Fluid Structure Interaction of Flow Induced Vibration: A study of Laminar and Turbulent flow fields*

Nadeem Ahmed SHEIKH**, Qamar IQBAL***, Shahab KHUSHNOOD*** and Ali El GHALIBAN***
** wolfson School, Loughborough University, UK
E-mail: nadeem_sheikh@hotmail.com
*** University of Engineering & Technology, Taxila, Pakistan

Abstract
The flow around a circular cylinder is a traditional problem of fluid dynamics, knowledge of which is essential for basic understanding as well as for technical applications, such as large buildings, bridges, standpipes, heat exchanger tubes, rods, transport pipelines, poles and cables, all of which attracted widespread attention. A circular cylinder usually experiences boundary layer separation. In certain Reynolds number range, a periodic flow motion develops in the wake as a result of boundary layer vortices being shed alternatively from either side of the cylinder leading to unwanted structural vibrations. In order to calculate the cylinder response to the flow, a computational method to solve the flow around the body and its resultant vibration using Fluent® is previously developed and validated. The present study details the extension and verification using the same method by incorporating flow turbulence through its modeling. The incoming free stream flow is uniform with Reynolds number based on diameter of 3.8 and 12.7mm. Results for the unsteady shedding flow behind a circular cylinder and its vibration are presented with experimental comparisons, along with a comparison of two-dimensional laminar as well as turbulent models of the flow for fully coupled interaction. The Strouhal number and structural displacements are in good comparison with the experimental data of [2] showing the capability of FSI method using Fluent® to tackle laminar as well as turbulent flows.

Key words: Computational Fluid Dynamics, Finite Volume Method, Vortex Shedding, Fluid Structure Interaction

1. Introduction
The flow around a circular cylinder is a classical problem of fluid dynamics, knowledge of which is essential for basic understanding as well as for technical applications, such as grand buildings, bridges, standpipes, transport pipelines, heat exchangers, nuclear fuel rods, tubes poles and cables, all of which attracted widespread attention. 
Flow vibrations are mostly induced by

- Vortex shedding
- Fluid elastic instability
- Turbulence and
- Acoustic resonance.

Among the above four vortex shedding is the principal excitation mechanism for flow-induced vibration in cross flows, producing alternating forces, which occur more frequently if the flow velocity is increased. In certain Reynolds number range, a periodic
flow motion will develop in the wake as a result of boundary layer vortices being shed alternatively from either side of the cylinder. This regular pattern of vortices in the wake is called a Von Karman vortex street. It creates an oscillating flow at a discrete frequency that is correlated to the Reynolds number of the flow. The periodic nature of the vortex shedding phenomenon can sometimes lead to unwanted structural vibrations, especially when the shedding frequency matches one of the resonant frequencies of the structure.

Complex fluid-structure interactions resulting from the free vibrations of a two-dimensional elastic cylinder in a cross flow are not well understood. The periodic nature of the vortex shedding phenomenon can sometimes lead to unwanted structural vibrations, especially when the shedding frequency matches one of the resonant frequencies of the structure [1].

Experimental data pertaining to the interaction behavior is rather scarce [2]. The motivation of the research is to employ flow-structure interaction methods based on solving the Navier-Stokes and structural dynamics equations of motion to provide predictions of the forces and responses of structures. The technique Computational Fluid Structure Interaction (CFSD) attempts to examine this problem numerically using a commercial CFD code (Fluent®) to assess the shedding frequencies, force and using structural solver to find the resultant displacements and velocities in the wake [3].

The present study examines turbulent flows over a two dimensional circular cylinder in addition to laminar flow calculation [3], although the technique can be extended to three dimensions as well. Numerical runs are carried out over a range of reduced velocities. The reduced velocity was varied by using cylinders of two different diameters. The near-wake flow behind the elastic cylinder, at three different Reynolds numbers in the sub-critical range, is studied in detail for the Strouhal number calculations in comparison with experimental results [2].

2. Computational Fluid Structure Interaction

There are generally two types of methods to calculate the fluid-structure interaction problems: the fluid and structure governing equations are

**Loosely Coupled:** The loosely coupled model means that the structural response lags behind the flow field solution. Within a time step for the loosely coupled method, the structure solver calculates the response after the flow solver is converged. This kind of methods may be limited to first-order temporal accuracy only regardless of the temporal accuracy of the individual solvers.

