Numerical Stress Analysis of Selected Materials in Solid Expandable Tubular Repair for Casing Damaged Well

by

Mohammad Zakuan bin Ahmad

Dissertation submitted in partial fulfilment of

the requirement for the

Bachelor of Engineering (Hons)

(Mechanical Engineering)

SEPTEMBER 2011

Universiti Teknologi PETRONAS

Bandar Seri Iskandar

31750 Tronoh

Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Numerical Stress Analysis of Selected Materials in Solid Expandable Tubular Repair for Casing Damaged Well

by

Mohammad Zakuan bin Ahmad

A project dissertation submitted to the

Mechanical Engineering Programme

Universiti Teknologi PETRONAS

in partial fulfilment of the requirement for the

Bachelor of Engineering (Hons)

(Mechanical Engineering)

Approved by,

....

(Dr Mohamad Zaki bin Abdullah)

Project Supervisor

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

September 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 ant that the original work contained herein have not been undertaken or done by unspecified sources or persons.

....

(Mohammad Zakuan bin Ahmad)

ABSTRACT

Solid expandable tubular is the technology of casing design which enables operator to reach the total depth required with a larger hole while starting with smaller hole compared to a conventional casing approach. The practice of solid expandable tubular in repairing casing damaged well will be described in this project. The demand of SET technology is huge despite of it is lacking of theoretical basis. The purpose of this project is to model solid expandable tubular and analyze the stress distribution for linear and non-linear behaviour using finite element method. This work produces axisymmetric modelling and analysis of the tubing which is developed using finite element software ANSYS to determine the displacement and stress for three materials which are aluminium, stainless steel, and titanium. These three materials are selected due to their significant differences in mechanical properties. Successful implementation of finite element analysis will allow the stress analysis to be conducted confidently without being too dependent on experimental work which is time and cost consuming.

The finite element analysis is preceded by modelling the geometry of the solid tubing, applying material's properties and appropriate boundary conditions. This project focuses on the use of ANYS software and understanding of linear and non-linear behaviour of metal to produce the required results. Axisymmetric analysis is chosen because the tubing is having axisymmetrical geometry. The analysis can reduce the computation time since the nodes and elements to be analyzed are lesser. The results obtained from the simulation are then compared and validated through theoretical calculation using Lame's theory on thick-wall cylinder for linear analysis while the non-linear analysis is based on the simplification of the stress-strain curve of each materials selected. The theory and simulations done justify the behaviour of the tubing where the diameter of the tubing increases while the thickness of the tubing decreases after expansion process. The stress distributions were proved to be different for linear and non-linear analysis.

ACKNOWLEDGEMENT

First and foremost, I would like to acknowledge my own Final Year Project Supervisor, Dr Mohamad Zaki bin Abdullah who had greatly supported this project. My highest gratitude for his every word of advices and knowledge which had helped completing my project.

I would also want to thank Dr Sonny Irawan, Dr Mokhtar Awang and Dr Hassan Fawad for their indispensable knowledge in helping me finishing my simulation successfully. Not to forget Mr Irfan for spending a huge amount of time assisting in my project.

Lastly, I will never forget the support from my very own family who have forever giving me strength to push forward and finish this project. Thank you to all my friends and Universiti Teknologi PETRONAS's staff for the assistance and facilities which had superbly easing my project.

TABLE OF CONTENT

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1

INTRODUCTION

1.1 Background Study

The Solid Expandable Tubular (SET) technology is one of major technologies in well construction and repairing in recent years. SET is an expandable metallic tube made of special material such as alloy steel with good ductility. The technology of casing design enables operator to reach the total depth with a larger hole than a conventional casing while beginning the well with a smaller hole. The SET technology uses a cold-working process to permanently deform the expandable tube into plastic region staying below the ultimate yield strength. The internal and external diameters of the tube will be expanded by specially designed mandrels or expansion cones. An example of tubular expansion process is shown in Figure 1.1.

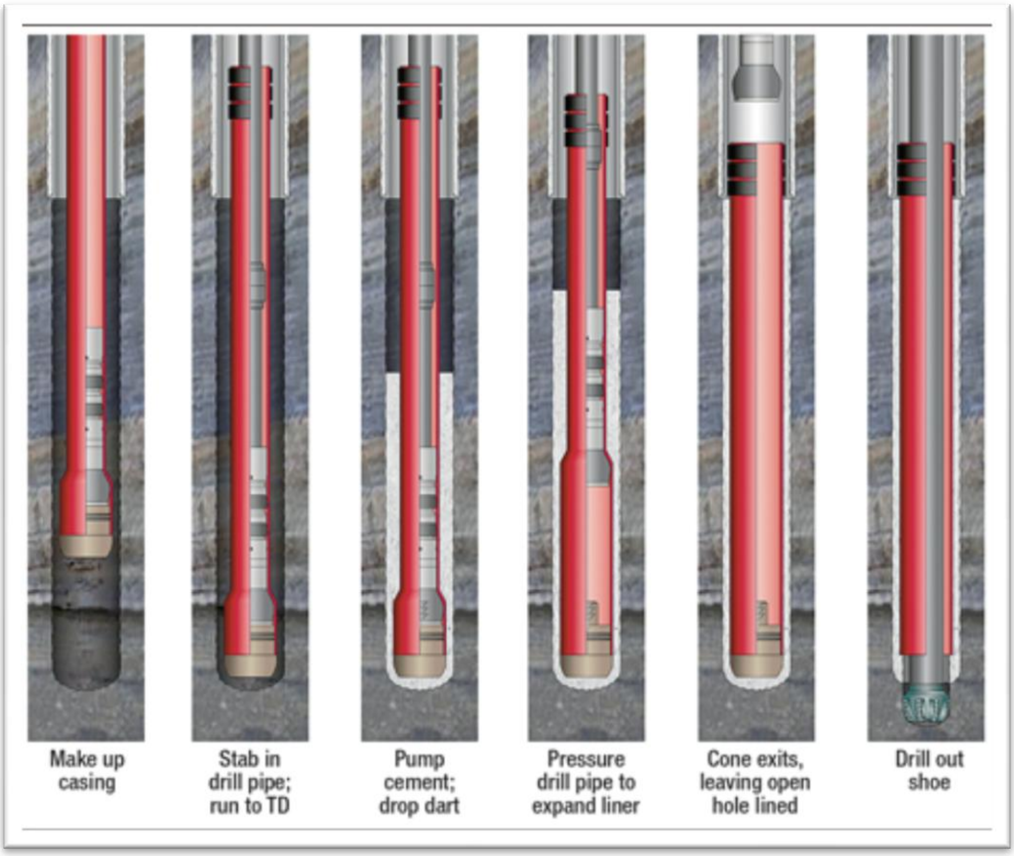

Figure 1.1: Tubular expansion process.

The expansion process will result in diameter increase, thickness decrease and a variation in length of the expanded tube. Deformation of the tube occurs when the internal wall of the tube is forced by the expansion cones to expand which increases its diameter and decreases its thickness as the process goes on. For the downhole expansion process in oil and gas industries, the tubular must be expanded until it meets the desired diameter without fracturing, bursting or damaging the tubular. The integrity of the tubular must be maintained after expansion to resist burst and collapse load in service.

The SET technology is widely used in repairing damaged wells due to cost effectiveness and high reliability. Analysis must be done on the stress formation to guide the repairing process. We will need to do simulations based on actual parameters from a well which had applied the SET technology for repairing purpose.

The well chosen is the Well Jing 708 of Huabei Oilfield, China where it was experiencing high-water cut (96.4%) in the production field in 2008. The casing was identified to be damaged and it had gone through a repairing process where SET technology was used [1]. Figure 1.2 represents the casing repairing operation.

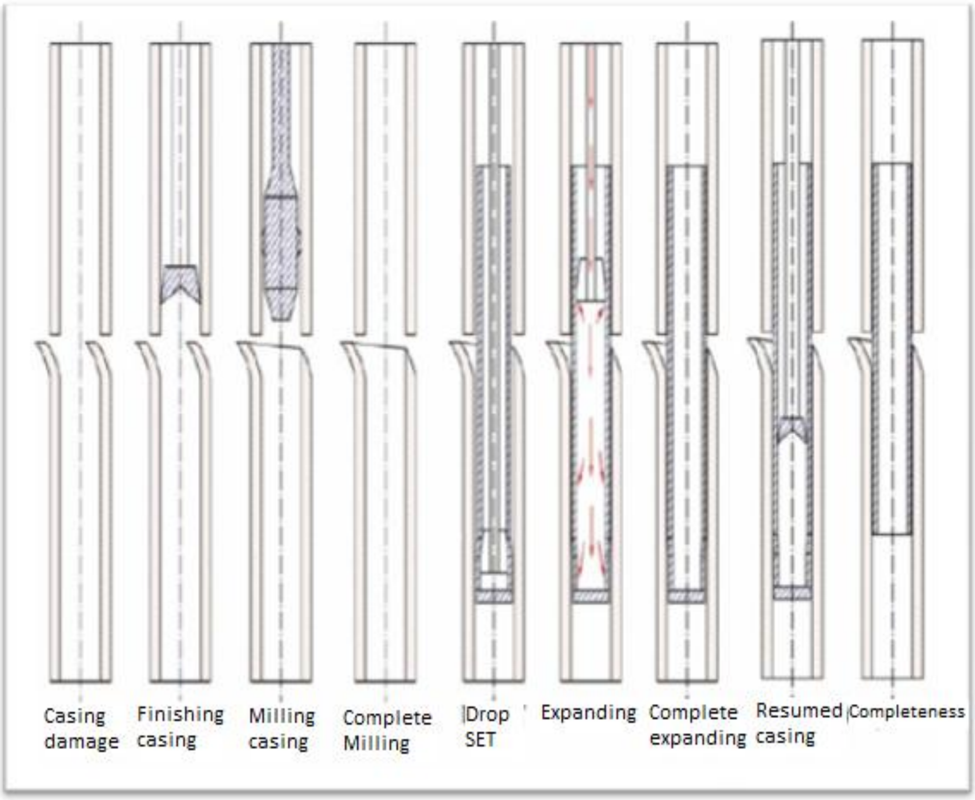

Figure 1.2: Operation process of casing repairing.

1.2 Problem Statement

1.2.1 Problem Identification

The demand of SET technology is huge despite of it is lacking of theoretical basis. Therefore, numerical simulation using ANSYS is proposed to analyze the stress formation for the application of SET since experimental analysis is expensive and time consuming.

1.2.2 Significance of the Project

Stress analysis result from the numerical simulation can be used to determine the integrity of the well after SET repairing took place. The analysis will also include different type of materials used for SET which can help determining the right amount of internal pressure applied and the change in thickness of the tubing for each material. This will provide theoretical support to ensure safe and reliable operation. Furthermore, the advancement of finite element method has made it reliable in doing analysis and it is a cheaper way compared to experimental work.

1.3 Objectives

This project is expected to meet the following objectives:

- To develop a finite element model of solid expandable tubular using ANSYS.
- To determine the effect of material properties on the stresses developed in solid expandable tubular wall and the changes in its radius due to internal pressure for linear and non-linear analysis using finite element method.
- To validate the results from simulations for linear and non-linear analysis.

1.4 Scope of Study

The scope of study is mainly to produce a numerical simulation using ANSYS software. The software should be able to simulate a model of the SET according to optimized boundary conditions which had gone through the repairing process. It is expected to produce stress diagram and displacement results for different materials. Mathematical calculations will also be done to verify the simulation results. For the first part of the project, research has to be done using journals, relevant research papers and ANSYS software to acquire knowledge on the SET technology, ANSYS application and related mathematical formulas. It is really necessary to be equipped with good skills to use ANSYS as the simulation is the main part of the project. Fail to do so will result in the failure of the whole project. Model of the well is created and formulas are studied and tested on ANSYS. For the second part of the project, when the model is perfected and the right formulas are produced, the simulation will commence and the results will be verified with the analytical solutions. It is important to be precise and accurate from the beginning of the project as the slightest error can cause bad result in the end.

1.5 Project Relevancy

This project is relevant to the oil and gas industry for safety precaution in operation. Stress analysis is a part of mechanical field and finite element method is one of the methods widely used in the engineering field. As mentioned before, The SET technology had been widely used in well repairing process but it is lacking in theoretical basis. The application of numerical analysis can provide excellent theoretical support and ensure safe and reliable operation of the repairing process

1.6 Project Feasibility

The project is fully computer based since I will be using ANSYS to create the model and simulate it. Microsoft Office Excel 2007 will also be needed for the mathematical calculations. The given time of the project should be enough to finish the project if all the plans and procedures go smoothly.

The given period of approximately seven months to complete this project is sufficient. The first half of the project will be filled with studies and research on the ANSYS software, gathering of information on SET parameter, and finding the right formula for analytical solutions. These will be done in the first three months of the project. The second half will be focused on the creation of axisymmetric model, running the simulation, mathematical calculation, results validation and conclusion of the results gained. The last month will be the crucial month of producing the final report which will include all the work done from the beginning and the discussion on the final result. All in all, it is believed that this project can be done in time since the worst thing that could happen throughout the project will be the unavailability of the ANSYS software but it is unlikely to happen here.

CHAPTER 2

THEORY AND LITERATURE REVIEW

2.1 ANSYS

There are two methods to build model. The first is to build a solid model in ANSYS directly. The second is to use computer-aided design and drawing software to build a geometric model, which is put into ANSYS software to form a finite element model.

In terms of the structural features of SET, 2D and 3D models are built and preliminary analysis is carried out. The results will show that 2D model has a short computing time and it reflects the stress and strain distribution in any axial plane, but it cannot reflect all forces on the whole SET. While the 3D model analysis could reflect all forces on the whole SET and contains the results of 2D model. Therefore, 3D finite element model is selected to analyze stress of SET [1].

According to geometric size of SET and expansion cone in application, direct method is used to build 3D finite element model of SET, expansion cone and the combination of them. On building model and meshing grid, the convenience for applying boundary conditions and loads is considered. The model must be optimized accordingly [2].

For the finite element model built, boundary conditions should be applied based on the field requirement. First, according to the actual demand, select the type of face-to-face contact, define the contact surfaces and the dynamic/static friction coefficient, and determine the contact surface and the target surface. Second, based on the boundary conditions of the expansion cone and SET in field application, the constraints are applied in the finite element model. Boundary conditions mainly include the constant velocity of the expansion cone, the type and attributes of contact surfaces, and the time step recording the stress–strain data.

In addition, considering the specific conditions of field application and the requirements of numerical simulation analysis, the boundary conditions are optimized, which is beneficial to the application of follow-up constraints and the introduction of the relative velocity of expansion cone, so as to make sure that the pre-processing accords with specific conditions. In short, after optimization for the combination model of SET, 3D finite element model is selected, which could directly reflect the changes of all SET parameters [3].

2.2 Finite Element Analysis

The Finite Element Analysis (FEA) is computational technique used to obtain approximations of boundary value problems in engineering. Boundary value problems are also sometimes called field problems. The field is the domain of interest and most often represents a physical structure. The boundary conditions are the specified values of the field variables on the boundaries of the field. Finite element analysis for SET repairing technology is in-line with the demand for oil field operation and contributes to more mature application of this technology [4].

The main features of FEA are:

- The entire domain is divided into small finite segments.
- Over each element, the behaviour is described by the displacement of the elements and the material laws.
- All elements are assembled together and the requirements of continuity and equilibrium are satisfied between neighbouring elements.
- Provided that the boundary conditions of the actual problem are satisfied, a unique solution can be obtained to the overall system of linear algebraic equations.
- The FEA is very suitable for practical engineering problems of complex geometries. To obtain good accuracy in regions of rapidly changing variables, a large number of small elements must be used.

Stress is generally defined as the average force (F) per unit area (A). This definition assumes that the stress is uniform over that particular area, but in reality stresses are seldom uniform over large areas. Therefore, it is more accurate if this area is made very small. The concept of stress at a point is physically valid because a small area would carry a small amount of force. The analysis results will have guiding significance in field operation [5].

In a three-dimensional Cartesian axes system there are six components of stress:

- Three direct (tensile or compressive) stresses (σ_{xx}, σ_{yy}, σ_{zz}) caused by force normal to the area
- Three shear stresses (σ_{xy}, σ_{xz}, σ_{yz}) caused by shear forces acting parallel to the area

The first subscript refers to the direction of the outward normal to the plane on which the stress acts, and the second subscript refers to the direction of the stress arrow [6].

A 'stress matrix' or a 'stress vector', which contains all stress components, can be conveniently expressed as

$$[\sigma] = \left[\begin{matrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{xz} \\ \sigma_{yz} \end{matrix} \right] \qquad \text{(Eq. 2.1)}$$

Similarly, a 'strain vector' can be defined as

$$[\varepsilon] = \left[\begin{matrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \varepsilon_{xy} \\ \varepsilon_{xz} \\ \varepsilon_{yz} \end{matrix} \right] \qquad \text{(Eq. 2.2)}$$

2.3 Solid Expandable Tubular (SET)

There are two problems when comes to casing sizes in well drilling activities which can increase the well costs and lead to the failure of reaching the required well's objective. First problem is during anticipated conditions that require an additional casing string of casing after the well has been started. Second problem is during known conditions that require multiple casing strings for a well before it has been started.

