ANALYSIS OF VORTEX INDUCED VIBRATION USING IFS

Prateek Chaturvedi[1], Ruchira Srivastava [1], Sachin Agrawal [3], and Karan Puri[4]

[1]Department of MAE, Amity University, Greater Noida, India
[3]Department of ME, Skyline Institute of Engineering and Technology, Gr. Noida, India
[4]Ex-student, Department of ME, DIT School of Engineering, Greater Noida, India

ABSTRACT

Interaction of fluid structure (IFS) is one of the upcoming field in calculation and simulation of multi-physics problems. IFS play an important role in calculating offshore structures deformations caused by the vortex induced loads. The complexity interaction nature of fluid around the solid geometries pose the difficulties in the analysis, but IFS analysis technique overshadow the challenges. In this paper, Analysis is done by considering a cylindrical member which is similar to the part of offshore platform. The IFS analysis is done by using the commercial package ANSYS 14.0. The Vortex induced loads simulation with IFS is purely a mesh dependent, for that we have to simulate many problems for getting optimum grid size. Computational Fluid Dynamics (CFD) analysis of a two dimensional model have been done and the obtained results were validated with the literature findings. CFD analysis is performed on the extruded version of the two dimensional mesh and the results were compared with the previously obtained two dimensional results. Preliminary IFS analysis is done by coupling the structural and fluid solvers together at smaller time steps and the dynamic response of the structural member to the periodically varying Vortex induced vibrations (VIV) loads were observed and studied.

KEYWORDS

Interaction of Fluid Structure (IFS), Computational Fluid Dynamics (CFD), Vortex Induced Vibrations (VIV) Introduction (Heading 1)

1. INTRODUCTION

Interaction of Fluid structure (IFS) is a highly versatile engineering field, it happens where fluid flow whether laminar or turbulent deform the structure which further cause the variation in the boundary layer of fluid system. It play a significant role in designing and analyzing system. For example, IFS simulations are applied to stop swing effects on aircraft and turbo-machines, and to determine the environmental loads and dynamic response of offshore structures. The simulations of multi-engineering problems have become more important aspect for the past ten years in the field of numerical simulations and analysis. Sudharsan, et al. [1] conducted a IFS simulation to study the response of an offshore structure in a numerical flow tank. Various surrounding and induced loads cause deformation to offshore structures.

Sumer and Fredsøe [2] briefly explained an experiment done by Drescher [3] by estimating drag and lift forces from the measured pressure distribution around the body. The flow pattern of fluid past a cylindrical element of high sub-critical Reynold number of the range 200 - 105 was studied by Wan and Raghavan [4]by numerical simulation technique. It becomes convenient for the designer if he knows about the fluid flow pattern around the element. In this work, one-way IFS analysis is performed on a vertical cylindrical member fixed at one end and the other end is free to vibrate to the loads due to vortex induced vibrations. Consider the fluid flow with Reynolds

DOI: 10.5121/ijci.2017.6205

number of 1200 is flowing through the element. The flow around the cylinder is considered to be laminar and is unsteady. The CFD simulation is done and the corresponding results are exported to the structural model. The dynamic response of the structural element to the fluid flow and vortex-induced vibrations are analysed.

2. PROBLEM FORMULATION

The governing equations for all kinds of fluid flow and transport phenomena are derived from the basic conservation principles such as conservation of mass, momentum and energy. All these conservation principles are solved according to the fluid model which gives set of governing equations of the fluid. The continuity equation can be written as follows:

$$\partial \rho/(\partial t)\ \ \mathrm{div}\ (\rho u)=0 \tag{1}$$

Navier Stokes equations derived from Newton's second law for a Newtonian fluid includes the body and surface forces acting on the body. These equations are given as follows:

$$\partial \rho u/\partial t+\mathrm{div}\ (\rho uu)=-\partial p/\partial x\ \mathrm{div}\ (\mu\ \mathrm{grad}\ u)+Smx \tag{2}$$

$$\partial \rho v/\partial t + \mathrm{div}\ (\rho v\square) = - \partial p/\partial y + \mathrm{div}\ (\mu\ grad\ v) + Smy \tag{3}$$

