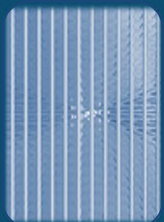
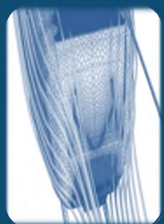
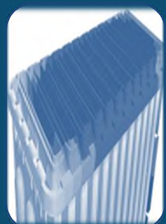
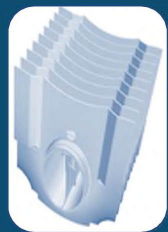
OpenFOAM®
in wastewater applications:
5 – Hands-on: Clarifier

nelson.marques@bluecape.com.pt

13-14th June 2015




bluecaPe
Computer Applications
in Science & Engineering

Progress

Introduction

- OpenFOAM characterization
- blueCFD installation

Getting started

- Simulation process overview

Meshing

- Meshers in OpenFOAM

Simulation

- How to setup and run a case

Hands-On: Clarifier

- Solids settling problem

Hands-On: UV disinfection

- Simple 2-lamps system

Hands-On: Splitter box

- Monitoring flow repartition

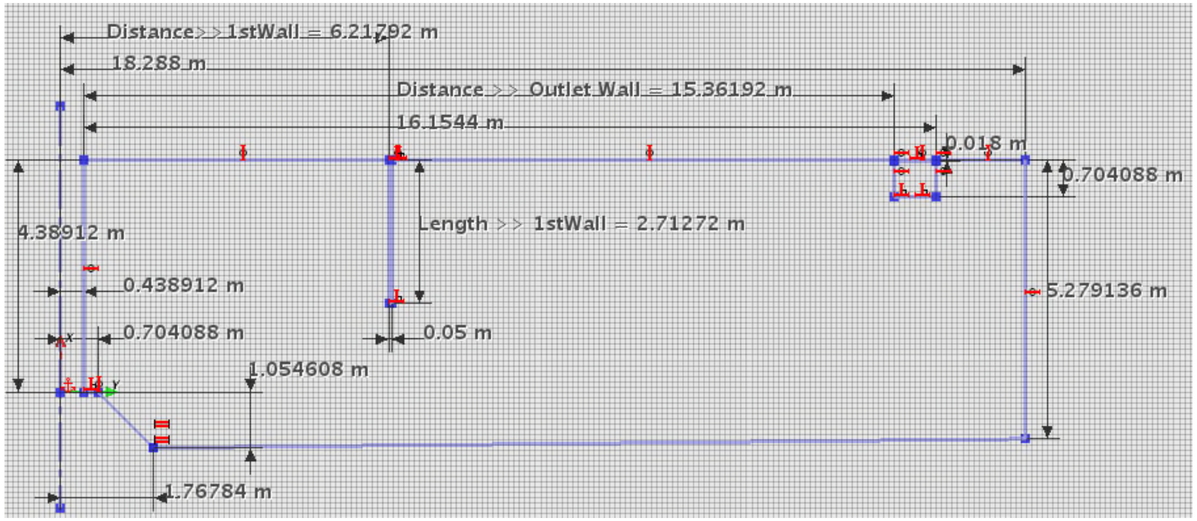