For the first problem, the sizes and depths are already selected, and one or more strings may be solid expandable tubing before using additional strings. Unstable bore-hole conditions and pressure may call for additional string that was not originally planned for. Another problem which is quite similar is the possibility that a planned casing string may stuck before reaching its planned depth. This condition will require an additional string. Both two cases above must use solid expandable tubing as the additional string to reach the total depth.

As for the second problem, it had been very common that the drilling operations may find it is necessary to run five or six strings or even as many as 10 strings to reach its objective. A conventional approach to this problem requires some very large well bores and casing to reach total depth with a final casing string size that can accommodate acceptable amount of production. In each case, size and clearance become serious problem [7].

The new technology encounter to solve these problems is using SET. SET is a type of tubing which can be run through conventional casing or another SET then expanded to a larger diameter than a conventional casing run through that same size pipe.

2.3.1 Tubing Expansion Process

The system is run through existing casing or liner and is positioned in open hole, then expanded from the bottom up. When the expansion cone reaches the overlap between the expandable tubing and the existing pipe string, the cone expands a special hanger joint to provide a permanent seal between the two strings.

There are two reasons why the solid expandable tubing system is expanded from the bottom up. The first reason is related to the shortening of the liner during expansion. Liners often are difficult to position at their planned total depth, and consequently may be positioned somewhat higher. A top down expansion would first anchor the expandable liner in the previous pipe string, and the ensuing expansion would shorten the liner from the bottom up. The shortened, expanded liner then may not cover an adequate interval at the bottom of the hole. On the other hand, a bottom up expansion first spots the expandable liner at its lowest depth, and the subsequent shortening experienced during the remaining expansion occurs in the overlap. Liner coverage at the bottom of the hole is thus ensured.

The second reason for using bottom-up expansion for solid expandable tubing is related to inner-string operations. It is easier to generate greater forces by pumping through and pulling on the inner-string than it is by adding weight to it. Because the inner-string already is being pulled out of the hole as a part of the bottom-up expansion operation, additional tensional forces can be added to the string, if necessary, to serve as a secondary force to drive the expansion. Whereas, with top-down expansion, any downward force or additional weight applied to the inner-string would serve as the secondary expansion force, placing the inner-string (drill pipe) in compression. Drill collars and heavyweight drill pipe would be needed as part of the inner-string to supply additional weight. This would only add time to inner-string makeup with minimal compression forces being added in comparison to the tensional forces available. The size of tubing and its mechanical properties typically determine the propagation forces required to expand it. The three materials to be analysed using the numerical simulations are Aluminium 2014-T6, Stainless Steel 304 and Titanium (Ti-6Al-4v) [8.

2.3.2 Advantages of Solid Expandable Tubular Technology

2.3.2.1 Slimming the Wellbore Design to Enhance Drilling Economics in Field Development

It is very clear if solid expandable tubular technology is plan inside casing design, it means to slim the wellbore. By incorporating the solid expandable tubing system at the beginning, the operator achieves a slimmer well profile while still maximizing hole size at total depth, reduces drilling time to total depth and reduces drilling costs.

By using SET to achieve a large wellbore at total depth resulted in the following features, advantages and benefits [9]:

- Higher rate-of-penetration (ROP) in long intermediate casing section (36% enhancement).
- Overall drilling cost savings (15-20%) using slimmed wellbore vs. big bore pipe program.
- Improved drilling performance and lower equivalent circulation density (ECD) below the solid expandable tubing system.

2.3.2.2 Constructing Extended-Reach Wells

In cases for which engineering analysis or previous experience indicates a potential problem with pressure differential, hole stability, pressure gradient, or geomechanical interaction, the standard solution is to solid expandable tubing casing. Because of the increased lateral distance drilled in an extended-reach well, much longer casing sections are run and additional casing strings may be needed. This condition can lead to the use of far greater initial hole and casing size.

Solid expandable casing was designed primarily to combat these concerns. By covering a swelling shale or lost-circulation zone with expandable casing, drilling could continue with less hole-size loss than experienced with a conventional casing string. The result was a smaller casing size at the surface, shorter drilling times, and lower completion costs [10].

2.3.2.3 Enable Practical Well Re-entry in Mature Field

The problem in mature fields is the lack of available slots on offshore platforms. It usually needs a new approach for a well re-entry program. An operator needs to explore a more economical and practical approach that reduces drilling time and maximizes the use of existing solid expandable tubings. The traditional methods are retrieving and milling operations but often time consuming and usually took longer than forecasted. For this problem, two method considered which are new-drills with slot recovery options and re-entry with solid expandable tubing. The slot recovery option is rejected for this application because it required studies about the compatibility of the slot-recover deflecting tool and the platform. When the timeframe that required to put the well back on production become a factor in preliminary analysis, it is shown that this approach is not as economic feasible as using solid expandable tubing. Therefore, the solid expandable system identified as the most viable alternative considering its cost effectiveness and technical contribution.

A significant benefit becomes apparent when offshore operators use solid expandable tubing in conjunction with sidetracking technology. In case of old platforms with no remaining slots, the only economic feasible method to reactivate a field may require re-entering existing wells using solid expandable tubing to preserve a larger inside diameter. The cost of drilling a new well may not be the operator’s economic parameters. Using the solid expandable tubing in conjunction with window exit procedures increases the probability of reaching the planned total depth with desired casing size. An optimized hole diameter in production zones result in higher flow rates, an attractive benefit in not only initial field development but in field re-development as well [11].

2.3.2.4 Facilitate Intelligent-Well Technology Application in Existing Multilateral Wells

A major Middle Eastern operator has been exploiting long reach, openhole, multi-lateral technology to maximize oil recovery from a large field producing from a limestone reservoir. Significant production increases have been attained, but the recovery efforts have been hampered by an inability to re-enter openhole sidetracks for remedial purposes. The operator has also been encountered water production problems with no real way to identify the source of the water or to remediate it if the source is found.

These problems have been overcome with the introduction of solid expandable tubing in combination with intelligent well technology. An openhole multi-lateral involves sidetracking out of the main bore while in the open hole. In wells where the technology has been applied, production rates have been generally been higher and recovery has improved. A lower unit development/operating cost is also a key driver in this type of well. The new adaptation of using solid expandable tubing and intelligent well technology allows for re-entry into lateral for remedial work and real-time pressure, temperature and flow data without the need for well intervention. Intelligent well components allow for quick identification of any water-producing zones and provide a means for shutting off the water production, again without the need for well intervention [12].

2.3.2.5 Deepwater Application

Drilling margins (the difference between pore pressure gradient and fracture pressure gradient) narrow as operations move into deeper water. These narrow drilling margins require more casing strings to drill to an equivalent depth below the mudline compared to a well drilled in a shallower water depth. In some cases, using conventional casing programs with an 18-3/4 in. blowout preventer (BOP) stack and a 21 in. outside diameter (OD) drilling riser, well objectives cannot be reached with sufficient hole size for evaluation and production operations.

An ultra-deepwater well, in water depths over 5,000 ft, can reached it's objectives by using a 13 3/8 x 16 in. solid expandable tubing system. This enabling technology can also provide contingency casing deeper in the well. Traditionally, as water depth increases, the size of the drilling vessel and equipment capacities increases. Water depth, ocean conditions, BOP, and riser size affect the size of the rig. The well objectives and casing program determine the minimum BOP stack and riser size. The riser size affects the following systems on a drilling rig [13]:

- Deck load
- Deck space
- Riser tension capacity
- Hoisting system capacity
- Mud system volumes
- Bulk volumes

2.3.2.6 Optimizing Well Design

Several avenues can be considered to optimize well design with expandable casing. An obvious opportunity for realizing savings in expenditures and resources comes by reducing the casing size itself and by drilling in the most efficient size ranges. Savings are realized in the following topside costs:

- Location or platform costs
- Tubing costs
- Mud products costs

Current costs from operators in the Middle East indicate that savings on the order of 20% to 40% are realized by eliminating one full casing size. As technology for placing wells in the optimum spot in a pay zone and as the length of producible sections continues to improves, accepting deliverability with a small hole at total depth is an antiquated compromise.

A conventionally drilled slim hole design also lack some flexibility to cope with unexpected well problems. If a lost circulation or overpressure zone is encountered, there may not be enough diameters left in the existing hole to drill to total depth or enough room left to effectively deal with corrosion or cement isolation problems.

Hole sizes exist that are more cost effective to drill. It is generally accepted that hole sizes below 7-7/8 in. are more difficult to drill than larger sizes due to the following:

- Less durability of smaller bit and other BHA parts because of their smaller mass.
- Flexibility of the assemblies which can lead to drag and buckling problems.
- Lack of space available to design engineers that prevents the removal of stress concentrations.

It is difficult to scale down items with moving parts, like roller cone bits, roller reamers and other downhole tools, due to size strength and heat dissipation issues. Scaling down can lead to problems dissipating heat that reduces bearing and cutting structure life.

On the opposite end, hole sizes above 12-1/4 in. tend to be slow to drill. Providing energy to break the rock at the bit face becomes more difficult. Drilling a larger hole requires [13]:

- More drilling mud
- Bigger volumetric flow rate
- Bigger pumps
- More solids control equipment
- More waste disposal
- More expensive BHA parts
- More steel for casing
- More cement for zonal isolation.

2.4 The Lame's Equation Theory

In continuum mechanics, stress is a measure of the internal forces acting within a deformable body. Quantitavely, it is a measure of the average force per unit area of a surface within the body on which internal forces act [14].

The linear stress analysis results obtained from the numerical simulations will be verified using Lame's equation. The assumption of the equation is the material of the cylinder is homogenous and isotropic.

For a 3D stress analysis of a hollow cylinder, there are three types of stress to be focused on which are:

- Radial Stress
- Hoop/Circumferential Stress
- Longitudinal Stress

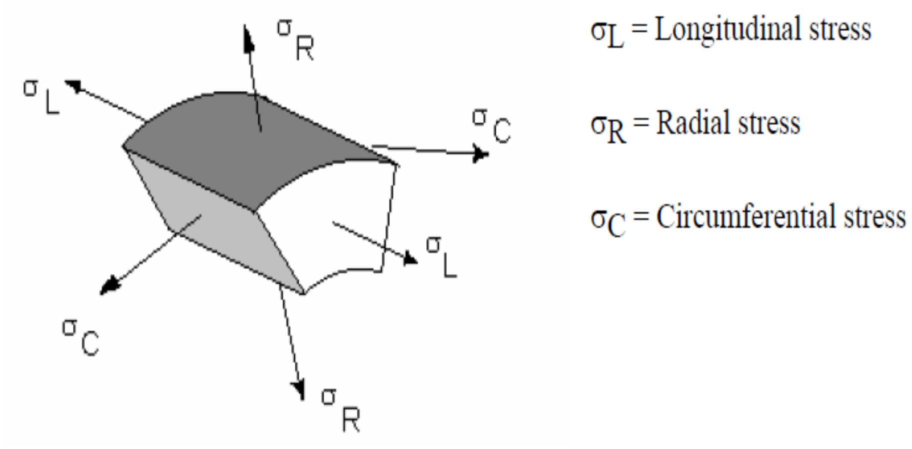

Figure 2.1: Stress diagram.

The stress diagram is shown in Figure 2.1. The three stresses can be determined based on Lame's equation of thick wall cylinder. The tubing is considered as a thick wall cylinder when the thickness is greater than one-twentieth of its inner diameter. The stress along the thickness or radial stress of the cylinder must be taken into account for thick wall cylinder because it varies significantly between the inner and outer radius. Figure 2.2 shows a thick wall cylinder diagram.

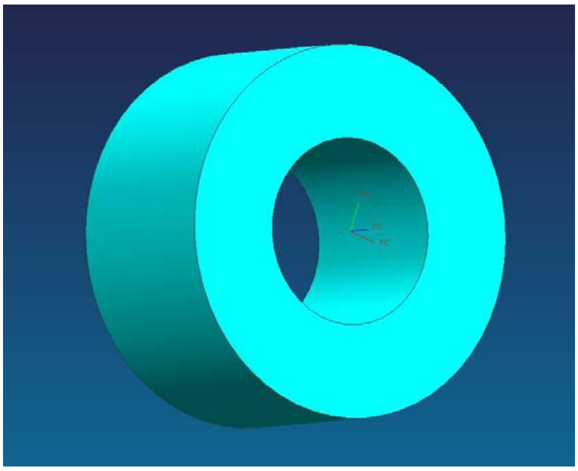

Figure 2.2: Thick wall cylinder.

In the case of thick wall cylinder subjected to uniform internal and external pressure, the deformation is symmetrical about the axial axis but for this project, the cylinder is subjected to internal pressure only for the expansion process. There is no axial stress since the cylinder is open ended.

So, for internal pressure only case, based on the Lame's equation theory, the radial, circumferential and longitudinal stress, and change in radius can be determined using below equations respectively [15].

$$\sigma_r = \frac{p_i a^2}{b^2 - a^2}\left(1 - \frac{b^2}{r^2}\right) \qquad \text{(Eq. 2.3)}$$

$$\sigma_\theta = \frac{p_i a^2}{b^2 - a^2}\left(1 + \frac{b^2}{r^2}\right) \qquad \text{(Eq. 2.4)}$$

$$\sigma_l = \frac{\pi a^2 p_i - \pi b^2 p_0}{\pi b^2 - \pi a^2} \qquad \text{(Eq. 2.5)}$$

p_i = internal pressure, N.
σ_r = radial stress, N/mm^2.
σ_Θ = hoop stress (also known as tangential stress or circumferential stress), N/mm^2.
a = inner radius of the cylinder, mm.
b = outer radius of the cylinder, mm.
r = radius at a point in the cylinder wall where stresses are to be determined, mm.

$$u = \frac{a^2 p_i r}{E(b^2 - a^2)}\left[(1 - v) + (1 + v)\frac{b^2}{r^2}\right] \qquad \text{(Eq. 2.6)}$$

E and v are Young Modulus and Poisson's Ratio, respectively.

Another equation is also used to calculate discrepancy between simulation values with theoretical values that obtained from Lame's equation.

$$\varepsilon_{A-T} = \frac{\text{Simulation Value} - \text{Theoretical Value}}{\text{Simulation Value}} \qquad \text{(Eq. 2.7)}$$

2.5 Isotropic Hardening

Linear finite element analysis assumes that there is no change in Young's modulus throughout the loading process. This assumption is acceptable as long as the stresses and strains always stay elastic. For larger loads, the linear model fails to take into account the plastic effects which the material experiences which becomes a poor model for reference. To account for the plastic effects, Young's modulus must be altered based on the known behaviour of the material. In bilinear analysis, additional values must be obtained to increase the ability of the analysis to produce accurate result [16]. Figure 2.3 shows a typical stress-strain curve.

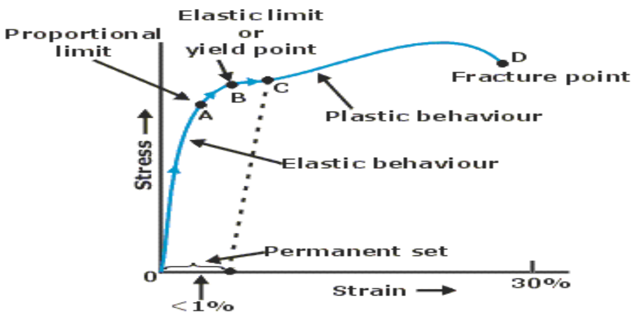

Figure 2.3: Typical stress-strain curve.

For bilinear analysis using finite element method, the stress-strain curve is simplified by finding a new value called tangent modulus, E_t. Young's modulus is referred as the slope for the elastic region until the yield strength of the material. Tangent modulus is the average slope representing the stress-strain curve after the elastic region. Figure 2.4 shows the simplify stress-strain curve for bilinear analysis.

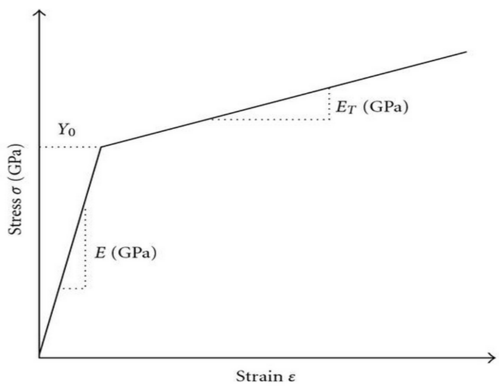

Figure 2.4: Bilinear stress-strain curve.

The value of tangent modulus can be calculated using the following equation [17].

$$E_T = \frac{\left(\left(1+\frac{\%EL}{100}\right)(S_{uts})\right)-S_y}{\ln\left(\frac{1}{1-\frac{\%RA}{100}}\right)-\frac{S_y}{E}} \qquad \text{(Eq. 2.8)}$$

E_T = Tangent modulus, GPa.

E = Young's modulus, GPa.

S_{uts}= Ultimate tensile strength, MPa.

S_y= Yield strength, MPa.

