Numerical simulations of fluid flow and heat transfer in a heater 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

January 4, 2020

Abstract

This case study demonstrates the simulation of a heater. It is an example of heat transfer between two parallel plates (heater). The present case also describes the conjugate heat transfer in multi-region (solid and water) and solver (chtMultiRegionFoam) in open source CFD package OpenFOAM. 3D case model (geometry and mesh) is made with blockMesh meshing tool/utility. The flow is considered transient, non-isothermal and laminar. The simulations are performed using OpenFOAM-5.x. The hydrodynamics of flow between two parallel plates is investigated. The temperature profile is analyzed obtained from the simulation.

Keywords: Fluid flow, Heat transfer, Heater, CFD, OpenFOAM

1 Introduction

The purpose of this case study is to learn OpenFOAM software [1] to the new users and to understand conjugate heat transfer in three dimensional flow in a heater[2, 3].. In this simulation, a pressure-based finite volume method is used for incompressible, transient, non-isothermal and laminar flow. The simulations are carried out using OpenFOAM-5.x [1]. In this case, conjugate heat transfer simulation are carried out for incompressible transient flow through a heater with using **chtMultiRegionFoam** solver in OpenFOAMO-5.x [1]. This case study demonstrates how to do the following:

- Set up a problem case;
- Creating a 3D mesh by using blockMesh utility;
- Create the geometry and import the geometry in OpenFOAM;
- Set up the properties of the fluids;
- Initialize the flow;
- Consider the laminar model for laminar flow regime;
- Set boundary/initial conditions (BC/IC);
- Set numerical schemes, solver parameters and control parameters;
- Brief explanation of solver (chtMultiRegionFoam);
- Post processing the case for results.

2 Problem statement

This case considers the transient simulation of conjugate heat transfer in a heater. Figure 1 shows the details of the geometry considered in the present study. The domain is considered a three-dimensional case by assuming that the geometry has a height of heater in the y-direction. The geometric parameters of the domain such as height, width and depth are considered with 60x40x40 (units, 0.40 m x 0.16 m x 0.20 m) respectively. The solid and water region are defined with topoSet in a 3D environment. The fluid is considered water. Table 1 shows the solid and fluid properties for the present study.

Initially, temperature is patched with value of 300 K. At time zero, the water starts entering and moving to leftwards in the heater. The velocity of the water at inlet are consider 0.001, 0.005 & 0.01 m/s (+ve, x-direction).

Δt and write intervals are 0.001 s and 100 s considered respectively. The time needed to stabilize (to achieved the steady state) the system is about t sec (2000 s).

Table 1: Details of solid (heater) and fluid(water) properties

Solid property	
Parameters	Value
Density (ρ), kg/m^3	8000
Specific heat (C_p), J/(K.kg)	450
Thermal conductivity (κ), W/m.K	80
Fluid property	
Dynamic viscosity (μ), Pa.s	959e-6
Density (ρ), kg/m^3	1000
Specific heat (C_p), J/(K.kg)	4181
Thermal conductivity (k), W/m.K	0.6

