

# **Fluid jet impinged into another fluid matrix**

**Kriti Chaturvedi**

Department of Chemical Engineering

Indian Institute of Technology, Bombay

## **Abstract**

This case study demonstrates how a fluid (ink) jet acts when impinged into another fluid matrix. A 3D geometry is made in FreeCAD and meshed using the snappyHexMesh utility of OpenFoam. The flow is considered laminar and in the absence of gravity as a very thin stream is injected and surface tension effects would cancel out the gravity effects. A uniform velocity profile of the injected fluid is considered at the inlet. The study is carried out using OpenFoam (version – 07).

## **1. Introduction**

This case study aims to analyse the behaviour of a fluid (ink) jet when impinged into another fluid matrix. Through this analysis, the maximum distance a fluid jet can travel within the fluid matrix without breaking up into streams is calculated. This analysis is useful in the design of an organic fluid 3D printer. It helps us to decide where to keep the printing plate in the fluid matrix so that we get a proper strip layer. The ink and the fluid matrix are considered to be immiscible. The injector is modelled as a cylindrical pipe and the fluid matrix domain is modelled as a cubical tank.

## **2. Problem Statement**

The geometry consists of a cylindrical pipe ( $d=0.5\text{m}$ ,  $h=50\text{m}$ ) and a cubical tank ( $300\text{m} \times 300\text{m} \times 300\text{m}$ ). Ink is injected from the inlet at the top of the cylindrical pipe. The flow is laminar and in the absence of gravity.

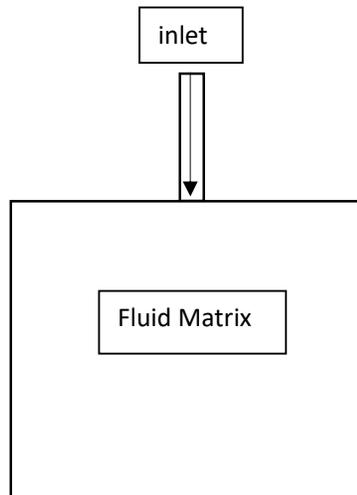


Fig. 1: Ink jet entering into fluid matrix

The fluids are immiscible. The following table demonstrates the fluid properties of the two fluids used:

Surface Tension (Ink/Fluid) – 0.03 N/m

Table1: Fluid Properties

Property	Ink	Fluid
Kinematic Viscosity (m <sup>2</sup> /s)	10 <sup>-6</sup>	4x10 <sup>-6</sup>
Density (kg/m <sup>3</sup> )	1000	1000

We simulated the initial condition when the tip of the cylindrical injector just touches the surface of the fluid matrix. No other medium is taken in between. The following figure shows the initial setup.

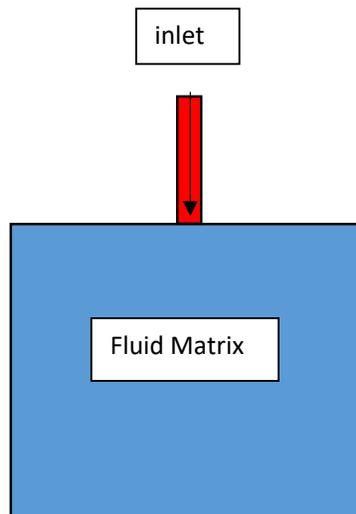


Fig. 2: Initial Condition

### 3. Governing Equations

The governing equations of the interFoam solver of OpenFoam are:

1. Continuity Equation:

$$\nabla \cdot \mathbf{u} = 0 \quad (1)$$

2. Momentum Equation:

$$\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \cdot \rho \nu [2S] + F \quad (2)$$

3. Volume of Fluid:

$$\rho = \alpha \rho_1 + (1 - \alpha) \rho_2 \quad (3)$$

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{u}) + \nabla \cdot (\alpha (1 - \alpha) \mathbf{u}_r) = 0 \quad (4)$$

In the above equations  $\mathbf{u}$  denotes the velocity of the fluid,  $\rho_1$  signifies density of the ink jet,  $\rho_2$  is the density of the fluid matrix,  $\mathbf{u}_r$  refers to the velocity at the interface,  $\alpha$  denotes the fraction of the ink in a cell, when  $\alpha = 1$ ;  $\rho = \rho_1$ , when  $\alpha = 0$ ;  $\rho = \rho_2$ .

### 4. Simulation Procedure

Firstly, the solver was validated using and the existing dam break tutorial in OpenFoam. The case files consist of 3 folders case, geometry, and mesh. The .stl files made in FreeCAD are stored in the geometry folder. First, go to the mesh folder through the terminal and copy the .stl files of the geometry folder into the triSurface folder in the constant folder using the command, `cp ../geometry/*.stl constant/triSurface`. Then mesh the geometry in the mesh folder with these 3 commands:

1. `surfaceFeatureExtract`
2. `blockMesh`
3. `snappyHexMesh -overwrite`

After the meshing is done go to the case folder through the terminal and copy the meshed geometry details into the case folder using the command, `cp -r ../mesh/constant/polyMesh constant`. Then run the case using the following commands:

1. setFields
2. interFoam

Then view it in paraview using the paraFoam command.

