NUMERICAL ANALYSIS OF MARINE RISER VORTEX-INDUCED VIBRATION WITH HELICAL STRAKE SUPPRESSION DEVICE

DINIESH NAIDU S/O THANNIMALAI

FACULTY OF ENGINEERING
UNIVERSITY OF MALAYA
KUALA LUMPUR

2019

NUMERICAL ANALYSIS OF MARINE RISER VORTEX-INDUCED VIBRATION WITH HELICAL STRAKE SUPPRESSION DEVICE

DINIESH NAIDU S/O THANNIMALAI

RESEARCH REPORT SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENT FOR THE DEGREE OF MASTER OF ENGINEERING

FACULTY OF ENGINEERING
UNIVERSITY OF MALAYA
KUALA LUMPUR

2019

UNIVERSITY OF MALAYA

ORIGINAL LITERARY WORK DECLARATION

Name of Candidate: Diniesh Naidu S/O Thannimalai

Registration/Matric No: KQK 160008

Name of Degree: Masters of Mechanical Engineering

Title of Project Paper/Research Report/Dissertation/Thesis ("this Work"): Numerical Analysis of Marine Riser Vortex-Induced Vibration with Helical Strake of Suppression Device

Field of Study: Fluid Dynamics

I do solemnly and sincerely declare that:

(1) I am the sole author/writer of this Work;
(2) This Work is original;
(3) Any use of any work in which copyright exists was done by way of fair dealing and for permitted purposes and any excerpt or extract from, or reference to or reproduction of any copyright work has been disclosed expressly and sufficiently and the title of the Work and its authorship have been acknowledged in this Work;
(4) I do not have any actual knowledge nor do I ought reasonably to know that the making of this work constitutes an infringement of any copyright work;
(5) I hereby assign all and every rights in the copyright to this Work to the University of Malaya ("UM"), who henceforth shall be owner of the copyright in this Work and that any reproduction or use in any form or by any means whatsoever is prohibited without the written consent of UM having been first had and obtained;
(6) I am fully aware that if in the course of making this Work I have infringed any copyright whether intentionally or otherwise, I may be subject to legal action or any other action as may be determined by UM.

Candidate's Signature Date:

Subscribed and solemnly declared before,

Witness's Signature Date:

Name:

Designation:

NUMERICAL ANALYSIS OF MARINE RISER VORTEX-INDUCED VIBRATION WITH HELICAL STRAKE OF SUPPRESSION DEVICE

ABSTRACT

Since offshore drilling rig is exploring oil and gas into depth water and marine riser facing the problem of vortex induced vibration (VIV) due to sea current. This VIV cause the marine riser to vibrate leads to failure. One of the way to reduce the vibration by attached helical strakes at marine riser to reduce the occurrence of vortex shedding. In this research report was studied about the numerical analysis of marine riser vortex-induced vibration with helical strake of suppression device. On this study, two types of models generated which are two dimensional and three dimensional. Each one has two models to be simulated which are laminar and Turbulence for two dimensional; with and without helical strake models for three dimensional. The Lift and Drag coefficient was simulated by analyzing the flow passes through a marine riser using ANSYS software. The results obtained from ANSYS are used as the input to MATLAB for produced results in a graph. The simulation was done for two models with and without helical strake. The model with helical strake was disturbed the vortex shedding at downstream, so the vibration of marine riser reduced. The results show the model with helical has decreased the Lift coefficient up to 96%.

Keywords: marine riser, vortex-induced vibration, helical strake, ANSYS software

ANALISIS BERANGKA MARIN RISER GETARAN VORTEKS YANG DISEBABKAN DENGAN HELIKS STRAKE PERANTI PENINDASAN

ABSTRAK

Kerana rig penggerudian luar pesisir sedang mengeksplorasi minyak dan gas di kedalaman air dan riser marin yang menghadapi masalah getaran teraruh vorteks akibat arus laut. Getaran teraruh vorteks ini menyebabkan riser marin gagal sebab getaran. Salah satu cara untuk mengurangkan getaran dengan heliks strakes yang dilekatkan pada riser marin untuk mengurangkan kejadian vortex shedding. Projek penyelidikan ini adalah kajian mengenai analisis berangka marin riser getaran vorteks yang disebabkan dengan heliks strake peranti penindasan. Pada kajian ini, dua jenis model yang dihasilkan adalah dua dimensi dan tiga dimensi. Masing-masing mempunyai dua model untuk simulasi yang lamina dan pergolakan untuk dua dimensi; dengan dan tanpa heliks strake model untuk tiga dimensi. Pekali Angkat dan Drag adalah simulasi dengan menganalisis aliran melalui satu marin riser menggunakan perisian ANSYS. Keputusan yang diperolehi daripada ANSYS akan digunakan sebagai masukan untuk MATLAB untuk keputusan yang dihasilkan dalam graf. Simulasi telah dilakukan untuk dua model dengan dan tanpa heliks strake. Model dengan heliks strake terganggu vorteks menumpahkan di hiliran, maka getaran marin riser dikurangkan. Keputusan menunjukkan model dengan heliks telah mengurangkan pekali Angkat sehingga 96%.

Kata Kunci: marin riser, getaran yang disebabkan oleh vortex, heliks strake, perisian ANSYS

ACKNOWLEDGEMENTS

My sincere gratitude goes to god for the success of this research. This research report represents a part of my master program studies. The work of this research report has been supervised by Prof. Madya Dr. Nik Nazri bin Nik Ghazali. Therefore, I thanks to Dr. Nik Nazri who provides great support, advices and helpful feedback during the entire research project. My deepest appreciation and would not successfully complete the research report without him.

I deeply appreciate the guidance of my project manager and discipline engineers. I wish to thank them for patient explanation of basic principles, for providing new ideas and for spending time to discuss my problems, it is really important for my study. Finally, a special thanks to my dear mother, my brother Punithan Naidu and especially my wife Thurga for the continuous motivation all the way.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF SYMBOLS AND ABBREVIATIONS

A	:	Reference area
CFD	:	Computational fluid dynamics
CPU	:	Central processing unit
C_D	:	Drag coefficient
C_L	:	Lift coefficient
D	:	Diameter
ϵ	:	Rate of dissipation of turbulence energy
F_2	:	Blending function
F_D	:	Drag force
G_ω	:	Generation of ω
G_k	:	Generation of turbulence kinetic energy
σ_k	:	Prandtl numbers for *k*
σ_ω	:	Prandtl numbers for ω
k	:	Turbulence kinetic energy
L	:	Body length
L'	:	Lift fluctuations r.m.s
ℓ	:	Cylinder total length
ℓ_c	:	Section of length
LES	:	Large eddy simulation
m/s	:	Meter per second
ρ_f	:	Fluid Density
PDE	:	Partial Differential Equations
PISO	:	Pressure-Implicit with Splitting of Operators

RANS	:	Reynolds-averaged Navier-Stokes
Re	:	Reynolds number
R.M.S	:	Root Mean Square
RNG	:	Renormalization-group
RSM	:	Reynolds stress model
St	:	Strouhal number
SST	:	Shear-Stress Transport
Γ_k	:	Effective diffusivity of k
Γ_ω	:	Effective diffusivity of ω
U	:	Stream speed
μ_t	:	Turbulent viscosity
VIV	:	Vortex-Induced Vibration
ν	:	Kinematic viscosity
a_1, α	:	Constants
ω	:	Specific rate of dissipation
Y_k	:	Dissipation of k
Y_ω	:	Dissipation of ω

CHAPTER 1: INTRODUCTION

1.1 Background of the study

In production of petroleum from offshore, marine riser is playing important role in transferring crude oil or gas (also known as hydrocarbon) from below seabed to top offshore platform, carrying mud and comprising drill strings in drilling operations as shown in figure 1.1 for marine riser and figure 1.2 for schematics of marine riser to floating rig. In addition, the processed fluids are taken to storage to stocking terminals or to pipelines leading to shore facilities by the export risers. The crucial of a right design of marine risers is approaching into sharper focus of the oil and gas corporations as they venture into deeper water. The costs of proper maintenance and proper intervention are also rising with slim line and depth intervention from simple support vessels would make good intervention economic in numerous instances where the hiring of a drill ship or a large rig may be economically prohibitive which leads the wells to produce less efficiently than required.

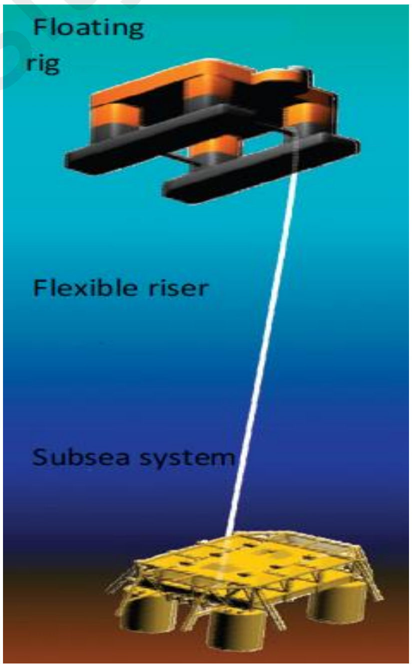

Figure 1.1: Marine riser (Ajayi et al., 2017)

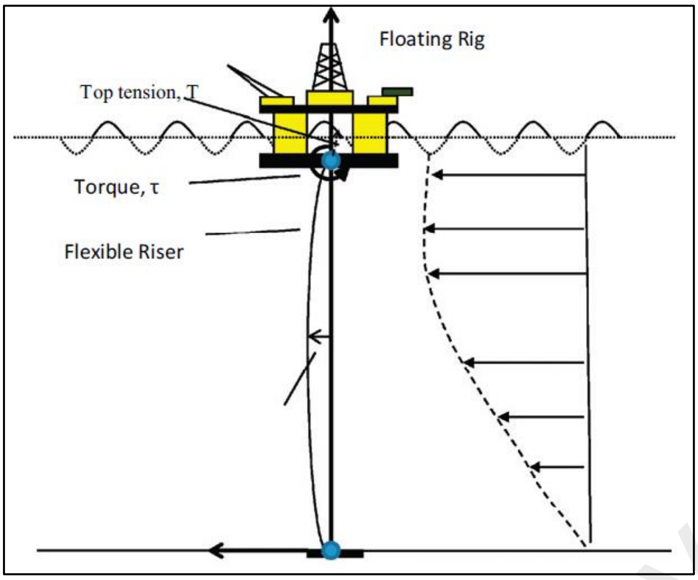

Figure 1.2: Schematic of Marine riser (How et al., 2009)

As the seawater current flows around the marine riser, pressure differences occur and vortices created at boundary layer leads to detach at the end of downstream of the marine riser as shown in Figure 1.3. This situation causes the structure to oscillate normally to the seawater flow due to flow instability happened at downstream and the phenomena known as Vortex-induced vibration (VIV). When the vibration of marine riser closer to the natural frequency of riser then the resonance takes place by increasing the oscillation amplitudes and creating phenomena known as lock-in.

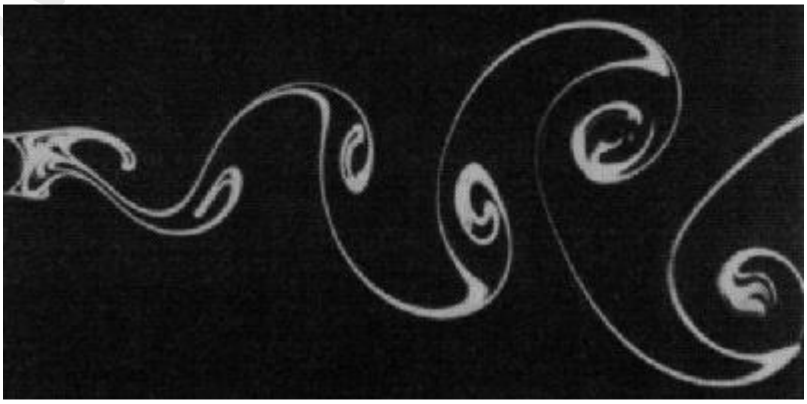

Figure 1.3: Von Karman Vortex Street (Lionel E. & Stephane P., 2002)

Vortex-induced vibration (VIV) is a critical reason for the major failure of the offshore marine riser as well as the thin structures, for example, bridge decks, towers and chimneys. For the offshore structures, especially risers are installed an array arrangement and there are high possibilities to hit one another when the vibration or oscillation started.

1.2 Problem statement

Risers are subjected to vortex-induced vibrations (VIV) when exposed to marine streams. The vortex shedding occurrence instigates in-line and cross-flow forces, prompting a vibration of the structure. The reaction of flexible structures is portrayed by multi-modular and multi-recurrence vibrations. Contingent upon the structural attributes and excitation streams, this vibration may cause exhaustion damage on the marine structure prompting failure at joint or surface by another structure when collision. We expect to advance the comprehension of the material science associated with vortex induced vibration (VIV) and apply this information to flexible structures such as marine risers. Along these lines, this research is on modelling of vortex induced vibration (VIV) to understand the effect of suppression device with and without. The comprehension of this stream structure is vital for future Research and Development in marine riser configuration in related fluid dynamics.

1.3 Objectives

The objectives of this research study are:

a) To analyse the vortex induced vibration passes through a marine riser.
b) To perform a simulation of the flow passes through a marine riser using ANSYS (FLUENT) Software package.
c) Understand the effectiveness of helical strake in suppressing the VIV of the marine riser.

1.4 Significant of the study

In the oil and gas industries, the performance of the marine riser is the most important thing that been focused. It included the delivery speed of crude oil, lifetime of marine riser, the capacity of delivery and many other things. This study is referring to the lifetime of marine riser into the deep sea. In order to improve this need, a suppressing device has been approached. This project is to study the vortex-induced vibration of the marine riser in order to reduce the vortex shedding. Hence, the lifetime of the marine riser can be increased due to the low vibration. So that, it can reduce the fatigue joint between the riser and other joints. Furthermore, can increase the capacity of delivery from sub-sea to the surface.

1.5 Scope of study

Theoretical analysis of the governing equation of vortex induced vibration (VIV), which the fundamental equation is Karman Vortex Street [4]. The study only incorporated of governing of vibration, which is the vortex flow, not included the vibration of the body. The study mainly concerns on 2 dimensions and 3 dimensions. The Geometry that using in this study are with and without suppressing device of the marine riser.

The software which is going to create and do meshing is Gambit. The vortex flow after hit the marine riser is simulating by using the ANSYS software (FLUENT). The simulation involved two different models such as laminar and turbulence. For both situations, the same Reynolds Number is applied to obtain different vortex flow. Meanwhile, the drag and lift coefficient can be obtained from the results.

CHAPTER 2: LITERATURE REVIEW

2.1 Vortex Induced Vibration

Vortex shedding is a phenomenon that leads to vortex-induced vibrations. Vortices can be created when a bluff body put inside a water stream. A boundary layer forms when the fluid flows around the bluff body. Contingent upon the stream attributes and geometry of the body, boundary layer separation happens. Because of the free stream due to separated boundary layer from outside edge and boundary layer from inside edge moves in higher speed, fluid rotates and a shear layer frames. Results of rotating fluid or stream, the separation from the body occurs and also known as vortices which are shed, in various shapes all through the wake. At the point when the non-symmetrical at bottom and top of vortices behind any bluff body and the subsequent pressure differential prompts lift force opposite to the course of the liquid stream. Change of lift force with time due to an intermittent arrangement of vortices cause the wavering movement of the bluff body. Conduct of vortex shedding governs by flow characteristic is called as Reynolds number. The Reynolds number is specifically identified with the stream speed (U), the body length of characteristic (L) the kinematic viscosity of the fluid (v) (Ball, I. et al.,2012).

$$\mathrm{Re} = \frac{U \times L}{v} \tag{2.1}$$

Bearman P.W. (2011), presents that when two number of cylinders in a circular shape are organized in such that their stream area able interfere to one another quantity of fascinating situation become visible. Bearman has focussed on two number of circular cylinders that positioned in line where the cylinder in rear is allowed to react the shedding extending across way. In respect to a VIV response, the downstream cylinder encounters a wake -induced vibration caused by associations between vortices shed from the two circular cylinders. It is suggested this is another sort of vortex -induced vibration.

Mukundan, H. et al. (2009) have conducted varies methodologies to estimate fatigue life of marine riser due to the vortex induced vibration by ocean current flow. They have reconstruct the marine riser and used the previous experiment data which is Norwegian Deepwater Programme (NDP). By using the reconstruction method, they have observed the fatigue life estimation least insufficient and influence of sensor availabilities and bandwidth. Also they have found that by using the Van der Pol oscillator model able to determine the rough estimation of fatigue life for marine riser.

