Isothermal incompressible fluid flow through a T-Junction

Vipinkumar c p
M.Tech ,Thermal and Fluids Engineering
Department of Mechanical Engineering
Lal Bahadur Shastri college of Engineering, Kasaragod, Kerala, India

Abstract

This case study demonstrates the simulation of isothermal incompressible water flow through a T-junction. In this study the T-junction with 38.1 mm NB is used. The geometric model is created using ANSYS Design Modular and the Meshing is done using ANSYS Meshing Academic R3. The simulations are performed using OpenFOAM-v6.The velocity profiles and pressure distributions are simulated for this case.

Problem statement

The incompressible, isothermal flow through a T-junction as shown in Fig. 1 to 3 for different cases is analysed. Water enters into T junction through inlet and flow through outlet, which are specified separately for each case. The steps followed for the analysis are

1. Creating 2D Geometry by using Ansys Design modular utility.
2. Creating 2D mesh by using Ansys Fluent Meshing utility.
3. Mesh generating using fluentMeshToFoam in to OpenFOAM.
4. Set boundary/initial conditions (BC/IC).
5. Solution using IcoFoam Solver .

The three different cases for the analysis are.

Case 1

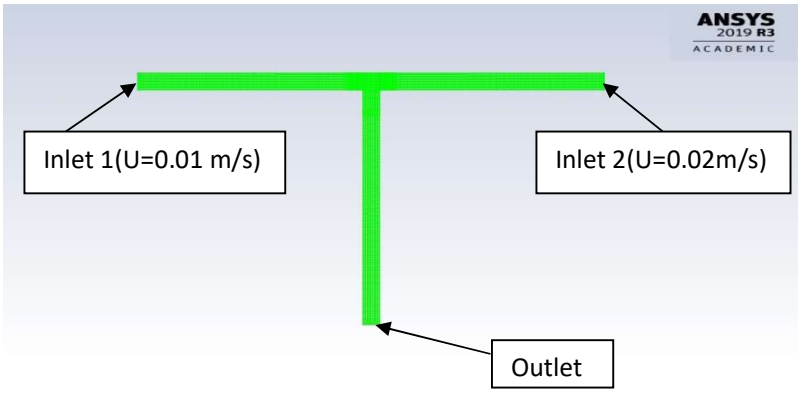

Fig.1

Case 2

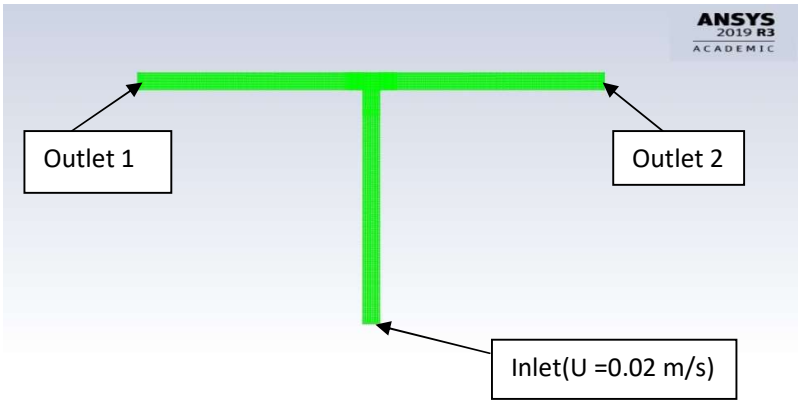

Fig.2

Case 3

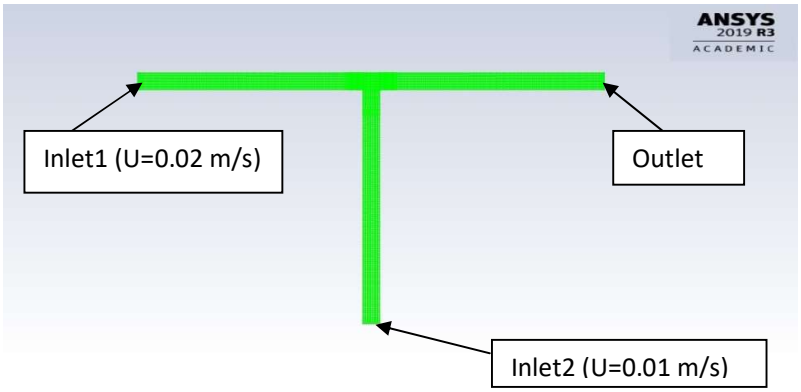

Fig.3