Numerical Investigation of Flow Induced Vibration for the Triangular Array of Circular Cylinder

Hardik R Gohel\textsuperscript{a}, Balkrushna A Shah\textsuperscript{b}, Absar M Lakdawala\textsuperscript{a,b,*}

\textsuperscript{a}Mechanical Engg. Dept, Gandhinagar Institute of Technology, Gandhinagar, 382721, India
\textsuperscript{b}Mechanical Engg. Dept, Institute of Technology, Nirma University, Ahmedabad, 382481, India

Abstract

Flow Induced Vibration (FIV) in Cross Flow around Triangular Array of Circular Cylinders with two-degree of freedom is studied numerically, where the cylinder was allowed to vibrate in the transverse (cross-flow) and longitudinal directions. Computational domain, Grid, Time step was optimized by performing Domain independent study, Grid independent study, Time step independent study. The computations were carried out at high Reynolds number range of 500, 1000, 2000, 5000 and $10^4$ with multi-cylinder with change in upstream cylinder positions at 20°, 30°, 45° and 60° and varying pitch at $1.5D$ and $2D$, Using FLUENT (version 6.3). Effects of Reynolds number on lift and drag force at various multi-cylinder arrangements were studied. The effects of Reynolds number on flow parameters such as drag coefficient lift coefficient, pressure coefficient, Strouhal number, lift and drag forces and vorticity were established. The lift and drag force on cylinder was used to solve the vibration equation, hence displacement of the cylinder which as validated with analytical as well as experimental results. A program was developed to calculate cylinder displacement and it has been employed successfully for the calculation and prediction of induced vibration of a circular cylinder.

© 2013 The Authors. Published by Elsevier Ltd. Selection and peer-review under responsibility of Institute of Technology Nirma University, Ahmedabad.

Keywords: Flow Induced Vibration, Cross-Flow over a cylinder, Effect of lift and drag on cross flow over a tube bundle, Karman Vortex Shedding

Nomenclature

\begin{itemize}
  \item $C_D$: Drag co-efficient
  \item $C_L$: Lift co-efficient
  \item $D$: Diameter of cylinder (m)
  \item $F_D$: Drag force (N)
  \item $F_L$: Lift force (N)
  \item $\mu$: Viscosity of free stream fluid (Pa·sec)
  \item $\rho$: Density of fluid (kg/m$^3$)
  \item $\Theta$: Orientation angle (degree)
  \item $f$: frequency of vibration (Hz)
  \item $k$: spring constant (N/m)
  \item $m$: mass per unit length (kg/m)
\end{itemize}

E-mail address: hardik.gohel@git.org.in.
1. Introduction

The problem of vortex-induced vibration of structures is important in many fields of engineering. It is a cause for concern in the dynamics of riser tubes bringing oil from the seabed to the surface, in flow around heat exchanger tubes, in the dynamics of civil engineering structures such as bridges and chimneys, and also in many other situations of practical importance. The wide range of problems caused by vortex-induced vibration has led to a large number of fundamental studies.

The viability and accuracy of large-eddy simulation (LES) with wall modeling for high Reynolds number complex turbulent flows is investigated by Pietro Catalano (2003), considering the flow around a circular cylinder in the super critical regime ($5 \times 10^4$ and $6 \times 10^5$) [1]. Guilmineau (2004) present some numerical results from a study of the dynamics and fluid forcing on an elastically mounted rigid cylinder with low mass-damping, constrained to oscillate transversely to a free stream [2]. Z. Huang (2006) had performed a systematic study of flow around one cylinder, two side-by-side cylinders, one row of cylinders and two rows of cylinders [3]. Experimental measurements and large eddy simulation (LES) technique were used by K. Lam to study the turbulent flow characteristics in a staggered tube bundle arrangement in 2010 at a sub-critical Reynolds number of $Re = 7500$ [4]. K. Lam presents the results of an investigation on the effects of wavy cylindrical tubes in a staggered heat exchanger tube bundle. The aim of this investigation is to compare the flow characteristics of a new configuration of cylindrical tubes with that of a similar arrangement which comprises purely circular cylinders. For the flow induced vibration problem some numerical work also carried out. A numerical study has been carried out by S. Mittal to study the flow-induced vibrations of a pair of cylinders in tandem and staggered arrangements at $Re = 100$ [5]. Detailed numerical results for the flow patterns for different arrangements of the cylinders, at a Reynolds number, $Re = 800$, are presented. Several qualitatively distinct wake regimes were observed experimentally as well as numerically by F.L. Ponta (2006) [6]. In year 1996, C.H.K. Williamson had studied the three dimensional vortex behavior of flow past a bluff body [7]. Results predict that for a low Reynolds number flow ($Re < 260$) in wake region of the bluff body the flow remains two dimensional behavior. For $Re > 260$ the vortex generated due to flow past a bluff body having a three dimensional nature of flow.

