International Journal of Civil Engineering and Technology (IJCIET)
Volume 9, Issue 4, April 2018, pp. 275–283, Article ID: IJCIET_09_04_030
Available online at http://www.iaeme.com/ijciet/issues.asp?JType=IJCIET&VType=9&IType=4
ISSN Print: 0976-6308 and ISSN Online: 0976-6316

PREDICT OF RISER'S RESPONSE TO EFFECTS OF FLOW MOTION

Mohammed J. Mawat
Civil Engineering Department/University of Basra, Basra, Iraq

Mudhar Hassan Gatea
Civil Engineering Department/University of Basra, Basra, Iraq

ABSTRACT

After installation phase, the riser will be exposed to an effects resulting from indetermination forces in design phase like drag force and lift force or from vortex induced vibration(VIV) because of flow movement around riser. Those effects might cause to failure, therefore a required a knowledge to study it to asses accurate fatigue life of structure else, designs might produce hazardous or expensive in some points because error elimination might not permanently take place. An accepted approach depending on computational fluid dynamic(CFD) theory is performed in this paper to simulate a three dimensional model of fluid domain with rigid riser at high Reynolds number and turbulence regime are solved by fluent ANSYS program. The CFD work on a riser's strip is done to predict the physics of the flow about rigid riser. An analysis of two models are adopted. First model was Standard Spalart-Allmaras (S-A) and the second was Large Eddy Simulation (LES).

Keywords: Computational Fluid Dynamic, Drag Force, Large Eddy Simulation, Lift Force, Riser Analysis, Spalart-Allmaras.

Cite this Article: Mohammed J. Mawat and Mudhar Hassan Gatea, Predict of Riser's Response to Effects of Flow Motion, International Journal of Civil Engineering and Technology, 9(4), 2018, pp. 275–283.
http://www.iaeme.com/IJCIET/issues.asp?JType=IJCIET&VType=9&IType=4

1. INTRODUCTION

Problem of fluid motion around cylindrical structures is one of most fundamental challenge in Computational Fluid Dynamic area. The riser, a vertical part of pipeline perpendicular to flow motion direction uses to extraction oil and/or gas from sea wells to surface platforms, figure 1, considers an important application for this kind of problems. The riser have length reach to hundreds of meters in some cases especially in deep water projects, where ratio of length to diameter (L/D) be very large[1, 2]. Therefore representation entire length of structure seems impossible since the number of elements in model will be huge and need more time for anatomy. There is promising way to simulate the long riser by taking a strip instead of entire length to build the model and credence short aspect ratio (L/D) to get limits of fluid domain of three dimensional model[3, 4]. In real world projects, deepwater risers can simply exceed aspect ratios of L/D = 1000. For example, Lucor, Mukundan and Triantafyllou (2006) consider a flexible cylinder of aspect ratio L/D = 2028[5]. They have improved a systematic methodology to extract the vortex induced vibrational modes of a riser, based on information from CFD to a long tensioned beam. Yiannis Constantinides and et al(2007)[2], several simulations of long scale model risers consisted of the L/D=1407 and L/D = 4137 are investigated. Computational Fluid Dynamics (CFD) tools is use for widespread in engineering field to provide studies on fluid-induced motions of structures[6]. The fatigue life of the riser is often conquered by Vortex-Induced Vibrations (VIV) as a result flow motion around it especially in water depths up to 3,000 m[7]. If the vibration of vortex is equal to one of the natural frequencies of riser the resonance phenomena accrue and fatigue life decreases rapidly. Therefore the analysis should be based on full 3-Dimensional CFD simulations. The CFD work on a riser's strip is done to predict the physics of the flow about rigid riser.