%RA = Percent reduction area.

%EL= Percent elongation.

The value of %EL can be obtained from books and internets but the value of %RA can be approximated using the following equation [17].

$$\%RA = \frac{200\times\%EL}{100+(2\times\%EL)} \qquad \text{(Eq. 2.9)}$$

CHAPTER 3

METHODOLOGY

3.1 Research Methodology

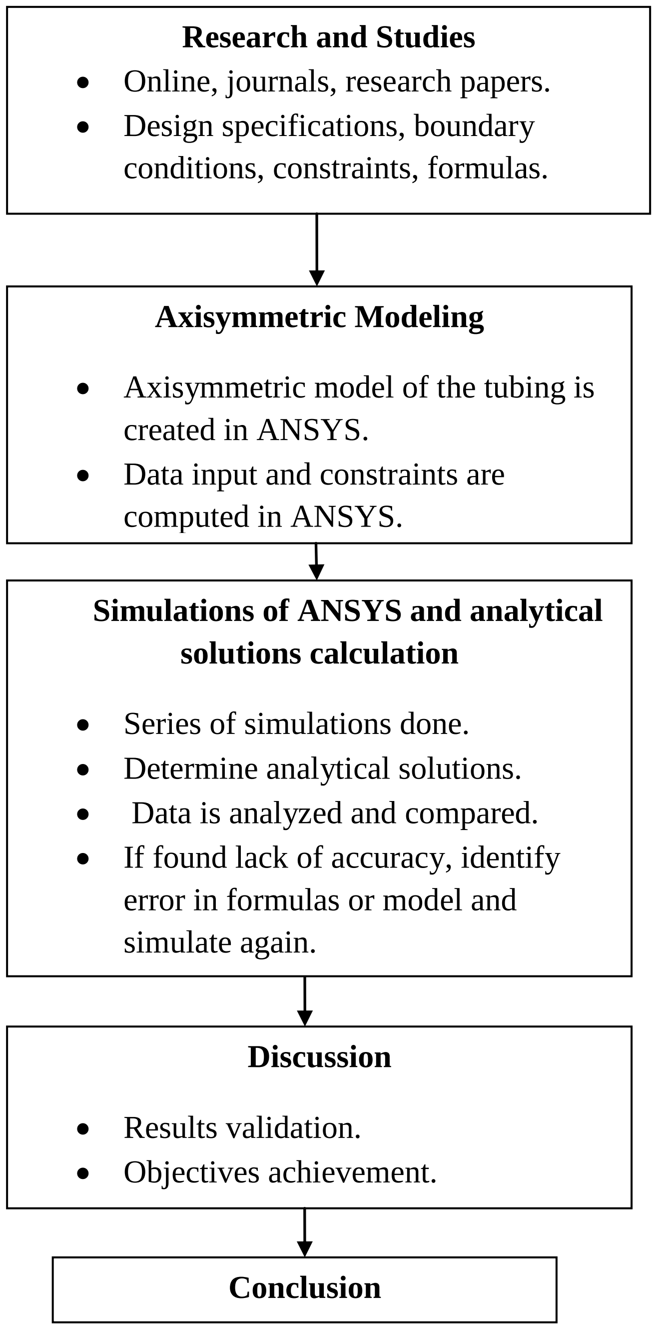

Figure 3.1: Research methodology flowchart.

3.2 Meshing Size Selection

Choosing the right meshing is important to produce accurate results at the end of the simulations. A series of simulations using meshing size of 10, 5, 1, 0.5, 0.1 and 0.05 was done using Stainless Steel 304 with internal pressure of 40 MPa. The inner radial stress produced by ANSYS for each size is then compared with the theoretical inner radial stress which is -40 MPa to find the suitable meshing size. Table 3.1 shows the comparison.

Table 3.1: Percentage error for different meshing size.

Size	Inner Radial Stress (MPa)	Percentage Error (%)
10	-38.377	4.06
5	-39.546	1.14
1	-39.969	0.08
0.5	-39.992	0.02
0.1	-40	0
0.05	-40	0

The table above shows that meshing size 0.1 and 0.05 has the most accurate result with 0% error but meshing size 0.05 will consume longer time to solve. So, 0.1 is used for the whole project.

3.3 Finite Element Modeling and Simulations for Linear Analysis Using ANSYS

Finite Element Method was performed by using ANSYS. Modeling was done after completing the data gathering. Well Jing 708 of Huabei Oilfield, China was taken as the case study. The well was inserted with SET with design parameter as shown in Table 3.2.

Table 3.2: Design parameter for SET [1].

Parameter	Value
Inner radius	0.049 m
Outer radius	0.057 m
Internal pressure	40 MPa

Below is the procedure by using ANSYS software.

3.3.1 Preliminary Decisions

The analysis, model and element type must be defined first. Structural analysis will be done to determine deformations, stresses, and reaction forces. Axisymmetric was used to model the geometry of the solid expandable tubular.

3.3.2 Preprocessing

Quadrilateral element was chosen for axisymmetric analysis and tetrahedron element was chosen for 3D modeling. PLANE82 is used for axisymmetric analysis since the element can be used either as a plane element or axisymmetric element. Figure 3.2 is an example of choosing the element type in ANSYS.

Command: ***Preprocessor – Element Type – Add/Edit/Delete – Define Element Types***

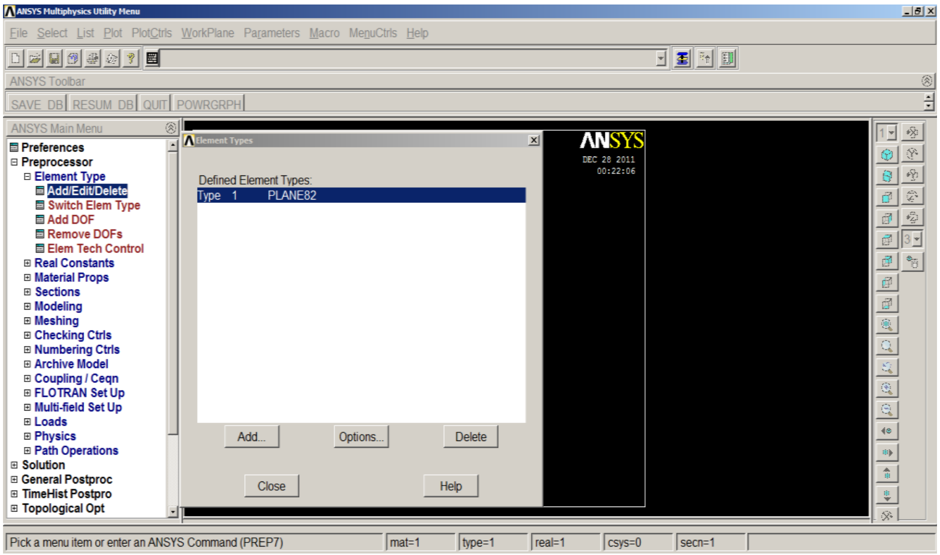

Figure 3.2: Selection of PLANE82 in ANSYS software.

The next step is to select axisymmetric as element behaviour for PLANE82 which is shown in Figure 3.3.

Command: ***Click Options – Select 'Axisymmetric' for K3 (Element behaviour)***

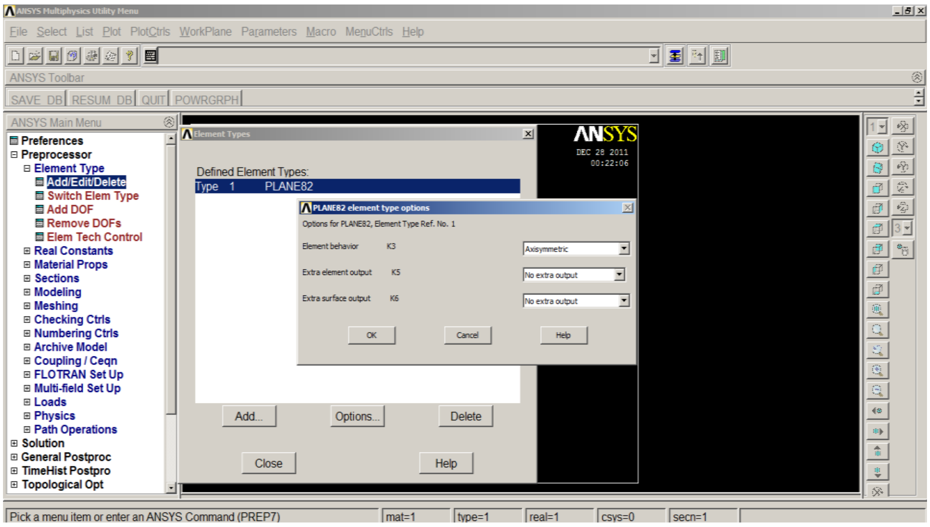

Figure 3.3: Selection of axisymmetric as element behaviour.

Then, three more steps are involved in preprocessing which are defining material properties, creating or importing model geometry and meshing the geometry. The material is defined by entering its material properties which include modulus of elasticity and Poisson's ratio for linear analysis and true stress/strain value for non-linear analysis. The materials selected for the project were listed in Table 3.3.

Table 3.3: Mechanical properties of materials (SI Units) [4].

Material	**Yield Strength (MPa)**	**Ultimate Tensile Strength (MPa)**	**Modulus of Elasticity, *E* (GPa)**	**Poisson's Ration, *v***
Aluminium 2014-T6	414	469	73.1	0.35
Stainless Steel 304	207	517	193	0.27
Titanium (Ti-6Al-4v)	924	1000	120	0.36

Figure 3.4 shows the material model of the element where the modulus of elasticity and Poisson's ratio are computed.

Command: ***Preprocessor – Material Props – Material Models – Linear – Elastic – Isotropic***

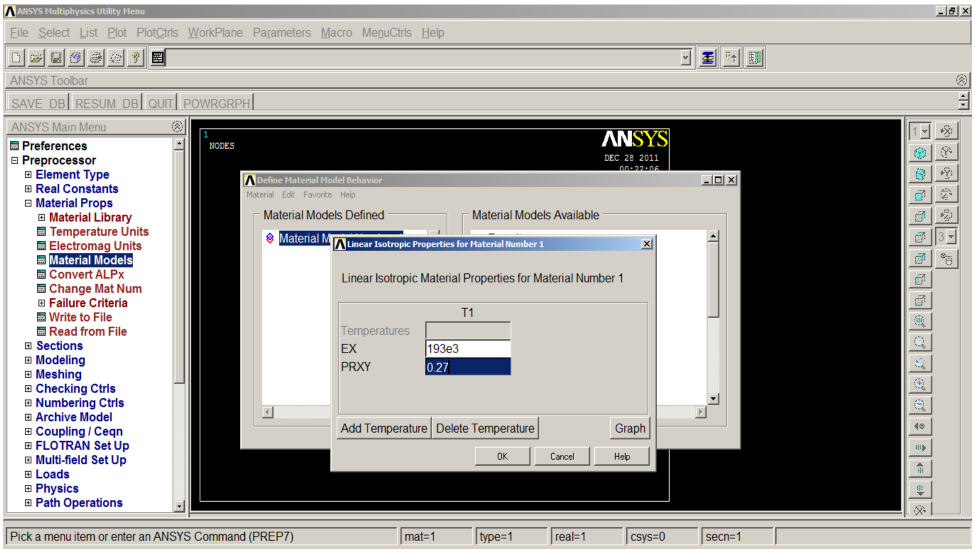

Figure 3.4: Material Model.

Tubing that had an inner radius of 49 mm and outer radius of 57 mm will be taken for the simulation. The length of the tubing is considered arbitrary. The tubing will be studied as a long segment and open-ended cylinder representing the tubular in well. So, a segment of 1000 mm long is used for the finite element model. Figure 3.5 shows the 3D model of the tubing. The tubing is created using below command.

Command: ***Preprocessor – Modeling – Create – Volume – Cylinder – Hollow Cylinder***

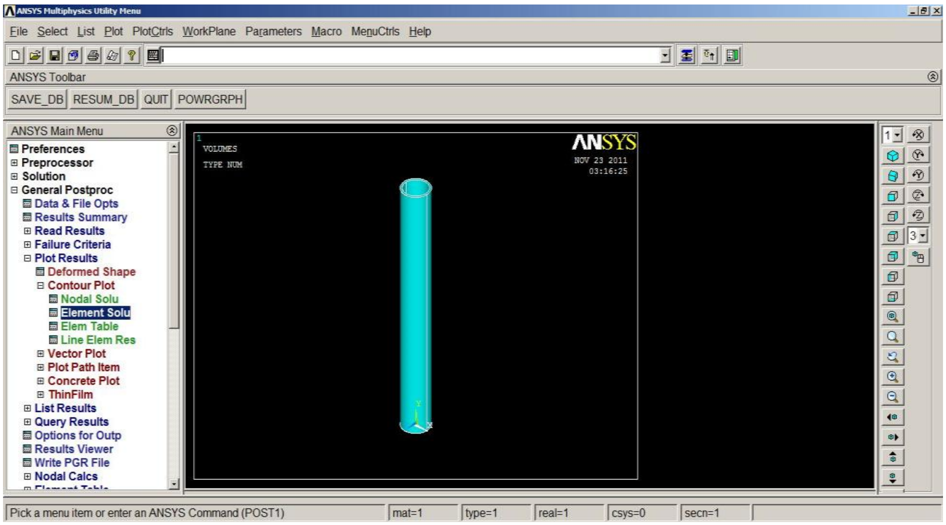

Figure 3.5: 3D Modelling.

As in the case of axisymmetric analysis, there is no need to create a full geometry of the tubing. Only a quarter of the tubing with relatively small height is enough for the analysis. Thus, a segment of 5 mm in height was used for the finite element model. Figure 3.6 shows the axisymmetric modelling where y-axis is the axis of symmetry. Since the axisymmetric behaviour have been selected for the element chosen, the model will have axisymmetric behaviour on the left side of y-axis.

Command: ***Preprocessor – Modeling – Create – Areas – Rectangle***

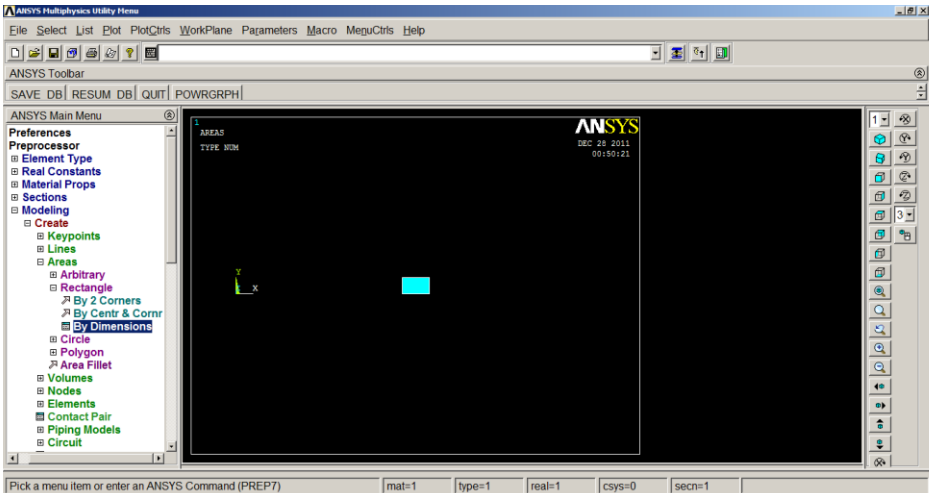

Figure 3.6: Axisymmetric model.

The next step is meshing which is the process of dividing the geometry into nodes and elements. The ANSYS can automatically generate the nodes and elements by computing the element type, real constants and material properties. The element size controls the fineness of the mesh. The smaller the size, the finer the mesh and the simulation accuracy will improve. Proper size must be chosen since the smaller the size, the longer it will take for the simulation to run. In this project, the element size was manually specified by entering an element edge length of 0.1 units. Figure 3.7 shows the finite element model after meshing process.

Command: ***Preprocessor – Meshing – MeshTool – Under 'Size Control : Global' click set – Enter 0.05 for element edge length – Mesh***

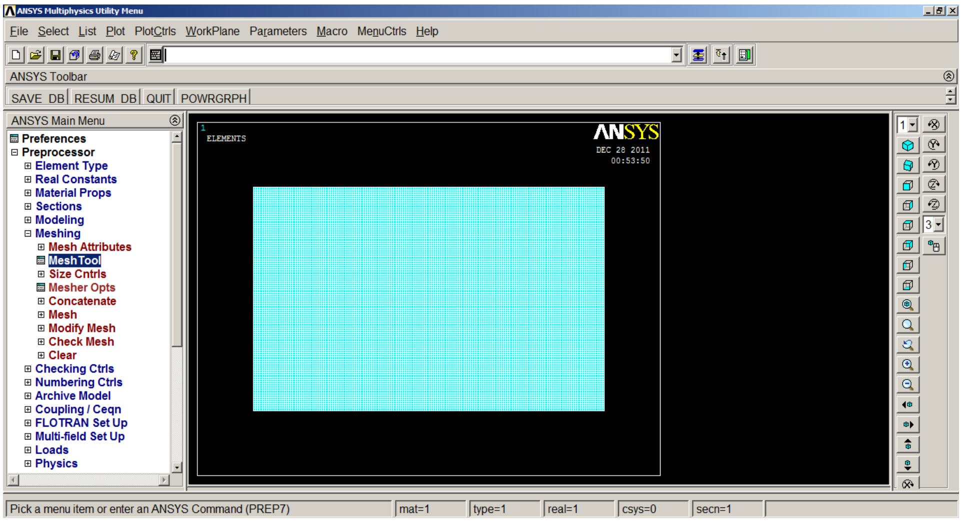

Figure 3.7: Meshing.