**Fully Coupled:** The fully coupled model is that the flow field and structure always respond simultaneously by exchanging the aerodynamic forces and structural displacements at each iteration in a single physical time step. Obviously, only the fully coupled model is rigorous in physical sense. Due to the complicated fluid-structure interaction phenomenon such as transonic stall flutter, oscillating shock waves and flow separation, etc., the fully coupled model between the fluid and structure system is necessary and is selected for this research to achieve high accuracy. Here a fully coupled scheme is implemented for all the cases.

Here Computational Fluid Structure Dynamics utilizes an implicit approach to the solution of the unsteady two-dimensional Navier-Stokes equations for computation of flow parameters. This is accomplished using constant physical time stepping in the calculations. Solutions computed using the multi-block scheme in which the dual time stepping loops has also been specified. Calculations are performed in parallel using a domain re-meshing/deforming technique with communication requirements [3]. The block diagram shows the procedure of coupling in figure 01.
Details of finite volume technique for computational fluid dynamics as well as different computational model for structural response can be found in [2, 3].

3. Structural Model

The cylinder motion can be approximated by a spring-damper mass model which permits translational motion along the stream (x) and transverse (y) directions only. Under these simplifications the complete model is supposed to represent the first bending mode in both coordinate directions of a long cable, pipe or such long cylindrical object in water or air. Two types of models are used for the calculation structural response based on the degrees of freedom available for the cylinder.

3.1 Single DOF Model

In this case the cylinder is connected to a single linear springs along longitudinal axes. Since both lift and drag are functions of time, displacements in Y directions will be excited by the unsteadiness in the flow.

\[ m \ddot{y} + C_y \dot{y} + K_y y = L \]
3.2 Double DOF Model

In this case the cylinder is connected to two linear springs along each of the coordinate axes. Since both \(L\) and \(D\) are functions of time, displacements in both coordinate directions will be excited by the unsteadiness in the flow.

![Double DOF Model](image)

The governing equations of the structural model are simply:

\[
mx'' + C_x x' + K_x x = D
\]

\[
my'' + C_y y' + K_y y = L
\]

It is shown that a two dimensional model produces more rational results in comparison with a one dimensional model [3]. So the results presented here are all based on two dimensional structural model.

4. Results

A set of simulations of flow past an elastically-mounted cylinder was run to determine the response. Two cylinders of same material, brass, but of different diameters are considered for simulation. The brass pipe is assumed to be of uniform diameter internally and externally with the density of \(8.9 \times 10^3 \text{ kg/m}^3\). The cylinders are fixed at both ends and the first or principal natural frequencies are 102 and 341 Hz respectively in bending mode. The cross section under consideration is taken at the half length of the structure. The damping ratios \(\zeta\) are taken as 0.0004 and 0.020 for the two cylinders respectively. The details of experimental computation of natural frequency and damping methods are presented in [2]. The summary of the inputs for simulation as utilized in [2] is as given in Table 1:
Table 1 Geometry Input

<table>
<thead>
<tr>
<th>Material</th>
<th>Brass</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Geometric parameters</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Nomenclature</strong></td>
<td><strong>Cylinder a</strong></td>
</tr>
<tr>
<td>Major Diameter</td>
<td>3.8 mm</td>
</tr>
<tr>
<td>Minor Diameter</td>
<td>3.1 mm</td>
</tr>
<tr>
<td>Length</td>
<td>500mm</td>
</tr>
</tbody>
</table>

The summary of inlet flow parameters are as tabulated in Table 2.

Table 2 Flow parameters

<table>
<thead>
<tr>
<th>Reynolds Number</th>
<th>Cylinder a</th>
<th>Cylinder b</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Velocity (m/s)</strong></td>
<td><strong>Velocity (m/s)</strong></td>
<td></td>
</tr>
<tr>
<td>2500</td>
<td>9.93</td>
<td>3.01</td>
</tr>
<tr>
<td>4600</td>
<td>18.0</td>
<td>5.76</td>
</tr>
<tr>
<td>6000</td>
<td>23.7</td>
<td>7.10</td>
</tr>
</tbody>
</table>

The multi – block structured mesh is used for simulation. As convention the flow inlet is on the left side of the figure 04. The right side and upper and lower boundaries are outlets. Three different mesh dimensions are used to conduct the mesh refinement study for the stationary cylinder consisting of 27504, 34832 and 35,742 cells respectfully. The mesh size of 34832 and 35742 find better approximation for the flow parameters. The second mesh size of 34832 is used as the baseline mesh for all the computations involving flow past a cylinder. The 2D grid consisted of 34,832 cells and 6 blocks, and extended 50 diameters into the far-field. Strouhal number is calculated through FFT of the lift force [3].

