FOSSEE, IIT Bombay
OpenFOAM Case Study Project
March 12, 2021

Turbulent Flow in a Diffuser

Nikhil Sharma

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

Abstract

The objective of this project is to study a turbulent flow in a diffuser using open source CFD package OpenFOAM. A diffuser is a device that reduces the kinetic energy and raises static pressure of the fluid passing through it. In this project, a turbulent flow through an asymmetric 2D diffuser is studied. Two different turbulence models: **k-ϵ and k-ω SST** are used and the results are compared with experimental data.

1. Introduction

Diffusers are regarded as steady-flow engineering device and have been used extensively to alter the flow rate at no extra cost by just widening the outlet cross section of the pipe[1]. Diffusers convert the kinetic energy into potential energy by reducing the velocity and increasing the static pressure of fluid passing through it. Diffusers find common applications in heating, ventilating, and air-conditioning (HVAC) systems and automobiles to improve it's aerodynamic properties.

2. Problem Statement

Primary objective of the project is to simulate turbulent flow inside a diffuser ,validate velocity and turbulent kinetic energy and also observe the effects of different turbulence models. Figure 1 shows the geometry and dimensions of the physical domain. It has three major sections an inlet, an angled expansion channel and an outlet channel. The dimensions of the geometry is taken [4] as L1=60 m, H1=2 m, L2=70 m and H2=9.4 m. Reynolds number is 17,000 based on inlet velocity and inlet dimension (H1). To study this turbulent flow, simpleFoam solver is used.

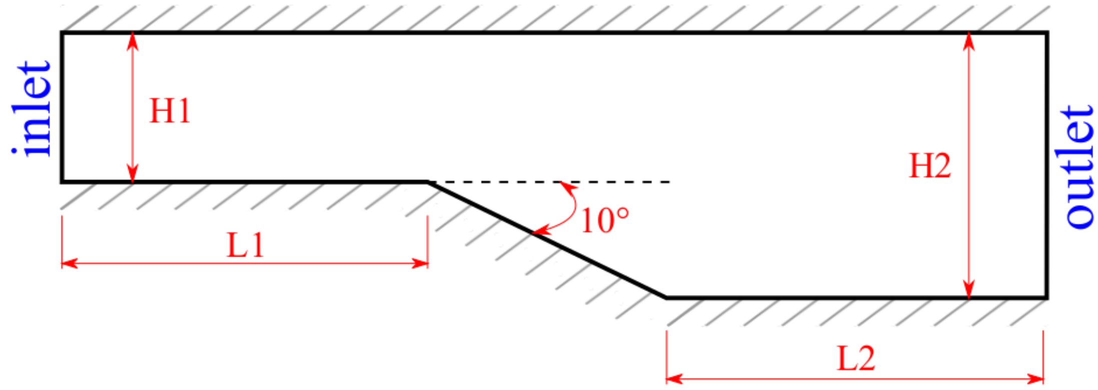


Figure 1: Geometry used for the simulation

3. Governing Equations

Following governing equations are solved by the simpleFoam solver:

Navier Stokes equations

$$\nabla.U = 0 \tag{1}$$

$$\nabla.(U \otimes U) - \nabla.\mathbf{R} = -\nabla p + \mathbf{S}_u \tag{2}$$

Here, **R** is the stress tensor and $\mathbf{S}_u$ is the momentum source.

k-ϵ model equations

$$\frac{D}{Dt}(\rho k) = \nabla.(\rho D_k \nabla k) + P - \rho\epsilon \tag{3}$$

$$\frac{D}{Dt}(\rho\epsilon) = \nabla.(\rho D_\epsilon \nabla\epsilon) + \frac{C_1\epsilon}{k}(P + C_3\frac{2}{3}k\nabla.U) - C_2\rho\frac{\epsilon^2}{k} \tag{4}$$

$$\nu_t = C_\mu\frac{k^2}{\epsilon} \tag{5}$$

Here, D_ϵ is the effective diffusivity for ϵ and ν_t is the turbulent viscosity.
The default value of model coefficients[2]: C_1, C_2, C_3 and C_μ have been used.

