



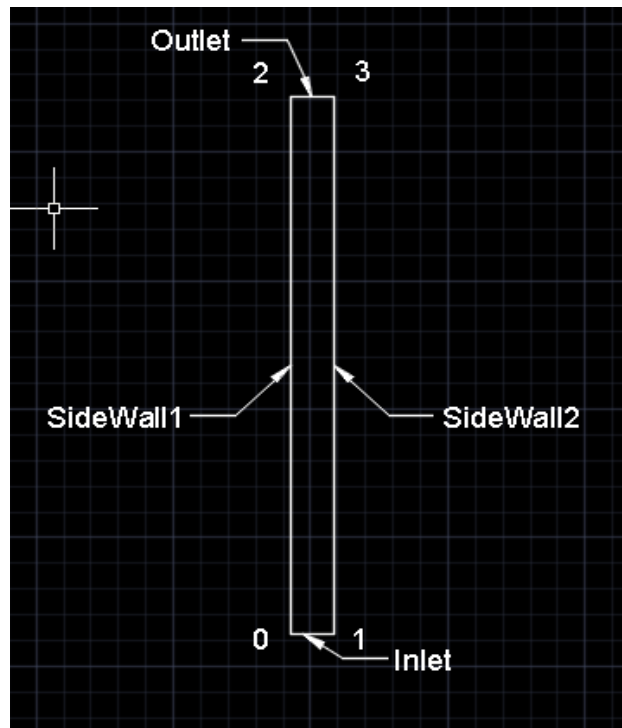
## “CFD Analysis of Natural Convection Flow through Vertical Tube”

Bor Abubakr

M.Tech, IIT Bombay

### Synopsis

The goal of this research migration project is to use OpenFOAM to perform numerical simulations of natural convection through asymmetric two-dimensional vertical tubes. The blockMesh tool was used to create the geometry and mesh. In the simulation, a steady-state SIMPLE algorithm-based buoyantSimpleFoam solver was utilized. As a reference, Prashant M Khanorkar et al. [1] used commercial CFD code Fluent to do their research. The geometry is 1 metre long and 16 millimeters in diameter.



**Figure (1): The Geometry**

## References

- [1] Prashant M Khanorkar and R E Thombre. "CFD ANALYSIS OF NATURAL CONVECTION FLOW THROUGH VERTICAL PIPE". In: International journal of Mechanical Engineering and Robotics Research (2013), ISSN: N 2278 – 0149 w. Vol. 2, No. 3.URL:  
<https://citeseerx.ist.psu.edu/viewdoc/download?doi=10.1.1.360.222&rep=rep1&type=pdf>

## 1 Introduction

In the reference paper [1], an asymmetric Tube case was analyzed to compare numerical results against experimental data. CFD analysis of the vertical tube is conducted. A vertical copper tube having constant cross section area is used for representing the medium through which natural convection of water takes place. The current research is mostly applicable to the design of water tube boilers and solar water heating systems. For the formation of steam in these applications, vertical copper tubes are employed, with water flowing exclusively due to natural convection flow. For the heating of copper tubes in various applications, a steady heat flux is given. Copper is primarily utilized to provide the best heat transmission to water. As the temperature of the water inside the vertical tube rises due to the steady heat flux, the density of the water falls, making it lighter and allowing it to flow upwards. At the same time, cool liquid replaces the hot water. As a result, a natural convection flow in the upward direction is being created. After reaching steady state, we have a specific mass flow rate, velocity, and temperature at the outlet. At come to a conclusion, the velocity at outlet, temperature at outlet, and mass flow rate at outlet were all compared for three different values of heat flux. This CFD analysis can be used to perform the analysis of natural convection flow characteristics like velocity, temperature and heat transfer coefficients.

