Numerical Investigation of Flow Control past a Slotted Cylinder for Lower Critical Transitional regime

Biswaranjan Pati\textsuperscript{1*}, Bishwajit Sharma\textsuperscript{2}, and Rabindra Nath Barman\textsuperscript{3}

\textsuperscript{1,2,3} National Institute of Technology, Durgapur, West Bengal, India.
*biswaranjan96@gmail.com/bp.20150817@btech.nitdgp.ac.in

Abstract. The phenomenon of fluid flow past a bluff body can be ubiquitously perceived both in nature as well as a multitude of engineering applications. The pragmatic and real-world significance of this problem combined with its simple yet intricate physics has turned the heads of quite a few researchers in the past century. The present study deals with the numerical investigation of flow characteristics around a modified circular cylinder with a slit. The numerical simulations were performed with a finite volume-based solver at a Reynolds number (Re) = 2500 based on the cylinder diameter (D) and the centerline velocity at the inlet, which can be considered as the lower critical value for the dynamic shift from laminar to turbulent flow regime. The upstream and downstream stagnation points on the cylinder surface were connected by a slit. The slits implemented were parallel, converging, and diverging with respect to the direction of flow and amalgamated with one cylinder having no slit, a total of four cases were solved. The windward width of parallel and converging slits was 0.1D and the end-width ratio ($\Phi$) of the converging and diverging slits was 1.5. The results indicated that incorporation of slits in the modified bluff body contributes to drag reduction by the creation of self-issuing jets which affect the wake vortex shedding process downstream of the cylinder. The parallel slit being the most efficacious one leads to the highest reduction of drag due to the formation of smaller sized vortices as compared to convergent or diverging slits in the wake region of the cylinder.

Keywords: Bluff body, Reynolds Number, Self-issuing jet, Wake vortex shedding, slotted cylinder.

1 Introduction

Flow past bluff bodies can be veraciously dubbed as one of the most quotidian phenomena, considering its occurrence and application in our everyday lives. Any object that operates in the flow field of a certain fluid, inevitably experiences drag which ultimately decreases the system’s performance. Many researchers and scientists have analyzed the aforementioned situation closely and have devised various methods to reduce the drag experienced by a system. Notable luminaries of the past decades such as Thom (1936) [1], Grove et al. (1964) [2], and Hamielec & Raal (1969) [3] have extensively investigated viscous flow past circular cylinders in the domain of high and low Reynolds numbers, and the coherence in their results is conclusive proof...
that indeed, the drag across a cylinder can be reduced by delaying flow separation from surface of bluff body. Although limited by the technologies of their times, these researchers have laid down the essential groundwork in the field of aerodynamics.

Recent endeavors have been focused on alternative and otherwise effective methods to manipulate the wake vortex shedding from a bluff body, reduce the drag, and suppressing the lift fluctuation. Chen et al. [4], [5] have experimented with suction flow control method to suppress the vortex-induced vibrations and reduce the unsteadiness of wind loads acting on a circular cylinder. The vibrations experienced by the cylinders were first reproduced using a spring-mass system and then the fluctuating pressure coefficients were analyzed. Feng et al. [6], [7] have implemented synthetic jets to control the wake vortex synchronization as well as delay the flow separation in circular cylinders. A synthetic jet generated by a non-sinusoidal waveform is used to control flow separation around a circular cylinder at a low Reynolds number. Plasma actuators [8] can also be used as effective tools to control flow through a body-force external field that couples with momentum in the external flow, based on a single-dielectric barrier discharge (SDBD) mechanism. Other simplistic but equally ingenious methods like heating [9] and counter-rotation of cylinders [10] have also been employed to control vortex shedding and reduce drag. Increasing the heat input results in a change of airflow patterns which leads to vortex suppression and drag reduction. Chan et al. [10] have experimentally substantiated the relationship between the degree of unsteady wake suppression and the speed and direction of rotation. Their experiments have also insinuated the existence of a critical rotation rate where complete suppression of flow unsteadiness can be achieved. Posdziech et al. [11] have used electromagnetic control of seawater flow around circular cylinders as another interesting avenue for research.

Passive methods of flow control include careful placement of obstacles in the path of fluid flow to suppress vortex formation and delay separation. Still others such as Gao et al. [12] have modified cylinder geometry by incorporating slits and analyzed flow around it by using Particle Image Velocimetry (PIV). This paper intends to compare and contrast the parameters of flow past a circular cylinder at a Reynolds Number = 2500 by implementing slits of different geometries like converging, parallel, & diverging and juxtapose the results with respect to a cylinder having no slit for determining the most efficient flow system in terms of drag coefficient and pressure drop.

