UNIVERSITI
TEKNOLOGI
PETRONAS

Riser Stress Analysis with Finite Element Approach

by

Lim Kha Loon

Dissertation submitted in partial fulfilment of
The requirements for the
Bachelor of Engineering (HONS)
(Mechanical Engineering)

JANUARY 2009

Universiti Teknologi PETRONAS
Bandar Sri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Riser Stress Analysis with Finite Element Approach

by

LIM KHA LOON

A project dissertation submitted to the

Mechanical Engineering Programme

Universiti Teknologi PETRONAS

in partial fulfillment of the requirement for the

BACHELOR OF ENGINEERING (Hons)

(MECHANICAL ENGINEERING)

Approved by,

____

RAHMAT ISKANDAR KHAIRUL SHAZI

Project Supervisor

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

JANUARY 2009

CERTIFICATE OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

____

LIM KHA LOON

ABSTRACT

This report is written with the main objective to identify the stress distribution along the riser with the inclusion of external loading conditions. This is a further study of riser stress analysis of the data generated with Bentley AutoPIPE as the data received only available in numerical values but not shown in 3-dimensional stress distribution diagram. This can be overcome with ANSYS Multiphysics as 3-dimensional stress distribution result can be generated which are shown in the chapter 4 of this report. PIPE59 element in ANSYS element library is chosen for the modelling process of this project. PIPE59 element is suitable to be practiced for modelling task related to immersed pipe. The simulation is commenced with the input data of a project located at Persian Gulf region. Through the simulation, analysis of the output on how loading conditions affect the riser stress analysis was obtained. The result which includes Von Mises stress and bending stress obtained were compared with the result generated from AutoPIPE, commercial finite element software using in pipeline design industry. This is to justify that ANSYS Multiphysics capable to produce result similar to Bentley AutoPIPE besides output the result in 3-dimensional stress distribution. Lastly, the knowledge gathered from this project can be used to perform riser stress analysis for commercial purpose.

ACKNOWLEDGEMENT

The author would like to express his utmost gratitude to Mr. Rahmat Iskandar Khairul Shazi for the advice, assistance, help and experience. It was with his guidance that a topic that so alien to author became familiar and interesting.

The author is ever grateful to multitude of persons he has come across who have shown the way in tackling this project, or at least have been a good friend and lend their ears and hands. The author is much obliged to Pipeline Engineer – Ir. Muhamad Hazlalin Ibrahim and Ezhani Esa from Ranhill WorleyParsons for their kindness and overwhelming support by providing all the input parameters required of this project. Kind support from Ezhani Esa whose experience in Bentley AUTOPIPE has prepared author with adequate pipeline design knowledge and methodology has helped in obtaining good progress throughout the project. A great thank you is also in order to Mr. Julendra for his experience and expertise in using ANSYS Multiphysics and assisting the author in navigating the new software.

The author wishes to express gratitude to Fan Yan Wei, a PhD student from Yangtze University for sharing knowledge related PIPE59 element and modelling technique. The assistance lent by him has allowed this project to be finished within the allocated time frame.

All the hard work of the author will not have been successful if not with the support, consideration and understanding from the family and friends when the author needed it most. Thank you all.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

1.1 Background of Study

The oil and gas industry has evolved rapidly throughout the years to meet the world demand of energy resources. As an increasing portion of oil and gas production is coming from offshore fields, more offshore platforms are built. Offshore platform is a very large structure as it consists of facilities such as living quarters, separators, water injection system and the machinery to extract oil and natural gas through wells in the ocean bed.

The standalone offshore platform in the middle of the sea is not a complete system as pipelines are required to connect between platforms and to crude oil terminal or floating tankers. Riser is an essential component in the pipeline system as it connects the subsea pipelines to the top side piping of the platform. There are two types of riser where rigid riser is applied on steel jacket platform; flexible riser is applied on deepwater condition where it connects the subsea wellhead to the floating production storage and offloading unit, FPSO.

In this study, the main focus is focused a shallow water rigid riser. A riser designed for construction in Persian Gulf is being referred to perform the riser stress analysis with finite element approach.

1.2 Problem Statement

The design of an offshore rigid riser is a complex task requiring the consideration of several factors, which need to be incorporated to arrive at an optimum topology or configuration of the pipeline system. In the pipeline design industry, stress analysis is among of the several analyses to be performed to ensure the riser configuration meets the production and site specified requirements. It is crucial to arrange the riser system configuration so that the external loading is kept within allowable limits with regard to tension, bending, torsion, compression and interference forces. Bentley AutoPIPE is one of the finite element software used in the industry to determine stress magnitude on the nodes modelled on a riser but there is a limitation with the software where it cannot present the stress distribution in 3-dimensional view such as what ANSYS Multiphysics software capable of. It is important to have the 3-dimension view of the stress distribution on the riser as this can help engineers understand the effect of external loadings acting along the riser. The development of a finite element stress analysis can significantly help in improving the multitudes of decision-making involved in a riser system, hence contributing to an enhanced overall design accuracy.

1.3 Objectives

The objectives of the project are as follows:

1. To study how stress distribution on the offshore rigid riser is affected by loading conditions.
2. To perform the analysis using finite element modelling software in order to determine the stress distribution on the offshore rigid riser in operating condition.

1.4 Scope of Study

The purpose of this study is to present the stress analysis for conventional riser carrying hydrocarbon gas installed on a steel jacket platform. Riser is an extension of a subsea pipeline which generally connects the topsides piping, often leading to a launcher or receiver, with the end of a pipeline or expansion spool piece on the seabed. The stress analysis is to be carried out with four loading conditions shown below:

- Dead load (weight of pipe, coating)
- Live load (weight of product)
- Thermal expansion at the end of the pipeline
- Hydrostatic/hydrodynamic loadings

The stress level under the worst combination of loading shall be checked against the allowable limits according to the design criteria given in Appendix D of ASME B31.8.

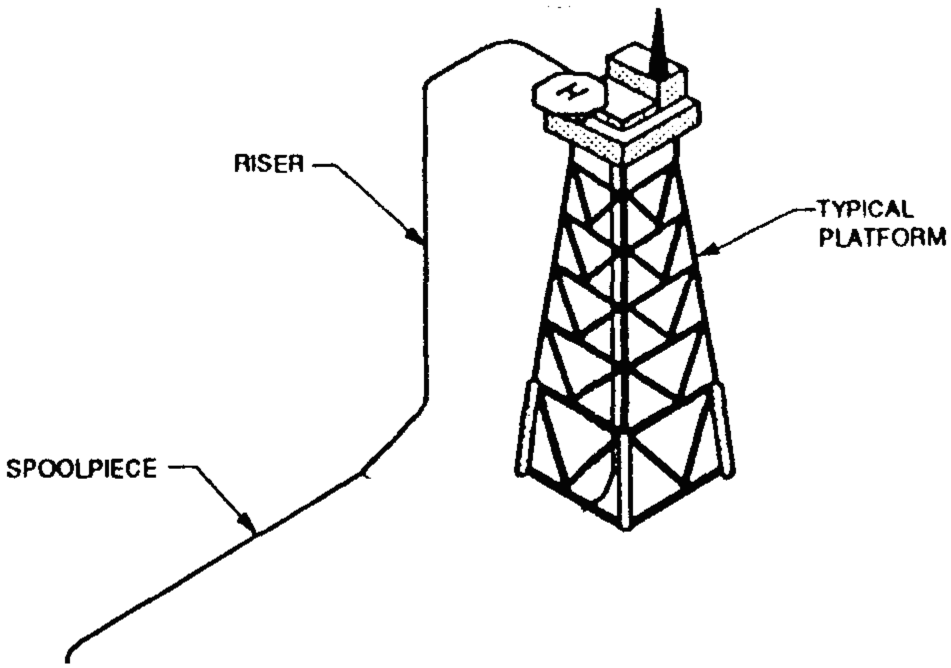


Figure 1.1: Conventional Riser on a Steel Jacket Platform

1.5 Feasibility of the Project

This project is suitable to be selected as final year project title after the evaluation of economic, technical skills and input parameters required.

Economic Feasibility

The first aspect of the feasibility being studied is in term of economic basis. The software required throughout this project is all available in the Mechanical department computer labs. Hence, no extra expenditure is required.

Technical Feasibility

The technical capability of author on the available software package is being considered. Author's decent knowledge background of software, ANSYS concludes that no further prerequisite course required to enroll before proceeding in this project. Hence, it is feasible for author to carry out this project in term of technical capability.

Data feasibility

The input parameters for this project are gathered from an actual field data with permission. The output of this project is compared with the result output obtained through Bentley AutoPIPE, pipeline stress analysis software including a state-of-the-art CAD-like graphical interface with unique object technology, fast analysis, realistic animation and visualization tools, and international design codes. This has helped in measuring the effectiveness of ANSYS in performing riser stress analysis compared to commercial software.

Based on the three aspects evaluated above, the project is proved to be feasible to be selected as final year project title.

CHAPTER 2

LITERATURE REVIEW

The design of an offshore pipeline system, which is a high cost involvement project is difficult task mainly due to the severe environmental conditions. The stress analysis on a pipeline system is highly important to ensure the structure able to reach the expected life cycle. H.-Y. Guo[1] and Bo Yu [2] have reported that the stress distribution on a riser is affected by the production and site specified requirements, installation method, static and dynamic loadings acting on it.

Various studies were completed in the open literature of pipeline engineering on problems related to riser stress analysis, they can be categorized into:

- Vortex-induced vibration on a riser
- Thermal impact on the pipeline
- Hydrodynamic force or marine pipelines including waves and currents

All of these studies are important and must be considered because these factors contribute to the equivalent stress acting on the riser.

2.1 Studies on factors affecting the output of equivalent stress

2.1.1 Effect of internal flow on vortex-induced vibration of risers

Vibration induced in elastic structures by vortex shedding is of practical importance because of its potentially destructive effect on marine risers. When the vortex shedding frequency approaches the material natural frequencies, large resonant oscillations occur. Large responses give rise to oscillatory stress. If these stress values persist, significant fatigue damage may occur. The vortex-induced vibration (VIV) response of a marine riser is a complicated process involving both the hydrodynamic and the structural properties of the riser.

H.-Y. Guo and M. Lou mentioned on the work has been done for VIVs, a system with the inclusion of internal flow inside the pipe has rarely been considered [1]. Thus, an experiment simultaneously involving internal fluid flow and external current is to be carried out. Several conclusions are drawn after processing the experiment data shown as below

- In a current, with the increase of internal flow speed, the amplitude of the strain in the in-line vibration and the cross-flow vibration will both increase, while the oscillation frequencies decrease. The cross-flow oscillation frequency follows the Strouhal relation, and the well-known frequency doubling between in-line oscillation and cross-flow vibration is obvious.

  $$f_s = \frac{S_t U}{D} (1) \qquad \text{(Equation 2.1)}$$

  where f_s is the shedding frequency, S_t is the Strouhal number, U is the current speed, and D is the diameter of the pipe.
- The effect of internal flow on the correlation coefficient of cross-flow vibration and in-line vibrations between sections of the riser model is more obvious with higher relative internal flow speed. The correlation efficient of cross-flow vibration and in-flow vibration between sections decreases when the internal flow speed increases.

2.1.2 Thermal impact of the products pipeline on the crude oil pipeline laid in one ditch

Crude oil basically can be classified into four classes which are Class A: light and volatile oils, Class B: non-sticky oils, Class C: heavy oils and finally Class D: non fluid oils. More than 80% of crude oils produced in China are Class C crude oil which either waxy crude oil with high pour points or viscous heavy crude oil, whose flowability is poor. Bo Yu said that one of the effective ways to transport the poor flowability crude oil in the pipelines is to heat it at the station [2]. A new technology of laying two pipelines in one ditch appears as before 2005, the crude oil pipelines were constructed independently in one ditch. Yi Wang mentioned the temperature of the crude oil is a key parameter for safe transportation, the most crucial problem in the design and operation of the double pipelines laid in one ditch is the thermal impact on the hot crude oil pipeline [2].

Jinjun Zhang mentioned the complete thermal system of the buried pipelines should contain the convective heat transfer of the oil in the pipelines and the heat conduction outside the pipelines [2]. The balance of heat flux is used to couple the convective heat transfer in the pipeline and the soil heat conduction. A numerical study on the heat transfer of the oils, wax deposition, steel pipes, corrosion protective covering and soil is to be carried out. The mass conservation equation is as below:

$$\frac{\partial}{\partial \tau}(\rho A) + \frac{\partial}{\partial z}(VA) = 0 \qquad \text{(Equation 2.2)}$$

The momentum conservation equation:

$$\frac{\partial V}{\partial \tau} + V\frac{\partial V}{\partial z} = -g \sin \propto - \frac{1}{\rho}\frac{\partial p}{\partial z} - \frac{f}{D}\frac{V^2}{2} \qquad \text{(Equation 2.3)}$$

The energy conservation equation:

$$\frac{\partial V}{\partial \tau}\left[(\rho A)\left(u + \frac{V^2}{2} + gs\right)\right] + \frac{\partial}{\partial z}\left[(pVA)\left(h + \frac{V^2}{2} = gs\right)\right] = -\pi D q_o \qquad \text{(Equation 2.4)}$$

The heat transfer equation of the oil flow can be obtained from the three equations listed above:

$$C_p \frac{dT}{d\tau} - \frac{T}{\rho}\beta\frac{dp}{d\tau} - \frac{fV^3}{2D} = -\frac{4q_0}{\rho D} \qquad \text{(Equation 2.5)}$$

where q_0 represents the axial heat flux density of the oil flow, and it also stands for the heat loss of the oil flow on the cross-plane of the pipeline [2].

