Simulation of Flow Over Three Circular Cylinder of Different Size To Predict Vortex Shedding

Gupta Deepa Dineshkumar[1], Manikant Raman[2], Pankaj Chaudhary[3], Asst. Prof. Mukesh Kumar Rout[4]
[1, 2, 3, 4] Dept of Mechanical Engineering
[1, 2, 3, 4] Dr.D.Y.Patil Institute of Technology, Pimpri Pune-411018

Abstract- *In this paper, two-dimensional flow over three circular cylinders of decreasing diameters arranged in side-by-side, with different spacing ratios and at low Reynolds numbers (Re=100) has been numerically investigated. Flow over single cylinder experiences vortex shedding at low Reynolds number which creates instability on the structures. But when more than one cylinder present at the flow field, their vortex shedding affect each other depending upon surface to surface distance (s/d). So based on above concept, the effect of vortex shedding on three circular cylinders arranged at spacing ratios(s/d) from 1 to 2 has been analyzed to find the optimum distance where the effect of vortex shedding is minimum on the cylinders, with the help of different flow pattern around the cylinders. The results of this experiment has been concluded from the variation of drag and lift coefficients. From s/d ratio 1 to 2, it is seen that as the spacing changes, time signal co-efficient of drag and lift also differs.*

In future, this concept can be analyzed to minimize the effect of vortex shedding on number of cylinders arranged parallel with each other at different Reynolds numbers. This area of research finds major application in the field of heat exchanger, suspension bridge, boiler, radiator, offshore structures, buildings, chimneys and power lines.

Keywords- Reynolds number, Vortex shedding, lift and drag coefficient, Strouhal number.

I. INTRODUCTION

The flow around single cylinders were studied theoretically in fluid mechanics and various flow parameters like vortex shedding, drag and lift coefficient has been analyzed effectively. But for past several years, flow around the multiple cylinders becomes the growing area of research due to the combine effect vortex shedding on each other. So to analyze the effect of vortex shedding, a prototype of three circular cylinders arranged side by side in decreasing order as look like aero foil shape at low Reynolds number (100) is simulated and presented in this paper. The spacing between the cylinders is also different according to s/d ratio. The spacing ratios between the cylinders are considered in three cases (s/d = 1, 2) to study the vortex shedding and turbulent properties in the flow field and the behavior of vortex shedding is analyzed with the help of numerical and graphical simulation. The main results are focused on the drag and lift coefficients, the vortex shedding frequency, the coherent structure, and the scale properties. The change in vortex shedding is very well described by the drag and lift signal with respect to picture of vortex shedding and strouhal number.It is shown that when S D/ is equal to 1., The vortex shedding of the main cylinder is strongly suppressed by the small cylinder, the drag and lift coefficients of the main cylinder are smaller than those in other two cases. While S/D is equal to 1, the vortex shedding of the main cylinder can be improved, the drag and lift coefficients of the main cylinder are larger than those in other three cases. It is shown that there is a linear relationship between the mean period and the mode in the semi-log coordinates. The vortex shedding period of the main cylinder is consistent with the period of the restructured coherent structures quantitatively.

II. COMPUTATIONAL DETAIL AND PROBLEM SPECIFICATION

2.1. Numerical description

The physical model considered here is three different circular cylinders arranged in row shown in Fig.1.1. Let the cylinder diameters (d_1, d_2 and d_3) and the incoming flow velocity U_o be the non-dimensional characteristic length and velocity respectively. The non-dimensional horizontal surface distance between each two cylinders s/d is used to recognize geometrical configuration, where‘s’ is the surface distance between two cylinders. The coordinate origin is located at first cylinder’s centre, and the x-axis is the mean flow velocity direction, while y-axis velocity is kept zero. Initial condition of x-velocity to start the simulation considered as a zero. The differential equation governing incompressible, viscous, unsteady and 2d-fluid flow comprises the continuity equations which are written as follows:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \qquad (2.1)$$

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{\partial p}{\partial x}+\frac{1}{Re}\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right] \quad (2.2)$$

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{\partial p}{\partial x}+\frac{1}{Re}\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right] \quad (2.3)$$

Where u and v are the velocity components, $Re=\frac{\rho u d}{\mu}, \mu$ *is* the dynamic viscosity of the fluid and ρ is the fluid density. The density is assumed to be constant in calculation due to the incompressibility of fluid. Air is used as fluid medium.

