Numerical simulations of Sieve Plate Pulsed Column using OpenFOAM

Dr. Raj Kumar Saini

Ph.D, Indian Institute of Technology, Bombay (IIT Bombay)

M.Tech, Indian Institute of Technology, Madras (IIT Madras)

Email : raj.km.saini@gmail.com

December 19, 2019

Abstract

This case study demonstrates the hydrodynamics of the pulse sieve plate column. The geometry (3D) and meshing of the column are created using Ansys workbench ('Design-modeler & Fluent Meshing'). The aim of this study is to investigate the hold-up distribution and velocity profile in the pulse sieve plate column. The simulations are performed using OpenFOAM-4.x. The dynamic behavior of hold-up distribution is captured and the simulations results of the column are analyzed.

Keywords: Hydrodynamics, Pulse flow, Volume of Fluid, Pulse sieve plate column, Two-phase flow, CFD

1 Introduction

The purpose of this case study is to use OpenFOAM software [1] to the new users for the advance system. The aim of this work is to understand the pulsatile flow in a three-dimensional sieve plate pulsed column (SPPC) [2, 3]. In this simulation, a pressure-based finite volume method is used for incompressible flow. The simulations are carried out using OpenFOAM. In this case, two phases of flow simulation approaches are considered for incompressible transient flow with using **interFoam** solver in OpenFOAMO-4.x [1]. This case study demonstrates how to do the following:

- Set up a problem case;
- Creating a 3D mesh by using ANSYS workbench (sppc.msh);
- Mesh imported in to OpenFOAM (fluentMeshTofoam);
- solve a transient problem using the VOF model;
- Create the geometry and import the geometry in OpenFOAM;
- Set up the properties of the fluids;
- Initialize the flow;
- To implement time dependent boundary condition, $v(t) = V0 * sin(2\pi f * t)$;
- Consider the turbulent model to capture the turbulence;
- Post processing the case for results.

2 Problem statement

This case considers the transient simulation of hydrodynamics behavior of sieve plate pulsed column (SPPC). Figure 1 shows three-dimensional geometry of SPPC considered in the present study. The dispersed and continuous phases are is considered toluene and water respectively. The fluid property for the present study is shown in Table 1.

Initially, the domain is filled with continuous phase and the bottom is filled with dispersed phase and the disperse phase is patched upto a height between 0 - 0.12 cm respectively.

The inlets velocities of the dispersed phase and continuous phase are 0.0156 m/s & 0.0468 m/s (+ve, x-axis) respectively. At time zero, the disperse phase is patched and will start entering the SPPC.

The geometrical parameters and operating conditions are following:

- Plate spacing : 50 mm
- Column diameter : 50 mm
- Hole diameter : 4.8 mm
- No of hole in plate : 25
- Plate thickness: 1 mm
- Dispersed phase: **toluene**
- Continuous phase: **water**
- Ratio of flow rate continuous phase/dispersed phase: 3:1
- The intensity of flow (Af, mm/s), A and f are amplitude (0.010 m) and frequency (1, f Hz) of pulse
- The intensity of flow (Af) =5 to 10

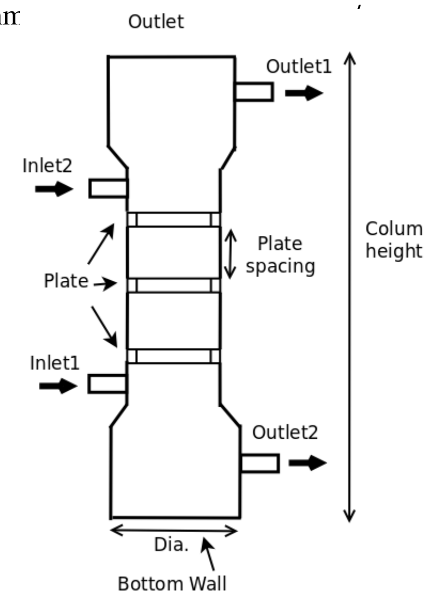


Figure 1: Sieve plate pulsed column

Table 1: Details of fluids properties

Parameters	Value
Dynamic viscosity, Continuous phase (μ_1), Pa.s	1e-06
Dynamic viscosity, Dispersed phase (μ_2), Pa.s	0.7e-06
Density, Continuous phase (ρ_1), kg/m^3	1000
Density, Dispersed phase (ρ_2), kg/m^3	870
Surface tension, Continuous phase-Dispersed phase (σ), N.m	0.07

3 Mathematical modeling

InterFoam (solver) solves the Navier-Stokes equations for isothermal and incompressible flow. It means that all material properties are constant in the region filled by one of the two fluid except at the interphase.