Jing Xu et al. (2017), have perform the analysis to predict the fatigue life of marine drilling riser by using the numerical methods such as generalised integral transform technique (GITT) and nonlinear wake oscillator model. Based on this, they have proved the relation between outer diameter of riser and sea current which is riser diameter influence the vortex induced vibration and after extend certain limit of diameter the fatigue life of riser effected.

According to Liu Xiuquan et al. (2016), the deep-water drilling riser which connected to platform and subsea well head have the major fatigue damage especially at upper and lower part of riser due to wave induced fatigue and vortex induced fatigue. This statement was based mode on riser modes in such condition of installation, connection and hard hang-off modes.

Mao Liangjie et al. (2015) have found the distribution of vortex shedding along the marine riser is not even under shear flow. This experiment data was analysed by modal analysis method.

2.2 Suppression Devices of Vortex-induced vibration

Frequently, due to vortex induced vibration (VIV), the marine riser will neglect to face the weakness of design criteria. There are few factor designers may consider for such cases as below:

- Remodel the marine riser by altering the mass and density, increase the tension, or fundamentally altering the design of riser.
- To reduce the riser oscillating, add vortex induced vibration suppression device.

When discussing about the strakes, fairing or composite of the strakes and fairing may reduce the vortex induced vibration around the marine riser. In Mexico gulf region, the sea water current is high

VIV in steel riser systems is typically mitigated using strakes, fairings or a combination of both to the steel risers. In high current regions such as the Gulf of Mexico, which is most likely to harmed or influence by the high-velocity sea current. Fully submerged and seawater currents as gulf region have to install suppression devices which cauterised by two main devises as fairing and stakes. This two types of suppression devices (fairings and helical strakes) contributes very effective suppression in vortex induced vibration. The figure 2.1 and figure 2.2 are shows the two types of supressing devices.

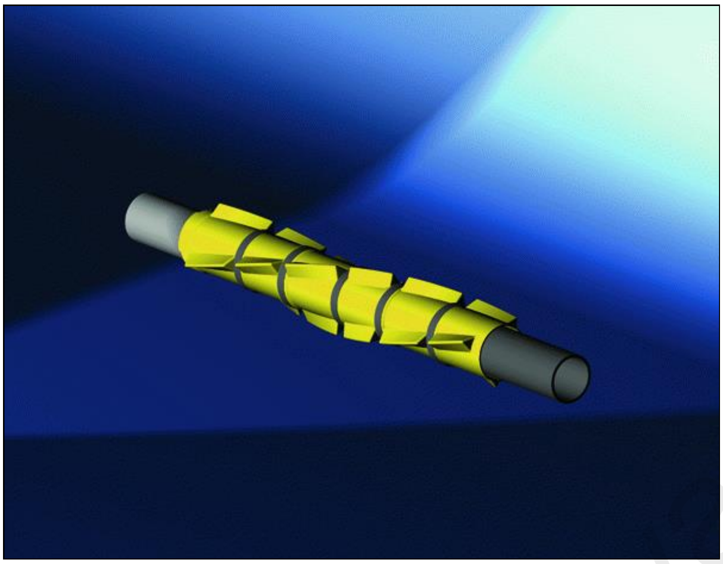

Figure 2.1: Helical stakes (Tiratsoo J.,2011)

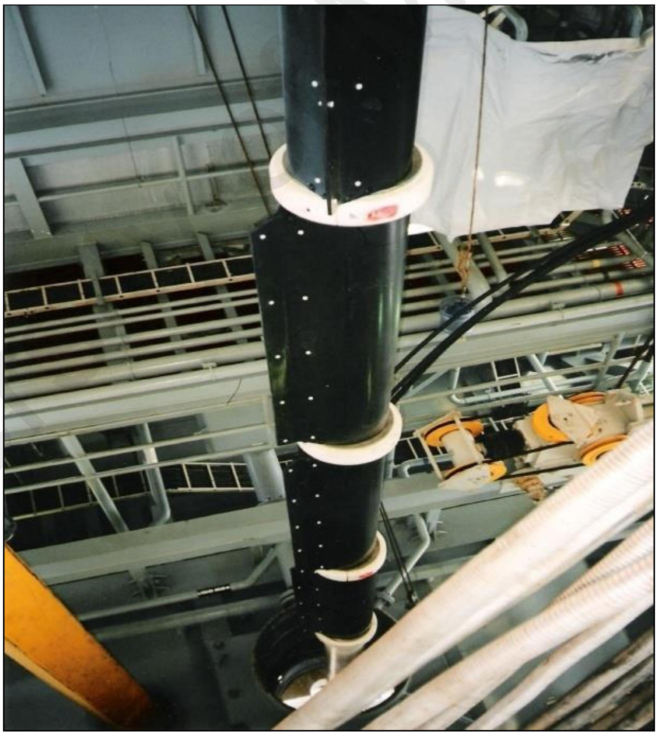

Figure 2.2: Fairing

The vortex phenomena which experience along marine riser height reduced by the Strakes suppress. Besides of strakes, the fairings diminish vortex occurrence as well as vortex shedding. 80 percentage of VIV can be reduced by implementing both strakes and fairings. Helical strakes are settled or fix and contain minimum significates issues than installed with fairings, however, have the additional weakness of expanding marine riser drag force. Compare to strakes, fairings give benefit in such way that minimum drag. In any case, rotation of fairings is required along the sea water current direction. The outline multifaceted nature of fairings may restrict their utilization particularly when fairing to be installed over a substantial segment of the whole length of riser.

The requirement to utilize suppression system over large portions of the riser length is very costly, in a manner of cost of installation and capital. Viability of the suppression device on the object is governs by outstanding curvature and different edges of tendency to the stream.

A big scale of experiments and testing was conducted to study the effectiveness and characters of the suppression device on marine risers. The mention experiments was consist of man-made equipment’s such as towing tank is a testing facility of hydrodynamic at indoor and outdoor water scale testing. By using the indoor towing water tank, it made possible to identify the geometry of suppression strake, coverage of riser, flow in cross direction and in-line direction. Furthermore, small scales of testing or experiments able to leads to significant benefits which produce a results that helpful for a bigger and detailed testing with sophisticated equipment’s (Zhang H. & Lim F., 2010).

Based on Min Lou, Wu-gang Wu and Peng Chen (2017), fairing type suppression devise reduce the vortex shedding occurring at upstream. They have conducted the experiments at hydrodynamic lab and obtained the data’s. Furthermore, they have analysis the effectiveness of fairing in tandem and side by side condition as well as different fairing caudal horns angle. Fairing with 45 degree and 60 degree of caudal horns angle effectively reduce the VIV regardless the riser configuration.

According to Paolo Simantiras and Neil Willis (1999), suppressing device that effectively works for the vortex-induced vibration case in first mode was helical strakes. The helical strakes proved even in the higher degree of slope in angles. That particular part of helical strakes contributes excellent performance in dispersing force on the experiment pipes when exposed to different angle of position regardless of properties, weight and flow direction. It is purely depend on the helical shape (Simantiras P. and Willis N., 1999).

According to Baarholm G.S. et al (2005), straked riser takes control of the VIV behavior. And from the model tests indicate a different physical behavior depending on the pitch and height of the strake triple start. The behavior seems to be strongly dependent on the pitch and height of the strakes (Liea, H.et al., 2005).

Kumar N. et al., (2018) have conducted experiment of circular cylinder as riser mounted with substructure known as Ventilated Net (VN) inside towing tank as well as at high Reynolds number (0.22-2.50) x 10^5. The result outcome from the experiment have shown the cylinder with substructure reduce the vortex induced vibration by 98 percentage and drag force by 40 percentage at Reynold number 1.2 x $10^{5.}$

2.3 Fluid Flow around a cylinder

The physical mechanism like laminar, transition, and the turbulent flow are fascinating issues in fluid mechanics. Due to the issues in fluid mechanics as mention before, creates two types of flow which are unsteady and steady vortices close to a body. Succeeding is molecular dispersion, follows by convection of kinematic and dynamics also dispersal of sea current at downstream. The results of distribution creates the pressure and velocity difference close to the bluff body. It can be understand the unpredictability of the physical mechanism issues by testing the behaviour of a true stream flows around the circular cylinder. Reynolds number influences the Karman vortex street and the shedding process. The separate phase of the process for increasing Reynolds number is shown below.

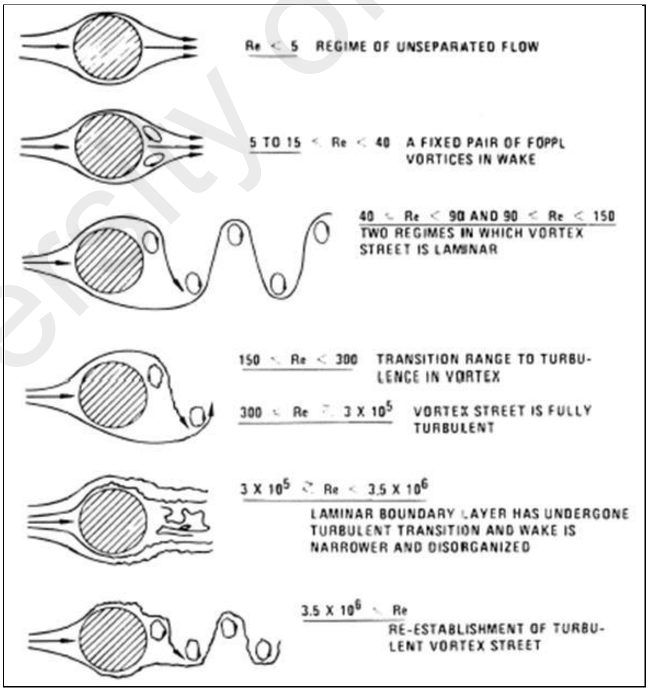

Figure 2.3: Flow regimes and corresponding wake at increasing Reynolds number (Blevins, R. D.,1990)

From Figure 2.3, we observed that flow separation happens early. Two symmetric eddies are formed at the downstream of the flow separation which called as boundary layer due to Reynolds number about 5 to 40. At Reynolds number of 40, the steady and symmetrical of eddies stay same but size changes according to Reynolds number. For Reynolds number further increasing 40 to 150, unstable wake and laminar staggered vortices are created at opposite signs. The turbulent vortices happens between ranges 150 to 300 of Reynolds number, while the laminar boundary layer remains on the cylinder. The subcritical of flow regime in range of 300 to $3x10^5$ is reach when the flow velocity further increased. The cyclic forces are created in x and y-direction when flow separation is near to 80-degree cylinder driving edge with periodic and strong vortex shedding. The Reynolds number between ranges $3x10^5$ to $3.5x10^6$ is transitional flow and the turbulent boundary layer which gradually slows the separation point approximately 140 degrees of the leading edge. As a result, the drag force is decreased. When the Reynolds number crossing the value of $3.5x10^6$, boundary layer and vortex street of turbulent is established with regular vortex shedding.

Carmo B.S. et al. (2011) presents a numerical simulation in two and three-dimensional of flow around two circular objects in tandem arrangements. In two dimensional case the Reynolds number is maintained 150 and for three-dimensional case is 300 and the velocity is varies based on structural stiffness. The simulation results show the cylinder flows changes when compares between flows around a cylinder in tandem arrangements to isolated from each other.

Mittaland and Kumar (2001) explore regarding the vortex-induced vibrations of the light circular cylinder set at Reynolds number with the range of $10^3 – 10^4$ in uniform flow. Various values of results are exhibited for a stationary cylinder on the vortex-shedding frequency. The wakes are shows organized when fluid flow at lower Reynolds numbers and unorganized wake occurs at higher Reynolds numbers. In some cases, various values of vortex shedding are observed. This article explores the Reynolds number which governs the fluid-structure interaction.

2.4 Steady and unsteady flow

In the case of subsidiaries stream of area disappear, that stream are taken into account for the gradual stream. Steady conditional stream alludes to the state wherever that liquid characteristic at some extent inside the framework don't alter extra duration. One thing else, stream is named unsteady. a selected stream is steady or unsteady, will depend upon the chosen definition of reference. For incidence, streamline flow over a circle is steady inside the definition of reference that is stationary with reference to the circle. During define of point which the motionless reference for a foundation stream, the stream is insecure. Turbulence streams is known as unsteady flow by its character. The turbulent stream will be that because it could, be factually stationary.

These concept implies for all the applied mathematics characters are remain same in all time. Normally, a mean area is that interest of the object and it can be persist about an exceedingly motionless of stream. Compare with steady flow over unsteady flow, steady flow normally easy to control compare to the unsteady flow. The steady flow governing equation have drawback that one dimensional time than the similar governing equation drawback while not exploiting the strength of the stream field. The figure 2.4 shows the Rayleigh–Taylor instability for simulation of hydrodynamics.

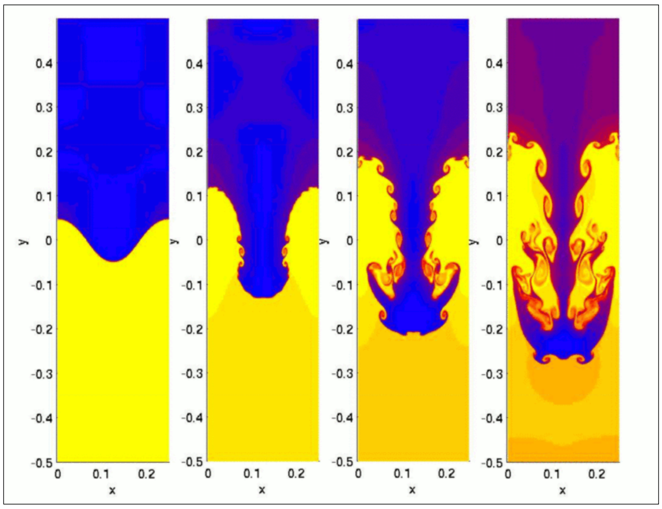

Figure 2.4: Hydrodynamics simulation of the Rayleigh–Taylor instability.

(Li, Shengtai & Hui Li, 2006)

Keramati Farhoud et al. (2012) studied about fluid flow at Reynolds number of 150 flows around a circular cylinder in unsteady and incompressible state at different alternative vortex times. To study and observe the flow passes around the circular cylinder were utilize by two equations such as continuity and Navier –strokes equation with the used of finite volume methodology. They have compared the results with different numerical and testing method outcomes which hav2.5e the reliable results. The outcome of testing have shown the fluid stream in entry region boundary become fully unsteady when passing around the cylinder and flow parameters in zero and one hundred and eighty degrees have fewer changes in vortex times. Conjointly it had been shows that the lift coefficient is more than drag coefficient by oscillation of amplitude. In other hand, the drag coefficient is as double as that of lift coefficient by frequency of the oscillation

2.5 Vortex Shedding

Vortex shedding is a common phenomenon in fluid dynamics. A fluid such as water or air passes around a bluff body creates the unsteady flow at certain velocities, but depend on dimension and body form. When fluid flow passes around bluff body, the vortices occur at behind of the body and separation occur around the body. Because of this fluid flow separation, low pressure region occur around the circular cylinder and the cylinder was weak in pressure and pushes toward the low pressure region.

When the bluff body is not fixed rigidly at both end, the frequency of resonance will take place and matches the vortex shedding of the structure. The body will gradually oscillate and the cylinder moves due to harmonic oscillations which generated by fluid flow energy. The same vibration phenomena can be seems in daily human life. Example the overhead electrical power cable moves with sound in a strong wind condition and radio antennas at vehicles flapping at high speed when the vehicles is in moving situation. The most common issue that faced by industries is tall chimneys. These tall chimneys will encounter vary of wind speed that will impact the flow around the chimney. These flow separation and vortex shedding created around chimneys and oscillates at very dangerous condition which harmful to the chimney base and tie-in components. The movement of the chimney due to vortex shedding can be prevented by installation of suppression device or shield at higher point or 20 percentage of overall length. The suppression device that commonly installed is helical strakes. This additional shield work well at vortex shedding situation and reduce the frequencies of separation. The ideal of the pitch for the design vortex shedding are 5D pitch where‘d’ stand for diameter of stack. In the year 1940, the incident Tacoma Narrows Bridge which collapse cause of Vortex shedding and these reason contributes the main effect on these incidents.

Zdravkovich , M.M. (1996), conducted experiment related to the modes of vortex shedding at different flow condition such as laminar and transition, low and high speed range of fluid flow. Study the oscillation of cylinder in full submerged stream and the effects. It shows that the oscillation amplitude of cylinder is caused by sudden changes in 180 degrees of phase in vertex shedding.

2.6 Phenomena of induced vibration

The phenomena possible to happen at tall chimneys, submarine periscopes, and electrical suspension lines and atomic fuel bars are found to vibrate fiercely under some state of fluid flow around them. So also, water and oil pipelines and tubes in air compressors experience bad vibrations under some state of fluid flow through them.