3.3.3 Solution

The next step is to apply appropriate boundary conditions and proper loading. There are two ways of applying the boundary conditions and loading to the model in ANSYS. The conditions can be applied to the solid model which are keypoints, lines and areas, or the conditions can be directly imposed on the nodes and elements. The first approach is preferable since there will be no need to reapply the boundary conditions and the loads when changes of meshing are needed.

Since the tubing is not moving downward or upward during the expansion process, the displacement should be zero in this direction where constraints of Y direction is applied to the bottom line of the rectangle as shown in Figure 3.8.

Command: ***Solution – Define Loads – Apply – Structural – Displacement – On Lines***

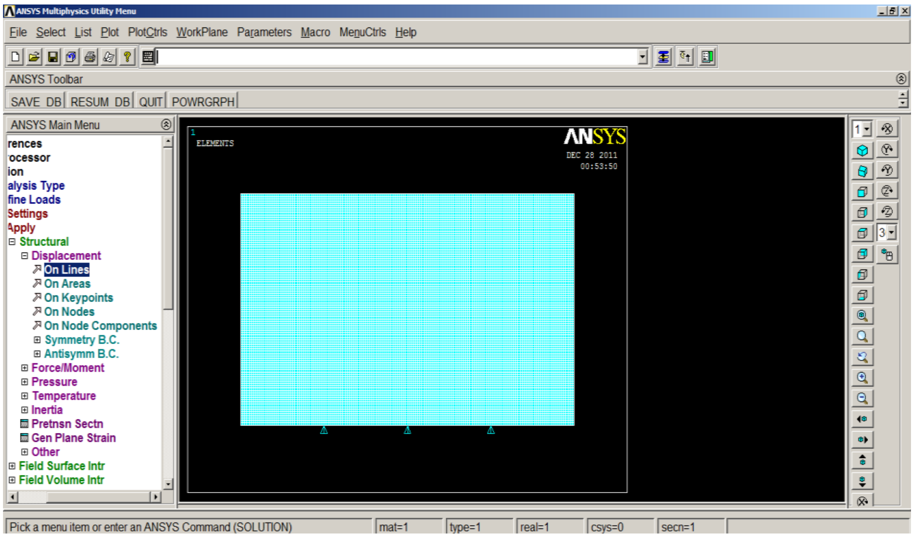

Figure 3.8: Constraints.

In order to apply force or load, the node or keypoint number, force magnitude and direction of force are needed. In this analysis, the internal pressure of 40 MPa was applied at the left line of the rectangle which is represented by a red arrow as shown in Figure 3.9.

Command: ***Solution - Define Loads – Apply - Structural – Pressure - On Lines***

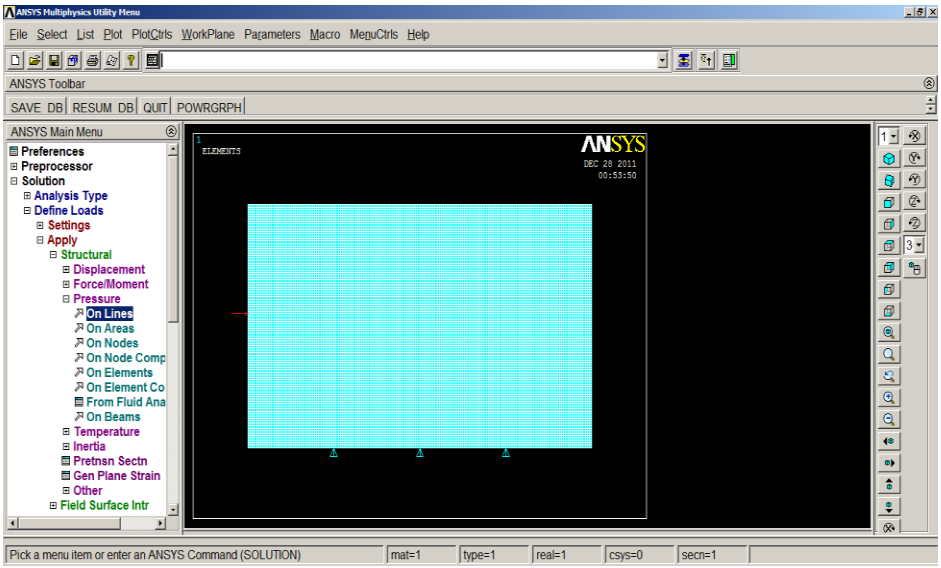

Figure 3.9: Applying internal pressure.

Once the model is completed and the boundary conditions together with appropriate loads are applied, the model can be solved. Figure 3.10 shows the simulation is successfully done.

Command: ***Solution – Solve – Current LS***

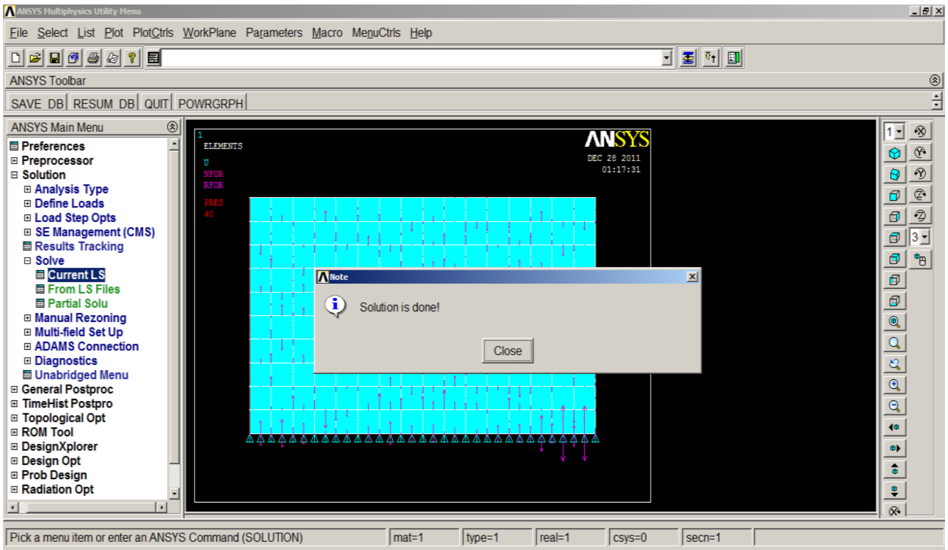

Figure 3.10: Solving the simulation.

3.3.4 Post Processing

Post processing is the final step in the finite element analysis process. This step involves reviewing the results and to validate them. Reviewing the results means generating the stress diagram where it will give proper visual result on the build up stress. To validate the result, this is where the mathematical solution will be taken into account. The stress created at a point of study will be compared between the simulated one and the calculated one. The error will be further evaluated to decide whether the simulation is acceptable or not. Figure 3.11 and Figure 3.12 illustrate the example of the result which shows the deformed and undeformed shape for 3D and axisymmetric analysis.

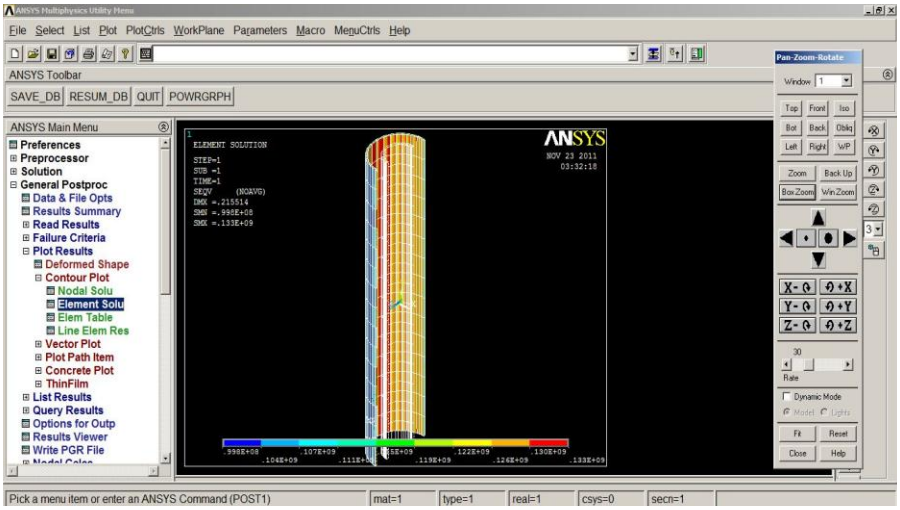

Figure 3.11: 3D stress diagram.

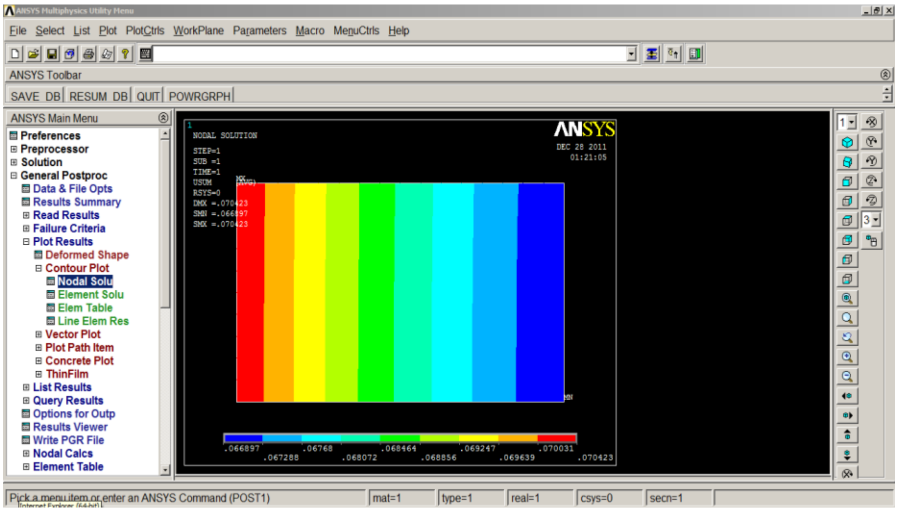

Figure 3.12: Axisymmetric stress diagram.

3.4 Finite Element Modeling and Simulations for Non-linear Analysis Using ANSYS

For non-linear analysis, the steps required are similar with the linear analysis except that the material model will require computation of additional values which are tangent modulus and yield stress as mentioned in Chapter 2. Figure 3.13 shows the computation of these two values.

Command: ***Preprocessor – Material Props – Material Models – Nonlinear – Inelastic – Rate Independent – Isotropic Hardening Plasticity – Mises Plasticity – Bilinear***

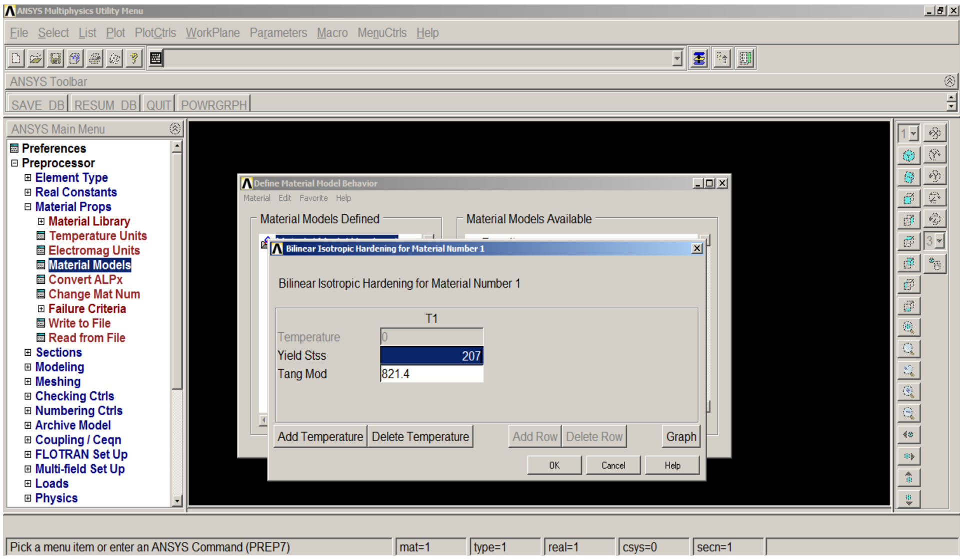

Figure 3.13: Computing tangent modulus and yield stress values.

The values of tangent modulus can be calculated using Eq. 2.8 and Eq 2.9. Table 3.4 shows the tangent modulus for the three selected materials.

Table 3.4: Tangent modulus for the three selected materials.

Material	**Tangent Modulus, E_T (GPa)**
Aluminium 2014-T6	0.5767
Stainless Steel 304	0.8214
Titanium (Ti-6Al-4v)	0.9453

Since all the three materials will only exhibit non-linear behaviour if the stress formation is above their yield strength, the internal pressure applied must be different for each of them. Eq. 2.4 was used to find the suitable internal pressure that will produce stress exceeding material's yield strength. So, for Stainless Steel 304, an internal pressure of 40 MPa is applied while 65 MPa and 140 MPa are applied using Aluminium 2014-T6 and Titanium (Ti-6Al-4v), respectively.

3.5 Project Activities

Table 3.5: Project Activities.

Activities	Starting Date	Finishing Date
Studies on Methods to use ANSYS	13 June 2011	31 July 2011
Gathering data on the well and SET parameter	13 June 2010	31 July 2011
Generating 2D and 3D model	11 July 2011	14 August 2011
Studies on constraints related for simulation	15 August 2011	11 September 2011
Studies on formulas for mathematical calculation	15 August 2011	11 September 2011
Model and input computation into ANSYS	12 September 2011	25 September 2011
Run simulations and determine analytical solutions	26 September 2011	23 October 2011
Stress diagram generations	24 October 2011	30 November 2011
Result Analysis and Comparison	14 November 2011	30 November 2011
Report Documentation	28 November 2011	31 December 2011

3.6 Project Milestone

Table 3.6: Project Milestone.

	2011						
Activities	June	July	Aug	Sept	Oct	Nov	Dec
Studies on Methods to use ANSYS							
Gathering data on the well and SET parameter							
Generating 2D and 3D model							
Studies on constraints related for simulation							
Studies on formulas for mathematical calculation							
Model and input computation into ANSYS							
Run simulations and determine analytical solutions							
Stress diagram generations							
Result Analysis and Comparison							
Report Documentation							

3.7 Tools and Equipments

In this project, the tools used are ANSYS software for numerical simulation and Microsoft Office Excel 2007 for analytical solution.

CHAPTER 4

RESULTS AND DISCUSSIONS

The linear and non-linear simulations using ANSYS were done using three different materials namely Stainless Steel 304, Aluminium 2014-T6 and Titanium(Ti-6Al-4v). Axisymmetric analysis was chosen due to the geometry of the tubing which makes it possible. The results are shown and discussed into two sections in this chapter which are linear and non-linear.

4.1 Linear Finite Element Analysis of Stress Distribution for Thick Wall Cylinder

4.1.1 Stainless Steel 304

Figures 4.1 and 4.2 show the deformed and undeformed shape of 3D and axisymmetric, respectively. It can be seen that the wall of the tubing expanded uniformly with increase in diameter and decrease in thickness. The 40 MPa internal pressure applied had forced the wall to expand radially. This justifies the expansion process of the solid expandable tubular.

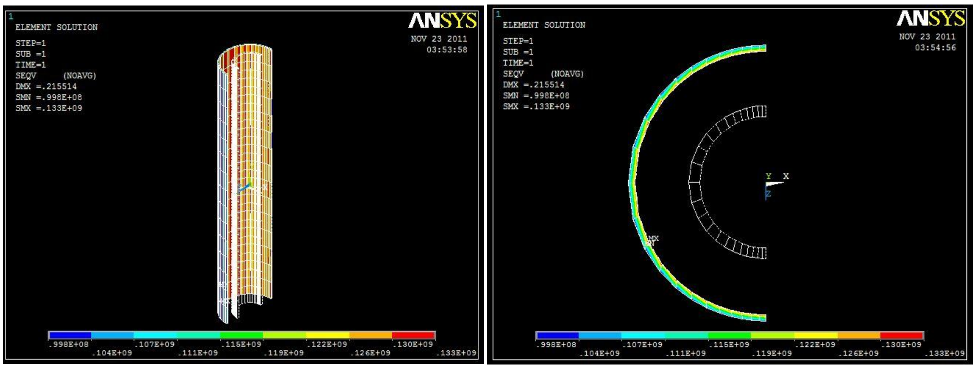

Figure 4.1: Deformed and undeformed steel expandable tubular (3D).

