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.
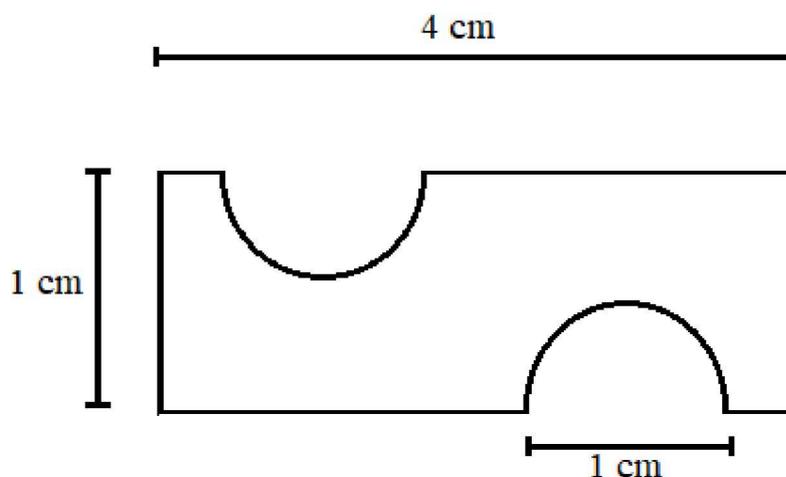

Fig.1

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.