FINITE ELEMENT ANALYSIS OF RESIDUAL STRESSES IN WELDED STRUCTURES

by

Syahirhatim Bin Sholihotdin

(10322)

Dissertation submitted in partial fulfillment of the requirements for the

Bachelor of Engineering (Hons)

(Mechanical Engineering)

September 2011

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Finite Element Analysis of Residual Stresses in Welded Structures

By

Syahirhatim B Sholihotdin

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, 2011

CERTIFICATION OF ORIGINALITY

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

____

(SYAHIRHATIM B SHOLIHOTDIN)

ABSTRACT

One of the problems during welding process is residual stress or welding distortion. This situation affects the integrity of welded structures because the zone that has high residual stresses will have a greater tendency to crack or fail. The objective of this project is to simulate the residual stresses in the welded structure by using finite element analysis. The ANSYS software will be used to simulate the welding process. In this simulation, Gas Tungsten Arc Welding (GTAW) had been chosen as welding type and the welded plate will undergo butt joint process to join them together. Birth and death elements were used to simulate the movement of the welding process.

The temperature-dependent material properties have being applied to the model. Bias method has been applied at the area of concern especially at the area where the heat flux is being applied. The heat convection need to be applied to the model and for the welding process, heat flux will be applied in the weld zone element from the first element until the last element at the welding zone. The prediction of welding residual stresses using finite element method starts with thermal analysis and followed by structural analysis.

A few tests will be done by varying the plate thickness. The results from the variation of plate thickness will be compared. Data that will be interpreted from the model are stress versus distance along the welding line, fusion line and along the mid section. From these result, the residual stresses will be investigated.

ACKNOWLEDGEMENT

I would like to express my sincere gratitude to my project supervisor Dr. Saravanan Karuppanan for his technical and moral support which made this final year project successful. His continuous feedback and support throughout the semester helping the student by giving encouragement, suggestions and ideas throughout the final year project are invaluable. I appreciate the time he spent on reviewing my writing and improving my technical writing skills. I would also like to thank Dr. Mokhtar Awang who helped me in ANSYS Software work.

My heartiest appreciation goes to my examiners for theirs constructive comments and criticisms, also not forgetting all my friends who had given me support towards completing the Final Year Project.

I would like to express my gratitude towards my parents, my brothers and my sister for their unconditional love, support and trust for providing me the best of educational opportunities to my study in Universiti Teknologi PETRONAS.

Thank you.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF APPENDICES

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

1.1 Project Background

Welding is one of the most reliable permanent joining processes in the industry. Welding is carried out by the use of heat or pressure or both and with or without added metal. There are many types of welding including Metal Arc, Atomic Hydrogen, Submerged Arc, Resistance Butt, Flash, Spot, Stitch, Stud and Projection. Highly non-uniform field applied during welding give rise to the residual stresses and distortions in final geometry. In huge steel fabrication industries such as marine structures, pressure vessels and piping in chemical industry the problem of residual stresses has been a major issue.

Residual stresses are defined as the stresses which exist in the structures without the application of external load and are supposed to be self-balancing within the bulk. Presence of residual stresses may be beneficial or harmful for the structural components depending on their nature and magnitude. During the welding process, a very complex thermal cycle is applied to the weldment which causes irreversible elasto-plastic deformation and subsequently gives rise to the residual stresses in and around fusion zone and heat affected zone (HAZ).

It is quite difficult to study the instantaneous residual stresses in the weld zone by using the normal available instrumentation and measuring facilities. For that reason, finite element method (using ANSYS software) was adopted in this work in order to find out the instantaneous distribution of residual stresses.

1.2 Problem Statement

Welding is a fabrication process that joins materials by causing coalescence. This is often done by melting the workpieces and adding a filler material to form a pool of molten material that cools to become a strong joint. The main issue of the welded structures is the residual stresses due to heat gradient. During welding of two components structure together, the complex thermal cycles will result in formation of residual stresses in that region and affect the integrity of the welded structures. It can quite impair the performance and the reliability of the welded structures. When the welding process being done on the duplex stainless steel, the Heat Affected Zone (HAZ) will experienced high gradient of residual stresses. The zone that has high residual stresses will have a greater tendency to crack or failure.

1.3 Objective and Scope of Study

Objective of this study is to simulate and investigate the residual stresses in the welded structure by using finite element analysis. This study aims to define the residual stresses behavior of the duplex stainless steel. ANSYS software will be used as a finite element analysis tool to study the residual stresses in welded duplex stainless steel. In this project, different plate thickness will be studied.

CHAPTER 2

LITERATURE REVIEW

Ihab, Maher, and Abdalla [1] studied the effect of boundary conditions on residual stresses in welding using element birth and element movement techniques. They used ABAQUS to simulate the models. On element birth techniques model, they simulated basic arc welding of two coplanar plates along with the parting line with the addition of a filler material between them as illustrated in Figure 2.1. On the study of element movement techniques, the meshing is similar to the element birth techniques but the elements of the weld pool are separated from the base plate.

They used Inconel Alloy 600 in their research to study the effect of boundary conditions on residual stresses. In developing a general purpose model for the welding process, they considered the moving heat source, heat loss, temperature-dependent material properties and metal deposition. A moving heat source is modeled by setting a heat flux distribution that varies with time applied to the top surface of the weld pool zone.

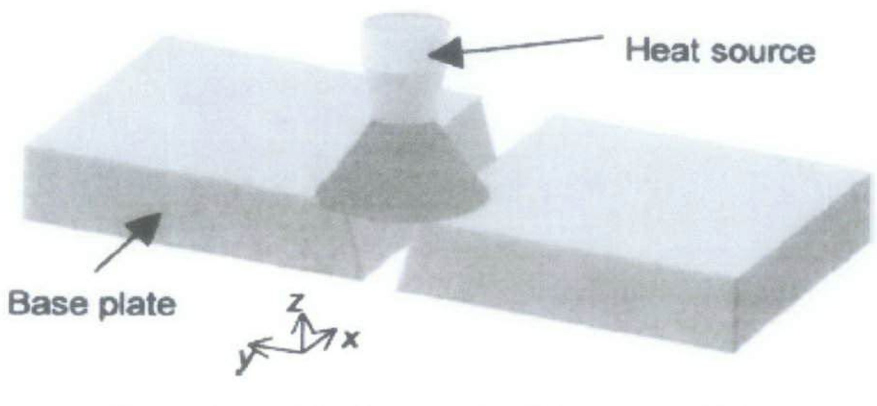


Figure 2.1: The diagram of welding process [1]

According to Ihab, Maher and Abdalla [1]: "The stresses generated during the welding process are not highly affected by the change in the boundary conditions. Anyhow, the residual stresses had some differences. Welding a free plate caused a much lower residual stresses than welding a plate that is part of a larger structure. The change in the thermal load had insignificant effect on the residual stresses. After comparing the element movement technique with the element birth technique, they found that these two methods have no significant effect on residual stresses of the welded structures."(p.439)

Research conducted by Li, Wang, Chen and Shen [2] have successfully illustrated the finite element analysis (FEA) of residual stress in the welded zone of a high strength steel. High strength steels have a great tendency to crack during welding. This steel has high hardenability, and the crack tendency can also be caused by the presence of hydrogen. The stresses present at the weld joint also lead to the crack of the high strength steels.

The steel that they used in this analysis is HQ130 steel that contained small amounts of alloying elements. They applied gas shielded arc welding (GMAW) in this simulation with a heat input of 16 kJ/cm. The tensile strength of this steel is 1300MPa. The test plate was welded using mixed gas shielded arc welding. The filler wire used was low strength ER100-G. They used two plates butt joints as a weld sample. This FEA consist of two parts. The first part is calculation of thermal analysis and the other one is calculation of instantaneous stress. From the analysis, they found that the HQ130 steel without pre-heating has a strong influence on the cold cracks for the stress, σ_x, that is perpendicular to the weld direction.

From the calculated results, they indicated that the weld metal and the fusion zone experience small amount of tensile stress at the beginning but increases gradually with time. At the beginning, the maximum σ_x is in the fusion zone but when σ_x increases gradually, the maximum restrain stress, σ_x shifts to the upper surface in HAZ near the fusion zone. Phase boundaries, impurities and micro defec ts present in the weld metal act as the source for crack formation near the fusion zone.

B. Gideon, L. Ward and D.G. Carr [3] studied the residual stress in duplex stainless steel (DSS) welds. They also studied on the susceptibility of DSS to intergranular corrosion (IGC). They performed various tests on DSS and studied the stress/strain levels within the various region/phases of the DSS welds and the correlation of DSS susceptibility to IGC. Stress/strain levels were determined using Neutron Diffraction techniques. ASTM A262 was used to assess the susceptibility of DSS welds to IGC. The joint configuration adopted was a double bevel single V bevel. They have done test on two condition of welding. First condition will have 1 weld root pass, 3 weld fill pass and 1 weld cap pass. The other condition will have 1 weld root pass, 2 weld fill pass and 1 weld cap pass.

According to Gideon, Ward and Carr [3]; "As a consequence of the varying heat inputs from 1.6 kJ to 2.0 kJ, for the two different weld conditions, there was a move towards the development of a structure containing 35-47% ferrite. This suggests that the control of heat input was not a significant factor in influencing the specific weld metal mechanical properties within the conditions specified. It was established that there was no relationship between the percentage ferrite content in the notch region and the corresponding toughness." (p.11)

By using neutron diffraction techniques, they found that for ferritic and austenitic steel, a maximum tensile stress is formed in the fill section of the weld and decreases in the root and cap regions for the transverse direction.

Robert N Gunn [4], mentioned that the carbon element have negative influence in alloying of DSS. The carbon causes precipitation of chromium carbides with accompanying chromium depleted zones. The maximum addition of carbon is at 0.03% of total composition. Chromium element contributes stability to the passive film of DSS but higher chromium content will increase the risk of intermetallic precipitation. Increase in Molybdenum content in DSS also enhances the risk of intermetallic precipitation.

D. Stamenkovic and I. Vasovic [5] carried out finite element analysis of residual stress in butt welding of two similar plates. They used ANSYS software to simulate the welding process. In their work, the welding process computation was split into two solution steps which are thermal and mechanical analyses. They also made the following assumption and features about the model:

i. The displacements of the parts during the welding do not affect the thermal distribution of the parts themselves;
ii. All the material properties are described till to the liquid phase of metal;
iii. Convection and radiation effects are considered;
iv. The element birth and death procedure is used;
v. Changes in the mechanical state do not cause a change in thermal state but a change in the thermal state causes a change in the mechanical state

According to M Sundar, G Nandi, A Bandyopadhyay and S C Roy [6]; "As phase change starts deviatoric stress becomes zero and considerable plastic deformation occurs in the weld metal and the base metal. As temperature decreases during the cooling phase, the stress in the solidifying material increases and becomes tensile due to the positive temperature gradient. The plastic strains resulting from the heating causes stress, which inturn produce internal forces that may cause buckling, bending and rotation. The residual stress combined with distortion and degradation of the material mechanical properties influences the buckling strength and fatigue life of welded structure." (p.1)

For this simulation, gas tungsten arc welding has been chosen. GTAW is an arc welding process that uses a non-consumable tungsten electrode to produce the weld. The weld area is protected from atmospheric contamination by a shielding gas and a filler metal is normally used. A constant-current welding power supply produces energy which is conducted across the arc through a column of highly ionized gas and metal vapors known as plasma. Figure 2.2 shows the detail of GTAW process.