#### 4.1 Geometry and Mesh

The geometry and the patches were created in the FreeCAD software. The geometry consisted of 2 parts the cylindrical pipe and the cubical tank. The following table shows the dimensions of the geometry.

Table 2: Dimensions of the geometry

Cylinder	Diameter= 10m, Height= 50m
Cubical Tank	300m x 300m x 300m

Firstly, a block covering the entire geometry was made using blockMesh then it was further refined using snappyHexMesh.

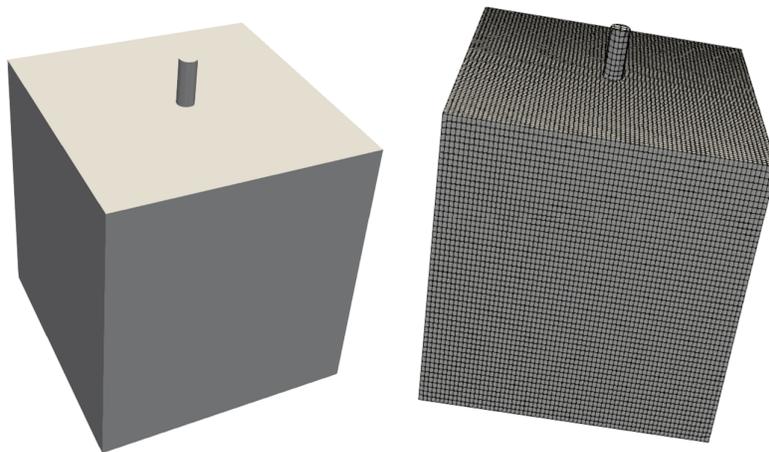


Fig. 3: isometric view of the geometry and the mesh

A grid independence study was performed to obtain the optimum size of the mesh. Three grids were made by changing the values of number of cells in the blockMeshDict file which is available in the 'system' folder in the mesh directory (~/.mesh/system). The refinement level in the snappyHexMeshDict (~/.mesh/system) file is taken to be (1,1). The following table shows the no. of blocks in each of the grids:

Table 3: Grid Specifications

Grid	Specification	No. of cells
Grid 1 (coarse)	(25, 25, 40)	100240
Grid 2 (moderate)	(30, 30, 40)	140508
Grid 3 (fine)	(35, 35, 40)	181094

Velocity (u) was plotted at t=5secs and error analysis was done.

