CFD and EXPERIMENTAL ANALYSIS of VORTEX SHEDDING BEHIND D-SHAPED CYLINDER

Chandrakant D. Mhalungekar
Department of Mechanical Engineering,
MIT College of Engineering, Pune 411038,
Pune University, India.

Swapnil.P.Wadkar
Department of Mechanical Engineering,
MIT College of Engineering, Pune 411038,
Pune University, India.

Abstract—The flow around bluff bodies is an area of great research of scientists for several years. Vortex shedding is one of the most challenging phenomenon in turbulent flows. This phenomenon was first studied by Strouhal. Many researchers have modeled the various objects as cylinders with different cross-sections among which square and circular cylinders were the most interested sections to study the vortex shedding phenomenon. The Vortex Shedding frequency depends on different aspects of the flow field such as the end conditions, blockage ratio of the flow passage, and width to height ratio. This case studies the wave development behind a D-Shaped cylinder, at different Reynolds numbers, for which we expect a vortex street in the wake of the D-Shaped cylinder, the well known as von Kármán Street. This body typically serves some vital operational function in aerodynamic. In circular cylinder flow separation point changes with Reynolds number but in D-Shaped cylinder there is fix flow separation point. So there is more wake steadiness in D-Shaped cylinder as compared to Circular cylinder and drag reduction because of wake steadiness.In the present work CFD simulation is carried out for flow past a D-Shaped cylinder to see the wake behavior. The Reynolds number regime currently studied corresponds to low Reynolds number, laminar and nominally two-dimensional wake. The fluid domain is a two-dimensional plane with a D-Shaped cylinder of dimensions B=90mm, H=80mm and L=200mm. CFD calculations of the 2-D flow past the D-Shaped cylinder are presented and results are validated by comparing with Experimental results of pressure distribution on cylinder surface. The experimentation is carried out using small open type wind tunnel. The flow visualization is done by smoke visualization technique. Results are presented for various B/H ratios and Reynolds numbers. The variation of Strouhal number with Reynolds number is found from the analysis. The focus of the present research is on reducing the wake unsteadiness.

Keywords—D-Shaped cylinder, Drag Co-efficient, Lift Co-efficient, Vortex Shedding, SIMPLE, Pressure Distribution, Strouhal Number, Flow Visualization.

I. INTRODUCTION

If velocity difference exists between a particle and its surrounding fluid, then fluid will exert a resistive force on the particle and this exerted force by the fluid on the particle is called drag.A wind tunnel is a research tool developed to study the effects of air moving over or around bluff bodies. It is used for technical support of any major development process involving aerodynamic parts. Mostly it is used in automotive industry, laboratory research and construction. An advantage of using wind-tunnels is that experiments there can be performed under well controlled flow circumstances compared to experiments in the open environment. The most common experiment using wind tunnel is investigation of vortex shedding. This is because the phenomenon of regular vortex shedding behind bodies in a stream of fluid is often observed in nature. Generally the study of vortex shedding in aerodynamics is to avoid the flow induce vibration around the body that results failure to the structure.

The present research involves the phenomenon of vortex shedding behind D-Shaped cylinder. This experiment is based on existing wind tunnel that is already developed. For the simulation part, the number of meshing and blocks should beconsidering as well for better result accuracy. The boundary conditions and wall condition around the bluff model should be considered and applied corresponding to the experimental. The focus of the present research is on reducing the wake unsteadiness, surface pressure calculation, Strouhal number calculation, Flow Patterns visualization.

A. Objective of the research project:

The objectives of this project are:

1. To study the characteristics of vortex shedding behind D-Shaped cylinder.
2. To reduce the drag using D-Shaped cylinder.
3. To make comparison between CFD simulation and experimental results of pressure distribution.

