ISSN: 2349- 5197
Impact Factor (PIF): 2.138

INTERNATIONAL JOURNAL OF RESEARCH SCIENCE & MANAGEMENT

NUMERICAL SIMULATION OF FLOW PAST TWO CYLINDERS ARRANGED SIDE BY SIDE

Gokul.R[1*],Aby Alias[2]
[1*2]School of Mechanical and Building Sciences,VIT University, Chennai Campus
Correspondence Author: gokul.r2013@vit.ac.in

Keywords: Reynold's number, tandem,wake,vortex

Abstract
Pairs of circular cylinders immersed in a cross-flow are encountered in many engineering applications. The cylinders may be arranged in tandem, side-by-side, or staggered configurations. Wake and proximity interference effects, which are determined primarily by the longitudinal and transverse spacing between the cylinders, and also by the Reynolds number, have a strong influence on the flow patterns, aerodynamic forces, vortex shedding, and other parameters. This paper reviews the understanding of the flow around two circular cylinders of equal diameter immersed in a cross-flow at low Reynolds number (Re).

Introduction
There range of applications for flow past two cylinders are various in day to day life. In many of these engineering applications. Cylinder-like structures can be found both alone and in groups in the designs for heat exchangers, cooling systems for nuclear power plants, offshore structures, buildings, chimneys, power lines, struts, grids, screens, and cables. Due to the importance of the problem in many engineering applications and because of the complex flow physics involved, considerable amount of research work, largely experimental [1–3] is available.
In both air and water flow Karman vortex shedding is responsible for problems with flow induced vibration and noise Sumner (2010). For understanding the vortex dynamics in the wake behind a bluff body and, thus, a considerable number of studies on the uniform flow have been performed so far, Williamson (1996) and Zdravkovich (1997). Two cylinders can be arranged in infinite number of ways, each arrangement having its own flow features. The dominant characteristics of the flow over a side by side arrangement of two cylinders are the complex interaction between the shear layers from the cylinders and vortex formation. Low Reynolds number flow have been the subject of investigation in recent years. Wang et al (2013) performed CFD analysis for unsteady, incompressible 2D flow around a circular cylinder for Re ranging from 50 to 250. In addition to the Reynolds number, the gap size, i.e., the shortest distance between the surfaces of the two cylinders, strongly influences the flow.

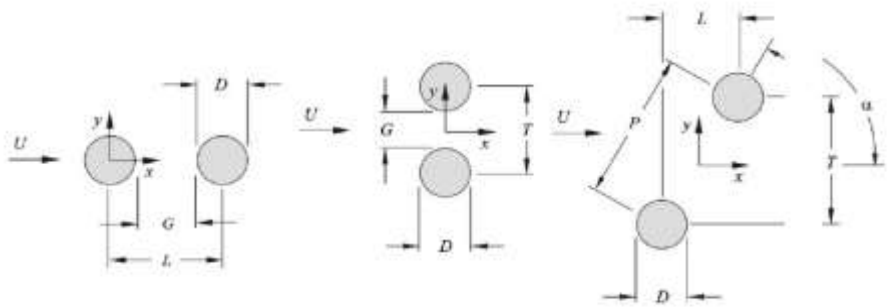


Fig1. Different arrangement for cylinders

Where the geometry is described by the center-to- center longitudinal distance, Diameter by D and G represent shortest distance between the cylinders. U is the upstream velocity of flow.

Numerical simulation
As incompressible viscous flow with constant fluid properties was assumed. The sole governing parameter except for geometrical domain restrictions, is the Reynolds number ,Re, traditionally based on the cylinder diameter and the uniform oncoming free stream velocity. Computational domain is selected upon Reynolds's number. Downstream distance is kept high than the upstream from the cylinder so that flow stabilizes.For the validation single cylinder kept in the computational domain is taken. Lixia Qua et al(2013) uses a O grid to solve single cylinder problem. In this study the tetra meshes are used.

Computational domain

Here computational domain is rectangular duct of 260mm * 170mm. Cylinder is kept 85mm from inlet ,where cylinder diameter is 10mm. Mesh generation depends upon Reynolds number. For high Reynolds number node count should be high. Whereas for a low Reynolds number a continuous mesh is required. In the case require a mesh that is continuous and uniform. In 2 Dimension tetragonal the fig mesh was developed using ICEM.

