CHAPTER 1

INTRODUCTION

1.1 Background of Study

Residual stresses are a type of stress that remains after the original cause of the load (external force/thermal gradient) has been removed. The stress remain along the cross sectional area of the structure, even without the external cause. Residual stresses occur due to many reasons. It can possibly occur due to inelastic deformation and heat treatment. Heat comes from the welding arc may cause localized expansion at the region of weld, which is taken up by either the molten metal or the placement of parts being welded. After the welding process end, some area was affected by the heat, called heat affected zone (HAZ) experience uneven cooling compared to other region, creating a different phase transition that leaves residual stresses.

Residual stresses are known to influence a material's mechanical properties such as creep or fatigue life. Sometimes, the effect is very deleterious. Therefore, it is very important to be able to monitor and control the residual stresses. Almost no technical materials, components or structures are available completely free of residual stresses. Consequently, if residual stresses distributions are not quantitatively known, considerable uncertainty exist is designing components correctly.

Residual stresses are more difficult to predict than the in-service stresses on which they may come together with. These include the axial stress, hoop stress and radial stress that may come up along with residual component. The simulation on behaviour of the residual stresses will be done using ANSYS software and the personnel involved will observe the trend of the stress concentration spot.

Heat affected zone (HAZ) is a region of the base material, which had its microstructure and properties altered due to the thermal load which include welding, heat treatment or intense cutting environment. The change in base material may

include the change of microstructure and phase transition after being applied with severe heat. The extent and magnitude of the effect is mainly affected by the thermal diffusivity that is dependent on type of materials. If the diffusivity is high, the cooling rate is high and creates smaller HAZ region, while low diffusivity lead to slower cooling result in a bigger HAZ region.

1.2 Problem Statement

There are many practical engineering problems for which the exact solution cannot be obtained. The incapability to obtain an exact solution may be attributed to either the complex nature of governing differential equation or the difficulties that arise from dealing with the boundary and initial conditions. Numerical solutions approximate exact solutions only at discrete points called nodes. The amount of residual stresses is hardly obtained after the welding process. This project is conducted to simulate the effect of welding process on Carbon Steel pipe to obtain the amount of the residual stresses. Since welding is widely used for pipeline connection for oil and gas industry, so it is very important to make sure that the process conducted properly to prevent any unwanted incident to happen. Welding process relies on an intensely localized heat input, which tends to generate undesired residual stress and deformations in welded structures. Since heat is applied, concerns on the formation of residual stress due to thermal load arise. Therefore, estimating the magnitude of welding deformations and characterizing the effects of the welding conditions are deemed necessary.

With modern computing facilities, the Finite Element Analysis (FEA) technique has become an effective method for prediction and assessment of welding residual stress and distortion. However, the welding deformations are various with production variations such as dimension, welding materials and welding process parameters. Therefore, rapidly and accurately predicting welding induced distortion for real engineering applications is more challenging. The pipeline for oil and gas transportation may be located on the seabed which exposed to the corrosive environment of saltwater or at the plant which exposed to personnel working in the plant and if there is any leakage can cause major impact to the personnel, company and also the environment. So, it is really necessary to perform analysis to simulate

the residual stress activity at the welding region especially on HAZ which is caused by thermal load.

1.3 Objective

- ▪ To determine the amount of residual stresses and its consequences at the Heat Affected Zone (HAZ) due to thermal load in Carbon Steel welded pipe.

1.4 Scope of Study

The analysis of residual stress due to thermal load will be on the carbon steel pipe. The heat variable from the welding process will be set at the melting point of the material and will be moved up for the next test. The type of welding selected is the Shield Metal Arc Welding (SMAW) which is widely used for pipeline welding connection. The simulation will be performing using UTP's ANSYS 11 software.

1.5 Relevancy of the Project

This project is regarding the Finite Element Analysis on the residual stress of welded connection between pipes. The software that being used is ANSYS software. The project can be done since this project is mostly on Engineering Material which studied about the mechanical properties of material and the course is taken by all Mechanical Engineering students. The knowledge during the study can be applied in this project to ensure that the project run smoothly and successfully.

Welds are often a fundamental part of engineering structures. Residual stress introduced in the welded regions, due to the nonlinear thermal processes during welding, can have harmful effects, such as stress corrosion cracking, hydrogen-induced cracking and reduced fatigue strength. Residual stresses can be responsible for altering the performance of their engineering components. It is therefore important to simulate the process of welding to predict the behaviour of welded structures from Finite Element Analysis. In this project, the FEA method is used to simulate the residual stresses in the weld region and HAZ on Carbon Steel welded pipe.

CHAPTER 2

LITERATURE REVIEW

Tubing made with High Strength Low Alloy steel (HSLA) is used to construct oil and gas pipelines for transporting pressurized hydrocarbons over long distances. The pipelines are subjected to high stresses due to internal fluid pressure, overlaying soil, surface traffic and environmental attack that can lead damage, especially under failed passive protective coating. The main causes of this damage are stress corrosion cracking, wall thickness reduction and the presence of stress concentrators [1]. Moreover, during the manufacturing process of the pipe, thermal and mechanical deformations always produce residual stresses. The generated residual stresses are usually very high, sometimes approaching the material yield strength, but their effects are not apparent until the structure is fully loaded or exposed to service environment [2, 3]. Tensile surface residual stresses are harmful as they increase the vulnerability of the component to fatigue damage, stress corrosion and even fracture. There has been a trend to improve the material mechanical properties in term of yield strength and corrosion resistance. In the last three decades, the steel yield strength has been increased from 250 MPa to more than 760 MPa [4, 5].

Since line pipe comes in spool, the proper connection between the spools is very important to prevent any bad incident in the future. The most convenient method is using welding. Welding is a fabrication or sculptural process that joins materials, usually metals or thermoplastics, by causing coalescence. Welding is preferred as pipe connection since the method is a very efficient joining process. There are a lot of advantages using welding method which are it is simple and easy, flexible and low cost, and also the efficiency is high. Although the method gives many advantages, but this method will cause nonlinear temperature distribution during the process. Initially, this temperature distribution caused a rapid thermal expansion and followed by the thermal contraction in the weld region and other area mainly HAZ. This generates plastic deformation and residual stresses in the welded area when it is cooled.

Welding process is an integral manufacturing procedure in the production of many engineering and structural components having a direct influence on the integrity of the components and their thermal and mechanical behaviour during service. Due to the high temperatures introduced during welding and the subsequent cooling of the welded metal, welding can produce undesirable residual stresses and deformations. Such stresses can be simulated for the process of welding to define the consequent of residual stresses and deformations and for use in the prediction of the behaviour of welded structures [6]. Figure 2.1 shows the distribution of tensile stress at the welded steel plate using the neutron and X-ray method. The peak tensile stress is located at the weld centre.
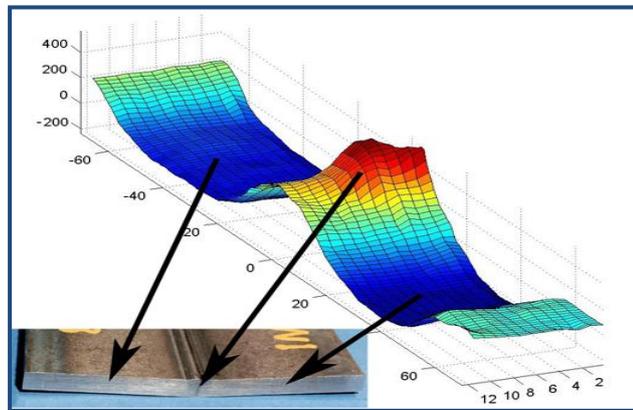
Figure 2.1 – Residual stress in a steel fusion weld measured using neutrons [14]

