FINITE ELEMENT ANALYSIS OF RESIDUAL STRESSES IN BUTT WELDING JOINT OF ASTM A36 PLATE AND ITS EFFECT ON FATIGUE CRACK GROWTH

by

Muhammad Hafizuddin Adha bin Arizam

(13225)

Dissertation submitted in partial fulfilment of

the requirement for the

Bachelor of Engineering (Hons)

(Mechanical Engineering)

SEPTEMBER 2013

Universiti Teknologi PETRONAS

Bandar Seri Iskandar

31750 Tronoh

Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

FINITE ELEMENT ANALYSIS OF RESIDUAL STRESSES IN BUTT WELDING JOINT OF ASTM A36 PLATE AND ITS EFFECT ON FATIGUE CRACK GROWTH

by

Muhammad Hafizuddin Adha Bin Arizam

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. Saravanan Karuppanan)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

September 2013

CERTIFICATION OF ORIGINALITY

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

____

(MUHAMMAD HAFIZUDDIN ADHA BIN ARIZAM)

ABSTRACT

Welding is a process to bond and joint materials especially metals for various purpose. Welding has become the method of choice for its strength and efficiency rather than riveting, screwing or bolting. Unfortunately, welding can also affect the structure's mechanism in the welded materials. This is because of the thermal load due to heat in welding, that create residual stresses in weld region, resulting fatigue crack growth in the materials. In construction, the ASTM A36 plays a vital role. This plate is used to weld construction of bridges, buildings and oil rigs. For simulation of welding, ANSYS software will be used in this project. The objectives of this investigation were to determine the variation of residual stresses due to thermal load in a welded plate as well as to estimate the fatigue crack growth caused by residual stresses. Three different heat flux was applied in this project. Theoretically, welding processes will produce a temperature distribution due to the heat applied. The temperature distribution will create heat affected zone (HAZ) that is caused by rapid thermal expansion, followed by thermal contraction. The residual stresses exist when the metal is cooled down rapidly to ambient temperature. Three different magnitude of heat flux, 122.4 MW/m^2, 163.2 MW/m^2 and 204 MW/m^2, were used in this project and they were varied based on current applied. Based on the results, high residual stresses were accumulated near the center of butt welded plate. This indicated that the stresses were distributed from the welding site when the plate was cooled down. Welding distortion and deformation occurred because of non-uniform heating and cooling of the joint. The higher the heat flux, the higher the deformation of the welded plate. In the fatigue crack growth analysis, result shows that the fatigue strength was affected by operational load or cyclic stresses. From the results, at lower cyclic load, the number of cycle to failure will be high. Thus, it takes longer time for material to experience fatigue damage. Whereas at high cyclic load, the number of cycle to failure will be less. As a result, it takes faster time to fatigue damage.

ACKNOWLEDGEMENT

In the name of Allah, the Most Gracious, the Most Merciful. Praise to Him the Almighty that in his will and given strength, the final year project is successfully completed within the allocated time.

Upon completing the final year project, author owe many thanks to many people for their help and support, as well as their contribution in time, effort, to advice, supervise, discuss and help. First and foremost, deepest thanks goes to the Universiti Teknologi PETRONAS and final year project coordinator that have given the author the biggest opportunity to study and conduct a project.

Author would like to thank a dedicated supervisor, Dr. Saravanan Karuppanan. The supervision and support that he gave truly helped the progression and smoothness of the final year project. The co-operation is indeed appreciated. High appreciation also goes to PhD student, Mr Santosh Patil. He guided the author on how to deal with many problems. Not to forget, a deep sense of gratitude goes to author's parents for their support and motivation along the project.

Lastly, the author would like to express his gratitude to the fellow colleagues especially Mr Zulkifly bin Abu Hamin for his help and ideas throughout the completion of this project.

Thank you.

Muhammad Hafizuddin Adha Bin Arizam
Mechanical Engineering Department

CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1

INTRODUCTION

1.1 Background of Study

The history of welding is started since the Bronze Age and Iron Age. During that time, forge welding was used to joint two pieces of metal by heating them with high temperature and hammering it together. Next, the electric fusion process, electric arc welding and the investigation of welding's effect such as residual stresses and deformation has started in eighteenth and nineteenth century with the analytical process [1].

Welding is a process that is used to joint materials especially metal and plastic. The aim is to bond together the material for various purpose of scope such as construction of building, connecting pipelines, automotive industries and many more. Although there is also other way to joint the metal such as riveting, welding has become the method of choice for its strength and efficiency.

The welding works when the electrode creates an electric arc that melts the metal and filler rod to create a pool of molten metal that hardens to fuse the two piece of metal together. To ensure the integrity of the joint, the welder needs to decide on the type of joint, welding process and value of heat input before welding the metals. There are five basic type of weld joints which are butt joint, T joint, Lap joint, corner joint and edge joint.

During welding, a lot of heat will be produced and thus generate residual stress. Residual stress can be defined as the stress that remains after welding process. When all external loads or reactions are removed, residual stress occur within the structure of welding area [2]. The place where residual stresses remain is called the heat affected zone (HAZ). This area was affected by the heat during welding and experienced uneven cooling compared to other area. Thus, it will create different phase transition that remain as the residual stresses there. These residual stresses can give negative effect to the structure of the metal such as crack growth, creep, distortion and fatigue.

1.2 Problem Statement

1.2.1 Problem identification

During the process of welding, a lot of heat is applied to the welding materials. The heat from the welding process will create temperature distribution around welded area. As a result, heat affected zone (HAZ) is generated. When the materials is cooled down rapidly to room temperature, residual stresses remain in the structure of the materials. These stresses may harm the structural components of the materials by encouraging fatigue crack growth. Fatigue crack growth needs attention because it can become a liability to the materials in term of life span.

1.2.2 Significance of the project

ASTM A36 plate is a low carbon steel alloy. This type of steel is very important and used widely in construction especially buildings, bridge and offshore structures. This project is useful to weld industry for their future research on residual stresses and fatigue crack growth. From this project, important parameters that and need to be considered before welding have been identified. For example the magnitude of heat input, the current and the type of joint applied can affect the structure of the welded materials.

Nowadays, simulation is used to get more accurate results and can save time. It allows the user to observe an operation through simulation without actually performing that operation. Besides, the results from the simulation can be applied to the real situation.

1.3 Objective

The objectives of this project are:

1. To determine the variation of residual stresses due to thermal load in a welded plate.
2. To estimate the fatigue crack growth caused by residual stresses.

1.4 Scope of Study

The scope of analysis is on residual stresses caused by thermal load with different heat input during welding. The fatigue crack growth condition of the material, ASTM A36 will be determined through Paris Law. The type of welding is shield metal arc welding (SMAW). The simulation will be done using ANSYS.

1.5 Relevancy of the project

This project can give awareness for welders to study the effect of welding to the materials. Besides, this investigation can also give useful information about formation of residual stresses that can affect the microstructure of the materials.

CHAPTER 2

LITERATURE REVIEW

2.1 Welding

Welding processes produce transient thermal heat source, which is used to melt and solidify base metal in localized fusion zone. Temperatures around the weld vary with distance from the weld centreline. This temperature distribution caused a rapid thermal expansion and followed by the thermal contraction in the weld region or heat affected zone (HAZ). The fusion zone (FZ) and heat affected zone (HAZ) regions experience high temperature, which cause phase transformations and alterations in the mechanical properties of the welded metal. The temperature around the weld that are not uniform is caused by intense concentration of heating in localized zone and consequent cooling during welding. Local plastic strain forms due to temperature gradients and thus lead to residual stress [3]. The residual stress caused by welding can have dissimilar kind of effect on welded structure for example increasing the sensitiveness of a weld to fatigue damage, stress corrosion cracking and fracture [4].

2.2 Residual Stresses

Existence of residual stresses may be beneficial or harmful for the structural components depending upon the magnitude and character of residual stresses. Residual stresses can be specified as the stresses that persist inside the structure without the application of external loads and supposed to be self-balancing within the bulk [5]. These residual stresses arise when the structure is cooled down rapidly to ambient temperature. Besides, metal deformation caused by residual stress was also affected by internal and external restraints. The internal restraint are controlled by the temperature-dependent thermo-physical and mechanical properties of materials such as Young's modulus, yield strength, thermal expansion coefficient, specific heat and thermal conductivity of the material [6].

Brar G.S stated in his study using Finite Element Analysis that the nearer the distance from weld centre, the higher the residual stresses. The residual stresses increase, with increase in the plate thickness. On top of that, Brar G.S concluded that heat input play an important role in welding. The lower value of welding current resulted in lower heat inputs and lower residual stresses.

Research on residual stress has been done continuously in order to improve the fabrication during welding. The techniques used by researchers to evaluate residual stresses consists of destructive and non-destructive techniques, namely X-ray diffraction method [7], neutron diffraction method [8], layer removal method, sectioning method, ultrasonic and magnetic method and hole drilling method [9]. Unfortunately these methods will not be used in this project because of expensive expenditure. Therefore, finite element analysis will be used to predict the residual stress as it is approved to be one of the proficient tool with good accuracy [10-11].

When the residual stresses from welding reach the yield stress of the material, some of elastic strain from residual stresses are converted into plastic deformation. Inherent strains can be defined as the residual plastic strain caused by welding process [12]. It exists in the welds and nearby where the structure undergo a large thermal cycle. Besides, it is also considered as a source causing the welding deformation.

2.3 Shield Metal Arc Welding

Shield metal arc welding (SMAW) is a type of arc welding that uses a flux-coated consumable rod electrode [13]. The electrode creates an electric arc that melts the metal and filler rod to create a pool of molten metal that hardens and fuse the two piece of metal together. Successful welding depends on many factors which are selection of electrode's size, welding current, travel speed, arc length and good preparation. Too low of current will experience difficulty in maintaining a stable arc. Excessive current will create overheating of the electrode [14]. It will cause burning and rough surface through material.

2.4 Birth and Death Simulation

In order to simulate the welding process exactly to the real welding, “birth and death” technique can be applied. This technique simulate the weld filler variation with time [15]. The heat input would be allocated to each of selected nodes so that weld heat input work as single load at any given time. According to Armentani E., Esposito R. And Sepe R. (2007), the elements are deactivated by multiplying their stiffness by a severe reduction factor. When elements are activated, they are not actually added to the model, but are simply reactivated. When an element is reactivated, its stiffness, mass, element loads and others return to their full original values.

2.5 Heat flux

Heat flux is the type of heat input that will be used in the welding simulation. Higher magnitude of heat flux can give large area of heat affected zone (HAZ). There are expensive and modern process of welding such as laser beam welding that give limited amount of heat with high concentration. This type of welding can result in small amount of HAZ but very expensive in term of cost. To calculate the heat flux for arc welding, the following formula can be used:

$$Q = \eta \frac{V I}{v d} \qquad (1)$$

where Q is the heat input, η is the efficiency, V is the voltage, I is the current, v is the welding travel speed and d is the diameter of electrode used.

2.6 Fatigue crack growth

In finite element, the effect of welding residual stresses on fatigue behaviour can be simulated by modelling the fatigue crack propagation under cyclic loading for two dimensional compact tension specimens. To predict the fatigue crack growth under residual stresses, a cohesive zone model with evolutionary damage law was applied [16]. Based on the result analysis of welding residual stresses by Goo B.C, Seo J.W and Yang S.Y. (2010), the locations of fatigue crack initiation is dependent on the magnitude of the applied load, because residual stresses are redistributed, which change the weakest point under applied loading.

Life prediction for fatigue crack growth was generated by Paris [17]. He relates the stress intensity factor *K*, to the sub critical crack growth under the fatigue stress. On top of that, Paris studied a number of alloys and realised that plots of crack growth rate against stress intensity factor range straight lines on log-log scales:

$$log\left(\frac{da}{dN}\right) = m\log(\Delta K) + \log C \tag{2}$$

$$\frac{da}{dN} = C\Delta K^{m} \tag{3}$$

where *a* is the crack length, *N* is the number of load cycles, $\frac{da}{dN}$ is the crack growth rate and *C* and *m* are constants.