ISSN: 2349- 5197
Impact Factor (PIF): 2.138

INTERNATIONAL JOURNAL OF RESEARCH SCIENCE & MANAGEMENT

The adapted numerical method with different spatial order of convergence has different requirements with regard to the location of the first grid point. The convergence criterion was set to 10^{-5}.Tests with smaller values of this precisione.g. $10.^{-6}$ showed no changes in the results.

Governing equations.

Problems of flow are solved based on continuity equations and momentum equations. Continuity equation is represented by

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$$

For momentum equation it is better to use Navier Stokes equations.

X momentum.

$$\rho(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z})=-\frac{\partial p}{\partial x}+\mu\nabla^2 u+\rho g_x$$

And Y momentum

$$\rho(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z})=-\frac{\partial p}{\partial y}+\mu\nabla^2 v+\rho g_y$$

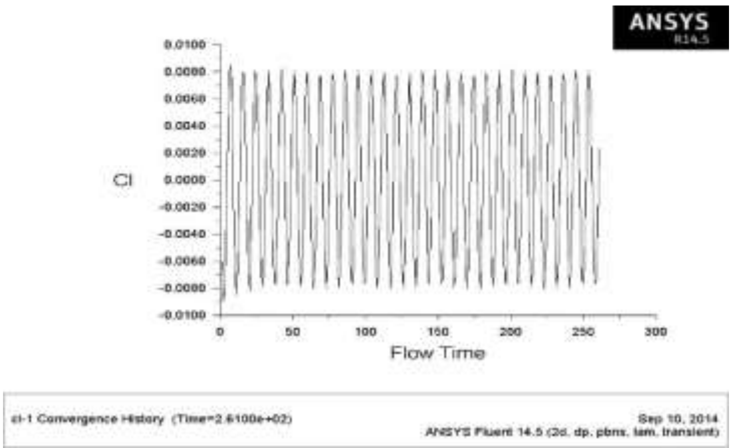
Where µ is coefficient of viscosity, ρ is density, u is velocity in x direction, v is velocity is y direction and w is velocity in z direction. Rest of the symbols has usual meaning.

Flow past a single circular cylinder

The boundary conditions for single circular cylinder area are adopted from literatures. The flow past circular cylinder is analyzed by using ANSYS FLUENT. Circular cylinder is considered as stationary wall condition with no slip shear condition. The boundary walls are also kept at no shear condition. Fluid is assumed to be air. A constant time step Δ t =0.05 is used for calculations. The tolerance level for the validation is 5%. The results are compared with [1].

Definition of global quantities

Generally for the computational fluid problems behavior of bluff body is described using drag and drift parameter. These parameters are used for validating previous known results from the literature. For Validation, use coefficient of drift and drag C_d and C_l . Park et al. (1998) and Fujun Wang et al (2013) describes behavior of single bluff bodies depending upon different Reynolds number .The ratio between C_d and C_l is denoted as 'r'. For the purpose of validation single bluff body similar to problem is taken. The above mentioned coefficientsare obtained from the plots. Generally root mean square (rms) values of the plots are used to compare the results. C_d' and C_l' are used to represent the rms values. Below are the plots of C_d and C_l.



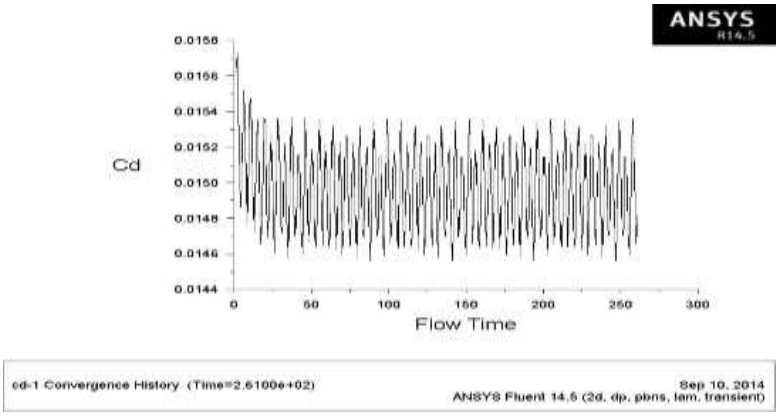


Fig.2 Plots of C_d and C_l

From the plots the rms values of C_d'and C_l 'are 0.0063 and 0.232151 . Comparison for the results with earlier works is shown in in Table.The study gives values of C_d and C_l within the tolerance of 5%.

