Virtual Tracer Tests: Coupling CFD and CREng to Simulate WRRFs Unit Processes

High Rate Algal Pond case study

teshome@etu.unistra.fr

1st September 2019

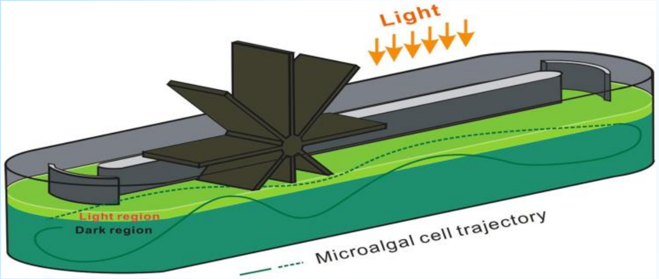
Overview

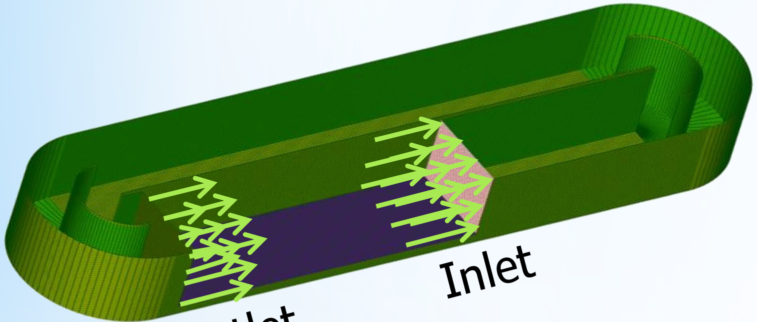
* Introduction to OpenFOAM and installation
* **Getting started:** Simulation process overview
* Meshing tools in OpenFOAM
* Setting up a case and run simulation
* Hands-On Coupling Computational Fluid Dynamics and Chemical Reaction Engineering

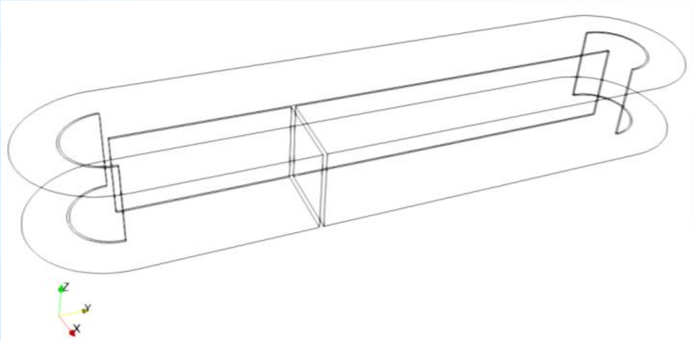
Contents

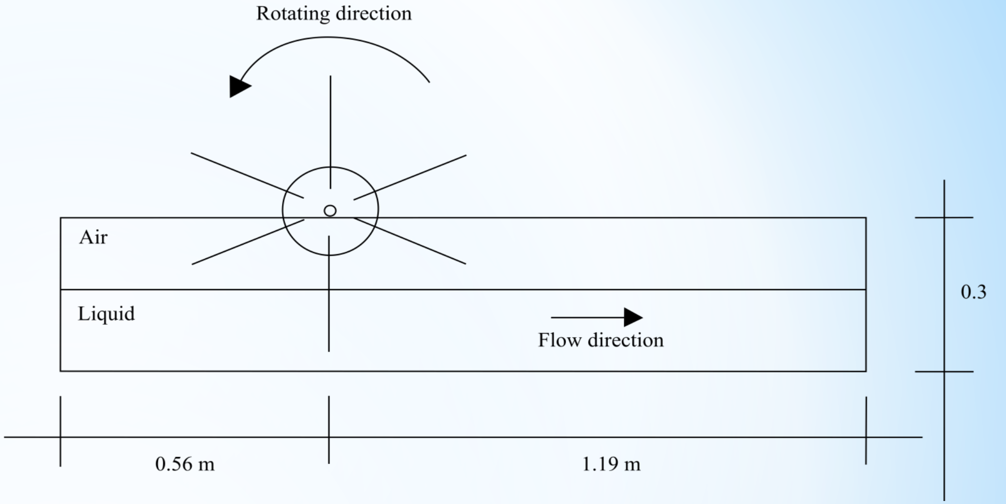
1. Geometry and Flow Properties
2. Solver selection
3. Meshing
4. Boundary Conditions
5. Properties and initialization
6. Run preparation
7. Simulation
8. Post-processing

