Design and Analysis of In-Field Liner (IFL) and IFL Pulling Head for Underwater Pipeline

by

Muhammad Hazwan Bin Desa

Dissertation submitted in partial fulfillment of
the requirements for the
Bachelor of Engineering (Hons)
(Mechanical Engineering)

MAY 2012

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Design and Analysis of In-Field Liner (IFL) and IFL Pulling Head for Underwater Pipeline

by

Muhammad Hazwan Bin Desa

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfillment of the requirements for the
BACHELOR OF ENGINEERING (Hons)
(MECHANICAL ENGINEERING)

Approved by,

____

(Dr. Mokhtar Bin Awang)

UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK
May 2012

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.

MUHAMMAD HAZWAN BIN DESA

ABSTRACT

In-Field Liner (IFL) is generally a flexible composite liner pipe, where it is commonly used nowadays to replace the degraded oil and gas steel pipeline. This project is aimed to determine the feasibility and the capability of IFL pipe to sustain the force exerted by the pulling cable in the pulling process to replace the degraded underwater steel pipeline for oil and gas application. There are two main concerns for this project, which are to design a pulling head for IFL pipe and analysis of the IFL pipe in term if its reaction to a certain amount of pulling force used in the pulling process. For design purposes, CATIA V5R20 software is used whereas ANSYS v14.0 software is used for analysis and simulation purposes. The pulling head design is created using CATIA software according to the specifications of IFL pipe, so that the pulling head will fit properly on the IFL pipe when it is attached. For analysis of IFL pipe, the Finite Element Analysis (FEA) model of IFL pipe in its folded configuration (U-shaped form) will be simulated using ANSYS software in order to determine the ability of IFL pipe to withstand the pulling force exerted by the pulling cable and its reaction to the pulling load will be the priority of this project as this will determine the relevancy and the feasibility of the replacement process by pulling the IFL pipe through the inner side the degraded steel pipeline.

ACKNOWLEDGEMENT

First and foremost, I would like to take this opportunity to express my greatest appreciation to my Final Year Project supervisor, Dr. Mokhtar Awang for his valuable guidance and advices throughout my entire project. His supports, patience, and willingness to assist me in the problem or difficulties that I faced in my project have contributed tremendously to my project. A special thanks goes to Mr. Ehsan Mohammadpour, who has been a great help to me in understanding and mastering the Finite Element Analysis. He has lend me his help by teaching me the simulation works in this project I also wish to take this opportunity to express my utmost gratitude to the individual and parties that have contributed their time and efforts in assisting me to complete this project, directly or indirectly. Without their helps and assistances, there is no doubt that I would face some difficulties throughout the whole project. This project would not be in success without all of their supports, guidance and patience. I also would like to take this opportunity to thank the Final Year Project coordinator, Dr. Hasan Fawad for his effort in ensuring the project is progressed smoothly within the time frame given. Last but not least, I would like to give my utmost gratitude and appreciation to my family and fellow friends for their support and inspiration so that this project can be completed successfully.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1

1. INTRODUCTION

1.1 PROJECT BACKGROUND

The underwater or subsea pipeline is usually used to transport fluids such as crude oil, natural gas and other hydrocarbon products which have been extracted from the wells deep in the sea to the offshore platform or to the onshore processing plant. The pipeline has long been considered as the most reliable and safest mode of long distance, high volume transport of crude oil, natural gas or other hydrocarbon products [1]. Despite this fact, the pipeline can and do fail due to the degradation on that pipeline after being exposed for a long period of time during its useful service life.

The deterioration on the crude oil transport pipeline for example, can lead to massive environmental hazards as the deteriorating pipeline may potentially resulting in leakage of the fluids transported using that pipeline. This is due to the fact that the material selection for the pipeline, which is usually carbon steel will be degraded greatly as the unprocessed hydrocarbon fluids are corrosive in nature [2]. Besides, the aggressiveness and the compositions of the subsea environment also could lead to the degradation of the transporting pipeline, where it contains chloride (Cl^-) ions which will react with the metal pipeline, especially carbon steel pipeline causing corrosion on it. In order to prevent these environmental hazards from occurring, the degraded pipeline needs to be upgraded or replaced with a new one. The replacement of that degraded pipeline will be done by implementing a trenchless pipeline rehabilitation method, where the high strength, flexible composite In-Field Liner (IFL) will be used by inserting it into the existing corroded pipeline and pulled through that pipeline without having to remove the degraded pipeline from its location.

Figure 1.1 shows the schematic view of the degraded steel pipeline replacement with the folded IFL pipe on the inner side of that steel pipeline.

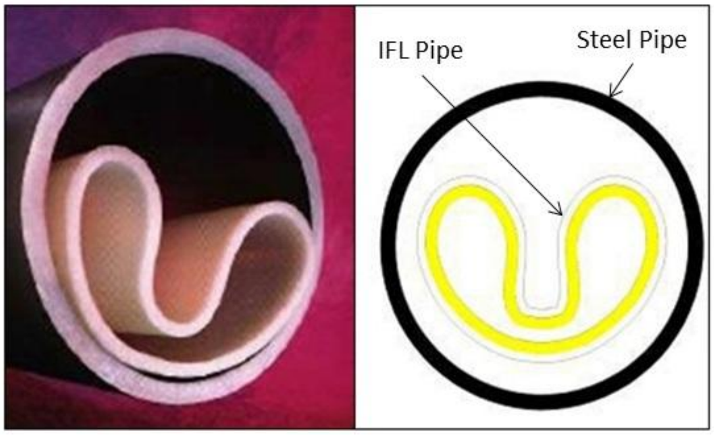

Figure 1.1: Schematic view of folded IFL pipe inside the steel pipeline

This trenchless pipeline rehabilitation method has been widely practiced nowadays, especially for underwater or underground pipeline as compared to the conventional method by replacing the whole corroded pipeline with a new one. The latter method is not being considered anymore today as it is obviously requires higher cost than the trenchless rehabilitation method. Figure 1.2 shows the model of the degraded underwater pipeline which has been constructed based on its actual configuration.

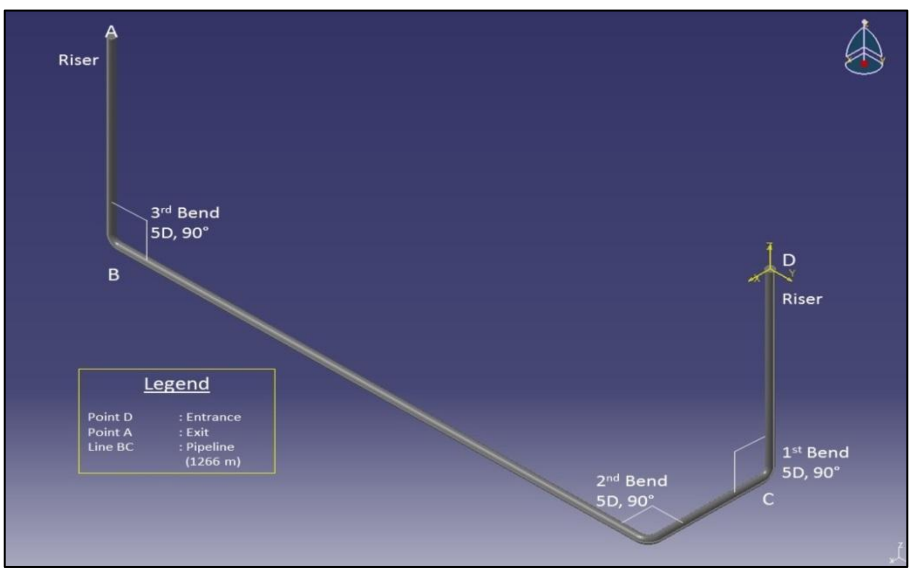

Figure 1.2: Model of the degraded underwater pipeline

This project is the actual industrial-based problem in the oil and gas industry, specifically in the replacement of the degraded steel pipeline with the pulling of IFL pipe through the inner side of the degraded pipeline. Due to a very long pipeline with the length of 1266 m, a certain amount of pulling force is needed in order to pull the composite IFL pipe with the same length of 1266 m through the inner side of the degraded pipeline. The main equipment that will be used in the IFL installation process comprise of IFL drum, IFL Folded machine, winch, auto sensing load controller, pulling cable, riser and IFL proofing tool. Briefly, the insertion process mainly consisted of transferring the pulling cable by using a pigging robot. One end of the pulling cable will be connected to the winch, where it will pull the IFL pipe by applying a very large pulling force and the other end of pulling cable will be attached to the IFL pulling head. Prior to the insertion process, the composite IFL pipe will be folded into "U-shaped" form by using a U-Folded machine. The process of folding the IFL pipe into "U-shaped" form is shown in the Figure 1.3.

Figure 1.3: Folding process for IFL pipe into "U-shaped" form

Then, the process followed by the IFL pull-in with the pulling cable. The required longitudinal pulling load for the installation of IFL has already been hand-calculated with the amount of 79151.3 N.

1.2 PROBLEM STATEMENT

This project will utilizes the concept of trenchless pipeline rehabilitation method, where it involves the insertion and pulling process of In-Field Liner (IFL) pipe into the degraded underwater pipeline without having to remove the existing pipeline from its location on the seabed. Thus, before conducting and implementing the insertion process for IFL pipe into the underwater pipeline in the real situation, it is very important to determine and predict the mechanical responses of the composite liner pipe including stress and strain behavior of the composite IFL pipe during the pulling process by simulating the pulling process of the liner itself. This simulation will provide the information on the safety of IFL physical conditions and the feasibility of the configuration for IFL pipe itself during the installation process.

Before the composite liner pipe is inserted into the pipeline, it will pass through the folding machine to fold the liner into "U-shaped" form. By folding the liner into "U-shaped" form, the friction force between the liner pipe and the inner wall of the steel pipeline will be at minimum during the pulling process. The folded composite liner pipe then will be inserted into the steel pipeline and it will be pulled by a steel cable through the entire length of the steel pipeline. The liner pipe is a fiber-reinforced composite pipe which exhibits complicated mechanical properties such as the orthotropic property of fiber-reinforced material. Due to this fact, the sustainability of the liner pipe which comprise of composite materials must be determine whether it can withstand the maximum pulling force exerted on it by the pulling cable. The main concern in this project is to determine the effects on the IFL pipe and its mechanical responses under a very large pulling force, either it manage to sustain the applied load or fail to sustain it which will eventually lead to the failure of the IFL pipe. Besides, the pulling cable will be attached to the pulling head and therefore, a suitable design for IFL pulling head also must be proposed and produced.

1.3 OBJECTIVES

The objectives of this project are to determine the effects and mechanical responses of the composite In-Field Liner (IFL) pipe in term of its physical condition when it is subjected to the pulling force by the pulling cable and to produce a suitable design for the IFL pulling head. The mechanical responses of the composite liner pipe when it is subjected to the pulling force will be simulated by using Finite Element Analysis (FEA) method in order to get all the pertinent information regarding its sustainability and effects on its physical elements when it is being pulled by the pulling cable with the designated longitudinal pulling force. The ability of the liner pipe to withstand the pulling force with no failure or fracture occurred can be verified from this simulation to ensure the feasibility of the liner pipe configuration during the replacement process, installation method used and the safety of the liner pipe during the pulling process. Furthermore, the IFL pipe needs a pulling head to enable it to be pulled into the degraded pipeline by the pulling cable and the suitable design of the IFL pulling head should be produced.

1.4 SCOPE OF STUDY

This project is actually a design and simulation-based project in which it emphasizes the designing skill and analyzing the problem by means of simulation of the real model using Finite Element Analysis (FEA) method so that the characteristic or mechanical responses of that particular model under certain condition can be obtained. This project will focus on the ability to understand the problem encountered and to use all the necessary tools needed in order to verify the feasibility and relevancy of the proposed solution to the problem. In this case, the problem is the effect on the physical condition and the sustainability of the composite IFL pipe when it is subjected to the pulling force to pull the liner pipe through the inside of degraded underwater pipeline by using the pulling cable. In order to determine the effects on the liner pipe and to verify whether the composite liner pipe can withstand the pulling force exerted on it, a simulation of that composite liner model must be done by using the actual parametric values of the properties for that liner pipe so that the simulation result obtained will be accurate to verify the feasibility on the implementation of this pipeline replacement method. Besides, this project also will focus on the ability to utilize and hone the design skill by producing a suitable and proper design for the IFL pulling head using the CAD software. Overall, this project mainly uses the Finite Element Analysis (FEA) method to obtain the required results from the simulation on the IFL pipe during the pulling process.

CHAPTER 2

2. LITERATURE REVIEW

2.1 TRENCHLESS PIPELINE REHABILITATION METHOD

Wrobel et al. [3] explained that the trenchless pipeline rehabilitation is basically referring to the replacement, repair as well as installation of the underground or underwater pipeline without the need for excavation of a continuous trench from the surface to successfully install the new flexible pipeline within the original alignment. When the underwater pipeline for transporting crude oil, natural gas, or other hydrocarbon fluids degraded as a result from the gradual exposure to the corrosive environment of the subsea, that pipeline should be replaced immediately as to avoid the leakage of the fluids flowing inside the degraded pipeline and causing massive environmental hazards.

The cost for replacing the entire pipeline structure is very high [3]. Besides, it will also distort and affect the processing or production progress in the onshore processing facilities as the operation of the transport pipeline that supply the crude oil for example, have to be shut down in order to enable the replacement and installation process for the new pipeline. The advancement of the trenchless pipeline rehabilitation with the pull-in-place method nowadays has caused it to be widely practiced in oil and gas industry. By pulling the flexible composite liner pipe through the degraded steel pipeline, it was proven to be the most cost-efficient pipeline rehabilitation method as compared to the conventional method by installing sets of a new pipeline [4].

