HEFAT2014
10th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics
14 – 26 July 2014
Orlando, Florida

Flow Past A Cylinder With Upstream Splitter Plate

Rajkumar M.R*[1],Rijas A.S[1] and Venugopal.G[2].
* Department of Mechanical Engineering, College of Engineering, Trivandrum, Kerala, India
1. Department of Mechanical Engineering, College of Engineering, Trivandrum, Kerala, India
2. Department of Mechanical Engineering, Govt: Rajiv Gandhi Institute of Technology, Kottayam, Kerala, India,
E-mail: rajkumar@cet.ac.in

ABSTRACT
This paper reports combined experimental and numerical investigations performed to study the effect of upstream splitter plate on drag reduction of a circular cylinder. The influence of spacing between the cylinder and splitter plate (aspect ratio) and thickness of the splitter plate (thickness ratio) on the flow characteristics of the cylinder is analyzed and discussed. The mathematical model governing the problem is numerically solved using a commercial computational fluid dynamics package ANSYS 14. The simulations have been carried out for Re=20,150 and 6×10^4 so as to cover the laminar regime. In order to validate the numerical model, experiments are performed by placing the cylinder along with the splitter plate in a low speed subsonic wind tunnel.

Keywords: - Reynolds number, aspect ratio, thickness ratio, coefficient of drag.

NOMENCLATURE

A	[m^2]	Projected area
C_D	[-]	Coefficient of drag
C_P	[-]	Coefficient of pressure
C_{PC}	[-]	Pressure coefficient at the front stagnation point of the cylinder
C_{PS}	[-]	Pressure coefficient at the trailing edge of the upstream splitter plate
ΔC_P	[-]	Difference between C_{PC} and C_{PS}
D	[m]	Diameter of cylinder
L	[m]	Length of the plate
P	[Pa]	Pressure at different point on the surface of the cylinder.
P_∞	[Pa]	Free stream pressure
Re	[-]	Reynolds number
S	[m]	Spacing between plate and cylinder
T	[m]	Thickness of plate
U_∞	[m/s]	Free stream velocity
u	[m/s]	X-component of velocity at different locations measured along the Y- direction at the downstream of the cylinder.
AR	[-]	Aspect Ratio (S/D)
TR	[-]	Thickness Ratio (T/D)
Greek Symbols		
ρ	[kg/m^3]	Density
ν	[m^2/s]	Kinematic viscosity
θ	[degree]	Angular position on the surface of the cylinder.

INTRODUCTION
Flow past a circular cylinder has been investigated by various researchers because of its practical applications in variety of engineering flows, such as offshore risers, bridge piers, periscopes, chimneys, towers, masts, stays, cables, antennae, wires etc. The flow past a cylinder is associated with very rich vortex dynamics, and different flow patterns are observed for various regimes of flow. The flow separation on the surface of the cylinder causes pressure drop in the rear part of the cylinder, resulting in considerable increase in pressure drag. Reducing the drag is critically important in many engineering applications, and different techniques have been proposed to achieve this goal. Among the different techniques the most frequently used methods are modification of the shape of the cylinder or placing additional surfaces in the flow stream either at the rear or the front of the cylinder.

Alam et al. [1] conducted experiments to study interaction of fluctuating aerodynamic forces on two circular cylinders of same diameter arranged in tandem in uniform flow. The study revealed that the effect of aerodynamic forces on the downstream cylinder will be significant only when the spacing between the cylinders is 3 times diameter of the cylinder.

The same authors [2] also studied experimentally the effect of T-shaped plate placed upstream of tandem cylinders considered in their earlier studies. They reported that for a tail length of T-shaped plate in the range 0.7 - 1.00 times the diameter of the cylinder, there was significant reduction in the fluid forces acting on the cylinder.

The effect of the splitter plate in the downstream of a vertical circular cylinder placed in shallow water was studied experimentally by Akilli et al. [3] .They observed that significant vortex suppression, when the distance between cylinder base and leading edge of the plate was kept in the range of 0-1.75 times the diameter of the cylinder.

Numerical investigation was performed by Hwang et al. [4] to study drag reduction on a circular cylinder by using two plates. One plate was placed in the upstream and other in the downstream of the cylinder along the centreline. The upstream plate reduces pressure in the vicinity of front stagnation point of the cylinder with a secondary effect of increasing the base pressure, while the downstream plate effectively suppresses vortex shedding in the near wake region by increasing pressure in the rear of the cylinder. They obtained minimum drag if distance between plates and cylinder were 1.5 and 2.4 times the diameter of the cylinder for upstream and downstream plates respectively.

