FOSSEE, IIT Bombay

OpenFOAM Case Study Project

May 18, 2020

Numerical Simulations Of Water Bottle Filling In OpenFOAM

Nandha Kumar.R*

B.E –Mechanical Engineering,

Sri Ramakrishna Engineering Collage.

Abstract

This case study numerically simulates the behavior of the water inside the water bottle. The flow behavior of the water when it filling the water bottle in the atmospheric condition is analyzed. The simulation is carried out in openFOAM v7 using the solver interFOAM. The geometry is a 3-D geometry and the flow is laminar and transient.

1.Introduction

It is aimed to simulate the water bottle filling when the water flows from the top of the inlet patch. The interFoam solver is used to simulate the case file. openFOAM v7 is the software used for this study purpose. The water bottle is considered to be a wall. Interfoam is a solver used for incompressible isothermal immiscible fluids using volume of fluid method (VOF).

Structure of the case file

```
──────┬── 0        ────── alpha.water , p_rgh , U .
      ├── Constant ── polyMesh , triSurface , g , transport properties , turbulence properties .
      └── System   ──  blockMeshDict , controlDict , fvschemes , fvsolution
                       SetFieldsDict , snappyHexMeshDict , surfaceFeaturesDict
```

2.Case Setup

2.1. Geometry

The geometry is done by salome-meca. the length of the bottle in the y direction(height) is 0.3 m with the radius of 0.05 m. the inlet was fixed at the top with 0.1 m radius. it’s a 3-D geometry. Figure 2.1.A and 2.1.B shows the geometry.

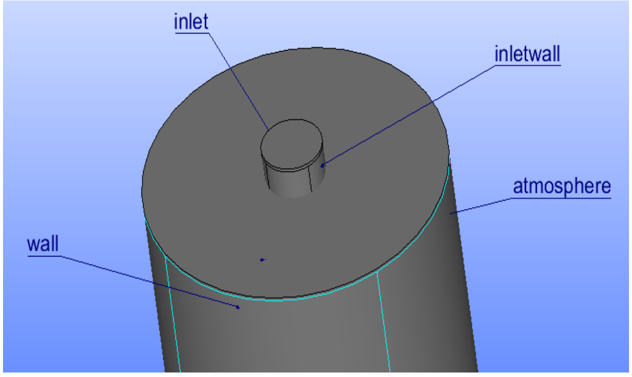


Figure 2.1.A

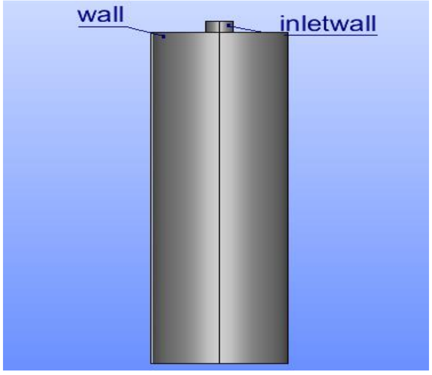


Figure 2.1.B

2.2. Mesh

The mesh is created using the Salome-Meca. The mesh was created with Hex mesh.The mesh was exported using UNV format . ideasUnvToFoam command is used to import the mesh file into the case file. figure2.2.A shows the mesh of the geometry.

