International Journal of Engineering Research-Online
A Peer Reviewed International Journal
Articles available online http://www.ijoer.in

Vol.3., Issue.5., 2015
(Sept.-Oct.)

REVIEW ARTICLE

ISSN: 2321-7758

TWO DIMENSIONAL FLOW ACROSS A CIRCULAR CYLINDER: A REVIEW

ABDULLAH SALEEM
PG Student, Department of Chemical Engineering
Aligarh Muslim University, Aligarh

ABDULLAH SALEEM

ABSTRACT

Over last five decades, the flow across bluff bodies has been a centre of intense research largely due to its wide range of applications. The most common type of bluff body which is encountered in engineering problem is circular cylinder. This review article will provide an insight of various parameters which play a crucial role in flow past a circular cylinder. Flow around cylinders is studied from a number of perspectives such as pressure distribution, force coefficients (lift and drag), vortex shedding, flow patterns, Strouhal number, Nusset number. The variation of these parameters with Reynolds number is investigated either experimentally or by using simulation software. The numerical simulation is carried out by using CFD solver (FLUENT), which utilizes finite volume method (FVM) for solving model equations. Flow patterns are represented by means of instantaneous streamlines, vorticity magnitude, velocity magnitude and pressure contours. Heat transfer results are presented by the variation of Nusset number with time for the given range of Reynolds number.

Keywords—Drag and lift coefficients, Finite Volume Method, Velocity Contour, Reynolds number, Nusset number, Strouhal number.

1. INTRODUCTION

Numerous issues related to hydrodynamics and aerodynamics are studied extensively because of their practical applications. Many of these problems are placed in category of incompressible flow and in many applications flow is unsteady. Flows around blunt objects such as circular cylinders are widely encountered in engineering problems. Examples of such application are cooling tower, offshore risers, sensors, chimneys, hot wire anemometer, heat exchanger tubes, cable in cable-stayed bridge, bridge pier, pipeline under ocean, solar heating systems, natural circulation boilers, nuclear reactor, etc.

There are other bluff body shapes such as sharp edged rectangular cross-section cylinders, square cylinders, etc. But the circular cylinder is the most widely used shape in engineering making appearances in chemical, civil, electrical, mechanical, offshore, aeronautical, nuclear, and wind engineering. Hence, study of flow across a circular cylinder is vital from researcher point of view. Most of these researches are based on study of various parameters like drag and lift coefficients (C_d, C_l), vortex shedding patterns, Strouhal number (St), Nusset number (Nu). The effect of variation of Reynolds number (Re) is observed on these parameters. The range of Reynolds number determines whether flow is laminar or turbulent.

When flow passes over a bluff obstacle, wake formation takes place in the downstream due to flow separation and formulation of recirculation zones. At a very low Reynolds number ($Re < 1$) the flow across a cylinder is steady and symmetrical in both upstream and downstream. As Reynolds number increases, the upstream and downstream symmetry disappears leading to the formation of two attached eddies behind cylinder. The size of these eddies becomes bigger with increase ofReynolds number. When Reynolds Number further increases (Re > 45), unsteadiness arises spontaneously despite of all operating conditions being held steady. This leads to the appearance of vortex shedding behind circular cylinder [1]. The vortex shedding causes drag force to increase and the body suffers from a periodic forcing in the normal direction to the main stream.

Gandikota and Amiroudine [2] studied the effect of aiding and opposing buoyancy on two dimensional laminar flow and heat transfer across a circular cylinder for various Reynolds number (Re =50-150). The critical Richardson number (Ri) is found to increase with Reynolds number in chosen range for both confined and unconfined cases. The Nusset number increases with increase in Reynolds Number. Also, the Nusset Number increases at a faster rate beyond critical Richardson Number, whereas for negative values of Ri, it almost remains constant.

Igor M. Kozlov et al. [3] simulated two dimensional flow across a circular cylinder by using RES for Reynolds number range from 5 to 200. In this method, fast fourier transformation (FFT) is used for solving the Poisson equation in rectangular meshes.

Bai and Li [4] investigated hydrodynamic characteristics of a circular cylinder in two-dimensional unsteady uniform cross flow and simulated numerically by the laminar model using FLUENT software. The pressure distribution, drag and lift coefficient and Strouhal number is calculated at Re =200. The results showed that the vortex shedding of flow over a circular cylinder could be well controlled by placing the baffle at the downstream medial axis of the cylinder, which could reduce drag and resist vibration.