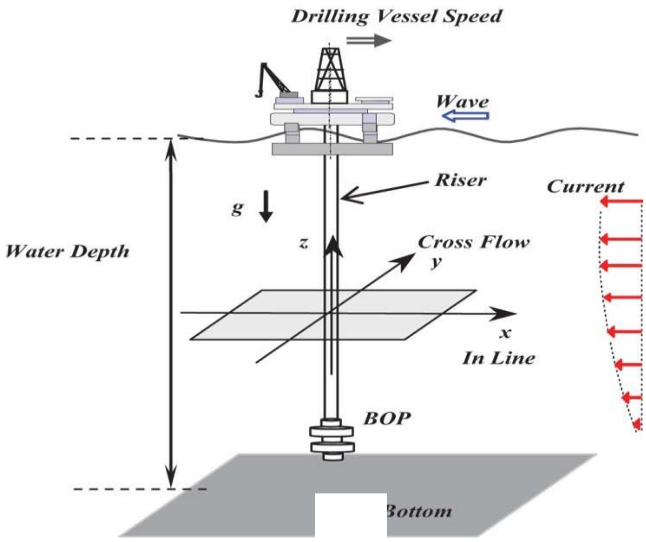
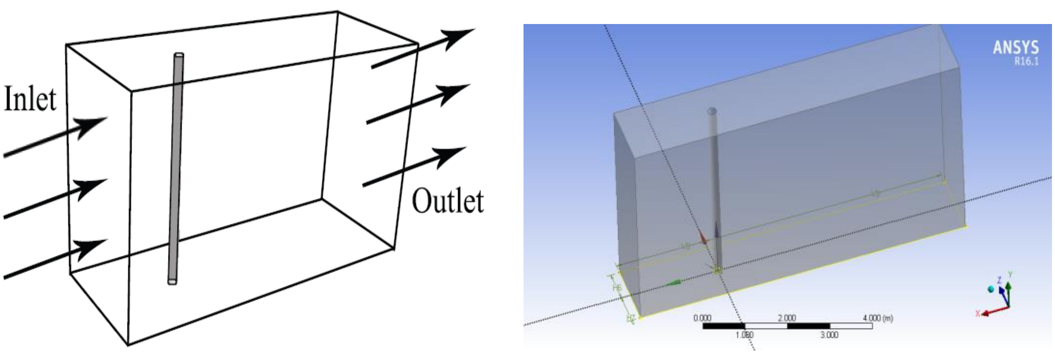


Figure 1 Sketch of Riser and environment about it

2. ANALYSIS MODEL

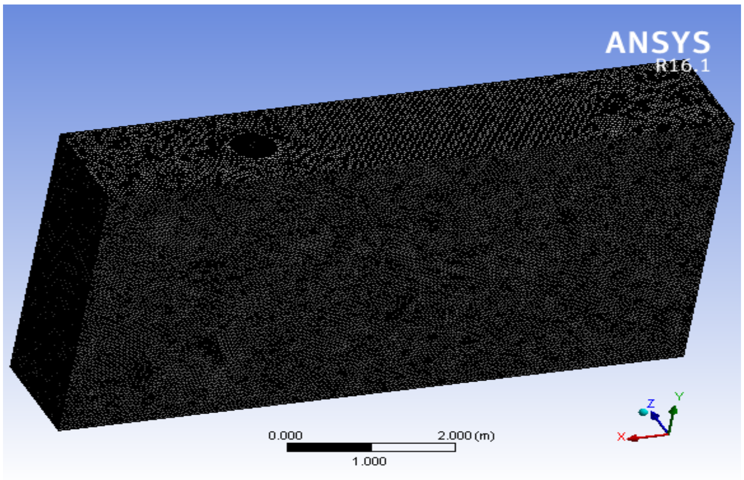
Depending on the diameter of the rigid riser section (D) a simple sketch of 3-Dimensional fluid domain is built for CFD strip. The riser centre in the model is located at 10D from inlet and 20D to the outlet boundaries with flow direction with equal distance to the lateral boundaries. The width of 7D in transverse direction is chosen for fluid domain. The length of riser section is 2.191m, since it is determined depending on aspect ratio (L/D) equals to 20. Figure 2 shows the Sketch of Geometry view for fluid domain Created in ANSYS FLUENT.