Total Number of volumes -26854

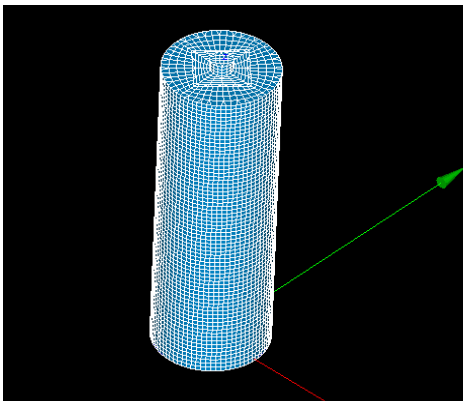


Figure 2.2.A

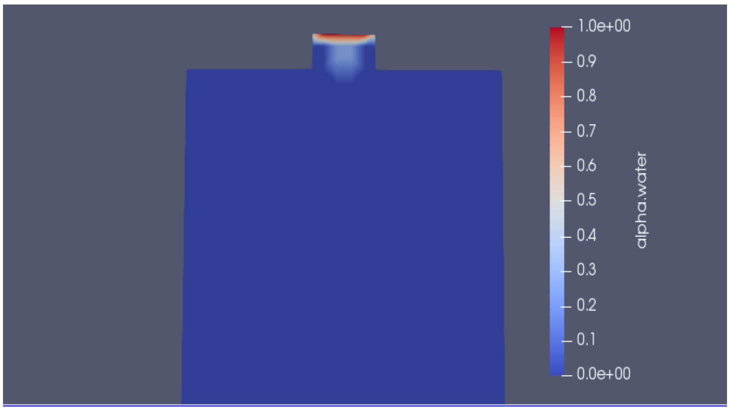


Figure 2.4.A

2.3. Boundary Conditions

Boundry name	Alpha.water	p_rgh	U
inlet	fixedValue(uniform 1)	fixedFluxPressure	FlowRateInVelocity (volumetric flow rate constant 0.1)
inletWall	zeroGradient	zeroGradient	noSlip
Wall	zeroGradient	zeroGradient	noSlip
Atmosphere	inletOutlet(uniform0)	totalPressure P0-uniform 0	pressureInletOutletVelocity (uniform (0 0 0))
defaultFaces	zeroGradient	zeroGradient	noSlip
Internalfield	uniform 0	uniform 0	uniform(0 0 0)

'0' file folder contains boundry condtions and initial conditions. The boundary conditions are written for U , p_rgh , water .

2.4. SetField

alpha.Water fixed as 1

Point 1 - (0 0 3)	Point 2 - (0 0 3.5)	Radius - 0.01

these lines in set fields dictionary will fix the inlet with water. Figure 2.4.A shows the water fixed at the inlet using setFields command.

2.5. Transport Properties

the transport properties dictionary contains the viscosity(nu) and the density of the fluid(rho).The model of the fluid is defined in this dictionary.

Fluids	Model	nu	rho
Air	Newtonian	1.48e-05	1
Water	Newtonian	1e-6	1000

Surface tension coefficients sigma 0.07 - (air and water)

2.6. g and turbulence properties

g – gravitational force – (0 0 -9.81)

which acts on the negative Z direction

turbulence properties – the flow is laminar

2.7. Equations

the equation used are –

Constant-density continuity equation

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

density equation

$$\rho = \alpha\rho_1 + (1-\alpha)\rho_2$$

Momentum equation

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

Equation for interphase

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

2.8. Calculations

Volume of the bottle = 2.356194 m^3

Area of the inlet = 0.036194 m^2

Mass flow rate $\dot{m}$ = 0.1 m^3/s

figure 2.8.A

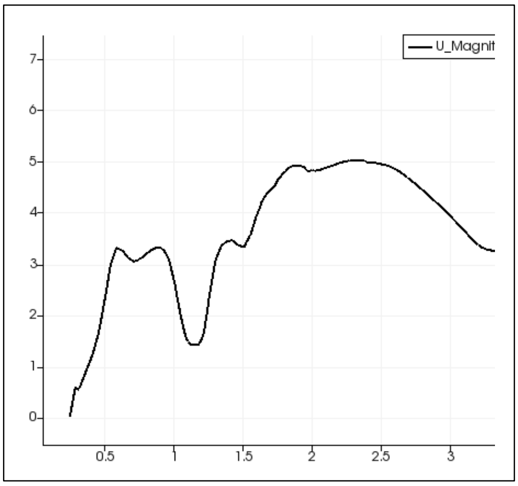
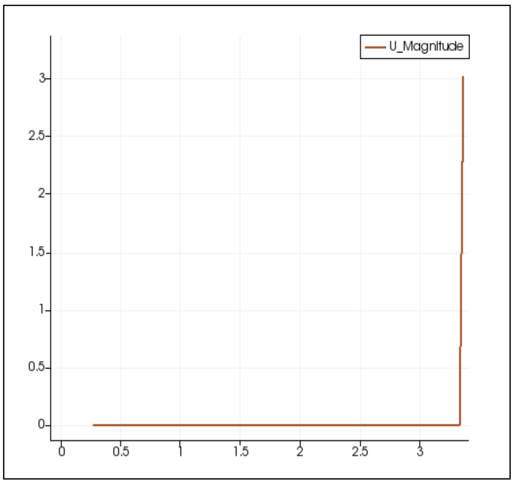


