THERMO MECHANICAL SIMULATION OF CORNER JOINT WELDED PLATE

by

Mohamad Hafiz Bin Mohd Hasan Abdullah

(9264)

Dissertation (FYP Part II) submitted in partial fulfillment of the requirements for the:

Bachelor of Engineering (Hons)

(Mechanical Engineering)

December 2010

Universiti Teknologi PETRONAS

Bandar Seri Iskandar

31750 Tronoh

Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Thermal Mechanical Simulation of Corner Joint Configuration

By

Mohamad Hafiz Bin Mohd Hasan Abdullah

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. Mokhtar Awang)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

December, 2010

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.

____

(MOHAMAD HAFIZ BIN MOHD HASAN ABDULLAH)

ABSTRACT

Welding is widely used in manufacturing industries to assemble various products. It is well known that welding process relies on an intensely localized heat input, which tends to generate undesired **residual stresses** and deformations in welded structures. Residual stresses are stresses that remain after the original cause of the stresses such as heat gradient during welding process. They remain along a cross section of the component even without external cause. This study has been conducted with objectives to develop finite element model of welding and compute temperature distribution and deformation of welded plate. In this study, finite element analysis (FEA) based on numerical method will be applied. FEA is used to study the thermal effect of heat input on residual stresses of corner joint plate and predict the stress field and its gradient around the fusion zone of welded plates. ANSYS simulation software will be used by implementing birth and death element for the analysis of welding problems using FEA. The prediction of welding residual stresses using finite element method can be simplified by uncoupling the thermal and mechanical aspects of the problem. Thermal and mechanical analyses will be solved sequentially. Result shows that, maximum temperature on the plate is 1768^{o}C which beyond melting temperature of carbon steel, 1500^{o}C. Residual stress is significantly high with average of 103MPa. These results are with an agreement with published data from Dragi Stamenkovíc in journal of “Finite Element Analysis of Residual Stress in Butt Welding Two Similar Plates”.

ACKNOWLEDGENT

I would like to express my sincere gratitude to my project supervisor Dr. Mokhtar Awang for his technical and moral support which made this final year project report possible. 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.

Lastly, 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.

TABLE OF CONTENT

LIST OF TABLE **PAGE**

LIST OF APPENDICES **PAGE**

LIST OF FIGURE **PAGE**

CHAPTER 1
INTRODUCTION

1.1 Project Background

Welding is widely used in manufacturing industries to assemble various products. In spite of the vast application of welding joints in the industry, still they have been considered as a weak point in the mechanical engineering design. This is due to the shortcomings of welding technology and mismatch of the mechanical properties at the joints. Besides, it is well known that, welding process relies on an intensely localized heat input, which tends to generate undesired residual stresses and deformations in welded structures.

Therefore, estimating the magnitude of welding deformations and characterizing the effects of the welding conditions are deemed necessary. Many techniques have been used for measuring residual stresses in metal including stress relaxation techniques, diffraction techniques, cracking techniques and techniques by use of stress sensitive properties. However, the application of these techniques in practice is usually expensive, destructive and limited by accuracy. Therefore, a general trend to study these stresses is by using finite element analysis (FEA).

The FEA method is the conventional means of calculating residual stresses. With modern computing facilities, FEA technique has become an effective method for prediction and assessment of welding residual stresses and distortions. However, the welding deformations are various with production variations such as dimension, welding materials and welding process parameters. Therefore, rapidly and accurately predicting welding induced distortion for real engineering applications is more challenging.

1.2 Problem Statement

Residual stresses are stresses that remain after the original cause of the stresses has been removed such as heat gradient during welding process. They remain along a cross section of the component even without external cause. Residual stresses occur for a variety of reasons, including inelastic deformation and heat treatment. Heat from welding may cause localized expansion, which is taken up during welding.

Since the heat generated during a welding process is dissipated through convection, conduction and radiation, a severe temperature gradient would exist around the welding point. This gradient, together with the rapid quenching and the phase transformation of the melted filler will cause the residual stresses along the joint.

These stresses not only cause unwanted deformation, but also reduce the fatigue and creep lifetime of the weld plate. Because of the inherent complexities of the welding process, factors including process related and geometry dependent affect the final residual stresses.

Fatigue behavior of welded plate is complicated due to many factors such as stress concentration, environment and residual stresses. Residual stresses that arise in welded plate by heating and cooling cycles during welding process are an important factor in fatigue failure. Besides, tensile residual stresses in welded plate can be as high as the yield stress and can have detrimental effect on the fatigue life of welded structures.

1.3 Objectives

This project has been conducted with objectives to:

a) Develop a finite element model of a welding process

b) Study the temperature and residual stresses profile during welding process.

1.4 Scope of Study

This study will concentrate on simple welding configuration which is corner joint. Corner joint is joining edges of two plates perpendicular to each other. Figure 1 shows a schematic representation of corner joint welding configuration. On the other hand, type of welding will be applied in this study is Shielded Metal Arc Welding (SMAW) because of the versatility of the process and the simplicity of its equipment and operation. Carbon steel is used as both, workpiece and filler metal. Table 1 shows their temperature dependent properties. In order to get temperature distribution and deformation, ANSYS 9 will be used with utilizing birth and death element.

