# CFD simulation and Experimental Validation of Flow through a Mouthpiece using Variable Head Method

**Rajat Walia**, OpenFOAM team, FOSSEE Project, IIT Bombay.

VIT Chennai, Vellore Institute of Technology

**Abstract** - This report aims to describe the calculation of actual and theoretical discharge through the mouth piece using software Salome, ICEM CFD, OpenFOAM and Experimental Setup. It also aims to study the flow velocity, discharge through mouthpiece which can be used to calculate coefficient of discharge and comparison of that with real time experiment. Coefficient of discharge is the ratio of the amount of water discharged to the amount theoretically discharge rate.

**Keywords** – Mouthpiece, Coefficient of Discharge, InterFoam, SnappyHex Meshing, ICEM CFD

## 1. INTRODUCTION

A *mouthpiece* is a tube of length not more than two to three times its diameter, which is mounted to a tank for measuring discharge from the tank. The presence of a mouthpiece will introduce pressure drop will tell the rate of flow rate *Mouthpieces* are classified on the basis of their position, shape and discharge conditions. According to the shape, they may be classified as, cylindrical, convergent, divergent and convergent-divergent. According to the position, they may be external or internal mouthpieces with respect to tank to which it is connected. Based on discharge conditions, they may be classified as running full and running free mouthpieces.

## 2. GEOMETRY

The geometry consists of a Balancing tank which is connected of external Mouthpiece. Flow through mouthpiece is directed by a rectangular open channel situated below to the mouthpiece towards Collection Tank. The modelling of Geometry was done in Salome Software using commands create a box, cylinder, Translation of cylinder & box, Fuse and Create group to create surface for giving boundary conditions.

Length of Collecting & Balancing Tank = 60 cm

Breadth of Collecting & Balancing Tank = 60 cm

Height of Collecting & Balancing Tank = 60 cm

Diameter of Mouthpiece = 5.5 cm

Length of Mouthpiece = 12.7 cm

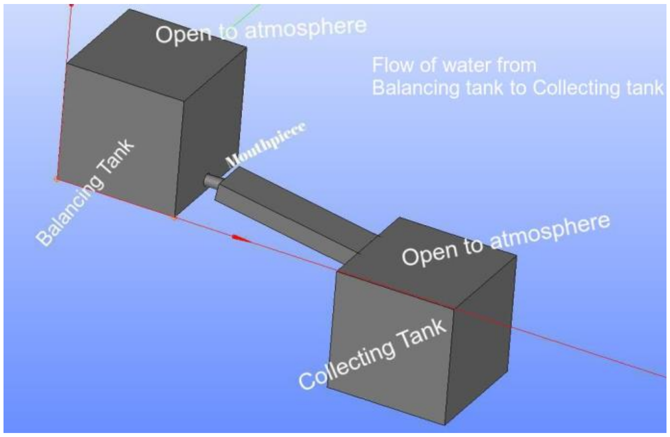


**Figure 1: Salome Model**

## 3. EXPERIMENTAL PROCEDURE (Variable Head Method)

Water is pumped to the Balancing tank via motor, which is connected to Sump filled with water situated below the Balancing tank. A Piezometer is attached to the Balancing tank, next to Piezometer a scale is fixed to visually locate height of water collected in Balancing tank. Coefficient of Discharge at variable head is calculated. Pump is switched off in order to make water still in balancing tank. After that water level start falling due to the gravity effect, Time taken by water level to drop **5 cm** height noted down, Reading of time and heads at different head noted down, Using Head drops and time taken for that drop in Balancing tank, Coefficient of Discharge is calculated.

Coefficient of discharge $$C_d = \frac{2A\left(\sqrt{H_1} - \sqrt{H_2}\right)}{at\sqrt{2g}}$$

Here 'H1'is initial head, '$H_2$' is final head, 'A' is Balancing tank cross-section area, 'a' is Mouthpiece cross-section area, 't' time taken for head drop.

A = 3600 $cm^2$

a = 23.74 $cm^2$

**Figure 2: Experimental Setup**

## 4. MESHING

Meshing of the model done using **ICEM CFD**, converting mesh file to **Fluent,** reading mesh file using command **fluentMeshToFoam.exe** and S**nappyHexMesh** utility in openFOAM.

