ANALYSIS OF STRESS AND DISPLACEMENT PROFILE ON 30" PIPELINE RISER AT MLNG TIGA

Final Report

FYP 2

NAME : HUDZAIFAH BIN ZOL HAMIDY

ID : 10076

COURSE : PETROLEUM ENGINEERING

Supervisor: Mrs. Mazlin Binti Idress

04th May 2011

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

ANALYSIS OF STRESS AND DISPLACEMENT PROFILE OF 30" PIPELINE RISER AT MLNG TIGA

By

Hudzaifah Bin Zol Hamidy

A project submitted to the
Petroleum Engineering Programme
Universiti Teknologi Petronas
In partial fulfillment of the requirement for the
Bachelor of Engineering (Hons)
Petroleum Engineering

Approved by,

(Mrs Mazlin Binti Idress)

UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK
May 2011

CERTIFICATION 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.

HUDZAIFAH BIN ZOL HAMIDY

ABSTRACT

This project is mainly concerns about the analysis on the finite element simulation of a 30-inch pipeline riser. The area of interest is focused on the displacement of the riser due to internal and external loads applied on the pipeline riser. This project is in collaboration with PETRONAS Carigali Sdn. Bhd (PCSB). Pipeline riser is subjected to internal pressure caused by the fluid inside the pipeline and external pressure, which is the underwater hydrostatic pressure. This could cause the pipeline to experience some displacement or dislocation. Thus, the aim of this study is to simulate the pipeline riser under operating condition and analyze the profile of the pipeline riser using a 3D Finite Element Analysis software. The scope of the study basically covers four main phases starting from literature review, data gathering, modeling and simulation run using software, and finally analyzing the profile of the pipeline riser model prior to the result obtained from the simulation. The methodology of this project demonstrates on how a pipe simulation is done using a 3D Finite Element Analysis software. The simulation used is based on a piping network model provided by the software and the output of the simulation is used for the analysis at the final stage of the study. This study has come out with results that were found to be inaccurate. The displacements of the pipeline riser have been determined from the simulation. The results obtained from the simulation are however reasonable considering the limitations that was set earlier in the project. The author has successfully managed to simulate the 3D model using the software identified.

ACKNOWLEDGEMENT

All praises toward Allah the Almighty and peace be upon Prophet Muhammad SAW. First of all, special gratitude goes to the supervisor, Ms Mazlin Binti Idress for the guidance throughout the project completion period. Great thanks convey to the Final Year Project Coordinators for their attentions and concerns on progress of the project, and to Computer Aid Laboratory technician for the assistance and trust of using the facilities and software during the simulation work. In addition, thanks to all lecturers in UTP especially in Petroleum Engineering & Geoscience Department and also the rest of UTP staffs whoever has provided guidance and assistance.

The gratitude is extended to author's beloved parents, Mr Zol Hamidy Bin Che An and Mrs Faridah Binti Abdul Hamid for full encouragements and supports, to the rest of family, friends and individual not mentioned above who involved directly or indirectly.

May Allah bless all of you.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

MLNG	Malaysia Liquefied Natural Gas Sdn. Bhd
PCSB	PETRONAS Carigali Sdn. Bhd
UTP	Universiti Teknologi Petronas
FEA	Finite Element Analysis
FEM	Finite Element Method
ANSI	American National Standard Institution
ASME	American Society Mechanical Engineering
PTS	PETRONAS Technical Standards
API	American Petroleum Institute
SI	Standard International Unit

LIST OF NOMENCLATURES

3D	Three Dimensional
2D	Two Dimensional
In	Inch
mm	Millimeter
cm	Centimeter
m	Meter
N	Newton
Pa	Pascal
M	Mega
G	Giga
Psi	Pound per square inch
Kg	Kilogram
°C	Celsius

CHAPTER 1

INTRODUCTION

1.1 Project Background

Malaysia is currently world's third largest exporters of liquefied natural gas (LNG).The PETRONAS LNG Complex in Bintulu, Sarawak comprises three LNG plants owned and operated by PETRONAS' joint venture companies; Malaysia LNG Sdn Bhd, MLNG Dua Sdn Bhd and MLNG Tiga Sdn Bhd respectively. An integrated world-class LNG production complex spread over 276 hectares of land, it receives its gas supply from upstream facilities offshore Sarawak. With a total of eight production trains and a combined capacity of over 23 million tonnes per annum (mtpa), the complex is one of the world's largest LNG production facilities at a single location.

MLNG Tiga plant has a total annual capacity of 7.8 Mtpa for two trains (Train 7 & 8), which was among the largest ever built at that time. Moreover, several innovative ideas made the plant one of the most significant achievements in the LNG industry.

There are several line of pipeline which connects the platform and the processing plant. The plant is located near the shore so that it is easy to get the supply of gas and condensate from the platforms located at Bintulu offshore.

The analysis on pipeline stress and displacement was initially conducted by PETRONAS Carigali and has been assigned to Protek Engineers Sdn. Bhd (Protek) to review and revise the development scheme. The design of pipeline and associated riser are generally in accordance with DnV OS-F101, Submarine Pipeline System, 2000. Since the offshore pipeline is beyond Malaysia territory, hence the pipeline should be designed accordance with PTS20.196, which specifies ANSI/ASME B31.8 code. [3]

The data of MLNG Tiga was provided as reference and the focus are would be analysis of stress and displacement profile on the pipeline riser from M1 platform to E11R-C.

1.2 Problem Statement

Underwater distribution pipeline is connected to offshore platform using a special pipeline that has bends at its end called a pipeline riser. Pipeline riser at normal operating conditions transfers high pressure and high temperature fluid that is pumped to transport the fluid to other places such as onshore terminal or another receiving platform. Pipeline riser is the most critical structure along the pipeline systems and could cause major problems to daily operations.

Transportation of oil and gas from offshore platforms to shore, vessels or other offshore locations is generally accomplished by means of subsea flow lines. Products obtained or processed on an offshore platform are transferred to a seabed flow lines through a pipeline riser running from the platform deck to the seabed.

During recent years, increase in water depth and important changes in operational requirements have brought significant alterations both in the types of pipeline riser systems and in the methods employed in their installation. These changes have presented many new stress problems which had not been either encountered, or, in certain instances, not even considered in previous pipeline riser designs. This project presents methods for computerized stress analysis of pipeline riser systems in their operating conditions.

Thus, this project is to study on the stress and displacement experienced by the pipeline riser. The result of the study could be use by LNG engineers or pipeline engineers especially when they are designing new pipeline structure in the near future. Under operating conditions, the pipeline riser is primarily subjected to fluid thermal expansion that may cause higher stress on the riser itself.

1.3 Objectives and Scope of study

The objectives of my research are:

1) To conduct simulation of the pipeline riser and establish its expansion profile
2) To find the maximum stress and displacement of the pipeline riser under operating condition

The scope of study basically covers the three main parts:

- **Pipeline Design and Standard Codes**

  Each of the pipeline system designed and conducted in Malaysia shall comply with all the rules and regulations by Malaysian government as laid down in Malaysian Petroleum Measures Act 1984 and PETRONAS Technical Standard Code (PTS). [4]

- **Design and Static Analysis of A Riser**

  The static analysis used for design utilizes the finite element method to come out with the maximum stress value.

- **Finite Element Analysis**

  One of the numerical method that can be used to find the solutions of engineering problems involving stress applied, thermal expansion, heat transfer and fluid flow.

CHAPTER 2

THEORY & LITERATURE REVIEW

2.1 Pipeline Transport Overview

There is some argument as to when the first crude oil pipeline was constructed. However, some say pipeline transport was pioneered by Vladimir Shukhov and the Branobel company in the late 19th century. Others say oil pipelines originated when the Oil Transport Association first constructed a 2-inch (51 mm) wrought iron pipeline over a 6-mile (9.7 km) track from an oil field in Pennsylvania to a railroad station in Oil Creek, in the 1860s. Pipelines are generally the most economical way to transport large quantities of oil, refined oil products or natural gas over land. Compared to shipping by railroad, they have lower cost per unit and higher capacity. Although pipelines can be built under the sea, that process is economically and technically demanding, so the majority of oil at sea is transported by tanker ships.

Oil pipelines are made from steel or plastic tubes with inner diameter typically from 4 to 48 inches (100 to 1,200 mm). Most pipelines are buried at a typical depth of about 3 to 6 feet (0.91 to 1.8 m). The oil is kept in motion by pump stations along the pipeline, and usually flows at speed of about 1 to 6 metres per second (3.3 to 20 ft/s). Multi-product pipelines are used to transport two or more different products in sequence in the same pipeline. Usually in multi-product pipelines there is no physical separation between the different products. Some mixing of adjacent products occurs, producing interface. At the receiving facilities this interface is usually absorbed in one of the product based on pre-calculated absorption rates.

Crude oil contains varying amounts of wax, or paraffin, and in colder climates wax buildup may occur within a pipeline. Often these pipelines are inspected and cleaned using pipeline inspection gauges *pigs*, also known as *scrapers* or *Go-devils*. Smart pigs are used to detect anomalies in the pipe such as dents, metal loss caused by corrosion, or other mechanical damage. These devices are launched from pig-launcher stations and travel through the pipeline to be received at any other station down-stream, cleaning wax deposits and material that may have accumulated inside the line. [6]

For natural gas, pipelines are constructed of carbon steel and varying in size from 2 to 60 inches (51 to 1,500 mm) in diameter, depending on the type of pipeline. The gas is pressurized by compressor stations and is odorless unless mixed with a mercaptan odorant where required by a regulating authority.

2.2 Structural Analysis of Pipeline Risers

Transportation of oil and gas from offshore platforms to shore, vessels or other offshore locations is generally accomplished by means of subsea flow lines. Products obtained or processed on an offshore platform are transferred to a seabed flow line through a pipeline riser running from the platform deck to the seabed.

During recent years, increases in water depth and important changes in operational requirements have brought significant alterations both in the types of pipeline riser systems and in the methods employed in their installation. These changes have presented many new stress problems which had not been either encountered, or, in certain instances, not even considered in previous pipeline riser designs. This paper presents methods for computerized stress analysis of pipeline riser systems in their 'as installed' operating conditions. It does not deal with stresses which may be encountered during installation.

Code compliance check tables compiled for easy cross referencing are included. These tables have been compiled from American and Norwegian Codes of Practice. [9]

2.3 Large Deformation 3D Static Analysis of Deep Water Marine Risers

The problem of static three-dimensional, nonlinear, large deformation of a marine riser is formulated within small strain theory and solved numerically. This type of analysis is necessary, for the new generation of drilling and production risers. The mathematical model takes properly into account the effects of internal and external pressure and complete nonlinear boundary conditions, without linearizing the follower forces. The extensibility or inextensibility condition is used as the constitutive relation in the tangential direction. Torsion and bending are coupled. The external load and the boundary conditions are deformation dependent. A solution method is developed based on an incremental finite element algorithm, which involves a prediction-correction scheme. In the correction phase deformation dependent quantities are updated. The extensibility or in-extensibility condition is used to reduce the degrees of freedom of the system. The numerical results of the developed computer code compare very well with available semi-analytical and numerical solutions. Three numerical applications are used to demonstrate the importance of large deformation, nonlinear and three-dimensional analysis. [3]

2.4 Thermal Expansion Analysis

The calculation procedure for determining the longitudinal pipeline response can be formulated on the basis of strain. The longitudinal strain component can be determined through linear superposition of Poisson effect due to internal pressure (εp), thermal expansion effects (εT), soil restraint (εs), and residual lay tension (εH). The corresponding pipeline axial or longitudinal displacement can be obtained by integrating the strain expression over the pipeline length.

The longitudinal stress component may or may not be statically determinate and is dependent on the imposed boundary condition. The boundary conditions can include the effects of soil reaction loads, anchor restraints, pipeline bend resistance and residual pipeline tension forces.