In ice-covered electrical transmission lines and the unbalance vibration, called as flutter, of airfoil sections. Singing of transmission line, also known as high-frequency vibration where the vortex shedding phenomena happen. Vortex shedding was the first reason behind the failure of the Tacoma Narrows suspension bridge within the state of Washington in 1940 as appeared in figure 2.5 and 2.6.

Figure 2.5: The original Tacoma Narrows Bridge roadway twisted and vibrated violently

Figure 2.6: The 1940 Tacoma Narrows Bridge collapse.

On the off chance that the frequency of the vortex shedding is in resonance with the natural frequency of the part that produces it, also developing a large stress product of vibration with large amplitudes. Reynolds number with the range of 60 to 5000 are found to have on some experiments that normal vortex shedding ability to happens strongly. The terms of a function of the Reynolds number and Strouhal number (St) are used as the frequency of shedding based on some experimental results obtained.

2.7 Modelling Turbulence

The fluctuating velocity fields are described as Turbulent flows. Mix transportation of fluctuations contents like energy, species concentration and momentum, also leads to fluctuation because of transported quantities. Because of the oscillation may be in small or big range of frequency, seems to be difficult to do simulation in low cost as well as in numerical analysis. To reduce the computational cost to solve by modify the set of an equation such as governing equations of ensemble-averaged, time-averaged and also remove the small scales by manipulation. Certain unknown values occur as results of equation modification and to determine the values in known quantities required to use turbulence models as a solution.

FLUENT provides the following choices of turbulence models:

- Spalart-Allmaras model
- k-ϵ models
  - ✓ Standard k-ϵ model
  - ✓ Renormalization-group (RNG) k-ϵ model
  - ✓ Realizable k-ϵ model
- k-ω models
  - ✓ Standard k-ω model
  - ✓ Shear-stress transport (SST) k-ω model
- Reynolds stress model (RSM)
- Large eddy simulation (LES) model

2.7.1 Turbulence Model

Turbulence flow is the common issues for the all the categories and there is no even one demonstration. For the choice of turbulence demonstration are consider few aspects as discussed below. Science enveloped within the stream is one of the consideration. Also, lesson for particular issues are setup hone and required precision of higher level. Function of computer for operation and for simulation the total duration also one of the consideration. To choose the most optimize demonstration for implementation, the restriction and abilities of the different choices have to obtain. The reason of these part are to deliver model of turbulent in form of issue diagram which is relevant in ANSYS Fluent. Examination of specific model memory and computer performance input as well as time consumption. Meanwhile, there are few difficulties that experience to identify the classes which is suitable for the turbulence analysis and guideline for the selection of the optimize turbulence demonstration.

2.7.1.1 The Standard k-$\boldsymbol{\omega}$ Model

In FLUENT, model of common k-$\boldsymbol{\omega}$ are depends on model of Wilcox k-$\boldsymbol{\omega}$ which consolidates alterations for impact of Reynolds-number with low value and spreading of a shear stream. In situation of assumption for expanding rates of shear stream such as estimation of close agreement, film combination and far wakes are using Wilcox model and also available for flow of free shear and wall bounded.

2.7.1.2 The Shear-Stress Transport (SST) k-$\boldsymbol{\omega}$ Model

The aim of purpose for successfully mix the vigorous and accurate definition for model of k-ω demonstrate within the region wall close to the shear-stress transport (SST) k-ω model was created with the free-stream independence of the k-ϵ model in the far field. The k-ω formulation is changed from k-ϵ model to achieve above matter. The k-ω mode for SST shows the comparative k-ω model standard show, unlikely incorporates to below the following clarification:

- The both standard k-ω model and the transformed k-ϵ model are added together and multiplied by a blending function. The design purpose of the blending function is to be near wall region, which activates model of standard k-ω and away from surface at zero as well as transformed k-ϵ model.
- The SST model incorporates a damped cross-diffusion derivative term in the ω equation.
- Turbulence viscosity definition is changes by the transportation of the turbulent shear stress.
- The difference in modelling constants

All the above features make the Shear-Stress Transport k-$\boldsymbol{\omega}$ model more precise and valid for the fluid cases than compare to the model of standard k- ω.

2.7.1.3 Transport Equations for the SST k-ω Model

The similar form of function for the SST k-ω model and standard k-ω model:

k- equation
$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right)+G_k-Y_k+S_k \tag{2.2}$$

and,

ω- equation
$$\frac{\partial}{\partial t}(\rho \omega)+\frac{\partial}{\partial x_i}(\rho \omega u_i)=\frac{\partial}{\partial x_j}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right)+G_\omega-Y_\omega+D_\omega+S_\omega \tag{2.3}$$

Where, G_ω represents the generation of ω of, G_k presents the generation of turbulence kinetic energy due to mean velocity gradients. Γ_k for effective diffusivity of k and Γ_ω for effective diffusivity of ω. Because of turbulence Y_k and Y_ω represent the dissipation of k and ω. D_ω represents the cross-diffusion term. S_k and S_ω are user-defined source terms.

The Shear-Stress Transport k-$\boldsymbol{\omega}$ model of the effective diffusivities shows as below:

Effective diffusivities of k

$$\Gamma_k = \mu + \frac{\mu_t}{\sigma_k} \tag{2.4}$$

Effective diffusivities of ω

$$\Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega} \tag{2.5}$$

Where σ_k and σ_ω are the turbulent Prandtl numbers for *k* and ω, respectively.

The turbulent viscosity μ_t is computed as follows:

$$\mu_t = \frac{\rho k}{\omega}\frac{1}{\max\left[\frac{1}{\alpha^*}, \frac{\Omega F_2}{a_1 \omega}\right]} \tag{2.6}$$

2.8 Computational Fluid Dynamics

The numerical estimation to the mathematical solution models of the liquid stream and heat transfer. One of the tool or software to solve the dynamic of fluid issues are thru utilizing Computational fluid dynamics (CFD). Numerous areas like physics, meteorology, geology, and engineering are approach these Computational fluid dynamics (CFD) due to the boundary condition and stream strategies. The solution of the coupled algebraic equations, modelling, and visualization, grid work, coordinate transformation and finally discretization are significant components in computational fluid dynamics.

Partial differential equations is a numerical solution which required continual equation of nature in a very distinct type. There are few part consist in equation of discretization and the main part known as grid generation or subdivided the cells. Also the equation represents in separate types at finite volume, finite element methods and finite difference for each point. For more details of the types of finite can be expressed as structured grid arrangement are required for finite difference method and for finite elements able to use structures grids as well as unstructured grids. It is more flexible than finite different. The finite volume method is also similar as the finite element method.

There are a many different methodologies for settling the fluid turbulence phenomena. By utilizing the fluctuating parts and decomposing the velocity into mean the Reynolds-averaged Navier-Stokes (RANS) equations are defined. The equation that coincidence with the sub grid turbulence model is the large eddy simulation (LES) that is one of obtain and tackles the Navier-Stokes. Solving the Navier-Stokes equations by using direct numerical simulation which solves turbulent flows in many cases and rectify all the measurement length in the turbulence flow by made the meshing into high quality condition. But the drawback of the direct numerical simulation is for low-Reynolds-number flows and restricted to basic geometries due to the constrained limit of even the most advanced supercomputers.

The ultimate step is to imagine the outcomes from the simulation. Very high graphical station and visualization software allow for produces of velocity and pressure contours, the vector of velocity, streamline, vorticity and movement in animation form of unsteady calculations. In spite of the modern equipment’s or hardware’s, it is troublesome to three-dimensional visualization and for unsteady flows. Besides, numerous modern visualization methods tend to be subjective, and the foremost important visualization regularly comprises of basic x-y plots comparing the numerical solution to the hypothesis or test information.

The application of Computational fluid dynamics (CFD) is wider in many areas such as hydraulics issues, aerodynamics, environmental fluid dynamics, atmospheric and oceanic dynamics. Also, the CFD useful in changing of millimetre with seconds to kilometre with year in terms of length and time scales respectively. Automobile optimal design related to hydrodynamics that analysis more on the driving force within the enhancement of computational fluid dynamics and this CFD also widely used in stream analysis involving ships, airships as well as vehicles.

2.9 ANSYS (FLUENT)

ANSYS is the world's main provider in service of software system simulation. There is so numerous computer program that used by ANSYS. FLUENT (Computational Fluid System software) is one of the software. FLUENT could be a progressive computer program for modelling fluid flow and heat transfer in advanced geometries. FLUENT provides complete mesh flexibility, determination issues with unstructured meshes that able to generated by advanced geometries with relative ease. Mesh is classified as the 2D triangular/quadrilateral, 3D tetrahedral/ hexahedral/ pyramid/ wedge, and mixed meshes. Own solution is available in FLUENT. The C computer language is used to program FLUENT and utilize the capabilities and abilities of C computer language. All the dynamic memory distribution, productive information structures, and adaptable solver control made the program possible. FLUENT also able to operate as the separate concurrent process for proficient application, inherent control, and adjusted to new conditions of the machine or servicing framework compose. Furthermore, all the abilities needs to identify the answer and produce the results in FLUENT. The collected dialect known as scheme is the user interface, a vernacular of LISP. The interface can be revise and enhance by the propelled owner thru composing menu macros and capacities

2.10 Drag and lift coefficient

The fluid flow able to separates into lift and drag forces after the forces acting on the circular cylinder. The forces in steady flow condition highly rely on fluid density, velocity of fluid and reference area of bluff body. The life force and drag forces are depends on drag coefficient C_D and lift coefficient C_L respectively. Force coefficient that fluctuation because of periodicity of vortex shedding and results to drag and lift coefficient. It is easier by counting the root mean square (r.m.s) for any types of flows in fluctuation condition.

In real flow situation, the drag force is not zero value but in ideal flow case, the drag force is become zero due to no pressure and shear forces. Equation below shows the drag force formula on a fixed cylinder or any other shape. The drag coefficient C_D is measured in different shapes and Reynolds numbers.

$$F_D = C_D \frac{1}{2} \rho_f U^2 A \tag{2.7}$$

The above drag force, F_D formula, consist of the drag coefficient, density of fluid ρ_f, velocity of the flow U and area of references, A. In the case of smooth fixed cylinder has drag coefficient in laminar flow boundary layer are reasonably persist in the range of $10^4 < Re < 5 \times 10^5$.

The results are no longer convincing in the event of vortex induced vibration and required to obtain a new demonstration. In VIV flow, few sorts of inquire about have attempted to discover a calculation demonstration on the amplification of drag factor. The below equation is express the multiplication of drag amplification factor with the local drag coefficient for fixed body. The ratio of A/D can be large as 2.31 when ratio A/D equal 1.

$$C_{D,amp} = 1 + 1.043\left(\frac{2y_{rms}}{D}\right)^{0.65} \tag{2.8}$$

The lift coefficient of r.m.s is describe as below:

$$C_L' = \frac{2L'}{\rho U^2 D \ell_C} \tag{2.9}$$

Where L' is define as the lift fluctuations r.m.s that acting on a span wise section of length ℓ_c. The $\ell_c = \ell$ is expressed as total fluctuation of lift and the ℓ is defined as cylinder total length.

CHAPTER 3: METHODOLOGY

3.1 Introduction

In this study, there are few kinds of methods that been applied to reach the desire of these research. The first method is doing the analysis of the parameter study on the vortex-induced vibration of the marine riser. This step can enhance the progress of the study. For other method is using the GAMBIT software for modelling and meshing. The solver was done by using the ANSYS FLUENT to perform the simulation of vortex induced vibration. For the post-processing method, the MATLAB is using to plot the graph as a result of this simulation.

3.2 Flowchart

The chronology of design, relevant calculation, sampling, and analysis are as detailed in Figure 3.1.

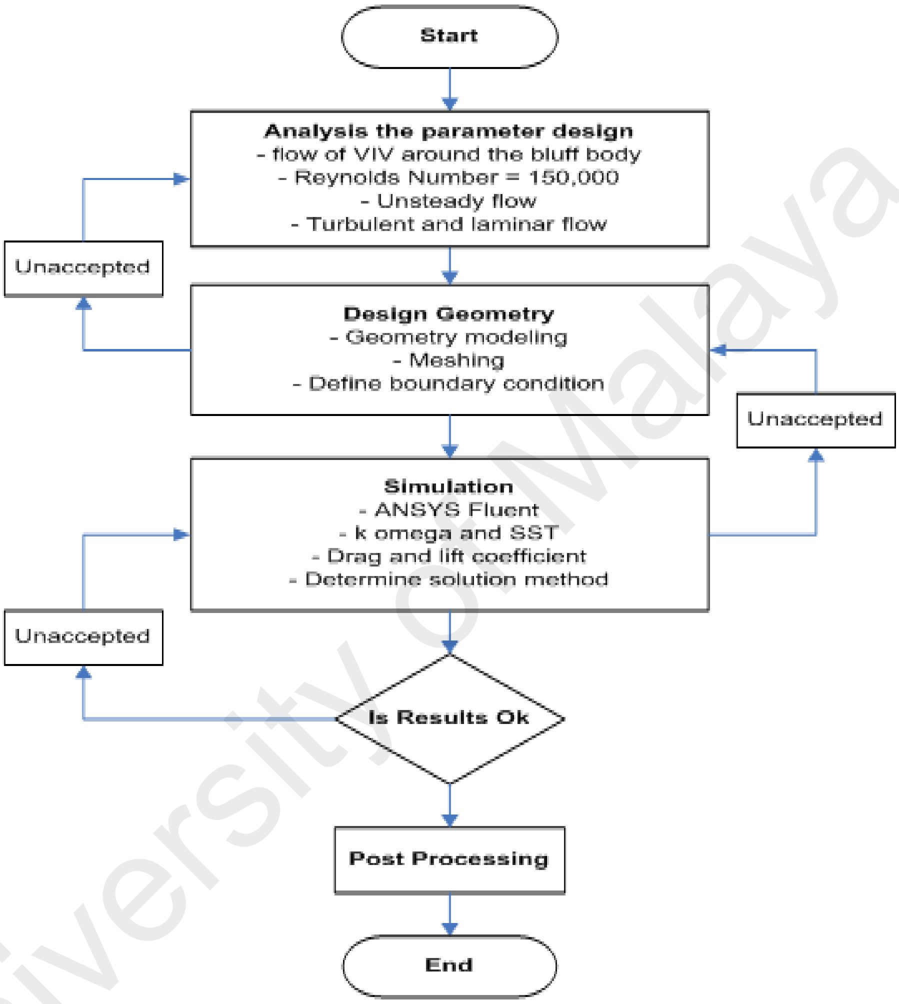

Figure 3.1: Research Methodology Flowchart

3.3 Computational Details

This research model configures the simulation by numerical methods for analyse the circular cylinder or marine riser for study the connection that associate with laminar or turbulent flow and component of marine. The marine part was configures into vertical position with particular diameter scale in the computational domain for obtain the result of the downstream after the interaction. There are 2-dimension and 3- dimensions to be analysed. The two dimensions consist of laminar and turbulent flow. On another hand, three dimensions consist of a marine riser with and without helical strake. For the viscosity section, the laminar and SST k-omega model will consider to analyse the difference and effects of the fluid flow passes a bluff body also known as cylinder for our study with 150,000 value of Reynolds number.

In the current work, viscosity models such as the laminar model and the k-omega model have been chosen to test the suitability and the applicability of the models on the flow past a circular cylinder for Reynolds number of 150,000. Testing data are utilize to compare the similarities and dissimilarities of the analysis between drag coefficient, life coefficient and Strouhal number. The brief details of the simulations are as follows:

3.4 Set-up Numerical Simulation

Yogini Pater (2010), has describe about the numerical simulation. This type of simulation required assist of computational fluid dynamics which are different from physical testing and theoretical methods. Computation fluid dynamic method consist of algorithms to settle the equations and numerical way with assist of computer. Pre-processing, the solver and post-processing are the main three concept in the computation fluid dynamic (CFD).

3.5 Pre-processing

In the pre-processing section, CFD programs received inputs of flows details acting as operator-friendly interface and upcoming transformation of the inputs into readable inputs by solver. Computational domain is known as fluid region where to analysed and the domain creates the individual numbers of elements that is known as grid or mesh in better words. The fluid properties have to define after the mesh or grid creation and the boundary condition of the domain specify appropriately.

3.5.1 Building a Model

3.5.1.1 Two Dimensional

The geometries were carried out in Gambit. Two geometries were created for three dimensional and one geometry created for one dimensional. Figure 3.2 shows the computational domain in circular shape and circular cylinder at centre in two dimension. The diameter (D) of the circular cylinder for these simulation is 1 meter. This simulation consider the domain size is 60 times bigger than the cylinder size at downstream and upstream.

