

Analysis of Vortex-Induced Vibration of Riser using Spalart-Almaras Model

Jaswar Koto^{a,b*}, Abdul Khair Junaidi^a

^aDepartment of Aeronautic, Automotive and Ocean Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia

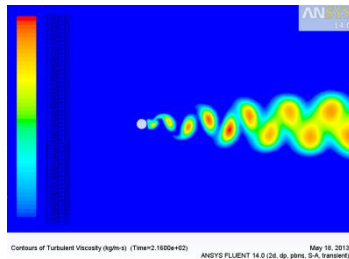
^bOcean and Aerospace Research Institute, Indonesia

*Corresponding author: jaswar@fkm.utm.my and jaswar.koto@gmail.com

Article history

Received :20 April 2014
Received in revised form :
10 May 2014
Accepted :1 June 2014

Graphical abstract



Abstract

Vortex-induced vibration is natural phenomena where an object is exposed to moving fluid caused vibration of the object. Vortex-induced vibration occurred due to vortex shedding behind the object. One of the offshore structures that experience this vortex-induced vibration is riser. The riser experience vortex-induced vibration due to vortex shedding caused by external load which is sea current. The effect of this vortex shedding to the riser is fatigue damage. Vortex-induced vibration of riser becomes the main concern in oil and gas industry since there will be a lots of money to be invested for the installation and maintenance of the riser. The previous studies of this vortex-induced vibration have been conducted by experimental method and Computational Fluid Dynamics (CFD) method in order to predict the vortex shedding behaviour behind the riser body for the determination of way to improve the riser design. This thesis represented the analysis of vortex induced vibration of rigid riser in two-dimensional. The analysis is conducted using Computational Fluid Dynamic (CFD) simulations at Reynolds number at 40, 200, 1000, and 1500. The simulations were performed using Spalart-Allmaras turbulent model to solve the transport equation of turbulent viscosity. The simulation results at Reynolds number 40 and 200 is compared with the other studies for the validation of the simulation, then further simulations were conducted at Reynolds number of 1000 and 1500. The coefficient of lift and drag were obtained from the simulations. The comparison of lift and drag coefficient between the simulation results in this study and experiment results from the other studies showed good agreement. Besides that, the in-line vibration and cross-flow vibration at different Reynolds number were also investigated. The drag coefficient obtained from the simulation results remain unchanged as the Reynolds number increased from 200 to 1500. The lift coefficient obtained from the simulations increased as the Reynolds number increased from 40 to 1500.

Keywords: Vortex-induced vibration; CFD; turbulent model

© 2014 Penerbit UTM Press. All rights reserved.

1.0 INTRODUCTION

Vortex-induced vibration (VIV) becomes one of the problems that concern everyone in marine industry as the marine riser system is designed to oil and gases from sea bed. The riser systems may have potential in experiencing a highly level of fatigue damage in a relatively period of time due to exposure of the riser to harsh current environments because of vortex-induced vibration. Vortex-induced vibration is a phenomenon where objects such as cylinder interact with moving fluid through it. When an object is immersed in moving water, it will experience a phenomenon where the sea currents caused the object to be excited by forces that caused by vortex shedding.

Vortex Shedding is an unsteady flow that occurred in special flow velocities depend on size shape of the object. The vortex shed occurred at the back of the body and separated periodically from each side of the body causing the time varying different pressure around the object explained that the vortex shed are not occurred symmetrically. This unsymmetrical vortex shedding cause the different lift force around the object resulting vibration

of the object in both parallel and perpendicular to the moving flow cause the cylinder body to move. The lock-in phenomena is associates with this vortex-induced vibration since it related with Eigen frequency of the riser and also frequency of the sea current.

2.0 LITERATURE REVIEW

2.1 Offshore Field Component

Offshore field can be divided into several areas in upstream sector and also known as the exploration and production sector that working on the searching for potential underground and underwater oil and gas fields. As shown in Figure 1, the offshore field components that involved in this upstream sector are including platform structure and process system, subsea system, and transport system. All of in these offshore field components are concerning the exploration of oil and gas and also the production process. This offshore field is also associated with

downstream sector where this sector is commonly involved in the process of refining, marketing, and selling the oil and gas product.