The operating temperature profile for pipeline transportation systems is not constant along the length. The temperature gradient is dependent on a number of factors that include product type (i.e. oil or gas), internal pressure profile, physical thermal properties (e.g. coating, pipeline and soil), metocean conditions (e.g. temperature, current speed) and pipeline cover (e.g. entrenched, or non-entrenched). One objective of a flow assurance study is to determine the pipeline pressure and temperature profile. The analysis must evaluate the pipeline response on a systematic basis and consider the loading history for the respective loading conditions such as:

- Prior to installation
- As-laid
- Flooded
- Hydro-test
- Operation
- Change in operational parameters
- Shut-in
- Shutdown and restart

The fundamental expressions to determine the pipeline behavior in terms of the circumferential and longitudinal stress-strain response, longitudinal reaction loads and longitudinal pipeline displacement are presented. [2]

2.5 Pipeline Mechanical Design

The mechanical design of pipelines usually requires considerations of several factors. Carneiro and Ferrante (1982) described the elements that important to the design [8]. These include:

- Internal Pressure
- External Pressure
- Stability
- Free Spans
- Expansion Stress
- Risers

2.5.1 Internal Pressure

Internal pressure is often the governing design consideration for pressurized pipelines. The magnitude of the internal pressure along with the pipe characteristics determines the magnitude of stress in the pipe wall (due to only internal pressure), which in turn determines the required wall thickness. This stress (or the associated wall thickness) is calculated using an equation called *Barlow's* formula:

$$P = \frac{2St}{D}$$

Where;

P = pressure (typically, pounds per square inch)
D = outside diameter (typically, inches)
S = allowable stress (typically, pounds per square inch, yield or tensile depending upon application)
t = wall thickness (typically, inches)

If the generated stress in the pipe wall too large, the pipelines will yield circumferentially, and continuous yielding will lead to thinning of the pipe wall and ultimately to rupture.

2.5.2 External Pressure

External pressure requires complex calculations both in determining actual loadings and the pipe responses to those loadings. Soil loads, wave current, buoyancy and changing bottom conditions must also be considered. A large external pressure tends to make a pipeline oval, and eventually causes it to collapse. This is mainly of concern for deep-water pipelines, where the external hydrostatic head is an important factor. The use of higher-grade steel or thicker wall pipe would protect an offshore pipeline against the external hydrostatic pressure.

2.5.3 Stability

A pipeline has to be stable on the seabed. If it is too light, it will slide sideways under the action of currents and waves. Additional weight may be provided by either increasing the wall thickness or by adding concrete weight coating. Another way to reduce the environmental loads on the pipe is by lowering it into a trench or burying it.

2.5.4 Free Spans

A pipeline laid on an uneven seabed does not usually conform to the seabed profile, but instead forms free spans. These are concern because of possible fatigue damage induced by vortex oscillations and because the spans are vulnerable to hooking by fishing gear and ship anchors.

2.5.5 Expansion Stress

Expansion stresses may arise from the difference between the pipeline operating temperature and the installation temperature. If sufficient flexibility is not built in, for example, by providing an expansion loop, buckling may occur.

2.5.6 Risers

Risers are vertical sections of pipe, which are used to connect an offshore pipeline on the seabed to the production facilities, normally located on a platform. Riser design must account for variations in temperature, internal pressure and external environmental loads anticipated throughout the lifetime of the system.

2.5.7 Safety Factor

Different values of design factor, F, are being used depending on the type of service and pipeline route. For oil pipelines, a design factor of 0.72 is being used for all locations, while for gas pipeline DF=0.72 is being used for remote and sparsely populated areas, such as deserts and tundra. For low populated areas, such as fringe areas of towns, industrial areas, a design factor 0.60 is being used. For well populated areas, such as residential and industrial areas and shopping centers, DF of 0.50 is being used. And for areas with multistory buildings, or where traffic is heavy, a design factor of 0.40 is being used. For offshore risers, a design factor of 0.50 and 0.60 are being used depending on local regulations.

2.6 Elements of a Pipeline Riser System

A pipeline riser can be defined as the piping system connecting a pipeline on the seabed to the processing equipment or piping on the platform deck. Until several years ago, most systems were designed as 'external risers' starting with a bend at seabed followed by a vertical riser pipe connected to the legs or horizontal bracing of the jacket with clamps (Fig, 1). With the recent development in concrete and steel platform having large diameter vertical columns, a new method of riser installation has emerged. This involves the pulling of the pipeline into the platform base via a tunnel. The tunnel is then dewatered and the internal riser is welded to the pipeline under atmospheric conditions. This process minimizes the use of expensive diver operations external to the platform for riser tie-ins. The use of external risers on concrete platforms is not popular at the present time but may never the less be used as a back-up system in-case internal riser installation fails or tunnels become inaccessible due to operational problems. Fig. 2 shows the general form of an internal riser approach. [9]

2.6.1 Pipeline at Seabed

For stress analysis purposes, the pipeline at seabed is defined as the part of the pipeline riser system in full contact with, or buried into the seabed, fully restrained against movement in all three Cartesian axes (Fig. 1.a).

2.6.2 Transition Piece

That part of the riser system connecting the pipeline at seabed from a point where the seabed pipe is fully restrained against expansion and not affected by the platform movements, to the vertical riser on the platform (Fig.1.b).

2.6.3 Vertical Riser

That part of the pipeline riser system between the transition piece and the deck piping system (Fig. l. c and 2.c).

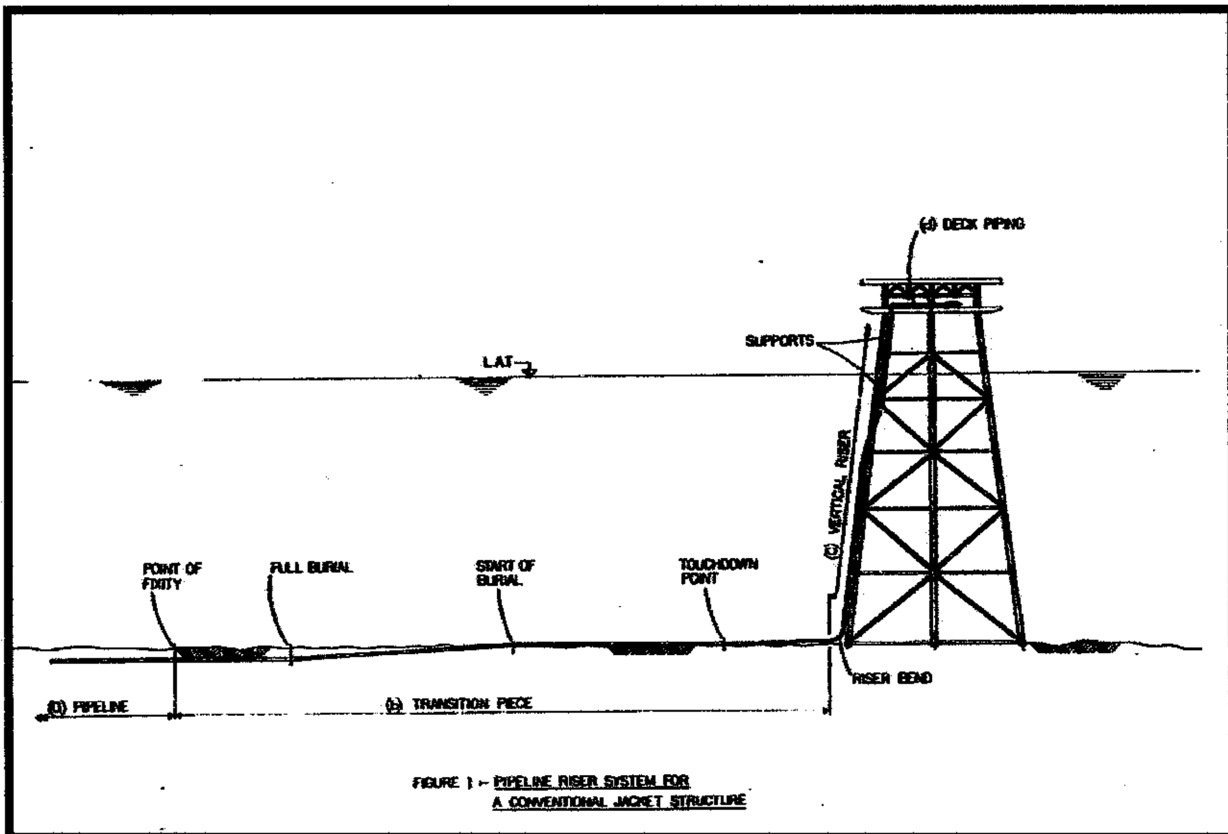


Figure 2.1 : Pipeline Riser System for a Conventional Jacket structure

2.6.4 Deck Piping

Deck piping contributes to the stiffness at the top of the riser. This part of the system generally has a major effect on the stresses at the top of external risers (Fig. 1. d). However, since most internal riser systems end at a vertical anchor (Fig.2), they are not usually affected by the deck piping stiffness.

2.6.5 Riser Supports

Riser supports are restraints provided along the pipeline riser. These may be touchdown points at the seabed, sliding guides, spring hangers or anchors on the platform. Accurate simulation of the support system is very important when calculating the riser stresses due to expansion.

Design analysis of a pipeline riser system involves analyzing and checking the stresses in the transition piece, the vertical riser and applicable parts of deck piping against allowable stresses specified in various design codes, while satisfying compatibility with boundary conditions. The following sections outline the procedure for structural analysis of these riser elements. [9]

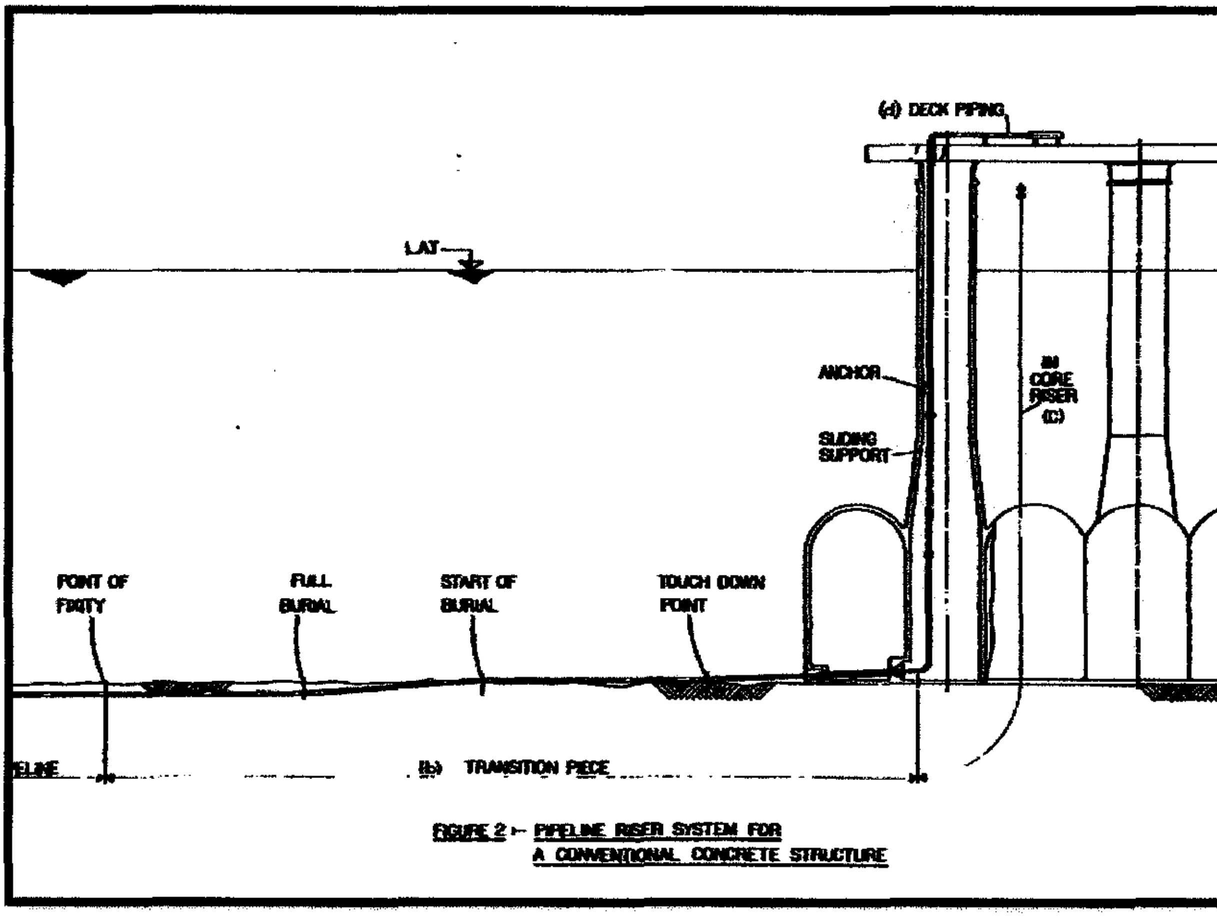


Figure 2.2 : Pipeline Riser System for a Conventional Concrete Structure

2.7 Introduction to FEM & FEA

The finite element method is a numerical procedure that can be used to obtain solutions to a large class of engineering problems involving stress analysis, heat transfer, electromagnetism and fluid flow.

In general, engineering problems are mathematical models of physical situations. Mathematical models of many engineering problems are differential equations with a set of corresponding boundary and initial conditions. The differential equations are derived by applying the fundamental laws and principles of nature to a system or a control volume. These governing equations represent balance of mass, force or energy. When possible, the exact solution of these equations renders detailed behavior of a system under a given set of conditions.

