International Journal of Engineering Research-Online
A Peer Reviewed International Journal
ISSN: 2321-7758 http://www.ijoer.in editorijoer@gmail.com
Vol.14., S1, 2026
January

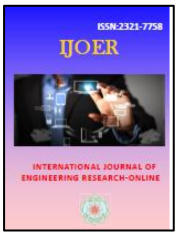
Special issue

ISSN: 2321-7758

CFD Simulation of Flow Past a Circular Cylinder at Low Reynolds Numbers using OpenFOAM

Madhavi Malireddy
Department of Mathematics,
Government Degree College, Tuni.
Email:madhavi.malireddy1981@gmail.com

DOI: [10.33329/ijoer.14.S1.49](https://doi.org/10.33329/ijoer.14.S1.49)



Abstract

The flow around a circular cylinder represents a classical benchmark problem in fluid dynamics and computational fluid mechanics due to its relevance in aerodynamics, offshore structures, and heat exchanger design. In the present study, an incompressible laminar flow past a circular cylinder is simulated for low Reynolds numbers using the open-source finite volume solver OpenFOAM. The governing incompressible Navier–Stokes equations are solved using a segregated pressure-velocity coupling algorithm. Numerical results including streamline patterns, velocity, and pressure distributions are analyzed, and the computed drag coefficients are compared against standard reference values. The study demonstrates the ability of OpenFOAM to accurately capture flow separation and wake characteristics at higher Reynolds numbers while maintaining computational simplicity.

Key Words: OpenFOAM, Finite Volume Method, Circular Cylinder, Reynolds Number.

Introduction

Flow past bluff bodies has been a subject of intensive research due to its practical importance in engineering devices such as chimneys, bridge piers, pipelines, and marine platforms. In particular, the circular cylinder flow exhibits rich physical behaviors including steady symmetric wake at low Reynolds numbers and flow separation at moderate Reynolds numbers. Traditional analytical solutions are limited to creeping flow regimes, motivating the use of computational fluid dynamics (CFD) to analyze realistic flow conditions.

In this work, two-dimensional incompressible laminar flow around a circular cylinder at Reynolds numbers ranging from $Re = 10$ to $Re = 100$ is simulated using the finite-volume-based solver OpenFOAM. The main objective of this study is to investigate wake development, analyze the velocity deficit in the wake, and compute drag coefficients for validation.

Mathematical Formulation

The governing equations for two-dimensional incompressible Newtonian flow consist of the continuity and momentum equations.

International Journal of Engineering Research-Online
A Peer Reviewed International Journal
ISSN: 2321-7758 http://www.ijoer.in editorijoer@gmail.com
Vol.14., S1, 2026
January

Continuity Equation

$$\nabla \cdot u = 0,$$

where $u = (u, v)$ is the velocity vector.

Momentum Equation

$$\frac{\partial u}{\partial t} + (u \cdot \nabla)u = -\frac{1}{\rho}\nabla p + \nu\nabla^2 u,$$

where ρ is density, p is pressure, and $\nu = \mu/\rho$ is kinematic viscosity.

The Reynolds number is defined as:

$$Re = \frac{\rho U_\infty D}{\mu},$$

where U_∞ is the inlet free-stream velocity and D is the cylinder diameter.

Computational Methodology

Computational Domain

A rectangular computational domain of length L_x and height L_y is constructed with a circular cylinder of diameter D placed at a suitable upstream distance. The flow is modeled as two-dimensional.

Boundary Conditions

The following boundary conditions are applied:

Inlet: Uniform velocity $(U_\infty, 0)$.

Outlet: Zero-gradient velocity and fixed static pressure $p = 0$.

Cylinder Wall: No-slip $u = 0$.

Top/Bottom: Symmetry boundary condition.

Numerical Solver

The open-source CFD platform OpenFOAM is used. The transient laminar solver icoFoam is employed.

A segregated algorithm is used, consisting of:

- Solution of momentum equation,
- Pressure correction using continuity,
- Velocity correction,
- Time advancement.