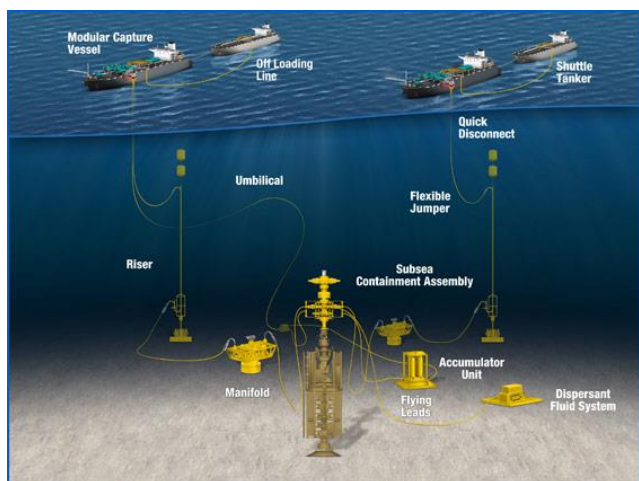


Figure 1 Offshorefield components
(http://marineoilwellcontainment.com/expanded_system.php)

2.2 Platform Structure

An oil platform is often referred as an offshore platform or platform structure since it is a large structure. This huge structure is used for exploration and process of oil and gas, and also used to store product temporarily until it can be transport to shore for the further process including refining and marketing process. The subsea system is controlled through the platform structure by using umbilical that used to control the activities of the subsea system. This platform structure is connected from upstream to downstream through riser and pipeline. Most of platform structures are contains house for worker (living quarters), drill well, helipad, oil rig etc. These facilities are installed depend on the demand and requirements for the platform structure. The size of the platform structure is depending to the facilities to be installed on the top side of this structure.

2.3 Process System

The offshore process system is normally referred to offshore platform production and the subsea processing system. This process system receives 3-phase (oil, gas, water) well fluid from subsea system and processed at platform. This process are consisting four major processing module which are separation process, gas compression and dehydration process, produced water conditioning, and sea water processing and injection system. These processes are also have the fire detection and suppression system, power generation, well services or drilling modules, water maker or utilities or sewage treatment, and living quarters.

2.4 Subsea System

Subsea system is normally associated with all process in oil and gas exploration and production. This system is also normally used for exploration and production in deep-water. This system can be divided into several parts including subsea drilling system, subsea process system, subsea production, and subsea control system.

2.5 Transport System

Transport system in offshore field also can be defined as a system that involve with the transportation process of oil and gas from the sea floor to the surface facility like floating structure. This system is concerning the transport from the well heads to the platform. The components involve are subsea jumpers, pipelines, and risers.

2.6 Deep-water Riser

Riser system is one of the most important systems used in marine industry especially for offshore application on deep-water. The system is used to extract oil and gas from seabed as mentioned above. This riser system is also extremely important conductor pipes that used to connect the floaters from surface and the wellheads at seabed. There are fundamentally two kinds of risers introduced such as rigid riser and flexible riser. The combination of both rigid and flexible risers are called hybrid riser. As shown in Figure 2, there are many possible type of configuration for marine riser such as catenary riser, catenary riser, top tensioned production riser, lazy S riser, steep S riser, lazy wave riser, steep wave riser and pliant wave riser. These configurations of risers are widely used for deep-water and basically it is configured due to the requirement of the deep-water production. New configurations of riser are also available for deep-water production such as Compliant Vertical Access Riser (CVAR), (multi-bore) hybrid riser (Yong Bai).