k-ω SST model equations

$$\frac{D}{Dt}(\rho k) = \nabla.(\rho D_k \nabla k) + \rho G - \frac{2}{3}(\rho k(\nabla.U)) - \rho\beta^*\omega k + S_k. \tag{6}$$

$$\frac{D}{Dt}(\rho\omega) = \nabla.(\rho D_\omega \nabla\omega) + \frac{\rho\gamma G}{\nu} - \frac{2}{3}(\rho\gamma\omega(\nabla.U)) - \rho\beta\omega^2 - \rho(F_1 - 1)CD_{k\omega} + S_\omega, \tag{7}$$

$$\nu_t = a_1 \frac{k}{\max(a_1\omega, b_1 F_{23} S)} \tag{8}$$

The default values of the model coefficients [3]: α_{k1}, α_{k2}, $\alpha_{\omega1}$, $\alpha_{\omega2}$, β_1, β_2, γ_1, γ_2, β^*, a_1, b_1 and c_1 have been used.

4. Simulation Procedure

OpenFOAM generally requires three folders namely 0, constant and system in the case setup. Simulation is run by typing in the required commands in the terminal. Steady state, incompressible, turubulent solver simpleFoam is used for the simulations. Table 1 below shows the fluid properties and turbulence parameters used in the simulations.

	Unit	Value
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

4.1 Geometry and Mesh

Figure 2 shows the computational domain used which basically is a 2D geometry in x-y plane. Cell thickness in the *z-direction* is one in our mesh and *empty* boundary condition has been implemented on boundary surfaces normal to the *z-axis*. A structured mesh having only hexahedra cells is used. Meshing was done using *blockMesh* utility. The mesh has been sufficiently refined near the walls to resolve the sub viscous layer.

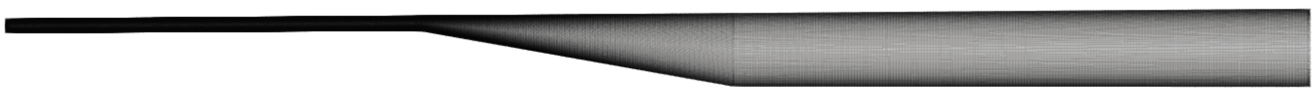


Figure 2: 2D view of Computational Domain

Mesh Sensitivity Analysis

A mesh sensitivity analysis is carried out to obtain most optimal results with minimum number of cells in order to reduce computational cost. Three non-uniform structured meshes were generated with 60,000, 90,000 and 120,000 cells. The meshes produced similar results for static pressure distribution[1]. Figure 3 below shows the variation of static pressure with length of diffuser.

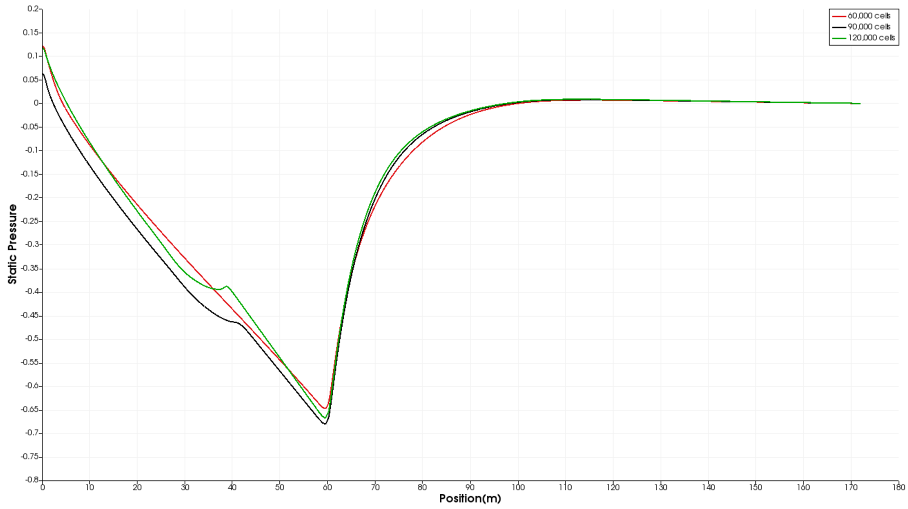


Figure 3: Mesh Sensitivity Analysis

4.2 Boundary Conditions

Boundary Conditions are defined for the following properties:

- **Kinematic Pressure (p , m^2s^{-2})** :

Inlet :	Zero Gradient
Outlet :	Fixed Value type (Uniform zero pressure)
Lower Wall :	Zero Gradient
Upper Wall :	Zero Gradient
Front and Back :	Empty

- **Velocity (U , ms^{-1})** :

Inlet :	Fixed Value Type (1.25 ms^{-1})
Outlet :	Zero Gradient
Lower Wall :	No Slip type
Upper Wall :	No Slip type
Front and Back :	Empty

[1] For doing mesh sensitivity analysis, static pressure 1m below the upper wall has been plotted.

- **Turbulent Kinetic Energy (k , m^2s^{-2})** :

	"turbulentIntensityKineticEnergyInlet" type
Inlet :	Intensity = 3.25%
	Value = 1.8 * 10^{-3} m^2s^{-2}
Outlet :	Zero Gradient
Lower Wall :	Fixed Value Type (Uniform Zero Value)
Upper Wall :	Fixed Value Type (Uniform Zero Value)
Front and Back :	Empty

- **Turbulent kinetic energy dissipation rate (epsilon , m^2s^{-3})** :

	"turbulentMixingLengthDissipationRateInlet" type
Inlet :	Mixing Length = 0.0035 m
	Value = 9.63 * 10^{-5} m^2s^{-3}
Outlet :	Zero Gradient
Lower Wall :	Fixed Value Type (Uniform Zero Value)
Upper Wall :	Fixed Value Type (Uniform Zero Value)
Front and Back :	Empty

- **Turbulence Specific Dissipation Rate (omega , s^{-1})** :

	"turbulentMixingLengthFrequencyInlet" type
Inlet :	Mixing Length = 0.0035 m
	Value = 22.13s^{-1}
Outlet :	Zero Gradient
Lower Wall :	Fixed Value Type (Uniform Zero Value)
Upper Wall :	Fixed Value Type (Uniform Zero Value)
Front and Back :	Empty

- **Kinematic Eddy Viscosity (nut , m^2s^{-1})**[2] :

Inlet :	"calculated" type Value = 0
Outlet :	"calculated" type Value = 0
Lower Wall :	"nutkWallFunction" type Value = 0
Upper Wall :	"nutkWallFunction" type Value = 0
Front and Back :	Empty

[2] This file contains no new information from physical point of view, it is included only because of technical reasons.

4.3 Solver

We need to analyze a steady state turbulent flow, to do so we have used simpleFoam solver. simpleFoam is a steady-state solver for incompressible, turbulent flow. The solver utilizes "Semi-Implicit Method for Pressure-Linked Equations" (SIMPLE) algorithm, which is iterative in nature.

5. Results and Discussions

Steady state simulations were performed using SIMPLE algorithm for both of the turbulence models. The obatined CFD results have been compared with the experimental results of modified velocity and modified turbulent kinetic energy. Post-processing was done using paraview.

5.1 Result 1: Modified Velocity (10*U + X-60)

Figures 4 and 5 show the plot of modified velocity obtained using k-ϵ and k-ω SST models respectively. From the plots it is seen that the k ϵ model gives close results near the region where expansion just starts but poorly predicts the flow further downstream of the flow. The SST model is observed to predict velocity close to experimental values velocities in the recirculating region and near the walls, but over predicts the flow in the region where re-circulation is not dominant.