Ali et al. [5] numerically investigated flow over square cylinder by placing a flat plate horizontally in the downstream. They identified significant reduction of drag force when the plate is placed at a distance of 2.3 times side length of the cylinder. It was also estimated that a total lift cancellation was possible if a plate of length equal to 0.26 times diameter of the cylinder is placed at a distance of 5.6 times diameter of the cylinder. Numerical investigations are performed by Alex et al [6]to study the effect of flat ground boundary near a cylinder on the suppression of vortex shedding. The analyses demonstrated that the presence of plane ground will produce additional circulation caused by the fluid viscosity with the Venturi effect. The additional circulation around the body is responsible for increase in lift force acting on body surface, while the venture effect is responsible for the decrease in drag force acting on body surface. Qui et al [7] experimentally investigated, the characteristics of wind loads acting on a circular cylinder with attached splitter plates at the front and rear of the cylinder. The study revealed that vortex shedding from the cylinder can be suppressed by a splitter plate having length equal to 3 times diameter of the cylinder in the wake.

From the above literature survey, it can be seen that many of the previous investigators have focused their attention on the effect of splitter plate placed either at upstream or downstream of the cylinder on the drag reduction. However most of these investigations are numerical. Experimental investigations relating to the study of the effect of splitter plate on the drag reduction of a cylinder has not been found else were in literature. The aim of this work is to experimentally investigate the effect of upstream splitter plate on the drag characteristics of a cylinder. Numerical analysis has also been performed and the numerical results are validated by comparing with experimental results. The influence of Reynolds number, aspect ratio and thickness ratio of the splitter plate on the drag characteristics of the cylinder is analyzed and discussed.

EXPERIMENTAL SETUP

The physical system consists of a cylinder having a diameter of 100mm with an upstream splitter plate of length equal to diameter of the cylinder placed upstream of the cylinder as shown in Figure 1. In the present work the main emphasis to study the effect of detached upstream splitter plate on the drag reduction on the cylinder, depending on the AR and TR, while the flow topology remains unchanged. To achieve this objective the length of the splitter plate is made equal to diameter so that the flow past the detached plate is laminar and the momentum thickness over the splitter plate boundary layer is thin, (for details see [4, 8]) The cylinder along with splitter plate placed in the test section of dimensions 500mm × 500mm × 2000mm of a low speed subsonic wind tunnel as shown in Figure 2. Pressure tappings (18 numbers) are provided along the circumference of the cylinder at the mid plane of the cylinder. These pressure tappings are connected to a differential manometer to measure the pressure distribution over the circumference of the cylinder. To measure the free stream velocity i.e., the velocity at the inlet of the test section a pitot static tube is provided at the inlet of the test section. The ends of the pitot static tube are connected to a projection type manometer having least count 0.1mm and the free stream velocity is calculated using the expression