2.2 Problem Specification

As Flow is incompressible with =1.225 and throughout the current study. The flow configuration is shown in Figure 2.1. Three fixed two-dimensional circular cylinders with diameter d_1=0.02 m ,d_2=0.015 m and d_3=0.01m which is also characteristic length scale, are exposed to constant and uniform velocity U^{∞}. The Length of computational domain in stream wise direction is taken to be $L_x = L_u + d + L_d$, where L_u is the upstream length and L_d the downstream length from the origin. In this case L_u = 7d and L_d =10d are used for the computational domain. The number of points along the lateral direction is L_y = (3*(8d+s)), where s is the spacing between two cylinders. Using above two formula of Lx and Ly the different boundary dimension are created for s/d=3 to s/d=1. At the inlet of the flow field, the boundary condition are applied (u= U^{∞}=0.22 m/s, v=0) where Uo is the free stream velocity, and at the outlet the pressure P=0. Symmetric condition is applied on top and bottom boundary. No-slip boundary condition is imposed on all the cylinder surfaces.

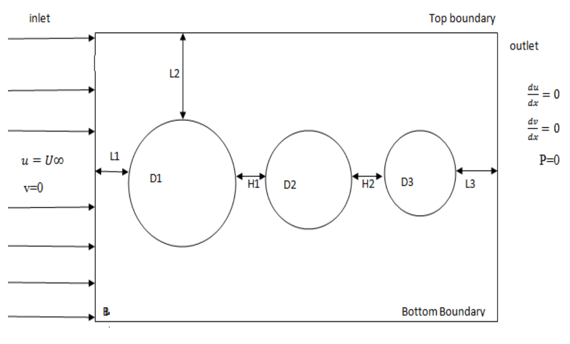


Fig.2.1 Computational domain for three side-by-side cylinder

III. OBJECTIVES

Here forth, main objectives shall to be finding the following four parameters and compile them with previous different research results:

$$C_l=\frac{F_L}{\frac{1}{2}\rho A u^2} \quad C_d=\frac{F_D}{\frac{1}{2}\rho A u^2} \quad St=\frac{f\,d}{U_\circ} \quad Re=\frac{\rho\, d\, U_\circ}{\mu}$$

Where F_L and F_D denote lift and drag force respectively. Re is Reynolds number and St is the non-dimensional form of vortex shedding frequency, called Strouhal number, d is the cylinder diameter, and *f* is the frequency of vortex shedding, which can be calculated from oscillation frequency of lift force.

(1) To study the effect of surface facing ratio at Reynolds number on three circular cylinders arranged in tandem (with four different arrangements).
(2) To examine the wake interactions behind the cylinders in each case.
(3) To examine the effect of Reynolds number, spacing ratio, mean drag coefficient, root mean square of lift and strouhal number on cylinders

To compare and validate the results with other such experiments.

(Note: Assumed data shall be the diameters of the cylinders, their distances both horizontal and vertical in a controlled boundary zone, the initial velocity of fluid, distance between two cylinders and s/d factor.)