During welding process, heat input in GTAW will be applied to the welded structure. This heat input is interpreted as heat flux in ANSYS simulation. In order to describe the movement of the welding process, the birth and death elements are used. The heat input can be calculated by using equation 1 below.

$$Q = \frac{n \text{ x } V \text{ x } I}{S \text{ x m}}$$ [Equation 1]

where

Q = heat input (W/mm^2),

V = voltage (V),

I = current (A),

S = welding speed (mm/s).

m = electrode diameter (mm)

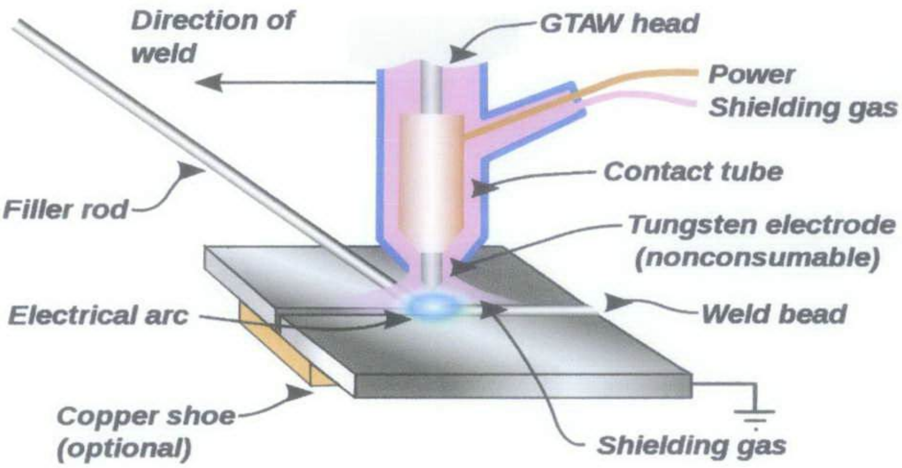


Figure 2.2: Detail of gas tungsten arc welding process

CHAPTER 3

METHODOLOGY

In order to complete this project, a few steps are involved. The project starts with the literature review regarding the residual stresses on welded structures and the residual stresses due to thermal load by using finite element analysis. Next, factors that contribute to residual stresses will be reviewed in order to decide which factor to be considered for this project. After that, finite element simulation of residual stresses in welded structures will be developed using ANSYS software. When the simulation process is finished, the data and result gained will be interpreted and investigated.

Gas tungsten arc welding (GTAW) has been choosen in this project. All standard and calculation for the heat input will be based on GTAW standard. The material that has been selected is Duplex Stainless Steel 2205.

3.1 ANSYS Work

ANSYS software is a finite element analysis code widely used in the computer-aided engineering (CAE) field. Every industry nowadays recognizes that a key strategy for success is to incorporate computer-based engineering simulation. This software will allow engineers to refine and validate designs at each stage where the cost of making changes is minimal. It permits an evaluation of a design without having to build and destroy multiple prototypes in testing. ANSYS software will have a big role throughout this project. Pre-ANSYS learning will take place once the literature review and theories are completed.

The project starts by modeling a set of welded plate. The material used for the first simulation is Duplex Stainless Steel 2205. Before meshing the model, the temperature-dependent thermophysical and thermal-structural properties such as conductivity, specific heat, density, Young's modulus, Poisson's ratio, and thermal expansion coefficient and also yield strength will be assigned to the model for thermal analysis.

Next, the model will be meshed. The bias method will be use in order to get more accurate result in desired area. The surface heat convection will be applied on the welded plate and heat flux will be applied on weld pool. The birth and death techniques were used to simulate a moving heat source.

After applying the heat flux on one element, the model will be solved and it will be continued by applying the heat flux on the next element until the last element of the weld pool. This operation describes the moving heat process of the welding. After that, the result will be interpreted in the forms of graphs such as stress distribution along the midsection, along the welding line and along the fusion line. Those results will be compared with the journal results. The result must have almost similar result to the journal result in order to verify the method.

If the result is the same with the referenced journal, the same method will be carried out for other material. The method will be used for other model. The thickness of the welded will be manipulated and the results from those simulations will be compared and analyzed. The residual stresses that occur in the welded plate will be investigated and the factors that significantly affect the stress distribution of the welded plate will be analyzed.

3.2 Sample Design

Before constructing the model in ANSYS, the sample of the model needs to be designed. The weld samples (two plates, 305 mm x 100 mm) will be selected in this project as in Figure 3.1. The thickness of the samples is 5mm, 10 mm and 15mm. The detailed geometry of the model is as in Figure 3.2.

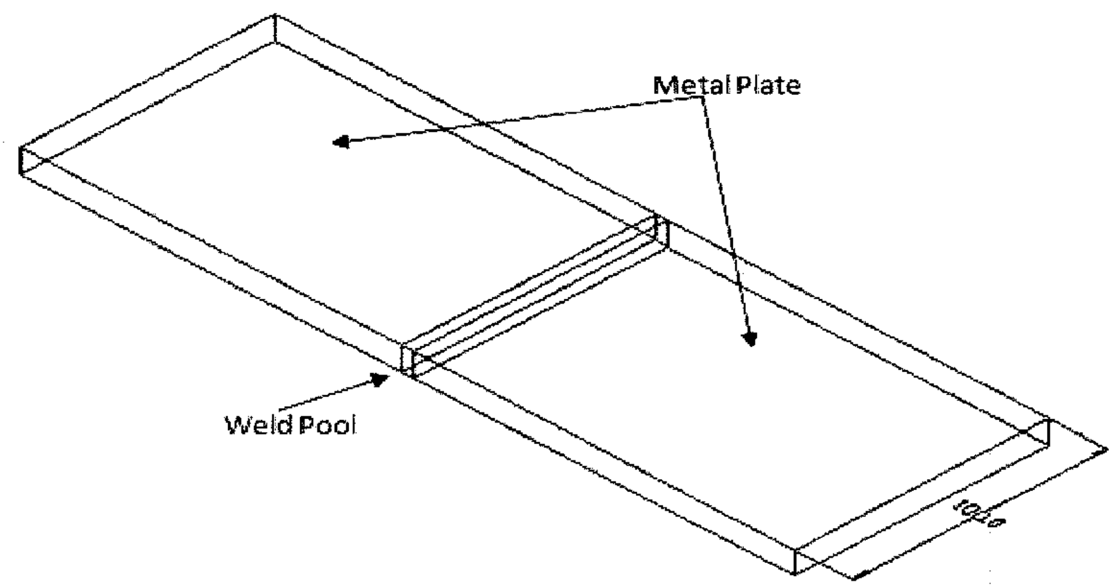
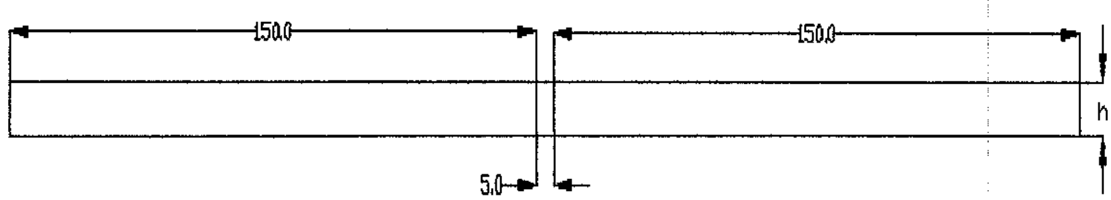


Figure 3.1: Schematic diagram of the welding test plate



Model	Thickness, h
1	5mm
2	10mm
3	15mm

Figure 3.2: Geometry detail of the model

3.3 Duplex Stainless Steel 2205 Parameters

The type of duplex stainless steel used in this project is Alloy 2205. Alloy 2205 is widely used in the chemical industry and oil and gas industry. This type of alloy is often exposed to the heat source. Therefore, the effect of heat exposed to this alloy need to be studied in order to make sure that the application of this alloy is safe in those industry. The mechanical and physical properties of Alloy 2205 are given in Table 3.1 and Table 3.2.

Table 3.1: Mechanical properties of Alloy 2205 at elevated temperature

Temperature (°C)	**50**	**100**	**200**	**300**
Yield Strength(0.2%), MPa	413.68	358.53	310.26	282.69
Tensile Strength, MPa	661.90	620.53	572.27	558.48

Table 3.2: Physical properties of Alloy 2205 at elevated temperature

Temperature (°C)		**20**	**100**	**200**	**300**
Density	kg/m^3	7695	7695	7695	7695
Modulus of Elasticity	GPa	40	37	35	30
Linear Expansion	$10^{-6}/°C$	-	13.0	14.0	14.5
Thermal Conductivity	W/m.K	110	100	80	70
Heat Capacity	J/kg.K	470	500	530	560
Poisson's Ratio		0.32	0.32	0.32	0.32

3.4 Parameters and Assumptions

In order to simulate the welding process, several parameters and assumptions have been set to simplify the process. The purpose of these assumptions is to reduce the complexity of the complex process of thermal cycle with acceptable accuracy. Parameters and assumptions that have been made are as follows:

i. Convection boundary condition is applied on the top surface of welded plate
ii. Heat source is moving at a constant speed, 5 mm/s.
iii. Material properties of the filler are the same with the material properties of the welded plate.
iv. Electrode diameter is 5 mm.
v. Displacement of plate during welding does not affect temperature distribution.
vi. Initial temperature of plate and electrode is at room temperature, 27°C.
vii. Estimated heat transfer coefficient, h is 15 $W/(m^2.K)$.
viii. Emissivity, ε is neglected.

3.5 Flow Chart

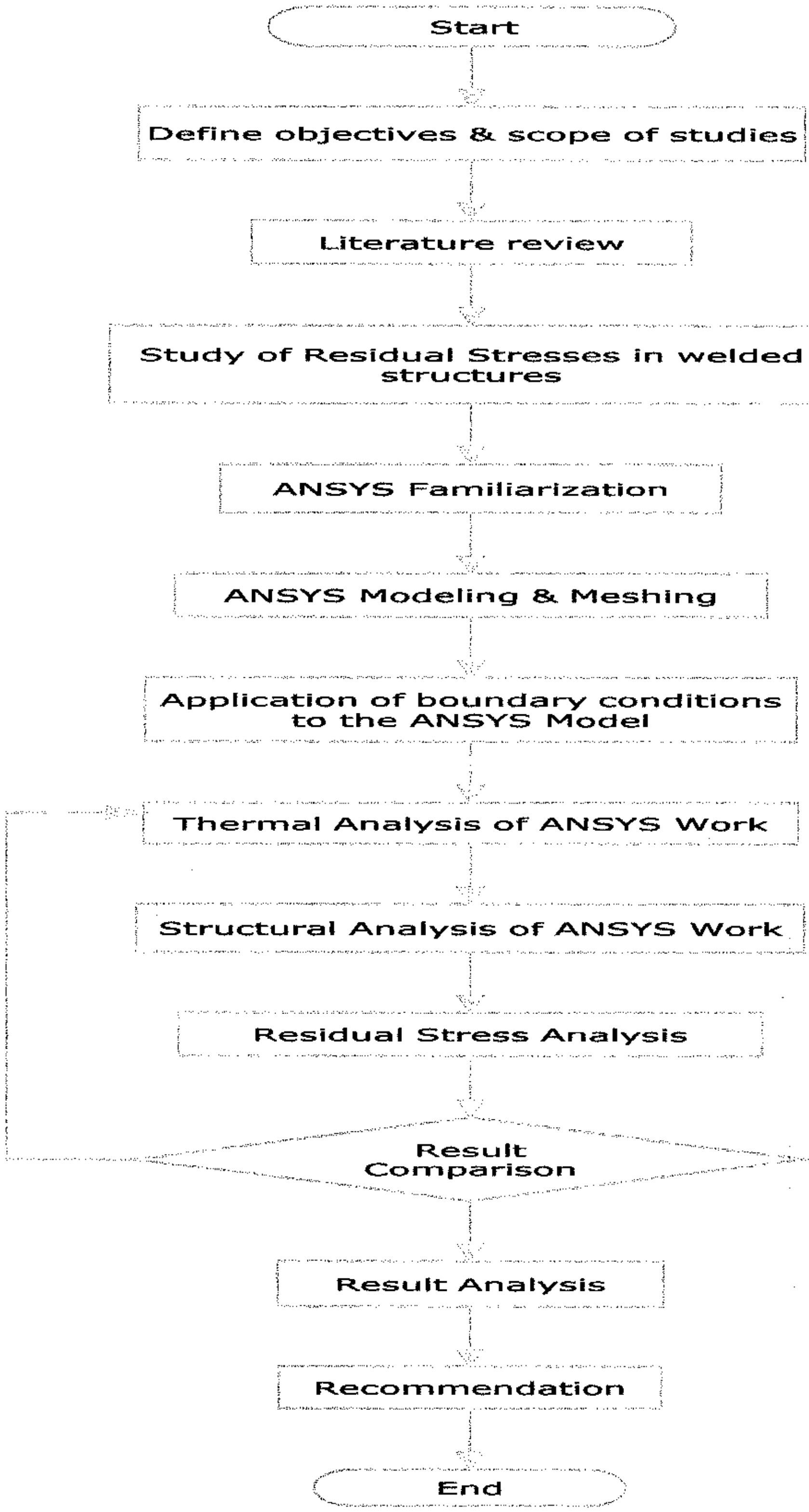


Figure 3.3: Flow chart of the project planning

CHAPTER 4

RESULTS & DISCUSSIONS

4.1 ANSYS Modeling

Figure 4.1 shows the basic model of the welded structure that contains two plates with weld pool that welded the two plates together. On this model, the temperature-dependent material properties were applied such as Young's modulus, Poisson ratio, thermal conductivity, specific heat and density. The element used was 8-noded brick element or SOLID70 that can perform a coupled displacement-temperature analysis which has single degree of freedom.

