

LES Simulations of turbulent flow in a branched channel

Rozy Kumari¹, Dr. Manaswita Bose², Manjil Sitoula³

¹Ph.D Research Scholar, Indian Institute of Technology (ISM) Dhanbad

²Professor, Indian Institute of Technology Bombay

³Research Assistant, FOSSEE

Abstract

Impactors are the simple device, which can be used to collect particles. There are different types of impactors which can be trap the particles. In this study T shape channel is used to understand the flow pattern in different stages. When air flows with the channel the direction of flow changes at the junction. In this channel air hit the wall when enter the branched channel. In this phenomenon heavy particles are settle and particle having less weight flow with the streamline. This study employs computational fluid dynamics technique (CFD) technique by using OpenFOAM solver. Simulation process encompasses geometry, mesh generation, solver setup and post processing. Large eddy simulation (LES) turbulence model is used for better understanding the flow behaviour at different location and dynamicKEqn is used for sub grid scale model. After post processing analysis it could be possible to find out the location where the particle can settle and where particle will move with the streamline. The finding of this study gives insights to optimize the channel.

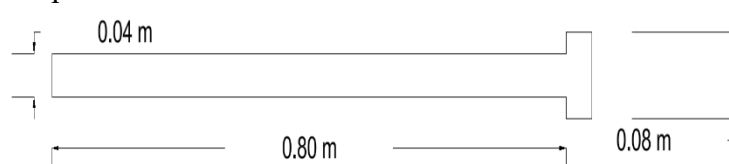


Fig 1 Represent the line diagram of T -channel

Parameter	Value
Length of main channel	0.8 m
Length of branched channel	0.08 m
Width of channels	0.04
Depth	0.01 m

Table 1 Represent the dimension used in T -channel