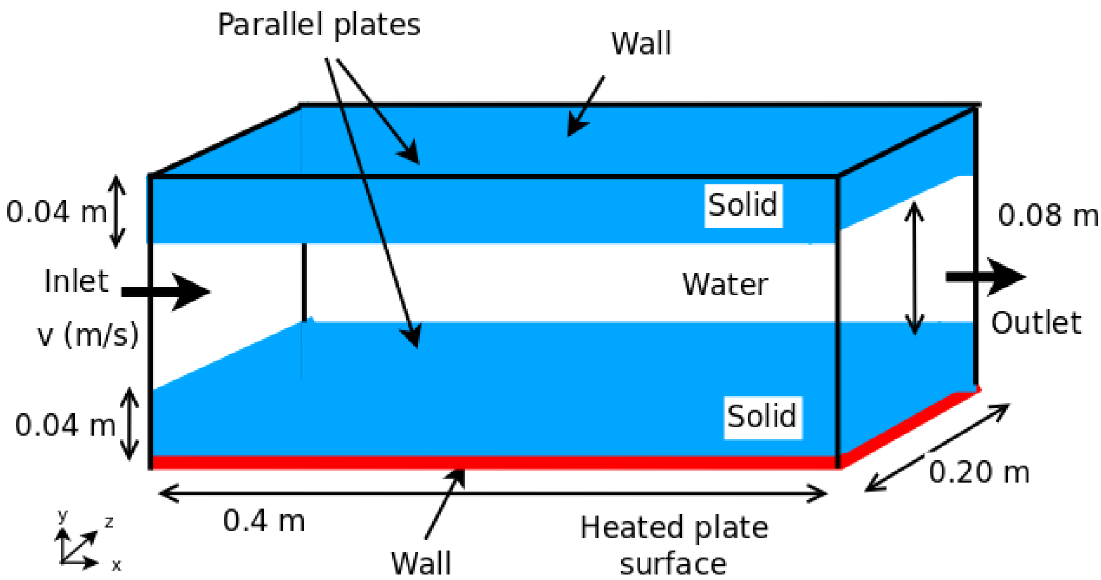


Figure 1: Schematic diagram of a heater

3 Mathematical modeling

For each region defined as fluid and solid, the equation for the fluid is solved as well as the solid region. The solid and fluid regions are coupled by a thermal boundary condition [2, 4, 3].

3.1 Equations for fluid

3.1.1 Continuity equation

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

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

3.1.2 Momentum equation

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

where, u represents the velocity, ρ density of the fluid, p_{rgh} = p - ρgh, the pressure minus the hydrostatic pressure , g_i the gravity and $\tau_{t_{ij}}$, and τ_{ij} are the turbulent, and viscose stresses respectively.

3.1.3 Energy conservation

The energy equation or the change rate of the total energy can be written as (Equation- 3):

$$\begin{aligned}\frac{\partial(\rho h)}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j h) + \frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j k) = \\ -\frac{\partial(q_i + q_{ti})}{\partial x_i} + \rho S + Rad + \frac{\partial p}{\partial t} - \rho g_j u_j + \frac{\partial}{\partial x_j}(\tau_{ij} u_i)\end{aligned} \tag{3}$$

where, kinetic energy (k = 0.5 $u_i u_i$), internal energy (e); heat transferred to fluid element by diffusion and turbulence $q_i + q_{ti}$; heat source term S; the heat source by radiation Rad; the enthalpy(h), h = e + p/ρ; density of fluid ρ .

3.2 Equations for solid

For the solid regions, the energy equation can be written as (Equation- 4):

$$\frac{\partial(\rho h)}{\partial t} = \frac{\partial}{\partial x_j}\left(\alpha \frac{\partial h}{\partial x_j}\right) \tag{4}$$

where, h, ρ and α denote the specific enthalpy, density and thermal diffusivity (κ/c_p) respectively.

3.3 Coupling between fluid and solid

The temperature (T) is same at interface both phases .i.e solid (s) and fluid (f).

$$T_f = T_s \tag{5}$$

The heat flux entering one region at one side of the interface, it should be equal to the heat flux leaving the other region on the other side of the domain.

$$Q_f = -Q_s \tag{6}$$

If, radiation is neglected then the above expression can be written as:

$$\kappa_f \frac{dT_f}{dn} = -\kappa_s \frac{dT_s}{dn} \tag{7}$$

where, n represents the direction normal to the wall. κ_f and κ_s are the thermal conductivity of the fluid and solid respectively.

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	
	heater (solid)	water (fluid)
maxY	wall	
minX	wall	Velocity
maxX	wall	Pressure
minY	wall	
minZ	wall	wall
maxZ	wall	wall
heater_to_water	coupled	coupled
water_to_heater	coupled	coupled

4 Simulation procedure

This case deals with there-dimensional laminar simulation in the heater. First step in setting up of an OpenFOAM case is 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 OpenFOAM/(username)-5.x/run/tutorials/heatTransfer/chtMultiRegionFoam/. This study is considered with transient and laminar case.