B. Scope of the research project:

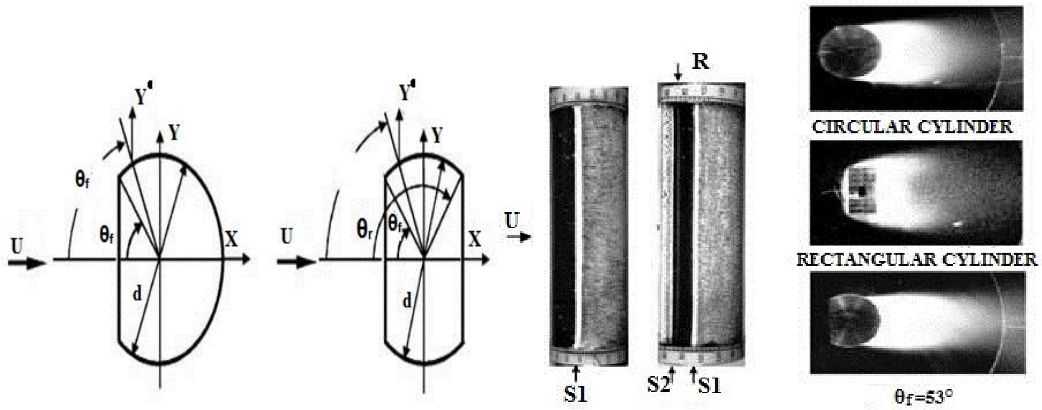
The scope of this project consists of this below:

1. CFD simulation of air flow around D- Shaped cylinder.
2. Fabrication of D-Shaped cylinder.

II. LITERATURE SURVEY

Many researchers have modeled the various objects as cylinders with different cross-sections such as square, circular, and diamond sections. Cox(18) in 1998 studied circular cylinder and showed that choosing the right turbulence model is highly important.

Aiba, S. And Watanabe (1), (2) reported that the drag force decreased by 50% and 25% for a circular cylinder and a sphere, respectively, by cutting out at circumferential angle Θ_f=53º on the front surface of the bluff bodies. However, the pressure distribution, Strouhal number, and flow patterns are not clear. Based on the above work above the authors Tamotsu IGARASHI*** and Yoshihiko SHIBA*** (3) investigated the drag reduction of D-Shape and I-Shape cylinders Fig.1(a).



(a) (b) (c)

Fig.1 (a) Configuration of test models and symbols, (b)Surface oil-flow patterns and (c) visualization of the wake flow around D-Shape cylinder (3).

They found that the drag co-efficient of bluff bodies is small for a large Strouhal number S and small negative pressure co-efficient $-C_{pb}$(3).Aiba et al. (1), (2) also did Flow visualization around a D-Shaped cylinder. Figure 1(b) presents the surface oil-flow patterns on the circular arc of a D-Shape cylinder and Figure 1(c)presents photographs of the wake flow obtained behind Circular, I-Shape and D-Shape cylinder.

Stalnov, Palei, et al. (4) investigated the effectiveness of a small array of body-mounted sensors, for estimation and eventually feedback flow control of a D-Shaped cylinder wake. The research was aimed at suppressing unsteady loads, a low dimensional Proper Orthogonal Decomposition (POD, Holmes et al. 1996) procedure was applied, and linear stochastic estimator (LSE) was employed to map the surface mounted hot film pressure signals. The results show that the estimates of the first two modes of the wake flow field are accurate to within 35% RMS error, with respect to the measured POD time coefficients. SubhankarSen et al. (8),(10), used finite-element method to predict in two-dimensions, the steady, laminar, incompressible flow past a stationary cylinder, results were presented for $Re \leq 40$ and blockages between 0.000125 and 0.80. It is found that the separation initiates from the base point, the separation bubble elongates approximately linearly with *Re*.

Sabra Razughi et al. (9) has been carried out the Hydrodynamic analysis of flow around square cylinders at different angles of attack at low Reynolds numbers, both in water and air using ANSYS CFX software, Twenty different cases were modeled and the Strouhal numbers were extracted after 10 stable oscillations. It was proved that the results of the derived formulas in EUREQA software are in good agreement with the computed results of the Ansys-CFX software at each of the considered angle.