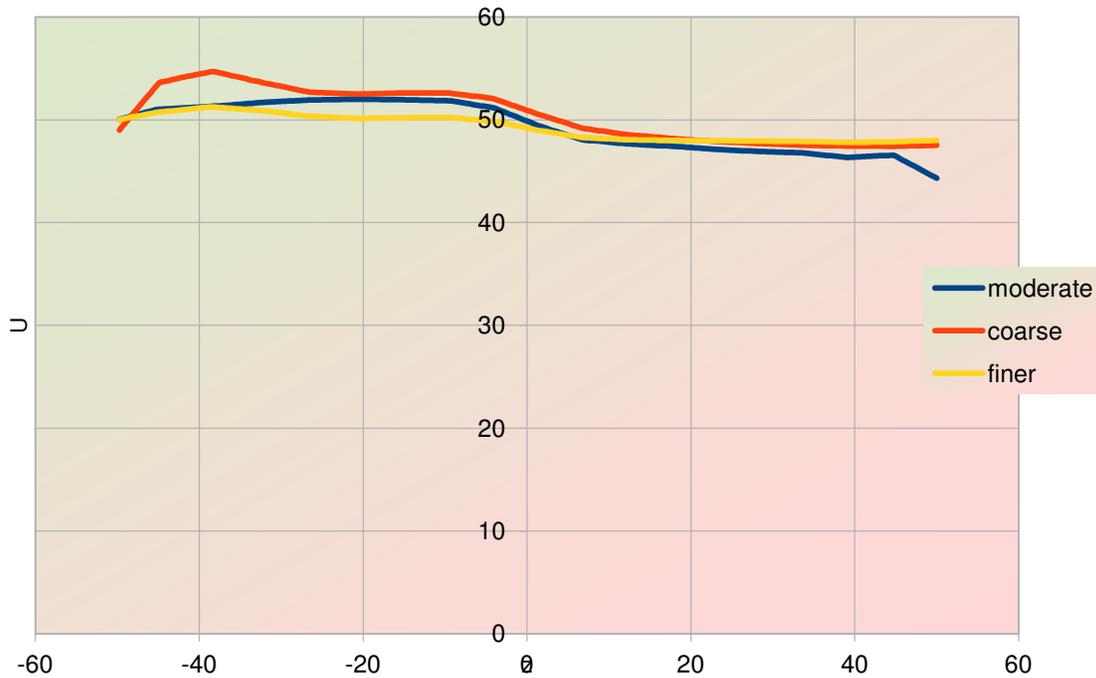


Fig. 4: U v/s z for three grids

Average absolute error was calculated for each of the pair of meshes. It was observed that error reduced as we moved towards the finer mesh. The following table lists the values for error for each of the pairs:

Table 4: Grid Optimisation

Coarse-Fine	Coarse-Moderate	Moderate-Fine
1.4463	1.1964	1.0173

Hence the finer mesh was used for further calculation.

## 4.2 Initial and Boundary Conditions

The initial conditions are mentioned in the files in the 0 folder of the case folder (~/.case/0).

Table 5: U (m/s) initial conditions

Patch	Type	Value
inlet	fixedValue	uniform (0 0 50)
outlet	pressureInletOutletVelocity	uniform (0 0 0)
volume	noSlip	-

Table 6: p\_rgh (kg/m.s<sup>2</sup>) initial conditions

Patch	Type	Value
inlet	fixedFluxPressure	uniform 0
outlet	totalPressure	uniform 0
volume	fixedFluxPressure	uniform 0

Table 7: alpha initial conditions

Patch	Type	Value
inlet	fixedValue	uniform 1
outlet	inletOutlet	uniform 0
volume	zeroGradient	-

In the constant folder changes were made in 3 files:

1. **g** - acceleration due to gravity (g) was taken to be (0 0 0)
2. **transportProperties** - Transport Properties are mentioned according to Table 1. Surface tension is taken to be 0.03 N/m.
3. **turbulenceProperties** – simulationType is taken to be laminar.

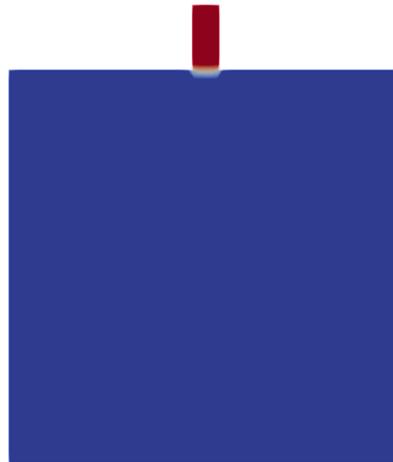


Fig. 5: Ink and the fluid matrix at  $t=0$  (sliced view)

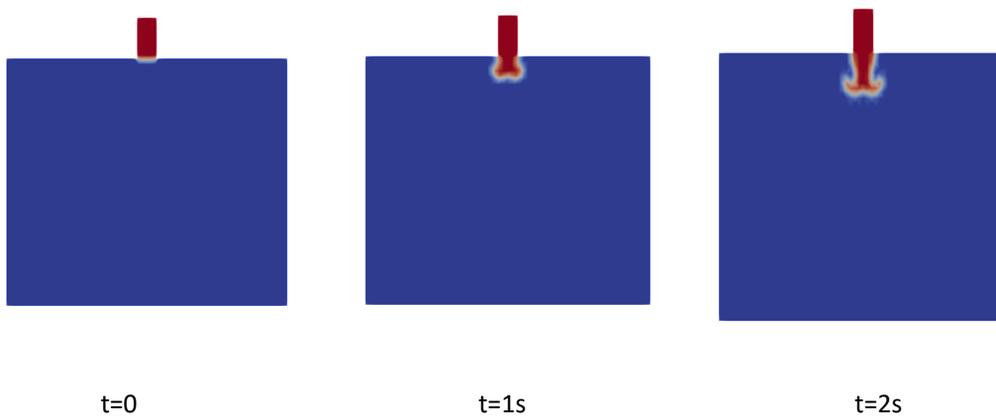
### 4.3 Solver

InterFoam solver is used to simulate this case. It is a transient solver and is used for incompressible and immiscible fluids under isothermal conditions. It can be used for both laminar and turbulent cases but we are using it for laminar case. It uses the Volume of Fluid (VOF) method to track the details of the interface.

The simulation can be simply run by executing the setFields and the interFoam commands.