$$U_\infty = \sqrt{\frac{2(P - P_\infty)}{\rho}} \tag{1}$$

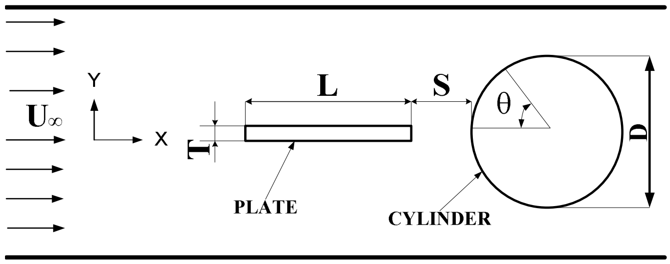


Figure 1 Schematic diagram

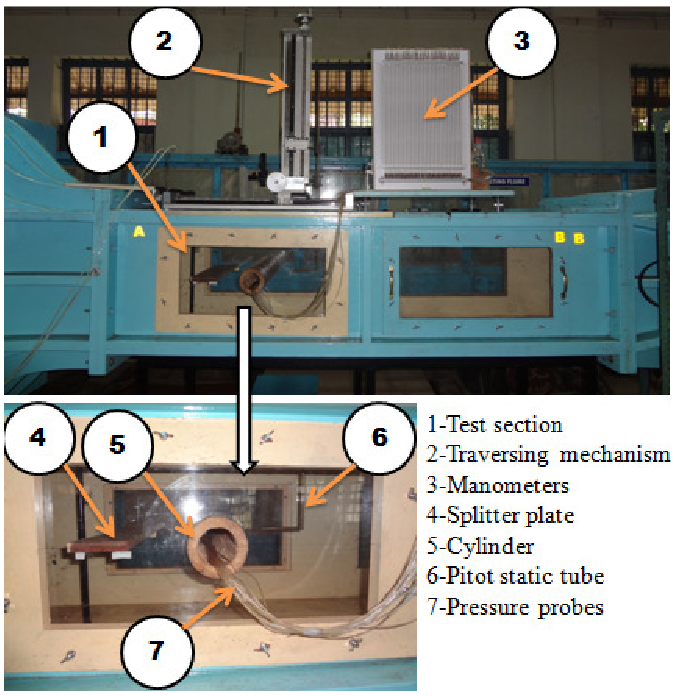


Figure 2 Experimental setup

In order to measure the wake velocity at the downstream of the cylinder a separate pitot static tube is provided. This pitot static tube is placed downstream of the cylinder at a distance of approximately eight times diameter of the cylinder and is connected to the traversing mechanism. The traversing mechanism allows the transverse movement of the pitot static tube (Y-direction).

The important dimensionless parameters considered in this study are

Reynolds number, $$Re = \frac{U_\infty D}{\nu} \quad (2)$$

Coefficient of pressure, $$C_P = \frac{P - P_\infty}{\frac{1}{2}\rho U_\infty^2} \quad (3)$$

Coefficient of drag,

Based on wake velocity, $$C_D = 2\int_{-\infty}^{\infty} \frac{u}{U_\infty}\left(1 - \frac{u}{U_\infty}\right)dy \quad (4)$$

Based on pressure distribution, $$C_D = -\frac{1}{2}\int_0^{2\pi} C_P \cos\theta d\theta \quad (5)$$

NUMERICAL ANALYSIS

The mathematical modelling governing the problem is numerically solved in a two dimensional computational domain using commercially available computational fluid dynamics package ANSYS FLUENT 14 [9]. The flow is assumed to be incompressible and laminar. The simulation have been performed for steady (Re=20) and unsteady (Re=150 & 6×10^4) cases. The computational domain consists of cylinder, upstream splitter plate and the region surrounding them. The size of the computational domain has been fixed based on the results of the earliest studies[10]. The governing equation of flow pertinent to this problem can be seen in Kundu et al. [11]

A. Boundary conditions and grid

No slip boundary conditions are employed on surfaces of cylinder and plate. At inlet, top and bottom boundaries of the domain, the velocity inlet (free stream velocity) boundary condition is specified, whereas pressure outlet condition is given at exit. The computational domain is discretised using structured non-uniform quadrilateral cells. Fine grids were used in the regions near the cylinder and plate so as to capture the gradient of the field variables. In order to arrive at an optimum grid, a grid sensitivity study has been conducted, at Re=20 (steady) as shown in Table1.The results are found to be insensitive to the grid beyond the one with beyond the one with 21500 cells consisting of 43330 faces and 21830 nodes. The final computational domain along with grid used for numerical simulation is shown in Figure 3.

Table 1 Grid Independence study (Re=20,AR=0.01, TR=1)

No of cells	Maximum coefficient of drag C_D
13000	1.30418
21500	1.28377
32000	1.28307

B. Numerical solution procedure

In the present computation the steady and transient solver options available in the FLUENT solver has been used for solving the governing partial differential equations for steady and unsteady cases respectively. Pressure based option with second order upwind discretization scheme has been selected for the present computation. The coupling between velocity and pressure is resolved by selecting the SIMPLE (Semi-implicit method for pressure linked equations) [12] algorithm. Since the equations are non linear the solution has to be progressed in a controlled manner. This is achieved by using relaxation factors. The under relaxation factors used in the present study are 0.3 for pressure and 0.7 for momentum. Convergence of the solution is checked by examining the residues of discretised conservation equations of mass and momentum. For unsteady case (Re=150) time step size is specified as 0.01s. The iteration is terminated only when the maximum of all the residues reaches less than 1×10^{-6} to assure conservation of quantities.