## 2 Governing Equations and Models

OpenFOAM software was used to replicate the results of Prashant M Khanorkar [1]. The simulation is governed by the Navier-Stokes equations for single-phase flows, the following are the governing continuity ,momentum, and energy equations for 2-D steady state incompressible flow. [2]

### 2.1 Governing Equations

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (1)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho_0} \frac{\partial p}{\partial x} + \gamma^2 \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \quad (2)$$

$$u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{1}{\rho_0} \frac{\partial p}{\partial y} + \gamma^2 \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) - \frac{\rho}{\rho_0} g \quad (3)$$

$$u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} = k \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right) \quad (4)$$

## 2.2 Boussinesq approximation (buoyancy)

The Boussinesq approximation is employed in the field of buoyancy-driven flow in fluid dynamics (also known as natural convection). It ignores density differences unless they exist in terms multiplied by  $g$ , or gravity's acceleration. The Boussinesq approximation states that the difference in inertia is minimal, yet gravity is powerful enough to cause the specific weights of the two fluids to differ significantly.

Nature (atmospheric fronts, oceanic circulation, katabatic winds), industry (dense gas dispersion, fume cupboard ventilation), and the built environment all have Boussinesq flows. For many of these processes, this approximation is highly precise, and it saves time.

The Boussinesq approximation is used to solve problems when the temperature of a fluid fluctuates from one location to another, causing fluid movement and heat transfer. The fluid meets the principles of mass conservation, momentum conservation, and energy conservation. Variations in fluid properties other than density are neglected in the Boussinesq approximation, and density emerges only when it is multiplied by  $g$ , the gravitational acceleration. [3]

The buoyancy term  $\rho g$  written as:

$$\rho g = (\rho_0 + \Delta\rho)g = \rho_0(1 - \beta(T - T_{ref}))g \quad (5)$$

$$\beta = \frac{1}{v} * \left( \frac{dv}{dT} \right)_p \approx - \frac{1}{\rho_0} * \frac{\rho - \rho_0}{T - T_{ref}} \quad (6)$$

This approximation valid only when:

$$\Delta\rho \ll \rho_0$$

Where:

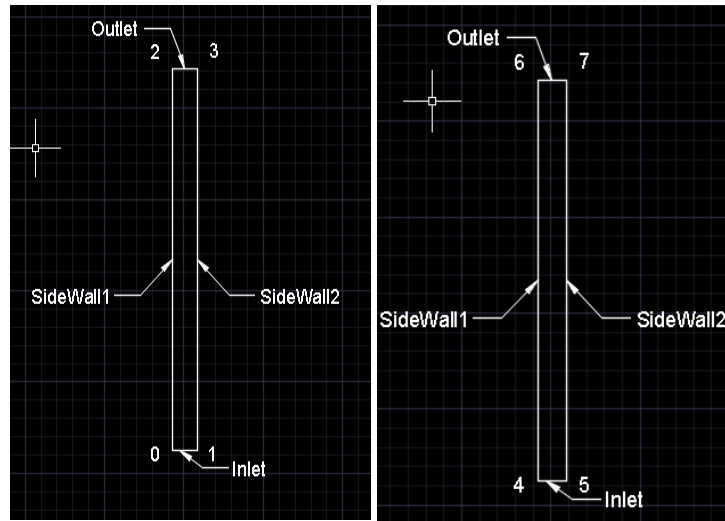
$\beta \equiv$  Coefficient of thermal expansion

