RUMO À REDUÇÃO DO ESFORÇO COMPUTACIONAL NAS PREDIÇÕES DE VIBRAÇÕES INDUZIDAS POR VÓRTICES DE UM RISCADOR CILÍNDRICO

TOWARDS REDUCING COMPUTATIONAL EFFORT IN VORTEX INDUCED VIBRATION PREDICTIONS OF A CYLINDRICAL RISER.

VIDYA, Chandran¹; SHEEJA, Janardhanan²; SEKAR, M^{3*};¹Department of Mechanical Engineering, Karunya Institute of Technology and Sciences, Tamil Nadu, India²Department of Mechanical Engineering, SCMS School of Engineering and Technology, Kerala, India³Department of Mechanical Engineering, AAA College of Engineering and Technology, Tamil Nadu, India

* Correspondence author

e-mail: mailtosekar@gmail.com

Received 23 August 2019; received in revised form 30 October 2019; accepted 18 November 2019

RESUMO

As vibrações induzidas pelo fluxo geralmente denominadas vibrações induzidas por vórtices são de grande importância no projeto de *risers* marinhos. Esses *risers* cilíndricos flexíveis sofrem vibrações de amplitude muito alta quando a frequência de derramamento de vórtice corresponde à frequência natural do riser. Tais vibrações são capazes de colocar a segurança da tripulação trabalhando em plataformas *offshore* em questão. Portanto, a previsão de resposta de tais estruturas é considerada muito importante. Embora muito trabalho numérico tenha sido feito neste campo, tratando o problema como uma interação fluido-estrutura bidirecional, o fato de esses trabalhos exigirem esforços computacionais muito altos não o tornou pertinente quando os recursos computacionais de ponta não estão prontamente disponíveis. Uma rápida previsão da resposta estrutural de tais estruturas esbeltas precisa ser útil para os engenheiros em momentos de necessidade. Este artigo aborda uma técnica de solução para esse problema através de um método econômico para previsão rápida e confiável da resposta do riser sob vibração induzida por vórtice, utilizando o esforço computacional mínimo para o número moderado de Reynolds ($Re \leq 3 \times 10^5$). As simulações de fluxo bidimensionais são realizadas usando CFD baseado em RANSE, seguido pelo mapeamento uniforme das forças hidrodinâmicas no *riser* tridimensional. A grade usada para a simulação numérica foi validada com relação aos resultados experimentais do túnel de vento para $Re = 5,3 \times 10^4$. As forças hidrodinâmicas correspondentes aos três primeiros harmônicos da frequência natural do *riser* foram usadas como entrada no solucionador estrutural para analisar a resposta usando o método dos elementos finitos. Obtiveram-se trajetórias do cilindro nos três primeiros modos de vibração, um padrão típico de oito algarismos, característico da vibração de bloqueio. Verificou-se que o método é bastante eficaz no cálculo rápido de problemas de vibração induzidos por fluxo para números de Reynolds baixos e moderados.

Palavras-chave: CFD; cilindros de fluxo passado; lock-in; resposta estrutural; vibrações induzidas por vórtices

ABSTRACT

Vibrations induced by flow, generally referred to as vortex induced vibrations, are of great importance in the design of marine risers. These flexible cylindrical risers undergo vibrations of very high amplitude when the vortex shedding frequency matches the natural frequency of the riser. Such vibrations are capable of putting the safety of crew working on offshore platforms in question. Hence the prediction of response of such structures is considered very important. Although a lot of numerical work has been done in this field treating the problem as a two-way fluid structure interaction, the fact that these works demand very high computational efforts has not made it pertinent where high end computing resources are not readily available. A quick prediction of the structural response of such slender structures needs to be handy to the engineers at times of need. This paper addresses a solution technique for such a problem through an economical method for quick and reliable prediction of riser response under vortex induced vibration utilizing minimum computational effort for moderate Reynolds number ($Re \leq 3 \times 10^5$). Two dimensional flow simulations are carried out using RANSE based CFD followed by the uniform mapping of hydrodynamic forces on to the three dimensional riser. The grid used for the numerical simulation has been well validated against wind-tunnel experimental results for $Re = 5.3 \times 10^4$. Hydrodynamic forces corresponding to the first three harmonics of natural frequency of the riser have been used as input in the structural

solver to analyse the response using finite element method. Trajectories of the cylinder in the first three modes of vibration have been obtained, a typical eight figure pattern which is characteristic for lock-in vibration. It is found that the method is quite effective in the quick computation of flow induced vibration problems for low and moderate Reynolds numbers.

Keywords: CFD; flow past cylinders; lock-in; structural response; vortex-induced-vibrations.

1. INTRODUCTION

Vortex shedding around bluff bodies is a natural yet a phenomenon that consumed years of comprehensive studies, for it is well known for the imminent catastrophes it brings with it. Tacoma Narrows Bridge disaster is the worst case one could recall while thinking about vortex shedding. With the ever-rising demand for petroleum products, the development of offshore oilfields has been growing fast over the past century. The drilling facilities are designed in such a way that it enables a prolonged offshore operation for a large period of time starting from a few months to several decades. Numerous studies are being carried out in this field for the proper design of the slender marine risers in ocean. The stability of structures especially those carrying pressurised fluid in them is a topic of research interest. (Ramírez *et al.* 2017, Pezzini, *et al.*, 2017) If the bluff structure is not mounted rigidly and the frequency of vortex shedding matches the natural frequency of the structure, the structure begins to resonate, vibrating with harmonic oscillations of large amplitude (Bourguet, 2011). This phenomenon is known as lock-in. During lock-in, vortex shedding frequency shifts to the natural frequency of the structure leading to large amplitude vibrations.