On the other hand, there are parameters that produce disturbance in a system. These types of parameters are summarized in **Table 2.1**. Examples of these parameters include external forces, moments, temperature difference across a medium and pressure difference in a fluid flow.

Table 2.1 : Parameters Causing Disturbance in Various Engineering Systems

Problem Type	Example of Parameters
Solid Mechanics	External forces and moments; support excitation
Heat Transfer	Temperature difference; heat input
Fluid Flow and Pipe Networks	Pressure difference; rate of flow
Electrical Network	Voltage difference

The analytical solutions are composed of two parts, a homogenous part and a particular part. In any given engineering problem, there are two sets of design parameters that influence the way on which a system behaves. First, there are those parameters that provide information regarding the natural behavior of a given system. These parameters include material and geometric properties such as modulus of elasticity, thermal conductivity, viscosity and area and moment of area. [1]

There are many practical engineering problems for which we cannot obtain exact solutions. This inability to obtain an exact solution may be attributed to either the complex nature of governing differential equations or the difficulties that arises from dealing with the boundary and initial conditions. To deal with such problems, it resorts to numerical approximations. In contrast to analytical solutions, which show the exact behavior of a system at any point within the system, numerical solutions approximate exact solutions only at discrete points, called nodes. The first step of any numerical procedure is discretization. This process divides the medium of interest into a number of small sub-regions and nodes. There are two common classes of numerical methods that are finite differential method and finite element methods.

With finite difference method, the differential equation is written for each node, and the derivatives are replaced by different equations. This approach results in a set of simultaneous linear equations. Although finite difference methods are easy to understand and employ in simple problems, they become difficult to apply to problems with complex geometries or complex boundary conditions. This situation is true for problems with non-isotropic material properties.

In contrast, the finite element method uses integral formulations rather than difference equations to create a system of algebraic equations. Moreover, a continuous function is assumed to represent the approximate solution for each element. The complete solution is then generated by connecting or assembling the individual solutions, allowing for continuity at the inter-elemental boundaries.

The origin of the modern finite element method may be traced back to the early 1900s when some investigators approximated and modeled elastic continua using discrete equivalent elastic bars. However, R. Courant, 1943, has been credited with being the first person to develop the finite element method. In a paper published in early 1940s, Courant used piecewise polynomial interpolation over triangular sub-regions to investigate torsion problems.

The next significant step in the utilization of finite element method was taken by Boeing Company in early 1950s when Boeing, followed by others, used triangular stress elements to model airplane wings. Yet, it was not until 1960 that R.W. Clough made the term finite element popular. During the 1960s investigator began to apply the finite element method to other areas of engineering such as heat transfer and seepage flow problems. [7]

Structural analysis is probably the most common application of the finite element method. The term structural or structure implies not only civil engineering structures such as bridges and buildings, but also naval, aeronautical and mechanical structures such as ship hulls, aircraft bodies and machine housings as well as mechanical components such as pistons, machine parts, and tools.

Finite Element Analysis is a mathematical representation of a physical system comprising a part/assembly, model, material properties and applicable boundary conditions, collectively referred to as pre-processing, the solution of that mathematical representation, solving and the study of results of that solution, post-processing. Simple shapes and simple problems can be, and often are, done by hand. Most real world parts and assemblies are far too complex to do accurately, without use of a computer and appropriate analysis software.

With today's technology, detailed 3D finite element models are practical, and have more accuracy over traditional finite element analysis methods. FEA is powerful tool to access design adequacy for the expected force field applied to a structure. It is equally valid for analysis of dynamic and static loading.

Complex structures have numerous natural frequencies and modes which can be excited by operating machinery or external forces. FEA is applicable to individual machine components as well as entire multi-level structures such as offshore platforms. The structure can be modeled with the boundary conditions and forces can be applied. The resulting deflection and stress is then calculated for evaluation and comparison to applicable engineering criteria. The reliability of the component for the applied conditions can be determined.

FEA is useful for failure analysis as well as design audits. The load conditions on a failed part can be modeled to determine if an overload condition is responsible. These analyses can be combined with metallurgical examination to develop a complete picture of the conditions leading to the failure. Identification of failure modes leads to successful redesign of the component.

The typical approaches involved in any finite element analysis done by the FEA software consist of the following: [1]

Preprocessing Phase:

1. Creation and discretization the solution domain into finite elements, that is, subdividing the problem into nodes and elements.
2. Assuming a shape function to represent the physical behavior of the element.

3. Developing equations for an element.
4. Assembling the elements to present the entire problem.

Solution Phase:

5. Applying boundary conditions, initial conditions and loadings, on the model.
6. Solving a set of linear or non-linear algebraic equations simultaneously to obtain nodal results, such as displacement values at different nodes or temperature values at different nodes in a heat transfer problem.

Post-processing Phase:

7. Obtaining other important information. At this point, values such as principal stresses, heat fluxes and displacement may be of interest.

FEA software typically uses a CAD representation of the physical model and breaks it down into small pieces called finite "elements". This process is called "meshing". The higher the quality of the mesh, the better the mathematical representation of the physical model. The primary purpose of an element is to connect nodes with predictable mathematical equations based on stiffness between nodes, the type of element used often depends upon the problem to be solved. The behavior of each element, by itself, is very well understood. By combining the behaviors of each element using simultaneous equations, one can predict the behavior of shapes that would otherwise not be understood using basic "closed form" calculations found in typical engineering handbooks.

There are many different types and classes of elements, most created for specialized purposes. For example, cable, piping, beams, truss structures, e-mag and etc. A one dimensional element represents line shapes, such as beams or springs. A 2D element, also known as quadrilateral element, will represent triangles and squares. 3D elements represent solid shapes and are usually in 2 basic shapes: brick, hexahedrons or "hex" and pyramids, tetrahedrons or "tets".

Examples of applications for specialized elements would be: scaffolding consisting of connecting 1D line elements. Car bodies and other stamped or formed sheet metal parts are typically very thin relative to their overall size and are usually best represented by 2D shell/plate elements. Many thin shapes can, and are, meshed with 3D solid elements but at the cost of increased processing time and sometimes a loss in accuracy because of the special formulation of 2D shell elements. The tradeoff is that, in order to mesh with 2D shell elements, there is often significant modification and preparation required to the CAD geometry in order to obtain a mesh able surface model, or models in the case of an assembly. In other words, the pre-processing requirement increases substantially.

The guidelines are, can it be meshed and solved in 3D solids because sometimes the resultant model is simply too large. If yes and the user is not looking for ultimate accuracy but only trending and behavioral information, then 3D solid meshing is often appropriate due to human time savings. 3D elements are ideal for thick and chunky parts and assemblies such as engine blocks, machine components, etc. Common element types used in FEA simulation is shown in figure below.

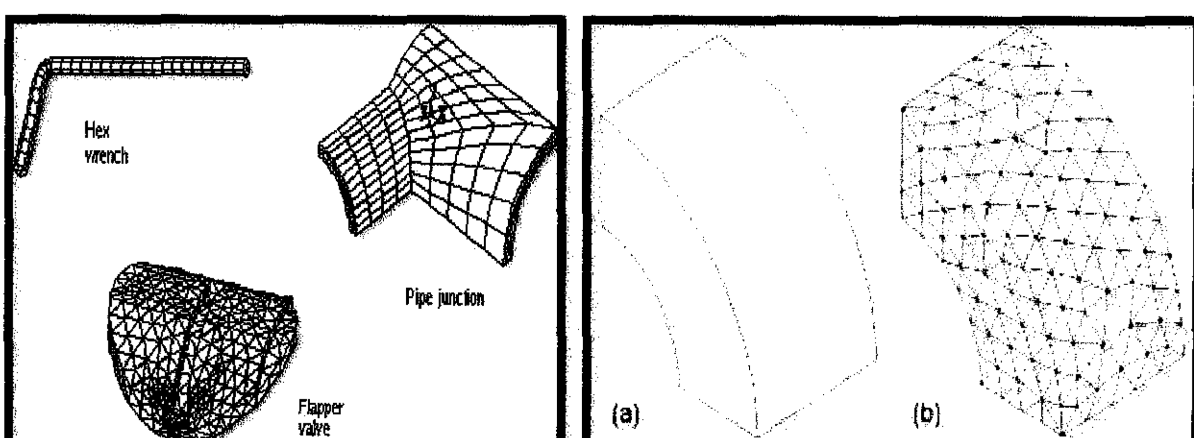


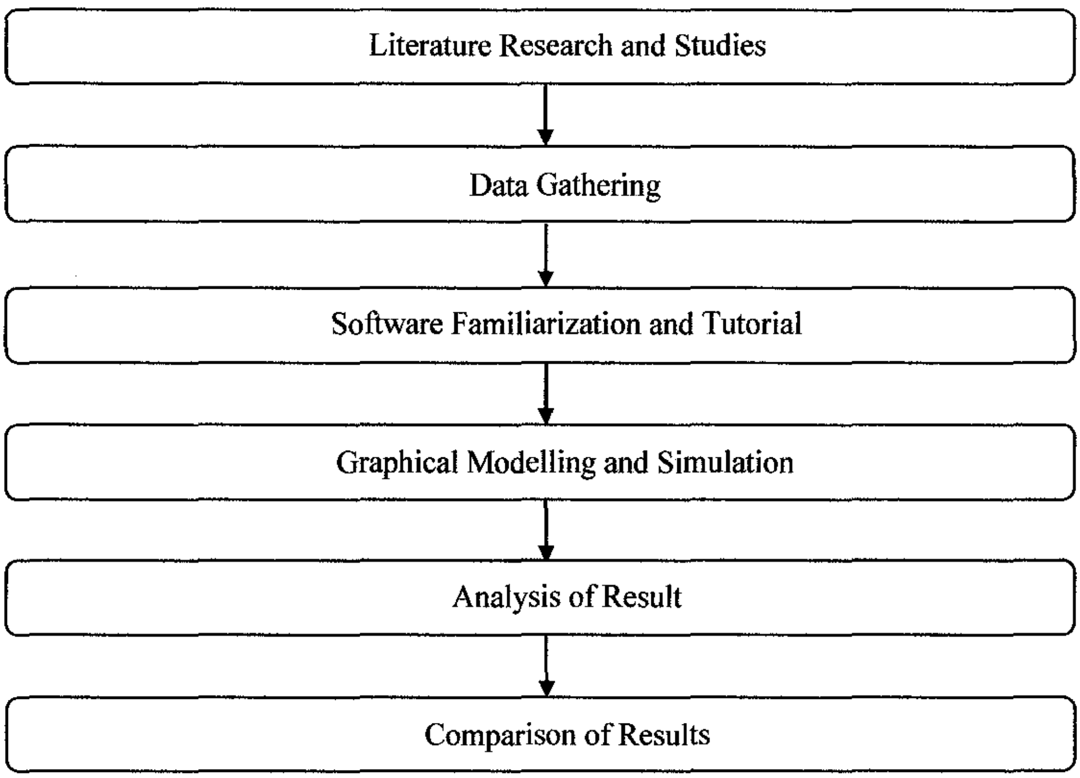
Figure 2.3 : FEM Common Model from ANSYS

CHAPTER 3

METHODOLOGY

3.1 Workflow of Project

Several main procedures have been identified towards accomplishment of the project. The following diagram summarizes on the tasks to be perform accordingly.



First step during the kick start of the project would be literature review and research by author through journals, websites, textbooks, SPE papers and Google. Next step would be data gathering from MLNG Tiga (PETRONAS) or from previous student data. Since the author have no experience on Finite Element Analysis and on the pipeline stress modeling, he needs to familiarize very well with the software and do tutorials by himself in order to get full understanding on using the software. After that, when the author finally could do the modeling nicely, he will proceed with the graphical modeling for the project and simulation on the pipeline riser stress and displacement profile. Next, the author will analyze the result and compare it with the previous student results and also with the result obtained from PCSB. The procedures identified in order to complete the study are:

1) Identification of structures and constraints to be modeled and simulated
2) Modeling of the structure using the software
3) Define loads and boundary conditions on the model
4) Run the simulation and generate results
5) Analysis of the results

3.2 Tools Required

Some software have been identified to be utilized during the work flow of the project is running such as ANSYS, CATIA V5, AutoCAD 2002. All of the software are provided by UTP in Computer Aid Laboratory in Block 15 and Block 16.

ANSYS is the modeling software for pipeline which could calculate stress and displacement profile. This software also could be replace or try out using other software such as AutoPIPE or CAESAR-II.