Advanced topics

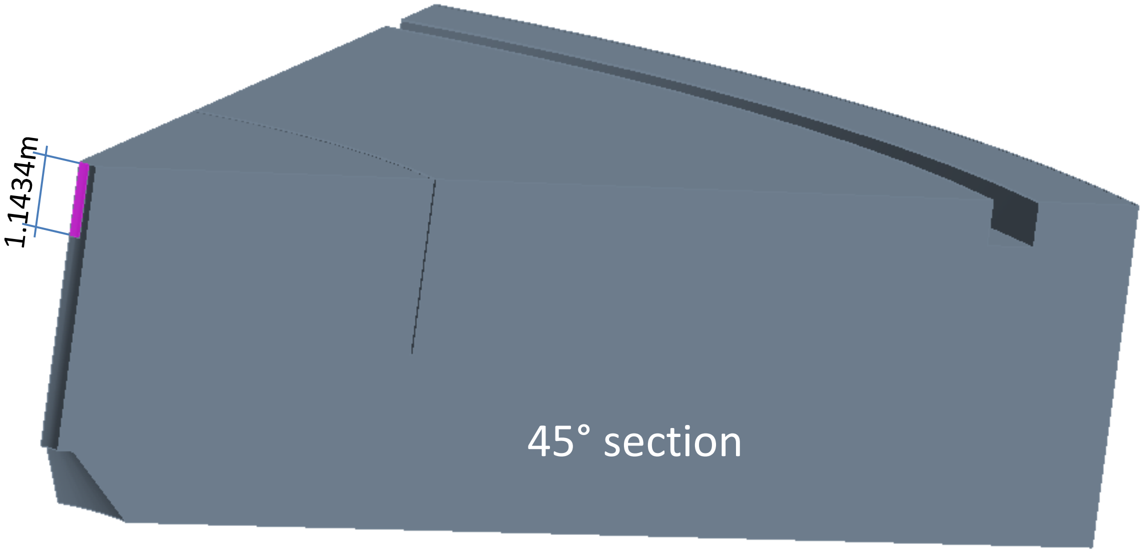
Contents

1. Case Description
   - Overview
   - Solver selection
2. Meshing
3. Boundary Conditions
4. Properties and Initialization
5. Run preparation
6. Simulation
7. Results processing

Case description (1/2)

Overview:





Inlet mass flow rate:

Inlet solids concentration:

- 3.52E-3

Water properties:

- ρ = 1000 kg/m3
- Newtonian fluid
    - ν = 1.78E-6 m2/s

Sludge properties:

- ρ = 1042 kg/m3
- Non-newtonian fluid
    - Bingham plastic

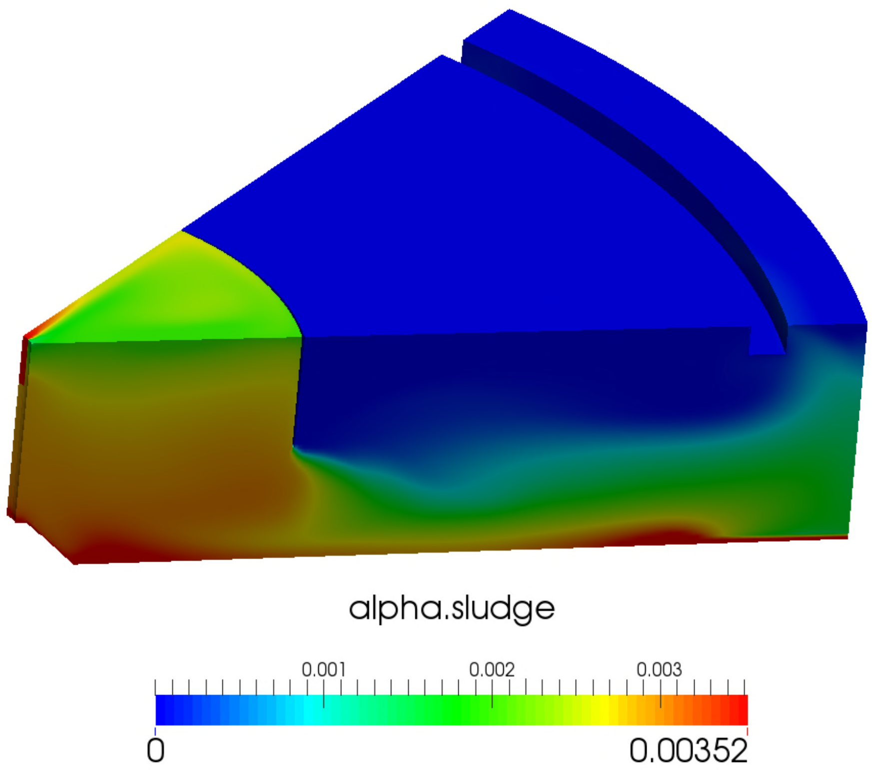
Case description (2/2)

Solver selection:

`driftFluxFoam`

Solver for 2 incompressible fluids using the mixture approach with the drift-flux approximation for relative motion of the phases.