The crack intensity factor is defines as:

$$\Delta K = \Delta\sigma Y\sqrt{\pi a} \tag{4}$$

where $\Delta\sigma$ is the range of cyclic amplitude and *Y* is constant. Besides, the equation can be solved by differential equation:

$$\frac{da}{dN} = C\Delta K^{m} = C(\Delta\sigma Y\sqrt{\pi a})^{m} \tag{5}$$

$$\int_{0}^{Nf} dN = \int_{ai}^{ac} \frac{da}{C(\Delta\sigma Y\sqrt{\pi a})^{m}} \tag{6}$$

A study by Benachour, M., Benachour, N., & Benguediab M. (2012), found that as number of cycles increase, the crack growth and length increase. Heat caused by welding and subsequent rapid cooling can cause tensile residual stresses at the weld toe of butt-welded joints. These tensile residual stresses were considered as one of the major influence on fatigue strength. The result of their study is shown in Figure 2.1. Case 1, 2 and 3 have different magnitudes of cyclic load of 200 MPa, 300 MPa and 400 MPa respectively.

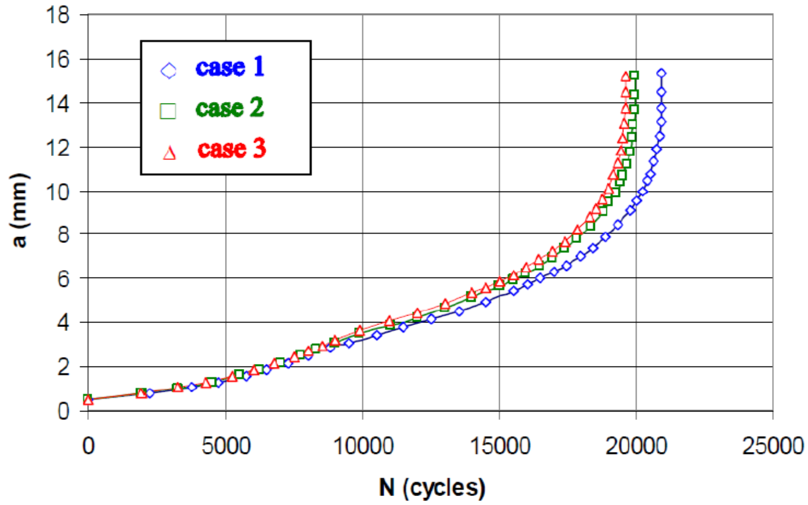


Figure 2.1 : Graph of crack length versus number of cycle [20]

2.7 ASTM A36

ASTM A36 is a low carbon steel used in a variety of industries. This type of steel is mainly used in structural and construction especially offshore [18]. The grades represent the yield strength of 36 ksi [19]. This type of steel will be used as the material in this project.

2.8 Finite Element Analysis

The usage of finite element simulations on residual stress and distortion are increasing. Finite element analysis (FEA) is based on the approximation solution to any complex engineering problem. The analysis is being used in virtually all engineering discipline.

CHAPTER 3

METHODOLOGY

3.1 Research Methodology

Several steps are required in this project. These steps are important to achieve a better result.

Step 1: Identify the Problem

In this project, the residual stresses in butt weld joint of ASTM A36 plate will be studied by using Finite Element Analysis. Besides, the effect of residual stresses on fatigue crack growth must also be considered.

Step 2: Conduct research on Residual stress and fatigue crack growth

Conduct a research to find techniques to simulate the residual stress and fatigue crack growth. In ANSYS, there are many technique that can be used to find residual stresses in welding, for example birth and death technique.

Step 3: Project Simulation and Data Gathering

The butt weld plate welding will be simulated using ANSYS software. Data gathering such as the thermal and mechanical properties of the material is important to run the simulation.

Step 5: Data evaluation and Analysis

Evaluate and analyse the data by plotting graph of simulation analysis.

Step 6: Produce Final Report

Prepare final report after all the simulation is done.

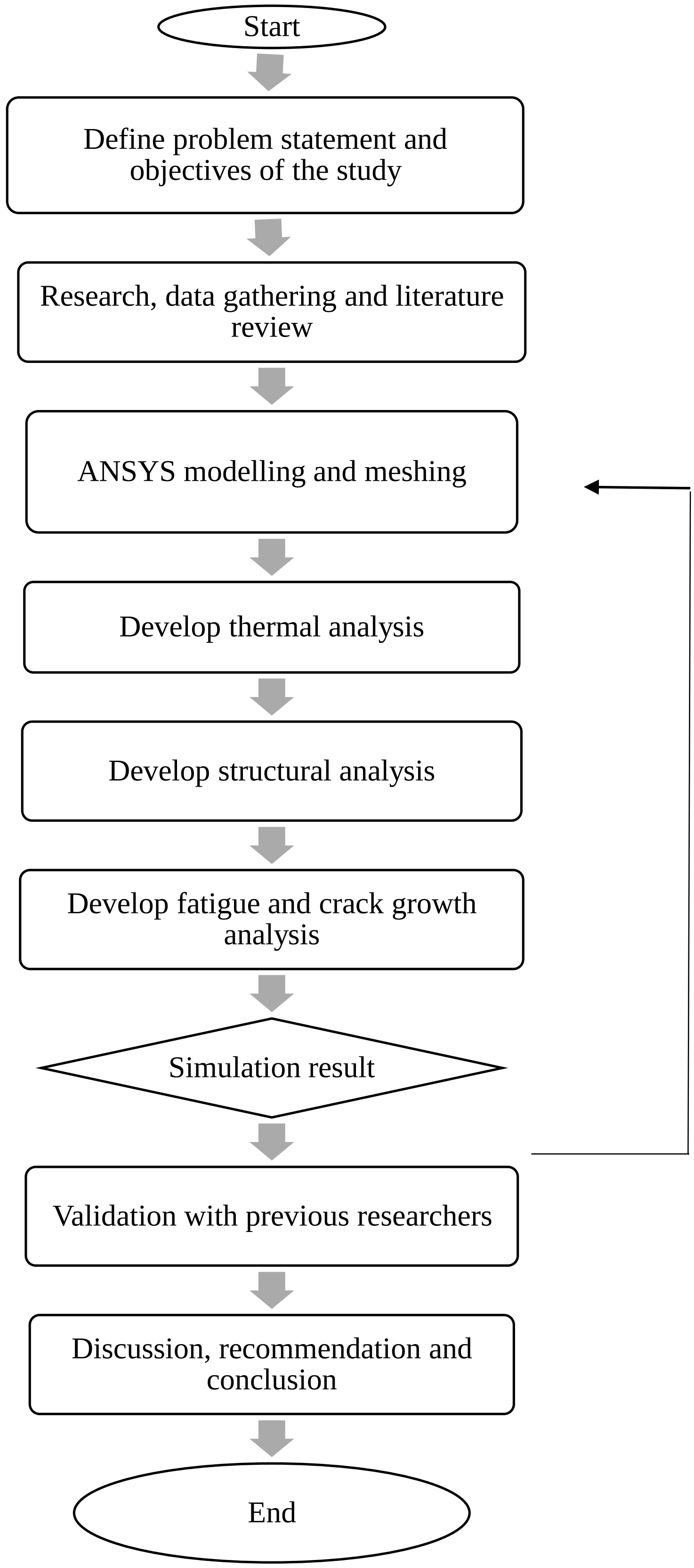


Figure 3.1: Project flowchart

Figure 3.1 shows the flowchart of this project. The results must be compared with previous research to ensure that the result is valid.

3.2 Finite Element Analysis

In finite element analysis, three steps will be involved:

3.2.1 Pre-processing phase

In this phase, several things need to be defined which are:

- Element types
- Material properties
- Model geometry
- Meshing controls

3.2.2 Applying boundary conditions

Boundary conditions are the information that need to be set to the model. There are three analysis that need to be done in this project. The boundary conditions for each analysis are:

i) Thermal Analysis:

- Ambient temperature
- Thermal conductivity
- Specific heat
- Convection surfaces
- Heat flux

ii) Structural Analysis:

- Temperature distribution
- Yield stress
- Young modulus
- Thermal expansion
- Density

iii) Fatigue and crack growth analysis

- Residual stresses
- Crack initiation
- Operational load

3.2.3 Post Processing

In post processing, several results need to be displayed which are:

- Shape and contour displays
- Tabular listings results
- Calculation results
- Error estimations

3.3 Develop simulation using ANSYS Software

3.3.1 Assumption

The assumptions that need to be defined before simulation are:

- Material properties of the filler and electrode are the same as the material properties of the base metal
- Initial temperature will be set to 300 K
- Radiation heat transfer is neglected
- Heat input is moving at a constant speed which is 5mm/sec
- Heat convection,

$$h_c = 15 \frac{W}{m^2 K} \tag{7}$$

3.3.2 Thermal Analysis

1) Define Element

The simulation start with thermal analysis. Firstly, in the ANSYS Main Menu, set the preferences as thermal. For the element, SOLID70 was used as the element type. SOLID70 is applicable to a 3 Dimension, steady state or transient thermal analysis. It contains eight nodes with a single degree of freedom at each node. Graphical user interphase method is the simulation tool provided in ANSYS main menu.

Graphical User Interphase Method:

Main Menu>Preprocessor>Element Type> Add/Edit/Delete

2) Define Properties of Materials

Table 3.1: Variation of properties of the material with temperature [6]

No	Temp (K)	Thermal Conductivity (W/m.K)	Specific Heat (J/kg.K)	Yield Stress (MPa)	Thermal Expansion Coefficient (x 10^{-5}/K)	Young's Modulus (GPa)	Density (kg/ m^3)
1	298	60	480	380	1.10	210	7880
2	373	50	500	340	1.15	200	7880
3	473	45	520	315	1.20	200	7800
4	673	38	650	230	1.30	170	7760
5	873	30	750	110	1.42	80	7600
6	1073	25	1000	30	1.45	35	7520
7	1273	26	1200	25	1.45	20	7390
8	1473	28	1400	20	1.45	15	7300
9	1673	37	1600	18	1.45	10	7250
10	1823	37	1700	15	1.45	10	7180

Table 3.1 shows the properties of ASTM A36 steel. These properties vary with temperature and need to be used in the Material Models.

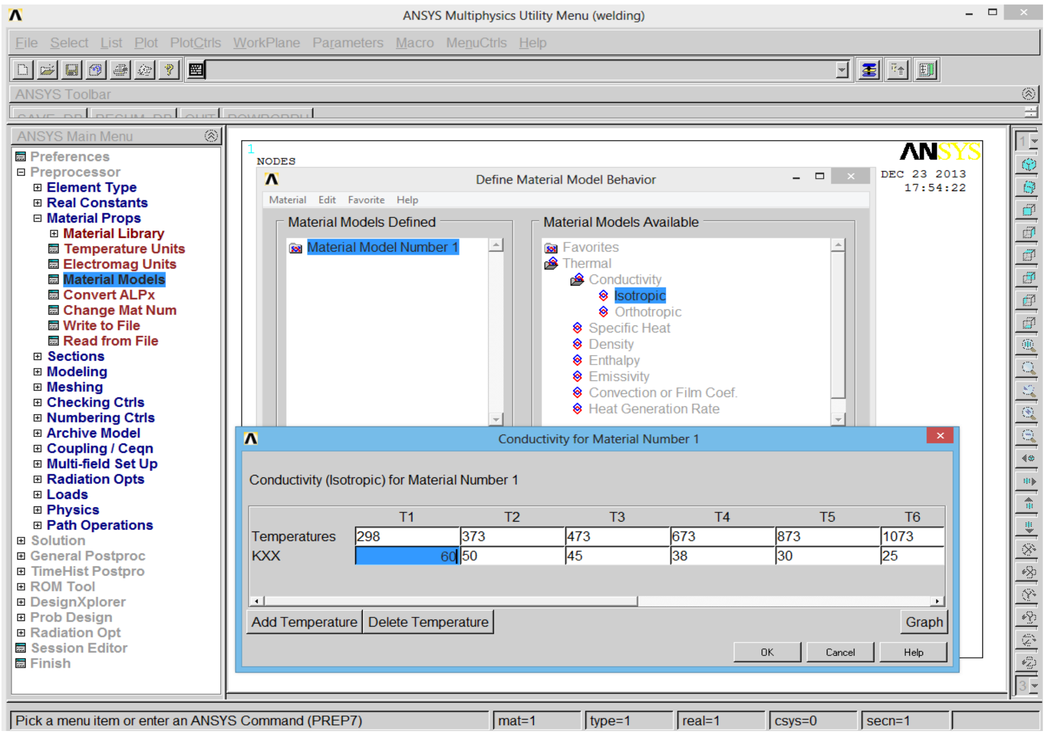


Figure 3.2: Define Material Properties

Figure 3.2 shows how the properties of material A36 is inserted into the material models.