$T_{ref} \equiv$  Reference temperature

### 3 Simulation Procedure

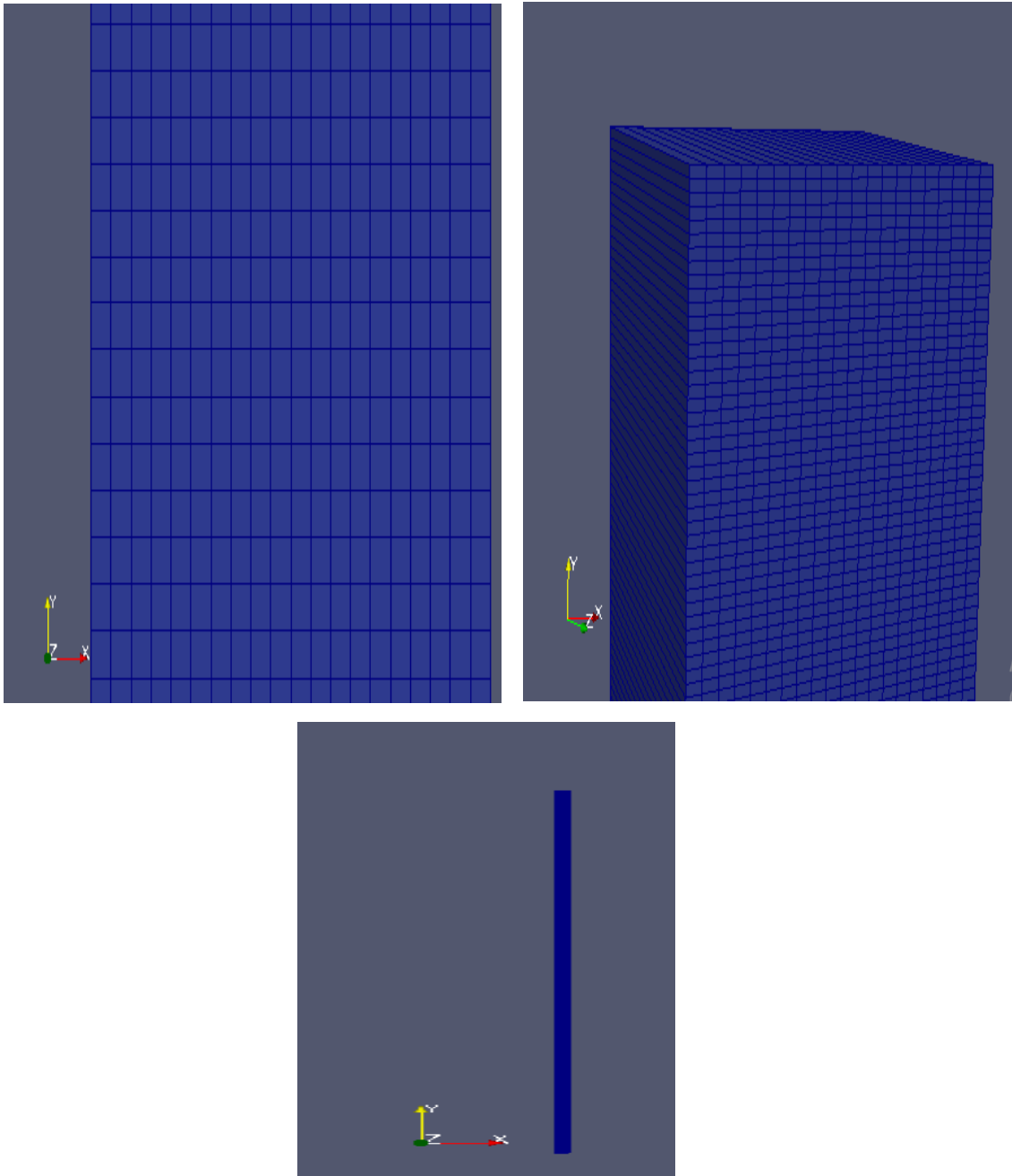
#### 3.1 Geometry and Mesh

The Geometry was created by using blockMeshDict utility, 7 vertices was specified to form the geometry as shown in Fig (2). Hexahedral block was used to connect the vertices and to create the geometry. Since the hexahedral block is three dimensional block, the faces that used to represent the third dimension (Z-axis) are kept as empty. The geometry consists of inlet, outlet, and walls.



**Figure (2): Vertices distributions**

Since the diameter of the Tube (X-axis) is very small compared to its length, a structured grid consisting of  $20 \times 1000$  cells was used. Simple grading with 0.2 expansion ratio was used for (Y-axis) to ensure accurate results at outlet. And since the geometry is asymmetric the tube has been converted to channel by using hydraulic diameter concept [4]. Fig (3) shows the geometry and mesh in ParaView software.