In this study, the concern of the residual stress is on Carbon Steel pipe which commonly used in oil and gas industry. This type of steel has high tensile and yield strength also the impact toughness. But, the weld crack easily formed in the HAZ and caused the possibility of brittle fracture in the welded zone when it is in service. HAZ is shown in the Figure 2.2. HAZ always has given close attention by many investigators. The welding joint of this steel also experience different weld thermal phase, which cause change of microstructure in the welded affected region, mainly in HAZ.
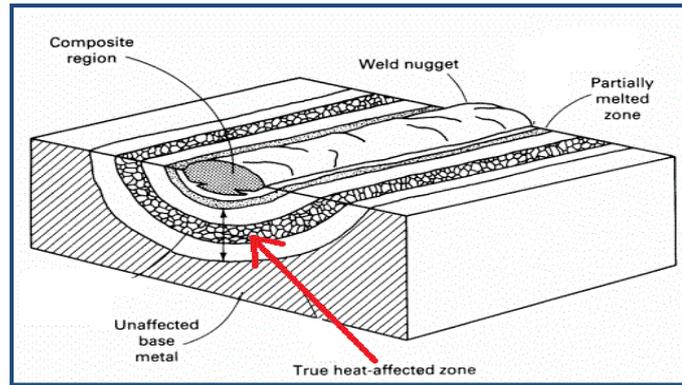
Figure 2.2 – HAZ on a welded area of steel plate [15]

The control on the weld heat input has been discussed in many researches. The weld heat input must be at a range where the molten metal is able to constantly dissipate the heat for cooling process without severe uneven cooling. By applying the heat input under the range of 20 kJ/cm, the metal microstructure will change in a ductile phase. If it is higher, the tendency of plastic distortion happening is high and the existence of brittle fracture could happen. It has been proved that the larger the weld heat input, the longer the cooling time and thus, it is easier for the deterioration of the metal and resulting in more severe residual stresses. Several experimental methods have been conducted to test and analyze the residual stresses and microstructure behaviour of the material. There are several non destructive technique and destructive technique for directly measuring the residual stresses which have been developed inside a specimen. These techniques include x-ray diffraction method, neuron diffraction method, layer-removal method, sectioning method, ultrasonic and magnetic method and also holes drilling method. However, the applications of these various experimental techniques are limited by either their cost or accuracy. Method like x-ray diffraction is very accurate but the method is not economical [7].

For this project, ANSYS software being used for the simulation and modelling and also to conduct the FEA. The finite element method is a numerical procedure that can be applied to obtain the solutions to a variety of problems in engineering. Steady, transient, linear, or nonlinear problems in stress analysis, heat transfer, fluid flow, and electromagnetism problems may be analyzed with FEA. ANSYS is a comprehensive general purpose finite element computer program that contains over

100,000 lines of code [8]. ANSYS has been a leading FEA program for well over 20 years. Finite Element weld simulation in principle consists of a thermal analysis, which represents the thermal process during welding result in revealing the temperature contours associated with welding, followed by a structural analysis which is based on the thermal findings. The structural analysis takes the temperature contours, made available by the thermal simulation, and uses them as input data to calculate a range of stress contours at the end of the analysis which remain in the modelled component as residual stresses [6]. To conduct this type of FEA, a sequentially coupled thermal-stress analysis is adopted since the stress or displacement solution is dependent on a temperature field with no inverse dependency [9].

Typical SMAW is shown in Figure 2.3. There are many important parameters that needed to be looked after to obtain good welding result. Top priority in doing welding process would be the safety issue. Safety equipments must be worn during the welding process [10]. SMAW is one of the world's most popular welding processes, accounting for over half of all welding in some countries. Because of its versatility and simplicity, it is particularly dominant in the maintenance and repair industry and, and is heavily used in the construction of steel structures and in industrial fabrication [11]. SMAW is often used to weld carbon steel , low and high alloy steel, stainless steel, cast iron, and ductile iron. The thickness of the material being welded is bounded on the low end primarily by the skill of the welder, but rarely does it drop below 0.05 in (1.5mm). No upper bound exists with proper joint preparation and use of multiple passes, materials of virtually unlimited thickness can be joined. Furthermore, depending on the electrode used and the skill of the welder, SMAW can be used in any position [12].
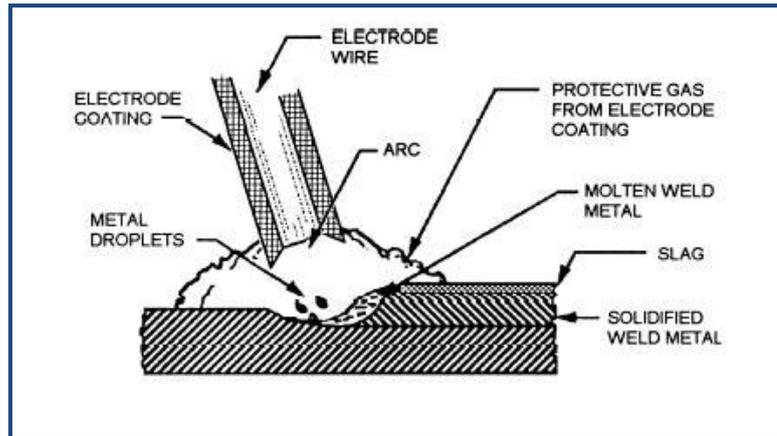
Figure 2.3 – Typical operating principles of SMAW welding process [16]

The most common quality problems associated with SMAW include weld spatter, porosity, poor fusion, cracking, and shallow penetration. Weld spatter not alter the integrity of the weld, but it damages the appearance and increases cleaning costs. It can be caused by high current, a long arc, or arc blow, a condition associated with direct current characterized by the electric arc being deflected away from the weld pool by magnetic forces. Arc blow also can cause porosity to the weld. Porosity normally not detectable without the use of advanced non-destructive testing methods. It is a severe concern since porosity can weaken the weld. Another defect affecting the strength of the weld is poor fusion. It is caused by low current, contaminated joint surfaces, or the use of improper electrode. Shallow penetration is harm to the weld strength. The problem can be overcome using a smaller electrode, increasing the current and reduce the welding speed. All these factors can lead to crack in the weld area. Another factor that can cause crack is high carbon, alloy or sulphur in the base material. in addition, the weld metal should not be excessively restrained, as it introduces residual stresses into the weld area and can cause cracking [13].

CHAPTER 3

METHODOLOGY

3.1 Research Methodology

In this project, there are several steps that needed to be follow in order to get the correct and relevant result. If the method that being used is wrong, then the result will be wrongly interpreted. There are three major steps for this project. The first step is to understand the project by interpreting the literature review. During this step, the understanding of the literature review is very important. It is to prevent misunderstanding of the project. For the second step is the modelling and simulation part. All the data for the SMAW welding and the material properties for carbon steel pipe are gathered to be put in the ANSYS software for the simulation. There are two analysis that needed to be completed which are structure analysis and thermal analysis. The last part is about data interpretation.

3.2 Steps involved for the Finite Element Analysis:

3.2.1 Pre processing Phase:
1) Define element types and options.
2) Define material properties
3) Create model geometry
4) Define meshing controls
5) Mesh the object created

3.2.2 Applying Boundary Conditions, Loads, and the Solutions

There are two methods that can be used to set the boundary conditions and loading to the model. The first one is to apply the conditions to the solid model (key points, lines, and areas) or the second one is to apply the boundary condition to the nodes and the elements. The method that will be

used is the first method because if there are changes in the meshing, there is no need to reapply the boundary conditions and the loads to the new model.

Required boundary conditions:

1) **Thermal Analysis:** Ambient Temperatures, Heat Transfer Rates, Convection Surfaces, Internal Heat Generation.
2) **Structural Analysis:** Displacements, Forces, Distributed Loads (Pressure), Temperature for thermal expansion.

3.2.3 Results of Finite Element Model: Post Processing

The post processing part is the part where the result of the analysis is displayed. The results that can be display are:

1) Deformed shape displays and contour displays
2) Tabular listings of the results data of the analysis
3) Calculations for the results data and path operations
4) Error estimations.