3.1 Continuity equation

The continuity equation (constant-density) is defined as:

$$\frac{\partial u_j}{\partial x_j} = 0 \tag{1}$$

3.2 Momentum equation

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j u_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\tau_{ij} + \tau_{t_{ij}}\right) + \rho g_i + f_{\sigma i} \tag{2}$$

where, u represents the velocity, ρ density of the fluid, p the pressure, g_i the gravity and $\tau_{t_{ij}}$, τ_{ij} and $f_{\sigma i}$ are the turbulent, viscose stresses and the surface tension respectively.

The density (ρ) is defined in the Equation- 3:

$$\rho = \alpha\rho_1 + (1-\alpha)\rho_2 \tag{3}$$

The value of α is 1, it means that the fluid 1 occupied with the density ρ_1 and the fluid 2 is occupied with the density ρ_2 with 0 the value of α. The α varies between 0 and 1 at the interphase between the two fluids.

The Continuum Surface Force (CSF) approach is used to modelled the surface tension ($f_{\sigma i}$) [4, 5]. It can be calculated using Equation- 4:

$$f_{\sigma i} = \sigma\kappa\frac{\partial\alpha}{\partial x_i} \tag{4}$$

σ and κ are the surface tension constant and radius of the curvature respectively. The curvature can be estimated using Equation- 5 [4, 5].

$$\kappa = -\frac{\partial n_i}{\partial x_i} = -\frac{\partial}{\partial x_i}\left(\frac{\partial\alpha/\partial x_i}{|\partial\alpha/\partial x_i|}\right) \tag{5}$$

3.3 Equation of interphase or Volume fraction

In order to investigate where the interphase between the two fluids is, an additional equation for α is to be solved.

$$\frac{\partial\alpha}{\partial t} + \frac{\partial(\alpha u_j)}{\partial x_j} = 0 \tag{6}$$

3.4 Boundary conditions

Details of boundary name and corresponding boundary conditions are presented in Table 2.

Table 2: Boundary conditions

Boundary Name	Boundary condition
topInlet	Velocity
bottomInlet	Velocity
topOutlet	Velocity
bottomOutlet	Velocity
pressureOutlet_1	Pressure
movingWall_1	sine wave; $v(t) = V0 * sin(2\pi f * t)$
Wall	No slip

4 Simulation procedure

This case deals with three-dimensional turbulence flow simulations in sieve plate pulsed column. First step in setting up of an OpenFOAM case is to copy to present working directory. We need to set all require input parameters before starting the simulation. Mesh generation and implementation of boundary conditions are adopted from a base nozzleFlow2D tutorial. This study is considered with transient, turbulence case. The solver settings are imported from nozzleFlow2D tutorial (OpenFOAM/(username)-4.x/run/tutorials/multiphase/interFoam/les/nozzleFlow2D/) from **interFoam**.

4.1 Creating geometry and mesh

- Geometry for the present problem is considered 3D dimensional geometry of sieve plate pulsed column. The geometry and mesh are generated using ANSYS workbench ('design-modeler & Fluent Meshing'), the file (sppc.msh) is available in 'case/' folder in user directory (∼/case/).
- Mesh can be imported to in OpenFOAM format by using the command '**fluentMeshToFoam**' in the terminal.
- Figure 2 shows the isometric view of the generated mesh using fluentMeshToFoam utility.
- All modifications for mesh with proper boundary condition with defining inlet and outlet is to be done as velocity patch and its neighbor patch (∼/case/constant/polyMesh/boundary).