With ANSYS, the study will be focusing to determine:

a) Temperature distribution along the welded plate

b) Stress distribution along welded plate

c) Deformation on the plate after heat has been remove

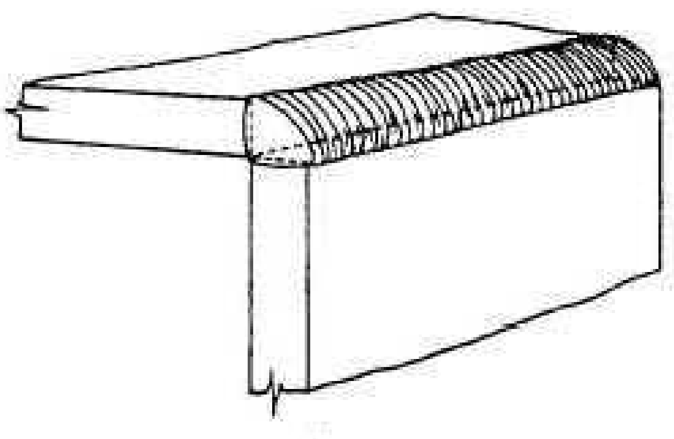


Figure 1: Corner joint configuration

CHAPTER 2
LITERATURE REVIEW

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. 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 as reported by *JF Lancaster* [1].

The heat input supplied by the welding arc produce complex thermal cycles in the weldment and these, in turn cause changes in the microstructure of the heat-affected zone. The high heat concentration is necessary because metallic materials diffuse the heat and cause transient, inhomogeneous thermal stress and metal movement. The transient temperature field causes thermal expansion, stress and strain that usually plastically deforms in the weld neighborhood and result in residual stresses and strain that remains when the weldment cools and the structure is distorted from its original shape based on *Z. Barsoum* [2].

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 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.

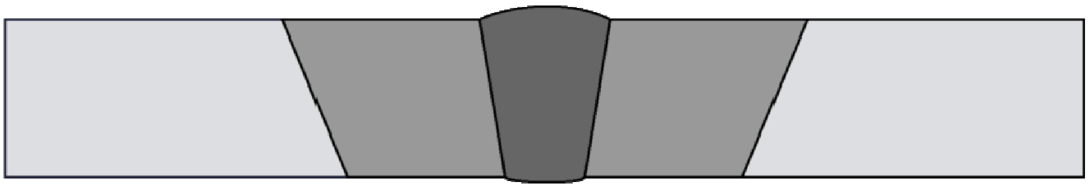


Figure 2: Cross section area of a welded joint

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 where once steady condition has been achieved, the temperature distribution relative to the heat source is stationary based on *N.R Mandal* [19]. 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 as moving.

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 six main types which are longitudinal shrinkage, transverse shrinkage, angular distortion, bowing and dishing, buckling, and twisting.

During the rapid heating cycle of a fusion welding process, material in the vicinity of the weld heats, expands in all directions and is compressed by the constraints of the much larger and cooler surrounding structure. The heated volume has a lower yield point than the cooler surrounding structure and is more readily upset to a smaller dimension, the heated volume yields in compression.

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 shown in Figure 3. The level of welding distortions depends on many variables, such as heat input, material thickness, electrode speed, material type and etc.

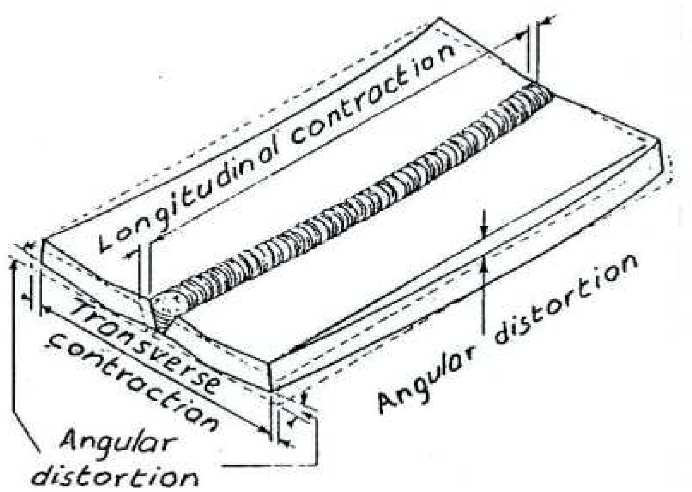


Figure 3: Welded plate distortion

Based on *Michaleris* [20], 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.

Residual stresses have a significant influence on the fatigue strength of welded structures, and it is well known that high tensile residual stresses have a detrimental effect on fatigue life and compressive residual stresses could have a favorable effect on fatigue life. The combination of welding residual stresses with operating stresses to which engineering structures and components are subjected can promote failure by fatigue as reported by *Z. Barsoum* [3].

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. Stress distribution for a complex welded structure is usually not known, and conservative assumptions are made of the residual stresses distribution when fatigue life predictions are assessed based on *Martinsson* and *Finch D.* [4, 5].

Several experimental destructive and non-destructive techniques for directly measuring residual stresses have been developed. These techniques include X-ray diffraction method, neutron diffraction method, layer-removal method, sectioning method, ultrasonic and magnetic methods, and hole drilling methods. However, the application of these methods in practice is usually limited by either cost or accuracy as reported by *E. Armentani et al* [6].

