Simulation of Flow over Fin and Heat Transfer through Fin in Forced Convection by Cyclic Boundary Condition

Onkar Balasaheb Patil

MTech, IIT Bombay

Synopsis

This case study project aims to study and simulate water flow over fin at constant wall Temperature subjected to forced convection using OpenFOAM – V2021. The geometry and mesh were defined using blockMeshDict file. A steady state, PIMPLE algorithm based solver chtMultiRegionFoam was used in the simulation. The analysis was executed by TryfonC. Roumpedakis et al [1] using fluent was taken as reference.

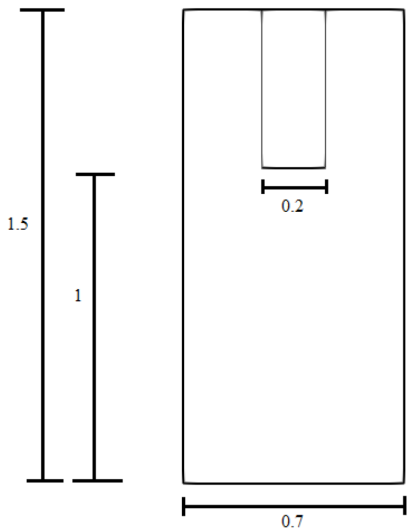

Fig.1

Dimensions of geometry are Length 0.7 m and Height 1.5 m, Water enters at a velocity of 5 m/s from the Left side. Inlet and Outlet had given cyclic boundary conditions.

References

[1] TryfonC. Roumpedakis et al. "Experimental Investigation and CFD Analysis of Heat Transfer in Single Phase Subcooler of a Small Scale Waste Heat Recovery ORC." Energy Procedia 129(2017) 487-494.

1. Introduction

In reference paper [1] Flow and Heat transfer in finned heat exchanger was studied. The Heat exchanger in a paper is having R134a as hot fluid, finned wall, and water as a cold fluid. Fins are on water side. Ansys fluent is used for simulation in research paper. In this case study. Water side of the finned heat exchanger is modeled using OpenFOAM. Single fin is considered with constant base temperature, to study flow over multiple fins in a row, the cyclic boundary condition is applied on inlet and outlet side. Convective heat transfer coefficient of flow is calculated and Temperature profile in the fin is verified by analytical solution. chtMultiRegionFoam Solver in OpenFOAM is used in this case study.

2. Governing Equations and Models

To simulate case study OpenFOAM – V2021 was used, Heat equation governs conduction in Fin and Fluid flow is governed by Continuity, Navier – Stokes equation ,and Energy equation.

2.1 Governing Equation for Solid

Heat equation for the fin is, at steady state and for the fin of uniform cross section,

$$\frac{d^2T}{dx^2} - \frac{hp}{kA}(T - T_0) = 0$$

Where, h - Convective heat transfer coefficient of outside fluid

k - Thermal conductivity of fin material

A - cross section area of fin

p - perimeter of fin.

2.2 Governing Equations for water

For incompressible flow, 2 Directional flow at steady state,

The Continuity equation is,

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

X – direction momentum equation (Navier – Stokes equation),

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial P}{\partial x} + \vartheta\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right)$$

Y – direction momentum equation (Navier – Stokes equation),

$$u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{1}{\rho}\frac{\partial P}{\partial y} + \vartheta\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right)$$

The Energy equation is, (with no source term)

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

In above equations viscous dissipation is neglected.

u,v,P,T are x direction velocity, y direction velocity, pressure, and temperature respectively. ϑ and α are kinematic viscocity and thermal diffusivity respectively, such that,

$$\vartheta = \frac{\mu}{\rho} \quad \text{and} \quad \alpha = \frac{k}{\rho C}$$

3. Simulation procedure

3.1 Geometry and Mesh

Geometry presented in fig. 1 has two sections, fluid and metal. Height of fluid section is kept large for observing variation of temperature along y direction. A 2 directional geometry is considered, fluid is water and metal is aluminium. Geometry of single block is created using blockMeshDict file under system folder.

