International Journal of ChemTech Research
CODEN (USA): IJCRGG, ISSN: 0974-4290, ISSN(Online):2455-9555
Vol.10 No.6, pp 209-214, 2017

Numerical Investigation on Control of Vortex Shedding Behind a Circular Cylinder Using Passive Techniques

R.Jaganraj, G. Boopathy*

Department of Aeronautical Engineering, Veltech Dr.RR & Dr.SR University, Avadi, Chennai-600062.

Abstract : Present numerical study aims at control and suppression of vortex shedding formed over a circular cylinder using V splitter. The present work is done with commercially avialble software ANSYS-Fluent^{TM} with the flow condition of unsteady, two–dimensional laminar conditions at a Reynolds number (Re) of 150. Numerical simulations with different passive controlling methods have been carried out to reduce the vortex shedding frequency or to suppress it completely by using the V splitter. It is found that V splitter with (0.5D) is suppress the shedding immediately. V splitter with 1D control the shedding initially and then supress, where 0.5D splitter is directly control the shedding. The results of vorticity and streamline contours, C_l, C_d and Strouhal number for better understanding the flow and shedding characteristics.
Keywords : V splitter, Vortex shedding, Strouhal Number, Shedding Frequency, vorticity

1. Introduction

Variety of Cylinder applications are commonly found in engineering stream due its unique nature of geometrical construction as well as its nature of symmetry and simplicity (Ex. Chimney, Stay cables, Pipe lines). Most of the physical geometry will be in cylindrical shape only. The flow around a circular cylinder is unique in nature, due its nature of separation, wake, and turbulence at wake region occurs when it interacts with different velocity of flow. This phenomenon makes a flow around a cylinder becomes more oscillatory which originally creates the vibration in host material due its elasticity and inertia. Most of the physical structures like mountain, trees are bluff bodies. Even bluff bodies take place in many building structures like stay cables and chimes. We can classify the bluff body into streamline body by its characteristic length to diameter ratio. Due to its unique nature of response with airflow, it may subject to uneven flow pattern around it. The researchers continue their efforts towards finding the flow properties of bluff bodies in low speeds after the first accident Tacoma Narrows Bridge, which collapsed in 1940 due to low speed instability problems. During the construction of the bridge, it was experience series of vertical motions in specific time interval. So the designers considered to use stringers for control the oscillatory motion. Two inclined cable as pair, stays that connected the main cables to the bridge deck at mid-span. These remained in place until the collapse, but were also ineffective at reducing the oscillations.

The structure was equipped with hydraulic buffers installed between the towers and the floor system of the deck to damp longitudinal motion of the main span. The effectiveness of the hydraulic dampers was nullified, however, because the seals of the units were damaged when the bridge was sand-blasted before being painted. To drill holes in the lateral girders and along the deck so that the air flow could circulate through them to reduce aerodynamic vertical loads. This was suggested by Professor Farquharson and his students. With this long historical background and the present studies on low speed aerodynamics studies over bluff bodies is having unique place in research field. This paper explains the fundamentals of flow control techniques over bluff bodies and how passive control can control the flow over a cylinder. The different methods have been applied and the Numerical analysis of cylinder flow as well as control over the flow was done by many researchers.

Flow around a circular cylinder is a fundamental fluid mechanics problem of practical importance. It has potential relevance to a large number of practical applications such as submarines, off shore structures, bridge piers, pipelines etc. The laminar and turbulent unsteady viscous flow behind a circular cylinder has been the subject of numerous experimental and numerical studies, especially from the hydrodynamics point of view [1].

It is found that the two-dimensional finite volume method computes hydrodynamic forces and captures vortex shedding very well. Even at high Reynolds number, the method is very much applicable without loss of accuracy. It is also observed that standard k-epsilon model computes drag coefficients accurately, where the realizable k-epsilon turbulence model is more effective for visualization of vortex shedding [2]. Flow control is broadly classified in to two methods, 1. Active and 2.Passive. The Active methods require external energy to control the flow field. In case of smooth cylinder, the separation angles for 2D or 3D numerical calculation are found to be around 80~90° in either side of the cylinder from the upstream stagnation point. The drag coefficients for smooth surface are 0.771 and 0.533 for 2D and 3D numerical calculation respectively and subsequent changes in drag due to introducing surface roughness are demonstrated. The critical surface roughness is found to be around 0.004 with coefficient of drag 0.43. Though the wake structure was vaguely visible, they are not as periodic as in Karman street. This article is a review of numerous analyses focused on modelling the flow over a cylinder at higher Reynolds number. This study contains more than 30 research papers on circular cylinder as well as on elliptical cylinder. Most of the scientists and researchers used CFD codes to analyze the models subjected to various conditions and compared results with measurements [3]. The LES solutions are shown to be considerably more accurate than the RANS results. They capture correctly the delayed boundary layer separation and reduced drag coefficients consistent with experimental measurements after the drag crisis. The mean pressure distribution is predicted reasonably well at ReD ¼ 5 105 and 106 [4].