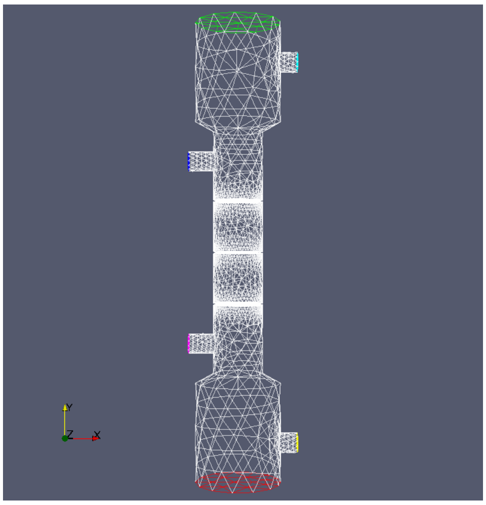


Figure 2: Computational geometry of column

4.2 Setting boundary conditions (BC)/Initial conditions (IC)

Files present in '0' folder (∼/case/0) has been kept 'alpha.water', 'alpha.water.org', 'p_rgh' and 'U' files for turbulent and transient flow. Boundaries are assigned and added seven boundaries of present case in three the files, i.e. 'alpha.water', 'p_rgh', 'U' and other respectively. Details of the boundary conditions are listed in Table 3.

Table 3: Details of boundary conditions

Boundary	p_rgh	U	alpha.water
topInlet	fixedFluxPressure	fixedValue (0.0156 0 0)	zeroGradient
bottomInlet	fixedFluxPressure	fixedValue (0.0468 0 0)	zeroGradient
topOutlet	fixedFluxPressure	fixedValue (0.0468 0 0)	zeroGradient
bottomOutlet	fixedFluxPressure	fixedValue (0.0156 0 0)	zeroGradient
pressureOutlet_1	fixedValue	inletOutlet	zeroGradient
movingWall_1	fixedValue	uniformFixedValue(sine wave)	zeroGradient
Wall	zeroGradient	No slip	zeroGradient

4.3 Solver details

The volume of fluid (VOF) method for phase fraction is used for incompressible, isothermal, immiscible fluids based on interface capturing approach with optional mesh and geometrical reconstruction technique is used (piecewise linear interface calculation) in simulation. In the present study, unsteady state and turbulent flow are considered. Turbulent flow model can be applied in the OpenFOAM by using 'simulationType' option in the 'turbulenceProperties' file in constant folder. The 'simulationType' option can be used as '**LES**'. In order to run unsteady state simulations, controlDict, decomposeParDict (for parallel computation), fvSchemes,fvSolution, and setFieldsDict files are kept in the system directory folder. '**./run**' command executes in terminal to run computations.

4.4 Post-processing

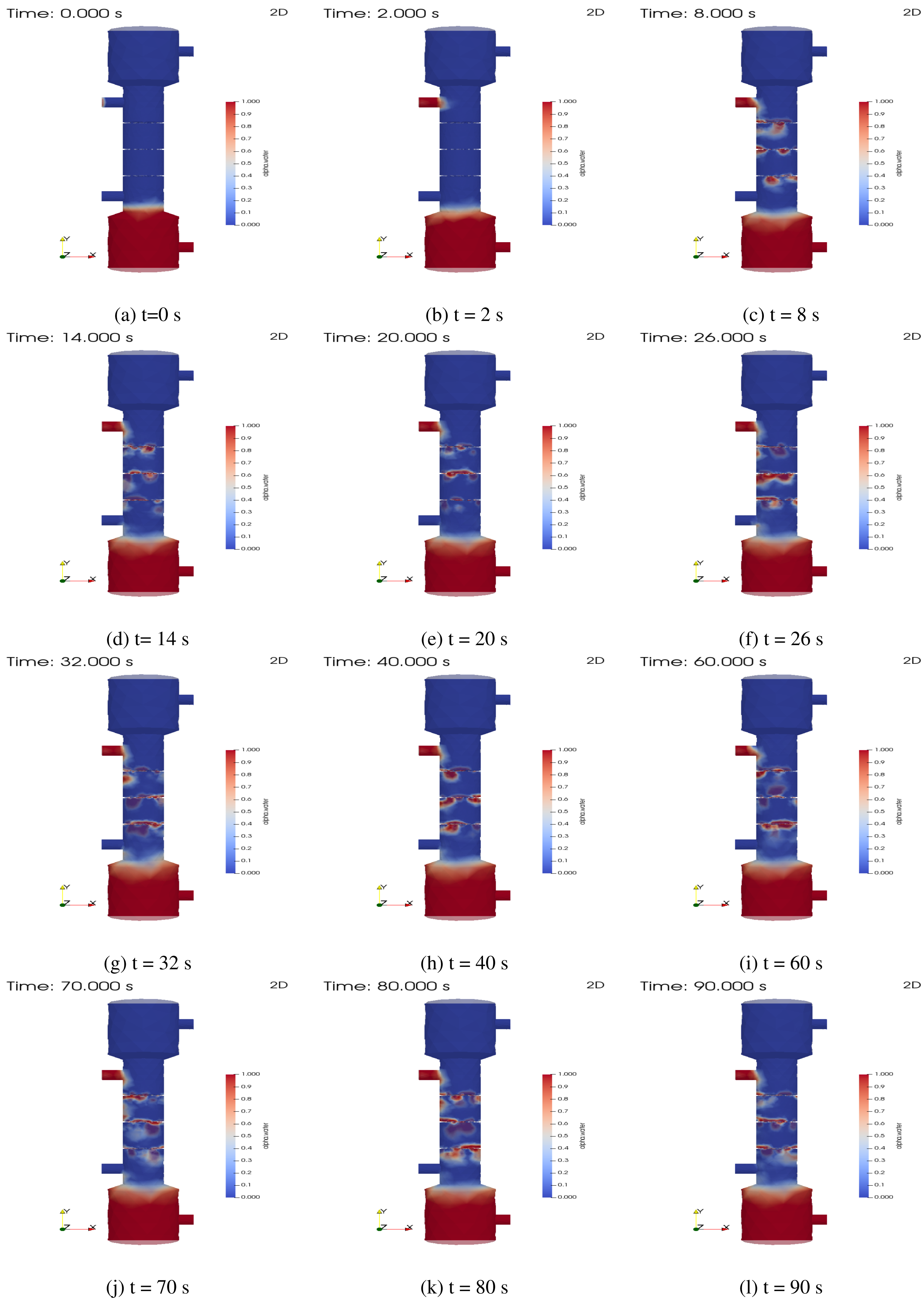
The paraFoam can be used to visualize the solution results. This can be done by typing the following command line in the terminal **paraFoam** to open the paraview software and upload the case.

