

Getting Started with OpenFOAM: Fundamentals and Practical Coating Applications

Session 3: Hands-On Coating Case Study

J. Vidal and J. Miguel Nóbrega

jpovidal@gmail.com / mnobrega@dep.uminho.pt

Outline

14:00 – 15:00	Session 1: Introduction to OpenFOAM and Simulation Fundamentals
15:00 – 15:30	Session 2a: Geometry, Mesh Generation, and Case Setup in OpenFOAM
15:30 – 16:00	Coffee-Break
16:00 – 16:30	Session 2: Geometry, Mesh Generation, and Case Setup in OpenFOAM
16:30 – 17:30	Session 3: Hands-On Coating Case Study

interFOAM solver

Two-phase Flow (Volume of Fluid - VoF)

- **Continuity**

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

- **Momentum**

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

- **Phase**

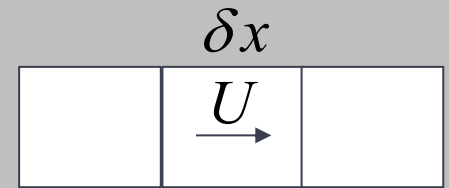
$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{U}) + \nabla \cdot (\alpha (1 - \alpha) \mathbf{U}_r) = 0 \quad \begin{cases} \alpha = 1 - \text{water} \\ \alpha = 0 - \text{air} \end{cases}$$

- **Properties**

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

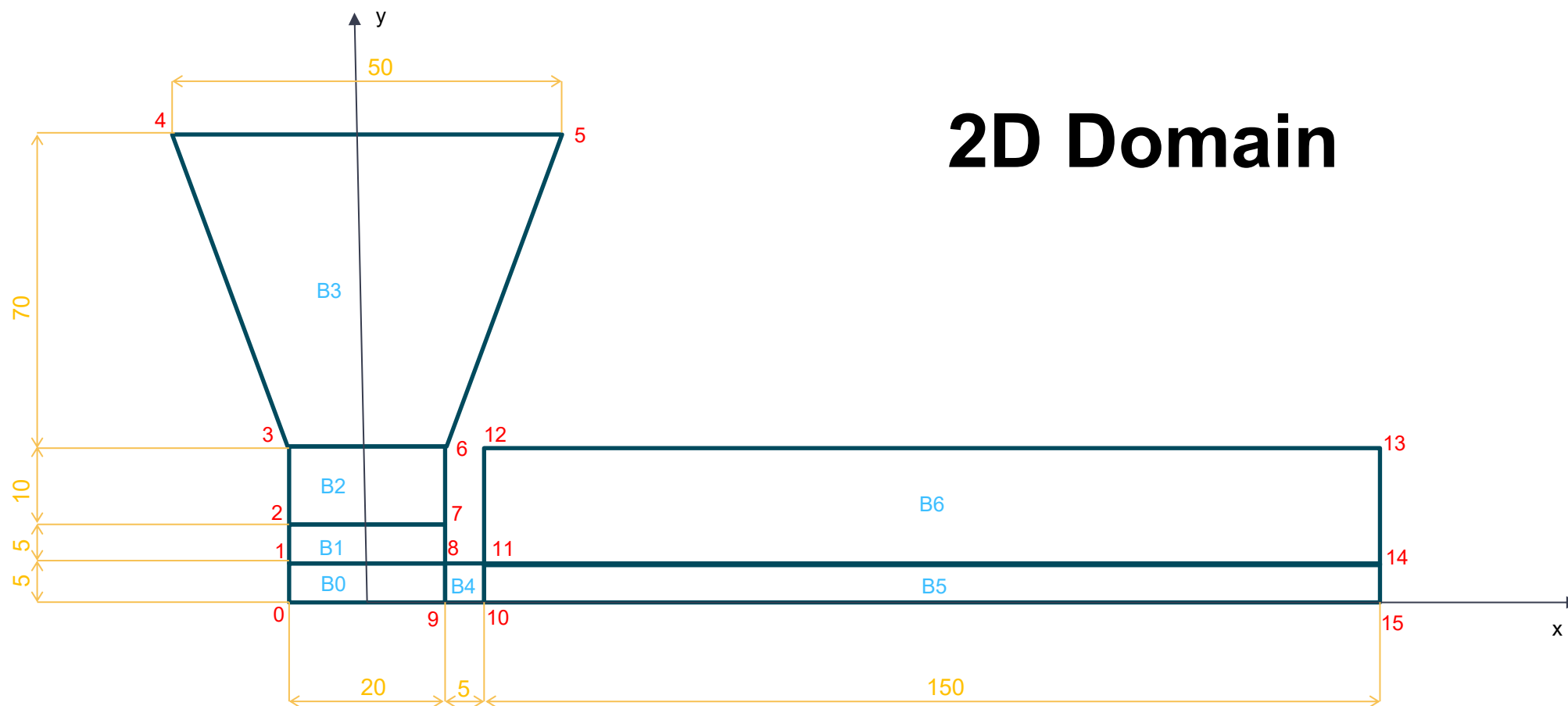
Courant Number

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



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

Case 3 - Geometry



Points
Blocks

Case 3 – Phase 1

1. Change to the case folder
2. Open VSCode
3. Check the case files
4. Run blockMesh
5. Check the Mesh
6. Change blockMeshDict to get a mesh like this:



Case 3 – Phase 2

1. Use Allrun script to run the case
2. Check the results...
3. Investigate de files to identify and correct the origin of the observed problem

Case 3 – Phase 3

1. Study the effect of some problem parameters
 - Fluid Viscosity and density
 - Roller Velocity