## Methodology

### 2.1 Problem setup

The setup consists of a circular cylinder placed in a rectangular computational domain where the origin of the Cartesian coordinate system coincides with the center of the cylinder. For the case of no-slip sidewalls, the velocity is set to zero at the lateral boundaries and a parabolic velocity profile is specified at inlet: 

\[ U = V \left[ 1 - (2y/H)^2 \right]; \]

where \( y \) is the vertical distance measured from the center of the cylinder and \( V \) is the
centerline velocity. The blockage ratio \((D/H)\) is taken as 0.015 based on previous literature [3]. The Reynolds number of this flow is based on the centerline velocity and diameter of the cylinder as the length scale. In all cases, the upstream and downstream boundaries are located sufficiently far from the cylinder to have no consequential influence on the overall flow pattern. These distances are \(X_U = 50D\) and \(X_d = 100D\), respectively, measured from the center of the cylinder (see Fig. 1).

![Fig. 1. Schematic of flow domain with different cylinder configurations](image)

### 2.2 Governing equations

K-\(\varepsilon\) turbulence model is employed to solve the viscous flow using the commercially available finite volume based solver, ANSYS Fluent [13]. The continuity, momentum and transport equations for the flow domain are given below.

**Continuity equation**

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{1}
\]

**Momentum equation**

\[
\frac{\partial u}{\partial t} + \frac{1}{\rho} \frac{\partial P}{\partial x} = \frac{\partial}{\partial x} \left[ \mu \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) \right] + \frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \right] \tag{2}
\]

Where \(\Phi\) = Newton's gravitational potential.
k-ε Transport equations

\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + \frac{G_k}{\rho} - \rho \frac{\partial \varepsilon}{\partial t} + Y_k + S_k
\]  

(3)

\[
\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[(\mu + \frac{\mu_t}{\sigma_\varepsilon})\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{\mu} \varepsilon) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon
\]  

(4)

Here \( k \) is the turbulent kinetic energy and \( \varepsilon \) is its rate of dissipation. The model constants have the following default values:

\[ C_{1\varepsilon} = 1.44; \quad C_{2\varepsilon} = 1.92; \quad C_{\mu} = 0.09; \quad \sigma_k = 1.0; \quad \sigma_\varepsilon = 1.3 \]

### 2.3 Validation of method

The formulation and implementation of finite volume analysis has been validated by comparing the coefficient of pressure \( C_p \) on surface of the cylinder at \( Re = 15 \) and 30 with those reported from Sen et al. [14] and Hamielec & Raal [3]. As expected, the maximum pressure occurs at the forward stagnation point and all the computations are made with the same flow domain and boundary conditions.

![Fig. 2. Comparison of predicted \( C_p \) with data from literature at \( Re = 15 \) and 30.](image)

**Fig. 2.** Comparison of predicted \( C_p \) with data from literature at \( Re = 15 \) and 30.

### 2.4 Mesh and Grid independence

Grid independence test is performed for total drag experienced by the cylinder for different grid counts. The model espoused for the grid independence was the cylinder having a diverging slit having 0.1D as the width of the smaller orifice and 1.5 as the end-to-end width ratio (\( \Psi \)). The meshing data is given in Table 1.
Cylinder configurations     | Nodes     | Elements     
--- | --- | --- 
Without slit       | 313066  | 300679  
With slit          | 342726  | 340950  

Table 1. Meshing data.

3 Results and discussion

All the computations are carried on a system with an Intel i5 5th generation processor with four CPUs having a clock speed of 2.2 GHz. The aforementioned meshing requires 8 GB of ram with approximately 7 hours of CPU time to achieve a convergence of $10^{-10}$ utilizing a single core. The efficiency of the system in terms drag reduction and point of boundary layer separation is evinced through the plots of streamlines, the coefficient of pressure ($C_p$), drag coefficient ($C_d$) and angle of separation ($\alpha$). Figures 4 (a) and (b) represent the comparison between the streamlines at Re=1500 and 2500 respectively, engendering some interesting results. The primary vortices seem to get detached due to the incorporation of slits and the formation of small secondary vortices is the result of self-issuing jets propelling out of the slit. As the Reynolds number increases however, the primary vortices seem to disappear out of the wake region and the secondary vortices are much more prominent (See Fig. 4 (a) and (b)). Least value of $\alpha$ (See Fig. 5(b)) for parallel slit shows that detached secondary vortices of small size contributed towards this result whereas in case of the cylinder without slit, there are attached vortices causing early flow separation over the surface of the cylinder for both the cases.

The size of primary vortices (bubble length) seems to directly proportional to the Reynolds number as evident form the bubble lengths at Re = 1500 and 2500.
Fig. 4. Streamlines around different slotted cylinders at (a) Re = 1500, and (b) Re = 2500.
Fig. 5. (a) Drag coefficient (Cd) at Re = 2500 and (b) Angle of boundary layer separation (α) at Re = 2500.

However, the primary vortices reach a maximum bubble length at a critical Reynolds number surpassing which the vortices seem to detach in its wake. The bubble length can be measured by attaching longitudinal probes along the centerline of the schematic as shown by Sen et al. [14].