The vortex shedding occurs at a discrete frequency and is a function of the Reynolds number, defined by Equation 1.

$$Re = \frac{\rho V D}{\mu} \quad (\text{Eq. 1})$$

The dimensionless frequency of the vortex shedding, the shedding Strouhal number, $St = f_v D/V$, is approximately equal to 0.2 when the Reynolds number is greater than 1,000. When vortices are shed from the cylinder, uneven pressure distribution develops around the upper and lower surfaces of the cylinder, generating an oscillatory hydrodynamic loading (lift) on the cylinder. This unsteady force given by Equation 2

can induce significant cross flow vibrations on a structure, especially if the "resonance" condition is met.

$$F_L = \frac{1}{2} C_L \rho A V^2 \quad (\text{Eq. 2})$$

C_L is the coefficient of lift. The cylinder also experiences a net force along the flow direction and is called the drag force and is given by Equation 3.

$$F_D = \frac{1}{2} C_D \rho A V^2 \quad (\text{Eq. 3})$$

where C_D is the drag coefficient.

The phenomenon of lock-in was first observed by Feng during his classical experiment (Feng, 1968). He described the phenomenon observed as matching the frequency of cylinder vibration and fluid force to the natural frequency in a vacuum. Later it was observed that this matching of frequency holds good only for higher values of mass ratio, m^* (Blevins, 1990). Further research in the field explains the phenomenon as either large amplitude vibration of the cylinder (Sarpkaya, 1977) or matching of the frequency of cylinder vibration and fluid force (Khalak and Williamson, 1999). Synchronization and lock-in, are often used synonymously, but from the experiments it was shown that, for zero damping and sinusoidal motion, synchronization ($f = f_n$) occurs at only one condition, effective stiffness ratio, $k_{\text{eff}}^* = 0$ (Sheils *et al.*, 2001), where k_{eff}^* is defined as in Equation 4.

$$k_{\text{eff}}^* = \frac{m^*}{V_R^2} \quad (\text{Eq. 4})$$

Matching of vortex shedding frequency with one of the natural frequencies of the structure may not always be the only (sole) reason for lock-in. Contrary to classical lock-in, whereby the

oscillation frequency matches the structural natural frequency, in the experiments on stationary cylinder free to oscillate, it was found that the oscillation frequency increases markedly above the natural frequency, through the excitation regime while at the same time it is below the vortex shedding frequency (of the non-oscillating structures) (Khalak and Williamson, 1999). Even though vortex shedding and the vibration induced by it has been a topic of extensive research for several years, due to its intrinsic nature, the researchers are still not able to confidently define the phenomenon and describe the flow physics behind shifting of shedding frequency. Numerous analytical, experimental, and numerical investigations have been carried out in the field of vortex induced vibration (VIV) of long flexible cylinders. Most of these studies focused on the structural response of the cylinders rather than the phenomenon of lock-in (Trim *et al.*, 2005; Vandiver *et al.*, 2009). Freely oscillating cylinders were modelling as a spring mass system with single or two degrees of freedom (Sekar *et al.*, 2009). Experiments in wind tunnels using particle induced velocimetry (PIV) have proved to predict the flow characteristics and structural response with much accuracy (Wang *et al.*, 2015). Laboratory experiments and offshore large scale experiments have also been recognized as effective tools for analysis of VIV (Domal and Sharma, 2017; Gao *et al.*, 2017). However, for case specific analysis of the problem, experiments are not always possible, and hence most of the researchers rely on computational fluid dynamics (CFD) as a tool for predicting VIV (Daniels *et al.*, 2016). Unlike the experiments CFD facilitates detailed study of the flow physics which is otherwise impossible. Three dimensional (3D) numerical simulations are widely accepted in the research community as capable of predicting VIV characteristics accurately. Researchers were under the notion that two dimensional (2D) simulations are acceptable only for lower Reynolds numbers ($Re < 250$) because of the inherent 3D characteristics of vortices. Later on several researchers have proved that 2D simulations are capable of accurately predicting VIV phenomenon in rigid structure cases (Xie *et al.*, 2012). 3D numerical simulation demands high computational resources for flow analysis and also for generation of the 3D computational grid. The objective of the present work is to test the accuracy of 2D numerical simulations in predicting flow characteristics of VIV and to develop an easy and economical tool for comprehensive analysis of vortex shedding and the structural response during VIV. RANSE-based Commercial solver

ANSYS Fluent -15, as well as ANSYS Workbench -15, have been used as the tools for the study.

2. MATERIALS AND METHODS

The grid generated for the analysis has been validated using the experimental method. Experiments have been carried out on a subsonic wind tunnel, and the results have been compared with those of numerical studies for the same value of Reynold's number. Details of the material and dimensions of the test cylinder are shown in Table 1.