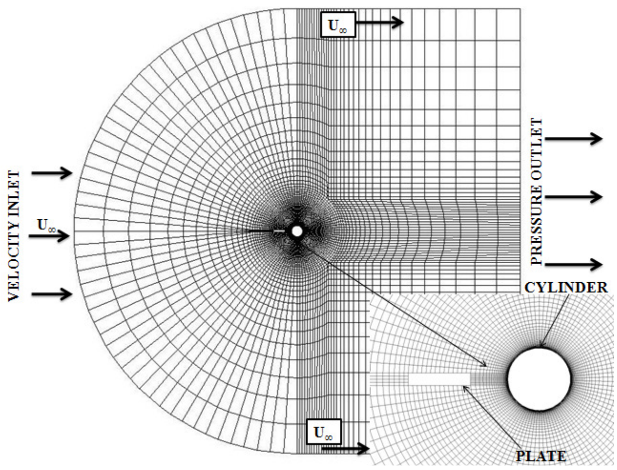


Figure 3 Computational domain and grid

RESULTS AND DISCUSSION

Experiments are conducted for Re=6×10^4 , aspect ratio 0.1, thickness ratio 0.2.The wake velocity at the downstream of the cylinder was measured by using pitot static tube connected to the differential manometer. Knowing the wake velocity distribution, drag force experienced by the cylinder was calculated by using equation (4). In order to check the validity of the above methodology the drag, based on the pressure distribution on the surface of the cylinder, by using equation (5) the coefficient of drag was calculated. These two results found to agree well as shown in Table2.

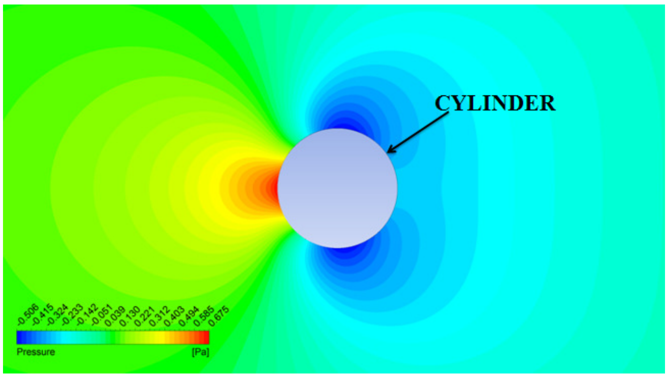
To check the validity of the proposed numerical methodology in solving this problem, numerical simulations were conducted for the same conditions as that of experiments. The comparison of the drag force experienced by the cylinder depicted in Table2 shows reasonably good agreement. The deviation in the values due to the experimental uncertainties which are inherent to any experimental setup. This comparison exercise shows the efficacy of the proposed numerical methodology in solving this type of problem. Due to experimental limitations, further numerical simulations were performed for Re=20(steady flow) and Re=150(unsteady flow), aspect ratio (0.1 - 25) and thickness ratio (0.01-9).

Table 2 Comparison of C_D obtained in experiment with numerical prediction

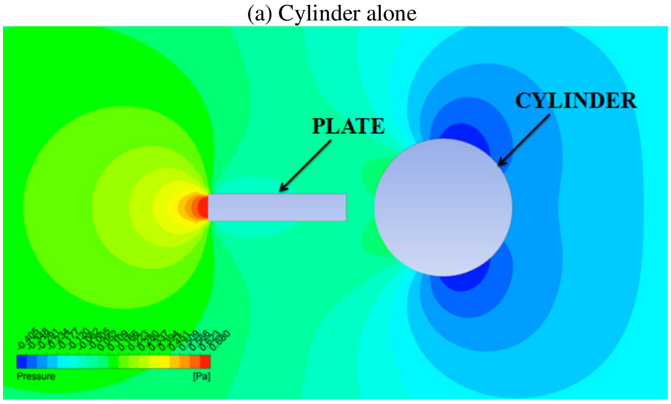
Re	Coefficient of drag (C_D)		
	Experimental		Numerical
	Based on wake velocity	Based on pressure distribution on the cylinder surface	
6×10^4	1.2	1.25	0.9

A. Effect of aspect ratio

Initially numerical simulations were performed with and without placing the splitter plate upstream of the cylinder. The pressure contour shown in Figure 4(a and b) reveals that due to the splitter plate the stagnation point at the upstream of the cylinder get shifted to the upstream of the splitter plate. This shift in the stagnation point causes the pressure at the upstream of the cylinder to reduce thereby decreasing the drag on the cylinder .