These equations describes the flow around a smooth circular cylinder, using a non- dimensional hydrodynamic number called Reynolds number (Re). The Reynolds number by definition and is formulated as

$$\mathrm{Re} = DU\ v \tag{4}$$

Where, D is the diameter of the cylinder, U is the flow velocity and v is the kinematic viscosity of the fluid. Flow regimes are obtained as the result of enormous changes of the Reynolds number. The vortex flow around the cylindrical element caused due to the changes of the Reynolds number in wake region of the cylinder, which are called vortices. At low values of Re (Re < 5), no separation of the flow occurs but when the Re is further increased the separation starts to occur and these vortices becomes unstable and initiates the phenomenon called vortex shedding phenomenon at certain frequency. The vortex shedding process is defined by the Reynolds number and the shedding frequency by the Strouhal number, St

$$\mathrm{St} = fs\mathrm{D}\ U \tag{5}$$

Where, fs is the shedding frequency, D the diameter of the cylinder and U the free stream velocity. As the result of the periodic change of the vortex shedding, a pressure difference is also created periodically and due to these forces acts on the cylinder in the in-line or cross-flow direction.

3. CFD METHODOLOGY

Computational Fluid Dynamics techniques used to study the behaviour of the fluid across the structural element by conducting the numerical simulation. In the pre-processing stage, the fluid flow problem is defined by giving the input in order to get the best solution of the problem. This stage is influenced by the following factors:

3.1. Solution Domain

The solution domain defines the abstract environment where the solution is calculated. The 2D domain has upstream length of 5D and the downstream side of length 27.5D. The 3D domain is the extruded version of the 2D domain for a length of 10 meters. The dimensions of the domain are represented in meters.

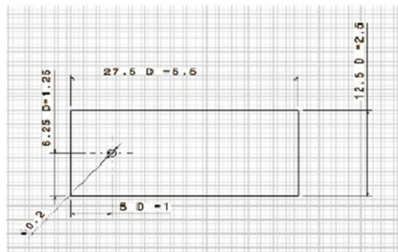


Fig.1 Drafted view of the 2D domain

3.2. Mesh Generation

A 2D block-structured mesh is generated using GAMBIT 2.4.6 for the domain. A grid independency study is performed determine the grid size precisely to produce an accurate result. The domain face is divided into 10 divisions consisting of 19150 elements shown in Fig.2. The grid is adapted in FLUENT at the specific region where the dominance of vortex shedding is quite high than other surrounding face.

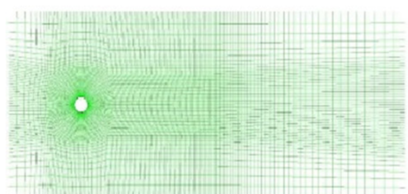


Fig.2 Structured mesh of the 2D domain

3.3. Fluid Properties

A Pressure-based solver is taken as the solver scheme for the model. The fluid flow is considered to be unsteady (transient). Since the case of Re=1000 is taken, the fluid flow is solved as a laminar flow. The fluid flowing with Re= 1000 undergoes a transition from laminar to turbulent flow in this region, but the turbulence is small compared to the laminar flow. So, the turbulence effect is neglected. The fluid properties are considered as follows

Density - 998.3 kg/m3
Viscosity - 0.1 kg/m-s

3.4. Boundary Conditions

The inlet boundary condition of the domain has a velocity inlet defined at the rate of 0.5 m/s. For the outlet boundary condition, outflow condition is considered. The Solution algorithm chosen for pressure- velocity coupling equation is the SIMPLE scheme for solving the governing equations with the specified boundary conditions. QUICK method is taken as the momentum spatial discretization.