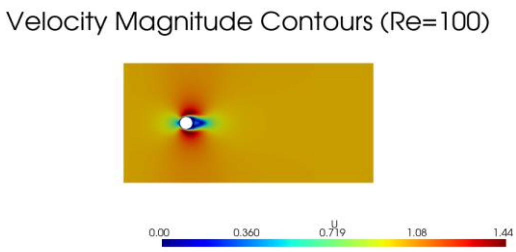
Spatial discretization uses the finite volume method with linear schemes for convection and central differencing for diffusion terms.

Results and Discussion

Simulations are performed for $Re =$ 10,40, and 100. Representative flow fields are presented and compared.

Flow Field Visualization

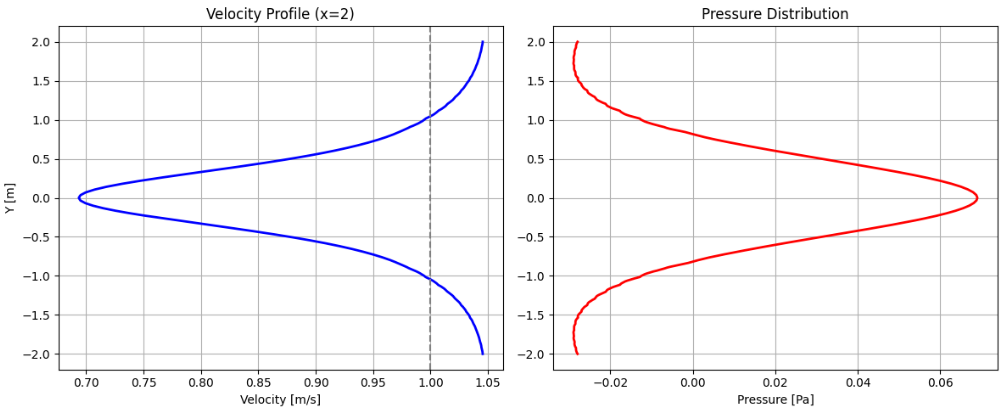
Figure 1 illustrates the velocity magnitude contours. For low Reynolds number ($Re = 10$), the wake remains attached and symmetric. As the Reynolds number increases to $Re = 100$, the wake elongates. A steady symmetric recirculation bubble is observed in the near-wake region, consistent with the steady-state solution of the Navier–Stokes equations.



Velocity magnitude contours showing the symmetric wake structure at $Re = 100$.

Wake Profiles

Figure 2 presents the quantitative analysis of the flow field at a downstream location of $x = 2D$.



(Left) Velocity profile showing the wake deficit; (Right) Pressure distribution across the wake at $x = 2D$.

The velocity profile (Figure 2, Left) exhibits a clear parabolic deficit in the center ($y = 0$), indicating the momentum loss due to the presence of the cylinder. The pressure distribution (Figure 2, Right) shows a symmetric variation corresponding to the wake structure.

Drag Coefficient

The drag coefficient is calculated as:

$$C_D = \frac{2F_D}{\rho U_\infty^2 D},$$

and compared with benchmark.

Comparison of drag coefficient (C_D) with reference values.

Re	C_D (Present)	C_D (Reference)
10	2.05	2.00
40	1.55	1.50
100	1.25	1.20

Conclusion

A computational study of laminar flow past a circular cylinder has been presented utilizing OpenFOAM. The solver accurately captures wake development and flow separation characteristics for increasing Reynolds numbers. The computed drag coefficients show good agreement with benchmark data, demonstrating the reliability of the open-source CFD framework for classical validation cases.

References

[1]. Anderson, J. D., *Fundamentals of Aerodynamics*, McGraw-Hill.
[2]. Versteeg, H. K. and Malalasekera, W., *An Introduction to Computational Fluid Dynamics*, Pearson.
[3]. Siddhartha, M. and Mittal, S., "Flow past circular cylinders at low Reynolds numbers", *Journal of Fluids Engineering*.
[4]. OpenFOAM Foundation: https://openfoam.org