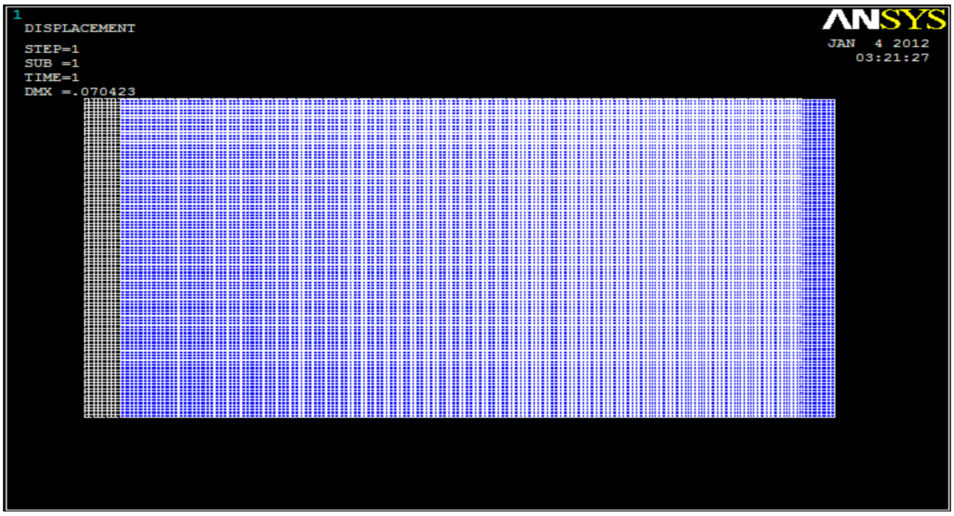

Figure 4.2: Deformed and undeformed steel expandable tubular (Axisymmetric).

Figure 4.3 shows the expansion process causes the diameter of the tubing increases while decreasing the thickness of the tubing. The inner radius displacement is 0.070423 mm and the outer radius displacement is 0.066897 mm which gives the final thickness of 7.996474 mm. The thickness decreased 0.003526 mm from its original which is 8 mm. Theoretically, by applying Eq. 2.6, the change in inner radius is 0.070405 mm and change in outer radius is 0.066897 mm.which gives the final thickness of 7.996492 mm. Note that the inner radius displacement is higher than the outer radius displacement due to the existence of the internal pressure only.

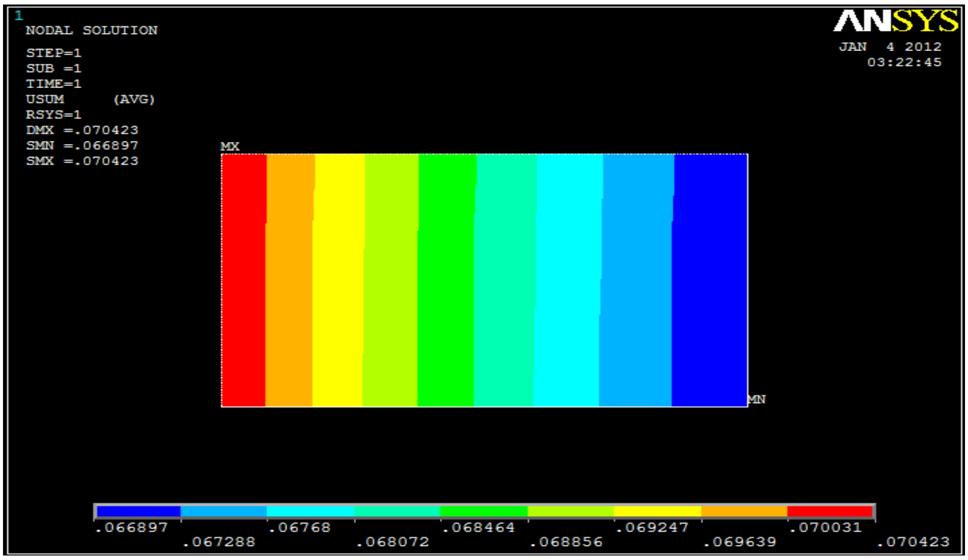

Figure 4.3: Change in radius of steel expandable tubular.

Figure 4.4 shows the radial stress, S_x is equal to the internal pressure applied which is -40 MPa on the interior of the tubing and 0.0000436 MPa on the exterior. Theoretically, by applying Eq. 2.3, the radial stress for the interior of the tubing is -40 MPa and 0 MPa for the exterior of the tubing.

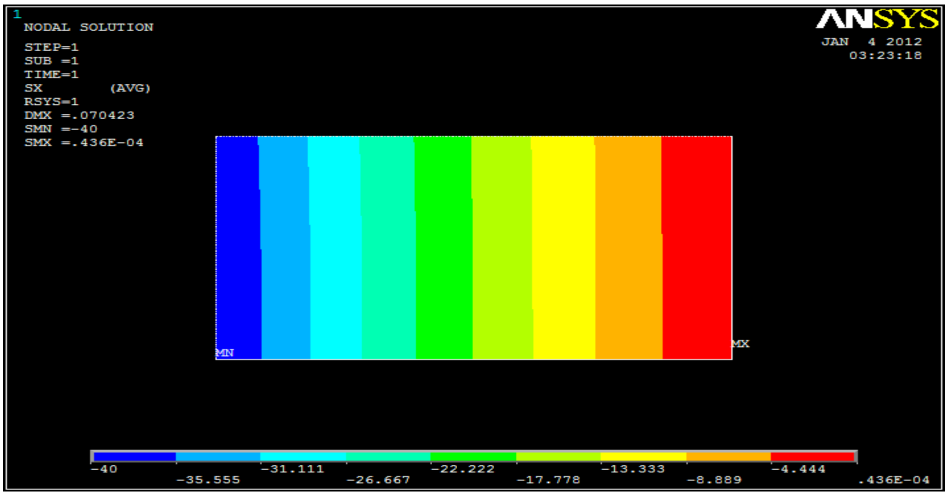

Figure 4.4: Radial stress, S_x of steel expandable tubular.

Figure 4.5 shows the hoop or circumferential stress, S_z where it is 266.509 MPa on the interior of the tubing and 226.509 MPa on the exterior. Theoretically, by applying Eq.n 2.4, the hoop stress for the interior of the tubing is 266.509 MPa and 226.509 MPa for the exterior of the tubing which are totally similar with the simulation values.

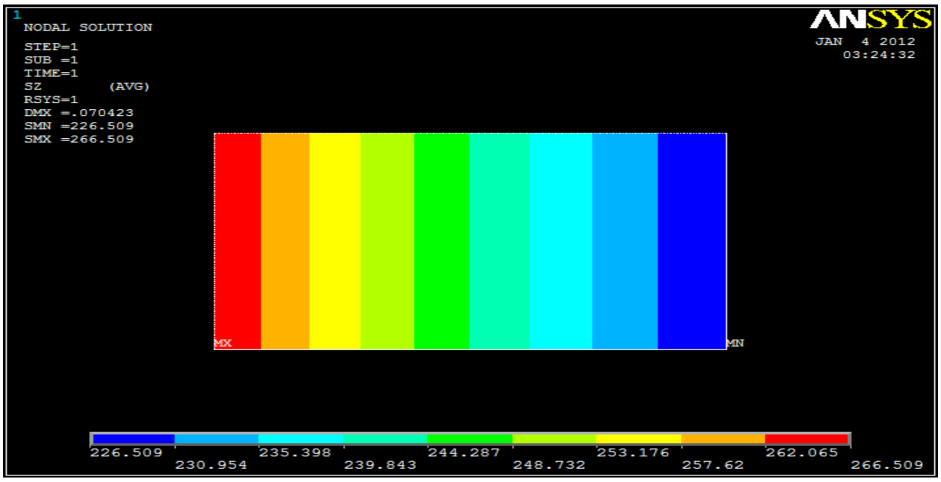

Figure 4.5: Hoop stress, S_z of steel expandable tubular.

Figure 4.6 shows the axial stress, S_y where it is -0.4122 MPa on the interior of the tubing and 0.0000003 MPa on the exterior. In theory, the axial stress should be zero due to open-ended cylinder.

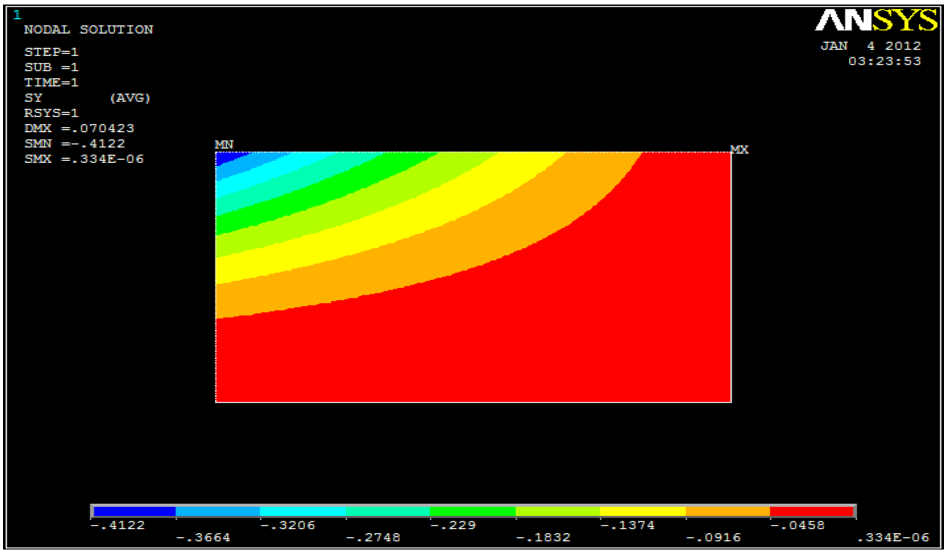

Figure 4.6: Axial stress, S_y of steel expandable tubular.

The theoretical values obtained give accurate validation for the ANSYS simulation values. The discrepancy between ANSYS's value with theoretical value is shown in Table 4.3.

4.1.2 Aluminium 2014-T6

Figure 4.7 shows the expansion process causes the diameter of the tubing increases while decreasing the thickness of the tubing. The inner radius displacement is 0.188108 mm and the outer radius displacement is 0.176622 mm which gives the final thickness of 7.988514 mm. The thickness decreased 0.011486 mm from its original which is 8 mm. Theoretically, by applying Eq. 2.6, the change in inner radius is 0.18803 mm and change in outer radius is 0.176622 mm.which gives the final thickness of 7.988592 mm. Note that the inner radius displacement is higher than the outer radius displacement due to the existence of the internal pressure only.

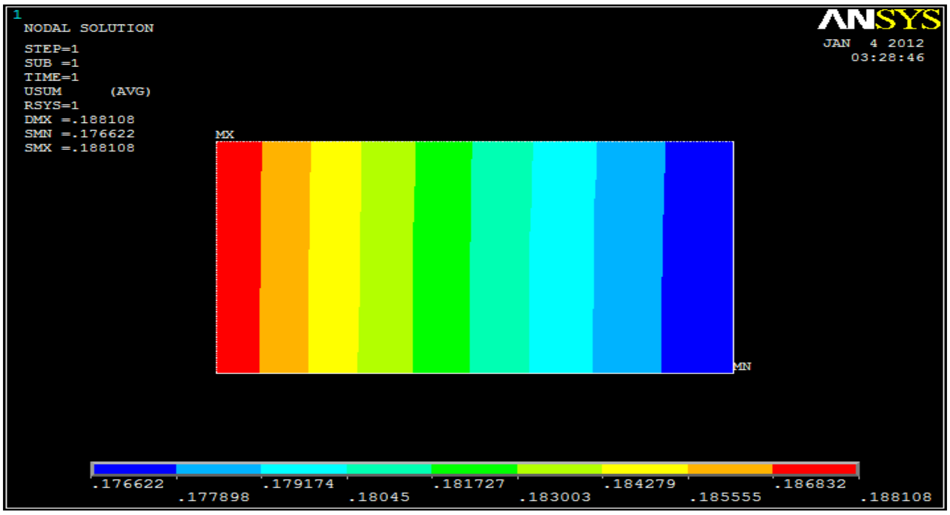

Figure 4.7: Change in radius of aluminium expandable tubular.

Figure 4.8 shows the radial stress, S_x is equal to the internal pressure applied which is -40 MPa on the interior of the tubing and 0.0000436 MPa on the exterior. Theoretically, by applying Eq. 2.3, the radial stress for the interior of the tubing is -40 MPa and 0 MPa for the exterior of the tubing.

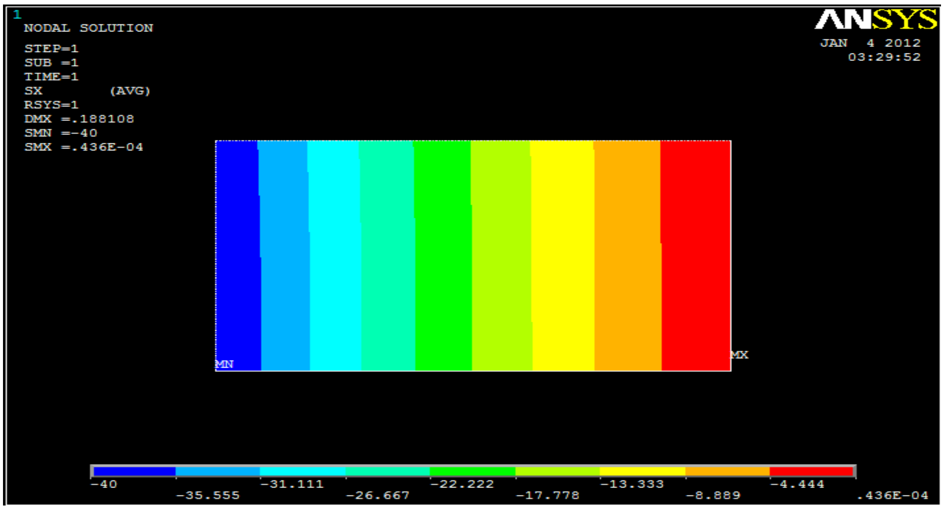

Figure 4.8: Radial stress, S_x of aluminium expandable tubular.

Figure 4.9 shows the hoop or circumferential stress, S_z where it is 266.509 MPa on the interior of the tubing and 226.509 MPa on the exterior. Theoretically, by applying Eq.n 2.4, the hoop stress for the interior of the tubing is 266.509 MPa and 226.509 MPa for the exterior of the tubing which are totally similar with the simulation values

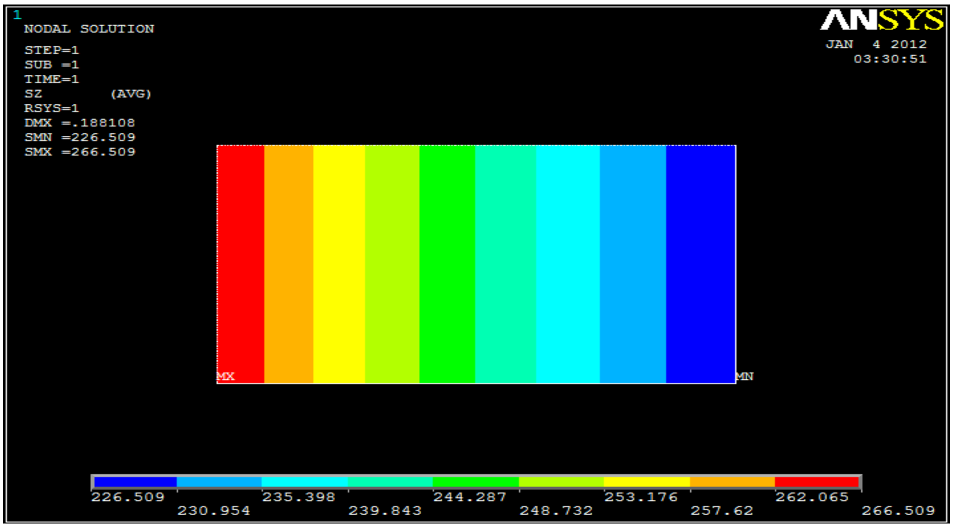

Figure 4.9: Hoop stress, S_z of aluminium expandable tubular.

Figure 4.10 shows the axial stress, S_y where it is -0.4122 MPa on the interior of the tubing and 0.0000003 MPa on the exterior. In theory, the axial stress should be zero due to open-ended cylinder.