3.3 Identification of Modeling Parameters

Project documents prepared by PCSB have been reviewed and the data of the pipeline are summarized in table below:

Table 3.1 : The 30" Pipeline Properties

No	Properties	Unit	Value
1.	Pipeline Length	Km	155.55
2.	Outer Diameter	Meter	0.7714
3.	Inner Diameter	Meter	0.7284
4.	Wall Thickness	Meter	0.0215
5.	Insulation Thickness	Meter	0.0955
6.	Pipe Material Density	Kg/m^3	7850
7.	Product Density	Kg/m^3	116.18

8.	Young's Modulus	GPa	207
9.	Poisson's Ratio		0.3
10.	Thermal Expansion Coefficient	Mm/mm/°C	1.17 x 10^5
11.	Design Maximum Temperature	°C	65
12.	Design Pressure	Bar	119.6
13.	Seabed Temperature	°C	21
14	Seawater Density	Kg/cu m	1025
15.	Pipe Elbow Radius	Meter	3.857
16.	Corrosion Allowance	mm	1.0
17.	Material Grade		API 5L-X65
18.	Maximum Flow Rate	MMscfd	700
19.	Product Velocity	m/s	5.46
20.	Corrosion Coating Thickness Riser	Mm	5.5
21.	Corrosion Coating Density Riser	Kg/ m^3	1280
22.	Corrosion Coating Thickness Expansion Spool and Pipeline	Mm	90
23.	Corrosion Coating Density Expansion Spool and Pipeline	Kg/ m^3	3091
24.	Concrete Coating	2.1	1260

Table 3.2 : Hydrodynamic Force Coefficients

Force Coefficient	Riser Design
Drag Coefficient (Cd)	0.7
Lift Coefficient (Cl)	0.0
Inertia Coefficient (Cm)	2.0

Table 3.3 : Design Factors for Equivalent Stress

Design Condition	Allowable Stress (%)
Temporary (Installation)	1.00
Operation	0.90

(DnV OS-F101, SUBMARINE PIPELINE SYSTEMS, 2000)

3.4 Define Loads on the Model

Load was applied on every nodes for the input data after the author has finished the modeling. By using hydrostatic pressure formula, the external load defined by author being calculated.

$$P = \rho g h$$

Where P = hydrostatic pressure

g = gravitational acceleration (9.81 m/s^2)

h = height below sea level

ρ = the density of the sea water (1025 kg/m^3)

Table 3.4 shows the summarization of the hydrostatic pressure of each node.

Table 3.4 : Pressure calculated on each node

Node	Water Depth (m)	Pressure (kPa)
1	0	0
2	0	0
3	4.5	45.25
4	21.0	211.16
5	21.9	220.57
6	42.0	422.32
7	78.9	793.18
8	102.0	1025.64
9	131.6	1323.31
10	136.2	1369.72
11	136.2	1369.72
12	136.2	1369.72
13	137.7	1384.79
14	137.7	1384.79
15	137.7	1384.79
16	137.7	1384.79
17	137.7	1384.79
18	137.7	1384.79

19	137.7	1384.79
20	137.7	1384.79
21	137.7	1384.79
22	137.7	1384.79
23	137.7	1384.79
24	137.7	1384.79
25	137.7	1384.79
26	137.7	1384.79
27	137.7	1384.79
28	137.7	1384.79
29	137.7	1384.79
30	137.7	1384.79
31	137.7	1384.79
32	137.7	1384.79
33	137.7	1384.79
34	137.7	1384.79

3.5 Project Planning – Gantt Chart for FYP 1

No	Detail/Week	1	2	3	4	5	6	7		8	9	10	11	12	13	14
1	Topic selection/confirmation															
2	Preliminary Research Study															
3	Submission of Preliminary Report															
4	Literature Review on pipeline riser and pipeline stress and displace.															
5	Data Gathering															
6	Submission of Progress Report															
7	FYP 1 Seminar (Compulsory)															
8	Software Familiarization															
9	Continue Literature Review and Software Familiarization															
10	Report Preparation															
11	Submission of Interim Report															
12	Oral Presentation (study week)															

3.6 Project Planning – Gantt Chart for FYP 2

No	Detail/Week	1	2	3	4	5	6	7	8	9	10	11	12	13
1	Continue software familiarization													
2	Conduct modelling													
3	Submit progress report													
4	Continue modelling													
5	Analysis of result													
6	Pre-EDX													
7	Result and conclusion													
8	Submit final draft report													
9	Oral presentation													

CHAPTER 4

RESULTS AND DISCUSSIONS

In this chapter, all the results from the simulation will be discussed in detail. Since the author still in learning with the simulation software, ANSYS, it has taken some times for the author to familiarize with the software. The author needs to do several tutorials from manuals and do a lot of try and error exercise in order to further the case study towards next step which is gaining simulation results. But, the author has already identified several aspects that the author must get from the simulation which are:

1) Pipeline Displacement
2) Comparison Result
3) Calculation of Static Analysis Riser
4) Calculation of Pipeline Expansion Analysis
5) Errors and Uncertainties

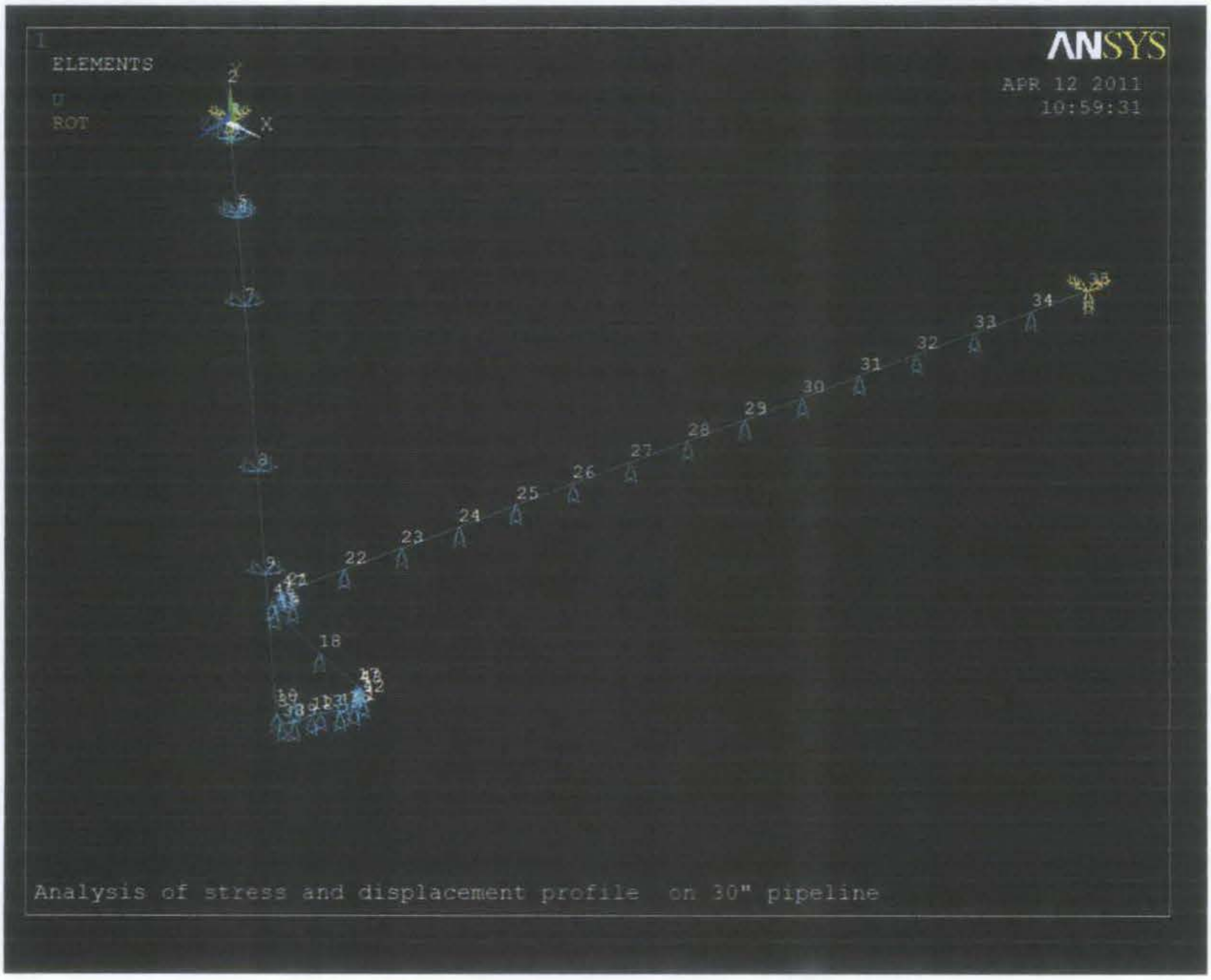


Figure 4.1 : Completed Pipe Model with Nodes

NLIST Command

File

LIST ALL SELECTED NODES. DSYS= 0

NODE	X	Y	Z	THXY	THYZ	THZX
1	0.0000	0.0000	0.0000	0.00	0.00	0.00
2	0.0000	725.20	65.800	0.00	0.00	0.00
3	0.0000	0.0000	0.0000	0.00	0.00	0.00
4	0.0000	-450.00	-40.830	0.00	0.00	0.00
5	0.0000	-2100.0	-190.53	0.00	0.00	0.00
6	0.0000	-2193.6	-199.03	0.00	0.00	0.00
7	0.0000	-4200.0	-381.07	0.00	0.00	0.00
8	0.0000	-7888.0	-715.67	0.00	0.00	0.00
9	0.0000	-10200.	-925.43	0.00	0.00	0.00
10	0.0000	-13160.	-1194.0	0.00	0.00	0.00
11	0.0000	-13622.	-1235.9	0.00	0.00	0.00
12	162.80	-13622.	-1946.7	0.00	0.00	0.00
13	203.32	-13622.	-2123.6	0.00	0.00	0.00
14	295.34	-13772.	-2525.4	0.00	0.00	0.00
15	335.86	-13772.	-2702.3	0.00	0.00	0.00
16	447.48	-13772.	-3189.7	0.00	0.00	0.00
17	-39.910	-13772.	-3301.3	0.00	0.00	0.00
18	-1229.1	-13772.	-3528.7	0.00	0.00	0.00
19	-2582.9	-13772.	-3838.7	0.00	0.00	0.00
20	-3070.3	-13772.	-3950.4	0.00	0.00	0.00
NODE	X	Y	Z	THXY	THYZ	THZX
21	-2958.7	-13772.	-4437.8	0.00	0.00	0.00
22	-2731.3	-13772.	-5627.0	0.00	0.00	0.00
23	-2504.0	-13772.	-6816.2	0.00	0.00	0.00
24	-2276.6	-13772.	-8005.4	0.00	0.00	0.00
25	-2049.3	-13772.	-9194.6	0.00	0.00	0.00
26	-1822.0	-13772.	-10384.	0.00	0.00	0.00
27	-1594.6	-13772.	-11573.	0.00	0.00	0.00
28	-1367.3	-13772.	-12762.	0.00	0.00	0.00
29	-1139.9	-13772.	-13951.	0.00	0.00	0.00
30	-912.61	-13772.	-15141.	0.00	0.00	0.00
31	-685.27	-13772.	-16330.	0.00	0.00	0.00
32	-457.93	-13772.	-17519.	0.00	0.00	0.00
33	-230.59	-13772.	-18708.	0.00	0.00	0.00
34	-3.2500	-13772.	-19897.	0.00	0.00	0.00
35	224.09	-13772.	-21087.	0.00	0.00	0.00
36	451.43	-13772.	-22276.	0.00	0.00	0.00
37	86.446	-13236.	-1578.3	0.00	0.00	0.00
38	0.0000	-13270.	-1204.0	0.00	0.00	0.00
39	22.684	-13521.	-1325.8	0.00	0.00	0.00
40	78.832	-13622.	-1580.1	0.00	0.00	0.00
NODE	X	Y	Z	THXY	THYZ	THZX
41	-14.589	-13772.	-2899.8	0.00	0.00	0.00
42	361.38	-13772.	-2813.7	0.00	0.00	0.00
43	312.14	-13772.	-3104.8	0.00	0.00	0.00
44	71.513	-13772.	-3275.8	0.00	0.00	0.00
45	-2608.2	-13772.	-4240.2	0.00	0.00	0.00
46	-2694.3	-13772.	-3864.3	0.00	0.00	0.00
47	-2935.0	-13772.	-4035.3	0.00	0.00	0.00
48	-2984.2	-13772.	-4326.3	0.00	0.00	0.00

Figure 4.2 : List of Nodes

Table 4.1 : Coordinates for each nodes

Nodes No	X	Y	Z
1	0	7.252	0.658
2	0	-7.252	-0.658
3	0	-4.5	-0.4083
4	0	-16.5	-1.497
5	0	-0.936	-0.085
6	0	-20.064	-1.8204
7	0	-36.88	-3.346
8	0	-23.12	-2.0976
9	0	-29.604	-2.686
10	0	-4.615	-0.4187
11	1.628	0	-7.108
12	0.4052	0	-1.769
13	0.9202	-1.5	-4.018
14	0.4052	0	-1.7692
15	1.1162	0	-4.8739
16	-4.8739	0	-1.1162
17	-11.892	0	-2.2737
18	-13.538	0	-3.1005
19	-4.8739	0	-1.1162
20	1.1163	0	-4.8739
21	2.7234	0	-11.892
22	2.7234	0	-11.892
23	2.7234	0	-11.892
24	2.7234	0	-11.892
25	2.7234	0	-11.892
26	2.7234	0	-11.892

27	2.7234	0	-11.892
28	2.7234	0	-11.892
29	2.7234	0	-11.892
30	2.7234	0	-11.892
31	2.7234	0	-11.892
32	2.7234	0	-11.892
33	2.7234	0	-11.892
34	2.7234	0	-11.892

4.1 Pipeline Displacement

In this particular chapter, the results from ANSYS simulation are discussed in detail. Then the results were compared with the previous student studies. Some calculations are conducted to support the simulation outcome. The errors of the results are discussed at the end of the chapter.