2.2 FLEXIBLE COMPOSITE IN-FIELD LINER (IFL) PIPE

A composite is a combination of two or more distinct materials, where each of the constituent materials can take advantage of individual properties to create synergy in the newly formed material [5]. The composite liner pipe have to be flexible enough because during the pulling process of liner pipe into the steel pipeline by the pulling cable, there are some bending sections in the underwater pipeline configuration and the flexible characteristic of that liner pipe will enable it to be pulled smoothly throughout the entire pipeline configuration even at the bending sections. The type of composite In-Field Liner pipe that will be used in this project is Primus Line® DN 200 PN 25. In Figure 2.1, it shows a schematic illustration of Primus Line® DN 200 PN 25, where it consisted of 3 layers of composite material with the inner layer is made of Thermoplastic Polyurethane (TPU), reinforced with the seamless, woven aramid fibers layer (Kevlar®), and coated with the outer layer of abrasion-resistance Polyethylene (PE) sheath [6].

Figure 2.1: Schematic of Primus Line® DN 200 PN 25 by layers.

2.2.1 Thermoplastic Polyurethane (TPU)

Qi and Boyce [7] explained that TPU is a thermoplastic polymer that exhibits the mechanical performance characteristics of rubber but it can be processed like a thermoplastic material. This particular characteristic of TPU causes it to have high elasticity and high abrasion resistance. With the modified TPU as the inner layer of the composite IFL pipe, it contains an oil-resistant interior coating based on TPU so that the flow and transportation process of the hydrocarbon fluids like crude oil in the IFL pipe become smooth due to its resistant to aromatic and aliphatic hydrocarbons [6]. The use of TPU for inner layer of this liner pipe also promotes less deposit on the inner wall of the composite liner as it has a smooth surface of internal coating.

2.2.2 Kevlar® Aramid Fiber

Kevlar® Aramid Fiber is one type of an organic fiber from the aromatic polyamide family but it is quite different and unique among polyamides. Strong [8] revealed that the unique properties and chemical composition of wholly aromatic polyamides (aramids) distinguish the aramid or also known as Kevlar in the fiber form from the other commercial fibers, especially when a part of the back bone stiffens and strengthens the Kevlar beyond any of the other polymeric material. In addition, Yue et al. [9] added that Kevlar is very well-known for its high-strength reinforcement fiber for plastic composites. Besides, it has a very remarkable combination of high modulus, thermal stability and toughness as compared to the other organic fibers [10]. In term of weight, Kevlar still has the highest specific tensile strength of all the fibers available in the industry. This is due to the fact that their molecular structure is developed during the production process which is based on liquid crystal technology, where the rigid molecular chains form a mesophase in solution and the spinning process aligns the molecular chains parallel to the fiber axis leading to a highly ordered structure with a high degree of crystallinity [11].

2.2.3 Polyethylene (PE)

Farshad [12] stated that polyethylene is categorized as one of the common synthetic homopolymer along with polypropylene (PP), polystyrene (PS) and polyvinyl chloride (PVC). With its high molecular weight, polyethylene exhibits the property of insensitive relatively to most solvent [8]. This property gives polyethylene an advantage when it is used in the underwater pipeline application due to its critical inertness. In addition, a modified outer layer of composite IFL pipe is made from wear and abrasion resistant polyethylene, so that it can withstand the corrosive and aggressiveness of the environmental condition in the subsea and keep the long service life of the composite liner pipe by protecting the other layers of liner as it is resistant to the wear and abrasion, even when the IFL pipe is fully exposed as the steel pipeline is totally degraded.

2.3 FINITE ELEMENT ANALYSIS (FEA)

Finite Element Analysis (FEA) has been widely practiced nowadays as it is able to solve numerous kinds of problem, from the numerical solutions to a very complex and complicated engineering problem [13]. Antal et al. [14] also added that Finite Element Analysis (FEA) is applicable in wide range of engineering principles, including hydrodynamics, mechanical, heat transfer and gas diffusion phenomena. Basically, in the Finite Element Analysis, it involves the model generation of some material or design that is stressed or analyzed for obtaining a specific results. It is commonly used to evaluate and analyzed the new product design or existing product for refinement works in order to verify that the proposed design will be able to function according to the client's specifications prior to manufacturing of the proposed design.

However, despite the countless advantages that FEA possessed, there is a quite small flaw with this method. Roylance [13] explained that FEA approach does not necessarily reveal how the stresses are influenced by the important problem variables such as materials properties and geometrical features, with an error in input data can produce inaccurate results that may be overlooked by the analyst.

2.3.1 Assumptions for Finite Element Analysis (FEA) on Liner Pipe Pull-In Process

Finite Element Analysis (FEA) has been extensively practiced to solve many complex problems because of its ability to easily analyze the problem and its flexibility when dealing with the complicated geometry and boundary conditions. There are several key issues that should be considered in the finite element analysis of the liner pipe problem. As proposed by El-Sawy and Moore [15], the liner plane strain is assumed to prevail. In the case where two dimensional analysis involves, the higher stresses in the liner for a given external fluid pressure and reduction in stability is relative to those predicted using three-dimensional analysis. In addition, Bathe [16] also suggested that when the updated Lagrangian incremental approach is used, the load is applied in an incremental fashion and the geometry of the problem is updated after the application of each load increment.

When the polymeric material is used for the liner pipe, the behavior of those polymer materials is significantly time dependent, and ultimately the finite element analysis should accurately model the rheology of the material. For design purposes, a suitable approximation is to use the long-term Young's modulus, where the value is taken from the end of the lifetime for liner pipe. Generally, a better approximation is to consider a time-dependent Young's modulus of the polymer material [17]. Another assumption that can be made when conducting the analysis of liner pipe insertion process is that the interaction between the liner pipe and the rigid cavity strongly influences the behavior of the encased liner. Therefore, a conservative assumption of a smooth interface is used [18].

2.3.2 Non-Linear Finite Element Analysis (FEA) of Liner Pull-In Process

In a non-linear finite element analysis of liner pull-in process, there are 3 key points that can be treated with this level of complexity; the pre-installation folding, the reformation process, and the bending that occurs in the pipeline with bend sections during installation. Among these 3 instances, the liner core pipe and the adhesive tape that are most susceptible to failure. The application of 2-D plane strain solid type elements captured the material stress-strain behavior in a more accurate manner than it would by using 3-D shell elements [19].

Analysis of the Folding Process

In the form of folded "U-shaped", the liner pipe has been simulated to ensure that the folding process not detrimental to the strength of the core liner pipe. Figure 2.2 illustrates the final shape of the liner in the U-forming process by using the FEA simulation.

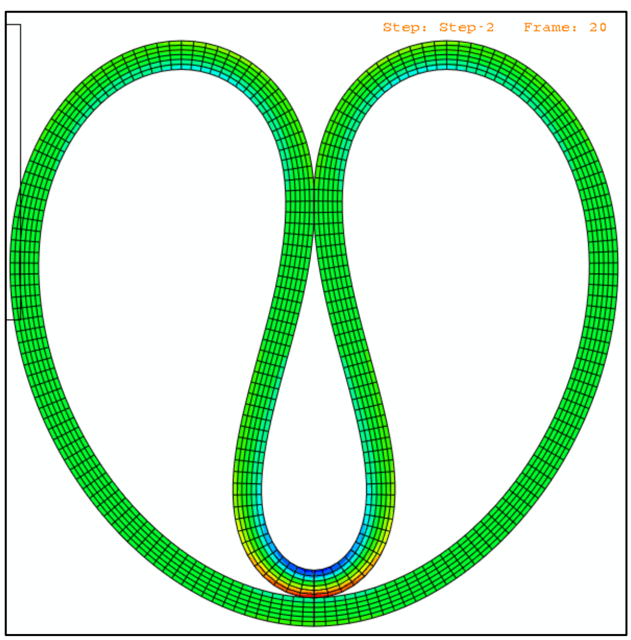

Figure 2.2: FEA simulation of the final shape in a U-forming process.

Bethel et al. [19] explained that the most heavily deformed region in the U-forming process is due to the bending at the root of the region. Even so, the maximum strain level at the large deformations on U-forming process is not large enough to cause a failure on the liner pipe.

Reformation Analysis

After the U-forming process is completed, the deformed cross section of the liner pipe is wrapped by adhesive tapes in order to prevent the elastic spring back during the insertion of the liner into the pipeline [19]. Immediately after the installation, the internal pressure is used to break the adhesive tapes to allow the liner pipe to expand to its original round shape again and fill up the internal of the pipeline.

Bending Analysis

Bethel et al. [19] added that the liner pipe model has been used to analyze the performance of the liner pipe itself when it is pulled through the pipeline during the installation in U-shape form at the bend location subsequently. A special constitutive behavior model of Marlow material is used that allow them to carry a tensile load only. During the installation, it is possible for the liner pipe to rotate as it is pulled-in and all possible orientations must be considered to ensure that the installation can be made safely even with the liner pipe in its most highly stressed condition.

CHAPTER 3

3. METHODOLOGY

3.1 PLANNED PROGRESS FLOW OF THE PROJECT

This chapter focused on the planned progress flow for this project. After the project title was selected, a preliminary research work was done including consultation with the supervisor to get the overview and basic understanding of the selected project. The details and information of the project which was given by the supervisor have been studied in order to understand the project background, problem statement, objectives and scope of study for this particular project. After fully understand the basics of the project, a more thorough research work was done for the literature review of this project. During this stage, all related materials and information including the previous researches and journals were retrieved as much as possible for a more thorough and deep understanding on the materials used for the composite IFL pipe and the mechanism of the pulling process for the composite IFL pipe into the degraded underwater steel pipeline.

Then, after all the research works have been completed, the main stage of this project was initiated. Firstly, in order to be able to design a good and suitable IFL pulling head, I have to familiarize myself with the design software first. In this case, the design software which I have used was CATIA V5R20 because of its ability to produce a three-dimensional (3D) model design easily on the basis of surface modeling and it can be exported to the simulation software like ANSYS for the simulation works due to its compatibility with the ANSYS software. After I have familiarized and mastered the CATIA software, the next step in this project was to produce a suitable design for the liner pipe pulling head. All parameters and data related to the composite IFL pipe and the steel pipeline were taken into consideration during the design stage in order to produce an acceptable and good design for the liner pipe pulling head. This has ensured that the IFL pipe pulling head design has met the requirement as in the real life design and maintained the credibility of the simulation results obtained.

After completing the design for IFL pulling head, the design for a three-dimensional (3D) model of “U-shaped” folded IFL pipe was carried out immediately by using CATIA V5R20 software. The IFL pipe was analyzed in the same manner as a three-dimensional (3D) beam element with an equivalent cross-section of the IFL pipe as to consider the complexities of the composite material and the shape of cross-section for the “U-shaped” folded IFL pipe during the replacement process by the pulling method. With the completion of the design task, then the simulation of the IFL pipe in the form of FEA models was done by using ANSYS Version 14.0 software. But still, I have to familiarize myself with the ANSYS software first before any simulation was done. After I have fully understood how the ANSYS software works, the FEA model of “U-shaped” folded IFL pipe was imported from the CATIA software to simulate the model and determined the sustainability of the composite liner pipe even when it was subjected to the maximum pulling force exerted by the pulling cable. The FEA model of IFL pipe was analyzed as it is in the form of beam element, where it was fixed on one end and the longitudinal pulling force with the magnitude of 79151.3 N was applied on the other end of IFL pipe in the simulation.

When the simulation has been completed, the results obtained were discussed and analyzed in order to verify the feasibility and the safety of the composite IFL pipe if it is happened to be used to replace the degraded pipeline in the real situation. The results obtained from the simulation have determined the effects and mechanical responses of the composite liner pipe including their ability to resist the failures when it was subjected under a very high pulling force by the pulling cable. Finally, the final report was produced and compiled based on the overall study and the results from the simulation of IFL pipe model. Figure 3.1 shows the planned progress flow for this project.