IV. SOLUTION PROCEDEURE

The discretization of the governing equation is done in the FLUENT using finite volume method, where first order implicit approach is used for discrimination of the equation. The SIMPLE pressure-velocity coupling approach is applied to the grid cell for simulating the problem. The gradient are solved using least square cell method. Pressure equation is standard and second order upwind method is used to solve the momentum equation. The SIMPLE approach include guess of pressure field, solution of momentum equation, calculation of total residual , pressure velocity correction and iteration of all steps until the full convergence is reached. This residual includes residual of momentum and velocity correction residual from continuity equation. Hence choosing very small value of residual, 10-6 in this paper, seems appropriate and adequate. As matter of fact, that this value represent the intrinsic error of discretization which is unavoidable .considering the present approach for discretization of governing equation, lower value than this would have no significant effect on the accuracy of present solution.

V. SIMULATION STUDY

Case 1: s/d=2

Here we observe sharp changes in the wake and vortex formations. The wake of the upstream cylinder interferes with the flow characteristics of the downstream cylinder. It is also observed that, the downstream cylinder undergoes a negative drag due to full submergence in the relatively low pressure wake region behind the upstream cylinder. The presence of the downstream cylinder leads to the pressure increase in this region, causing a reduction in upstream cylinder drag force.

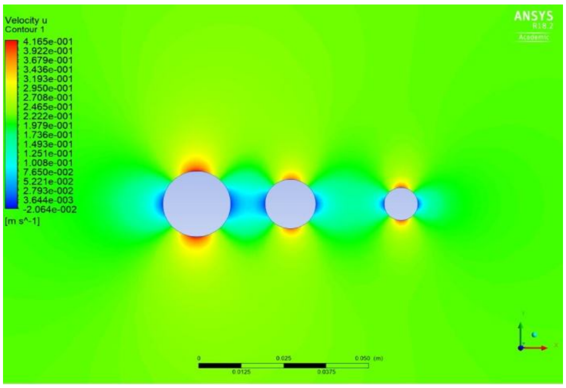
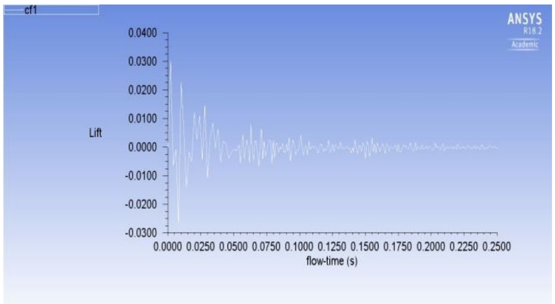
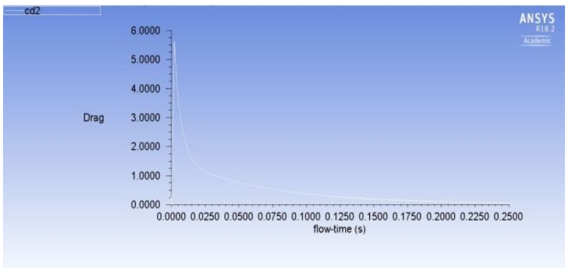


Fig.5.1 Contours of Velocity Magnitude



(a)



(b)

Fig 5.2 (a) Lift coefficient (b) Drag coefficient

Case 2: s/d=1

We have now reduced the distance between two successive cylinders from 20 mm to 10 mm. The interference between the fluid characteristics of each cylinder with other further increases. The presence of the downstream cylinders leads to the increase of pressure before them and causing a reduction in the upstream cylinder drag force. This saves the upstream cylinder from damage but the downstream cylinders are severely affected.