The heat conductive equations of the wax deposition, pipeline wall and corrosion protective covering are listed below:

$$\rho_i C_i \frac{\partial T_i}{\partial \tau} = \frac{1}{r}\frac{\partial}{\partial r}\left(\lambda_i r \frac{\partial T_i}{\partial r}\right) + \frac{1}{r^2}\frac{\partial}{\partial \theta}\left(\lambda_i \frac{\partial T_i}{\partial \theta}\right) \quad i = 1,2,3 \qquad \text{(Equation 2.6)}$$

with boundary condition:

at r = D/2,

$$\lambda_i \frac{dT_1}{dr} = -\propto_0 (T - T_0) \qquad \text{(Equation 2.7)}$$

The heat conductive equation of the soil is as follow:

$$\rho_s C_s \frac{\partial T_s}{\partial \tau} = \frac{\partial}{\partial x}\left(\lambda_s \frac{\partial T_s}{\partial x}\right) + \frac{\partial}{\partial y}\left(\lambda_s \frac{\partial T_s}{\partial y}\right) \qquad \text{(Equation 2.8)}$$

with boundary conditions:

At $y = 0,\ \lambda_s \frac{dT_s}{dy} = \propto_a (T_a - T_s)$

At $y = -H,\ \ T_s = T_n$

At $x = \pm L,\ \frac{\partial T_s}{\partial x} = 0$

Zhengwei Zhang mentioned that the existence of the products pipeline changes the soil temperature field on one side of the crude oil pipeline and changes the heat transfer [2]. That is to say when there is only a single crude oil pipeline, the heat is totally absorbed by the soil. When the products pipeline coexists in one ditch with the crude oil pipeline, the heat is partly absorbed by the soil and partly absorbed by the products oil. The gradient of the soil temperature on the very side of the products pipeline decreases and the heat loss to the environment lessens.

Kai Wang concluded that the temperature drop is not notable when the pipeline interval is not less than 1.2m. However, the temperature decreases a lot when the pipeline interval is less than 1.2m. Therefore, generally speaking the pipeline interval more than 1.2m is relatively safe for the pipeline operation [2].

2.1.3 A finite element solution of wave forces on submarine pipelines

Submarine pipelines serving as media for transporting offshore oil and gas to land are widely used in the practical engineering. They are placed on the ocean floor with different ways such as bottom seated or buried under the seabed. The forces on these pipelines are influenced by many factors, namely the position of the pipe with respect to bottom boundary and free surface, water depth, marine growth, and wave-current conditions. The design of these pipelines requires a careful prediction of the wave forces acting on them.

Forces on a submerged cylinder or a pipeline due to waves have been investigated by many researchers and the main methods are experiments. The hydrodynamic forces on cylinders under the action of waves are usually evaluated using the Morison's equations. This equation involves two hydrodynamics coefficients of drag C_D, and inertia C_M, which have to be necessarily determined from experiments [3]. Sarpkaya has tested the hydrodynamic coefficients of CD, CM, CL with different clearances between a cylinder and a plane boundary in a sinusoidally oscillating fluid [4].

The numerical model for simulating wave forces on a circular cylinder is rare. The two-dimensional Navier-Stokes equations for an incompressible flow are used as governing equations. For simplification of numerical procedure, the pressure p is dvided into static pressure p_0 and the pressure die to waves p_w, namely

$$p = p_0 + p_w \qquad \text{(Equation 2.9)}$$

where the static pressure is defined as $p_0 = -\rho g y$, g being the gravitational acceleration.

After the division procedure of the pressure, the dimensional Navier-Stokes equations can be written as

$$\frac{qu}{qt} + u\frac{qu}{qx} + v\frac{qu}{qy} + \frac{1}{\rho}\frac{qpw}{qx} = v\left(\frac{q^2u}{qx^2} + \frac{q^2u}{qy^2}\right), \qquad \text{(Equation 2.10)}$$

$$\frac{qv}{qt} + u\frac{qv}{qx} + v\frac{qv}{qy} + \frac{1}{\rho}\frac{qpw}{qx} = v\left(\frac{q^2v}{qx^2} + \frac{q^2v}{qy^2}\right), \qquad \text{(Equation 2.11)}$$

$$\frac{qu}{qs} + u\frac{qv}{qy} = 0, \qquad \text{(Equation 2.12)}$$

where u and v are the velocity components in the x and the y directions respectively; v = 1.0 x 10^{-6} m^2/s is the kinematic viscosity of water.

As for the boundary conditions, the free surface is controlled by the kinematic boundary condition

$$\frac{\partial n}{\partial t} + u\frac{\partial n}{\partial x} = v \qquad \text{(Equation 2.13)}$$

where n is the wave surface elevation. The non-shear stress condition in the tangential direction of the surface requires

$$\frac{\partial n u_s}{\partial n} = 0 \qquad \text{(Equation 2.14)}$$

where u_s is the velocity component in the tangential direction of the boundary, and n is the normal direction of the wave surface.

In the case of irregular waves, it is difficult to determine the phase velocity at the outlet boundary. Wei Bai and Liang Zheng have set a spongy layer in front of the outlet boundary to absorb the wave energy [3]. When waves pass through the spongy layer, nearly all the energy is absorbed. For the implementing of the spongy effect, after each time step, the velocity and the surface elevation of the nodes in spongy layer are divided by the spongy coefficient. The spongy coefficient is expressed as

$$\mu(x, y) = exp\left[\left(a^{-S/\Delta S} - a^{-W/\Delta S}\right) \ln b\right],$$

In which the μ is the spongy coefficient, S is the distance from the nodal point to the open boundary, ΔS is the size of gird, W is the thickness of the spongy layer which is about twice the incident wave length, and a and b are two constants, which are set 1.11 and 5.0 by the researchers respectively [3].

A two-dimensional non-linear viscous numerical wave tank has been established for simulating the interaction between a submarine pipeline and the seabed. The Navier-Stokes equations are discretized in a moving mesh system by the finite element method. The deffered correction second-order upwind scheme is employed for discretizing the convective fluxes. The present model appears to work well and the results of calculations for the wave forces on a submarine pipeline are compared with the experimental results and potential theory value. The effects of clearance between the pipeline and seabed, wave height and water depth on wave forces acting on a submarine pipeline are studied [3].

The main conclusions are:

1)

Table 2.1: Relationship between C_D and C_M with respect to *E/D*

e/D (gap to diameter ratio)	C_D (Drag Coefficient)	C_M (Inertia Coefficient)
< 0.2	Increases with *e/D* increasing	Decreases with *e/D* increasing
0.2 < x < 0.5	Decreases with *e/D* increasing	
> 0.5	Nearly maintained at a constant	

From Table 2.1 above, it is shown that the effect of the seabed on the horizontal wave force is not significant.

2) The horizontal force increases linearly with the wave height parameter *H/2a* increasing with decreases with the water depth parameter *d/a* increasing, within the range of calculations.
3) The vertical force increases non-linearly with *H/2a* increasing, and decreases with *d/a* increasing. The non-linear variation becomes pronounced at larger value of *H/2a* or smaller value of *d/a* or both.

where *e/D* is defined as the gap-to-diameter ratio, is the considered parameter and ranges from 0.1 to 1.5. *e* is the clearance between the pipeline and the seabed whereas *D* is the diameter of the pipeline.

2.1.4 Hydrodynamic forces on marine pipelines including waves and currents

One of the major tasks in the design of submarine pipelines is the analysis of the hydrodynamic stability of the pipeline. This analysis is important to ensure that during the construction and operation stages, the pipeline will remain stable under the action of the hydrodynamic forces produced by the waves and currents. Sabag said that in order to reach this stability, the horizontal and lift forces are balanced against the minimum submerged weight of the pipeline [5]. The gravitational and friction forces act together to resist the hydrodynamic forces of the waves and currents.

Edge and Sabag mentioned for a pipeline resting on the sea bed, the total forces acting on the pipeline are W, total submerged weight of pipe including concrete coating and wrap; F_d, F_I, and F_L, drag, inertia and lift force; N, normal force and F_r, friction resistance [5].

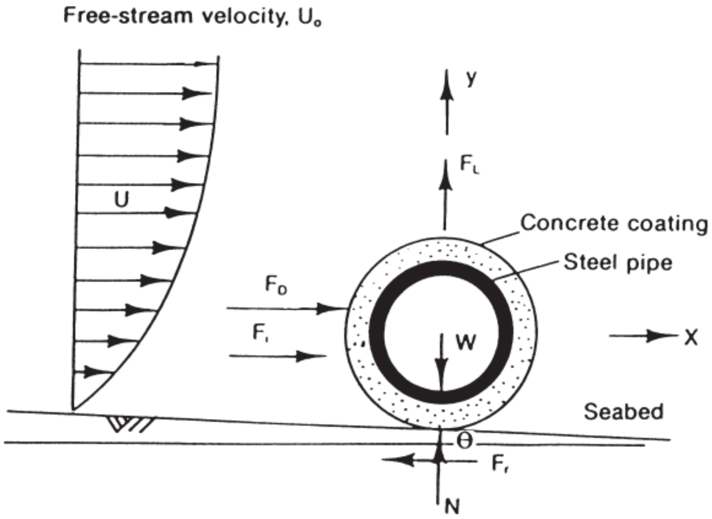


Figure 2.1: Hydrodynamic forces on pipe

In order to incorporate in the model the wake velocity behind the cylinder and time dependent hydrodynamic coefficients, Lambarokos (1987) proposed a model, the Wake I Force Model which uses time-dependent drag and lift coefficients [5].

The expressions for the drag, lift and inertial forces are:

$$F_D = 0.5\rho D C_D(t)|U_e|U_e \quad \text{(Equation 2.15)}$$

$$F_L = 0.5\rho D C_L(t)U_e^2 \quad \text{(Equation 2.16)}$$

$$F_I = \frac{\pi D^2}{4}\rho\left[C_M\frac{\mathrm{d}U}{\mathrm{d}t} - C_{AW}\frac{\mathrm{d}U_w}{\mathrm{d}t}\right] \quad \text{(Equation 2.17)}$$

where ρ is water mass density, D is pipe diameter, $C_D(t)$ and $C_L(t)$ are time dependent drag and lift coefficients. The horizontal force is the sum of F_D, and F_I. C_M is the inertia coefficient for the ambient flow. C_{AW} is the added mass coefficients associated with the wake flow passing the pipe.

Soedigdo found that Wake I Force Model is not accounted for by the conventional force model where the lift force shows a large phase difference relative to the velocity and the hydrodynamic forces in a give velocity half cycle depend strongly on the velocity magnitude in the preceding half cycle [5]. He has developed the Wake II Force Model, which is a continuation of the Wake I Force Model, is based upon a closed form correction by solving the linearized Navier-Stokes equation for oscillatory flow. It assumes that the eddy viscosity in the wake is time dependent and of a harmonic sinusoidal form. The wake velocity correction affecting the pipe in periodic flow, according to the Wake II Force Model is:

$$U_w = \frac{\sqrt{\pi} erf\left[\frac{1}{2} C_2 sin^n(\omega t + \emptyset)\right] U_m C_1}{C_2} \quad \text{(Equation 2.18)}$$

Where U_w is the wake maximum velocity correction affecting the pipe in periodic flow, U_m is the peak velocity in present half cycle, C_1, C_2, $\emptyset$ and n are empirical parameters that are determined from comparisons with field data. C_1 and C_2 decay of the wake velocity correction, $\emptyset$ is the phase angle and n is the exponent that determines the sharpness of the wake velocity correction. The wake correction is used to modify the velocity in the calculation of the hydrodynamic forces.