In summary, thus, many experimental as well as numerical work at higher Reynolds no is done in FIV area. Some of them have calculated the FIV and suggested the techniques to damp the vibration, where as some work are done experimentally by changing orientation of tube bundle in triangular array. Literatures also suggest the appropriate condition to setup the model of FIV. In spite of all these studies, the investigations of flow induced vibrations as a fully coupled problem are still incomplete. In addition, little theoretical work has been done for the simulation and control of flow induced vibrations. The numerical studies have been carried out using FLUENT (version6.3).

2. Problem Statement and Formulation

Consider unsteady, two-dimensional, viscous, incompressible flow past triangular array of circular cylinder placed in a uniform stream, as shown schematically in fig 1. The flow is bounded by the plane at upper and lower side boundaries. These are treated as symmetry boundary condition, while vertical plane at left side and vertical plane right of the domain are
considered as the flow inlet and outlet planes respectively. Free stream velocity is considered as unity. Three stationary cylinders are placed in triangular array, two upstream sides and one downstream side. The cylinder wall is considered as no slip boundary condition (fig-1). Consider upper upstream side cylinder as cylinder-1, lower upstream side cylinder as cylinder-2 and downstream cylinder as cylinder-3 hereafter. All three stationary circular cylinders are of diameter $D$. It is observed from fig 1 that upstream cylinders were placed at $10D$ and $45D$ distance away from the inlet and outlet boundaries, respectively. The downstream cylinder-3 position ($15D$ from upper and lower boundary) is located with respect to upstream cylinders 1 and 2, by varying pitch $p$ and orientation angle $\theta$. Two different pitch (i.e. $1.5D$ and $2D$) and four different orientation angle $\theta$ ($20^\circ$, $30^\circ$, $45^\circ$ and $60^\circ$) are considered in the present study.

2.1. Mathematical Modeling and Numerical Approach

Due to two-dimensional nature of flow, there is no flow in z-directions and no flow variables depend upon the z coordinate. Under these conditions, the equations of the continuity and momentum for an incompressible fluid reduce to:

Continuity equation:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$ (1)

$X$ momentum equation:

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

$Y$ momentum equation:

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

The variables for length, x and y velocity, time, pressure and temperature in governing equations were converted to dimensionless form as:

$$x = \frac{x'}{D}, \quad u = \frac{u'}{U_e}, \quad v = \frac{v'}{U_e}, \quad t = \frac{t_U t'}{D}, \quad P = \frac{P'}{\rho U_e^2}$$ (4)

Vibration equations,

$$m\ddot{x} + c\dot{x} + kx = F$$ (5)

Force in x direction,

$$F_x = m\ddot{x} + c\dot{x} + kx$$ (6)

Force in y direction,

$$F_y = m\ddot{y} + c\dot{y} + ky$$ (7)

Where, the forces $F_x$ and $F_y$ are the total drag and lift forces acting on the cylinder and $m$, $c$ and $k$ are the tube mass per unit length, structural damping coefficient and spring constant respectively.

Drag and Lift co-efficient,

$$C_D = \frac{F_D}{0.5 \rho U_e^2 D}, \quad C_L = \frac{F_L}{0.5 \rho U_e^2 D}$$ (8)

Where, $F_D$ and $F_L$ are the total drag and lift forces exerted by the fluid acting on cylinder per unit cylinder length. Therefore $C_D = C_{pD} + C_{fD}$ where $C_{pD}$ and $C_{fD}$ are pressure and friction drag coefficients respectively. The frequency of vortex shedding ($f$) in wake region is given by Strouhal number (St) and it is defined as
Vortex shedding from a smooth, circular cylinder in a steady flow is a function of Reynolds number. The Reynolds number is based on free stream velocity $U$ and cylinder diameter $D$,

$$Re = \frac{DU}{\mu}$$  \hspace{1cm} (10)

The cylinder has been modeled as a two degree-of-freedom system with independent responses in x - the drag direction, and y - the lift direction. The initial conditions of the cylinder are zero displacement and velocity. As discussed in the introduction, the response is assumed to be two-dimensional with symmetry along the cylinder axis. The following properties are applied:

- $k$ - Stiffness is 4334.73 $N/m$
- $\zeta$ - Structural damping is 0.01 or 1%, giving a damping co-efficient ($c$) of 0.46 $Ns/m$
- $m$ - Mass per unit length is 0.122 $kg/m$

2.2. Modeling in GAMBIT:

A bottom up modeling approach is followed for the present work. The flow domain is rectangular with multi-cylinder (fig 2). GAMBIT was used to generate a structured multi-block mesh around a cylinder of diameter $D = 1$.

2.3. Computational procedure in FLUENT:

A segregated, unsteady solver with 1st order implicit formulation, laminar viscous model was applied. Fluid properties were defined to adjust the Reynolds number. Operating pressure was selected as atmospheric pressure. Controls for solution were set to PISO second order central difference scheme with no skewness neighbor coupling. Model was initialized at inlet boundary. Animation for each time step was applied for vorticity contours. Total no of time step 34375 and time step size 0.016 was executed for analysis. Vorticity contour, pressure contour, $xy$ plot of total pressure verses curve length, force vectors in drag and lift directions, solving equation of vibration by inserting lift and drag forces to find out the displacement in lift and drag directions is done as a part of post processing work.

