Turbulent Flow in a Diffuser

Nikhil Sharma

Department of Mechanical Engineering
Malaviya National Institute of Technology Jaipur, Rajasthan India

Abstract

Aim of this case study is to study turbulent flow through a diffuser using OpenFoam software. Two cases, one using Standard k-ϵ turbulence model and the other using -ω SST model needs to be used to study the problem. Numerical results such as the Velocity profile, and turbulent kinetic energy should be compared against experimental data. A 2D asymmetric diffuser is shown below in the figure. It has three major sections an inlet, an angled expansion channel and an outlet channel. The dimensions of the geometry is taken [1] as L1=60 m, H1=2 m, L2=70 m and H2=9.4 m. Flowing fluid is entering from inlet with velocity of 1.25 m/s and exiting from outlet. Fluid properties and boundary conditions are given in the table

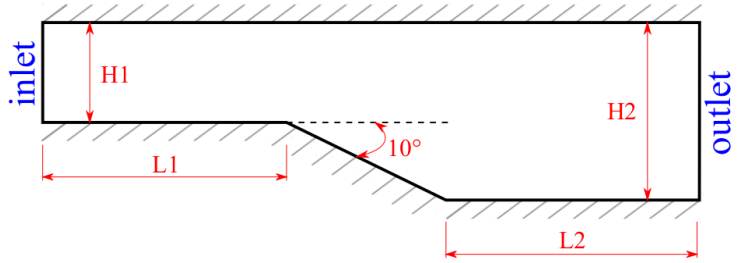

Figure 1: Geometry of the plain wall jet

Variable	Unit	Value
Velocity at Inlet(U)	ms^{-1}	1.25
Pressure at outlet(p)	Pa	0
Density(ρ)	kgm^{-3}	1
Dyanamic viscosity(μ)	$kgm^{-1}s^{-1}$	$1.47 * 10^{-4}$
Turbulent Kinetic Energy (k)	m^2s^{-2}	$1.8 * 10^{-3}$
Turbulent Dissipation Rate (ϵ)	m^2s^{-3}	$9.63 * 10^{-5}$
Turbulent Intensity (I)	%	3.25
Turbulent Mixing Length (L)	m	$3.5 * 10^{-3}$

Table 1: Fluid Properties and Turbulence Parameters

REFERENCE

1.Timur Dogan, Michael Conger, Maysam Mousaviraad, Tao Xing, Fred Stern, Simulation of Turbulent Flow in an Asymmetric Diffuser. Iowa City, IA 522421585