III. EXPERIMENTAL METHOD

In this case the experimentation is carried out by using small open type wind tunnel.

A. Types of Tests

The types of tests conducted in the wind tunnel are as follows:

1. *Force Test: -*

The strain gauge balance detects the forces and is capable of separating the components of total wind force. The motion of air around the body produces the pressure and velocity variations, which produces the wind forces, and moments, that can be experimentally detected and measured in wind tunnel. The forces acting on the model are:-

Lift: - The force component acting upwards, perpendicular to the direction of the flight. The aerodynamic lift is produced primarily by the pressure forces acting on the vehicle surface.

$$F_l= (C_l \times A \times \rho \times V^2)/2 \quad (1)$$

Drag: - The net aerodynamic force acting in the same direction as the undisturbed free-stream velocity.

$$F_d = (C_d \times A \times \rho \times V^2)/2 \quad (2)$$

Where, Fl = Lift Force (N)
Fd = Drag Force (N)
C_l=Co-efficient of Lift
C_d = Co-efficient of Lift
ρ = Density of air (Kg/m^3)
A = Total surface area in contact with the fluid (m^2).

2. *Pressure Test*

In this test, insert tiny tubes into the model surface and connect them to a pressure measuring device. Here the pressure measuring device used is S-10 Pressure transmitter, it measures pressure in terms of water column. Fabrication of D-Shaped cylinder with S-10 in wind tunnel is shown in Fig. 2.

Fig.2 Fabrication of Model in wind tunnelwith S-10 transmitter.

The pressure is calculated by using following empirical formula,

$$P = \rho \times g \times h \quad (3)$$

Where, P is Pressure in Pascal
ρ is density of air kg/m^3
h is height of water column in mm.

3. *Flow Visualization*

Allow the streamlines of air flow to look at the body's surface. It is the study of methods to display dynamic behavior in liquids and gases. The laboratory flow visualization has become more and more exact, with careful control of the particulate size and distribution. There are several flow visualization techniques.

B. Non-Dimensional Numbers

1. *Reynolds Number: -*

The vortex shedding from a bluff body is a function of the Reynolds Number (Re). The flow characteristics of wind passing across bluff body are depend on magnitude of inertial to viscous within the flow (the parameter are called Reynolds Number). The Reynolds Number is defined as:

$$\mathrm{Re} = \frac{\rho \times V \times L}{\mu} \text{ or } \mathrm{Re} = \frac{V \times D}{\nu} \quad (4)$$

Where, *V* is the wind velocity,
D, *L* is the lateral dimension of the body,
υ is kinematics viscosity of air,
µdynamic viscosity of air and

Different *Re* will affect the formation of vortex shedding over bluff bodies.

2. *Strouhal Number: -*

When considering a long cylinder, the frequency of vortex shedding is given by the empirical formula

$$St = \frac{f \times D}{V} \quad (5)$$

Where, f = Vortex shedding frequency
St= Strouhal Number
V= Wind Velocity

International Journal of Innovative Research in Advanced Engineering (IJIRAE) ISSN: 2349-2163
Volume 1 Issue 5 (June 2014) http://ijirae.com

IV. NUMERICAL FORMULATION

A. Computational Fluid Dynamics (CFD):-

Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process. The use of (CFD) to predict internal and external flows has risen dramatically in the past decade.

