FOSSEE, IIT Bombay
OpenFOAM Case study
April 23, 2022

# Study of Convective Heat Transfer over Series of Tubes by Cyclic and Symmetric Boundary Condition

Onkar Balasaheb Patil

MTech, IIT Bombay

Synopsis

This case study project aims to study and simulate water flow over series of tubes at constant wall Temperature subjected to forced convection using OpenFOAM – V2021. The geometry and mesh were imported from Ansys Fluent. A steady state, SIMPLE algorithm based solver buoyantSimpleFoam was used in the simulation. The analysis in Ansys fluent tutorial was taken as reference.

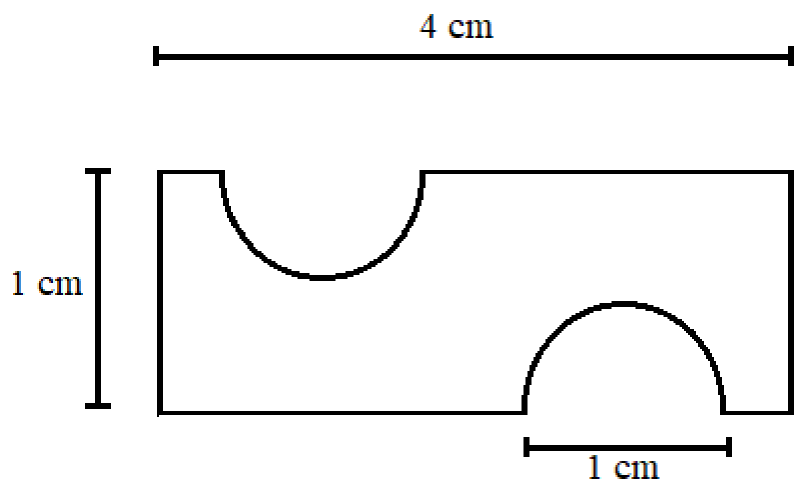


Fig.1

Dimensions of geometry are Length 4 cm and Height 1 cm, Water enters at velocity 5 m/s from Left side. Inlet and Outlet had given cyclic boundary condition and walls other than curves are at symmetric boundary condition.

## References

[1] Ansys fluent Tutorial on symmetric and periodic boundary condition.

# 1. Introduction

In many Heat exchangers there is flow over series of tubes. In this case study, such flow is modelled using OpenFOAM. A small part of heat exchanger is considered with tubes are at constant Temperature, Cyclic and symmetric boundary conditions are used to visualize flow for entire geometry. Temperature variation in domain is ploted and Temperature profile is verified by Ansys fluent. buoyantSimpleFoam Solver in OpenFOAM is used in this case study.

# 2. Governing Equations and Models

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

## 2.1 Governing Equations for fluid flow

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),

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

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. $\vartheta$ and $\alpha$ 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 is of fluid flow domain. Geometry is small part of heat exchanger geometry, this small part can be analysed by cyclic and symmetric boundary condition. A 2 directional geometry is considered, fluid is water. Geometry and mesh is created using Ansys Fluent, and it imported in OpenFOAM.

