Universidad de los Andes Colombia

39th International Conference of the Polymer Processing Society

ICIPC® INSTITUTO DE CAPACITACIÓN E INVESTIGACIÓN DEL PLÁSTICO Y DEL CAUCHO

Introduction to OpenFOAM® Computational Library and Viscoelastic Fluid Flow Simulation

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

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)**

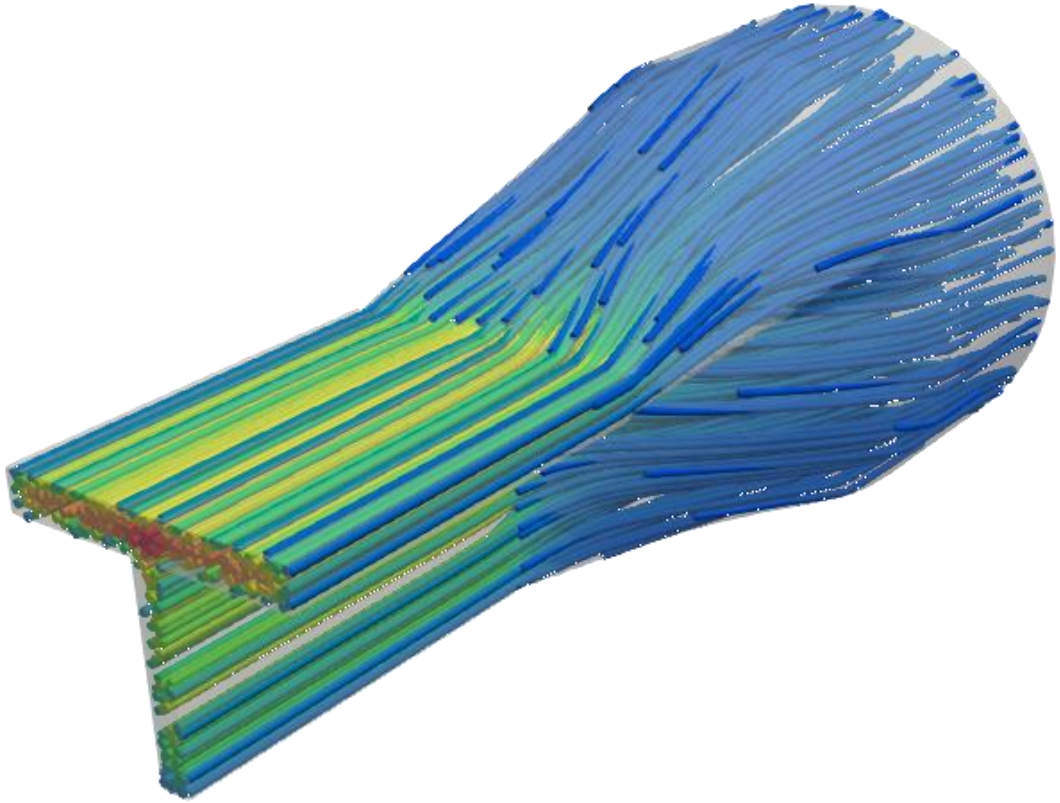
PPS-39
May 19-23, 2024
Cartagena de Indias, Colombia