(a) Cylinder alone



(b) Cylinder with upstream splitter plate (AR=0.2, TR=0.2)

Figure 4 Static pressure contour (Re=20)

To further investigate the effect of the spacing between the splitter plate and the cylinder on the drag experienced by the cylinder simulations were performed for different Reynolds number and aspect ratio keeping the thickness ratio fixed as 0.2. Figure 5 shows the C_D of the cylinder plotted as a function of aspect ratio for Re=20 and Re=150.Here Re=20 corresponds to steady laminar flow and Re=150 corresponds to unsteady laminar flow. Their exist minimum C_D and this is at AR=8 for

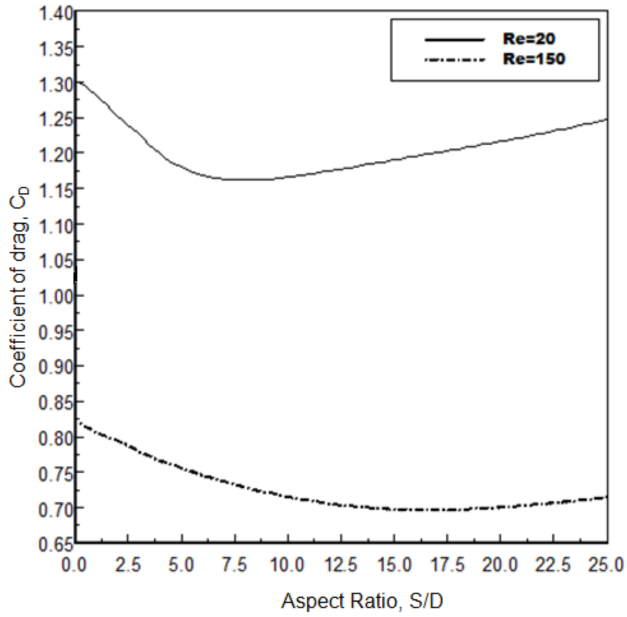


Figure 5 Variation of coefficient of drag with aspect ratio(TR=0.2)

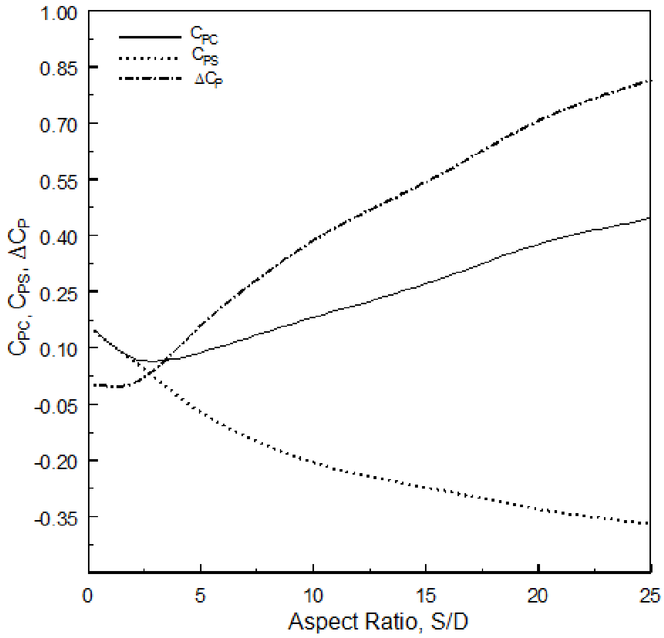
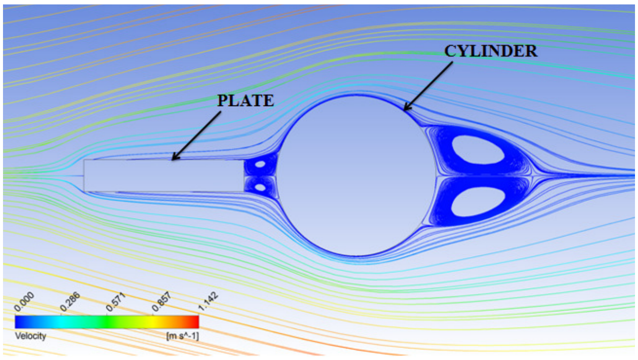


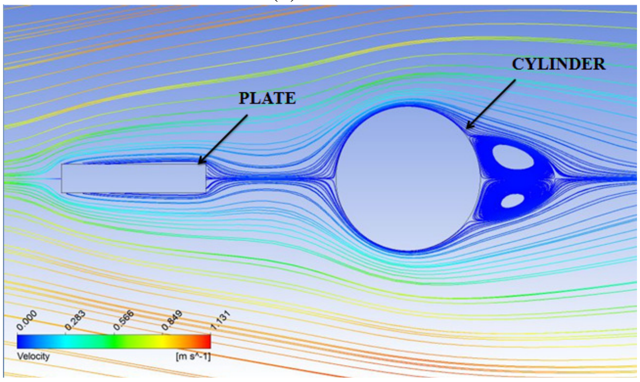
Figure 6 Variations of C_{PC},C_{PS} and $\Delta C_P = C_{PC} - C_{PS}$ with AR (Re=20,TR=0.2)