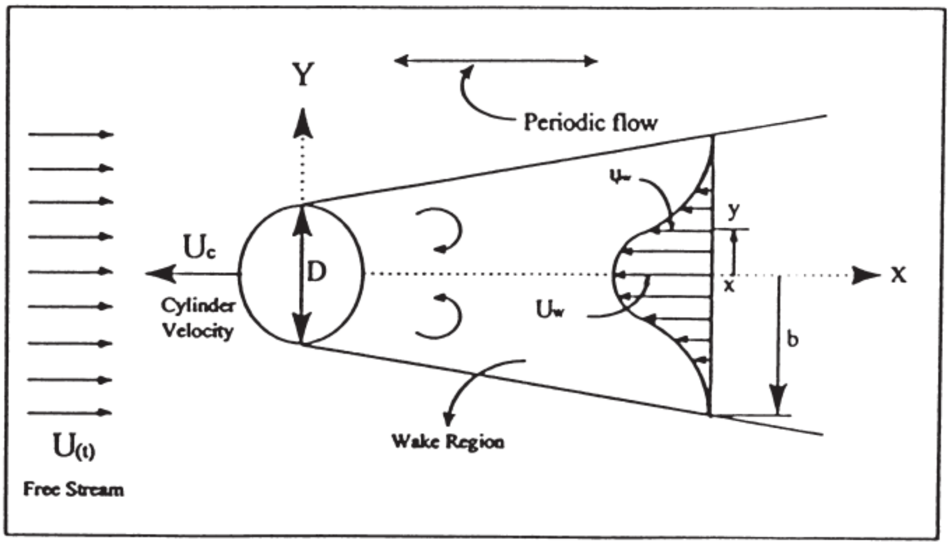


Figure 2.2: Wake flow parameters for a cylinder in harmonic oscillatory flow in space [7]

Sabag and Edge [7] mentioned the concept of the Wake II Model for hydrodynamic forces has been extended to include the case of waves plus steady currents. This model gives satisfactory results when applied to wave plus current cases and offers a substantial improvement over predictions with the conventional model. In the measurement of horizontal forces, the Wake II Model gives very accurate results for all cases and there is a significant improvement when compared with the conventional model although the conventional model gives satisfactory results in predicting the force peaks.

2.2 Analytical method for stress analysis of pipelines

Stress analysis of pipelines is now fairly established such that commercial computer packages are available to accomplish this purpose. However due to the complexity of loading on pipelines, an exact method of stress analysis is still open to research and the current methods, such as [8], neglect the effect of shearing and axial forces on the design section. A. Nourbakhsh mentioned that neglecting the effect of shearing forces in stress analysis is a common practice among mechanical engineers [11]. Shearing force complicates the stress analysis to a large extend such that the problem cannot be solved unless by the application of computer simulation techniques, such as finite element methods. K. Abhary proved that, under certain conditions, this may introduce up to about 50% error in the maximum stress of around structural member [11].

An exact analytical method has been developed which takes not only the shearing force, but also the axial force into account. This method leads to a trigonometric polynomial equation of the fourth order, which can be solved easily by any commercial, or specifically developed mathematical computer program. A study carried out on the state of stresses at an arbitrary point of pipeline shows that the maximum value of axial stress, σ_a and shear stress, τ is on the external surface of the pipe, unlike hoop stress, σ_h and radial stress, σ_r whose maximum values are on the internal surface of the pipe. The study is further researched with maximum-distortion –energy (Mises-Henckey) theory as the failure criterion which leads to a 4th order polynomial equation below.

$$(B-D)X^4 + 2(C-2A)X^3 - 6BX^2 + 2(2A+C)X + (B+D) = 0 \qquad \text{(Equation 2.19)}$$

where

$$A = 2(K_b M)^2 \cos 2\alpha - 1.5b^2 \cos 2\beta$$

$$B = -2(K_b M)^2 \sin 2\alpha - 1.5b^2 \sin 2\beta$$

$$C = K_b M[4a + (p-c)/m]\sin\alpha - 3bK_t T \sin\beta$$

$$D = K_b M[4a + (p-c)/m]\cos\alpha - 3bK_t T \cos\beta$$

Then the study is continued with upper-bound approach where all terms in the equation is arranged in positive terms shown in the equation below

$$\sigma_P = \left[(\sigma_a - \sigma_h)^2 + \sigma_a\sigma_h + 3\tau^2 + p(\sigma_a + \sigma_h) + p^2\right]^{\frac{1}{2}} \qquad \text{(Equation 2.20)}$$

from the original equation

$$\sigma_P = \left[\sigma_a^2 + \sigma_h^2 - \sigma_a\sigma_h + 3\tau^2 + p(\sigma_a + \sigma_h) + p^2\right]^{\frac{1}{2}} \qquad \text{(Equation 2.21)}$$

The maximum upper-bound stress in the pipeline is then overestimated by the greatest value of the upper-bound stresses on all of the nominated design sections. This approach leads to a very rapid stress analysis of the pipeline network because it does not consider lower-bound stresses in the calculation. Thus, lesser time required to complete the analysis. A compromise between the above two approaches, namely the exact and the upper-bound, is to determine the greatest upper-bound stress in the network and then determine the exact maximum stress on its corresponding cross section.

L. L. H. S. Loung said the analytical method developed above helps designers determine accurately and rapidly the maximum stress in a pipeline network with just any simple computer program capable of solving a polynomial of the fourth order [11].

The analytical method developed above is not suitable to be used in industry applications when compared to finite element package. This is due to many pre-requisite calculations need to be completed prior using equation above to generate the required stress data on the pipelines. With finite element package, stress values generated include all the loading conditions input into the software but with analytical method, all respected stress (eg. Bending stress due to dead load, live load, current and wave load) must be calculated manually before input into the analytical equation developed above. Errors may occurred during the process of manual calculation, where engineers with high understanding of the equations are required to perform the process.

CHAPTER 3
METHODOLOGY

This section of the report describes the methodology being applied to accomplish the objectives of this project and it is summarised in the flow chart below.

3.1 Preliminary Research and Literature Review

The strong understanding in engineering theory of riser stress analysis must be achieved at early stage of the project. It has helped the author understand the objectives of the project and the direction heading to. Factors affect stress analysis output must be researched before commence the finite element analysis. This can be accomplished by undergo literature review through sources available. The sources available in the campus include online resources, internet and books collection in Information Resource Centre.

3.2 Finite Element Analysis

Finite element analysis is performed with ANSYS software after the completion of all data gathering. The analysis is involving the simulation of riser stress under four loading conditions mentioned in the scope of study. The result later is compared with the result generated from AutoPIPE, commercial finite element software using in pipeline design industry to determine the success rate of the project.

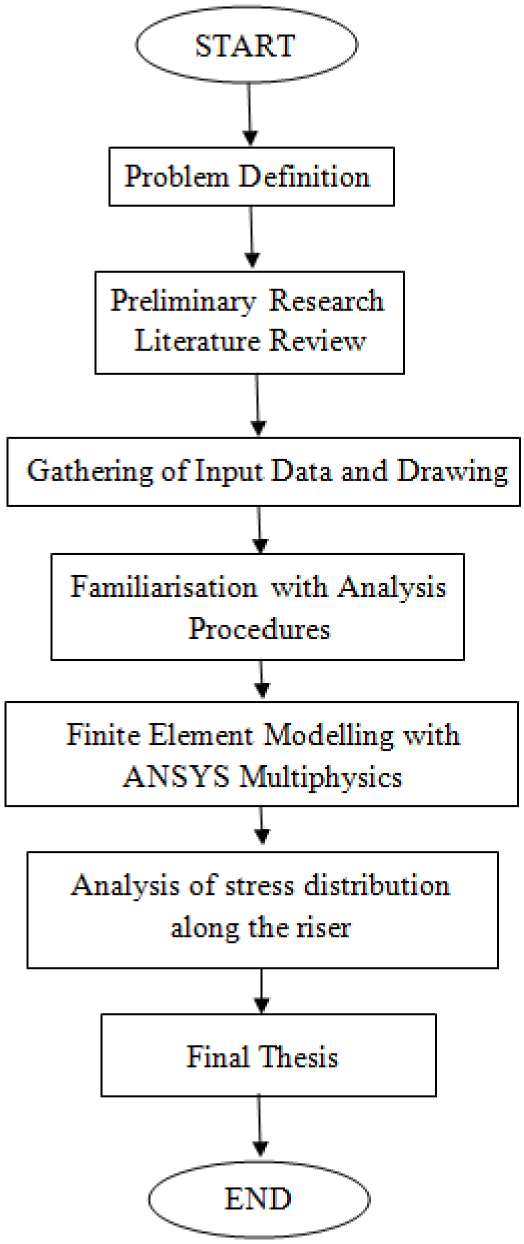


Figure 3.1: Flow Chart

3.3 Gantt Chart

July 2008 Semester

Table 3.1: Gantt Chart

No.	Detail/Week	1	2	3	4	5	6	7	8	9	10		11	12	13	14
1	Selection of Project Topic	■	■									Mid-Semester Break (27 September 2008 – 7 October 2008)				
2	Preliminary Research Work		■	■	■											
3	Submission of Preliminary Report				●											
4	Project Work					■	■	■	■	■	■					
5	Submission of Progress Report								●							
6	Seminar (compulsory)								●							
7	Project Work Continue												■	■	■	■
8	Submission of Interim Report Final Draft														●	
9	Oral Presentation															●

January 2009 Semester

Table 3.2: Gantt Chart

No.	Detail/Week	1	2	3	4	5	6	7	8	9		10	11	12	13	14	15	16	17
1	Project Work Continue	■	■	■							Mid-Semester Break (21 March 2008 – 29 March 2008)								
2	Submission of Progress Report 1				•														
3	Project Work Continue				■	■	■	■	■	■		■							
4	Submission of Progress Report 2								•										
5	Seminar (compulsory)								•										
6	Project Work Continue												■	■	■				
7	Poster Exhibition											•							
8	Submission of Dissertation (soft bound)																•		
9	Oral Presentation																	•	
10	Submission of Project Dissertation (hard bound)																		•

The Gantt charts above represent the schedule to be followed throughout the final year project. During the July 2008 semester, the first two weeks were allocated to select a desired final year project title with lecturers in Mechanical department. As shown in Table 3.1, week 2 to week 4 was allocated to conduct the preliminary research and literature review of this final year project. In this activity, author has gained the necessary engineering knowledge related to project title. As instructed by Final Year Project coordinator, week 4 is due date for preliminary report submission while week 8 is the submission date for progress report. The project work is scheduled to start on week 5 till the end of the semester as shown in the table above. The 10 weeks time frame is allocated to proceed with the main part of the project, which includes obtaining a strong understanding of factors, affecting riser stress analysis output and familiarise with the design capability of ANSYS software. Interim report is scheduled to submit on week 13 while oral presentation to be conducted on the final week of the semester. In the Gantt chart for January 2008 semester as shown in Table 3.2, week 1 to week 13 was allocated to complete the finite element analysis of this project by using ANSYS software. In this activity, author has gained the necessary engineering knowledge and software skills required to achieve the objectives of this project. As instructed by Final Year Project coordinator, week 4 is due date for progress report 1 submission while week 8 is the submission date for progress report 2. The project work is scheduled to complete by week 11 of the semester as shown in the table above. The 11 weeks time frame are adequate to complete the project based on the objectives and scope of study, and comparing the stress output with the result generated by AutoPIPE software. Poster exhibition is scheduled to be held on week 10, while the oral presentation on week 16. Soft bound version of dissertation is scheduled to be submitted on week 15 while hard bound version of dissertation on final week of the semester.

3.4 ANSYS Multiphysics

3.4.1 Introduction

ANSYS Multiphysics provides the analysis industry's most comprehensive coupled physics tool combining structural, thermal, CFD, acoustic and electromagnetic simulation capabilities into a single software product. A broad range of applications involve everything from rotating machines (motors and alternators), sensors and actuators, power generators and transformer systems, and Micro Electro Mechanical systems (MEMS) can be completed with ANSYS Multiphysics. This software is a general purpose analysis tool allowing a user to combine the effects of two or more different, yet interrelated physics, within one, unified simulation environment [15].

ANSYS Multiphysics provides two methods to couple multiple physics together which are Direct and Sequential:

- Direct – solves all DOFs at the FEA coefficient matrix level.
- Sequential – solves DOFs for one physics then passes results at loads and boundary conditions to the second physics. At least two iteration, one for each physics, in sequence, are needed to achieve a coupled response.

In this project, Sequential Coupled Physics is method chosen as structural, thermal and fluid were the loads and boundary conditions applied on the shallow water rigid riser.

The purpose of this project is to study whether the latest version of ANSYS could churn out solutions to examine the stress distribution on the riser in 3-dimensional view which is not possible with Bentley AutoPIPE. Bentley AutoPIPE only able to generate stress data in numerical values only for validation task. The stress magnitude generated with ANSYS Multiphysics is to be compared with the result generated by Pipeline Engineer – Ezhani Esa using Bentley AutoPIPE. The result generated by her is proved to be dependable as the source of comparison for this final year project, as it is inspected and certified by a 3rd party inspection company appointed by the client of the project. Considering that the result generated using Bentley AutoPIPE is accepted by client as the data to be referred for construction stage in the future, it is safe to say the result from Bentley AutoPIPE is highly dependable to represent the experimental data for this final year project.