CRheo@IPC

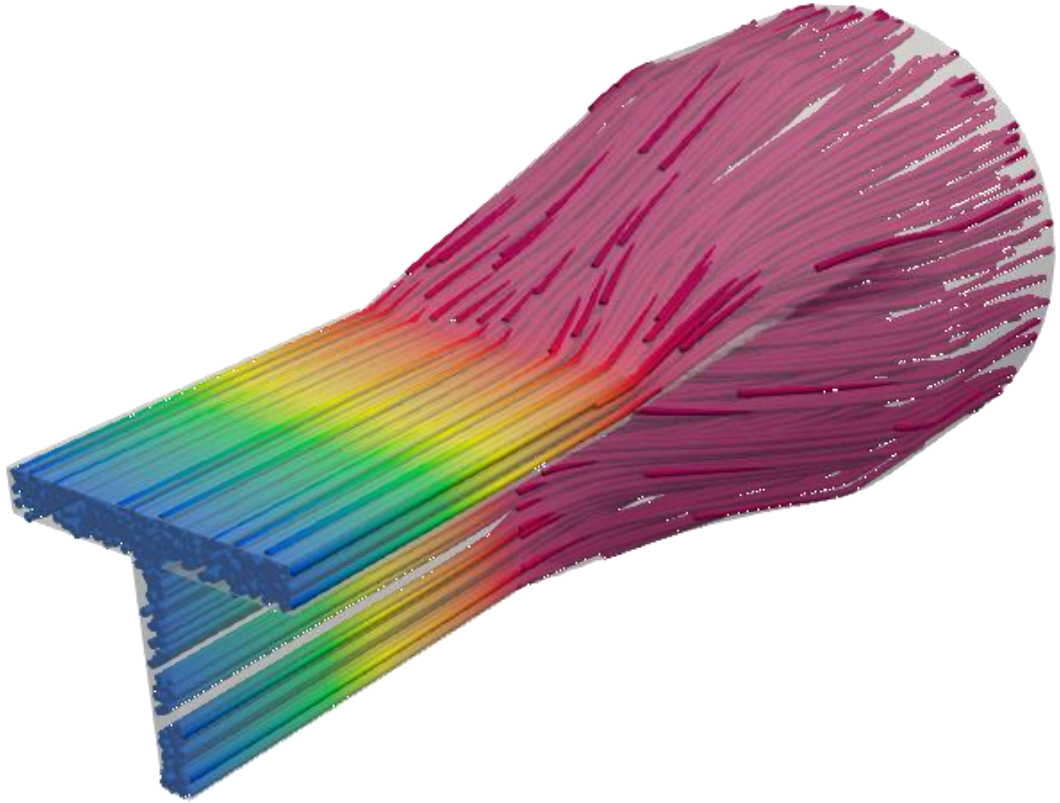
iPC

JKU IPPD Institute of Polymer Processing and Digital Transformation

P3 - Case 31



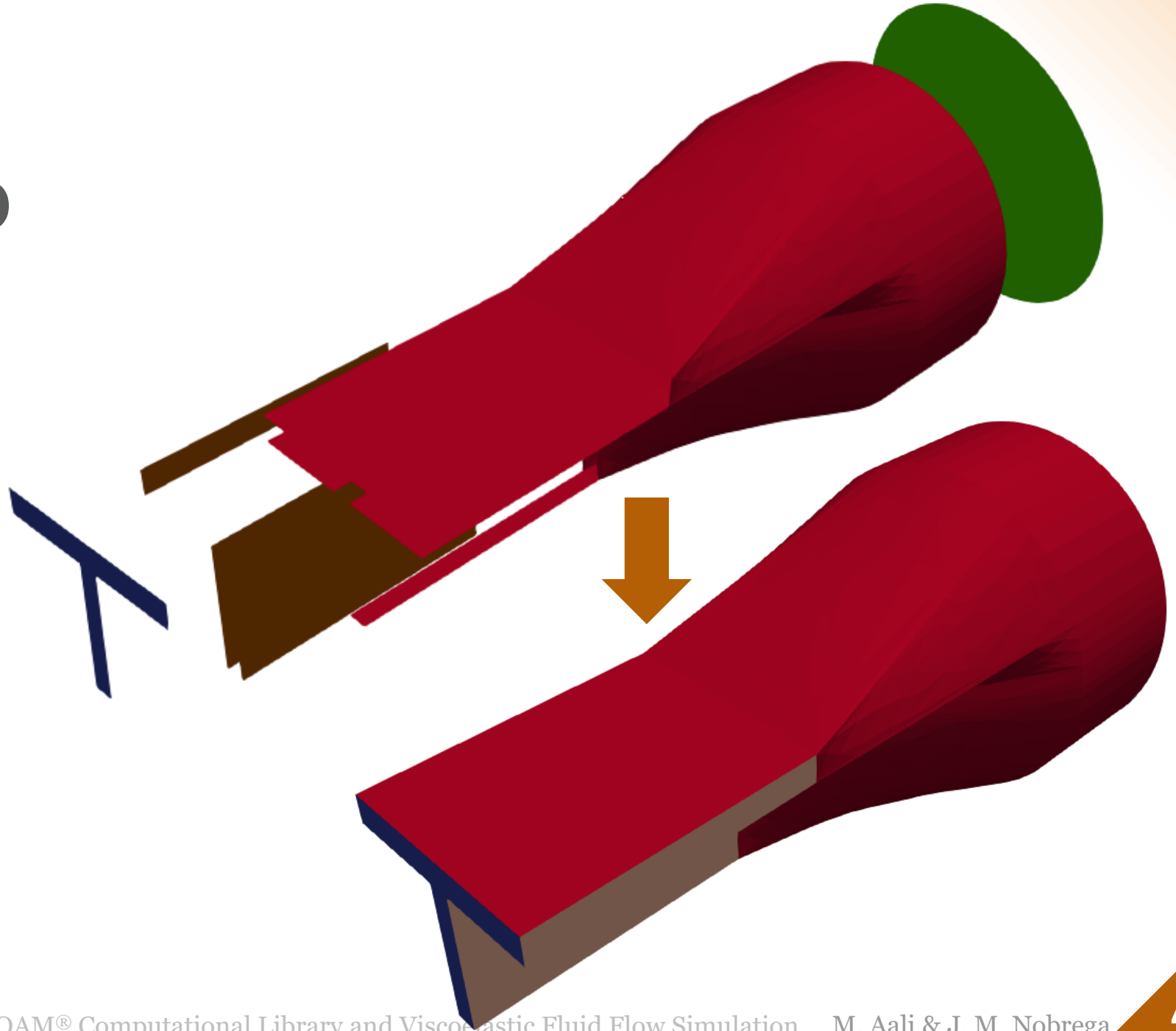
Velocity Field



Pressure Field

Single-phase Flow

P3 - Case 31
Geometry
(Profile extrusion die)



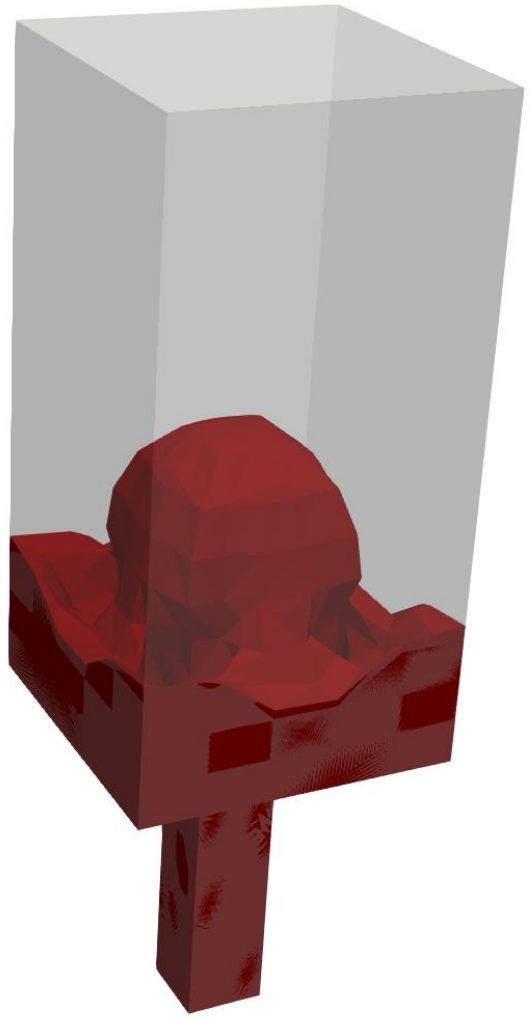
P3 - Case 31

1. Open Ubuntu terminal
2. of2206 //Load OpenFOAM variables
3. >> run
4. >> cd case31/c31Geo
5. Visualize *in.stl, wall01.stl, wall02.stl* and *out.stl* in paraview
6. *Check files myList*
7. *>> ./joinSTL.x*