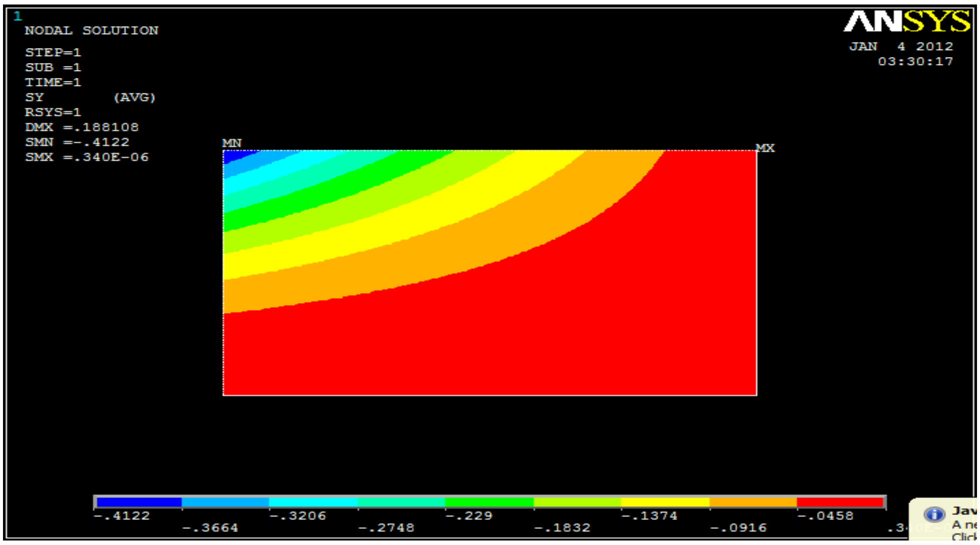

Figure 4.10: Axial stress, S_y of aluminium expandable tubular.

The theoretical values obtained give accurate validation for the ANSYS simulation values. The discrepancy between ANSYS's value with theoretical value is shown in Table 4.3.

4.1.3 Titanium (Ti-6Al-4v)

Figure 4.11 shows the expansion process causes the diameter of the tubing increases while decreasing the thickness of the tubing. The inner radius displacement is 0.114755 mm and the outer radius displacement is 0.107592 mm which gives the final thickness of 7.992837 mm. The thickness decreased 0.007163 mm from its original which is 8 mm. Theoretically, by applying Eq. 2.6, the change in inner radius is 0.114705 mm and change in outer radius is 0.107592 mm.which gives the final thickness of 7.992887 mm. Note that the inner radius displacement is higher than the outer radius displacement due to the existence of the internal pressure only.

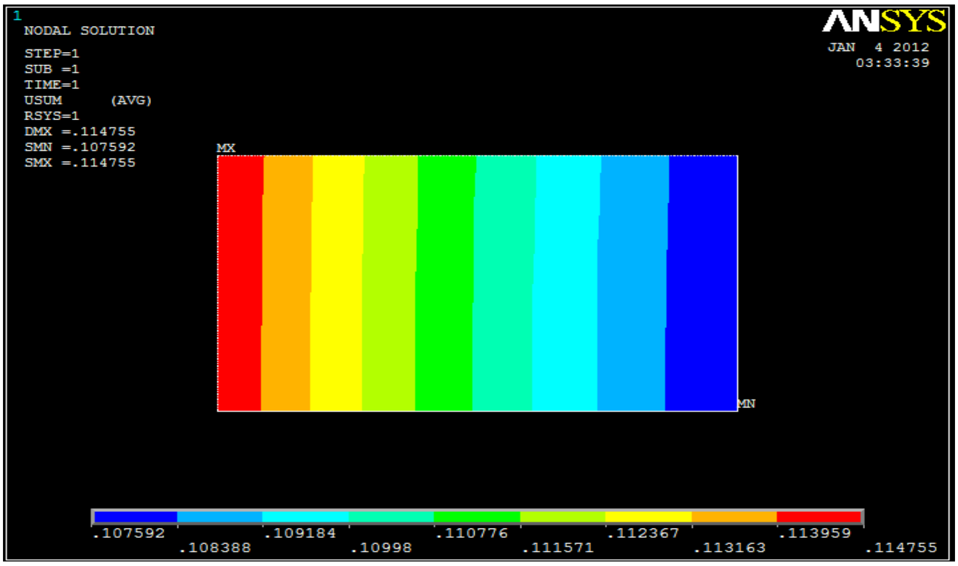

Figure 4.11: Change in radius of titanium expandable tubular.

Figure 4.12 shows the radial stress, S_x is equal to the internal pressure applied which is -40 MPa on the interior of the tubing and 0.0000436 MPa on the exterior. Theoretically, by applying Eq. 2.3, the radial stress for the interior of the tubing is -40 MPa and 0 MPa for the exterior of the tubing.

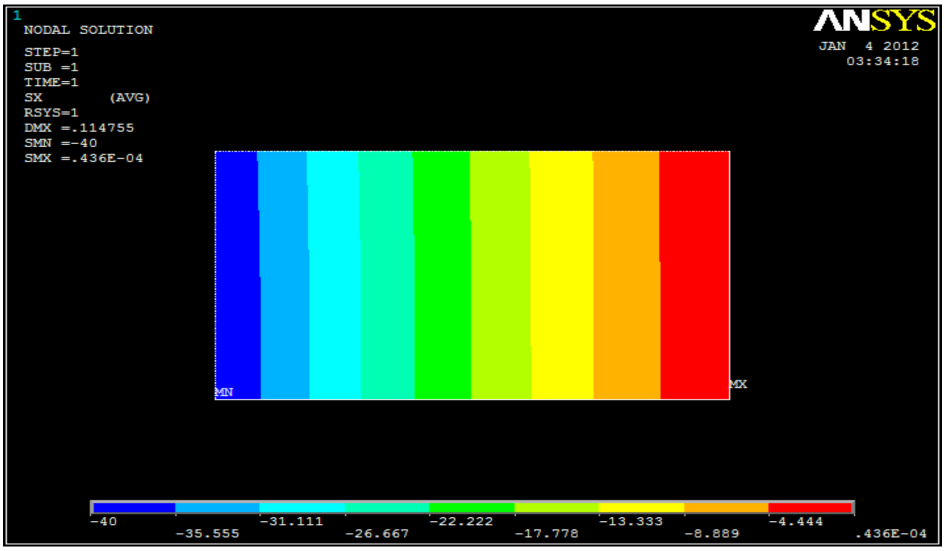

Figure 4.12: Radial stress, S_x of titanium expandable tubular.

Figure 4.13 shows the hoop or circumferential stress, S_z where it is 266.509 MPa on the interior of the tubing and 226.509 MPa on the exterior. Theoretically, by applying Eq.n 2.4, the hoop stress for the interior of the tubing is 266.509 MPa and 226.509 MPa for the exterior of the tubing which are totally similar with the simulation values

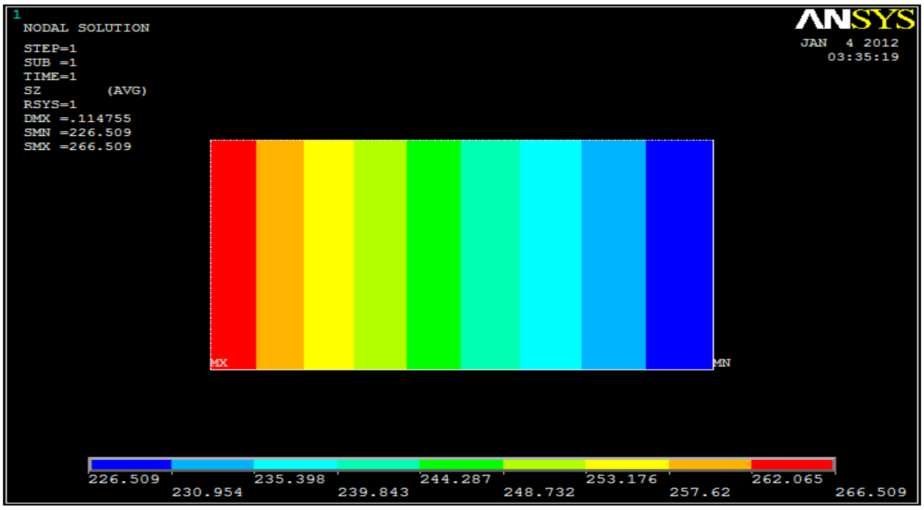

Figure 4.13: Hoop stress, S_z of titanium expandable tubular.

Figure 4.14 shows the axial stress, S_y where it is -0.4122 MPa on the interior of the tubing and 0.0000003 MPa on the exterior. In theory, the axial stress should be zero due to open-ended cylinder.

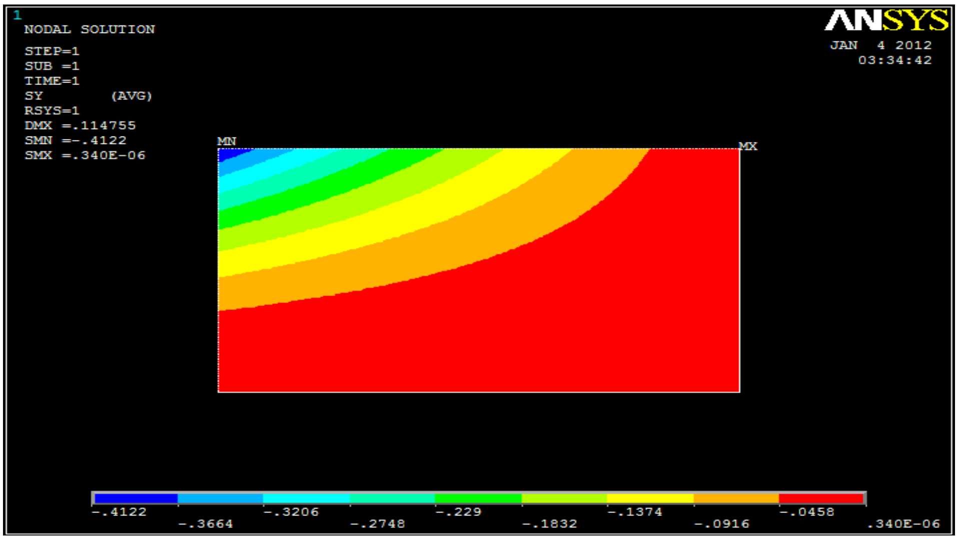

Figure 4.14: Axial stress, S_y of titanium expandable tubular.

The theoretical values obtained give accurate validation for the ANSYS simulation values. The discrepancy between ANSYS's value with theoretical value is shown in Table 4.3.

4.1.4 Validation of Linear Finite Element Analysis with Analytical Solution

The summary of the results obtained from the linear finite element analysis and analytical solution using The Lame's Equations are showed in Table 4.1 and Table 4.2, respectively.

Table 4.1: Results obtained from linear finite element analysis using ANSYS.

Material	Aluminium 2014-T6	Stainless Steel 304	Titanium (Ti-6Al-4v)
Internal Pressure (MPa)	40	40	40
Inner Radial Stress (MPa)	-40	-40	-40
Outer Radial Stress (MPa)	0.00004	0.00004	0.00004
Inner Axial Stress (MPa)	-0.4122	-0.4122	-0.4122
Outer Axial Stress (MPa)	0.0000003	0.0000003	0.0000003
Inner Hoop Stress (MPa)	266.509	266.509	266.509
Outer Hoop Stress (MPa)	226.509	226.509	226.509
Change in inner radius (mm)	0.188108	0.070423	0.114755
Change in outer radius (mm)	0.176622	0.066897	0.107592
Change in thickness (mm)	0.011486	0.003526	0.007163
Final thickness (mm)	7.988514	7.996474	7.992837

Table 4.2: Results obtained from analytical solution using the Lame's Equations.

Material	Aluminium 2014-T6	Stainless Steel 304	Titanium (Ti-6Al-4v)
Internal Pressure (MPa)	40	40	40
Inner Radial Stress (MPa)	-40	-40	-40
Outer Radial Stress (MPa)	0	0	0
Inner Axial Stress (MPa)	0	0	0
Outer Axial Stress (MPa)	0	0	0
Inner Hoop Stress (MPa)	266.509	266.509	266.509
Outer Hoop Stress (MPa)	226.509	226.509	226.509
Change in inner radius (mm)	0.18803	0.070405	0.114705
Change in outer radius (mm)	0.176622	0.066897	0.107592
Change in thickness (mm)	0.011408	0.003508	0.007113
Final thickness (mm)	7.988592	7.996492	7.992887

Both of the results obtained are compared for error estimation or discrepancy. The inner radial stress, inner and outer hoop stress, change in radius, change in thickness and final thickness are taken for comparison because the other properties have similar values for finite element analysis and theoretical.

Table 4.3: Comparison between linear analysis with theoretical values

(in %).

Material	Aluminium 2014-T6	Stainless Steel 304	Titanium (Ti-6Al-4v)
Inner Radial Stress (MPa)	0	0	0
Inner Hoop Stress (MPa)	0	0	0
Outer Hoop Stress (MPa)	0	0	0
Change in inner radius (mm)	0.04	0.03	0.04
Change in outer radius (mm)	0	0	0
Change in thickness (mm)	0.7	0.5	0.7
Final thickness (mm)	0.001	0.0002	0.001

The largest discrepancy is 0.7%. This number reflects the accuracy of finite element analysis which agrees greatly with the theoretical value because the acceptable error level would be 10%. The error can be reduced even more by using smaller element size when meshing. The low error percentage justifies the reliability of the simulations.

4.2 Non-linear Finite Element Analysis of Stress Distribution for Thick Wall Cylinder

Based on strong agreement between linear finite element analysis with theoretical values, the non-linear analysis was done confidently.

4.2.1 Stainless Steel 304 with 40 MPa Internal Pressure

As mentioned in Chapter 3, internal pressure of 40 MPa is applied in order to obtain stress formation that exceeds the material's yield strength so that it will exhibit non-linear behaviour. Figure 4.15 shows the bilinear stress-strain curve for Stainless Steel 304 after computing the value of tangent modulus and yield strength from Tables 3.3 and 3.4. The curve is the simplification of the typical stress-strain curve.

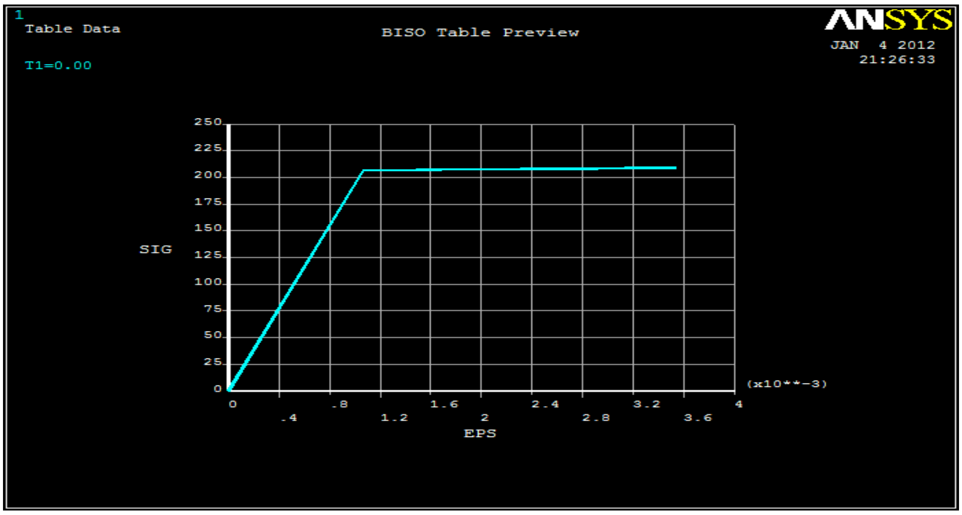

Figure 4.15: Steel bilinear stress-strain curve.

Figure 4.16 shows the displacement of the tubing wall. The inner radius displacement is 3.298 mm while the outer radius displacement is 3.03 mm. Note that the value is significantly different with the theoretical value for linear analysis. This shows that the thickness of the tubing is greatly reduced once the material started to yield.

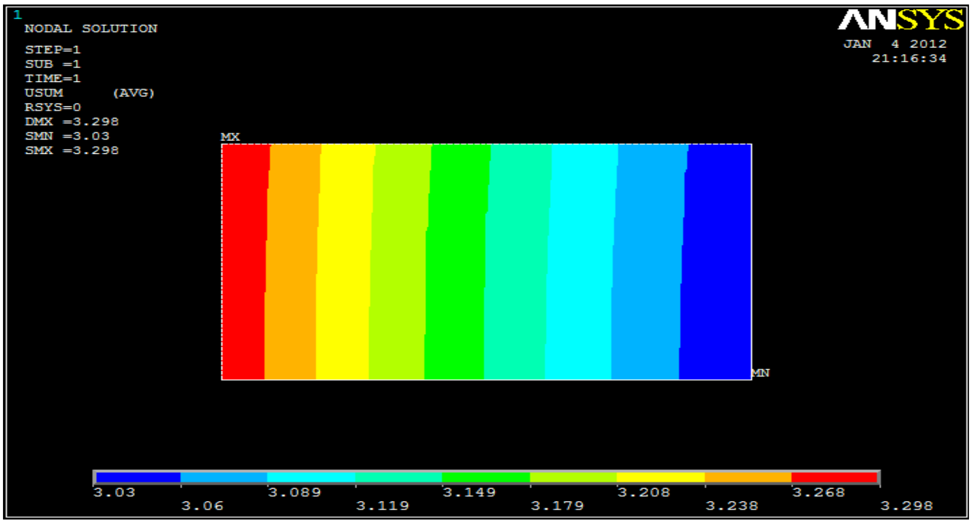

Figure 4.16: Change in radius of steel expandable tubular (non-linear).