Graphical User Interphase Method:

ANSYS Main Menu> Preprocessor>Material Props> Material Models.

3) Modelling

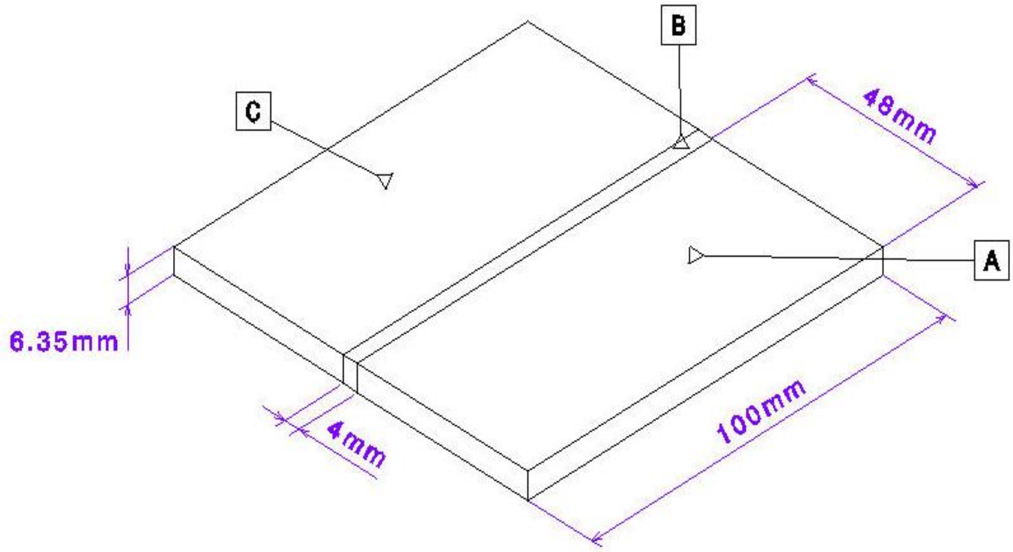


Figure 3.3: Geometry of the model

In Figure 3.3, there are three volumes for the model. A and C are the work piece of ASTM A36. B is the filler material. The model was created using key points, combine them into 3 volumes and glue it together.

4) Meshing

In finite element analysis, meshing is one of important step that need to be done. Any continuous object has infinite degree of freedom. By meshing in the finite element method, the infinite degree of freedom is reduced to finite degree of freedom by creating nodes and element in the model. The size of meshing will influence the accuracy of the results and it also depends on the type of analysis. In Figure 3.4, the size of element was set to 0.003m. Then, the volume is meshed using volume sweep.

Graphical User Interphase Method:

ANSYS Main Menu> Preprocessor> Meshing> Mesh> Volume Sweep> Sweep

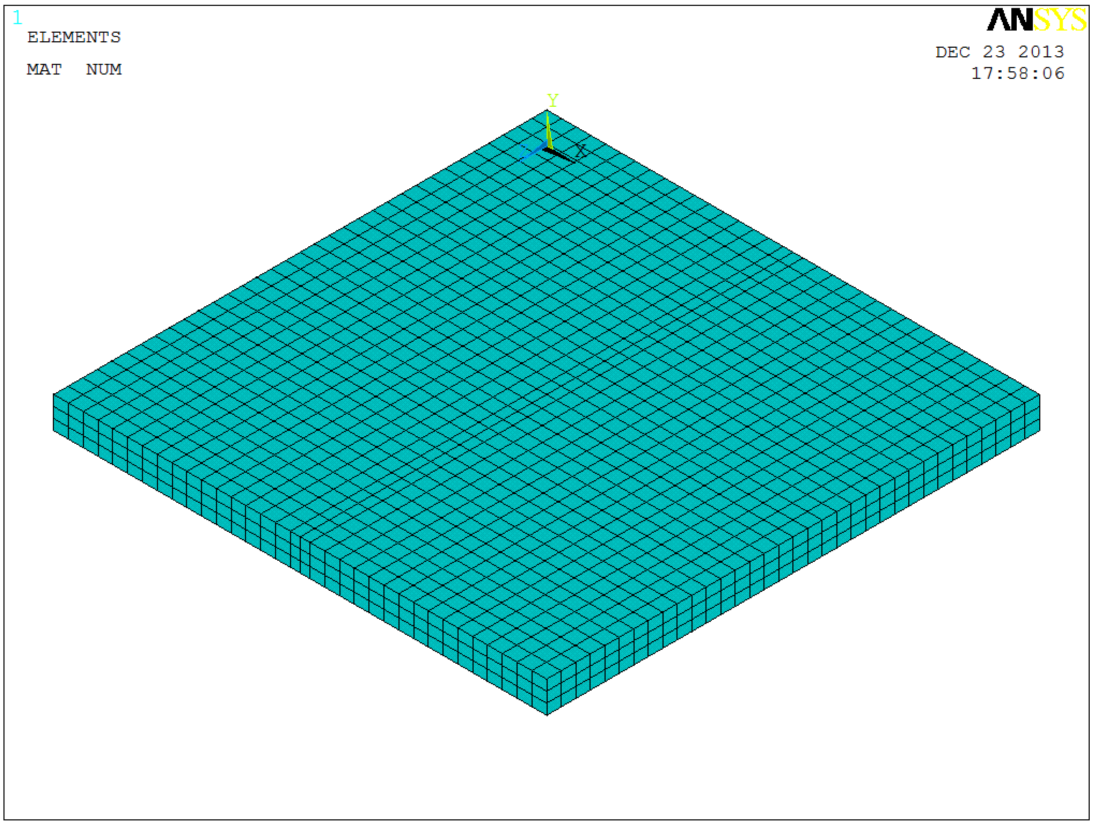


Figure 3.4 : Meshing the volume of the model

5) Setting the Analysis Type

Transient thermal analysis is set to the simulation. Later in the structural analysis, the temperature distribution from transient thermal analysis will be the input value. Newton-Raphson is applied as element killing in birth and death technique.

Graphical Interphase User Method:

ANSYS Main Menu> Preprocessor> Loads> Analysis Type> New Analysis> Transient> Full

6) Define initial constraint

To control the electrode travel speed, time of load step need to be set in solution control. Load step was set to 1 second to ensure the solution was done every 1 second at the end of each load step. On top of that, the initial temperature or ambient temperature was set as 300 K. The temperature is set because of the room temperature. For heat convection, it was applied on the surface of the plate. The value of heat convection was set to $h_c = 15\frac{W}{m^2K}$. In Figure 3.5 below, the heat convection is applied to the model.

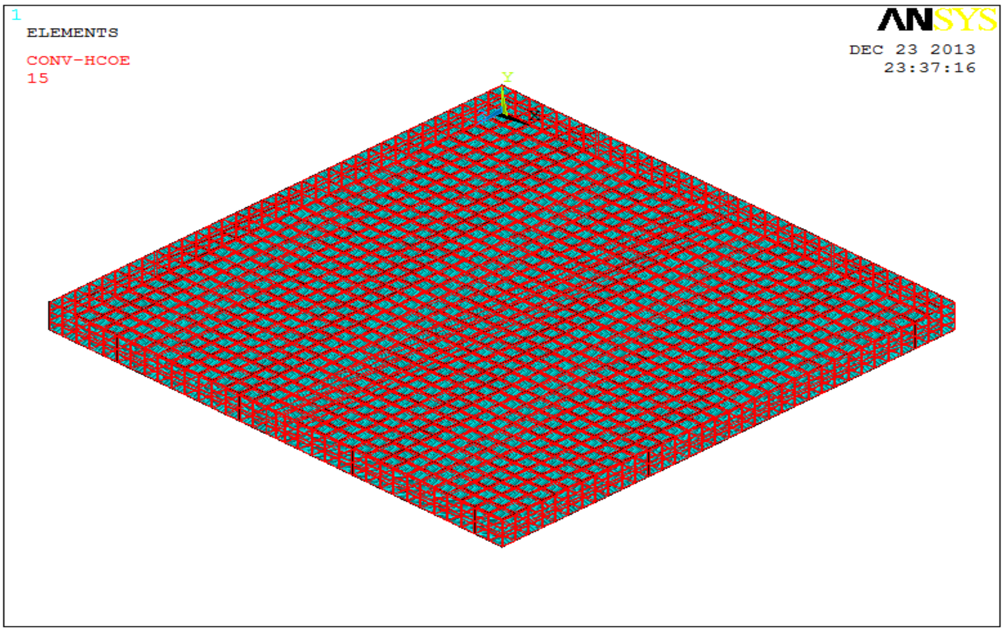


Figure 3.5: Applied heat convection

Graphical User Interphase Method:

Solution control:

ANSYS Main Menu> Solution> Analysis Type> Sol'n Controls

Uniform Temperature:

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

Convection:

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

7) Define Load

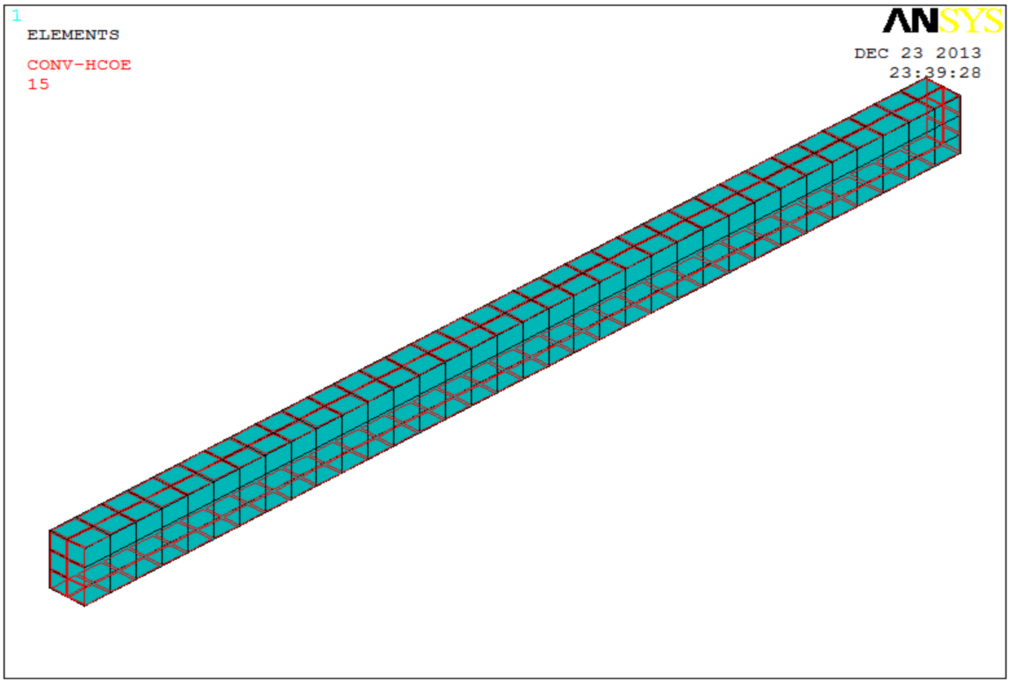


Figure 3.6: Applied heat flux

Figure 3.6 shows the application of heat flux to the filler material which is volume B. The heat flux was applied to volume B using birth and death technique. It was applied to the filler material in a certain amount of time. The heat flux would be allocated to each of the selected element so that weld heat input work as a single load at any given time. EKILL command was used to deactivate the element and EALIVE was used to activate back the element. In this simulation, three type of heat input was used as a parameter with different magnitude of current. The standard range of current that is suitable for this thickness are 110 A to 130 A [14]. High current will be applied to see the variation in the result.