Used for simulating the settling of the dispersed phase and other similar separation problems.



Brennan, D., "The Numerical Simulation of Two-Phase Flows in Settling Tanks", PhD Thesis, Imperial College, January 2001

Meshing (1/4)

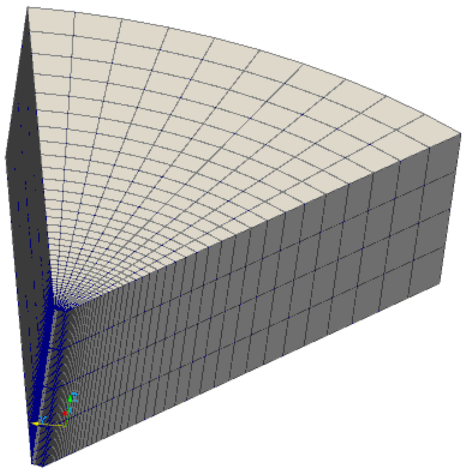
The mesh generation strategy is a bit more convoluted than usual, due to the geometrical "1/8th of a cylinder" shape.

The following steps are shown in the next 2 slides:

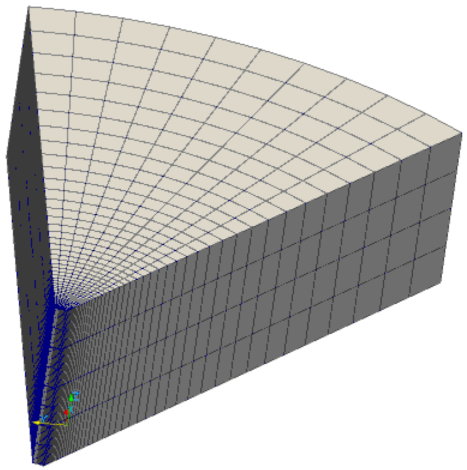
1. Generate the background mesh, as a cylindrical mesh.
   - Very fine near the cylindrical centre and becomes coarser in proportion to the distance to the centre.
2. In 6 passes, the mesh is refined only along the height, depending on the distance to the centre.
3. In 3 more passes, the mesh farthest from the centre is refined along all 3 directions (radius, angle, height).
4. Mesh is extruded.
5. "snappyHexMesh" in 2 passes: outer geometry, then deflector.
6. Cells next to the outlets (at the launder) are refined.

Meshing (2/4)