Figure 4.17 shows the radial stress, S_x is slightly different with the internal pressure applied which is -39.88 MPa on the interior of the tubing and -0.092519 MPa on the exterior.

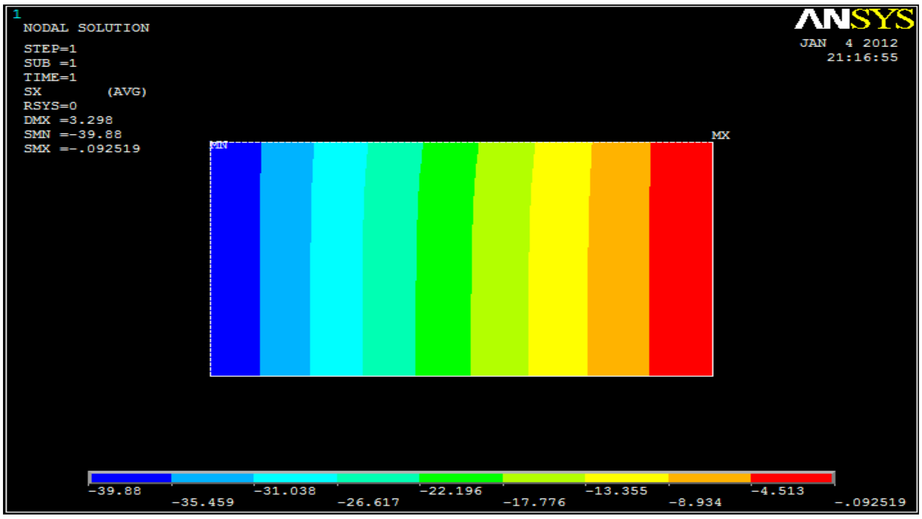

Figure 4.17: Radial stress, S_x of steel expandable tubular (non-linear).

Figure 4.18 shows the hoop or circumferential stress, S_z where it is 239.333 MPa on the interior of the tubing and 249.77 MPa on the exterior.

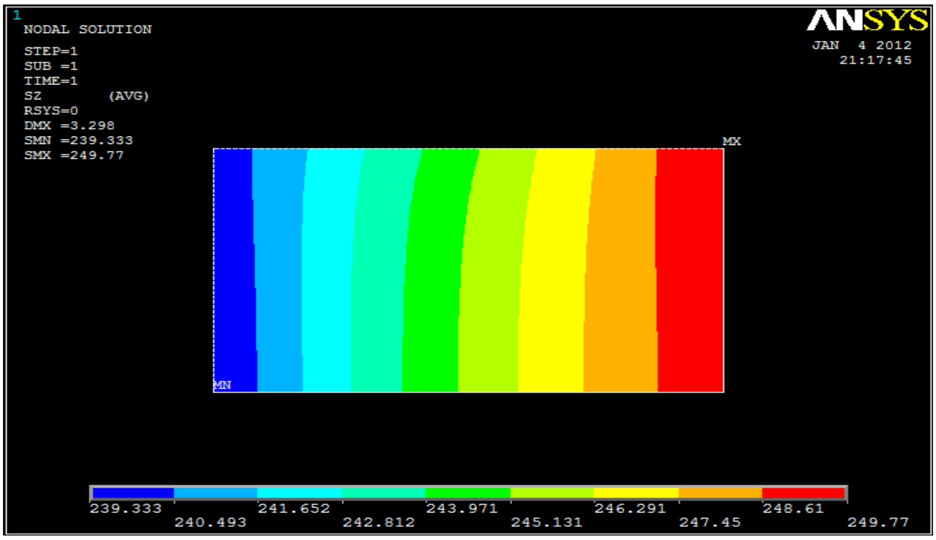

Figure 4.18: Hoop stress, S_z of steel expandable tubular (non-linear).

Figure 4.19 shows the axial stress, S_y where it is -0.643914 MPa on the interior of the tubing and 0.246913 MPa on the exterior. In theory, the axial stress should be zero due to open-ended cylinder.

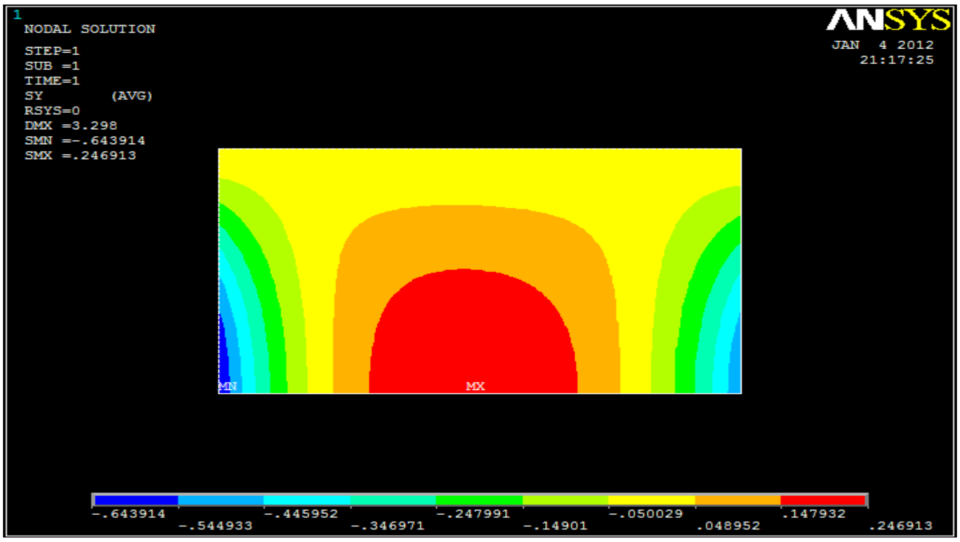

Figure 4.19: Axial stress, S_y of steel expandable tubular (non-linear).

The theoretical values obtained from the Lame's Equation are compared with the non-linear simulations value. The discrepancy between ANSYS's value with theoretical value is shown in Table 4.6.

4.2.2 Aluminium 2014-T6 with 65 MPa Internal Pressure

As mentioned in Chapter 3, internal pressure of 65 MPa is applied in order to obtain stress formation that exceeds the material's yield strength so that it will exhibit non-linear behaviour. Figure 4.20 shows the bilinear stress-strain curve for Aluminium 2014-T6 after computing the value of tangent modulus and yield strength from Tables 3.3 and 3.4. The curve is the simplification of the typical stress-strain curve.

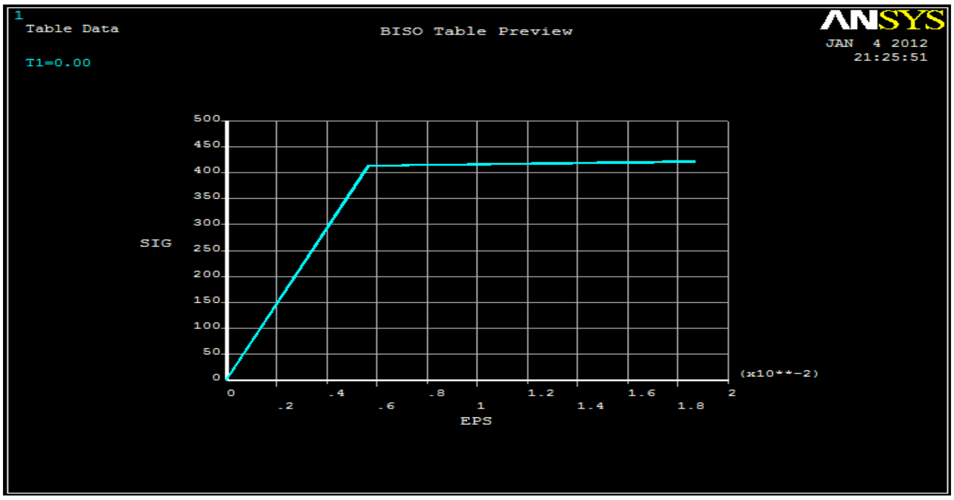

Figure 4.20: Aluminium bilinear stress-strain curve.

Figure 4.21 shows the displacement of the tubing wall. The inner radius displacement is 0.399455 mm while the outer radius displacement is 0.373066 mm. This shows that the thickness of the tubing is greatly reduced once the material started to yield.

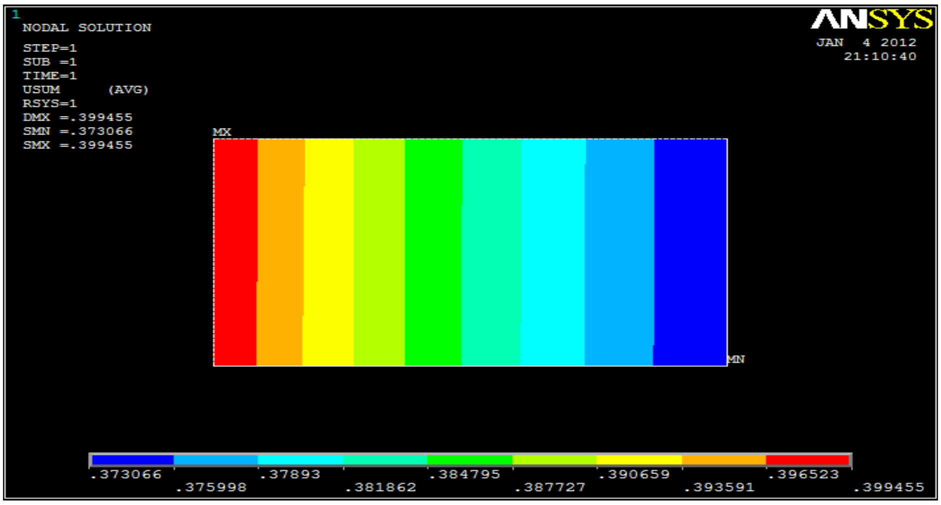

Figure 4.21: Change in radius of aluminium expandable tubular (non-linear).

Figure 4.22 shows the radial stress, S_x is slightly different with the internal pressure applied which is -64.809 MPa on the interior of the tubing and -0.152453 MPa on the exterior.

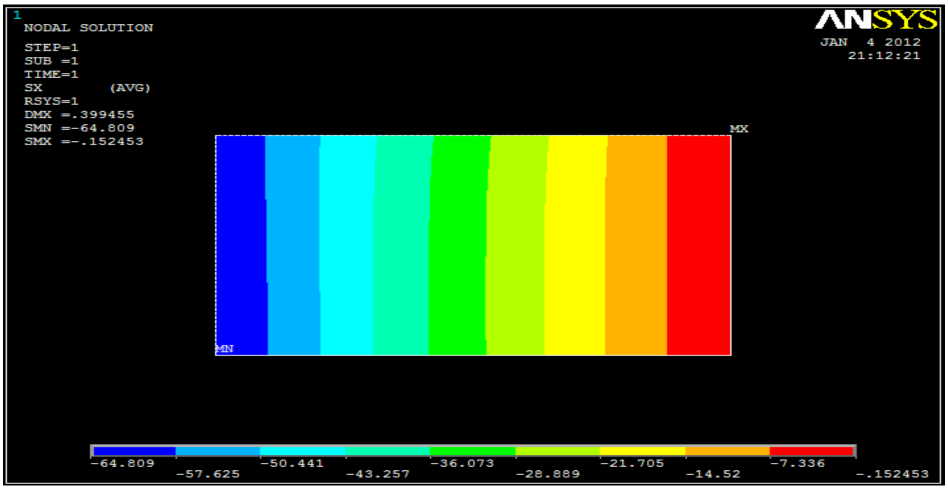

Figure 4.22: Radial stress, S_x of aluminium expandable tubular (non-linear).

Figure 4.23 shows the hoop or circumferential stress, S_z where it is 378.947 MPa on the interior of the tubing and 414.432 MPa on the exterior.

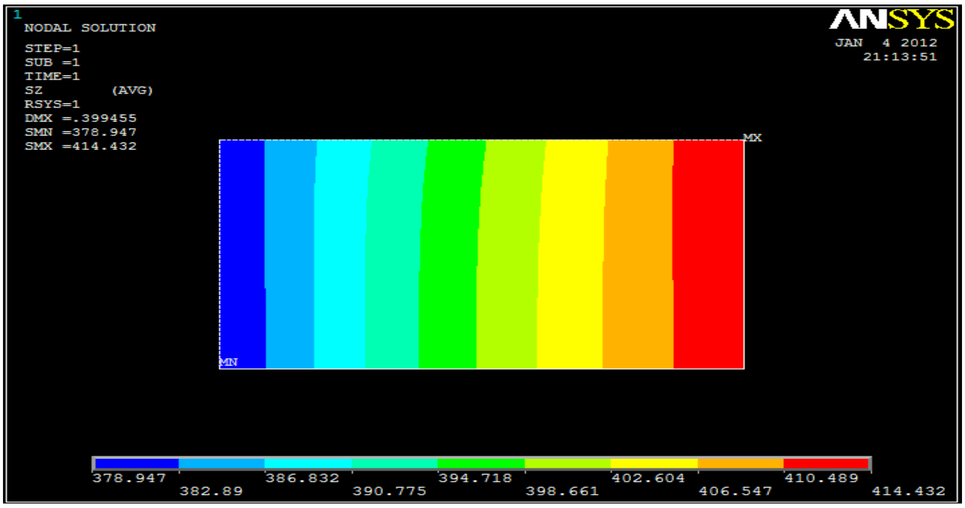

Figure 4.23: Hoop stress, S_z of aluminium expandable tubular (non-linear).

Figure 4.24 shows the axial stress, S_y where it is -1.027 MPa on the interior of the tubing and 0.379805 MPa on the exterior. In theory, the axial stress should be zero due to open-ended cylinder.

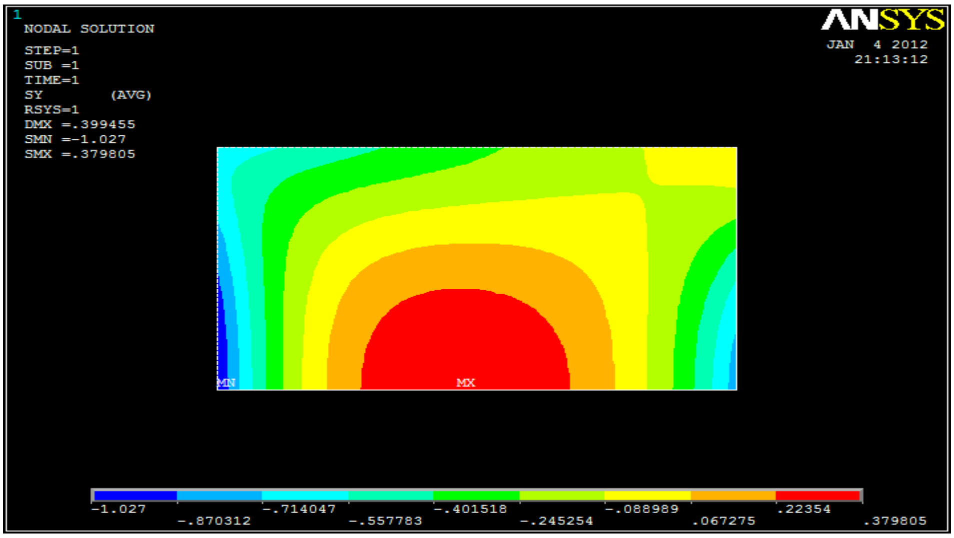

Figure 4.24: Axial stress, S_y of aluminium expandable tubular (non-linear).

The theoretical values obtained from the Lame's Equation are compared with the non-linear simulations value. The discrepancy between ANSYS's value with theoretical value is shown in Table 4.6.

4.2.3 Titanium (Ti-6Al-4v) with 140 MPa Internal Pressure

As mentioned in Chapter 3, internal pressure of 140 MPa is applied in order to obtain stress formation that exceeds the material's yield strength so that it will exhibit non-linear behaviour. Figure 4.25 shows the bilinear stress-strain curve for Titanium (Ti-6Al-4v) after computing the value of tangent modulus and yield strength from Tables 3.3 and 3.4. The curve is the simplification of the typical stress-strain curve.

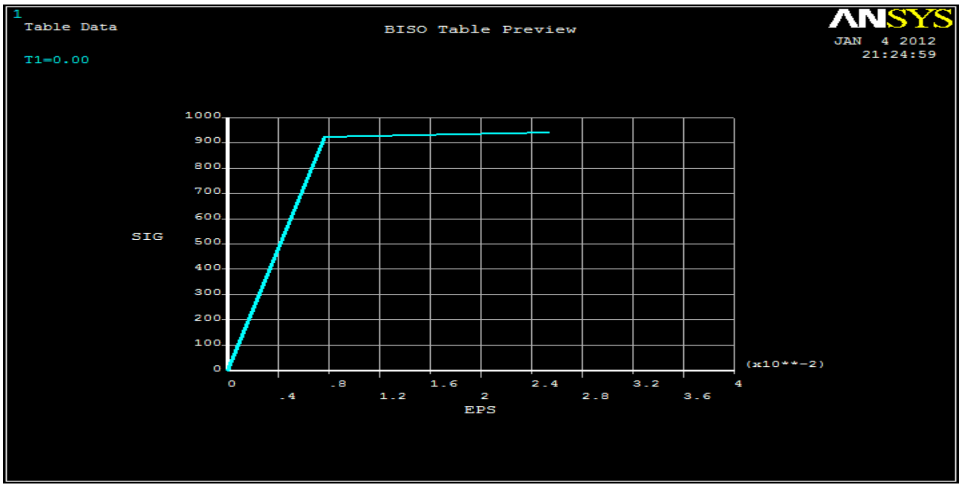

Figure 4.25: Titanium bilinear stress-strain curve.

