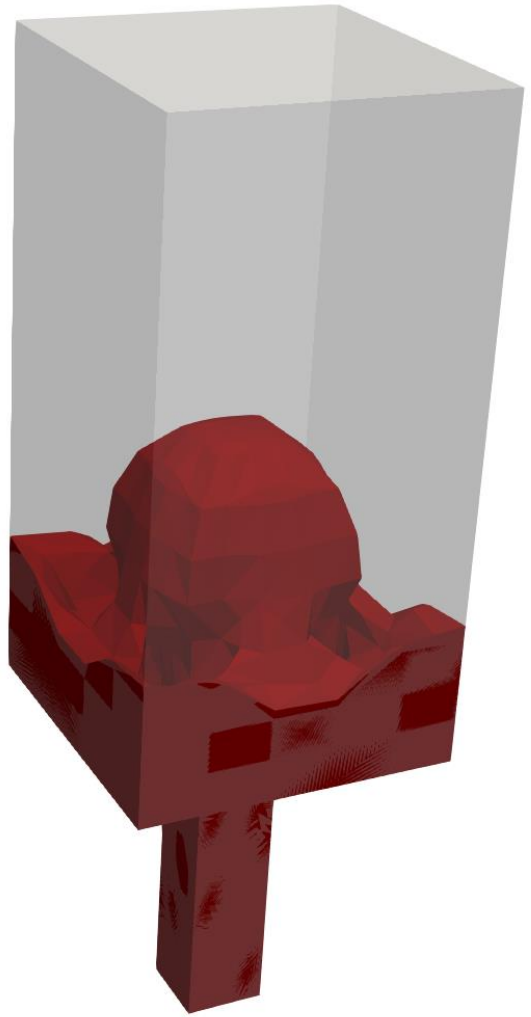
- **Properties**

$$\phi = \alpha \phi_l + \left(1 - \alpha \right) \phi_g$$

P3 - Case 32

1. Analyze the case study files with the instructor
2. Run the applications sequentially (blockMesh->setFields->interFOAM)
3. >> foamCleanTutorials
4. Analyze and the Allrun file
5. Visualize the results in Paraview (use Clip->Scalar)
6. >> foamCleanTutorials
7. refineMesh
8. Visualize the Mesh
9. >> refineMesh –overwrite
10. Run and Visualize results (save video)
11. Analyze/edit decomposeParDict
12. Run the case in Parallel with Allrun_parallel
13. To visualize results in paraview select decomposedCase
14. Test the problem with different inlet velocities

Results



Multiphase Flow

CRheo@IPC

PPS 2024
30 Sept.-03 Oct. | Ferrol · Spain

IPPD Institute of Polymer Processing and Digital Transformation