Table 1. Geometric specification of the riser and fluid domain

The diameter of the riser (D)	0.05 m
Distance from inlet to the riser	9 D
Distance from riser to outlet	27 D
Lateral distance from cylinder to both sides	7 D
Cylinder Material (Hollow)	PVC

2.1. CFD Prediction for Vortex Shedding

Geometric modeling and the generation of computational grids around the riser placed in a fluid domain, mimicking ocean environment (still water conditions) have been carried out using ANSYS ICEM CFD.

2.1.1. Creating a computational domain

The geometric specifications of the riser model and the fluid domain with respect to the diameter of the riser are given in Table 1. The length of the riser (L) has been chosen to be 0.4 m, so that the aspect ratio (L/D) is 20. At this value of aspect ratio, the model has been found to account for the three dimensional effects of vortex shedding (Vandiver *et al.*, 2009). Figure 1 shows the representation of fluid domain. Domain size has been fixed based on the published domain independency test results (Gutafsson, 2012).

Figure 1. Dimensions of the fluid domain with the riser

Meshing or generating the computational grid in the fluid domain around the riser effectively is most important in capturing the vortex shedding phenomenon. The vorticity transport equation represented by Equation 5 gives an insight into the mechanism of generation and transport of vortices.

$$\frac{\partial \omega}{\partial t} + \mathbf{u} \cdot \nabla \omega + \omega(\nabla \cdot \mathbf{u}) = \omega \cdot \nabla \mathbf{u} + \frac{1}{\rho^2} \nabla p \times \nabla^2 \omega \quad (\text{Eq. 5})$$

The third term on the right hand side of the equation represents the diffusion of vorticity by viscosity. Due to this term, vorticity is generated along solid wall boundaries because of steep velocity gradients. These steep gradients make vortical motion susceptible to numerical dissipation. But in the near wall region, where the mesh is usually very fine, this is not an issue since the fine mesh can capture viscous effects. It is a bigger issue in the far field, where poor resolution can severely weaken and distort the vertical structures (Kamkar, 2011). Hence importance must be given even to the far field wake, where the major concern is the mesh resolution.

2.1.2. Mesh Generation

Mesh element size near the surface of the cylinder is of great importance in case of turbulent flow compared to laminar flow. The interaction between the mean flow and the boundary layer flow is more in turbulent flow, and turbulence plays the most important role in the transport of momentum and hence must be properly resolved, especially at the boundary for better results. To accurately capture the features of flow near the boundary, the spacing of the first grid point should be such that it is well within the laminar sub layer of the boundary layer for turbulent flow and within the boundary layer for laminar flow. In the outset a mesh has been generated, and the drag force computed has been compared with the value

obtained through experiments. The experimental set up is described in the previous section. The flow over the cylinder corresponds to a $Re = 5.3 \times 10^4$ which is in the laminar flow regime. The boundary layer at $Re = 5.3 \times 10^4$ is laminar before separation, but during vortex shedding the wake of the cylinder turns turbulent in nature at any $Re > 300$. Hence while meshing the geometry due consideration must given to the possible influence of turbulence on the boundary layer.

From Blasius's solution of the equation for the boundary layer in laminar flow, represented by Equation 6, the boundary layer thickness, considering the boundary layer to be completely laminar, is 1×10^{-3} m.

$$\delta = \frac{4.91 D}{\sqrt{Re_D}} \quad (\text{Eq. 6})$$

If, for an additional factor of safety, we consider the influence of turbulence on the boundary layer, then the minimum element size near the cylinder wall must be chosen so that it is well with the laminar sub-layer of the boundary layer. The thickness of the laminar sub-layer is obtained from Equation 7.

$$\delta' = \frac{11.6 \nu}{V^*} \quad (\text{Eq. 7})$$

Where V^* is the frictional velocity given by Equation 8.

$$V^* = \sqrt{\frac{\tau_0}{\rho}} \quad (\text{Eq. 8})$$

And τ_0 the wall shear stress is obtained as in Equation 9.

$$\tau_0 = \frac{0.664 \rho V^2}{2\sqrt{Re_D}} \quad (\text{Eq. 9})$$

For $Re = 5.3 \times 10^4$, the thickness of the laminar sub-layer has been obtained from the Von-Karman momentum integral equation as 2.8×10^{-4} m. While generating mesh for computation, 3×10^{-4} m has been fixed as the global minimum seed element size with a scale factor of 1. An unstructured mesh has been generated with view that the same mesh may be used for analyzing the

variation in flow characteristics with oscillating cylinder. Quad dominant mesh type has been preferred in shell meshing parameters as it suits accurate meshing of curved cylindrical surface. Patch dependent mesh method has been selected since it gives the best quad dominant quality while capturing surface details. Tetra/Mixed mesh type and Robust (Octree) mesh method have been selected for surface meshing. Moreover it is to be observed that the first grid point should exhibit a Y^+ (wall normal dimensionless distance) value of less than 1 in case of RANS simulations. Near wall spacing or element size has been calculated as 2×10^{-5} from Equation 10.

$$\Delta S = \frac{\mu Y^+}{\rho V^*} \quad (\text{Eq. 10})$$

This value of near wall spacing is ensured by fixing maximum element size and height 3×10^{-5} m and height ratio 1.05 with 20 numbers of prism layers to cover the entire boundary layer in the part mesh set up of cylinder surface. After a thorough grid independency study, the final mesh for analysis has been chosen with 41,932 elements. Unstructured mesh used for the flow analysis using ANSYS 15 is shown in Figure 2.