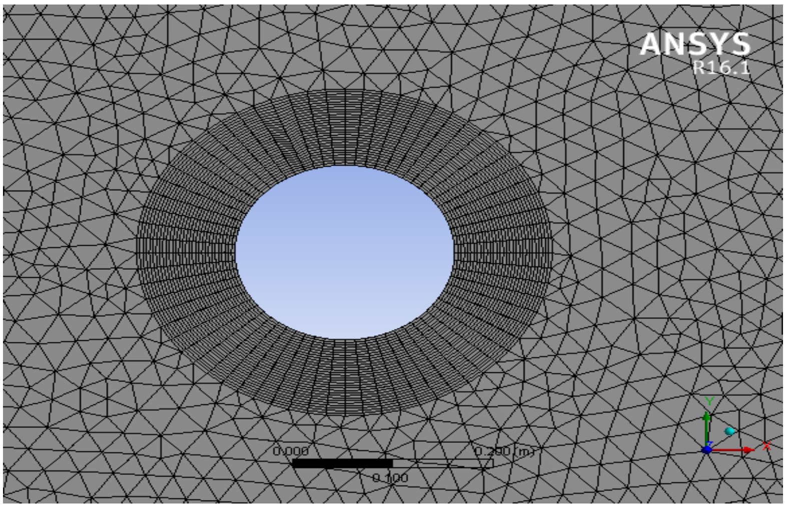
a b

Figure 2 a) sketch of 3D fluid domain. b) Ansys 3D fluid domain.

Since ANSYS Fluent is a state-of-the-art computer program for modeling fluid flow, heat transfer, and chemical reactions in complex geometries. Uses finite volume method, the number of cells used to mesh the environment directly affects the accuracy and length of time of the solution. The domain is divided into more than 2.5 million triangular elements with element's size equal to 0.03m, Fig. 3-a. an inflation(Refinement) have been made in the mesh steps by 30 layers around the riser with 0.01m of element size edge, Fig. 3-b, to get good performance of the model and more accurate data in this region[8].



a



b

Figure 3 Mesh sense of fluid domain. a) 3D mesh view. b) Top view of mesh.

The aim of this study is to simulate a riser section have length between 200-300m below water surface, so the density and viscosity of water are chosen to represent properties of flow at that level.

Physics parameters of flow material enabled in this simulation included:

- Flow material is water
- Flow material is incompressible liquid has constant density.
- Time model is Implicit unsteady.

This choice is used to observe the vortex shedding behind the riser, for this is a time-dependent problem (unsteady) and the governing equations used in this analysis are too complex to solve by explicit approach. In this method a certain convergence is achieved by using inner iterations. After each cycle of inner iteration, the time step going to update.

- Solver algorithm is simple
- Convection scheme is second order upwind
- Temporal discretization is second order.
- Viscous regime is turbulence with Re $=1.5*10^5$.

The parameters used for the main variables are listed in Table 1.

Boundary condition of model geometry shown in fig. 2 is illustrated in Table 2.

Table 1 Physics Values and Parameters for simulation.

Density	1025 kg/m^3
Dynamic Viscosity	0.0015 Pa*s
Diameter of Riser	0.2191m
Inlet Velocity	1m/s
Temporal Discretization	2nd-order
Turbulence model	Standard Spalart-Allmaras (S-A) for model 1 Large Eddy Simulation (LES) for model 2
Base Size of Mesh	0.01m
Time step	0.1s
Maximum Inner Iterations	20
Maximum Flow Time	20s
L/D	10

Table 2 Boundary Conditions for fluid domain

Inlet	Velocity inlet
Outlet	Pressure outlet
Upper and lower boundary	Symmetry plane
Lift and Right boundary	Symmetry plane
Riser	No slip wall

3. RESULTS AND DISCUSSION

Depending on approach of turbulence method, an analysis of two models can be adopted. First model is Standard Spalart-Allmaras (S-A) and the second is Large Eddy Simulation (LES).

