CHAPTER 1

INTRODUCTION

1.1   Background of Study

Welding is a very common process in manufacturing field. Welding process is a fabrication or sculptural process that joins materials together by causing coalescence. This is often done by melting the work pieces and adding a filler material to form a pool of molten material that cools to become a strong joint, with pressure sometimes used in conjunction with heat, or by itself, to produce the weld. This is in contrast with soldering and brazing, which involve melting a lower-melting-point material between the work pieces to form a bond between them, without melting the work pieces.

In ships fabrication industries, arc welding process is commonly welding type used in this sector. There are four types of arc welding that is Shielded Metal Arc Welding, Gas Tungsten Arc Welding, Shielded Metal Arc Welding, and Flux Cored Arc Welding. The arc produced from this welding type generates a very large amount of heat and a very bright light. This arc generated in the welding process is from the high voltage and current power supply.

The implementation of numerical techniques to model manufacturing processes has the advantage of improving the product, perfecting the process, reducing the scrap rates, and improves the efficiency of the manufacturing process. Modelling can be used as a tool in many stages of the life of a product.

The complex nature of the welding process causes difficulty in analysing and modelling by numerical methods. The complexity include the temperature dependent of the material properties, non-linear boundary conditions, moving heat sources, the phase changes and transformations, complex residual stress states and the difficulties of making experimental measurement at the high temperatures. As result from these complexities, finite element modelling of the weld process must include complex thermo-mechanical interactions, filler material deposition and moving heat sources when simulating a welding process.

These complexities of the welding nature lead to a problems to determine the causes of the distortion of the product. The residual stress can reduce the life cycle of the product. There are many factors that lead to welding distortion of the final product, not only from one source. Since welding is difficult to use an analytical solution in calculating the temperature distribution and stress distribution of the product, numerical solution is the best way to analysing the temperature and stress distributions.

The result of the simulation can be used to explain the physical essence of some complex phenomena in the welding process explicitly and also can be used as the basis for optimizing the welding parameters.

1.2   Problem Statement

When continuous welding is applied on 2 plates, local hot spots are formed. As a result, there are possibilities that the plate will distort due to the heat applied by welding. This project studies the cause of the distortion and how to minimize the deformation on the plate. Minimizing the deformation is important because it will reduce or eliminate the repair work after the welding on the plate finished. Distortion can pose a major problem, since the final product is not the desired shape.

1.3   Objective of This Project

The main objectives of this project are:

- Develop a finite element model of a welding process.
- Compute the temperature field in a welded plate.
- To determine the deformation of the plate.

1.4   Scope of Study

This scope of study is this project

- Study the temperature distribution along the welded plate in the welding process.
- Study the stress distribution on the plate.
- Study the deformation phenomena on the plate after the heat from the welding process applied.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

There are two main categories for welding which is fusion and solid phase welding process. In fusion welding, two edges or surfaces to be joined are heated to the melting point and, where necessary, molten filler material is added to fill the joint gap. For solid phase welding, two clean, solid metal surfaces are brought into sufficiently close contact for a metallic bond to be formed. Solid phase welding can be accomplished at temperatures as low as room temperature.

The basic equipment used in Shielded Metal Arc Welding is shown in Figure 2.1 below. By using a heat source with sufficient power it is possible to fuse through a complete section of very thick plate. The weld pool produced is difficult to control and the heat affected zone (HAZ) of such welds has a relatively coarse grain, adversely affecting the mechanical properties of the steel [1].
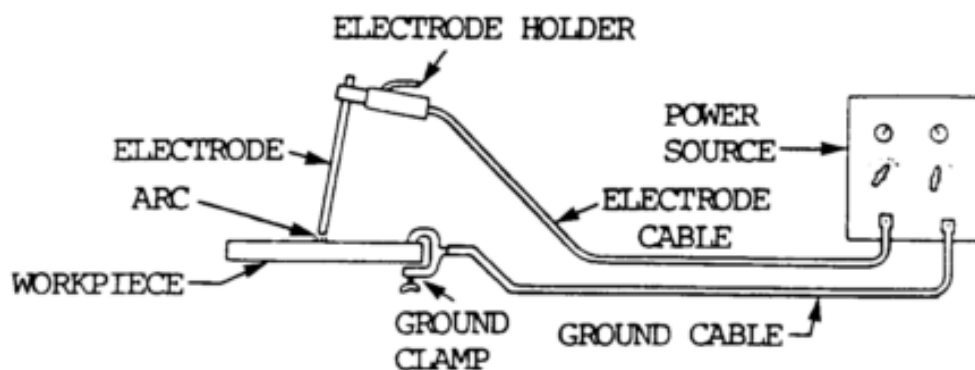

Figure 2.1: Shielded Metal Arc Welding equipment set-up.

2.2 Heat Transfer

An understanding of the nature of heat transfer is essential for the proper appreciation of the heat effect of fusion welding. Heat transfer theory can indicate the minimum heat input rate to form a weld of any given width, and the essential variables which govern the heating rate and cooling rate in the heat affected zone and the weld metal. Heat affected zone (HAZ) is the area of base metal which has had its microstructure and properties altered. Figure 2.2 below shows the cross-section of a welded butt joint, with the darkest gray representing the weld or fusion zone, the medium gray the heat affected zone (HAZ), and the lightest gray the base material.
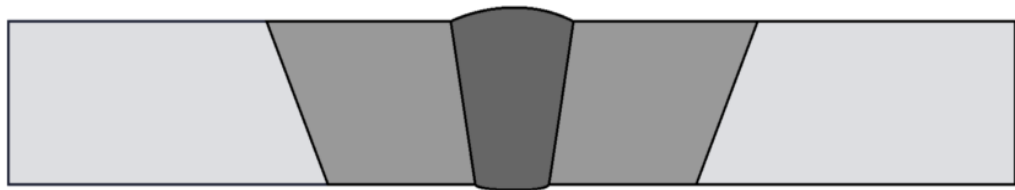

Figure 2.2: Cross section area of a welded butt joint.

The electric arc heat source is known as a surface source, which applies heat over a small area on the metal surface. In most fusion welding a continuous moving source is used. The continuous moving source has a special characteristic: once steady condition has been achieved, the temperature distribution relative to the heat source is stationary **[2]**. This condition is known as the quasi-stationary state and in most cases it is convenient in developing equation regarding the source as stationary and the heat flow medium (the base metal) as moving. Equation 2-1 shows the conductional heat in a homogeneous isotropic solid in term of rectangular co-ordinates **[3]**:

$$\frac{\delta^2 T}{\delta x^2} + \frac{\delta^2 T}{\delta y^2} + \frac{\delta^2 T}{\delta z^2} - \frac{1}{\alpha}\frac{\delta T}{\delta t} = 0 \qquad\qquad \text{Equation 2-1}$$

The power of the welding process is the product of the current *I* and voltage *V* passing through the arc. The power is converted to heat, but due to convection, conduction and radiation and spatter, heat losses occur. The effect of heat losses is expressed by the arc efficiency coefficient, ŋ$_{arc}$, in the calculation of the welding power.

$$P = \eta_{arc}VI$$ Equation 2-2

Welding arc is usually maintained between an electrode and work piece. Such an arc is constricted at the rod and spreads out towards the plate. The column temperature is highest near the electrode. Having a clear understanding of the temperature and heat flux distribution is very important for the load application in weld modelling. An accurate representation of the thermal flux in the finite element method (FEM) software package will help with more accurate and reliable results. Stamenković **[10]** used Equation 2-3 to model the heat input during welding by the equivalent heat input which includes body heat flux as follow:

$$Q = \eta \frac{UI}{V}$$ Equation 2-3

Where *ŋ* is the efficiency, *v* is the travel speed, *V* and *I* are the arc voltage and current. In this paper, the value of the voltage and current is 24V and 120A. The arc efficiency is 0.85% in SMAW, while the travel speed is 5mm/s and 10mm/s.

The heat input is modelled as a heat flux applied on the surface of the element model. The equation of heat flux in this model is:

$$q = \frac{Q}{m}$$ Equation 2-4

Where *q* is the heat flux (W/m^2), *Q* is the heat input from the arc (Ws/m) and *m* is the mesh size of the model.

2.3 Welding Distortion.

Welding distortion due to a weld in a plate arise primarily because the strip of material which has been melted contracts on cooling from melting point to room temperature. Welding distortion can be classified into three types of distortion, which is angular, longitudinal and transverse distortion **[2]**.