Numerical simulation based on finite element methods of residual stresses calculation in corner joints have been developed years ago and most of these studies were conducted based on either axisymmetric or two-dimensional plane assumption based on *Ueda Y, Rybicki E.F* and *Wilkening W.* [7, 8, 9]. In the other hand, it is also demonstrated that the residual stresses in welded joints are not always symmetric, *Karlsson* and *Dong P.* [10, 11]. High tensile and compressive residual stresses concentrations have been observed at welding start/end point based on *Shack* and *Josefson* [12, 13].

Different parameters determine the amount of the residual stresses and its distribution pattern in welded joints. Those parameters describe by *Leggatt R.H* [14] are:

- The geometry of the parts being jointed.
- The material properties of the weld and parent materials, including composition, microstructure, thermal properties and mechanical properties.
- Residual stresses which exist in the parts before welding

Fatigue failures in welded structures usually initiate at the stress concentration point based on *Xiangyang et al.* [15]. Hence, a two-dimensional simplified model cannot always simulate the fatigue failure of welded joints accurately. Therefore, it is essential to conduct a detailed three-dimensional analysis in order to simulate the fatigue responses of welded joints.

Accurate and reliable residual stresses predictions are essential for structural integrity and fatigue assessment of components containing residual stresses. However, finite element simulation of residual stresses is time consuming due to welding involves in many phenomena such as non-linear temperature dependent material behavior, three-dimensional nature of the weld pool and the welding processes and microstructural phase transformation. Despite the simplification by excluding various effects, welding simulation are still time consuming and complex. Therefore, simplified welding simulation procedures are required in order to reduce the complexity and thus maintain the accuracy of residual stresses prediction. Figure 4 shows the simulation scheme and coupling fields in welding analysis proposed by *Z. Barsoum* [2].

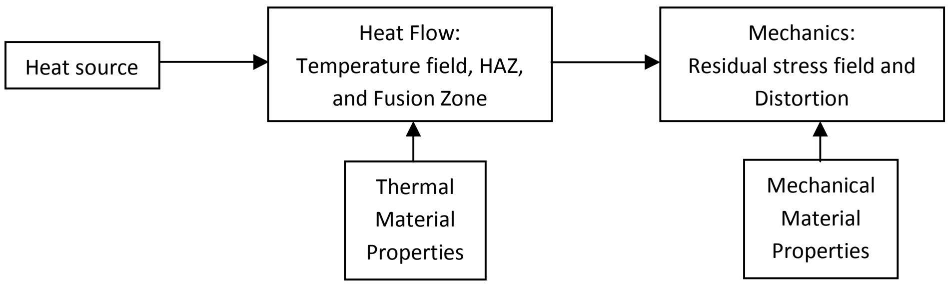


Figure 4: Simulation scheme in welding analysis

The numerical method for predicting residual stresses of welded joint consists of two parts, heat transfer and residual stresses analyses as reported by *Xiangyang et al.* [15]. The prediction of welding residual stresses using finite element method can be simplified by uncoupling the thermal and mechanical aspects of the problem. Figure 5 shows the individual process of thermal and mechanical model. To simulate the moving heat source, it is necessary to model the heat source during each time increment. Based on *Stamenkovic et al.* [16], the moving heat source is simplified by assuming that the welding arc stayed at an element with constant specific volume heat flux and then, moved to the next element at the end of the load step as the welding was finished.

Based on *Martinsson* [4], a parametric model is adopted in single-pass corner joint in order to simulate the weld filler variation with time. The effect of thermal properties and weld efficiency on transient temperatures during welding was studied and the residual stresses after welding were determined by the finite element method. A fully coupled thermal-mechanical two-dimensional analysis was performed with the commercial software program ANSYS 9 and heat flow was evaluated by a non-linear transient analysis.

In this study, the technique of element "birth and death" will be adopted. This technique can be used to simulate the process of filler metal addition. All elements must be created, including those weld fillers to be born in later stages of the analysis. To achieve the "element death" effect the elements are not actually removed. These are deactivated by multiplying their stiffness by a severe reduction factor. Likewise, when elements are "born", they are not actually added to the model, but simply reactivated. When an element is reactivated, its stiffness, mass, element loads and other properties return to their full original values.