Re=20 and AR=16 for Re=150.The C_D curve corresponding to steady flow has the same trend as those corresponding to unsteady flow. This implies that C_D reducing mechanism of the detached upstream splitter plate is strongly related to the momentum deficit just upstream of the plate rather than the wake flow downstream of the cylinder. The reason for the existence of optimum aspect ratio corresponding to minimum value of C_D can be explained based on Figure 6. Figure 6 presents variations of the pressure coefficient at the front stagnation point of the cylinder (C_{PC}), the pressure coefficient at the trailing edge of the upstream splitter plate (C_{PS}), and the difference between the two ($\Delta C_P = C_{PC} - C_{PS}$) for Re =20. ΔC_P

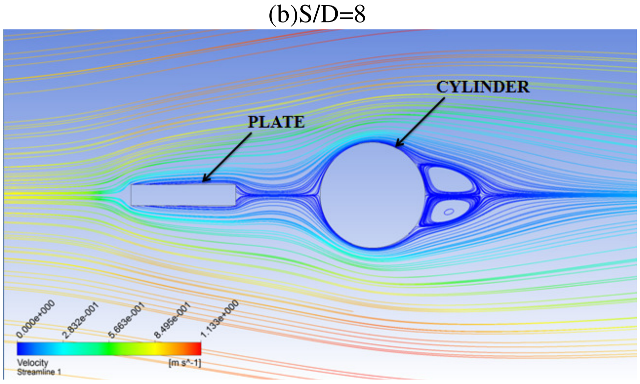
is a measure of the momentum recovery. It can be inferred from Figure 6 that as the AR increases up to the optimum AR the momentum recovery, ΔC_P is small (greater momentum deficit) compared to the increase in C_{PS}. This increase in C_{PS} i.e. stagnation pressure at the trailing edge of the upstream splitter plate is due to the blockage of flow by the cylinder. This blockage effect creates a recirculation zone in the space between the splitter plate and cylinder as shown in Figure 7a. Therefore the C_D decreases with increase in aspect ratio and has



(a)S/D=2



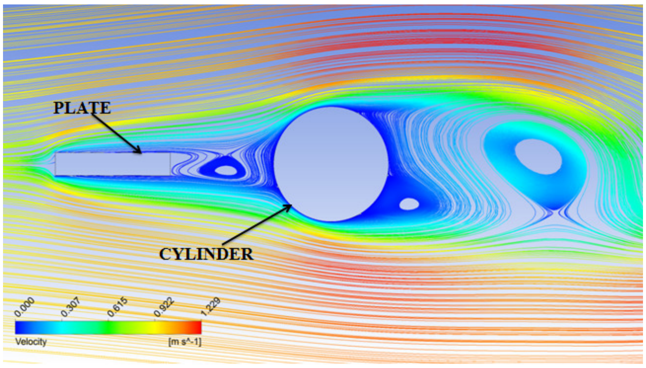
(b)S/D=8



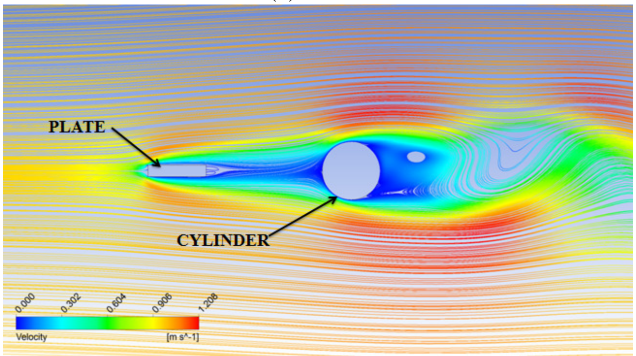
(c)S/D=9

Figure 7 Streamline pattern (Re=20 , T/D=0.2)