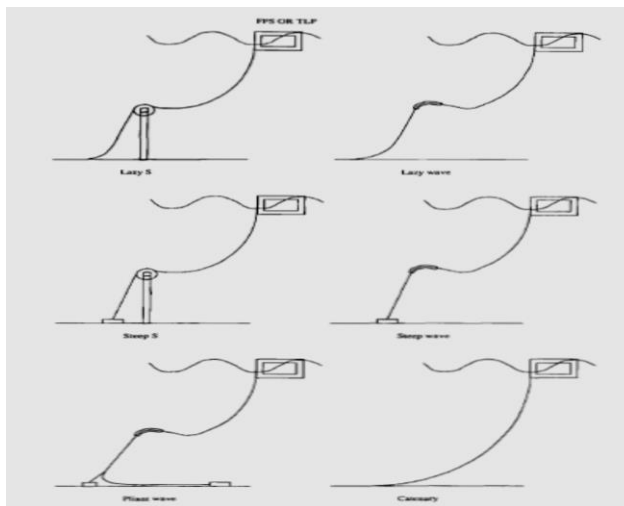


Figure 2 Riser configurations

2.7 Vortex-induced Vibration

Vortex-induced vibration is widely known in the oil and gas industry for several decades. The vortex-induced vibration is known as self-exciting, self-regulating, self-limiting highly phenomenon that received extensive attention in this oil and gas industry due to the flow-structure-interaction problem. This phenomenon commonly are excited by vortex shedding due to external load that is sea currents as the riser is experiencing sea currents in deep-water. This vortex-induced vibration also influenced by a large number of parameters including mass ratio, structure stiffness, damping, surface roughness and the most important parameters for this vortex-induced vibration are Reynolds number, Strouhal number, and reduced velocity. The body literature related to this vortex-induced vibration is massive. The body literature are covers all experimental and numerical

investigation methods, multi-degree or single degree of freedom motion, rigid or flexible riser motion, and vortex-induced vibration of riser in deep-water.

2.8 Analysis Method of Vortex-Induced Vibration

Vortex-induced vibration is widely known in the world of oil and due to its effect on riser used to extract oil and gas from sea floor that caused by external load. In the previous literatures, the problems related to vortex-induced vibration are studied and investigated. As the result, many researchers have been done to solve this problem. There were several methods of analysis conducted previously to predict, simulate, and solve this vortex-induced vibration since this problem is concerned in oil and gas industry. The methods used to investigate this problem are including empirical method and computational method.

2.9 Reynolds-Averaged Navier-Stokes (RANS)

Reynolds-Averaged Navier Stokes (RANS) is an approach used in the numerical method for the analysis of vortex-induced vibration. Reynold-Averaged Navier-Stokes are derived from time-averaging the Navier-Stokes equations. The idea behind these equations is the Reynolds decomposition which referred to the separation of the flow variable. There are numerous studies of vortex-induced vibration using this Reynolds-Averaged Navier-Stokes for the simulation by numerical method.

3.0 METHODOLOGY

3.1 Methodology of CFD

In general, CFD simulation is divided into three main stages, which are pre-processor, simulator or solver, and post-processor. The processing stage is the stage where the geometry of the problem is defined as the solution domain and the fluid volume is divided into discrete cells or mesh. The physical modeling, parameter chemical phenomena, fluid properties and boundary conditions are necessary to be defined. In this study, the engineering software used for this study is ANSYS ICEM. The second stage, which is the solver stage where the fluid flow problem is solved using numerical methods which applied in the engineering software called ANSYS Fluent. The numerical methods are either finite different method (FDM), finite element method (FEM), or finite volume method (FVM). The last stage is the post-processor. The post-processor is performed for the analysis and visualization of the resulting solution. Many CFD packages are equipped with versatile data visualization tools, for instance domain geometry and grid display, vector plots, 2D and 3D surface plot, particle tracking and soon.

3.1.1 Pre-processing Stage

Pre-processing stage is a stage where the flow fluid problem is defined by giving the input in order to get the best solution of the problem. There are many factors that affect the CFD solution accuracy and some of the factors are in this pre-processing stage. The pre-processing stages are divided into several stages which are:

- Define the solution domain
- Generation of mesh
- Parameters for physical modelling
- Properties of fluid
- Boundary conditions

A) Solution Domain

The solution domain can be defined as the abstract environment where the solution is calculated. The shape of the solution domain can be in any shape like circular and rectangular. For this study, the solution domain is in 2D which is illustrated as in Figure 3.

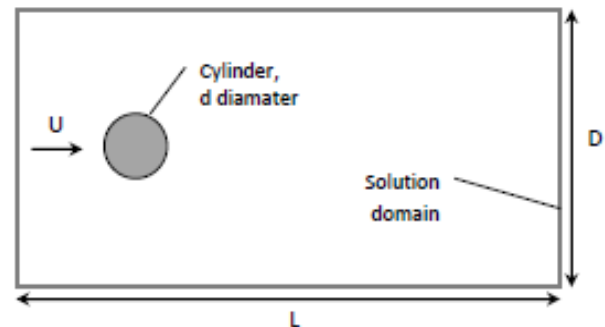


Figure 3 Rectangular shaped solution domain

B) Meshing

Meshing can be defined as the discrete locations at which the variables are to be calculated and to be solved. The grid divides the solution domain into a finite number of sub domains, for instance elements, control volumes etc. Meshing can be generated after the solution domain is defined. The term of mesh generation and grid generation is often interchangeably.