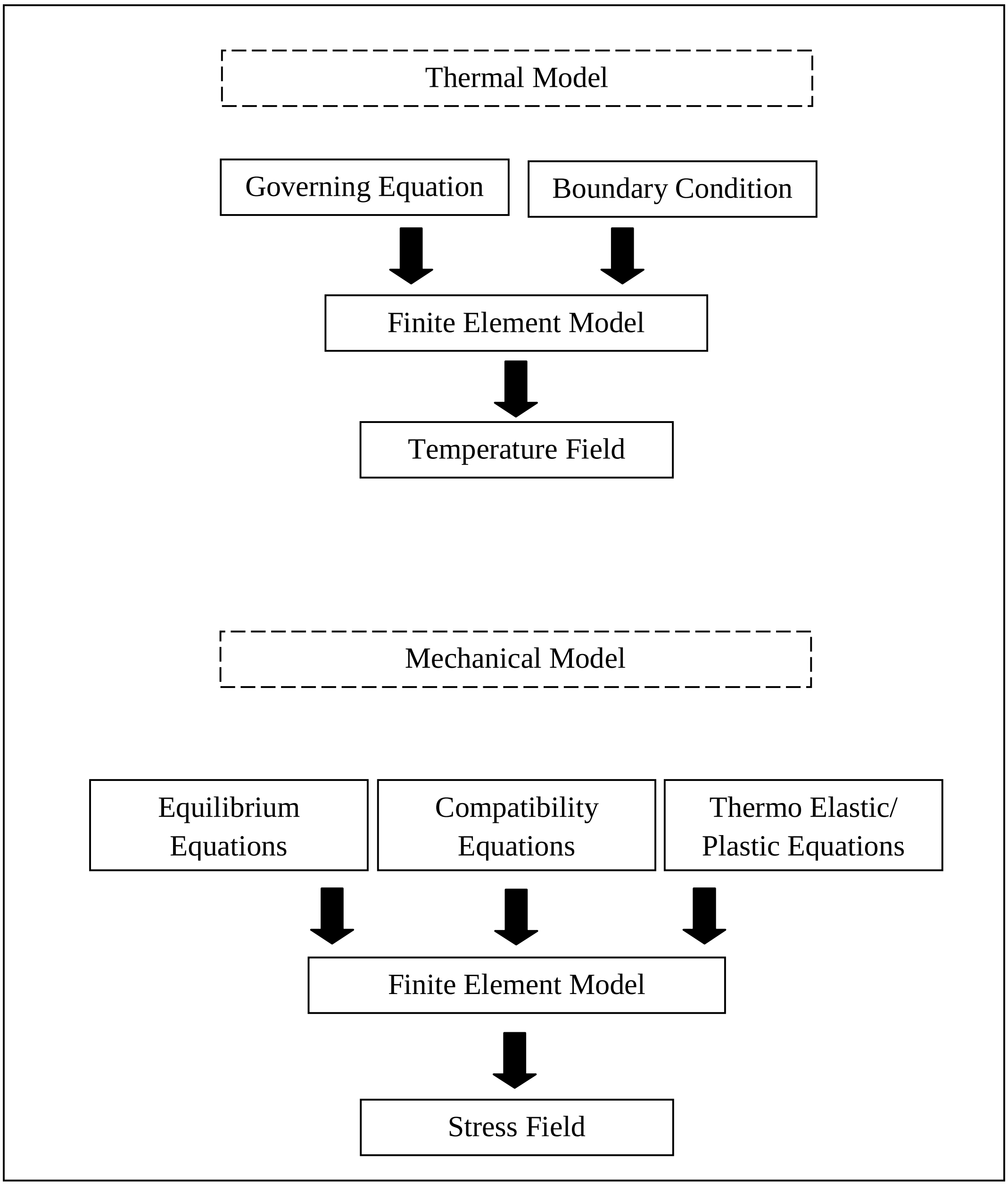


Figure 5: Thermal and Mechanical Model

CHAPTER 3
METHODOLOGY

3.1 Research and Data Gathering

To carry out this project, it is important to fully understand about residual stresses and technique that plan to be applied which is FEA. Therefore, reviewed on books, journals, papers and other technical documents related with FEA residual stresses analysis have been thoroughly done. This stage of research and data gathering beneficially by giving insight and basic knowledge about project conducted.

Besides, those documents are very good reference on methods and techniques that had been develop before such as X-ray diffraction method, neutron diffraction method and numerical method which is FEA. Results of the analysis provided by those documents are compared to determine the best technique or method in analyzing residual stresses on corner joint. Based on the result, it has been proved that FEA is best method to analyze residual stresses. Gantt chart on project planning and milestone are shown in Appendix A and Appendix B respectively.