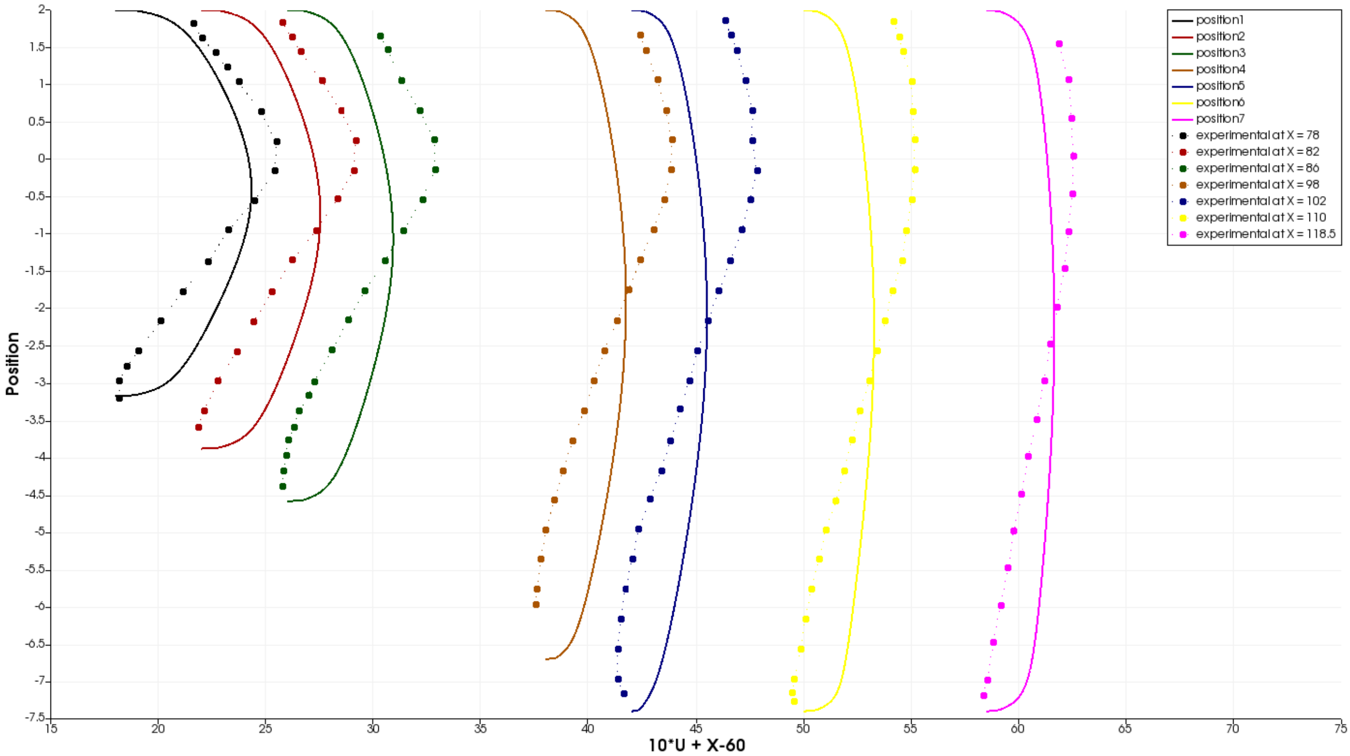


Figure 4: Plot of modified velocity obtained using k-ϵ model

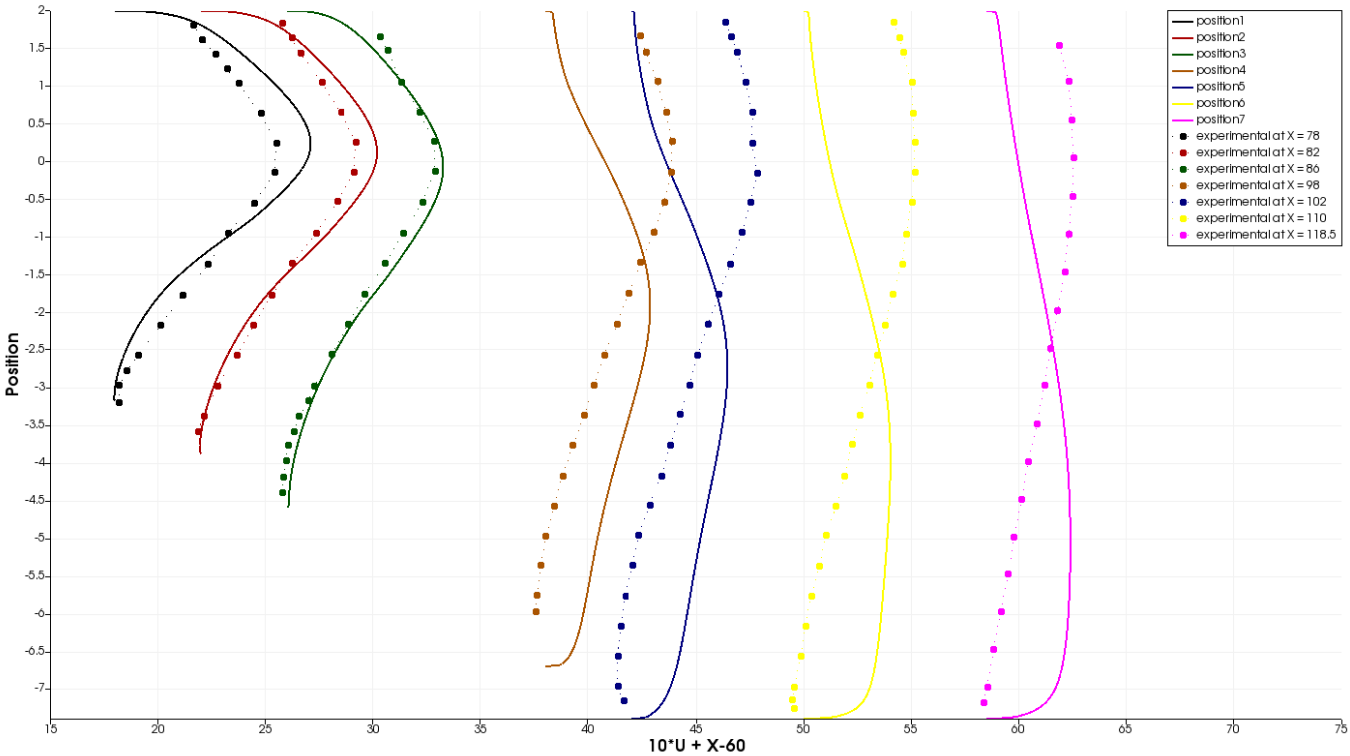


Figure 5: Plot of modified velocity obtained using k-ω SST model

5.2 Result 2: Modified Turbulent Kinetic Energy (500*K + X-60)