There are two types of grids by Ferziger and Peric used in this study which are structured grid and unstructured grid as shown in figure.4 and figure.5 respectively. A structured grid is a regular grid consists of groups of grid lines with the property that members of a single group do not cross each other and cross each member of the other groups only once. This is the simplest grid structure since it has only four neighbours for 2D and six neighbours for 3D. Even though it simplifies programming and the algebraic equation system matrix has a regular structure, it can be used only for geometrically simple solution domains.

Unstructured grid is a type of grid can be used for very complex geometries. It can be used for any discretization method, but they are best for finite element and volume methods. Even though it is very flexible, there is the irregularity of the data structure. Moreover, the solver for the algebraic equation systems is usually slower than for structured grids.

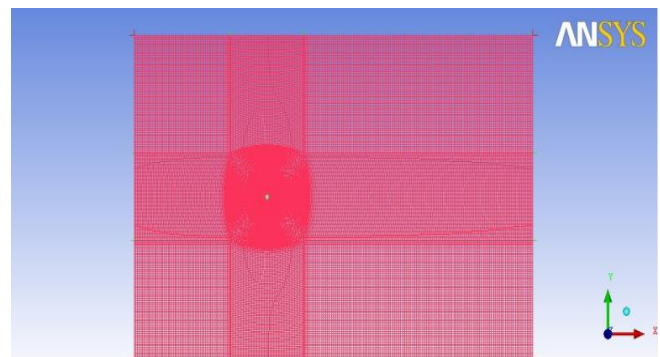


Figure 4 Structured grid

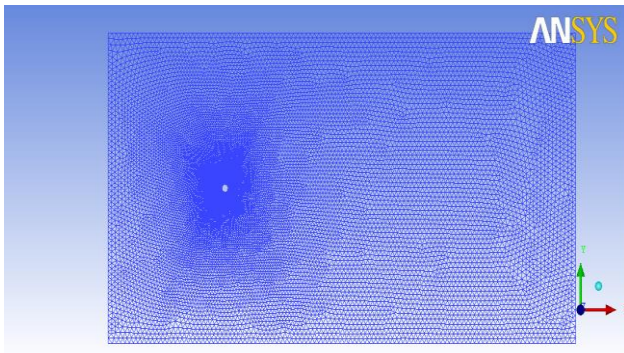


Figure 5 Unstructured grid

C) Boundary Condition

There are several boundary conditions for the discretised equations. Some of them are inlet, outlet, wall, prescribed pressure, symmetry and periodicity.

- Inlet Boundary Condition:** The inlet boundary condition permits flow to enter the solution domain. It can be a velocity inlet, pressure inlet or mass flow inlet.
- Outlet Boundary Condition:** The outlet boundary condition permits flow to exit the solution domain. It also can be a velocity inlet, pressure inlet or mass flow inlet.
- Wall Boundary Condition:** The wall boundary condition is the most common condition regarding in confined fluid flow problems, such as flow inside the pipe. The wall boundary condition can be defined for laminar and turbulent flow equations.
- Prescribed Pressure Boundary Condition:** The prescribed pressure condition is used in condition of external flows around objects, free surface flows, or internal flows with multiple outlets.
- Symmetry Boundary Condition:** This condition can be classified at a symmetry boundary condition, when there is no flow across the boundary.

3.1.2 Simulation or Solving Setup Stage

In this stage, the numerical techniques are used to solve the fluid flow problems where all of CFD simulations used this technique. This technique or method is commonly called the discretization method. The meaning of the discretisation method is that the differential equations are approximated by an algebraic equation system for the variables at some set of discrete location in space and time. There are three main techniques of discretization method, the finite difference method, the finite element method and the finite volume method. Even though these methods have different approaches, each type of method yields the same solution if the grid is very fine.

The Finite Difference Method (FDM) is one of the easiest methods to use, particularly for simple geometries. It can be applied to any grid type, whether structured or unstructured grids. FDM is very simple and effective on structured grid. It is easy to obtain higher-order schemes on regular grid. On the other hand, it needs special care to enforce the conservation condition. Moreover, for more complex geometry, this method is not appropriate. The advantage of FEM is its ability to deal with arbitrary geometries. The domain is broken into unstructured discrete volumes or finite elements. They are usually triangles or quadrilaterals (for 2D) and tetrahedral or hexahedra (for 3D). However, by using unstructured grids, the matrices of the

linearized equations are not as well ordered as for structured grids. In conclusion, it is more difficult to find efficient solution methods.