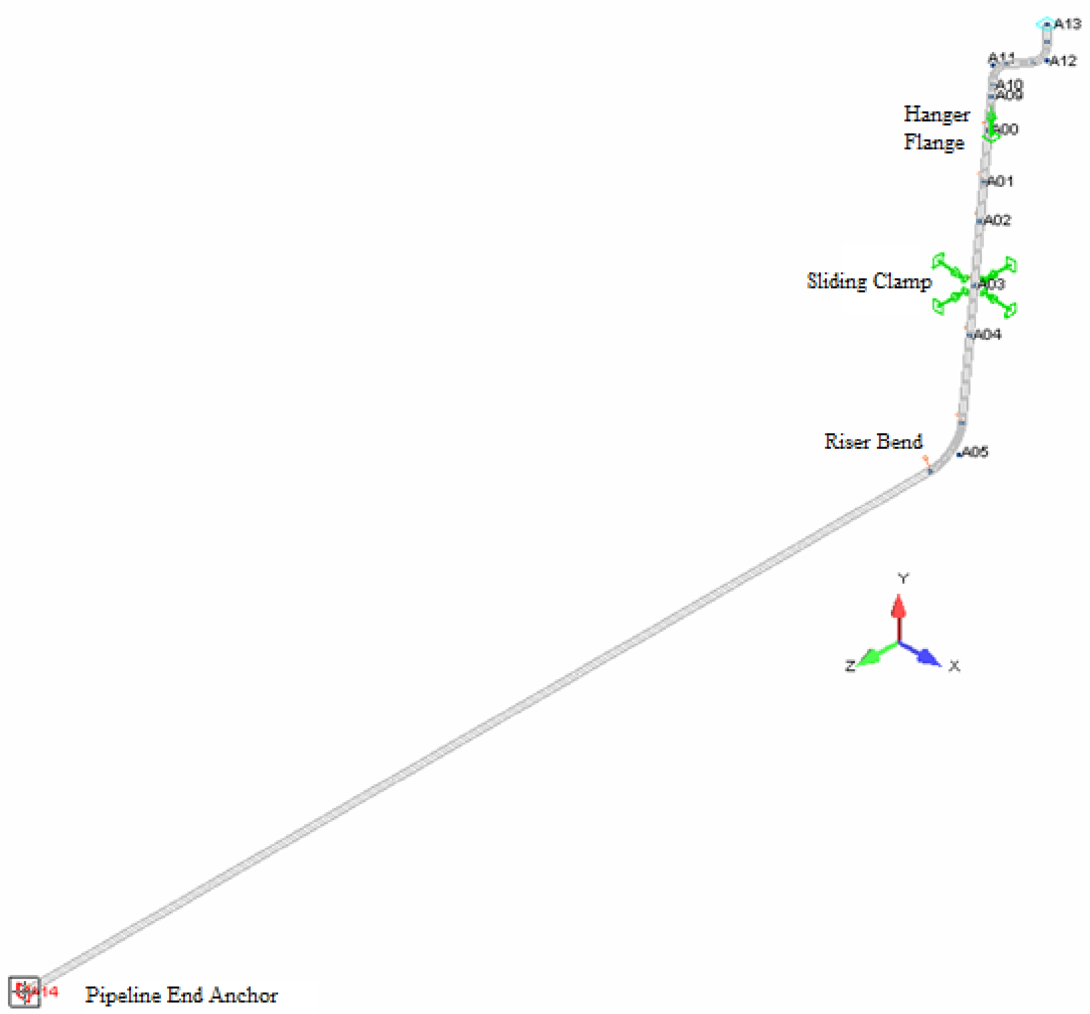


Figure 3.2: Pipeline 3-Dimensional Modelling in Bentley AutoPIPE

PIPE59 Element in ANSYS Multiphysics element library is chosen as the element for this project. PIPE59 is a uniaxial element with tension-compression, torsion, and bending capabilities, and with member forces simulating ocean waves and current. Unfortunately, when applying hanger flange and sliding clamp constraint on the riser with ANSYS Multiphysics, several limitations are in place, which degrades the accuracy of the prediction to the point that no directly observable correlation can be made with respect to the experimental results. In other words, the hanger flange and sliding clamp generated in Bentley AUTOPIPE cannot be modelled by the ANSYS

Multiphysics. The method used to replace the hanger flange and sliding clamp in ANSYS Multiphysics is by defining load constraints on the nodes involved.

3.4.2 Design Assumptions

It needs to be pointed out that due to several limitations in finite element analysis in simulating real life condition, several design assumptions need to be applied on the riser stress analysis.

i. The riser model has included a straight pipeline section (100m) to simulate the thermal expansion effect at riser/pipeline interface. The pipeline end is modelled as a fully constraint node imposed with expansion forces derived from end expansion study.
ii. Hanger flange is modelled as fully constraint at all DOFs. Sliding clamp is modelled as fully constraint at X and Y planes with no translational movements in the horizontal place. It is assumed that these restraint conditions define the hanger flange and sliding clamp behaviour in actual situation.
iii. Riser is filled with homogeneous maximum product density during operating case.
iv. The minimum seawater temperature is taken as the system ambient and pressure test temperature for analysis.
v. Wind loading is not considered.
vi. Waves and currents are assumed to be collinear.
vii. Maximum platform movements coincide with the wave direction.

As mentioned earlier, PIPE59 is element chosen to model the riser system and below are the assumptions and restrictions of this element:

i. The pipe must not have a zero length. In addition, the outer diameter must not be less than or equal to zero and the internal diameter must not be less than zero.
ii. Elements input at or near the water surfaces should be small in length relative to the wave length.

iii. Neither end of the element may be input below the mud line (seabed). Integration points that move below the mud line are presumed to have no hydrodynamic forces acting on them.
iv. If the element is used out of water, the water motion table need not be included.
v. The element should also be used with caution in the reduced transient dynamic analysis since this analysis type ignores the element load vector. Fluid damping, if any, should be handled via the hydrodynamic load vector than α (mass matrix) damping.
vi. The applied thermal gradient is assumed to vary linearly along the length of the element.
vii. The same water motion table should not be used for different wave theories in the same problem.

3.4.3 Modelling Methodology

Finite element analysis with ANSYS Multiphysics in this project has been divided into three stages which are:

- Pre-processing
- Solution
- Post-processing

3.4.3.1 Pre-processing

The goals of pre-processing are to develop an appropriate finite element mesh, assign suitable material properties, and apply boundary conditions in the forms of restraints and loads.

PIPE59 Element is chosen as the main element for this project as it is the only element available in ANSYS Multiphysics element library which can simulates the condition of an immersed pipe. The real constants parameter for PIPE59 such as outside diameter, wall thickness, coefficient of inertia and internal fluid density are input into the element library. The modelling stage is started by using pipe modelling option in ANSYS Multiphysics; all the input parameters (pipe geometries and properties, coating type and thickness, product density) are stored within the elements between nodes. There is no necessity to perform finite element mesh as nodes and elements already created.

After completing the modelling stage, it is essential to define the material properties of this project. The pipe material properties (density, young modulus and Poisson's ratio), viscosity of the seawater and hydrodynamics data are defined into the material library. Pipe material properties must be input in order to allow ANSYS Multiphysics to calculate the submerged weight of the riser itself. The input of seawater viscosity has helped the software to calculate Reynolds number at different water depth. Then, hydrodynamics data is input into the Water Motion Table. This has defined the current and wave loadings acting on the riser from the water surface to the mud line (seabed).

Boundary conditions must be applied on the riser in order to complete the pre-processing stage where equivalent constraint similar to hanger flange, sliding clamp, anchor point and seabed are defined.

3.4.3.2 Solution

While the pre-processing and post-processing phases of the finite element method are interactive and time-consuming for analyst, the solution is often a batch process, and is demanding of computer resource. The governing equations are assembled into matrix form and are solved numerically. The assembly process depends not only on the type of analysis, but also on the model's element types and properties, material properties and boundary conditions.

In this project, static is selected as the type of analysis. In Solution Controls, author can select necessary items to be included in the results file, which can be output to tables in post-processing section.

3.4.3.3 Post-processing

After a finite element model has been prepared and checked, boundary conditions have been applied and the model has been solved, it is time to investigate the results of the analysis. This activity is known as post-processing phase of the finite element method.

Post-processing begins with a thorough check for problems that may have occurred during solution. Most solvers provide a log file, which should be searched for warning and errors. As mentioned in PIPE59 element manual, it is required to define an element table for the types of stress that wanted to be included PIPE59 Stress Output table. Once the solution is verified to be free of numerical problems, the quantities of interest may be examined. Stress output can be obtained by listing all the element tables defined earlier.

3.5 ANSYS Multiphysics PIPE59 Equations

In this subsection, all the equations below are used in PIPE59 element – ANSYS Multiphysics to calculate the stress generated along the riser due to various loading conditions. PIPE59 is the element commonly used to model immersed pipe or cable. PIPE59 is similar to PIPE16 with the exception of two principal differences which are the mass matrix includes the:

a. Outside mass of the fluid (acts only normal to the axis of the element)
b. Internal structure components (pipe option only)
    i. Hydrostatic effects
    ii. Hydrodynamic effects

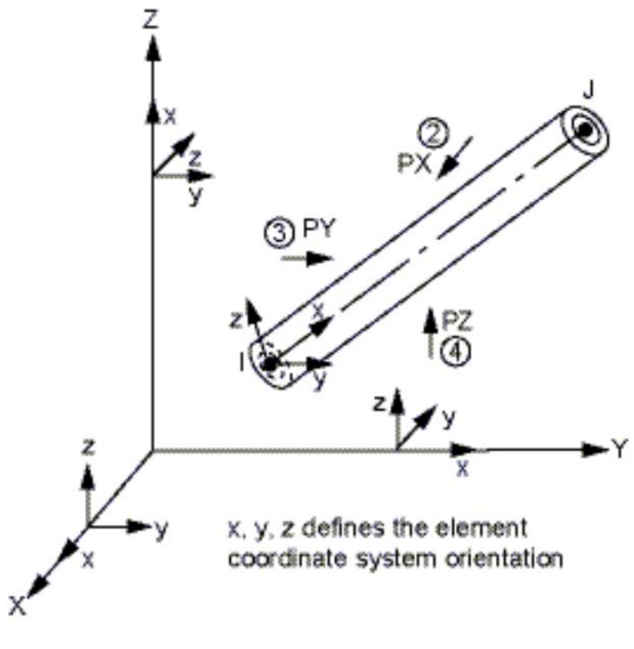


Figure 3.3: PIPE59 Geometry [6]

3.5.1 Location of the Element

The origin for any problem containing PIPE59 must be at the free surface (mean sea level). Further, the Z axis is always the vertical axis, point away from the centre of earth.

The element may be located in the fluid, above the fluid, or in both regimes simultaneously. There is a tolerance of only $\frac{D_e}{8}$ below the mud line, for which

$$D_e = D_0 + 2t_i \qquad \text{(Equation 3.1)}$$

where

t_i = thickness of external insulation

D_o = outside diameter of pipe/cable

3.5.2 Load Vector

The element load vector consists of two parts:

1. Distributed force per unit length to account for hydrostatic (buoyancy) as well as axial nodal forces due to internal pressure and temperature effects.
2. Distributed force per unit length to account for hydrodynamic effects (current and waves)

The hydrostatic and hydrodynamic effects work with original diameter and length while the conditions such as initial strain and large deflection effects are not considered.

where: $\{F/L\}_b$ = vector of loads per unit length due to buoyancy

C_b = coefficient of buoyancy

$\{g\}$ = acceleration vector

3.5.3 Hydrodynamic Effects

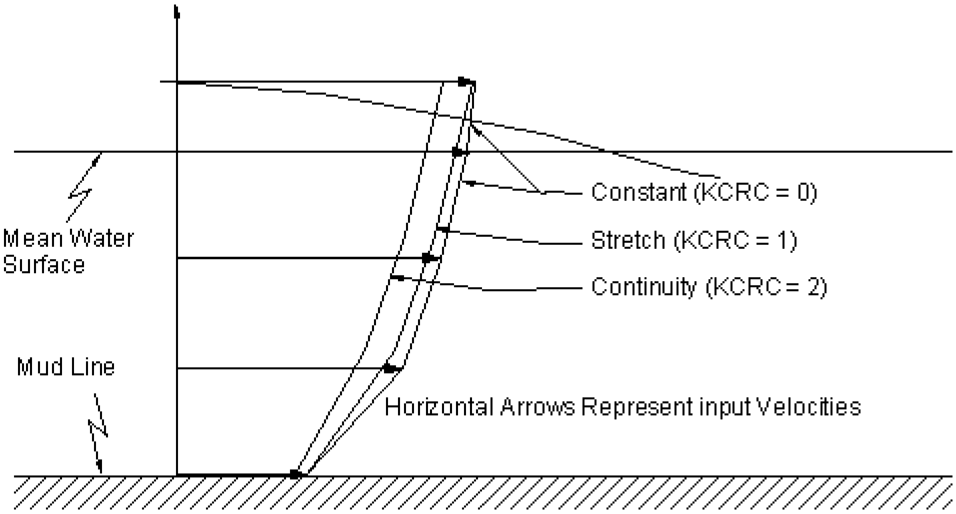


Figure 3.4: Velocity Profiles for Wave-Current Interactions [6]

It is necessary to compute the relative velocities as both the fluid particle velocity and the structure velocity must be available so that one can subtracted from the other. Finally, a generalized Morison's equation is used to compute a distributed load on the element to account for the hydrodynamic effects:

$$\left\{{}^{F}/_{L}\right\}_{d} = C_D \rho_w \frac{D_e}{2} |\{u_n\}|\{u_n\} + C_M \rho_w \frac{\pi}{d} D_e^2 \{v_n\} + C_T \rho_w \frac{D_e}{2} |\{\dot{u}_t\}|\{\dot{u}_t\} \qquad \text{(Equation 3.2)}$$

where:

$\{F/L\}_d$ = vector of loads per unit length due to hydrodynamic effects

C_D = coefficient of normal drag

ρ_w = water density (mass/length^3)

D_e = outside diameter of the pipe with insulation (length)

$\{\dot{U}_n\}$= normal relative particle velocity vector (length/time)

CM = coefficient of inertia

$\{\dot{V}_n\}$= normal particle acceleration vector (length/time^2)

C_T = coefficient of tangential drag (see below)

$\{\dot{U}_t\}$= tangential relative particle velocity vector (length/time)

CHAPTER 4
RESULT & DISCUSSION

In this chapter of the report, author will elaborate on all the input parameters and experimental data gathered, and the results of the simulations executed.