The contraction of weld metal as it cools after deposition causes shrinkage that takes place simultaneously in all directions, and therefore it causes several types of distortion as in Figure 2.3. The level of welding distortions depends on many variables, such as heat input, material thickness, electrode speed, material type and etc.
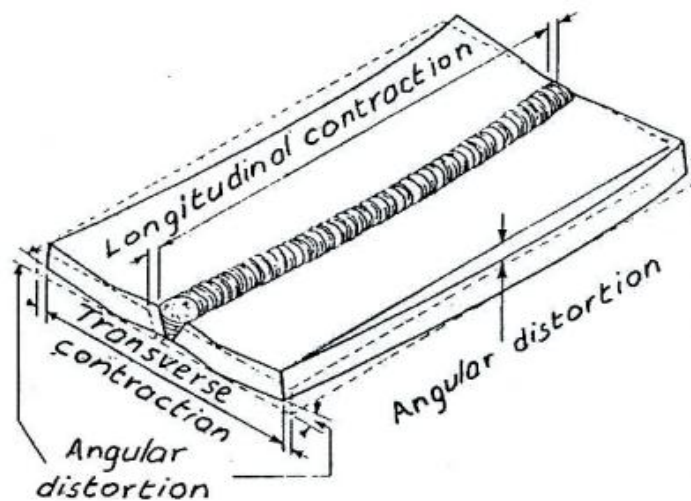

Figure 2.3: Welded plate distortion

If the contraction of the weld was continuously happen, the longitudinal contraction of the weld would be equal to αT_m where α is the thermal expansion and T_m the melting temperature. Assuming only elastic deformation the corresponding stress would be $E\alpha T_m$ where E is the young modulus of the material. The value of $E\alpha T_m$ is greater than the elastic limit, so that plastic deformation of the weld takes place during cooling and the residual stress in the weld exceeds the elastic limit.

2.4 Residual Stress

Residual stress is stress that remain in the base metal after the original cause of the stress has been removed. Residual stress is often the cause of fracture and degrading the buckling strength os a structure **[5]**. Residual stress distribution is very complex and difficult to predict because it is strongly affected with many parameter, such as geometry of the plate, thickness, joint type, mechanical and physical properties, arc properties and also the electrode travel speed **[10]**. Basically, there is two method used in welding residual analysis, which is destructive and non-destructive **[5]**. Recently, there many paperwork involved in order to solve this complex engineering problem using finite element method to calculate residual stress. Stamenković **[10]** use the finite element analysis to make a welding simulation and predict the residual stress in butt welding of two similar carbon steel plate. He obtained the values of the axial residual stress from the finite element method are very close to the experimental result. The result show in Figure 2.4:
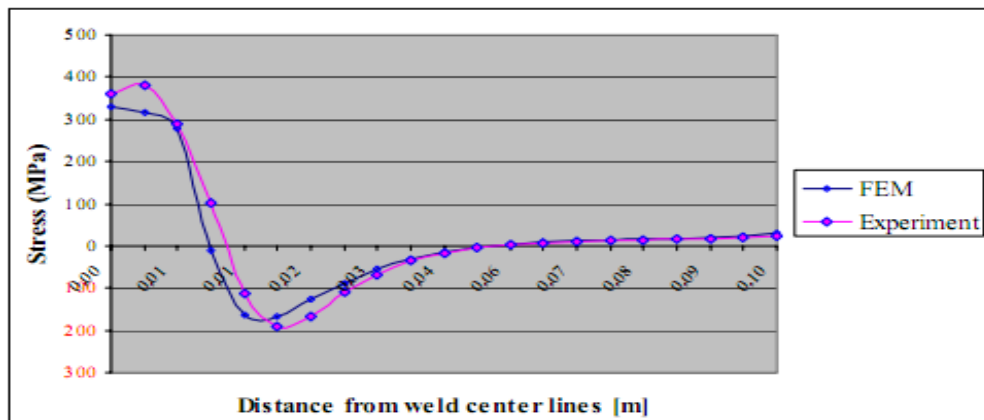

Figure 2.4: Comparison of the FEM with experimental result for axial residual stress.

2.5 Application of Finite Element Method in Welding Process

The finite element method is a computational technique used to obtain approximate solution of boundary value problem in engineering. A simple description of of FEM is the cutting of a structure into several elements, describing the behavior of each element in a simple way, reconnecting the element at nodes as if it were pin or drops of glue that held the elements together. There many thesis in the past using this method mainly focused on temperature field estimation and welding deformation. Weld modelling guidelines on element mesh, boundary conditions and material properties from the previous paperwork are be taken in accounting in this project.

2.5.1   Thermal and Structural Analysis.

Thermal and structural analysis is done separetely in simulating welding process. In Capriccioli and Frossi paperwork **[4],** and Stamenković **[10]**, they assumed that the thermal calculation at a given time time is independent from the strucutral results obtained at a previous time according to the point , so the thermal and the mechanical analysis can be uncouple. Change in mechanical result, do not change the thermal results, but change in thermal cause changes in mechanical result.So, thermal and mechanical analysis is done seperately, and thermal analysis is done first.

In Chang and Teng **[5]** paperwork, they done a multi pass welding sequences, and during each pass, thermal stress are calculated from the thermal distribution from the thermal model. The temperature field result is in Figure 2.5. Then, in the mechanical analysis, the thermal history from the thermal analysis obtained was an input for the structural anlysis as a thermal load. Then, the stress result will be calculated, and deformation of the plate can be seen.
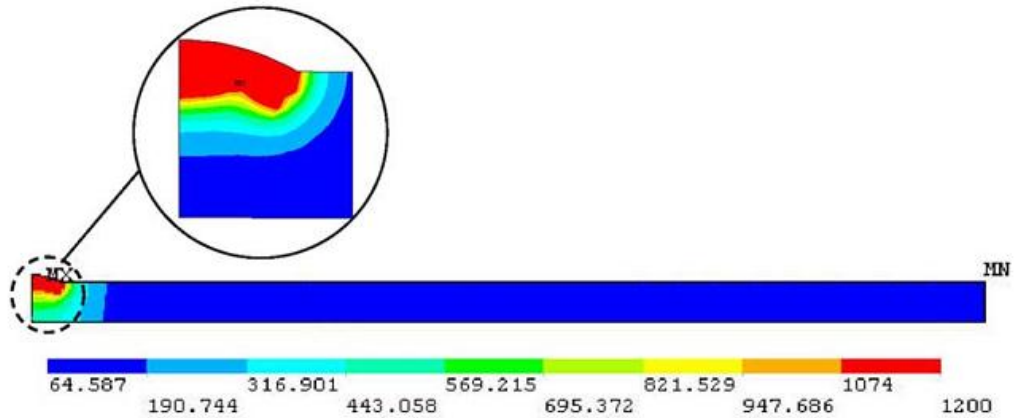
Figure 2.5 : Temperature field of the thermal analysis after third pass welding

2.5.2   Modelling Assumption

In the modelling of the welding process certain assumption were made to simplify the model. Some of the assumption are:

- Base plate and electrode had the same mechanical properties.
- In previous paperwork, they all assumed the mechanical properties are time dependent. Chang and Teng **[5]** assume that when temperature increase, the modulus of elasticity, yeild stress and thermal conductivity for ASTM A36 carbon steel decrease as illustrated in Figure 2.5:
- The analysis is based on quasi-steady state the heat source is moving at a constant velocity
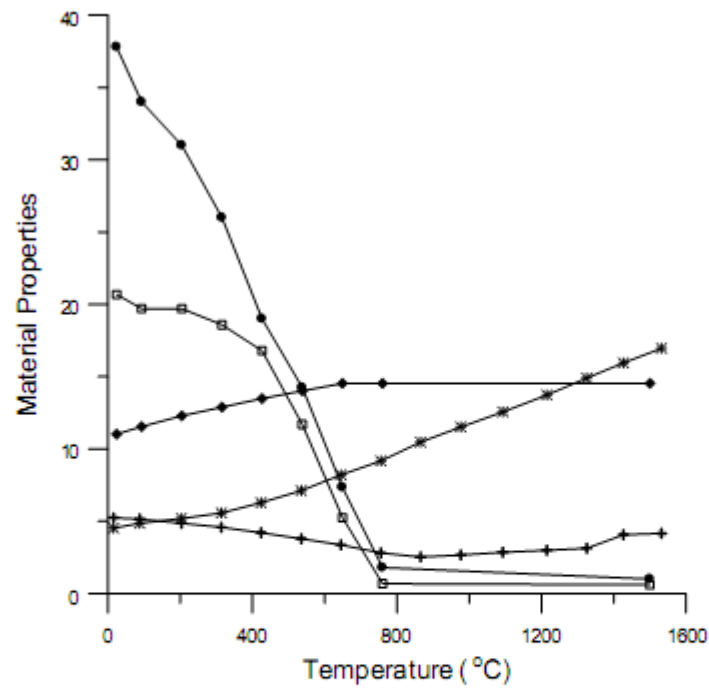
Symbol	Material Properties	Unit
●— σ_y	Yield stress	$\times 10^7$ Pa
□ —E	Young's modulus	$\times 10^{10}$ Pa
◆ — α	Thermal expansion	μ m/m °C
+—k	Conductivity	$\times 10$ W/m °C
*—c	Specific heat	$\times 10^2$ J/Kg °C

Figure 2.5: Temperature dependent material properties os ASTM A36 carbon steel.

- Convection and radiation effect are considered in Capriccioli and Frosi **[4]** model. For the other surface, except the top surface, the equation for the heat loss due to convection is,

$$q = h_c A(T - T_\infty) \qquad\qquad \text{Equation 2.5}$$

Where $h_c = 15$ W/m^{2o}C

- Displacements of the parts, during welding do not effect the thermal distribution of the parts themselves.