FEM is widely used in structural analysis of solids, but is also applicable to fluids. To ensure a conservative solution, FEM formulations require special care. The FEM equations are multiplied by a weight function before integrated over the entire solution domain. Even though the FEM is much more stable than finite volume method (FVM), it requires more memory than FVM. FVM is a common approach used in CFD codes. Any type of grid can be accommodated by this method. Indeed, it is suitable for complex geometries. This method divides the solution domain into a finite number of contiguous control volumes (CV), and the conservation equations are applied to each CV.

4.0 RESULTS AND DISCUSSIONS

4.1 Result of Grid Quality Evaluation

At this stage, a grid for the further step of simulation shall be selected based on two criteria which are element quality and orthogonal quality. Both of the quality criteria are obtained from ANSYS ICEM. A tabulation of grid quality for structured grid and unstructured grid is given in Table 1. The table indicates the values of different grid quality for each type of grid.

Based on the Table 1, it can be concluded that the best grid quality is the structured grid. The Table 1 shows that structured grid have better quality of element quality and orthogonal quality in term of minimum, maximum, and mean values. Therefore, the structured grid shall be used for further simulations.

4.2 Validation of CFD Simulations

The validation of the simulation is conducted by comparison between the simulation results with previous study. The comparison are made at $Re = 40$ and 200 .

A) Steady Laminar Case at $Re = 40$

The simulation is set at $Re = 40$ and a steady laminar flow condition. The number of iteration for this simulation is 2000 with reporting interval is 1 and profile update interval is 1. The solution is converged at 339 iterations and yield C_d equal to 1.632.

Table 2 shows the value of C_d from the results of CFD simulation and the experiment results from previous study. The comparison of C_d values between the CFD simulation results and experiment results from previous study indicates that the value obtained from the CFD simulation is in good agreement to other measurements. The value of C_d from CFD simulation from this study is within the range of the experiment values from the previous study. Therefore, the structured grid used for the CFD simulation shall be used for the simulations of vortex-induced vibration of 2D rigid riser at $Re = 200, 1000, \text{ and } 1500$.

B) Transient (Unsteady) Case at $Re = 200$

In this case, the rigid riser is exposed to the turbulent flow with Reynolds number of 200. The comparison is made with the experiment results as shown in Table 3. The Spalart-Allmaras turbulent model is selected for the turbulent model in modelling the transient flow case. 30 time steps/max iterations were chosen in one shedding cycle for Strouhal number of 0.2 (average estimation for flow past cylinder). The vortex shedding frequency for this case is 0.2 Hz and the time step is equal to 0.2 seconds

with number of time steps of 1080. The simulations yield the value of $C_d = 1.310$ and $C_l = 0.602$.

4.3 Case Study

Further simulations of 2D vortex-induced vibration at $Re = 1000$ and 1500 is conducted. The graphs of lift and drag coefficient is plotted against Reynolds number.

4.3.1 In-Line Vibration

Figure.6 shows the coefficient of drag against Reynolds number. The graph is plotted based on the values of coefficient of drag for each Reynolds number 40, 200, 1000, and 1500. The coefficient of drag is almost similar for each Reynolds number of 200, 1000, and 1500. The value of drag coefficient at Reynolds number 40 is higher compared to the value of drag coefficient at Reynolds number 200, 1000, and 1500. The Reynolds number of 40 is in the range of laminar flow where the vortex shedding will formed in a pair (two-attached re-circulating vortices) for the range of $40 < Re < 200$. The Reynolds number 200 is in the range of turbulent flow, where the wake region behind the rigid riser experience the transition from laminar to turbulent in range $200 < Re < 300$. The flow behind the rigid riser experiences complete turbulent for the Reynolds number 1000 and 1500 as the flow regime at range $300 < Re < 3.5 \times 10^5$ is complete turbulent flow.

For a laminar case ($Re = 40$), the drag coefficient is larger than the drag coefficient of turbulent flow ($Re = 200, 1000, \text{ and } 1500$). This indicates that the drag force for laminar flow is larger than turbulent flow for Reynolds number 40 to 1500. Therefore, the in-line vibration of rigid riser for $Re = 40$ is higher than in-line vibration at $Re = 200, 1000, \text{ and } 1500$.

