LES Simulations of turbulent flow in a branched channel

Rozy Kumari

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

Guide: Dr. Manaswita Bose

Professor, Indian Institute of Technology Bombay

Mentor: Mr. Manjil Sitoula

FOSSEE IIT Bombay

Abstract

The aim of this study is to simulate turbulent flow through a branched channel using the Large Eddy Simulation method as implemented on OpenFOAM. The simulation process encompasses geometry, mesh generation, solver setup and post processing. The dynamicKEqn is used as the sub grid scale model. Simulations are performed for Reynolds number 4000. Profiles of the mean and the rms velocities are obtained and are compared against the results published by Kim (1987).

1. Problem Statement

The objective of the project is to simulate flow through a branched channel with dimensions (0.08 m×0.04 m×0.01m). Figure 2 shows the schematic of the geometry. The dimension of the main duct is selected such that the flow develops before it separates into two branches. The entry length is determined based on (L=10D).

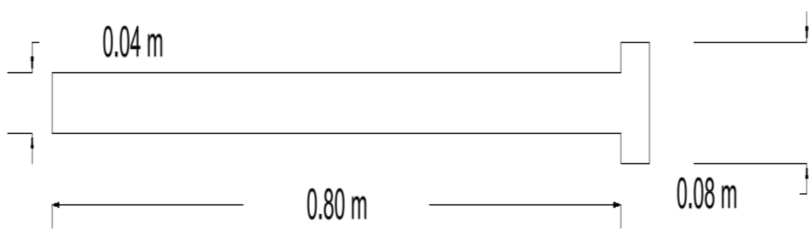
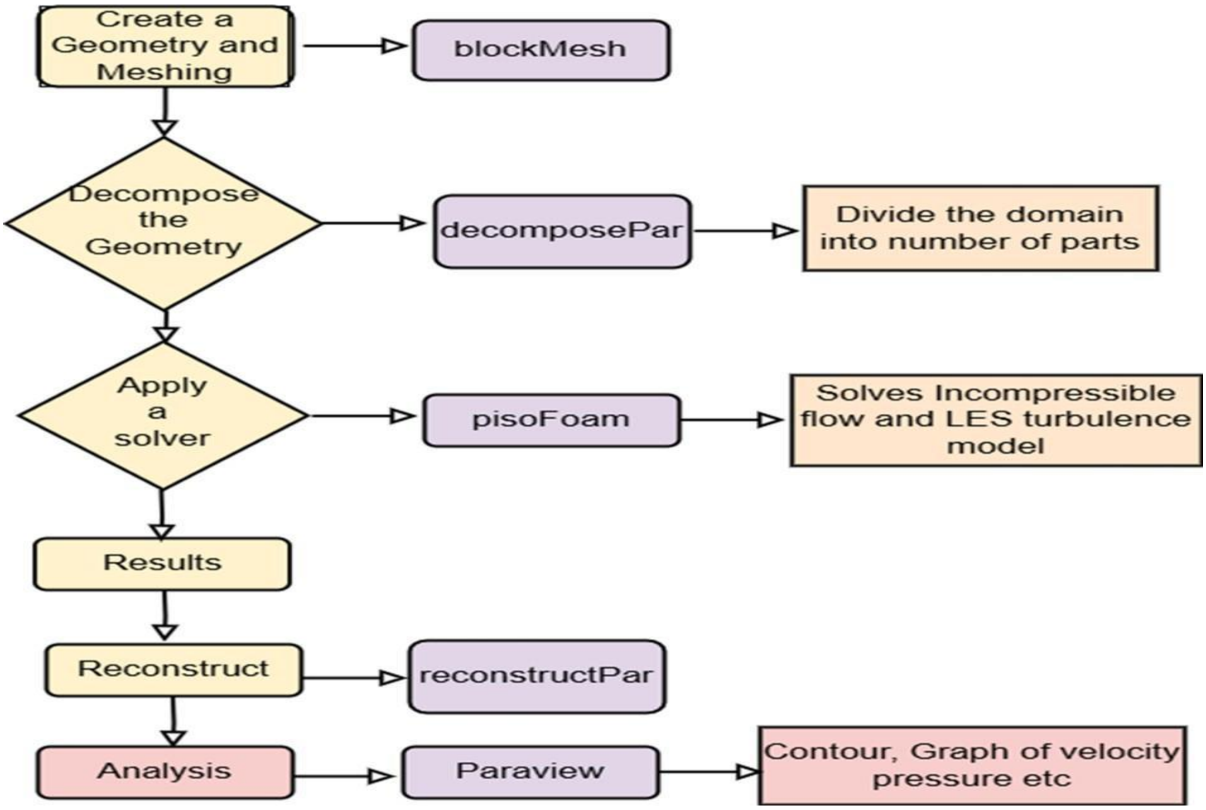