3.2 PROJECT FLOW CHART

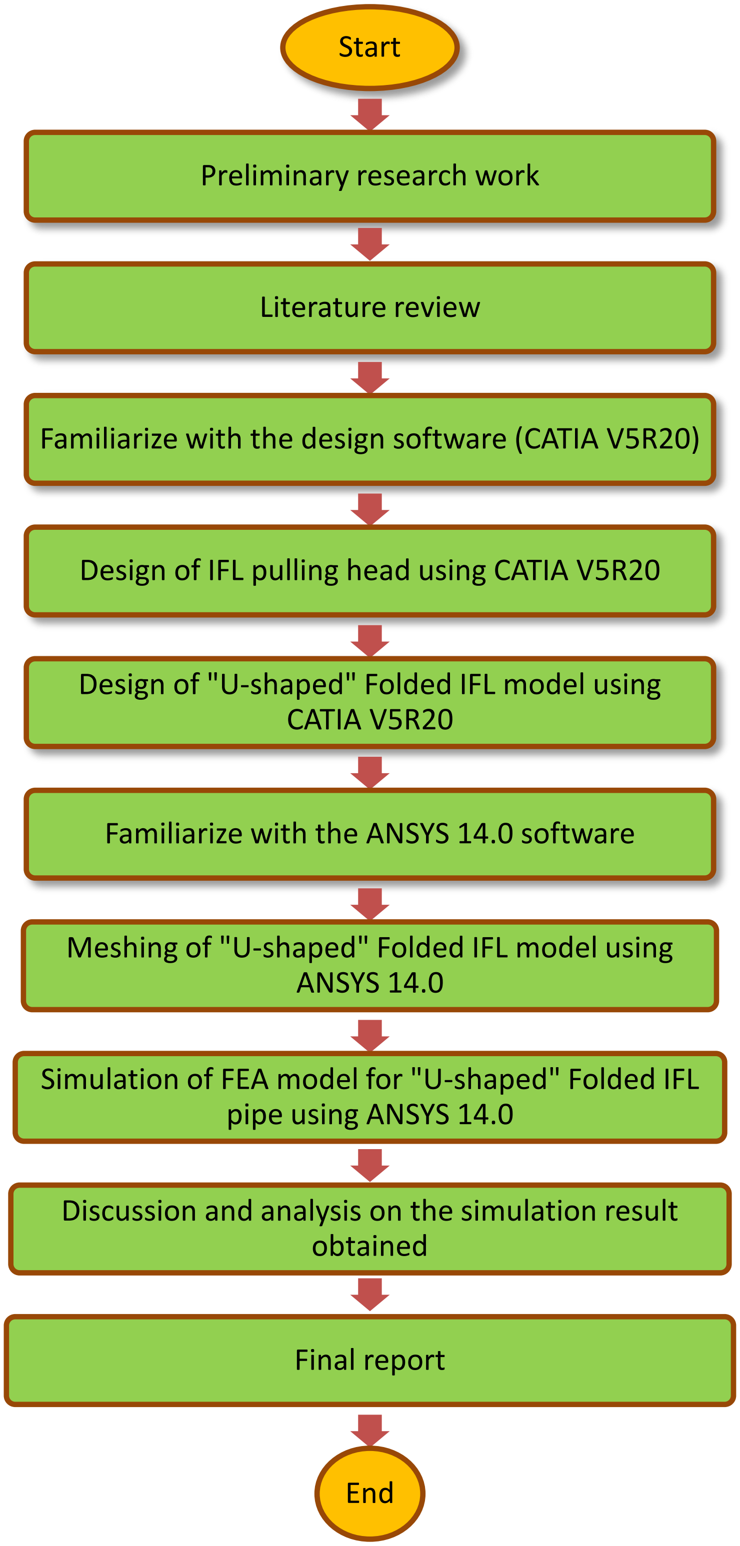

Figure 3.1: Project Flow Chart

CHAPTER 4

4. RESULTS AND DISCUSSIONS

4.1 DATA GATHERING

This project consisted of two main parts, namely the designing task and the simulation task. All parameters that were used in this project, including the design parameters and the simulation parameters were carefully chosen as to ensure that the end results were accurate and valid.

4.1.1 Design Parameters

The design parameters play an important role in determining the validity of the design produced for IFL pulling head and the folded configuration of IFL pipe. Table 4.1 shows the material properties of the existing steel pipeline which was very important in the process of designing the IFL pulling head as to ensure that the pulling head was fit nicely inside the steel pipeline, thus enable it to be pulled through the pipeline.

Table 4.1: Material properties of existing steel pipeline

ITEMS	UNITS	VALUES
Outside Diameter (OD)	m	0.2191
Inside Diameter (ID)	m	0.2
Wall Thickness	m	0.00953
Pipeline Length	m	1266
Steel Density	Kg/m^3	7800
Young's Modulus (E)	GPa	2.07
Poisson's Ratio (v)	-	0.3

The properties of IFL also were very important for the design task of the IFL pulling head and the folded configuration of the IFL pipe itself. The properties of the IFL pipe are shown in Table 4.2**.**

Table 4.2: Properties of IFL pipe

Type	Primus Line® DN200 PN25
Fluid	Crude Oil
Outside Diameter	182 mm
Inside Diameter	169 mm
Wall Thickness	6.5 mm
Internal Coating Material	TPU
Reinforced Layer Material	Kevlar®
External Coating Material	PE

4.1.2 Simulation Parameters

For simulation purposes of the IFL pipe, there were 3 different sets of data that should be taken into consideration when dealing with the simulation input parameters, which were the properties for each layer of the IFL pipe. There were also 3 different layers for the IFL pipe, in which the outer layer was made from Polyethylene, Kevlar as the reinforcement layer, and TPU as an inner layer of the IFL pipe. Among these 3 layers, the source of the strength for the IFL pipe was contributed mainly from the Kevlar layer. The inner layer from TPU concerns with the smooth flow of the crude oil as it is oil-resistant while the outer layer which is made from wear and abrasion-resistant polyethylene was used to protect the surface of the IFL pipe and prolonged its service life.

The simulation of this project was aimed to determine whether the IFL pipe has an adequate strength to sustain a pulling load of 79151.3 N during the pulling process without being damaged. Therefore, only the properties of Kevlar layer will be considered and was used as the simulation input parameters. The following Table 4.3 shows the material properties of Kevlar layer for IFL pipe.

Table 4.3: Material properties of Kevlar layer for IFL pipe.

Manufacturer	DuPont
Product Name	Kevlar 29
Young’s Modulus (E)	70.5 GPa
Poisson’s Ratio (v)	0.32
Yield Strength	1.24 GPa
Ultimate Tensile Strength	2.92 GPa
Shear Modulus (G)	1.8 GPa

4.2 DESIGN AND MODELING

For design and modeling part, there were 2 items that need to be designed and modeled, namely the IFL pulling head and the Folded IFL pipe. All designs and modeling tasks were carried out by using CATIA V5R20 software.

4.2.1 IFL Pulling Head Design

Figure 4.1 shows the illustration of the IFL pulling head design in multiple views created using CATIA V5R20 software.

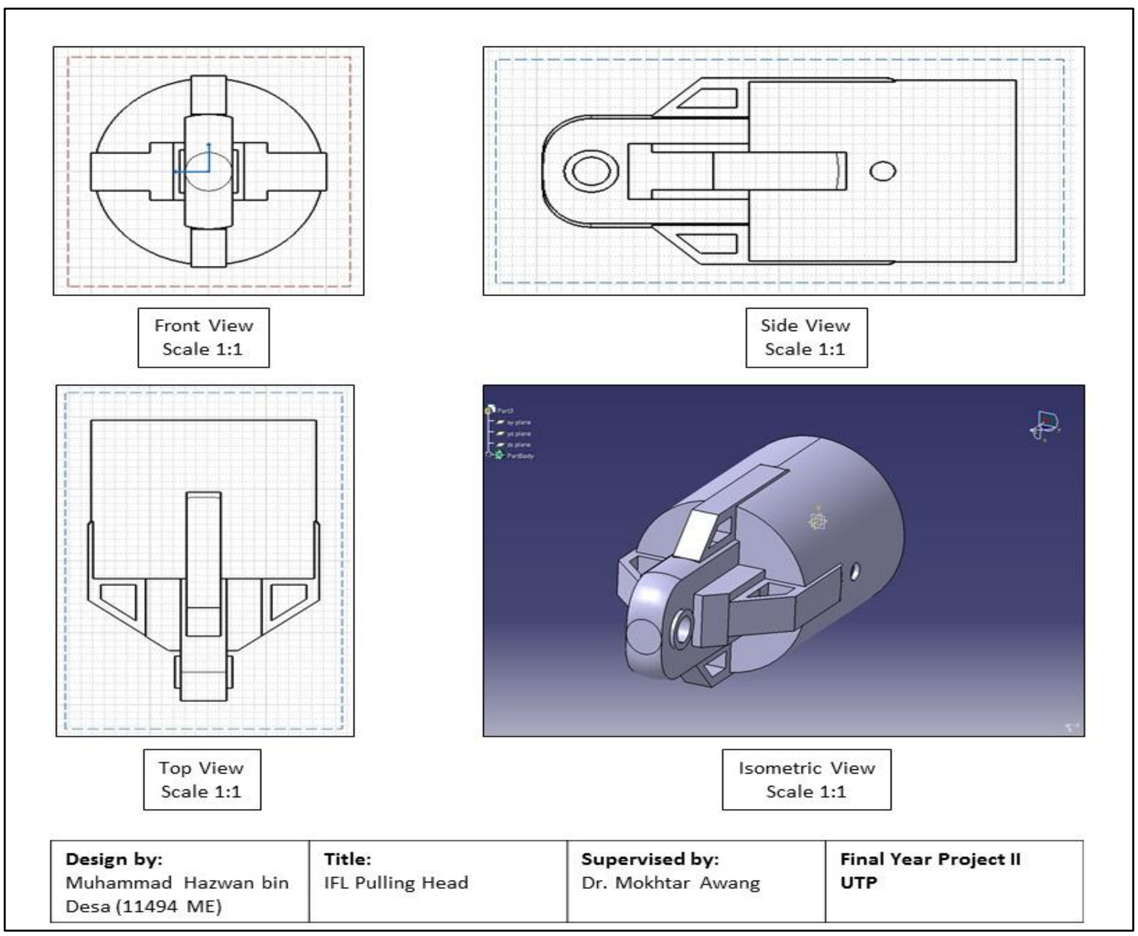

Figure 4.1: IFL Pulling Head design in multiple views.

4.2.2 Folded IFL Design

In this project, the most challenging task was to design the Folded IFL pipe model due to its complex cross-section and geometrical shape. In folded form, the outer diameter of IFL was smaller as compared to the outer diameter of IFL when it was expanded. In addition, the length of the IFL pipe circumference must be kept maintained in folded form and it should be exactly the same as in the expanded form. Figure 4.2 shows the illustration of Folded IFL design in multiple views created using CATIA V5R20 software.

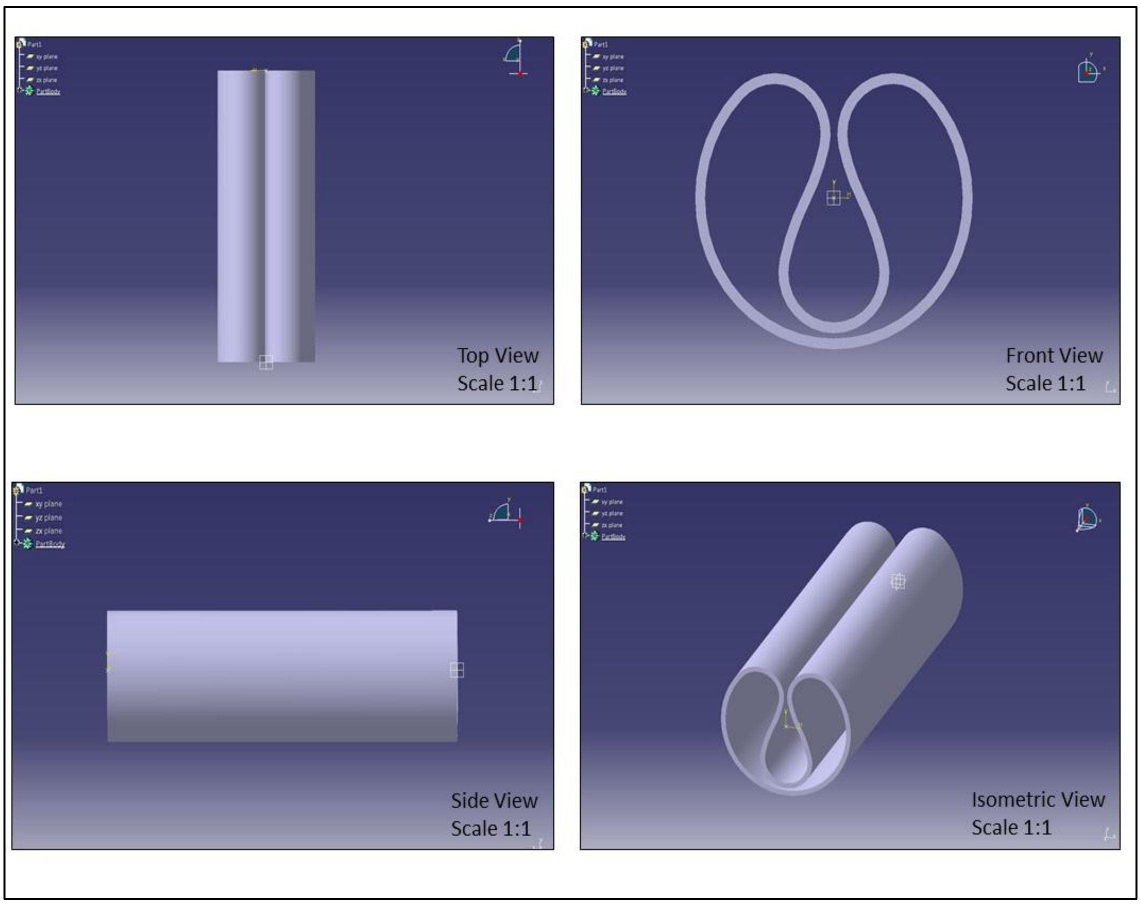

Figure 4.2: Folded IFL design in multiple views.

4.3 ANALYSIS AND SIMULATION OF IFL PULLING PROCESS

The Finite Element Analysis (FEA) of IFL pipe was done by using ANSYS v14.0 software. The analysis and simulation of the IFL pipe was done in order to determine the effects of pulling load applied on the IFL during the pulling process through the inner side of the existing steel pipeline.

4.3.1 Meshing for FEA Model of Folded IFL Pipe

In Finite Element Analysis (FEA) method, the model of folded IFL pipe was meshed first before the simulation of the pulling process for IFL pipe was done. Figure 4.3 and Figure 4.4 show the meshing for FEA model of Folded IFL pipe in isometric and front view respectively.