B. Governing equations:-

Governing equations can be derived using Lagrangian and Eulerian methods and converted to the other form. Following are the three Governing equations:

1. Continuity equation

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{6}$$

2. Momentum equation

$$\frac{\partial}{\partial x_i}(\rho u_i u_j - \tau_{ij}) = -\frac{\partial p}{\partial x_i} \tag{7}$$

3. Energy equation

$$\frac{\partial}{\partial x_i}\left(\rho u_i T - k\frac{\partial T}{\partial x_i}\right) = 0 \tag{8}$$

V. MODELLING andSIMULATION

A. Problem Definition: - CFD and Experimental analysis of Vortex Sheddingbehind D-Shaped Cylinderwith different B/H ratiosanddifferent Reynolds number is carried out.

B. Geometry: - The 2D Plane geometry of a D-shaped cylinder with radius R=40mm is used for validation with experimental results. ANSYSICEM CFD was used for making 2D geometry of D-Shaped cylinder with different B/H ratios. First for B/H=1.125 was modeled. The length of the cylinder is 0.2m. The flow domain is taken of H=80mm, L1=10H, L2=10H, L3=10H and L4=15H as any further increase in any direction produced no change in drag coefficient values.

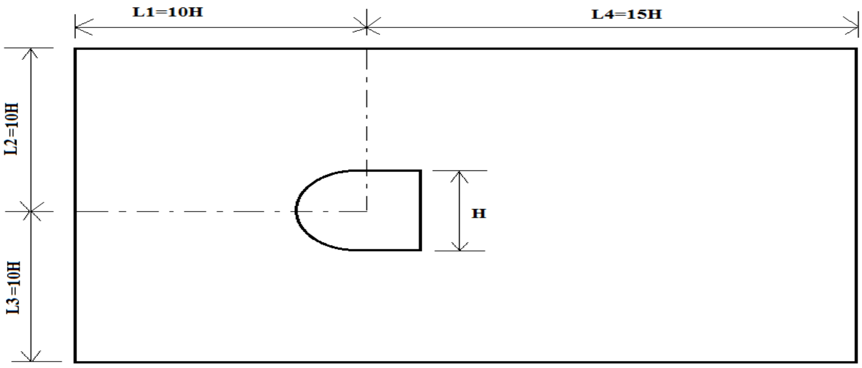


Fig.3Flow domain for CFD simulation of D-Shaped Cylinder.

C. Meshing: - 2D quad mesh is generated inICEM CFD fig. 4. ICEM CFD can generate both structured and unstructured meshes using structured or unstructured algorithms which can be given as inputs to structured as well as unstructured solvers, respectively.