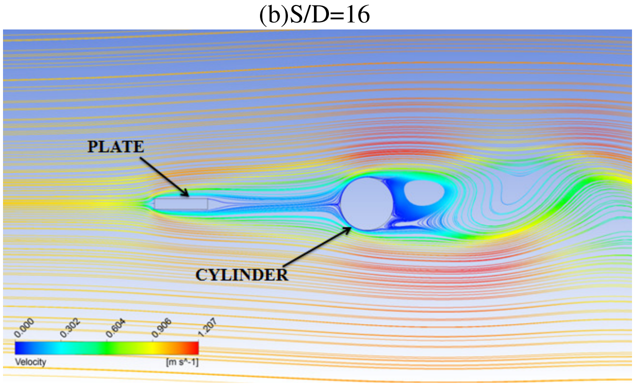
an optimum value AR=8. At the optimum AR the flow starts to reattach itself but the momentum recovery is low(Figure 7b). As the AR is further increased beyond the optimum the momentum recovery (ΔC_P) becomes dominant compared to the increase in C_{PS}. The flow fully reattaches itself (Figure 7c) and the drag continuously increases. However at Re=150 the optimum AR shift to AR=16. This shift in the location of optimum AR is due to the change in the flow pattern. At Re=150 the velocity of the incoming air is high and therefore it has to travel more distance before it reattaches itself. This reattachment begins to takes place at AR =16, but the momentum recovery is low and therefore the drag will be minimum. As AR further increases flow reattaches itself and the drag on the cylinder increases. The streamline pattern illustrated in Figure 8 clearly shows the above mentioned phenomena.



(a)S/D=8



(b)S/D=16



(c)S/D=20

Figure 8 Streamline pattern (Re=150 , T/D=0.2)

B. Effect of thickness ratio

Simulations have been performed by varying the thickness ratio (0.01-0.9) keeping Re=150 and AR=16 (optimum). As the thickness ratio increases, the coefficient of pressure at the trailing edge of the splitter plate becomes more dominant over the momentum recovery, resulting in continuous reduction in C_D as illustrated in Figure 9. This reduction in C_D

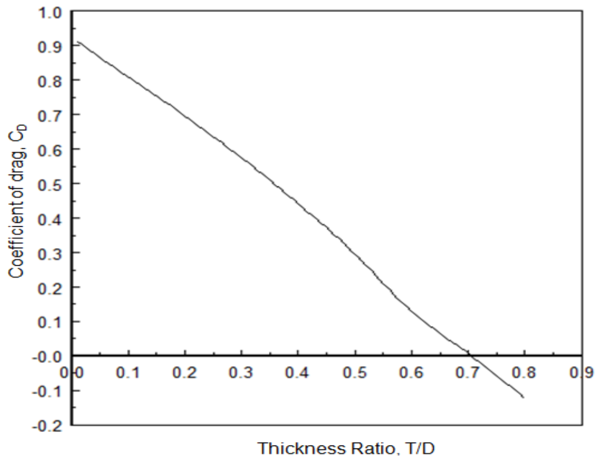
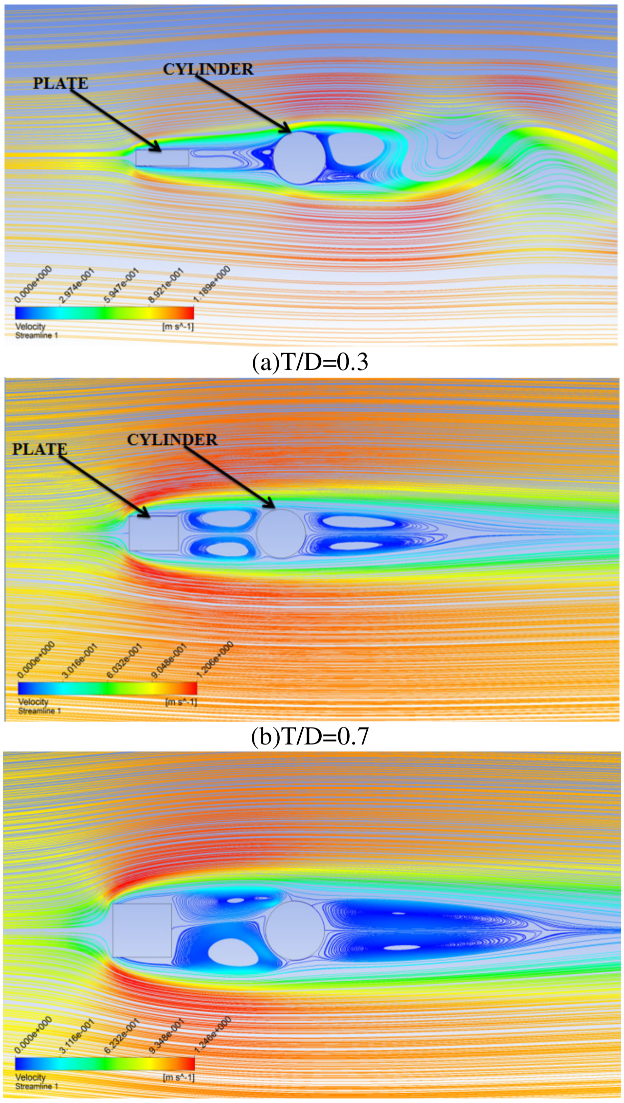