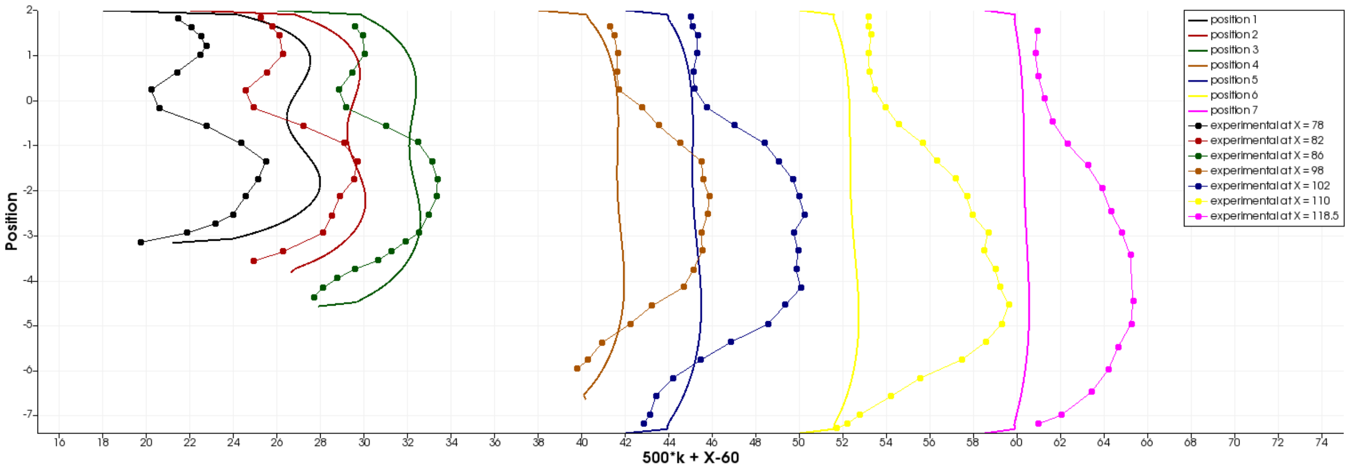


Figure 6: Plot of modified TKE obtained using k-ϵ model

Figures 6 and 7 show the plot of modified TKE obtained using k-ϵ and k-ω SST models respectively. From the plots it is seen that the k-ϵ model poorly predicts TKE at expanding region of