**Figure (3): Geometry and mesh in ParaView**

### 3.2 Boundary Conditions

There are four boundary conditions files for buoyantSimpleFoam solver, U-file, P-file, and P\_rgh- file. For U boundary conditions, uniform fixed value of y-component velocity was taken from the paper at inlet, no slip conditions was specified for walls, and zero gradient condition was specified for outlet.

For Pressure (P-file) all conditions were specified as calculated because pressure will be calculated from P\_rgh file in this solver.

For T boundary conditions, uniform fixed value was specified for inlet which was mentioned in the paper, zero gradient was specified for outlet, and fixed gradient was specified for the walls since 3 constant heat flux were provided to the walls. The value of this fixed gradient can be calculated from Fourier's law of conduction:

$$\frac{dT}{dx_{wall}} = \frac{q}{k_{water}} \quad (7)$$

Where:

$q$  = constant heat flux (796, 1194, 1592 w/m<sup>2</sup>)

$k_{water}$  = thermal conductivity of water at 20 C (0.59w/m k)

For P-rgh file (pressure without hydrostatic pressure), fixed flux pressure was assigned to walls and inlet which implies zero gradient for P file, and prghPressure boundary condition was specified for the outlet to calculate prgh condition at that location by using value of P which was specified in the same file as zero(atmospheric condition).

Table (1): Boundary conditions (case 1)

	Initial	Inlet	Outlet	walls
U (m/s)	0	Uy=0.00315	zeroGradient	noSlip
T (K)	293	293	zeroGradient	fixedGradient(1350 k/m)
P_rgh (Pa)	0	fixedFluxPressure	prghPressure (0 pa)	fixedFluxPressure
P (Pa)	0	calculated	calculated	calculated



Table (2): Boundary conditions (case 2)

	Initial	Inlet	Outlet	walls
U (m/s)	0	$U_y=0.0039$	zeroGradient	noSlip
T (K)	293	293	zeroGradient	fixedGradient(2025 k/m)
P_rgh (Pa)	0	fixedFluxPressure	prghPressure (0 pa)	fixedFluxPressure
P (Pa)	0	calculated	calculated	calculated

Table (3): Boundary conditions (case 3)

	Initial	Inlet	Outlet	walls
U (m/s)	0	$U_y=0.0043$	zeroGradient	noSlip
T (K)	293	293	zeroGradient	fixedGradient(2700 k/m)
P_rgh (Pa)	0	fixedFluxPressure	prghPressure (0 pa)	fixedFluxPressure
P (Pa)	0	calculated	calculated	calculated

### 3.3 constants and fluid properties

Table (4) shows the thermophysical properties of water at 20 C, since Reynolds number (Re) is very small, laminar flow will be consider. And since the ratio between Grashof number(Gr) and Reynolds number(Re) is much greater than one, then natural convection will be consider.

Table (4): thermophysical properties of water at 20 C [5]

property	value
Molecular Weight(g/mole)	18
Density(kg/m <sup>3</sup> )	1000
coefficient of thermal expansion(k <sup>-1</sup> )	$2.1 \cdot 10^{-4}$
Specific heat at constant pressure(kj/kg.k)	4.180
Thermal conductivity(w/m.k)	0.59
Dynamic viscosity(Pa.s)	$1 \cdot 10^{-3}$
Prandtl number(Pr)	7
Reynolds number(Re)	50.4
Maximum Grashof number(Gr <sub>max</sub> )	$3.7 \cdot 10^5$

### 3.4 Solver

The governing equations in the discretized domain are run using a steady-state for incompressible, turbulent flow-based buoyantSimpleFoam solver. buoyantSimpleFoam is steady state, one phase, turbulent, compressible solver employs the SIMPLE method (Semi-Implicit Method for Pressure Linked Equations). Boussinesq approximation has been used with this solver to make incompressible flow. [6]