Golani and Dhiman [5] studied the effect of Reynolds number (Re = 50 -180) on drag and lift coefficients, Strouhal number and average Nusset number. The numerical simulation was carried out using CFD solver Fluent. They found that the rms drag and rms lift coefficients increase with Reynolds number. Shedding frequency of vortex and the average Nusset number also increase with increase in Reynolds number. They also obtained correlations for time-averaged drag coefficient and time average Nusset number as a function of Reynolds number for given range of conditions covered in study.

Park [6] investigated flow past circular cylinder in Reynolds number range of more than 160. It was found that stagnation pressure monotonically decreases as the Reynolds number increases, while the base pressure shows a non-monotonic behaviour. It means that as Reincreases, the base pressure increases in steady flow, but it decreases when flow becomes unsteady. Also, as Re increases, the total drag coefficient monotonically decreases, while the pressure drag coefficient increases in unsteady flow but decreases in steady flow, indicating that vortex shedding significantly increases the pressure drag.

Ding et al. [7] described an efficient method for simulating the two-dimensional steady and unsteady incompressible flows. The method was based on a hybrid approach, which combines the conventional finite difference (FD) scheme and the mesh free least square-based finite difference (MLSFD) method. Numerical simulation was carried out for the steady flow with Reynolds numbers of 10, 20 and 40, and unsteady flow with Reynolds numbers of 100 and 200.

2. GOVERNING EQUATIONS

The governing equations comprise of continuity equation, Navier-Stokes equation (x and y components for 2D problem) and energy equation. These equations are based on fundamental principle of conservation of mass, momentum and energy respectively [5].

Equation of Continuity:

$$\frac{\partial V_x}{\partial x} + \frac{\partial V_y}{\partial y} = 0 \qquad (1)$$

Navier-Stokes equation:

For x component

$$\frac{\partial V_x}{\partial t} + \frac{\partial (V_x V_x)}{\partial x} + \frac{\partial (V_y V_x)}{\partial y} = -\frac{\partial p}{\partial x} + \frac{1}{Re}\left(\frac{\partial^2 V_x}{\partial x^2} + \frac{\partial^2 V_x}{\partial y^2}\right) \qquad (2)$$

For y component:

$$\frac{\partial V_y}{\partial t} + \frac{\partial (V_x V_y)}{\partial x} + \frac{\partial (V_y V_y)}{\partial y} = -\frac{\partial p}{\partial y} + \frac{1}{Re}\left(\frac{\partial^2 V_y}{\partial x^2} + \frac{\partial^2 V_y}{\partial y^2}\right) \qquad (3)$$

Energy Equation:

$$\frac{\partial T}{\partial t}+\frac{\partial (V_x T)}{\partial x}+\frac{\partial (V_y T)}{\partial y}=\frac{1}{Pe}\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right) \qquad (4)$$

Here V_x,V_y are velocity components in x and y directions. T is the temperature and Pe stands for Peclet number.

3. MESHES AND BOUNDARY CONDITIONS

The meshes (grids) are created by using commercial software GAMBIT or ICEM CFD. The computational fluid domain can be either rectangular or circular. The mesh is dense near to the wall of circular cylinder, whereas in the outer zone, the mesh becomes sparser. Uniform incoming fluid flowing velocity is used for entrance boundary and at the exit boundary FLUENT assumes a zero diffusion flux for all variables. No slip condition at the interface between solid and fluid on the wall surface is assumed. Fig. 1 shows the rectangular fluid domain for a circular cylinder along with the meshes [8].

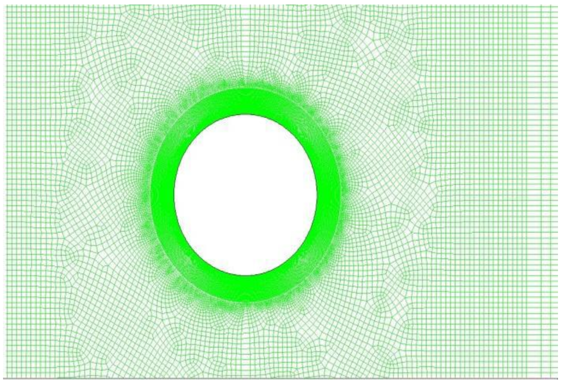

Figure1. Meshes and computational boundary condition around circular cylinder. Reproduced from Farhoud et al. [8]