3.2 Material Selection

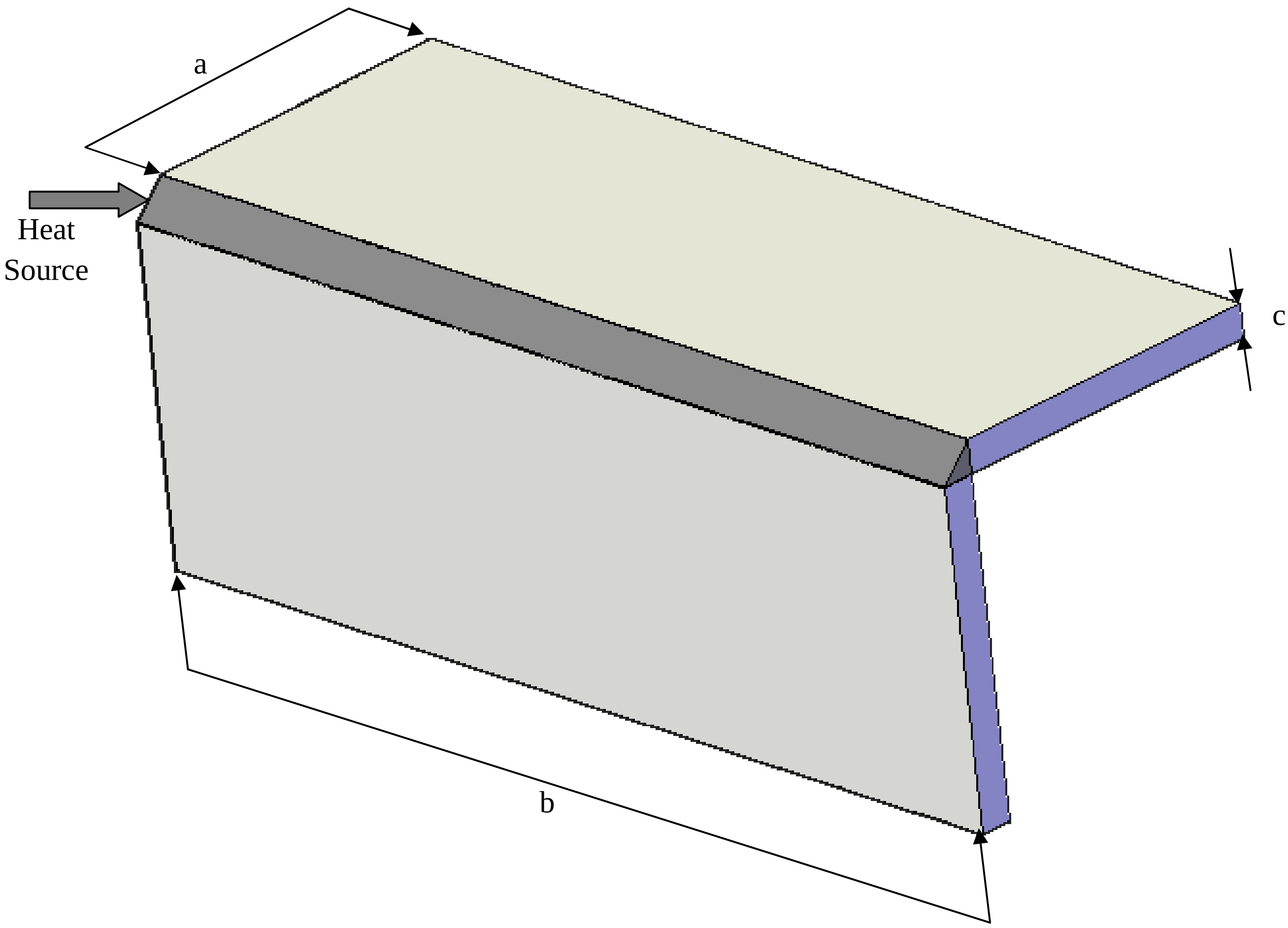
Different material has different properties. In this study, carbon steel (ASTM 36) will be used as the workpiece and filler metal. Their temperature dependent properties of carbon steel are shown in Table 1.

Table 1: Material Properties of Carbon Steel (ASTM 36)

Temperature (°C)	Density (kg/m^3)	Young's Modulus (GPa)	Poisson's Ratio	Thermal Expansion Coefficient ($10^{-5}/°C$)	Conductivity (W/m °C)	Specific Heat (J/kg °C)
100	7880	200	0.3	1.20	50	500
200	7800	200	0.3	1.30	45	520
400	7760	170	0.3	1.42	38	650
600	7660	80	0.3	1.45	30	750
800	7520	35	0.3	1.45	25	1000
1000	7390	20	0.3	1.45	26	1200
1200	7300	15	0.3	1.45	28	1400
1400	7250	10	0.3	1.45	37	1600
1550	7180	10	0.3	1.45	37	1700

3.3 Geometry Modeling

Welding plate and filler for this study is model with combination of three volumes consist of two symmetric workpiece plate and one filler. The geometry structure of the plate and weld filler is shown in Figure 6.



a = 50mm, b = 150mm, c = 4mm

Figure 6: Geometry modeling

3.4 Welding parameter

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. Equation below used to model the heat input during welding by the equivalent heat input which includes body heat flux as follow:

$$Q = \eta \frac{UI}{V\mathrm{m}}$$

Where η is the efficiency, V is the travel speed of electrode, U and I are the arc voltage and current while m is electrode diameter. For SMAW, η, V, U and I parameter commonly used are 0.85, 5mm/s, 24V and 90A respectively while electrode diameter used is $4\sqrt{2}mm$. Therefore, heat input will be:

$$Q = \eta \frac{UI}{V\mathrm{m}} = 0.85 \frac{24V \times 90A}{5mm/s \times 4\sqrt{2}mm} = 65W/mm^2 = 65e6W/m^2$$

3.5 Assumptions

Welding process is a very complex process of thermal cycles. Therefore, in order to conduct these analyses, several assumptions have been made to simplified welding simulation procedures by reducing the complexity with acceptable accuracy. Assumptions taken are as followed:

- Heat loss due to convection, conduction and radiation by arc efficiency coefficient η_{arc} in calculation of heat input. In this study, η_{arc} taken as 0.85 as reported in [15]
- Filler is a triangular volume
- Moving heat source at constant speed, 5mm/s as reported by Xiangyang [15]
- Displacement of plate during welding do not effect temperature distribution
- Apply temperature dependent material properties based on [15, 17]
- Metal used for filler is carbon steel ASTM36
- Electrode diameter is $4\sqrt{2}mm$
- Not taking into consideration of plastic deformation

3.3 Boundary Conditions

To simulate this project, a few boundary conditions have been considered which are as followed:

- Initial temperature of plate and electrode is room temperature, $27^{o}C$
- Estimated convection heat transfer coefficient, *h* for carbon steel is $15 w/(m^2.K)$
- Four selected node will be restricted to deformation due to clamping during welding.

3.7 Geometry Meshing

Meshing basically is a preprocessing step to breaking up physical domain or 3D model into simpler subdomains or so called elements. This study wills utilize hexahedra shape of meshing with bias along filler joint. The meshed models are shown Figure 10.

