Implementing Streamwise Periodic Boundary Condition in OpenFOAM

Harikrishnan S
Research Scholar
Department of Mechanical Engineering
Indian Institute of Technology Madras, Chennai, Tamilnadu, India

ABSTRACT

The objective of the present project is to describe the implementation of streamwise periodic boundary condition in open source CFD package OpenFOAM. Periodic boundary conditions are used in numerical investigations to save the computational time. Fully developed flow in plane channel has been considered in the present study as shown in Fig. 1. Implementation of streamwise periodic boundary adds a source term in the Navier-Stokes equation. This project also explains setting up OpenFOAM case from existing tutorials available in the OpenFOAM repository. Geometry and mesh has been generated by using 'blockMesh' utility. Source term has been added to the Navier-Stokes equation by using 'fvOption' utility. Steady, laminar flow has been solved by using 'simpleFoam' solver. Obtained results are validated with the analytical results available in the literature. Details regarding geometry and flow has been listed in Table-1.

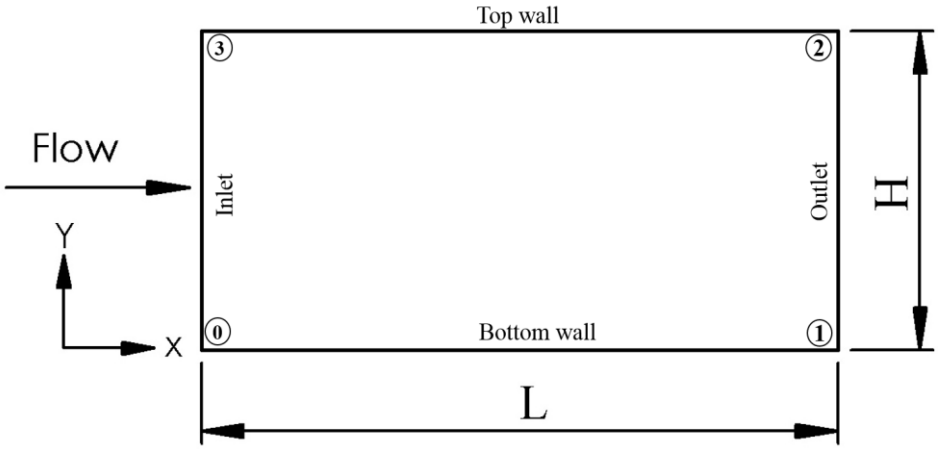

Figure1: Geometry of plane channel

Table-1:- Details of geometry and flow conditions

Geometry details	Length of the channel (L) = 1m
	Height of the channel (H) = 0.5m
	Hydraulic diameter (D_H) = 0.5m
Fluid Property	Kinematic viscosity = 0.01 m^2/s
Reynolds number	Re =50