1. Geometry and Flow Properties (1/2)

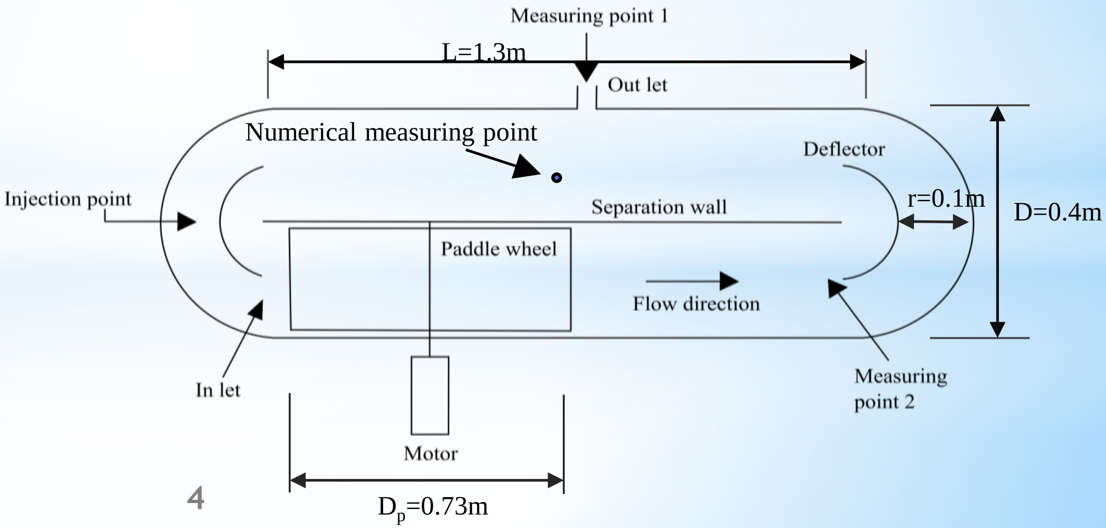








Side view



Top view

1. Geometry and Flow Properties (2/2)

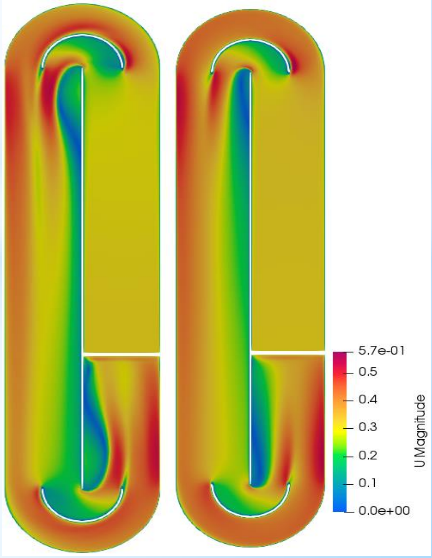
- Inflow velocity=0.311m/s in the Y-direction
- Steady state
- Single phase and Incompressible fluid
- Turbulence model κ-ε

2. Solver Selection

- **simpleFoam**
  - solve continuity and momentum equations to calculate the flow fields in single phase incompressible fluid.

$$\nabla \cdot \boldsymbol{U} = 0$$

$$\frac{\partial \boldsymbol{U}}{\partial t} + (\boldsymbol{U} \cdot \nabla)\boldsymbol{U} = \upsilon \nabla^2 \boldsymbol{U} - \nabla p + g$$



