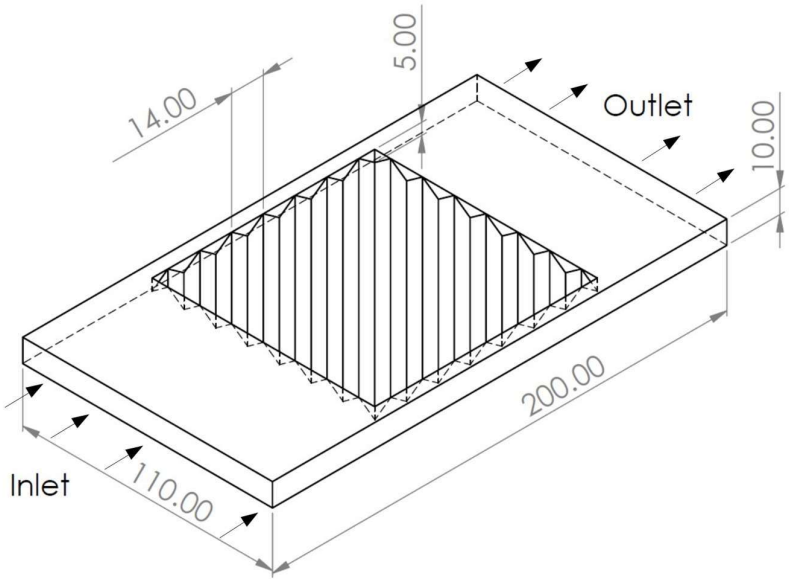
FOSSEE, IIT Bombay
OpenFOAM Research Migration Project
April 21, 2022

SIMULATION OF NARROW CHANNELS WITH CORRUGATED WALLS

Mohammed Danish

Synopsis

This research migration project aims to do numerical simulations of compact heat exchangers made of corrugated plates using OpenFOAM 7. The channel used for the simulation is formed by only one corrugated plate, while the other plate is flat. The mesh was defined using Ansys Meshing software. A steady-state, bouyantSimpleFoam solver was used in the simulation. A two-equation turbulence model $\kappa - \omega$ *SST* is used, in addition to isothermal flow, heat transfer simulations are conducted for a Reynolds number range (900–1400), for the case of hot water (60ºC) in contact with a constant-temperature wall (20ºC). The analysis executed by Kanaris et. al. using commercial CFD code, *CFX* was taken as a reference.



Geometry of the Heat Exchanger

There are 14 equal sized corrugations present with the distance between the plates at the conduit entrance is 10 mm. Flowing fluid (water) is entering from inlet with velocity (0.07 m/s – 0.1 m/s) corresponding to the Reynolds number and exiting from outlet.

References

Kanaris, Athanasios & Mouza, Aikaterini & Paras, Spiros. (2005). “Flow and Heat Transfer in Narrow Channels with Corrugated Walls”. Chemical Engineering Research & Design. Vol: 83. Page: 460-468. DOI: https://doi.org/10.1205/cherd.04162.