3.8 Thermal and Mechanical Model Simulation

This study will apply transient analysis where temperature distribution *T* of welded plate is a function of time *t* and coordinate *x*, *y*, *z* and the balance relation of heat flow of a volume bounded by an arbitrary surface *S* is given by following equation:

$$\rho C \frac{\partial T}{\partial t} = \frac{\partial}{\partial x}\left(\kappa_x \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y}\left(\kappa_y \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z}\left(\kappa_z \frac{\partial T}{\partial z} \right) + \dot{q}$$

where ρ, C, and κ_i are the density, specific heat capacity and thermal conductivity respectively in *i* direction of the plate material and they depending on the temperature, whereas $\dot{q}$ is the rate of internal heat generation.

Thermal and mechanical analyses will be done sequentially. First, the temperature and phase evolution are determined as a function of time in the thermal analysis. Element type of SOLID70 will be applied during thermal analysis of stainless steel plate. SOLID70 element type has capability of 3-D thermal conduction with eight nodes and one single degree of freedom, in this case temperature. This element is applicable to a 3-D steady state or transient thermal analysis. The specific heats are evaluated at each integration point to allow for abrupt changes. Figure 7 shows the thermal analysis procedure.

After finished with thermal analysis and obtain result for temperature distribution, the study proceeds with structure analysis. Element type of SOLID45 with eight nodes and three degree of freedom is applied. This element type has capabilities to analyze plasticity, creep, swelling, stress, stiffening, large deflection and large strain. Figure 8 shows the thermal analysis procedure.

