Eurasia Journal of Science and Technology	ISSN: 2663-1024	Volume 1, Issue 2,pp 8-13	June 2019

FLOW PAST A BLUFF BODY SUBJECTED TO LOWER SUBCRITICAL REYNOLDS NUMBER

Pankaj Kuma, Santosh Kumar Singh*
Department of Mechanical Engineering, SRM Institute of Science and Technology, Kattankulathur – 603203, India
** Corresponding author. Mob: +91 9433054519.*
E-mail address: fmsks84@gmail.com (S. K. Singh).

Abstract
The paper aimed to numerically study the flow structure at the wake of the bluff body with altered blockage ratio (BR) keeping the fixed aspect ratio (AR). The study was conducted by finite volume technique by means of commercial software ANSYS-FLUENT. The CFD analysis of the bluff body is mainly subjected to the lower subcritical Reynolds number range (5000 to 15,000) along with blockage ratio as an important factor. The flow parameters such as drag coefficient, pressure, and kinetic energy variations are analysed here with respect to the Reynolds number (Re) and BR. It was observed that at fixed Re, the drag coefficient (C_D) reduce with an increase in the BR while decrease with increasing Re for a fixed value of BR.
Keywords: Blockage ratio; Bluff body; Drag; Reynolds number

1. INTRODUCTION

The understanding of flow over bluff bodies has been a topic of substantial significance. It has implication to a number of engineering fluid flow situations in a different branch of engineering. The major characteristic of flow over bluff body is the development of a turbulent wake with a recirculation, which has a principal effect on the drag acting on the bluff-body. This type of flow structure is characterized by a flow separation zone, followed by turbulent eddies in the wake flow zone. A large number of numerical and experimental studies are accessible on flow over a cylinder. For example, Obasaju [1] studied the effect of cylinder orientation at high Re using hotwire anemometry and suggested a reduction in drag coefficient at a lower angle of attack. Lee and Budwig [2] studied experimentally, the effect of AR for a circular cylinder at low RE. They suggested that a different Strouhal number (Sh) value for AR > 60 in the different range of Re (64 < Re <130). Szepessy and Bearman [3] studied the upshot of AR for flow past a cylinder at high Re and suggested that the wake flow is 2D. The effect of AR for flow past a circular cylinder at wide range of Reynolds numbers was studied by Norberg [4]. The Re was varied from laminar to transition and finally turbulent flow. The constant vortex shedding was observed for AR > 40 and was delayed for lower ARs. In addition, at AR > 100, the Sh was observed to be sovereign of AR. The effect of blockage was investigated numerically by Behr et al. [5], Anagnostopoulos et al. [6], and Turki et al. [7]. It is shown that with increasing blockage parameter the Strouhal number and drag coefficient increase, while the base suction and stagnation pressure coefficients increase. Recently, Mallick and Kumar [8] suggested that the drag force increases with an increase in the diameter of the cylinder. Also, for a cylinder of a particular diameter, drag force has been found to increase with an increase in air velocity.

In summary, thus, while a large amount knowledge is accessible on the drag, vortex shedding, transition from one flow regime to another, etc., little is known about the extent of blockage at lower subcritical Reynolds number on drag and wake characteristics for a circular cylinder. The present study is an attempt to investigate the effects of blockage ratio at lower subcritical Reynolds number range (5000 to 15,000). In the present study the simulation is executed on a 2D mesh construction of the cylinder with varying blockage ratio having a foremost anxiety for pressure, drag and kinetic energy analysis. The analysis of flow at the wake of the bluff body is done by a assortment of pressure, velocity intensity in 2D and turbulent kinetic energy intensity data up to 3 times the diameter of the cylinder especially at the wake of the cylinder.