4.1 Creating geometry and mesh

- Geometry for the present problem is considered 3D dimensional domain. The geometry and mesh are generated by using the blockMesh utility.
- Figure 2 shows the isometric view of the generated mesh using blockMesh utility.
- Meshing a geometry with more than one region.
- All modifications for mesh with proper boundary conditions with defining inlet and outlet is to be done as velocity patch and its neighbor patch (∼/case/constant/polyMesh/boundary).

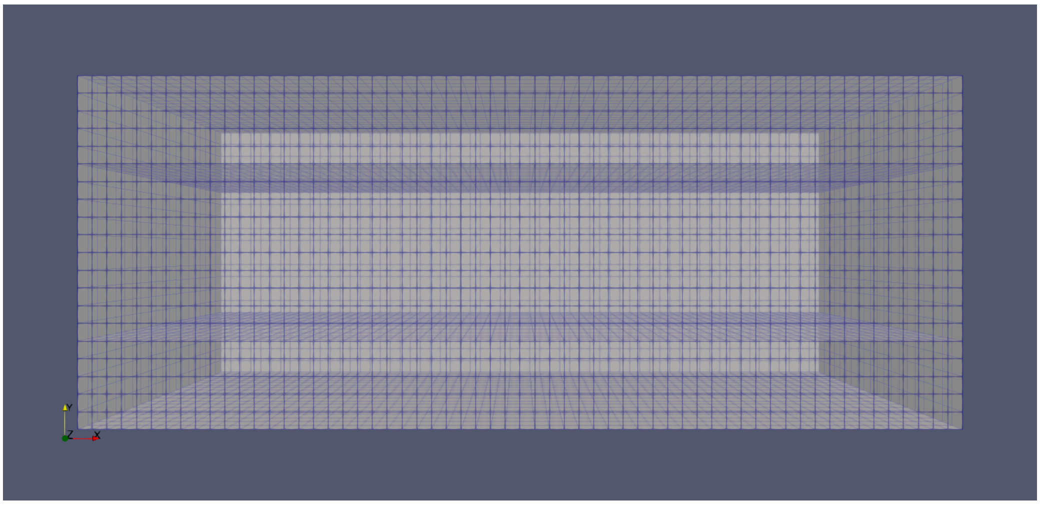


Figure 2: Computational geometry of heater

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

All the boundary conditions for fields variables are mentioned in ’0’ file folder. Boundary conditions are discussed and that remain same during simulation. Files present in ‘0’ folder (∼/case/0/water/ and ∼/case/0/heater/) has been kept ’cellToRegion’, ’p’, ’p_rgh’, ’T’ and ’U’ files for laminar and transient flow. Boundaries are assigned and added seven boundaries of present case in three the files, i.e. ’cellToRegion’, ’p’, ’p_rgh’, ’T’ and ’U’ respectively. Details of the boundary conditions are listed in Table 3 & 4.

Table 3: Details of heater (solid) boundary conditions

Boundary	p	p_rgh	U	cellToRegion	T
maxY	calculated	calculated	calculated	zeroGradient	fixedValue
minX	calculated	calculated	calculated	zeroGradient	zeroGradient
maxX	calculated	calculated	calculated	zeroGradient	zeroGradient
minY	calculated	calculated	calculated	zeroGradient	zeroGradient
minZ	calculated	calculated	calculated	zeroGradient	zeroGradient
maxZ	calculated	calculated	calculated	zeroGradient	zeroGradient
heater_to_water	calculated	calculated	calculated	calculated	compressible::turbulent TemperatureCoupled BaffleMixed

Table 4: Details of water (fluid) boundary conditions

Boundary	p	p_rgh	U	cellToRegion	T
minX	calculated	zeroGradient	fixedValue	zeroGradient	fixedValue
maxX	calculated	fixedValue	inletOutlet	zeroGradient	inletOutlet
minZ	calculated	fixedFluxPressure	noSlip	zeroGradient	zeroGradient
maxZ	calculated	fixedFluxPressure	noSlip	zeroGradient	zeroGradient
water_to_heater	calculated	fixedFluxPressure	noSlip	calculated	compressible::turbulent TemperatureCoupled BaffleMixed

4.3 Solver details

In the present study, unsteady state and laminar flow are considered. Laminar flow model can be applied in the OpenFOAM by using 'simulationType' option in the 'turbulenceProperties' file in constant folder. In order to run transient 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.