Assume that the efficiency, η = 0.85, voltage = 24V, current = 120 A, welding travel speed = 0.005 m/s, and electrode diameter used = 0.004 m.

An example calculation of heat flux is:

$$Q = \eta \, \frac{V\,I}{vd}$$

$$= \frac{0.85\,(24 \times 120)}{(0.005 \times 0.004)} = 122\,400\,000 \text{ W/m}^2$$

Table 3.2: Heat flux

Current, A	120	160	200
Heat Flux, kW/m^2	122 400	163 200	204 000

Table 3.2 shows the information of three heat flux with different amount of current applied.

Graphical User Interphase Method:

ANSYS Main Menu> Solution> Define Loads> Apply> Thermal> Thermal> Heat Flux

8) <u>Solve</u>

To solve the simulation, current LS was used. Next was to read the result of temperature distribution on the contour plot at General Postproc.

Graphical User Interphase Method:

Solve:

ANSYS Main Menu> Solution> Solve> Current LS>

Result:

ANSYS Main Menu> General Postproc> Plot Result> Contour Plot

3.3.3 Structural Analysis

The aim of structural analysis was to get the variation of residual stresses after welding has been done in the thermal analysis. The temperature distribution data from thermal analysis was applied as the input to structural analysis. The residual stresses will be compared for different heat flux.

1) <u>Switch element type</u>

The type of element will be switched from thermal to structural. Solid 185 was used as the element because it was applicable in 3 dimensional modelling of solid structures. The element contain eight nodes with three degrees of freedom at each node. Next step was to define the properties for structural analysis which are Young's Modulus, Poisson Ratio and yield stress.

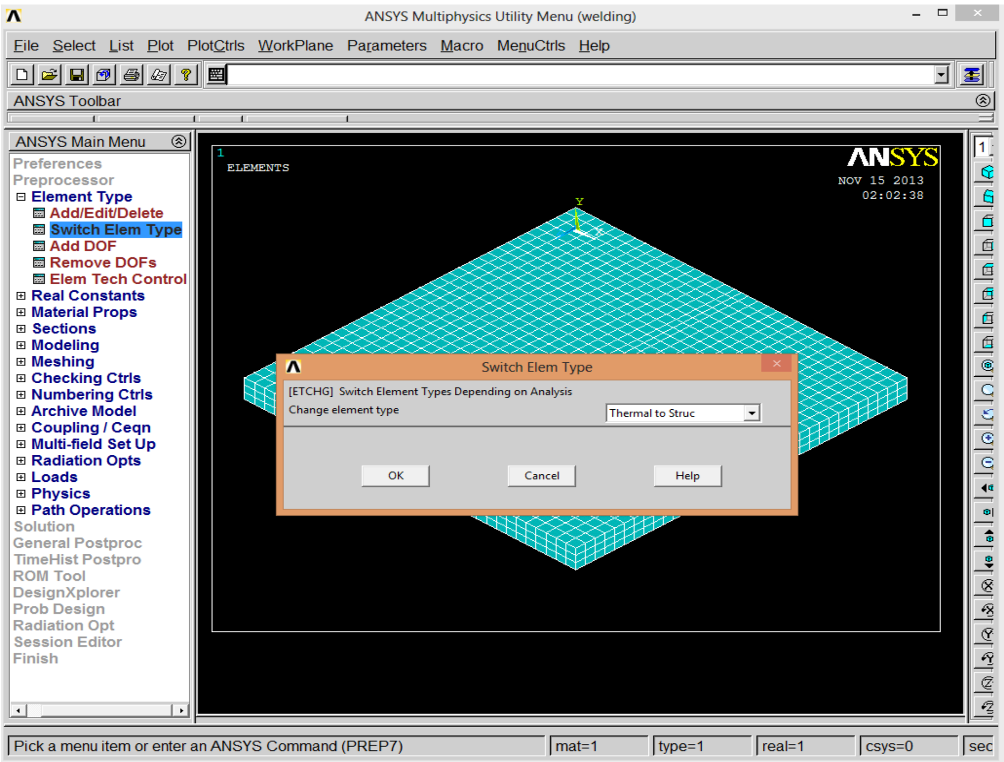


Figure 3.7: Switch element type

Figure 3.7 shows how to switch the element type from thermal to structural.

Graphical User Interphase Method:

Switch Element Type:

ANSYS Main Menu> Preprocessor> Element Type> Switch Element Type

Define Element:

ANSYS Main Menu> Preprocessor>Material Props> Material Models.

2) <u>Specify Initial Condition</u>

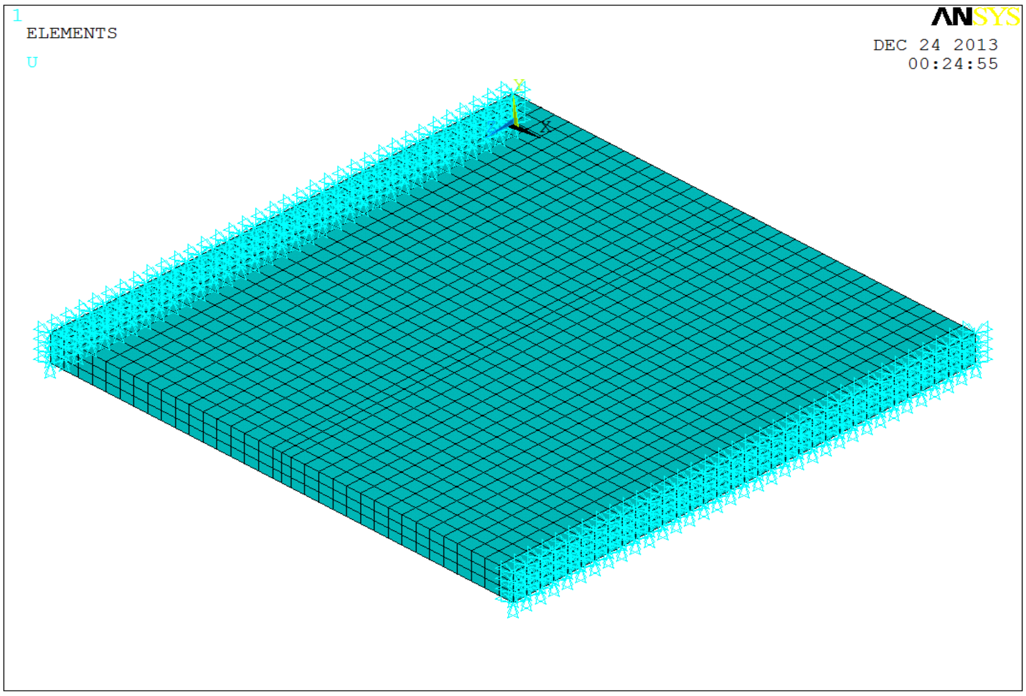


Figure 3.8: Clamped work pieces

Zero displacement is applied as shown in Figure 3.8. Selected nodes were chosen at each side and was set as zero displacement to indicate that both sides of plates were assumed to be clamped. Besides, both plates will have constant displacement. Next was to apply the temperature from thermal analysis. The data was taken from .nth file in ANSYS folder.

Graphical User Interphase Method:

Displacement:

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

Apply Temperature:

ANSYS Main Menu> Solution> Define Loads> Apply> Structural> Temperature> From Thermal Analysis

3) Solve and Obtain Result

After the temperature has been applied, the simulation is ready to be solved. Last step was to get the result for contour plot and plot the graph of residual stresses.

Solve:

ANSYS Main Menu> Solution> Solve> Current LS>

Result:

ANSYS Main Menu> General Postproc> Plot Result> Contour Plot

3.3.4 Fatigue Crack Growth Analysis

To investigate the effect of welding residual stresses on fatigue behaviour, operational load with and without residual stress was applied. This analysis is done by using Paris Law to calculate the fatigue life of the materials. The materials was tested with range of operating load from 100 MPa to 500 MPa.

Next, each stress of operating load will be added with the maximum residual stresses from three heat flux to see the pattern of fatigue curve. Last step was to obtain the number of cycle and to compare with operating load. Table 3.3 shows the number of cycle with different load.

Table 3.3 : Number of cycle with different load

Operational load (MPa)	Number of Cycles to failure for load 1	Load A (MPa)	Number of Cycles to failure for load A	Load B (MPa)	Number of cycles to failure for load B	Load C (MPa)	Number of cycles to failure for load C
500		762.5		794		799.3	
450		712.5		744		749.3	
400		662.5		694		699.3	
350		612.5		644		649.3	
300		562.5		594		599.3	
250		512.5		544		549.3	
200		462.5		494		499.3	
150		412.5		444		449.3	
100		362.5		394		399.3	

Load 1, A, B and C are defined as:

- Load 1 = Operational load without residual stresses.
- Load A = Operational load added with maximum residual stresses from heat flux 1.
- Load B = Operational load added with maximum residual stresses from heat flux 2.

- Load C = Operational load added with maximum residual stresses from heat flux 3.

There are few assumptions and parameters that need to be used to analyse this analysis:

- Fracture toughness of ASTM A36, K_{1C} = 62.6 MPa$\sqrt{m}$
- Constant C = 1.36 x 10^{-10} and m = 2.25
- Geometry dimensionless parameter, Y = 1.12
- Initial crack is 0.1 mm

Sample calculation

i. *Operating load without any residual stresses.*

The load used without any residual stresses was between 100 MPa to 500 MPa.

From Paris Law,

$$\frac{da}{dN} = C\Delta K^m$$

$$\frac{da}{dN} = 1.36 \times 10^{-10}\Delta K^{2.25}$$

In order to calculate the cyclic life, the critical crack size was determined.

Critical crack size, a_c :

$$K_{1c} = \Delta\sigma_{max} Y\sqrt{\pi a_c}$$

$$a_{c=}(\frac{K_{1C}}{Y\sigma_{max}\sqrt{\pi}})^2 = (\frac{62.6\ MPa\ \sqrt{m}}{1.12\ \times 500\ MPa\ \sqrt{\pi}})^2$$

a_c=3.978 X 10^{-3} m

a_c=3.978 mm

Hence, the limits on the Paris Law integration were between 0.1 mm to 3.978 mm. The first operational load was 500 MPa.

Rearranging the Paris Law gives:

$$\frac{da}{dN} = C\Delta K^m = C(Y\Delta\sigma\sqrt{\pi a})^m$$

$$\int_0^{Nf} dN = \int_{ai}^{af} \frac{da}{CY^m\Delta\sigma^m(\pi a)^{\frac{m}{2}}}$$

Then, the required life cycle can be obtained from the previous equation.

$$N_f = \frac{1}{CY^m\Delta\sigma^m\pi^{\frac{m}{2}}}\left(\frac{a_f^{1-\frac{m}{2}} - a_i^{1-\frac{m}{2}}}{1-\frac{m}{2}}\right)$$

$$= \frac{1}{1.36\times10^{-10}\times\ 1.12^{2.25}\times 500MPa^{2.25}\times\pi^{1.13}}\left(\frac{3.978mm_f^{-0.13} - 0.1mm_i^{-0.13}}{-0.13}\right)$$

N_f = 5220 cycles

ii. Operating load with residual stresses of first heat flux.

For the second test, the residual stresses from first heat flux was added to the operational load. The stresses range was between 112.5 MPa to 762.5 MPa.

From Paris Law,

$$\frac{da}{dN} = C\Delta K^m$$

$$\frac{da}{dN} = 1.36\ \times\ 10^{-10}\Delta K^{2.25}$$

In order to calculate the cyclic life, the critical crack size was determined.

Critical crack size, a_c :

$$K_{1c} = \Delta\sigma_{max} Y \sqrt{\pi a_c}$$

$$a_{c=}(\frac{K_{1C}}{Y\sigma_{max}\sqrt{\pi}})^2 = (\frac{62.6\ MPa\ \sqrt{m}}{1.12 \times 762.5\ MPa\ \sqrt{\pi}})^2$$

a_c = 1.71 X 10^{-3} m

a_c = 1.71 mm

Hence, the limits on the Paris Law integration were between 0.1 mm to 1.71 mm.