4.3.2 Cross-Flow Vibration

Figure.7 shows the graph of lift coefficient against Reynolds number. The graph is plotted with the Reynolds number of 40, 200, 1000, and 1500. The graph indicates that for a Reynolds number range 40 to 1500, the value of lift coefficient increases. As mentioned in the above session, the Reynolds number of 40 is within the range of a laminar flow where the wake of laminar flow where the vortex shedding will form in a pair (two-attached re-circulating vortices). The Reynolds number 200 is in the range of turbulent flow, where the wake region behind the rigid riser experience the transition from laminar to turbulent in range $200 < Re < 300$. The flow behind the rigid riser experiences complete turbulent for the Reynolds number 1000 and 1500 as the flow regime at range $300 < Re < 3.5 \times 10^5$ is complete turbulent flow.

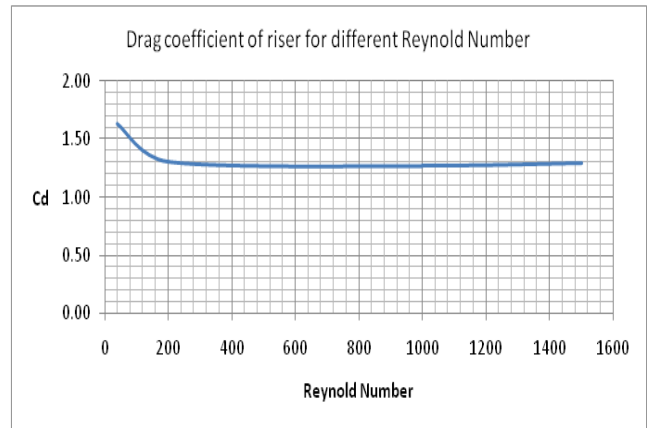


Figure 6 Graph of C_d versus Re range 40 to 1500

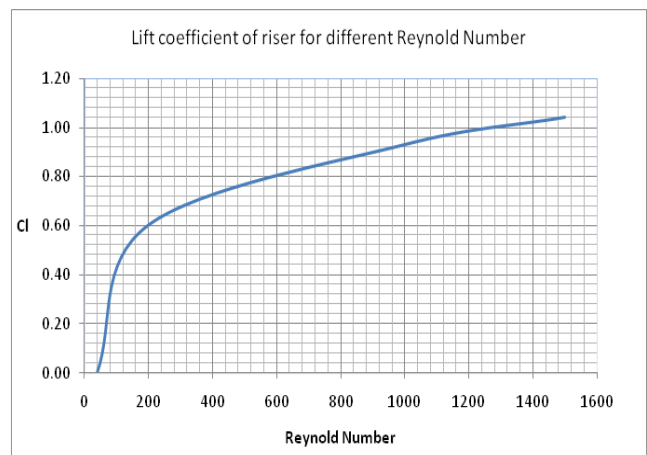


Figure 7 Graph of C_l versus Re range 40 to 1500

The simulation result of vortex-induced vibration of 2D rigid riser at $Re = 40$ yield the lift coefficient which almost zero that is 0.0018. This indicates that the cross-flow vibration at $Re = 40$ is very small as the vortex shedding in the flow regime behind the rigid riser body is symmetrical. Therefore, the pressure different is smaller as shown in Figure 8 in Figure 9. The value of lift coefficient increases significantly from $Re = 40$ to $Re = 200$ indicates that the cross-flow become significantly larger from the transition of laminar flow to turbulent flow. Therefore, the cross-flow vibration changes significantly during the transition of laminar flow to turbulent flow. The coefficient of lift increase (almost linear) from $Re = 200$ to $Re = 1500$.

Table 1 Grid quality measurement

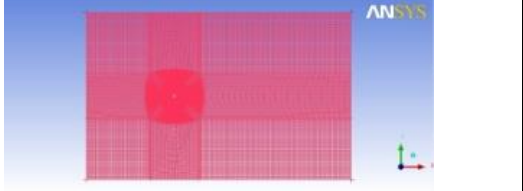
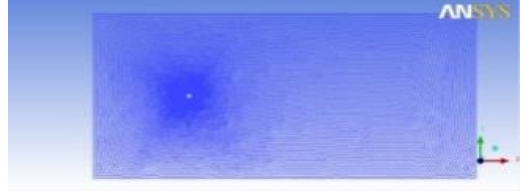
Criteria						
	Structured Grid			Unstructured Grid		
	Minimum	Maximum	Mean	Minimum	Maximum	Mean
Element Quality	0.928	1.000	0.988	0.431	0.999	0.950
Orthogonal Quality	0.923	1.000	0.995	0.698	1.000	0.996

Table 2 Values of Cd from experimental data obtained from previous journals and CFD simulation