3.3 Key Milestones

There are several key milestones in this project which are:

1) Obtain the temperature contours which indicate the region experiencing melting and the HAZ.
2) Residual stresses obtained from structural analysis is plotted to show sections of the FE models, at the weld region, exhibiting the most relevant stresses.
3) Obtain the value of Residual Stresses.

3.4 Project Activities / Project Flow

```
┌─────────────────────────────────────────────┐
│                    Start                      │
└─────────────────────────────────────────────┘
                       │
                       ▼
┌─────────────────────────────────────────────┐
│         Define objective and scope of study   │
└─────────────────────────────────────────────┘
                       │
                       ▼
┌─────────────────────────────────────────────┐
│                literature review              │
└─────────────────────────────────────────────┘
                       │
                       ▼
┌─────────────────────────────────────────────┐
│          Study of residual stress in welding  │
└─────────────────────────────────────────────┘
                       │
                       ▼
┌─────────────────────────────────────────────┐
│           ANSYS software familiarization       │
└─────────────────────────────────────────────┘
                       │
                       ▼
┌─────────────────────────────────────────────┐
│            ANSYS modelling and meshing         │
└─────────────────────────────────────────────┘
                       │
                       ▼
┌─────────────────────────────────────────────┐
│      Application of different type of loads in ANSYS │
└─────────────────────────────────────────────┘
                       │
                       ▼
┌─────────────────────────────────────────────┐
│                Thermal Analysis                │
└─────────────────────────────────────────────┘
                       │
                       ▼
┌─────────────────────────────────────────────┐
│               Structural Analysis              │
└─────────────────────────────────────────────┘
                       │
                       ▼
┌─────────────────────────────────────────────┐
│             Residual Stresses Analysis         │
└─────────────────────────────────────────────┘
                       │
                       ▼
┌─────────────────────────────────────────────┐
│                 Result Analysis                │
└─────────────────────────────────────────────┘
                       │
                       ▼
┌─────────────────────────────────────────────┐
│          Conclusion and Recommendation         │
└─────────────────────────────────────────────┘
                       │
                       ▼
┌─────────────────────────────────────────────┐
│                    End                        │
└─────────────────────────────────────────────┘
```
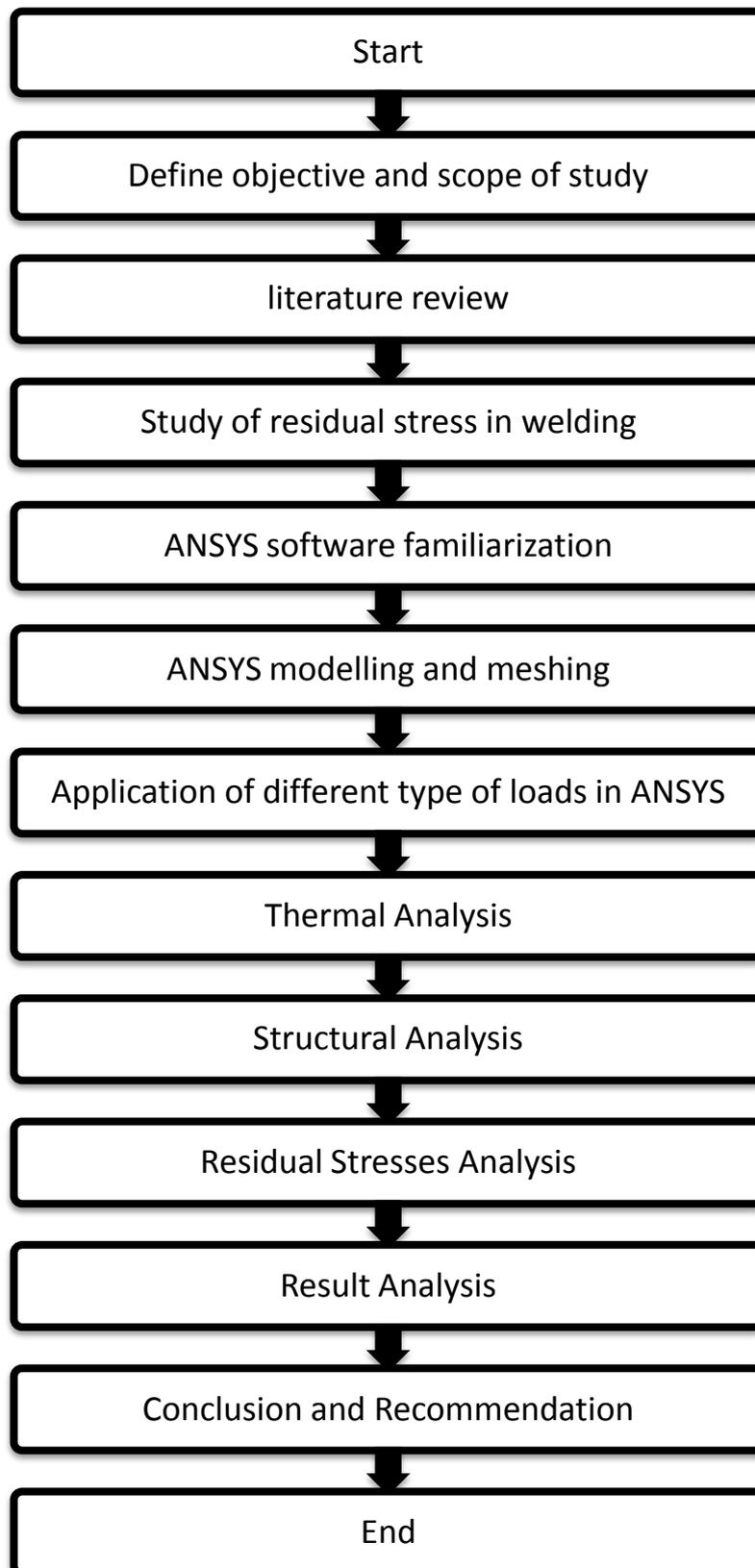
Figure 3.1: Project flow on FYP I & FYP II

3.5 Study Plan (Gantt chart)

Table 3.1: Study plan for FYP I

No	Detail/ Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Briefing/ Topic selection/Confirmation	▨	▨	▨											
2	Topic familiarization.			▨	▨										
3	Completion and submission of the Extended Proposal				▨	▨	▨								
4	Literature review on Finite Element Analysis and Residual Stress					▨	▨	▨	▨	▨	▨				
5	Proposal Defence							▨							
6	Understanding of work and all the tools and equation used						▨	▨	▨	▨					
7	ANSYS familiarization						▨	▨	▨	▨	▨				
8	Data gathering for Material Properties of Carbon Steel pipe and welding parameters							▨	▨	▨	▨				
9	Preparation of report												▨	▨	
10	Interim report submission													▨	▨

Table 3.2: Study plan for FYP II

No	Detail/ Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
1	Determine Carbon Steel A 36 thermal properties.	▨	▨	▨												
2	Calculation of important parameters. Eg. Weld heat input, welding speed, and etc.			▨	▨											
3	Determine Carbon Steel A 36 structural properties.					▨	▨									
4	ANSYS modelling and meshing.					▨	▨	▨	▨	▨	▨					
5	FYP 2 Progress Report Submission								▨							
6	Thermal Analysis, Structural Analysis and Residual Stress Analysis								▨	▨	▨	▨	▨	▨		
7	Pre-SEDEX & Presentation (Internal Examiner)											▨				
8	Submission of Draft Report to Supervisor												▨			
9	Submission of Dissertation (Soft bound) & Technical Paper													▨		
10	Oral Presentation 2 (VIVA)														▨	
11	Submission of Hard bound Report															▨

3.6 Welding Parameters and Model Geometry

Several assumptions needed to be made in order to simulate the welding process. There are the assumptions that will be applied in the ANSYS simulation:

1) Convection boundary condition is only applied on the outer surface of the model.