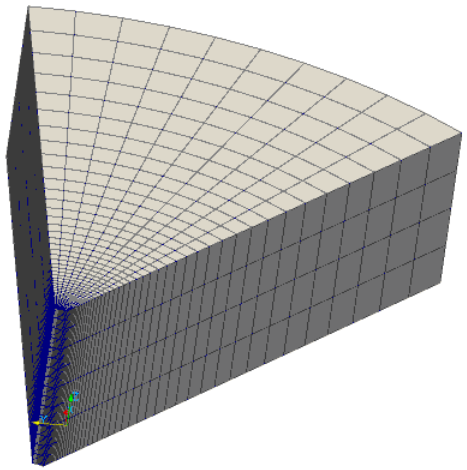
1. background Mesh



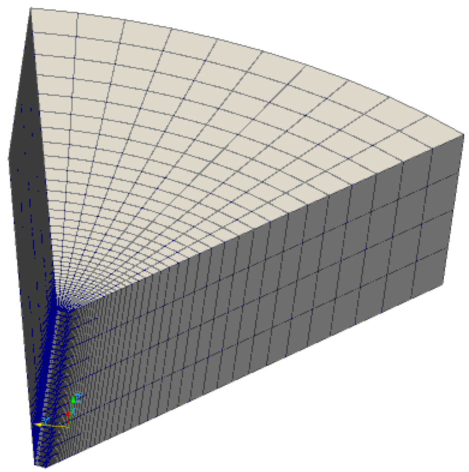
2. Refine Z, R<0.64m



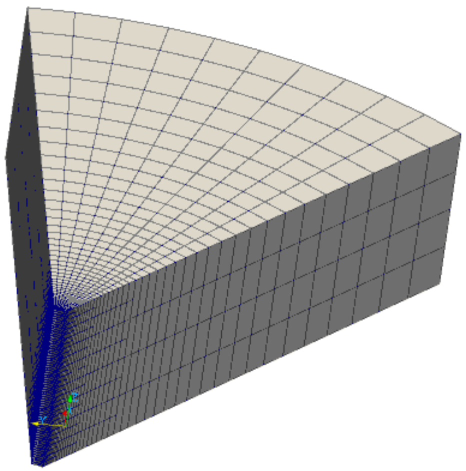
3. Refine Z, R<1.03m



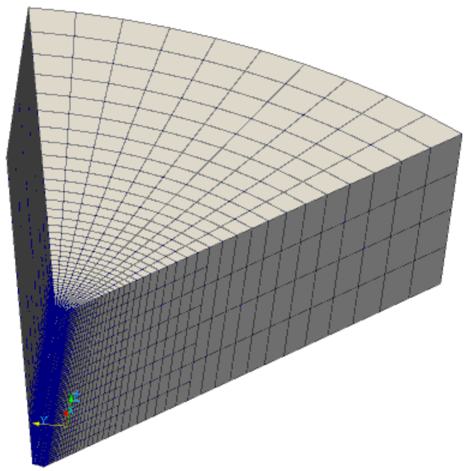
4. Refine Z, R<1.94m