4. NUMERICAL RESULTS

4.1 Pressure Coefficient Distribution

Fluid particles near the surface of wall experience resistance to the flow due to the adverse pressure gradient which acts in opposite direction to flow and also due to the wall friction. When the kinetic energy is not sufficient to sustain motion, some particles leave the surface of the cylinder, resulting in the formation of recirculation zones in the downstream of cylinder. Farhoud et.al [8] extensively studied the distribution of pressure coefficient and shear stress at the wall for Re= 150 at stagnation point (Θ=0 to Θ=360). Fig. 2 shows the plots for pressure distribution coefficient and wall shear stress. It is observed that the pressure coefficient and wall shear stress at zero and 180 at different vortex times have the same figure and do not vary at different times. The velocity gradient becomes zero at separation angle of about 116.5 which is called as separation point.At this point, flow momentum near the surface is not sufficient to overcome pressure gradient and continuing movement in downstream becomes impossible, because progressive stream prevent flow to return and boundary layer separation should takes place eventually leading to creation of wake in the downstream [9].

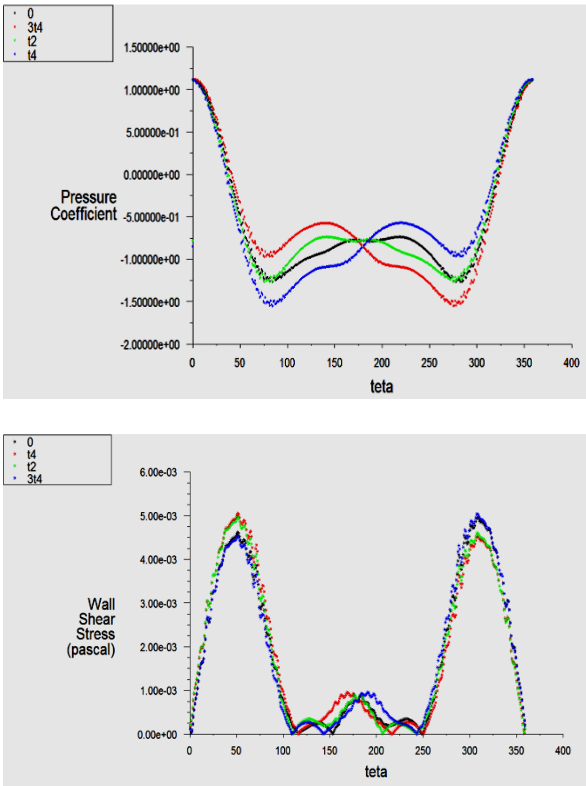

Figure2. Pressure coefficient and wall shear stress distribution at different vortex times and stagnation point (Θ=0 to Θ=360) [8].

4.2Drag coefficient and lift coefficient

The total drag coefficient consists of two components, i.e., skin friction drag (C_{DF}) and form drag (C_{DP}) coefficients. The skin friction drag is a result of the viscous forces acting on the body while the form drag is due to the unbalanced pressure forces on the body. The total drag coefficient is calculated by adding these two components [5]. Similarly total lift coefficient consists of friction lift (C_{LF}) and pressure lift (C_{LP}) coefficients. Farhoud et al. [8] found that for Re= 150, the lift coefficient (C_L) oscillates with larger amplitude than the drag coefficient (C_D). The drag coefficient varies almost twice as fast as the lift coefficient. The reason behind is that the drag coefficient is affected by vortex shedding process from both sides of the cylinder. This trend can be clearly observed in Fig. 3.