### ICEM CFD MESHING

Tetrahedral meshing is done using ICEM CFD. Model dimension was in meter. **Maximum global element** size was kept as **0.5**, **Scale factor** kept as **1**, names are given to each surface which can be used for defining the boundary conditions. **INLET** name is given to the top surface of Balancing tank, **OUTLET** name is given to the top surface of collecting tank and to the top surface of Channel. **MOUPTHPIECE** name given to the Mouthpiece surface, **CHANNEL1 & CHANNEL2** name given to Side and Bottom of channel surface, **WALL** name given to rest of the walls.

| Name | Maximum Element Size |
|---|---|
| INLET | 0.2 |
| OUTLET | 0.2 |
| WALL | 0.2 |
| CHANNEL1 | 0.03 |
| CHANNEL2 | 0.05 |
| MOUTHPIECE | 0.008 |

Refinement is given near to Mouthpiece interface. Manual Smoothing had done using **Smooth Mesh Globally** Command.

| | |
|---|---|
| Maximum no. of cell | 84038 |
| Maximum Skewness | 0.80 |

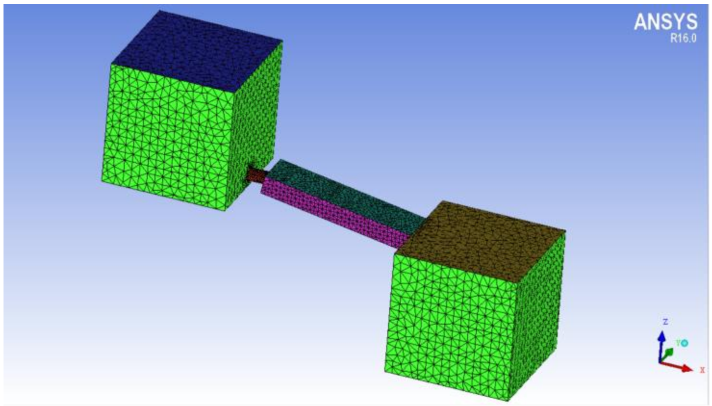


**Figure 3: ICEM Meshing**

### SnappyHex Meshing

**STL** files are generated of all faces of geometry and saved in folder naming '**triSurface**' inside Constant folder. **Block Mesh** is created first which is a simple rectangular meshed box. Geometry is bounded inside that rectangular box. **SurfaceFeatureExtract** command is used then to locate all sharp edges and face which can be used to locate where to snap the mesh. This will create a folder naming '**extendedFeatureEdgeMesh**' which consist information of all surfaces and edges. Point is defined in **snappyHexMeshDict** file which is located inside the fluid domain. This point enables the Snappyhex Mesh to understand where to keep the mesh and where to snap. Mesh at different mesh sizes, edge refinement and surface refinement is generated.

Command used for Meshing:-

**blockMesh.exe**

**surfaceFeatureExtract.exe**

**snappyHexMesh.exe**

**Mesh with 1.56 Lac Elements**

| Surface Name | Edge Refinement | Surface Refinement |
|---|---|---|
| inlet | 2 | (2 2) |
| outlet | 2 | (2 2) |
| wall | 3 | (3 3) |
| mouthpiece | 3 | (3 3) |

**hex (0 1 2 3 4 5 6 7) (20 20 20) simpleGrading (1 1 1)**

**nCellsBetweenLevels:** 2

**resolveFeatureAngle:** 30 degree

**Max Skewness** 4.53

High surface and edge refinement is kept at mouthpiece and wall to capture the sharp gradients, effect due to sudden contraction, expansion and Boundary layer Effect. **nCellBetweenLayers** is kept **2** for slow expansion between each high and low refinement zone.

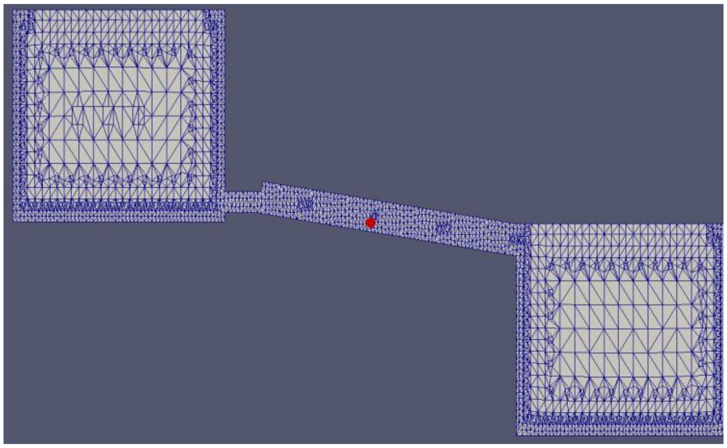

**Figure 4: Side view of Mesh having 1.56 Lac Elements**

**Mesh with 1.88 Lac Elements**

| Surface Name | Edge Refinement | Surface Refinement |
|---|---|---|
| inlet | 2 | (2 2) |
| outlet | 2 | (2 2) |
| wall | 3 | (3 3) |
| mouthpiece | 3 | (3 3) |