## 5. Results and Discussions

Contour of  $\alpha(\text{ink})$  at inlet velocity  $U=50\text{m/s}$ :



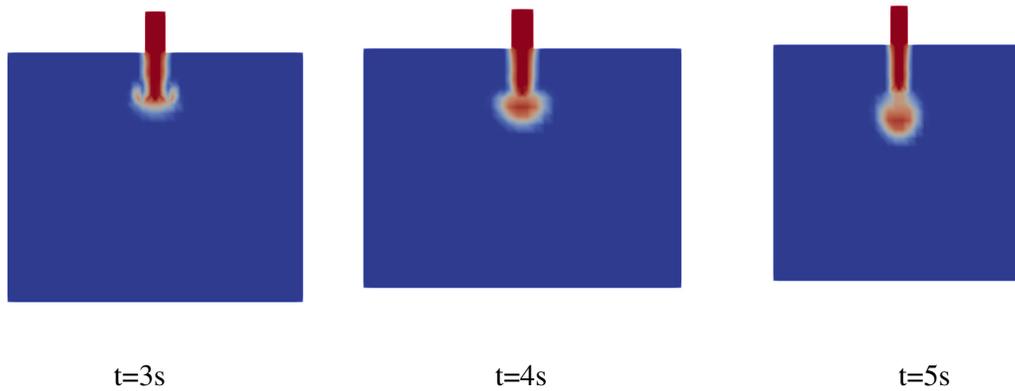


Fig 6: The volume fractions of the ink jet and the fluid matrix at  $t=0, 1, 2, 3, 4, 5$  s.

As we can see from the plots as the ink enters into the medium its interface first bifurcates into two streams then forms an umbrella-like interface and then starts forming a bulb-like structure with most of the ink concentrated at the centre of the bulb.

Different values of  $U$  were also taken and plotted to see how the profile varies.

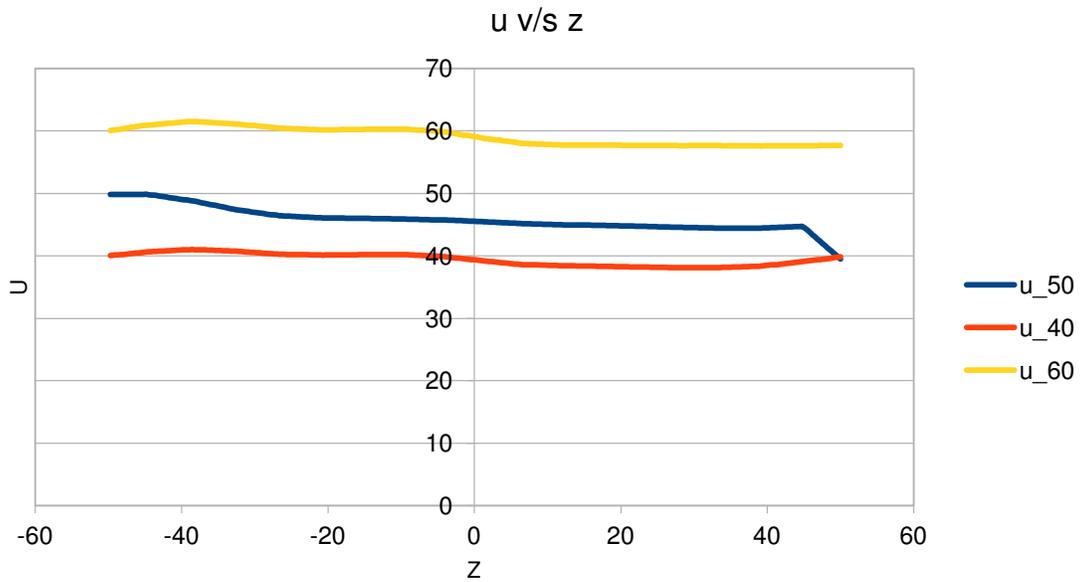


Fig 7:  $U$  v/z for 3 values of  $U$  .i.e, 50 m/s, 40 m/s and 60 m/s

As we can see from the plots the structure of the three graphs are almost similar they are just parallelly shifted depending on the value of  $U$ .

## References

- [1] Zhainakov, A.Z., Kurbanaliev, A.Y. Verification of the open package OpenFOAM on dam break problems. *Thermophys. Aeromech.* 20, 451–461 (2013).  
<https://doi.org/10.1134/S0869864313040082>
- [2] Multiphase (VOF) Simulation Project by Jozsef Nagy. (2020, July 28). OpenFOAM Wiki, . Retrieved 09:07, April 19, 2021  
from [https://wiki.openfoam.com/index.php?title=Multiphase\\_\(VOF\)\\_Simulation\\_Project\\_by\\_Jozsef\\_Nagy&oldid=3345](https://wiki.openfoam.com/index.php?title=Multiphase_(VOF)_Simulation_Project_by_Jozsef_Nagy&oldid=3345).
- [3] OpenFOAM v6 user guide: 2.3 breaking of a dam. (2018, July 10). Retrieved from <https://cfd.direct/openfoam/user-guide/v6-dambreak/>