8. Visualize *total.stl* in paraview
9. >> cp total.fms ../c31/ ## Copy *total.fms* to folder *c31*
10. *>> cd ../c31 #move to c31*

P3 - Case 31

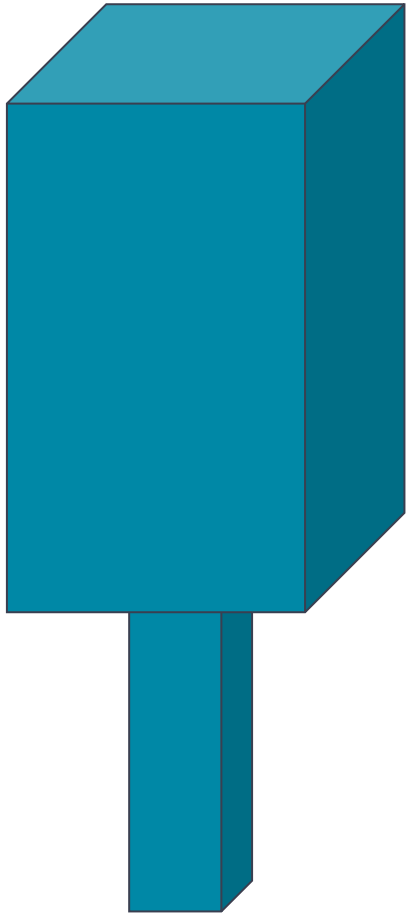
1. Study the Allrun script
2. >> ./Allrun
3. During the run check the contents of the different log files
4. After convergence check results in paraview (decomposed case)
5. Use the foamCleanTutorials utility if it is required to clean the case
6. >> reconstructPar –latestTime
7. Check results in paraview (reconstructed case)
8. Plot the streamlines