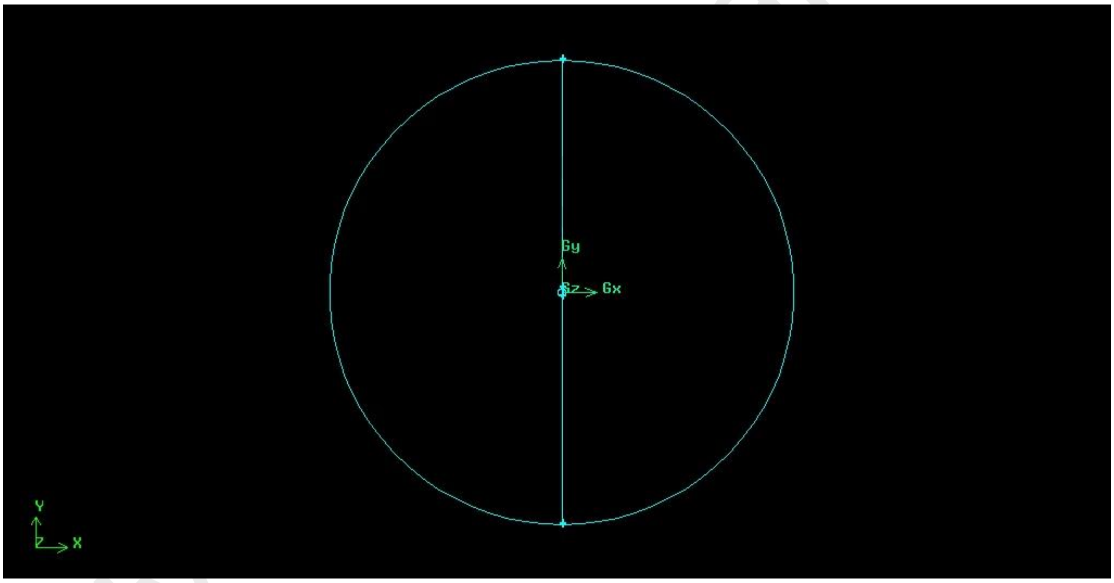

Figure 3.2: circle computational domain for 2D model

3.5.1.2 Three Dimensions

Two types of the model created which are with and without helical strake for three-dimensional models. Figure 3.3 shows the computational domain in rectangular shape and in two dimension model. The diameter (D) of the circular cylinder for these simulation is 1 meter and the length of the cylinder consider 10 times more than cylinder to obtain reasonable span wise results. For these computational domain, the downstream is 15 times more than cylinder diameter size as well as upstream is 5 times more.

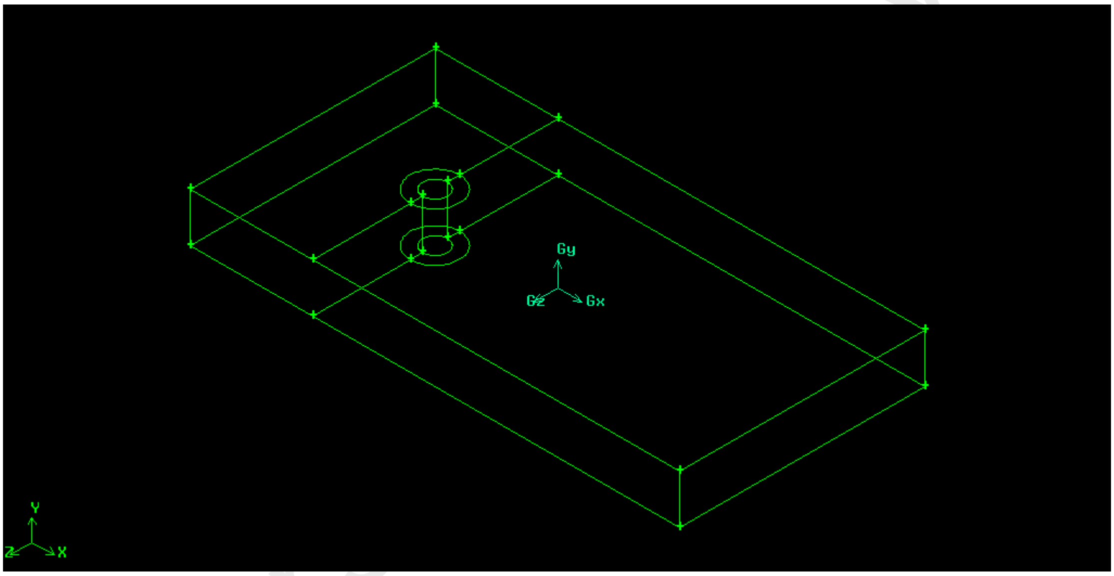

Figure 3.3: Riser without helical strake for 3D model

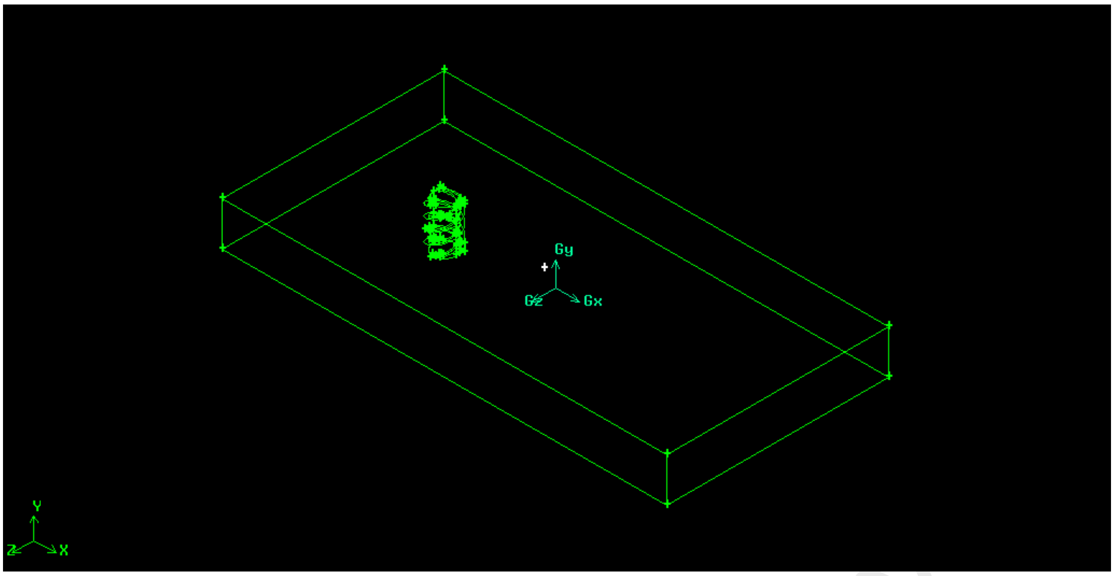

Figure 3.4: Riser with helical strake for 3D model

3.5.2 Meshing

Meshing is implemented by using GAMBIT 6.0. A structured hexahedral mesh is generated in the 2D model and unstructured tetrahedral mesh generated in the 3D model for the simulation. Type of grid structures occurred surround the circular cylinder is hexahedral cells. Figure 3.5 shows the structured mesh generation in the computational domain. In order to improve the quality of results on the wall of the fluid, in contact with the cylinder, had a mapped face to ensure that readings could be taken from 360° around the cylinder. The reason behind the low mesh quality on the domain is the larger cell numbers that taken due to unstructured meshing. Figure 3.6 and 3.7 are shows the unstructured mesh of riser for three dimensional.

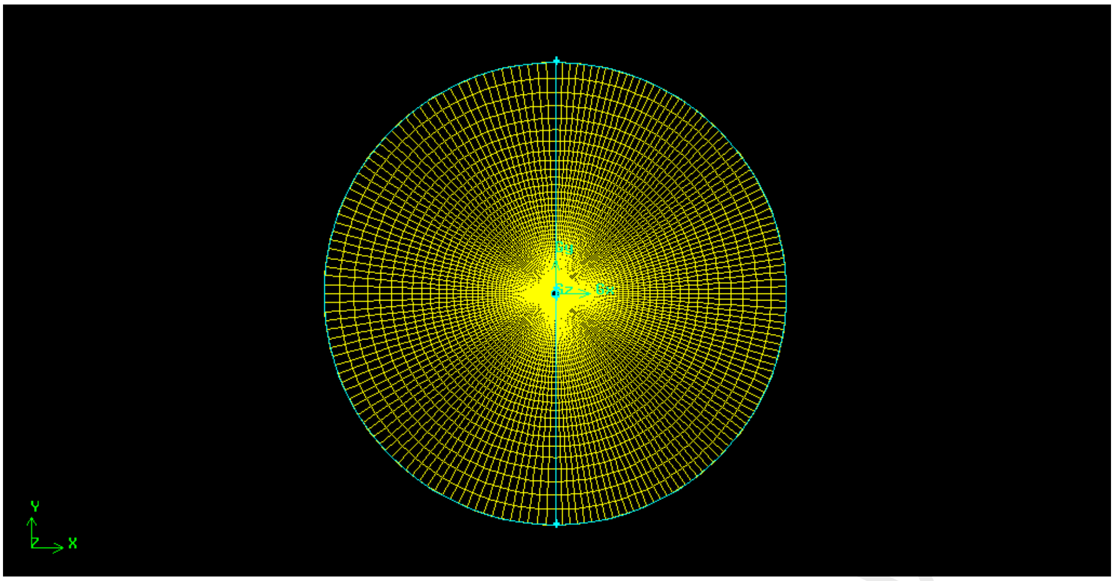

Figure 3.5: 2D Model Meshed using Hex-Sub Map

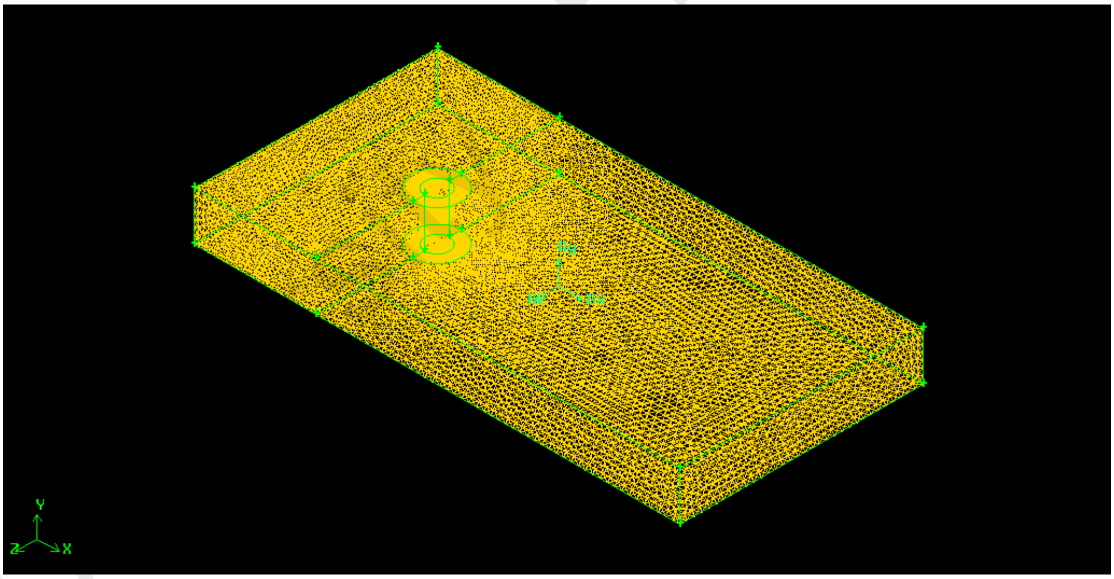

Figure 3.6: Riser without helical strake meshed using Tetrahedral-TGrid

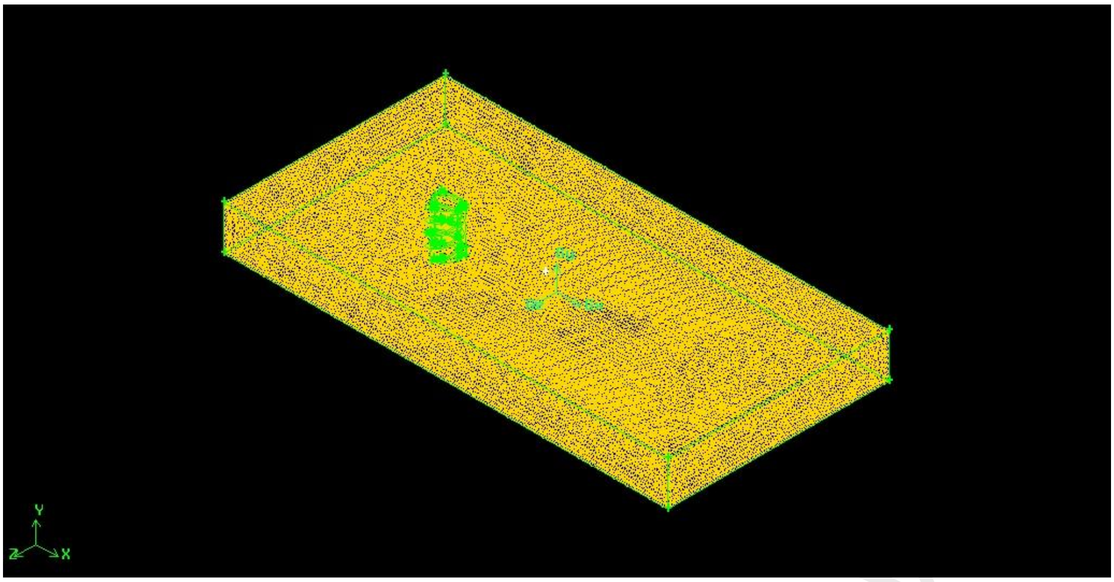

Figure 3.7: Riser with helical strake meshed using Tetrahedral-TGrid

3.5.3 Boundary Type

The boundary types are defined for each model for the analysis purpose. Boundary conditions provide FLUENT with information on flow or thermal conditions at the boundaries of the physical model. For the two dimensional model, the boundary types specified as velocity inlet, outflow, and cylinder as well with no slip. For the three dimensional model, all the boundary types are same but add one more boundary need to add which is symmetry. Because in three dimensional it has top, bottom and two sides.

3.6 Solver

The function of solver is to calculates and solving the governing equation of the CFD problems. Partial Differential Equations (PDE) is the governing equation for fluid motion. Flow variables and variables derivatives are the two things that made Partial Differential Equations (PDE). There are few limitation to PDEs to generate directly the outcome. Therefore, process called numerical discretization is use to transfer PDEs into equation of algebraic. On these process, there are four types of method were available such as

Finite element method, Finite difference method, Finite volume method and the last is Spectral method. For finite element method, equation was generates for all other element by each independent elements. Finite volume method generates solution to numerical equations for point of given according to neighbouring points values. Finite difference method is the same principle as finite volume method. On these research, finite volume method was used in FLUENT.

3.6.1 Pressure based solver

There are two types solver in Fluent such as density-based and pressure-based. Both solver function in different ways. For this research with 2 dimensional and 3 dimensional, we have chosen pressure based as solver. The projection method is important role in pressure-based solver in which utilize the algorithm that is belongs to projection method. Pressure equation is utilize to rectify the constraints of mass conservation (continuity) of the velocity field in projection method. The derivation of the pressure equation are from momentum and continuity equations in way that satisfying the continuity through corrected the velocity field by the pressure. The entire solution will involves iteration and solved repeatedly until solution converged due to the nonlinear and coupled to one another of governing equations.

3.6.2 Viscous model

In the viscous model, there are few models are available to perform a simulation. For this simulation, laminar and k-omega models chosen to perform simulation at Reynolds number at 150,000. Laminar model is employed to one of the two-dimensional models. The shear-stress transport (SST) is chosen with k-omega model. Standard k-omega model is not precise and reliable as SST k-omega model. Cross diffusion term was added in the ω equation as modification and by doing this, the model equation will function properly in zones of far field and near wall.

3.6.3 Spatial Discretization Scheme

For the purpose of convection for each governing equation, the scheme of discretization was employed in Fluent. For the laminar and turbulent models, the pressure implicit with the splitting of the operator (PISO) was chosen. In case of transient stream calculation, it is highly recommended neighbour correction with PISO algorithm. It is useful for larger time step used. PISO also suitable because it able to calculate the larger step time in stable condition and for both pressure and momentum has an under-relaxation factor of 1. Furthermore, for steady state and transient calculation are suggested to use PISO with skewness correction on meshing with high distortion degree

3.6.4 Boundary condition

In the simulation sequence, one boundary is configure as velocity flow which is at upstream of the computational domain. For two dimensional and three-dimensional model, the velocity is used was 1 meter per second (1m/s) and initial gauge pressure is 0 pascal. The boundary condition for opposite face of domain boundary is set as the flow outflow to allow the fluid flow freely. Meanwhile, the solid surface of the marine riser or circular cylinder is set as wall with no-slip boundary conditions. The reason of using wall boundary conditions is to make the bound between solid and fluid area. In viscous flows, the no-slip boundary condition is enforced at walls by default. The top, bottom and two sides are set to symmetry conditions and this is only for three-dimensional models because two dimensional don't have this kind of boundaries. Setting of symmetry for avoid velocity of fluid to reduce, to not cross the boundary and also act as mirror for boundary.