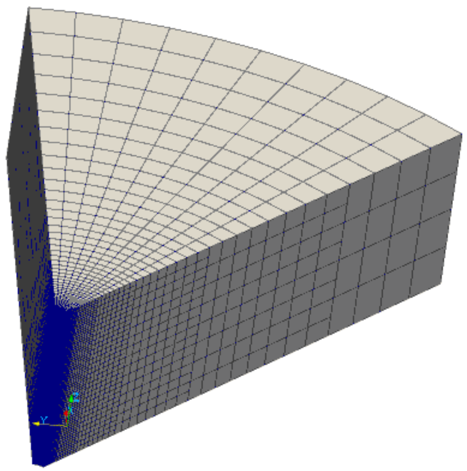
5. Refine Z, R<3.67m



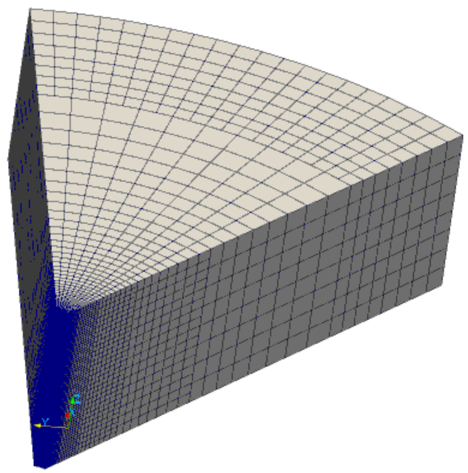
6. Refine Z, R<7.00m



7. Refine Z, R<13.36m

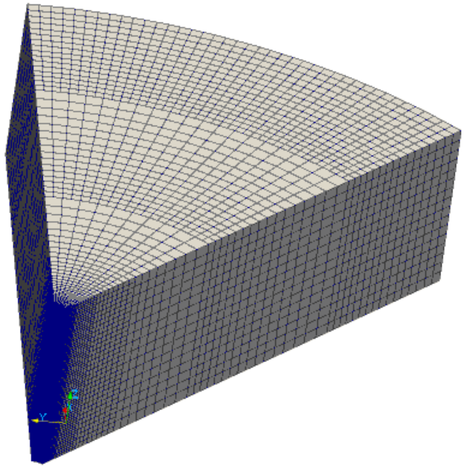


8. Refine RθZ, R>13.36m

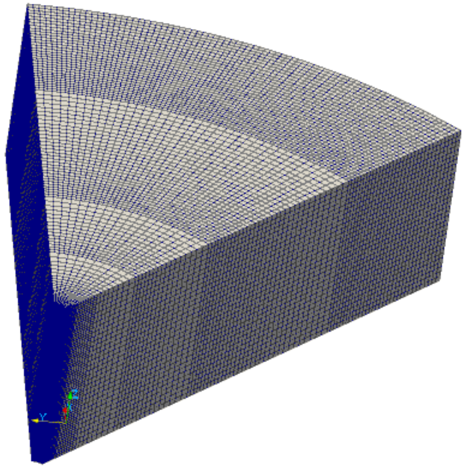


Meshing (3/4)