**hex (0 1 2 3 4 5 6 7) (21 21 21) simpleGrading (1 1 1)**

**nCellsBetweenLevels:** 2

**resolveFeatureAngle:** 30 degree

**Maximum Skewness:** 3.92

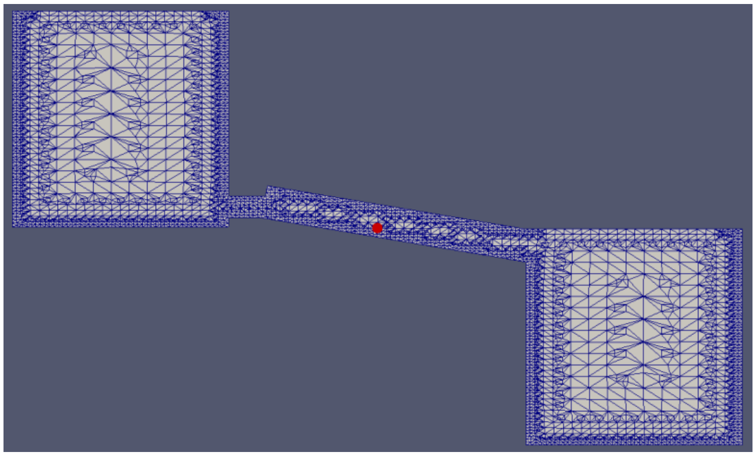

**Figure 5: Side view of Mesh having 1.88 Lac Elements**

**Mesh with 2.001 Lac Elements**

| Surface Name | Edge Refinement | Surface Refinement |
|---|---|---|
| inlet | 3 | (2 2) |
| outlet | 4 | (3 3) |
| wall | 4 | (3 3) |
| mouthpiece | 5 | (4 4) |

**hex (0 1 2 3 4 5 6 7) (12 12 12) simpleGrading (1 1 1)**

**nCellsBetweenLevels:** 10

**resolveFeatureAngle:** 160

**Max skewness:** 2.0490755

High surface and edge refinement is kept at mouthpiece and wall to capture the sharp gradients, effect due to sudden contraction, expansion and boundary layer effect. **resolveFeatureAngle** kept as **160** degree to resolve sharp angle i.e. applies maximum level of refinement to cells that can see intersections whose angle exceeds resolveFeatureAngle. **nCellBetweenLayers** is been kept **10** for slow expansion between each high and low refinement zone i.e gradual transition between level of refinement.

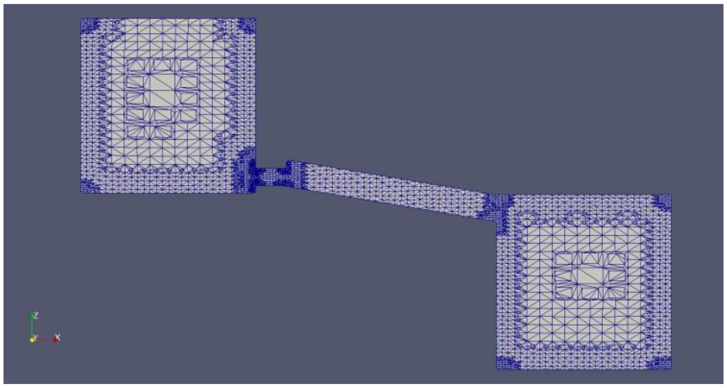

**Figure 6: Side View of Mesh having 2.001 Lac elements**

## 5. ANALYSIS

**OpenFOAM Multiphase interFoam** solver is used for solving the case using generated mesh files. The two-phase algorithm in interFoam is based on the **volume of fluid (VOF)** method in which a specie transport equation is used to determine the relative volume fraction of the two phases, or phase fraction α, in each computational cell. Physical properties are calculated as weighted averages based on this fraction. The nature of the VOF method means that an interface between the species is not explicitly computed, but rather emerges as a property of the phase fraction field. Since the phase fraction can have any value between 0 and 1, the interface is never sharply defined, but occupies a volume around the region where a sharp interface should exist.