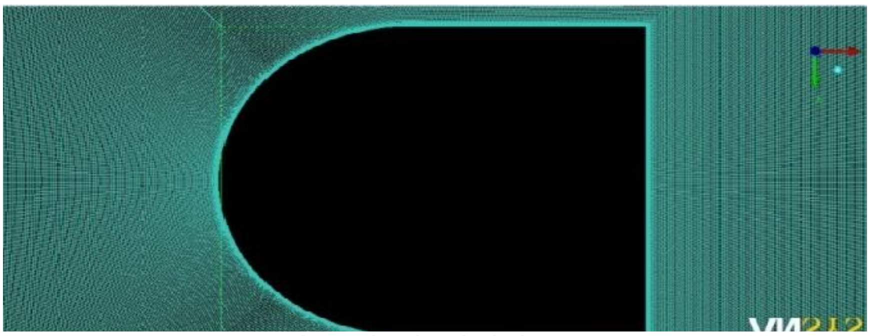


Fig. 4 O-typeMesh around D-Shaped cylinder in ICEM CFD

VI. RESULTSand DISCUSSION

A. CFD and Experimental Pressure calculation

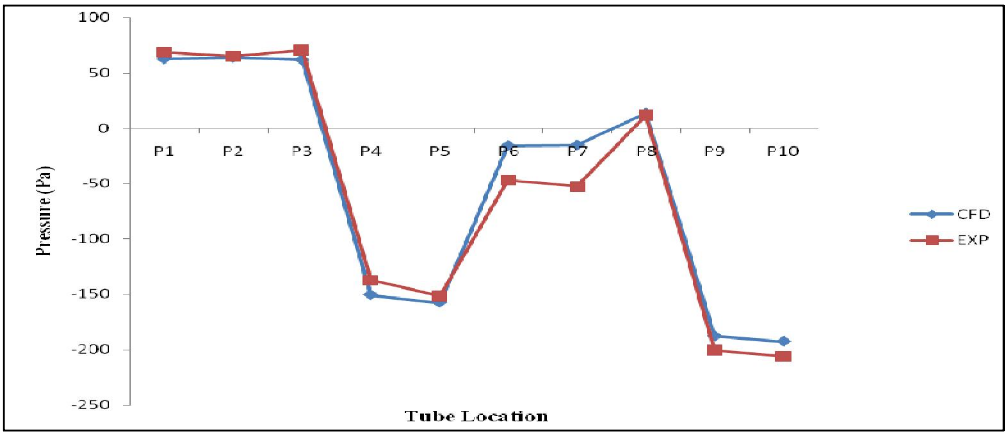


Fig. 5 Graph of CFD and Experimental pressure values.

Fig. 5 shows the graph of CFD and Experimental pressure values. From this graph it is clear that there is minor % of error which is acceptable between CFD and Experimental readings of Pressure values. From this graph we conclude that the CFD results are nearly same with the experimental results.

B. Drag Co-efficient Calculation

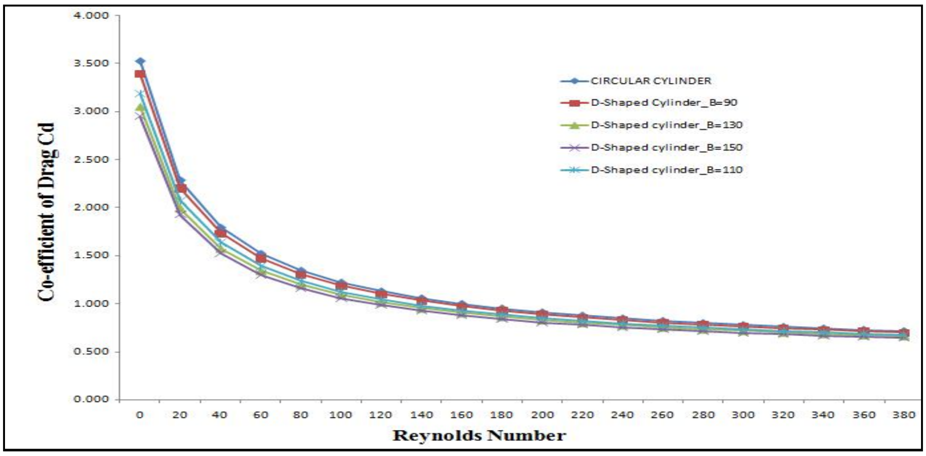


Fig. 6 Graph of Reynolds number vs Co-efficient of Drag

Fig. 6 showsthat drag co-efficient reduces as the Reynolds number increases. It is also clear that drag co-efficient is more for circular cylinder and it reduces for D-Shaped cylinder i.e. drag reduction by wake steadiness.