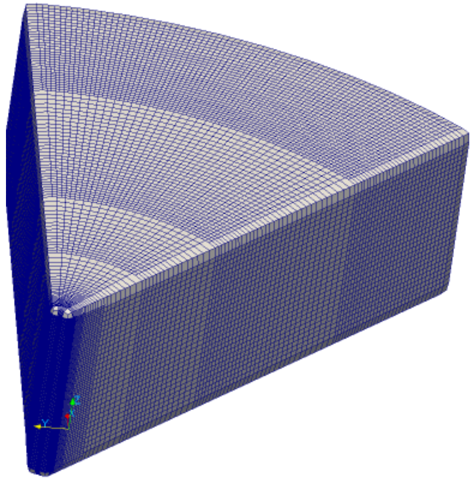
9. Refine RθZ, R>7.0m



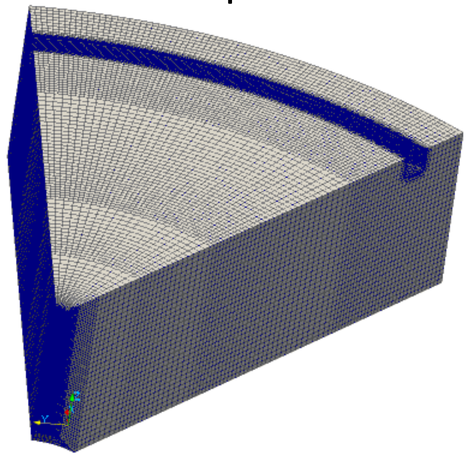
10. Refine RθZ, R>3.67m



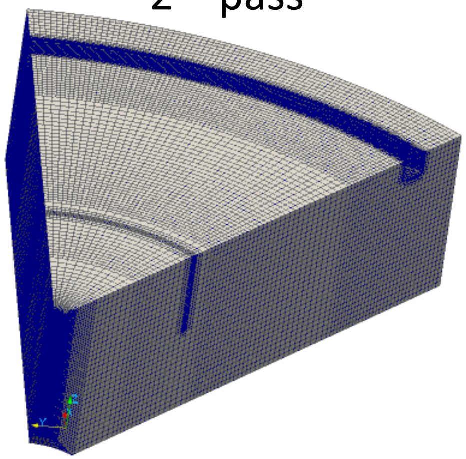
11. Extrude mesh



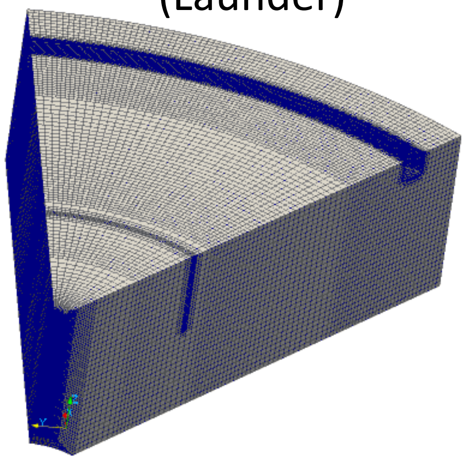
12. snappyHexMesh
1st pass



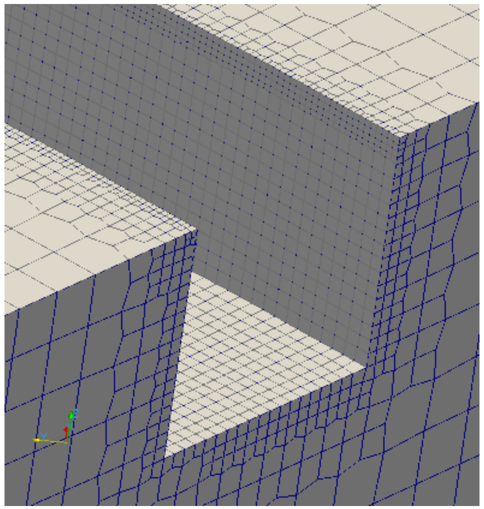
12. snappyHexMesh
2nd pass



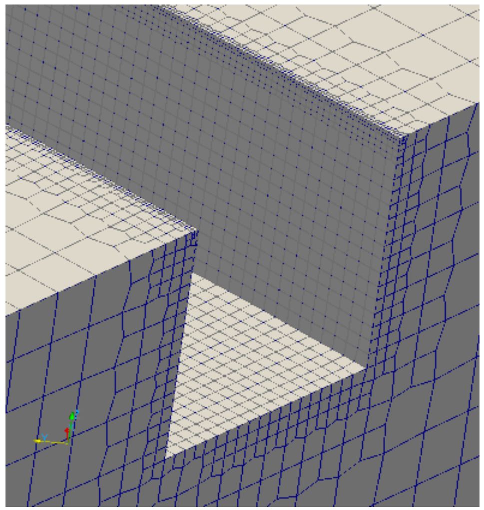
13. Refine Outlets
(Launder)



12. (Zoom-in)