Figure 04: 2D grid around Circular cylinder

For turbulent model of flows the standard “two equations Wilcox k-ω turbulence model” is used [4]. This model shows best results as compared to other two equation and one equation models present in Fluent. This is due to better wake treatment of the turbulence model. The inlet flow parameters are described in table 2 along with 0.2% turbulence intensity and turbulent viscosity is taken as 1% of the actual viscosity to introduce the inlet turbulent flow conditions. The fixed time step size a minimum of 60 iterations per time step is set for optimal convergence. The results are presented for the turbulent flow in figure 5 and 6, showing much improved wake formation.
Figure 5: Contours of turbulent Kinetic energy. These illustrate the formulation of vortex structure. The introduction of turbulence model significantly improves the formation of oppositely spinning vortex structures. The extension of these structures is up to 30 diameters downstream.

Figure 6 Contours of velocity magnitude.

Table 3 shows the two-dimensional motion in laminar and turbulent flow field, and introduction of turbulence improve the results reasonably. As indicated in figure 7 especially for the case a cylinder which is lesser stiff than the other one.

<table>
<thead>
<tr>
<th>Test Case</th>
<th>Re #</th>
<th>Shedding Frequency</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>FLUENT</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Laminar</td>
</tr>
<tr>
<td>a</td>
<td>2500</td>
<td>535</td>
</tr>
<tr>
<td></td>
<td>4600</td>
<td>1052</td>
</tr>
<tr>
<td></td>
<td>6000</td>
<td>1380</td>
</tr>
<tr>
<td>b</td>
<td>2500</td>
<td>48.7</td>
</tr>
<tr>
<td></td>
<td>4600</td>
<td>90.1</td>
</tr>
<tr>
<td></td>
<td>6000</td>
<td>117.5</td>
</tr>
</tbody>
</table>
The y-displacement results of the corresponding cylinders are further processed to generate the root mean square value of the cylinder displacement. Figure 8 and 9 compares flow induced vibrations of the laminar and turbulent flows. Root mean square values of case a cylinder shows significant change in vibration amplitudes in turbulent field. This reduction in the amplitude is due to the fluid damping and excessive inertial and viscous dissipation in turbulent field and can be termed as turbulent flow induced damping or “turbulent damping”. Although it is less significant for the stiffer cylinder in case ‘b’, figure 8, but as the Reynolds number is increased the exhibition of turbulent damping is observed.

Figure 7: Strouhal Number Comparisons

Figure 8: Comparison of the root mean square values for the Case a cylinder.

Figure 9: Comparison of the vibrating root mean square values for the two cylinders.
The formation of the wake structure and its pattern is important for the study of the S, P, P plus S and 2P vortex shedding. The wake structure with the turbulence model introduction is greatly improved as indicated in figure 10.

Figure 10: Different Wake Structure for the turbulent flow simulation

5. Conclusion

Results show that the root mean square displacement at mid-span y-rms increases as the Reynolds number increases. The vortex shedding frequency, which is the most prominent in the lift force generation and velocity output, is correctly measured by the FFT of the Lift force. Furthermore, the transverse displacement also shows the multiple frequencies which are characteristics of the unsteady drag indicating the generation of multiple orders of vortex structures. Thus, the findings show that fluid-cylinder interactions far away from synchronization are essentially a linear process. The flow field is significantly affected by the introduction of turbulent model. Notable effects include a large increase in wake structure and vortical motion. Even the resultant cylinder motion is also affected indicating the presence of turbulent structures introducing “Turbulent damping” due to higher viscous and inertial damping in turbulent flow field. Introduction of turbulence at low Reynolds number seems to reduce the cylindrical vibrations. Turbulence at low Reynolds number appears to dampen the energy levels by dissipation of fluctuating fluid energy. The findings are significant and more findings are possible for three dimensional simulations with Large Eddy Simulation (LES) or Detached Eddy Simulations (DES) in future.
Acknowledgements

The author is indebted to Air University Islamabad, Pakistan for providing support in completing the present research work.

Nomenclature

Re #  Reynolds Number
FIV  Flow-Induced Vibration
CFD  Computational Fluid Dynamics
y  Y-displacement
x  X-displacement
L  Lift Force
D  Drag Force
m  Mass
K_{x,y}  Spring Stiffness in x and y directions
C_{x,y}  Damping in x and y directions
FSI  Fluid Structure Interaction

Reference