The focused area on the pipeline was the bottom part of riser which connected to the expansion spool.

Figure 4.3 shows the simulation resulted a deformed line which is in blue color. Largest total displacement experienced by node number 27 with 0.57947 meter. The node is located at the pipeline expansion area.

Appendix C lists the result of displacement on each nodes in x, y, and z-directions. The maximum displacement for each direction also indicated.

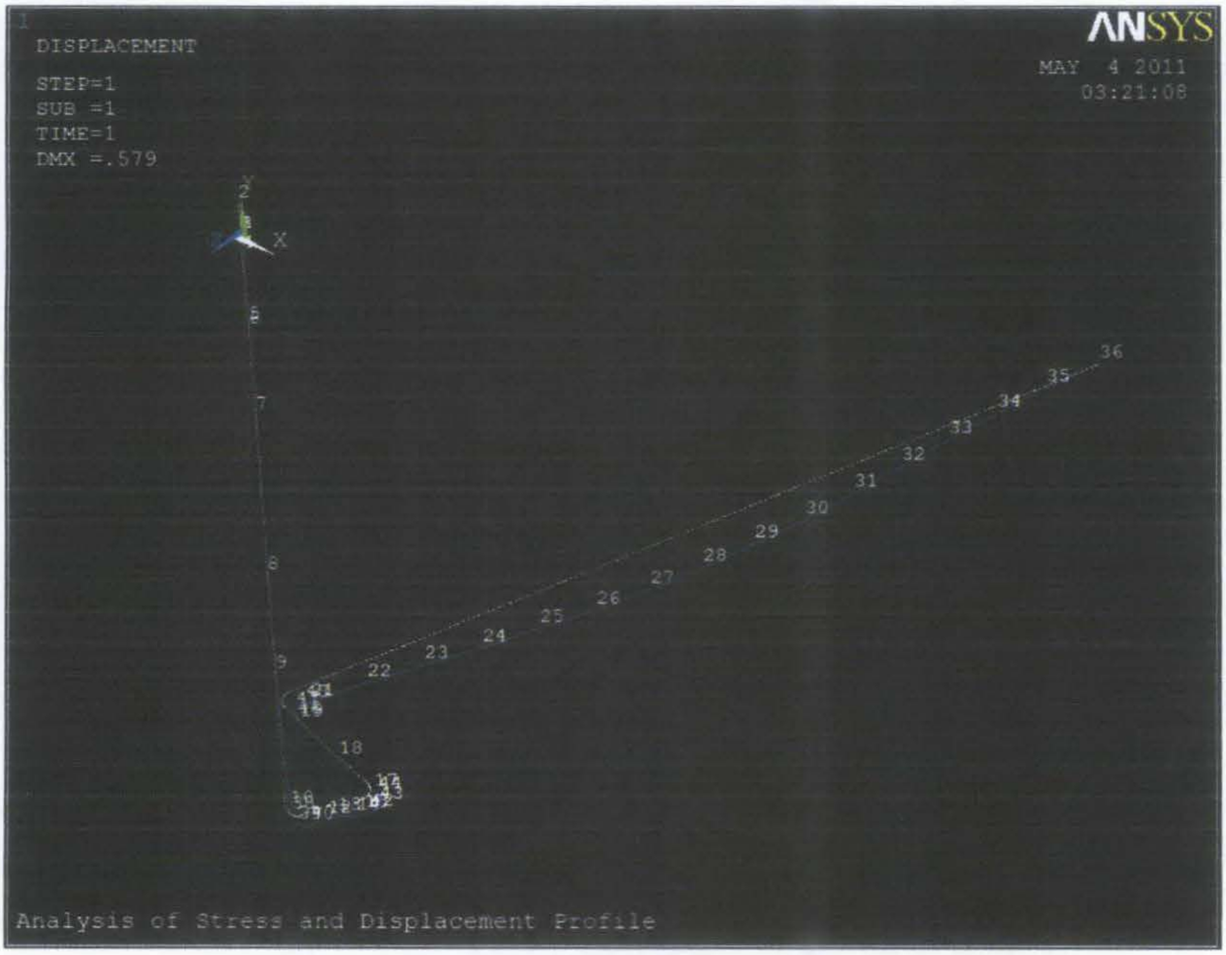


Figure 4.3 : Original and Deformed Pipeline after Simulation

4.1.1 Displacement on X-direction

Figure 4.4 shows the displacement of each node in x-direction. The maximum displacement occurred on node 27 along the pipeline which indicated by the red color line with value of 0.579 meters. Along the pipeline the displacement is only experienced very little displacement since the pipeline is located on the seabed and has been buried and clamped to the sea floor.

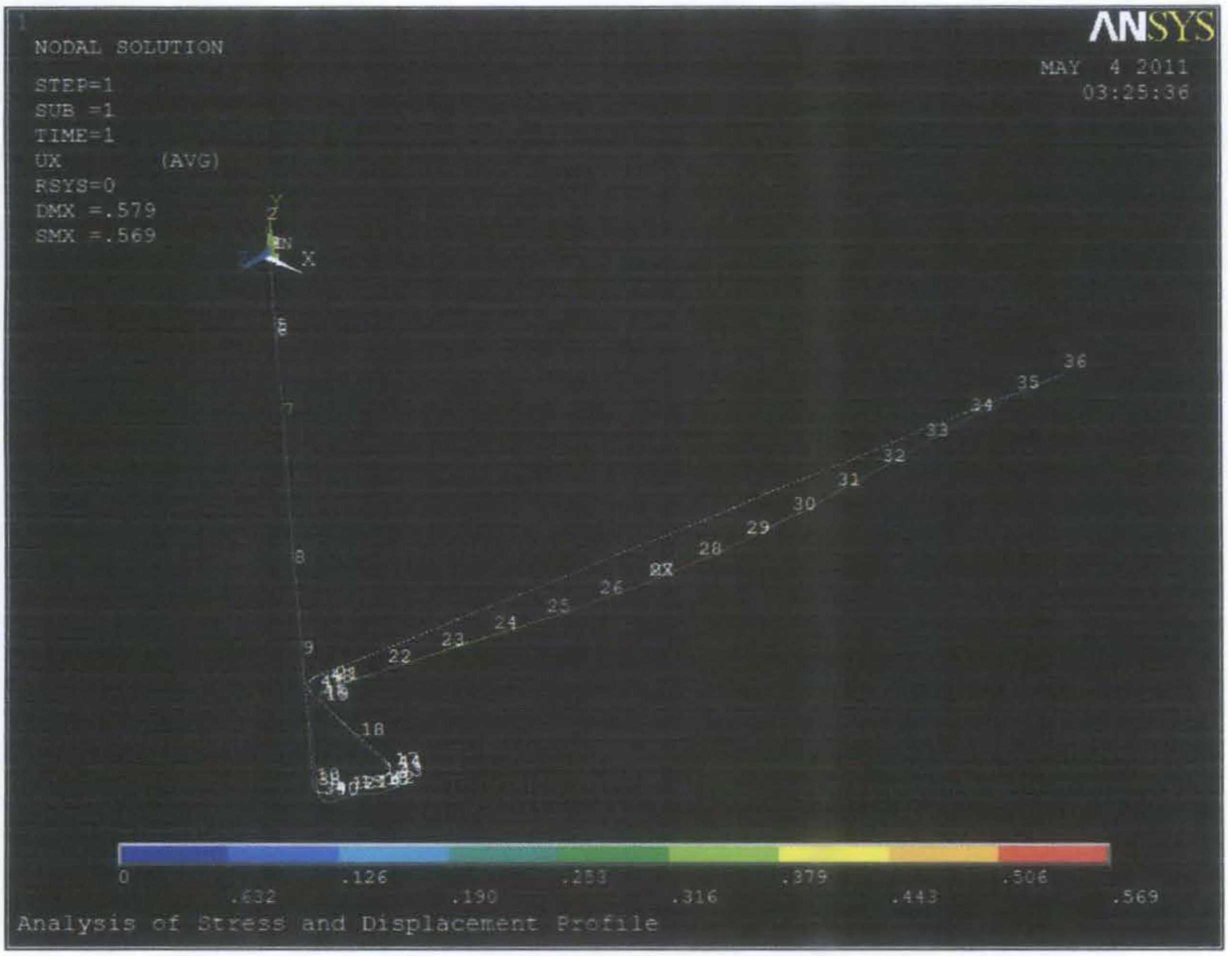


Figure 4.4 : Pipeline Displacement in X-direction with Magnitude Scale

4.1.2 Displacement on Y-direction

Figure 4.5 and 4.6 shows the displacement of each node in y-direction. The maximum displacement occurred on node 38 at the bottom of the riser with value of -0.542 meters. Along the expansion spool, the pipeline experienced only a very little displacement since the node is located at the first bend connected the riser and the expansion spool. At the expansion spool, the pipeline is not experiencing big displacement because the expansion spool has created friction which restricted the movement between the expansion spool and the seabed.