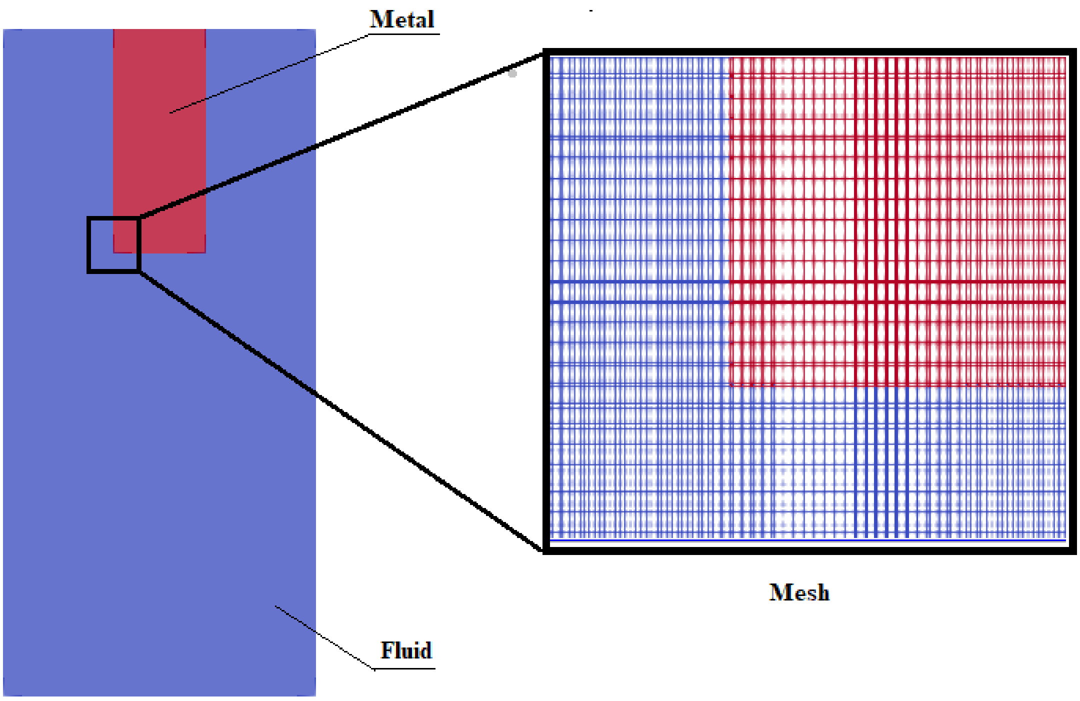

Fig.2

200 x 200 cells are taken in geometry. Fluid and the Metal are separately defined in topoSetDict file under system folder. blockMesh command is given to mesh geometry, topoSet command is given to create fluid and metal zones and regions are splited by command splitMeshRegions –cellZones –overwrite.

3.2 Initial and Boundary conditions

There are seven boundary conditions are to be specified for inlet, outlet, upper wall, front, back, lower wall and metal interface from which inlet and outlet are kept cyclic, front and back are kept empty and rest are at no slip boundary condition, the upper wall is maintained at 373 K and internal field is at uniform 300 K. velocity source term 5m/s is given in fvOptions.

Boundary Condition	Velocity	Temperature
Inlet	Cyclic (from outlet)	Cyclic
Outlet	Cyclic (to inlet)	Cyclic
Front	Empty	Empty
Back	Empty	Empty
Upper wall	No slip	Constant temperature
Lower wall	No slip	Adiabatic
Metal interface	No slip	Heat transfer from metal to fluid