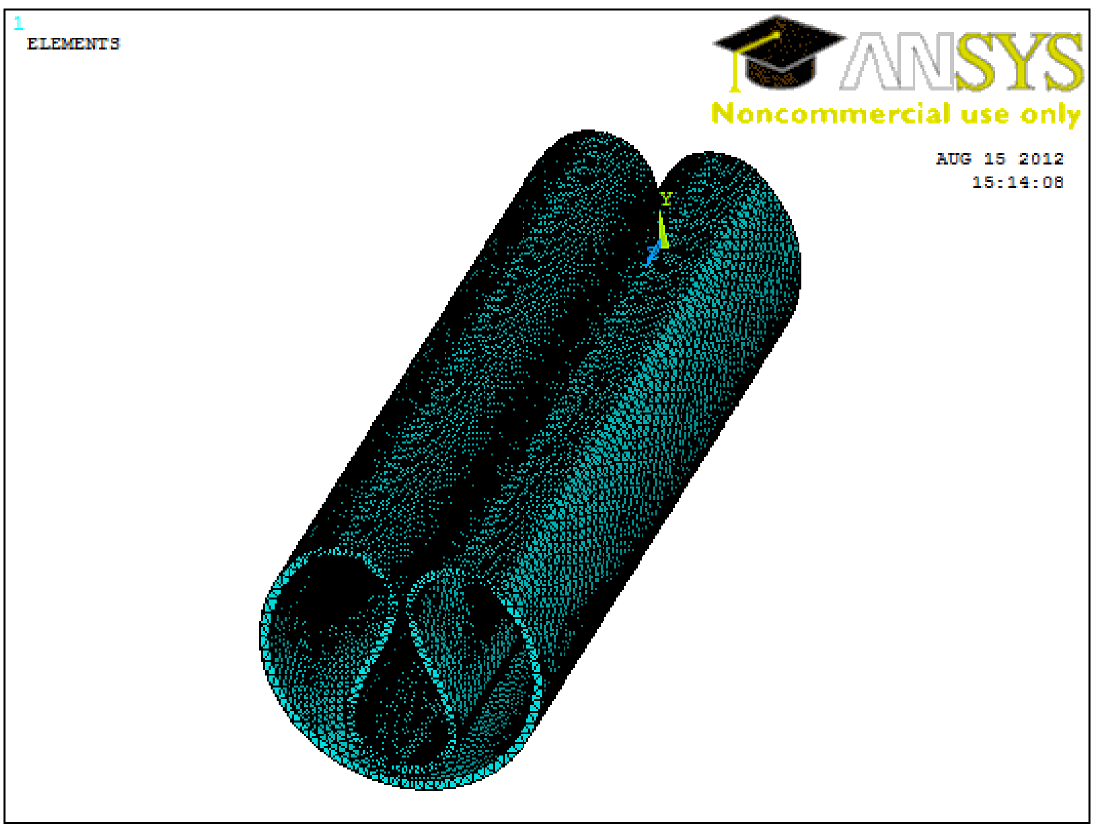

Figure 4.3: Meshing for FEA model of Folded IFL pipe in isometric view.

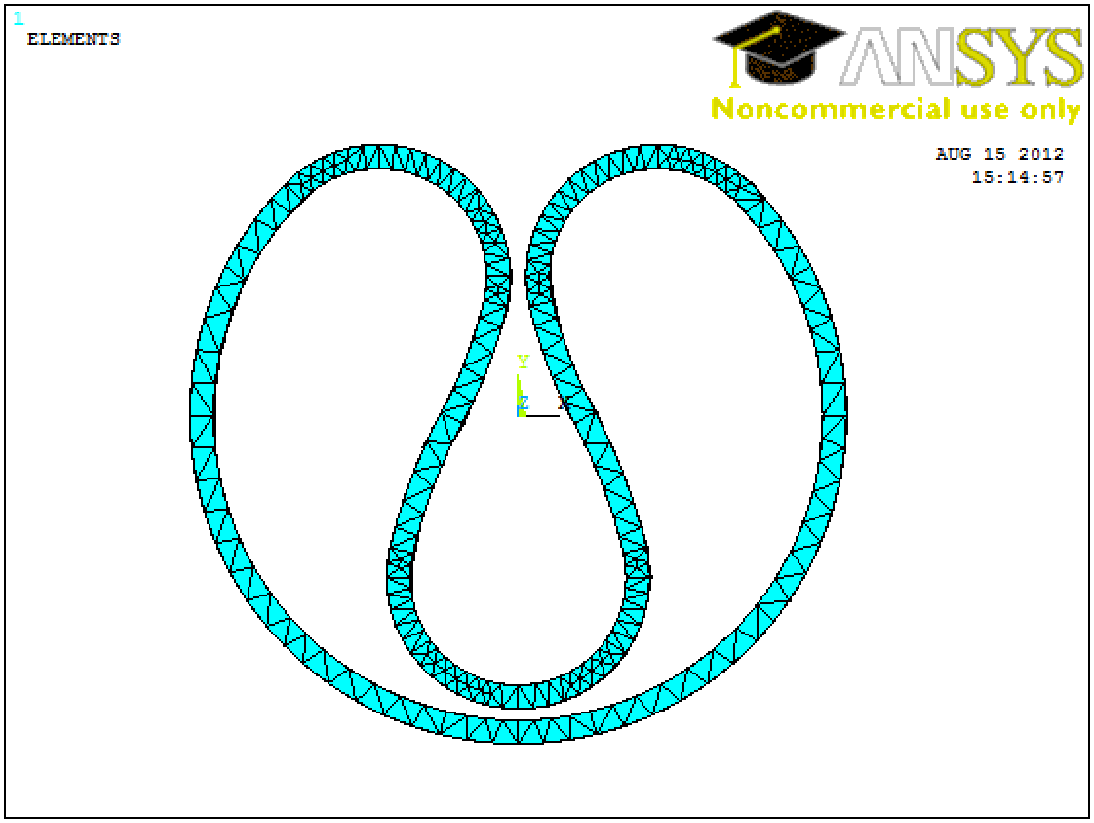

Figure 4.4: Meshing for FEA model of Folded IFL pipe in front view.

4.3.2 Simulation Results of Folded IFL Pipe

The simulation of Folded IFL pipe was done based on the static analysis method, where only a partial of the IFL pipe was used in this simulation. In this case, the IFL pipe model that was used in the simulation was about 500 mm in length. One end of the IFL pipe was fixed in all Degree of Freedom (DOF) while on the other end, the pulling force with the magnitude of 79151.3 was applied on IFL pipe. Due to the complex cross-section and geometrical shape of Folded IFL pipe, the results obtained were not really accurate where the distribution of the displacement on IFL pipe body was not as expected. Therefore, as an alternative, the simulation was done in a backward calculation manner. Instead of applying the pulling force on all 4 pinholes area for the IFL pulling head and the IFL pipe body, the displacement in the direction of the pulling force was applied. Then, once the simulation has been solved, the total reaction force acting on the body of IFL pipe was calculated from the simulation so as to ensure that the total reaction force was equivalent to the total amount of the actual pulling force required.

In this alternative simulation method, the applied displacement in the direction of the pulling force was about 0.119 mm and the resultant reaction force acting on the IFL pipe body was 79419 N. This value was almost the same as the actual pulling force required, which was 79151.3 N. Figure 4.5 shows the reaction force acting on IFL pipe as a result of the applied displacement of 0.119 mm in the direction of the pulling force.

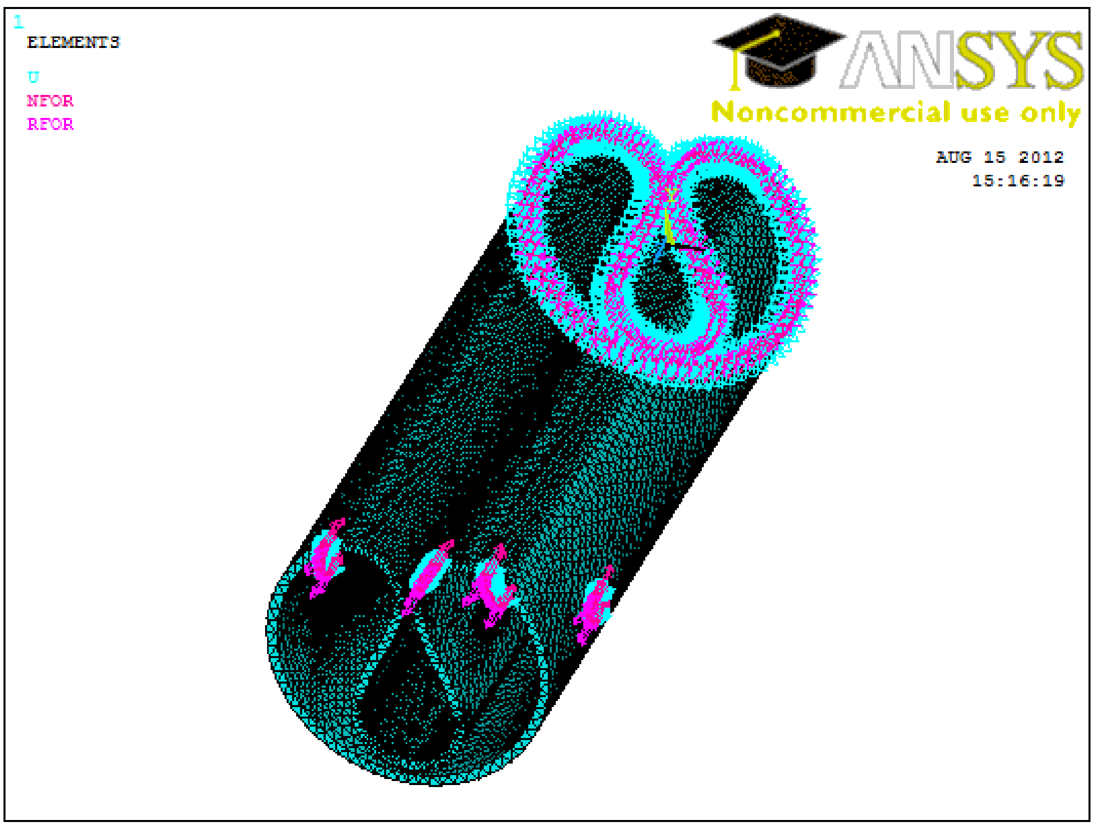

Figure 4.5: Reaction force acting on IFL pipe

The following figures show the simulation results obtained on the pulling process of the IFL pipe and they were represented in the form of Nodal Solution.

Displacement of IFL Pipe in z-Direction (Pulling Force Direction)

Based on Figure 4.6, it shows that the maximum displacement on the IFL pipe as a result of the applied pulling force on it was about 0.119 mm. The maximum displacement on the IFL pipe can be observed to concentrate mainly at the joint hole or the pinhole area for the attachment of the IFL pulling head on the IFL pipe body.

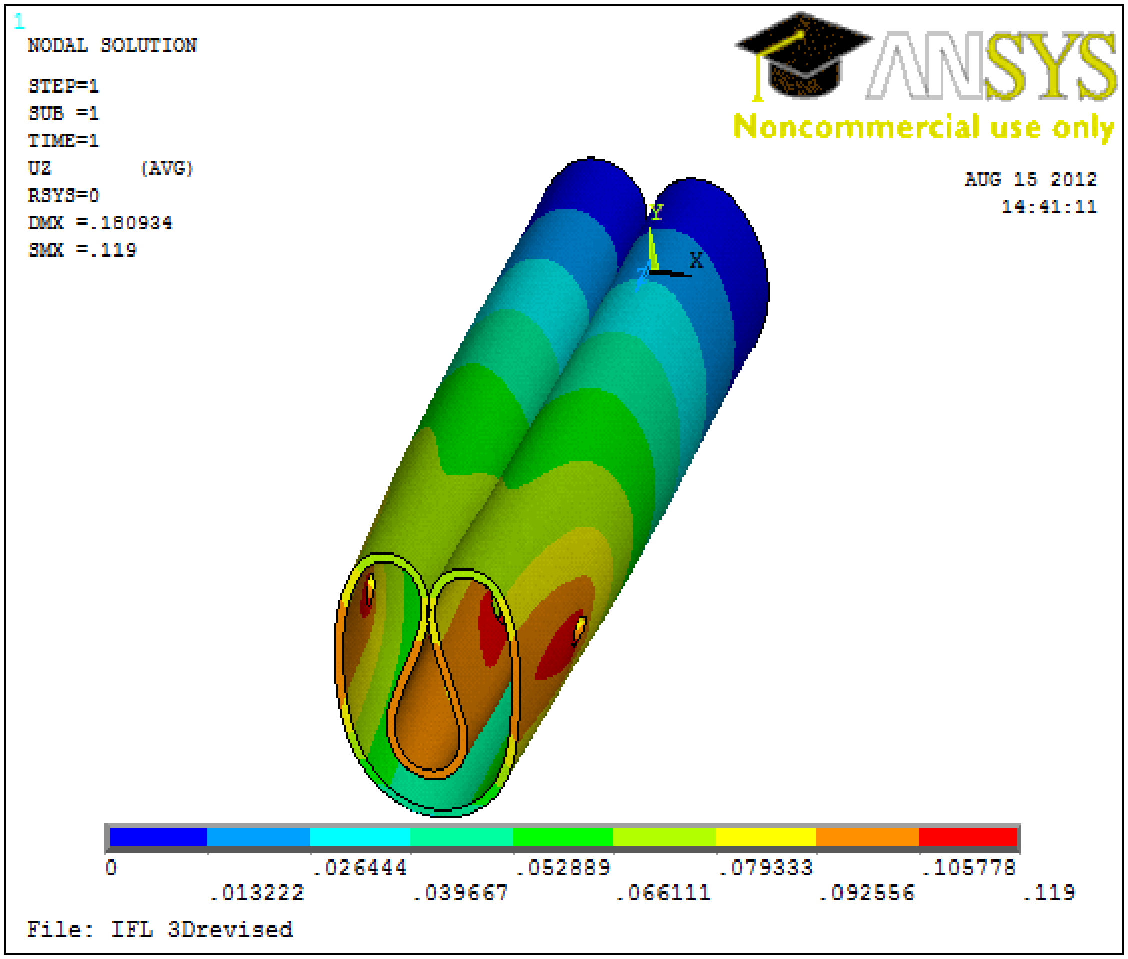

Figure 4.6: Displacement of IFL in z-direction.

