OpenFOAM®
in wastewater applications:
6 – Hands-on: UV disinfection

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
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:**
- 1 kg/s

**Radiation Intensity:**
- 40W/lamp

**Water properties:**
- Newtonian fluid
  - $\nu = 1.0 \times 10^{-6} \text{ m}^2/\text{s}$

The channel’s height is 23cm and its width is 10 cm.
The lamps diameter is 1 inch and their length is 1m.
Case description (2/2)

Radiation details:

wall

wall

1 m
Solver selection: buoyantBoussinesqPimpleFoam

Transient solver for buoyant, turbulent flow of incompressible fluids

...and it includes radiation.

Li, Chan, et al, “A numerical prediction on the reduction of microorganisms with UV disinfection”, Journal of Mechanical Science and Technology, 24(7), 21010
Meshing

1. Go inside case folder
   `cd s03_UVlamps`

2. Copy folder “s01_Start” into something with a name of your liking, e.g., “myUVlamps”
   `cp –r s01_Start myUVlamps`

3. Make sure the CAD is present in its proper location
   `ls constant/triSurface/UVlamps.org.stl`

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 (1/3)

inlet
{
    type flowRateInletVelocity;
    massFlowRate 1.0;
    rho rho;
    rhoInlet 999.0;
}

Wall

outlet
{
    type pressureInletOutletVelocity;
    value uniform (0.0 0.0 0.0);
}

top
{
    type slip;
}

Temperature:
boundaryField
{
    ".*"
    {
        type fixedValue;
        value uniform 0;
    }
}
Boundary conditions (2/3)

```
".*Support.*"
{
    type MarshakRadiationFixedTemperature;
    Trad 0;
    emissivityMode lookup;
    emissivity uniform 0.7;
    value uniform 0;
}
```

```
bottomLamp
{
    type MarshakRadiationFixedTemperature;
    Trad 300;
    emissivityMode lookup;
    emissivity uniform 1.0;
    value uniform 0;
}
```
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 G`
   `nano T`
   `nano U`
   `nano alphat`
   `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 RASProperties
   ```

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

5. Examine the radiation solver settings
   
   ```
   nano radiationProperties
   ```

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

   ```
   cd ../0.org
   cat U
   ```
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**
1. First mesh is coarse (57296 cells). Let’s create a finer one and use the fields just computed
2. Create new case directory and edit mesh resolution parameters
   `cp -r myUVlamps myUVlamps_2`
   `cd myUVlamps_2`
   `nano system/snappyHexMeshDict`
   `./Allclean; ./Allrun.pre`
3. Modify Allrun to interpolate fields between meshes
   `nano Allrun`
4. Change stopping criteria and submit run
   `nano system/controlDict`
   `./Allrun &`
5. Monitor run progress
Results processing

1. Contour and Section plots
   1. G and U
2. Integral values
3. Streamlines
   1. Filter: custom source
   2. Save Data to export
4. Y+ at walls
   
yPlusRAS
Thank you for your time.

Next:

7 – Hands-On: Splitter box