CHAPTER 3

METHODOLOGY

3.1 Research Methodology

In order to start this project, the author shall begin with identify all the work and thing that should be done. All the information gained is from the books and journals as reference, and the sources have to be verified that it is suitable for the research. In addition, the author must get familiarize with the meshing software that want to used in this project. This project is mainly using ANSYS software to simulate the model. The project flow for this project is illustrated on Figure 3.1.
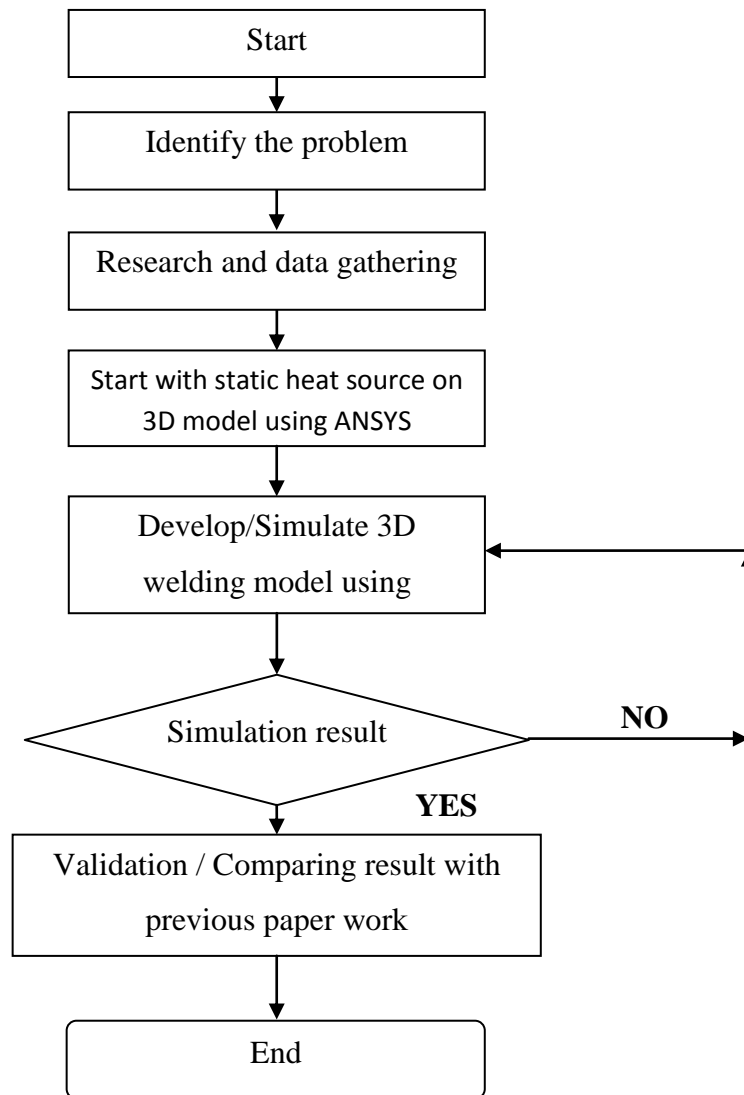
```
                    ┌─────────────────────────┐
                    │          Start          │
                    └─────────────────────────┘
                                 │
                                 ▼
                    ┌─────────────────────────┐
                    │    Identify the problem  │
                    └─────────────────────────┘
                                 │
                                 ▼
                    ┌─────────────────────────┐
                    │ Research and data gathering │
                    └─────────────────────────┘
                                 │
                                 ▼
                    ┌─────────────────────────┐
                    │ Start with static heat source on │
                    │    3D model using ANSYS  │
                    └─────────────────────────┘
                                 │
                                 ▼
                    ┌─────────────────────────┐
                    │   Develop/Simulate 3D    │◄──────────┐
                    │    welding model using   │           │
                    └─────────────────────────┘           │
                                 │                         │
                                 ▼                         │
                         ╱───────────────╲                 │
                        ╱  Simulation result ╲    NO        │
                        ╲                   ╱───────────────┘
                         ╲───────────────╱
                                 │ YES
                                 ▼
                    ┌─────────────────────────┐
                    │ Validation / Comparing result with │
                    │    previous paper work   │
                    └─────────────────────────┘
                                 │
                                 ▼
                    ╭─────────────────────────╮
                    │           End           │
                    ╰─────────────────────────╯
```

Figure 3.1: Flow Chart of the project.

3.2 Introduction to ANSYS Software.

ANSYS is engineering simulation software created by ANSYS, Inc. ANSYS is a general purpose finite element modelling package for numerically solving a wide variety of mechanical problem, used widely in industry to simulate the response of physical system to structural loading, thermal and electromagnetic effects. The software also includes solutions for both direct and sequentially coupled physics problems including direct coupled-field elements and the ANSYS multi-field solver. ANSYS use the finite element method to solve the underlying governing equations and the associated problem specific boundary conditions.

3.3 Element Birth and Death Technique in Welding Simulation

In ANSYS, the heat generated by the arc of the tip of the electrode is modelled using heat flux. The moving of the heat source or travelling time is modelled using element birth and death technique. In element birth and death, the desired element is turned off by means that the element is killed is not reacting on any of the input given. ANSYS program does not actually remove "killed" elements. Instead, it deactivates them by multiplying their stiffness by a severe reduction factor. This factor is set to 1.0E-6 by default, but can be given other values. Figure 7 shows the welding model that want to be developed in ANSYS. In order to simulate the electrode deposition to the weldment, birth and death technique is applied to the element along the welding direction. Initially all the element in the welding direction is being killed, the elements are not removed but are deactivated. After the filler material from the electrode is added, the element is activated by using birth technique. When activated, the element properties like stiffness, conductivity and all the other properties is return to their actual value.
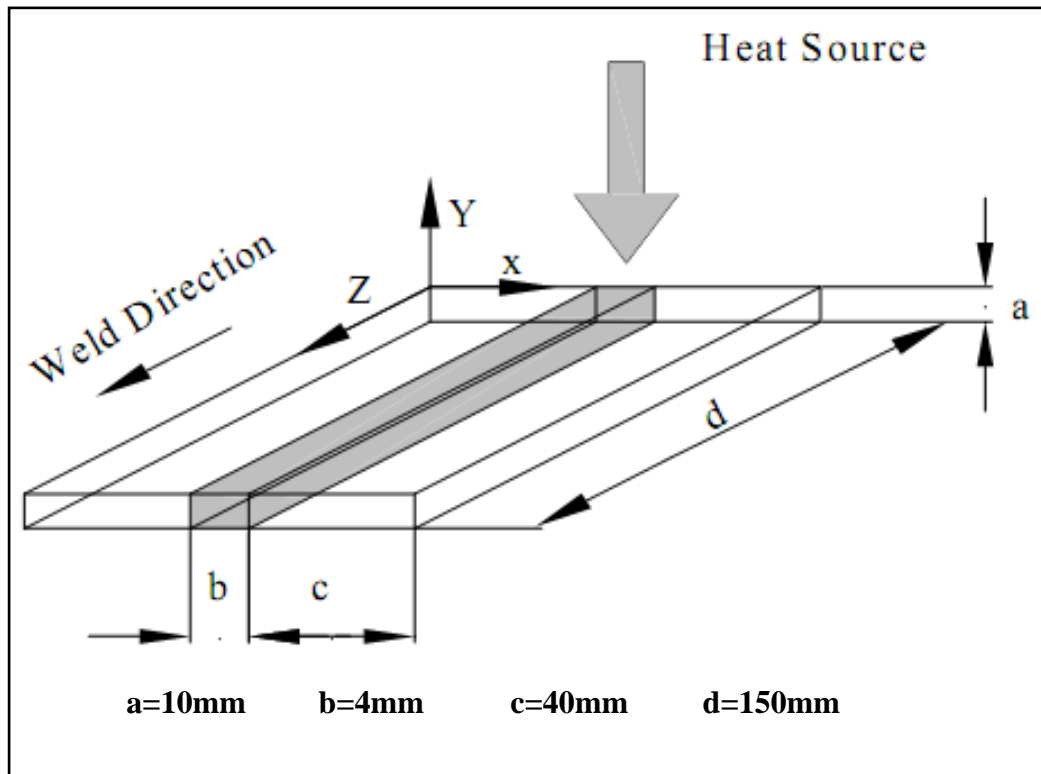
Figure 3.2: Welding model for this project

3.4 Steps in Developing Welding Model Using ANSYS

The step in modelling a simple 3D using ANSYS software:

1. Set the element type for the model. For thermal analysis, use Solid70, the characteristic for Solid70 is the element defined by eight nodes having three degree of freedom at each node.

2. Set the material properties for the model. The model used ASTM A36 low carbon steel with the mechanical properties of the material does change with temperature. The ASTM A36 material properties is as Table 3.1 below:

Table 3.1: Material properties of A36 low carbon steel.