### Turbulence Model:

Fluid is water. **Reynolds number** is calculated by using the formula given below:-

$$\mathrm{Re}_D = \frac{\rho V D}{\mu} = \frac{V D}{\nu}$$

V is Flow Velocity, D IS Hydraulic diameter, μ is Dynamic viscosity, ν is Kinematic viscosity

Hydraulic Diameter of Circular pipe = Diameter of Mouthpiece = 0.055 m

ν of Water = $1.14 \times 10^{-6}$ $m^2/s$

Theoretical Flow velocity is calculated using formula:-

$$V_t = \sqrt{2gH}$$

g = 9.81 $m^2/s$

H = 0.05 m

V = 0.99 m/s

Reynolds Number = **47763.15**

Since the Reynolds Number is more than 2300, so the flow is **Turbulent**. Hence, the **K- Epsilon Turbulent model** is been used for solving the case.

### Boundary Condition:

***Velocity Boundary Condition:***

| Name | Boundary Condition |
|---|---|
| INLET | pressureInletOutletVelocity |
| OUTLET | pressureInletOutletVelocity |
| WALL | noSlip |
| MOUTHPIECE | noSlip |

**PressureInletOutletVelocity** boundary condition is been used at inlet & outlet because inlet & outlet surface are open to the environment. **NoSlip** boundary condition is been used to fix velocity at the wall zero due to viscosity effect.

***Pressure Boundary Condition:***

| Name | Boundary Condition |
|---|---|
| INLET | totalPressure |
| OUTLET | totalPressure |
| WALL | fixedFluxPressure |
| MOUTHPIECE | fixedFluxPressure |

**totalPressure** boundary condition is been used at inlet and outlet because this is self-stabilizing boundary condition with **PressureInletOutletVelocity**. totalPressure boundary provides total pressure condition. **fixedFluxPressure** is been used at wall and mouthpiece to set the pressure gradient to the provided value such that the flux on the boundary is that specified by the velocity boundary condition.

### Solving the Case:

Water was filled in balancing tank up to **40cm** height by defining coordinates in '**setFieldsdict**' file. Acceleration due to gravity defined in vertical downward direction. At time t = 0 s, Water start flowing from balancing tank to collecting tank via mouthpiece due to Gravity effect. Analysis stops when water completely transferred from balancing tank to collection tank. Time is noted down for every **5 cm** drop of water in balancing tank.

| No. of Cells | TimeStep (Seconds) | Max. CFL Number |
|---|---|---|
| 84038 | 0.001 | 2.04 |
| 1.56 Lac | 0.003 | 1.59 |
| 1.86 Lac | 0.003 | 1.38 |
| 2.01 Lac | 0.0006 | 0.53 |

## 6. EXPERIMENT

| $H_1$ (cm) | $H_2$ (cm) | Time for fall in head, t Sec | $\sqrt{H_1}$-$\sqrt{H_2}$ | $C_d$ |
|---|---|---|---|---|
| 39 | 34 | 4.35 | 0.414 | 0.651 |
| 34 | 29 | 4.80 | 0.445 | 0.634 |
| 29 | 24 | 5.08 | 0.486 | 0.655 |

**Average $C_d$ = 0.646**

## 7. RESULT & DISCUSSION

**Experimental Results:**

Coefficient of discharge at **5 cm** head fall in balancing tank is calculated. Average Coefficient of discharge of flow through a mouthpiece is **0.646**.

**Analysis Results:**

Maximum velocity at the mouthpiece observed. The streamlines approaching the orifice converges towards it. Since an instantaneous change of direction is not possible, the streamlines continue to converge beyond the orifice until they become parallel, that point is known as **Vena-contractor.**