3.7 Post-Processor

In this stage, results which obtain from the simulation is to be developed in order to view the effect of the fluid flow around the cylinder or marine riser. By using the post processing method, provide us to see the contour plots of velocity, kinetic energy, pressure and other flow properties.

3.7.1 Results (Graphics)

The last step in this methodology is results of ANSYSY Fluent analysis and includes the association and clarification of the simulated flow data for the vortex induced vibration. The data is validated by applying Grid Independence Study. The design geometry was analysed and chosen from the dimension of the models that chosen based on vorticity occurs.

3.7.2 Graph

For this section, MATLAB R2012a software has been used to generate lift coefficient and drag coefficient graph to analysis the Vortex-induced vibration (VIV) of the seawater past thought cylinder and marine riser. For the four types of simulation had different types of graph occurs because of vary with the viscous model as well as dimensional differences.

3.8 Summary of Methodology

The study of vortex induced vibration flow past a marine riser should determine carefully the parameters and factor in order to obtain the reliable as well as optimise results. By using general literature, the quality of the meshing model, Reynolds number and boundary conditions are playing an important role in increasing the performance of the simulation. Basically, as mentioned earlier in the design process, drag and lift coefficient are the main part for this study, hence some data and information about the vortex induced vibration and marine riser are collected through the internet, journals, books, and articles. The simulation of the vortex induced vibration will be employed and analysed by using GAMBIT software and ANSYS software. Table 3.1 shows the properties which involve in this simulation and same data used for the four models. For each model of the two dimensional and three dimensional are conducted with the same Reynolds number, boundary conditions, and design parameter. But except for the two dimensional using different viscous model. For three dimensional analysis, 2 different aspect consider to compare with the actual condition at below sea stream. 1st aspect using marine riser independently without any additional suppression devices and 2nd aspect was attached the helical strake known as suppression device at marine riser and modelled accordingly. The two different types of marine riser will produce different vortex shedding and results which helpful to oil rig companies. All the details are summarized in Table 3.2, Table 3.3 and Table 3.4, for the simulation settings of 2D laminar model, 2D turbulent model and 3D turbulent models respectively.

Table 3.1: Properties for flow past a marine riser

Parameter	Value
Diameter	1.0 meter
Density	1025 kg/m3
Dynamic viscosity	0.0015 kg/m.s
Current velocity	1 m/s
Reynolds number	150,000

Table 3.2: Simulation settings for 2D laminar model

Settings	Choice
Simulation type	2D, Unsteady
Solver	Double precision, Pressure based and implicit
Temporal discretization	2nd order
Viscous model	Laminar models
Pressure	Standard
Pressure-velocity coupling	PISO
Momentum	2nd order upwind
Boundary conditions:	
Inlet	Velocity inlet
Outlet	Outflow
Circular cylinder	No-slip wall

Table 3.3: Simulation settings for 2D turbulence model

Settings	Choice
Simulation type	2D, Unsteady
Solver	Double precision, Pressure based and implicit
Temporal discretization	2nd order
Turbulence model	k-ω and SST models
Pressure	Standard
Pressure-velocity coupling	PISO
Momentum	2nd order upwind
Turbulent kinetic energy	2nd order upwind
Turbulent dissipation rate (for k-ω model)	2nd order upwind
Specific dissipation rate (for SST model)	2nd order upwind
Boundary conditions:	
Inlet	Velocity inlet
Outlet	Pressure outlet
Top, bottom and side wall	Symmetry
Riser wall	No-slip wall

Table 3.4: Simulation settings for 3D turbulence models of with and without helical strake

Settings	Choice
Simulation type	3D, Unsteady
Solver	Double precision, Pressure based and implicit
Temporal discretization	2nd order
Turbulence model	k-ω and SST models
Pressure	Standard
Pressure-velocity coupling	PISO
Momentum	2nd order upwind
Turbulent kinetic energy	2nd order upwind
Turbulent dissipation rate (for k-ω model)	2nd order upwind
Specific dissipation rate (for SST model)	2nd order upwind
Boundary conditions:	
Inlet	Velocity inlet
Outlet	Pressure outlet
Top, bottom and side wall	Symmetry
Riser wall	No-slip wall

CHAPTER 4: RESULTS AND DISCUSSION

4.1 Introduction

On this chapter will present the results of the simulation that were carried out. In this simulation, the main concerned of the vortex induced vibration is vortex shedding at the downstream. This situation can be seen through the contours of vortex and the oscillation reflected in graph which are drag and lift coefficient graph. There also been compared the results between laminar and turbulent for two dimensional. Meanwhile, compared the actual case with results of vortex shedding occur at downstream of marine riser between with and without helical strake. All the results will be discussed in this section. Data validation also provided for some results for clarification based on previous studies.

4.2 Analysis Vortex Induced vibration of Two Dimensional Model

4.2.1 Laminar flow past around cylinder

In this laminar model, the vortex shedding occurs at the downstream of the domain after the circular cylinder. Figure 4.1 shown the contours of vorticity magnitude for this laminar flow. The figure shows that vortex shedding is unsteady. This is because of the Reynolds number is 150,000 which means the regime is above the steady regime. This result is validated by referring the study of Farhoud R.K. et al (2012). They observed that, the cylinder fluctuate at small range of amplitude respect to the Reynolds number of 150. Furthermore, the drag coefficient is found smaller of half of the lift coefficient, the reason behind the drag coefficient amplitude lower or smaller than lift coefficient due to vortex shedding occur around circular cylinder at both direction since the sea flow changes. The drag and lift coefficient for this laminar flow is shown in figure 4.2 and 4.3 respectively.

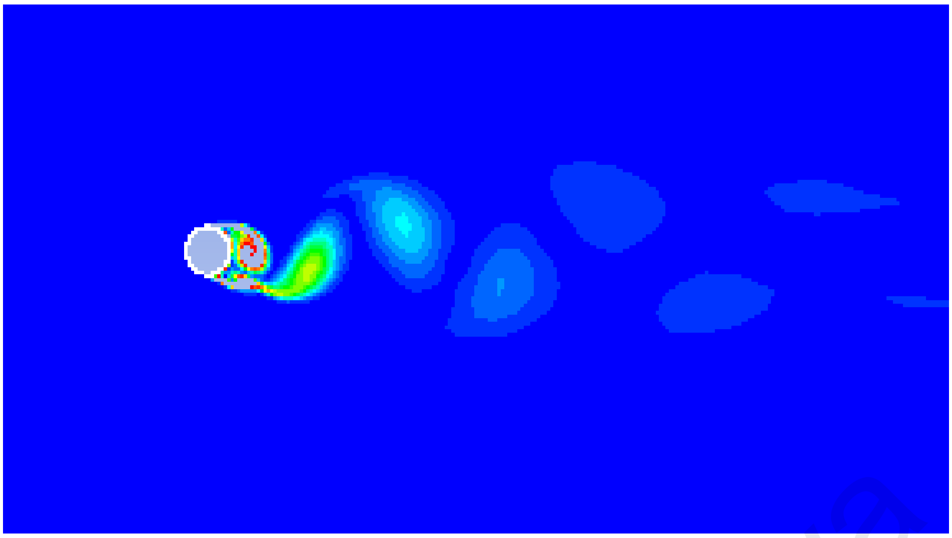

Figure 4.1: Contours of vorticity magnitude for this laminar flow

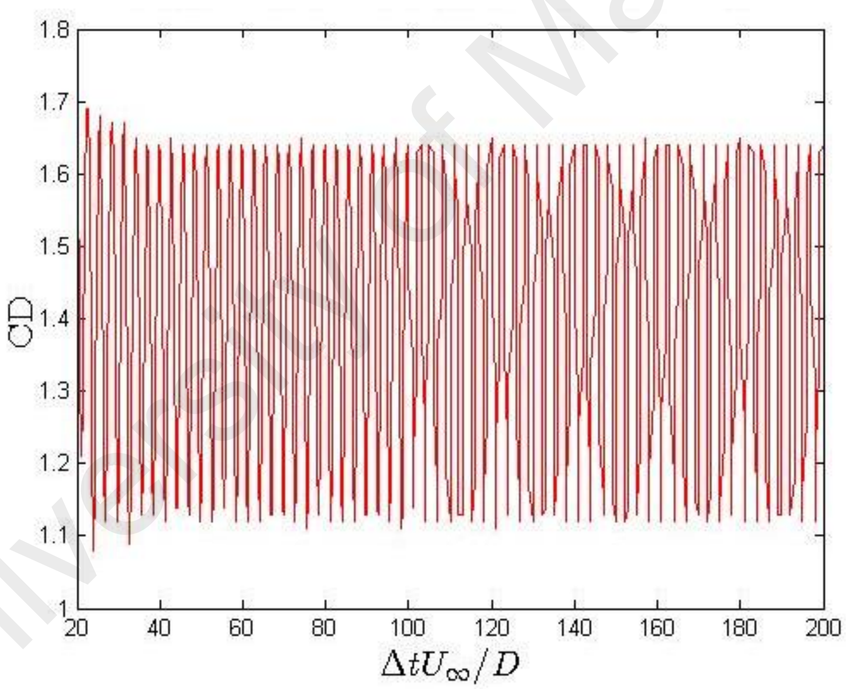

Figure 4.2: Drag coefficient graph for unsteady laminar flow

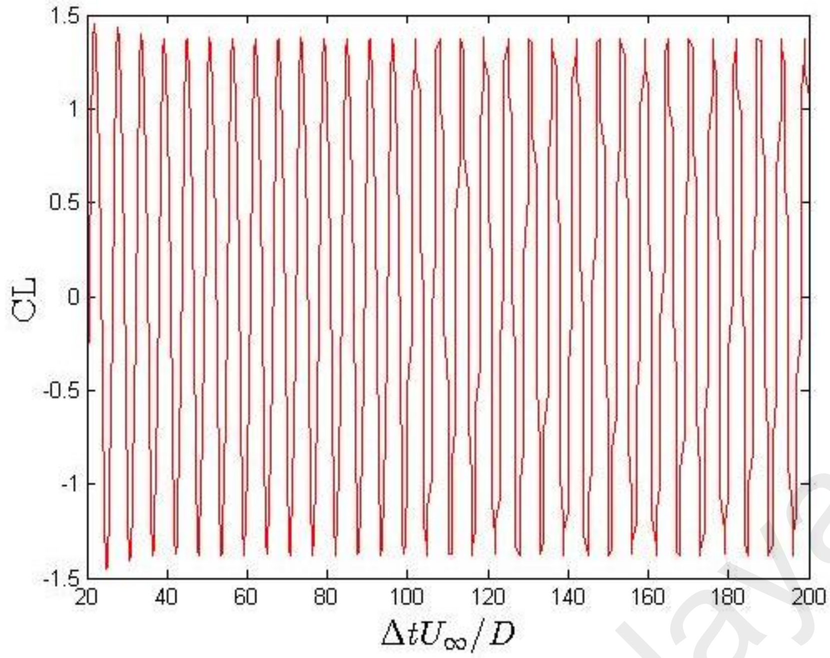

Figure 4.3: Lift coefficient of the unsteady laminar flow

There are other results also obtain from this simulation. These results reflect the unsteady laminar flow with the Reynolds number of 150,000. This results are contour of velocity magnitude, contour of stream function and velocity vectors coloured by vorticity magnitude are shown below.

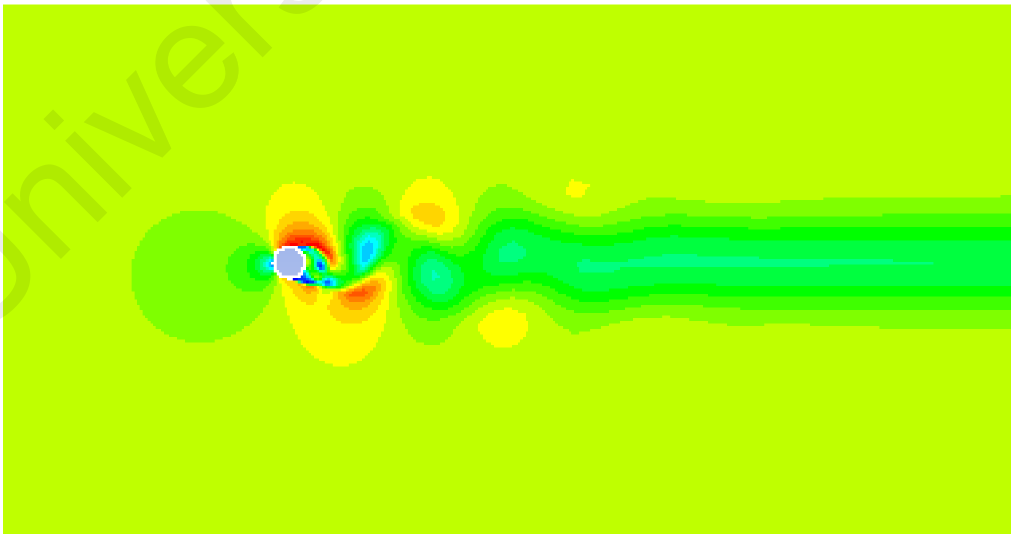

Figure 4.4: Contour of velocity magnitude

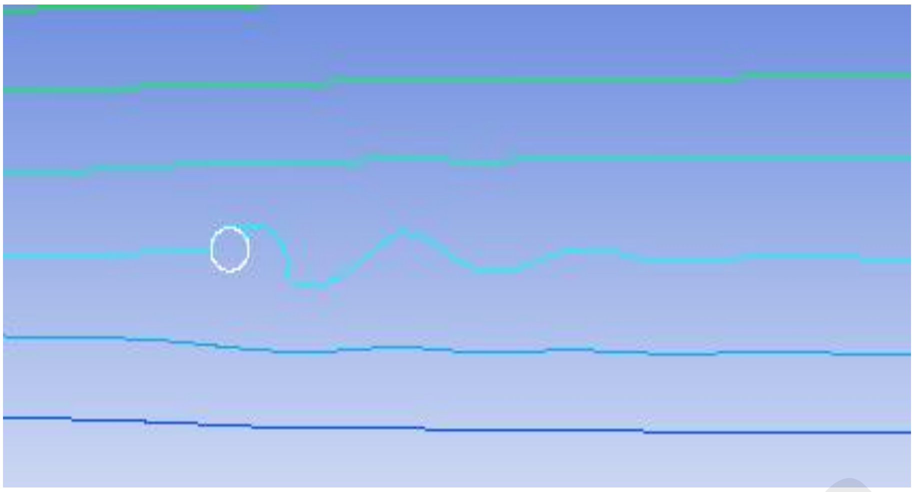

Figure 4.5: Contour of stream function

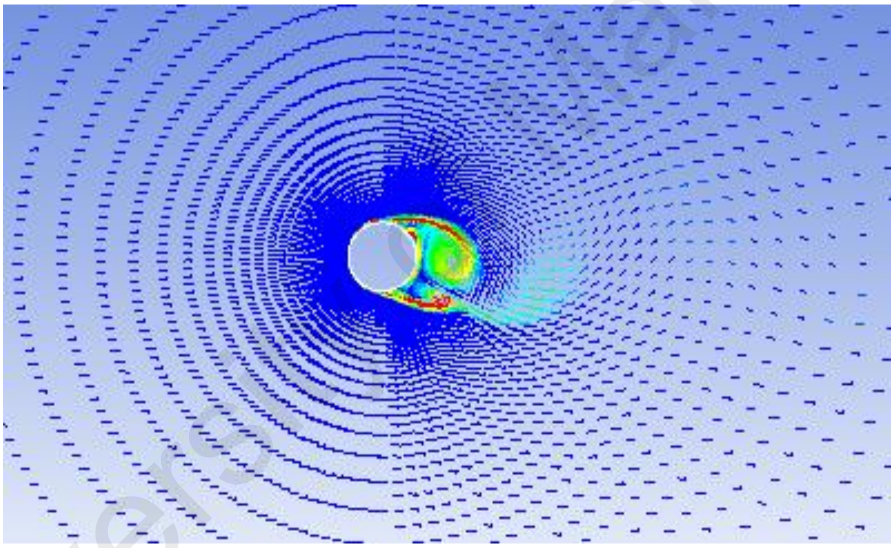

Figure 4.6: Velocity vectors coloured by vorticity magnitude

4.2.2 Turbulence flow past around cylinder

In this turbulent model, the vortex shedding occurs at the downstream of the domain. Figure 4.7 shown the contours of vorticity magnitude for this turbulence flow. The figure shows that vortex shedding is unsteady. This result is merely different from the previous results which is laminar flow. This is because of the forces and pressure at the circular cylinder. Also due to the different properties of model induced the different flow as well as different results. This result is validated by referring the study of Saghafian M.et al. (2003). By validating the results we obtain from ANSYS Fluent, proved that the simulation of two dimensional flow uses Reynolds number of 20,000 and due to the high Reynolds number the turbulent stream occur behind the circular cylinder. Meanwhile the Strouhal number also affected by the turbulence flow and separation position. The separation found to delay in the stream due to high turbulence effects. The drag and lift coefficient are shown at figure 4.8 and 4.9. Both drag and lift coefficient most likely similar with the result obtain by Saghafian M.et al. (2003).