Temperature (°C)	Specific heat (J/kg°C)	Conductivity (W/m°C)	Density (kgm-3)	Yield stress (MPa)	Thermal expansion coefficient (10-5/°C)	Young's modulus (GPa)	Poisson's ratio
0	480	60	7880	380	1.15	210	0.3
100	500	50	7880	340	1.2	200	0.3
200	520	45	7800	315	1.3	200	0.3
400	650	38	7760	230	1.42	170	0.3
600	750	30	7600	110	1.45	80	0.3
800	1000	25	7520	30	1.45	35	0.3
1000	1200	26	7390	25	1.45	20	0.3
1200	1400	28	7300	20	1.45	15	0.3
1400	1600	37	7250	18	1.45	10	0.3
1550	1700	37	7180	15	1.45	10	0.3

3. Create geometry of the model. Create 3 volumes for the model, volume number 1 and 3 as the work piece while volume number 2 as filler material. Dimension as per Figure 3.2. The generated model in ANSYS show in Figure 3.3 (*make sure to convert all units to m*).
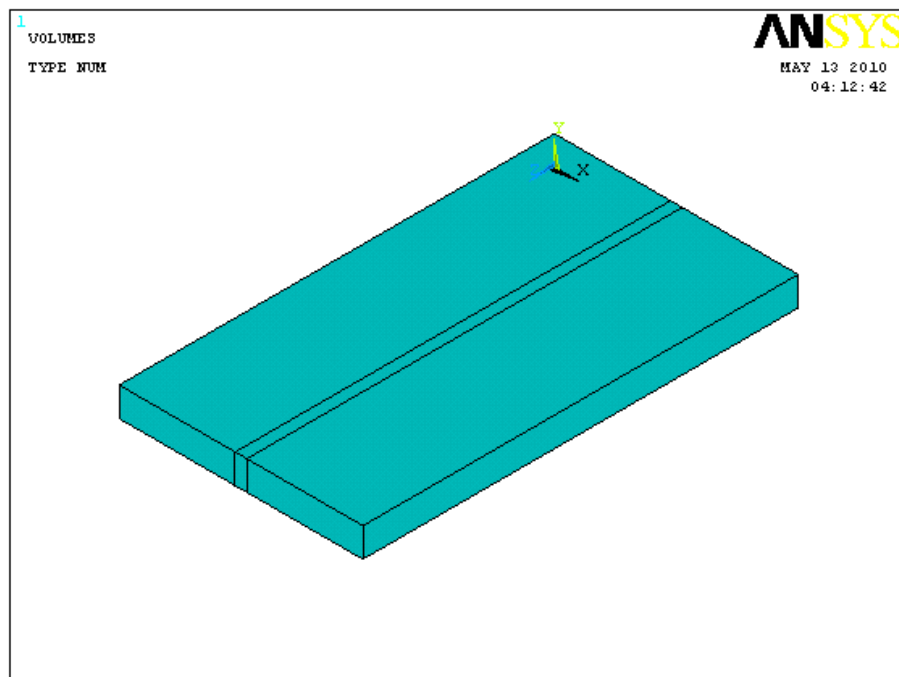
Figure 3.3: Model of 3D plate generated.

4. Glue all the volume.

5. Set the size of each element 0.005m in y-direction, 0.01 for z-direction and use bias command to set the meshed for x-direction as per Figure 3.4 below.
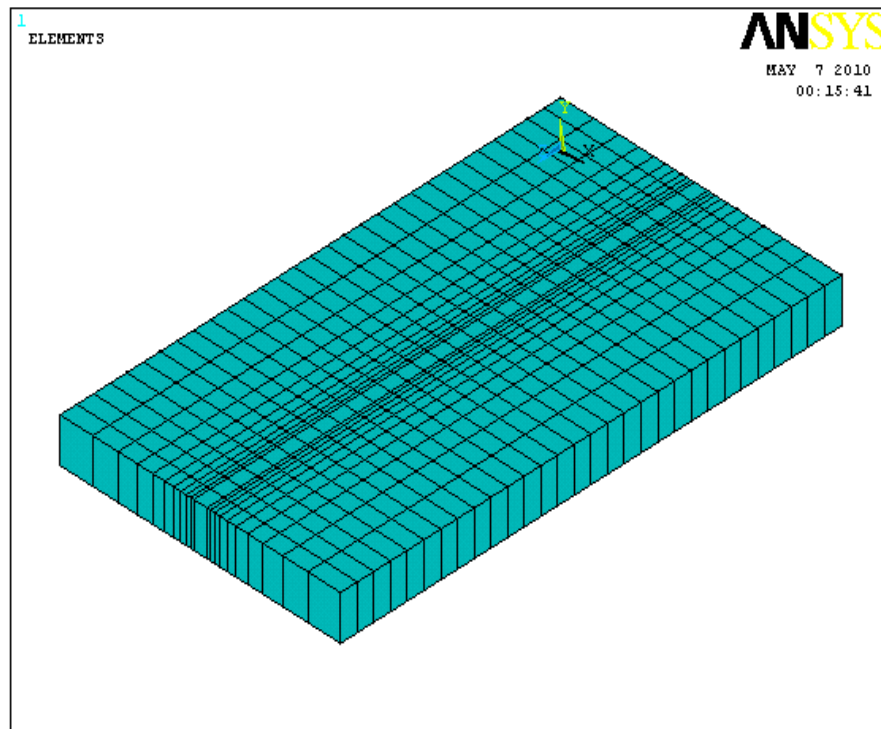
Figure 3.4: Meshed model generated.

6. Set the analysis type to transient.

7. Turn on the Newton-Raphson Solver. Due to a glitch in the ANSYS software, there is no apparent way to do this with the graphical user interface. Therefore, type NROPT,FULL into the command line. This step is necessary as element killing can only be done when the N-R solver is turned on.

8. Set the solution control: Solution>Analysis Type>Sol'n Controls, the the window in Figure 3.5 will pop up. Set the time at the end of the loadstep to 1s, and turn off the automatic time stepping. Set the number of substep to 1. This step is to set the solution done every 1second at the end of every loadstep. By this way we can control the electrode travelling speed.
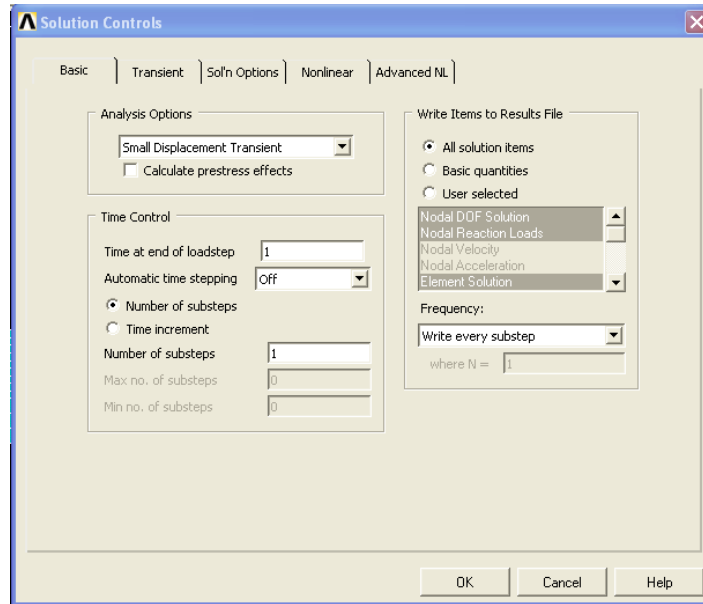
Figure 3.5: Setting the solution control.

9. Define the load input to the model. Set the initial temperature of the plate to 27°C. In this model, we assume the heat from the arc as a heat flux. Heat flux is applied to the element. In this model, the heat flux value is:

   - Using equation $Q=\eta VI/m^2$, where η, V and I for SMAW welding process is .85, 24V and 120A.

     $Q=0.85(24\times120)/(0.005\times0.004)$

     Heat input, $Q = 122.4 \times 10^6$ W/m^2

   - Applied the heat convection coefiicient and the bulk air temperature to 15W/m^{2o}C and 30°C.

10. Kill all elements in volume 2 in order to simulate the filler material deposition in welding process except for the first element.

11. Solve this current load step and plot the nodal results.

12. Restart this analysis from previous load step. Set the time at the end of laodstep to 2 second. Next, to simulate the electrode movement, apply the heat flux to the next element, and delete the previous heat flux on the previous element. Activate the new element (*birth*). Solve this load step. Repeat this step until the last element on volume 2.

13. Save the analysis in **.rth** file.

14. Next, move to the structural analysis. First, change the element type from thermal to structural. Set the analysis to static analysis.

15. Apply load to the model by applying the previous thermal analysis result that have been save in **.rth** file and solve the current load step.