Displacement of IFL Pipe in y-Direction

Figure 4.7 shows the displacement of IFL pipe in y-direction. The maximum displacement of the IFL pipe in y-direction was about 0.054 mm. It can be seen from the figure that the maximum displacement was concentrated mainly at the top of the folding area on the IFL pipe, where the pulling force was applied on that end of the IFL pipe.

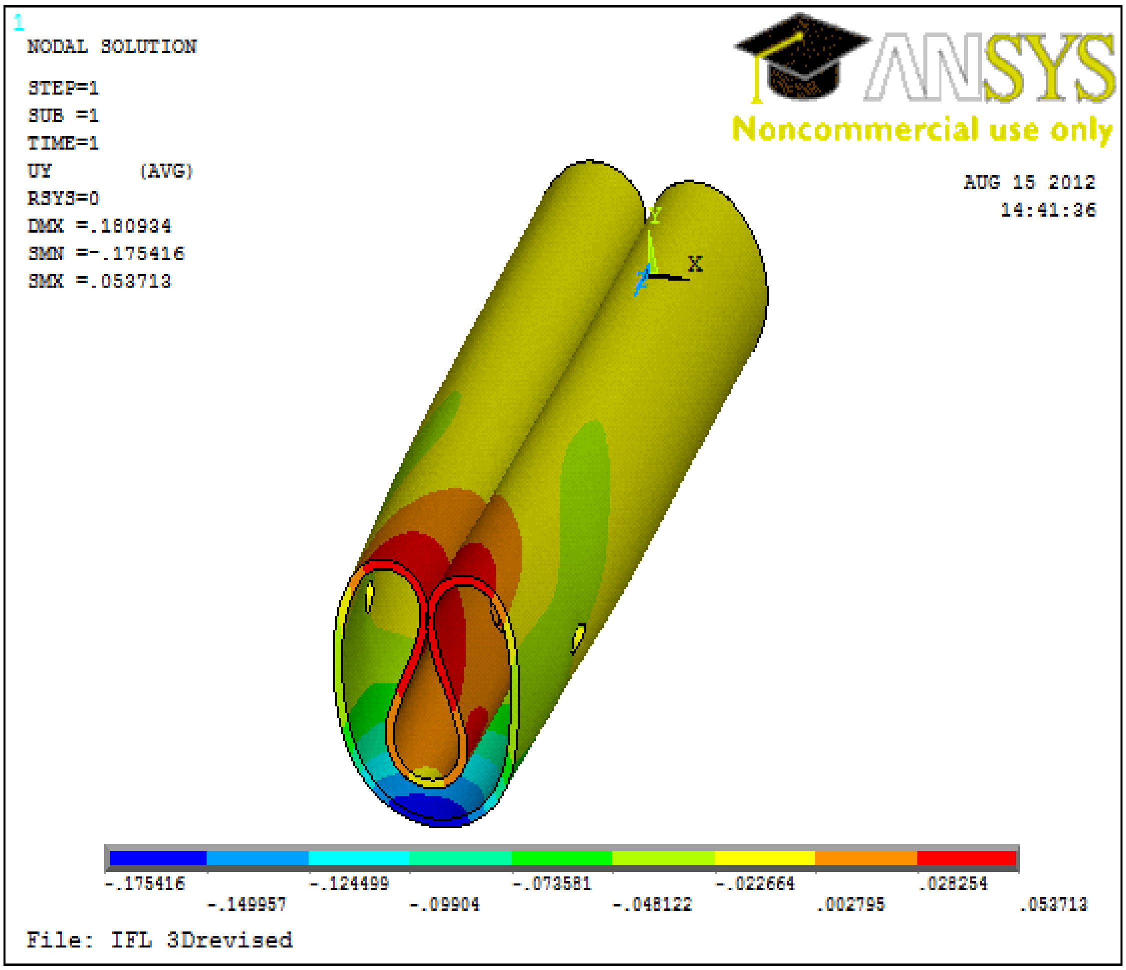

Figure 4.7: Displacement of IFL in y-direction.

Displacement of IFL Pipe in x-Direction

Figure 4.8 shows the displacement of IFL pipe in x-direction. The maximum value for the displacement of the IFL pipe was about 0.142 mm and it focused mainly at the front left side of the IFL pipe, where the pulling force was applied on that end of the IFL pipe.

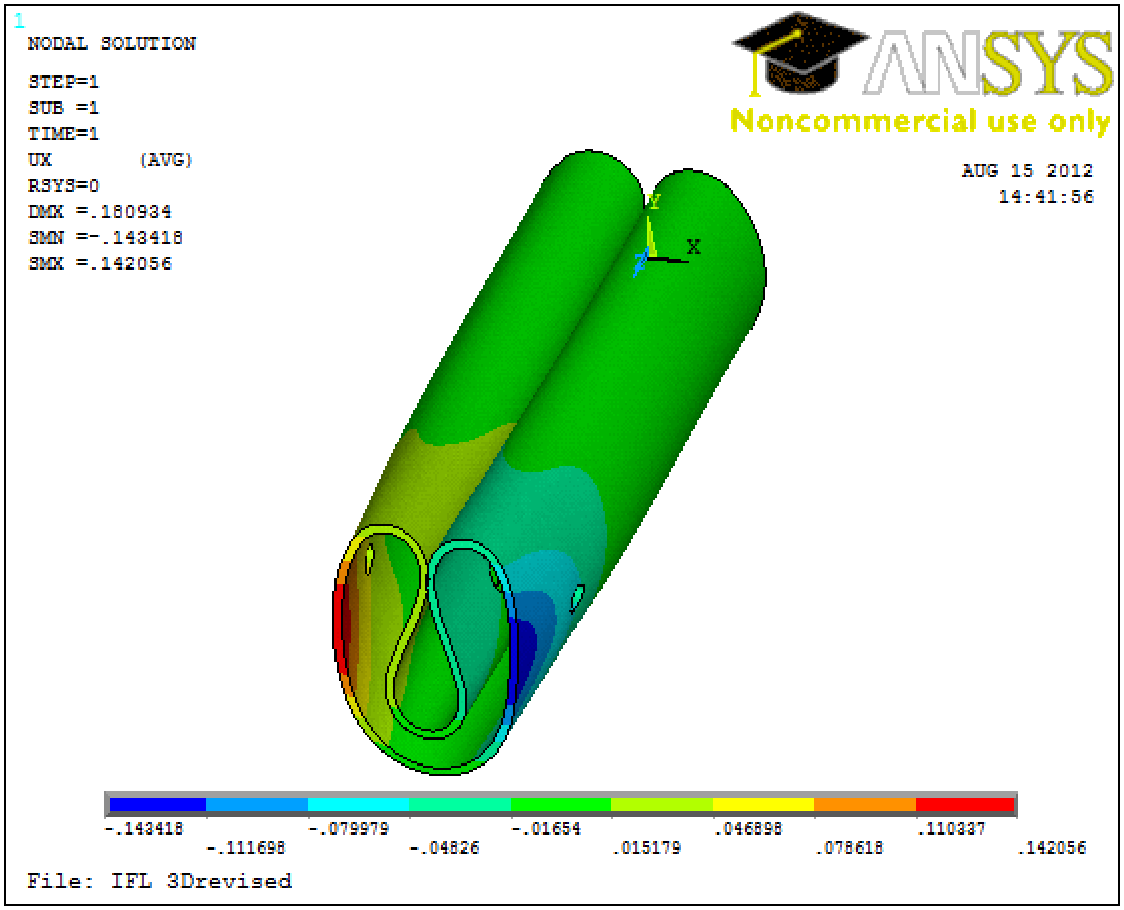

Figure 4.8: Displacement of IFL in x-direction.

IFL Pipe Displacement Vector Summation

Figure 4.9 shows the IFL pipe displacement vector summation. The maximum displacement on the IFL pipe was about 0.181 mm and it focused mainly at the front side on its end where the pulling force was applied during the pulling process simulation.

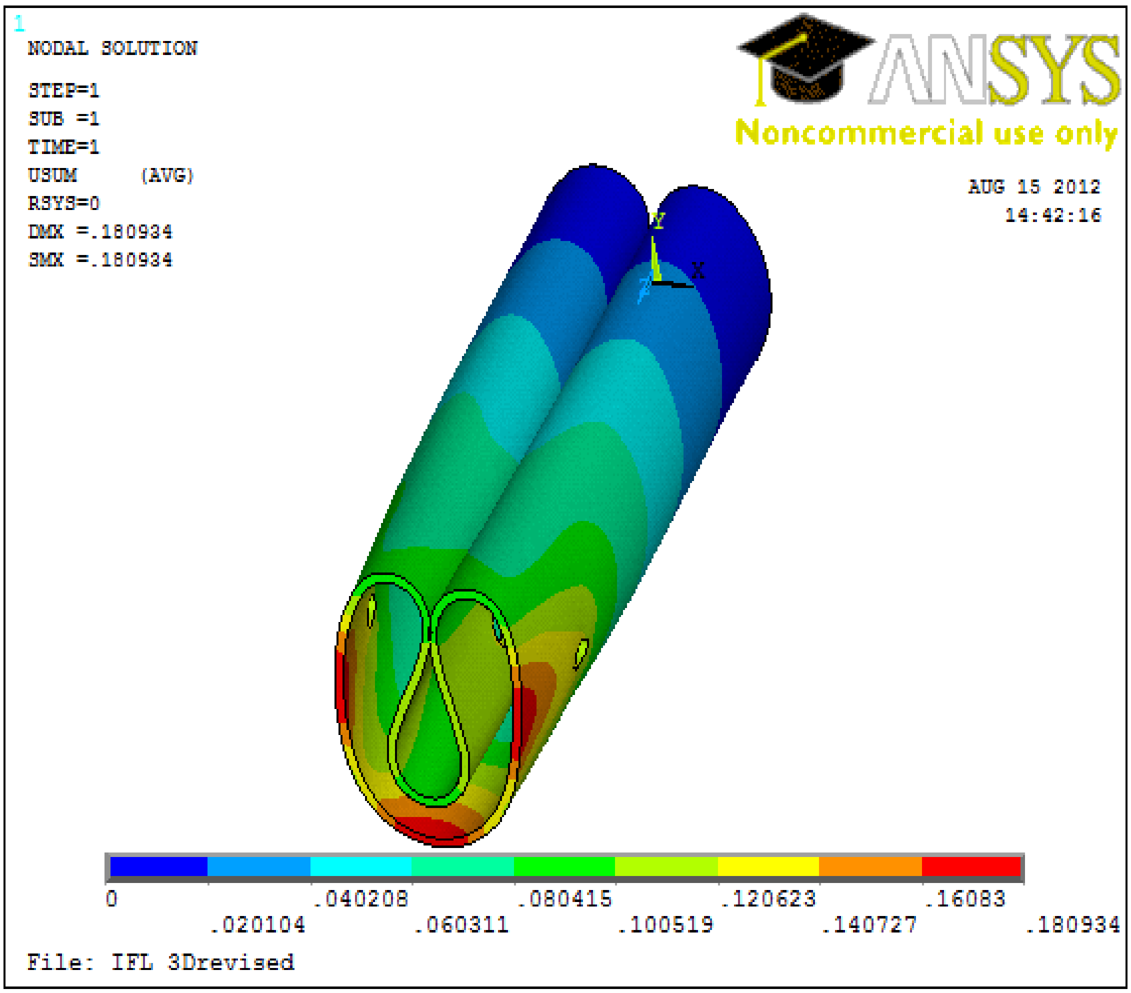

Figure 4.9: IFL displacement vector summation.

Stress on IFL Pipe in x-Direction

Figure 4.10 shows the stress on the IFL pipe in x-direction. The maximum stress value in x-direction was about 108.391 MPa.

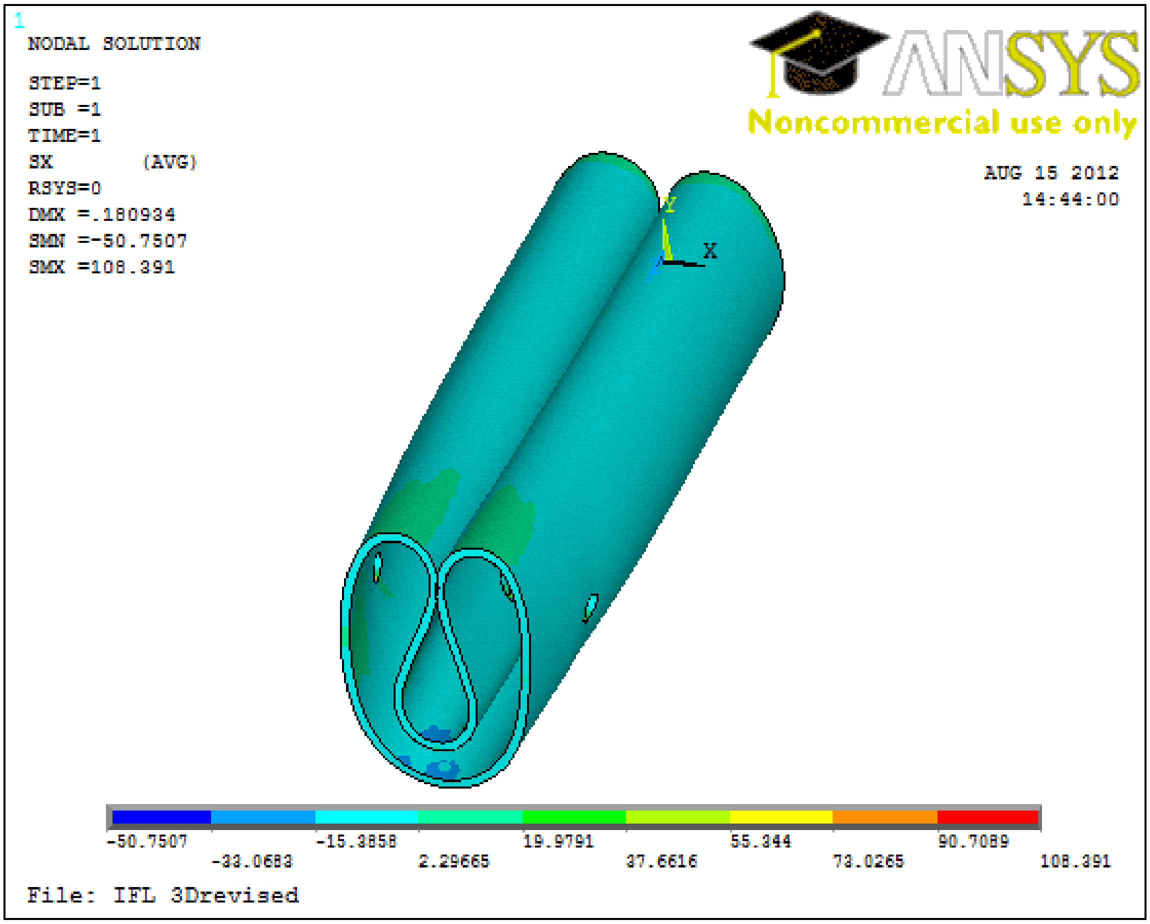

Figure 4.10: Stress on IFL in x-direction.