2) Heat input is moving at constant speed which is 5mm/sec

3) Material properties of the filler and electrode have the same properties with the base metal.

4) Displacement of the metal does not affect the temperature distribution

5) Initial Temperature will be set to 300K

6) Heat transfer coefficient, h is 50 W/m.K

7) Radiation Heat transfer is neglected

3.61 Weld heat input calculation

Heat input during welding is modelled in ANSYS by a distributed heat flux applying on individual elements. The amount of heat input is calculated as follow:

$Q = \mu \frac{UI}{V}$

Where:

μ - Arc Efficiency

V – Travel Speed

U – Voltage

I – Current

It is assumed that the current is 180 A, Voltage is 24 V and the welding speed is 5mm/sec. These values are in accordance with the Welding Procedure Specification (WPS) that has been written by Kavoshgharan Mechanics of Mersade Gharb Company for pipe welding. The arc efficiency is considered 85% [18].

Q = (0.85 x 180 x 24) / 0.005

   = 73.4 kW

3.62 Important Parameters of the A36 Carbon Steel pipe.

The pipe dimension is shown in Table 3.3. The thickness is based on Carbon Steel ANSI Schedule 40. The dimension needed to be set for the modelling part in the simulation. The weld direction is shown in Figure 3.2.

Table 3.3: Carbon Steel pipe dimension

Outside Diameter	OD = 16 " NPS (400 mm)
Inside Diameter	ID = 15 " (375 mm)
Thickness	0.5" (12.5 mm)
Length	1000 mm

The pipe geometry of the carbon steel pipe is as below:
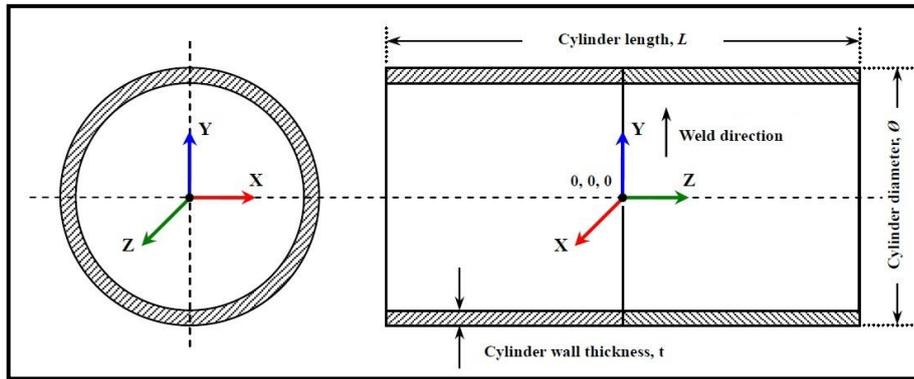
Figure 3.2: Geometry of Carbon Steel pipe

Thermal properties are required in order to simulate the welding process. The data is used in the early stage of the process which is to define the material properties in the ANSYS. Table 3.4 and table 3.5 shows the structural and thermal properties of the A36 Carbon Steel.

Table 3.4: Structural Properties of Carbon Steel [17]

Yield Strength, psi (MPa)	51000 (355)
Tensile Strength, psi (MPa)	71000 - 90000 (490 - 620)
Poisson Ratio	0.3
Young's Modulus (GPa)	202
Melting Temperature (K)	1698 - 1773
Density	7850 kg/m^3

Table 3.5: Temperature dependent thermal, physical and mechanical properties
of ASTM A36 carbon steel [17]

NO	Temperature (K)	Specific Heat (J/kg.K)	Conductivity (W/m.K)	Yield Stress (MPa)	Thermal Expansion Coefficient (X 10^{-5}/K)	Young's Modulus (GPa)
1	273	480	60	380	1.10	210
2	373	500	50	340	1.15	200
3	473	520	45	315	1.20	200
4	673	650	38	230	1.30	170
5	873	750	30	110	1.42	80
6	1073	1000	25	30	1.45	35
7	1273	1200	26	25	1.45	20
8	1473	1400	28	20	1.45	15
9	1673	1600	37	18	1.45	10
10	1823	1700	37	15	1.45	10

3.7   Simulation Methodology by Using ANSYS Software

3.7.1 Thermal Analysis

1) Define the element type

The element type that will be used for the thermal analysis is SOLID70. SOLID 70 have a 3-D thermal conduction capability. The element has eight nodes with a single degree of freedom, temperature, at each node. The element is applicable to a 3-D steady state or transient thermal analysis, which transient thermal will be used in this analysis. The element also can compensate for mass transport heat flow from a constant velocity field. Since this analysis will be followed by structural analysis, SOLID70 element can be changed to structural element.

Graphical User Interface (GUI) Method: **ANSYS Main Menu → Preprocessor → Element Type → Add/Edit/Delete**

2) Define material properties

There are several thermal properties that needed for the thermal analysis. These properties are based on the pipe properties which is Carbon Steel A36. The properties that are required are thermal conductivity, specific heat and density of the pipe.

GUI Method: **ANSYS Main Menu → Preprocessor → Material Props → Material Models**

3) Modelling

The pipe is modelled by using dimension. All the dimensions such as thickness, internal diameter, external diameter, and length are inserted into the software.

GUI Method: **ANSYS Main Menu → Preprocessor → Modelling → Create → Volume → Cylinder → By Dimension**

4) Meshing

Meshing is very important part in Finite Element Analysis. Inappropriate element shape and size will influence the accuracy of the results. The size of mesh is depending on the type of analysis. For this analysis, the mesh size is normal size. Fine mesh size will consume a lot of time during the solution part. Mesh tools used is volume sweep. The shape of the meshing is normal square shape since the welding line is based on nodes and it cannot be done if the shape is hexagonal.

GUI Method: **ANSYS Main Menu → Preprocessor → Meshing → Mesh → Volume Sweep → Sweep**

5) Analysis Type

The analysis type used for this thermal analysis is transient thermal analysis. This analysis determines the temperatures and other thermal quantities over time. The temperatures from transient thermal analysis will be used as inputs to the structural analysis for thermal stress evaluation.

GUI Method: **ANSYS Main Menu → Solution → Analysis Type → New Analysis → Transient → Full**

6) Define initial condition

The initial condition is very important for ANSYS to solve the problems without any error and without affecting the results. There are two initial conditions which are ambient temperature and the thermal convection. The ambient temperature is set to be 300K and for the convection heat transfer, it only applied on the outer surface of the pipe.

GUI Method:
   1) Convection

ANSYS Main Menu → Solution → Define Loads → Apply → Thermal → Convection → On Areas

2) Uniform temperature

ANSYS Main Menu → Solution → Define Loads → Apply → Thermal → Temperature → Uniform Temp

   7) Define Loads

The load is applied on the nodes on the surface of the pipe. The load is applied using heat flow since the heat source is moving throughout the pipe on the welding line. The heat is applied through birth and death technique. The birth and death option is used to deactivate and reactivate selected elements in the analysis. The amount of weld heat input is calculated as below:

$$Q = \mu \frac{UI}{V}$$

Where:

 μ - Arc Efficiency

 V – Travel Speed

 U – Voltage

 I – Current

It is assumed that the current is 180 A, Voltage is 24 V and the welding speed is 5mm/sec and the arc efficiency is considered 85% [18].

$$
\begin{aligned}
Q &= (0.85 \times 180 \times 24) / 0.05 \\
&= 73.4 \text{ kW}
\end{aligned}
$$

GUI Method: **ANSYS Main Menu → Solution → Define Loads → Apply → Thermal → Heat Flow → On Nodes**

   8) Solve

Solve command is initiated to solve after all the pre-processor command required for the thermal analysis is completed. This analysis is solve by two type of SOLVE command which are solve by using Current LS and Increment LS. The total number of steps is 25. This indicates 25 steps that needed to be solved.