Here is the brief introduction of **OpenFOAM**, a toolbox of CFD simulation: "OpenFOAM" is an open source toolbox for CFD simulations. "chtMultiRegionFoam" is an open source CFD solver of OpenFOAM. Multi-region OpenFOAM case structure is slightly adapted from the standard case structure. Within each "fluid" and "solid" subdirector, there exists the standard contents ("polyMesh", "transportProperties", "fvSchemes", "fvSolution", etc.)

The structure of the multi-region **case** is like this:

```
case
|----0/
     |----water/
               |----cellToRegion
               |----p
               |----p_rgh
               |----T
               |----U
     |----heater
               |----cellToRegion
               |----p
               |----p_rgh
               |----T
               |----U
     |----cellToRegion
     |----p
     |----p_rgh
     |----T
     |----U

|----constant/
     |----water/
               |----g
               |----radiationProperties
               |----thermophysicalProperties
               |----turbulenceProperties
     |----heater
               |----radiationProperties
               |----thermophysicalProperties
     |----regionProperties

|----system/
     |----water/
               |----changeDictionaryDict
               |----fvSchemes
               |----fvSolution
     |----heater
               |----changeDictionaryDict
               |----fvOptions
               |----fvSchemes
               |----fvSolution
```

```
        |----blockMeshDict
        |----controlDict
        |----fvSchemes
        |----fvSolution
        |----topoSetDict
|----clean.sh
|----run.sh
```

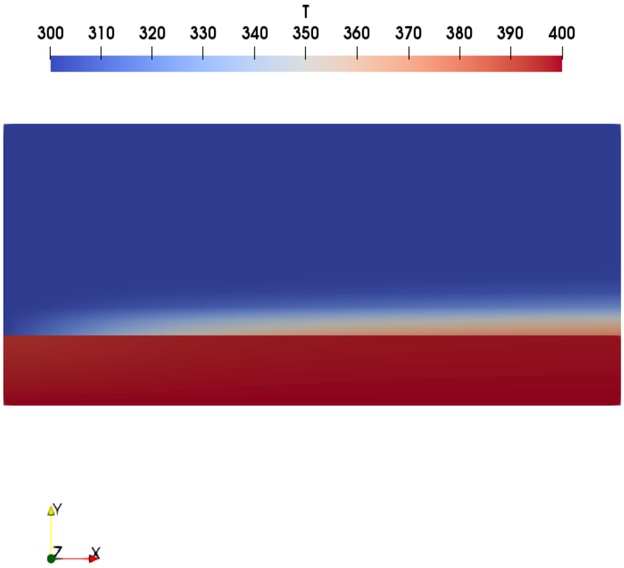
4.4 Post-processing

The paraFoam, it can be used to visualize the simulations results in OpenFOAM. This can be run by typing the following command line in the terminal **paraFoam -builtin** to open the ParaView software and upload the case.