16. Plot the result and save all results.

CHAPTER 4

RESULT AND DISCUSSION

4.1 Thermal Analysis for Electrode Travelling Speed, v=5mm/s
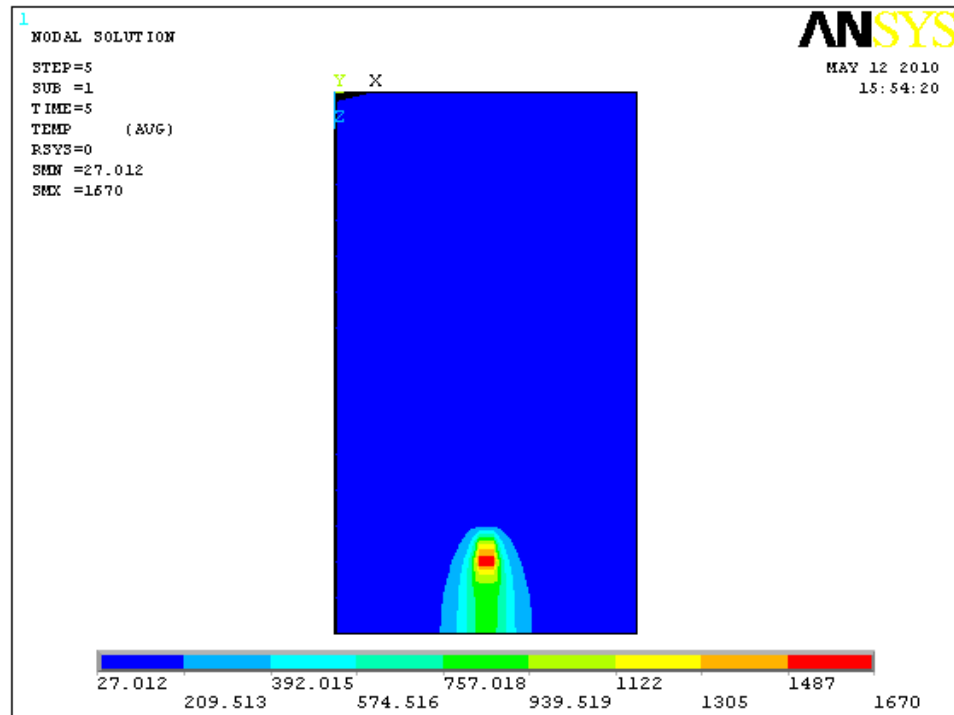
Figure 4.1: Temperature distribution of the plate at t=5s

Figure 4.1 represent the temperature distribution on the plate. The heat flux is set to 122×10^6 W/m^2 as the heat input to one element per unit area per time. The initial temperature of the plate set to 27^oC and the temperature of the surrounding air is set to 30^oC with convection coefficient of 15 W/m^2.K. In the Figure 4.1, we can see the temperature distribution at the welded plate at time, t=5s with welding speed of 5mm/s. The maximum temperature is 1670^oC. The maximum temperature located at the centre of the heat flux, which is in the real case is the arc produced by electrode. The temperature that is far from the welded area is maintained at 27^oC which is the minimum.

The temperature distribution at the area near the welding direction can be seen in Figure 4.1, or we can say this area is the HAZ area. Temperature distribution at the HAZ area can explain the behaviour of the HAZ area which is the microstructure and properties in the area is altered. The microstructure change because of the heat applied in the welding process. The HAZ area can be clearly seen in every second in this thermal analysis.
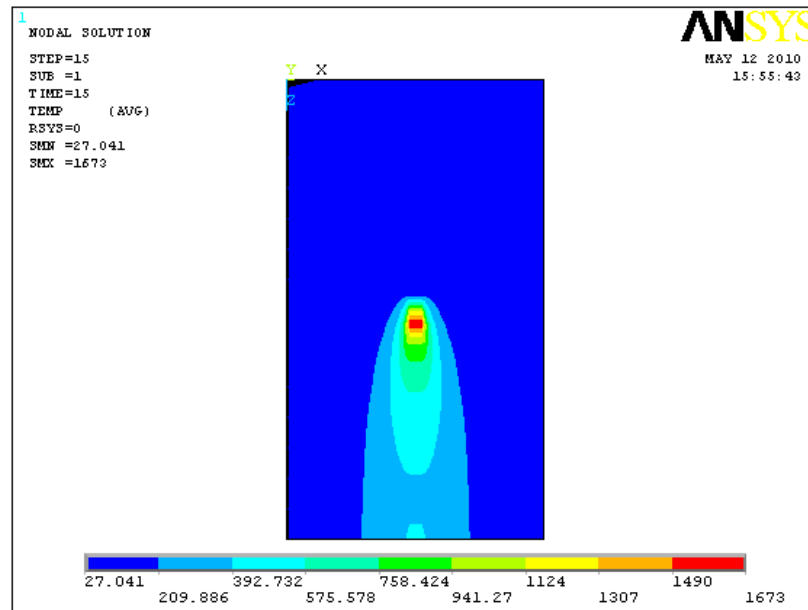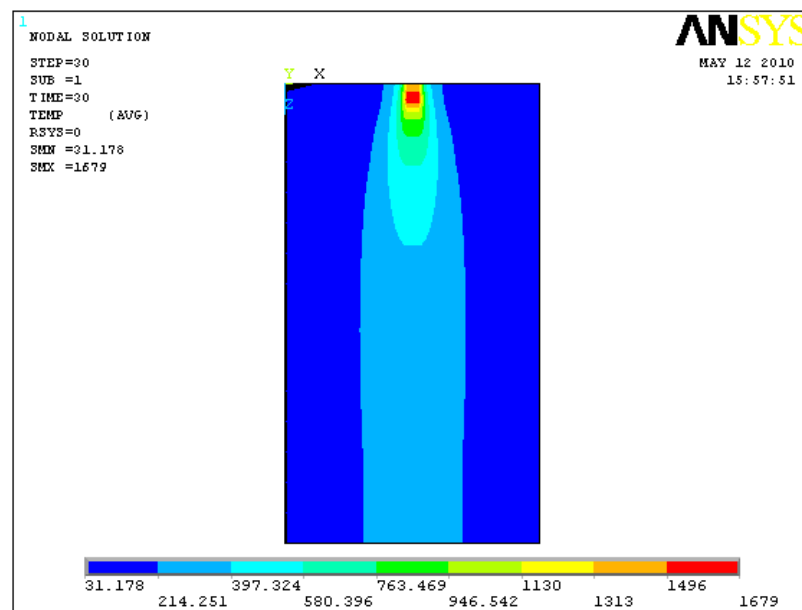

Figure 4.2: Temperature distribution at t=15s

Figure 4.3: Temperature distribution at t=30s.

21

Figure 4.2 and 4.3 shows the results for the simulation at t=15s and t=30s which is the electrode is at the halfway and at the end of the welding process for this model. There is not a big difference of the maximum temperature between t=15s and t=30s. At t=15s, the maximum temperature is 1973°C and at t=30s, the maximum temperature is 1670°C.

Mean while the minimum temperature value is increasing from t=15s to t=30s, that is from 27.041^{o}C to 31.178 ^{o}C. This increasing value of the minimum temperature is due the heat conduction in the work piece itself that transfer the heat from the hotter area to the cooler area.

For the welding speed of 5mm/s, the average of the maximum temperature is around 1670°C. This means that the work piece is that is subjected to the heat flux is melted since the melting temperature of A36 is around 1400°C. The temperature distributions at the plate along at each load step are attached in appendix.
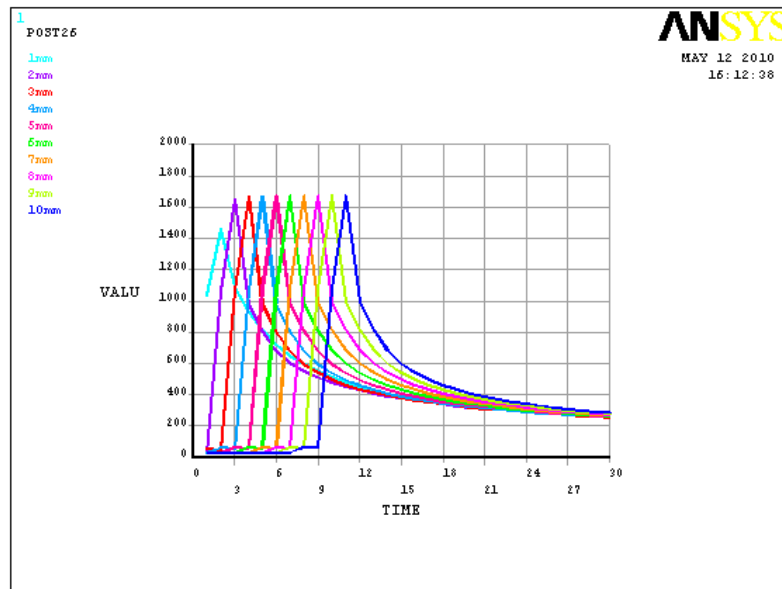

Figure 4.4: Temperature distribution at first 10 nodes along welding direction.