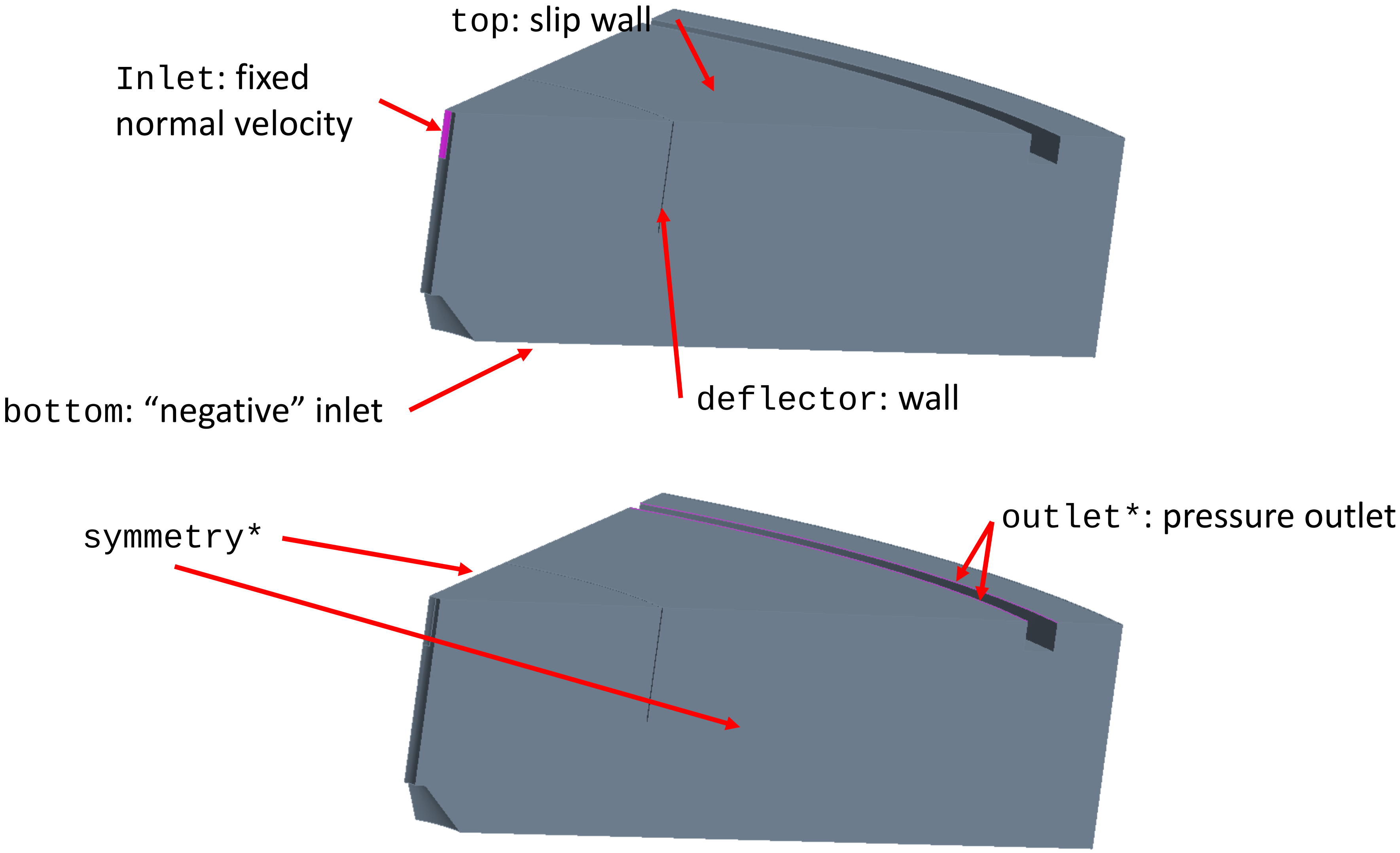
13. (Zoom-in)



Meshing (4/4)

1. Go inside case folder
   `cd s01_Clarifier`
2. Copy folder “s01_Start” into something with a name of your liking, e.g., “myClarifier”
   `cp -r s01_Start myClarifier`
3. Now edit the Allrun.pre script to see the operations involved
   `nano Allrun.pre &`
   - **calcRadiusField** – creates a field with the radius distances and optionally the refinement directions; source code present at “calcRadiusField”
   - **topoSet** – used for creating a cell selection list (*cellSet*)
   - **refineMesh** and **refineMeshRadial** – used for refining each *cellSet*; source code for the latter present at “refineMeshRadial”
4. Let’s launch the mesh generation operations and wait a bit
   `./Allrun.pre`
5. Examine the mesh with paraFoam

Boundary conditions (1/2)



Everything else: wall

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 p_rgh
nano alpha.sludge
nano k
nano epsilon
nano nut
```

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
   ```
4. Examine the transport properties settings
   ```
   nano transportProperties
   ```

Initialization, on this case, will just assume quiescent water everywhere at $t = 0\ s$. You can confirm this by doing:

```
cd ../0.org
cat alpha.sludge
```

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
   ```

3. Submit run
   ```
   ./Allrun &
   ```

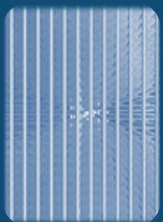
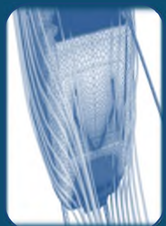
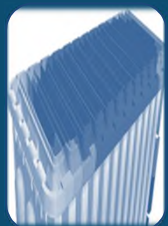
4. Monitor run progress
   - If and when the case crashes, adjust the time step and resume the run.

Results processing

1. Contour and section plots
2. Thresholds
3. Integral values

Thank you for your time.

Next:
6 – Hands-On: UV disinfection


bluecaPe
Computer Applications
in Science & Engineering