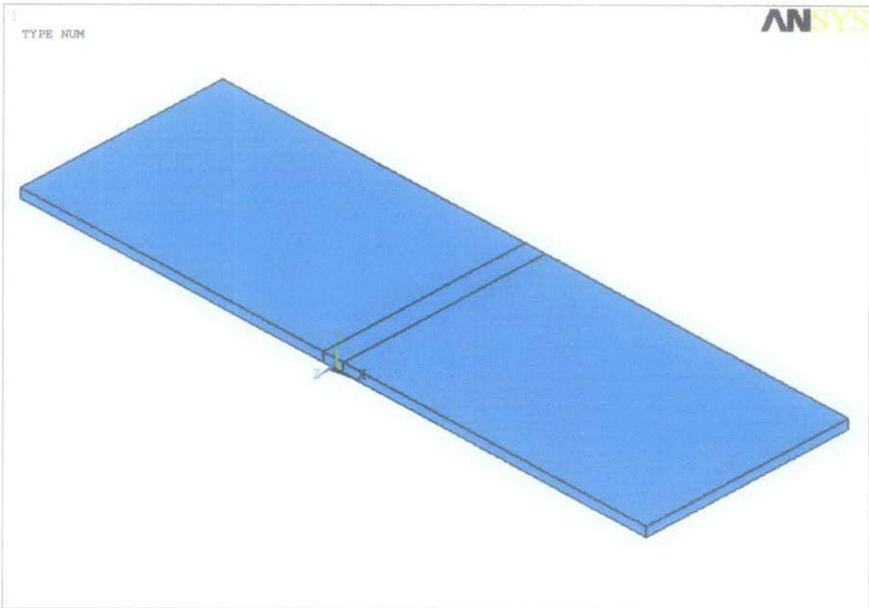


Figure 4.1: Model of welded plate in ANSYS

4.2 ANSYS Meshing

The next process after modeling is the meshing of the model. For this model, bias method was applied. The purpose of using bias method is to get more refine mesh in the area of concern. For this model, the area of concern is at the center on the model where the welding process will be simulated. The element at the center of the model will have refined size compared to the parts that are opposite to the welding area. By applying this bias method, the result will be more focused on concerned area. Figure 4.2 shows the model that has been meshed using bias method.

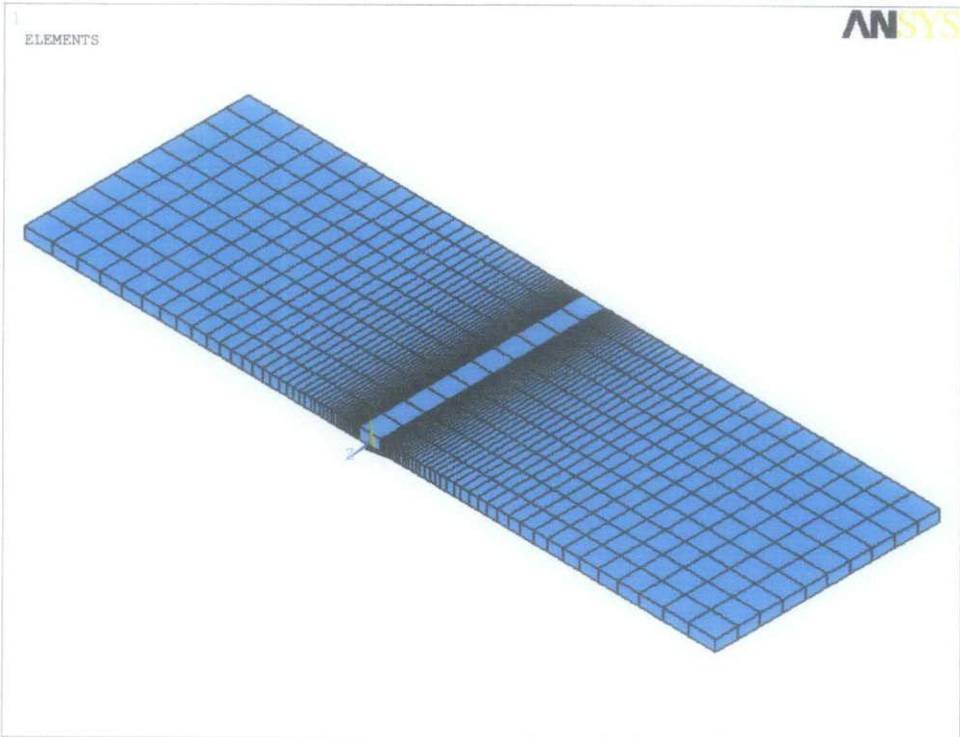


Figure 4.2: Meshed model of welded plate in ANSYS

4.3 Gas Tungsten Arc Welding (GTAW) Details

In order to simulate the welding process, equation 1 from the literature review will be used. For this simulation, the welding speed is 5mm/s. The electric input is 12V and 90A. The arc efficiency for gas tungsten arc welding is 80% or 0.8. The electrode used is tungsten alloy because it has the highest melting temperature among pure metals which is at 3422°C. The electrode diameter can vary between 0.5 and 6.4 mm and their length can range from 75 to 610 mm. From those details, the heat input was calculated as below.

$$Q = \frac{0.8 \text{ x } 12 \text{ x } 90}{5\frac{mm}{s} \text{ x } 5 \text{ } mm} = 35 \text{ W/mm}^2 = 35 \text{ x } 10^6 \text{ W/m}^2$$

4.4 Welding Simulation

4.4.1 Applying Boundary Condition

An important step before applying the heat input to the model is applying boundary conditions. A few boundary conditions need to be applied to the model such as heat convection, radiation and initial temperature. Convection has been applied to all top surface of model as shown in Figure 4.3. Heat transfer coefficient, h = 15 W/(m^2.K) is applied to the model. The emissivity, ε is neglected because the value is very small

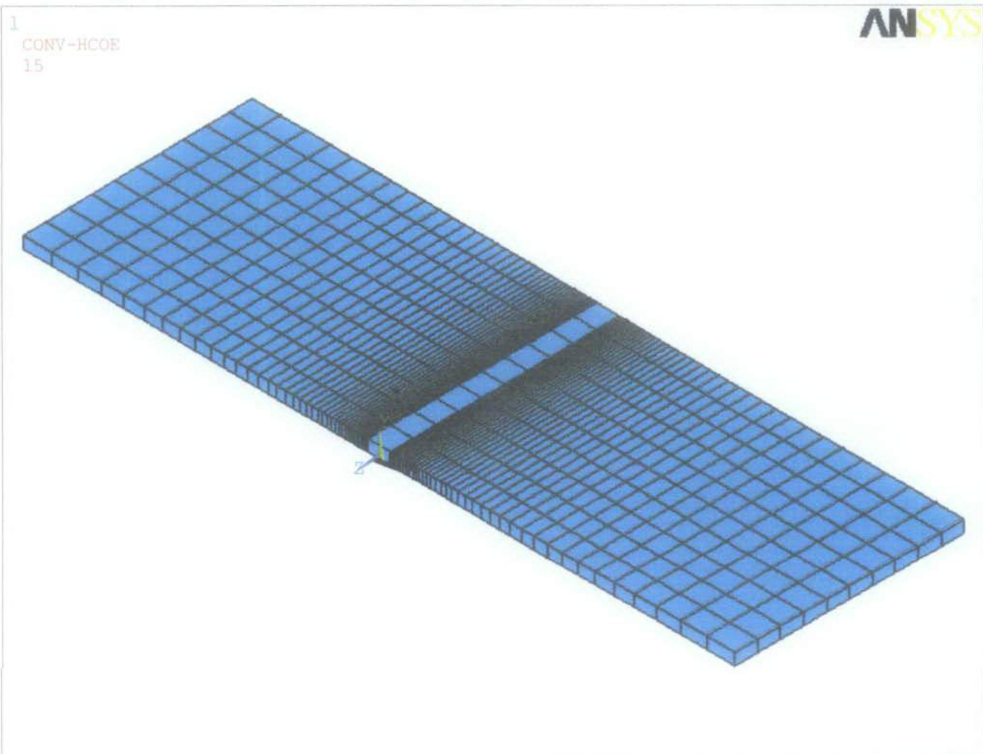


Figure 4.3: Heat convection at top surface of the model

4.5 Thermal Analysis

For thermal analysis, the temperature distribution along the welded metal was plotted. The welding speed is 5mm/s. In order to visualize this condition, the time taken for each loadstep is assign to be 5s. In this simulation, there are 19 loadsteps including the cooling process of the welded plate. Figure 4.4, Figure 4.5, Figure 4.6, and Figure 4.7 below show the temperature distribution at 5s, 25,s, 50s, and 950s respectively. Appendix A shows the full temperature distribution from 5s to 950s.

