EXPERIMENTAL AND NUMERICAL ANALYSIS OF DIFFERENT AERODYNAMIC PROPERTIES OF CIRCULAR CYLINDER

Kartik Chandra Bhagat[1], Subrato Kumar Soren[2], Sunil Kumar Chaudhary[3]
[1]Post Graduate Student, Department of Mechanical engineering, BIT, Sindri, Dhanbad, India
[2,3]Assistant Professor, Department of Mechanical engineering, BIT, Sindri, Dhanbad, India
kcb181091@gmail.com[1], sksoren2009@gmail.com[2], skchaudhary74@gmail.com[3]

---***---

Abstract *A sub-sonic wind tunnel testing machine is a tool used in various research of aerodynamics to study the effect of air moving past to the solid objects for laminar flow field (Re < 2×10^5). A circular cylinder generates a high mean drag and large fluctuating forces. In this research work the experimental and numerical investigation has been carried out to find drag force and pressure coefficient of different aspect ratio (L/D) circular cylinders (diameter 7.5 & 10 cm, length 65 cm). Qualitative technique of fluid flow separation point and quantitative technique of static pressure measurements have been used to obtain various characteristics of flow past over circular cylinder at different wind speed of 8.35, 9.27, 10.23 m/s. The concept of the pressure coefficient is illustrated and explained. Plotted graph between coefficient of pressure and angle of rotation produces very similar curves to those in the research literature. Variation of drag coefficient with increasing air velocity is also demonstrated. Numerical analysis is done using CAE software ANSYS FLUENT 14.5 to study pressure distribution on the periphery of circular cylinder, flow separation point and also to compare drag force. The cylindrical model and computational domain is prepared in ANSYS 14.5 workbench and pre-processing is done in Viscous-laminar model.*

Keywords - *circular cylinders, coefficient of pressure, drag coefficient, drag force, laminar flow, Reynolds Number (Re), sub-sonic.*

1. INTRODUCTION

The flow characteristics over a cylinder are one of the most applicatory topics of research. The cylinder subjected to low Reynolds number experiences flow separation, and oscillations in the wake region. The periodic nature of the vortex shedding phenomenon can sometimes results to unwanted structural vibrations, which leads to structural damage. Many studies had been carried out to evaluate drag force [1], investigated the range of Reynolds number 6×10^3 to 5×10^6 effect of relative roughness on drag for the circular cylinder [2], the drag coefficient of flow for various aspect ratios circular cylinder were calculated from data obtained by performing tests on an air flow bench [3]. The separation of particles of various sizes and shapes depend upon variations in the behavior of particles when subjected to the action of moving fluids [4]. The separation angles for smooth cylinder by experimentation calculation are found to be around 80~90° in either side of the cylinder from the upstream stagnation point [5]. The general aerodynamic objective for most sub-sonic wind tunnels is to obtain a flow in the test section that is assumed to be a parallel steady flow with a uniform speed throughout the test section. Viscous flow behaviour in the upstream half of the cylinder differed from that on its downstream half and the influence of the magnitude of the Reynolds number on the ability of the viscous flow to recover pressure on the downstream side of the cylinder [6]. The fundamental laws utilized in modeling low speed aerodynamic flows include mass conservation, force and motion relating to the Newton's Second Law and energy exchanges governed by the First Law of Thermodynamics. In considering low-speed flows, the assumption of incompressible flow is often adopted. Flow past a circular cylinder for Re=10^0to10^7 done numerically by solving the unsteady incompressible two dimensional Navier-Stokes equations and they described the shear layer instability and drag crisis phenomena [7].

Three dimensional simulation are performed using the CAE software ANSYS FLUENT. Computational fluid dynamics (CFD) is a tool to analyze the systems involving fluid flow, mass transfer, heat transfer, chemical reactions and associated phenomena by solving mathematical equations which govern the processes using a numerical method by means of computer based simulations. The effects of lift and drag on usually two-dimensional cylinders which is useful to describe the variation of numerical results between 2D and 3D analysis [8]. The fundamental basis of any CFD problem is the Navier-Stokes equations, which define any singlephase fluid flow. These equations can be interpreted by removing terms defining viscosity to get Euler equations.