From figure 4.4 we can see that the work piece undergo a rapid heating and cooling while in welding process. The maximum temperature for the first node is lower than all the other nodes. After the quasi-steady state is achieved at t=2s, the maximum temperature is maintain along the welding direction. The temperature raised by the time the heat flux applied, up to $1700^{o}C$, and drop rapidly to $600^{o}C$ after 6 seconds the heat flux removed. This trend of heating and cooling is all the same at all nodes. Figure 4.5 and 4.6 below the other 20 nodes temperature distribution along the welding direction.
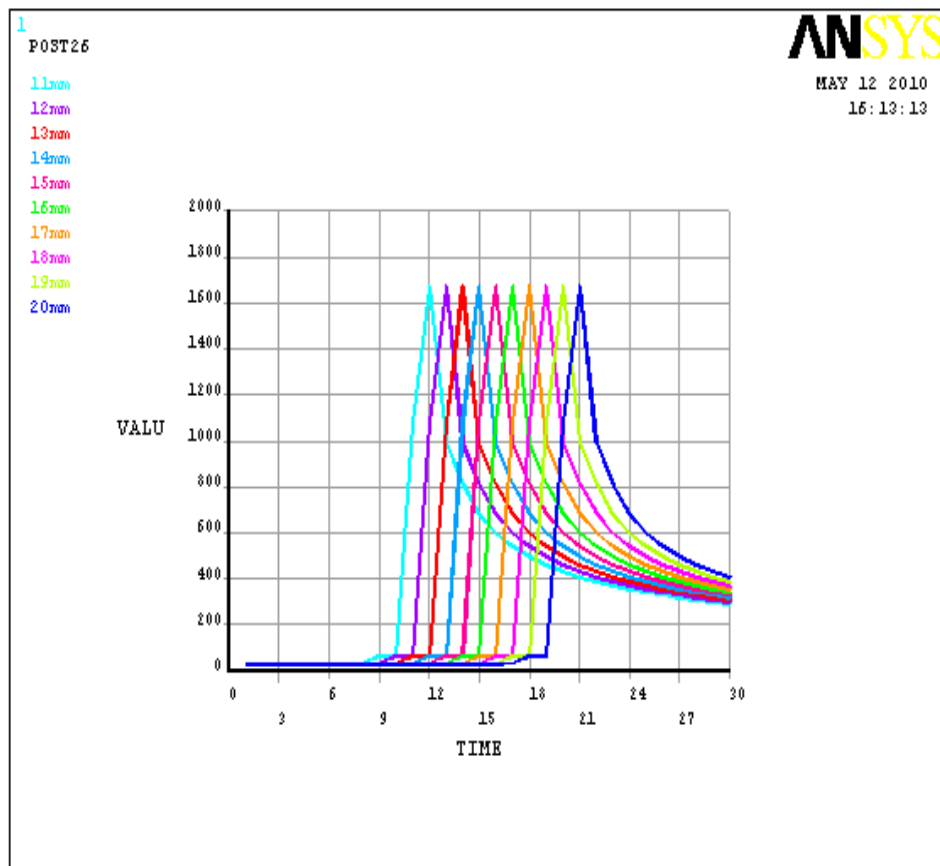
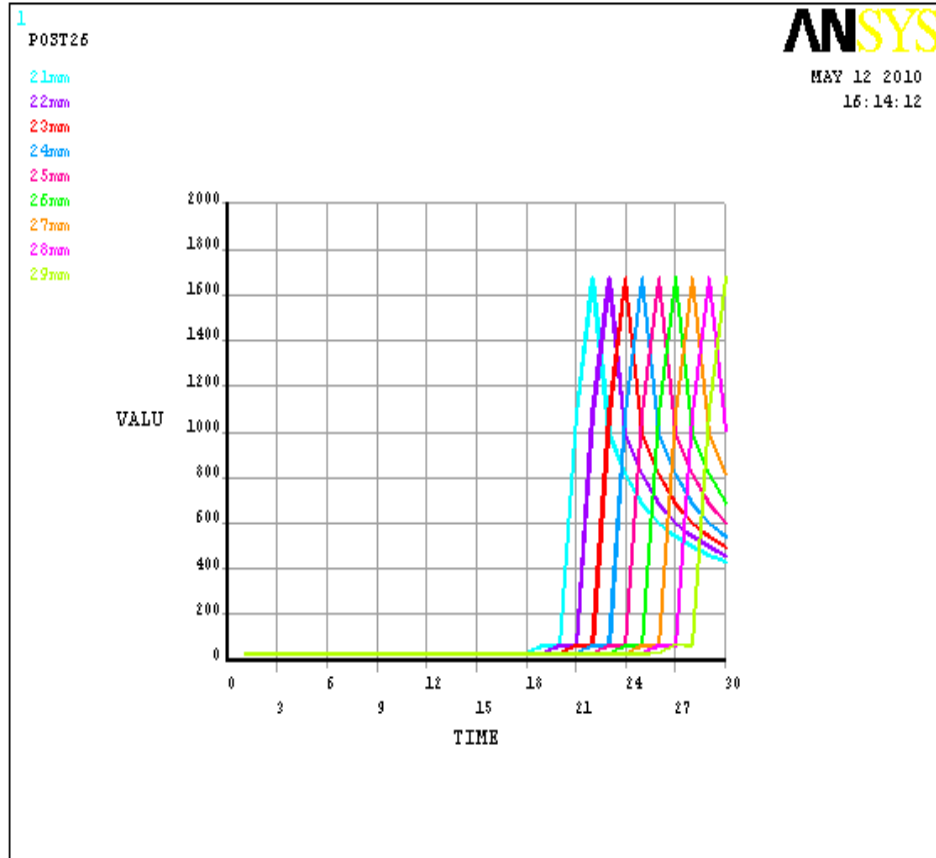**Figure 4.5: Temperature distribution for next 10 nodes along the welding direction.**

Figure 4.6: Temperature distribution for last 10 nodes along the welding direction.

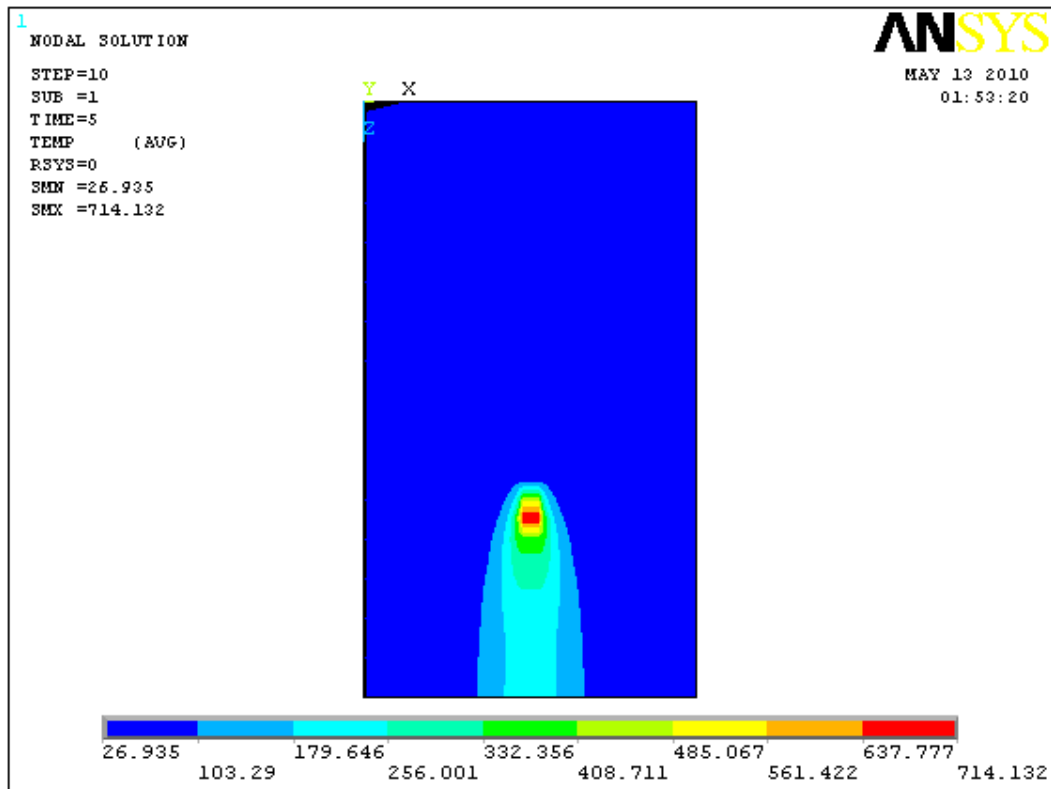## 4.2 Thermal Analysis Result for Electrode Travelling Speed, v=10mm/s

Figure 4.7: Temperature distribution of the plate at t=5s

Figure 4.7 above represent the temperature distribution on the plate at t=5s for the welding speed of 10mm/s. The maximum temperature achieved is 714.132^{o}C which is lower than maximum temperature for v=5mm/s. This is due to the heat input for v=10mm/s lower than v=5mm/s. Equation 2-3 stated that heat input is inversely proportional to the welding speed. The maximum temperature for v=10mm/s is also lower than the melting temperature of A36 carbon steel, which means that the work piece is not melted during welding and can result the welding joint between the plate is weak. So, for welding speed 10mm/s is not suitable to joint this work piece.