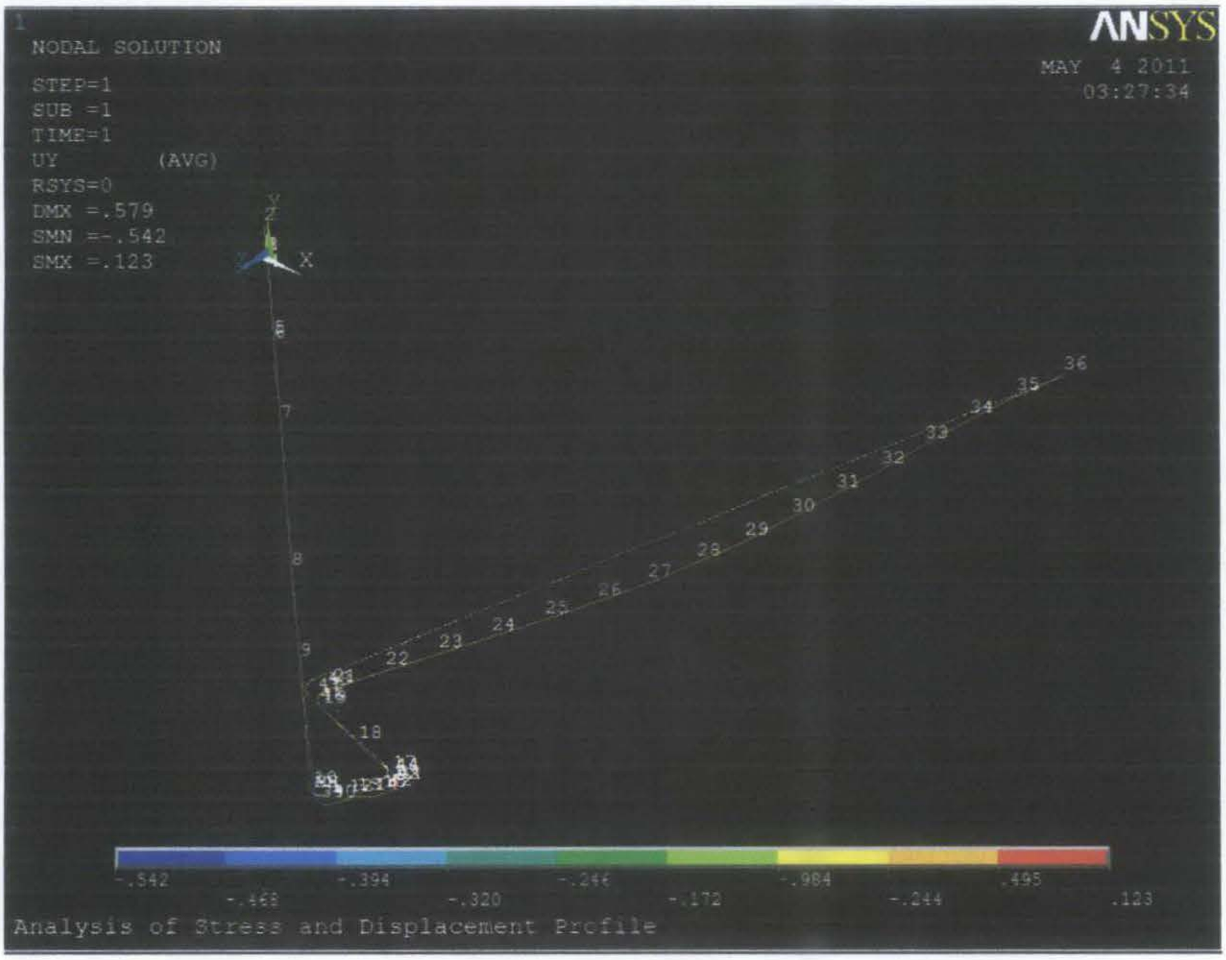


Figure 4.5 : Pipeline Displacement in Y-direction with Magnitude Scale

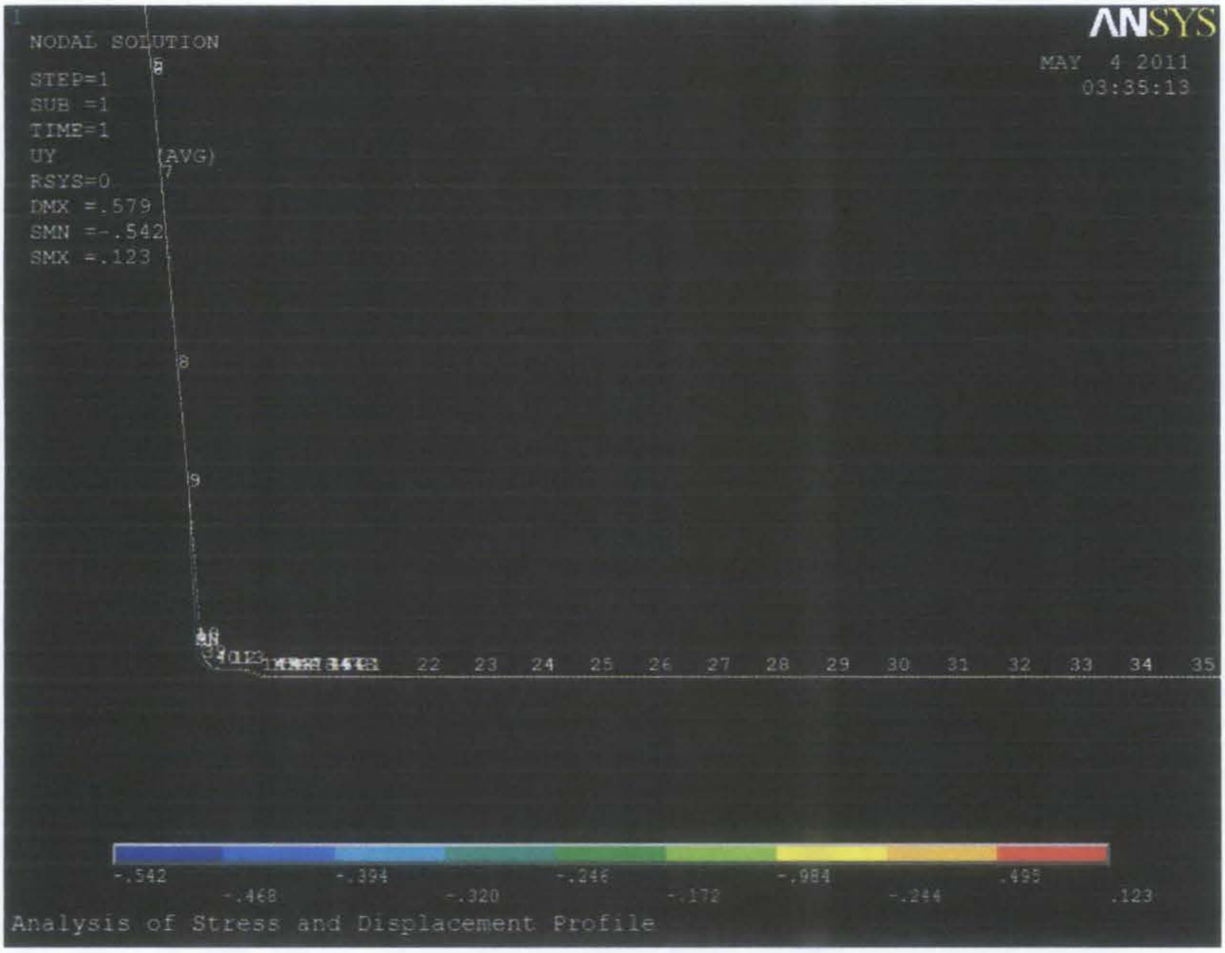


Figure 4.6 : Side View of the displacement in Y-direction

4.1.3 Displacement on Z-direction

Figure 4.7 shows the displacement of each node in z-direction. The maximum displacement occurred on node 27 with the value of 0.11 meters. For horizontal pipeline, the displacement is just a small displacement and this indicated that the stress on the pipeline is also quite low. The simulation did not indicate any value of stress.

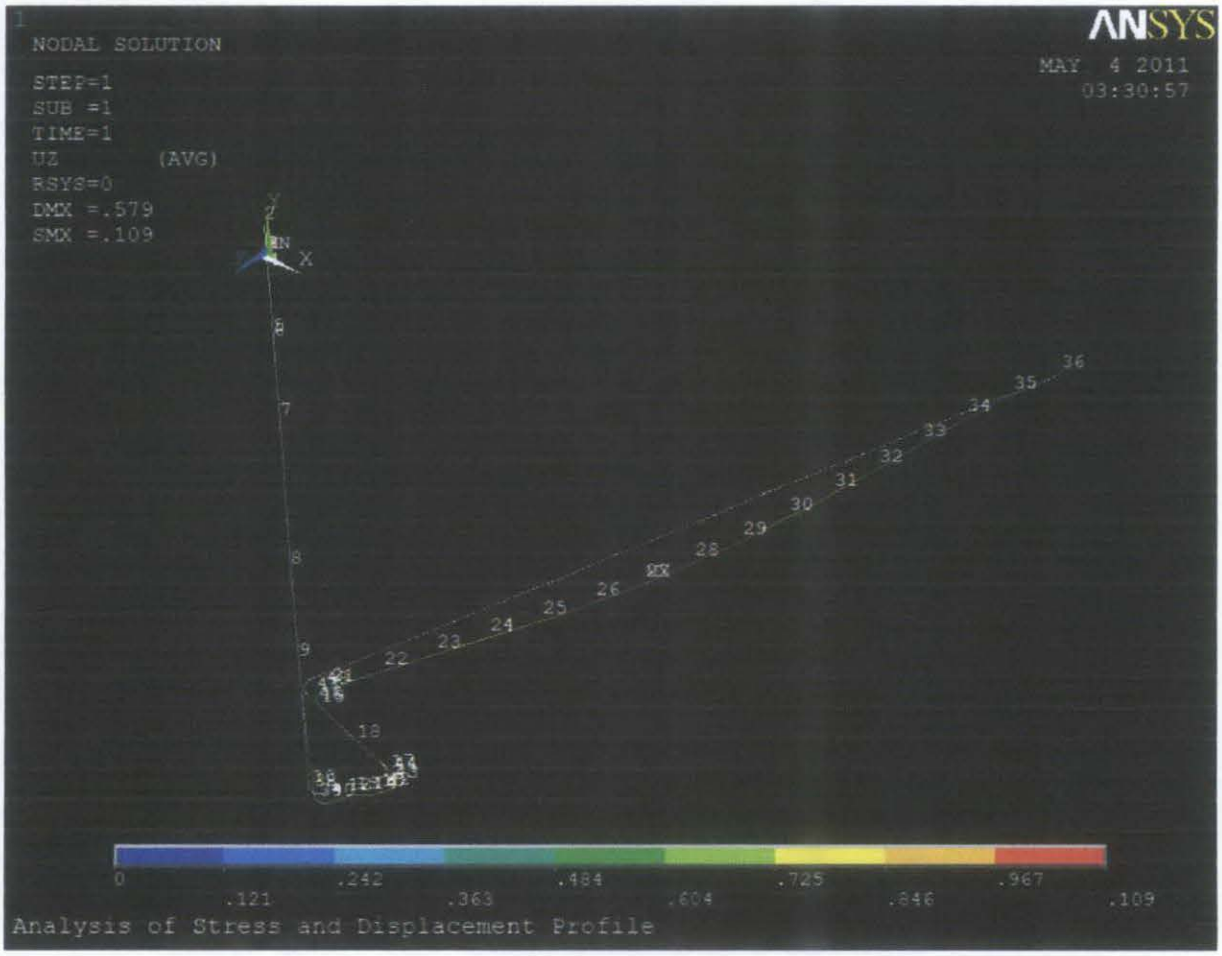


Figure 4.7 : Pipeline Displacement in Z-direction with Magnitude Scale

4.2 Comparison Result

4.2.1 Displacement of Pipeline

Table 4.2 shows the percentage error for result obtained from author ANSYS simulation to compare with the previous study result. The value of displacement of previous study has been referred to make the comparison. Only certain nodes were compared since author need to pick the nodes with similar coordinates.

$$\text{Formula of Percentage Error (\%)} = \frac{(\text{Previous} - \text{ANSYS}) \times 100}{\text{Previous}}$$

Table 4.2 : Comparison of Displacement Values on Specified Nodes

Node	ANSYS			Previous Study			Error in Percentage (%)		
	Ux (m)	Uy (m)	Uz (m)	Ux (m)	Uy (m)	Uz (m)	Ux	Uy	Uz
9	0	6113.3	0	0	0.72	0	0	84.8	0
12	0.182	0	0.672	0.01	0.96	0.05	17.2	96	12.4
14	0.205	0	0.723	0.02	0.98	0.09	18.5	98	63.3
15	0.209	0.51	0.733	0.24	0.98	0.1	12.9	47.9	6.33
17	0.221	0	0.685	0.23	0.98	0.14	3.9	98	38.9
18	0.222	0	0.641	0.09	0.98	0.16	14.6	98	30.1
21	0.242	0	0.462	0.21	0.98	0.22	15.2	98	11

All the errors are very high, which means the result obtained from ANSYS is inaccurate. The objective of the study is to obtain the best displacement result which to reduce the displacement values from previous student studies. Hence, the result that the author has obtained is slightly lower than the result obtained from previous study. Author has successfully decreased the displacement value and the error is just a comparison between author ANSYS simulation and previous study simulation.

The results might be difference due to the input displacement value on each direction of the pipeline. Author put zero displacement value on x and y direction since it was clamped to the platform directly and for the z-direction author put the displacement value of 10. The result shows that the movement of the pipeline has many restrictions from each of the directions and furthermore the restriction from the friction between pipeline and the seabed also has make great impact to the result of the simulation.

4.2.2 Forces and Bending Moments

Appendix D lists the values of forces, F and bending moment, M resulted from the simulation process. Table 4.3 shows that the comparison values of forces in x, y and z-direction exerted on certain nodes between ANSYS result from author and previous study.

Table 4.3 : Comparison of Forces Displacement Values on Specified Nodes.

Node	ANSYS			Previous Study			Error in Percentage (%)		
	Fx (kN)	Fy (kN)	Fz (kN)	Fx (kN)	Fy (kN)	Fz (kN)			
7	0.547	0	0.415	2.1	2.33	5.33	74	100	92.2
8	0.101	0	0.557	1.02	1.16	2.38	90.1	100	76.6
9	0.215	0	0.177	1.36	1.16	3.17	84.2	100	94.4
10	0	0	2.2576	0.47	3.14	8.78	100	100	74.3

The difference of forces are high because maybe author and previous student neglected several aspect and did not consider some of this criteria:

- Hydrostatic pressure
- Environmental forces
- Weight of pipeline and product

4.3 Calculation of Static Analysis Riser

A.J Ferrante (1982) wrote the static analysis used for the design that utilizes finite element method. The riser is subjected to loads to its weight, buoyancy force and hydrodynamic forces. The hydrodynamic forces consist of three elements; the viscous force, therefore depending on the Reynolds number, the inertial force due to the added mass and the lift which acts in a direction perpendicular to the plane defined by the riser and the direction of flow. The lift is a consequence of the vortex shedding and is considered only in the dynamical analysis.