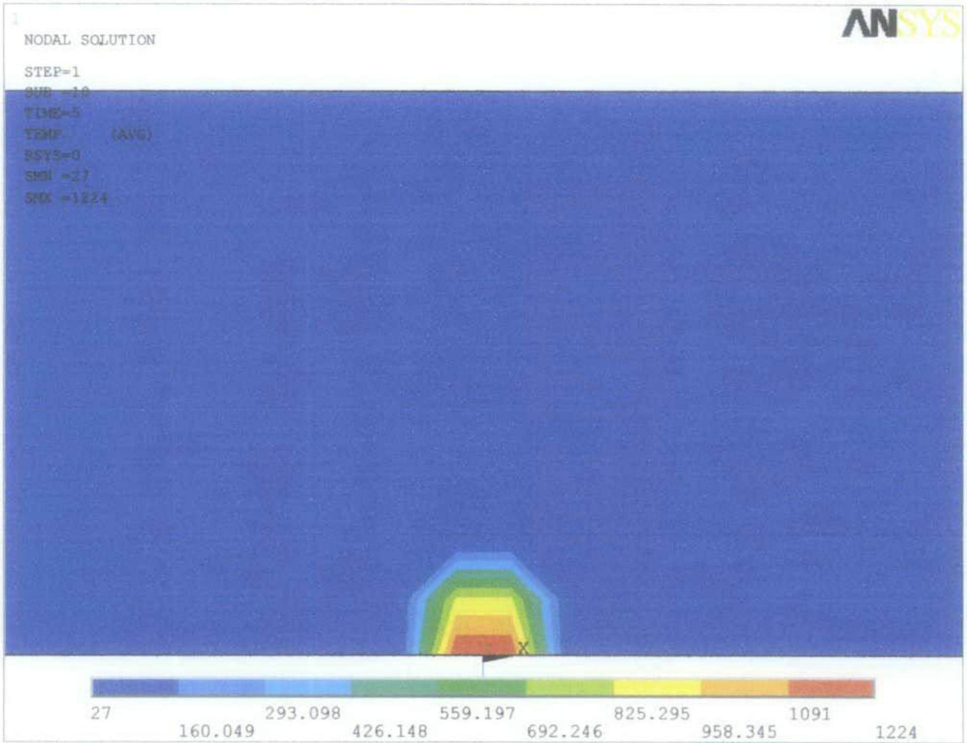


Figure 4.4: Temperature distribution at $t = 5$s

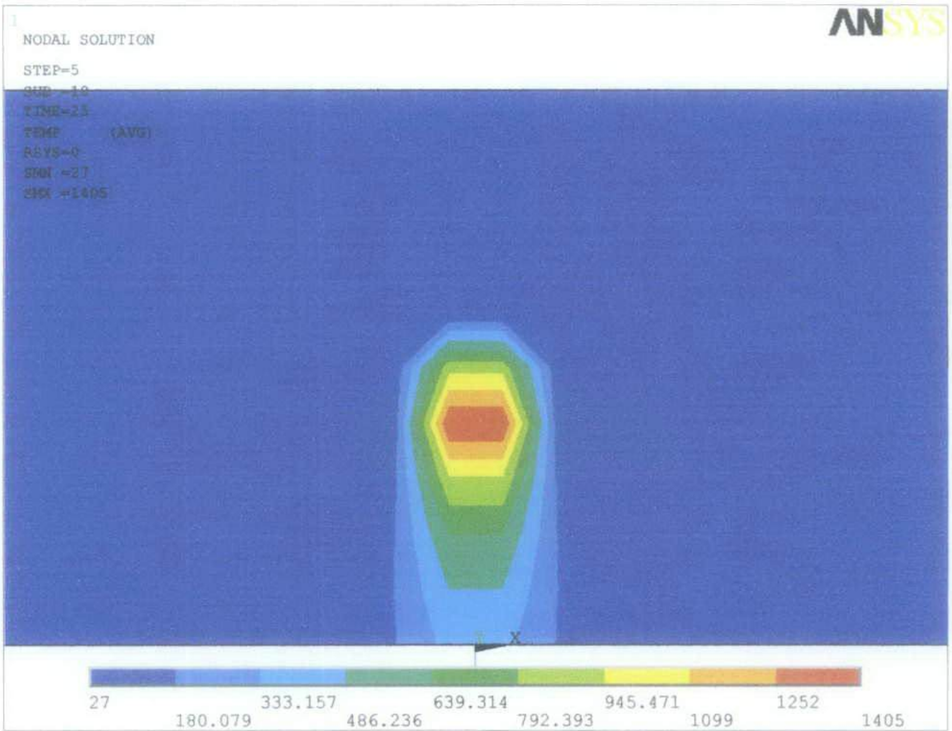


Figure 4.5: Temperature distribution at $t = 25$s

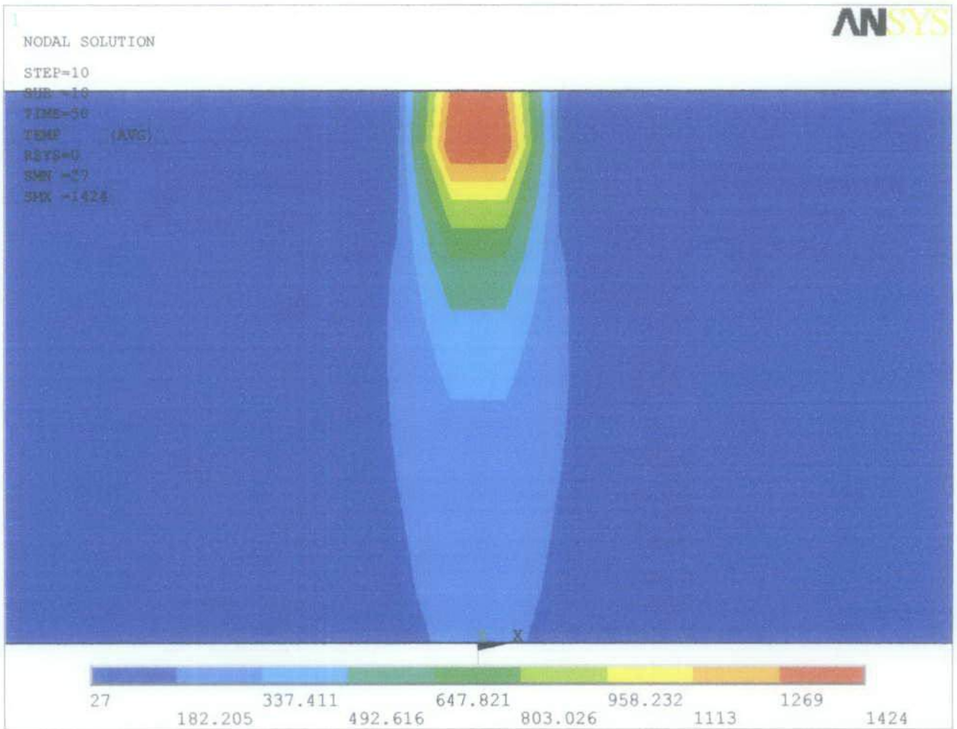


Figure 4.6: Temperature distribution at $t = 50$s

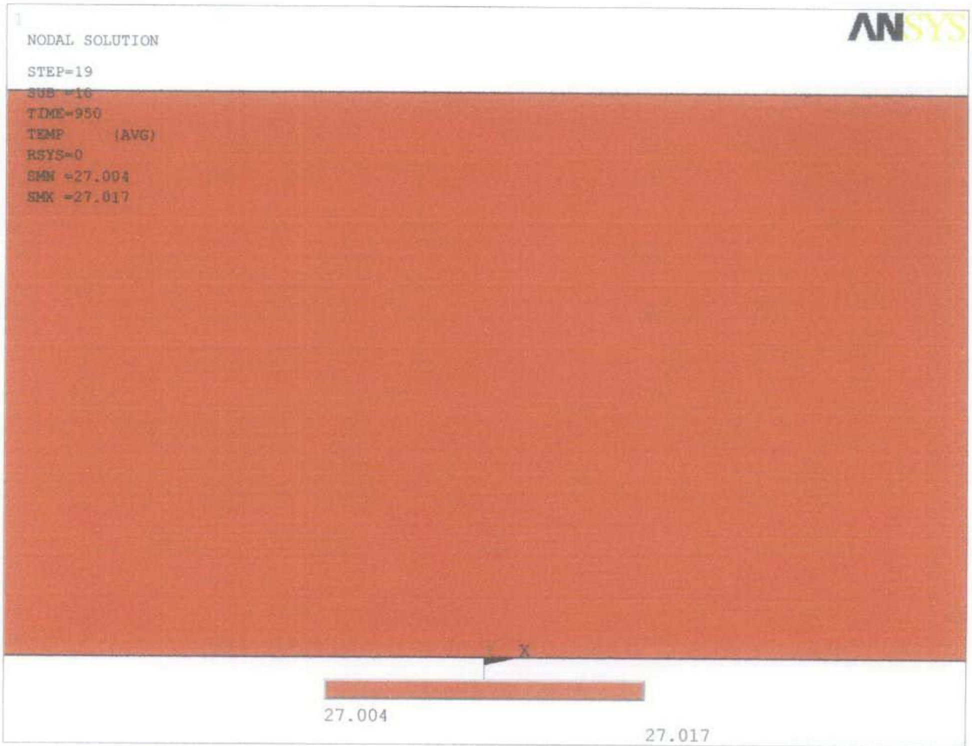


Figure 4.7: Temperature distribution at $t = 950$s

From the simulation, the time taken for the welded plate to cool down to room temperature is around 950 second or about 16 minutes. The maximum temperature of the welded metal during the welding process is 1424^0C. There also heat lost due to convection at the top surface of the welded plate.

For the thermal flux, Figure 4.8, Figure 4.9 and Figure 4.10 show the thermal flux distribution at time 5s, 15s, and 25s. The value of the heat flux that have been applied to the workpiece is 35x 10^6 W/m^2

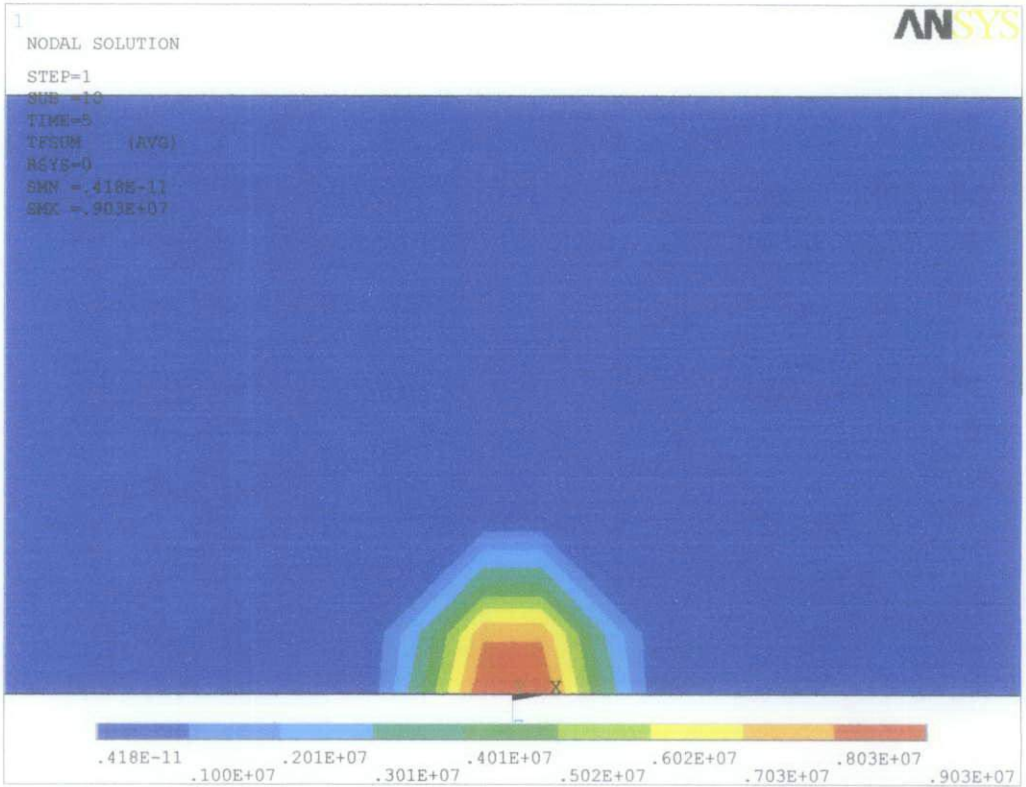


Figure 4.8: Heat flux distribution at t = 5s

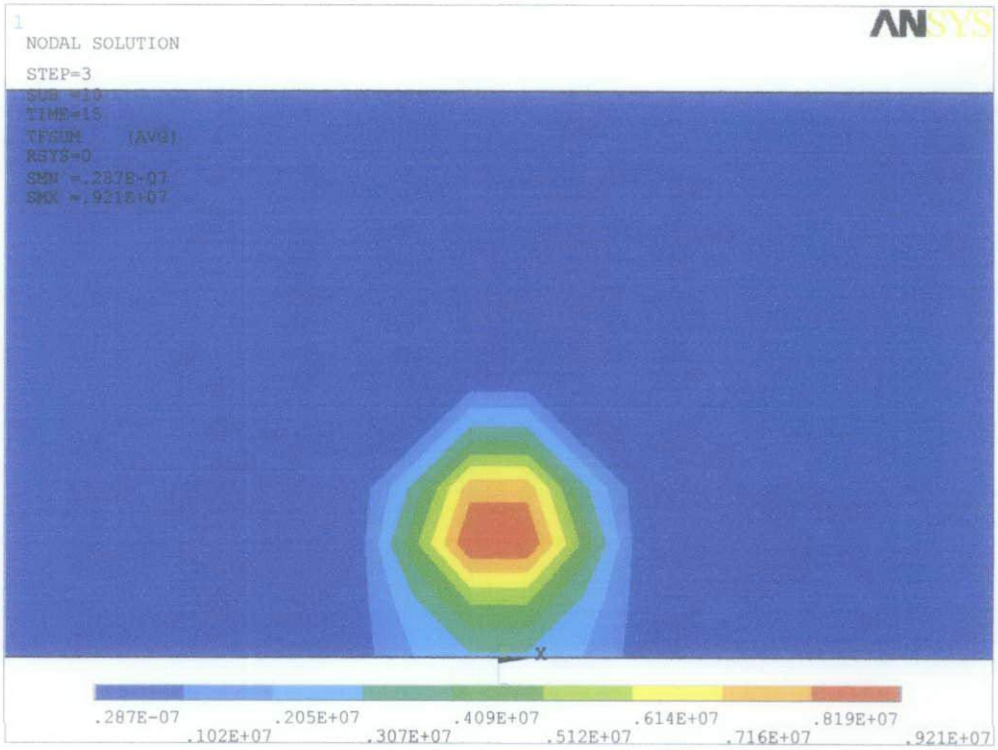


Figure 4.9: Heat flux distribution at t = 15s

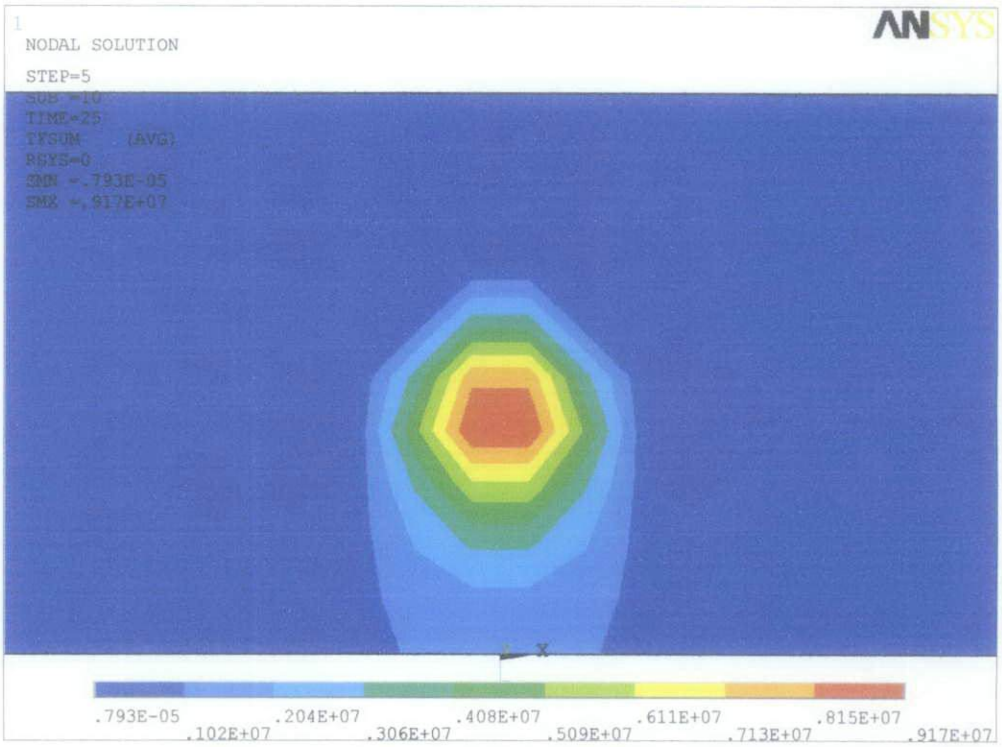


Figure 4.10: Heat flux distribution at t = 25s

4.6 Thermal Profile

In order to simulate the structural analysis, the temperature history of the welded metal must be identified. Every node in the welded metal will experience change in temperature during the welding process. Two nodes have been taken as reference for validation process. Figure 4.11 shows two nodes that have been indicated for thermal profile study. Figure 4.11(a) and Figure 4.11(b) show temperature profile at two chosen nodes.