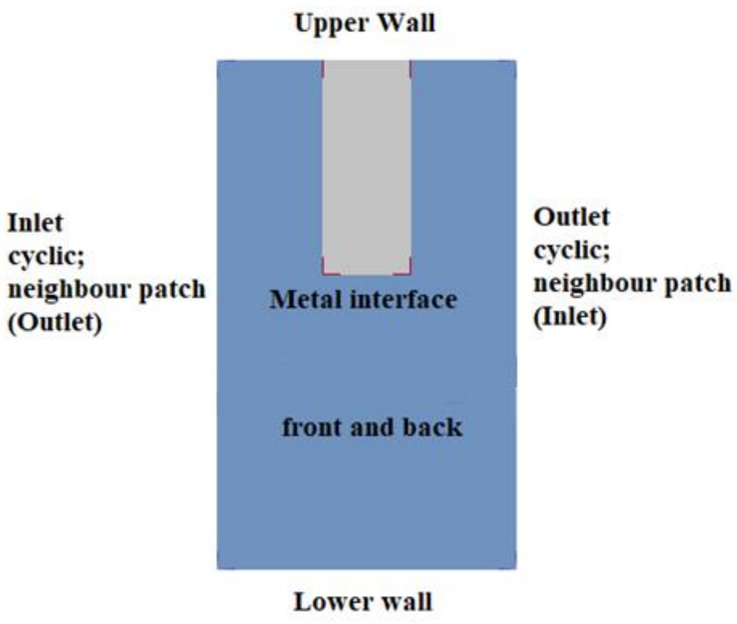

Fig.3

3.3 Solver

For steady state, incompressible fluid and metal problem, chtMultiRegionFoam solver is used, in this equation for each variable characterizing the system is solved sequence wise and the solution of preceding equation is inserted in subsequent equation. Same is used for fluid and metal coupling. First equation for fluid solved using temperature of solid of preceding iteration, this defines boundary condition for temperature in fluid. After this equation for solid is solved using temperature of fluid of preceding iteration. This loop continues until convergence is reached. PIMPLE algorithm is used in this solver.

4. Results and Discussion

4.1 Velocity Profile

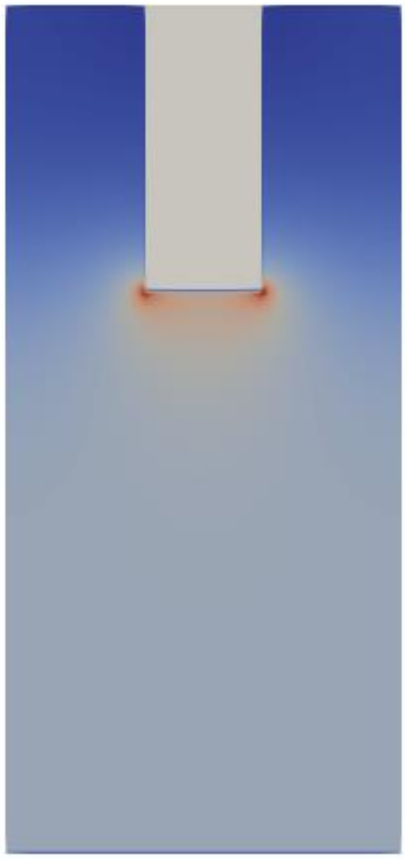

Fig.4

Increase in velocity can be observed at tip of fin, away from fin, towards wall velocity is almost equal to free stream velocity. in region between fins very low velocity can be observed.

4.2 Temperature Profile

Fig.5

Decrease in temperature from fin base towards negative y direction can be observed, fin tip is almost at fluid temperature.