For the convergence, conditional strategy used with 5000 maximum iterations or  $10^{-5}$  ( $10^{-4}$  for P\_rgh) convergence criteria. The solution had converged at 1393, 1342, 1342 iterations for the three cases respectively.

```
Time = 1393

DILUPBiCGStab: Solving for Ux, Initial residual = 7.50306e-06, Final residual = 7.50306e-06, No Iterations 0
DILUPBiCGStab: Solving for Uy, Initial residual = 2.02685e-07, Final residual = 2.02685e-07, No Iterations 0
DILUPBiCGStab: Solving for e, Initial residual = 9.95504e-06, Final residual = 9.95504e-06, No Iterations 0
DICPCG: Solving for p_rgh, Initial residual = 1.71596e-08, Final residual = 9.91589e-09, No Iterations 162
time step continuity errors : sum local = 9.97076e-08, global = -1.33742e-09, cumulative = 2.13994e-05
rho min/max : 1003.01 1005.67
ExecutionTime = 722.59 s  ClockTime = 827 s

SIMPLE solution converged in 1393 iterations

End
```

Figure (4): Convergence of simulation for case 1

```
Time = 1342

DILUPBiCGStab: Solving for Ux, Initial residual = 9.13786e-06, Final residual = 9.13786e-06, No Iterations 0
DILUPBiCGStab: Solving for Uy, Initial residual = 3.56178e-07, Final residual = 3.56178e-07, No Iterations 0
DILUPBiCGStab: Solving for e, Initial residual = 9.95123e-06, Final residual = 9.95123e-06, No Iterations 0
DICPCG: Solving for p_rgh, Initial residual = 1.55782e-07, Final residual = 9.28106e-09, No Iterations 477
time step continuity errors : sum local = 1.05792e-07, global = -1.94385e-11, cumulative = 1.43915e-05
rho min/max : 1002.13 1005.67
ExecutionTime = 742.01 s  ClockTime = 841 s

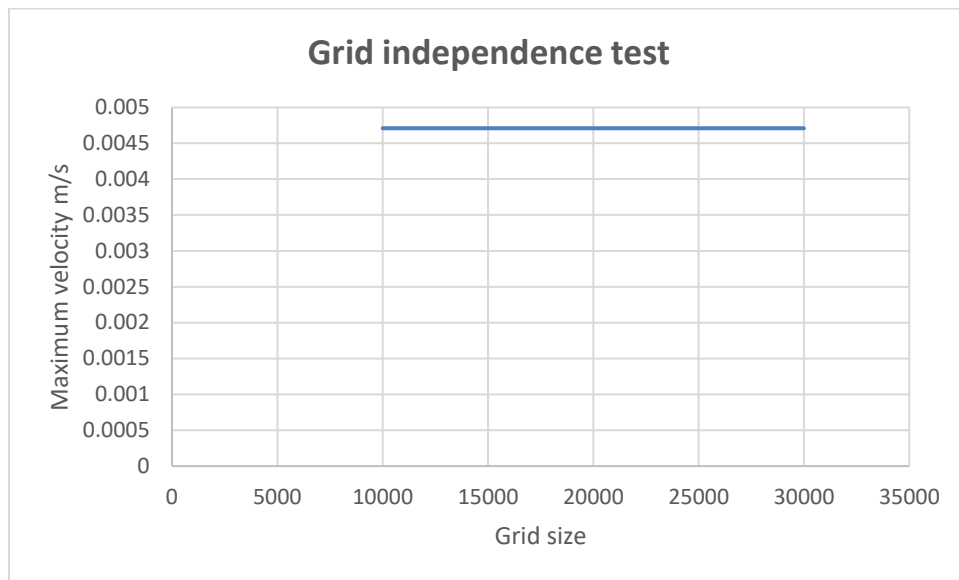
SIMPLE solution converged in 1342 iterations

End
```

Figure (5): Convergence of simulation for case 2 and case 3

#### 4 Grid independence test

The grid independence test is a process used to find the optimal grid condition that has the smallest number of grids without generating a difference in the numerical results based on the evaluation of various grid conditions [7]. For that 3 different grid size were used with case 1 to find the optimal grid size.