Table 1.Comparision of coefficient of forces

Author(s) (year)	Cd'	Cl'	r
Fujun Wang et al (2013)	.0064	0.225	36.5
Park et al(1998)	0.00625	0.235	36
Present	0.0063	0.23215	36.85

Grid independence study

Grid independence study is carried for three different mesh count. Each case of mesh yield different values of C_d and C_l.

Table 2. Grid Independence study

Mesh Count No. of node	Re	Drag Co-efficient (rms)	Lift Co-efficient (rms)	Percentage error (%)
24563	100	0.0073	0.34	7.0123
37482	100	.00701	0.29	3.56
44265	100	0.006345	0.23215	1.4974

On comparison with different C_d and C_l values for different mesh count. It can be inferred that mesh with highest mesh count yields precise results.

Flow past two cylinders in side by side arrangement.

The actual problem is modelled in ICEM CFD. The distance between the cylinders determines how the flow is affected. Center to center distance of the cylinders is kept changing. Remaining parameters i.e Reynolds number, boundary conditions are kept same as the validation case. Figure 3 is mesh diagram for the two cylinder problem. The domain is meshed using tetra mesh. Generating 88482

The actual problem is modelled in ICEM CFD. The distance between the cylinders determines how the flow is affected. Figure 2 is mesh diagram for the two cylinder problem. The domain is meshed using tetra mesh. Generating 44303 elements.

ISSN: 2349- 5197
Impact Factor (PIF): 2.138

INTERNATIONAL JOURNAL OF RESEARCH SCIENCE & MANAGEMENT

Fig. 3. Case 1

For two cylinder arrangements in consistent mean cross stream, the least complex of the methodologies is treatment by Zdravkovich (1987), who arranged the liquid conduct into two essential sorts of impedance, in view of the area of the downstream chamber concerning the upstream one.. Complex wake and vortex-street interactions occur when the cylinders are spaced between these two extremes, in particular an asymmetric, or biased, flow pattern that may be bistable in nature. Many experimental studies of side-by-side cylinders in steady cross-flow have been undertaken, from low Reynolds numbers (e.g., Williamson (1985) at Re=50) (Sun et al (1992)).

Results & discussions

It is known that the flows around multiple cylinders are more complex than that around a single cylinder. In the side by side arrangement of two circular cylinders, when the spacing between the two cylinder surfaces is larger than the diameter of the cylinder, the occurrence of two synchronized vortex streets. When two circular cylinders are arranged transverse to the mean flow in a side-by-side configuration, the wakes of the two cylinders interact with one another on either side of the gap between the two cylinders. If the cylinders are spaced sufficiently far apart, they may behave as two independent bluff bodies, although synchronization between the adjacent vortex streets may occur.

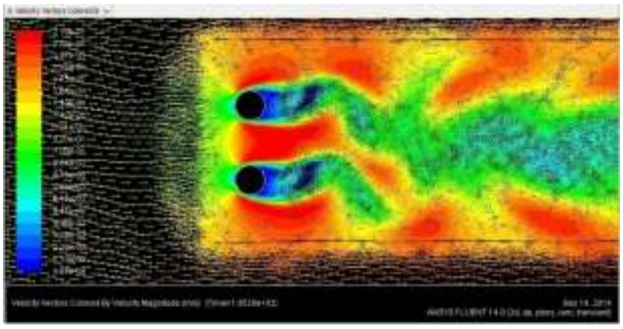


Fig 4.Velocity vector plot.(L/D=2.5)

Above is the velocity vector plot for side-by-side arranged cylinders with gap distance of 15mm. There is biased flow pattern. Plotting for the drag and lift coefficients in figures(3)and(4)

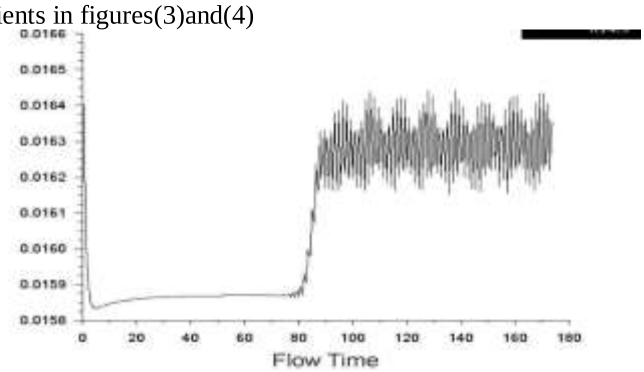


Fig 5. Plots for C_dVs.time(L/D=2.5)