2. NUMERICAL METHODOLOGY

The objective is to simulate fluid flow over a cylinder in 2D arrangements at the centre axis of a limited surrounding. **Figure 1** represents the designed geometry based on which the mesh is generated for various test cases. Origin of the space is selected as the centre of the cylinder for convenience in referring to wake region extent. The side AC on left state inlet while side BD on right state outlet, and y-axis and *x*-axis symbols point in their respective positive direction. To determine the cylinder diameter, three blockages (6, 10 and 20 %) of the flow region is considered here. For example, in the case of 10 % Blockage, the diameter of the cylinder is 6 mm ($1-\frac{(60\times60)-(60\times d)}{60\times60}=10\%$). The similar procedure was used for the calculation of diameter for the blockages 6 and 20 % and the respective diameters are 3.6 and 12 mm.

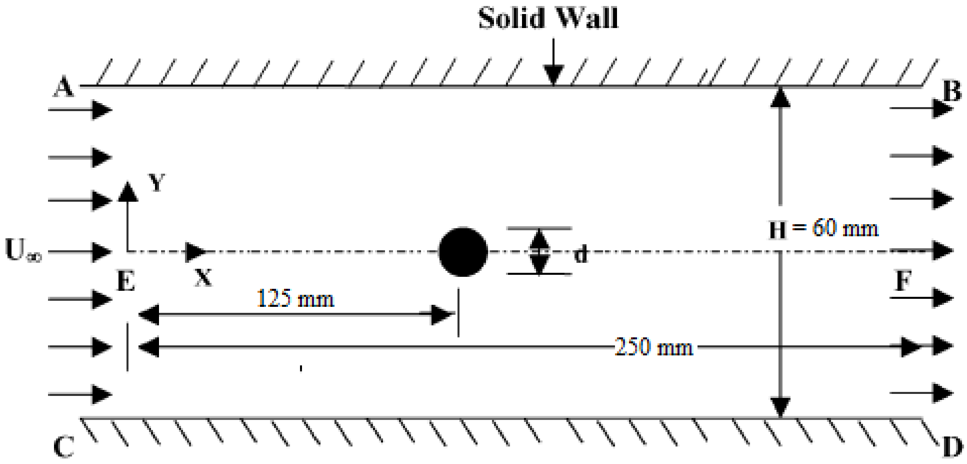

Fig 1 Schematic of the numerical computations domain

The inlet velocity was determined by the respective Reynolds number, $\mathrm{Re} = \frac{L \times U}{\upsilon}$, where, L, U and υ are the characteristics length, inlet flow velocity and kinematic viscosity respectively. For Re = 5000, the obtained velocities are 0.833, 0.416, and 1.388 m/s for the BR = 10, 20, and 6 % respectively. The ICEM ANSYS software was used to developed 2D mesh for various cases. Closed to the cylinder, O-grid arrangement mesh is structured for the improvement in simulation results. The developed unstructured 2-D mesh was exported for all conditions and simulated in ANSYS Fluent. The boundary conditions are inlet uniform flow (U = 1.0, V = 0.0), at the cylinder surface U = 0.0, V = 0.0, top and bottom are consider as a symmetry boundary condition and outlet boundary-continuative boundary condition can be expressed as (P = 0.0). Directly a 'fluent.msh' of the created unstructured mesh as above is imported in the setup module of fluent. Double precision computer number format option is selected which use 64-bit for each floating point and hence increases precision. Pressure based type solver is chosen with transient time setup and absolute velocity formulation in planar coordinate system 2D space. SST K-omega viscous model is used for defining the CFD transformed basic equations. Points are created in the wake region behind the cylinder shape. Each point is created with an increment equal to that of the cylinder radius in both x and y-direction as shown in **Figure 2**. Surface report definitions are created for extracting absolute pressure, turbulence kinetic energy, and x-velocity on these points in wake region. Surface report definition windows are shown in **Figure 2** The calculation at each node point is done for a maximum of 200 iterations for the solution to converge over a time lapse of 10^{-1} order seconds unit by multiplying the number of time steps over a time step size of 0.0001. The coefficient of drag and lift (C_D and C_L) values over cylinder surface are obtained through force reports by changing direction vector post-simulation.

