

Simulation of Flow Through a Y-Splitter

Chaitanya Apte
Veermata Jijabai Technological Institute

Abstract

This case study project aims to perform a simulation of flow through a Y-splitter using OpenFOAM. The geometry mentioned in Figure 1 has one inlet and two outlets both with different cross-sectional diameter. Fluid which is air enters with velocity of 1.56 m/s and leaves at the bifurcated outlets. Flow is incompressible since velocity is very low. In the referred Fluent Study, the exact geometry is used to monitor mass flow-rates at the bifurcated outlets to confirm if splitting is uniform. The results of that study are compared with the simulation results performed using OpenFOAM in this project. Additionally, mesh independence study is done using 4 meshes of 6k, 35k, 70k and 2.5L cells respectively. Transient solver pimpleFoam is used for the simulation. The k-epsilon turbulence model is applied. Similar study conducted at SASTRA University using Ansys Fluent is used as reference.

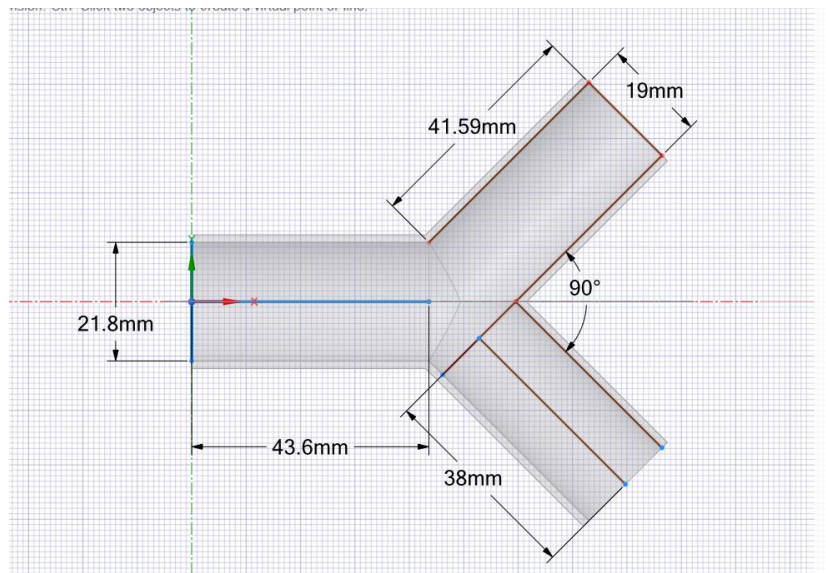


Figure 1:Geometry