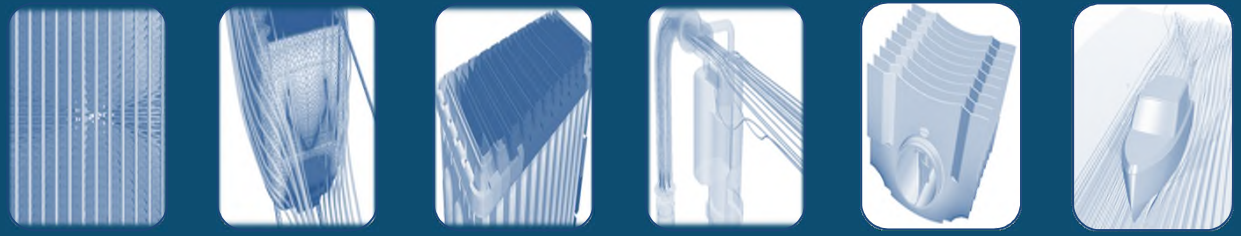
OpenFOAM®
in wastewater applications:
7 – Hands-on: Splitter box

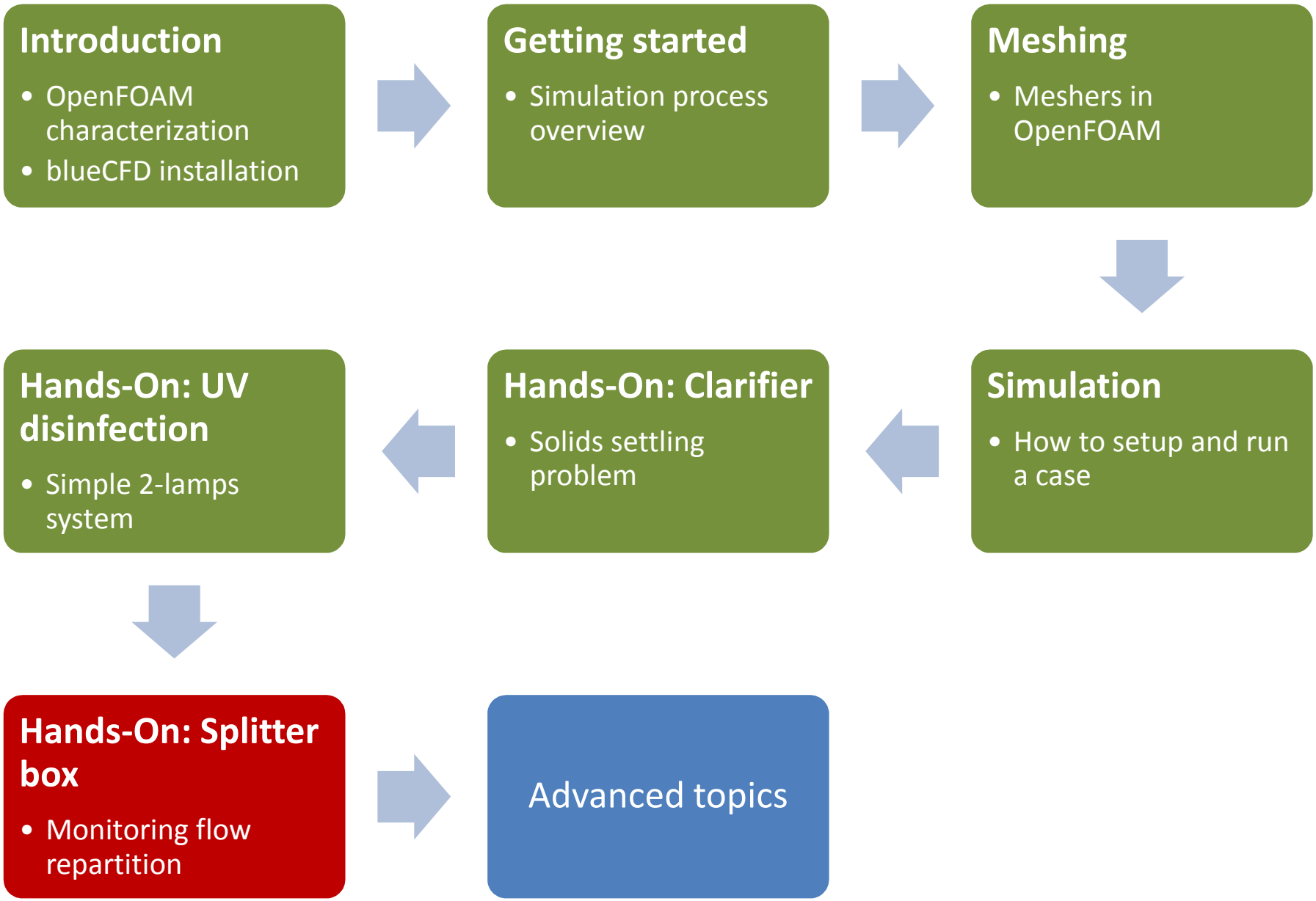
nelson.marques@bluecape.com.pt

13-14th June 2015



bluecape
Computer Applications