From the plot of drag force (See Fig. 5 (a)) it is evident that the cylinder without slit experiences the maximum value and the parallel slit cylinder experiences the minimum value of total drag. These results are consistent with the streamlines and the pressure coefficient plots; as the slits are incorporated the total surface area exposed to the flow increases and hence the viscous drag increases as evinced in the case of slotted cylinders. While increase in viscous drag may contribute to an increase in total drag which may be abated in various ways, the pressure drag, on the other hand, decreases due to the incorporation of slits as a result of self-issuing jets. The formation of small secondary vortices is the result of self-issuing jets propelling out of the slit. The location of boundary layer separation is determined by the size of secondary vortices. The separation angle (α) is measured in counter clockwise direction from the line of symmetry at leeward of the cylinder. Least value of α for parallel slit shows that detached secondary vortices of small size contributed towards this result whereas in case of the cylinder without slit has attached vortices causing early flow separation over the surface of the cylinder.

Fig. 6. The plot for Coefficient of pressure (Cp)
The plot for the coefficient of pressure ($C_p$) (See Fig.6) further interprets the coherence between the results manifested by the streamlines. Negative pressure implies a suction zone and as $C_p$ is dependent on pressure; a negative maximum reached in all the cases signifies that a maximum suction zone is being created. The proximity in the maximum value of suction in case of $Re = 2500$ in case of each plot signifies the propinquity between the values of separation angles which is vindicated by Fig. 5 (b). A larger magnitude represents larger vortices with minimum detachment; which in turn expedites boundary layer separation and increases drag force.

4 Conclusion

After careful analysis of all cases of flow past a cylinder with or without slits, it can be concluded that applying slits to a cylinder contributes to decrease in pressure drag by a detachment of primary vortices and formation of smaller secondary vortices. A few other concluding remarks can be stated as follows:

- The cylinder in parallel slits proves to be most efficacious in reducing drag and delaying boundary layer separation due to detachment of primary vortices and smallest size of secondary vortices.
- As the Reynolds number increases, the primary vortices disappear from the wake region and secondary vortices are much more prominent.
- The plot of drag coefficient implies that the cylinder without slit experiences highest value of drag, mainly manifested as pressure drag.
- The proximity in the values of pressure coefficients ($C_p$) in each case is consistent with the plot of ($\alpha$) accounting for the closeness in the values of angle of boundary layer separation.

Experimenting within a range of Reynolds number and different configurations of slits in terms of width and the end-to-end ratio can give a more comprehensive understanding of the aforementioned phenomena, which can be contemplated as future scope of work.

5 Appendix

| Symbols | Meaning                                |
|---------|----------------------------------------|
| $\Phi$  | End-to-end width ratio                 |
| $\alpha$ | The angle of boundary layer separation |
| $C_p$   | Coefficient of pressure                |
| $D$     | Diameter of cylinder                   |
| $H$     | Height of flow domain                  |
| $X_u$   | The upstream width of the flow domain  |
| $X_d$   | The downstream width of the flow domain|
| $U$     | Velocity profile                       |
| $V$     | Centerline velocity                    |
6 References

1. Thom, A. (1933): The flow past circular cylinders at low speeds. Proceedings of the Royal Society of London. Series A, Containing Papers of a Mathematical and Physical Character, 141(845), 651-669.
2. Grove, A. S., Shair, F. H., & Petersen, E. E. (1964): An experimental investigation of the steady separated flow past a circular cylinder. Journal of Fluid Mechanics, 19(1), 60-80.
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. Chen, W. L., Xin, D. B., Xu, F., Li, H., Ou, J. P., & Hu, H. (2013): Suppression of vortex-induced vibration of a circular cylinder using suction-based flow control. Journal of Fluids and Structures, 42, 25-39.
5. Chen, W. L., Li, H., & Hu, H. (2014): An experimental study on a suction flow control method to reduce the unsteadiness of the wind loads acting on a circular cylinder. Experiments in fluids, 55(4), 1707.
6. Feng, L. H., Wang, J. J., & Pan, C. (2010): Effect of novel synthetic jet on wake vortex shedding modes of a circular cylinder. Journal of Fluids and Structures, 26(6), 900-917.
7. Feng, L. H., & Wang, J. J. (2010): Circular cylinder vortex-synchronization control with a synthetic jet positioned at the rear stagnation point. Journal of Fluid Mechanics, 662, 232-259.
8. Corke, T. C., Enloe, C. L., & Wilkinson, S. P. (2010): Dielectric barrier discharge plasma actuators for flow control. Annual review of fluid mechanics, 42, 505-529.
9. Lecordier, J. C., Hamma, L., & Paranthoen, P. (1991): The control of vortex shedding behind heated circular cylinders at low Reynolds numbers. Experiments in Fluids, 10(4), 224-229.
10. Chan, A. S., Dewey, P. A., Jameson, A., Liang, C., & Smits, A. J. (2011): Vortex suppression and drag reduction in the wake of counter-rotating cylinders. Journal of Fluid Mechanics, 679, 343-382.
11. Posdziech, O., & Grundmann, R. (2001): Electromagnetic control of seawater flow around circular cylinders. European Journal of Mechanics-B/Fluids, 20(2), 255-274.
12. 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.
13. Fluent, Ansys: "12.0 Theory Guide." Ansys Inc 5 (2009).
14. 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.