C. Strouhal number

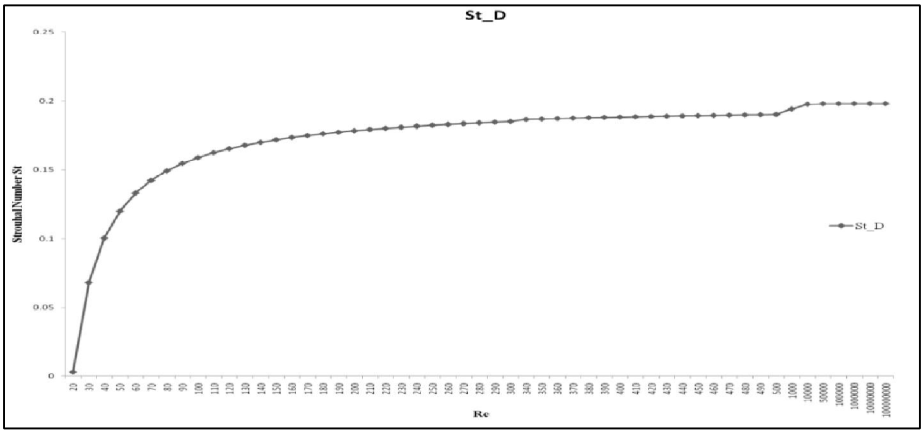
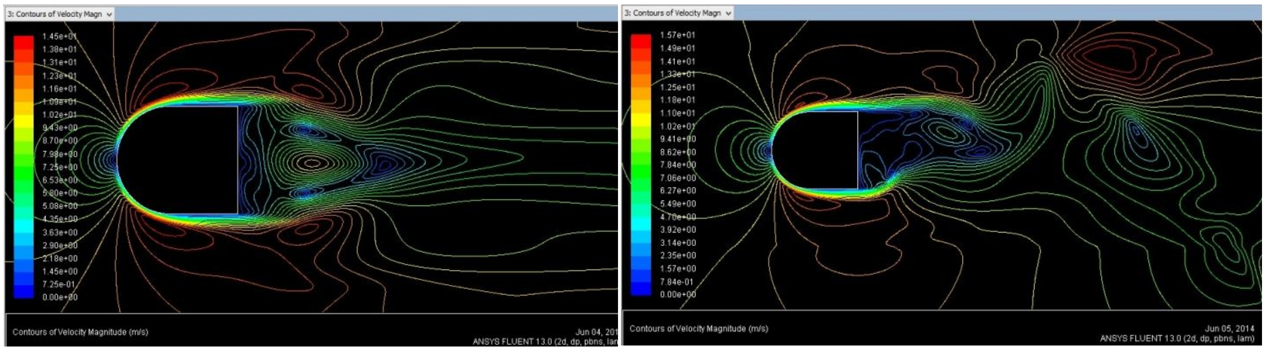


Fig. 7 Graph of Reynolds number vs Strouhal Number

From Fig.7 it is clear that Strouhal number increases with increase in Reynolds number. Hence Number of vortices increases with Reynolds number. Fig. 8 (a) and (b) shows the different vortices at different Reynolds numbers.

D. Vortices generated behind the D-Shaped Cylinder.



(a) (b)

Fig. 8 Vortices generated behind D-Shaped cylinder in CFD (a) at Re= 100, (b) at Re= 125

Fig. 9 Vortices generated behind D-Shaped cylinder in wind tunnel experimentally.

From figure 8 and figure 9 it is clear that there is fix flow separation point in D-Shaped cylinder.

VII. CONCLUSION

From the experimental and CFD results we get the following conclusions: -

1. Pressure at the upper side, bottom side and back side of cylinder is of negative pressure, it is because ofvortices generated at that surfaces.
2. In circular cylinder flow separation point changes with Reynolds number, so there is wake unsteadiness but in D-Shaped cylinder there is fix flow separation point, so there is wake steadiness as compared to circular Cylinder. Because of this drag co-efficient is more for circular cylinder and it reduces for D-Shaped cylinder i.e. drag reduction by wake steadiness.

ACKNOWLEDGMENT

I would like to convey my sincere gratitude to H.O.D. of mechanical department MIT COE, Pune, Prof Dr.B.S.Kothavale for his invaluable suggestions, constructive criticism, motivation and guidance for carrying out related experiments and for preparing the associated reports and presentations. His encouragement towards the current topic helped me a lot in this project work.

I would also like to thank Prof S.P.Wadkar for his help in the research done in this project.

I owe my thankfulness to Prof Dr. S.V.Dingare, PG Co-coordinator, Department of Mechanical for providing necessary facilities in the department and also to all other staff members ofMIT COE, Punefor their excellent coordination and arrangement towards the consistent evaluation of this project.