GUI Method: **ANSYS Main Menu → Solution → Solve → Current LS**

9) Read Results

The results from the welding process will show the temperature distribution and also the HAZ along the welding line. The contour plot will show the temperature distribution and the maximum temperature can be seen from the contour plot.

GUI Method: **ANSYS Main Menu → General Postproc → Plot Results → Nodal Solution**

10) Path Operation

Path is method that is used to collect specific data set. In this case, the data needed is the temperature along the affected area of the welding line. Path operation will define and record the data between the starting point and the finish point. The path in this analysis is including all the nodes at the welding line. The specific data set will be saved for the structural analysis.

GUI Method: **ANSYS Main Menu → General Postproc → path Operations → Define Path → By Nodes**

11) Data Storage

The data from the path created along the welding line is plotted into graph and the data will be saved to be used in structural analysis. The data will be saved in .rth file.

GUI Method: **ANSYS Main Menu → General Postproc → Path Operations → Archive Path → Store → Paths in file**

12) Finish

FINISH Command will be issued after all the process in thermal analysis is done. This is important for the ANSYS to start new analysis which is structural analysis.

3.7.2 Structural Analysis

The data from the thermal analysis such as temperature will be used for the next analysis which is structural analysis. Structural analysis is conducted to determine the effects of loads which is thermal load from the thermal analysis on physical structures of the metal. The results from the analysis are used to verify the structure fitness for use. Some structural properties such as yield strength, tensile strength, Poisson Ratio, young modulus and the melting point of the material is required in order to perform the structural analysis.

The element type which previously SOLID70 will be changed to SOLID185 which is structural element. The results will be evaluated from the contour plot that plotted to show the stress distribution caused by the welding process.

1) Change Element

In order to conduct the structural analysis, the element type must be reliable. The previously used is SOLID70. The element is only used for thermal analysis. SOLID70 will be replaced with SOLID185. SOLID 185 is used for 3-D modelling of solid structures. It is defined by eight nodes having three degrees of freedom at each node, translation in the nodal x, y, and z directions. The element has plasticity, hyper elasticity, stress stiffening, creep, large deflection, and large strain capabilities.

GUI Method: **ANSYS Main Menu → Preprocessor → Element Type → Switch Elem Type**

2) Define Properties

Some additional properties required to perform the structural analysis. Yield strength, tensile strength, Poisson Ratio, young modulus and the melting point will be inserted into the software.

GUI Method: **ANSYS Main Menu → Preprocessor → Material Props → Material Models**

3) Specify Initial Conditions

Initial conditions are required in order to obtain better results in this analysis. There are two initial condition required. The first one is both end of the pipe is assumed clamped and the displacement is constant which is zero. It means that no displacement at both end of the pipe. The second initial condition is the reference temperature which is same as the bulk temperature is set to 300K.

GUI Method:
   1) Reference Temperature

ANSYS Main Menu → Solution → Define Loads → Settings → Reference Temp
   2) Displacement

ANSYS Main Menu → Preprocessor → Loads → Define Loads → Apply → Structural Displacement → On Areas

4) Solve

The solve command is initiated after all the temperature profile, parameters and boundary conditions for structural analysis is inserted. The temperature from the thermal analysis will be load into the structural analysis. The effect on structural properties of the pipe affected by the thermal load from the welding process will be evaluated.

GUI Method:
   1) Load Temperature

ANSYS Main Menu → Solution → Define Loads → Apply → Structural → Temperature → From Therm Analy

   2) Solve

ANSYS Main Menu → General Postproc → Path Operations → Archive path → Retrieve → Path from file

3.8 Tools Required

The tools required for this project is only ANSYS software. ANSYS Version 11 is being used since the software is provided by UTP. The software is used to simulate the residual stress on carbon steel welded pipe. All the required data such as material properties and dimension is inserted into the system and the software will simulate and calculate the required data for the results.

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Results

The results from the thermal analysis and structural analysis will be evaluated to obtain the value of the residual stresses created by the welding process. The results will be evaluated from the contour plot and vector plot.

4.1.1 Temperature Distribution
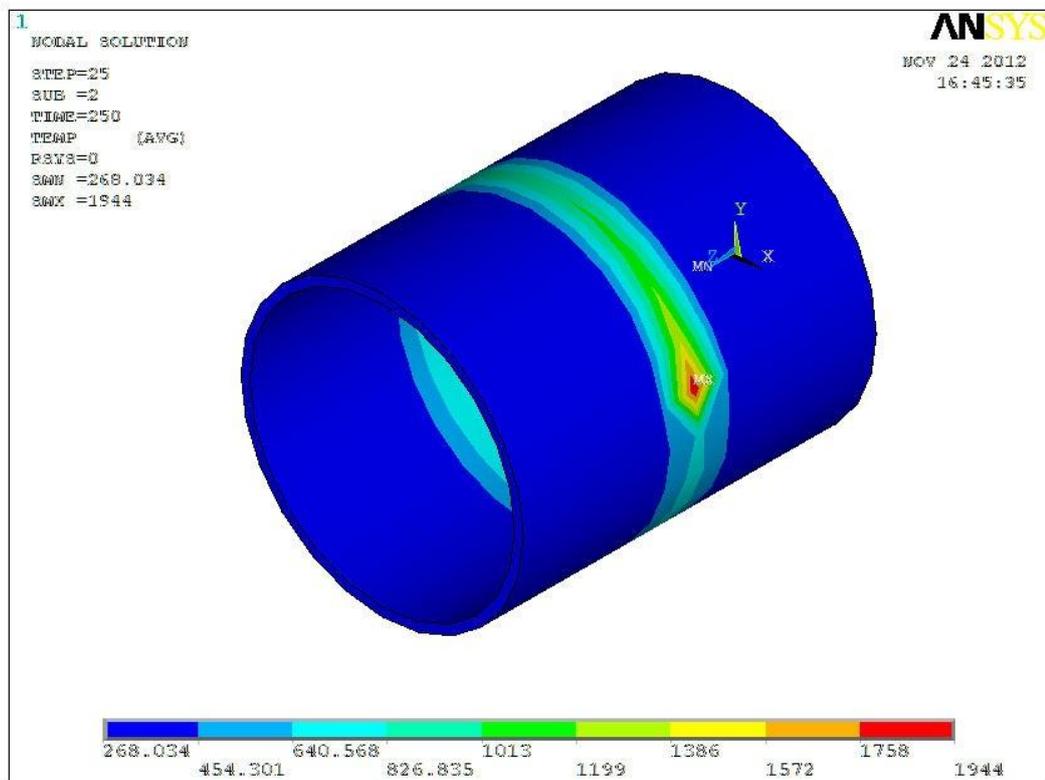
Figure 4.1: Temperature distribution along the welding line

The contour plot in Figure 4.1 show the temperature distribution and the maximum temperature can be seen from the contour plot. The maximum temperature shown is 1944K.