4.1 Data Gathering

Data gathering of this final year project is completed by obtaining the required input parameters and experimental data from previous industrial internship company. The input parameters gathered are essential because it serves as the input data for the pre-processing stage in the finite element analysis.

The input parameters gathered basically can be divided three main parts as below:

- Engineering Design Input Parameters
- Engineering Drawing
- Experimental Data

4.1.1 Engineering Design Input Parameters

In this project, the selected riser to be involved in this stress analysis is with the external diameter of 24 inch. The riser connects the pump platform to a pipeline at the ends at shore. The riser is carrying sour hydrocarbon gas as product with the total flowrate of 47 MMscfd.

The tables shown below are the necessary input parameters gathered which consist of:

- Design Process Data
- Design Mechanical Data
- APL 5L Steel Material Properties

Table 4.1: Design Process Data

PARAMETER	24” PIPELINE
ANSI / ASME Class Rating	300#
Maximum Design Pressure (1)	400 psig (2758 kPag)
Maximum Design Temperature (2)	95°C (above water)
	65°C (under water)
Minimum Design Temperature	0 deg C
Operating Pressure	126 psig (868.7 kPag)
Operating Temperature	34 deg C
Maximum Contents Density	10.5 kg/m^3
Minimum Contents Density	8 kg/m^3

Table 4.2: Design Mechanical Data

PARAMETER	24” PIPELINE
From / To	Pump Platform /Shore
Length	40.34 km
Pipeline Outside Diameter,	24 inch
Internal Corrosion Allowance	3.2mm
External Corrosion Allowance	0.0mm
Material Standard / Grade	API 5L X-52
External Corrosion Coating	FBE
External Corrosion Coating Thickness	0.5mm
Riser Splash Zone Coating	Monel
Riser Splash Zone Coating Thickness	0.25mm

Table 4.3: API 5L Steel Material Properties

PARAMETER	API 5L – X52
Steel Density	7850kg/m^3
Modulus of Elasticity	207GPa
Poisson’s Ratio	0.3
Coefficient of Thermal Expansion	11.7×10^{-6}
Thermal Conductivity	45 W/m°K
SMYS	359Mpa
Ultimate Tensile Strength	455Mpa

The three tables above show the necessary input parameters for the pre-processing stage in the finite element analysis. The appropriate data is chosen based on the design assumptions made.

4.1.2 Engineering Drawing

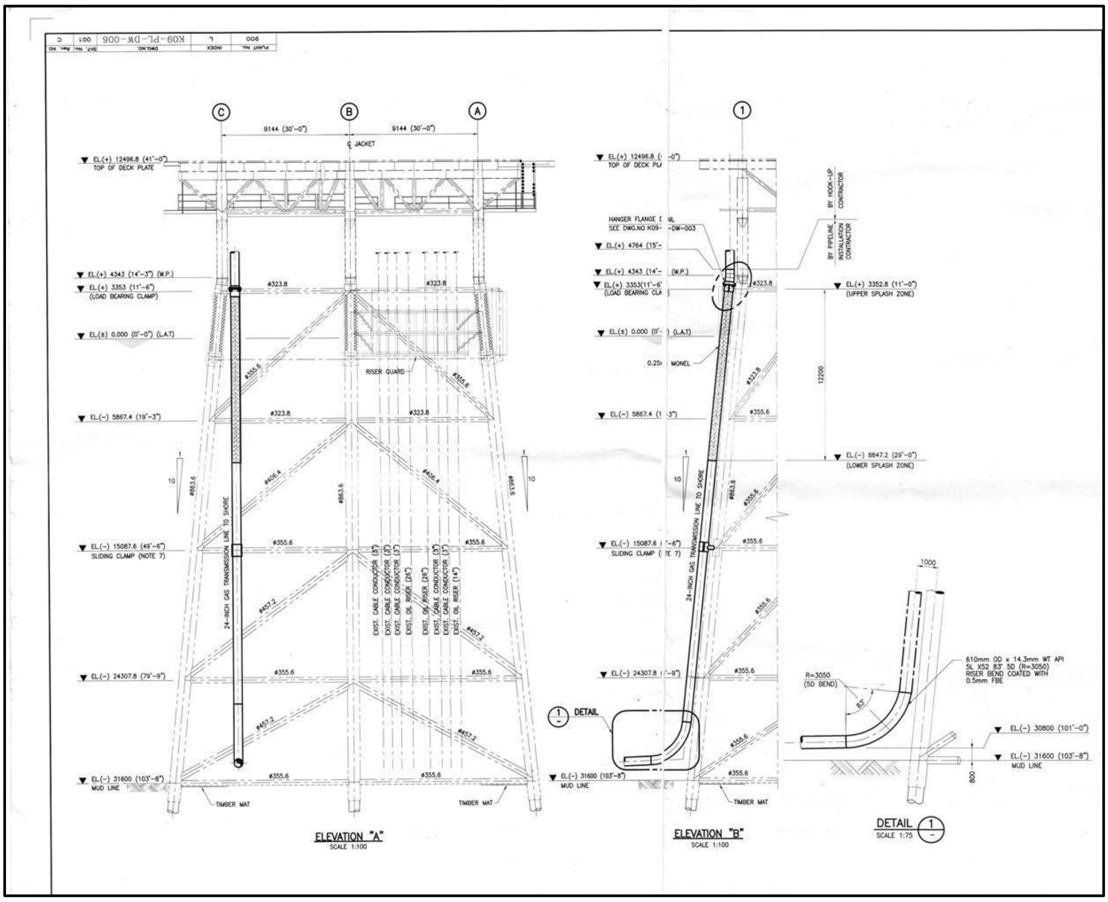


Figure 4.1: Riser General Arrangement Drawing

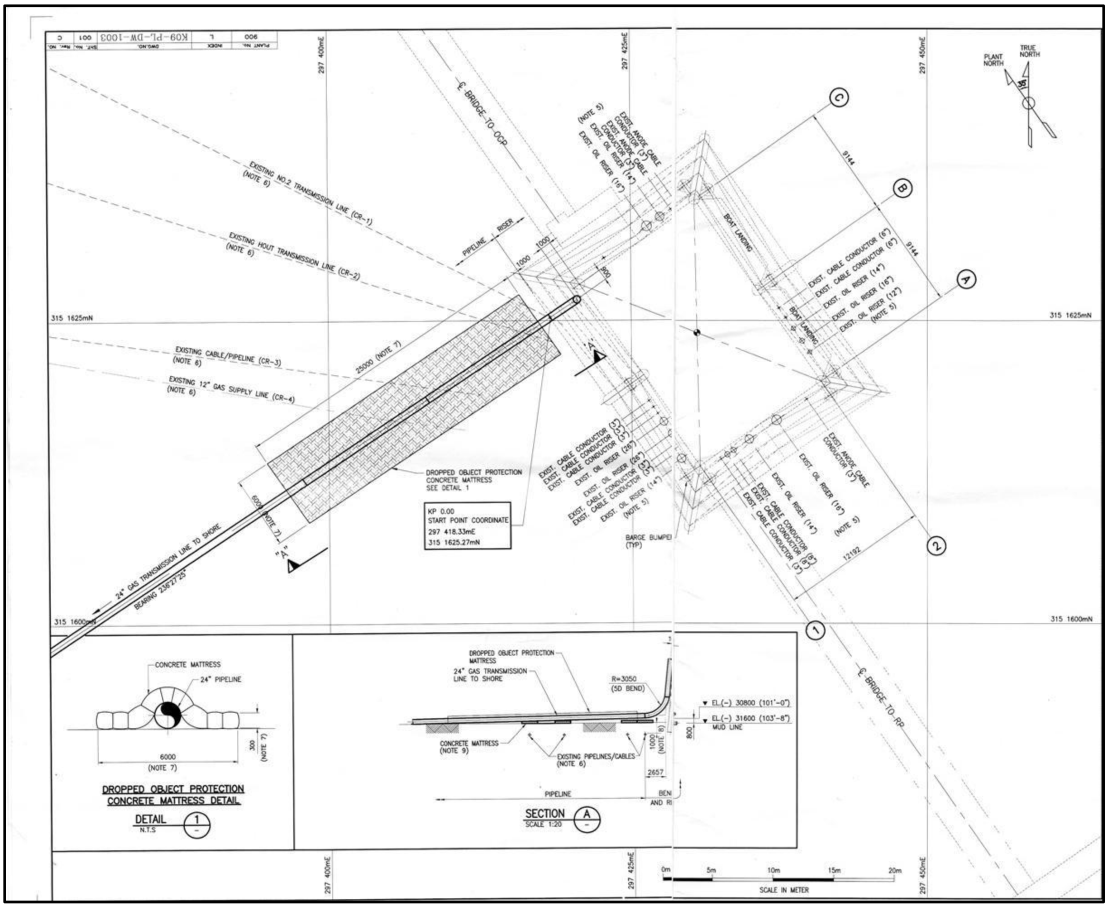


Figure 4.2: Pipeline Approach Drawing

4.1.3 Experimental Data

The experimental data of this riser stress analysis is gathered from industrial internship company with the permission. The stress analysis was performed by Pipeline Engineer – Ezhani Esa using Bentley AutoPIPE software. Bentley AutoPIPE is the most comprehensive piping analysis software for 20 years in small to high-end piping projects worldwide for the power, oil and gas, petrochemical and offshore markets. It has been used globally by leading EPCs (Engineering, Procurement and Construction). The result generated by her is proved to be dependable as the source of comparison for this final year project, as it is inspected and certified by a 3rd party inspection company appointed by the client of the project. Considering that the result generated using Bentley AutoPIPE is accepted by client as the data to be referred for construction stage in the future, thus it is safe to say the result from Bentley AutoPIPE is highly dependable to represent the experimental data for this final year project.