2. Solver Selection

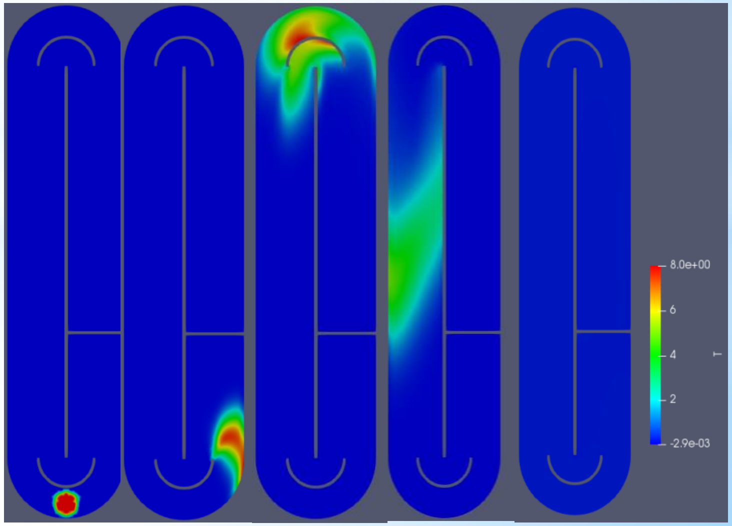
- **scalarTransportFoam**
  - Solves transport equation (convection-diffusion) on a given velocity fields.
  - No source term

$$\frac{\partial T}{\partial t} + (\nabla \cdot \boldsymbol{U}T) - \nabla^2(DT) = 0$$