2. EXPERIMENTAL SETUP

Experiments have been done using the sub-sonic wind tunnel at Birsa Instiitute of Tecchnology Sindri, Dhanbad. The wind tunnel has the test section of size 150 cm×60 cm and speed of range 8 to 10.23 m/s. It is an open circuit continuous flow wind tunnel in which air is sucked by the four bladed propeller located at the aft section of the tunnel which is driven by a 400/440 V, 15 HP induction motor. Experimental set-up of sub-sonic wind tunnel can be seen in figure 2.2 and schematic sketch showing the details of the tunnels is shown in figure.2.3. During the present investigation, tests have been made at three different free stream velocity of 8.35, 9.27, 10.23 m/s. Two circular cylinder having diameter (D= 7.5 &10 cm) and having the length of 65 cm are used. The cylindrical model is shown in figure 2.1 and details of working physical parameters are listed in Table.1. For the measurement of static pressure on the periphery of cylindrical surface 9 small holes are made at 40^0 interval, the pressure ports were connected to the multi-tube inclined manometer which is inclined at an angle of 33^0. The static pressures were measured using multi tube water manometer. Pitot tube is used to measure flow velocity. It is based on the principle that if the velocity of flow at a point becomes zero, the pressure there is increased due to the conversion of the kinetic energy into pressure energy. The velocity is determined by measuring the rise of liquid in the tube.

Fig 2.1: Experimental cylinders

Table 1: Working Parameters

S.NO	Parameters	Values
1.	Air density	1.225 kg/m^3
2.	Barometric pressure	1.03×10^5 N/m^2
3.	Dynamic viscosity	1.81x10^{-5} kg/m-s
4.	Gas constant	287.2

Fig 2.2: Experimental set-up of open air circuit sub-sonic wind tunnel

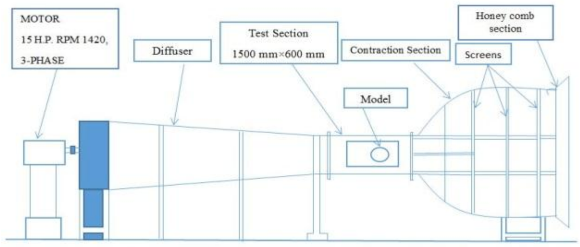

Fig 2.3: A schematic sketch showing the details of open air sub-sonic wind tunnel circuit

3. METHODOLOGY

1. The mass contained between any two steam lines remain constant throughout the flow field. We can use Bernoulli equation to relate the pressure and velocity along the stream line since no mass passes through the surface of cylinder.

According to Bernoulli equation:

$$\frac{P_{atm}}{\rho_{air}} + \frac{1}{2}V_0^2 + gZ_0 = \frac{P_\infty}{\rho_{air}} + \frac{1}{2}U_\infty^2 + gZ_\infty \quad (1)$$

$$U_\infty = \sqrt{\frac{2(P_{atm} - P_\infty)}{\rho_{air}}} \quad (2)$$

If inclined ($\theta^1 = 33^0$) water manometer is used then:

$$U_\infty = \sqrt{2\frac{\rho_w g \Delta h \sin\theta^1}{\rho_{air}}}$$

$$U_\infty = \sqrt{\frac{2 \times 1000 \times 9.81 \times \Delta h \sin\theta^1}{1.225}}$$

$$U_{\infty} = 126.55\sqrt{\Delta h \sin\theta^{1}}\ m/s \quad (3)$$

2. PRESSURE COEFFICIENT AND DRAG COEFFICIENT BY PRESSURE DISTRIBUTION METHOD

Pressure coefficient is denoted by C_P. In order to compare the variation of pressure around a bluff body for a variety of flow conditions, it is conventional to use a dimensionless ratio called the pressure coefficient which compares the pressure on the surface of the cylinder, P_S, to that at infinity, P_∞. Mathematically,

$$C_P = \frac{2(P_s - P_\infty)}{\rho U^2} = \frac{P_s - P_\infty}{P_0 - P_\infty} \quad (4)$$

If the shear is neglected because the pressure force is found to be dominated then coefficient of drag (C_D) can be calculated by:

$$C_D = \int_0^{2\pi} C_P \, Cos\theta \quad (5)$$

3. BOUNDARY LAYER SEPARATION

A fluid acts much the same way when forced to flow over a curved surface at high velocities. At sufficiently high velocities, the fluid stream detaches itself from the surface of body. This is called **flow separation.** For circular cylinders flow separation occurs at about **θ= 80°** (measured from the front stagnation point of a cylinder) when the boundary layer is laminar and at about **θ = 140°** when it is turbulent.

4. COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS

4.1 Geometry generation

Geometry of cylinder and computational domain (150 cm× 60 cm× 65 cm) is modeled in design modeler of ANSYS FLUENT. An outline of cylindrical geometry can be seen in figure 4.1.