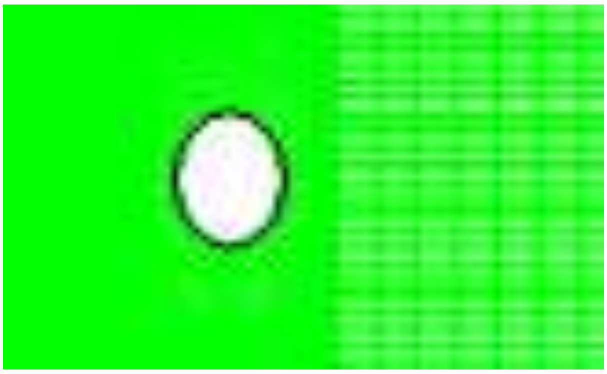
Results have shown that flow in entrance boundary after passing cylinder is completely unsteady in spite of steady flow and also results have shown changes periodically and have minimum value in symmetry line in the computational domain [5]. The key idea is to propose a method to calculate the interfacial force without ad hoc constants that should usually be adjusted for the type of flow and the type of the numerical method, when this kind of model is used.

In the present work, this force is calculated using the Navier–Stokes equations applied to the Lagrangian points and then distributed over the Eulerian grid. The main advantage of this approach is that it enables calculation of this force field, even if the interface is moving or deforming. It is unnecessary to locate the Eulerian grid points near this immersed boundary. The lift and drag coefficients and the Strouhal number, calculated for an immersed cylinder, are compared with previous experimental and numerical results, for different Reynolds numbers [6].

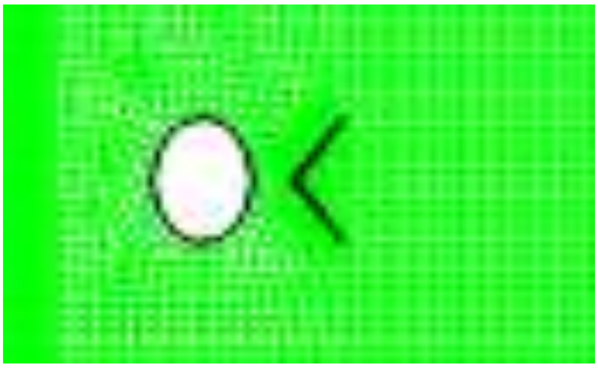
For each Reynolds number, several meshes with different grid and time step size resolutions were chosen to calculate the hydrodynamic quantities such as the time-averaged drag coefficient, root-mean square value of lift coefficient, Strouhal number, the coefficient of pressure on the downstream point of the cylinder, the separation angle. By comparing the values of these quantities of adjacent grid or time step size resolutions, convergence study has been performed. Solution validation is obtained by comparing the converged results with published numerical and experimental data. The deviations of the values of present simulated quantities from those corresponding experimental data become smaller as Reynolds numbers increases from 1×105 to 1×106. This may show that the standard k-ε model with enhanced wall treatment appears to be applicable for higher Reynolds number turbulence flow [7]. The response of the flow are investigated at a fixed Reynolds number, R = 200. The oscillation frequency was fixed to the vortex shedding frequency from a fixed cylinder, f0, while the amplitudes of oscillations were varied from 0 . 1 to 1 . 0 a, where a represents the radius of the cylinder. The response of the flow through the fluid forces acting on the surface of the cylinder are investigated [8].

2. Problem Statement

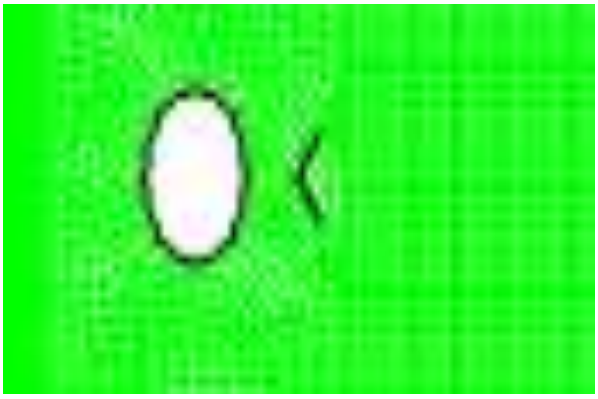
The 2D computational domain consist of, cylinder model with dia D is kept between walls of 50D and 20D dimension. The discretization is done by using a non-uniform structured grid generation techniques, as shown in Fig. 1 for this computational work. The boundary conditions are used as velocity and pressure outlet at left and right side of the domain. The no slip condition is given to the cylinder.