Fig 2 Represent the line diagram of T -channel

2. Simulation Procedure



2.1 Geometry and Mesh

Table 1 Represent the dimension used in T -channel

Parameter	Value
Length of main channel	0.8 m
Length of branched channel	0.08 m
Width of channels	0.04 m
Depth	0.01 m
Hydraulic diameter	0.016 m
Reynolds number	4000

Figure 2 represents the geometry of the T channel having the length of 0.04 m, width is 0.04 m and branched having dimension 0.08 and width 0.04 and depth is kept same throughout the channel. The geometry is created by using blockMesh.

(a)

(b)

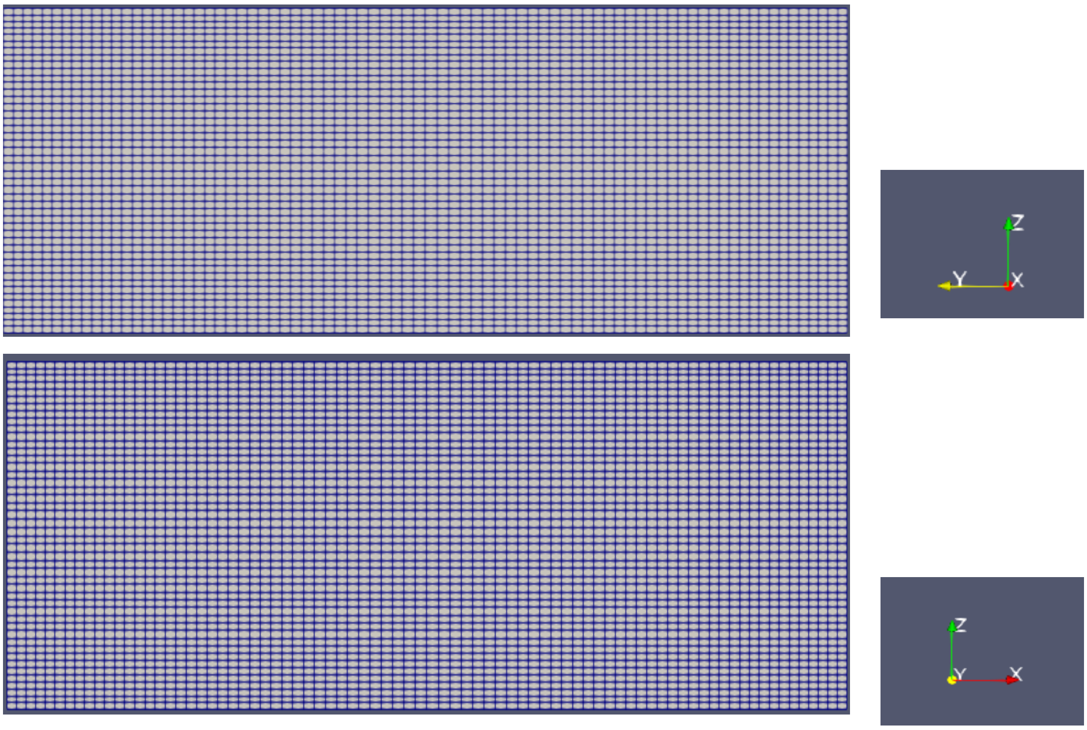


Fig 3 Meshing at the (a) Inlet (b) Outlet

figure 3 (a) and 3 (b) represents the meshing at the inlet and outlet with the cell expansion 0.83 near the sidewall, bottom wall and top wall.

2.2 Solver

The pisoFoam solver is used as it is a transient, incompressible flow solver in OpenFOAM that uses the PISO (Pressure Implicit with Splitting of Operators) algorithm. It is capable of handling Large Eddy Simulaltion (LES), and set the turbulence model and mesh resolution appropriately, dynamicKEqn LES model is used in this study. The model was run upto 3.05 s with the 0.00001s

2.3 Initial and Boundary Conditions

Initial and boundary conditions used in this study is given in table2.