**Figure (6): Grid independence test**

Since the grid size change not effected the maximum velocity for case 1, then the middle grid size was selected ( $20 \times 1000$ ) to save some computational costs.

#### 5 Results, Discussions, and Validation

Table (5) shows the OpenFoam results for Average outlet velocity, Average outlet temperature, and mass flow rate and comparison of them with fluent as well as with experimental results.

We can see that the CFD (OpnFoam) and experimental results are quite close, and there is only a small percentage of error.

We noticed that when the heat flux increased, both the outlet velocity and the outlet temperature rose.

Table (5): Comparision between CFD Results and Experimental Results

Parameters	Heat Flux (W/m <sup>2</sup> )	CFD Results(OpenFoam)	CFD Results(FLUENT)	Experimental Results	% FLUENT Error with Experimental	% OpenFoam Error with Experimental
Average outlet velocity(m/s)	796	0.003128	0.00322	0.00315	2.17	0.7
	1194	0.003875	0.00396	0.0039	1.5151	0.65
	1592	0.004274	0.00434	0.0043	0.921	1.54
Average outlet temperature(K)	796	302	319	314	1.6	4
	1194	304	323	320	0.93	6.25
	1592	306	328	328	0	7.20
Mass Flow Rate(kg/s)	796	0.000629	0.000644	0.000638	0.931	1.03
	1194	0.000779	0.000792	0.00079	0.2525	1.41
	1592	0.000859	0.000865	0.00087	0.578	1.28

Figure (5) and (6) show the velocity contours for fluent and OpenFoam respectively. We can see Maximum velocity found at center of the pipe. Also we can see velocity gradually increases from bottom to top outlet.

Figure (7) and (8) show the velocity profile along diameter for fluent and OpenFoam respectively. we can see outlet velocity profile along diameter is parabolic because of fully developed flow at outlet.

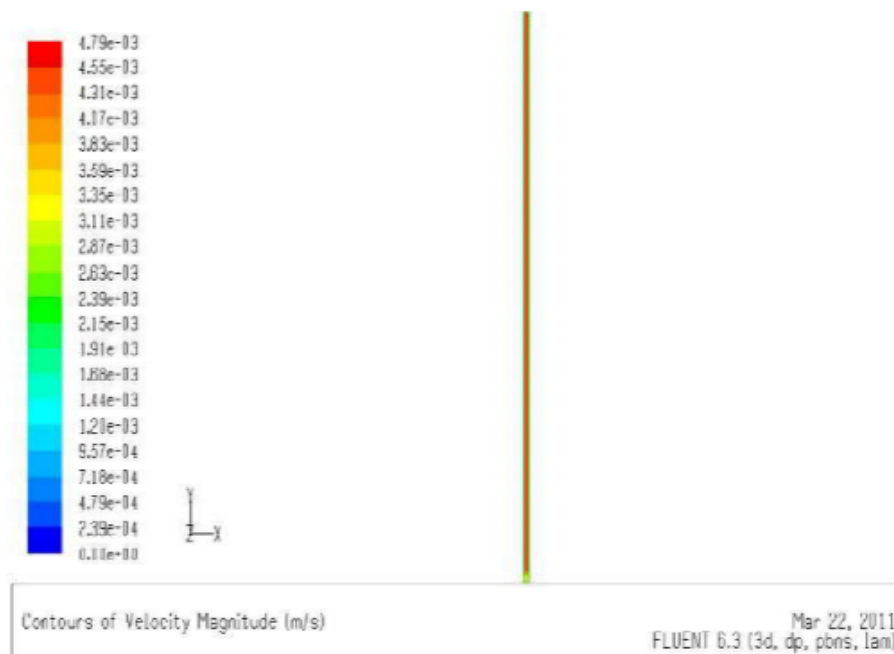


Figure (7): velocity contour for case1 (Fluent)

