Numerical Investigation of Separated Flow past Slotted Circular Cylinder at Critical Reynolds Number in Laminar Regime

Bishwajit Sharma¹, Rabindra Nath Barman¹*
¹National Institute of Technology Durgapur, West Bengal, India - 713209

Abstract. The numerical investigation of steady two-dimensional fluid flow past a slotted circular cylinder is carried out at the critical Reynolds number (Re) in the laminar regime. Based on diameter (D) of the cylinder and free stream velocity the critical Re is considered to be 50. The blockage by the cylinder is kept as 0.01 of total channel height. Three different types of slits are present axisymmetric with the horizontal axis of cylinder i.e. (1) Converging, (2) Diverging, and (3) Parallel. The converging and diverging slit width is kept as 0.1D and 0.05D for wide and narrow end respectively and 0.1D for the parallel slits. The upstream and downstream length of the channel from the center of the cylinder is 50D and 200D respectively with the channel height of 100D. The cylinder, slit and channel walls have no-slip boundary condition with uniform velocity at the inlet. A finite volume solver is used to simulate the aforementioned cases. Bubble length, separation angle and drag coefficients are reported to examine the effectiveness of self-propelled jets by the slits in the vicinity of the cylinder. The drag coefficient increases with the incorporation of slits in critical Re. The increase in friction drag due in slotted cylinder acting as the dominant role in the overall rise in drag coefficient.

1. Introduction

Flow past cylinder is a basic and classical example of bluff body flow but yet intricate. A very wide application with enriched physics has attracted researchers for many decades to explore the downstream wake formation past the cylinder. As Reynolds number increases a series of complex sequence of events appear. The series of flow behaviour was experimentally shown from Re 0.1 to 2000 with the help of aluminium dust suspension and water [1]. The twin vortices seemed to produce at Re 5 and could be accurately measured exceeding Re 7. The vortices start to shed the Karman Street above Re 45 with a long tail of attached twin vortices. Using the bend in quartz fibers the drag coefficients can be found out for a wide range of the Re [2]. Numerical solutions of incompressible Navier-Stokes equations were solved to visualize by using finite difference approximations [3] [4] [5] [6]. A semi-analytical method was also developed to provide an accurate and efficient solution of flow around a cylinder at low Re [7]. These investigations also incorporated the different boundary conditions for channel wall and blockage ratio of the flow domain [8] Some of the recent work on the flow past slotted cylinder [9] attracted the attention of many researchers to employ this passive flow control technique to manipulate the downstream wakes. Heat transfer analysis [10] was also carried out but these studies were limited to the flow with higher Reynolds number [11]. Low Re applications can be seen in the chemical industries, the flow of highly viscous fluid, lava flow, flow past very small bluff body etc. No past work is available...
which can effectively comment on the critical Reynolds number in the laminar zone for a slotted cylinder. This paper highlights the effect of incorporating slits inside a circular cylinder at critical Reynolds number (Re 50) with three different slit geometries i.e. converging, parallel, & diverging.

2. Methodology

2.1. Problem Setup
The two-dimensional rectangular domain is represented in the schematic diagram shown in Figure 1. The circular cylinder with diameter D is placed axisymmetrically with respect to the horizontal axis and the origin lies at the center of the cylinder. The cylinder, slit, top and bottom edges of the flow domain is a wall with no-slip boundary condition. The width H of the rectangular channel is taken as 100D. The cylinder is placed at an upstream distance L_u of 50D from flow inlet. Uniform velocity inlet and a pressure outlet are governing the flow inside the domain. The leeward distance L_d from the center of cylinder to the outlet boundary is taken as 200D.

![Figure 1. Schematic diagram of the domain.](image)

2.2. Governing Equations.
A two-dimensional, steady, incompressible and laminar Navier-Stokes equation is solved. A finite volume based solver Fluent [12] is used to discretize and solve the transport equations.

Continuity equation:

$$\sum_{j} \frac{u_i}{x_j} = 0$$

Momentum equation:

$$u_j \frac{u_i}{x_j} = 1 \frac{p}{x_i} + \frac{2}{x_j x_j} u_j$$
2.3. Grid and model validation

The non-uniform structured mesh with quadrilateral elements is generated with the prism layer in the vicinity of the cylinder surface. To capture the downstream flow phenomena very fine mesh elements with smooth transition are employed. The flow domain and the generated mesh in the flow domain are shown in Figure 2. The grid validation test is employed by observing the coefficient of drag for a various number of nodes for a cylinder with a parallel slit and the result is shown in figure 2. No significant variation in the drag coefficient can be seen beyond 71000 nodes.

![Figure 2. Grid distribution in the domain.](image)

Therefore, a different number of nodes i.e. 71216, 72917, 71453, and 70312 are taken for cylinders having converging, diverging, parallel and no slit respectively. The grid validation test is employed by observing the coefficient of drag for a various number of nodes for a cylinder with a parallel slit and the result is shown in figure 3.

![Figure 3. Grid validation result.](image)

No significant variation in the drag coefficient can be seen beyond 71000 nodes. Therefore, the different number of nodes i.e. 71216, 72917, 71453, and 70312 are taken for cylinders having converging, diverging, parallel and no slit respectively. The numerical model is validated with the past work [5] and the comparison is represented in figure 4 with a very good agreement. The two dimensional Navier-Stokes equations are discretized and solved with the finite volume approach. The steady and incompressible flow is considered as the downstream
instabilities in the flow just start beyond Re 50 over the slotted cylinder. All the cases are computed serially with a quad-core processor.