Table 2: Initial and Boundary Conditon

Setup			
Solver type		PisoFoam	
Gravitational acceleration		X=0, Y=0, Z=-9.81 m/s^2	
Patch		**Condition**	
Inlet		Zero Gradient	
Outlet		Fixed Value	
Top		Fixed Value	
Bottom		Fixed Value	
Front		Fixed Value	
Back		Fixed Value	
Turbulence model			
LES		dynamicKEqn LES model	
Material		Air	
Initial Conditions			
inlet		zeroGradient	
outlet		fixedValue	
top|bottom		fixedValue	
frontAndBack		fixedValue	
Boundary conditions for P		**Boundary Conditions for U**	
inlet	Fixed Value (3.75m/s)	inlet	zeroGradient
outlet	InletOutlet	outlet	fixedValue
top|bottom	fixedValue	top|bottom	fixedValue
frontAndBack	fixedValue	frontAndBack	fixedValue

4.Results and Discussion

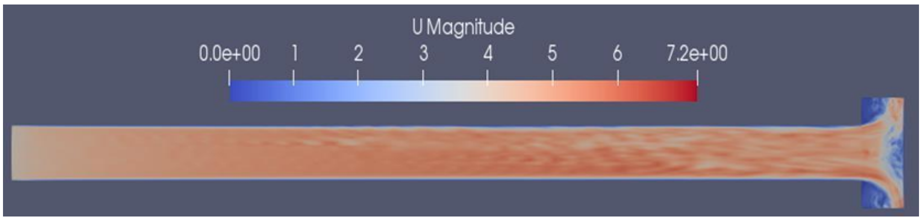


Fig 4 velocity along the length

Figure 4 represents the velocity along the length here it can be easily seen that velocity near the side wall is zero. Maximum velocity reaches up to 7.2 m/s which is found mid of the channel. As the air move to the junction, it separates in two direction and bend towards the both the outlets. Figure 5 (a) and (b) represents the mean velocity and vorticity near the end of channel and at the branched channel.