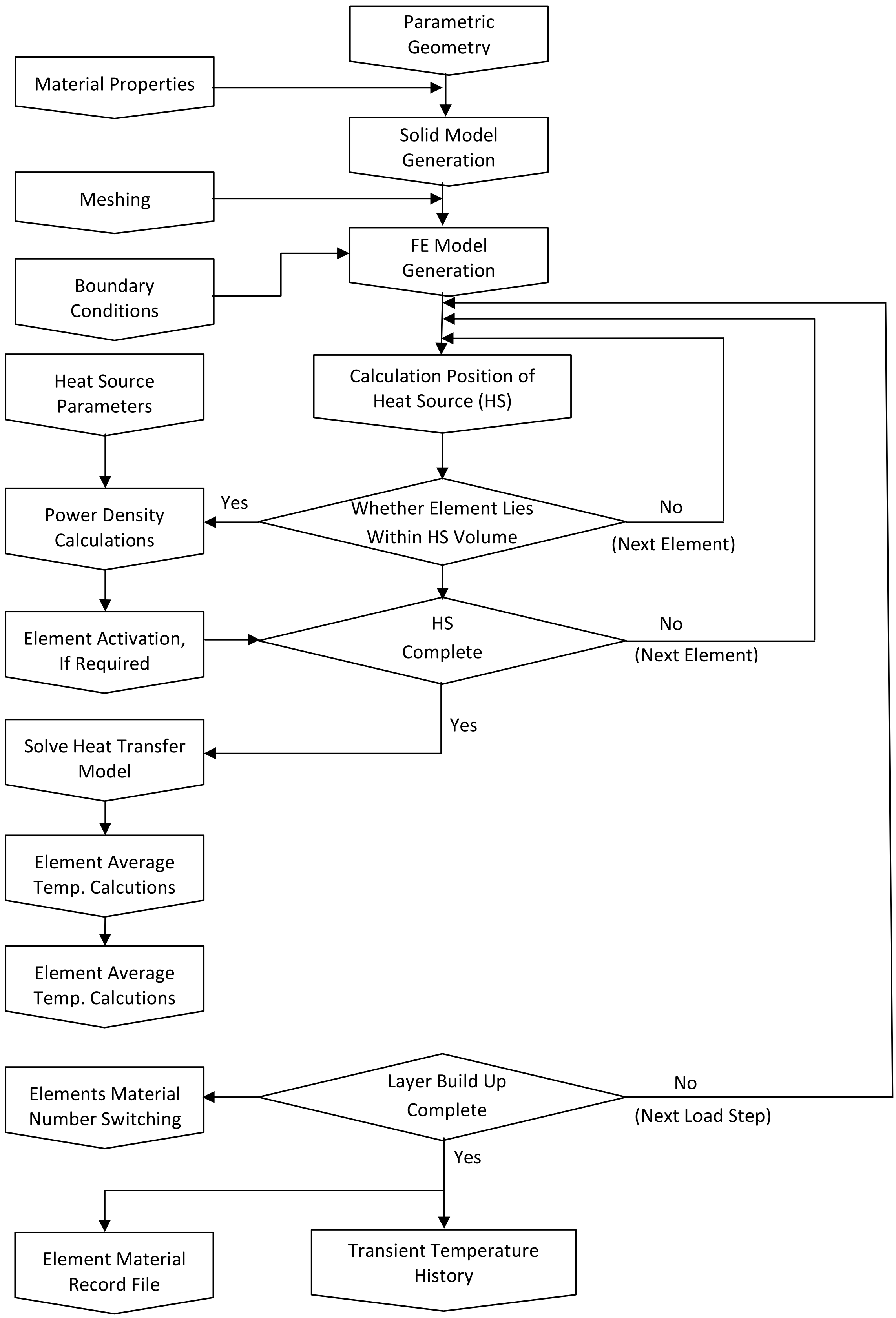


Figure 7: Procedure for Thermal Analysis

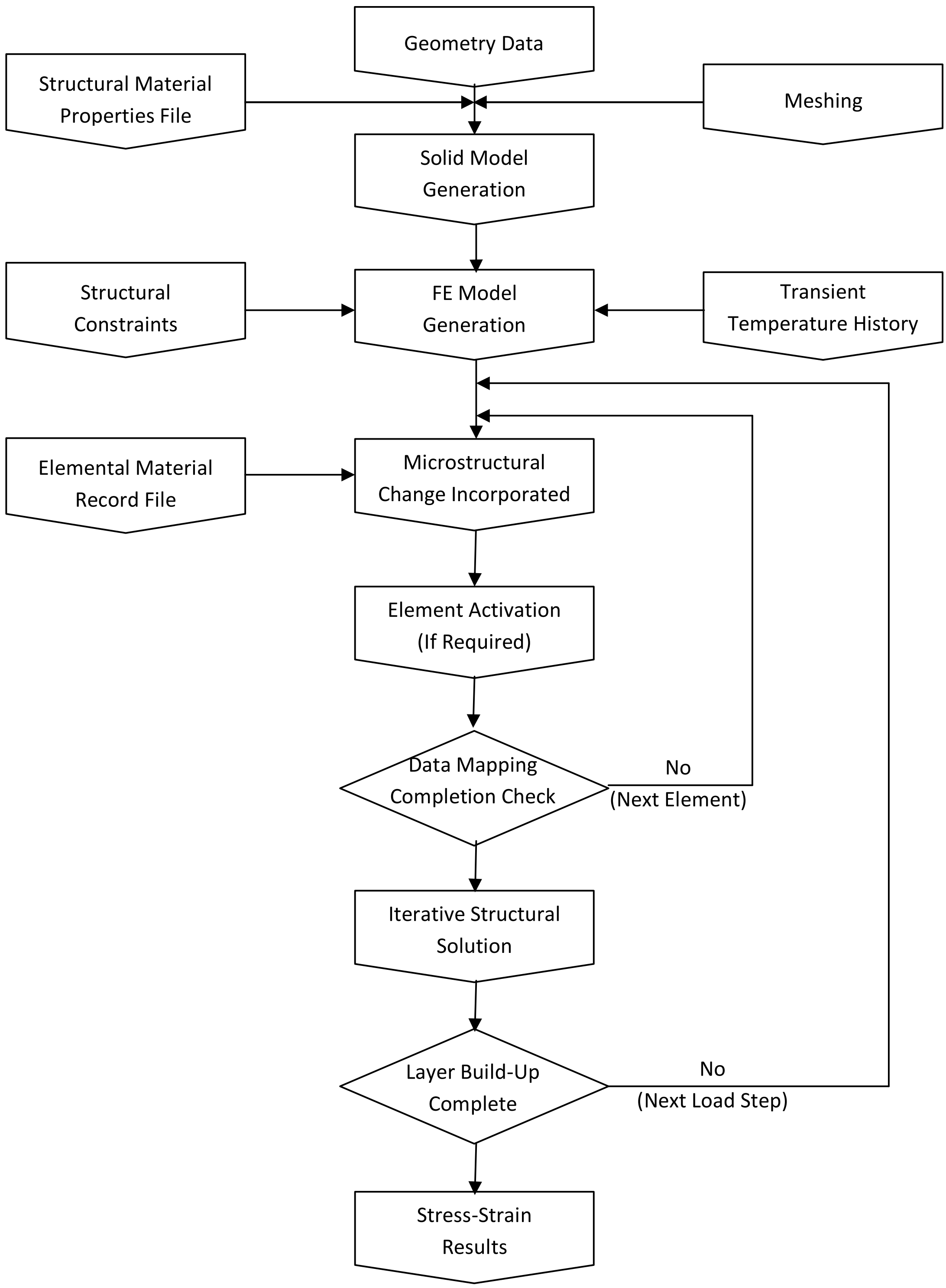


Figure 8: Flow diagram of Structural Analysis

CHAPTER 4
RESULT AND DISSCUSION

4.1 Geometry Modeling

Two plates of the joint are 4mm thick, 150mm long (along the welding direction) and 50mm width has been modeled using ANSYS. The element type SOLID70, which has a single degree of freedom, was used for thermal analysis. For the structural analysis, the element type SOLID45 with three translational degrees of freedom at each node was used. Figure 9 shows the geometry model used in the analysis.