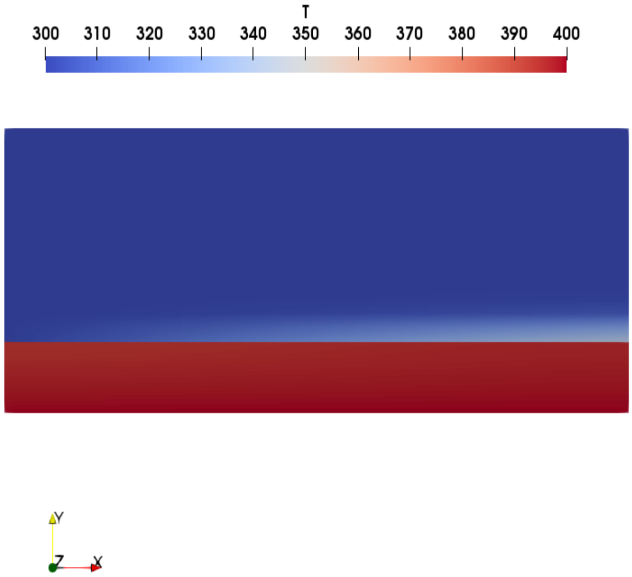
5 Results and discussion

In this section, the numerical results are shown for conjugate heat transfer (conduction and convection) in a heater. The physical properties of the heater (solid) and water (fluid) are given in Table 1.

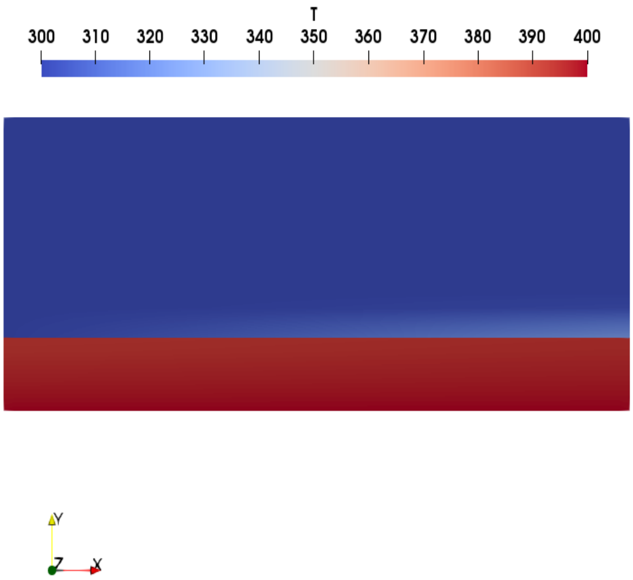
Figure 3 shows the temperature distribution in the heater, viewed along the z-axis. It is observed that the temperature distribution is more at low inlet water velocity, v=0.001 m/s, and less temperature distribution at high inlet water velocity, v=0.0 m/s.