diffuser but gives good prediction further downstream of the flow. The SST model is observed to highly over predict TKE at recirculating region. It gives close to experimental results at the walls.

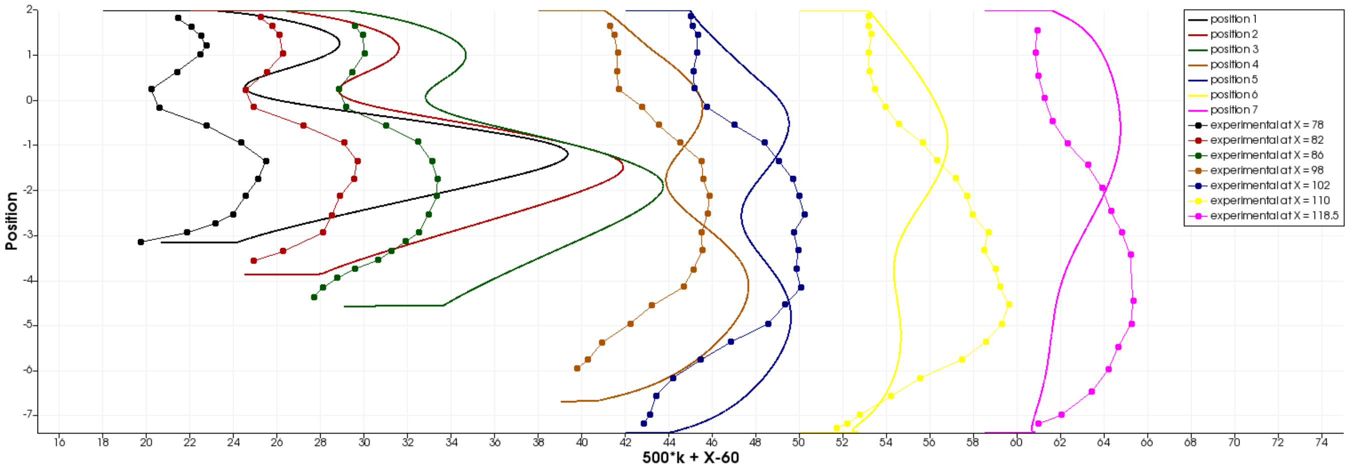
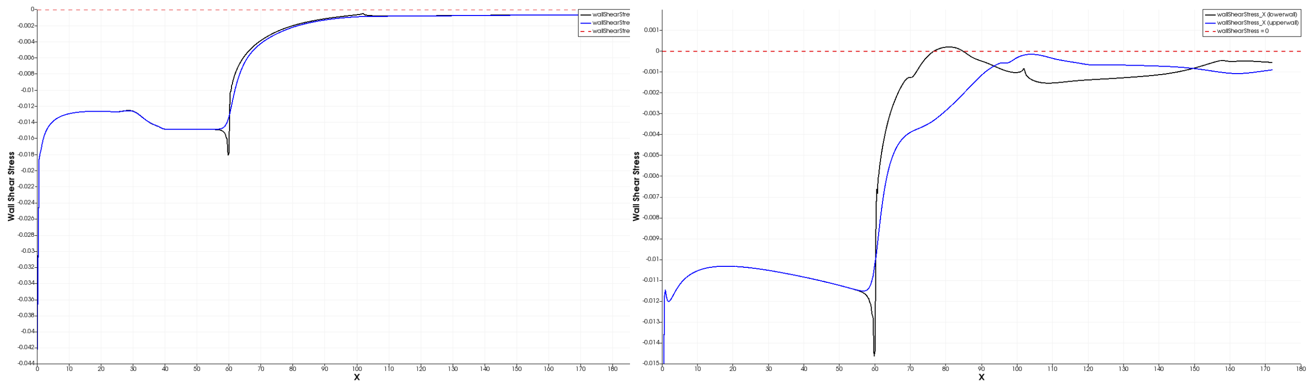