4.1.2   Von-Mises of Total Mechanical Strain
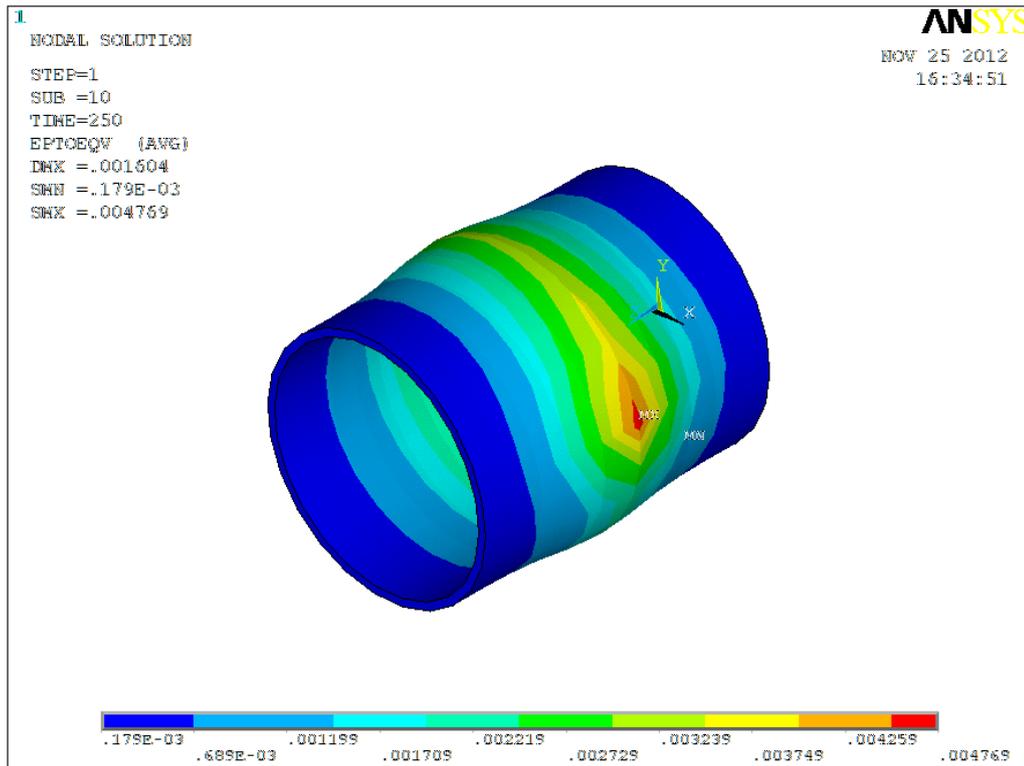
Figure 4.2: Von-Mises of Total Mechanical Strain acting on the pipe.

Region that undergoes mechanical strain is shown in the Figure 4.2. This plot is to show the formation of HAZ along the welding line. Maximum strain is occurred at node 333 which is 0.004769 m/m. Strain indicates that distortion occur and based on the stress value, the stress cause plastic deformation to the metal.

4.1.3   Stress in Z-axis
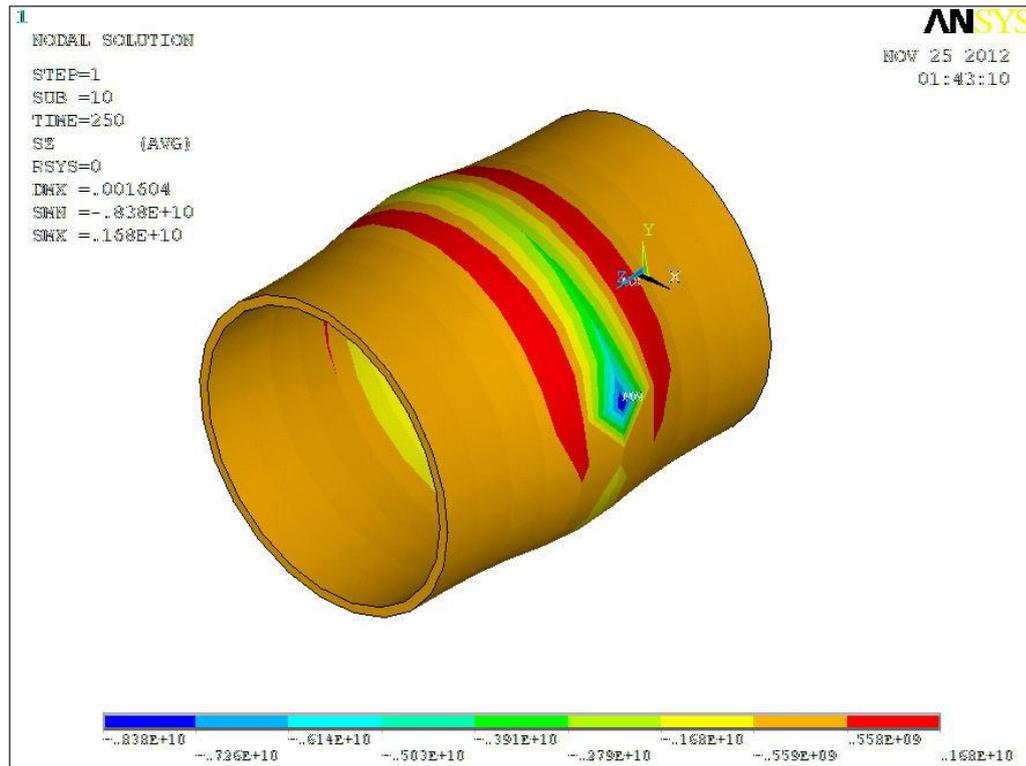
Figure 4.3: Stress in Z-axis acting on the pipe

Figure 4.3 shows the stress in Z-axis acting on the pipe after the welding process. Stress in Z-axis is the axial stress acting on the pipe. The maximum value of axial stress is -838 MPa which indicates it is a compressive stress. The axial stress gradually moves towards the end of the pipe and the maximum value at node 333 is caused by the heat input acted twice at the node. Distortion will occur at the point where the stress is exceeding the tensile strength. But, the value will be decrease after the cooling process. So, the value of axial stress will be lower depending on the cooling method.

4.1.4 Stress Intensity Plot
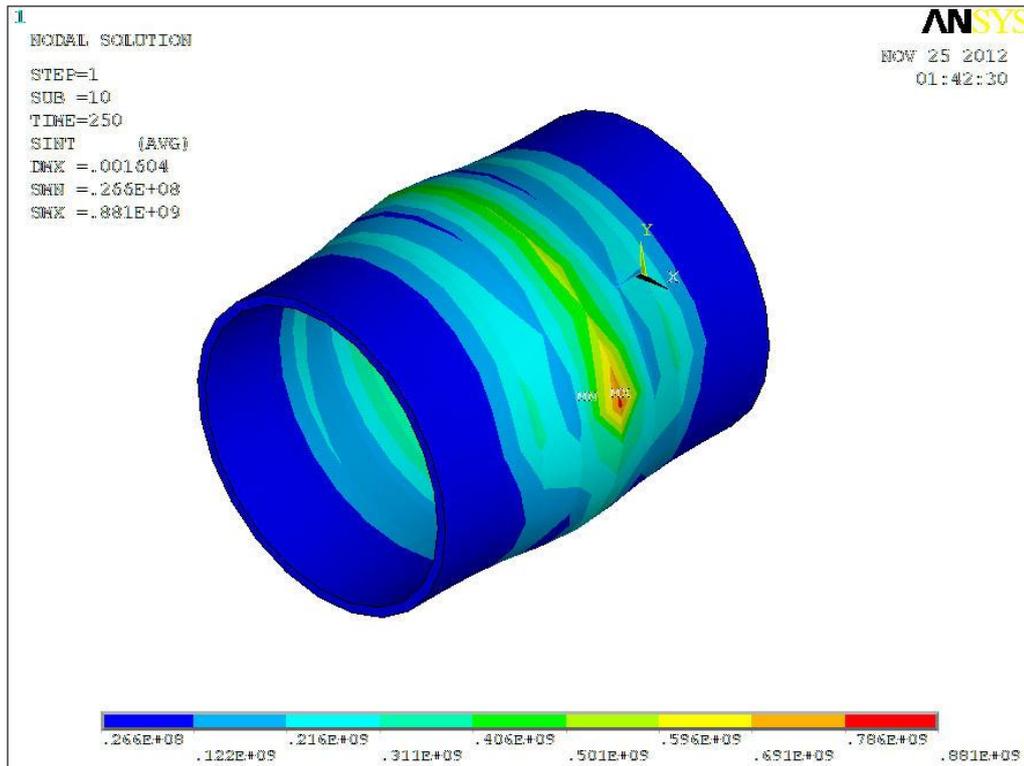
Figure 4.4: Stress intensity plot for the pipe

The stress intensity plot is shown in the Figure 4.4. The plot shows where is the accumulation of stress that caused by the welding process. The maximum accumulation is at node 333. This shows that the weakest point along the welding line is at that point. Any micro crack can lead to later crack propagation. Extra precaution and post weld heat treatment should be focused on that area to reduce the severity of the stress concentration at that area.