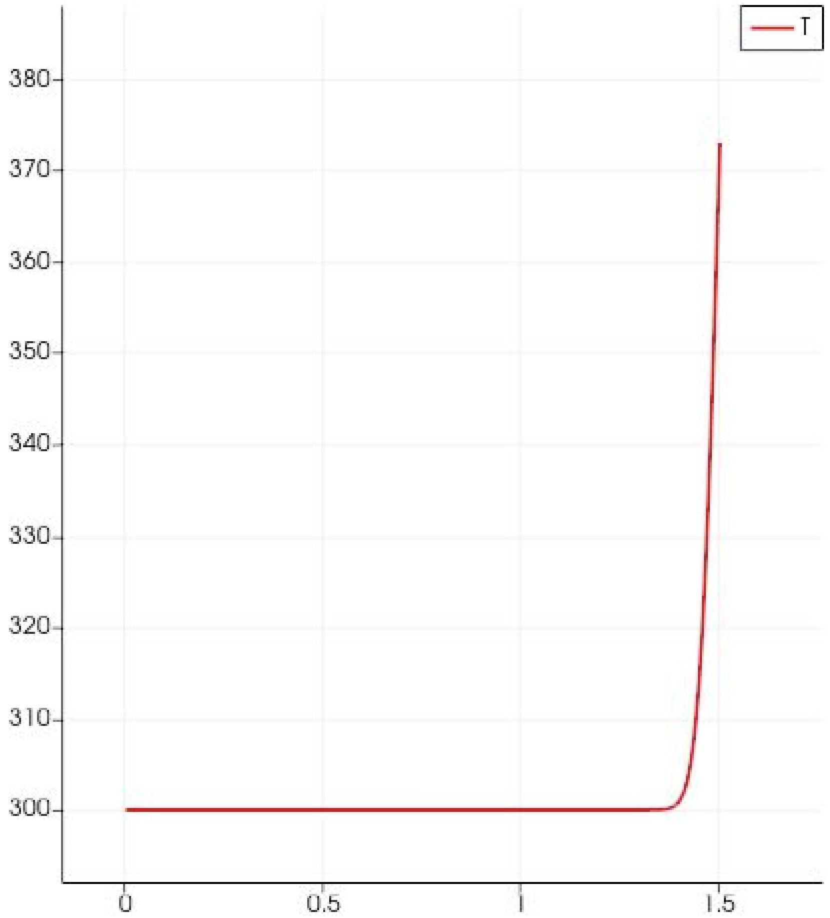

Fig.6

Above graph shows Temperature variation along y direction, exponential decrease in temperature along length of fin can be observed in above graph.

4.3 Analytical Solution

For validating solution given by OpenFOAM, analytical solution for temperature variation inside fin is considered.

Solution of differential equation,

$$\frac{d^2T}{dx^2} - \frac{hp}{kA}(T - T_0) = 0$$

$$T(0) = 373; \quad T(0.5) = 300; \quad x = -y$$

is, $T = T_0 + \frac{(T_b - T_0)}{1 - e^{\sqrt{\frac{hp}{kA}}}}\left(e^{\sqrt{\frac{hp}{kA}}x} - e^{\sqrt{\frac{hp}{kA}}(1-x)}\right)$

k = 200 W/m^2K; A = 0.02 m^2; p = 0.42 m;

chtMultiregionFoam postprocessing given heat transfer coefficient as h = 48 W/m^2.

Which gives plot of T vs x in matlab as shown below.