The under-relaxation factors are included to the pressure based solver to stabilize the iterative process. If the values are high, the model becomes unstable and may fail to converge and if the values are low, they might take up large number of iterations to converge to a value. Therefore, the default values are used before initiating the calculation. The time step size for the simulation is taken to be 0.2 sec and the solver is initialized. The solution runs for time step count of 600 and the corresponding CD and CL values are written to a file for plotting the FFT.

4. FSI METHODOLGY

The FSI (one way or two coupling) analysis is done in ANSYS Workbench by coupling fluid and structural system together and this process is known as system coupling. The fluid or structural solvers either receives or transmits data in the coupling analysis. In one way coupling, the solver initiates the first time step. FLUENT runs the initial set of simulations and the setup iterates till the convergence is reached. The data (fluid forces) is now transferred to ANSYS Mechanical, so that this solver begins the iterative process for the convergence to be received at the same time step. The simulations of various cases done in the previous section are related in bringing up reasonable coupling procedure by implementing the various appropriate schemes and techniques. Before initiating the coupling simulation, it is necessary to analyze each fields separately to obtain stable solution. The solution is influenced by the following factors:

4.1. Geometry and mesh

The geometry of the model is similar as mentioned in the previous section. The meshing technique i.e. hexahedral mesh done in the CFD simulation is not taken in the FSI analysis as it may end up in more computational time due to the grid size. So, Tetrahedrons meshing is done on the fluid domain of the fluid solver. The number of elements in the fluid domain is 756452.

4.2. Material properties

The properties of the fluid is taken similar to the previous section cases with density of 998 kg/m3 and viscosity of 0.1 kg/m-s. The material considered for the short riser is considered to be structural steel having a density of 7850 kg/m3. The yield strength of the material is 250 MPa. After performing the modal analysis of the model, sets of eigen value frequencies are obtained. The first eigen value frequency of the modal is 3.4537 Hz and which corresponds to a reduced velocity of Vr = 0.69. The total mass of the riser is 986.46 kg.

4.3. Boundary conditions

The boundary conditions used in this simulation is similar to the cases previously discussed. The fluid domain is solved with dynamic mesh due to free end of the riser at the top section. The deformation of the mesh will be however minimal and unnoticeable in this case. The solution scheme is taken as coupled for this simulation and time step size as 0.05 having 400 time steps. For the structural model, the bottom end is constrained to all DOF and the top end is constrained in y direction.

5. RESULTS AND DISCUSSION

5.1. Results for 2D domain

The solution of the model is generated based on the solving techniques utilized during the CFD simulation. At the Re=1000 which is predominately considered to be a laminar case, the lift and drag forces are computed acting upon the cylindrical structure. Fig.4 shows the contour view of the vortices are constantly shedding during the flow time of 60 sec.

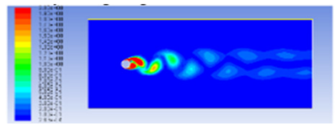
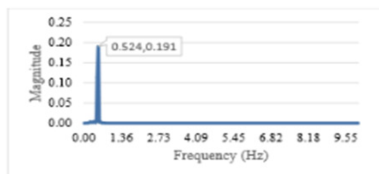


Fig.4 Vorticity Magnitude contour view (time t = 60s)

An FFT is generated to obtain the shedding frequency of the simulation and this plot requires the convergence history of the lift forces acting in the cross-flow directions. Fig. 5 shows the FFT plot of the spectral analysis of lift convergence and the obtained shedding frequency is 0.524 Hz



. Fig.5 Spectral Analysis of Lift Convergence (2D model)

The Strouhal number is obtained by substituting the obtained frequency in (5), and is found to be 0.2096. The obtained Strouhal number shows good agreement with the data obtained as in [5]. Fig. 6 shows the convergence plot of the forces acting on the cylinder in the cross-flow directions. Vortex shedding occurs at a time step size of two seconds which shows good agreement with the analytical value of shedding frequency.