4.3 Comparing Thermal Result with Previous Paper Work.


```
ANSYS 6.1
JUL 10 2005
20:14:27
NODAL SOLUTION
STEP=5
SUB =1
TIME=10
TEMP        (AVG)
RSYS=0
PowerGraphics
EFACET=1
AVRES=Mat
SMN =300.004
SMX =2275
     300.004
     519.402
     738.799
     958.196
     1178
     1397
     1616
     1836
     2055
     2275
```
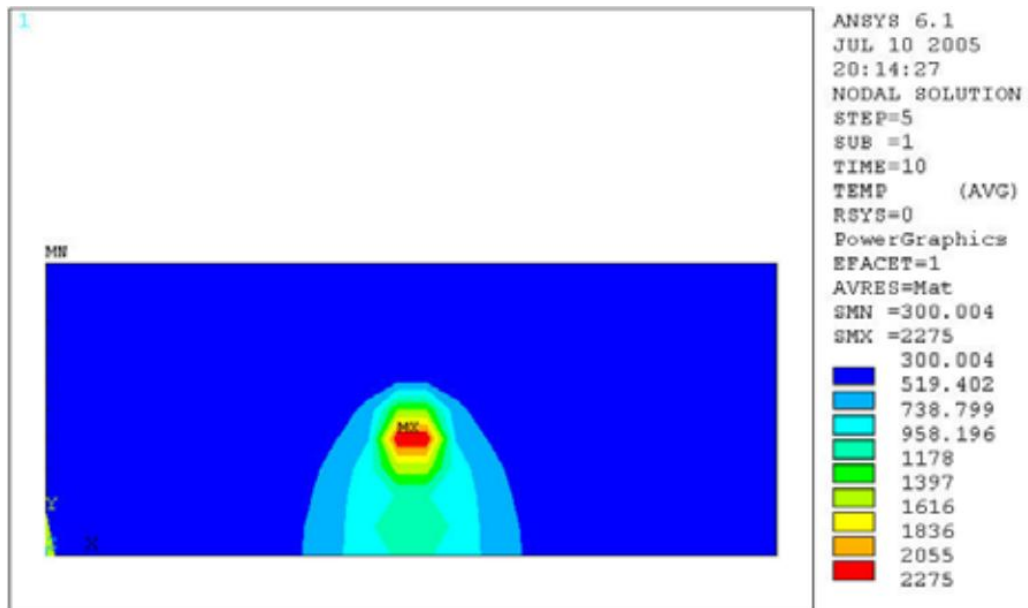
Figure 4.8: Thermal result from previous paper work at t=5s. [7]

Figure 4.8 shows temperature distribution for previous paper work at t=5s for v=5mm/s **[7]**. The maximum temperature is 2275K, which is higher due to some different in our analysis. In this paper work used a different heat input value, and also the heat flux is not constant. The meshing size is also different that lead to different result in maximum temperature. For the temperature profile, both our model is likely the same.
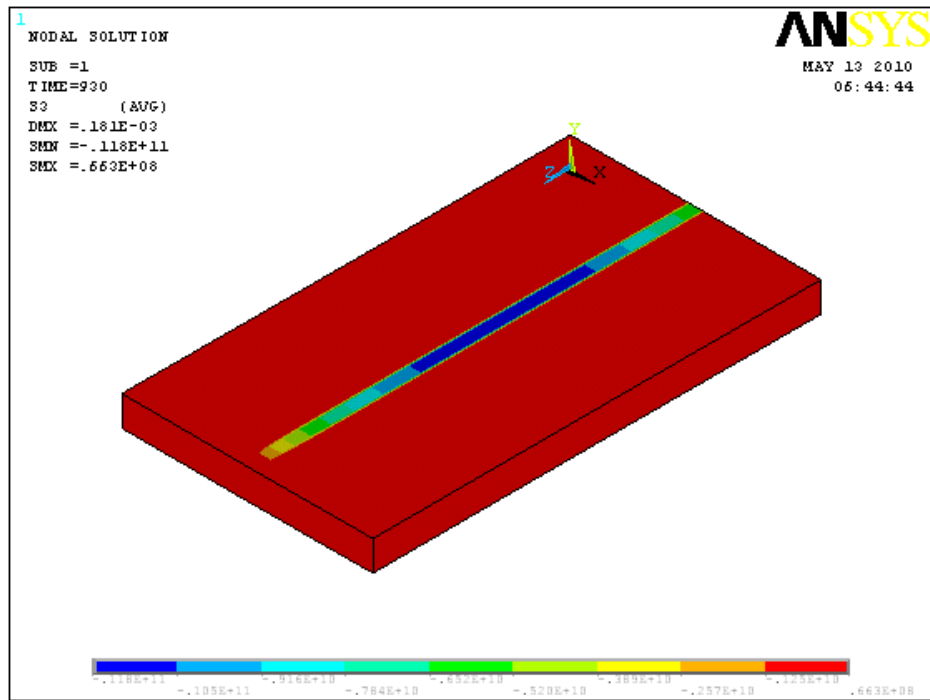
26

4.4 Structural Analysis

Figure 4.9: Stress distribution of the welded plate.

Figure 4.9 above show the stress distribution on the welded plate at t=930s. At this time, the electrode has been removed from the work piece, and the work piece is leaved for cooling for 15 minutes. Figure 4.9 above only show the stress distribution without the deformation of the plate. The highest stress is located along the welding direction with the maximum is at the centre of the weldment. The value of the compressive stress along the welding direction is between 2.5 GPa to 11.8GPa. The stress along the welding direction is high due to expansion of the material from the heat applied is constrain by the cooler volume of the work piece. The work piece that constraints the expansion of the metal undergoes a tensile stress with the value of 66.3 MPa.
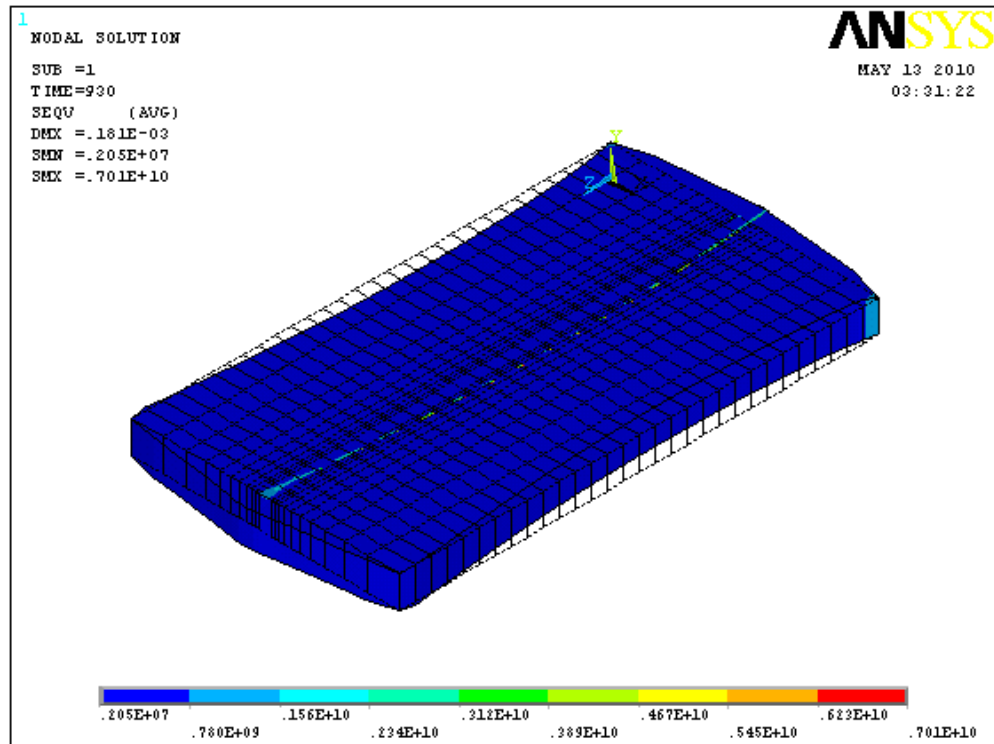
4.5 Deformation of Plate.

Figure 4.10: Deformation of the welded plate.

The plate is constraint at all four corner of the plate to represent the clamping of the work piece. Figure 4.10 shows the deformation that occur during t=930s on the welded plate. While Figure 4.11 shows the deformation of the plate from top view.
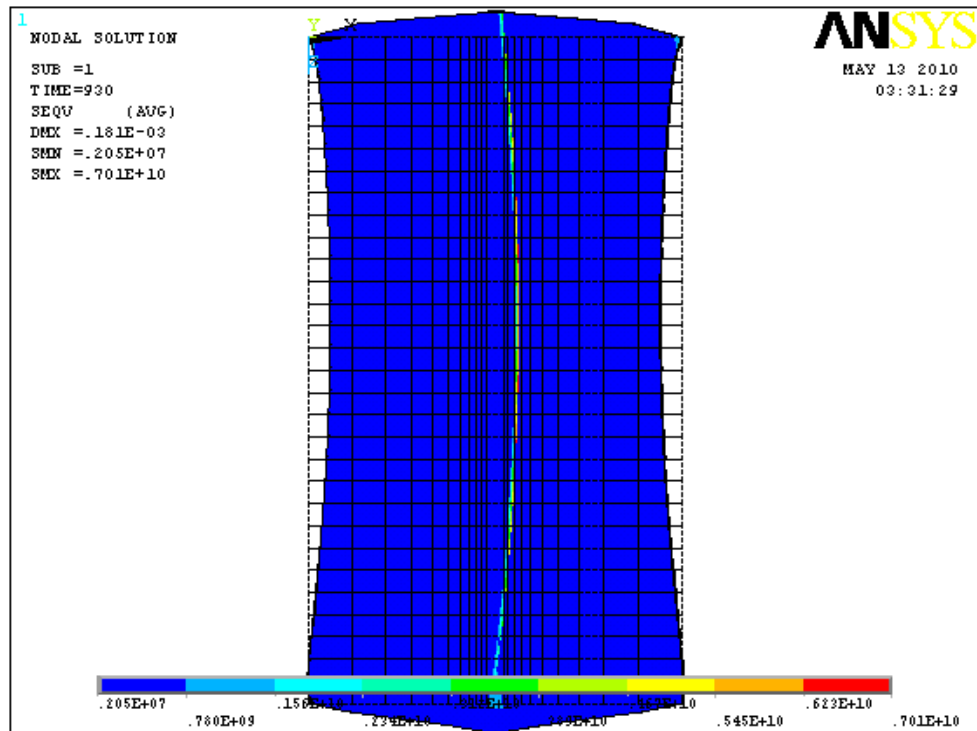
Figure 4.11: Deformation of the welded plate from top view.