4.2 Modelling Methodology and Result

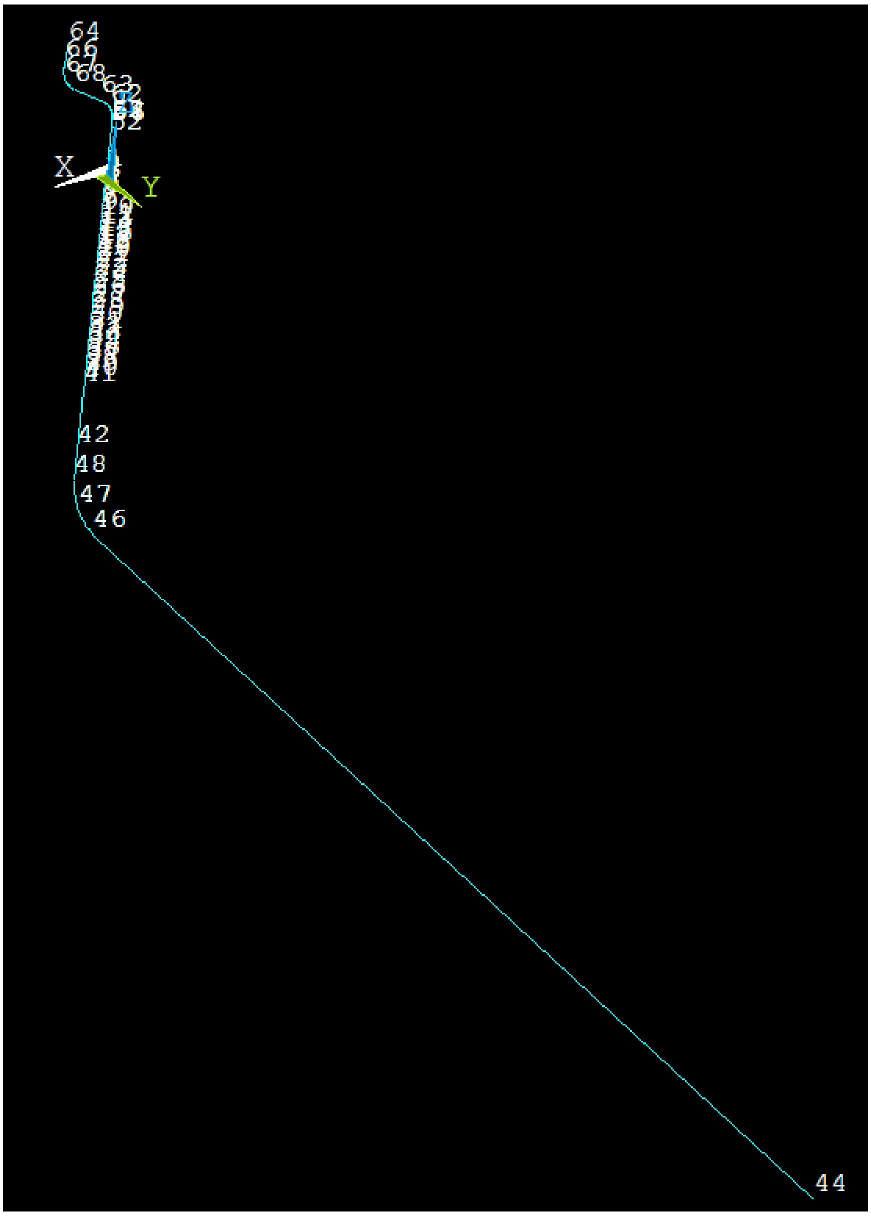


Figure 4.3: Line model of riser in ANSYS Multiphysics

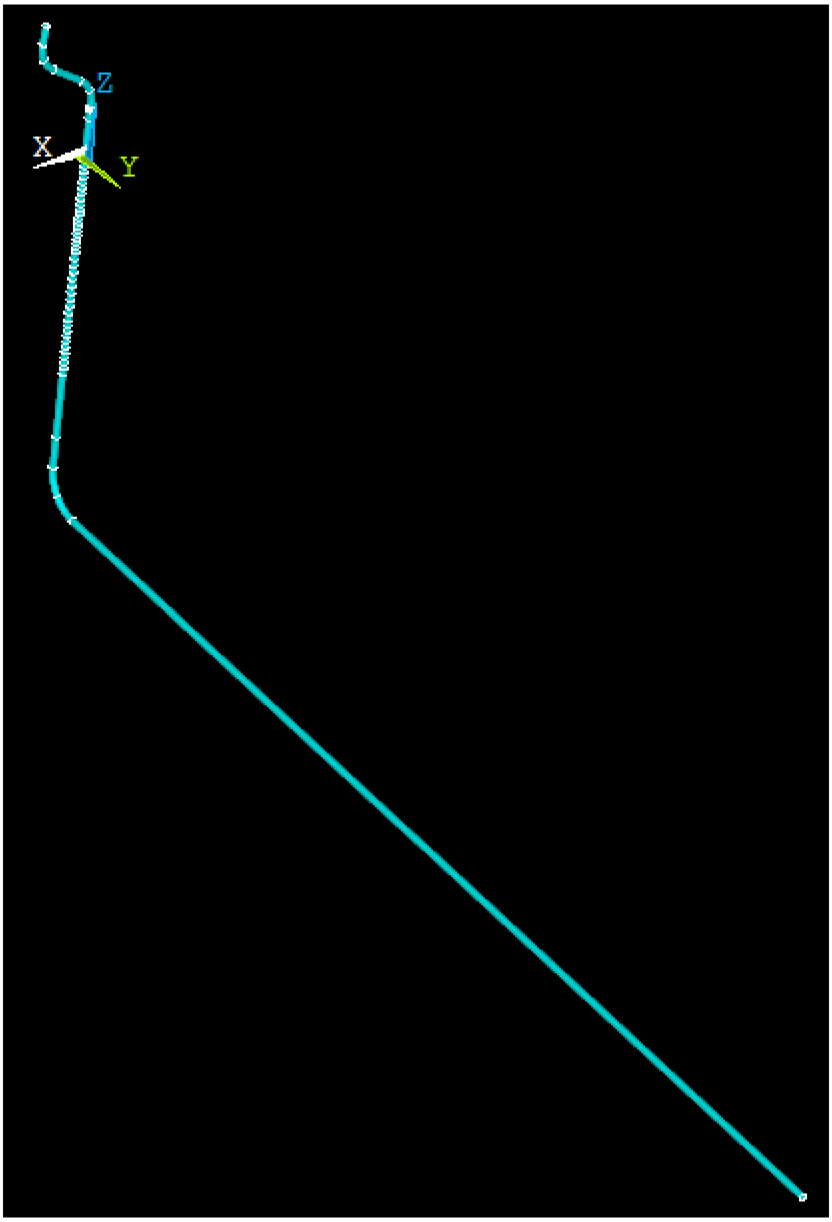


Figure 4.4: Three-dimension solid model of riser in ANSYS Multiphysics

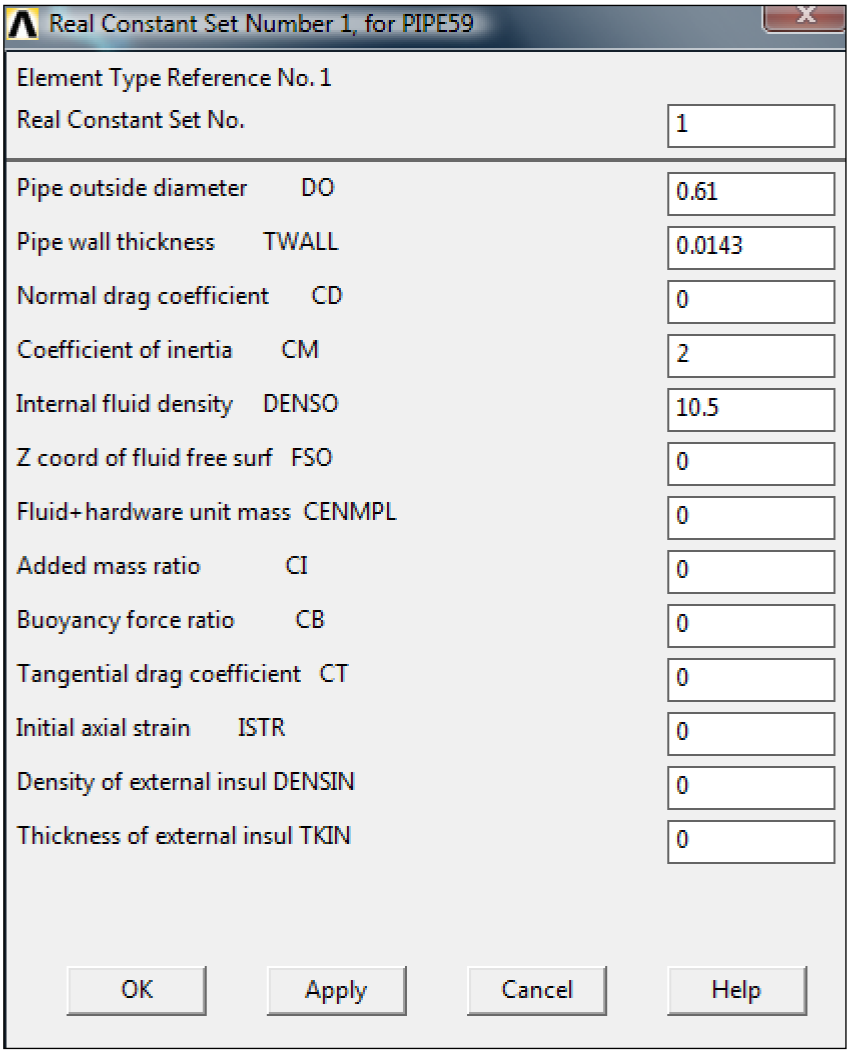
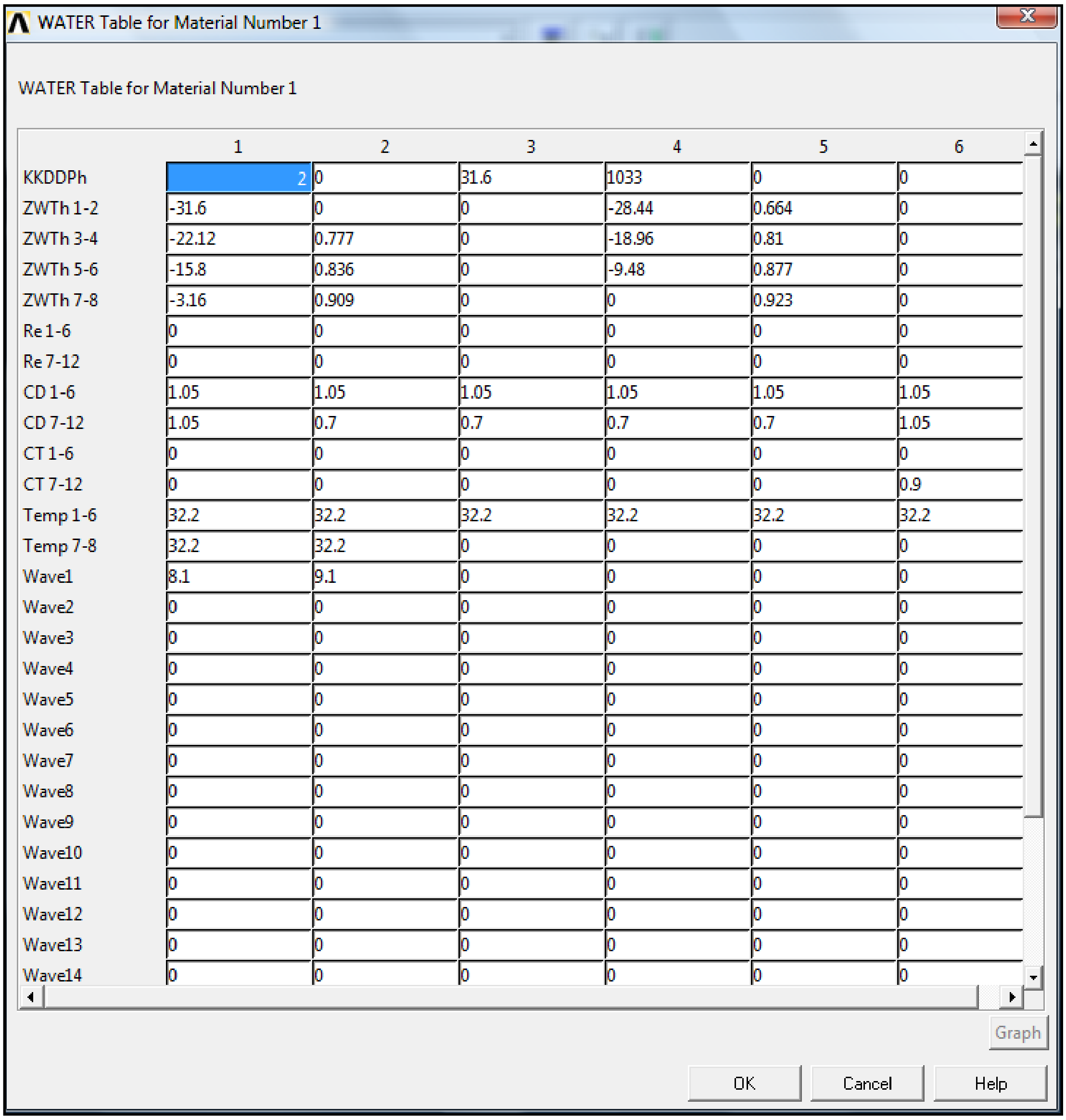


Figure 4.5: The GUI interface for PIPE59 real constants in ANSYS Multiphysics



WATER Table for Material Number 1

WATER Table for Material Number 1

	1	2	3	4	5	6
KKDDPh	2	0	31.6	1033	0	0
ZWTh 1-2	-31.6	0	0	-28.44	0.664	0
ZWTh 3-4	-22.12	0.777	0	-18.96	0.81	0
ZWTh 5-6	-15.8	0.836	0	-9.48	0.877	0
ZWTh 7-8	-3.16	0.909	0	0	0.923	0
Re 1-6	0	0	0	0	0	0
Re 7-12	0	0	0	0	0	0
CD 1-6	1.05	1.05	1.05	1.05	1.05	1.05
CD 7-12	1.05	0.7	0.7	0.7	0.7	1.05
CT 1-6	0	0	0	0	0	0
CT 7-12	0	0	0	0	0	0.9
Temp 1-6	32.2	32.2	32.2	32.2	32.2	32.2
Temp 7-8	32.2	32.2	0	0	0	0
Wave1	8.1	9.1	0	0	0	0
Wave2	0	0	0	0	0	0
Wave3	0	0	0	0	0	0
Wave4	0	0	0	0	0	0
Wave5	0	0	0	0	0	0
Wave6	0	0	0	0	0	0
Wave7	0	0	0	0	0	0
Wave8	0	0	0	0	0	0
Wave9	0	0	0	0	0	0
Wave10	0	0	0	0	0	0
Wave11	0	0	0	0	0	0
Wave12	0	0	0	0	0	0
Wave13	0	0	0	0	0	0
Wave14	0	0	0	0	0	0

Graph

OK Cancel Help

Figure 4.6: The GUI Interface for Water Motion Table in ANSYS Multiphysics

4.3 Simulation Results and Discussion

The simulation results obtained from this project mainly concentrated on the stress magnitude for nodes from the top of the riser till the end of pipeline anchored. The stresses to be discussed in this section are be Von Mises stress and bending stress along the riser. There is no shear stress inside the riser as it is filled with homogeneous maximum product density during operating case.