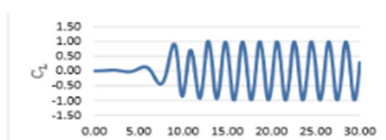


Fig.6 CL Convergence history (2D model)

Fig.7.shows the variation of forces acting in the inflow direction of the fluid flow.

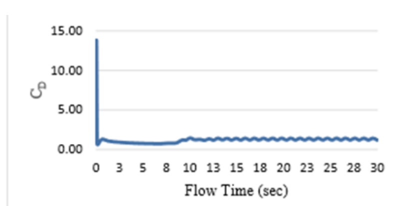


Fig.7 CD Convergence history (2D model)

Table 2 shows the validation of results of the present study with the previous studies having Re = 1000. The results shows good agreement with the literature studies for the computed data and slight discrepancies with the experimental data.

Table II Validation of Result

S. no.	Source of result	CD
1	Present Study	1.21
2	Anderson [5]	0.9
3	Yunus & John [6]	1.19

5.2. Results of 3D domain

The simulation of the riser is computed and evaluated with the convergence of lift and drag coefficients histories at corresponding flow time. The vortex shedding starts at 8 sec and attains a constant amplitude at about 17 sec. Fig. 8 shows the that the vortices formed behind the cylinder are consistent along the entire length of the cylinder which shows that shedding frequency would be the same along the entire length of the cylinder.

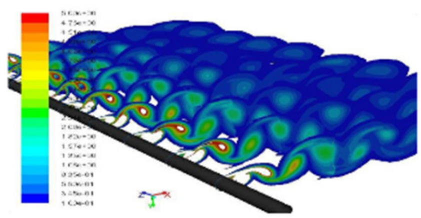


Fig.8 Contours of vorticity magnitude formed by rakes (3D model) time t = 18.75s

Figure.9 shows the convergence plot for the drag forces acting on the 3D riser and these forces attain a constant amplitude after certain period of time.

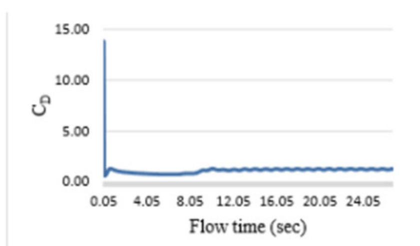


Fig.9 CD Convergence history (3D model)

The values of the vortex shedding frequencies and the Strouhal number remains the same since the vortex shedding period is the same for both the cases. The coefficient of drag forces acting on the cylinder is 1.29 and when compared with the coefficient of drag forces of the 2D model having the same geometry.

5.3. Results of FSI on a riser model

The comparison between the existing models cannot be done because the riser has a smaller L/D ratio and low stiffness value when compared to the real-time risers. The importance of this study is that the results obtained remains safe and goes well versed with the theory. Fig. 10 shows the formation of vortices at certain magnitudes behind the cylinder in the form of path lines.

Fig.10 Path lines colored by vorticity magnitude (IFS model)

Fig. 11, the stress developed due to the forces has a maximum value of 2.2 MPa which is well below the yield strength of the material. Thus, the 3D riser design is evaluated to be safe.

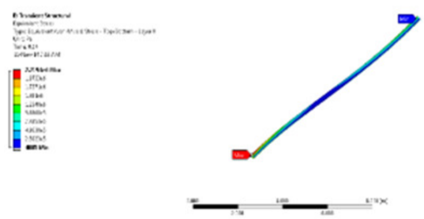


Fig.11 Equivalent (Von-Mises) stress on the structure

The deformation of the riser is defined by the VIV induced loads acting along the x and z coordinates. The forces acting along the z- direction are the lift forces from the fluid flow. Fig.12 shows the deformation due to the lift forces acting on the structure and it is seen that the maximum amplitude acts at the time of 19.14 sec of the flow and has a value of 0.89x10-6m.