(A)



(B)



(C)

Fig. 1. Grid Structure (A) Baseline (B) Baseline with v splitter of 1D X 1D (C) Baseline with v splitter of 0.5 D X 0.5 D

2.1 Numerical Methodology

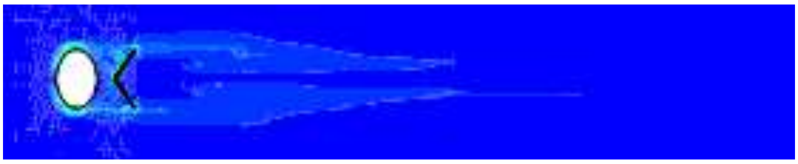
Table 1. Numerical methodology

Type	Unsteady Pressure based
Grid Type	Unstructured quad mesh
Pressure velocity couple	PISO
Software Used	ANSYS Fluent™

2.2 Results and Discussions

Models used in this investigation are V splitter with 1D and 0.5D cross section to find the vortex shedding suppress and control. The strohaul number *St* = *f* D/U (U velocity, D diameter, f- frequency) is calculated for finding the shedding behaviors of three models as shown in Fig. 1 As shown in Fig. 3 the shedding frequency (f) is calculated using the time history of lift and drag curves. The vorticity contours are plotted for all the cases in Fig. 2 and to see the flow path and von Karman vortex shedding patterns.