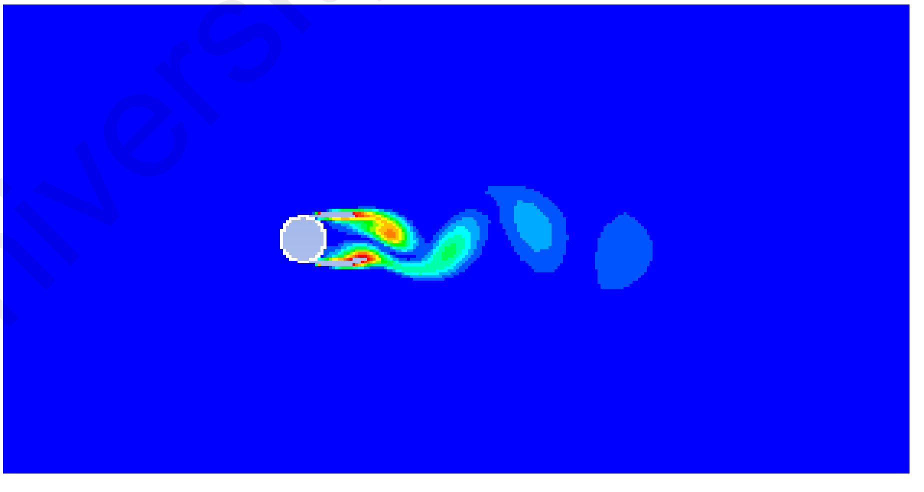

Figure 4.7: contours of vorticity magnitude for this turbulence flow

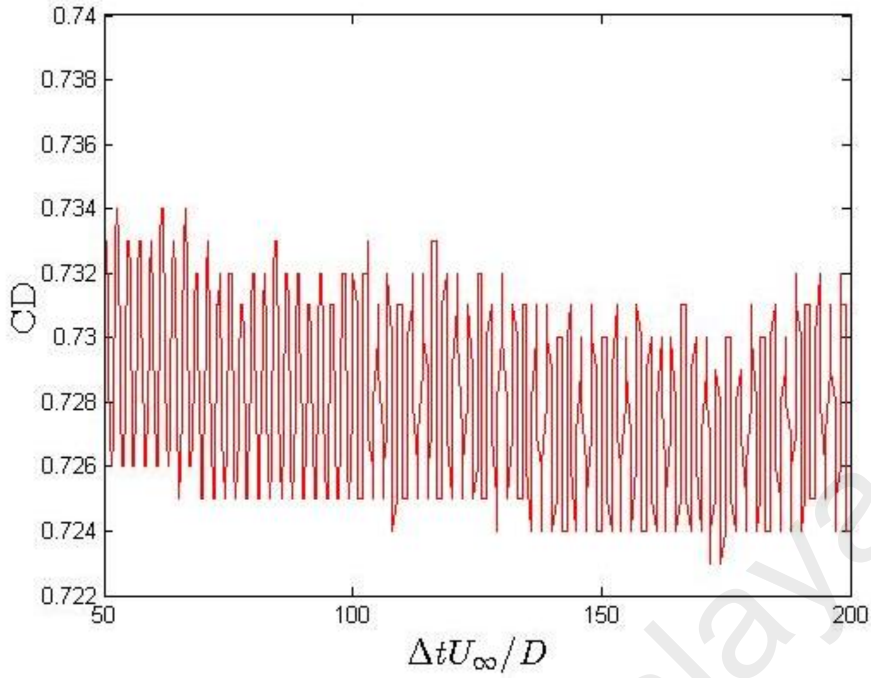

Figure 4.8: Drag coefficient of the turbulence flow

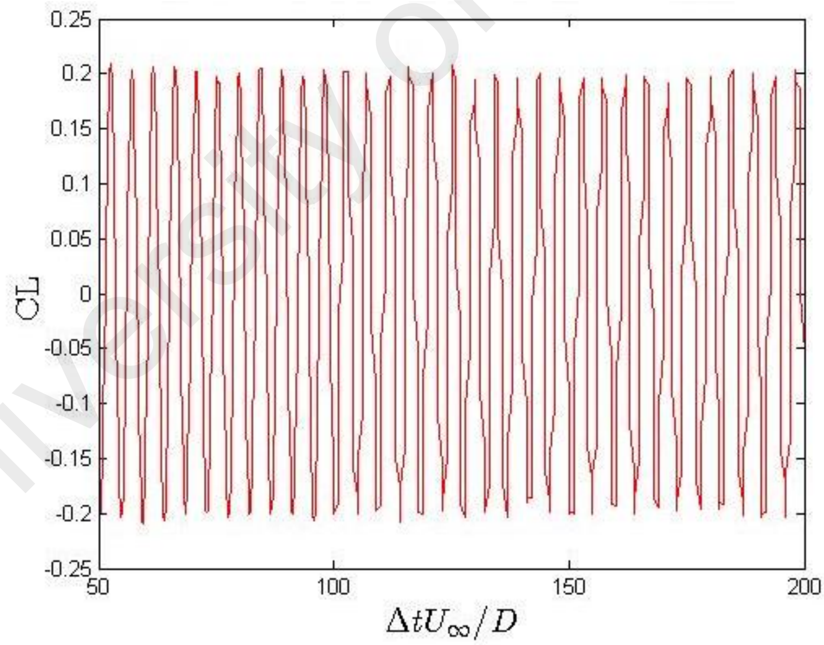

Figure 4.9: Lift coefficient of the turbulence flow

Simulation of turbulence flow provides other data such as contour of velocity magnitude, contour of static pressure and velocity vectors coloured by vorticity magnitude are shown figure 4.10, 4.11 and 4.12 respectively.

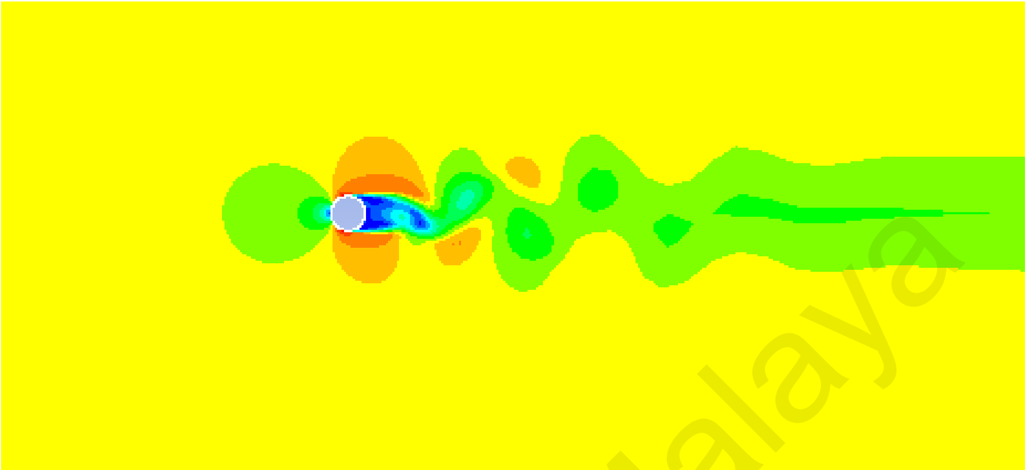

Figure 4.10: Contour of velocity magnitude foe turbulence flow

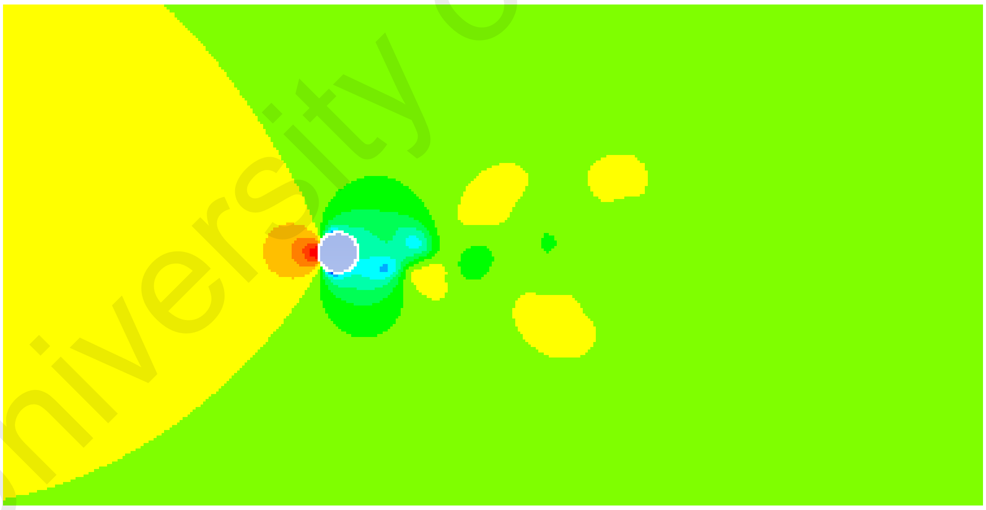

Figure 4.11: Contour of static pressure for turbulence flow

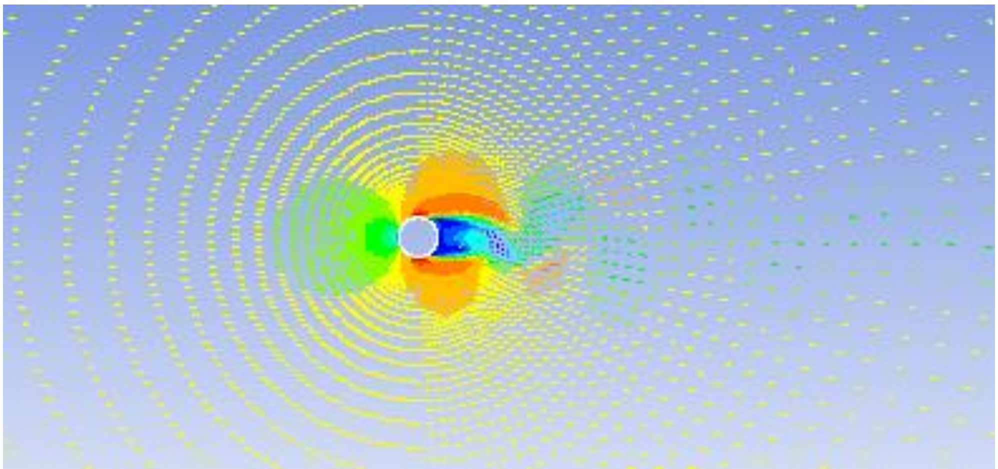

Figure 4.12: velocity vectors coloured by vorticity magnitude for turbulence flow

4.3 Analysis vortex Induced vibration of three dimensional models

This simulation of vortex induced vibration for three dimensional is validate based on the two dimensional results. Because lack of resources for the three dimensional model. The three dimensional simulation will be conducted for the marine riser with and without helical strakes. Following results will shows and discuss the different impact of fluid flow after helical strakes.

4.3.1 Marine riser without helical strake

For this section, the simulation carried out with marine riser without helical strake. All the data used similar with the two dimensional model of turbulence flow. The results of this simulation such as contours of static pressure, contours of velocity magnitude, contours of vorticity magnitude and contours of turbulent viscosity are shown in figure 4.13 to 4.16.

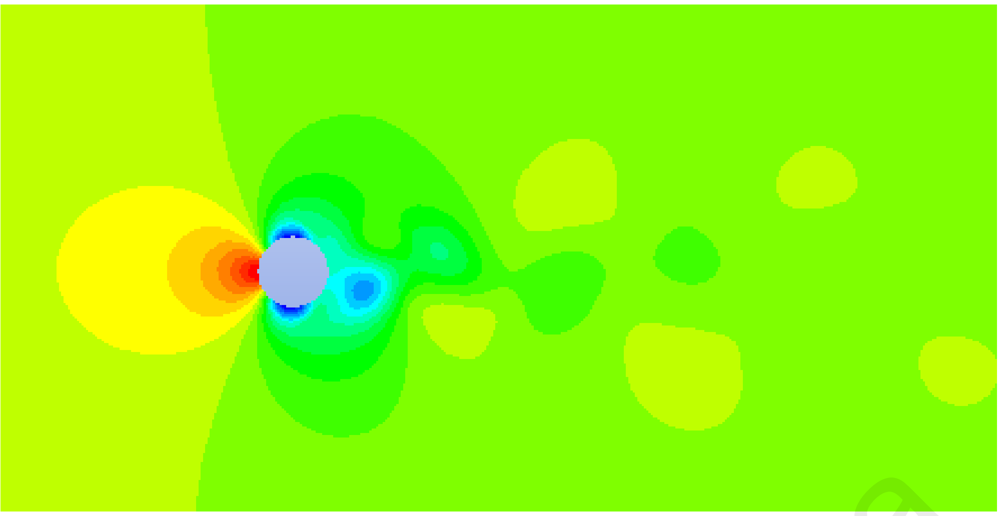

Figure 4.13: Contours of static pressure of without helical strake for 3D

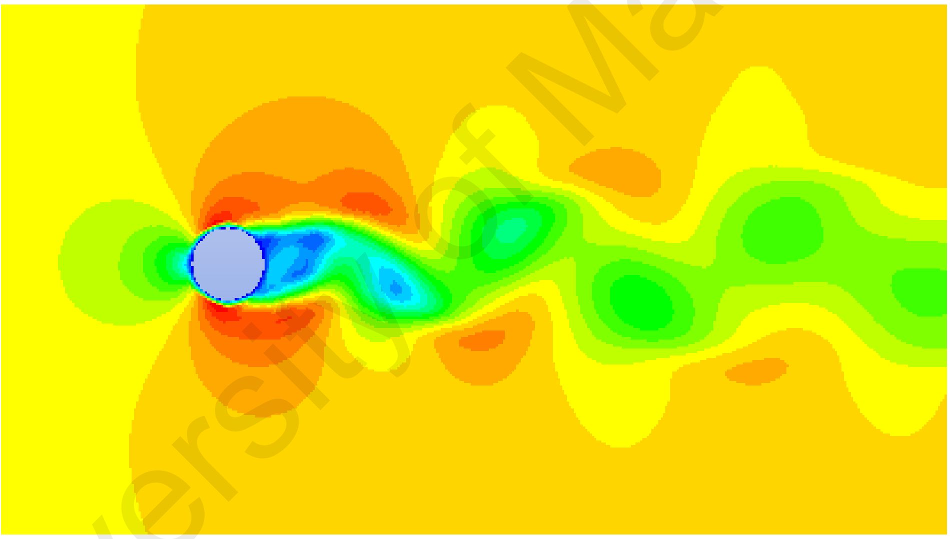

Figure 4.14: Contours of velocity magnitude of without helical strake for 3D

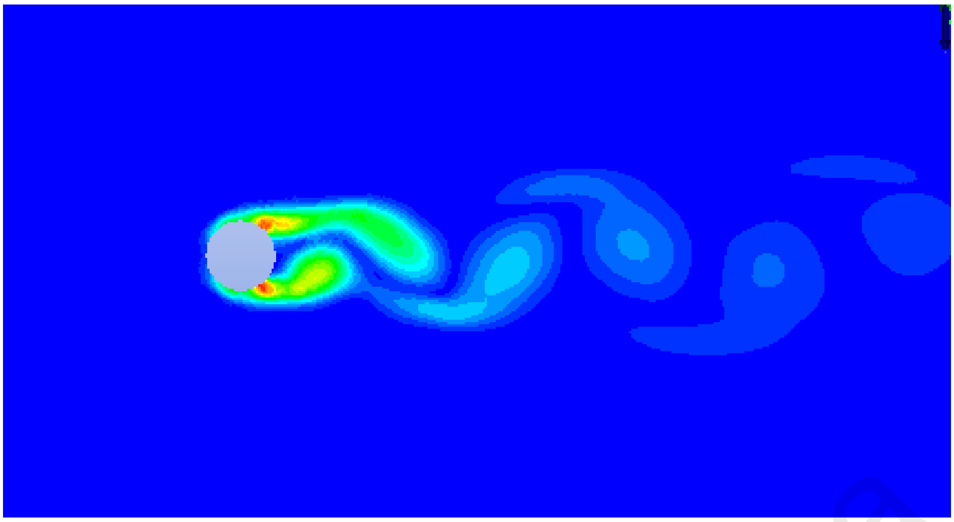

Figure 4.15: Contours of vorticity magnitude of without helical strake for 3D

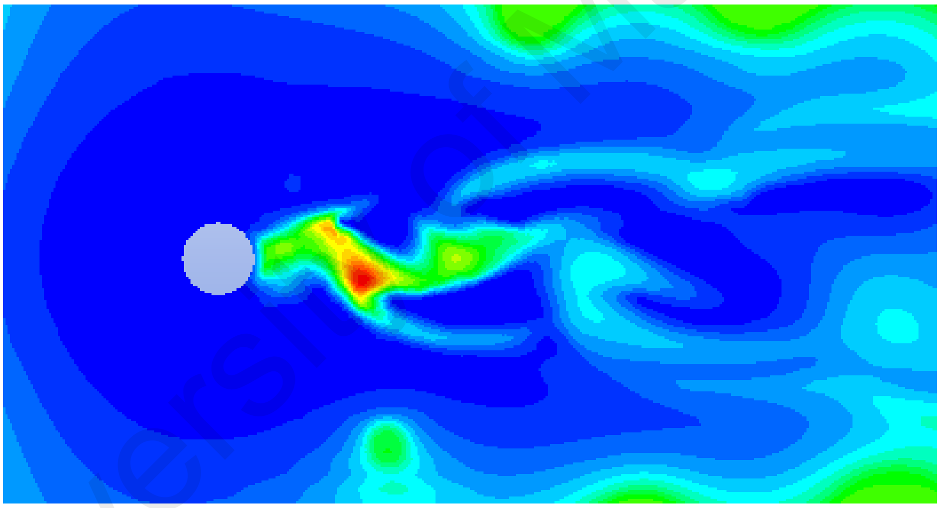

Figure 4.16: Contours of turbulent viscosity of without helical strake for 3D

4.3.2 Marine riser with helical strake

For this section, the simulation carried out with helical strake. This is the important part in this section, because this simulation indicates the efficiency of the helical strake on the vortex shedding and performance of the strake. The results of this simulation such as contours of static pressure, contours of velocity magnitude, contours of vorticity magnitude and contours of turbulent viscosity are shown in figure 4.17 to 4.20.