Figure 4.26 shows the displacement of the tubing wall. The inner radius displacement is 0.413373 mm while the outer radius displacement is 0.387312 mm. This shows that the thickness of the tubing is greatly reduced once the material started to yield.

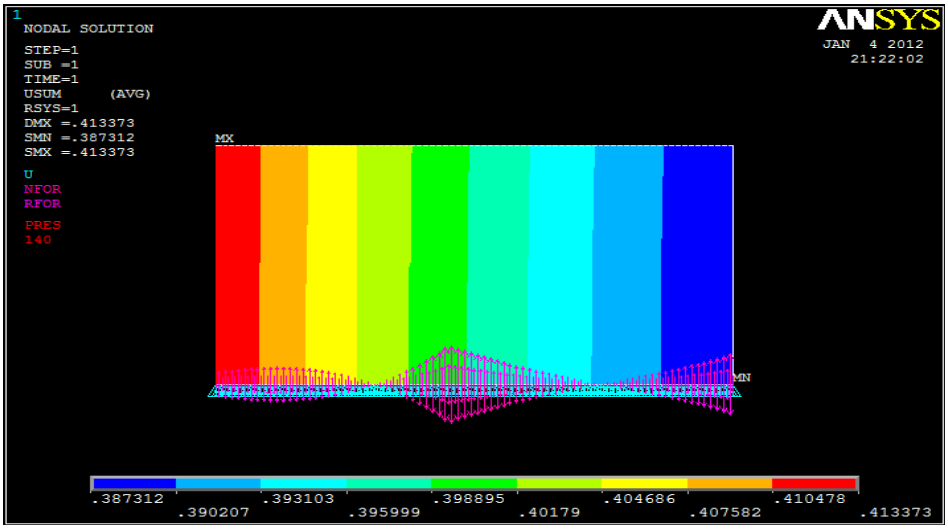

Figure 4.26: Change in radius of titanium expandable tubular (non-linear).

Figure 4.27 shows the radial stress, S_x is slightly different with the internal pressure applied which is -139.574 MPa on the interior of the tubing and 0.001932 MPa on the exterior.

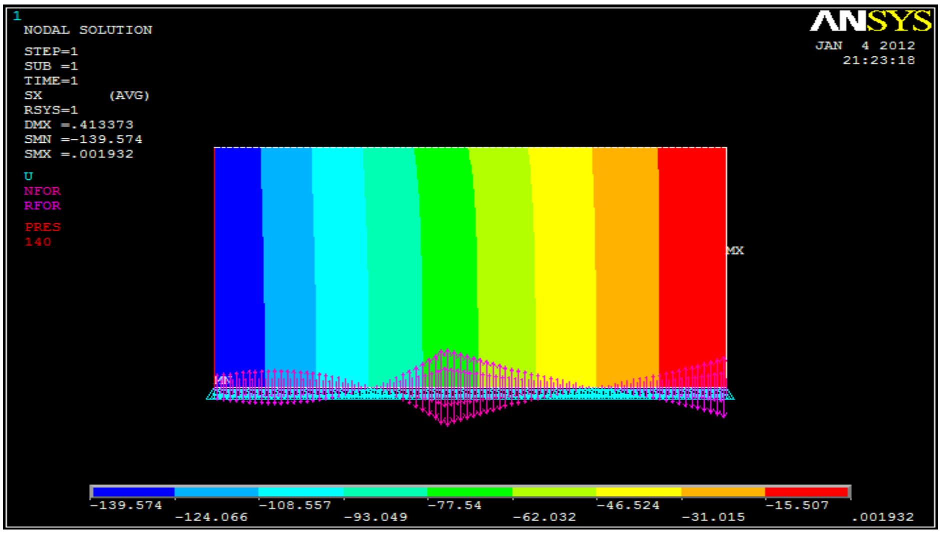

Figure 4.27: Radial stress, S_x of titanium expandable tubular (non-linear).

Figure 4.28 shows the hoop or circumferential stress, S_z where it is 847.264 MPa on the interior of the tubing and 815.297 MPa on the exterior.

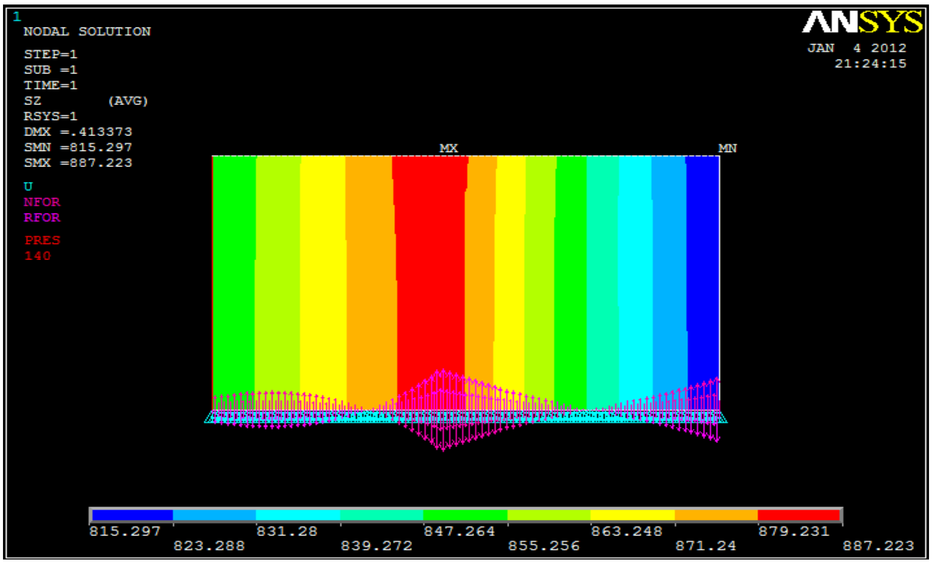

Figure 4.28: Hoop stress, S_z of titanium expandable tubular (non-linear).

Figure 4.29 shows the axial stress, S_y where it is -1.537 MPa on the interior of the tubing and -1.144 MPa on the exterior. In theory, the axial stress should be zero due to open-ended cylinder.

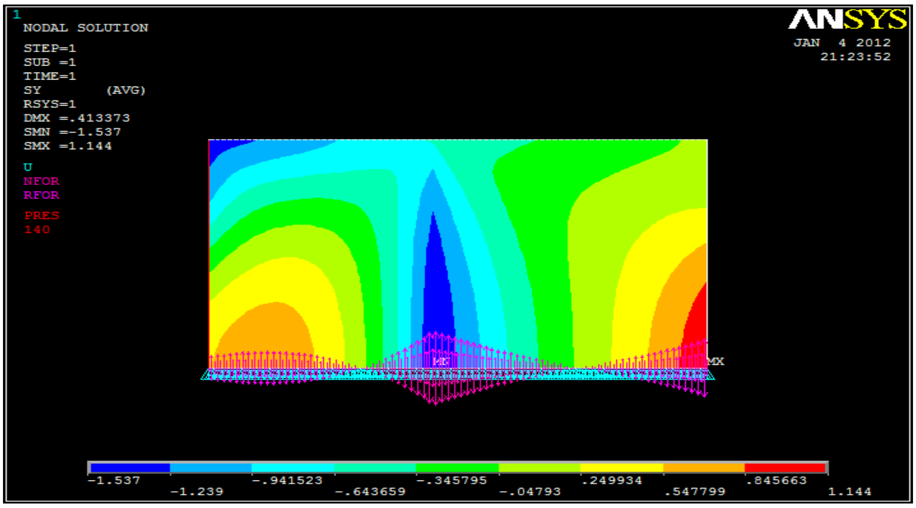

Figure 4.29: Axial stress, S_y of titanium expandable tubular (non-linear).

The theoretical values obtained from the Lame's Equation are compared with the non-linear simulations value. The discrepancy between ANSYS's value with theoretical value is shown in Table 4.6.

4.2.4 Validation of Non-linear Finite Element Analysis with Analytical Solution

The summary of the results obtained from the non-linear finite element analysis and analytical solution using The Lame's Equations are showed in Table 4.4 and Table 4.5, respectively.

Table 4.4: Results obtained from non-linear finite element analysis using ANSYS.

Material	**Aluminium 2014-T6**	**Stainless Steel 304**	**Titanium (Ti-6Al-4v)**
Internal Pressure (MPa)	65	40	140
Inner Radial Stress (MPa)	-64.809	-39.880	-139.574
Outer Radial Stress (MPa)	-0.152	-0.093	0.002
Inner Axial Stress (MPa)	-1.027	-0.644	-1.537
Outer Axial Stress (MPa)	0.380	0.247	-1.144
Inner Hoop Stress (MPa)	378.947	239.333	847.264
Outer Hoop Stress (MPa)	414.432	249.770	815.297
Change in inner radius (mm)	0.399455	3.298	0.413373
Change in outer radius (mm)	0.373066	3.03	0.387312
Change in thickness (mm)	0.026389	0.268	0.026061
Final thickness (mm)	7.973611	7.732	7.973939

Table 4.5: Results obtained from analytical solution using the Lame's Equations.

Material	**Aluminium 2014-T6**	**Stainless Steel 304**	**Titanium (Ti-6Al-4v)**
Internal Pressure (MPa)	65	40	140
Inner Radial Stress (MPa)	-65	-40	-140
Outer Radial Stress (MPa)	0	0	0
Inner Axial Stress (MPa)	0	0	0
Outer Axial Stress (MPa)	0	0	0
Inner Hoop Stress (MPa)	433.078	266.509	932.783
Outer Hoop Stress (MPa)	368.078	226.509	792.783
Change in inner radius (mm)	0.305548	0.070405	0.401466
Change in outer radius (mm)	0.28701	0.066897	0.376571
Change in thickness (mm)	0.018538	0.003508	0.024895
Final thickness (mm)	7.981462	7.996492	7.975105

Both of the results obtained are compared for error estimation or discrepancy. The inner radial stress, inner and outer hoop stress, change in radius, change in thickness and final thickness are taken for comparison because the other properties have similar values for finite element analysis and theoretical.

Table 4.6: Comparison between non-linear analysis with theoretical values (in %).

Material	**Aluminium 2014-T6**	**Stainless Steel 304**	**Titanium (Ti-6Al-4v)**
Inner Radial Stress (MPa)	0.3	0.3	0.3
Inner Hoop Stress (MPa)	12	10	9
Outer Hoop Stress (MPa)	3	10	3
Change in inner radius (mm)	30.73	4584.33	2.97
Change in outer radius (mm)	30	4429	3
Change in thickness (mm)	42.4	7539.7	4.7
Final thickness (mm)	0.10	3.31	0.01

Table 4.7 shows that the Lame's Equation can't validate the non-linear values with percentage error as big as 7539.7%. This proves that the Lame's Equation is only valid for linear behaviour.

Based on linear and non-linear analysis of the three selected materials, Stainless Steel 304 is found to be the most suitable because it only needs low internal pressure to expand compared to the other two materials. Although maybe in extreme condition, titanium would be the best option due to its high ultimate tensile strength. The decision must be based on the situation. The most common casing material is steel because it is easily formed. When exposed to environment, steel will build up a protective oxide layer which protect it under corrosive conditions and eventually last longer in service life. Under more extreme conditions, steel with special chemistries or alloys such as the stainless steel grades, are available fore greater life and protection. Steel possesses the high yield and tensile strengths required for water well use. Casings may be subjected to underground external forces after installation. Eartquakes or subterranean earth movements tend to displace them. Another important quality of steel is its weldability. This facilitates proper field installation.

The linear simulation results and calculation using Lame's equations proves that the diameter of the tubing increases while the thickness decreases during expansion process. These results satisfy the theory and application in oil well drilling and production. The linear finite element analysis accuracy is very good with the largest error percentage of 0.7% to the calculated values. Based on this, the non-linear analysis can be done confidently by taking the linear analysis as the base. It is also found that the Lame's Equation is only valid for linear analysis. Various sources of error during the simulation can contribute to inaccurate results such as wrong geometry and inappropriate meshing. Due to these errors, the post processing stage of finite element analysis might show unacceptable deformed shape and the percentage error will be very high. This will cause the results to be invalid.

Finite element analysis is considered as predictions of results based on known assumption though not actual. Badly shaped elements can give less accurate results than required. A mesh should have well-shaped elements and the size must be fine enough to give good detail. Another important step is to put in high consideration in applying boundary condition correctly. Although the simulation can run without error, wrongly applied boundary condition will result in error in the end of the simulation. The actual boundary condition must be studied thoroughly before applying it to the simulation.

The analysis was successfully done after all the problems were recognized and solved. Besides validating theoretically, the finite element analysis result can also be verified through experimental work. But experimental work will be very expensive and time consuming which would be a great disadvantage in a highly demanding oil and gas business. Furthermore, the approach of axisymmetric analysis had been proven to cut simulation time significantly. The success of this project shows the reliability of finite element analysis for stress analysis in oil and gas field.

CHAPTER 5

CONCLUSIONS AND RECOMMENDATIONS

5.1 Conclusions

The project's objectives were met successfully. The axisymmetric modeling of the solid expandable tubular was correctly build with all the boundary conditions applied accordingly. Linear analysis was first done and validated with analytical solutions before running the non-linear analysis. The largest percentage error for linear analysis is 0.7% which proves the finite element analysis agree really well with the theoretical values. This was taken as a basis for the non-linear analysis. Although there is no good justification for the non-linear analysis, the bilinear stress-strain curve produced for each materials can be accepted as the correct input for the simulations. The simulation results and calculation shows that the diameter of the tubing increase while the thickness of the tubing decrease. But the amount of displacement is different for the three selected materials where Aluminium 2014-T6 has the highest change in thickness and Stainless Steel 304 with the lowest change in thickness. The material properties do affect the change in radius of the tubing during expansion process. This project can provide guide for solid expandable tubular operation and much can be done to improve this project.

5.1 Recommendations

The stress analysis can be further improved by taking into account other factor such as friction in the analysis to gain more accurate result. Different diameters and design pressure of SET should be studied to see the effect on stress distribution. More materials should be selected to study their capability in well drilling operation which can widen the options when manufacturing casing or pipe. Analysis on non-axisymmetric and non-uniform expansion tubing should be done in the future for more complicated operation.

REFERENCES

1. Binggui, X., Yanping, Z., Hui, W., Hongwei, Y. And Tao, Jia., 2009, "Application of numerical simulation in the solid expandable tubular repair for casing damaged wells,", *Petrol. Explor. Develop*. **36 (5)**: 651-657.
2. Xucheng, W. and Min, S., 2002, "Finite element method and the basic principles of numerical methods," Beijing; Thinghua University Press, 183-184.
3. ANSYS, Inc. Theory Reference and Verification Manual, ansysinfo@ansys.com.
4. Changwu, H., 2007, "Future drilling technology", *Petrol. Explor. Develop.* **34 (6)**: 723.
5. Becker, A.A., 2004, *An Introductory Guide to Finite Element Analysis,* Series Editor Roger C. Baker. Professional Engineering Publishing.
6. Owoeye, O.O., Aihevba, L.O. and Hartmann, R. A., 2000, "Optimization of well economics by application of expandable tubular technology," SPE 59142.
7. Byrom, T.G., 2007, "Casing and Liners for Drilling and Completion," *Gulf Publishing Company*.
8. Dupal, K.K., 2001, "Solid expandable tubing Technology – A Year of Case Histories in the Drilling environment." IADC/SPE 67770, 1-16.
9. Tubes, D., 2006, "Slimming the Wellbore Design Enhances Drilling Economics in Field Development," SPE 102929.
10. Denny, D. 2004, "Solid expandable tubing for Constructing Extended-Reach Well," SPE.
11. Itoua-Konga, F., 2007, "Solid expandable tubing Enable Practical Wells Re-entry in Mature Fields," OTC 18647.
12. Rivenbark, M. and Abouelnaaj, K., 2006, "Solid expandable tubing Facilitate Intelligent-Well Technology Application in Existing Multilateral Wells," SPE 102934.
13. Rivenbark, M., 2005, "Real World Examples of Solid Expandable Casing Applications," AADE-05-NTCE-17.
14. Anonymous, 17th August 2011, http://www.en.wikipedia.org/wiki/Stress_(mechanic)/.

15. Ansel C. Ugural and Saul K. Fenstev, 2003, *Advance Strength and Applied Elasticity 4th Edition*, Prentice Hall.
16. Anonymous, 25th December 2011, <http://mpdc.mae.cornell.edu/>.
17. Anonymous, 16th December 2011, <http://www.xansys.org/forum/viewtopic.php?t=20805/>.