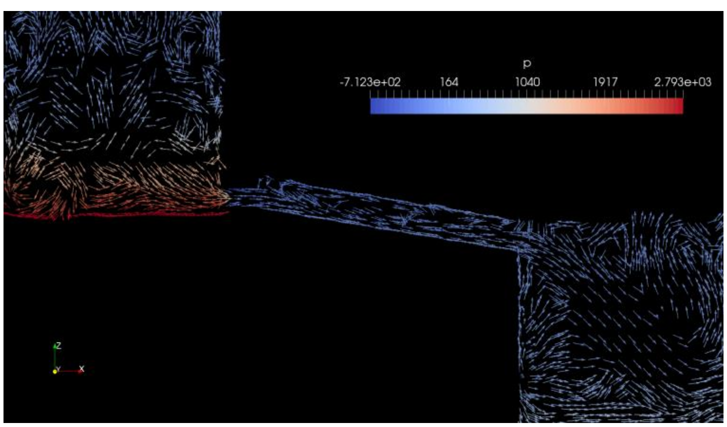


**Figure 7: Glyph of Model having 1.56 Lac cells at time t = 23 s**

Time for **5 cm** fall in head is taken and Coefficient of discharge is calculated.

Coefficient of discharge $$C_d = \frac{2A\left(\sqrt{H_1} - \sqrt{H_2}\right)}{at\sqrt{2g}}$$

**Coefficient of discharge for Model with 84038 cells:**

| $H_1$ (cm) | $H_2$ (cm) | Time for fall in head, t Sec | $\sqrt{H_1}$-$\sqrt{H_2}$ | $C_d$ |
|---|---|---|---|---|
| 39 | 34 | 3.5 | 0.414 | 0.810 |
| 34 | 29 | 4 | 0.445 | 0.761 |
| 29 | 24 | 3.7 | 0.486 | 0.899 |

**Average $C_d$ = 0.823**

**Coefficient of discharge for Model with 1.56 Lac cells:**

| $H_1$ (cm) | $H_2$ (cm) | Time for fall in head, t Sec | $\sqrt{H_1}$-$\sqrt{H_2}$ | $C_d$ |
|---|---|---|---|---|
| 39 | 34 | 3.01 | 0.414 | 0.941 |
| 34 | 29 | 3.91 | 0.445 | 0.779 |
| 29 | 24 | 4.23 | 0.486 | 0.899 |

**Average $C_d$ = 0.873**

**Coefficient of discharge for Model with 1.86 Lac cells:**

| $H_1$ (cm) | $H_2$ (cm) | Time for fall in head, t Sec | $\sqrt{H_1}$-$\sqrt{H_2}$ | $C_d$ |
|---|---|---|---|---|
| 39 | 34 | 3.6 | 0.414 | 0.787 |
| 34 | 29 | 3.9 | 0.445 | 0.781 |
| 29 | 24 | 5 | 0.486 | 0.665 |

**Average $C_d$ = 0.744**

**Coefficient of discharge for Model with 2.01 Lac cells:**

| $H_1$ (cm) | $H_2$ (cm) | Time for fall in head, t Sec | $\sqrt{H_1}$-$\sqrt{H_2}$ | $C_d$ |
|---|---|---|---|---|
| 39 | 34 | 3.498 | 0.414 | 0.810 |
| 34 | 29 | 5.002 | 0.445 | 0.609 |
| 29 | 24 | 5.200 | 0.486 | 0.640 |

**Average $C_d$ =0.686**

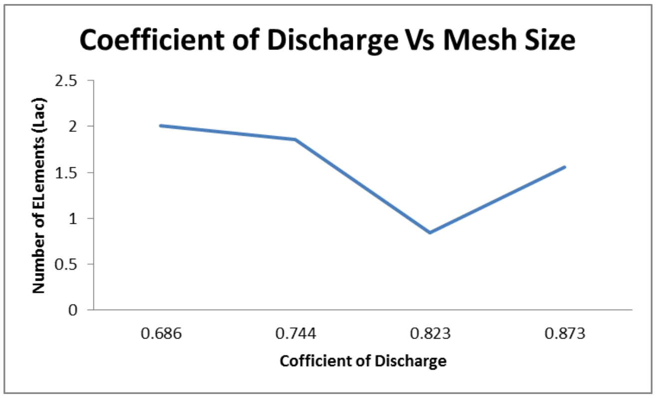


**Figure 8: Coefficient of Discharge Vs Mesh Size**

Figure 8 represents variation of Coefficient of Discharge with respect to total number of cell in Fluid domain. $C_d$ becomes closer to experimental value with the finer meshing.