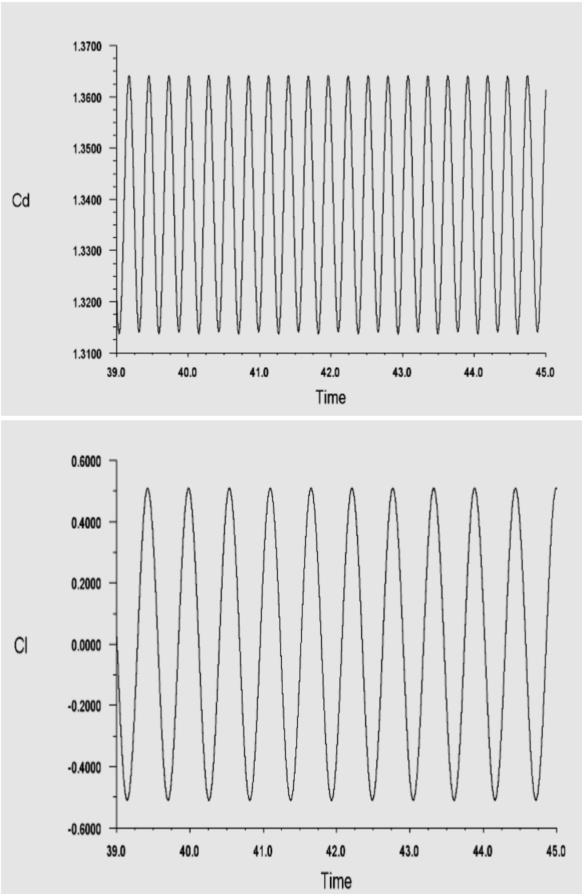

Figure3. The time variation of lift and drag coefficient for Re=150. Reproduced from Farhoud et al. [8].

Golani and Dhiman [5] studied the variation of time-averaged drag coefficient and rms (root mean square) value of drag and lift coefficients with Reynolds number. Fig. 4(a) and 4(b) represents this variation of time-averaged and rms value. It is found that the time-averaged drag coefficient decreases with increasing value of the Reynolds number. However, rms value of drag and lift coefficients are found to increase with increasing value of Reynolds number. But the magnitude of rms lift coefficient is one order higher than rms drag coefficient.

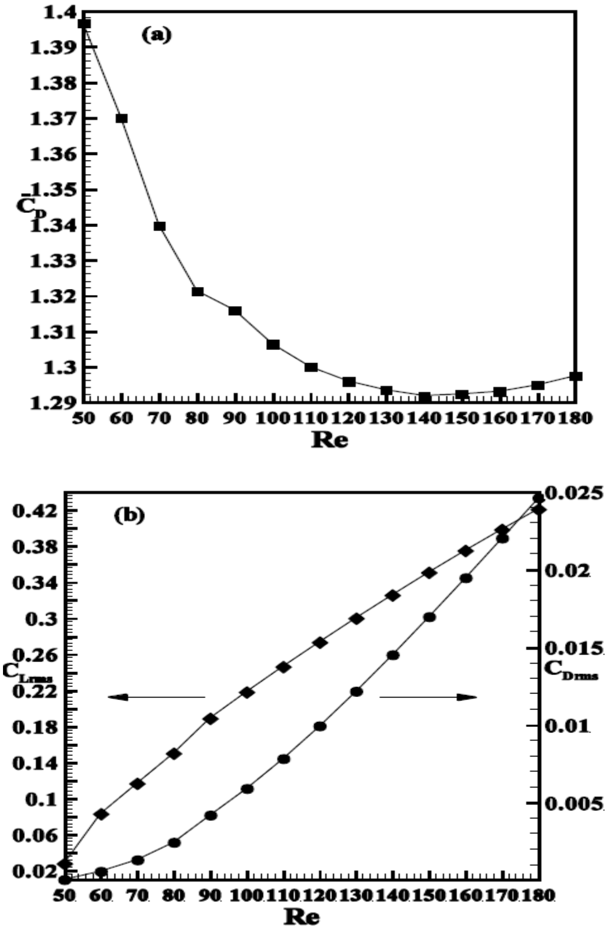

Figure4. Variation of (a) time-averaged drag coefficient and (b) rms value of drag and lift coefficients with Reynolds number. Reproduced from Golani et al. [5]

4.3 Strouhal Number

In unsteady periodic flow regime, vortices are shed by bluff body from one side, then from other side thus forming a vortex street. The frequency of shedding is expressed by a non-dimensional parameter called Strouhal number (St). It is given by St = fD/U, where f is the vortex shedding frequency, D is the diameter of cylinder and U is the free stream velocity.

Norberg [10] compiled a plot exhibiting variation of Strouhal number with a wide range of Reynolds number. The plot is given by Fig. 5, which covers laminar shedding as well as turbulent shedding phenomenon. It is found that Strouhal number increases with Reynolds number upto Re=

$5.1x10^3$ (laminar shedding regime), but after that it exhibits opposite trend and decreases with increase in Reynolds number in turbulent shedding regime. This aberrant pattern can be explained on the basis of transitional change in the three dimensionality of near-wake vortex shedding, more specifically with an increasing degree of spanwise waviness of primary vortices and by the inception of naturally occurring and random-positioned vortex dislocations.