3.1. S-A MODEL

When observing the frequency of lift coefficient as shown in fig. 4, one can see that the amplitude of (Cl) to some extend is unsteady and a considerable decline in the amplitude can be seen at t = 6s then it's to be steady and reaches ($-0.5*10^{-2}$) at end of flow time because of the convergence is achieved at first inner alteration between alteration N and N-1. The drag coefficient must be close to 0.82 corresponding to L/D = 10, [9], when it can be noted that the largest value for the drag coefficient is 0.65 and it is obtained at t = 1s as presented in fig. 5.

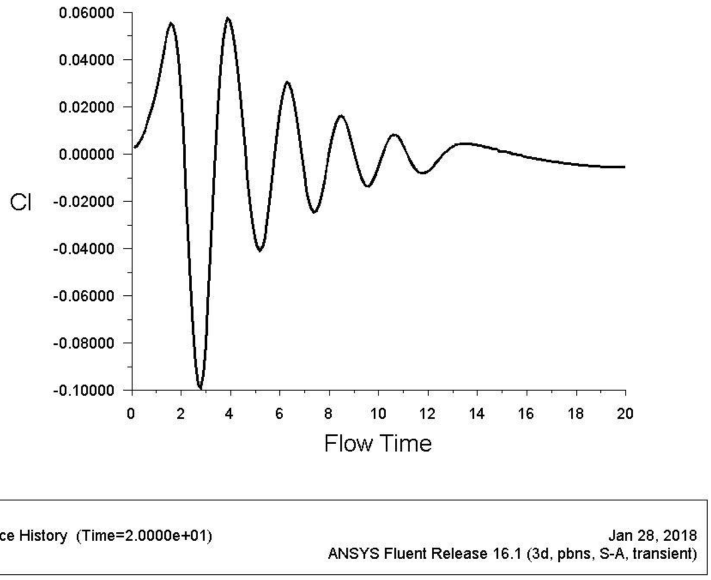


Figure 4 Convergence History of Lift Coefficient (Cl) of S-A model

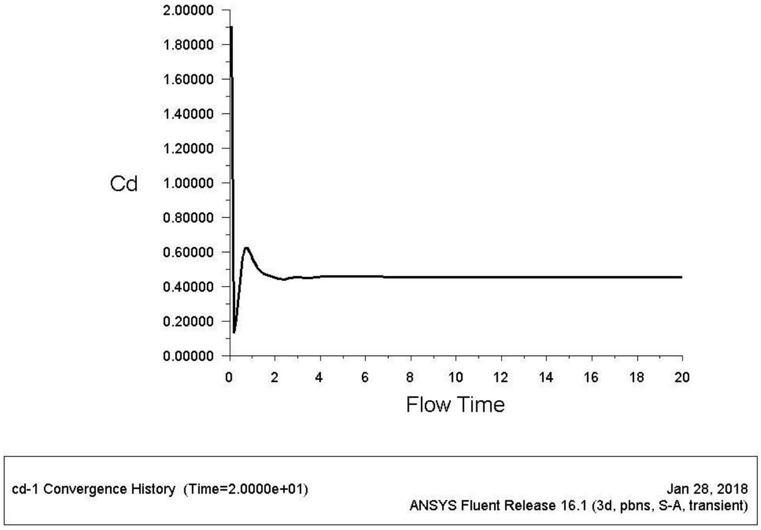
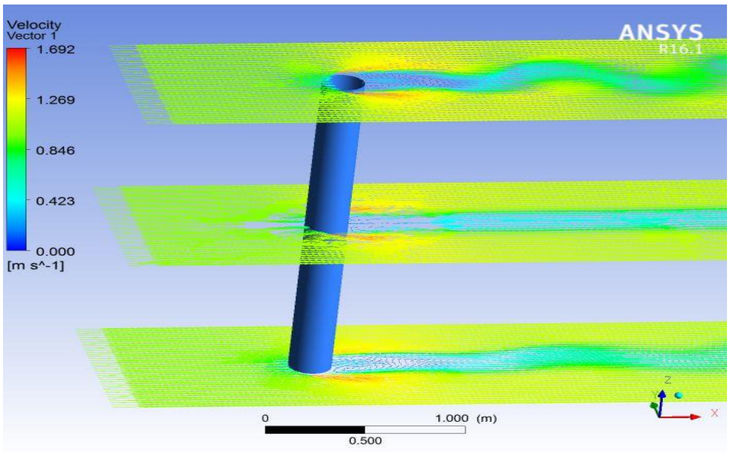