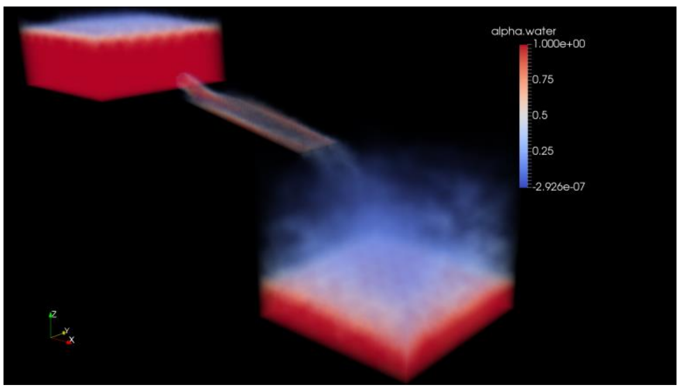


**Figure 9: Volume Water Fraction of model having 84 thousand elements at time t = 13 s**

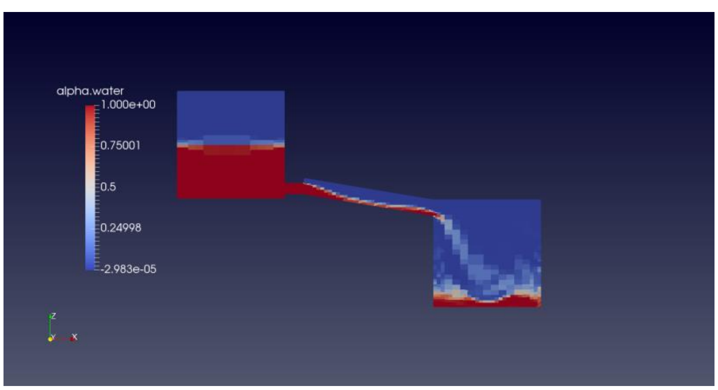


**Figure 10: Water Fraction Contour of model having 2.01 Lac cell at time t = 7.5 s**

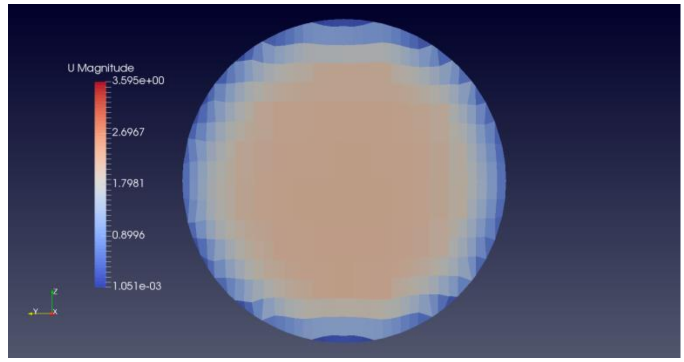


**Figure 11: Velocity Contour at Mouthpiece cross-section of model having 2.01 Lac cell at time t = 7.5 s**

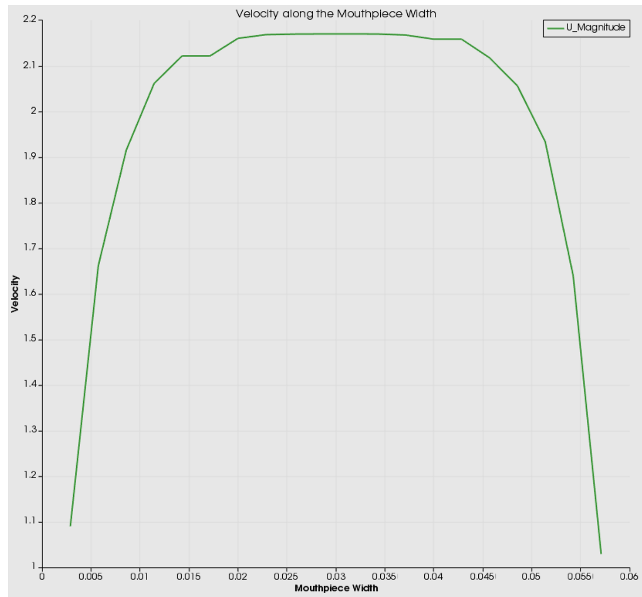


**Figure 12: Velocity variation along the mouthpiece cross-section of model having 2.01 Lac cell at time t = 7.5 s**

Maximum velocity around **2.5 m/s** is at the centre of mouthpiece and velocity at the mouthpiece walls is almost zero because of the **No slip** effect.