3. Results and Discussion

Computational domain, Grid, Time step was optimized by performing Domain independent study, Grid independent study, Time step independent study. Results obtained for $c_s$ and $c_l$ at different grid density, domain size and time step size were compared with the results available in the literature and found that $55D \times 30D$ size of domain, with finer grid and 0.016 time step size is optimum for the further computational work. Prior to presenting the new results obtained in this study, it is appropriate to establish the reliability and accuracy of present results.

3.1. Qualitative Results

The computations were carried out at high Reynolds number range of 500, 1000, 2000, 5000 and $10^4$ with multi-cylinder with change in upstream cylinders positions with respect to downstream cylinder at $20^\circ$, $30^\circ$, $45^\circ$ and $60^\circ$ with varying pitch ($1.5D$ and $2D$). Total 40 simulations on multi-cylinder array were performed for this study. Note that the total number of computations performed (40) on multi cylinder was arrived at by considering two different pitch (i.e. $1.5D$ and $2D$) × four different orientations (i.e. $20^\circ$, $30^\circ$, $45^\circ$ and $60^\circ$) × five different Reynolds number (i.e. 500, 1000, 2000, 5000 and $10^4$).

Fig 3 describes the vorticity generation and Karman vortex street phenomenon for multi cylinder triangular array with $Re = 5000$, domain size $55D \times 30D$, total no of elements 33752 and time step size $\Delta t = 0.016$. Fig 3 shows the multi cylinder arrangement with triangular array with $p = 2$ and angle between downstream cylinder $30^\circ$. Note that in these cases time step size is taken as $\Delta t = 0.016$, and Reynolds number taken as $Re = 5000$. 

On cross flow over multi cylinder it was observed that the flow got supported from upstream cylinder and no wake formation was observed between upstream and downstream cylinder. However, the length of recirculation zone increases as pitch between upstream and downstream cylinder increase from $1.5D$ to $2D$. It was also observed that, for all configurations the cylinder array acted as a bluff body with recirculation zone between upstream and downstream cylinder. It was observed that the vortex of negative vorticity was started from the upper upstream cylinder and the vortex of positive vorticity was started from the lower upstream cylinder with a gap flow in between the upper and lower upstream cylinder. However, because of the influence of gap flow between upper and lower upstream cylinder, two separate vortex street generated, due to single centre downstream cylinder these vortex street get separated from each other and wake becomes wider (Fig 3). Similar trend were observed at different orientation of angle and different pitch. It was also observed that the onset of two different vortex street formations from upper upstream cylinder and lower upstream cylinder was affected by orientation angle and pitch.

3.2. Quantitative Results and Discussion

For the flow past a bluff body, the drag and lift coefficients at the surface of body are two very important parameters. The time evolutions of these two characteristic parameters illustrate the variation of the flow field. It can be observed that lift coefficients show obvious periodic oscillations. This implies the periodic variation of flow field. It can also be found that the lift coefficient oscillates with larger amplitude than the drag coefficient. These phenomena are consistent with those observed by experimental results. The reason is that the lift coefficient is affected by vortex shedding process from both sides of the cylinder. Because of the periodic flow over the cylinder, the cylinder starts to vibrate and it was found that as Reynolds no. increases the displacement increases. Fig 4 shows the comparison of drag co-efficient with varying pitch of the cylinder ($1.5D$ and $2D$) with four different angle of arrangement $20^\circ$, $30^\circ$, $45^\circ$ and $60^\circ$. The drag co-efficient on the body tends to zero as the Reynolds number tends to infinity. As the Reynolds number increase the wake region becomes wider and short. In fig 4 circle represents the negative drag value. The negative drag on the cylinder number 3 (downstream cylinder) was observed which is considered to be the new outcome of the present study.

4. Conclusions

From the present study, it was observed that lift coefficients show obvious periodic oscillations. It was also be found that the lift coefficient oscillates with larger amplitude than the drag coefficient. It was found that the drag and lift co-efficient on cylinder-1 (upper upstream cylinder) and cylinder-2 (lower upstream cylinder) did not varying much with increase in the Reynolds number as there is attached recirculation zone downstream of both the cylinder. The drag co-efficient decrease and lift co-efficient increase with increase in Reynolds number for cylinder-3. However, the drag co-efficient did vary with orientation angle and pitch as both these factors affect the length of recirculation zone.

It was also observed that, because of influence of gap flow, vortex generated from upper and lower upstream cylinder interacts with the vortex generated from downstream cylinder and in between. Maximum vibration in lift direction was predicted on the available data. It is observed that max displacement of Cylinder-1 (upper upstream cylinder) is highest at triangular arrangement with $\theta = 30^\circ$ then the rest of the orientations (i.e. $20^\circ$, $45^\circ$ and $60^\circ$) at $Re = 5000$. While, the minimum displacement observed at $Re = 5000$ with $\theta = 45^\circ$. For cylinder-3, it is observed that maximum displacement at $Re = 5000$ measured at $\theta = 60^\circ$.
5. References