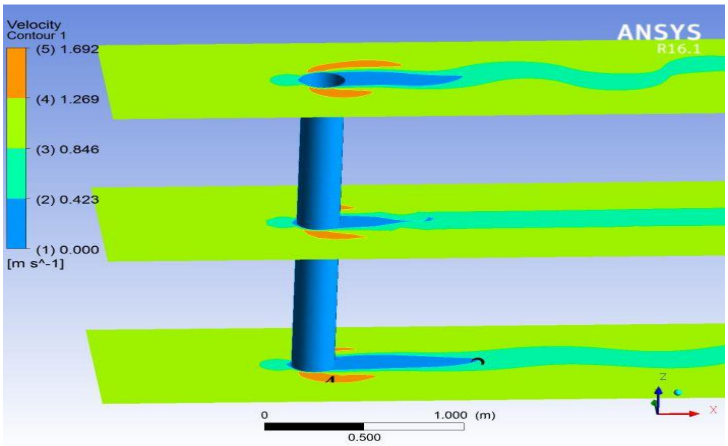
Figure 5 Convergence History of Drag Coefficient (Cd) of S-A model

The vortex pattern is investigated by the velocity contours. Figure 6 shows the velocity vectors and velocity contours in specified zones on the strip for the Riser section. The high values of velocity occur at both side of Riser and lower values at the rear, therefore, the maximum shear stresses can be seen at high velocity's area as appear in Fig. 7.

Pressure differences between front and rear of Riser lead the Riser to vibrate, referring to Vortex Induced Vibration (VIV); those differences are clearly appearing in Fig. 8. When the vortex's frequency meets with natural frequency of the Riser the resonance will occur.



a



b

Figure 6 a) Velocity vector at z = 0m, z = 1.1m and z = 2.2m

b) Velocity contour at z = 0m, z = 1.1m and z = 2.2m

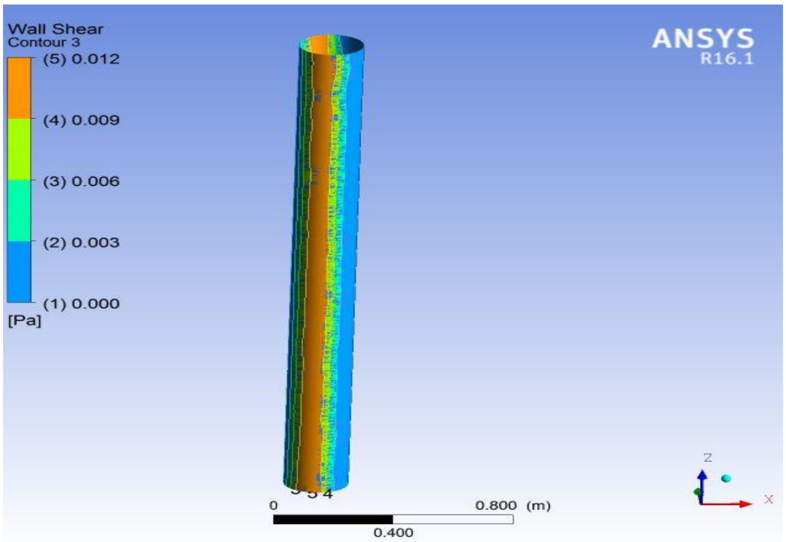


Figure 7 Shear stress contour of Riser's wall.