Stress on IFL Pipe in y-Direction

Figure 4.11 shows the stress on the IFL pipe in y-direction. The maximum stress value in y-direction was about 106.934 MPa.

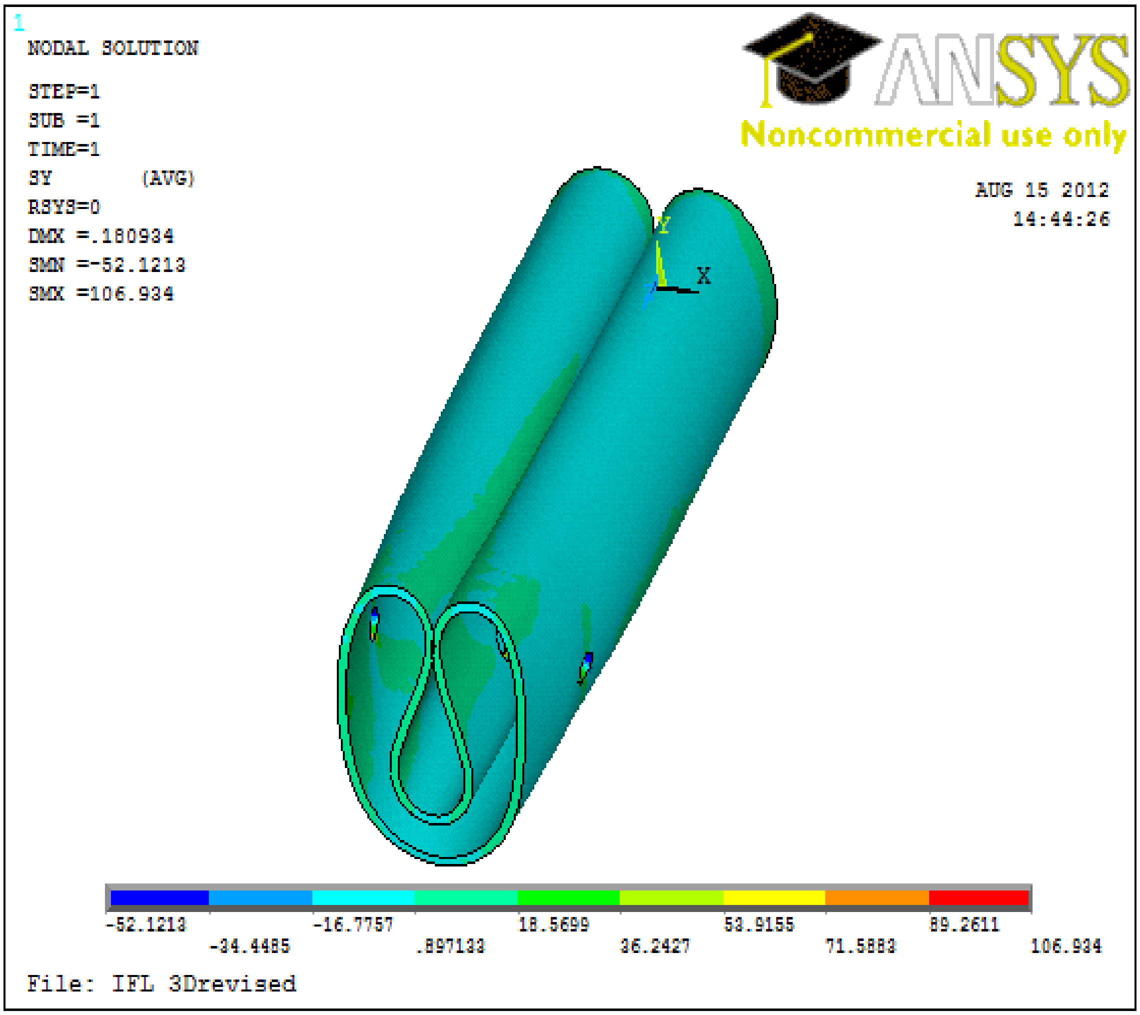

Figure 4.11: Stress on IFL in y-direction.

Stress on IFL Pipe in z-Direction

Figure 4.12 shows the stress on the IFL pipe in z-direction. The maximum stress value in z-direction was about 510.861 MPa.

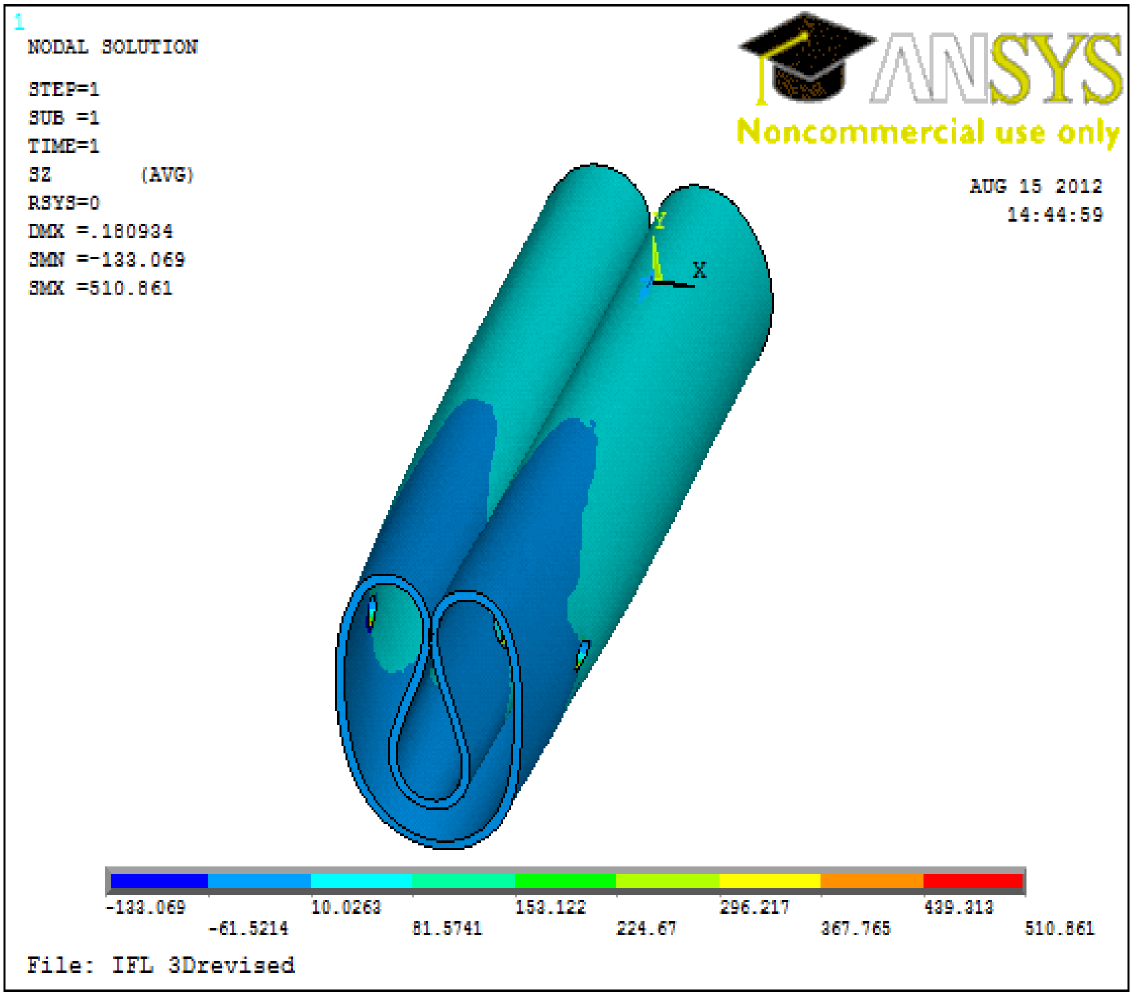

Figure 4.12: Stress on IFL in z-direction.

Shear Stress on IFL Pipe in XY-Plane

Figure 4.13 shows the shear stress on the IFL pipe in xy-plane. The maximum shear stress value in xy-plane was about 27.367 MPa.

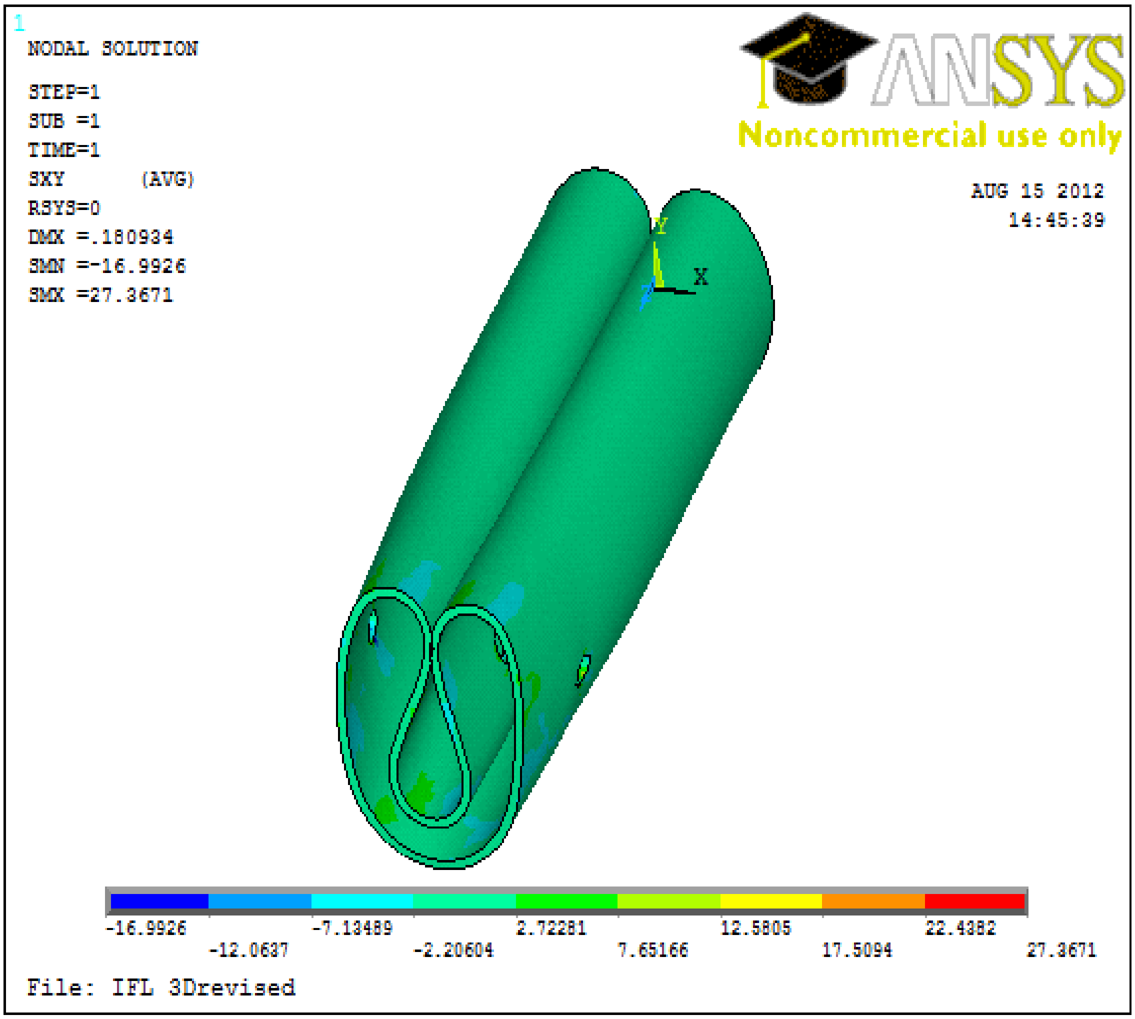

Figure 4.13: Shear Stress on IFL in xy-plane.

Shear Stress on IFL Pipe in YZ-Plane

Figure 4.14 shows the shear stress on the IFL pipe in yz-plane. The maximum shear stress value in yz-plane was about 152.346 MPa.