![Graph showing numerical model validation with past work][1]

**Figure 4.** Numerical Model validation with past work [5].

### 3. Results and Discussions

#### 3.1. Bubble length and flow separation

The phenomena behind the flow past cylinder are significantly affected by the downstream formation of the bubble. The bubble size and its shape influences the drag and lift forces applied over the surface of the cylinder. The non-dimensional bubble length is defined as

\[ \eta = \frac{x}{D} \]  

(3)

Where \( \eta \) is the non-dimensional bubble length and \( x \) is the distance of the bubble rear end to the center of the cylinder. The bubble length can be a very good parameter to predict the onset of the wake with a further increase in Re. The oscillation of the trail influences the elongated bubbles and thus leads to the Karman vortex formation. The bubble formations for different slit shapes are shown in figure 5.

![Graph showing bubble formation and length for different slits][2]

**Figure 5.** (a) Bubble formation and, (b) Bubble length for cylinders with, 1. Converging slit, 2. Diverging slit, 3. Parallel slit and, 4. No slit.
The perception can be made clear by observing the non-dimensional bubble length. It is found that the bubble in the diverging slotted cylinder is most elongated and least elongated in the cylinder with no slit. The self-issuing jet for the slotted cylinder causes the bubbles to elongate. The non-dimensional bubble length for different cases is shown in figure 5. The angle of separation for different cases is shown in table 1. The angle is measured from the upstream stagnation point to downstream stagnation point. It is found that the cylinder with no slit is able to delay the flow separation with respect to the slotted cylinder by 0.4 degree. The flow inside the slit faces produces the flow resistance which further causes an increase in the skin friction.

| Cases                          | Separation Angle (in degree) |
|-------------------------------|-----------------------------|
| Slotted cylinder (Converging, diverging and parallel) | 105.9 |
| No slit                       | 106.3 |

3.2. Coefficient of drag

Drag reduction is an important objective for considering the flow past bluff body. Drag coefficient defined as the ratio of force exerted on the body in the counter-flow direction to the force caused by the dynamics pressure. It is given by

\[ C_d = \frac{\frac{F_x}{1}}{\frac{u^2 D}{2}} \] (4)

The drag force has the contribution from viscous and pressure drag. The viscous force is significantly affected by the skin friction and the surface nature of the body. The pressure drag is produced due to the formation of eddies and flow separation causing a negative pressure gradient across the cylinder. It is very important to know about the contribution of these two forces on the overall drag. Figure 7 is an attempt to show the drag coefficients for different cases.

Figure 6. The coefficient of drag for cylinders with, 1. Converging slit, 2. Diverging slit, 3. Parallel slit and, 4. No slit.
Cylinder with parallel slit has the highest friction drag and least pressure drag among the slotted cylinders. Total drag for slotted cylinders is similar. The converging and diverging slots are showing a similar trend to each other. Total drag is least for the cylinder with no slit irrespective of having highest pressure drag. Due to the absence of slits, the viscous drag is minimum.

4. Conclusion
The finite volume solver is employed to solve the steady and viscous incompressible flow past the circular slotted cylinder. Three different slits shapes have been studied. The low separation is affected by the slits and the separation angle is delayed for the cylinder having no slit by 0.4 degree. The drag value match quite well with the past study. Viscous drag for the slotted cylinder is found to be increased due to the slit surfaces. Total drag was found to be least for no slit cylinder and almost 8 percent increment for all the geometries with slit.

References
[1] Taneda, S. (1956). Experimental investigation of the wakes behind cylinders and plates at low Reynolds numbers. Journal of the Physical Society of Japan, 11(3), 302-307.
[2] Tritton, D. J. (1959). Experiments on the flow past a circular cylinder at low Reynolds numbers. Journal of Fluid Mechanics, 6(4), 547-567.
[3] Hamielec, A. E., & Raal, J. D. (1969). Numerical studies of viscous flow around circular cylinders. The Physics of Fluids, 12(1), 11-17.
[4] Takami, H., & Keller, H. B. (1969). Steady two-dimensional viscous flow of an incompressible fluid past a circular cylinder. The Physics of Fluids, 12(12), II-51.
[5] Dennis, S. C. R., & Chang, G. Z. (1970). Numerical solutions for steady flow past a circular cylinder at Reynolds numbers up to 100. Journal of Fluid Mechanics, 42(3), 471-489.
[6] Fornberg, B. (1980). A numerical study of steady viscous flow past a circular cylinder. Journal of Fluid Mechanics, 98(4), 819-855.
[7] Nieuwstadt, F., & Keller, H. B. (1973). Viscous flow past circular cylinders. Computers & Fluids, 1(1), 59-71.
[8] Sen, S., Mittal, S., & Biswas, G. (2009). Steady separated flow past a circular cylinder at low Reynolds numbers. Journal of Fluid Mechanics, 620, 89-119.
[9] Wang, J., & Dong, Y. (2017). The numerical investigation of flow and heat transfer characteristics of flow past a slit-cylinder. Science China Technological Sciences, 60(4), 602-612.
[10] Gao, D. L., Chen, W. L., Li, H., & Hu, H. (2017). Flow around a circular cylinder with slit. Experimental Thermal and Fluid Science, 82, 287-301.
[11] Ordia, L., Venugopal, A., Agrawal, A., & Prabhu, S. V. (2017). Vortex shedding characteristics of a cylinder with a parallel slit placed in a circular pipe. Journal of Visualization, 20(2), 263-275.
[12] Fluent, A. (12). Theory guide, 2009. ANSYS Inc.