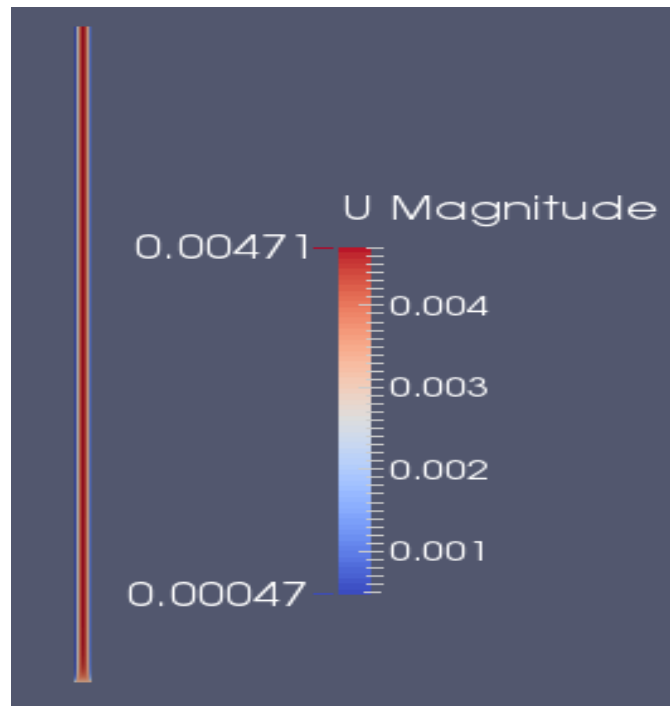


Figure (8): velocity contour for case1 (OpenFoam)

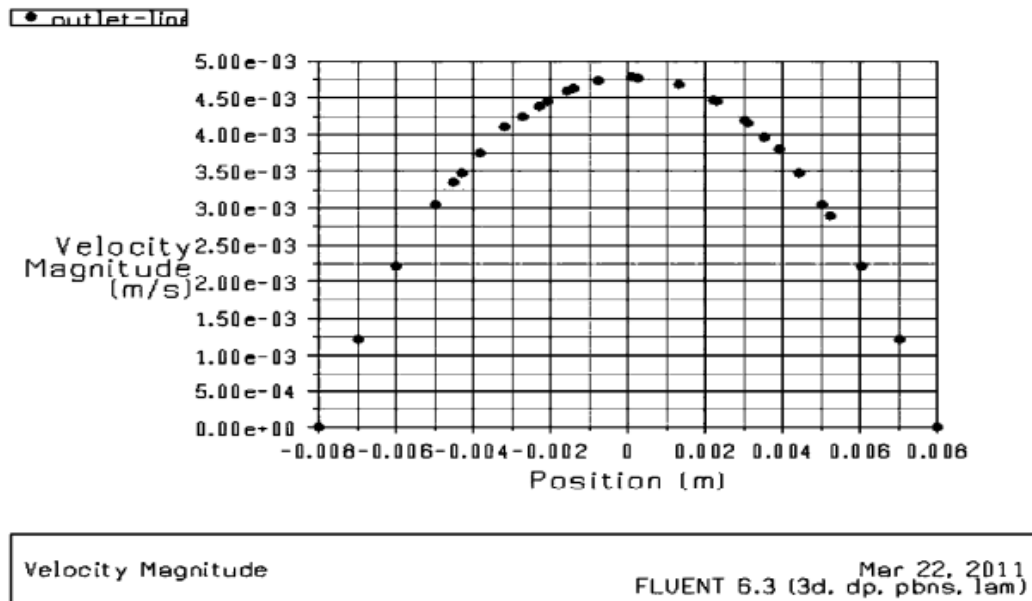
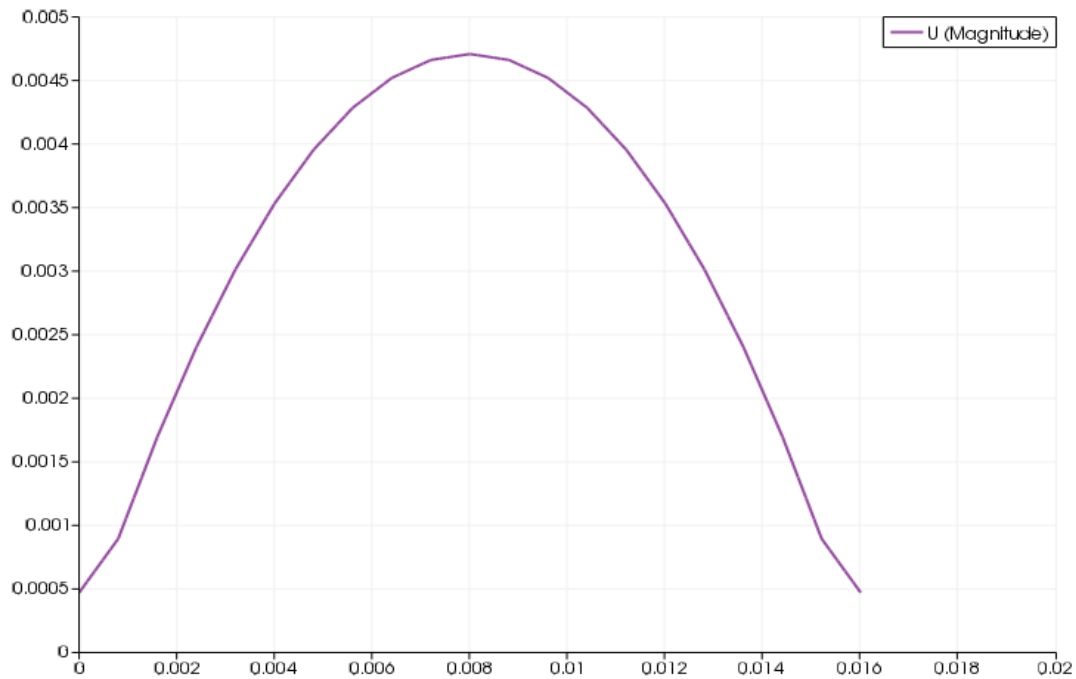