Figure 2. Unstructured 2D mesh generated in ANSYS ICEM CFD

2.1.3. Flow Analysis

Flow past the cylinder at $Re = 5.3 \times 10^4$ has been simulated using the generated unstructured mesh in ANSYS Fluent -15. Pressure based transient analysis has been carried out. Fluid flowing has been chosen to be water at density 998.2 kg/m^3 , and the inlet velocity 1.06 m/s in order to match the Re value. $k-\omega$ SST turbulence model has been selected, which has been tested against other models and proven to be the most adaptable model to predict the near cylinder and wake flow characteristics (Chandran *et al.*, 2018). Velocity inlet boundary condition has been given at the inlet, pressure outlet at the outlet boundary, and

for both sides of the domain, symmetry boundary condition (Chandran *et al.*, 2019). Pressure velocity coupling has been done using PISO scheme. Second order upwind spatial discretization has been selected for momentum and turbulent kinetic energy. Second order implicit transient formulation has been used. The spatial discretization gradient is least squares cell based. Since the simulation has been compared with the results obtained from wind tunnel experiments turbulent intensity has been calculated from the empirical correlation for a duct flow given by Equation 11.

$$I = \frac{V'}{V_{\text{avg}}} = 0.16 (Re_D)^{\frac{1}{8}} \quad (\text{Eq. 11})$$

Where V the root mean square of velocity fluctuations and V_{avg} is the mean flow velocity. For $Re = 5.3 \times 10^4$, the turbulence intensity will be 4 % according to Equation 11 and hence so chosen for analysis. The time step size for the transient simulation has been calculated based on vortex shedding frequency corresponding to $Re = 5.3 \times 10^4$. The time period of vortex shedding has been calculated from the definition of Strouhal number (St) given by Equation 12.

$$St = \frac{fD}{V} \quad (\text{Eq. 12})$$

where f is the vortex shedding frequency.

The time period has been obtained as 0.25 seconds from the value of frequency. For accurately capturing the shedding phenomenon one-time period should contain at least 20 time steps. Accommodating 25 time steps per time period, the time step size obtained is 0.01 seconds. Further for ensuring stability of the solution, Courant Friedrichs Lewy (CFL) condition must be satisfied as given by Equation 13.

$$\Delta t = \frac{C_m \Delta x}{V} \quad (\text{Eq. 13})$$

where C_m is the maximum allowable CFL number.

For explicit solver $C_m = 1$. Δx is the minimum element size. The time step size obtained from Equation 13 is 3×10^{-4} seconds. A time step size of 1×10^{-4} seconds has been selected for the transient analysis of the flow with a safety factor. Simulations have been performed for various time step sizes ranging from 1×10^{-3} to

5×10^{-5} . Above a value of 1×10^{-4} the results have been found to be independent of time step size. Simulations have been run for 5 seconds, and it took nearly 1.5 hours of physical time to complete the runs on an 8 GB RAM machine. Analysis of flow took 1.36 seconds computational time for reaching convergence. RMS value of coefficient of drag (C_D) obtained from the analysis is 0.63, and that of coefficient of lift (C_L) is 0.61. Shedding frequency has been calculated from the period of oscillation of the lift force. Lift force oscillates about zero mean value at the same frequency as that of vortex shedding. The frequency of oscillation of C_L , and hence that of shedding, has been found to be equal to be 35.7 Hz, which corresponds to a $St = 0.28$. A relationship between St and Re (Techet, 2005) it is observed that at $Re = 5.3 \times 10^4$ for a smooth cylinder, Strouhal number value is just above 0.3. Hence the obtained shedding frequency value from the numerical simulation has been proved to be in acceptable range. C_D and C_L time histories after convergence for a number of time periods are presented in Figure 3 and Figure 4.

Figure 3. Time history of the coefficient of drag (C_D) at $Re = 5.3 \times 10^4$

Figure 4. Time history of the coefficient of lift (C_L) at $Re = 5.3 \times 10^4$

2.2. Experiment - Flow Past Horizontal Cylinder

Experiments have been conducted with two folded objectives. The first one does a quantitative comparison, and the other one for a qualitative comparison. Quantitative comparisons with the numerical study have been established through a test at moderate $Re = 5.3 \times 10^4$ while the qualitative ones employed lower Re tests for flow visualizations, thus proving the efficacy of the mesh at moderate and low Re .

2.2.1. Quantitative Comparisons

A smoke test was carried out in the subsonic wind tunnel of Aerospace Laboratory of Karunya Institute of Technology and Sciences, Coimbatore, India. A horizontal cylinder of diameter 50 mm and length 600 mm fitted with pressure tapings has been used as the model for testing. The compressor of the wind tunnel unit was operated at 600 rpm, which corresponds to 16.4 m/s ($Re = 5.3 \times 10^4$) velocity at the test section. Smoke was inducted into the test section by burning liquid paraffin. Flow patterns and vortex shedding around the cylinder were captured using a high resolution camera. C_D and C_L values have been calculated from the measured pressure distribution. Details of the wind tunnel are as follows.