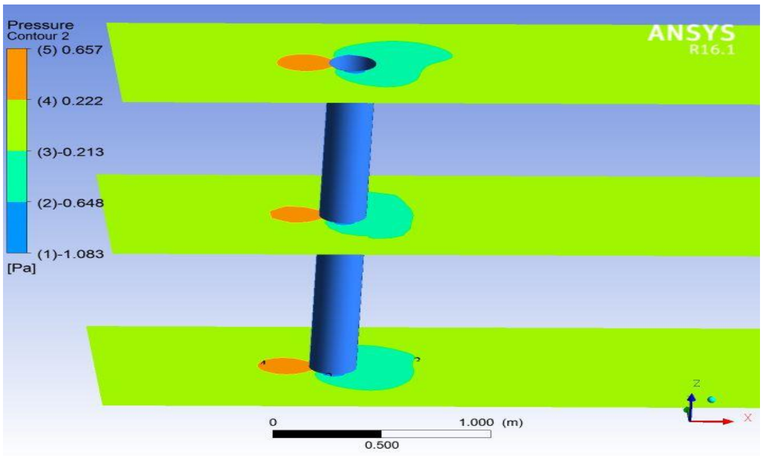


Figure 8 Pressure distribution around Riser at z = 0m, z = 1.1m and z = 2.2m

3.2. LES MODEL

Figures 9-14 show the results obtained from the model based on Large Eddy Simulation approach with remaining other parameters the same. The amplitude of (Cl) is the same behavior of Standard Spalart-Allmaras model but it has higher values Fig.9. This interpretation also applies on drag coefficient, where it is can see that maximum value of (Cd) is 0.9 at t = 1.6s as shown in Fig. 10.