Figure (9): velocity profile along diameter for case1 (Fluent)



**Figure (10): velocity profile along diameter for case1 (OpenFoam)**

## 6 Conclusion

Natural convection flow through vertical tube was simulated for three different values of heat flux. The OpenFoam results had compared with experimental results as well as with fluent results and the errors found to be very small. We found that when heat flux increases both outlet temperature and velocity increase. We saw that velocity increased from bottom to top due to buoyancy effect.

For residual control,  $10^{-5}$  difference was selected for momentum and energy equations, and  $10^{-4}$  for  $P_{\text{rgh}}$  (pressure without hydrostatic). 5000 maximum iterations was used and all cases had converged before reaching to 5000 iterations.

Grid independence test was carried out to find the optimal grid condition and since all selected grid size had given same results, middle grid size was selected (20\*1000) to save some computational costs.

## References

- [1] Prashant M Khanorkar and R E Thombre. “CFD ANALYSIS OF NATURAL CONVECTION FLOW THROUGH VERTICAL PIPE”. In: International journal of Mechanical Engineering and Robotics Research (2013), ISSN: N 2278 – 0149 w. Vol. 2, No. 3.
- [2] <https://caefn.com/openfoam/solvers-buoyantsimpleFoam>.
- [3] [https://en.wikipedia.org/wiki/Boussinesq\\_approximation\\_\(buoyancy\)](https://en.wikipedia.org/wiki/Boussinesq_approximation_(buoyancy)).
- [4] [https://www.cfd-online.com/Wiki/Hydraulic\\_diameter](https://www.cfd-online.com/Wiki/Hydraulic_diameter).
- [5] <https://www.engineeringtoolbox.com/>.
- [6] <https://www.openfoam.com/documentation/guides/latest/doc/guide-applications-solvers-heat-transfer-buoyantSimpleFoam.html>.
- [7] <https://www.hindawi.com/journals/ace/2020/8827936/>.

**DISCLAIMER:** This project reproduces the results from an existing work, which has been acknowledged in the report. Any query related to the original work should not be directed to the contributor of this project.