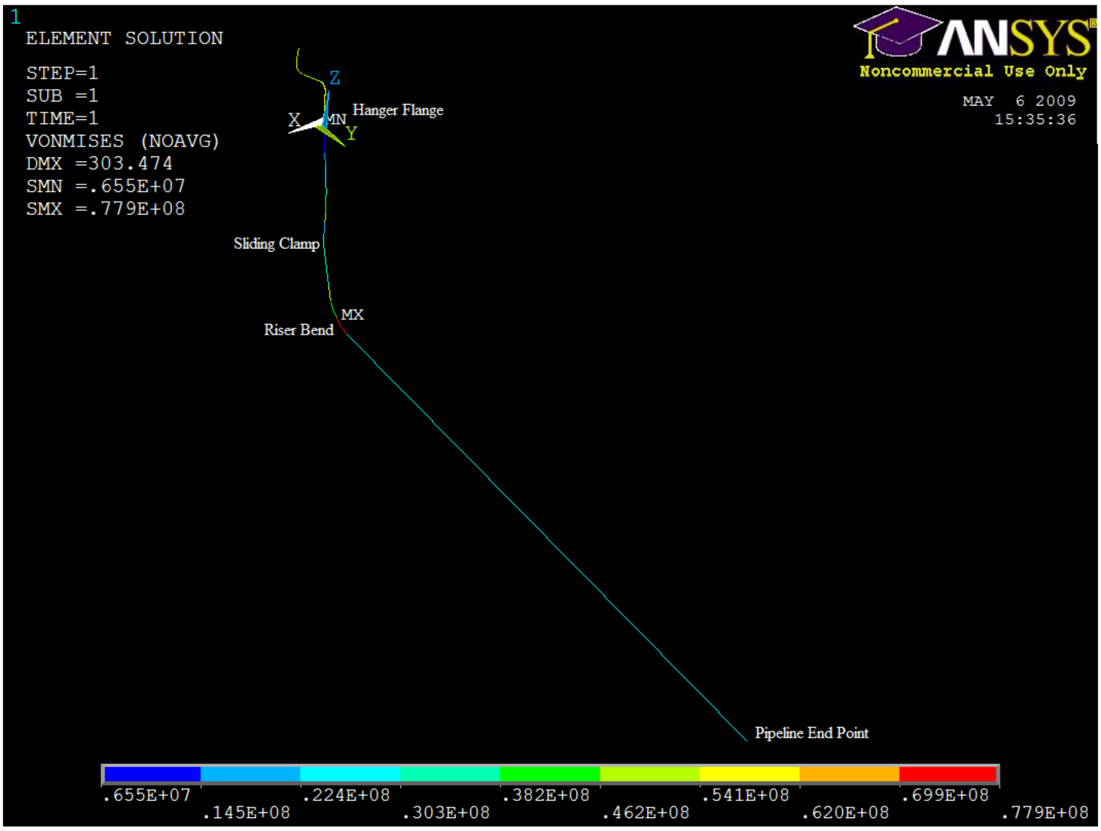


Figure 4.7: Von Mises Stress Output in ANSYS Multiphysics

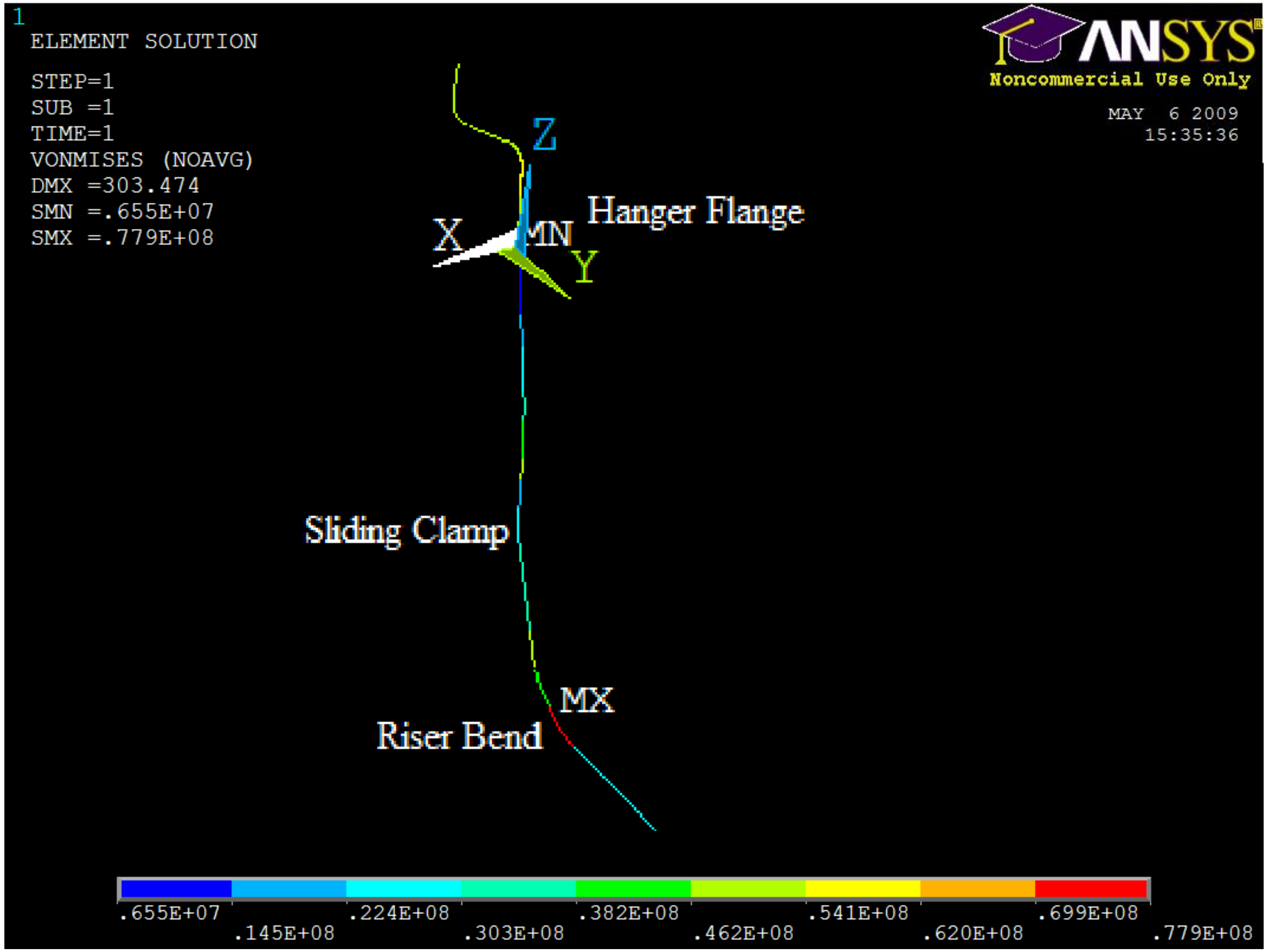


Figure 4.8: Von Mises Stress Output of Riser in ANSYS Multiphysics

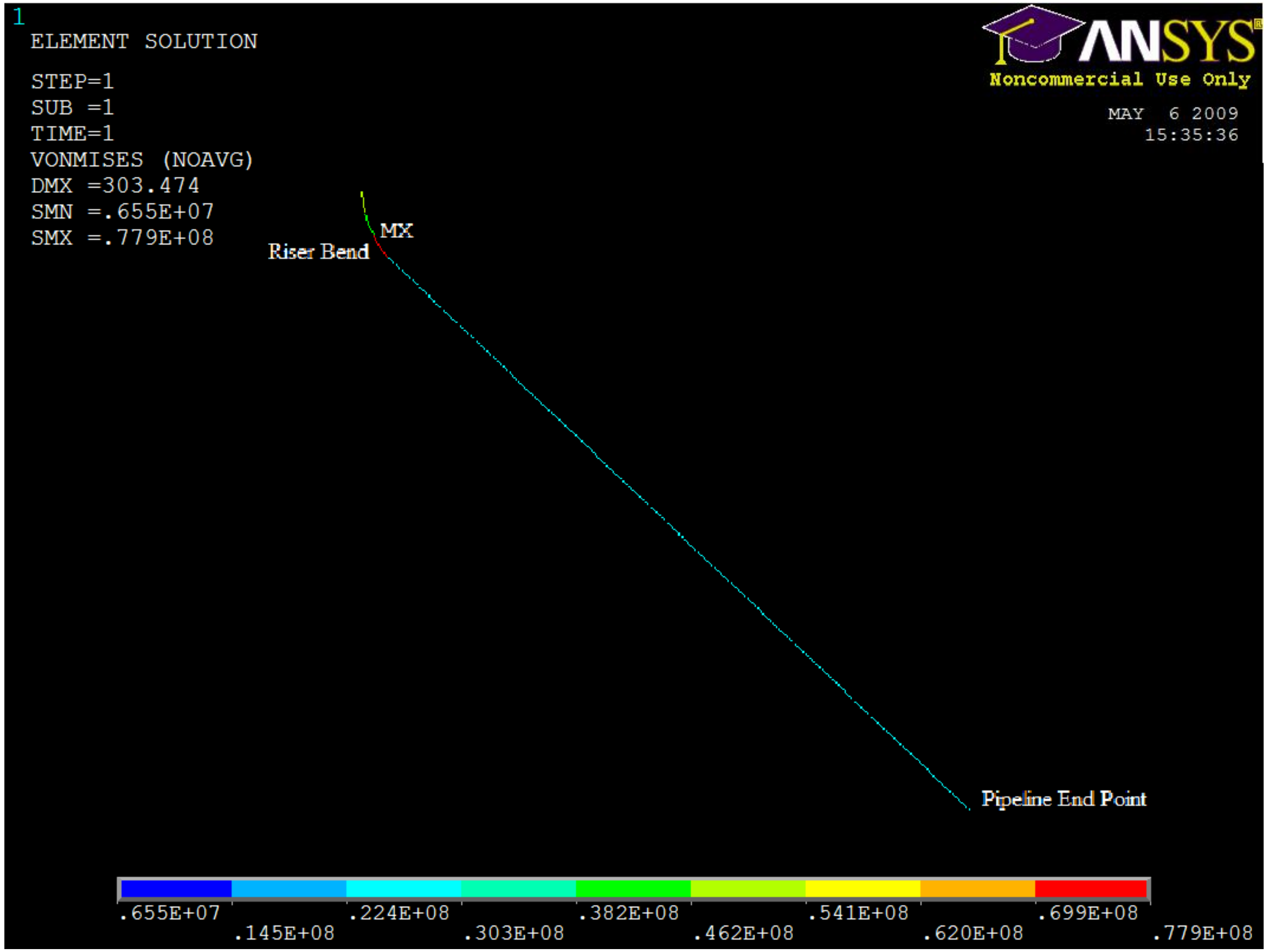


Figure 4.9: Von Mises Stress Output of Riser Bottom in ANSYS Multiphysics

As refer to the figure 4.7, 4.8 and 4.9 above, it shows the stress distribution along the riser with the inclusion of loadings condition (mentioned in scope of study). The highest stress value is recorded at riser bend where 77.9 Mpa is recorded. Riser bend area is the region with the highest stress as shown clearly in figure 4.7 and 4.8 where the stress value is in the range of 69.9 Mpa till 77.9 Mpa. This phenomenon happens at riser bend because at the particular node, bending stress is at the maximum value. As hoop stress is constant throughout the riser, bending stress plays a major role in affecting combined stress value throughout the riser. This can be further justified with figure 4.7, 4.8, 4.9, 4.10, 4.11 and table 4.4 and 4.5 where the stress at riser bend is the maximum among all the nodes along the riser. This can be concluded that the combined stress magnitude is directly proportional to the value of bending stress.