The welded plate undergoes a transverse contraction at the x-direction. During welding, the heats applied expand the work piece in all direction, but due to constraint of the cooler work piece that far from the welding direction, the work piece under a non-uniform expansion. The expansion process of the work piece occurs only at the y-direction. Then, during cooling, the expended metal tends to contract to return to the initial volume, but due to the non-uniform expansion of the metal, the dimension in the x-direction shrink. There is no deformation occur on the thickness of the plate due to constant heat applied along the thickness of the work piece.
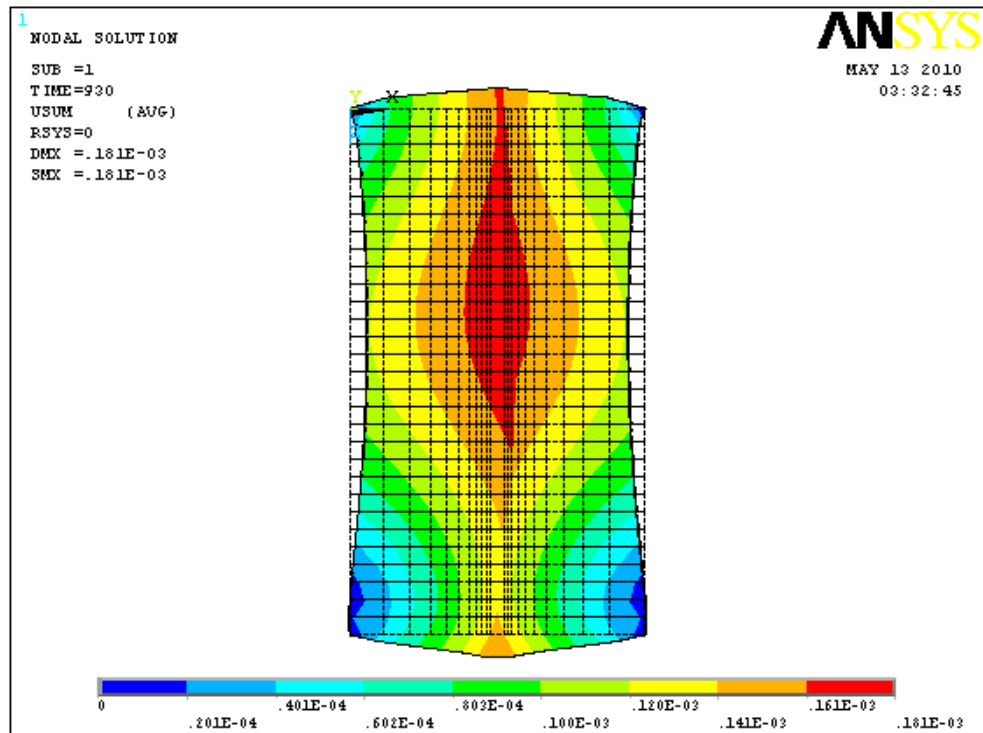
Figure 4.12: Displacement of work piece at t=930s.

Figure 4.12 shows the displacement that occur on the plate at t=930s. The maximum displacement is at the centre of work piece along the x-axis. It shrinks from total length of 84mm to 80mm. The total shrinkage in the x-direction is 0.3×10^{-3}m. The displacement in z-axis is different, where it expands from 150mm to 160mm. the displacement of the work piece is due to the heat input of the welding process.

CHAPTER 5

CONCLUSION

5.1   Conclusion

By uncoupled the thermal and structure analysis while doing the welding simulation generate more accurate results. In thermal analysis, the changes of any parameter would yield a different result in structural analysis; on the other hand, changes of parameter in structural analysis do not affect the thermal analysis result. By that, this thermal and structural analysis can be uncoupled.

In thermal analysis, the element used, Solid70, have eight nodes, and single degree of freedom which is the temperature. Electrode movement and speed in welding are simulated by restarting the loadstep, and simultaneously applied a new heat input while deleting the previous heat input. The heat input from the arc of the electrode is modelled as a heat flux in this analysis. The deposition of filler material is modelled used Element Birth & Death. All of these approaches are used in this thermal analysis, and yield a temperature distribution result which is consistent with different electrode movement speed. The welding simulation of a welding simulation is generated and met the first objective of the project which is to develop a finite element model of a welding process. The comparison of the thermal result with the previous paper work is also acceptable. The temperature distribution on the plate is analysed as part of this project's second objective.

The result from the thermal analysis is then used in the structural analysis. The result generated in the thermal analysis is saved, and start a new analysis (uncoupled analysis). Structural analysis used Solid45 as element type that having eight nodes and three degree of freedom which is the translation at x, y, and z axis. The temperature distribution history is used as a load in structural analysis. The result generated is the stress distribution in the plate. The third objective is achieved by generating the displacement of the welding model.

5.2 Recommendation

In order to get more accurate result, the heat flux can be treated as non-constant input, which is different from this project. The actual heat generated from the electrode is not constant. The heat input can be modelled by using Rosenthal equation. This equation can be inserted in the program, and the program will run the heat input as per equation inserted.

Next, the filler material can be treated to not having the same material with the work piece. In real life welding, the material properties of the electrode are different from the material of the work piece. Usually, the melting temperature of the electrode material is lower than the work piece. By this, we can yield more accurate results in the thermal and structural analysis.

Lastly, use an ANSYS command to generate the welding simulation, where we write every step that we want to apply in the analysis in the **.txt** file. ANSYSY will read the command, and generated the model as per command written in the file. By this, we can reduce the time used in making the welding simulation. This ANSYS command method also suitable for a longer welding time analysis.

CHAPTER 6

REFERENCES

[1] JF Lancaster, 1965, *The Metallurgy of Welding, Brazing and Soldering, 2nd edition* (George Allen and Unwin Ltd)

[2] N R Mandal, 2004, *Welding and Distortion Control, 1st edition* (Alpha Science International Ltd.)

[3] Binay K. Dutta, 2006, *Heat Transfer Principles and Application, Sixth Edition* (Prentice Hall of India Ltd)

[4] A. Capriccioli, P. Frosi, 2009, *Multipurpose ANSYS FE Procedure for Welding Process Simulation*, Fusion Engineering Design, Vol. 84, pg 546-553.

[5] P.H. Chang, T.L. Teng, 2004, *Numerical and Experimental Investigations on the Residual Stress of the Butt-Welded Joints*, Computational Material, Vol. 29, pg 511-522.

[6] A. Carpinteri, C Majorana, 1995, *Fully Three-Dimensional Thermomechanical Analysis of Steel Welding Process*, Journal of Material Processing Technology, Vol.53, pg 85-92.

[7] M. Sundar, G. Nandi, A. Bandyopadhyay, S.C. Roy, 2005, *Assessment of Residual Stress and Distortion by Finite Element Metho*d, Proceeding of International Conference on Mechanical Engineering 2005.

[8] F. V. Tahami, A.H.D. Sorkhabi, M.A. Saeimi-S, A. Homayounfar, 2008, *3D Finite Element Analysis of the Residual Stresses in Butt-Welded Plates with Modelling of Electrode-Movement*, Journal of Zhejiang University, Vol. 10, pg 37-43.

[9] L.E Lindgren, H.A. Haggblad, J.M.J McDill, A.S. Oddy, 1997, *Automatic Remeshing for Three-Dimensional Finite Element Simulation of Welding*, Computer Method in Applied Mechanics and Engneering, Vol. 147, pg 401-409

[10] D. Stamenkovic, I. Vasovic, 2009, *Finite Element Analysis of Residual Stress in Butt Welding Two Similar Plates*, Scientific Technical Review, Vol. LIX, No. 1, pg 57-60.

[11] S. Qingyu, L. Anli, Z. Haiyan, W. Aiping, 2002, *Development and Application of the Adaptive Mesh Technique in the Three-Dimensional Numerical Simulation of the Welding Process,* Journal of Materials Processing Technology, Vol. 121, pg 167-172.

APPENDICES

- **Thermal Result**
- **ANSYS Command List**