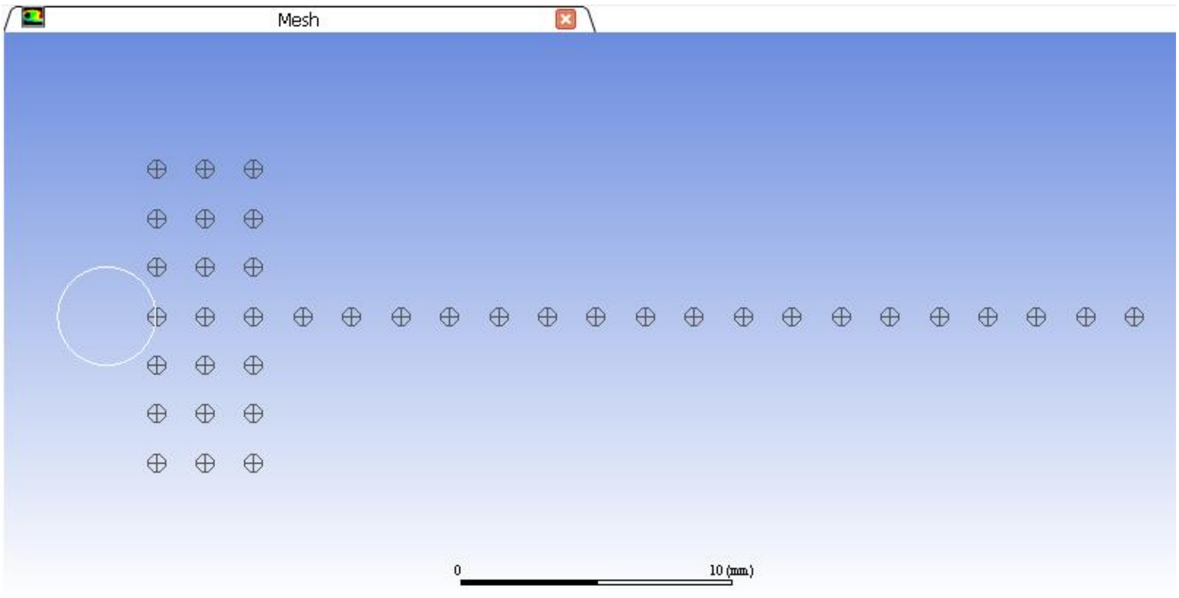

Fig 2 Points created for surface reports

2.1 Grid Independence

In the present study the mesh density was investigated to make sure that the obtained result not depended on the mesh nodes. The mesh node and the drag and lift coefficient are presented in **Table 1** for the grid independent test. Further, the variation of C_D and C_L in the **Figure 3** is presented here against number of nodes used in the mesh. The mesh at minimum no of node at which the C_D and C_L is constant was selected as a grid independent mesh. It is founded that a mesh with 31700 nodes is finest for the present study.

Table 1 Variations of C_D and C_L with number of nodes Reynolds number 5000

Nodes	C_D	C_L
35220	1.480763	1.07589
31700	1.465363	1.06451
24118	0.834492	0.028372
21168	0.831992	0.027572
18418	0.828968	0.026525