figure 2.8.B



Expected time to fill the tank = 2.35614 / 0.1 = **23.56 sec**

$$\dot{m} = A \times v$$

$$0.1 = 0.031416 \times v$$

Expected velocity at the inlet v = **3.183 m/s**

From figure 2.8.B we can verify the velocity. Figure 2.8.A represents the velocity profile at 15 s.

3.Results and Discussion

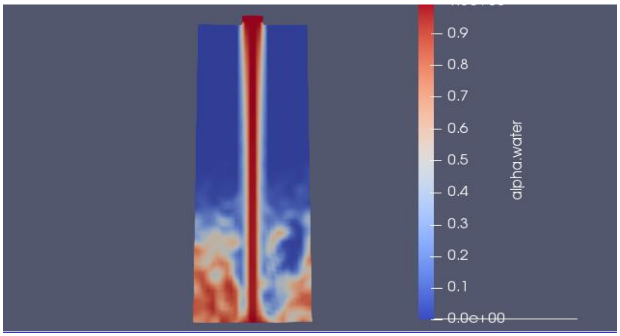
The simulation carried out with adjustable run time which adjusts the time step in accordance to Courant Number. The max courant number is fixed as 0.5. After the simulation the post processing is done.

3.1. Post processing

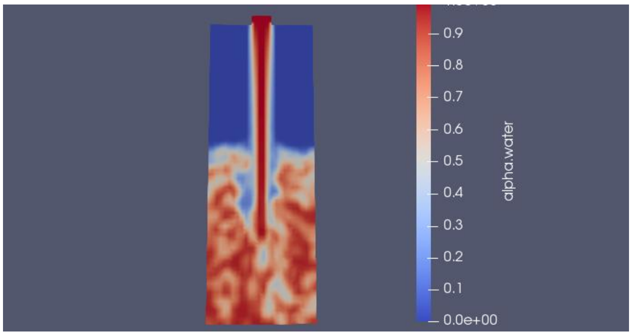
The post processing is done using paraview software. The fluid behaviour inside the Water bottle is analysed. The command paraFoam Is used to read the data. then paraview is opened up and the post process files are opened for visualizing the results.

3.2. Results

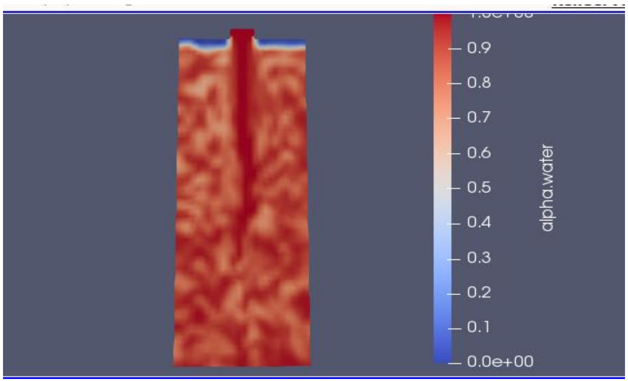
from the analysis we can able to examine the fluid flow at filling of water bottle and the fluid behavior of the water and air of fluid over the time. Figure 3.2.a , 3.2.b , 3.2.c shows the simulation for 5, 10 , 20 sec respectively.