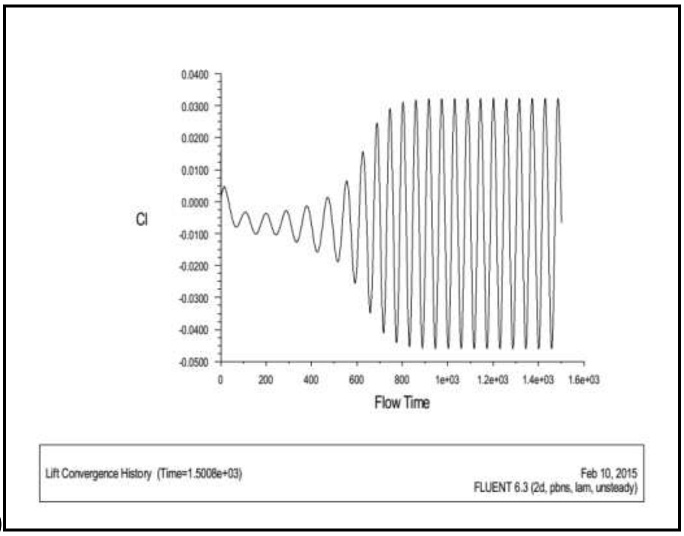
With 0.5 D



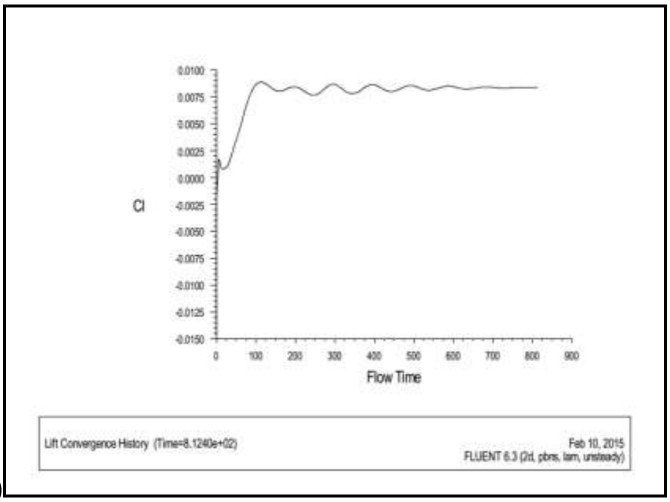
With 1 D

With Base Line

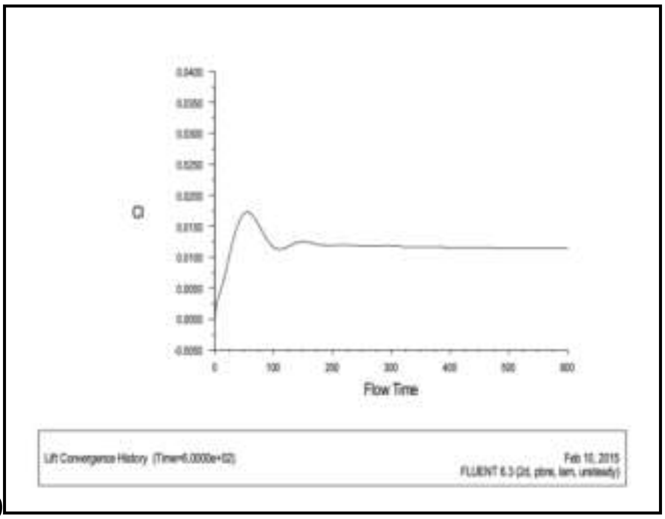
Fig. 2. Vorticity contours for baseline and baseline with splitter



(A)



(B)



(C)

Fig. 2. C_L Time history for (A) Baseline, (B) baseline with V splitter (1DX1D) (C) Baseline with V splitter (0.5DX0.5D)

2.3 Effects of V Splitter on Vortex Shedding

The simulation is made to find the behavior of vortex shedding for base diagram, and V splitter with two different geometries. It is found that the cylinder with the V (1D) Splitter has control the shedding by 0.07f initially and the shedding is suppressed after 400 seconds of the flow and becomes zero. V splitter with (0.5D) is suppress the shedding immediately. So 1D v splitter is control the shedding initially and then supress, where 0.5D splitter is directly control the shedding. Details are in Table 2.

Table 1. Comparison of flow over cylinder with different V Splitter configuration.

Configuration	**C_l**	**C_d**	**Shedding Timing(t)**	**Frequency (*f*=1/t)**	**St= *f* D/U**
Base Line	-1.64	7.51	70	0.07	0.182
V Splitter 1D	1.24	0.08	0	infinite	0
V Splitter 0.5D	0.001	0.075	0	infinite	0

3. Conclusion

The passive method of V splitter is applied for the present numerical study consist of two–dimensional unsteady to suppression of vortex shedding formed over a circular cylinder is discussed in detail using commercially available CFD software ANSYS-Fluent^{TM}. The vortex shedding is characterized by the Strouhal number, Renolds number. Frequency of the shedding is calculated using time history of lift coefficient curve. It is found that cylinder with V Splitter can suppress the flow shedding completely for 0.5D V splitter and control the flow shedding for 1D V splitter.

4. References:

1. Toukir Islam and S.M. Rakibul Hassan, Experimental and Numerical Investigation of Flow over a Cylinder at Reynolds Number 105, Journal of Modern Science and Technology Vol. 1. No. 1. May 2013 Issue. Pp.52-
2. Md. Mahbubar Rahman, Md. Mashud Karim and Md. Abdul Alim, Numerical Investi-gation Of Unsteady Flow Past A Circular Cylinder Using 2-D Finite Volume Method, June, 2007
3. Manish Rawat, Rajesh Gupta and R.M.Sarviya, High Reynolds number flow over cyl-inders: A Review, International Journal of Application or Innovation in Engineering & Management (IJAIEM), Volume 2, Issue 6, June 2013.
4. Pietro Catalano, Meng Wang , Gianluca Iaccarino , ParvizMoin , Numerical simula-tion of the flow around a circular cylinder at high Reynolds numbers, International Journal of Heat and Fluid Flow 24 (2003) 463–469.
5. Rahman Keramati Farhoud, Saber Amiralaie1, Ghader Jabbari, Solmaz Amiralaie1, Nu-merical Study of Unsteady Laminar Flow around a Circular Cylinder, Journal ofCivil Engi-neering and Urbanism Volume 2, Issue 2: 63-67 (2012)
6. A.L.F. Lima E Silva a, A. Silveira-Neto, J.J.R. Damasceno, Numerical simulation of two- dimensional flows over a circular cylinder using the immersed boundary method, Journal of Computational Physics 189 (2003) 351–370
7. Xian-tao Zhang, Zhi-yu Li1, Shi-xiao Fu, Muk chen Ong and Ying Chen, Study of the flow around a cylinder from the subcritical to supercritical regimes, Ocean Systems Engineering, Vol. 4, No. 3 (2014) 000-000.
8. Qasem M. Al-Mdallal, Numerical Simulation of Flow Past a Circular Cylinder Undergoing Figure-eight-Type Motion: Oscillation Amplitude Effect, Proceedings of the World Congress on Engineering 2014 Vol II, WCE 2014, July 2 - 4, 2014, London, U.K
9. L. G. Pack, R. D. Joslin , Overview of Active Flow Control at NASA Langley Research Center NASA Langley Research Center Hampton, Virginia 23681-0001.

*****