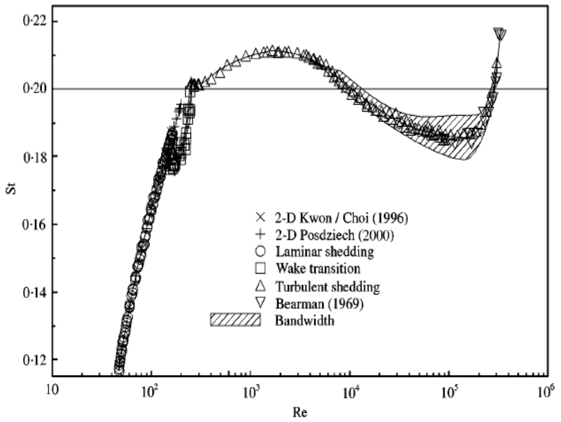

Figure5. Strouhal number versus Reynolds number. Reproduced from Norberg [10].

4.4 Nusset number

The concept of local Nusset number and average Nusset number is described by Golani and Dhiman [5]. The local Nusselt number (Nu) is defined as, $-\frac{\partial T}{\partial n}$, where n is the cylinder surface normal direction. The average Nusselt number is calculated by averaging the local Nusselt number over the surface of the cylinder. Fig. 6 represents the variation of average Nusset number with Reynolds number. It is found that average Nusset number increases almost linearly with increase in Reynolds number.

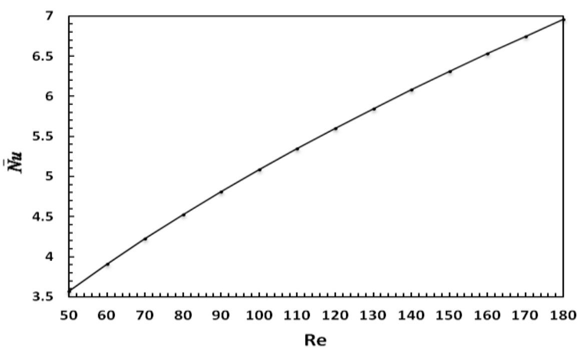

Figure6. Average Nusset number variation with Reynolds number. Reproduced from Golani et al.[5].

4.5 Flow pattern

The flow field around the circular cylinder is presented by way of instantaneous streamline, vorticity magnitude, and velocity magnitude and pressure profiles. Farhoud et al.[8] represented these flow patterns by using FLUENT software. Fig. 7(a), 7(b), 7(c) show pressure contour, velocity magnitude contour and vorticity contour respectively. It can be observed from Fig. 7(a), 7(b), that vortex initially grow in one side of cylinder and then separated and grow again on other side of the cylinder. As a result, position of separated vortex from the cylinder sides is observed in opposite phases [8]. From Fig. 7(c), it can be seen that vortex formed in the upper side of the cylinder and separated, and is growing in the lower side of the cylinder.

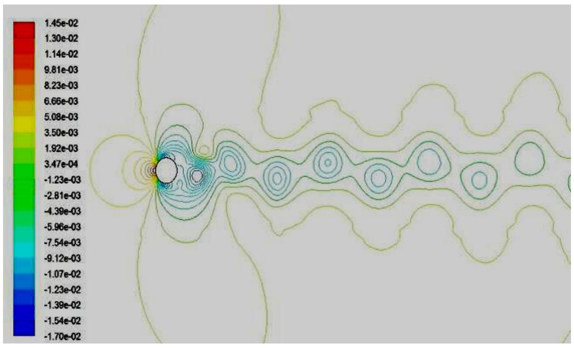

Figure 7(a). Pressure contour [8].

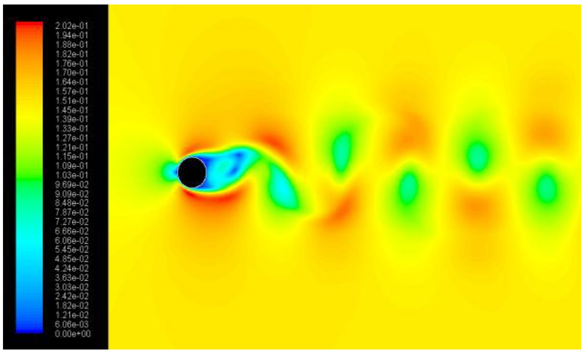

Figure 7(b). Velocity magnitude contour [8].