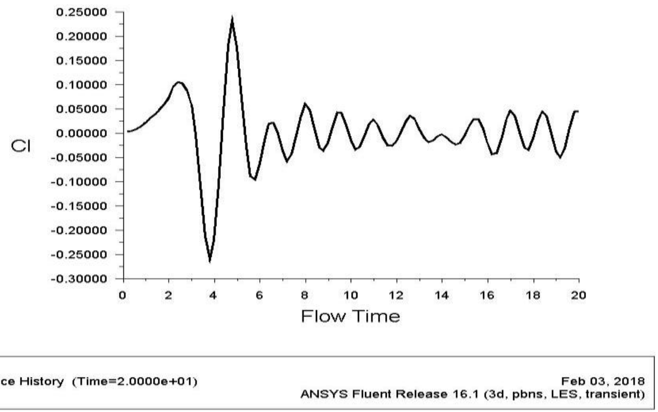


Figure 9 Convergence History of Lift Coefficient (Cl) of LES model

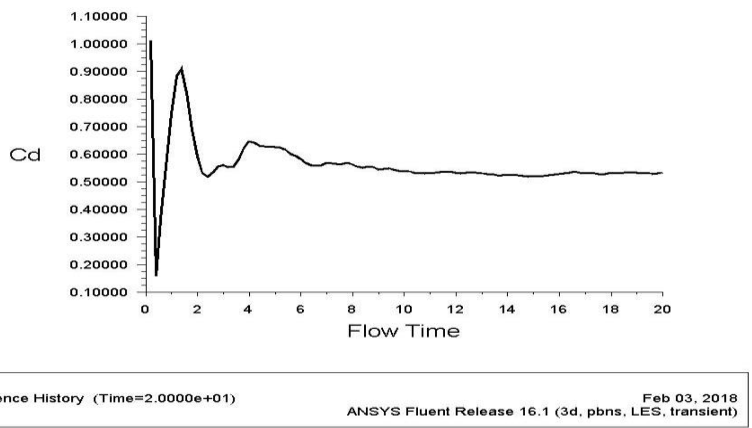
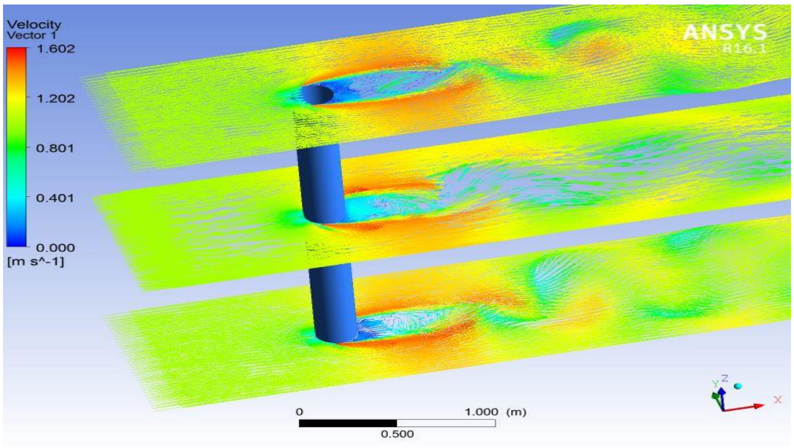
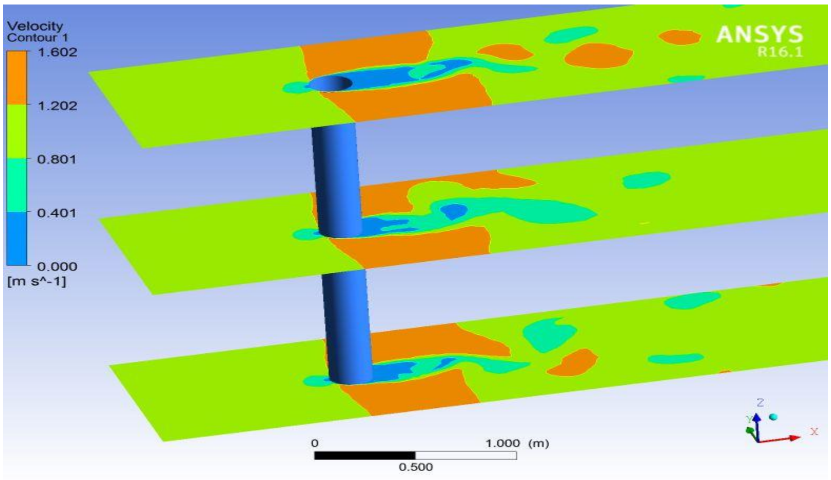


Figure 10 Convergence History of Drag Coefficient (Cd) of LES model

Calculated magnitudes of shear wall stress and pressure have obviously larger values than S-A model because of the velocity distribution around Riser will be less as plotted in Fig. 11, Fig. 12 and Fig. 13.



a



b

Figure 11 a) Velocity vector at z = 0m, z = 1.1m and z = 2.2m b) Velocity contour at z = 0m, z = 1.1m and z = 2.2m

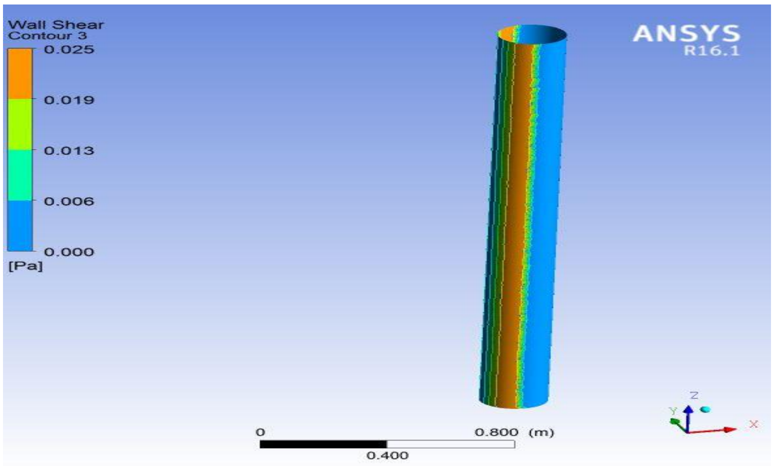


Figure 12 Shear stress contour of Riser's wall.