I thank my family and friends for being constant support throughout my life.

REFERENCES

[1] Aiba, S. And Watanabe, H., *"Flow Characteristics of a Bluff Body Cut from a Circular Cylinder"*, J. Fluids Eng., Vol.119 (1997), pp.453-454.

[2] Aiba, S., *"Flow Dynamic Drag of an Axially Symmetrical Bluff Body Consisting of a Plane Surface and a Spherical Surface"*, J. Fluids Eng. Vol.120 (1998), pp.851-853.

[3] Igarashi, T. and SHIBA, Y., *"Drag Reduction of D-Shape and I-Shape Cylinders (Measurements of Pressure Distribution and Flow Visualization)"*, Proc. 5th Bluff Body Aerodynamics and Applications, Ottawa, Canada, (2004), pp.181-184.

[4] White, F., *"Fluid Mechanics"*, McGraw-Hill, 2003.

[5] Collis S. et al, (2004),*"Issues in active flow control: theory, simulation and experiment"*. Prog Aero Sci V40:237-289 (previously AIAA paper 2002-3277).

[6] Roshko A (1954) – *"On the drag and shedding frequency of two-dimensional bluff bodies"*. NACA TM3169.

[7] Roussopoulos K, Monkewitz, PA (1996) *"Nonlinear modeling of vortex shedding control in cylinder wakes"*.Physica D (97):264-273.

[8] Sen, S. et al, 2010. *"Numerical Simulation of Steady Flow Past a Circular Cylinder"*, Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power, (2010), IIT Madras, Chennai, India.

[9] Sabra Razughi et al, *"Hydrodynamic Analysis of Flow around Square Cylinders and Determination of the Strouhal number as a function of angles of attack and Reynolds numbers"*, International Journal of Scientific World, 1(2) (2013) 37-43

[10] Sen, S. et al, 2009. *"Steady Separated flow past a circular cylinder at low Reynolds numbers"*, Journal of Fluid Mechanics 620 89-119.

[11] Oksana Stalnov et al, *"Experimental Estimation of a D-Shaped Cylinder Wake Using Body-Mounted Sensors"*, Exp Fluids (2007) 42:531-542.

[12] Md. Mahbubar Rahman[1] et al, *"Numerical Investigation of Unsteady Flow Past a Circular Cylinder Using 2-D Finite Volume Method"*. Journal of Naval Architecture and Marine Engineering (2007).

[13] Gera. B. et al, *"CFD Analysis of 2D unsteady flow around a square cylinder, International Journal of Applied Engineering Research"*, DINDIGUL, Vol. 1, No3, 2010.

[14] Cox, Jared et al. *"Computation of sound generated by viscous flow over a circular cylinder"*. NASA Technical Memorandum 110339. March 1997.

[15] Csiba, L.A., Martinuzzi., J.R.,*"Investigation of bluff body asymmetry on the properties of vortex shedding"*, J. Wind Engineering, August 2007.

[16] Anagnostopoulos, P., *"Numerical study of the flow past a cylinder excited transversely to the incident stream. Part2: timing of vortex shedding, Aperiodic phenomena and wake parameters"*, J. Fluids and Structure, March 2000.

[17] Bently, J.P., Nichols, A.R., *"The mapping of vortex fields around single and dual bluff bodies"*, Butterworth-Heinemann Ltd,1990.

[18] Dalton, C., Wu, Z., *"Numerical Solutions of a Viscous Uniform Approach Flow Past Square and Diamond Cylinders"*, 5th International Symposium on Fluid Structure Interaction, Aeroelasticity, and Flow Induced Vibration and Noise, 2003.

[19] Yunus A Cengel, *"Fluid Mechanics-Fundamentals & applications"*, McGraw-Hill companies, pp.817-868, 2006.