Experimental Data (Journal)	C _d	CFD Simulation (Present Study)
Linnick and Farel	1.540	C _d = 1.632
Herfjord	1.600	
Berthelsen and Faltinsen	1.590	
Russel and Wang	1.600	
Xu and Wang	1.660	
Calhoun	1.620	

Table 3 The values of Cl from experimental data obtained from previous journals and CFD simulation

Experimental Data (Journal)	C _d	Cl	CFD Simulation (Present Study)
Linnick and Farel	1.340	0.690	C _d = 1.310 Cl = 0.602
Herfjord	1.350	0.700	
Berthelsen and Faltinsen	1.370	0.700	
Russel and Wang	1.290	0.500	
Xu and Wang	1.420	0.660	
Calhoun	1.170	0.668	
Franke, <i>et al.</i>	1.310	0.650	
Rajani, <i>et al.</i>	1.337	0.424	

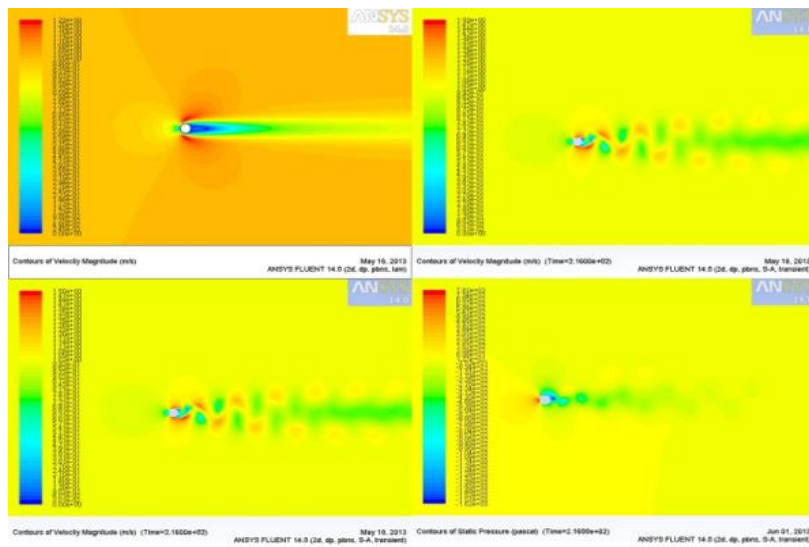


Figure 8 Contours of velocity magnitude (Re = 40, 200, 1000, and 1500)

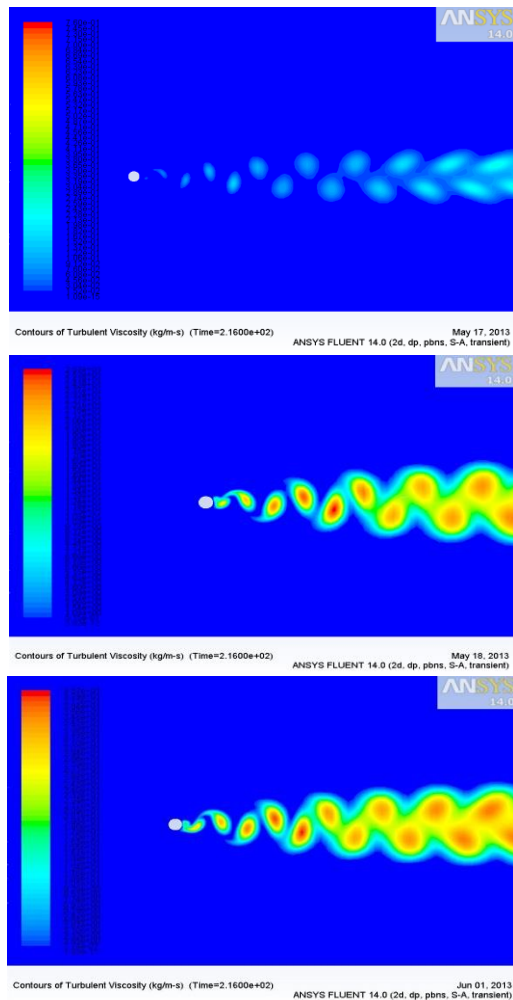


Figure 9 Contours of Turbulent Viscosity (Re = 200, 1000, and 1500)

5.0 CONCLUSIONS

The conclusion is the flow characteristic of the steady laminar case at $Re = 40$ which represented by the value of drag coefficient shows a good agreement with the experiment results from the previous studies. Therefore, the Spalart-Allmaras turbulent model give accurate result for the simulation of vortex-induced vibration of 2D rigid riser at laminar flow of $Re = 40$. The value of drag coefficient and lift coefficient at $Re = 200$ shows a good agreement with the experiment results. The simulation results in this study show the value of drag coefficient is almost similar for the Reynolds number 40 to 1500. The in-line vibration is almost similar for the Reynolds number range from 40 to 1500. The simulation results in this study at range of Reynolds number 40 to 1500 indicate that the coefficient of lift is increases as the Reynolds number increase.