The hydrodynamic force coefficients used for the riser pipes above the seabed are as per PTS 20.196 (Ref. 1), as follows:

- Inertia Coefficient : 2.0
- Drag Coefficient : 0.7
- Lift Coefficient : 0.0

This alternative method, calculation of static analysis was conducted to obtain the maximum stress applied on the pipeline,

The maximum stress, $\sigma_t = T_i + \sigma_i$ [4.1]

Where T_i = stress due tension

σ_i = stress due to bending moment

and

$$\sigma_i = \frac{|M_i|D}{2C_m} \quad [4.2]$$

$$T_i = \frac{4 P_i}{\Pi (D^2 - D_i^2)} \quad [4.3]$$

Where M_i = maximum stress due to bending moment (Nm)

D = external diameter (m)

D_i = internal diameter (m)

C_m =coefficient of the inertia of the riser (kgm^2)

P_i = internal pressure (kPa)

Thus, the maximum stress on the pipeline is $\underline{236 \times 10^6}$ Nm.

4.4 Calculation of Pipeline End Expansion Analysis

This method is conducted to get the pipeline end expansion or called pipeline displacement. It was provided in MLNG project document.

The distance (z) from the free end to the virtual anchor point is calculated using the following equation:

$$z = \left(\pi r_m^2 \frac{P}{f}\right)\left\{1 - 2\upsilon + \left(\frac{2E\alpha t(T_i - T_w)}{P r_m}\right)\exp\left(-z\beta\right) + \frac{2E\alpha t(T_w - T_{ins})}{P r_m}\right\}$$

While, the free end expansion, Δ, is estimated using the formula below

$$\Delta = \frac{\alpha(T_i - T_w)}{\beta}\{1 - \exp(-z\beta)\} + \frac{1}{E}\left\{(0.5 - \upsilon)\frac{P r_m z}{t} - \frac{f z^2}{4\pi r_m t}\right\} + \{\alpha z(T_w - T_{ins})\}$$

Details explanation about the equation and the calculation can be referred in Appendix F. The end expansion of pipeline value is 2.204m.

4.5 Error and Uncertainties

Result of maximum displacement from ANSYS simulation is gathered in combination of all three Cartesian axis which are x, y and z. While the displacement or expansion is obtained from the calculation of pipeline and expansion analysis. The result will be then compared with previous study. For stress on the pipeline, the result will be obtained through calculation of static analysis

The maximum combination of all the directions is 1.22 meters. While from the calculation from the pipeline expansion analysis shows the result of 2.204 meters and results from previous study is 1.209 meters.

For stress on the pipelines, calculation of static analysis resulted stress equal to 236×10^6 Nm and the stress allowable limit is fifty percent from 224 Nm which 336 Nm. All of the results obtained in this study have a very high difference compared to previous study result.

The main reason of the errors is because the software used in this study, ANSYS is not suitable due to some factors such as the software do not take in consideration of critical aspect such as wave current, seabed friction, submerge weight and other environmental loads in the analysis. It is not appropriate to offshore pipeline since it specify more on power facilities piping while offshore hydrocarbon pipeline should comply with ANSI B31.8. For pipe modeling in ANSYS, the application of meshing nodes was done automatically. Author did not have the chance to conduct it manually. Thus, it affected the result since high approximation contributes to inaccuracy of the result.

Possible errors and uncertainties might occur due to insufficient data of the MLNG Pipeline project such as the paper drawings of pipelines that indicates details measurement named

Pipeline General Agreement and documents of result obtained for pipeline displacement and forces by PCSB to make the comparison of the results.

The calculations required some criteria that were not provided in the project document such as temperature decay coefficient per meter pipe. Author did some research and took general temperature decay coefficient for steel pipe which excluded the coefficient for the pipeline insulation and concrete coating.

All the reasons stated above are some of the factors contributed to inaccuracy of result. Besides, lack of knowledge and understanding of the software, how to handle the software also affected the results of the simulation.

CHAPTER 5
CONCLUSION AND RECOMMENDATIONS

The main objectives of this study are achieved within one year period. Modeling and simulation using ANSYS were conducted in order to determine the displacement experienced by pipeline attached to riser named expansion spool. The result obtained from ANSYS for maximum displacement of expansion spool is 1.22 meter on node 27 which is located at the horizontal pipeline.

This study improved the result obtained in previous study because author did consider the thermal and fluid analysis in ANSYS instead of structural analysis alone. Author had revised the previous study report to extract more relevant data to be used in the simulation. Finally, the percentage error has been reduced compared to the previous study report.

From the early stage of the study, the author faced a lot of difficulties. Starting from the familiarization of the ANSYS software until the running of simulation several obstacles faced. Calculation of static analysis and expansion analysis were conducted to verify the simulation results. The result obtained for maximum stress and pipeline end expansion are 236×10^6 Nm and 2.204 meter respectively.

The methodology of this project has been going through smoothly until the fourth stage which is graphical modeling and simulation. The graphical modeling and simulation had taken the longest time to accomplish. Author faced a lot of difficulties using ANSYS since author has never been introduced to this software before. ANSYS is the biggest obstacle for author since author need to learn by himself through friends and through internet. Lack of knowledge and understanding of the software gave big impact to the result, and created lots of difficulties since there were some constraints using this software and among them is the software itself is not put in consideration of some important parameters such as environmental loads in the analysis

Based on this study, author could conclude that ANSYS is not suitable software to be use for offshore pipeline analysis study. Application of other software might be help such as AutoPIPE, PipeSIM , DIANA Pipe or CAESAR-II. All the software is not available in UTP except for PipeSIM, but for PipeSIM, author found that to conduct the simulation perfectly, we would required a lot more data such as environmental loads, power pumps and compressor.

The main conclusion could be made by the author is to obtain the best result of any simulation study, familiarization of the software is the most important part. Without further familiarization of the software, the study would not be completed. Hence, a lot of time needed to spend with the software in order to succeed successfully.

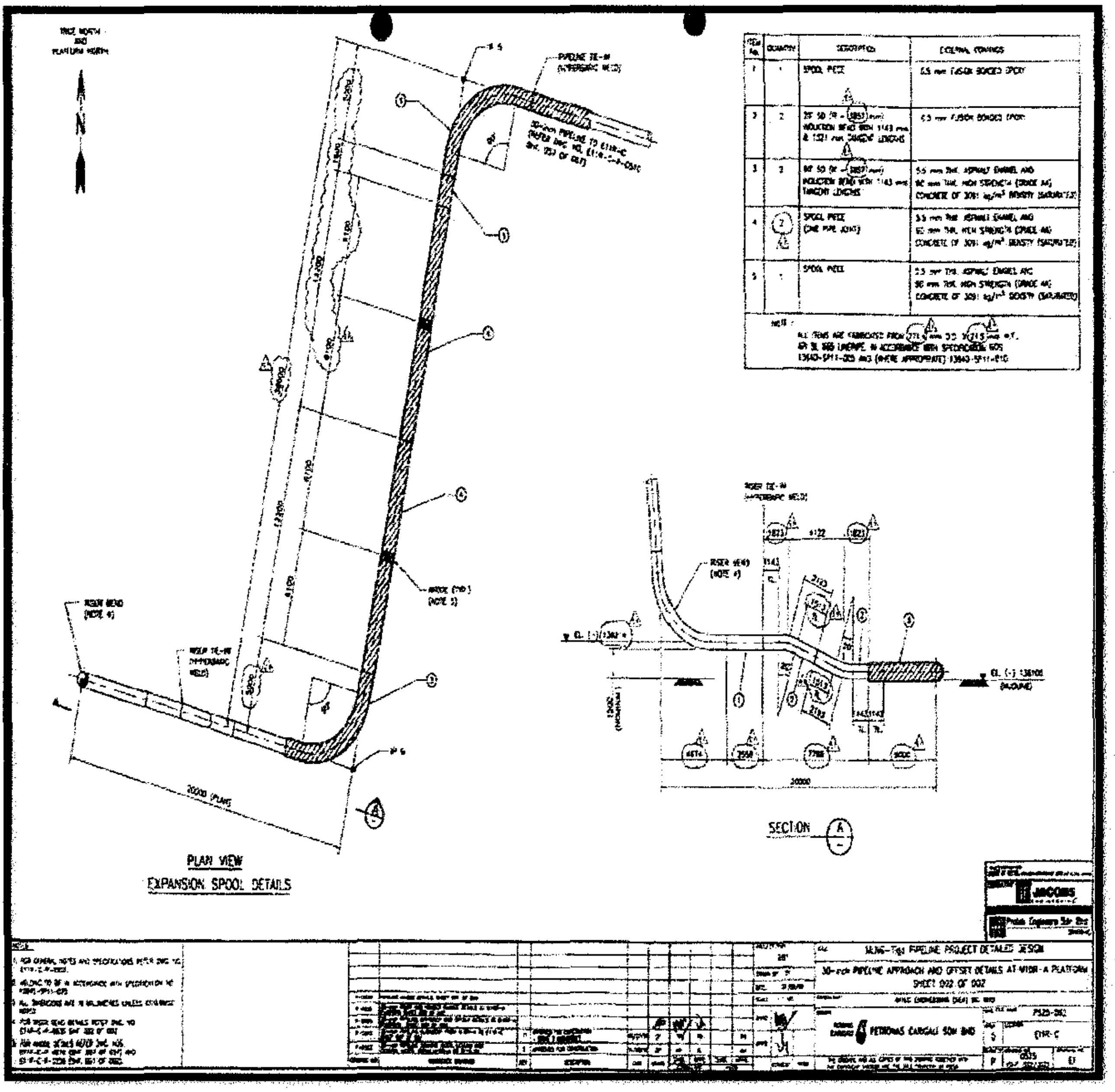
REFERENCES

1. Saeed Moaveni, ***Finite Element Analysis Theory and Application with ANSYS***, 2nd Edition, Prentice Hall, New Jersey, 2003.
2. Shawn Kenny, ***Thermal Expansion Analysis***, Faculty of Engineering and Applied Science, ENGI 8673 Subsea Pipeline Engineering, Memorial University, Winter 2009.
3. Michael M. Bernitsas, John E. Kokarakis and Ashjar Imron, ***Large Deformation 3D Static Analysis of Deep Water Marine Risers***, Applied Ocean Research, 1985, Vol. 7, No.4
4. ***ASME B31.4***, Liquid Petroleum Transportation Piping, American Society of Mechanical Engineers, New York, NY.
5. ***ASME B31.8***, Gas Transmission and Distribution Piping, American Society of Mechanical Engineers, New York, NY.
6. Bai.Y, ***Pipelines and Risers***, Elsevier Ocean Engineering Book Series, Vol.3, Elsevier.
7. Paul Jukes, Ayman Eltaher, James Wang, Billy Duron, J P Kenny.2008. ***The use of Advanced Finite Element Analysis Tools for the Design and Simulation of Subsea Oil and Gas Pipelines and Components***. PECOM November 11-13, 2008, Villahermosa, Tabasco, Mexico.
8. Trent Brown, SPE, ITP InTerPipe, Inc., and Paul Jukes, SPE, and Jason Sun, JP Kenny, Inc. ***Mechanical Design of Subsea and Buried LNG Pipelines.*** Offshore Technology Conference, 4-7 May 2009, Houston, Texas.
9. Demir Karsan MCE, ***Structural Analysis of the Pipeline Risers***, Brown & Root (UK) Ltd, Engineering Department, 1st March 1977.
10. PETRONAS Technical Standards, ***PTS 20.196, Pipeline Engineering***, 1994.
11. PETRONAS Technical Standards, ***PTS 20.082, Submarine Pipeline and Riser Design***, 1991.
12. ANSYS User's Manual: ***Procedures, Vol. I***, Swanson Analysis System, Inc.
13. ANSYS User's Manual: ***Commands, Vol. II***, Swanson Analysis System, Inc.
14. ANSYS User's Manual: ***Elements, Vol. III***, Swanson Analysis System, Inc.