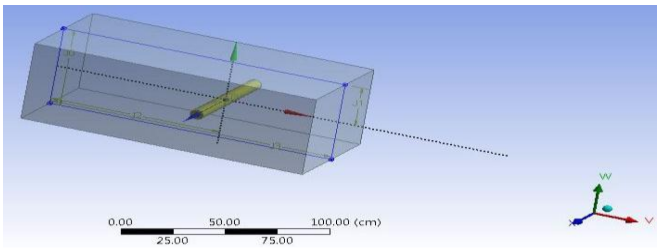

Fig 4.1: Computational domain (150cm × 60cm × 65cm)

4.2 Grid generation

The computational domain is sub-divided into small element or cell called grid. The generated mesh helps in solving the governing equation flow at each cell or nodal points. Hex dominant meshing method is adopted. The quality of mesh is generated during report quality check in solver. The number of nodes and elements created are 5192 & 6107 respectively. Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality. Minimum Orthogonal Quality = 6.97807e-02 and maximum Aspect Ratio = 6.54104e+01 is obtained.

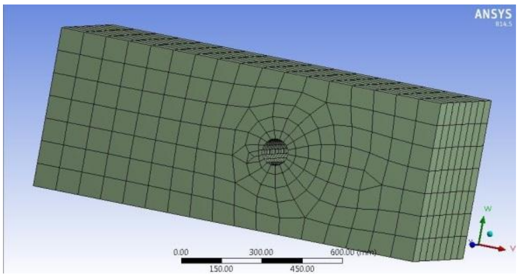

Fig 4.2: Hex Dominant mesh

4.3 ANSYS FLUENT Analysis

The pressure distribution on the periphery of cylindrical model is obtained at three different air velocity 8.35, 9.27, 10.23 m/s.

4.3.1 Pressure distribution contour

In all the pressure contours, distribution of pressure with varying velocities are shown on the periphery of cylinder surface & it is clear that pressure is maximum at front surface of cylinder that is at stagnation point.