time 5s figure 3.2.A

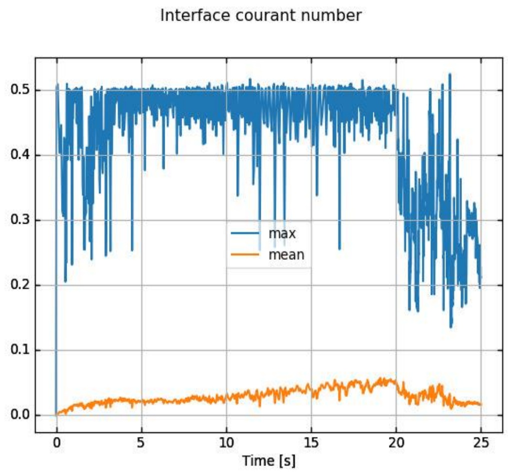


time 10s figure 3.2.B

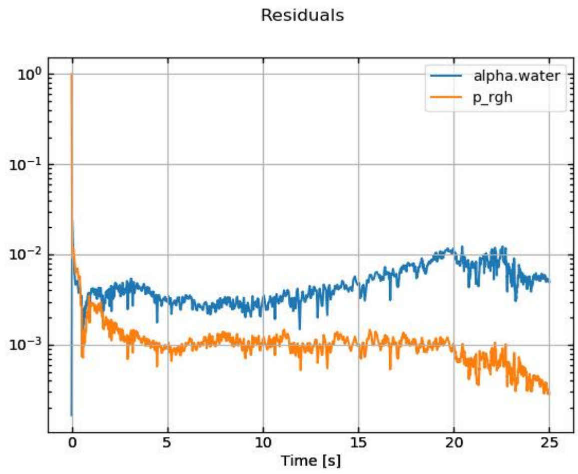


time 20s figure 3.2.C

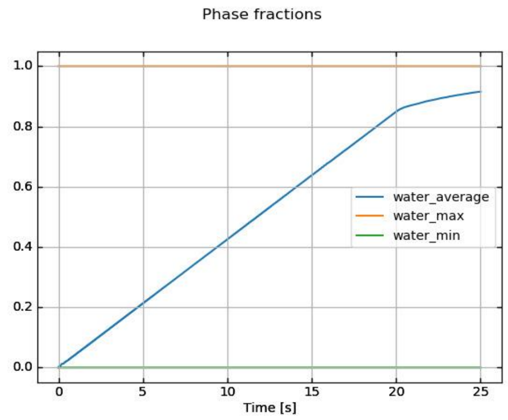
3.3. Plots



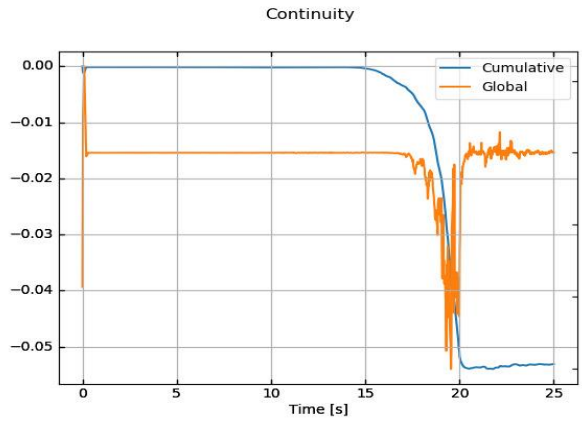
Plot 1



Plot 2



Plot 3



Plot 4

Plot 1 - Shows the mean and max courant number . from this we can able to know max courant number approximately not exceeded more then 0.5.

Plot 2 - shows the residuals of alpha.water and p_rgh.

Plot 3 - shows the phase fractions of water over the time.

Plot 4 - shows the continuity graph over time.

3.4. Conclusion

This case study explains the flow behaviour of the 2 incompressible immiscible fluids (air and water)

Using VOF fluid method using interFoam solver in open foam. This can be analysed for various fluids by varying the physical parameters.

3.5. Reference

1. interFoam in openFoam wiki – https://openfoamwiki.net/index.php/InterFoam
2. tutorials in open foam
   https://wiki.openfoam.com/Multiphase_(VOF)_Simulation_Project_by_Jozsef_Nagy