Rearranging the Paris Law gives:

$$\frac{da}{dN} = C\Delta K^m = C(Y\Delta\sigma\sqrt{\pi a})^m$$

$$\int_0^{Nf} dN = \int_{ai}^{af} \frac{da}{CY^m\Delta\sigma^m(\pi a)^{\frac{m}{2}}}$$

Then, the required life cycle can be obtained from the previous equation.

$$N_f = \frac{1}{CY^m\Delta\sigma^m\pi^{\frac{m}{2}}}(\frac{a_f^{1-\frac{m}{2}} - a_i^{1-\frac{m}{2}}}{1-\frac{m}{2}})$$

$$= \frac{1}{1.36 \times 10^{-10} \times\ 1.12^{2.25} \times 762.5\ MPa^{2.25} \times \pi^{1.13}}(\frac{1.71mm_f^{-0.13} - 0.1mm_i^{-0.13}}{-0.13})$$

N_f = 1638 cycles

3.4 Gantt chart

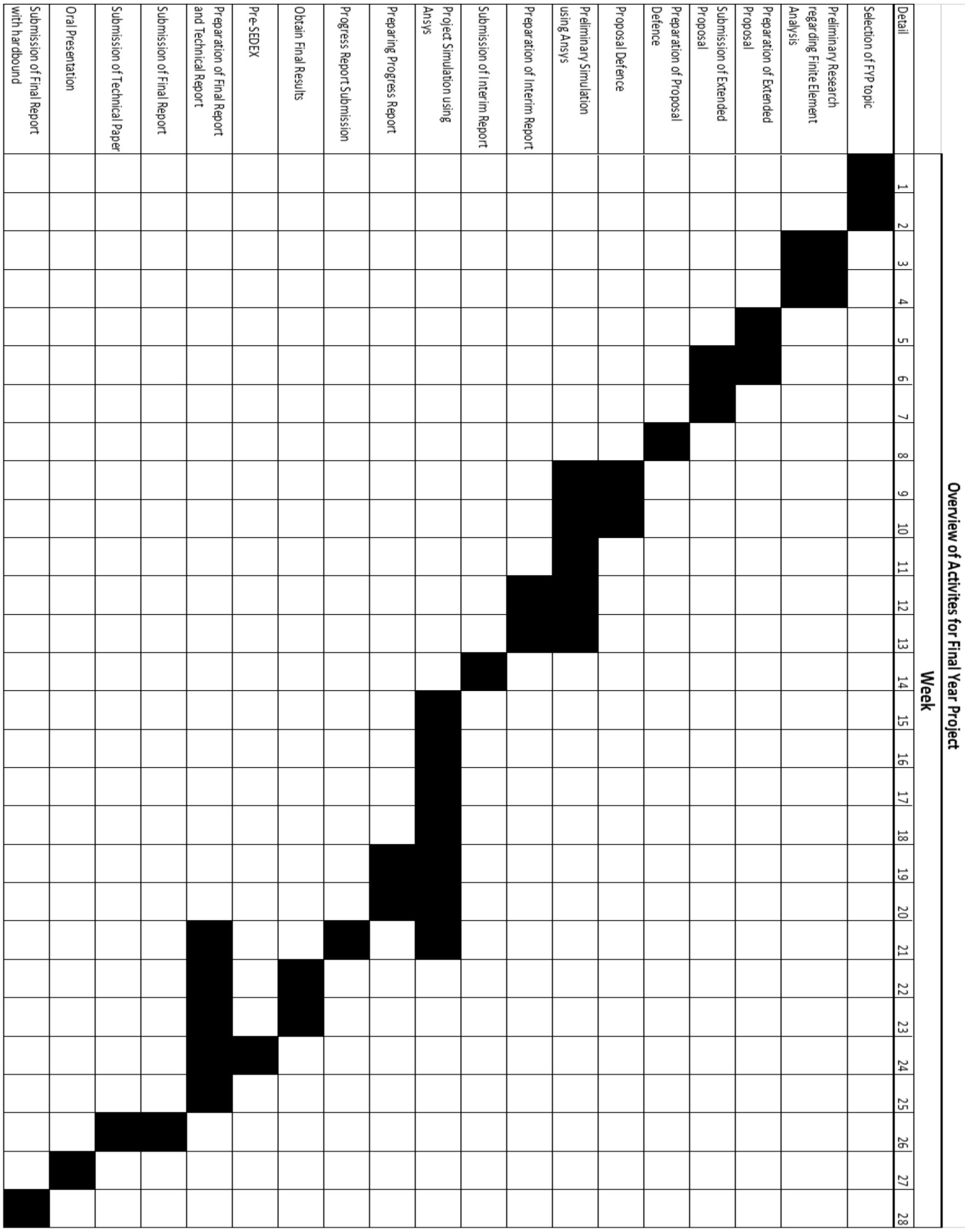


Figure 3.9: Gantt chart for 28 weeks

Figure above show the gantt chart of the project. The duration to complete this project took about 28 weeks.

3.5 Key milestone

Table 3.2: Key milestone

No.	Milestone	Date
1	Submission of Extended Proposal	June 2013
2	Proposal Defense	July 2013
3	Preliminary Simulation	August 2013
4	Data Collection	September 2013
5	Submission of Progress Report	October 2013
6	Final Results of Simulation	October 2013
7	Pre-SEDEX	November 2013
8	Submission of Final Report (softcopy)	November 2013
9	Submission of Technical Report	November 2013
10	Oral Presentation	December 2013
11	Submission of Final Report (hardbound)	December 2013

Table 3.2 shows the key milestone of this project. The project will finish in December 2013.

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Results of thermal Analysis

Three different magnitude of heat flux, 122.4 MW/m^2, 163.2 MW/m^2 and 204 MW/m^2, were used in this research and they were varied based on the current applied.

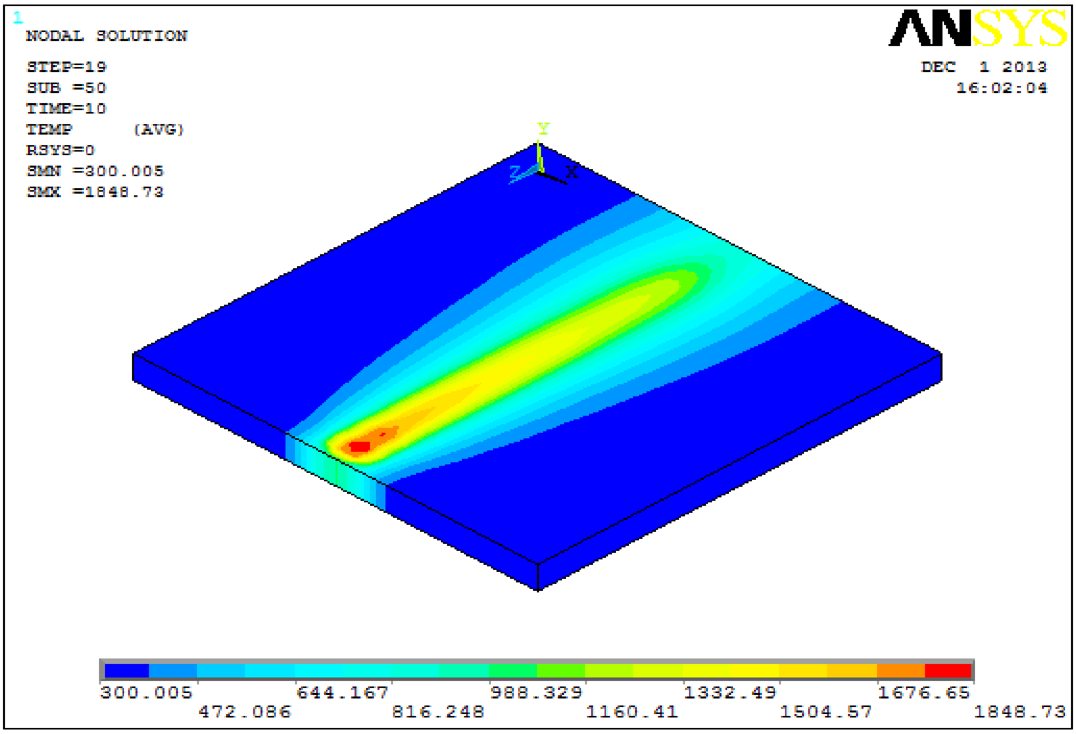


Figure 4.1 : Temperature distribution of first heat flux

The temperature distribution of the ASTM A36 plate after welding is shown in Figure 4.1. The result was taken from contour plot after welded with 122.4 MW/m^2 of heat flux by using 120 A current. The maximum temperature was 1848.73 K and it exceeded the melting point of this material, 1673 K. The temperature obtained indicated that the filler melted and the temperature distribution was taken at last step of welding. It took 10 seconds to finish this welding.

Table 4.1 : Tabulation of temperature distribution data

Distance (m)	Temperature for heat flux 1 (K)	Temperature for heat flux 2 (K)	Temperature for heat flux 3 (K)
0.000	300.00	300.00	300.00
0.005	300.00	300.00	300.00
0.010	300.02	300.04	300.09
0.015	301.00	300.50	300.17
0.020	302.00	302.70	300.40
0.025	304.00	306.00	309.00
0.030	307.00	310.00	311.00
0.035	355.00	373.00	405.00
0.040	531.50	520.00	508.51
0.045	1229.73	1310.00	1829.57
0.050	1848.73	2004.00	2306.00
0.055	1800.37	2163.12	2485.09
0.060	950.88	1231.56	1174.05
0.065	420.50	424.00	427.00
0.070	310.00	306.00	304.00
0.075	304.00	303.00	302.00
0.080	303.00	303.00	302.00
0.085	301.10	301.00	300.00
0.090	300.00	300.00	300.00
0.095	300.00	300.00	300.00
0.100	300.00	300.00	300.00

Data for temperature distribution is shown in Table 4.1. The third heat flux, 204.4 MW/m^2 gave the highest value of temperature which was 2306 K. While the first heat flux, 122.4 MW/m^2 resulted in smallest temperature which was 1848.73 K. This indicate that the higher the heat flux applied, the higher the temperature distribution. The temperature is high at the distance from 0.045 m until 0.05 m, which was at the filler material located between the two plates.

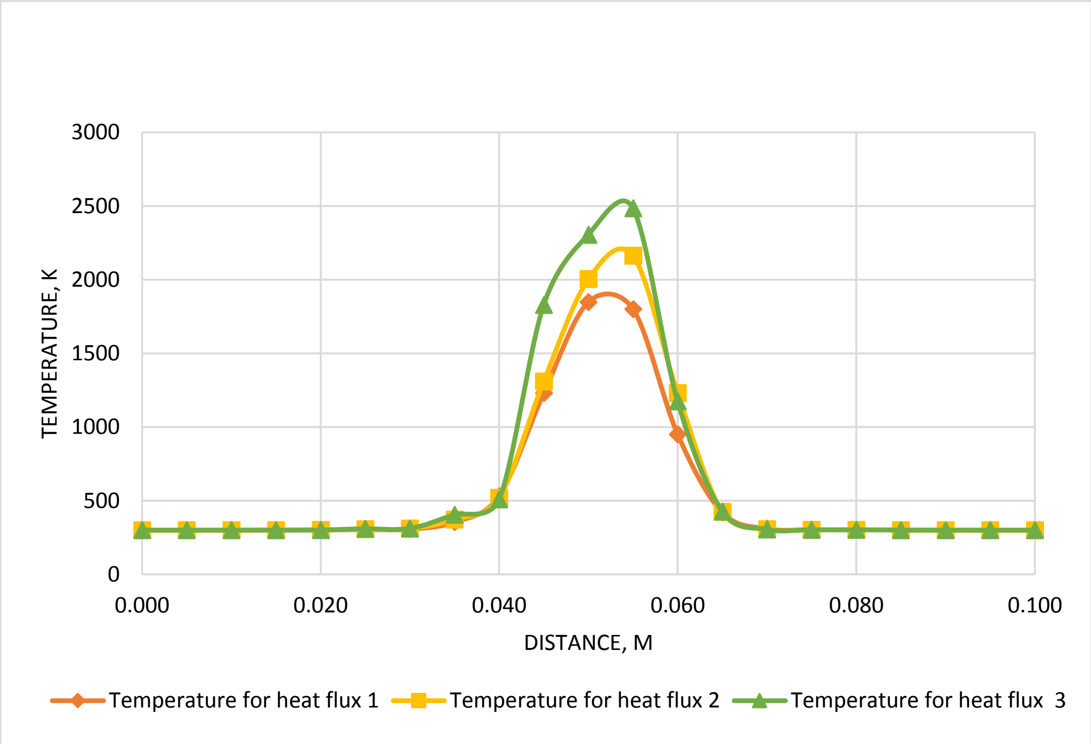


Figure 4.2 : Graph of temperature distribution versus distance

Tabulation data of three temperature distribution was plotted as in Figure 4.2. From the graph, it is clearly stated that the temperature is higher in the middle between two plates where molten material, filler was used to joint the plates. M. J. Park, et al [8] stated that the temperatures varied around the heat affected zone (HAZ). This zone experienced high temperature, which cause phase transformation in mechanical properties of the materials.