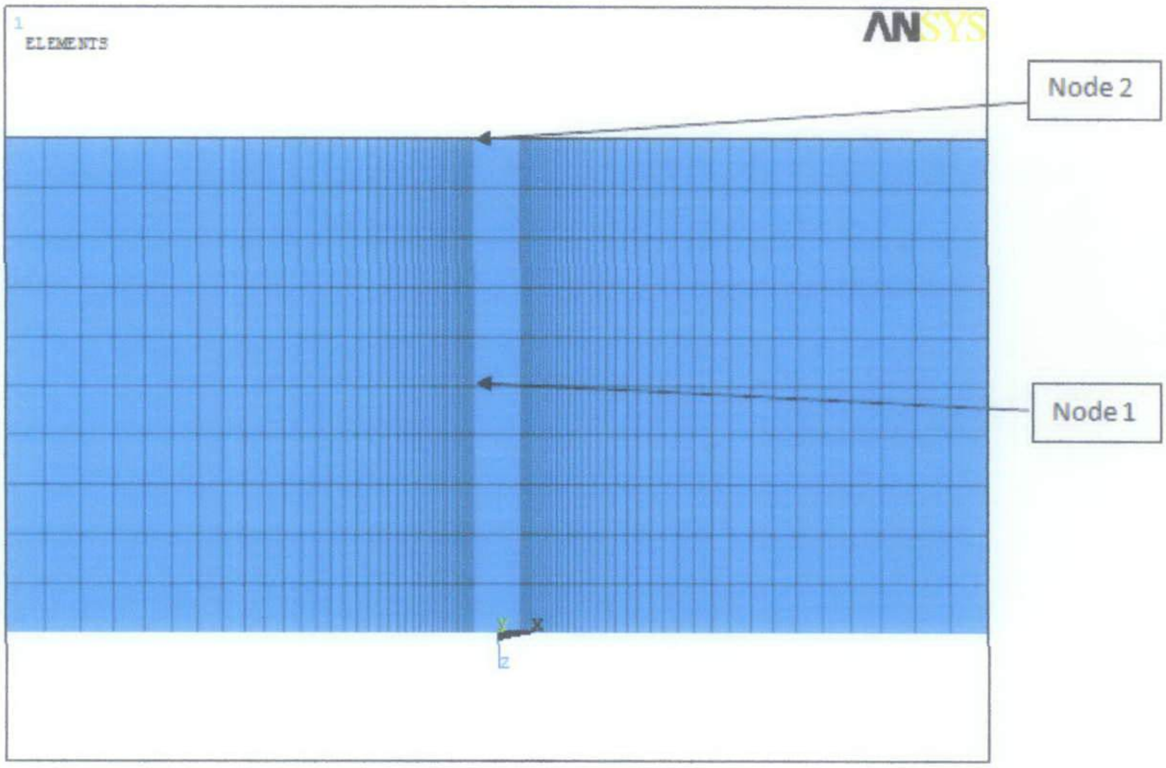


Figure 4.11: Two chosen nodes for temperature profile plot

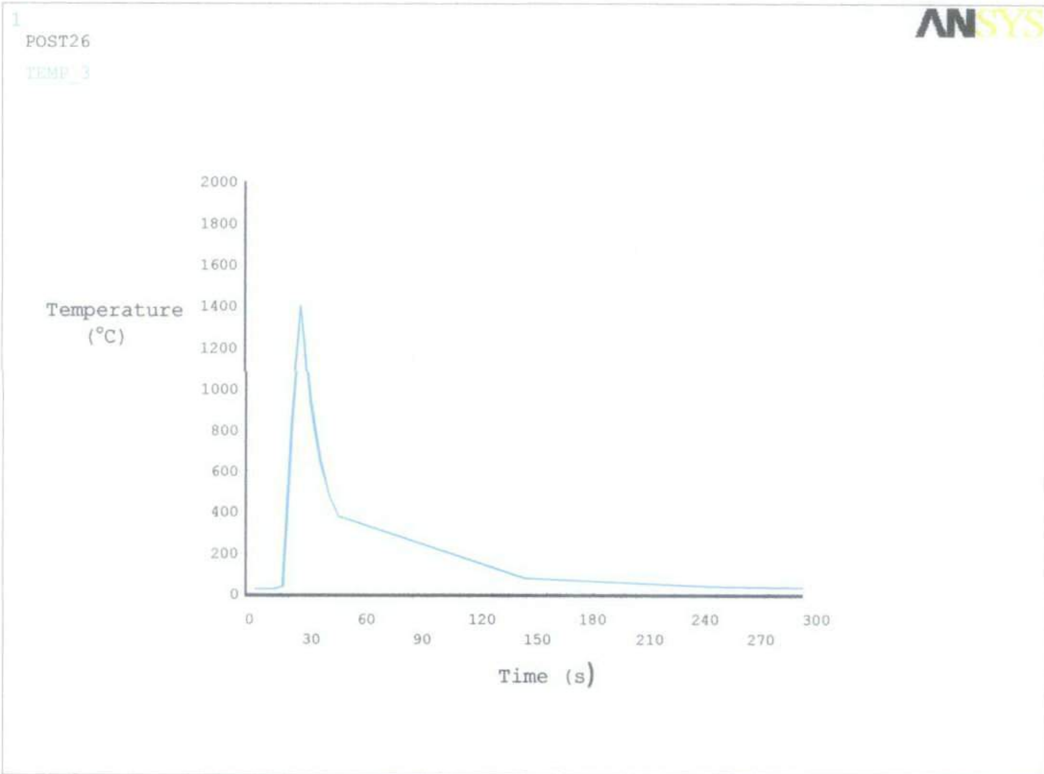


Figure 4.11(a): Temperature profile at node 1

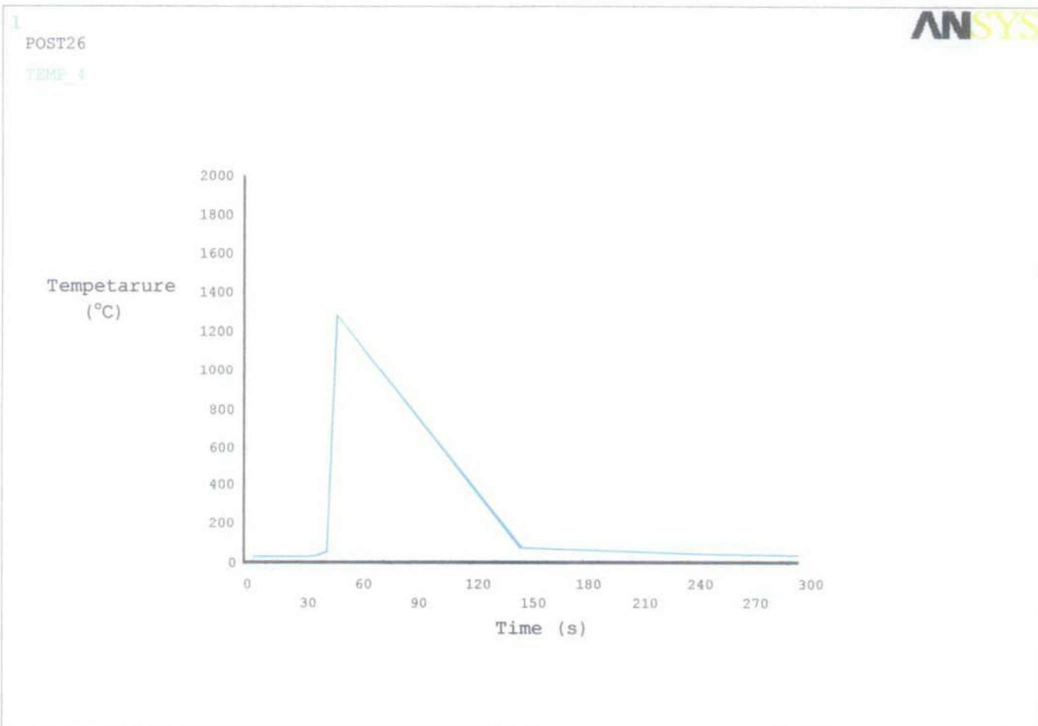


Figure 4.12(b): Temperature profile at node 2

From the temperature versus time graph that have been plotted, the temperature at the chosen nodes achieve the highest point at the time where the heat flux is being applied to those nodes. As the conclusion for the thermal profile study, the temperature profile history of each element is valid for the structural analysis based on two factors:

a) The temperature reaches the highest temperature at the same time where the heat flux has been applied.
b) The highest temperature that can be achieved by the workpiece is within the range of melted temperature of the material.

4.6 Structural Analysis

After completing the thermal analysis, the simulation continues with the structural analysis. In order to simulate the structural analysis, the element type is changed from SOLID70 to SOLID45 (linear 8-node brick element with three degrees of freedom at each node). In this simulation, the temperature history of the model is taken from the thermal analysis result.

The welded metal was clamped at four nodes in order to constraints the movement of the welded metal. From the structural result, the deformation of welded metal is insignificant. The deformation of the welded metal is shown in Figure 4.13. It is stated in the simulation that the maximum deformation in only about 1.19×10^{-3} m

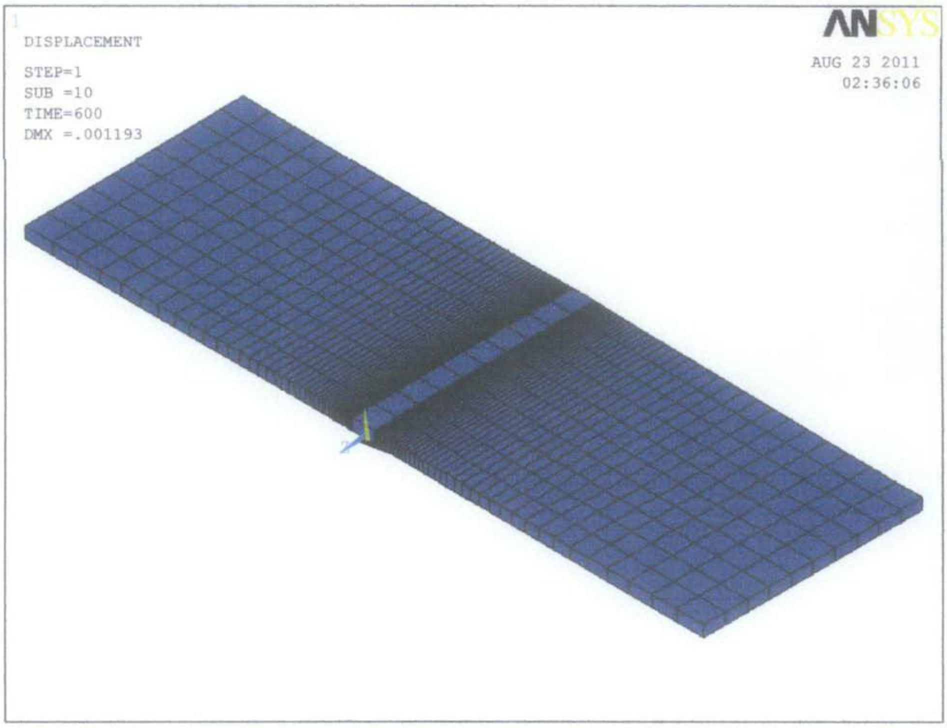


Figure 4.13: Deformation of welded plate

After applying the temperature from thermal analysis to the model, the stress distribution of the welded metal is plotted. Figure 4.14 shows the stress distribution in the welded metal. From the analysis, the average residual stress occurs at the fusion zone. Residual stress occurs at fusion zone is around 190 MPa.