2.2.1.1. Wind tunnel specification

Test Section Size	:	Cross section = 600 x 600 mm
Length	:	4000 mm
Maximum Speed	:	45 m/s
Contraction Ratio	:	6:1
Contraction Length	:	1.8 m
Entry Section	:	Bell mouthed.
Smoke	:	Provided in the contraction cone
Power	:	22 kW/30 HP AC motor.

An inclined manometer with ethanol as the manometric fluid is fixed on the tunnel. The two limbs of the manometer are connected to the static pressure holes one in the settling chamber just before the contraction and the other to that at the entrance of the test section. The reading on the manometer is very near to the dynamic head of the fluid in the test section, and it serves as a reference for keeping the tunnel speed constant. The tunnel is also provided with a pitot static tube

which can be traversed across the tunnel cross section.

Pressure readings have been observed from the pressure ports provided on the circumference of the cylinder. Static and stagnation pressure have also been observed. Plots of pressure distributions around the cylinder and coefficient of pressure are represented in graphs Figure 5(a) and 5(b). C_D has been calculated by integrating $C_p \cos \theta$ over 360° using Simpson's method of integration. The value of C_D obtained from the wind tunnel experiment is 0.62. A comparison of the results obtained from the simulations and experiment are given in Table 2.

a)

b)

Figure 5. (a) Pressure distribution around the cylinder at $Re = 5.3 \times 10^4$ (b) Coefficient of pressure around the cylinder at $Re = 5.3 \times 10^4$

2.2.2. Qualitative Comparisons

An experiment to visualize, study, and analyze the characteristics of wake behind the horizontal cylindrical model has been conducted through smoke injection tests in the wind tunnel. A 50 mm diameter cylinder with 600 mm length has been used for flow visualization. The model is exactly similar to the one used for pressure measurement but without pressure ports. The model which had been used for pressure measurement cannot be used in this scenario because of the risk of the pressure ports getting

clogged with smoke particles. The recommended maximum tunnel speed for smoke visualization experiments is 4 m/s. Smoke tests have been performed at 0.6 m/s which corresponds to $Re = 2000$. Numerical simulations also have been performed using ANSYS Fluent 15 for the same Reynolds number using the previously generated mesh. Results obtained from the experiment and simulations are compared in Table 2. St has been calculated from the frequency of vortex shedding using Equation 12. In numerical simulations the shedding frequency is taken to be equal to the oscillation frequency of lift coefficient. Shedding frequency is obtained as 0.99 Hz and Strouhal number as 0.1995 for $Re = 2000$. Vortex shedding frequency from the wind tunnel experiment has been calculated by repeatedly noting from the recorded video of shedding phenomenon, the time is taken for shedding 20 vortices from the upper boundary of the cylinder, and then taking the average time observed for the calculation. Results obtained are presented in Table 2 for comparison with those obtained from numerical simulations.

Figures 6(a) and 6(b) represent the flow pattern obtained from the numerical study after convergence at 1.6 seconds of flow and wind tunnel test respectively for flow past cylinder at $Re = 2000$. Distances to the lower pressure zones (vortices) shed from the cylinder are indicated on a scale of the diameter of the cylinder. The length of the line segment in both figures corresponds to the diameter of the cylinder. The angle of separation of the boundary layer is shown in the wake patterns given in Figures 7(a) and 7(b). The numerical value of the separation angle is given in Table 2. For verification of the results from the present grid, flow analysis has been carried out at $Re = 1000$, in order to validate the grid using other published numerical works. The RMS value of C_D is found to be 1.24, which is very much comparable with the published results at $Re=1000$ ($C_D= 1.15$) (Braza. *et al.*, 1986).

2.3. Force - Frequency Plot and Structural Analysis

Characteristics of flow past cylinder is a topic extensively investigated by researchers. Many have proposed plots from experimental, analytical, and numerical simulation results, which show the relationship between various flow parameters such as. with Re . The present work focuses on a marine riser model of specified dimension. A methodology has been developed to predict the structural response especially the trajectory of any section of the riser under vortex induced vibration. As the first phase, a data sheet has been created using the unstructured 2D mesh

given in Figure 2. The sheet gives the numerical results of C_D , C_L , and vortex shedding frequency (f_v) from the 2D simulations performed in ANSYS Fluent-15 for a range of Reynolds number up to the sub critical value and the values have been mapped over the entire length of a 3D cylinder

Simulations have been performed for $Re = 100$ to $20,000$. The velocity of flow has been assumed to be uniform along the span of the cylinder. Using the values of C_D and C_L obtained from the flow analysis lift and drag force acting on a cylindrical riser of 8 m length, and 0.05 m diameter has been estimated. Figures 8(a) and 8(b) show the plot for drag and lift forces on a cylindrical riser model of 0.05 diameter and 8 m length against its vortex shedding frequency.