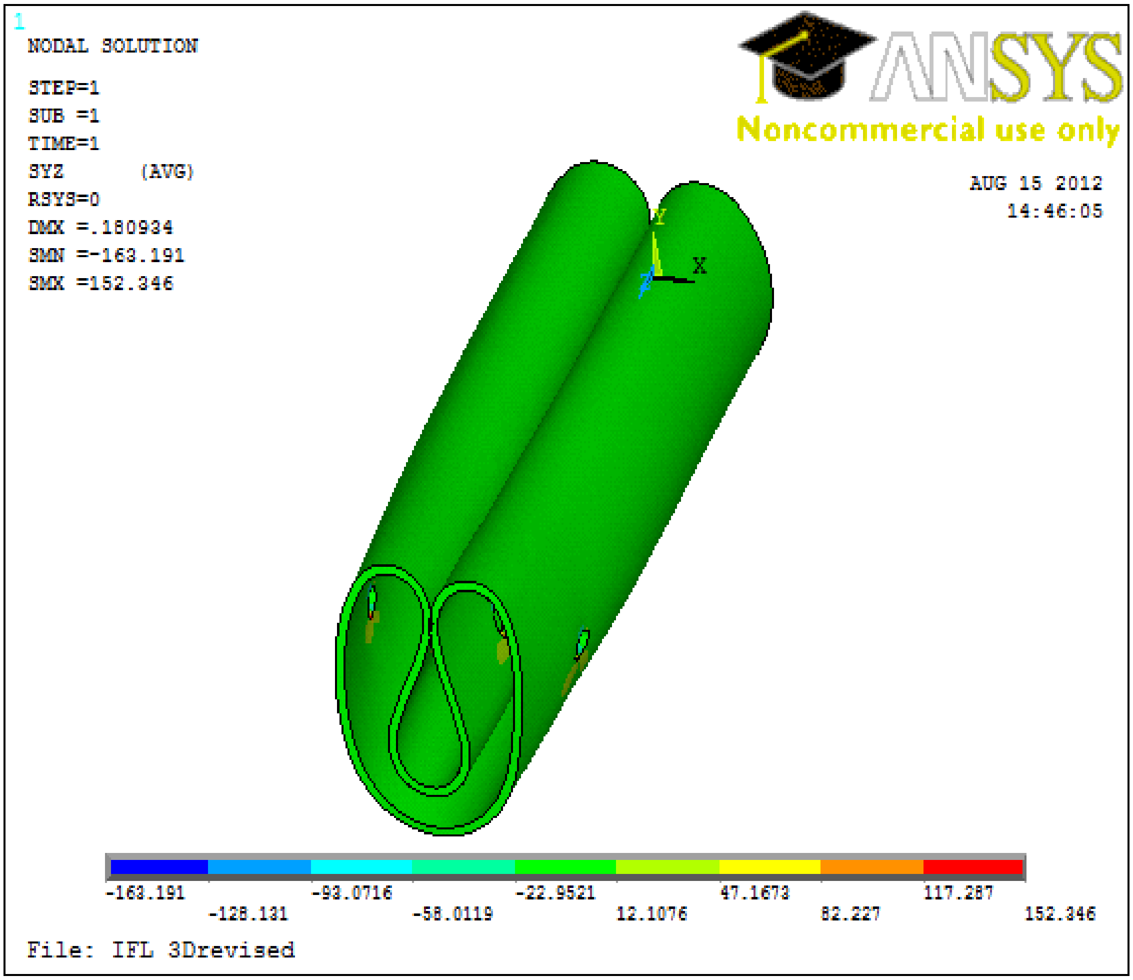

Figure 4.14: Shear Stress on IFL in yz-plane.

Shear Stress on IFL Pipe in ZX-Plane

Figure 4.15 shows the shear stress on the IFL pipe in zx-plane. The maximum shear stress value in zx-plane was about 48.523 MPa.

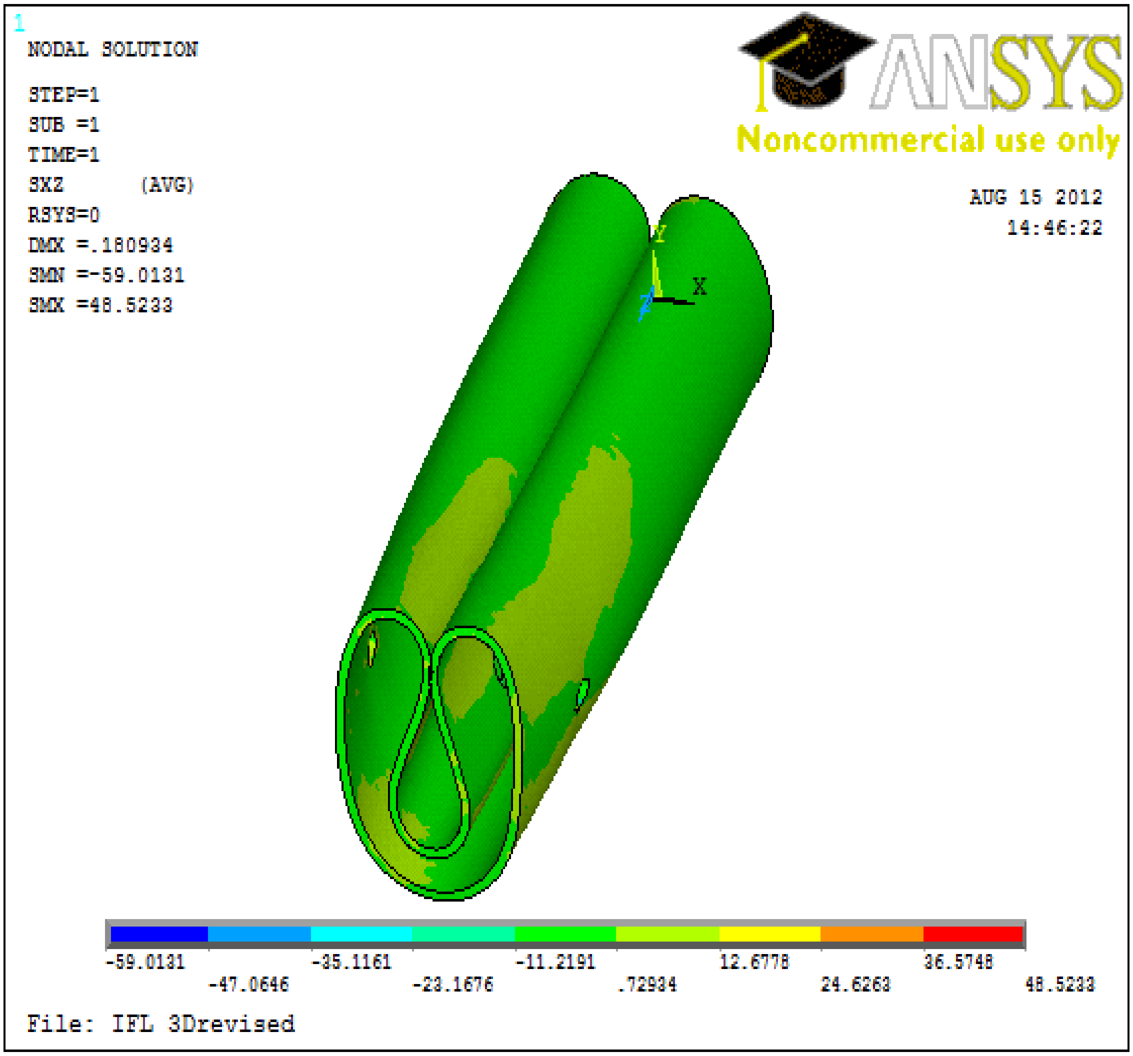

Figure 4.15: Shear Stress on IFL in zx-plane.

Von Mises Stress on IFL Pipe

Figure 4.16 shows the Von Mises stress on the IFL pipe. The maximum Von Mises stress value on the IFL pipe was about 424.461 MPa.

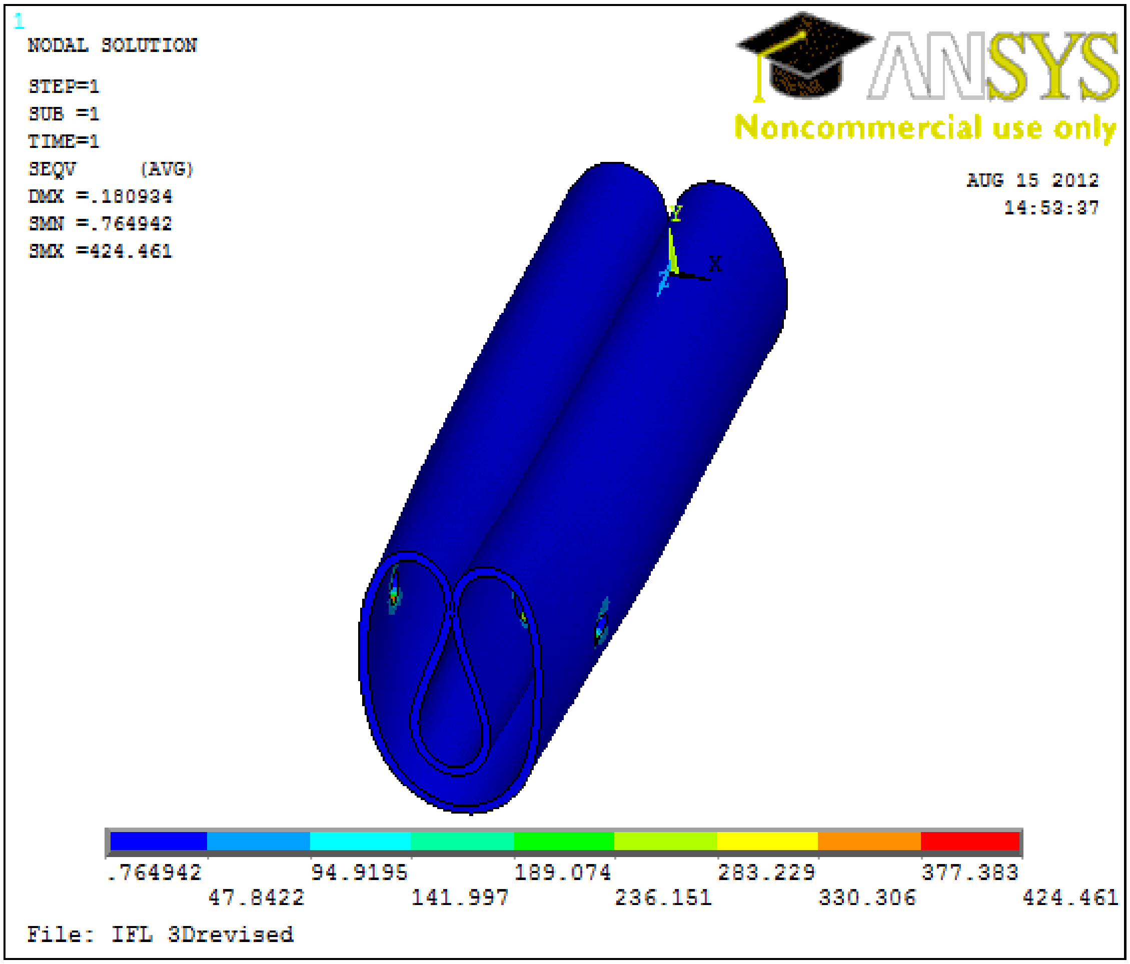

Figure 4.16: Von Mises Stress on IFL pipe.

The distributions of Von Mises stress on IFL pipe seemed to focus on the joint holes or pinholes area for the attachment of the IFL pulling head and the IFL pipe body. Figure 4.17 shows the closed-up view on the distribution of Von Mises stress on the IFL pipe. The maximum concentration of Von Mises stress on the pinhole area indicates that it could be the critical area for the initiation of the fracture or failure for the IFL pipe.

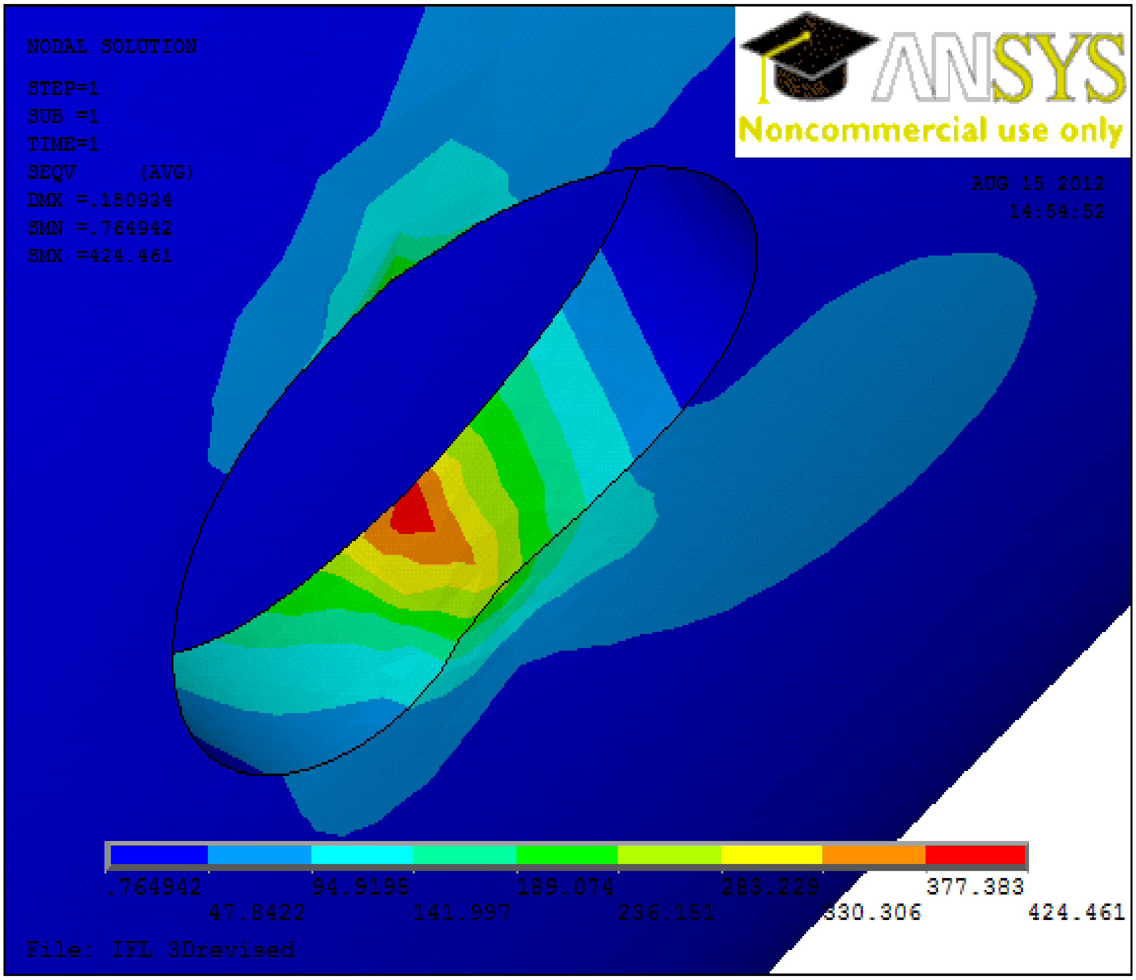

Figure 4.17: Von Mises Stress on IFL pipe (closed-up view).