It is also found that the maximum stress intensity of the welded metal occurs at the welding area and the stress is about 435 MPa and the minimum is 1.41 MPa. From the analysis, the maximum stress that can be achieved by the model is higher than yield strength of the material but still lower than the tensile strength of the material. It means the welded metal experienced the plastic deformation and tends to fail or crack.

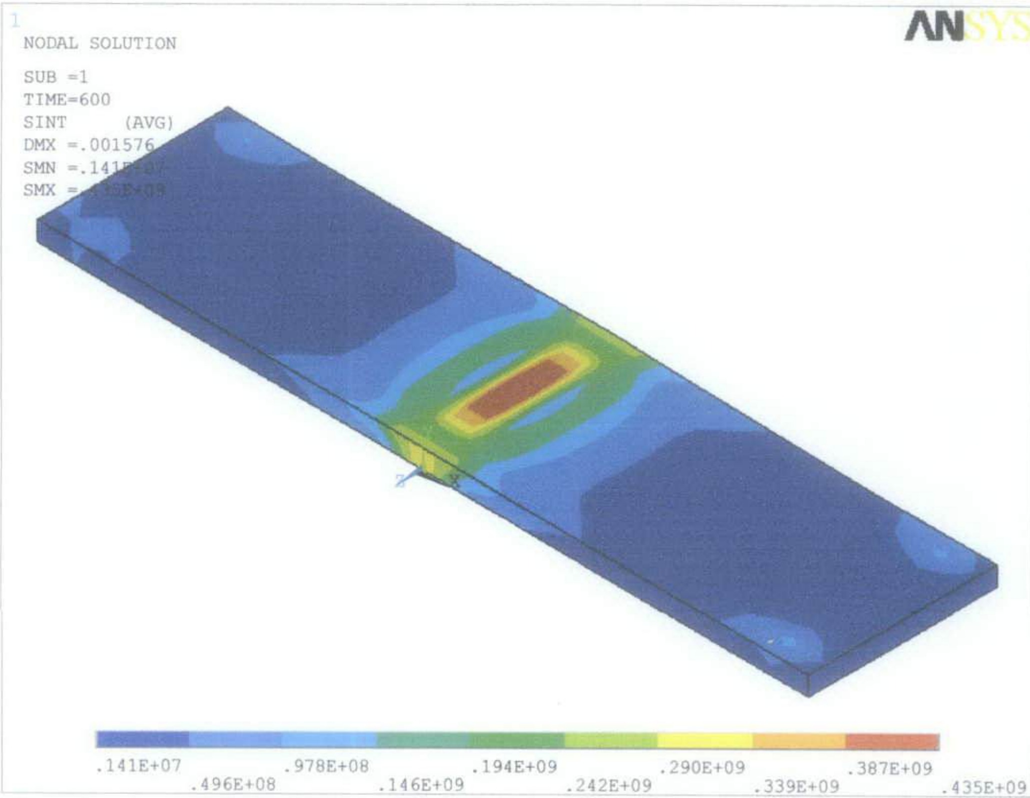


Figure 4.14: Stress distribution of welded plate

The stress distributions for three different thicknesses are being compared in Table 4.1. From the analysis, the result shows that the highest stress occurs at the 15 mm thickness of welded metal. It means the residual stress is affected by the factor thickness of the welded metal.

Table 4.1 Comparison of the maximum stress values

	Thickness 1 (5 mm)	Thickness 2 (10 mm)	Thickness 3 (15 mm)
Longitudinal tensile stress (MPa)	108	134	200
Longitudinal compressive stress (MPa)	1.17	1.32	1.49
Transverse tensile stress (MPa)	47.2	40.9	34.6
Transverse compressive stress (MPa)	2.68	3.79	4.3

From Table 4.1, the longitudinal tensile and compressive stress increase when the plate thickness increases. It shows that the transverse tensile stress decreasing when the thickness increases but the transverse compressive stress increase when the thickness increases.

In conclusion, the tensile and compressive stress increase significantly when the thickness of the welded plate increase. Although the transverse tensile stress shows the decreasing trend, the value between the maximum stress and minimum stress is much smaller than different in longitudinal tensile stress. It shows that, when we welding thick plate, the risk of the welded plate to crack or fail is increased.

The stress distributions of three different thicknesses have been plotted in Figure 4.15 to Figure 4.20. The stress distributions have being plotted in three concerned areas which are along the welding line, along the fusion area and along the mid section of the welded plate. The stresses that have been studied in this analysis are longitudinal stress and transverse stress.

