# Forced convection past a semi-circular cylinder at incidence with a downstream circular cylinder

Saptarshi Joshi
Department of Mechanical Engineering, IIT Bombay

Synopsis

This research migration project aims to do numerical simulations of forced convection past a semi-circular cylinder at incidence with a downstream circular cylinder [1] using `OpenFOAM`. The geometry and mesh were defined using ANSYS. The transient simulations were performed using the SIMPLE and PISO algorithms-based `pimpleFoam` solver. The thermo-fluidic transport is studied in the Laminar flow regime at a Reynolds number of $Re = 100$. The analysis executed by Sarkar et. al. [1] using commercial CFD code `Fluent` was taken as a reference for validation.
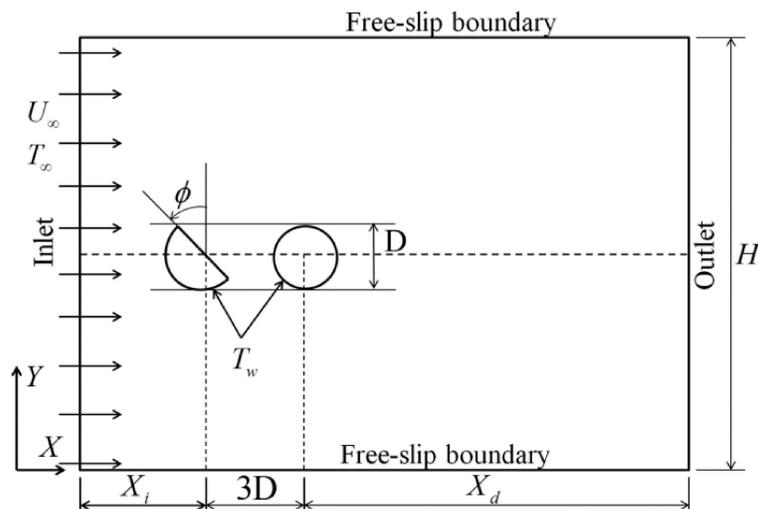
The angle of incidence of the semi-circular cylinder is $\phi = 45°$. The upstream, downstream distances and the height of the domain are chosen to be $X_i = 8D$, $X_d = 37D$ and $H = 20D$. Here, D is the the projected diameter of the semi-circular cylinder and the diameter of the tandem circular cylinder.