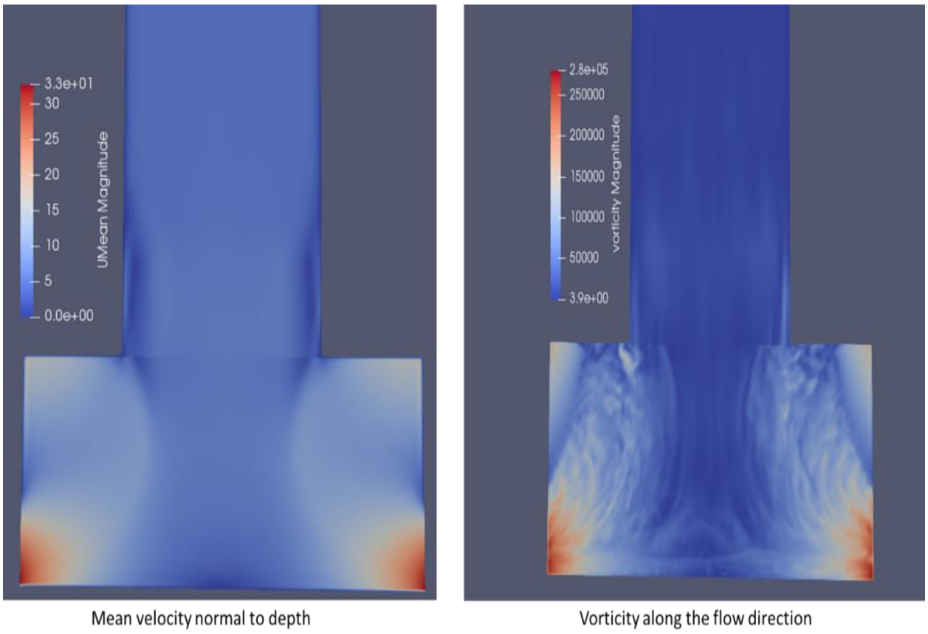


Fig 5 Representation of velocity and vortex

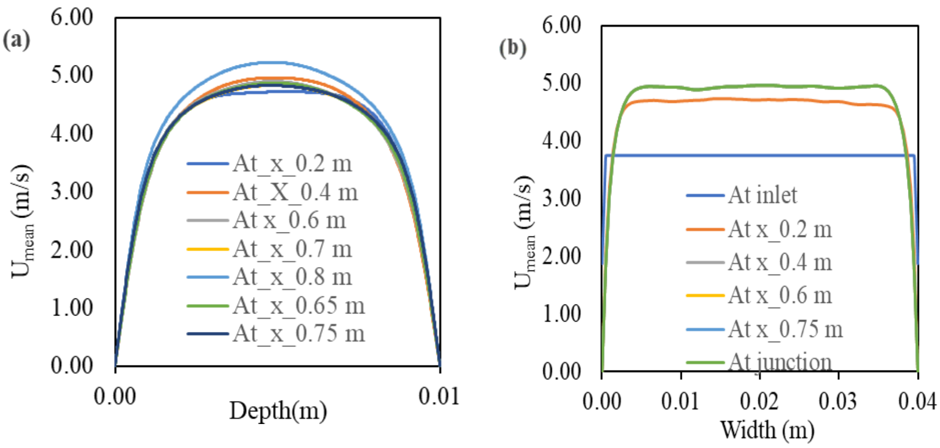


Fig 6 Representation of velocity at different location

Figure 6 represents the mean velocity at different location of the channel. Figure 6 a represents the variation of velocity with depth at different location of length. Results indicate that after 0.3 m length there is a development of flow and maximum velocity reach up to 5 m/s at mid of the depth and it is followed the same at different location of length. location of length of branched channel with respect to depth and width.

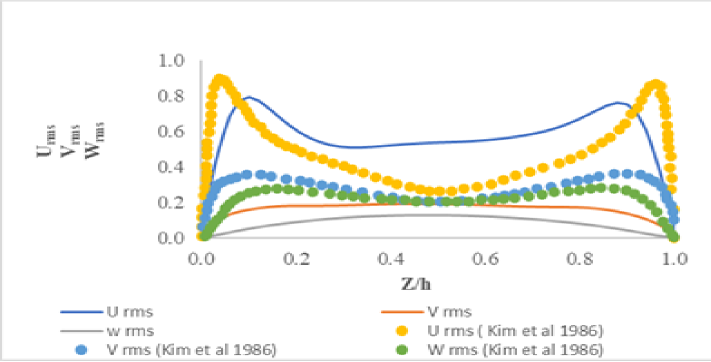


Fig 7 Root mean square velocity fluctuations normalized by wall shear velocity

Figure 7 represents the root mean square velocity fluctuations normalized by wall shear velocity. The results are qualitatively similar to the results of Kim et al. (1987). The simulated values from both models exhibit root mean square for U_{rms}, V_{rms} and W_{rms} is 0.58, 0.13 and 0.148 respectively.

Acknowledgement

I would like to extends my deepest gratitude to Dr. Manaswita Bose (Professor, Indian Institute of Technology Bombay), Mr. Sourav Ganguli, PhD research scholar (IIT Bombay department of Energy Science and Engineering) and Mr. Manjil Sitoula (Research Assistant (FOSSEE), Indian Institute of Technology Bombay) and for their relentless support and valuable critiques throughout this study. Their attention to detail and academic prowess made my experience of learning and working very insightful and exemplary.

References

Gurumurthy, A., & Kleinstreuer, C. (2021). Helical fluid-particle flow dynamics for controlling micron-particle deposition in a representative human upper lung-airway model. *Journal of Aerosol Science*, *151*, 105656.

Kim, W. W., & Menon, S. (1995, January). A new dynamic one-equation subgrid-scale model for large eddy simulations. In *33rd aerospace sciences meeting and exhibit* (p. 356).

Kim, J., Moin, P., & Moser, R. (1987). Turbulence statistics in fully developed channel flow at low Reynolds number. *Journal of fluid mechanics*, *177*, 133-166.

Kloss, C., Goniva, C., Hager, A., Amberger, S., & Pirker, S. (2012). Models, algorithms and validation for opensource DEM and CFD–DEM. *Progress in Computational Fluid Dynamics, an International Journal*, *12*(2-3), 140-152.

Le, T. C., & Tsai, C. J. (2021). Inertial impaction technique for the classification of particulate matters and nanoparticles: A review. *KONA Powder and Particle Journal*, *38*, 42- 63.

Lu, H., Han, Z., Li, H., Chang, X., Dong, L., Fan, M., ... & Jing, X. (2023). Simulation of turbulent flow structure and particle deposition in a three-dimensional heat transfer duct with convex dimples. *Coatings*, *13*(5), 900.

Marple, V. A. (2004). History of impactors—the first 110 years. *Aerosol Science and Technology*, *38*(3).

Kumari, R., Das, B. S., Devi, K., Khuntia, J. R., & Mohanty, M. P. (2024). Hydrodynamic performance of raceway pond using k-ω and LES turbulence models. *Journal of Renewable and Sustainable Energy*, *16*(6)