6.0 RECOMMENDATION

1. There are various factors that affect vortex-induced vibration which may lead to the success or failure of the analysis. Therefore, specific studies to investigate the most

factors that affect vortex-induced vibration are recommended.

2. The analysis of vortex-induced vibration can be conducted either by experiment or CFD simulations. For CFD simulation, the validation of the simulation is with other studies is necessary in order to validate the factors for the simulation such as domain shape, size, solver used, and etc. therefore, validation of the simulation result before performing further simulation is recommended.
3. There are other type of turbulent models can be used to perform the simulation of vortex-induced vibration to solve the transport equation of turbulent viscosity.

Acknowledgement

The authors also would like to acknowledge UniversitiTeknologi Malaysia (UTM) and Ministry of Higher Education (MOHE) of Malaysia for supporting this research.

References

- [1] Adi Maimun, M. Mobassher, Yasser, M. Ahmad. 2012. Fundamentals of Vortex Induced Vibration Analysis of Marine Riser. Offshore Mechanics and Hydrodynamics, Mechanical–Marine Technology, University Technology Malaysia, Johor, Malaysia.
- [2] Berthelsen. 2008. A Local Directional Ghost Cell Approach for Incompressible Viscous Flow Problems with Irregular Boundaries. *Journal of Computational Physic.* 227.
- [3] Feng. 1968. The Measurement of the Vortex Induced Effects in Flow Past Stationary and Oscillating Circular and D-Section Cylinders. The University of British Columbia.
- [4] J. T. Klamo. 2007. Effects of Damping and Reynolds Number on Vortex-induced Vibration. California Institute of Technology, Pasadena, California.
- [5] J. Deng. 2007. The Flow Between Stationary Cylinder and a Downstream Elastic Cylinder In Cruciform Arrangement. Department of Mechanics, Institute of Fluid Engineering, Zhejiang University, Hangzhou, China.
- [6] J. M. Dahl. 2006. Two-degree-of-freedom Vortex-induced Vibrations using a Force Assisted Apparatus. Department of Mechanical Engineering, Centre for Ocean Engineering, Cambridge, USA.
- [7] J. B. Wanderley. 2005. Vortex Induced Loads on Marine Risers. Ocean Engineering Department, Brazil.
- [8] Jaswar, M. K. R. A. Razak and C. L. Siow, 2013. Vortex-Induced Vibration of Riser. International Conference on Marine Safety and Environment, Johor Bahru, Malaysia
- [9] M. N. Linnick. 2005. A High Order Immersed Interface Method for Simulating Unsteady Compressible Flows on Irregular Domains. *Journal of Computer Physics* 204.
- [10] M. T. Asyikin. 2012. CFD Simulation of Vortex Induced Vibration of a Cylindrical Structure. Department of Civil and Transport Engineering, Norwegian University of Science and Technology.
- [11] R. Violette. 2007. Computational of Vortex-induced Vibration of Long Structures Using a Wake Oscillator Model: Comparison with DNS and Experiments. Department of Mechanic, France.
- [12] S. Yong. 2007. Vortex Induced Vibrations of Finned Cylinders. Technology Research Department, CNOOC Research, Beijing, China.
- [13] W. Wu. 2011. Two-dimensional RANS Simulation of Flow Induced Motion of Circular Cylinder with Passive Turbulence Control. Naval Architecture and Marine Engineering, The University of Michigan,
- [14] Z. Y. Pan. 2006. Numerical Simulation of Vortex-induced Vibration of a Circular Cylinder at Low Mass-damping Using RANS Code. Laboratory of Ocean Engineering, China Ship Scientific Research Centre, China.
- [15] Zhongjun Yin. 2012. Numerical Simulation of the Damping Influence on Vortex-Induced Vibration. School of Mechanical Engineering, University of Science and Technology, Beijing, China.