5 Results and discussion

Simulations are performed using OpenFOAM to study the hydrodynamics of sieve plate pulsed column (SPPC). Simulation results are analyzed with the help of 'paraview' (∼/case/ paraFoam). Figures 3 & 4 show the field α (volume fraction) for dispersed phase (toluene) and velocity profile at different times.

Figure 3 shows the volume fraction of the disperse phase (toluene) contours plots captured in paraview with different time span. At time 0, the dispersed phase is patched with the help of setfieldsDict. So the flow fields such as volume fraction of dispersed phase contour remain zero at that time (Figure 3a). After that these dispersed phase starts entering into the column.

Figures 3a to 3l, as time passes, the dispersed phase will start to entire into the column and will reach to bottom of the column. Figures 4a to 4l show the velocity distribution in the column at time. In this work, at the time of 90 sec are enough to understand the system get stable. The simulation can start further and will obtain the extract time value when the dispersed phase is completely distributed throughout the column.



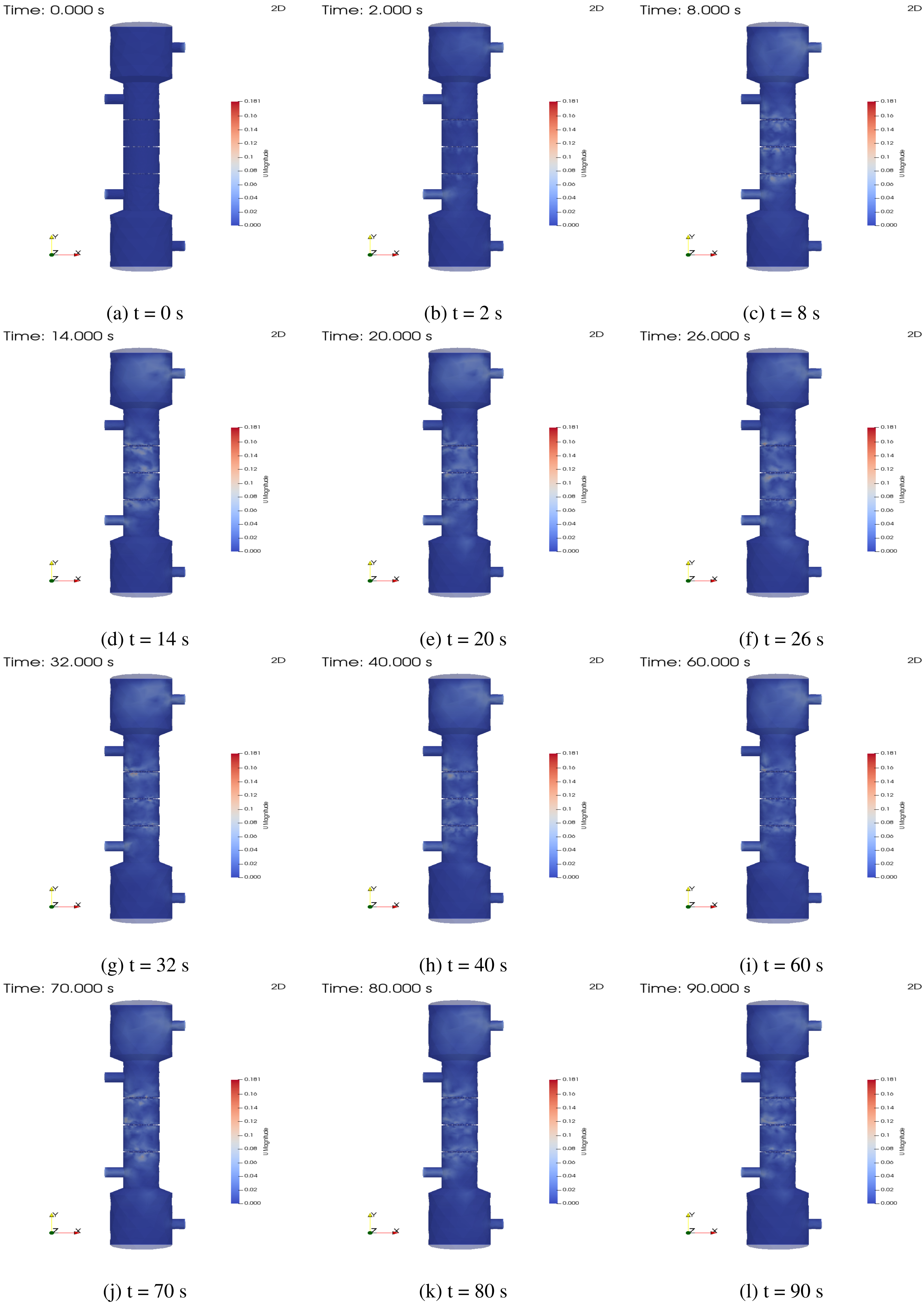
(a) t=0 s (b) t = 2 s (c) t = 8 s

(d) t= 14 s (e) t = 20 s (f) t = 26 s

(g) t = 32 s (h) t = 40 s (i) t = 60 s

(j) t = 70 s (k) t = 80 s (l) t = 90 s