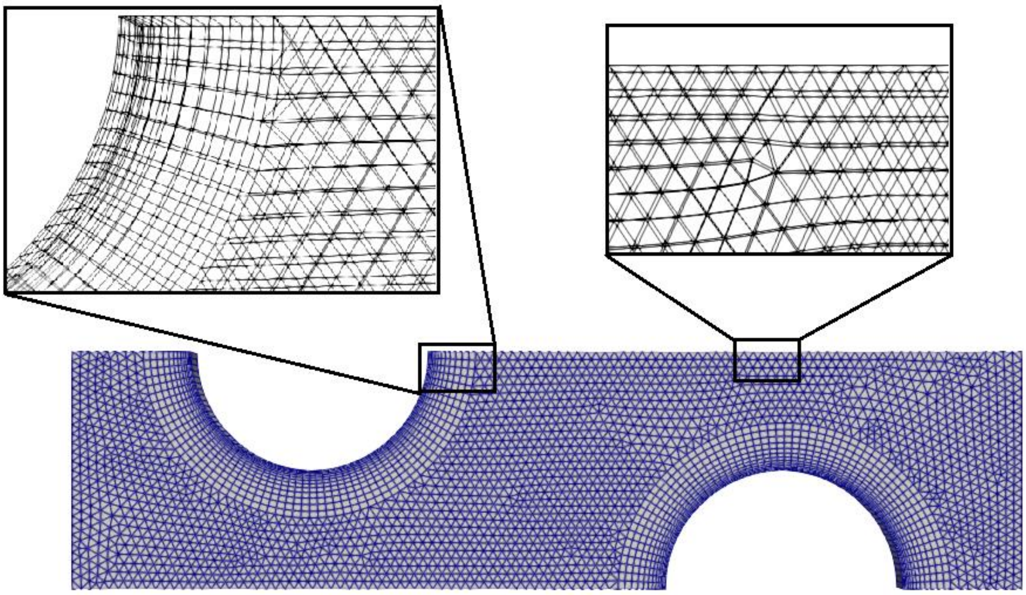

Fig.2

Tetrahedral cells were taken in geometry. After command FluentMeshToFoam <file.msh>, boundaries were created in polymesh folder in constant folder

### 3.2 Initial and Boundary conditions

There are eight boundary conditions are to be specified for inlet, outlet, walls and curved surfaces inlet and outlet are named as wall 12 and wall 9 respectively and cyclic boundary condition is given to inlet and outlet, curved surfaces are at no slip boundary condition and maintained at 400 K, rest walls are assigned with symmetric boundary condition. velocity source term 5m/s is given in fvOptions.

| Boundary Condition | Velocity | Temperature |
|---|---|---|
| Wall 12 (inlet) | Cyclic (from outlet) | Cyclic |
| Wall 9(outlet) | Cyclic (to inlet) | Cyclic |
| Wall 21 | no slip | Constant temperature |
| Wall 3 | no slip | Constant temperature |
| Wall 11 | symmetric | symmetric |
| Wall 13 | symmetric | symmetric |
| Wall 18 | symmetric | symmetric |
| Wall 24 | symmetric | symmetric |

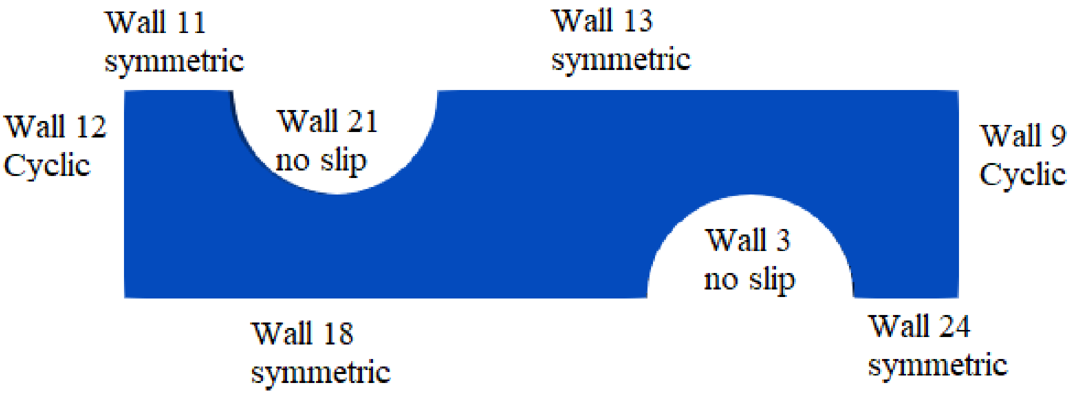


Fig.3

### 3.3 Solver

For steady state, heat transfer problem, buoyantSimpleFoam solver is used, it is steady state, compressible, turbulence, heat transfer solver. It uses SIMPLE algorithm. It has equations of SIMPLE algorithm and Hydrostatic pressure effects. buoyantSimpleFoam requires inputs as p (pressure), p_rgh (pressure-hydrostatic contribution), U (Velocity), T (Temperature). In SIMPLE algorithm pressure value is taken from previous iteration, checked for continuity and pressure value is modified. This loop continues until convergence is reached.