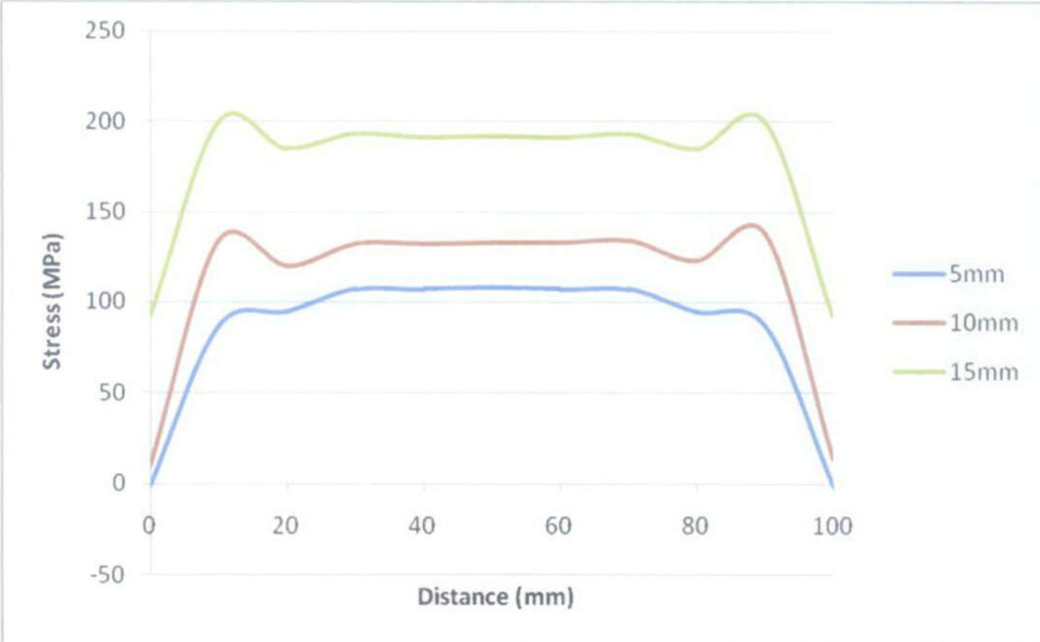


Figure 4.15: Longitudinal stress along the welding line

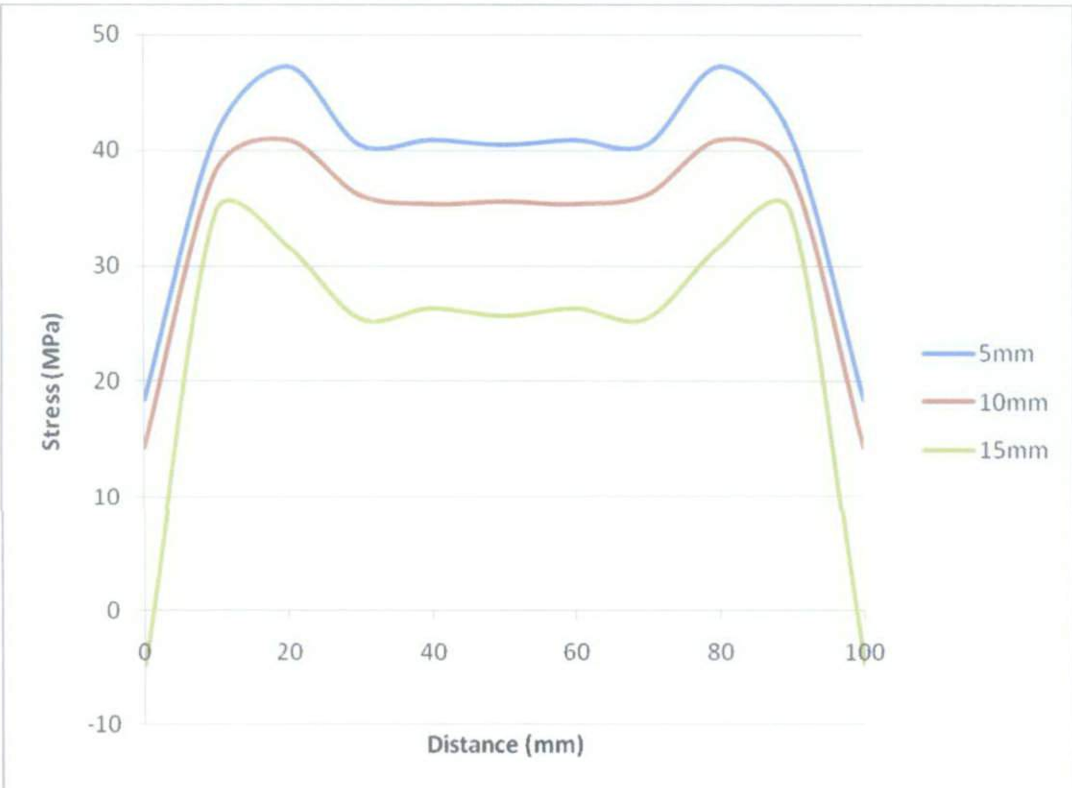


Figure 4.16: Transverse stress along the welding line

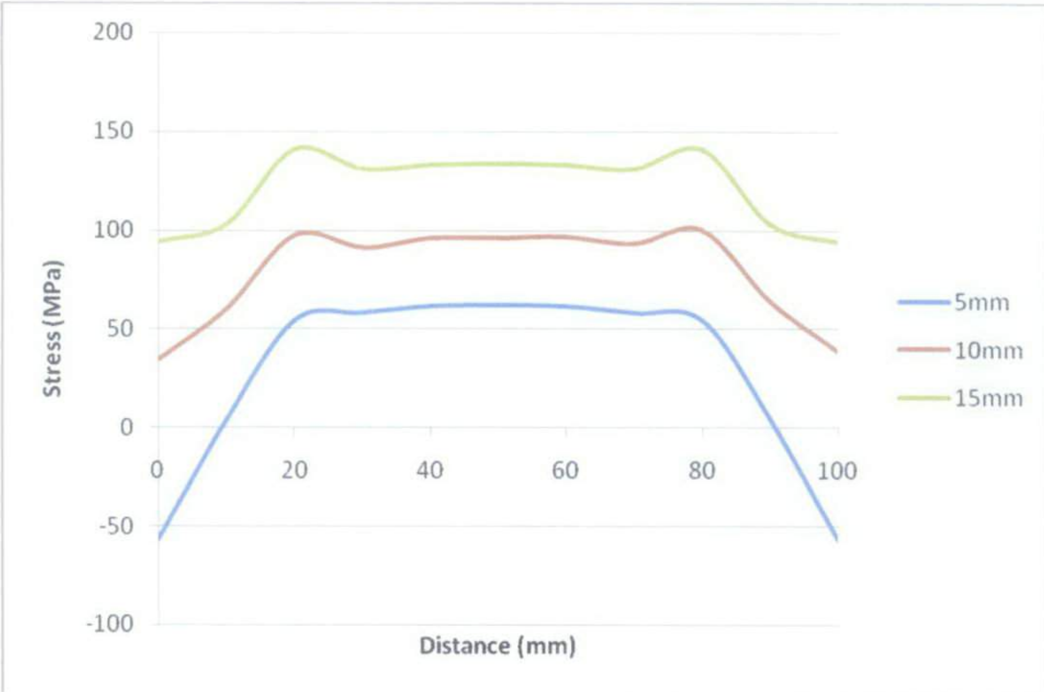


Figure 4.17: Longitudinal stress along the fusion line

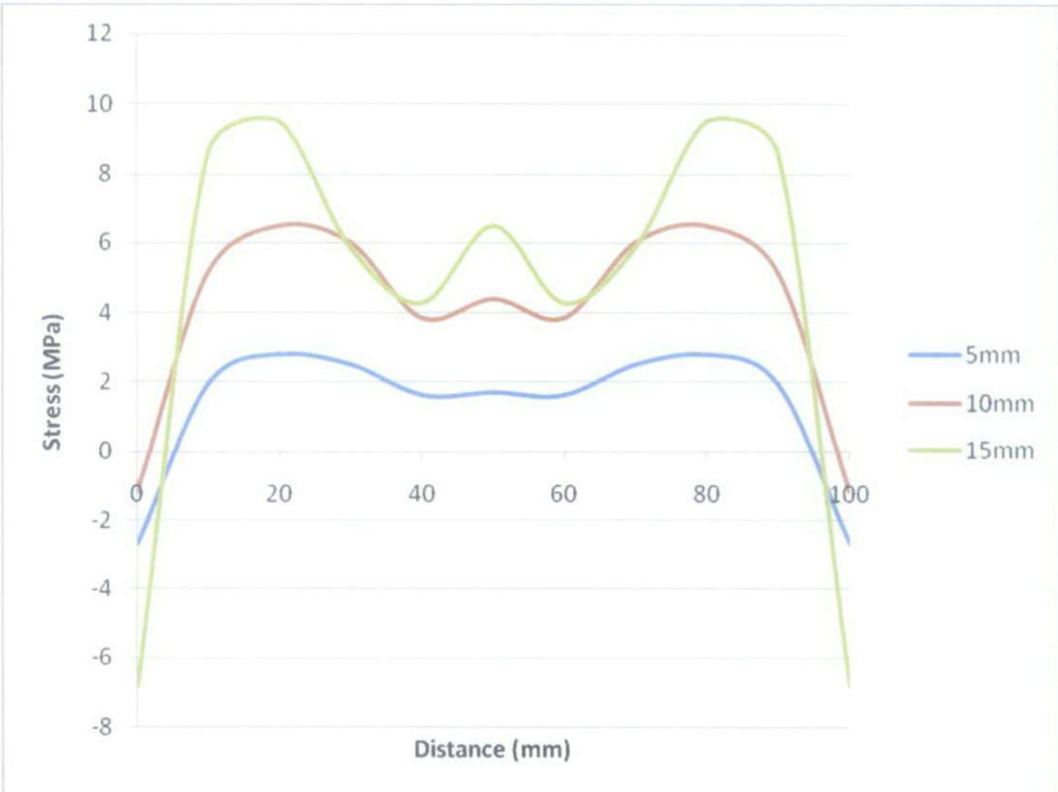


Figure 4.18: Transverse stress along the fusion line

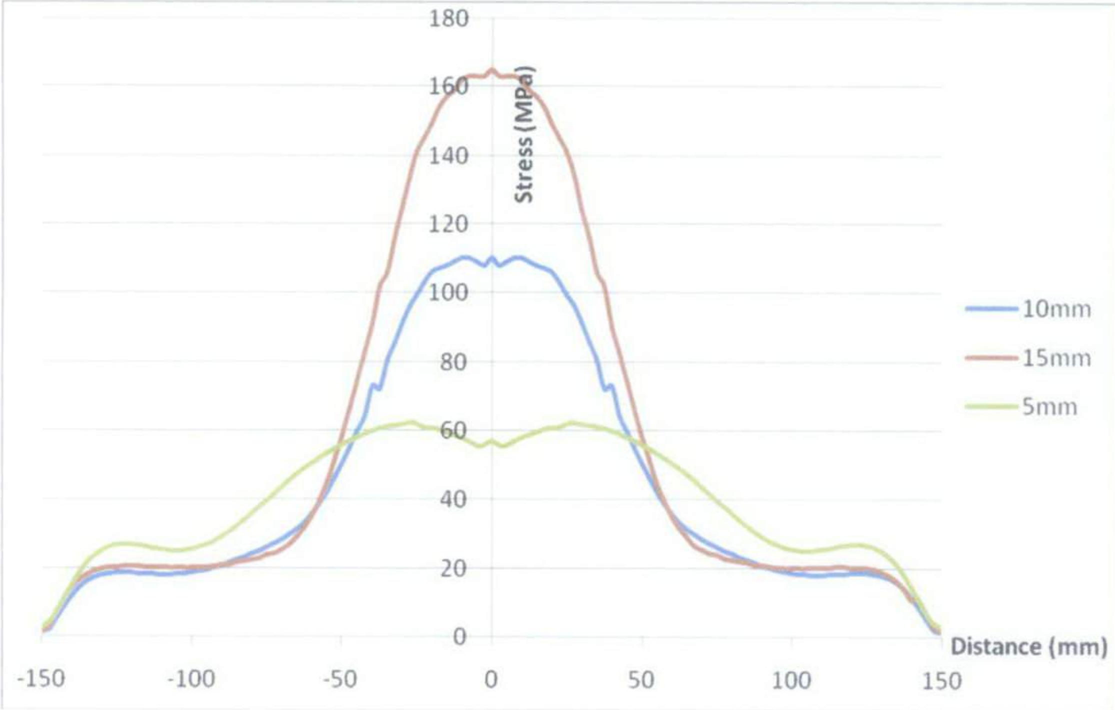


Figure 4.19: Longitudinal stress along the mid section

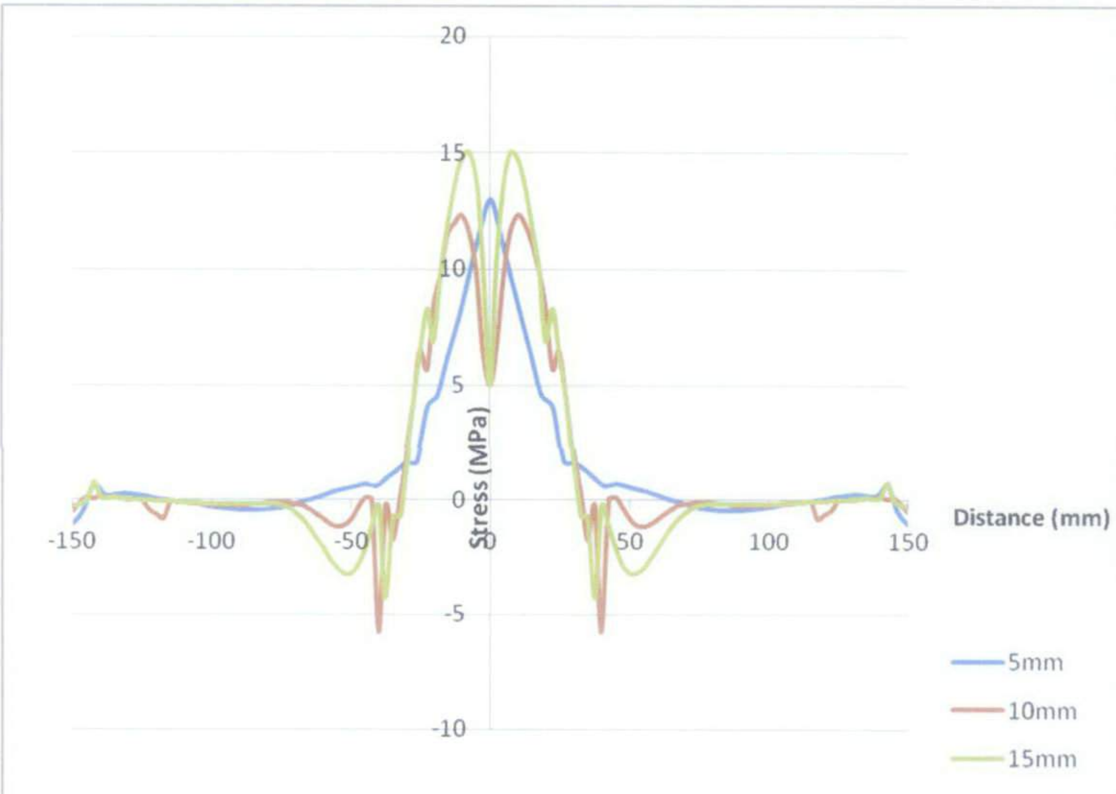


Figure 4.20: Transverse stress along the mid section

From the graph plotted, the highest tensile stress was found along the welding line as shown in Figure 4.15. This result shows that the welding area is the highest point where the heat loads were applied.

From the literature review, it is stated that the sensitization normally occurs at the fusion zone. From Figure 4.17, the value of longitudinal stress plotted is quite high. It means the residual stresses that affect the integrity also occur at this area and contribute to the material failure.

The highest stress occurs at the center of the welded plate and decreasing when we move farther from the welding area. From the Figure 4.19 and Figure 4.20, it shows that the highest stress point occurs at point 0 mm and keeps decreasing when the distance increases.

All plotted graph shows inconsistency where the increasing or decreasing trends do not occur smoothly. These situations shown in Figure 4.20 where there are some chaotic regions that occur from point -50 mm to 50 mm. It may occur because insufficient parameters applied to the model. In this simulation, a few parameters have been neglected in order to simplify the model, this assumption may affect the accuracy of the result. Every graph also shows increasing stress when the thickness is increased.

CHAPTER 5

CONCLUSION & RECOMMENDATION

5.1 Conclusion

Form the thermal analysis, it can be concluded that for the welding simulation with 5 mm/s welding speed and 35 x 10^6 W/m^2 of heat flux, it takes 16 minutes for the welded plate to cool down. For the structural analysis, the maximum stress that can be achieved by the welded is 435 MPa. The displacement of the welded plate is insignificant and it is about 1.19 x 10^{-3} m. It is found that if we choose thicker plate to be welded, the risk of residual stress to occur will increases.

As the conclusion, ANSYS can be used to simulate the welding process to analyze the thermal and structural analysis of the welded structure. This simulation is one of the cost effective method to analyze the welded structure. The boundary conditions and the material properties can be changed as desired to get different results for different type of material or different type of welding. The familiarization with the ANSYS software takes long time in order to understand many applications in ANSYS. All the applications and functions are useful to simulate the residual stresses on welded structure.

The conducted project has successfully simulated electrode travelling with filler and achieves expected result. For thermal analysis, all material properties, boundary condition and heat input have been applied. From the simulation, the model reaches its melting temperature successfully. For the structural analysis, the structural load is taken from the thermal result. The result obtained from the structural analysis is the residual stress distribution that remains on the welded plate after the welding process.

5.3 Recommendation

In order to get better results, a few recommendations for improvement can be done. In order to get better results, the size of element can be downsized. The simulation will analyze each element of the workpiece, it means if the elements have smaller size, the more result can be obtained from the analysis.

The filler material can also be treated by not having the same material with the work piece. It will affect the thermal and structural behavior of the filler material and more representative of the real life situation.

CHAPTER 6

REFERENCES

[1] Ihab F.Z. Fanous, Maher Y.A. Younan and Abdalla S. Wifi, November 2003, *Study of Effect of Boundary Conditions on Residual Stresses in Welding Using Element Birth and Element Movement Techniques,* Mechanical Engineering Department, The American University in Cairo, Egypt.

[2] Li Yajian, Wang Juan, Chen Maoai and Shen Xiaoqin, 30 July 2003, *Finite Elemet Analysis of Residual Stress in The Welded Zone of High Strength Steel,* Key Laboratory of Liquid Structure and Heredity of Materials, Ministry of Education, Shandong University, Jinan, China.

[3] B. Gideon, L. Ward, and D.G. Carr, 2006, *Residual Stress Determination of Duplex Stainless Steel Welds and Their Susceptibility to Intergranular Corrosion,* RMIT University, Melbourne, Australia.

[4] Robert N Gunn 1997, *Duplex Stainless Steels,* England, Abington

[5] Dragi Stamenkovic, and Ivana Vasovic, 2009, *Finite Element Analysis of Residual Stress in Butt Welding Two Similar Plates*, Scientific Technical Review, Vol.LIX, No.1.

[6] M Sundar, G Nandi, A Bandyopadhyay and S C Roy, 2005, *Assessment of Residual Stress and Distortion in Welding by Finite Element Method,* Preceedings of the International Conference on Mechanical Engineering 2005, Dhaka, Bangladesh.