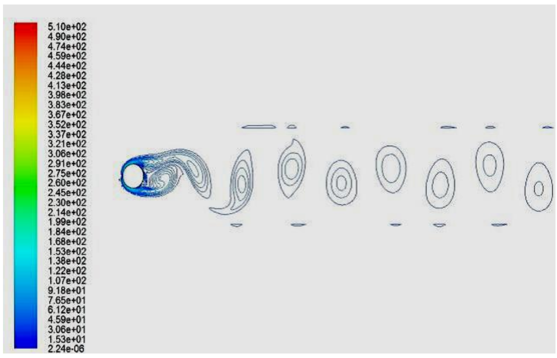

Figure 7(c). Vorticity contour [8].

5. Conclusion

The variation of flow and heat transfer parameters with Reynolds number is studied. It is found that the pressure coefficient and wall shear stress at stagnation point of 0 and 180 do not vary with respect to time. However, at other stagnation points, pressure and shear stress do vary with time. It is also found that the time-averaged drag coefficient decreases with increasing Reynolds number. However, rms value of drag and lift coefficients are found to increase with increasing value of Reynolds number. The lift coefficient oscillates at higher amplitude as compared to drag coefficient. Also the variation of lift coefficient with Reynolds number is one order higher in magnitude than variation of drag coefficient. The Strouhal number that signifies the frequency of vortex shedding is found to increase with Reynolds number in laminar shedding range (Re= $5.1x10^3$), but tends to decrease in turbulent shedding regime owing to three dimensional transition of shedding. The value of average Nusset number also increases with increase in Reynolds number indicating that heat transfer is enhanced as the transition from laminar to turbulent flow occurs. The flow field around the circular cylinder is presented by means of pressure, vorticity magnitude, and velocity magnitude contours.

References

[1]. Tritton D. J, Physical fluid dynamics, Oxford University Press, Oxford, 1987.

[2]. Gandikota, G., Amiroudine, S., Chatterjee, D., Biswas, G., 2010, The Effect of Aiding/Opposing Buoyancy on Two-Dimensional Laminar Flow Acrossa Circular Cylinder, Numerical Heat Transfer, Part A, 58: 385–402, Taylor & Francis Group.

[3]. Igor M. Kozlov, Kirill V. Dobergo, Nickolai, N. Gnesdilov, Application Of RES methods for computation of hydrodynamic flows by an example of 2D flow past a circular cylinder for Re=5-200, International Journal of Heat and Mass Transfer 54, 2011, 887-893 .

[4]. HuaBai, Jiawu Li, Numerical simulation of flow over a circular cylinder at low Reynolds number, Advanced Material Research Vols 255-260, 2011, pp 942-946.

[5]. Ravi Golani, A.K. Dhiman, Fluid flow and heat transfer across a circular cylinder in the unsteady flow regime, The International Journal Of Engineering And Science,2014, Volume 3, pp 08-19

[6]. Jeongyoung Park, Kiyoung Kwon, Haecheonchoi, Numerical solution of flow past a circular cylinder at Reynolds Numbers up to 160, KSME International Journal, Vol 12, No 6, pp 1200-1250, 1998.

[7]. H. Ding, C. Shu, K. S. Yeo, D. Xu, Simulation of incompressible viscous flows past a circular cylinder hybrid FD scheme and meshless least square-based finite difference method, Computational Method Applied Mechanics Engineering 193, 727-744, 2004.

[8]. RahmanKeramatiFarhoud, SaberAmiralaie, GhaderJabbari, SolmazAmiralaie, Numerical Study of Unsteady Laminar Flow around a Circular Cylinder, Journal of Civil Engineering and Urbanism, Volume 2, Issue 2: pp 63-67, 2012.

[9]. Frank P. Incropera, David P. De Witt, Introduction to heat transfer, volume one, Third edition, 1996.

[10]. C. Norberg, Flow around a circular cylinder: Aspects of fluctuating lift, Journal of Fluids and Structures, pp 459- 469, 2001.

ABOUT AUTHOR

Abdullah Saleemwas born in 1992 in the city of Aligarh, India. He received B.Tech degree in Petrochemical Engineering with honours in 2014 from Aligarh Muslim University. He is currently pursuing M.Tech in Chemical Engineering from Aligarh Muslim University. He is a member of IAENG (International Assosciation of Engineers). He has done one month summer training in Indian Oil Corporation Ltd. at Mathura refinery under the supervision of M.W Beg (Chief Production Manager). His final year B.Tech project was based on Techno-economic feasibility of plant design for production of Polyethylene Terephthalate (PET). He is currently working on a project titled “Numerical Study of unsteady laminar flow around a circular cylinder”.