4.1.5 Von-Mises Stress Distribution
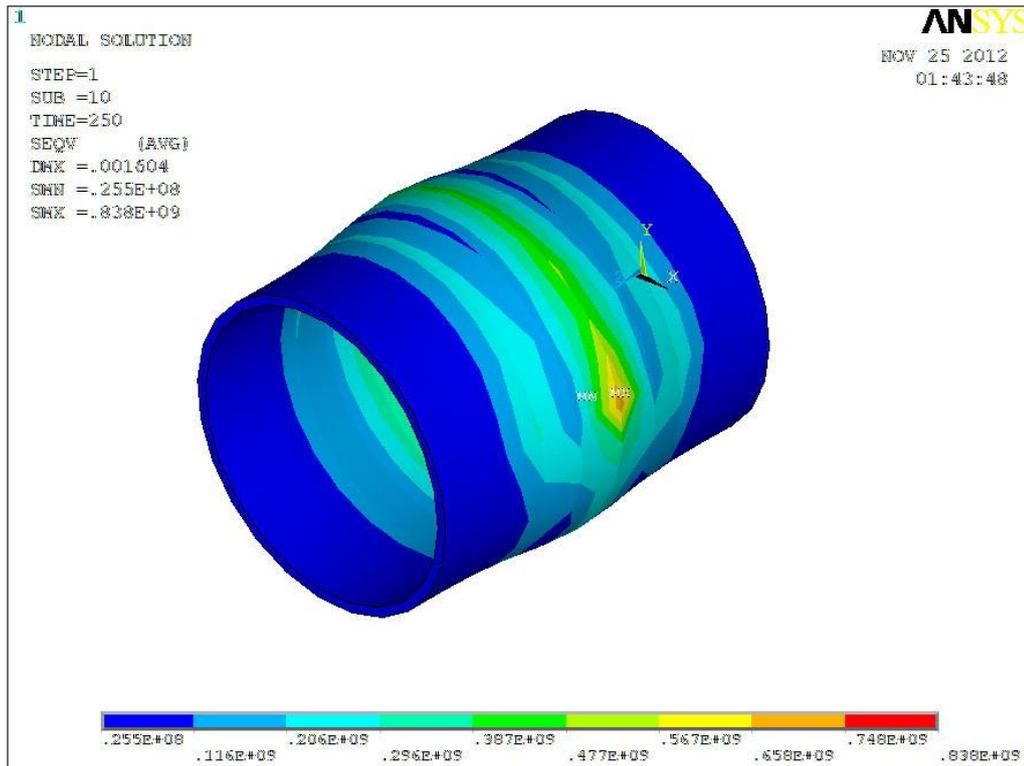
Figure 4.5: Stress distribution plot

The Von-Mises stress distribution shown in Figure 4.5 is used to predict the yielding of the materials due to the load applied from the welding process. The critical value is the yield strength of the material. In this case, the yield strength of the pipe is 355MPa. The maximum value for this Von-Mises Stress is 838MPa. Hence, the area that gives higher Von-Mises Stress value than the yield strength will undergo plastic deformation.

4.1.6   Sum of Displacement / Distortion
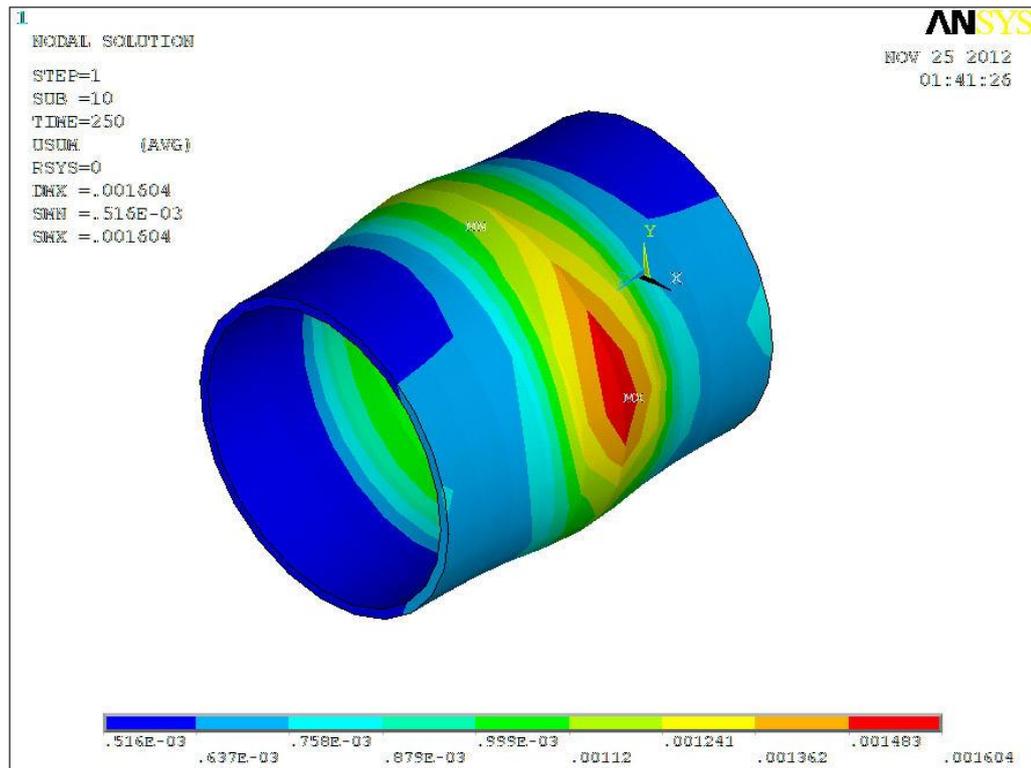
Figure 4.6: Sum of displacement / distortion of the pipe.

Welding distortion, deformation or warping of weldment during welding is a natural outcome of intrinsic non uniform heating and cooling of the joint. Sum of displacement shows in the Figure 4.6. The maximum displacement occurs at node 333. Node 333 is where the welding process starts and ends. The maximum value is 1.604mm. This shows that the thermal loads from the welding process can lead to the distortion of the metal. Distortion magnitude also affected by the Thermal Expansion Coefficient. Thermal expansion coefficient of this pipe is 1.15×10^{-5}/K.

4.1.7 Shear Stress in XY Plane
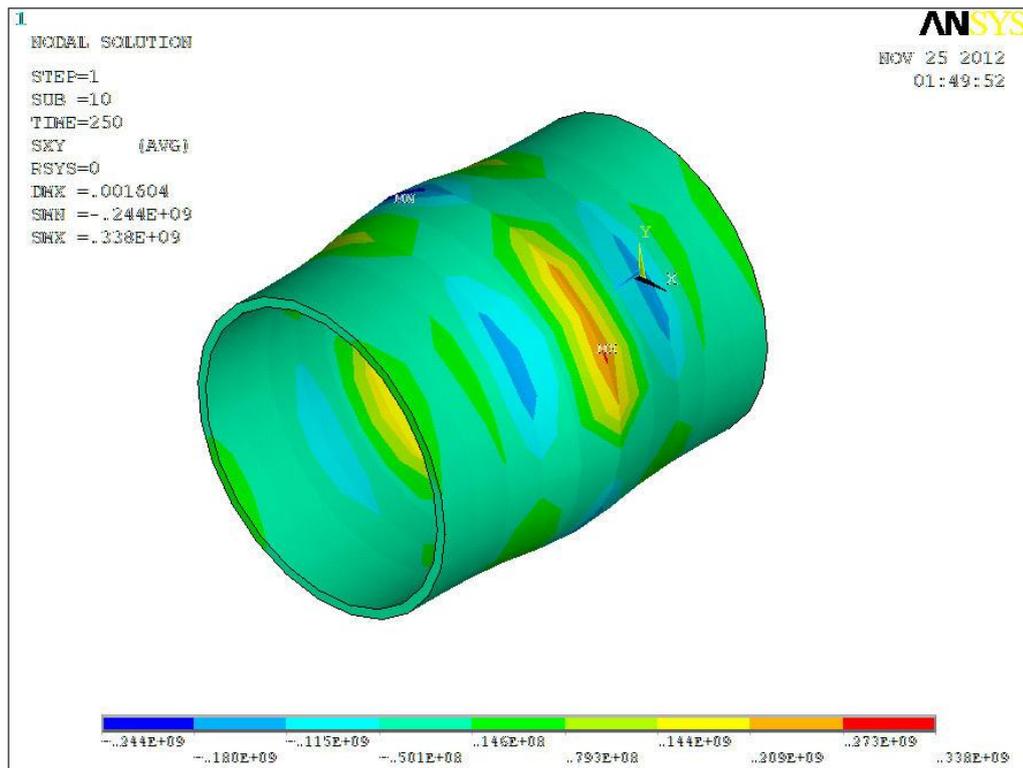
Figure 4.7: Stress vector plot of Shear Stress in XY Plane