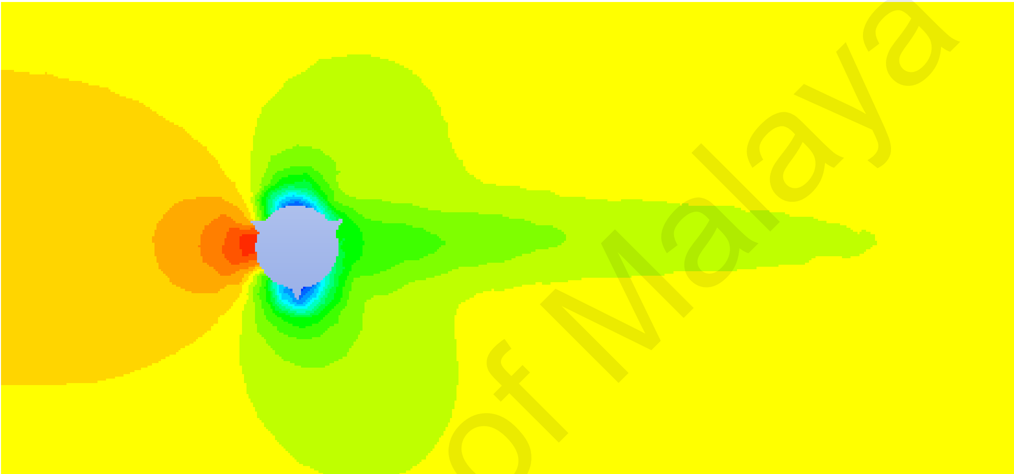

Figure 4.17: Contours of static pressure of helical strake for 3D

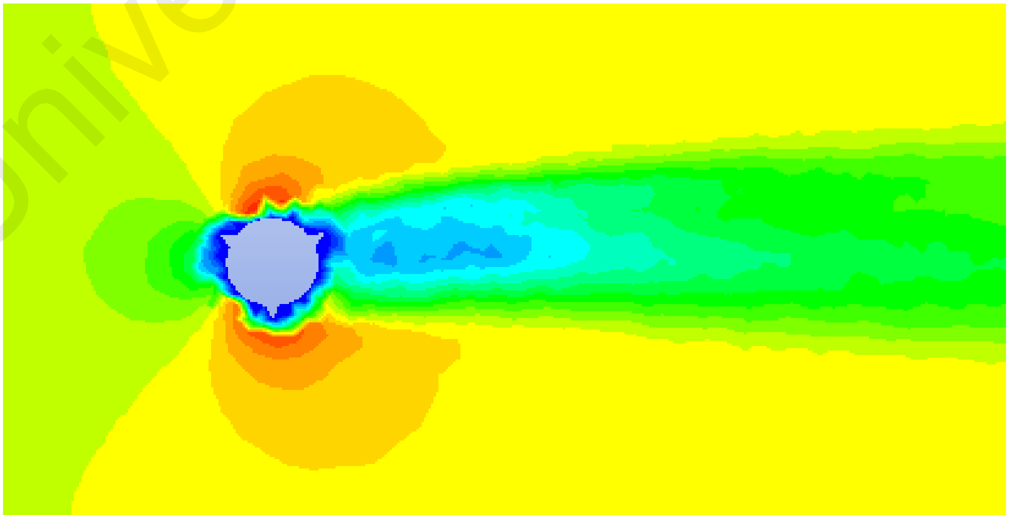

Figure 4.18: Contours of velocity magnitude of helical strake for 3D

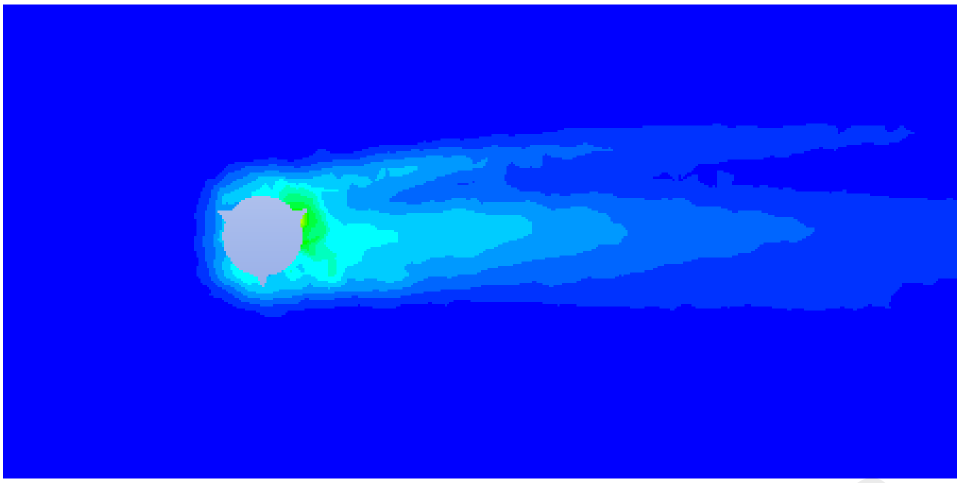

Figure 4.19: Contours of vorticity magnitude of helical strake for 3D

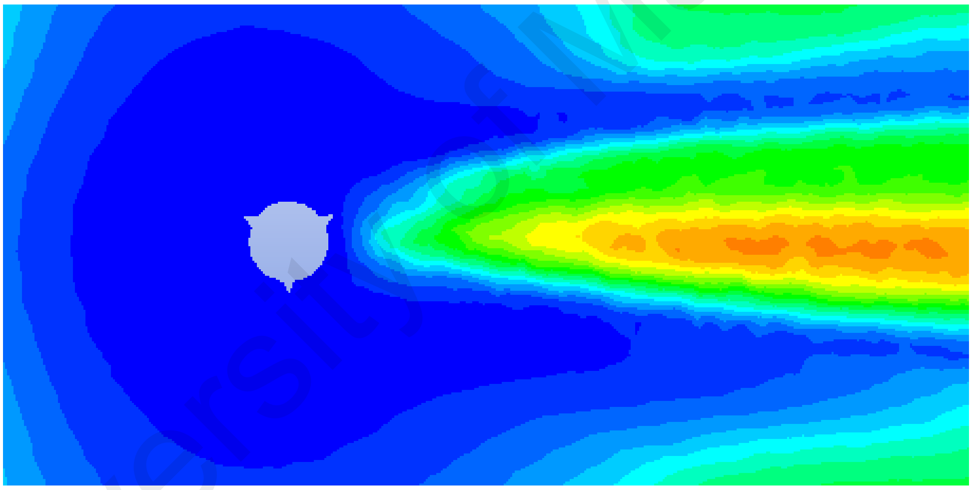

Figure 4.20: Contours of turbulent viscosity of helical strake for 3D

4.3.3 Analysis of vortex induced vibration with and without helical strake

Based on the results that obtains from the simulation of with and without helical strake, it is shows that effect of helical strake at the vortex shedding. When compared the contours of static pressure between with and without helical strake, the model with helical strake have lower static pressure. As well as contours of velocity magnitude are lower for the helical model. The main important result is vorticity magnitude. When compare the both results of contours of vorticity magnitude for with and without helical strake, it is proven that by using the helical strake the vortex shedding can be reduced as shown in Figure 4.19.

As mention earlier, this comparison can be identified by referring the lift and drag coefficient which is generated by MATLAB software. Figure 4.21 and Figure 4.22 are the results of drag coefficient flow past a marine riser without helical strake. The drag coefficient r.m.s from the results shows that 0.03073 and 0.01148 for with and without helical strake respectively. Drag coefficient r.m.s value for with helical strake is more than without helical strakes because helical strakes are relatively bluff to the incoming flow and often produce early separation of the incoming flow, they are associated with higher drag. Figure 4.23 and Figure 4.24 are the result of lift coefficient flow past a marine riser with helical strake. The lift coefficient r.m.s from the results showed that 0.007765 and 0.2284 for with and without helical strake respectively. Based on this graph, the helical strake has lower lift coefficient compare to without. Meanwhile, the drag coefficient increase due to the helical strake. The regime of Vortex Street for both condition can be seen from Figure 4.25 to Figure 4.28.