From the plot, the hydrodynamic forces acting on the cylinder at various frequencies can be obtained. The value of hydrodynamic loads obtained from the plot at a specific frequency has been used as the input for predicting the structural response of the cylinder and the trajectory of any point on the surface of the cylinder.

Figure 8. (a) Drag Force – Frequency plot
(b) Lift Force – Frequency plot

Structural analysis of the cylinder model has been performed in ANSYS Workbench-15. Drag and lift forces acting on the cylinder as a

result of vortex shedding have been given as input in the solver. It has been treated as an oscillating force having an amplitude equal to maximum lift in the direction perpendicular to the flow (cross flow) and equal to maximum drag in the direction of flow (in line). The frequency of oscillation of lift and drag forces has also been obtained from flow analysis. It has been observed that the drag coefficient is oscillating about a non-zero value at double the frequency of the lift coefficient. Vortices are shed behind the cylinder at the same frequency as that of the oscillating lift force. When the vortex shedding frequency matches the natural frequency of oscillation of the cylindrical structure, a resonance condition, well known as lock – in vibration occurs, which gives rise to oscillations of very high amplitude. This may bring in extensive damage to the structure and to the working crew. Hence the analysis of the response of structures vibrating at their natural frequencies due to vortex shedding is considered to be most desirable in design of offshore structures.

The cylinder has been modeled in ANSYS Workbench-15 as a hollow vertical, having a thickness of 2.5×10^{-3} mm. The material of the riser model has been chosen to be Poly Vinyl Chloride (PVC) of density 1400 kg/m^3 . The simulation has been performed as a transient case since it involves time varying forces and deformation. The top end has been modeled fixed in the x and y directions, and the bottom end with motion in x, y, and z direction arrested. Since the riser is a flexible structure, Mechanical ANSYS Parametric Design Language (APDL) has been used as the solver. Stiffness and mass coefficients were provided as input in the damping control of analysis settings of the solver. After performing a grid independency analysis, a mesh having 66,158 elements have been chosen for the response analysis. The meshing of the geometry has been done using tetrahedral elements. The time step size has been chosen to be 5×10^{-2} based on the oscillating frequency of hydrodynamic loads. Transient simulation has been performed for 10 seconds. Local displacements at estimated locations of maximum vibration amplitude at the first three modes of vibration were obtained by inserting a displacement probe at the respective locations on the cylinder. Time histories of displacement in the X and Y directions and the trajectory of the cylinder during vibration have been plotted from the probe data.

The first three modes of vibration and the corresponding natural frequencies of the riser in the air have been obtained as 2.99, 9.7, and 20.23

Hz, respectively, from the modal analysis. Natural frequencies of the riser in water have been obtained solving Equation 14, considering added mass to be 70% of the total system mass, $m_a = 1.12$ kg, and stiffness of the riser $k = 17.73$ N/m. The first three natural frequencies of the riser in water have been obtained as 2.88 Hz, 7.16 Hz, and 9.41 Hz, respectively.

$$\frac{1}{\omega_{n \text{ water}}^2} = \frac{1}{\omega_{n \text{ air}}^2} + \frac{m_a}{k} \quad (\text{Eq. 14})$$

Drag and lift forces, corresponding to the identified natural frequencies of the riser, have been obtained from the plotted frequency – force graph. The amplitude of variation of drag force has been observed to be much less compared to the lift force. Lift force has been considered as periodically varying force, which oscillates about zero and acting in the cross flow direction as given in Equation.15 in the structural solver.

$$F(t) = F_0 \cos(\omega_v t) \quad (\text{Eq. 15})$$

Drag force oscillates with double the frequency of lift force, and it oscillates about a non-zero value. Drag force has two components, viz. average drag and the fluctuating component of drag force. From the numerical simulation, both RMS value and fluctuation about RMS value of drag forces have been obtained. Drag force is applied to the cylinder in the structural solver, as given by Equation 16.

$$F_D(t) = F_{D \text{ avg}} + F'_D \cos(2 \omega_v t) \quad (\text{Eq. 16})$$

Where $F_{D \text{ avg}}$ is the RMS value of drag force, and F'_D is its fluctuating component. The lift and drag forces obtained from the flow analysis corresponding to the flow regimes at which vortex shedding frequency (ω_v) matches the first three harmonics of the natural frequency of the cylinder (ω_n) have been used for structural analysis. This condition is generally referred to as lock-in condition. The drag force oscillates at double the frequency of lift coefficient. This phenomenon is well established in published literature (Durbin, 2007).

For the identified natural frequencies, structural analysis has been carried out to study various response parameters. A deformation probe has been inserted at $z = 4$ m for first and third modes of vibrations and at $z = 2$ m for the second mode to observe the response of the

cylinder under oscillating load.

3. RESULTS AND DISCUSSION

It can be observed that the numerical simulation with the grid generated, as shown in Figure 2 in the present work, is capable of replicating the wake pattern behind the cylinder exactly in the way the experiments do. Wake dimensions are comparable in both cases in terms of the span and angle of separation. Quantitative comparisons between the experimental and numerical values presented in Table 2 indicate the reliability in the present numerical predictions. Here the important observation is that at lower $Re = 2000$, a better prediction of St is achieved (9% deviation from the experimental ones) while at higher $Re = 4000$, a deviation of 22% is observed from the corresponding experimental value.

