

Numerical Analysis of Flow Past a Blunt Heated Cylinder using OpenFOAM

Submitted by –

Mr. Mohammed Saqlain

UG Student, Department of Mechanical Engineering
The National Institute of Engineering, Mysore 570008

Under the guidance of –

Dr. Mrinal K R

Assistant professor, Department of Mechanical Engineering
The National Institute of Engineering, Mysore 570008

Abstract –

This study presents a comprehensive transient numerical analysis of the flow and heat transfer characteristics of flow past a 2D blunt heated cylinder using the open-source computational fluid dynamics (CFD) software OpenFOAM. This study focuses on the numerical analysis of flow over a blunt heated cylinder and investigates its heat transfer characteristics using chtMultiRegionFoam in openFOAM. A range of Reynolds numbers $10,000 < Re < 100,000$ was considered in the simulation. The analysis involved examining various parameters such as lift and drag coefficients, Nusselt number, Strouhal number, and the variation of pressure, temperature, and velocity contours. The results revealed interesting trends and phenomena. The lift and drag coefficients exhibited fluctuations at higher Reynolds numbers, indicating flow unsteadiness and turbulence effects. The Nusselt number provided insights into the heat transfer between the cylinder and the fluid stream, while the Strouhal number highlighted the shedding of vortices in the wake region. The pressure, temperature, and velocity contours offered a visual representation of the flow characteristics. Overall, this study enhances our understanding of the flow behaviour and heat transfer in the context of a blunt heated cylinder, contributing to the broader knowledge in fluid dynamics and thermal sciences.

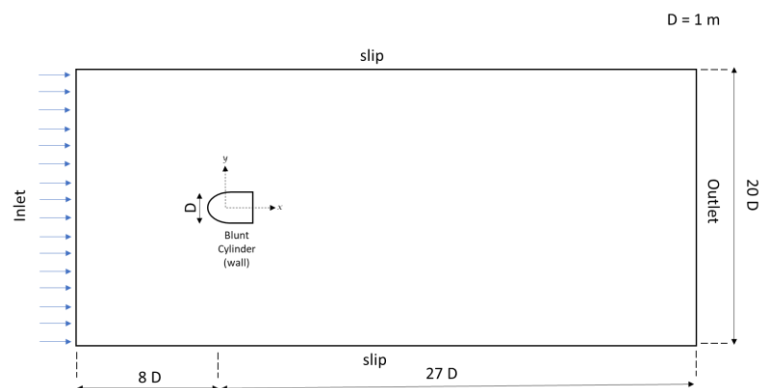


Figure 1 - Physical domain for present study