Numerical simulation of flow through 2-D channel with constriction using OpenFOAM

Syed Aaqib Ahmed A

Sr. Design Engineer, TTCADD, Vellore – India

Email: syedaaqibahmeda@gmail.com

Abstract

This report aims to present the characteristics of air flow through channel with constriction. The geometry of the flow channel is of 2m long with 0.1m x 0.01m. The span of constriction considered is 0.01m. For the analysis, the three geometry (0.01m, 0.02m and 0.03m) of the constriction are evaluated. Both the geometry and the mesh are developed and generated using the blockMesh utility of the OpenFoam V-12. The generated mesh is simple-graded. The simulations are carried out using the icoFoam solver, which is a transient laminar flow solver. The analysis are conducted for three Reynolds number conditions of Re = 20, 100 and 200. Accordingly, there are nine simulations. The velocity at the constriction, before the constriction and after the constriction are evaluated. Also, the pressure gradient across the flow is described. The velocity and pressure gradient contours are obtained using the ParaView software. From the analysis, it is observed that with the increase in the span of the constriction, the pressure gradient increases. This increase in the pressure gradient with the increase in the span of constriction is found to be substantial at higher Reynolds number. Further, the velocity profiles and velocity stream lines indicate the recirculation zones, in particular downstream of the constriction. The size of recirculation zones found to increase with Reynolds number and constriction size.

Keywords: *flow through constriction, pressure gradient, OpenFoam*