Figure 7: Plot of modified TKE obtained using k-ω SST model

5.3 Result 3: Shear Stress at the walls

Figure 8 shows the variation of shear stress at the lower and upper walls of he diffuser obtained using the two models. The flow either separates or reattaches at the location where shear stress is zero. The flow is observed to separate if the variation of shear stress is from negative to positive and is observed to reattach when the variation is opposite.
From figure 8a, it is seen that no flow separation is observed using k-ϵ model.
From figure8b, we observe the flow to separate from the lower wall at X = 76.36m and reattaches at X = 84.54m. Also it is seen that there is no flow separation at the top wall. The separation and reattachment can also be seen in the streamlines as shown in figure 9.



(a) using k-ϵ model (b) using k-ω SST model

Figure 8: Variation of shear stress

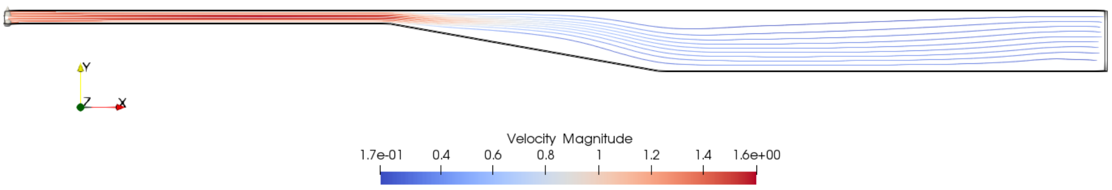


Figure 9: Streamlines with velocity scale obtained using k-ω SST model

Conclusions

In this this project, we simulated turbulent flow through a diffuser using two different turbulence models. The performance of the different turbulence models is compared with each other and experimental results. The k - ω SST model was observed to over predict the flow in most of the region but it was found to give highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients. The standard k - ϵ model used in this case was found to underpredict most of the flow properties. To predict flow separation accurately using k-epsilon model; other k-epsilon models like RealizableKE, RNG-KE can be compared.

It can be concluded from this project that k ω SST model predicts the flow better in the near wall region, the k - ϵ model was found to predict flow better in the far from wall region.

References

[1] R. S. Azad. Turbulent flow in a conical diffuser: A review. *Experimental Thermal and Fluid Science*, 13(4):318–337, 1996.

[2] D. Laurence, J. Uribe, and S. Utyuzhnikov. A robust formulation of the v2- f model. *Flow, Turbulence and Combustion*, 73(3):169–185, 2005.

[3] F. R. Menter, M. Kuntz, and R. Langtry. Ten years of industrial experience with the sst turbulence model. *Turbulence, heat and mass transfer*, 4(1):625–632, 2003.

[4] M. M. T. X. F. S. Timur Dogan, Michael Conger. Simulation of turbulent flow in an asymmetric diffuser.