Shear stress in XY plane of the pipe is shown in Figure 4.7. Shear stress is defined as the component of stress coplanar with a material cross section. Shear stress arises from the load vector component parallel to the cross section. The plot shows that the maximum stress is 338MPa.

4.2 Results Comparison

Table 4.1 shows the value of stresses at the different value of load applied during the welding process. The current is changed with the increment of 10 A and reduction of 10 A. The value of residual stresses is increased as the load applied increased and it decreased as the load decreased. Low heat input may cause lack of penetration and the welding process is not completed. While higher heat input will induced a lot of residual stresses that can weaken the properties of the metal. As a consequence, the load applied needed to be choosing very carefully.

Table 4.1: Summary and comparison of stresses from different load

Weld Heat Input (W)	69360	73400	77500
Max Stress Intensity (MPa)	793	881	970
Von Mises Stress	704	838	1156
Von Mises Total Mechanical Strain (m/m)	0.004260	0.004769	0.005200
Maximum Distortion / Displacement (m)	0.001500	0.001604	0.001704
Shear Stress in XY Plane (MPa)	227	338	416
Maximum Temperature (K)	1911	1944	2100

4.3 Discussion

Based on the stress diagram plotted, node 333 showed the maximum stress acting on the pipe due to the welding process. This is due to the starting point and end point of the welding is at node 333. It means that the node experienced two times of heat input. It also shows the maximum stress intensity is at that node. The highest temperature is at that point which is 1944 K.

Maximum axial stress acted on the same node which is node 333. Axial stress is the stress acted on the pipe on z-axis. The value is -838 MPa which is the compressive stress. The melted metal due to welding process will move towards the welding line and created huge amount of axial stress. The distortion is occurred at that point since the value exceeding the tensile strength of the metal which is 490 MPa – 620 MPa. But, the distortion will be reduced when cooling process is done.

Displacement vector plot shows the value of displacement after the welding process. Distortion indicates that stress acting on the pipe causes a constant movement towards both ends of the pipe. The distortion is not really becomes main concern since the maximum distortion only 0.001604 m that is around 1.6 mm. The distortion will form weld bead on the welding line. With the proper surface finishing and machining will remove the distorted metal.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

From the results, it was showed that the value of residual stresses increased as the heat load increased. There were several stresses that remain on the pipe that cause the changed in mechanical properties of the pipe. The value is obtained from the contour plot of the simulation.

The maximum stress intensity showed by the contour plot was 881 MPa. The total mechanical strain has the maximum value of 0.004769 m/m. The distortion showed that the maximum value was 0.001604m and this was on all degree of freedom. From the normal weld heat input that suggested by Welding Procedure Specification (WPS), the maximum temperature was found at node 333 which is 1944K.

.

5.2 Recommendation

There are some recommendations in order to improve the results quality and to make the simulation applicable for the industrial purpose. The recommendation is on the software used which is ANSYS software and also on the technique used.

The meshing size used for this simulation is medium size of mesh. The size can be changed to fine mesh so that the results will be more accurate. The user also needed to learn how to use command script in ANSYS so that not much time wasted on the simulation.

Moreover, the welding parameters used can be revised depends on the requirement needed and the type of material used. The electrode speed is constant in this simulation. In the manual welding process, it is very difficult to maintain the welding speed. But, in this simulation the welding speed is remain constant for ease of heat input calculation.

REFERENCES

[1] Leis B, Parkins RN. (1998) *Mechanics and material aspects in prediction serviceability limited by stress-corrosion cracking.* Fatigue Fract Eng, Mater Struct ;21:583–601.

[2] A. Amirat, A. Mohamed- Chateauneuf, K. Chaoui (2006) *Rliability assessment of underground pipelines under combined effect of active corrosion and residual stress.* International Journal of Pressure Vessel and Piping.

[3] Glinka G. (1986) *Residual stresses in fatigue and fracture: theoretical analyses and experiments*. In: Niku-Lari A, editor. Residual stresses generation, relaxation. UK : Pergamon Press

[4] Gray JM, Pontremoly M. (1987) *Metallurgical options for grades X70 and X80 line pipes.*International conference pipe technology, Rome; p.171–91.

[5] Endos S, Nagae M. (1992) *Development of X100 UOE line pipe.*The International conference on pipe line reliability, Calgary; p. 3:4.1–4.11.

[6] A. Yaghi, T.H. Hyde, A.A. Becker, W. Sun, J.A. Williams, and B. Pathiraj. (2006) *5th International Conference on Mechanics and Materials in Design*, School of Mechanical, Materials & Manufacturing Engineering, University of Nottingham,

[7] David Roylance. (2001) *"Finite Element Analysis",* Department of Materials Science and Engineering, Massachusetts Institute of Technology, Cambridge, MA 02139

[8] Saeed Moaveni. (1999) *Finite Element Analysis*, Theory and Application with ANSYS, Minnesota State University, Mankato.

[9] ABAQUS User Manual, (2002) version 6.3. Hibbitt, Karlsson & Sorensen, Inc.

[10] Weman, Klas. (2003) *"Welding Processes Handbook",* New York: CRC Press LLC. ISBN 0-8493-1773-8.

[11] Lincoln Electric (1994). *The Procedure Handbook of Arc Welding*. Cleveland: Lincoln Electric. ISBN 99949-25-82-2.

[12] Cary, Howard B. and Scott C. Helzer (2005). *Modern Welding Technology*. Upper Saddle River, New Jersey: Pearson Education. ISBN 0-13-113029-3.

[13] Ali Moarrefzadeh (2012), *Finite – Element Simulation for Thermal Profile in Shield Metal Arc Welding (SMAW) Process*, International Journal of Emerging trends in Engineering and Development.

[14] Withers PJ, Webster PJ (2008), *Neutron and synchrotron X-ray strain scanning.* Strain, Manchester Materials Science Centre, Manchester.

[15] WebCorr Corrosion Consulting Services, *Different Types of Corrosion, Recognition, Mechanism & Prevention.* Retrieved August 20, 2012, 04.40p.m, from http://www.corrosionclinic.com/types_of_corrosion/weld_decay_weldment_corrosion.htm

[16] Welding Process Weld Design, *Shield metal arc welding (SMAW),* Retrieved August 21, 2012, 12.30 p.m, from *http://www.corrosionist.com/Shielded_metal_arc_welding_(SMAW).htm*

[17] D. Gary. (2007) *Carbon Steel Handbook,* Electric Power research Institute (EPRI), California.

[18] Kou S. (2003), *Welding Metallurgy*, 2nd Edition, John Wiley & Sons, Inc. New Jersey.