FOSSEE, IIT Bombay
OpenFOAM Research Migration Project
August 2024

# Flow Around a Cylinder: A Comparative Study of Immersed Boundary and Arbitrary Lagrangian–Eulerian (ALE) Methods in OpenFOAM

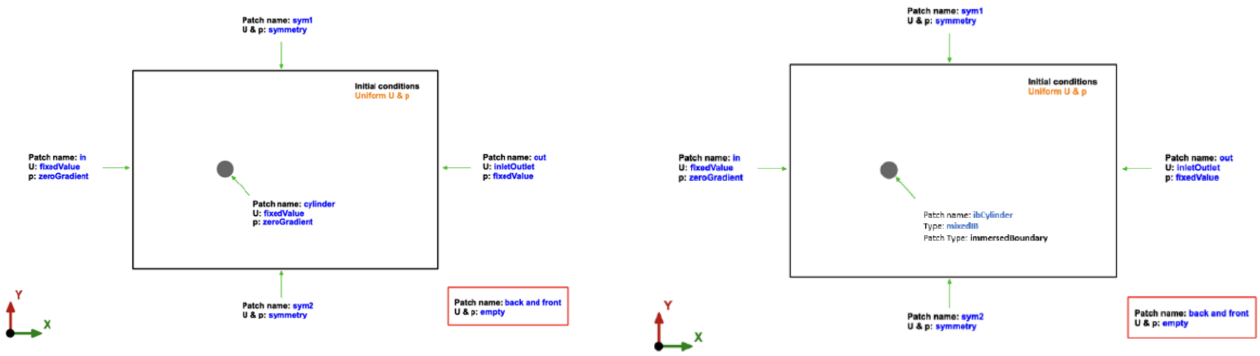
Antim Gupta[1], Chandan Bose[2]

[1] Faculty of Technology, Design and Environment, Oxford Brookes University, Oxford, OX33 1HX, United Kingdom

[2] Assistant Professor, Aerospace Engineering, College of Engineering and Physical Sciences, The University of Birmingham

## Synopsis

Understanding the flow physics around cylindrical objects is a fundamental problem in fluid dynamics with numerous engineering applications, including aerospace, civil, and mechanical engineering sector. Accurate simulation of such flows is crucial for designing efficient structures and devices subjected to fluid forces, as it directly impacts their aerodynamic performance, safety, and durability. Predicting vortex-induced vibrations and drag forces on cylindrical structures can lead to better designs that mitigate structural fatigue and failure caused by resonance frequency. In this study, a comparative analysis of flow around a cylinder using the Immersed Boundary Method (IBM) and the Arbitrary Lagrangian-Eulerian (ALE) approach within the OpenFOAM® (FOAM-Extend-4.1) computational fluid dynamics framework has been conducted. IBM simplifies mesh generation by utilizing a fixed Cartesian grid, whereas ALE accommodates dynamic mesh deformation. A performed 2D transient numerical simulations has been performed for Reynolds numbers (Re = 100, 200, 1000, 2000) to evaluate the accuracy, efficiency, and robustness of both methods in capturing key flow characteristics such as vortex shedding, lift, and drag coefficients. Both static and oscillating cylinder flows were studied. Initially, mesh sensitivity analysis was conducted using Richardson extrapolation with a refinement factor of 2 and was validated against existing literature for Re = 200 [1], [2], [3], [4], [5]. The results demonstrate that both methods can accurately simulate flow around a cylinder, though they differ in computational efficiency and ease of implementation. IBM required significantly more computational time to achieve result convergence compared to ALE. For static cylinders, ALE studies included IcoFoam-laminar, PimpleFoam-laminar, and PimpleFoam with the k-omega SST turbulence model. The laminar models were effective for low Reynolds numbers (≤200), producing a drag coefficient (Cd) of 1.4 for Re = 100 and 200 using IcoFoam, and 1.38 and 1.375 using PimpleFoam-laminar. The k-omega SST model estimated Cd values of 1.37 and 1.29 for Re = 1000 and 2000, respectively. Using IBM with the IcoFoam solver, Cd values were 1.5, 1.44, 1.287, and 1.143 for Re = 100, 200, 1000, and 2000, respectively. Studies on oscillating cylinder flows, considering a 0.1D amplitude in the y-axis at a frequency of 1Hz, are also presented and compared for IBM and ALE at Re = 100, alongside time-dependent and grid sensitivity tests for IBM. These findings provide insights into the strengths and limitations of IBM and ALE methods, emphasizing trade-offs between computational efficiency and implementation complexity.




(a) (b)

Figure 1: Computational domain chosen for CFD studies representing the Initial and Boundary conditions for a) ALE model and, b) IBM model.

## References

[1] D. Russell and Z. J. Wang, “A cartesian grid method for modeling multiple moving objects in 2D incompressible viscous flow,” J Comput Phys, vol. 191, no. 1, pp. 177–205, Oct. 2003, doi: 10.1016/S0021-9991(03)00310-3.

[2] C. Liu, X. Zheng, and C. H. Sung, “Preconditioned Multigrid Methods for Unsteady Incompressible Flows,” J Comput Phys, vol. 139, no. 1, pp. 35–57, Jan. 1998, doi: 10.1006/JCPH.1997.5859.

[3] J. Il Choi, R. C. Oberoi, J. R. Edwards, and J. A. Rosati, “An immersed boundary method for complex incompressible flows,” J Comput Phys, vol. 224, no. 2, pp. 757–784, Jun. 2007, doi: 10.1016/J.JCP.2006.10.032.

[4] D. Calhoun, “A Cartesian Grid Method for Solving the Two-Dimensional Streamfunction-Vorticity Equations in Irregular Regions,” J Comput Phys, vol. 176, no. 2, pp. 231–275, Mar. 2002, doi: 10.1006/JCPH.2001.6970.

[5] M. Braza, P. Chassaing, and H. Ha Minh, “Numerical study and physical analysis of the pressure and velocity fields in the near wake of a circular cylinder,” J Fluid Mech, vol. 165, pp. 79–130, 1986, doi: 10.1017/S0022112086003014.