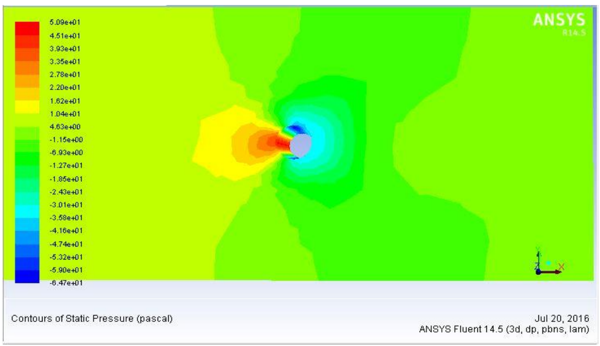

Fig 4.3: pressure contour at V=8.35m/s for D=7.5 cm

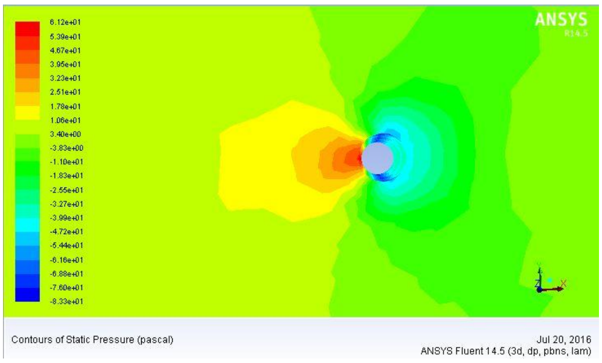

Fig 4.4: pressure contour at V=9.27m/s for D=7.5 cm

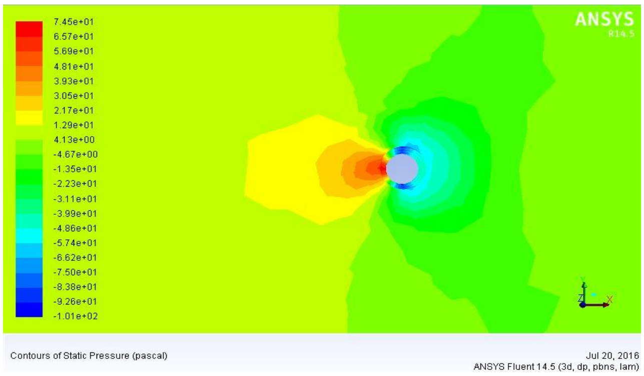

Fig 4.5: pressure contour at V=10.23 m/s for D=7.5 cm

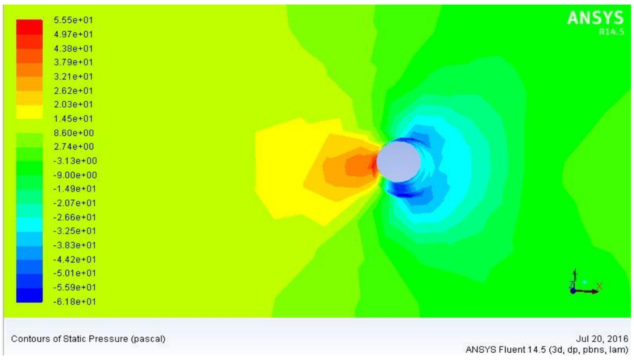

Fig 4.6: pressure contour at V=8.35m/s for D=10 cm

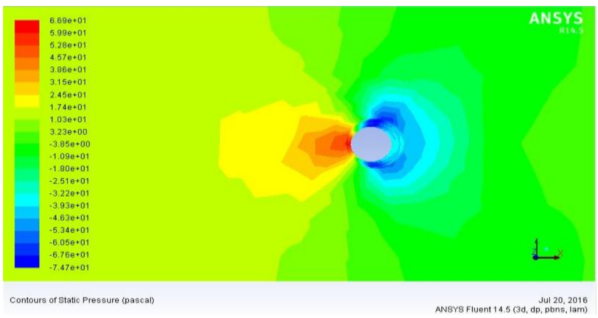

Fig 4.7: pressure contour at V=9.27m/s for D=10 cm

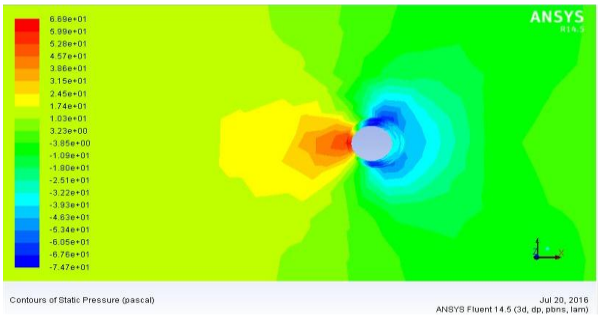

Fig 4.8: pressure contour at V=10.23m/s for D=10 cm

4.3.2 Velocity vector plots

The corresponding velocity vector plots are presented for the varying velocity of 8.35, 9.27, 10.23 m/s of varying diameter of the cylinder.

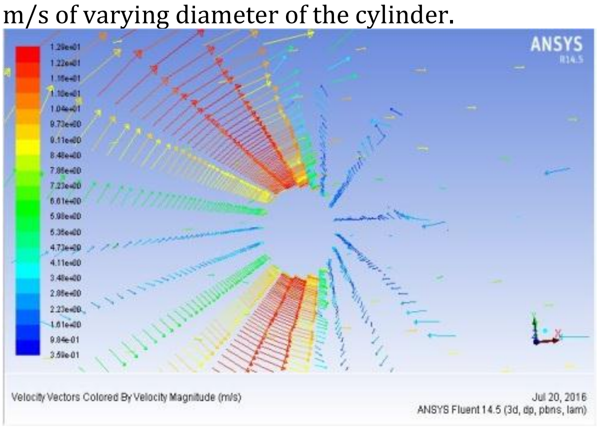

Fig 4.9: velocity vector plot on the periphery of cylinder having D= 7.5 cm

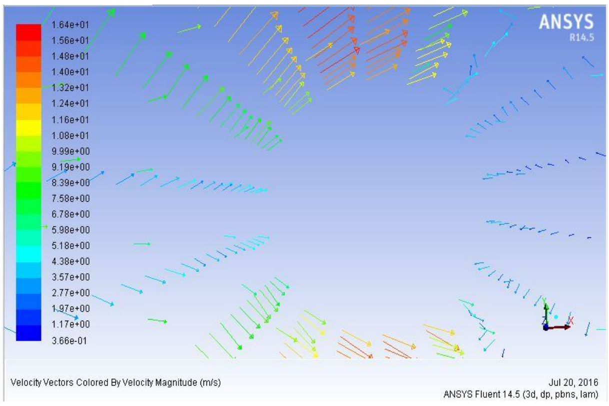

Fig 4.10: velocity vector plot on the periphery of cylinder having D= 10 cm

In velocity vector plots, the magnitude of velocity on the upper surface of the cylinder is found to be maximum and low velocity at the rear of the cylinder due to formation of wake. The velocities are zero at front surface of cylinder i.e at stagnation point shown in figure 4.9 and figure 4.10.