Figure 3: Field α, isometric view of volume fraction of dispersed phase at different times (red color correspond to α=1, water and blue color corresponds to α= 0, toluene).



(a) t = 0 s (b) t = 2 s (c) t = 8 s

(d) t = 14 s (e) t = 20 s (f) t = 26 s

(g) t = 32 s (h) t = 40 s (i) t = 60 s

(j) t = 70 s (k) t = 80 s (l) t = 90 s

Figure 4: Flow field v, isometric view of velocity profile at different times .

References

[1] C. J. Greenshields, *OpenFOAM: The OpenFOAM Foundation. User Guide Version 4*. CFD Direct Limited, June. 2016.

[2] N. Sen, K. K. Singh, A. W. Patwardhan, S. Mukhopadhyay, and K. T. Shenoy, "Cfd simulations of pulsed sieve plate column: Axial dispersion in single-phase flow," *Separation Science and Technology*, vol. 50, no. 16, pp. 2485–2495, 2015. [Online]. Available: https://doi.org/10.1080/01496395.2015.1064136

[3] C. Jiao, S. Ma, and Q. Song, "Mass transfer characteristics in a standard pulsed sieve-plate extraction column," *Energy Procedia*, vol. 39, pp. 348 – 357, 2013. [Online]. Available: http://www.sciencedirect.com/science/article/pii/S1876610213013088

[4] J. Brackbill, D. Kothe, and C. Zemach, "A continuum method for modeling surface tension," *Journal of Computational Physics*, vol. 100, no. 2, pp. 335 – 354, 1992.

[5] J. A. Heyns and O. F. Oxtoby, "Modelling surface tension dominated multiphase flows using the vof approach," 2014.