From the history of cylinder displacement during the lock-in, it can be observed that, as the frequency of shedding increases, the amplitude of displacement in cross flow direction goes on decreasing and that in the inline direction increases. This is because, at higher Re corresponding to higher shedding frequencies, the value of C_D is also high. A huge number of researchers are focusing on the cross flow response of cylinder. The observation made here emphasizes the need for investigating the inline response during lock-in at higher harmonics of natural frequency.

By plotting the non-dimensional amplitude of the cylinder in IL and CF directions, the trajectory of the probe location has been traced, as shown in Figures 9 (a) – (c) for all three frequency regimes. It can be observed that the cylinder point follows an eight figure trajectory during lock - in. The cylinder is expected to follow an eight figure trajectory due to oscillating lift and drag forces induced by vortex shedding at its wake. In lock-in condition the frequency of the inline vibration is twice that of cross flow vibration, and the trajectory of the cylinder corresponds to “Lissajou figure 8” (Vandiver *et al.*, 2009). Trajectory obtained from the numerical study is very much comparable with the response of an 8 m riser model obtained from the experimental study (Liangjie *et al.*, 2004). The trajectory from the experiment is given in Figure 10.

The trajectories traced show the increasing importance of accounting for IL response at higher harmonics of natural frequency. The maximum amplitude in the CF direction is obtained to be equal to $2.5D$, where D is the diameter of the cylinder when the cylinder locks on to the first

natural frequency. During the first natural frequency lock-in vibration, the cylinder traces the same path repeatedly for several cycles of oscillation. But moving to higher harmonics, there is considerable uncertainty in the case of exact path traced. The magnitude of the non-dimensional amplitude of oscillation and the pattern of trajectory matches reasonably well with the published experimental and numerical results.

a)

b)

c)

Figure 9. Trajectory of the cylinder (a) shedding frequency equal to the first harmonic of natural frequency (b) shedding frequency equal to second harmonic of natural

frequency (c) shedding frequency equal to third harmonic of natural frequency

Figure 10. Trajectory obtained from an experimental study of the response of cylindrical riser

4. CONCLUSIONS

This paper presents a method that is computationally economic at the same time efficient in predicting VIV of cylindrical risers. Two dimensional flow analysis is capable of predicting the numerical values of hydrodynamics loads, pressure patterns during vortex shedding, the amplitude of oscillation and the trajectory as well for VIV problems. Vortex induced vibration of slender structures especially petroleum risers and mooring cables are one of the important aspects that should never be neglected in their design. Hence a thorough understanding of the loads and responses of such slender structures is essential before its design and deployment. Numerical analysis using two way fluid structure interaction (FSI), which can predict the parameters with considerable accuracy is always not handy for everyone dealing with this type of problem because of the heavy computational requirement needed for such solvers. This method has been proposed in a view to supporting young researchers in the field of the intrinsic flow phenomenon of vortex shedding, who lack high computational facility but may also understand the phenomenon without compromising much accuracy.

The method has been well validated in two stages, initially the grid, which yields the same flow physics as that of conventional experiments and then the structural behaviour which also is in good agreement with similarly published works (Liangjieet al.,2004 ,Vandiveret al.,2009). On the whole, it is found that the method is quite effective in the quick computation of VIV problems for low and moderate Re. At high Re, 3D studies find their

use as 2D predictions do not suffice in providing the minute details of flow physics.

5. ACKNOWLEDGEMENT

The authors of this paper wholeheartedly express their gratitude and acknowledgement to Mr. K. Alfred Sunny and Mr. Kumar of Karunya Institute of Science and Technology, Coimbatore, India, for the technical support rendered by them while conducting experiments on the sub-sonic wind tunnel. We also extend our deepest sense of gratefulness to Prof. M. Madhavan, former Principal, SCMS School of Engineering and Technology, Eranakulam, India, for his timely advice and support.

