



Synopsis

Divyesh Variya
FOSSEE Team, IIT Bombay

Prediction of Turbulence in Separated Flow using Asymmetric Diffuser Geometry

This research migration project aims to do numerical simulations of the turbulent flow in an asymmetric two-dimensional diffuser using `OpenFOAM foamExtend-4.1`. The geometry and mesh were defined using `blockMesh` utility. A steady-state, SIMPLE algorithm-based `simpleFoam` solver was used to simulate the problem. For accurate turbulence predictions, various models of the $\kappa - \epsilon$ turbulence family were used and compared with the experimental data. The analysis executed by Gianluca Iaccarino [1] using commercial CFD code `CFX`, `Fluent`, and `Star-CD` is taken as a reference.

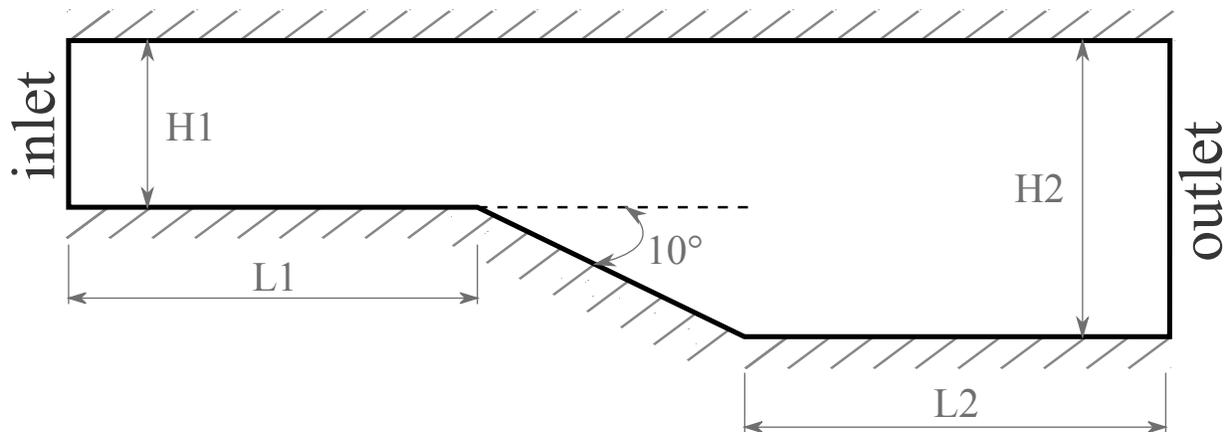


Figure 1: Geometry and Dimensions

The dimensions of the geometry stated in the figure 1 are: $L1=60$ 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 discussed in the report.

References

- [1] Gianluca Iaccarino. “Predictions of a Turbulent Separated Flow Using Commercial CFD Codes”. In: *Journal of Fluids Engineering* 123.4 (May 2001), pp. 819–828. ISSN: 0098-2202. DOI: 10.1115/1.1400749. URL: <https://doi.org/10.1115/1.1400749>.