where T is the transported scalar

U is fluid velocity and

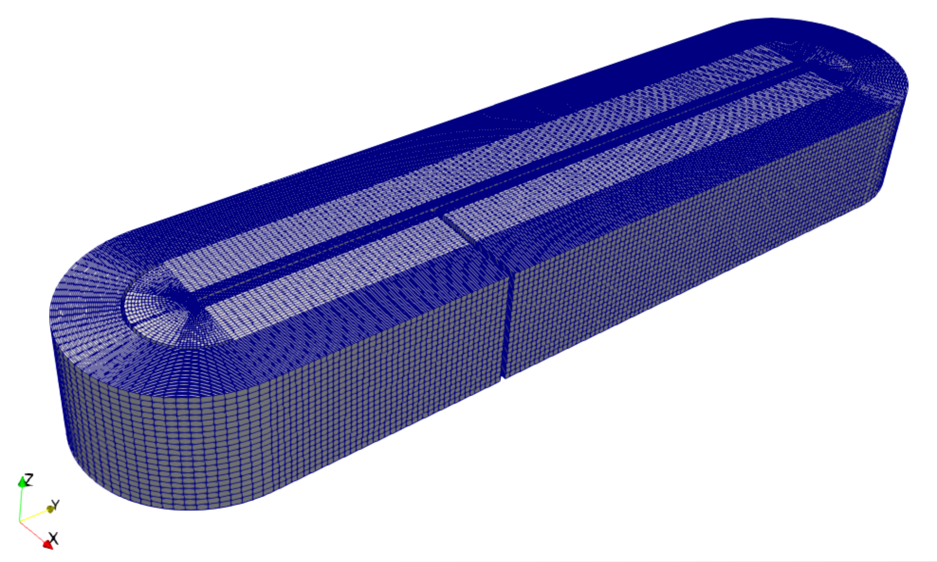
D is the diffusion coefficient



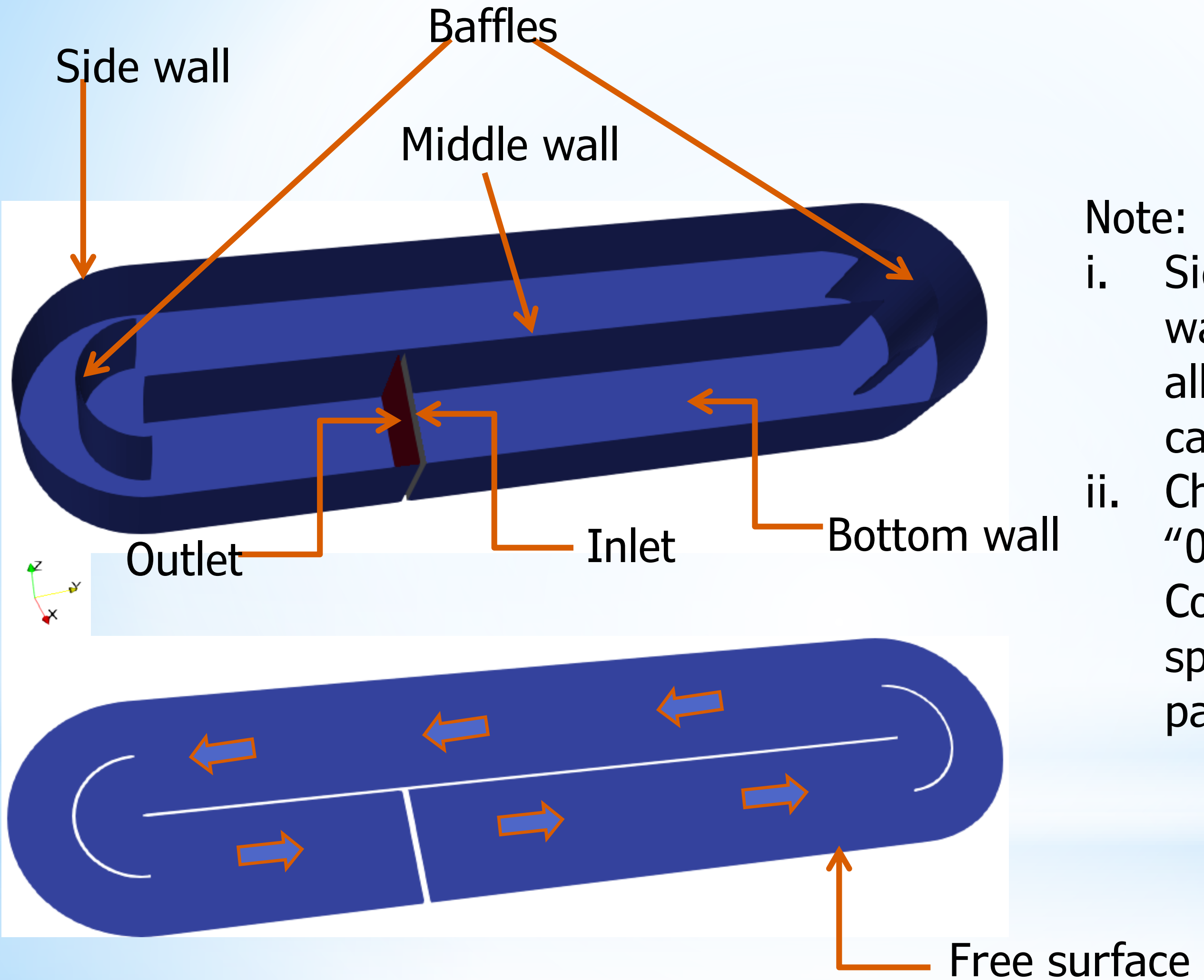
3. Meshing

- **Block Mesh**
  - Basic mesh generating tool in OpenFOAM
  - Dictionary “blockMeshDict” in “system” folder
  - Command: blockMesh