5. RESULT AND DISSCUSSION

With the help of Pressure distribution technique, the pressure coefficients are found and pressure distribution over the cylinders observed. The coefficient of pressure is plotted against different pressure points located at 40^0 intervals on the surface cylinder. Integrating area under this curve gives the C_D. In pressure distribution method, variation of pressure coefficient is occurred through different parameters such as velocity, diameter. C-oefficient of pressure versus angle of rotation graph signifies that fluid is separating between 90^0 and 100^0 shown in figure 5.1, 5.2 and 5.3 which is nearly similar to theoretical values for laminar flows.

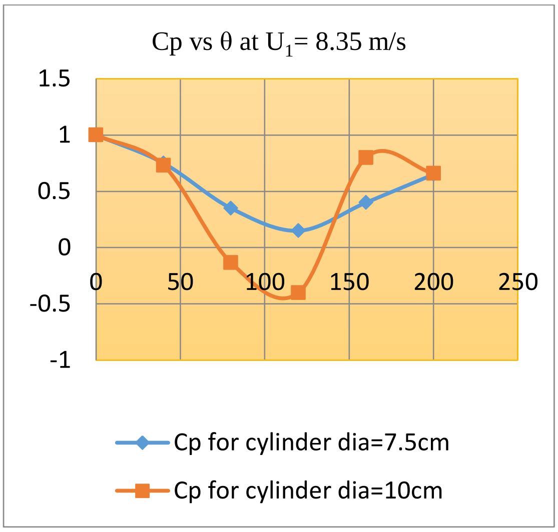

Fig 5.1: Variation of coefficient of pressure at V = 8.35 m/s

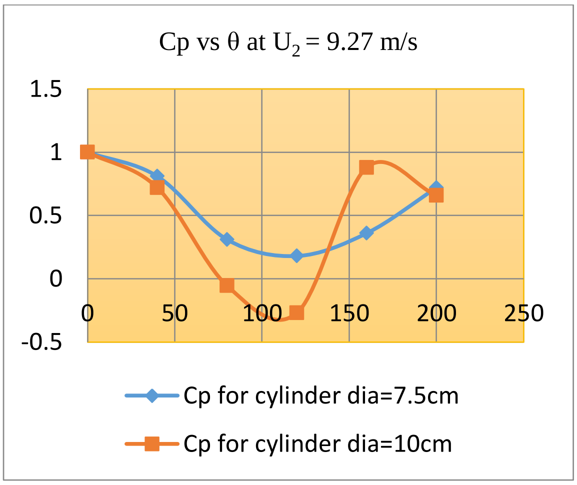

Fig 5.2: Variation of coefficient of pressure at V = 9.27 m/s

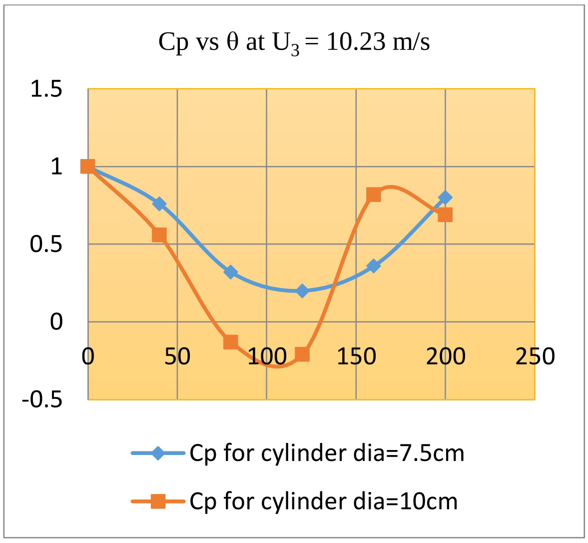

Fig 5.3: Variation of coefficient of pressure at V = 10.23 m/s

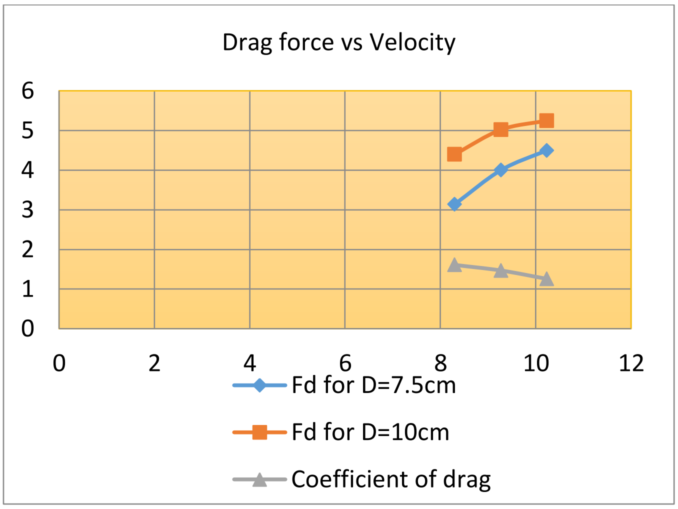

Fig 5.4: Variation of drag force and coefficient of drag with three velocities

Table 5.1: Numerical and experimental values for drag force in the cylinder surface region

Zone	Cylinder -wall				Drag force Error%
Velocity m/s	Force (N) for cylinder D=7.5cm		Force (N) for cylinder D=10cm		
	Num.	Exp.	Num.	Exp.	
8.35	2.33	3.14	3.41	4.4	13.57%
9.27	3.85	4.00	4.22	5.02	
10.23	3.47	4.47	5.13	5.25	

Drag force is increasing after air velocity 8.35m/s it is because drag force is directly proportional to velocity and also projected area (frontal area) for laminar flow field. **13.57%** error is noticed in drag force by comparing result experimentally and numerically.

6. CONCLUSION

The drag force, pressure coefficient and pressure distribution on the periphery of cylinder surface calculated by ANSYS FLUENT software, experimental method has achieved the agreeable and satisfied solution. Drag variation and its dependence on the flow and physical properties of cylinder.

- The nature of drag force is listed as it reacts to the different parameter considering the fluid properties and geometric properties of the cylinder.
- When the area of the cylinder is increased by increasing the diameter or length, the drag force found to be increasing.