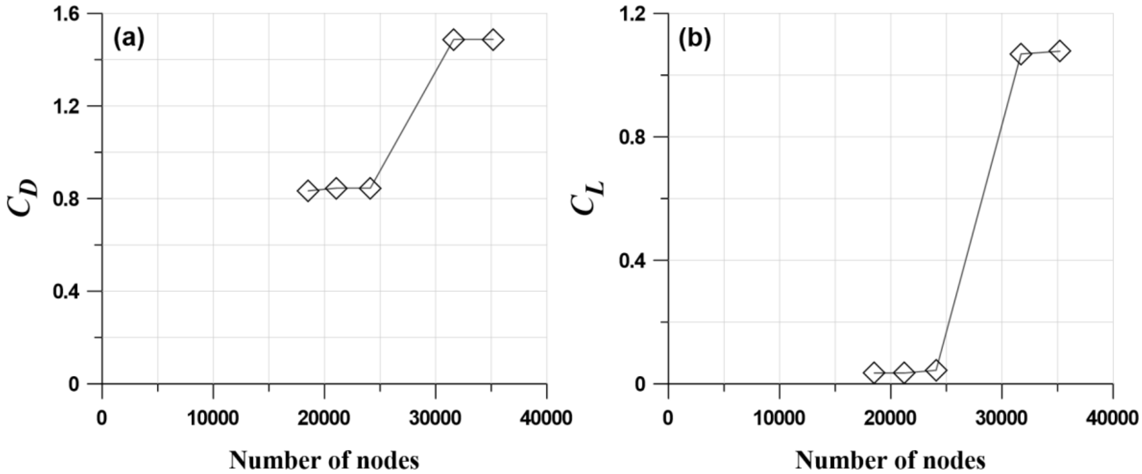

Fig 3 Variation of C_D and C_L with the number of nodes

3. RESULT AND DISCUSSIONS

Pressure variation is significant in the study of flow over the circular cylinder. Pressure varies accordingly with the vortices motion in the locality of the cylinder. Flow detaches alternately about symmetrical bodies to form vortices around the cylinder. This typically initiates intermittent energies on the body due to the pressure varies. This circumstance is mainly important inflow linking fluid and structure interaction such as the flow around a tall building or suspension bridge. Even though pressure persuaded force does not influence the simulation on a fastened cylinder very much. Vortex structure and evolution provoke forces on the bodies wrapped in the flow. A vortex produces a negative pressure suction area nearby to the surface where it progresses. Thus, the analysis of pressure variation is imperative in the study of the aerodynamic forces around a body. Therefore, here **Figure 4** is provided to shows the variation of pressure on the surface (x/r = 1.8, where r is the radius of the cylinder) and wake region (x/r = 3.6 and 5.4) of the cylinder. The intensity of pressure variation at the centre (y = 0) is very low particularly at the surface for Re = 15000. This is probably because the pressure allotment in the vicinity of the cylinder, flow energy is rather low due to viscous effects and thus, receptive to the amends of the pressure gradient. However, with low Reynolds number, the variation in pressure is uniform for all the selected x-locations. Further, it is observed (y = 0) that on the surface of the bluff body pressure is low compared to the other two locations but it increases with the change in position along the y-direction. For clear representation of pressure distribution, **Figure 5** shows the contour plot of pressure variation at a fixed aspect ratio for a different Reynolds number.

