FOSSEE, IIT Bombay
OpenFOAM Research Migration Project
June 23, 2024

Synopsis

Evan Fernandes

Department of Aerospace Engineering, Amity University Mumbai

Low Reynolds Number Flow Over Non-Rectangular Bluff Bodies

This study presents a comprehensive transient numerical analysis of the flow around a 2D bluff body using OpenFOAM, a robust open-source computational fluid dynamics (CFD) software. The investigation focuses on understanding the behaviour of low Reynolds number flows over a bluff body, specifically using the PISOFoam solver within OpenFOAM. The accuracy of the solver was initially verified against established research findings on rectangular bluff bodies operating at low Reynolds numbers. The study systematically explores the influence of angle of attack (AOA) on flow characteristics, conducted at a Reynolds number of 150. Multiple angles of attack ranging from 0° to 30° were simulated to capture the diverse aerodynamic responses of the bluff body. Key parameters such as lift coefficient (Cl), drag coefficient (Cd), Strouhal number, and velocity contours were meticulously analysed to uncover insightful trends and phenomena. Results revealed distinct patterns in the variation of lift and drag coefficients across different AOAs, highlighting the complex interaction between flow separation, vortex shedding, and wake dynamics. The fluctuating nature of these coefficients underscored the unsteady nature of the flow, influenced significantly by the changing AOA. Notably, the Strouhal number computations vividly illustrated the periodic shedding of vortices in the wake region, emphasizing the body's aerodynamic instability. Visual representations through pressure and velocity contours provided a clear depiction of flow structures and boundary layer characteristics, offering valuable insights into flow behaviour around bluff bodies. These contours served as essential tools in identifying regions of high and low-pressure gradients, thereby enhancing the understanding of flow separation and reattachment phenomena. Overall, this study contributes significantly to advancing our comprehension of bluff body aerodynamics using CFD. By exploring a range of AOAs and rigorously analyzing key parameters, the research deepens insights into flow behaviour under low Reynolds number conditions. The findings not only validate the computational approach but also enrich the broader knowledge of flow physics, benefiting applications in aerodynamic design, wind engineering, and vehicle performance optimization.

References

[1] Yuce M, Kareem D, "A Numerical Analysis of Fluid Flow Around Circular And Square Cylinders". In *American Water Works Association* (2016). DOI: 108(10) E546-E554

[2] Chatterjee D, Gupta S, "Numerical Study Of The Laminar Flow Past A Rotating Square Cylinder At Low Spinning Rates". *In Journal of Fluids Engineering, Transactions of the ASME* (2015). DOI: 10.1115/1.4028500