APPENDICES

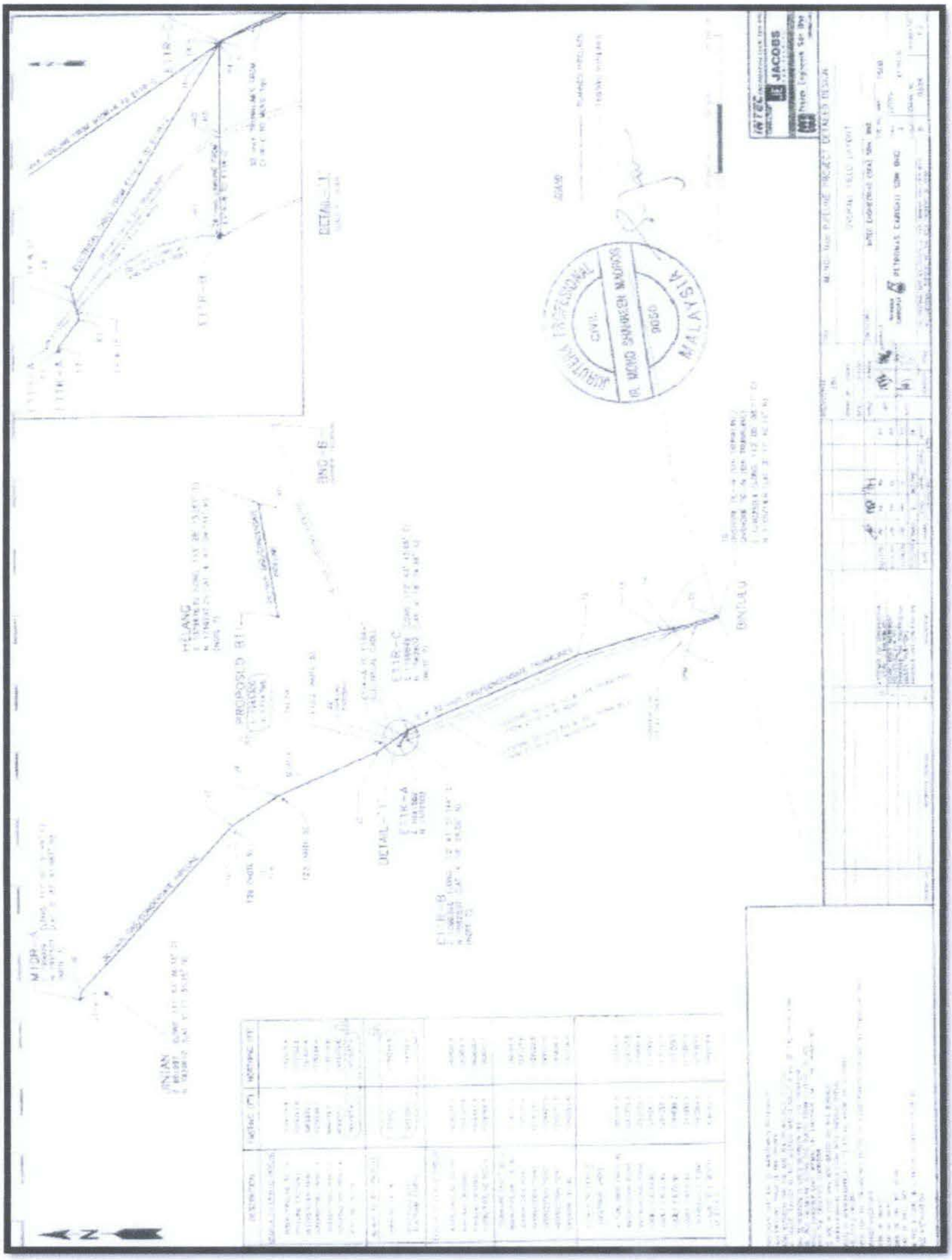
APPENDIX A

DETAIL DRAWING OF EXPANSION SPOOL



APPENDIX B

PROJECT LAYOUT DRAWING



APPENDIX C

DISPLACEMENT ON NODES

TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

NODE	UX	UY	UZ	USUM
1	0.0000	0.0000	0.0000	0.0000
2	0.65639	-0.53579	0.59093	0.88482
3	0.16315	-0.13113	0.14536	0.21891
4	0.0000	993.87	0.0000	993.87
5	0.0000	1858.0	0.0000	1858.0
6	0.0000	1907.6	0.0000	1907.6
7	0.0000	2958.9	0.0000	2958.9
8	0.0000	4904.7	0.0000	4904.7
9	0.0000	6113.3	0.0000	6113.3
10	0.13237	-0.53386	0.58841	0.14495
12	0.18228	0.0000	0.67113	0.19424
13	0.18844	0.0000	0.68525	0.20052
14	0.20464	0.0000	0.72235	0.21702
15	0.20944	0.50095	0.73333	0.22191
17	0.22095	0.0000	0.68506	0.23133
18	0.22180	0.0000	0.64091	0.23087
19	0.22405	0.23038	0.54246	0.23053
21	0.24152	0.0000	0.46172	0.24590
22	0.29788	0.0000	0.56946	0.30328
23	0.36919	0.0000	0.70577	0.37587
24	0.44173	0.0000	0.84446	0.44973
25	0.50445	0.0000	0.96435	0.51358
26	0.54887	0.0000	0.10493	0.55881
27	0.56917	0.0000	0.10881	0.57947
28	0.56212	0.0000	0.10746	0.57230
29	0.52714	0.0000	0.10077	0.53668
30	0.46623	0.0000	0.89130	0.47468
31	0.38406	0.0000	0.73421	0.39101
32	0.28787	0.0000	0.55033	0.29309
33	0.18756	0.0000	0.35856	0.19096
34	0.95626	0.0000	0.18281	0.97358
35	0.27191	0.0000	0.51982	0.27684
36	0.0000	0.0000	0.0000	0.0000
38	0.13808	-0.54217	0.59756	0.15056
39	0.15452	-0.48266	0.61783	0.16648
40	0.16871	-0.24825	0.64005	0.18046
42	0.21217	0.96461	0.73959	0.22469

***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

LOAD STEP= 1 SUBSTEP= 1

TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

NODE	UX	UY	UZ	USUM
43	0.21814	0.12349	0.73070	0.23005
44	0.22069	0.34836	0.69681	0.23143
46	0.22453	0.16079	0.52172	0.23051
47	0.22863	0.17017	0.46635	0.23334
48	0.23757	-0.12537	0.45267	0.24184

MAXIMUM ABSOLUTE VALUES

NODE	27	38	27	27
VALUE	0.56917	-0.54217	0.10881	0.57947

APPENDIX D

FORCES AND BENDING MOMENT ON NODES

PRNLD Command

File

PRINT SUMMED NODAL LOADS

***** POST1 SUMMED TOTAL NODAL LOADS LISTING *****

LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING X,Y,Z SOLUTIONS ARE IN THE GLOBAL COORDINATE SYSTEM

NODE	FX	FY	FZ	MX	MY	MZ
1	0.98835E+08	0.54115E+10	0.58065E+09	0.35047E+11	-0.12331E+12	-0.50141E+11
4	0.11441E+09		0.10470E+09			
5	0.10805E+10		0.74077E+09			
6	-0.10842E+10		-0.71062E+09			
7	0.54651E+09		0.41495E+09			
8	-0.10140E+10		-0.55603E+09			
9	0.21460E+10		0.17645E+10			
10			2.2576			
12	13.155	-0.19451E+11	-94.100			
13	17.507	0.12420E+11	51.403			
14	3.7342	0.30555E+10	24.746			
15	-22.459		179.18			
17	111.40	-0.78826E+09	76.124			
18	72.133	-0.46709E+09				
19	-75.926		142.04			
21	-28.420	0.18180E+09	-83.525			
22	-3.8505	0.11557E+09	-12.075			
23	-3.2400	0.89180E+08	-6.0849			
24	2.2030	0.96250E+08	-17.437			
25		0.94356E+08	-24.008			
26		0.94864E+08				
27	-2.1844	0.94728E+08	-17.995			
28		0.94764E+08				
29	-5.7448	0.94754E+08	-8.6247			
30	2.4723	0.94757E+08	-39.928			
31	3.6851	0.94756E+08	-31.969			
32	3.8425	0.94756E+08	1.9185			
33		0.94756E+08				
34		0.94756E+08	-9.7826			
35		0.94756E+08	2.6745			
36	0.10820E+10	0.47378E+08	-0.77567E+09	-0.98158E+10	0.31450E+13	0.43031E+09
39				-63.866		
40	-13.188		61.101	59.641		
42	-9.1150		-124.36			
44	-475.85		-22.414			
46	-43.877		-45.249			
48	-35.173		-15.336			

***** POST1 SUMMED TOTAL NODAL LOADS LISTING *****

LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING X,Y,Z SOLUTIONS ARE IN THE GLOBAL COORDINATE SYSTEM

NODE	FX	FY	FZ	MX	MY	MZ
TOTAL VALUES						
VALUE	0.29700E+10	0.17526E+10	0.15632E+10	0.25231E+11	0.30217E+13	-0.49711E+11

APPENDIX E

CALCULATION OF STATIC ANALYSIS

The maximum stress, $\sigma_t = T_i + \sigma_i$ [4.1]

Where T_i = stress due tension

σ_i = stress due to bending moment

and

$$\sigma_i = \frac{|M_i|D}{2C_m} \quad [4.2]$$

$$T_i = \frac{4\,P_i}{\Pi\,(D^2 - D_i^2)} \quad [4.3]$$

Where M_i = maximum stress due to bending moment (Nm)

D = external diameter (m)

D_i = internal diameter (m)

C_m =coefficient of the inertia of the riser (kgm^2)

P_i = internal pressure (kPa)

$$\sigma_t = T_i + \sigma_i = \frac{4(119.6 \text{ x } 10^5)}{3.142(0.7714^2 - 0.7284^2)} + \frac{(0.302)(0.7714)}{2(2.0)}$$

Thus, the maximum stress on the pipeline is <u>236×10^6 Nm.</u>

APPENDIX F

CALCULATION OF END EXPANSION ANALYSIS

The distance (z) from the free end to the virtual anchor point is calculated using the following equation:

$$z = \left(\pi r_m^2 \frac{P}{f}\right)\left\{1 - 2\upsilon + \left(\frac{2E\alpha t(T_i - T_w)}{P r_m}\right)\exp(-z\beta) + \frac{2E\alpha t(T_w - T_{ins})}{P r_m}\right\} \qquad [4.4]$$

The free end expansion, Δ, is estimated using the formula below :

$$\Delta = \frac{\alpha(T_i - T_w)}{\beta}\{1 - \exp(-z\beta)\} + \frac{1}{E}\left\{(0.5 - \upsilon)\frac{P r_m z}{t} - \frac{f z^2}{4\pi r_m t}\right\} + \{\alpha z(T_w - T_{ins})\} \qquad [4.5]$$

where:

α	=	thermal expansion coefficient of steel pipe
	=	1.17×10^{-5} mm/mm/°C
β	=	temperature decay coefficient (per m)
υ	=	Poisson's ratio, 0.3 for steel
E	=	Young's modulus, 2.07×10^{11} N/m^2
f	=	friction resistance per unit length
P	=	design pressure (N/m^2)
r_m	=	mean radius of steel pipe (m)
t	=	wall thickness of steel (m)
T_i	=	pipeline inlet temperature (°C)
T_w	=	seawater temperature during operation (°C)
T_{ins}	=	installation temperature (°C)

$$z = \left(\pi r_m^2 \frac{P}{f}\right)\left\{1 - 2\upsilon + \left(\frac{2E\alpha t(T_i - T_w)}{P r_m}\right)\exp(-z\beta) + \frac{2E\alpha t(T_w - T_{ins})}{P r_m}\right\}$$

$$z = \left(3.142(0.375)^2 \frac{1196 \times 10^4}{0.48}\right)\left\{1 - 2(0.3) + \left(\frac{2(2.07 \times 10^{11})(1.17 \times 10^{-5})(0.0215)(65 - 21)}{1196 \times 10^4 (0.375)}\right)\right\}$$

$Z = 6844245.112$

$$\Delta = \frac{\alpha(T_i - T_w)}{\beta}\{1 - \exp(-z\beta)\} + \frac{1}{E}\left\{(0.5 - \upsilon)\frac{P r_m z}{t} - \frac{f z^2}{4\pi r_m t}\right\} + \{\alpha z(T_w - T_{ins})\}$$

$$\Delta = \frac{(1.17 \times 10^{-5})(65 - 21)}{0.01}\{1 - \exp(-684 \times 10^4 (0.01))\} + \frac{1}{207 \times 10^9}\left\{(0.5 - 0.3)\frac{1196 \times 10^4 (0.73)(684 \times 10^4)}{0.0215} - \frac{0.49(684 \times 10^4)^2}{4(3.142)(0.73)(0.0215)}\right\} + \{1.17 \times 10^{-5}(684 \times 10^4)(21 - 2$$

The free end expansion, $\Delta = 2.204$ m