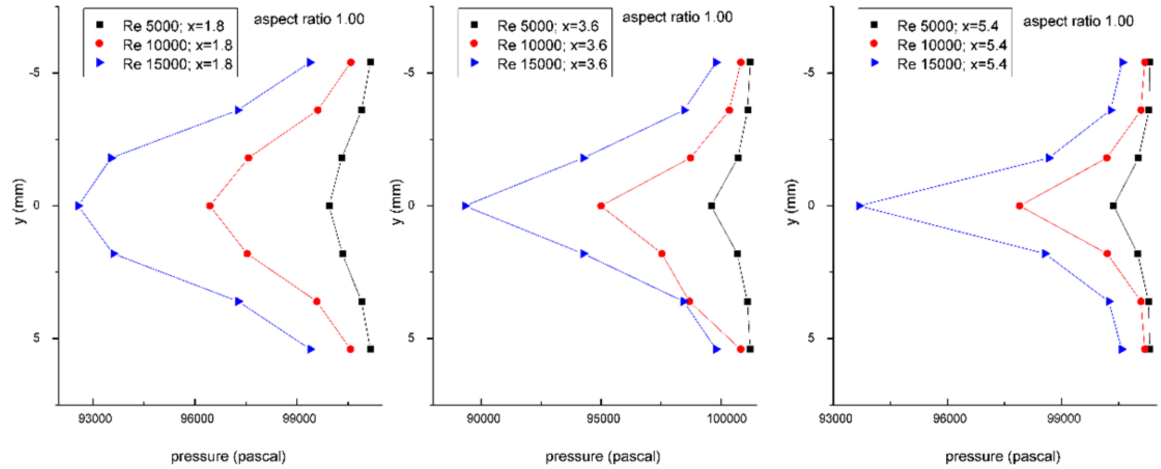

Fig 4 Variation of pressure at different *x* and Reynolds number keeping a fixed aspect ratio

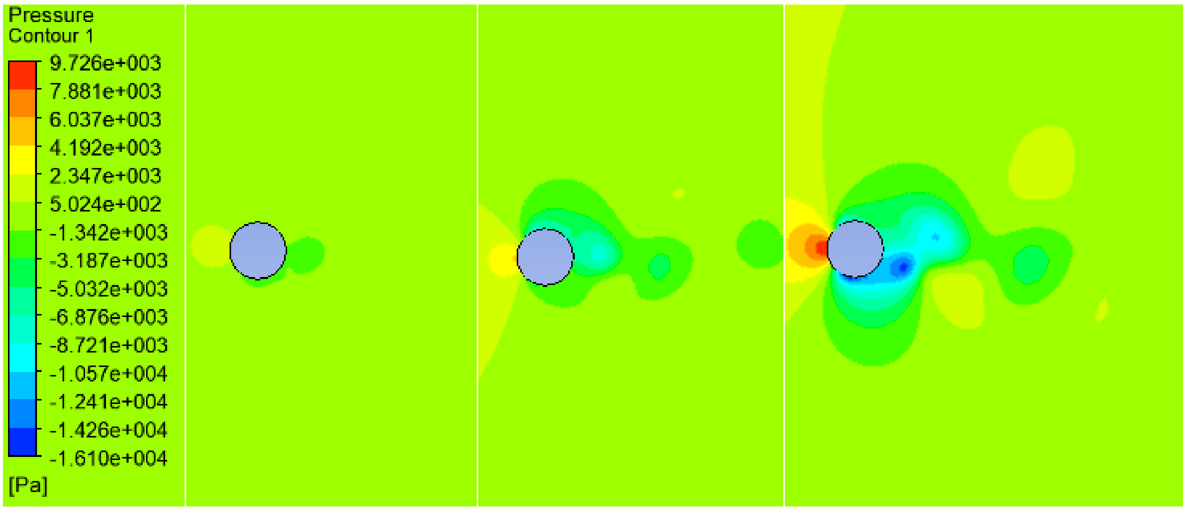

Fig 5 Contour plot of pressure variation at different Reynolds number keeping a fixed aspect ratio

Figure 6 shows the variation of pressure intensity at different blockage ratio keeping fixed Reynolds number and aspect ratio. It is observed from **Figure 6** that at 6% blockage ratio the intensity of pressure is minimum particularly at a distance of $x = 2r$ and 3r. With the increase in blockage ratio, pressure intensity increases successively which is similar for all three locations. This observation is clearly visible from the contour plot of pressure intensity as shown in **Figure 7**. Further, negative pressure is observed behind the cylinder at a blockage ratio of 10%. With the increase in pressure ratio, this negative pressure zone shifted a little away from the bluff body.