P3 - Case 32

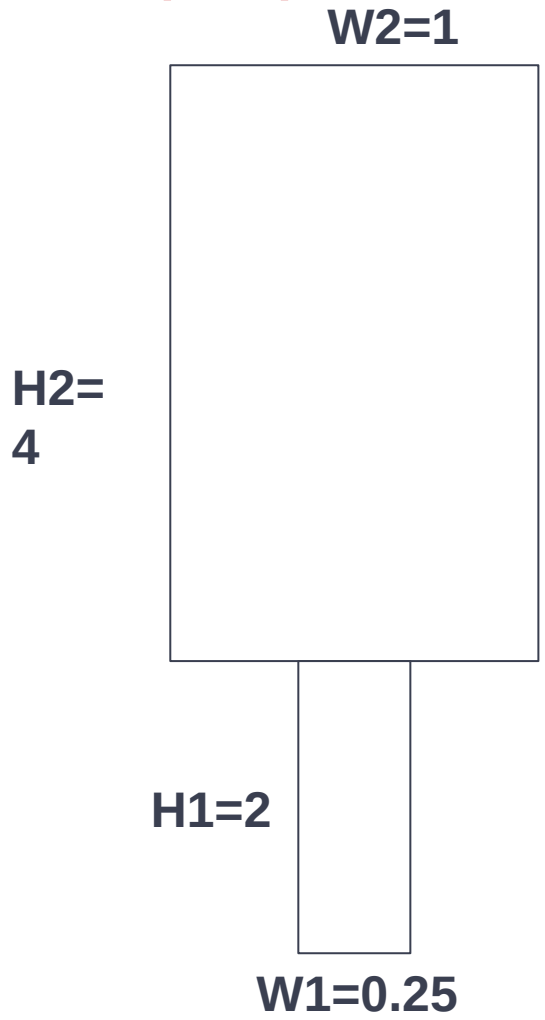


Multiphase Flow

P3 - Case 32 Geometry

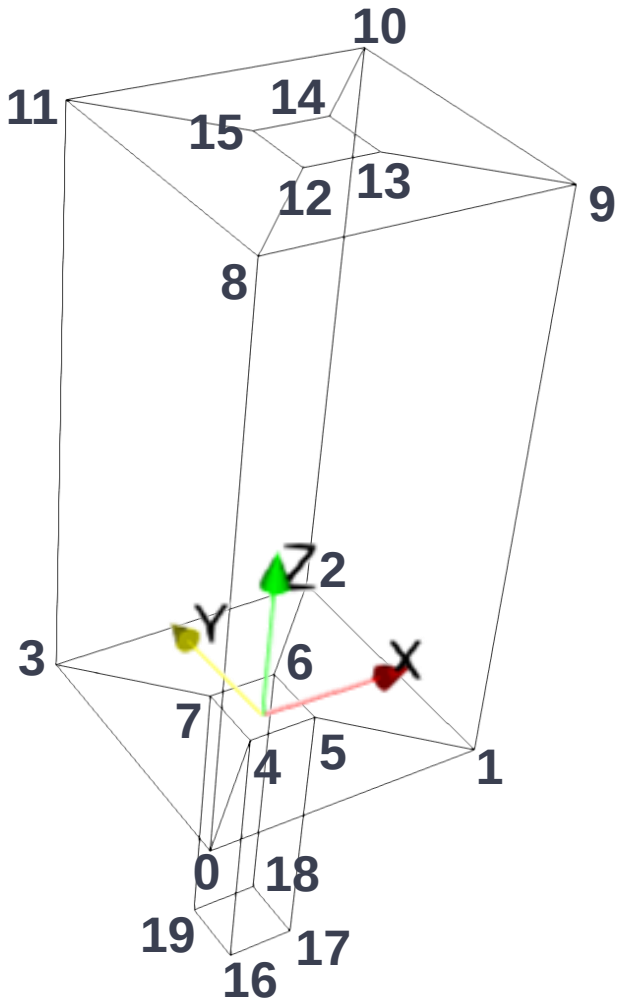


Dimensions (in m)