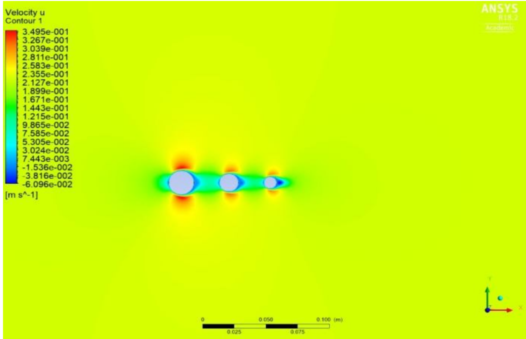
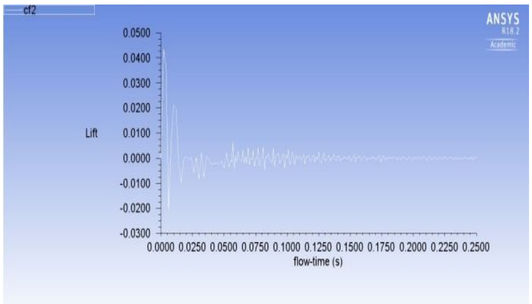
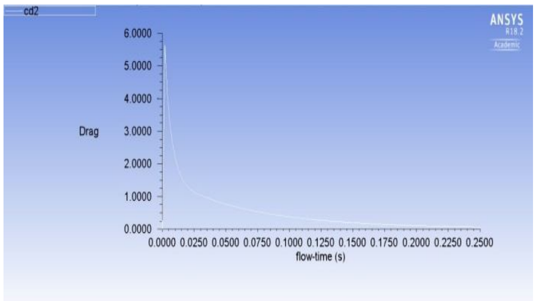


Fig.5.3 Contours of Velocity Magnitude



(a)



(b)

Fig 5.4 (a) Lift Coefficient (b) Drag Coefficient

VI. CONCLUSION REMARK

A Numerical simulation of flow over cluster of three circular cylinders in side-by-side as an object of study carried out using ANSYS/CFD finite volume method were performed for different surface spacing ratios: S/D=3,2,1 and Reynolds number of 100. Comparing to other methods, it has advantage of higher accuracy. The aim of this project to understand the physics of flow around more than one circular cylinders interaction in flow field, the flow pattern for same spacing ratio but with cylinders of different diameters have been studied numerically, and several new flow patterns with respect to time signal have been found. The present result of circular cylinder is compared with square cylinder for Re=100. At Re=100, it is evident that as the gap between cylinders increases, the upstream cylinder shows the same behaviors as a single cylinder and its lift and drag time histories have confirmed this.

VII. FUTURE SCOPE

1. Air flow and heat transfer around chimney stacks and cooling towers, heat losses from tall buildings and heat transfer in heat exchangers and boilers are just some of the applications. Measurement of aerodynamic forces acting on a body like automobile or flying object is very important and effective in design and manufacturing processes.

   Examples of its various applications in practical engineering areas include: offshore platforms, transmission cables, cooling towers, heat exchanger tubes, chimney stacks and marine risers.
2. Since many high-rise buildings are affected by other nearby buildings, their design must consider the aerodynamic forces acting on the structures. It is also important to investigate the characteristics of vortex shedding caused by the flow interference of other structures since this may be connected to structural vibration problems.
3. 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, in both air- and water-flow.

VIII. ACKNOWLEDGEMENTS

We are grateful towards our project guide Prof. Mukesh Rout for their support and guidance to carry out such project.

REFRENCES

Reference From books

[1] Hydraulics and Fluid mechanics by Dr. P.N. Modi and Dr.S.M.Seth

Papers:

[2] Control of vortex shedding behind circular cylinder for flows at low Reynolds numbers by S. Mittal and A. Raghuvanshi.(International Journal For Numerical Methods In Fluids Int. J. Numer). Meth. Fluids 2001; 35: 421-447.

[3] Flow past a cylinder: shear layer instability and drag crisis S. P. Singh, S. Mittal S. P. Singh, S. Mittal Department of Aerospace Engineering Indian Institute of Technology Kanpur.

[4] Numerical simulation of the flow around rows of cylinders,Z. Huang a,b, J.A. Olson a,b, R.J. Kerekes b,c, S.I. Green a,b,*.Computers & Fluids 35 (2006) 485–491

[5] Measurements of the flow fields around two square cylinders in a tandem arrangement, Journal of Mechanical Science and Technology 22 (2008) 397~407)

[6] Two circular cylinders in cross-flow: A review by D. Sumner, Journal of fluids and structures 26 (2010) 849-899.