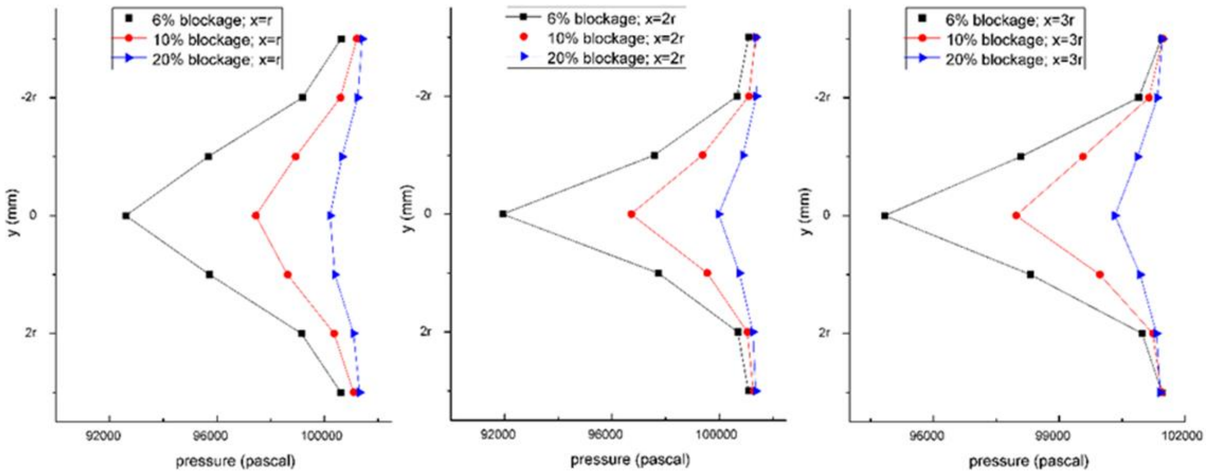

Fig 6 Variation of pressure at different blockage ratio keeping a fixed Reynolds number

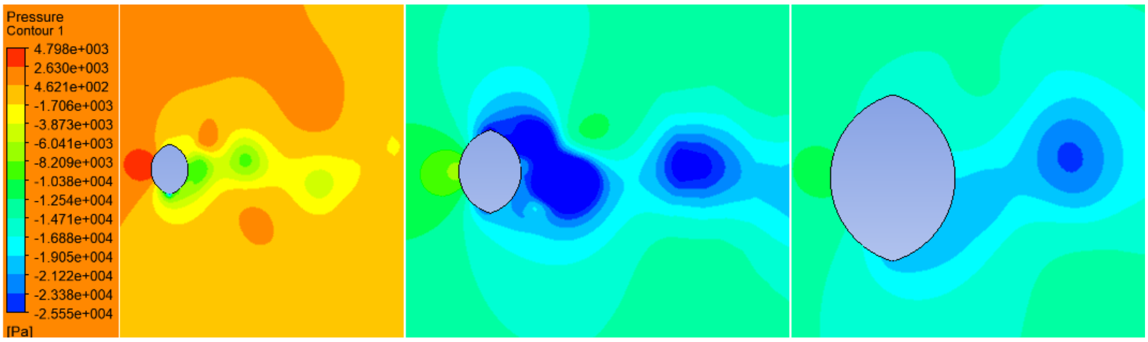

Fig 7 Contour plot of pressure variation at different blockage ratio keeping fixed Re

Figure 8 shows the variation of turbulent kinetic energy at different blockage ratio keeping fixed Re and AR. The variation of turbulent kinetic energy is maximum at $x = r$, particularly at 6% of blockage. It is to be noted here that on the surface of the bluff body a clear double peak is observed in the y-direction with equal in magnitude. With an increase in distance, the double peak value diminishes. It is also revealed from **Figure 8** that the TKE value decreases with an increase in blockage ratio as well as with increased distance from the bluff body. To shows the clear visualization of TKE distribution with blockage ratio keeping aspect ratio fixed contour plot is also plotted in **Figure 9**. The zone of TKE is bigger for a lower blockage ratio at $x = r$ and $2r$.