4.2 Results of Structural Analysis

Results from thermal analysis were used in structural analysis to find the residual stresses.

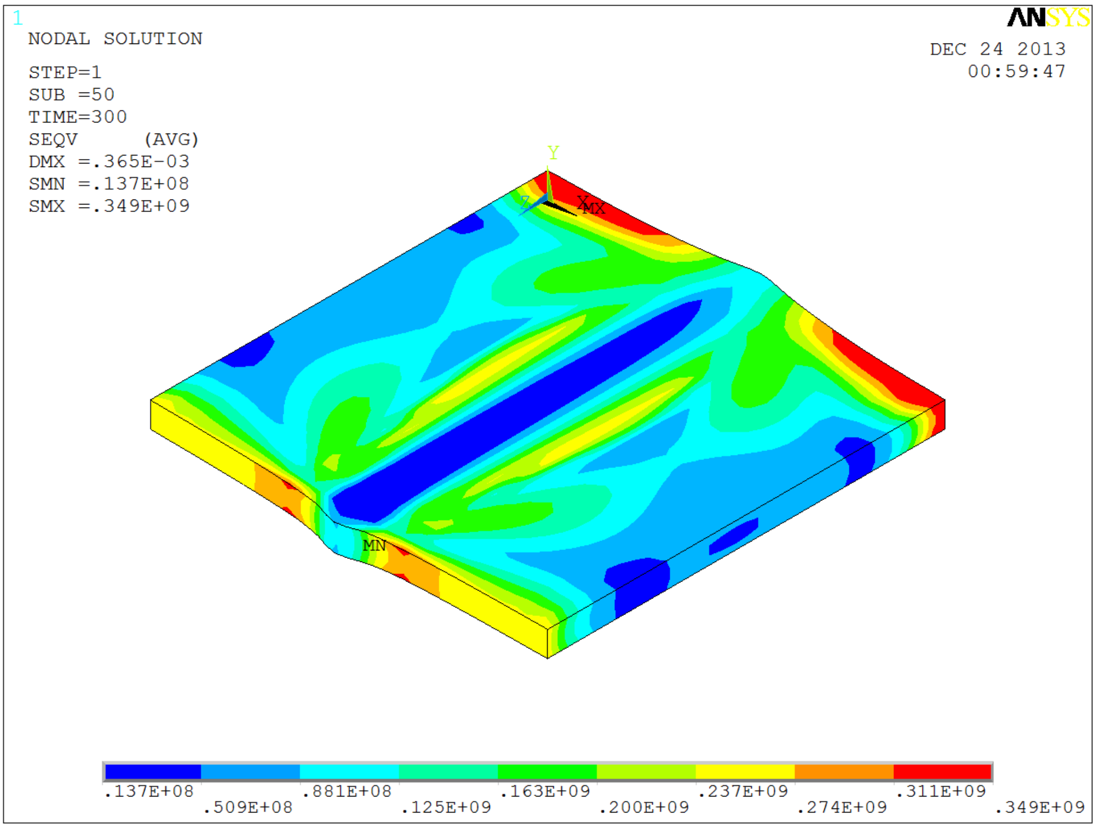


Figure 4.3: Von Mises Stress of first heat flux

Figure 4.3 shows the von Mises stress for heat flux of 122.4 MW/m^2 with current of 120 A. The contour plot showed the formation of distortion along the heat affected zone (HAZ). The results were taken after 300 seconds to ensure that the welded metal is cooled down. Value of the stresses is taken at selected nodes across the middle of the plates. The maximum stresses plotted was 262.5 MPa. High residual stresses were accumulated near the centre of the plate. This result is consistent with the investigation of residual stresses by Chang, K., & Lee, C., (2009) [3] that stated that the stresses were distributed when the thermal load from welding was cooled down and the stresses spread from the welding site. In this case, the maximum stresses value has exceeded the yield strength of the material, 250 MPa and thus resulted in plastic deformation.

Table 4.2: Tabulation data of residual stresses from three different heat flux

Distance (m)	Residual Stresses for heat flux 1 (MPa)	Residual Stresses for heat flux 2 (MPa)	Residual Stresses for heat flux 3 (MPa)
0.000	54.84	59.49	61.30
0.005	52.00	51.20	50.50
0.010	57.00	57.44	58.00
0.015	69.20	73.32	77.00
0.020	93.00	87.10	85.00
0.025	82.50	58.10	46.40
0.030	46.20	75.70	107.60
0.035	262.50	294.08	310.43
0.040	190.30	138.50	90.00
0.045	67.13	31.90	25.00
0.050	21.75	18.13	14.90
0.055	21.75	18.13	14.90
0.060	69.92	45.72	24.00
0.065	214.29	156.13	96.00
0.070	262.00	292.08	308.40
0.075	45.20	88.40	109.00
0.080	91.30	73.30	42.00
0.085	93.50	87.10	84.00
0.090	56.00	58.40	62.00
0.095	53.00	51.70	50.00
0.100	54.00	58.50	63.00

From Table 4.2, the first heat flux gave the lowest residual stress which was 262.5 MPa. Whereas third heat flux gave the highest residual stress which was 310.43 MPa. The results illustrated that higher heat flux results in higher residual stresses.

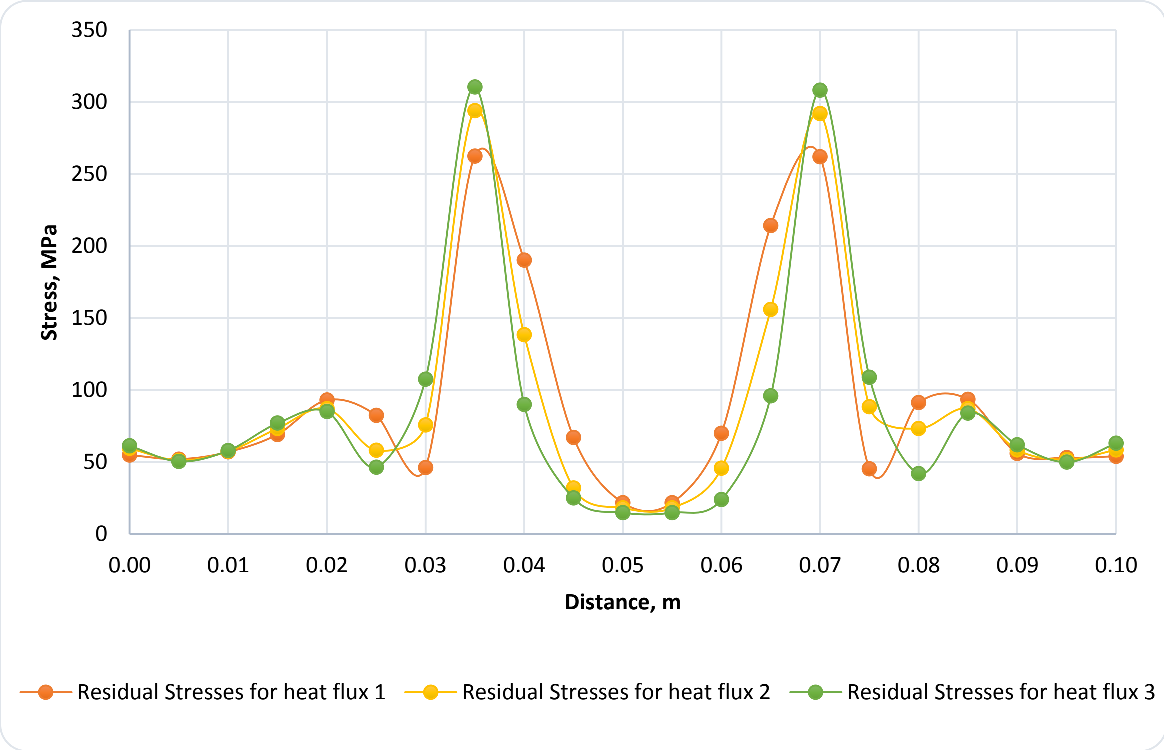


Figure 4.4: Graph of residual stresses for three different heat flux

Graph of the model of residual stresses is shown in figure 4.4. From the graph, it can be seen that the stress is low at the center which is at the welded area. This is due to thermal load that distribute around the middle of the plate. According to Jeyakumar, M., Christoper, T., Narayanan, R., & Rao B. N., (2011) [6] the stresses are higher at heat affected zone (HAZ) because the residual stress and distortion is caused by metal deformation in HAZ area of the weld joints.

Based on figure 4.4, it is conclude that high heat flux gives high residual stresses. This is because, high heat flux makes the welded metal to experience high temperature distribution in heat affected zone (HAZ).

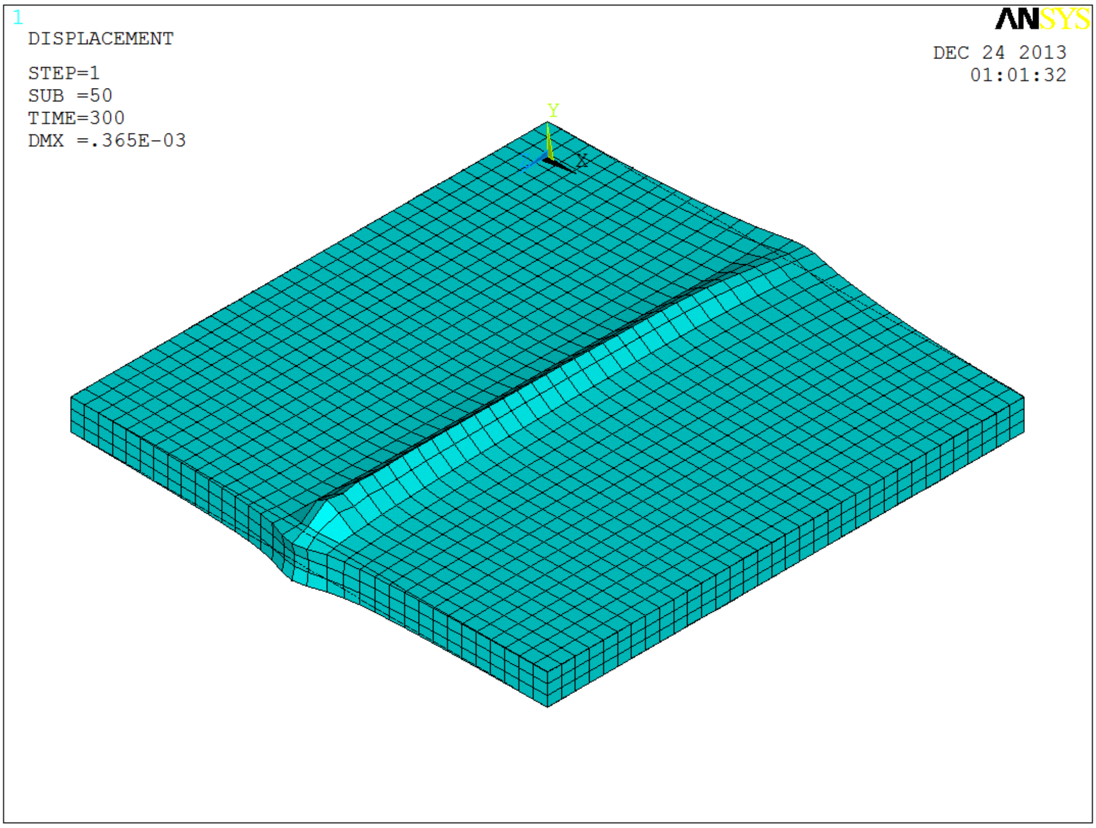


Figure 4.5: Deformation of materials in third heat flux

Figure 4.5 shows the highest distortion of 3.65 X 10^{-4} m. However the distortion is small in value and can be removed through proper surface finishing. Welding distortion and deformation after welding is natural because of non-uniform heating and cooling of the joint. Most deformation occurred at welded area and some of it in heat affected zone (HAZ). Another reason for material deformation was also due to high thermal load acting on the surface of the materials. Distortion indicated the stresses accumulation towards the outer side of plate. The distortion formed the weld beads at the middle of welded line.