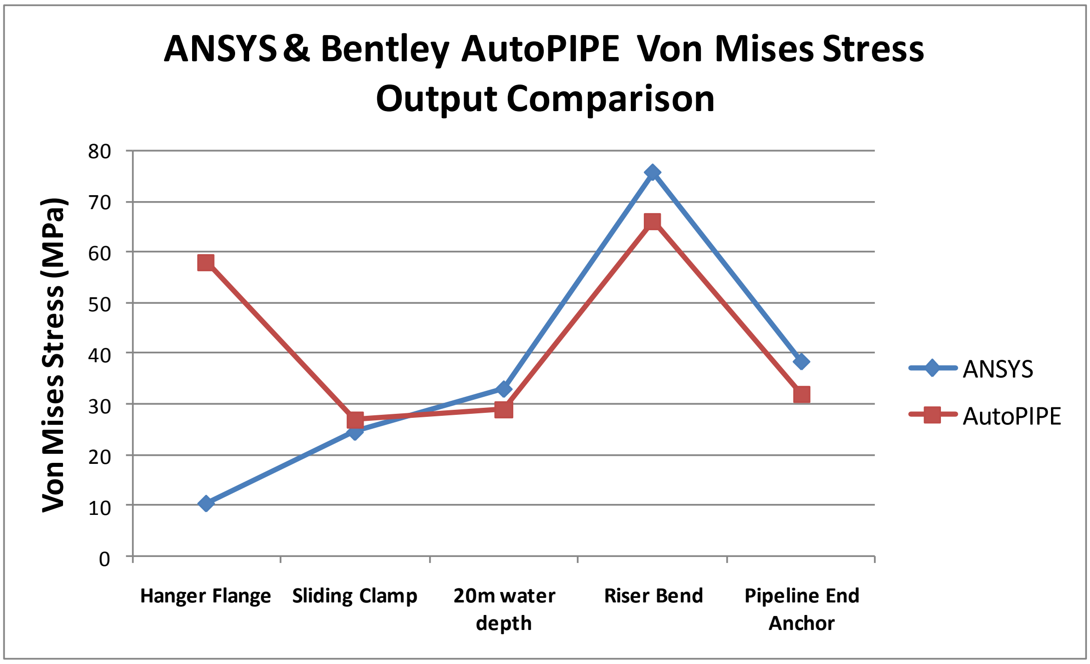


Figure 4.10: ANSYS & Bentley AutoPIPE Von Mises Stress Output Comparison

Table 4.4: ANSYS & Bentley AutoPIPE Von Mises Stress Output Comparison

Von Mises Stress (Mpa)		
	ANSYS	AutoPIPE
Hanger Flange	10.425	58
Sliding Clamp	24.457	27
20m water depth	32.916	29
Riser Bend	75.558	66
Pipeline End Anchor	38.325	32

Based on figure 4.4 and table 4.4, the Von Mises stresses generated by ANSYS Multiphysics at particular coordinates are comparable to the experimental data generated with Bentley AutoPIPE except at hanger flange. This can be explained as ANSYS Multiphysics not able to model the hanger flange similar to actual condition as Bentley AutoPIPE capable of. In actual condition, hanger flange allows vertical upward expansion of the riser due to the thermal expansion but restraint any vertical downwards expansion. Thus it is explainable that the stress value generated for hanger flange is different to the correct data generated by Bentley AutoPIPE. According to the result generated as tabulated above, the highest Von Mises stress is located at the riser bend, which is justified by ANSYS Multiphysics and Bentley AutoPIPE.

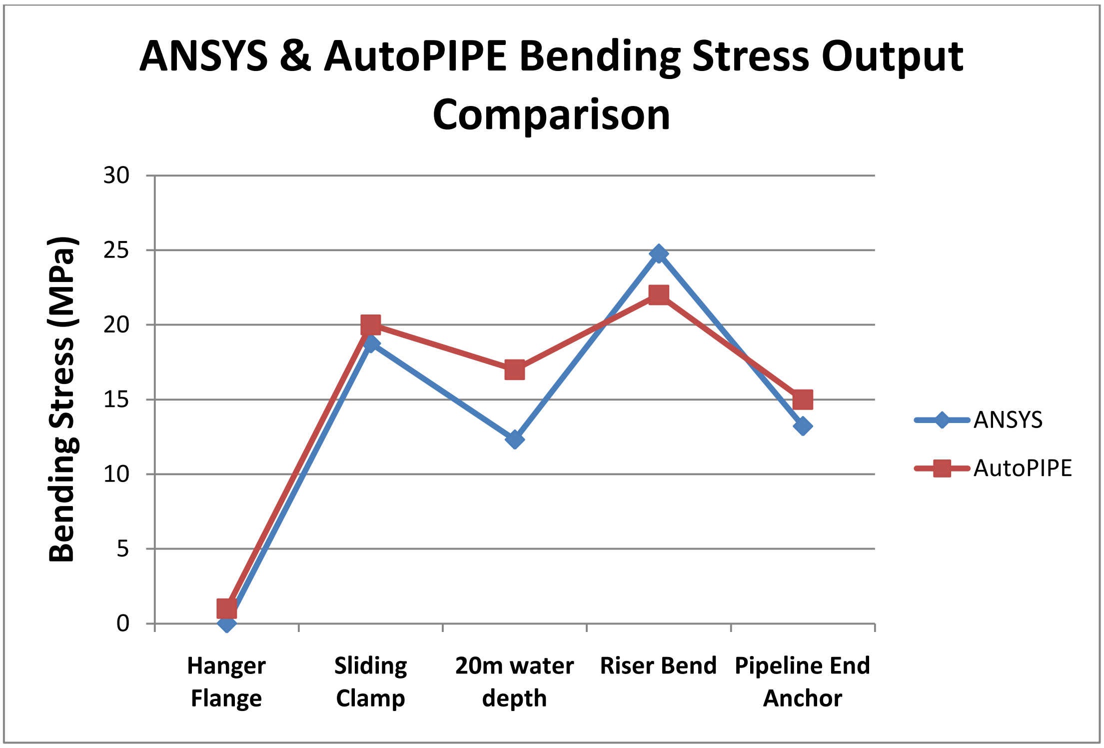


Figure 4.11: ANSYS & Bentley AutoPIPE Bending Stress Output Comparison

Table 4.5: ANSYS & Bentley AutoPIPE Bending Stress Output Comparison

Bending Stress (Mpa)		
	ANSYS	AutoPIPE
Hanger Flange	0.020079	1
Sliding Clamp	18.758	20
20m water depth	12.314	17
Riser Bend	24.142	22
Pipeline End Anchor	13.214	15

Based on figure 4.5 and table 4.5, the bending stress generated by ANSYS Multiphyiscs at particular coordinates are comparable to the experimental data generated with Bentley AutoPIPE except at hanger flange. This has been explained briefly in previous page. According to the result generated as tabulated above, the highest bending stress is located at the sliding clamp, which is justified by ANSYS Multiphysics and Bentley AutoPIPE.

After obtaining magnitude value for Von Mises stress and bending stress, hoop stress of the riser can be calculated using analytical method because the hoop stress is uniform for all coordinates on the riser as the operating pressure; inner diameter and wall thickness are constant throughout the length of the riser system.

This riser system is considered as a thin wall cylinder system, thus the formula to calculate the hoop stress is below:

$$\sigma_h = pr/t$$

where *p* is the operating pressure, *r* is the inner radius and *t* is the wall thickness.
The hoop stress calculated based on the formula above for this riser system is 59 MPa.

As discussed earlier in this report, both Von Mises stress and bending stress are obtainable with ANSYS Multiphysics but it is essential to justify the Von Mises stress throughout the riser and pipeline system shall not exceed the allowable stress limit in ASME B31.8:

Hoop Stress, $S_h \leq F_1 S_y$
Bending Stress, $S_b \leq F_2 S_y$
Von Mises stress, $S_{eqv} \leq F_3 S_y$

where, F_1 = design factor for hoop stress
F_2 = design factor for bending stress
F_3 = design factor for Von Mises stress
S_y = pipe specified minimum yield strength

Table 4.6: Design Factor for Offshore Pipeline System

Analysis Cases	Pipeline			Riser & Platform Piping		
	F1	F2	F3	F1	F2	F3
Operating Case	0.72	0.8	0.9	0.6	0.8	0.9

The design factor shown in the table 4.6 above is determined from the ASME B31.8 where the design factor of riser & platform piping is to be used to determine whether the stress stay within the allowable stress limit.

Table 4.7: Stresses vs Allowable Stress Limit

	Maximum Stress Magnitude	Stress Magnitude * Design Factor	Within Allowable Stress Limit
Hoop Stress	59	215.4	Yes
Bending Stress	24.142	287.2	Yes
Von Mises Stress	75.558	323.1	Yes

According to Table 4.7, it is shown that all the stresses meet the allowable stress limit criteria in ASME B31.8 [14]. This has concluded that all the stresses throughout the riser system are well below the allowable stress limit and no behaviour of overstress, which have been proved with Bentley AutoPIPE and ANSYS Multiphysics.

CHAPTER 5
CONCLUSIONS AND RECOMMENDATIONS

Using ANSYS Multiphysics has been a very challenging exercise in its own even though it is a very user-friendly package. The problem lies not with the software interface itself but the multitude of options, and inputs that a user has to familiarise with as soon as possible.

Although extensive manuals for ANSYS Multiphysics are available for reference, it is imperative for a user knows why a particular model is chosen over another. There are numerous options to choose from, and selection of the appropriate model reduces the time spent running simulations.

5.1 Conclusions

Several conclusions can be made about the results of this project:

- ANSYS Multiphysics has been able to model the hydrodynamics load on offshore riser with PIPE59 element, which can be used to model immersed pipe.
- ANSYS Multiphysics capable of generating the bending stress; Von Mises stress magnitudes which are comparable to the experimental data within the scope of study of this project.
- ANSYS Multiphysics not able to model the hanger flange as Bentley AutoPIPE capable of, this can be solved with ANSYS AQWA which is not available in the university computer lab.
- ANSYS Multiphysics not able to model the soil properties at the mud line (seabed), which can affect the stress distribution on the pipeline.
- Only a value of lift coefficient can be input for the whole length of the riser system, which is undesirable as lift coefficient only applicable to pipeline resting on the seabed, not for the riser off seabed.

5.2 Recommendations

Some of the suggested work that could be carried out in the future for the riser stress analysis are as below:

- Further validation of the result of this project should be carried out in order to determine the repeatability of the results when applied to other riser system configurations.
- The riser stress analysis can be furthered study with the boundary conditions and loads for hydrotest and installation conditions, besides the operating condition apply in this project.
- To include the soil properties of the seabed, jacket deflections occur at the hanger flange and sliding clamp into the stress analysis to achieve result similar to the real time stress data.
- As the riser covered in this project is installed on a conventional steel jacket, it is recommended to model the steel jacket with the riser installed with ANSYS Multiphysics to study how it will affect the stress distribution as steel jacket is not model in this study.

ANSYS Multiphysics is a very powerful finite element software package and with its PIPE59 element, it is capable to model an offshore riser system with the inclusion of hydrodynamic loading. However there are some limitations with PIPE59 element compared to commercial pipe stress analysis software such as Bentley AutoPIPE and COADE Caesar, hopefully with new versions of ANSYS Multiphysics that offers superior PIPE59 element will able to close the apparent gap.

REFERENCES

1. H.-Y. Guo and M. Lou, 2006, "Effect of internal flow on vortex-induced vibration of risers," Ocean University of China, Qingdao, China.
2. Bo Yu, Yi Wang, Jinjun Zhang, Xin Liu, Zhengwei Zhang and Kai Weng, 2007, "*Thermal impact of the products pipeline on the crude oil pipeline laid in one ditch*," China University of Petroleum, Beijing, China.
3. Zhi-Peng Zang, Bing Teng, Wei Bai and Liang Cheng, 2007, "*A finite element solution of waves forces on submarine pipelines,*" Dalian University of Technology, Dalian, China.
4. Morison, J.R., O'Brien, M.P., Johnson, 1950, "*The force on a submerged cylinder near a plane boundary*," Transactions of the American Institute of Mining Engineers 189, 149-154.
5. Said R. Sabag, Billy L. Edge, Iwan Soedigdo, 1999, "*Wake II model for hydrodynamic forces marine pipelines including waves and currents*," Department of Civil Engineering, Texas A&M University, USA.
6. Mouselli, A.H., 1981, "*Offshore Pipeline Design, Analysis, and Methods*," PennWell Publishing Company, Tulsa, OK.
7. Lambrakos, 1987, "*Wake model of hydrodynamic forces on pipelines*," J. Ocean Engineering.
8. ANSYS Multiphysics Chapter 14 Element Library, "*PIPE59 – Immersed Pipe or Cable*".
9. Yong Bai and Qiang Bai, 2005, "Subsea Pipelines and Risers," Elsevier Limited.
10. Mohammad Iranpour, Farid Taheri and J. Kim Vandiver, 2008, "*Structural life assessment of oil and gas risers under vortex-induced vibration*," Dalhousie University, Halifax, Canada.
11. Andrew C. Palmer and Michael T.S. Ling, 1981, "Movement of Submarine Pipeline Close To Platforms," *Offshore Technology Conference 4067*
12. K. Abhary, Anourbakhsh and L. L. H. S. Loung, 1999, "*An exact analytical method for stress analysis of pipelines*," Tehran University, Tehran, Iran.
13. King RC and Crocker S. 1967, *Piping handbook. 5 Chapter 4, p. 63, 65,* New York, McGraw Hill.
14. ASME B31.8 – Gas Transmission and Distribution Piping Systems.

15. ANSYS Multiphysics, ANSYS Inc, Cannonsburg, USA.