ISSN: 2349- 5197
Impact Factor (PIF): 2.138

INTERNATIONAL JOURNAL OF RESEARCH SCIENCE & MANAGEMENT

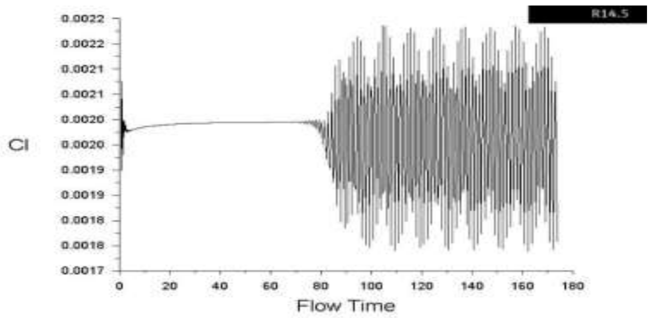


Fig 6. Plot for C_lVs.time(L/D=2.5)

Plotting for C_d and C_l , thus obtaining above plots. Similar to the section B, root mean square values are calculated. C_d' and C_l' values are 1.421223 and 0.2272.
Comparing the rms values obtained to the literatures available.

Table.3.Comparison of rms values of C_d and C_l

Author(s) (year)	Cd'	Cl'	r
Lee et al(2009)	1.45	0.273	0.188
Yan Bao(2010)	1.429	0.2666	0.189
Present	1.421223	0.2272	0.1610

Following are plots for C_l different ratios of L/D.

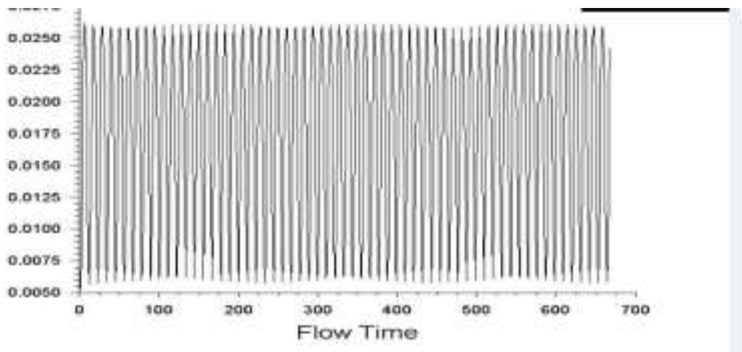
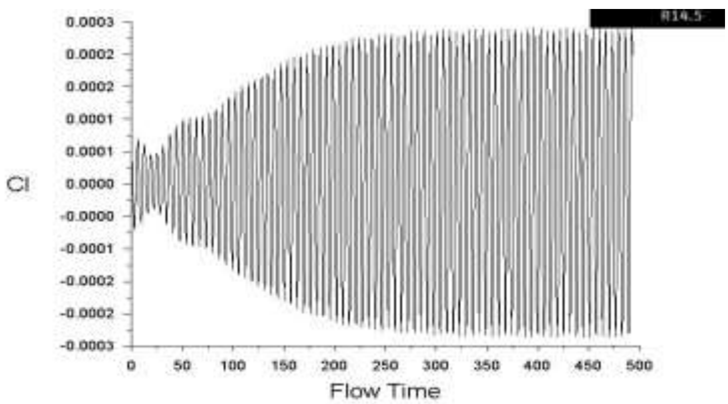


Fig.7 Cl plot for L/D =1



ISSN: 2349- 5197
Impact Factor (PIF): 2.138

INTERNATIONAL JOURNAL OF RESEARCH SCIENCE & MANAGEMENT

Fig8.Cl plot for L/D=1.5

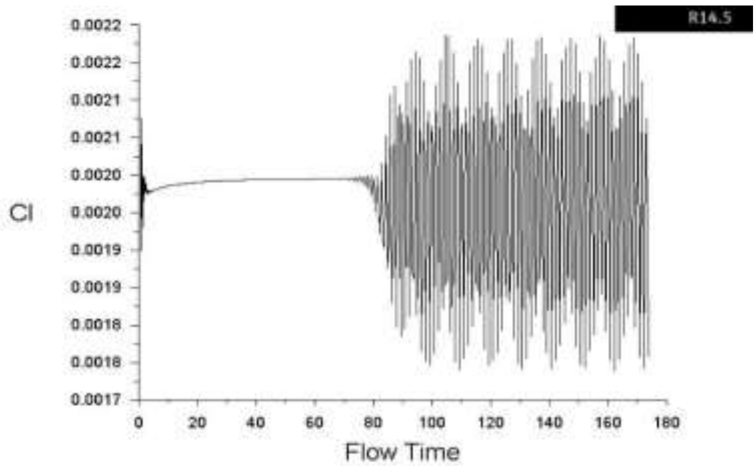


Fig 10. Plot for C_l Vs. time(L/D=2.5)