Table 4.3: Summary of thermal and structural analysis results

Current (A)	120	160	200
Heat Flux (MW/m^2)	122.4	163.2	204
Max Temperature (K)	1848.73	2163.12	2484.06
Min Temperature (K)	300	300	300
Max Stresses (MPa)	262.5	294	310.3
Min Stresses (MPa)	21.7	18.1	14.9
Deformation (m)	7.43×10^{-6}	4.91×10^{-5}	3.65×10^{-4}

From table 4.3, three heat flux values were applied in the welding simulation to see the variation of residual stresses. Current with three different magnitudes were also varied in this experiment. From the results, it can be seen that the temperature exceeded the melting point of the filler material. So, all the heat flux can be used to weld the two plate.

The third heat flux generated highest temperature, stresses and deformation. This was because, it generated the most thermal load during the welding and thus gave high residual stresses after the materials were cooled to the ambient temperature. On the other hand, more deformation occurred due to excessive residual stresses. In summary, as the heat flux increased, the values of temperature, residual stresses and deformation were also increased.

4.3 Results of Fatigue Crack Growth Analysis

Fatigue crack growth analysis was done by using the Paris Law equation. The result for the remaining number of cycle based on the different load is as shown in Table 4.4. The effect of welding residual stresses on fatigue behaviour was investigated based on operational load with and without residual stress.

Table 4.4: Number of cycles to failure based on operational load

Operational load 1 (MPa)	Number of Cycles to failure for load 1	Load A (MPa)	Number of Cycles to failure for load A	Load B (MPa)	Number of cycles to failure for load B	Load C (MPa)	Number of cycles to failure for load C
500	5220	762.5	1638	794	1460	799.3	1433
450	6616	712.5	1908	744	1691	749.3	1657
400	8624	662.5	2248	694	1977	699.3	1936
350	11647	612.5	2682	644	2339	649.3	2287
300	16475	562.5	3248	594	2806	599.3	2739
250	24830	512.5	4005	544	3419	549.3	3332
200	41023	462.5	5046	494	4248	499.3	4130
150	78369	412.5	6527	444	5401	449.3	5237
100	195141	362.5	8730	394	7066	399.3	6829
-	-	312.5	12191	344	9589	349.3	9228
-	-	262.5	18047	294	13654	299.3	13064
-	-	212.5	29032	244	20769	249.3	19710
-	-	162.5	53090	194	34793	199.3	32615
-	-	112.5	121435	144	68034	149.3	62470

Load 1, A, B and C are defined as:

- Load 1 = Operational load without residual stresses.
- Load A = Operational load added with maximum residual stresses from heat flux 1
- Load B = Operational load added with maximum residual stresses from heat flux 2
- Load C = Operational load added with maximum residual stresses from heat flux 3

The stress range of operational load was between 100 MPa to 500 MPa. The operational load without residual stresses experienced larger number of cycles to failure which was 195141 cycles. The lowest cycle to failure was 1433 cycles when operating or cyclic load of 149.3 MPa to 799.3 MPa was applied to the welded material.

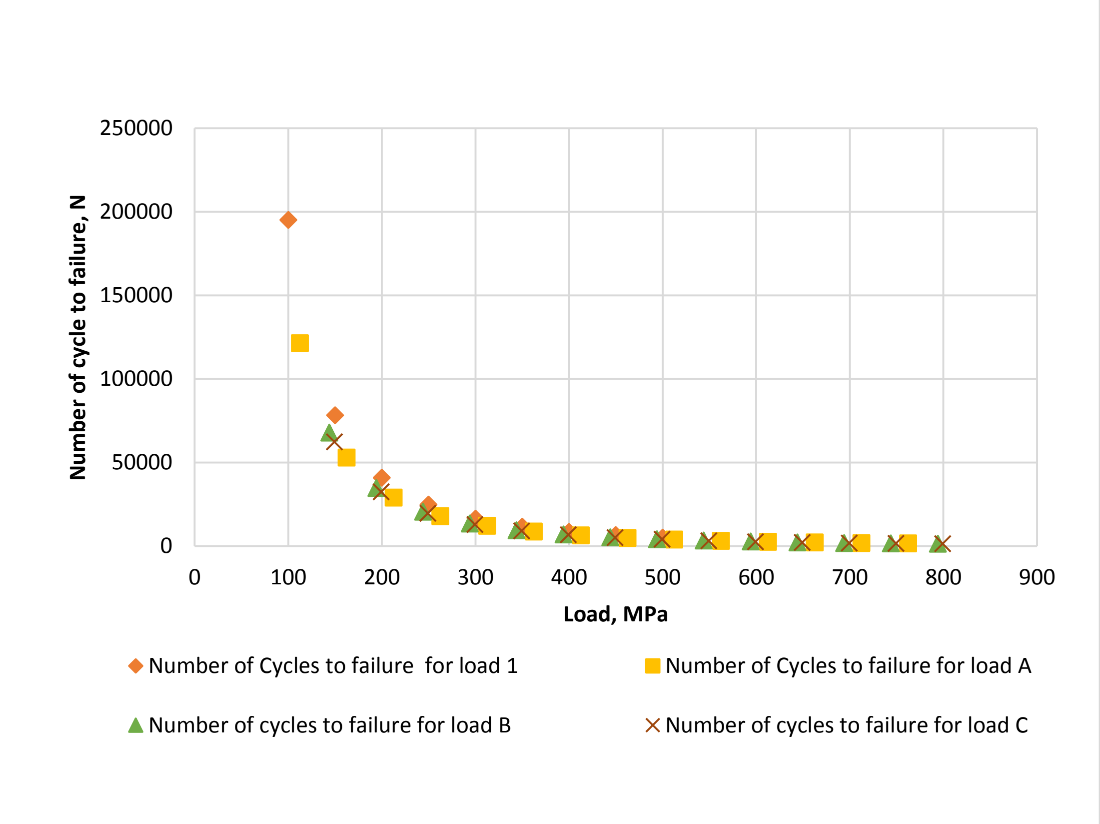


Figure 4.6: Graph number of cycles to failure versus load

Figure 4.6 shows the graph number of cycle versus load. We can see that fatigue damage is clearly affected by operational load or cyclic stresses. The pattern clearly describe that at lower cyclic load, number of cycles to failure will be high.

Table 4.5: Data of crack length with different number of cycles to failure

Number of Cycles for load 1	Crack length for load 1 (mm)	Number of Cycles for load A	Crack length for load A (mm)	Number of cycles for load B	Crack length for load B (mm)	Number of cycles for load C	Crack length for load C (mm)
5220	0.108	1638	0.103	1460	0.105	1433	0.105
6616	0.111	1908	0.104	1691	0.106	1657	0.106
8624	0.114	2248	0.105	1977	0.107	1936	0.107
11647	0.119	2682	0.105	2339	0.108	2287	0.109
16475	0.129	3248	0.107	2806	0.110	2739	0.111
24830	0.146	4005	0.108	3419	0.112	3332	0.113
41023	0.190	5046	0.110	4248	0.116	4130	0.117
78369	0.358	6527	0.114	5401	0.120	5237	0.122
195141	3.978	8730	0.119	7066	0.128	6829	0.129
-	-	12191	0.127	9589	0.140	9228	0.142
-	-	18047	0.143	13654	0.162	13064	0.165
-	-	29032	0.180	20769	0.210	19710	0.215
-	-	53090	0.305	34793	0.362	32615	0.371
-	-	121435	1.710	68034	1.577	62470	1.557

Table 4.5 gives information on crack length. The crack length was identified for different number of cycle. The largest crack length is 3.978 mm. Meanwhile, the smallest crack length is 0.105 mm.

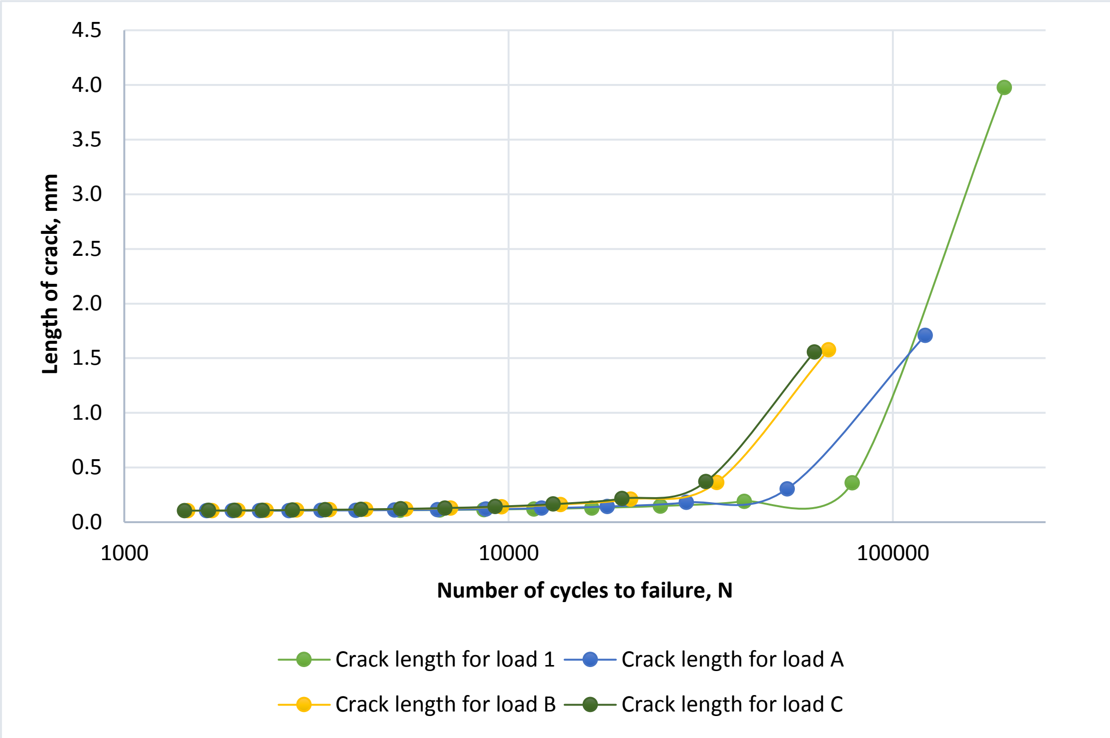


Figure 4.7: Graph of crack length versus number of cycles to failure.

Figure 4.7 shows the crack length based on number of cycle. The basic operational load gives highest number of cycles which was 195141. It was because, the cyclic load acting on material was low, so the number of cycle to failure will be high.

The third heat flux that gave maximum residual stresses has lower number of cycle to failure. As a result, the time to failure for the material will be fast. In a nutshell, the results obtained followed the theory by Paris, P., & Erdogan F. (1963) [17] where as the number of cycle decreased, the crack growth will be decreased.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

As the conclusion, the objectives of this project have been achieved which were to determine the variation of residual stresses as well as to estimate the fatigue crack growth due to residual stresses.