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  5                                     |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      blockMeshDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

convertToMeters 1;

vertices

(
    ( 0 -0.2 0) //0
    ( 0.2 0 0) //1
    ( 0.2 1.3 0) //2
    ( 0 1.5 0) //3
    ( 0 1.5 0.2) //4
```



4. Boundary Conditions (1/2)



Note:

i. Side wall, Baffles, Middle-wall and Bottom wall are all merged to **Walls** for this case.

ii. Check inside the directory “0” if the Boundary Condition is correctly specified for all the field parameters (nut, T and U).

5. Properties and Initialization

1. In the case directory, go inside folder “constant”

   cd constant

2. Examine the turbulence model settings

   cat turbulenceProperties

   cat RASproperties

3. Examine the transport properties settings

   nano transportProperties

The passive scalar material is initialized by “**setFields**” dictionary in the “system” folder. In this case ‘sphereToCell’ method is used to defined the initial position and shape of the scalar material.

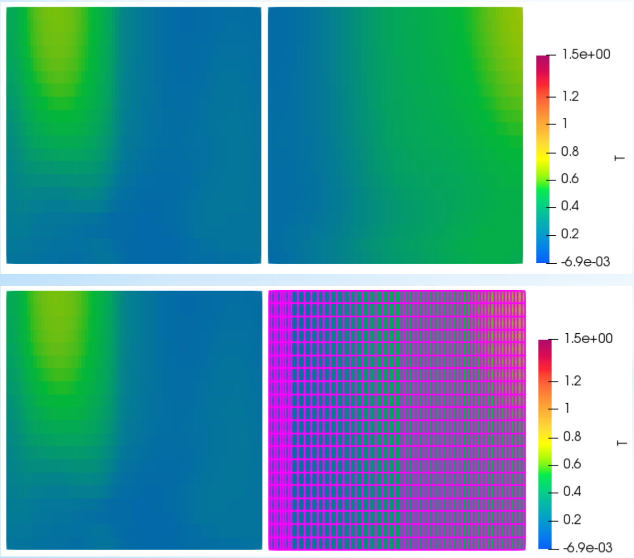
6. Simulation

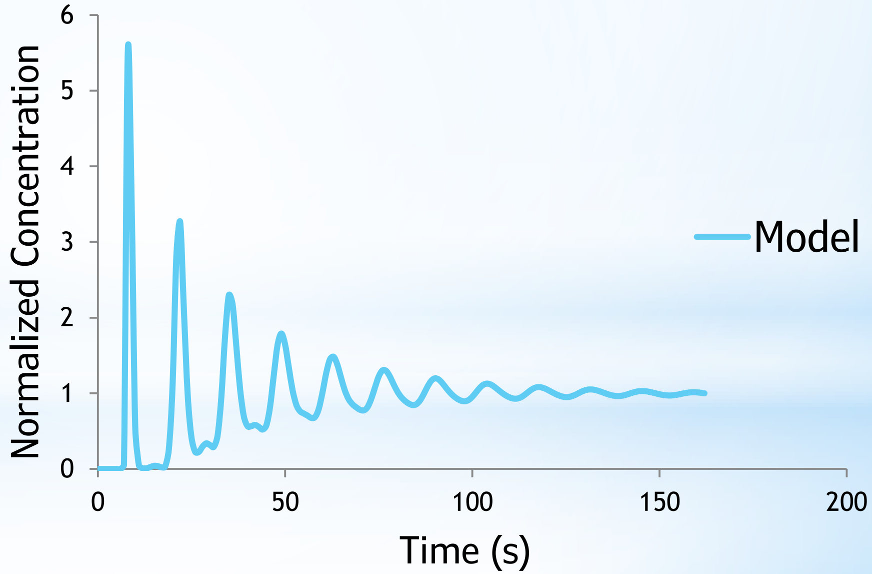
1. Review **fvSchemes** and **fvSolution**
   nano system/fvSchemes
   nano system/fvSolution
2. Review controlDict
   nano system/controlDict
3. Submit run

4. Monitor run progress

8. Post Processing

1. Horizontal slice – passive scalar convection and diffusion
2. Vertical Slice and cell selection
3. Time serious plot





Thank you for your attention!