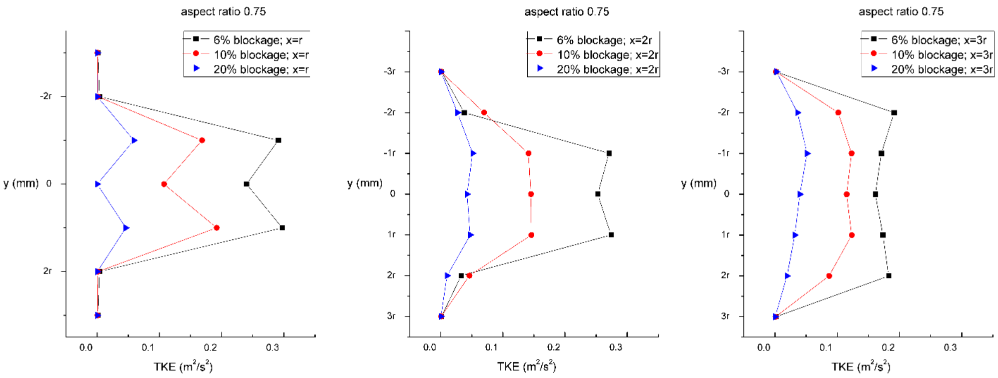

Fig 8 Variation of TKE at different BR keeping a fixed Re

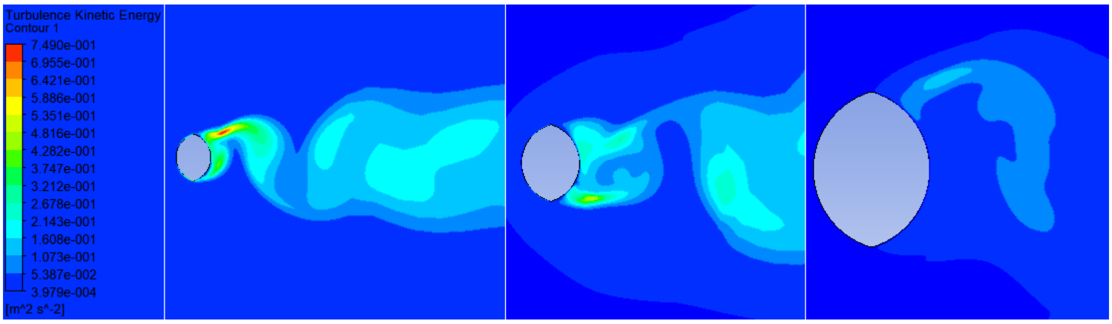

Fig 9 Contour plot of TKE variation at different blockage ratio keeping fixed Re

For a fixed value of blockage ratio, the reliance of drag coefficient on the Re is qualitatively dependable with the trends observed from the previous study (not shown here). Further examination of these results suggests that the drag force is maximum at low Reynolds number and it decreases with an increase in Reynolds number. Further, as the blockage ratio changed the C_D also changes and the minimum value of C_D is observed at a higher blockage ratio.

Figure 10 shows the variation of the drag coefficient with different aspect ratio and Reynolds number. It can be seen from **Figure 10** that the drag coefficient gradually decreases with an increase in Reynolds at aspect ratio 0.75 and 1 and it becomes steady for aspect ratio 1.50. In fact, slightly variation is observed at low aspect ratio with a change in the Reynolds number i.e., it appears that at higher aspect ratio the effect of Reynolds number is insignificant at a particular location. Drag coefficient decrease with an increase in Reynolds number for each aspect ratio. Aspect ratio affects drag more prominently than Reynolds number. For aspect ratio 1.5 Reynolds number has an almost null effect and for that of aspect ratio, 1 and 0.75 decrease gradient increases with Reynolds number. It is observed from **Figure 6-8** that the lift coefficient is maximum for aspect ratio 0.75 and it decreases significantly with an increase in Reynolds number. Aspect ratio 0.75 and 1.5 exhibit a continuing initial increase and decrease relation with increasing Reynolds number, whereas 0.75 having a more clear difference. However, aspect ratio 1 indicates a gradual decrease with an increase in Reynolds number.