## 4. Results and Discussion

### 4.1 Velocity Profile

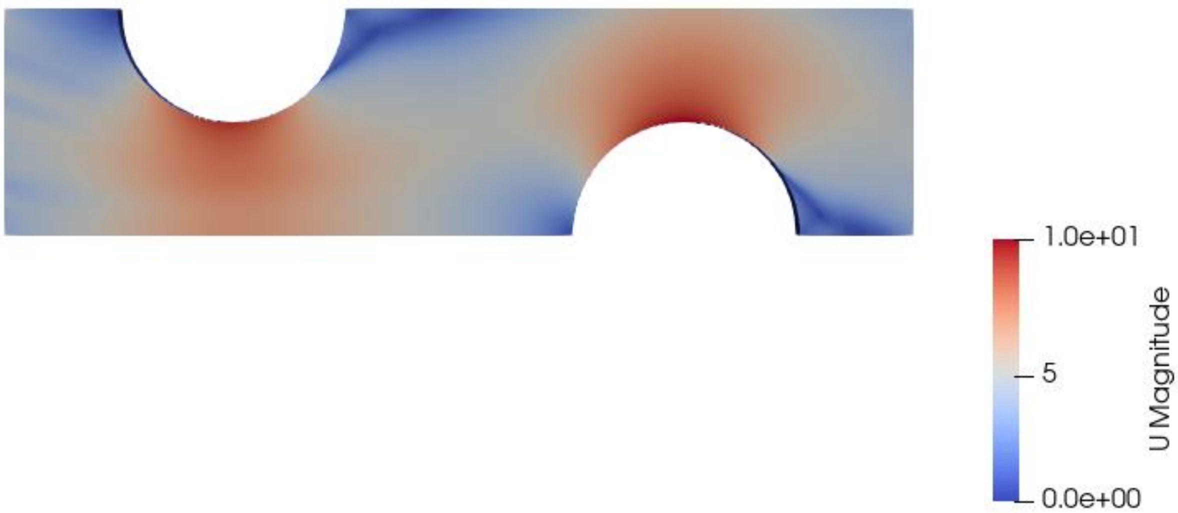


Fig.4

Increase in velocity can be observed near curved surfaces, stagnation effect is visible at upstream side of curved surface

### 4.2 Temperature Profile

Fig.5

Increase in Temperature towards downstream side of curve was observed. Change in temperature is low due to high velocity.

### 4.3 Velocity and Temperature profile given by Ansys fluent

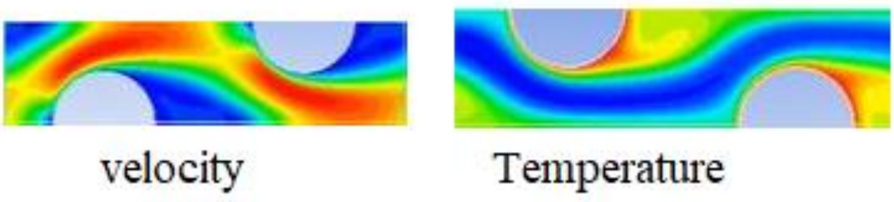


Fig.6

### 4.4 Validation of Result

From figures 4,5 and 6, it is observed that profiles are not exactly same, velocity profile is approximately same but there is significant difference in Temperature profile.

## 5. References

[1] Ansys fluent Tutorial on symmetric and periodic boundary condition.