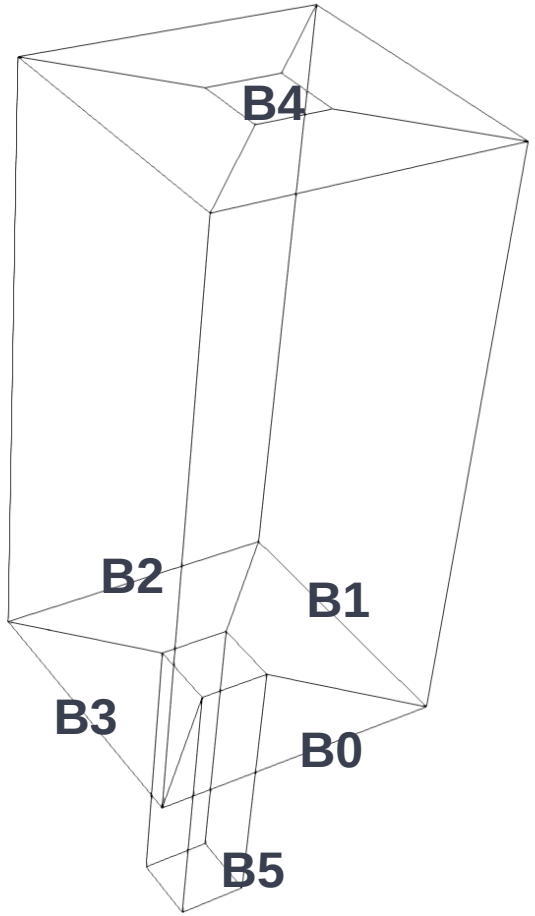


P3 - Case 32
Geometry

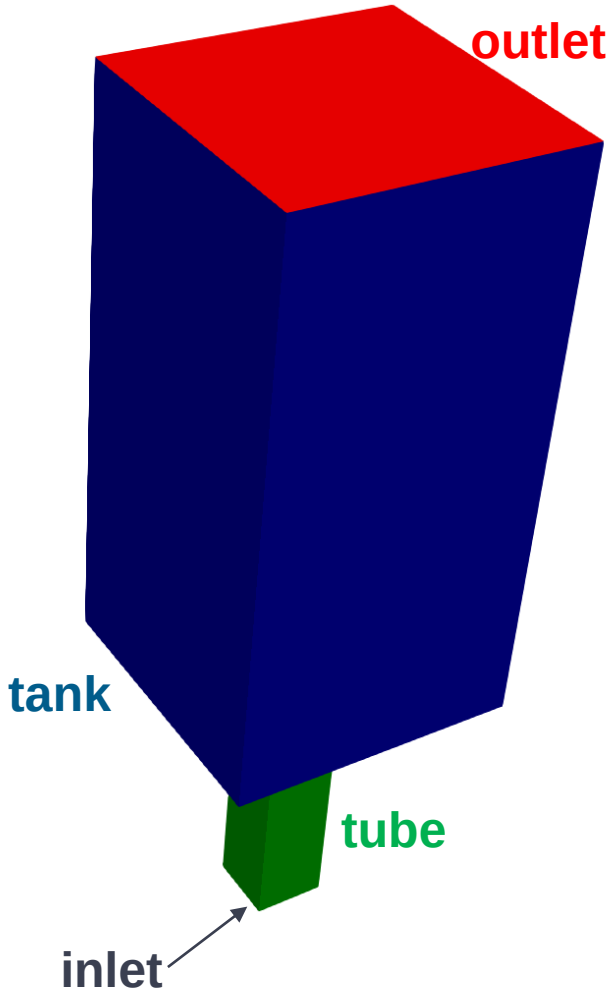
Vertices



Blocks



Patches



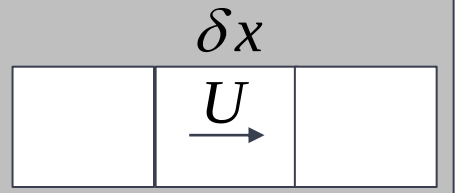
P3 - Case 32

Governing Equations

Multiphase Flow (Volume of Fluid - VOF)

Courant Number

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



$$\text{Stability} \Rightarrow C_o < 1$$

- **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