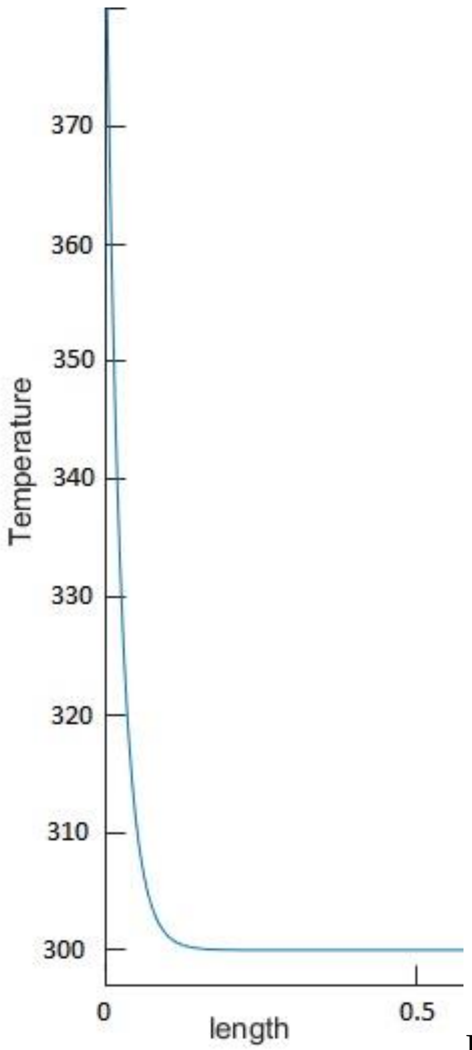

Fig.7

4.4 Validation of Result

In our geometry x = -y, hence image is flipped vertically for comparing graphs.

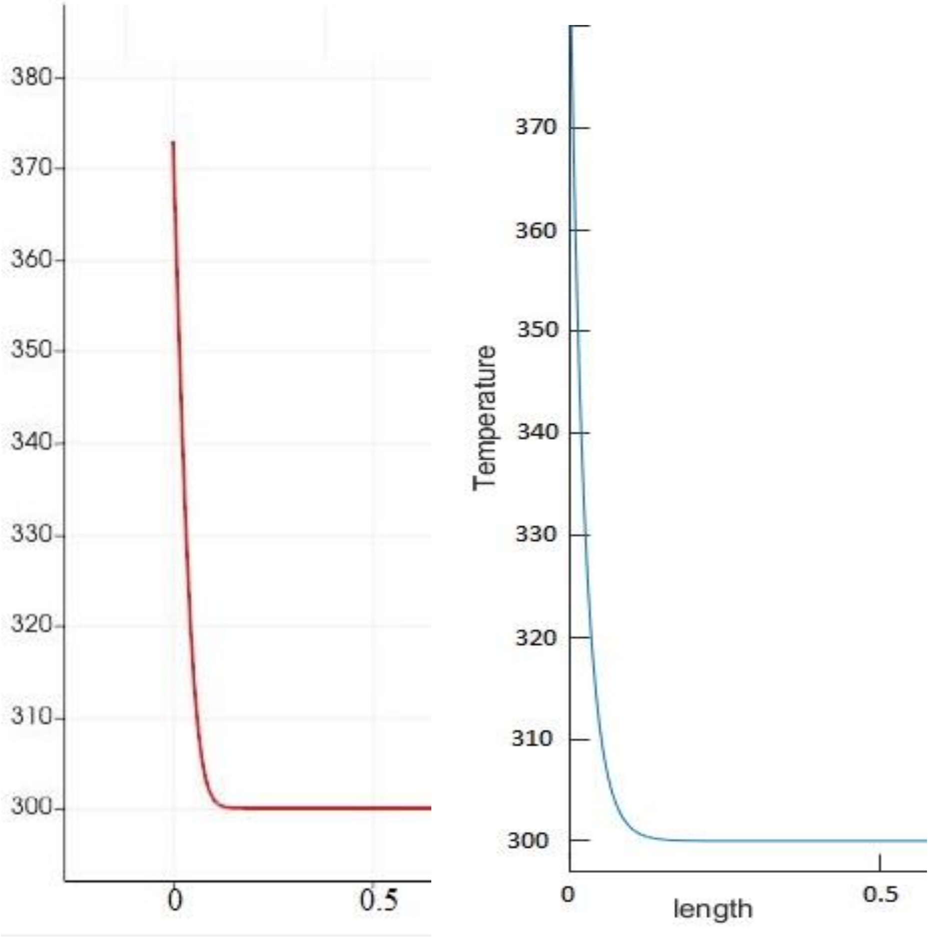

Fig.8

OpenFOAM **Analytical**

Two graphs are approximately same. It validated result

5. References

[1] TryfonC. Roumpedakis et al. “Experimental Investigation and CFD Analysis of Heat Transfer in Single Phase Subcooler of a Small Scale Waste Heat Recovery ORC.” Energy Procedia 129(2017) 487-494.

[2] Incropera’s Principles of Heat and Mass Transfer, WILEY, Indian edition(2019).