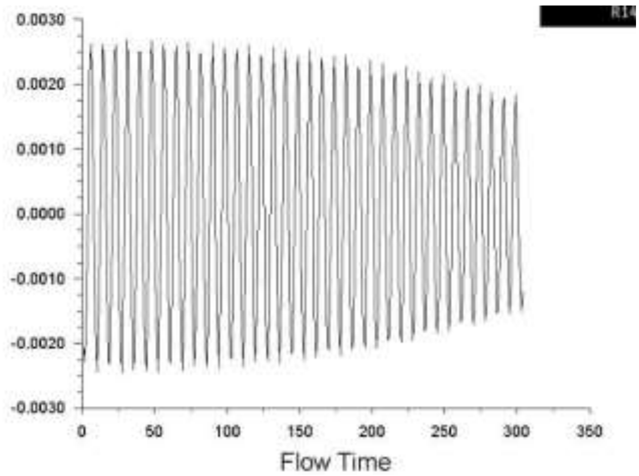


Fig9. Cl plot for L/D =4

Conclusion

This paper is an attempt to discuss the flow around two circular cylinders of equal diameter and a comparison with flow around single cylinder. Unlike an ideal in viscid fluid, a viscous flow past a cylinder, no matter how small the viscosity, will acquire vorticity in a thin boundary layer adjacent to the cylinder. Boundary layer separation can occur, and a trailing wake will occur behind the cylinder. The pressure will be lower on the wake side of the cylinder, than on the upstream side, resulting in a drag force in the downstream direction.

For the case of flow past two cylinder the flow is At intermediate values of T/D the two side by side circular cylinders have long known to exhibit an asymmetrical or biased flow pattern. The deflection of the biased gap flow varies with T/D. The trend is toward a smaller degree of deflection with increasing T/D.

References

1. Lixia Qua, ChristofferNorberg, LarsDavidson,Shia-HuiPeng,Fujun Wang(2013) , Quantitative numerical analysis of flow past a circular cylinder at Reynolds number pp (50-200).
2. Masami Sato, Takaya Kobayashi,SIMULIA (2012) Design & Analysis Corporation. A fundamental study of the flow past a circular cylinder using Abaqus/CFD.
3. Sumner, Journal of Fluids and Structures 26 (2010), Two circular cylinders in cross –flow: A review Department of Mechanical Engineering, University of Saskatchewan, pp(849–899)

INTERNATIONAL JOURNAL OF RESEARCH SCIENCE & MANAGEMENT

4. Yan Bao ,Dai Zhou.,2010 Cheng Huang aNumerical simulation of flow over three circular cylinders in equilateral arrangements at low Reynolds number by a second-order characteristic-based split finite element method. Computers & Fluids882–899.
5. Liu kun, Ma Dong-Jun, Sun De-Jun, In Xie-Yuan., Ser. B, 2007, Journal of hydrodynamics Science Direct ,Wake patterns of flow past a pair of circular cylinders in side by side arrangements at low reynolds numbers, Department of modern mechanics, university of science and technology of china,pp(690-697).
6. Hadan2, Saha, Busan(2006), Journal of Fluids and Structures 22 Elsevier,Uniform-shear flow over a circular cylinder at low Reynolds numbers S. Kang Department of Mechanical Engineering, Dong-A University, , Republic of Korea, 604-714.
7. Park, J., Kwon, K., Choi, H., 1998. Numerical solutions of flow past a circular cylinder at Reynolds numbers up to 160. KSME International Journal 12 (6), 1200–1205. Persillon, H., Braza
8. Zdravkovich , M.M., 1997. Flow around Circular Cylinders, vol. 1: Fundamentals. Oxford University Press, Oxford, UK. Williamson, C.H.K., 1996. Vortex dynamics in the cylinder wake. Annual Review of Fluid Mechanics 28, 477–539.
9. Jendrzejczyk J. A., and Chen, S. S., 1986, Fluid Forces on Two Circular Cylinders in Cross Flow,PVP Am.Soc. Mech. Engg, 104, pp. 1–13.
10. Lee K, Yang KS, Yoon DH. Flow-induced forces on two circular cylinders in proximity. Compute Fluids 2009;38:111–20.