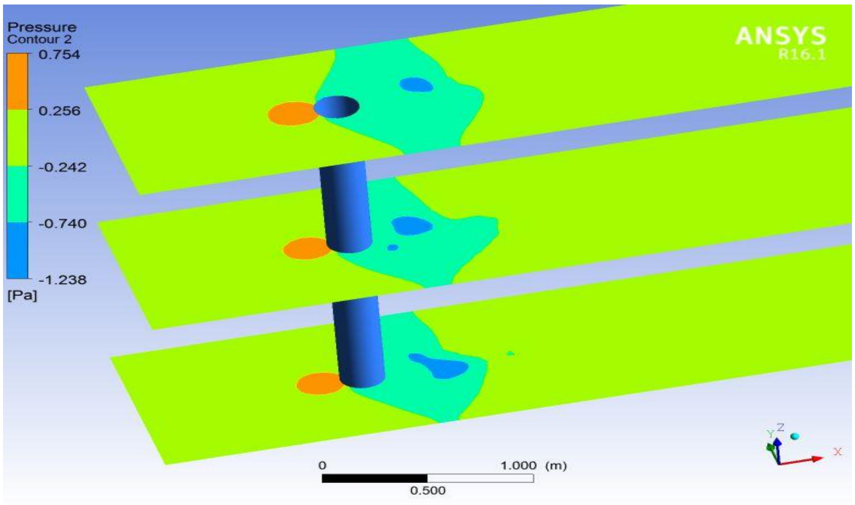


Figure 13 Pressure distribution around Riser at z = 0m, z = 1.1m and z = 2.2m

4. CONCLUSIONS

Probable, the most decisive parameter in CFD simulations is how the flow should be solved. The two turbulence models are used to model the turbulent flow; Spalart-Allmaras model gives low values of Cd and Cl with fairly steady compared with Large Eddy Simulation. With respect to Spalart-Allmaras model can be noted that the largest value for the drag coefficient is 0.65 and it is obtained at t = 1s, while the maximum value of (Cd) is 0.9 at t = 1.6s for Large Eddy Simulation and A calculated magnitudes of shear wall stress and pressure have obviously larger values than S-A model because of the velocity distribution around Riser will be less.

REFERENCES

[1] Constantinides, Y. and O.H. Oakley. Numerical prediction of VIV and comparison with field experiments. in ASME 2008 27th International Conference on Offshore Mechanics and Arctic Engineering. 2008. American Society of Mechanical Engineers.

[2] Constantinides, Y., O.H. Oakley, and S. Holmes. CFD high L/D riser modeling study. in ASME 2007 26th International Conference on Offshore Mechanics and Arctic Engineering. 2007. American Society of Mechanical Engineers.

[3] Gustafsson, A., Analysis of vortex-induced vibrations of risers. 2012.

[4] Wu, H., et al., Numerical simulation of vortex-induced vibration of two long flexible circular cylinders in tandem. 2016.

[5] Lucor, D., H. Mukundan, and M. Triantafyllou, Riser modal identification in CFD and full-scale experiments. Journal of Fluids and Structures, 2006. 22(6-7): p. 905-917.

[6] Chen, H.-C., et al., Cfd simulation of riser viv. OTRC Project Final Report (Project Report Nos. 32558/22820/CE and 32558/2282A/SC), 2007.

[7] de Wilde, J.J. and R.H. Huijsmans. Laboratory investigation of long riser VIV response. in The Fourteenth International Offshore and Polar Engineering Conference. 2004. International Society of Offshore and Polar Engineers.

[8] Rosetti, G.F., G. Vaz, and A.L. Fujarra. Modeling transition for the cylinder flow with verification and validation. in ASME 2015 34th International Conference on Ocean, Offshore and Arctic Engineering. 2015. American Society of Mechanical Engineers.

[9] White, F.M., Fluid Mechanics fourth edition, McGraw and Hill. International Edition, Singapore, 1994.