4.3.3.1 Drag Coefficient

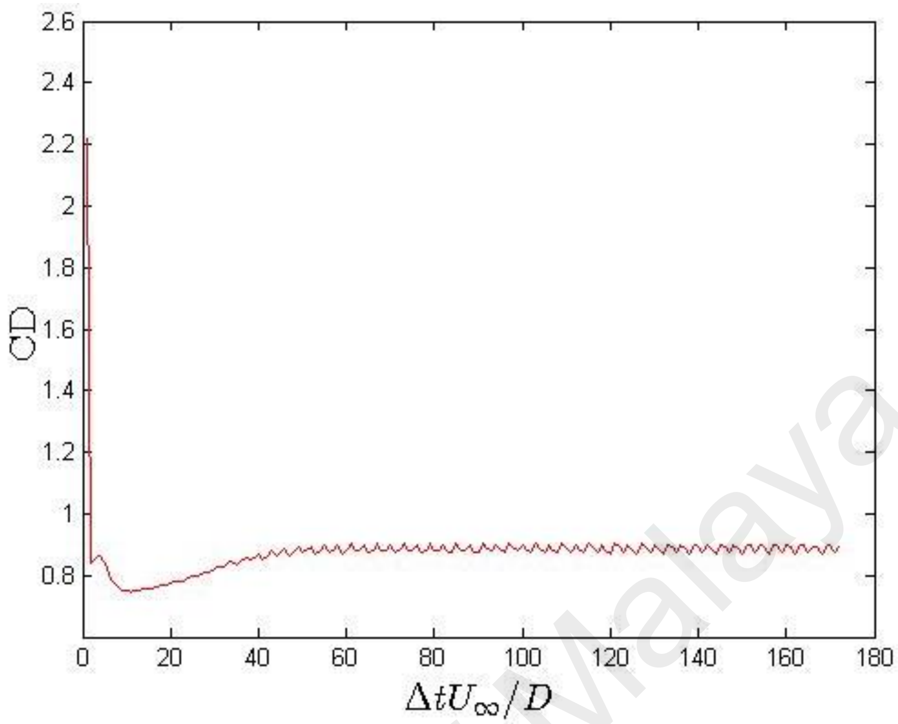

Figure 4.21: Drag coefficient for without helical strake

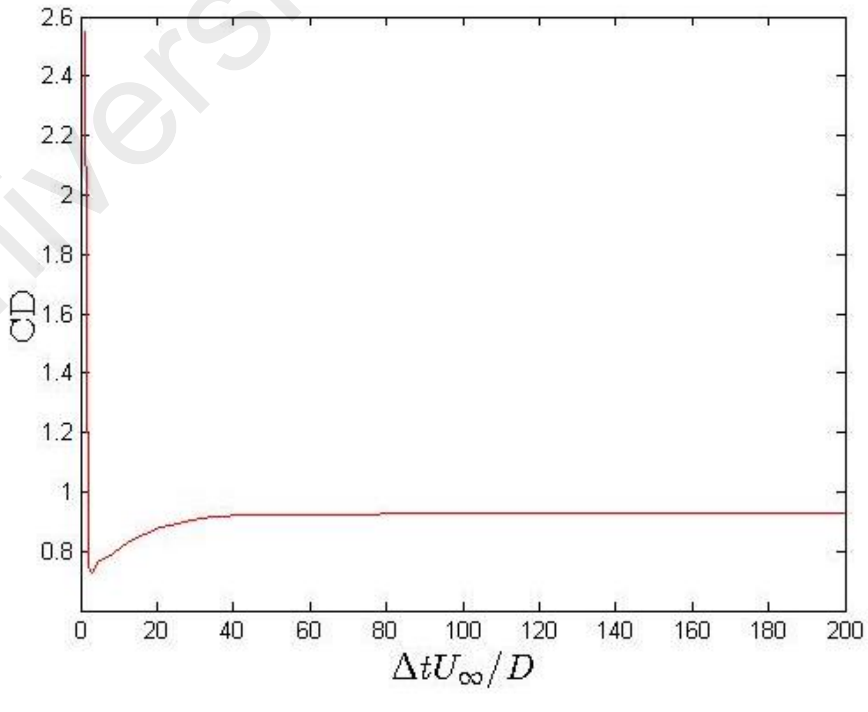

Figure 4.22: Drag coefficient for with helical strake

4.3.3.2 Lift Coefficient

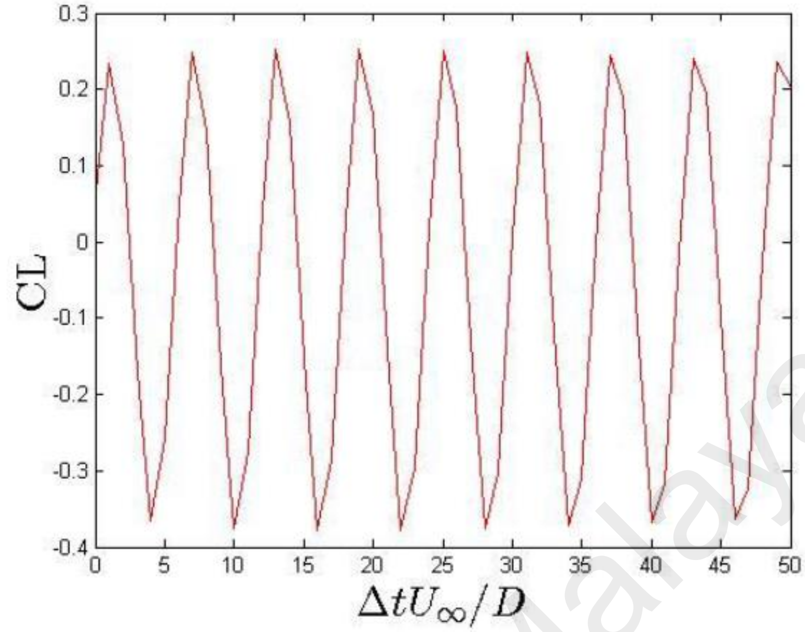

Figure 4.23: Lift coefficient for without helical strake

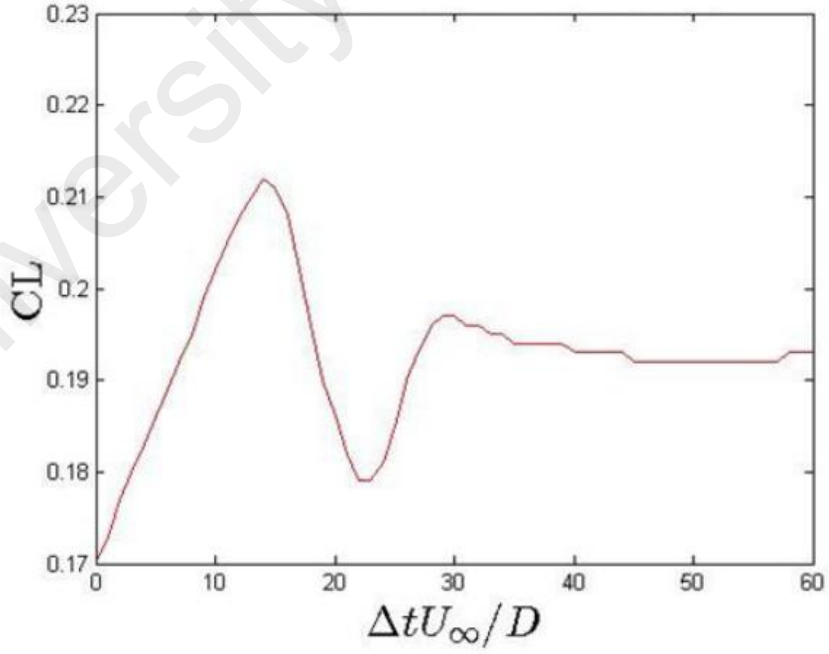

Figure 4.24: Lift coefficient for with helical strake

4.3.3.3 Vortex Regime for without Helical Strake

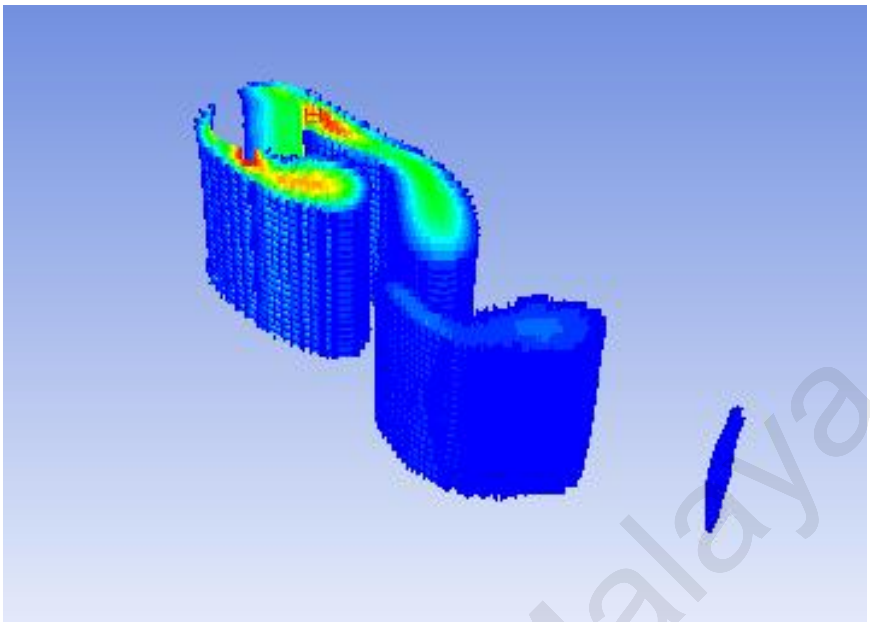

Figure 4.25: Vorticity magnitude of vortex regime for without helical strake

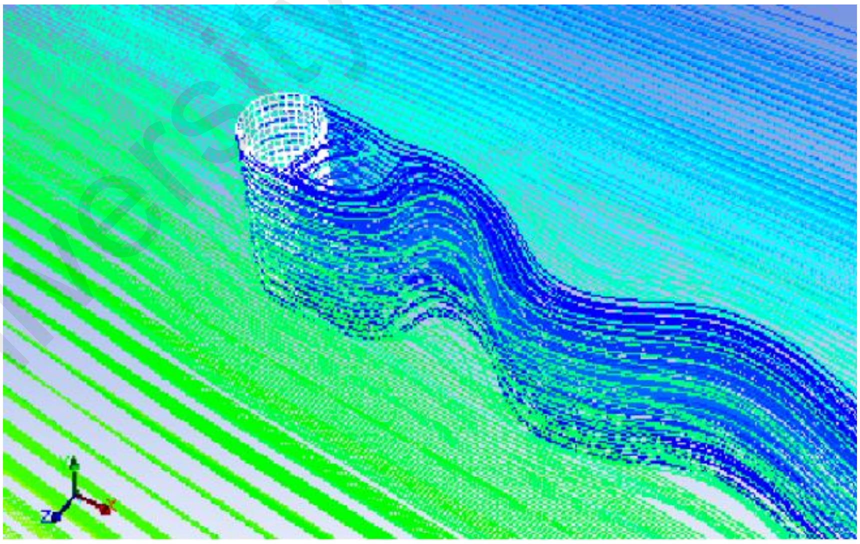

Figure 4.26: Path line coloured of vortex regime for without helical strake

4.3.3.4 Vortex Regime for with Helical Strake

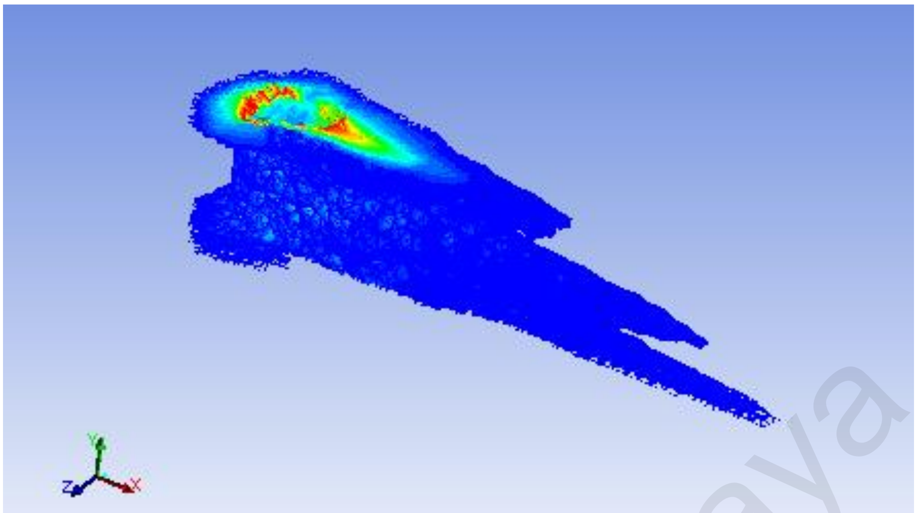

Figure 4.27: Vorticity magnitude of vortex regime for helical strake

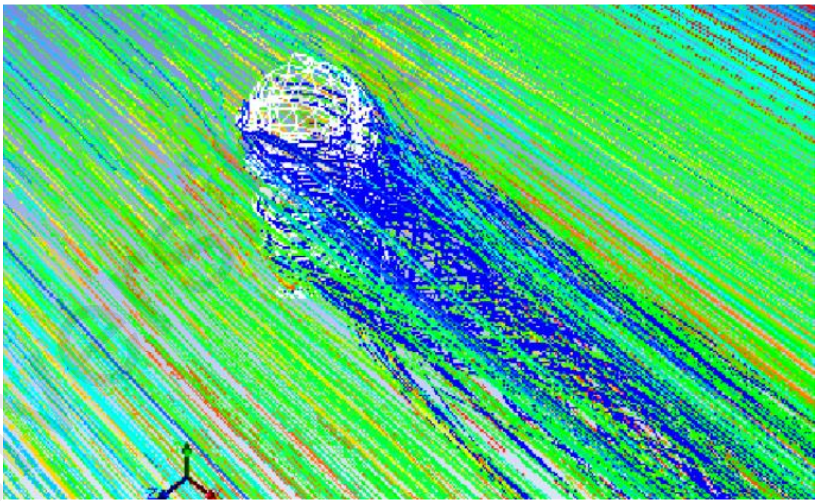

Figure 4.28: Path line coloured of vortex regime for with helical strake

The simulation results above as shown in figure 4.26 for vortex regime without helical strakes and figure 4.28 for vortex regime with helical strakes. Without the helical strakes the marine riser oscillates at certain amplitudes due to vortex shedding. The colours shows the perfect vortex shedding occur inside the domain. Vortex shedding more at this situation because the flow separation leads to pressure different. The tremendous changes can be obtain by implementing helical strakes at marine riser. The fluid flow was disturb by the helical blades and reduce the vortex shedding.

4.4 Summary

From the results that obtain from the simulation, which are two dimensional and three dimensional results are analysed. For the two dimensional model, the laminar flow and turbulent flow results are analysed. From this, it shows that for both conditions have same types of vortex shedding and graph method. But have different values for both conditions. For the three dimensional model, two types of simulation conducted such as with and without helical strake. From this results, with using the suppressing device which is helical strake reduced the vortex induced vibration compare to without helical strake. The graph of drag coefficient shows the higher drag for with helical strakes. This because of helical strakes increase the frontal area of the marine riser. Hence the flow of sea current hit the surface of helical strakes which is bigger than the without suppression device and creates higher drag coefficient. Other hand, lift coefficient of with helical strake has the lower fluctuating value compare to without suppression device cause of the flow separation was disturbed and vortex shedding creation was reduced.

CHAPTER 5: CONCLUSION AND RECOMMENDATION

5.1 Conclusion

In conclusion, the study was conducted to understand the effectiveness of with and without helical strake on vortex induced vibration. This study is simulated by using ANSYS Fluent software for both two dimensional and three dimensional. Hence the objectives of this study are achieved. From the model that analyzed, the comparison has been made. It is proven that, by using the suppressing device which is helical strake the vortex induced vibration can be reduced. Graph lift coefficient shows that the fluctuating of a marine riser is lower by employed the helical strake. From the graph of without helical strake, the r.m.s lift coefficient was 0.2284 and for the graph with helical strake has the value of r.m.s life coefficient is 0.007765. So that, this lift coefficient is decreased by 96 percentage. The lift and drag coefficient have influence the result of VIV in way that oscillation of marine riser due to drag and lift forces acting on it. The major cause of higher oscillation marine riser cause by lift force perpendicular to inflow velocity and drag is in parallel. The excitation of lift force on riser creates vortex shedding that leads to vibration.

Hence, the helical strake can reduce the vibration of the marine riser to increase the lifetime of the riser. Helical strakes contain part by part components and installed along the marine riser in spiralled condition. The helical shape of fins break the vortices effectively which occur behind the marine riser along the certain length. Sea current flow hit the strakes and separation happened early on the wake. The stakes faces high drag force due to the frontal area is bigger compare to other types of suppressing devices. Meanwhile, these type helical strake can be application in all direction of sea current, unlike the fairing type device. However, many oil & gas production companies applying these helical stakes as main suppressing devise on marine riser since they found the outcome on minimising the vortex-induced vibration (VIV) in marine industry. Generally, the helical strakes will not installed full length of any marine structure due to cost and inefficient. Only will install at certain high above from the seabed and below from the water level to meet the optimize design.

5.2 Recommendations

There are few numbers of recommendations could be considered as an improvement for the current study. They are as following,

a. Should using the Fluid-Structure Interaction to be closer to the real situation that happens in sup-sea. So that can solve real situation problem and can predict the fatigue.

b. For the effectiveness of helical strakes can be decrease by the marine growth also known as species or plant that stick or grow at riser. To minimize or reduce such marine growth to grow at helical stakes, required to apply a layer of coating at surface of helical and by doing these the performance of stakes can be protected.

c. To perform the Computational Fluid Dynamics (CFD) simulation in higher accurarcy, there are few factors need to be consider. Domain shape is vital in fluids and mesh or grid related to size and shape as well as solver to calculate the data. All the mention factors are highly effected the CFD simulation with data and time period. Thus, each factor required detailed studies prior to produce optimize setup on analysis.

d. More shape of stakes or fairing should design and analysis the efficiency in installation method to reduce time and hazard, materials to prevent corrosions, and mainly in vortex induced vibration (VIV) situation to reduce the vortex formation and wake.

REFERENCES

Ajayi O.O., Agarana M.C., Adeniyi M.O., & Mercy Ogbonnaya (2017). Modelling and control of Flow Induced Vibration of Top-Tensioned Marine Risers using Analytical Methods. *Journal of Procedia Manufacturing*, 7, 609-615.

Ball I. et al. (2012). *Maximizing Vortex Induce Vibrations Through Geometry Variation*. Unpublished thesis, Worcester polytechnic institute, United States.

Blevins, R. D. (1990). *Flow-Induced Vibration*, 2nd edition, Van Nostrand Reinhold. New York, United States.

Carmo,B.C. et al., (2011). Flow-induced vibration of a circular cylinder subjected to wake interference at low Reynolds number. *Journal of Fluids and Structures*, 27, 503-522.

Farhound, R.K. et al. (2012). Numerical Study of Unsteady Laminar Flow around a Circular Cylinder. *Journal of Civil Engineering and Urbanism*, 2(2), 63-67.

Fuller R.G., Lang C.R. & Lang R.H (2000). *Twin views of the Tacoma Narrows Bridge Collapse*. USA: One Physics Ellipse College Park, MD 20740-3845

How B.V.E, Ge S.S. & Choo Y.S (2009). Active control of flexible marine risers. *Journal of Sound and Vibration*, 320, 758-776.

Knardahl, G.M. (2012). *Vortex Induced Vibrations of Marine Risers.* Unpublished master's thesis, Norwegian University of Science and Technology, Norway.

Kumar N. et al., (2018). Suppression of vortex-induced vibrations using flexible shrouding-An experimental study. *Journal of Fluids and Structures*, 81, 479-491.

Lou M. et al., (2017). Experimental study on vortex induced vibration of risers with fairing considering wake interference. *International Journal of Naval Architecture and Ocean Engineering*, 9, 127-134.

Li, Shengtai & Hui Li, (2006). *Rayleigh–Taylor instability*. Retrieved from https://en.wikipedia.org/wiki/Rayleigh%E2%80%93Taylor_instability#cite_note-17

Liea H., Baarholma G.S. and Larsen C.M. (2005). Reduction of VIV using suppression devices an empirical approach. *Marine Structures*, 18, 489-510.

Lionel E. and Stephane P. (2002). *General purpose fluid mechanics hands-on.* Retrieved from: http://hmf.enseeiht.fr/ travaux/CD0102/travaux/optmfn/gpfmho/01-02/grp1/presenta.htm.

Liu Xiuquan et al. (2016). Analysis on the Operation Fatigue of Deepwater Drilling Riser System. *The Open Petroleum Engineering Journal*, 9, 279-287.

Mao Liangji et al., (2015). Vortex-induced vibration mechanism of drilling riser under shear flow. *Petroleum Exploration and Development*. 42(1), 112-118.

Mittal S. and Kumar V. (2001). Flow-induced vibrations of a light circular cylinder at reynolds numbers 10^3 to 10^4. *Journal of Sound and vibration*, 245(5), 923-946.

Mukundan, H. et al. (2009). Monitoring VIV fatigue damage on marine risers. *Journal of Fluids and Structures,* 25, 617-628.

Richard P.O., Tom M.A., & David L.G.,(2007). The Mechanical Universe, Mechanics and Heat, Advanced Edition. New York, NY: Cambridge University Press

Saghafian, M.et al. (2003). Simulation of turbulent flows around a circular cylinder using nonlinear eddy-viscosity modelling: steady and oscillatory ambient flows. *Journal of Fluids and Structures,* 17(8), 1213-1236.

Simantiras P. and Willis N. (1999). Investigation on vortex induced oscillations and helical strakes effectiveness at very high incidence angles. *Proceedings of the International Offshore and Polar Engineering Conference*, May 30 – June 4, Brest, France.

Tiratsoo J. (2011). *Trelleborg launches new subsea vibration suppression system*. Retrieved from pipe lines international: http://pipelinesinternational.com/news/trelleborg_launches_new_subsea_vibration_suppression_system/055021/

Xu J, et al (2017). A vortex-induced vibration model for the fatigue analysis of a marine drilling riser. *Ships and Offshore Structures*, 12(1), S280-S287.

Yogini Patel, (2010). Numerical Investigation of flow past a circular cylinder and on a staggered tube Bundle using various turbulence models. Unpublished master's thesis, Lappeenranta University of technology, Finland.

Zdravkovich M.M. (1996). Different modes of vortex shedding: An Overview. *Journal of Fluids and Structures*, 10 (5), 427-437.

Zhang H. and Lim F. (2010). *VIV Suppression Device Effectiveness for Steel Catenary Risers.* Proceedings of the Twentieth International Offshore and Polar Engineering Conference. June 20-25 .Beijing, China