(a) v = 0.001 m/s



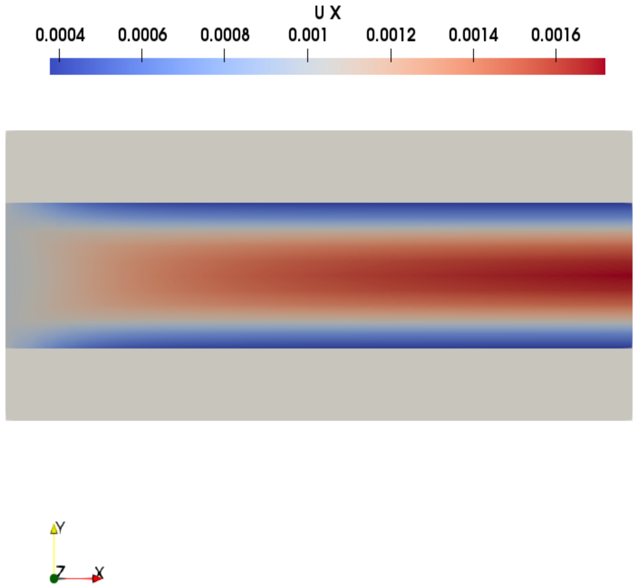
(b) v = 0.005 m/s



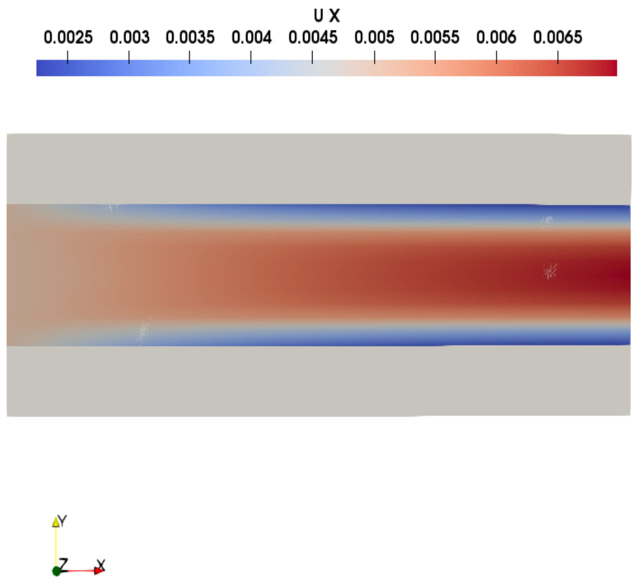
(c) v = 0.01 m/s

Figure 3: Temperature contour plots in heater for different inlet velocities at time t=2000 s (steady-state achieved)

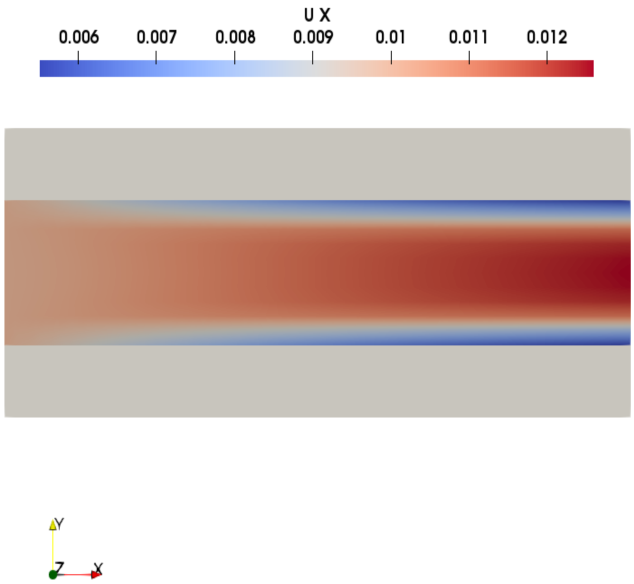
Figure 4 shows the velocity contour plots captured for three different inlet velocity of water, v=0.001, 0.005 & 0.01 m/s at time t=2000 s. It can be seen in Figure 4 the velocity tends to zero at near the wall so it is verified no-slip condition and maximum velocity at the center.



