“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.

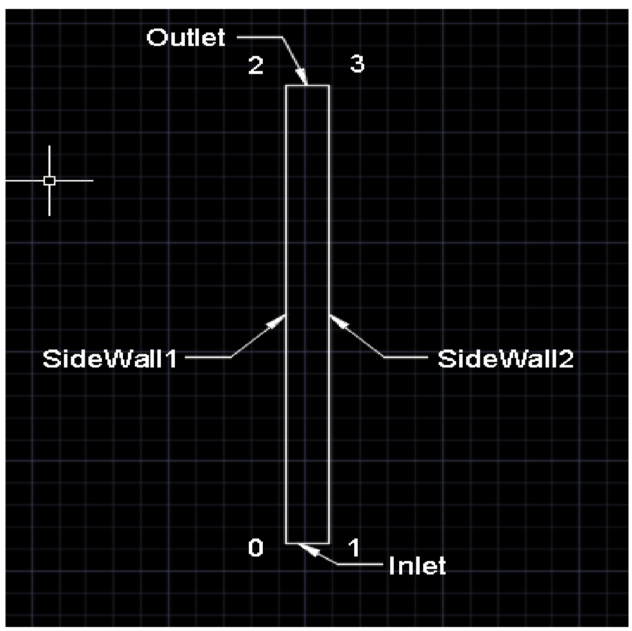

Figure (1): The Geometry

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