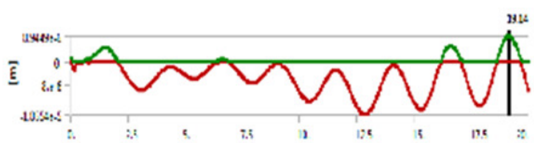


Fig.12 Directional Deformation along z-axis

The forces acting along the x- direction are the drag forces which act in the flow direction. Fig.13 shows the deformation due to the drag forces having a maximum value of 0.00237 m of the riser at the corresponding time step.

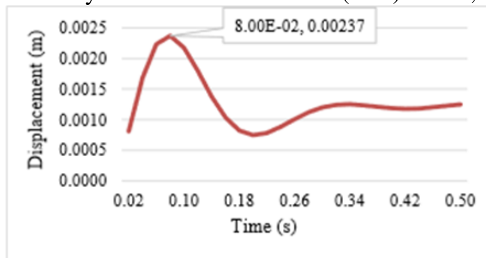


Fig.13 Directional Deformation along x-axis

6. CONCLUSIONS

The CFD simulation of a 2D cylindrical structure was performed with a flow having a Re of 1200. Grid Independency study was performed for a 2d model having structured mesh by adapting the region around the cylinder. So, the block structured mesh was preferred for the future simulations having the similar geometry. The flow characteristics and patterns were recorded for the laminar flow around the cylinder. The Strouhal number, vortex-shedding frequencies, coefficients of lift and drag forces were obtained from the simulation. The mean CD values were validated with the previous studies and they proved a good agreement with the literature studies. A CFD simulation was done on a 3d riser and the results were validated with the 2d model which showed no significant change in the Vortex shedding frequency and the Strouhal number. Finally, Preliminary IFS simulation is done on a 3d riser using hexahedral mesh in order to reduce the computational time. The amplitude of vibration due to the lift and drag forces were studied and the effect of the structural response was observed. The results show that the design of the model is safe and has a high value of factor of safety. Two-way IFS analysis of the model will be performed and the dynamic response would be studied.

REFERENCES

[1] N. M. Sudharsan, M. Kantharaj and K. Kumar, "Preliminary investigations on non-linear fluid-structure interaction of an offshore structure", Journal of Mechanical Engineering Science, Vol. 217, No. 7, 2003, pp. 759-76

[2] Sumer B.M. and Fredsøe J., "Hydrodynamics around Cylindrical Structure", World Scientific Publishing Co Pte. Ltd. Singapore, 2006

[3] Wan, S.I. and V.R. Raghavan, "Numerical Simulation of High Sub-critical Reynolds Number Flow Past a Circular Cylinder", International Conference on Boundary and Interior Layers, unpublished, 2006

[4] Yunus A.C., John M.C., "Fluid Mechanics: Fundamentals and Applications", 1st Ed., Published by McGraw-Hill, ISBN 0–07–247236–7, 2006

[5] Henry, H.B., "Pressure vessel design handbook", 2nd Ed., Van Nostrand Inhold Company, ISBN-0-89464-503-X, 1991

AUTHOR

Prateek Chaturvedi is research focus is in the field of Automobiles and Refrigeration. In the beginning, I worked in different fields like Tribology and Productivity of Machine tools. Further, I worked in the field of Fluid Mechanics and Technical Innovations in MSME sector as well. Later, I worked in the field of Automobile, mechanical and electrical areas. I filed three patents as well regarding the same. I believe to work in research field in such a way that its result and findings may invent something new for the mankind technically. Two out of my three patents are in the process of commercialization. Currently, I am working on the same topics. I am a member in the Editorial Board of International Journal of Multidimensional Research. In my future research, I will be working in the field of refrigeration. Recently, my Ph.D. got enrolled. There I will be working in the field of Composite Materials. I want to explore myself towards automation using Mechatronics as well.