(a) v = 0.001 m/s



(b) v = 0.005 m/s

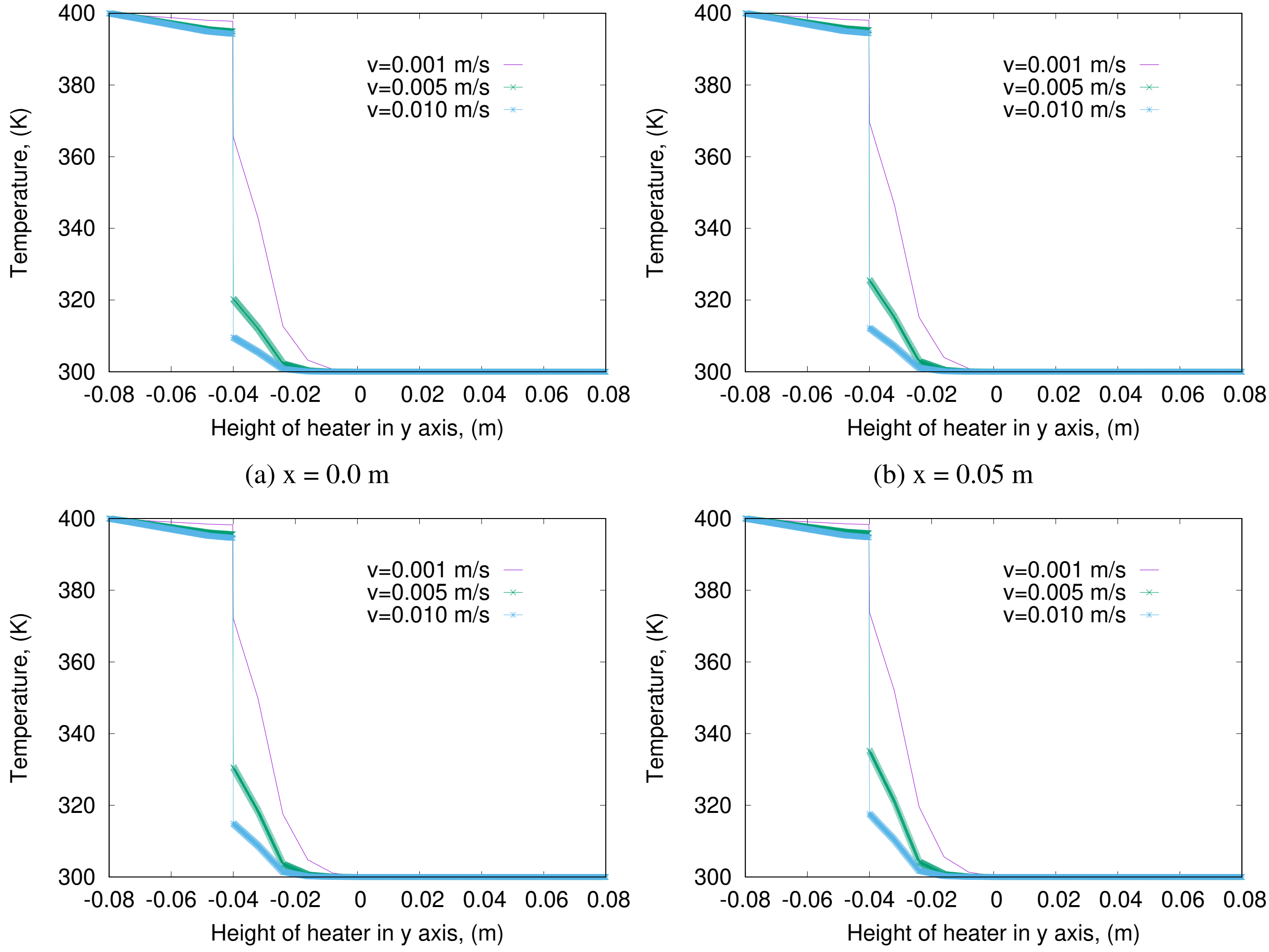


(c) v = 0.01 m/s

Figure 4: Velocity contour plots in heater for different inlet velocities at time t=2000 s (steady-state achieved).

Figure 5 shows the comparison of the field T (Temperature profile) for three velocities, v=0.001, 0.005 & 0.01 m/s at a different axis. i.e. x=0 , 0.05, 0.1 & 0.15 m. The variation in temperature is between 300 K to 400K. In Figure 5a, the temperature changes more along with the height of the heater at less inlet velocity of water for x=0. It is verified more heat transfer from the heated surface to the cold surface at less inlet velocity of the water. It means that the fluid will stay more time with the contact of the heated surface at less inlet velocity of the water.

It is found that the similarity in the results, which are obtained from the simulations for x=0.05, 0.1 & 0.15 m (Figure 5b, c & d). The simulation results are analyzed when the system is archived steady-state (t=5000 s).



(a) x = 0.0 m

(b) x = 0.05 m

(c) x = 0.10 m

(d) x = 0.15 m

Figure 5: Temperature profile variation along with the height of heater for different inlet velocities at time t=2000 s (steady-state achieved)

References

[1] C. J. Greenshields, *OpenFOAM: The OpenFOAM Foundation. User Guide Version 5*. CFD Direct Limited, July. 2017.

[2] L. M. Moukalled, F. and M. Darwish, *The finite volume method in computational fluid dynamics*. An Advanced Introduction with OpenFOAM and Matlab, 2016.

[3] P. J. P. Robert W. Fox, Alan T. McDonald, *Introduction to Fluid Mechanics (8th ed)*, 8th ed., 2011.

[4] M. Darwish, F. Moukalled, *A unified formulation of the segregated class of algorithms for fluid flow at all speeds*. Numerical Heat Transfer: Part B: Fundamentals, 2000.