Figure 9 Variation of coefficient of drag with thickness ratio (Re=150,S/D=16)



(a)T/D=0.3

(b)T/D=0.7

(b)T/D=0.9

Figure 10 Streamline pattern (Re=150, S/D=16)

is due to the increase in the strength of the vortex with increase in thickness ratio as shown in figure 10a and b. Furthermore its worth to mention here that as the TR is increased beyond 0.7 vortex shedding takes place in the space between splitter plate and the cylinder Figure 10c, and the cylinder experience a drag in the opposite direction.

CONCLUSION

The effect of upstream splitter plate on the drag reduction of a circular cylinder has been investigated experimentally and numerically. Experimentally measured C_D were found to agree well with numerical results there by establishing the proposed numerical methodology. From the results it was observed that there exists an optimum AR at which the drag experienced by the cylinder was minimum. The location of optimum aspect ratio is different for steady Re=20 and unsteady Re=150. This is because the fluid has to travel more distance before it reattach itself. The increase in thickness ratio of the splitter plate continuously decreases the drag up to 0.07 due to the increase in the stagnation pressure at the trailing edge of the splitter plate. As the thickness ratio increases beyond 0.07 the cylinder experience a drag in the opposite direction due to the vortex shedding in the space between splitter plate and cylinder. The thickness ratio of the splitter plate continuously decreases drag up to 0.7 and thereafter the cylinder experiences a drag in the opposite direction.

REFERENCES

[1] Md. Mahbub Alam, M. Moriya, K. Takai, H. Sakamoto. Fluctuating fluid forces acting on two circular cylinders in a tandem arrangement at a subcritical Reynolds number. Journal of Wind Engineering and Industrial Aerodynamics, 91: 139–154 ,2003A.

[2] Md. Mahbub Alam, H. Sakamoto, Y. Zhoua. Effect of a T-shaped plate on reduction in fluid forces on two tandem cylinders in a cross-flow. Journal of Wind Engineering and Industrial Aerodynamics, 94:525–551, 2006.

[3] Huseyin Akilli, Besir Sahin, N. Filiz Tumen. Suppression of vortex shedding of circular cylinder in shallow water by a splitter plate. Flow Measurement and Instrumentation , 16: 211–219,2005.

[4] Jong-Yeon Hwang, Kyung-Soo Yang, Drag reduction on a circular cylinder using dual detached splitter plates. Journal of Wind Engineering and Industrial Aerodynamics ,95: 551–564,2005.

[5] Mohamed Sukri Mat Ali ,Con J. Doolan. Vincent Wheatley, Low Reynolds number flow over a square cylinder with a detached flat plate. International Journal of Heat and Fluid Flow , 36:133–141,2012.

[6] Alex Mendonca Bimbato , Luiz Antonio Alcantara Pereira , Miguel Hiroo Hirata , Suppression of vortex shedding on a bluff body. Journal of Wind Engineering and Industrial Aerodynamics, 121: 16–28 ,2013.

[7] Y.Qiu , Y.Sun , Y.Wu, Y.Tamura. Effects of splitter plates and Reynolds number on the aerodynamic loads acting on a circular cylinder. Journal of Wind Engineering and Industrial Aerodynamics, 127: 40–50 ,2014.

[8] H.Schlichting, K.Gersten ,Boundary Layer theory, Springer-Verlag, 2000.

[9] ANSYS FLUENT 14,275 Technology Drive , Canonburg.
[10] https://confluence.cornell.edu/display/SIMULATION/FLUENT+Learning+Modules
[11] Pijush K.Kundu, Ira M.Cohen, Fluid Mechanics, Academic Press,79-99,2002.
[12] S.V Patankar, Numerical Heat Transfer and Fluid Flow, McGraw Hill Book Company, Washington, 126-131,1980.