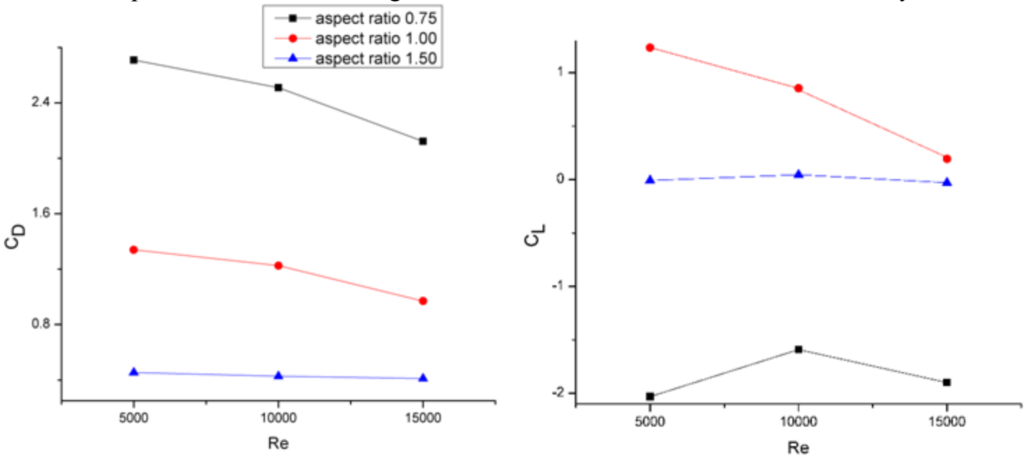

Fig 10 Variation of drag coefficient with different aspect ratio and Reynolds number

4. CONCLUSIONS

The objective is study the flow characteristics numerically at the wake of the circular cylinder with different blockage ratio and Re. This study was made with help of finite volume technique using commercial software ANSYS-FLUENT at Re = 5000 10000 and 20000, along with blockage ratio as an important factor. The basic results lead to the following findings:

- It is observed that the drag coefficient, C_D reduced with an increase in *Re* for a fixed value of aspect ratio.
- The lower magnitude of pressure which correlated with the wake zone of the cylinder is slowly diminishes with an increase in BR.
- At AR = 1, the drag coefficient has a constant value especially for initial BR and thereafter, C_D raises linearly with further enhance in BR.
- It was observed that pressure is well associated with mean TKE and it reorganizes the flow intensity.

DECLARATION OF CONFLICTING INTERESTS

The authors would like declare that no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

FUNDING

The authors received no financial support for the research, authorship, and/or publication of this article.

REFERENCES

[1] Obasaju ED. An investigation of the effect of incidence on the flow around a square section cylinder. Aeronaut. Q, 1983, 34, 243–259.
[2] Lee T, Budwig R. A study of the effect of aspect ratio on vortex shedding behind circular cylinders. Phys. Fluids A, 1991, 3(2), 309–315.
[3] Szepessy S, Bearman PW. Aspect ratio and end plate effects on vortex shedding from a circular cylinder.” J. Fluid Mech., 1992, 234, 191–217.
[4] Norberg C. An experimental investigation of the flow around a circular cylinder: Influence of aspect ratio. J. Fluid Mech., 1994, 258, 287–316.
[5] Behr M, Hastretier D, Mittal S, Tezduyar T. Incompressible Flow past a circular cylinder: dependence of the computed flow field on the location of the lateral boundaries. Comput. Meth. Appl. Mech. Engg., 1995, 123, 309-316.
[6] Anagnostopoulos P, Illiadis G, Richardson, S. Numerical study of the blockage effects on viscous flow past a circular cylinder. Int. J. Numer. Methods Fluids, 1996, 22, 1061-1074.
[7] Turki S, Abbassi H, Nasrallah SB. Effect of the blockage ratio on the flow in a channel with a builtin square cylinder. Comput. Mech., 2003, 33, 22–29.
[8] Mallick M, Kumar, A. Study on Drag Coefficient for the Flow Past a Cylinder. Int. J. Civ. Eng., 2014, 5, 301-306