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

nelson.marques@bluecape.com.pt

13-14th June 2015
**Progress**

**Introduction**
- OpenFOAM characterization
- blueCFD installation

**Getting started**
- Simulation process overview

**Meshing**
- Meshers in OpenFOAM

---

**Hands-On: UV disinfection**
- Simple 2-lamps system

**Hands-On: Clarifier**
- Solids settling problem

**Simulation**
- How to setup and run a case

---

**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
Overview:

Inlet mass flow rate:

**Inlet solids concentration:**
- 3.52E-3

**Water properties:**
- \( \rho = 1000 \text{ kg/m}^3 \)
- Newtonian fluid
  - \( \nu = 1.78E-6 \text{ m}^2/\text{s} \)

**Sludge properties:**
- \( \rho = 1042 \text{ kg/m}^3 \)
- Non-newtonian fluid
  - Bingham plastic
Case description (2/2)

Solver selection:

\texttt{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.

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
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)

- **Inlet**: fixed normal velocity
- **Top**: slip wall
- **Bottom**: "negative" inlet
- **Deflector**: wall
- **Symmetry**:
- **Outlet**: pressure outlet
- **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”
   ```bash
cd constant
   ```
2. Examine gravity
   ```bash
cat g
   ```
3. Examine the turbulence model settings
   ```bash
cat turbulenceProperties
   ```
4. Examine the transport properties settings
   ```bash
   nano transportProperties
   ```

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

```bash
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.
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**