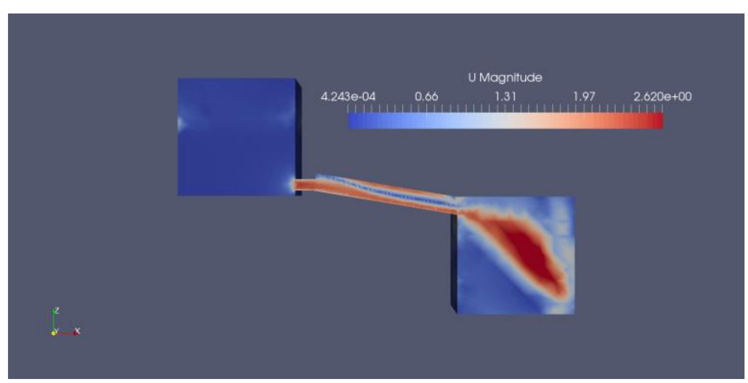


**Figure 13: Velocity Contour of model having 84 thousand elements at time t = 4 s**

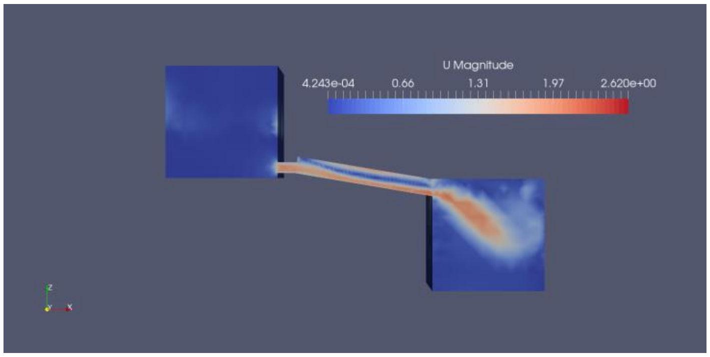


**Figure 14: Velocity Contour of model having 84 thousand elements at time t = 13 s**

Fluid is accelerated along the mouthpiece whereas Velocity of fluid at the top of balancing tank is approximately negligible as compare to velocity along the mouthpiece because top surface of balancing tank is open to environment. Maximum Velocity at mouthpiece is **1.55 m/s** for t = 13 s.

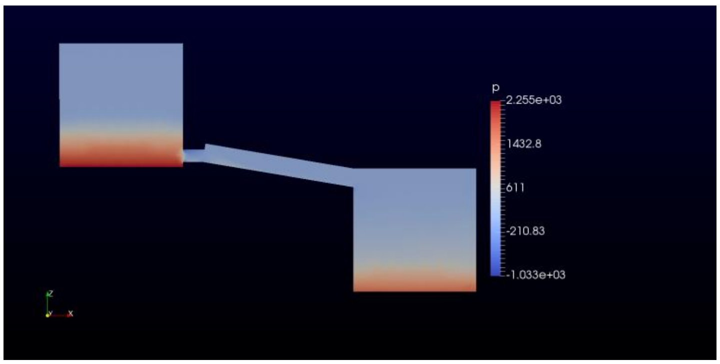


**Figure 15: Pressure Contour of model having 84 thousand elements at time t = 13 s**

Figure 15 shows the pressure contour inside the model. We can see that fluid is still at the bottom surface of balancing tank and experience maximum pressure.

## 8. CONCLUSION

The CFD result become close to experimental value with the finer meshing. With the decrease in time step CFL number is maintained close to **0.53** which makes the data transfer from one cell to other more accurate. With the more number of elements inside fluid domain, gradients captured accurately. Experimental $C_d$ is **0.646** where the average $C_d$ calculated via CFD is **0.781**. This difference cause because Material properties such as surface roughness, wall friction, rusting etc. are not considered. Other reasons are computational error, experimental setup and CAD model are not exactly same, improper grid spacing, few highly skewed cells, unstable time step, unpredictable real time environmental condition and round off error as computer can handle on fixed number of digits.

## 9. REFERENCE

[1] https://nptel.ac.in/courses/105103095/49

[2] https://cfd.direct/openfoam/user-guide/v6-snappyhexmesh/

[3] https://www.openfoam.com/documentation/user-guide/standard-boundaryconditions.php

[4] https://www.scribd.com/doc/71243629/Flow-Through-Orifice-and-Mouthpiece

[5] https://www.slideshare.net/Dheerajsoni20/flow-through-mouthpieces