in Science & Engineering

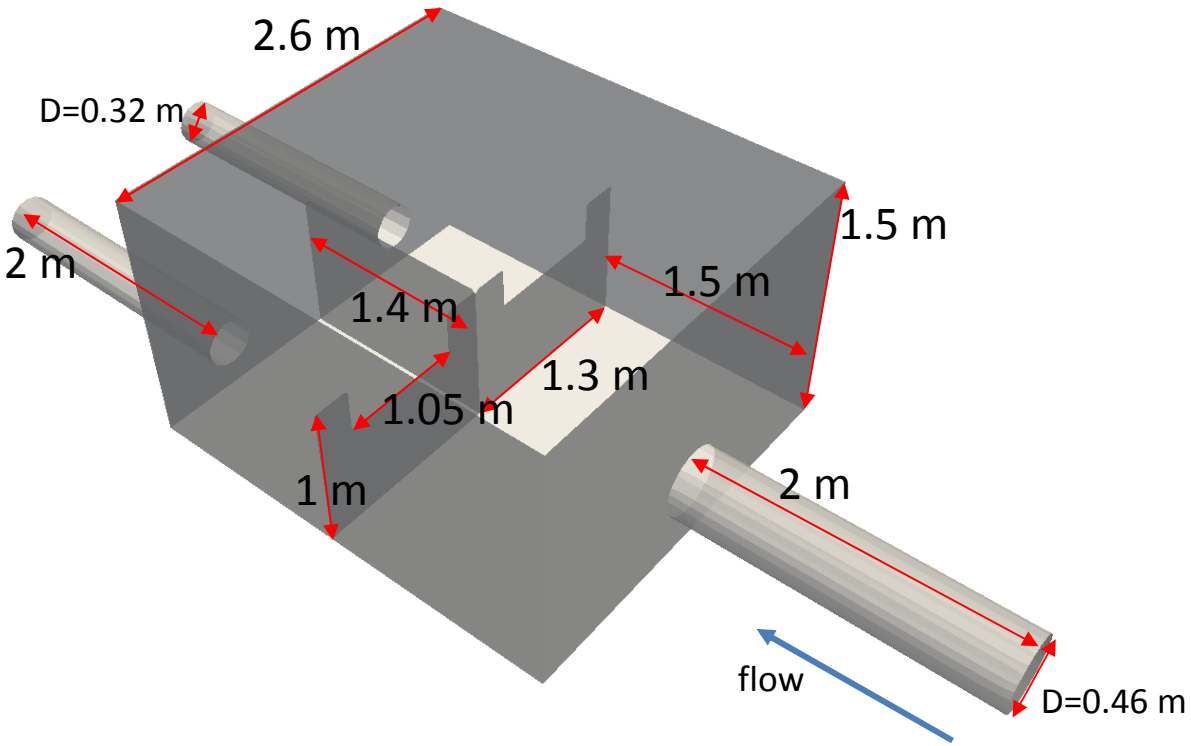
Progress



Contents

Case description (1/2)

Overview:



Inlet mass flow rate:

- 25 kg/s

Water properties:

- ρ = 999 kg/m3
- Newtonian fluid
  - ν = 1.131E-6 m2/s

Air properties:

- ρ = 1.18 kg/m3
- Newtonian fluid
  - ν = 1.572E-5 m2/s

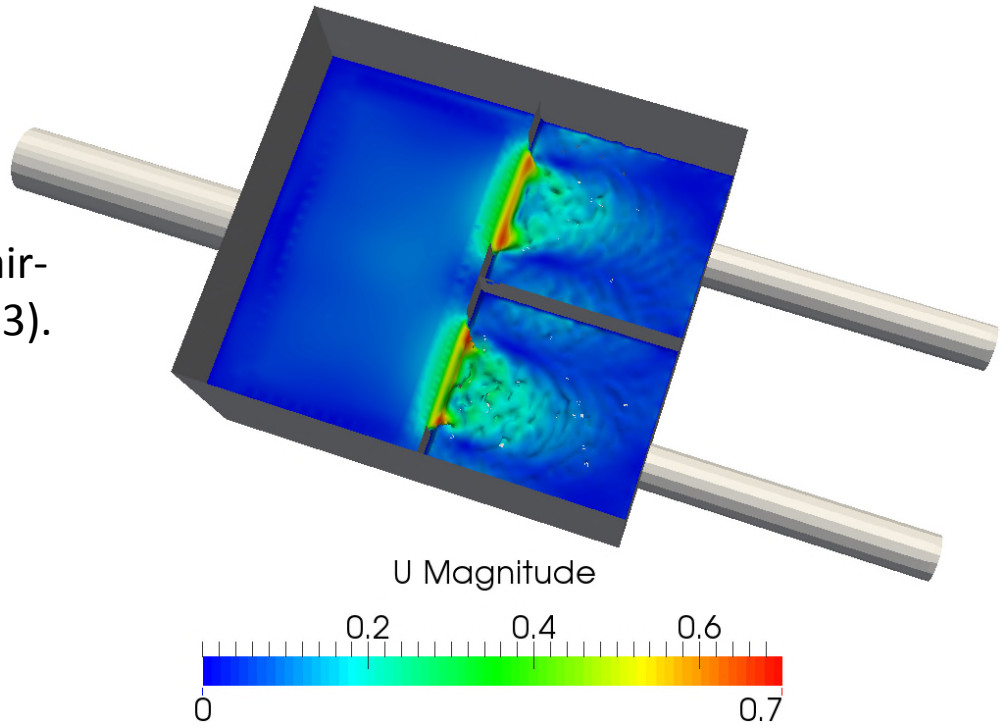
Moreover, each duct is centered on their respective faces except in the vertical direction, where they are placed at a distance of D from the box's floor.
The weirs only differ on their respective height: 0.7 m on the weir to the left of the inlet duct and 0.72 m on the other.

Case description (2/2)

Solver selection:
`interFoam`

Transient solver for the flow of two incompressible and immiscible fluids using the Volume Of Fluid (VOF) approach.

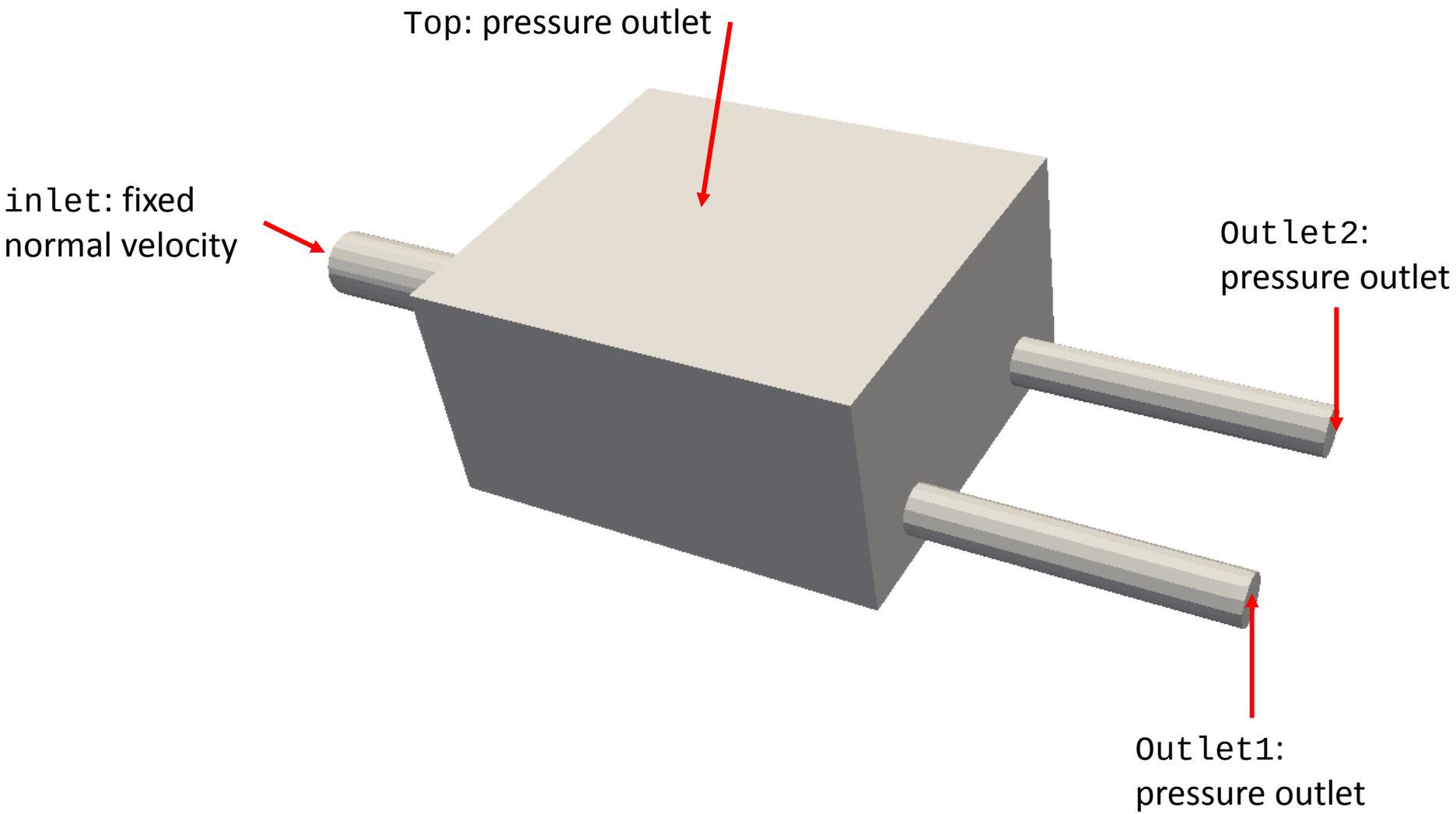
Lopes, Pedro. "Free-surface flow interface and air-entrainment modelling using OpenFOAM." (2013).