4.4 SIMULATION RESULTS VERIFICATION

In order to ensure that the simulation results obtained were accurate, the results were verified. So, in this case, the verification method used was the analytical method, where the maximum value of Von Mises stress was calculated from the value of maximum stresses obtained from this simulation.

<u>Principal Stresses</u>

Cubic stress equation (3D state of stress)

$$\boxed{\begin{aligned}&\sigma^3 - \sigma^2 (\sigma_x + \sigma_y + \sigma_z) + \sigma (\sigma_x\sigma_y + \sigma_y\sigma_z + \sigma_z\sigma_x - \tau_{xy}^2 - \tau_{yz}^2 - \tau_{zx}^2) - \\ &(\sigma_x\sigma_y\sigma_z + 2\,\tau_{xy}\tau_{yz}\tau_{zx} - \sigma_x\tau_{yz}^2 - \sigma_y\tau_{zx}^2 - \sigma_z\tau_{xy}^2) = 0\end{aligned}}$$

Roots to the above equation were the principal stresses σ_1, σ_2, σ_3

<u>Von Mises Stress</u>

Octahedral shearing stress criterion

$$\boxed{\sigma_H = \frac{1}{\sqrt{2}}\sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}}$$

Maximum Stresses Values For IFL Pipe

All the maximum stresses values are shown in Table 4.4. These maximum stresses values were very important in order to calculate the maximum Von Mises stress by using the analytical method, so that the simulation results obtained was verified to be accurate.

Table 4.4: Maximum stresses values for IFL pipe

Stress, σ_x	108.391 MPa
Stress, σ_y	106.934 MPa
Stress, σ_z	510.861 MPa
Shear Stress, τ_{xy}	27.3671 MPa
Shear Stress, τ_{yz}	152.346 MPa
Shear Stress, τ_{zx}	48.5233 MPa

Principal Stresses:

$\sigma^3 - \sigma^2\,(108.391 + 106.934 + 510.861) + \sigma\,[(108.391)(106.934) + (106.934)(510.861) + (510.861)(108.391) - 27.3671^2 - 152.346^2 - 48.5233^2] - [(108.391)(106.934)(510.861) + 2(27.3671)(152.346)(48.5233) - (108.391)(152.346)^2 - (106.934)(48.5233)^2 - (510.861)(27.3671)^2] = 0$

Roots to the above equation were the principal stresses $\sigma_1, \sigma_2, \sigma_3$

σ_1 = 568.385 MPa

σ_2 = 104.158 MPa

σ_3 = 53.6433 MPa

Von Mises Stress:

$$\sigma_H = \frac{1}{\sqrt{2}}\sqrt{(568.385 - 104.158)^2 + (104.158 - 53.6433)^2 + (53.6433 - 568.385)^2}$$

$= 491.44$ MPa.

The value obtained for Von Mises stress by using the analytical method was 491.44 MPa as compared to the Von Mises stress value obtained from the simulation result, which was 424.461 MPa. The difference between these two values is small and therefore the results obtained from the simulation have been verified to be accurate.

4.5 DISCUSSIONS

Based on the results obtained from the simulation of IFL pipe, it can be seen that the displacement of the IFL pipe in the direction of the applied pulling force (79151.3 N) was about 0.119 mm. The displacement was maximum at the point where the pulling force is applied, namely at the pinholes where the pulling head was attached to the IFL pipe. This value of displacement was still acceptable without any major effect on the IFL pipe. This is due to the fact that the maximum value of Von Mises stress developed on the critical area of the IFL pipe was about 424.461 MPa, which was less than the yield strength of Kevlar material which was 1240 MPa. Therefore, under the influence of the pulling force with the magnitude of 79151.3 N, the IFL pipe was only deformed elastically due to the lower value of Von Mises stress as compared to the Yield Strength of the Kevlar material. In addition, based on the simulation results obtained, the IFL pipe will not be fractured or failed during the pulling process as the value of Ultimate Tensile Strength for Kevlar material, which was about 2.92 GPa, was way too high as compared to the value of Von Mises stress developed on the critical area of the IFL pipe.

The Ultimate Tensile Strength determined the fracture limit of that IFL pipe, where it can be fractured if it was exceeded and vice versa. The Von Mises Stress was observed to develop mainly in the area perpendicular to the direction where the pulling force was applied. In other word, the critical area which could potentially be the initial point of the failure for IFL pipe was perpendicular to the direction of the pulling force. The IFL pipe was considered to be in the safe level during the pulling process to pull the IFL pipe through the inner side of the degraded steel pipeline.

CHAPTER 5

5. CONCLUSION & RECOMMENDATIONS

5.1 CONCLUSION

As a conclusion, this project is a comprehensive study and research upon the utilization and the usage of proper tools and software for designing the In-Field Liner (IFL) pulling head model and Folded IFL model, followed by the simulation of FEA model for IFL pipe during the pulling process. For designing tasks of the IFL pulling head and folded IFL model, these tasks were carried out successfully with the full utilization of the design software, where CATIA software was used in this case. The simulation on the FEA model of IFL pipe revealed the reactions and mechanical responses on the physical aspect of the composite IFL pipe material including their capability to withstand the pulling force exerted by the pulling cable. This can be visualized from the simulation results obtained and the simulation results verification by using the analytical method to calculate the maximum Von Mises stress on the critical area of IFL pipe. Under the influence of pulling force with the magnitude of 79151.3 N, the IFL pipe only undergone a displacement of 0.119 mm in the same direction as the applied pulling force. The IFL pipe was said to be deformed elastically as the maximum value of Von Mises stress developed on IFL pipe was less than the Yield Strength of the Kevlar material. The critical area on IFL pipe has been identified to be developed perpendicularly to the direction of the pulling force. The critical area can be the potential initial point of failure for the IFL pipe during the pulling process. The result of this study will correlate on the feasibility and the safety of the composite IFL pipe if it is happened to replace the degraded steel pipeline with this composite IFL pipe in the real life situation. Overall, the objectives of this project have been successfully achieved within the time frame given.

5.2 RECOMMENDATIONS

For the suggested future works, another simulation on the IFL pipe can be done in order to validate further the simulation results obtained from this simulation. For the validation purposes, the suggested method is to use a contact element in the form of a rigid body and it will be placed in the pinhole of the IFL pipe. The pulling force will be applied on both ends of that rigid body contact element and the results obtained will be compared.

REFERENCES

[1] Liou, J. (1997). *Pipeline Research Needs: Real-Time Pipe Integrity Monitoring Research Needs,* ASCE, [p.41-43].

[2] Hall, S. J., Hill, D. J. and Dang, P. (2009). *Plastic Liner for Hydrocarbon Transport: A Qualified and Cost Efficient Alternative to CRAs,* OTC 19937.

[3] Wrobel, G., Pusz, A., Szymiczek, M. and Michalik, K. (2009). *Journal of Achievements in Material and Manufacturing Engineering: Swagelining as a Method of Trenchless Pipeline Rehabilitation,* vol 33:1, [p.27-34].

[4] Li, Z., Wang, L., Guo, Z., Shu, H. (2011). *Thin Wall Structures 51: Elastic Buckling of Cylindrical Pipe Linings with Variable Thickness Encased in Rigid Host Pipes*, [p.10-19], Elsevier, (retrieved from http://www.elsevier.com/ on 22/2/2012).

[5] Budinski, K. G. and Budinski, M. K. (2010). *Engineering Materials: Properties and Selection,* Ninth Edition, Pearson.

[6] Flexible High-Pressure Pipelines for Trenchless Pipe Renewal, Primusline Brochure, (retrieve from http://www.primusline.com/download/ on 23/2/2012).

[7] Qi, H. J. and Boyce, M. C. (2004). *Stress-Strain Behavior of Thermoplastic Polyurethane,* MIT.

[8] Strong, A. B. (2006). *Plastics: Materials and Processing,* Third Edition, Pearson Prentice Hall, [p.225], [p.269].

[9] Yue, C. Y., Sui, G. X. and Looi, H. C. (2000). *Composites Science and Technology 60: Effect of Heat Treatment on Mechanical Properties of Kevlar-29 Fibre,* Elsevier, [p.421-427], (retrieved from http://www.elsevier.com/ on 22/2/2012).

[10] Kevlar Aramid Fiber, Technical Guide (2011). (retrieved from http://www.dupont.com/ on 23/2/2012).

[11] Bencomo-Cisneros, J. A., Tajeda-Ochoa, A., Garcia-Estrada, J. A., Herrera-Ramirez, C. A., Hurtado-Macias, A., Martinez-Sanchez, R. and Herrera-Ramirez, J.M. (2011). *Journal of Alloys and Compounds: Characterization of*

Kevlar-29 Fibers by Tensile Tests and Nanoindentation, Elsevier, (retrieved from http://www.elsevier.com/ on 22/2/2012).

[12] Farshad, M. (2006). *Plastic Pipe System: Failure Investigation and Diagnosis,* Elsevier.

[13] Roylance, D. (2001). *Finite Element Analysis: An Introduction*, Massachusetts Institute of Technology (MIT), Cambridge, MA.

[14] Antal, S., Nagy, T. and Boros, A. (2003). *Improvement of Bonded Flexible Pipes According to API Standard 17K*, OTC 15167.

[15] El-Sawy, K. and Moore, I. D. (1998). *Journal of Structural Engineering: Stability of Loosely Fitted Liners Used to Rehabilitate Rigid Pipes.*

[16] Bathe, K. J. (1982). *Finite Element Procedures in Engineering Analysis.* Prentice-Hall, Englewood Cliffs, N.J.

[17] Boot, J. C. and Welch, A. J. (1996). *Creep Buckling of Thin-Walled Polymeric Pipe Linings Subject to External Groundwater Pressure.* Thin-Walled Structure, 24, 191-210.

[18] El-Sawy, K. and Moore, I. D. (1996). *A Two-Level Iterative FEM Technique for Rigorous Solution of Non-Linear Interaction Problems Under Large Deformations.* Components and Structures, 61(1), 43-54.

[19] Bethel, K., Catha, S. C., Ekelund, A., Kanninen, M. F. and Stonesifer, R. B. (2008). *Smart Pipe: An Innovative Trenchless Technology For Rehabilitating High Pressure Gas or Liquid Pipelines.* The Journal of Pipeline Engineering, Houston, TX, USA.

APPENDICES

APPENDIX I: Project Gantt Chart for FYP I

No	Detail/Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1.	Selection of Project Title: Design and Analysis of IFL and IFL Pulling Head for Underwater Pipeline														
2.	Preliminary Research Work: Understanding the basics information of the project (Project background, problem statement, objectives, etc.)														
3.	A Thorough Research Work: Study and research on the literatures of the project.														
4.	Familiarizing with the design software (CATIA V5R20).														
5.	Design of IFL pulling head model using CATIA V5R20 software.														
6.	Design of "U-shaped" Folded IFL model using CATIA V5R20 software.														
7.	Familiarizing with FEA software (ANSYS v14.0).														
8.	Meshing of IFL pulling head model using ANSYS v14.0 software.														
9.	Meshing of U-Folded IFL model using ANSYS v14.0 software.														
10.	Submission of Interim report														

APPENDIX II: Project Key Milestone Chart for FYP I

No	Detail/Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1.	Completion of IFL pulling head design.								●						
2.	Completion of FEA model of U-Folded IFL design.										●				
3.	Completion of meshing for IFL pulling head model.												●		
4.	Completion of meshing for U-Folded IFL model													●	
5.	Completion and submission of Interim Report														●

APPENDIX III: Project Gantt Chart for FYP II

No	Detail/Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1.	Simulation of FEA model for “U-shaped” Folded IFL during the pulling process using ANSYS v14.0 software.														
2.	Discussion and analysis on the simulation results obtained.														
3..	Produce a final report.														

APPENDIX IV: Project Key Milestone Chart For FYP II

No	Detail/Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1.	Completion of simulation on the pulling process for folded IFL pipe										●				
2.	Completion of discussion and analysis on the simulation results obtained.												●		
3.	Completion and submission of final report														●