- With the increase in velocity of the fluid flow, the drag increases but its coefficient gets reduced.
- The drag force gets increased when the Re value is increased, but the C_D drops down.
- The velocity profile observed explains the formation of boundary layer over the cylinder surfaces.
- The flow separation occurred as the consequence of distributed pressure over the cylinder was found at an angle 90^0-100^0.
- All the method employed are compared and found to be good and acceptable value with error percentage of 13.57%.

ACKNOWLEDGEMENT

The author wishes to acknowledge the assistance provided by the professors of Birsa Institute of Technology Sindri, Dhanbad for conducting experiments.

REFERENCES

[1] Chhabra R.P, Agarwal L. Sinha N.A, Drag on Non-Spherical Particles: an evaluation of available methods, Elsevier Science, 101(1999), pp. 288-295

[2] Achenbach, E., and E. Heinecke. "On vortex shedding from smooth and rough cylinders in the range of Reynolds numbers 6× 103 to 5× 106." Journal of fluid mechanics 109 (1981): 239-251.

[3] Monalisa Mallick, A. Kumar (2014) "Study on Drag Coefficient for the Flow Past a Cylinder," International Journal of Civil Engineering Research, vol.5, pp. 301-306.

[4] Gabitto J, Tsouris C., Drag coefficient and settling velocity for particles of cylindrical shape, Powder Technology, 183 (2008), pp. 314-322

[5] Toukir Islam and S.M. Rakibul Hassan "Experimental and Numerical Investigation of Flow over a Cylinder at Reynolds Number 10^5" Journal of Modern Science and Technology Vol. 1. No. 1. May 2013 Issue. Pp.52-60

[6] Libii, Josué Njock. "Using wind tunnel tests to study pressure distributions around a bluff body: the case of a circular cylinder." World Transactions on Engineering and Technology Education 8.3 (2010)

[7] Mittal, Sanjay, and Saurav Singh. "Vortex-induced vibrations at subcritical Re. "Journal of Fluid Mechanics 534 (2005): 185-194.

[8] Mittal, R. and S. Balachandar."Effect of three-dimensionality on the lift and drag of nominally two dimensional cylinders." Physics of Fluids (1994-present) 7.8 (1995): 1841-1865.

[9] Cengel Y.A, Cimbala J.M, Fluid Mechanics: fundamentals and application, New York, Tata McGraw-Hill Education, 2010