Meshing

1. Go inside case folder
   ```
   cd s01_Clarifier
   ```
2. Copy folder “s01_Start” into something with a name of your liking, e.g., “mySplitterBox”
   ```
   cp –r s01_Start mySplitterBox
   ```
3. Make sure the CAD is present in its proper location
   ```
   ls constant/triSurface/
   ```
4. Now edit the `Allrun.pre` script to have a look at the operations involved
   ```
   nano Allrun.pre &
   ```
5. Edit the blockMeshDict file
   ```
   nano Allrun.pre
   ```
6. Edit snappyMeshDict file
   ```
   nano Allrun.pre
   ```
7. Let’s launch the mesh generation operations and wait a bit
   ```
   ./Allrun.pre
   ```
8. Examine the mesh with paraFoam

Boundary conditions (1/2)



Boundary conditions (2/2)

Check the files inside directory 0.org to make sure everything is correct:

1. Move into directory 0.org

```
cd 0.org
```

2. Edit each file in turn and examine it

```
nano U
nano alpha.water
nano epsilon
nano k
nano p_rgh
```

Properties and Initialization

1. On case directory, go inside folder "`constant`"
   ```
   cd constant
   ```
2. Examine gravity
   ```
   cat g
   ```
3. Examine the turbulence model settings
   ```
   cat turbulenceProperties
   cat RASproperties
   ```
4. Examine the transport properties settings
   ```
   nano transportProperties
   ```

Initialization, on this case, will just assume quiescent water up to a certain height level at $t = 0\ s$. You can confirm this by doing:

```
cd ../system
nano setFieldsDict
```

Run preparation

1. Examine `Allrun` for an overview of operations
   ```
   nano Allrun &
   ```
2. The basic steps are
   1. Parallel decomposition
      ```
      cat system/decomposeParDict
      ```
   2. Renumber mesh
   3. Remember to have the same number of partitions everywhere
3. Results can be merged back when done if necessary. This step is mandatory if there will be a change in the number of partitions during the run.

Simulation

1. Review `fvSchemes` and `fvSolution`

   ```
   nano system/fvSchemes
   nano system/fvSolution
   ```

2. Review `controlDict`

   ```
   nano system/controlDict
   ```

   - Note function objects!

3. Submit run

   ```
   ./Allrun &
   ```

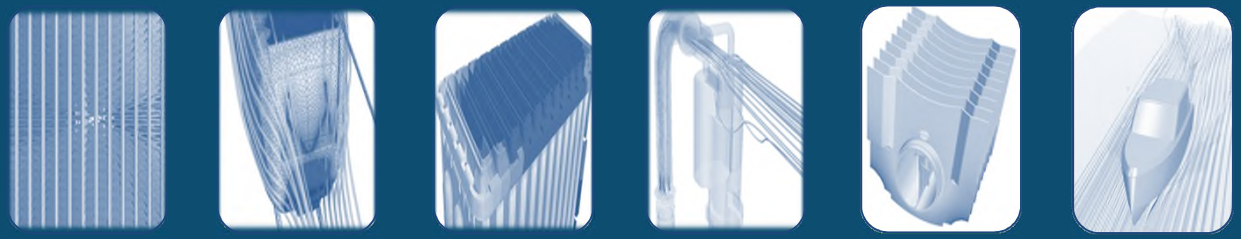
4. Monitor run progress

Results processing

1. Isosurface
    - VOF
2. Animation
3. Integral values
    - Function objects: Flux across outlets

Thank you for your time.

Next:
8 – Advanced topics



bluecape
Computer Applications
in Science & Engineering