6. REFERENCES

1. Gustafsson, A. Analysis of Vortex-Induced Vibration of Risers, Master's thesis in Applied Mechanics, Chalmers University of Technology, Sweden, **2012**.
2. Blevins, R.D. Flow-Induced Vibration, second ed., Van Nostrand Reinhold, New York, **1990**.
3. Bourguet, R., Karniadakis, G.E., Michael, S., and Triantafyllou, M.S. Lock-in of the vortex-induced vibrations of a long tensioned beam in shear flow, *Journal of Fluids and Structures*, **2011**, 27, 838–847.
4. Braza, M., Chassaing, P., and Minh, H.H. The Numerical Study and Physical Analysis of the Pressure and Velocity Fields in the Near Wake of a Circular Cylinder, *Journal of Fluid Mechanics*, **1986**, 165, 79–130.
5. Chandran, V., Sekar, M., Janardhanan, S., and Menon, V.A. numerical study on the influence of mass and stiffness ratios on the vortex induced motion of an elastically mounted cylinder for harnessing the power. *Energies*, **2018**, 11, 2580.
6. Chandran, V., Sekar, M., Janardhanan, S., Influence of Frequency Ratio on Hydroelastic Response of a Cylinder with Degrees of Freedom under Vortex Induced Vibration., *International Journal of Innovative Technology and Exploring Engineering*, **2019**, 8, 9S3, 307 -312.
7. Daniels, S.J., Castro, I.P., and Xie, Z. Numerical analysis of free stream turbulence effects on the vortex-induced vibrations of a rectangular cylinder, *Journal of Wind Engineering and Industrial Aerodynamics*, **2016**, 153, 13–25.
8. Domala, V., and Sharma, R. An experimental study on vortex-induced vibration response of marine riser with and without semi-submersible, *Journal of Engineering for the Maritime Environment*, **2017**.
9. Durbin, P.A., and Medic, G. Fluid dynamics with a computational perspective, **2007**, Cambridge University Press, Newyork.
10. Feng, C.C., The measurement of vortex-induced effects in flow past stationary and oscillating circular and D-section cylinders, Master's thesis, **1968**, University of British Columbia.
11. Gao, Y., Fu, S., Xiong, Y., Zhao, Y., and Liu, L. Experimental study on response performance of vortex-induced vibration on a flexible cylinder, *Ships, and Offshore Structures*, **2017**, 12, 116–134.
12. Kamkar S.J. Mesh Adaptation Strategies for vortex dominated flows, Ph. D Thesis, **2011**, Stanford University.
13. Keshavarzian, B., and Khosravi, M. Numerical Investigation of the Structural Frequencies Effects on Flow Induced Vibration and Heat Transfer, *Journal of Material and Environmental Science*, **2015**, 6 (7), 1949–1956.
14. Khalak, A., and Williamson, C.H.K. Motions, forces, and mode transitions in vortex-induced vibrations at low mass-damping, *Journal of Fluids and Structures*, **1999**, 13, 813–851.
15. Liangjie, M., Qingyou, L., and Shouwei, Z. Experimental Study of the Vortex-Induced Vibration of Drilling Risers under the Shear Flow with the Same Shear Parameter at the Different Reynolds Numbers, *PLoS ONE*, **2004**, 9, e104806.
16. Pezzini, A., Brião, V.B., and DE BONI, L.A. B. Preliminary study for a cleaning and water reuse system, *PERÍODICO TCHÊ QUÍMICA*, **2017**, 28, 127–132.
17. Ramírez, Ann-Katrin, P., Wolff, and Carl, F. Building quality infrastructure services for water and sanitation providers in latinamerica – examples from german development cooperation, *PERÍODICO TCHÊ QUÍMICA*, **2017**, 28, 18–28.
18. Sarpkaya, T., Transverse Oscillations of

Circular Cylinder in Uniform Flow, Part I
Report No. NPS-69L 77071-R, **1997**, Naval
Postgraduate School, Monterey, Calif.

19. Sekar, M., Srinivas, J., Kotaiah, K.R., and Yang, S.H. Stability analysis of turning process with the tailstock-supported workpiece, *Int J Adv. Manuf. Technol*, **2009**, 43, 9–10, 862–871.
20. Shiels, D., Leonard, A., and Roshko, A. Flow-Induced Vibration of a Circular Cylinder at Limiting Structural Parameters, *Journal Of Fluids And Structures*, **2001**, 15, 3–21.
21. Techet, A., 13.42 Lecture: Vortex induced vibration – MIT open courseware, **2005**, Massachusetts Institute of Technology, USA.
22. Trim, A.D., Braaten, H., Lie, H., and Tognarelli, M.A. Experimental investigation of vortex-Induced vibration of long marine risers. *Journal of Fluids and Structures*, **2005**, 21, 335–361.
23. Vandiver, J.K., Jaiswal, V., and Jhingran, V. Insights on vortex-induced, traveling waves on long risers. *Journal of Fluids and Structures*, **2009**, 25: 641–653
24. Wang, Q., Li, M., and Xu, S., Experimental study on vortex induced vibration (VIV) of a wide-D-section cylinder in a cross flow, *Theoretical and Applied Mechanics Letters*, **2015**, 5: 39–44.
25. Xie, F., Deng, J., Xiao, Q., and Zheng, Y.A. Numerical simulation of VIV on a flexible circular cylinder, *Fluid Dynamics Research*, **2012**, 44, 21.

Table 2. Comparison between wind tunnel experiments and numerical simulations.

Reynolds Number (Re)	Coefficient of Drag (C_D)		Strouhal Number (St) (Non-dimensional Frequency)		Wake Dimension	
	Experiment	Numerical	Experiment	Numerical	Experiment	Numerical
53000	0.62	0.63	NA		NA	
4000	-	-	0.178	0.22	$\theta = 126^\circ$	$\theta = 120^\circ$
2000	-	-	0.183	0.1995	D = 0.05m	D = 0.02m

a)

b)

Figure 6. Pressure pattern at the wake of the cylinder at $Re = 2000$ (a) obtained from numerical simulations (b) obtained from the smoke test conducted in the subsonic wind tunnel

a)

b)

Figure 7. Boundary layer separation at the wake of the cylinder at $Re = 4000$. (a) obtained from numerical simulations (b) obtained from the smoke test conducted in the subsonic wind tunnel