[7] Publishing.TMR Stainless, 2009, *Practical Guidelines for the Fabrication of Duplex Stainless Steel,* International Molybdenum Association (IMOA), London, UK.

[8] Sand Meyer Steel Company 2008, date accessed: 27 Sept 2010, <http://www.sandmeyersteel.com/2205.html#5>.

[9] The A to Z of materials 2010, date accessed: 6 Oct 2010, <http://www.azom.com/details.asp?ArticleID=958#_Physical_Properties>.

[10] W. Fricke, 22 Nov 2005, date accessed: 24 August 2010, <http://onlinelibrary.wiley.com/doi/10.1002/mawe.200500933/>.

[11] Mato Peric, Dragi Stamenkovic, and Vili Milkovic, 2010, *Comparison of Residual Stresses in Butt-Welded Plates Using Software Packages Abaqus and Ansys*, Scientific Technical Review, Vol.60, No 3-4, pp.22-26.

[12] Changhee Lee, Jong-Bong Lee, Dong-Hwan Park and Suck-Joo Na, June 2008, date accessed: 12 Sept 2010, <http://www.scientific.net/MSF.580-582.439>.

[13] Karlsson C.T. 1998, *Finite Element Analysis of Temperature and Stresses in a Single-Pass Butt-Welded – Influenced of Mesh Density and Material Modeling,* Engineering Computational Volume 6.

[14] O.C Zienkiewicz and T.L Taylor, 2005, *The Finite Element Method: Its Basis and Fundamental,* Sixth Edition.

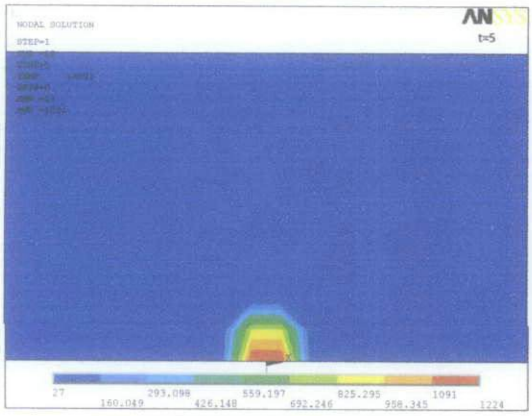
[15] Panagiotis Michaleris and Andrew Debicarri, April 1996, *Prediction of Welding Distortion,* Journal of Edison Welding Institute, Columbus, Ohio.

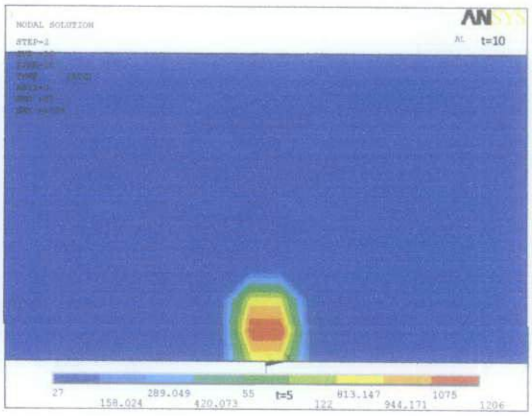
[16] Goldak, J., 1990, *Keynote Address: Modeling Thermal Stresses and Distortion in Welds,* Recent Trends in Welding Science and Technology, ASM International.

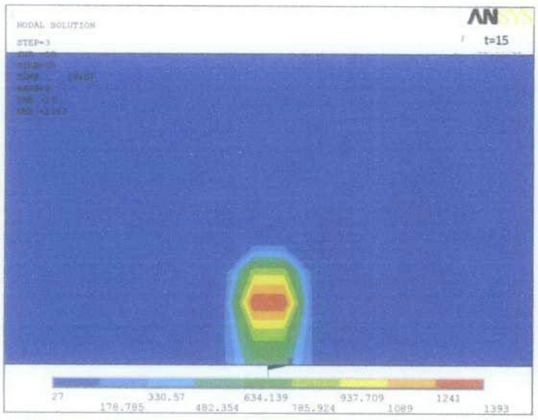
[17] Wilkening, W. W., Snow. J. L., 1993, *Analysis of Welding-Induced Residual Stresses with the Adina System,* Computers & Structures Volume 47

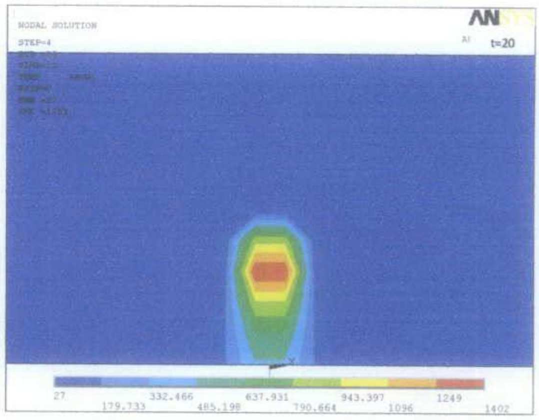
Appendix A

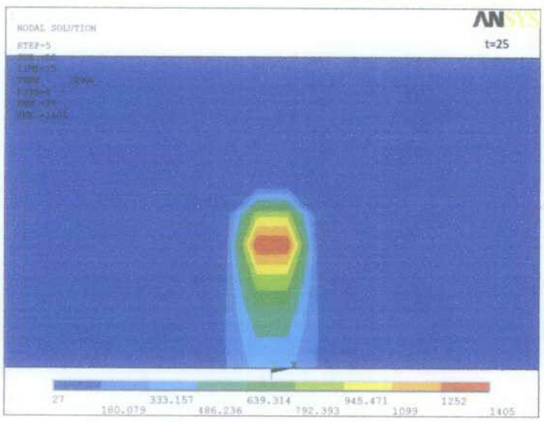
Temperature distribution of welded plate

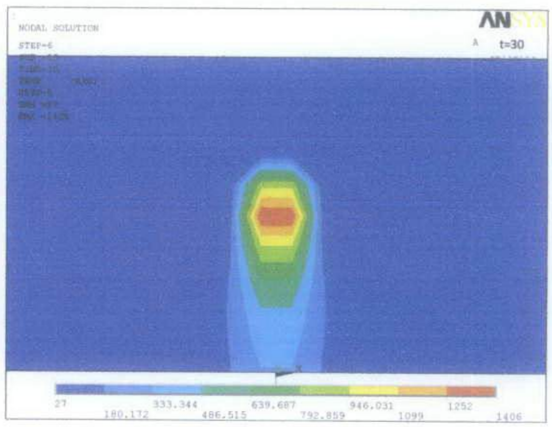


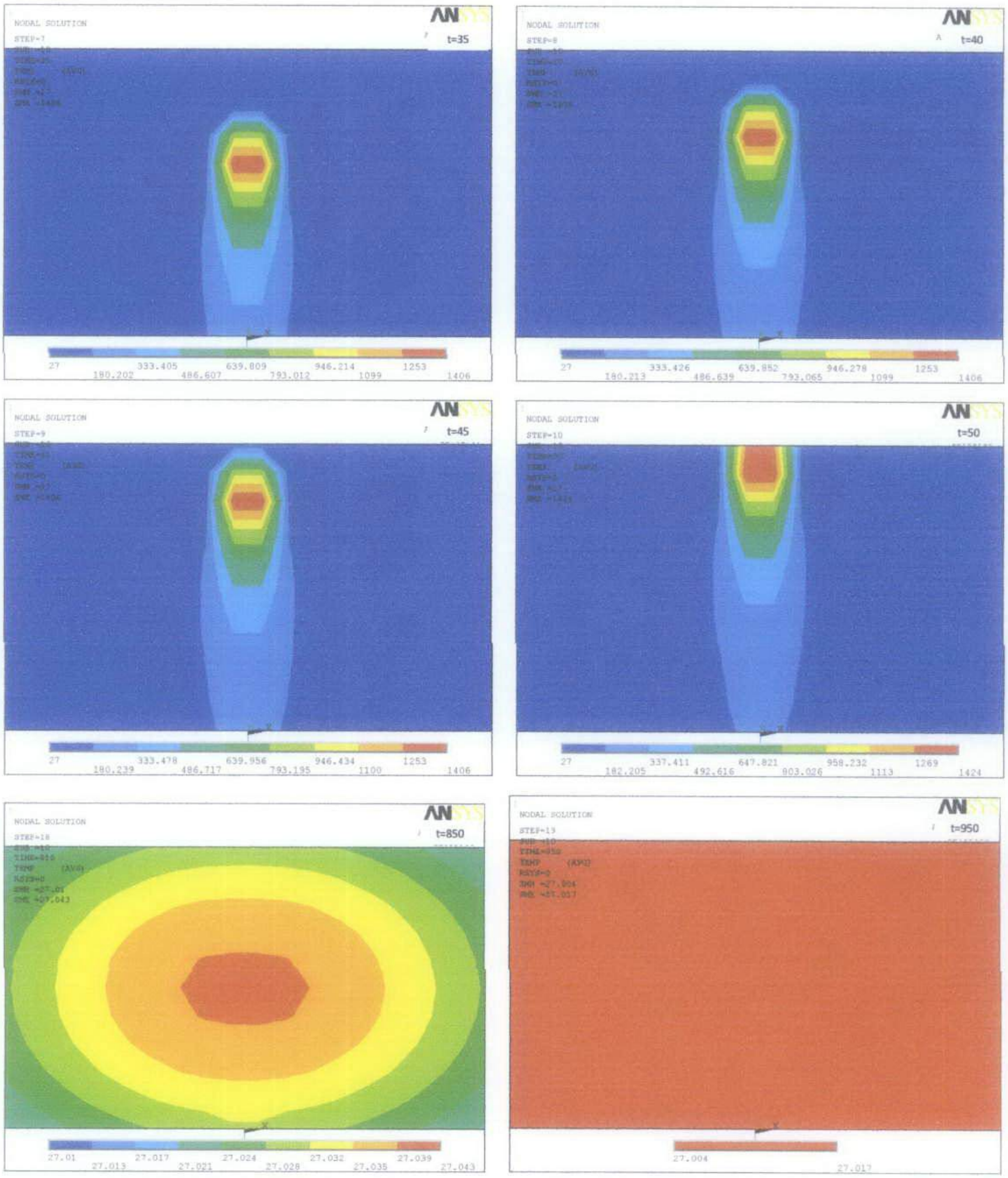











NODAL SOLUTION
t=35
27
180.202
333.405
486.607
639.809
793.012
946.214
1099
1253
1406
NODAL SOLUTION
t=40
27
180.213
333.426
486.639
639.852
793.065
946.278
1099
1253
1406
NODAL SOLUTION
t=45
27
180.239
333.478
486.717
639.956
793.195
946.434
1100
1253
1406
NODAL SOLUTION
t=50
27
182.205
337.411
492.616
647.821
803.026
958.232
1113
1269
1424
NODAL SOLUTION
t=850
27.01
27.013
27.017
27.021
27.024
27.028
27.032
27.035
27.039
27.043
NODAL SOLUTION
t=950
27.004
27.017

Appendix B

The Gantt chart for FYP I

No	Activities	Week														
		1	2	3	4	5	6	7		8	9	10	11	12	13	14
1.0	Selection of Project Topic								Mid-Semester Break							
2.0	Project title award															
3.0 3.1 3.2	Preliminary Research Gathering resources and information Journal study & review															
4.0 4.1 4.2	Research continuation Further research on residual stresses in welded structures Identify project methodology															
5.0 5.1 5.2 5.3 5.4	Familiarization of ANSYS Familiarize with the basic of ANSYS 2D & 3D modeling in ANSYS Familiarization on meshing process Familiarize with load application and boundary conditions															

The Gantt chart for FYP II

No	Activities	Week 1	2	3	4	5	6	7		8	9	10	11	12	13	14
1. 1.1 1.2	Research continuation Further research on welding simulation in ANSYS Study on boundary condition of welding simulation								Mid-Semester Break							
2. 2.1 2.2 2.3 2.4 2.5	Modeling Model 1 (Thermal analysis) Meshing the Model 1. Applying material properties Applying boundary conditions. Applying heat flux. Interpret result.															
3. 3.1 3.2 3.3 3.4 3.5	Modeling model 2 (Structural analysis) Meshing the Model 2. Applying material properties Applying boundary conditions. Applying temperature load. Interpret result															
4. 4.1 4.2	Stress Analysis Plotting stress graph Interpret result															
6.	Compare FEA result with the references journal															