Based on thermal analysis, the temperature was higher at the middle of two plates where molten material, filler was used to joint the plates. The temperature distribution varied along the heat affected zone (HAZ). Thermal analysis also concluded that the higher the heat flux, the higher the temperature experienced by welded material.

From structural analysis, high magnitude of residual stresses accumulated near the center of the plate which was at the HAZ. The stresses were distributed when the thermal load from welding was cooled down to ambient temperature. Residual stresses also caused deformation on the plate. When the maximum stresses value has exceeded the yield strength of the material, plastic deformation occurred. In structural analysis, it can be concluded that as the heat flux increased, the residual stresses and deformation were also increased.

Last but not least was the fatigue crack growth analysis. At lower cyclic load, number of cycles to failure will be high. This illustrated that the higher the number of cycles to failure, the longer time for fatigue damage to occur. Meanwhile, crack growth was high when the number of cycles high.

5.2 Recommendation

Molzen, M, S., and Hornbach D. (2000) has concluded in their study that welding residual stress can be treated through shot peening and heat treatment. Heat treatment can be done to the welded materials to relieve the stresses by reducing hardness and increase ductility. Thus residual stresses can be relieved from heat treatment. Shot peening is used to produce a compressive stress layer from cold work process. The surface of material is shot with force to create plastic deformation. The compressive stress layer from shot peening will resist the fatigue crack growth from developing in material. Crack growth will be delayed when shot peening is applied.

American Welding Society (AWS) has created a manual to give information on the requirement before welding. Selection of electrode is important in welding to get a similar composition to parent metal. In term of size, the smaller diameter of electrode is recommended for thin type of plate. Correct current selection is an important factor in welding. If the current is set too low, it will be difficult to strike and maintain the stability of arc. The electrode tend to stick to the work piece and thus result in poor penetration and formation of beads. The type of joint is also crucial in welding. Square butt weld is suitable for plate up to 6 mm in thickness as implemented in this experiment. However, for thickness up to 10 mm and above, the suitable type of joint is single V-groove butt. This is because, the shape can make the filler to grip more on the welded plate.

Further study on the simulation of welding need to be improved. Using ANSYS as the simulation software is the best selection because this option is inexpensive and save time. However, many assumptions need to be reconsidered. It plays an important role to get more accurate results. More and detailed analysis can be done to get as much information on the effect of welding. Error during the simulation also need to be avoided. The heat flux override an element of 1643. Lastly, the results must also be validated with other research to see the comparison of results and get to know which part that need to be improved.

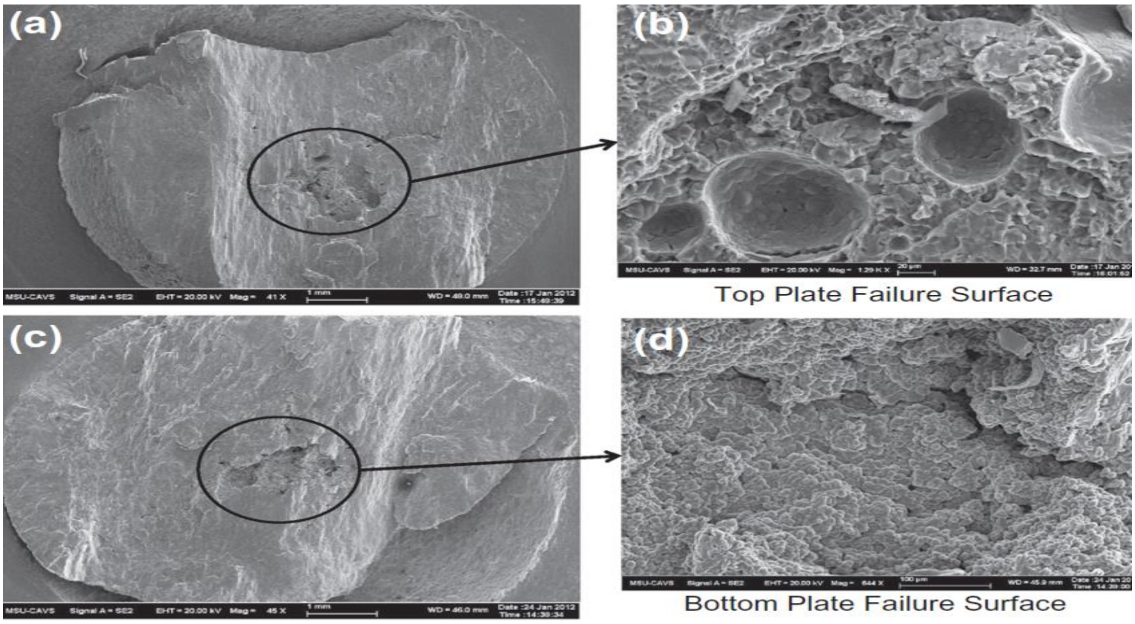
REFERENCES

1. Abid M., & Ullah S. (2012), "Investigation of residual stresses and distortion in welded pipe-flange joint of different classes". *IIUM Engineering Journal, vol. 13,No. 2.*
2. Ramakrishna G., Kumar, CH. A., Govindarao, P. & Ramana S.V. (2013)," A finite element based analysis of residual stresses of welded joint through mechanical vibrations". *International Journal of Mechanical and Production Engineering (IJMPE),*ISSN No: 2315-4489, vol. 2,Iss-1.
3. Chang, K., & Lee, C., (2009), "Finite element analysis of the residual stresses in T-joint fillet welds made of similar and dissimilar steels". *Introduction advance manufacturing technology ,* 41:250-258.
4. Taljat, B., Radhakrishnan, B., Zacharia, T., (1998), "Numerical analysis of GTA welding process with emphasis on post-solidification phase transformation effects on the residual stresses". *Material Science Engineering A*, 246:45–54.
5. Brar, G. S., "Finite element simulation of residual stresses in butt welding of two AISI 304 stainless steel plates". ISSN: 2319-3182, Vol. 2, Issue 1.
6. Jeyakumar, M., Christoper, T., Narayanan, R., & Rao B. N., (2011), "Residual stress evalution in butt-welded steel plates". *Indian journal of engineering & material sciences, Vol. 18,* pp. 425-434.
7. Withers, P.J., and Bhadeshia, H. K. D. H., Residual stress Part 1,"measurement techniques". *Materials Science and Technology*, vol. 17, pp. 355-365, 2001.
8. M. J. Park, H. N. Yang, D. Y. Jang, J. S. Kim, and T. E. Jin, ―Residual stress measurement on welded specimen by neutron diffraction, *Journal of Materials Processing Technology*, vol. 155–156, pp. 1171–1177, 2004
9. Schajer, G. S., "Measurement of non-uniform residual stresses using the hole-drilling method Part. I-II". *Journal of Engineering Materials and Technology*, vol. 110, pp. 338–343, 1988.
10. Sarkani, S., Tritchkov, V., and Michaelov, G. (2000) "An efficient approach for computing residual stresses in welded joints". *Finite Elements in Analysis and Design*, vol. 35, pp. 247-268.

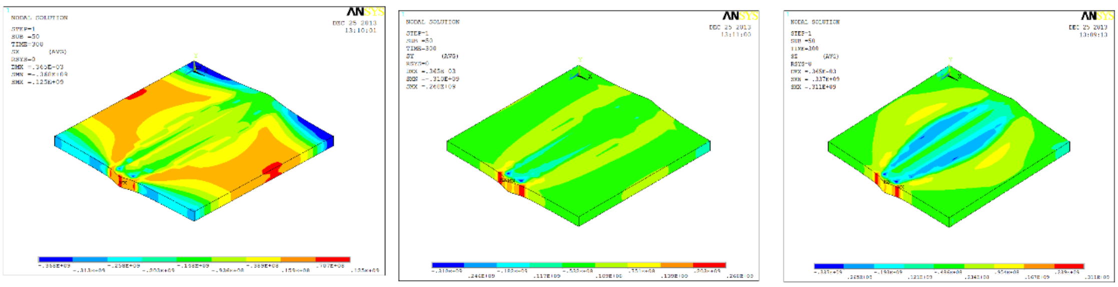
11. Khurram, A., & Shehzad, K. (2012), “FE simulation of welding distortion and residual stresses in butt joint using inherent strain”. *International journal of applied physics and mathematics,* Vol. 2, No. 6.
12. Yuan M.G., & Ueda, Y., (2006),*J Engineering Material Technology,* 118 pp. 229.
13. SMAW applications. [Online]. Available : http://www.toolingu.com/definition-650210-82769-shielded-metal-arc-welding.html
14. Fundamentals of Manual Metal Arc [Online]. Available : http://www.bocworldofwelding.com.au/media/pdf/file/library/WOWLibrary-Fundamentals%20MMA%20welding.pdf
15. Armentani, E., Esposito, R., & Sepe, R. (2007), “The effect of thermal properties and weld efficiency on residual stresses in welding”. *Journal of achievements in materials and manufacturing engineering.* Vol. 20, Issue 1-2.
16. Goo B. C., Seo J. W., & Yang S. Y. (2010). Analysis of Welding Residual Stresses and Its Applications, Finite Element Analysis, David Moratal (Ed.), ISBN: 978-953-307-123-7, InTech, Available: http://www.intechopen.com/books/finite-element-analysis/analysis-of-welding-residual-stresses-and-itsapplications.
17. Paris, P., & Erdogan F.(1963), “A critical analysis of crack propagation laws”, *Journal of Basic Engineering*, Transactions of the American Society of Mechanical Engineers, pp.528-534
18. *Structural, Carbon & HSLA Steel Plate*. Chapel Steel. [Online] Available: http://www.chapelsteel.com/A36.html
19. Metals Handbook, 10th ed., vol. 1, ASM International Handbook Committee., ASM International, Materials Park, OH, (1990) Information provided by Bethlehem Steel , 1997.
20. Benachour, M., Benachour, N., & Benguediab M.(2012) “Fatigue crack initiation and propagation through residual stress field” *Journal of World Acedemy of Science,Engineering and Technology.* V71 pp.222
21. Florea, R.S., Bammann, D. J., Yeldell A., Solanki, K.N., & Hammi Y. (2013) “Welding parameters influence on fatigue life and microstructurein resistance spot welding of 6061-T6 aluminum alloy” *Journal Materials and Design*, Vol 45 pp. 456-465.

22. Molzen, M, S., and Hornbach D. (2000), "Evaluation of welding residual stress levels through shot peening and heat treating" *copyright society of automotive engineers, inc.*

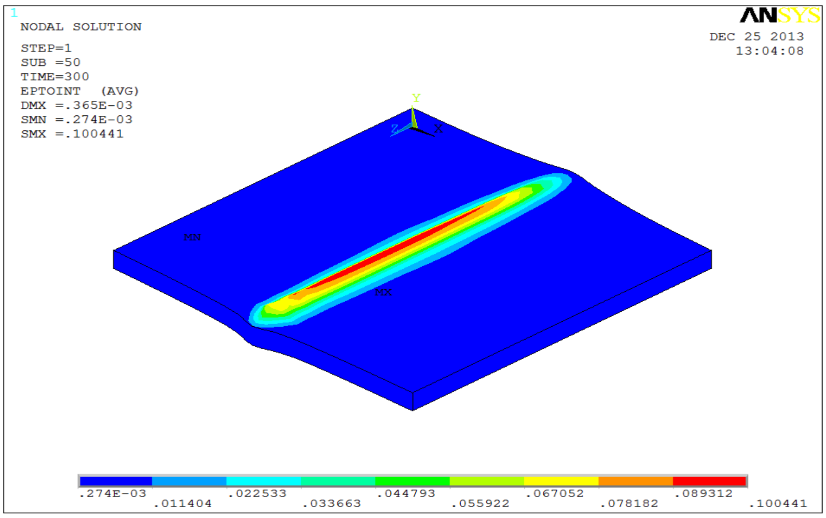
APPENDICES



Failure due to welding [19]



X, Y and Z component of stress in first heat flux



Total mechanical strain intensity for first heat flux