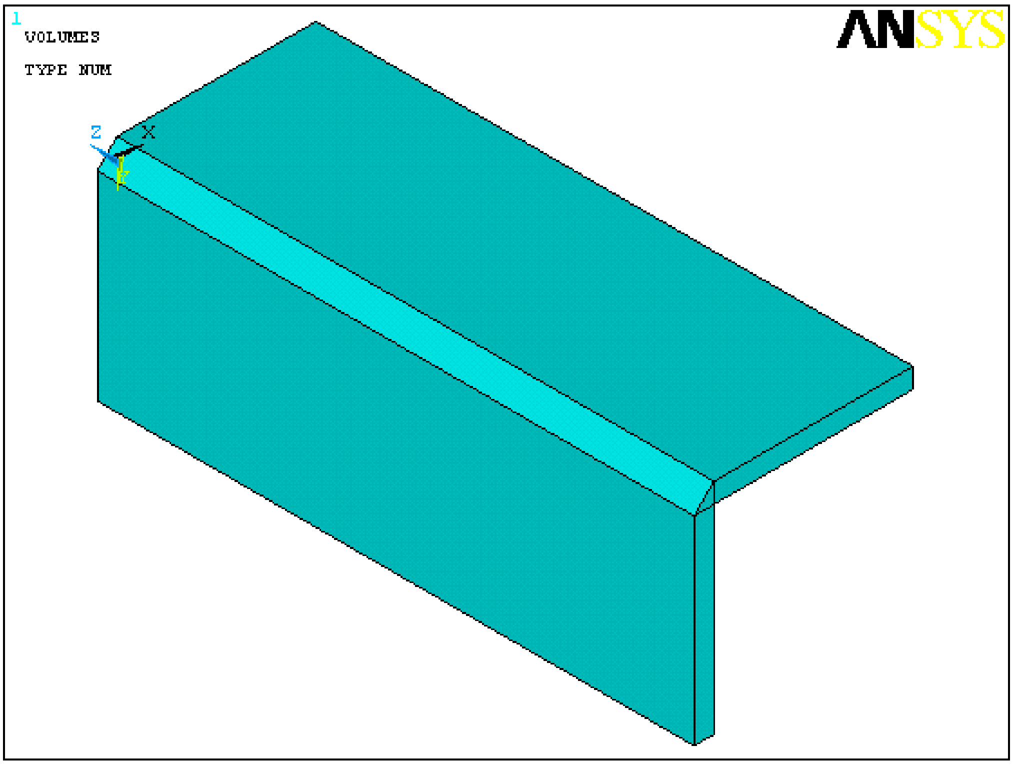


Figure 9: Geometry model in ANSYS

4.2 Meshed Model

To increase the accuracy of analysis, bias mashed has been applied along the filler-workpiece contact because deformation and stress are critical at that area. Meshed model is shown in Figure 10 below.

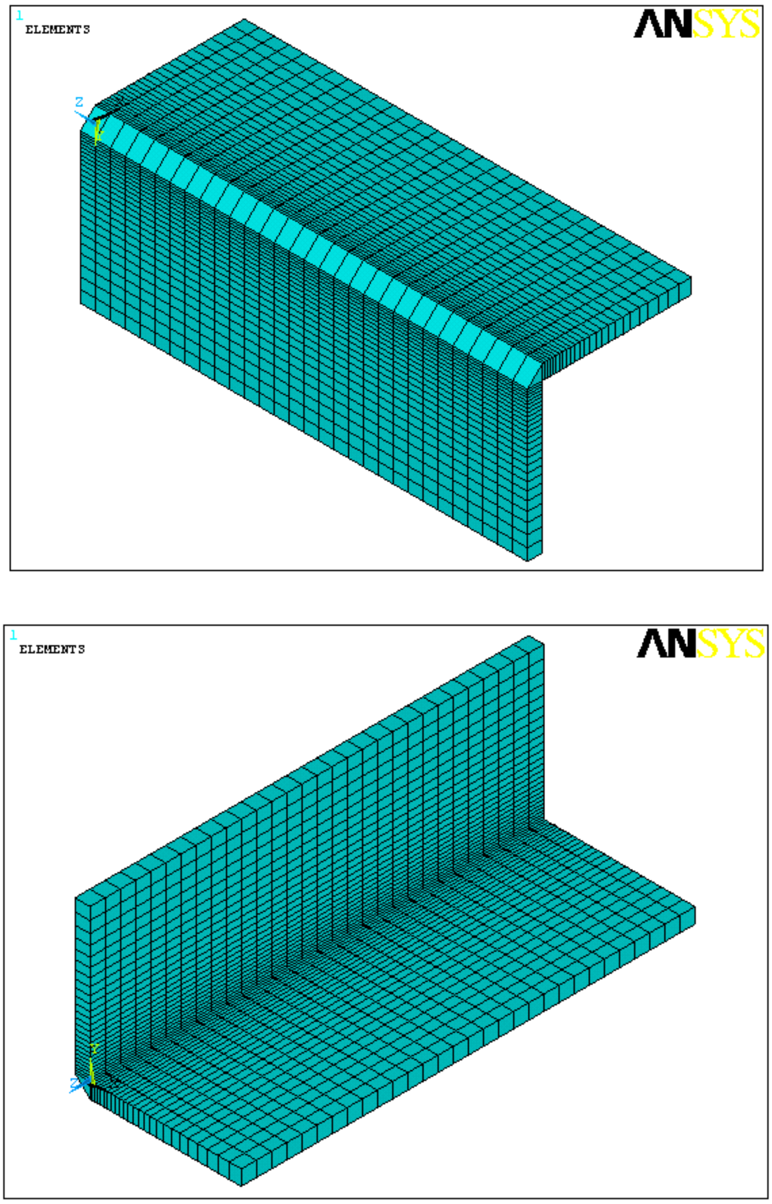


Figure 10: Isometric view of corner joint welded plates

4.3 Thermal Analysis

Thermal analysis for this project has been conducted to compute the temperature distribution along the workpiece. Time taken for each loadstep is assign to be 1s since the welding speed is 5mm/s. For this project, there are 30 loadsteps which mean time for the whole analysis is 30s. Figure 11a), 11b), 11c) and 11d) below show temperature distribution at 1s, 10s, 20s and 30s respectively.

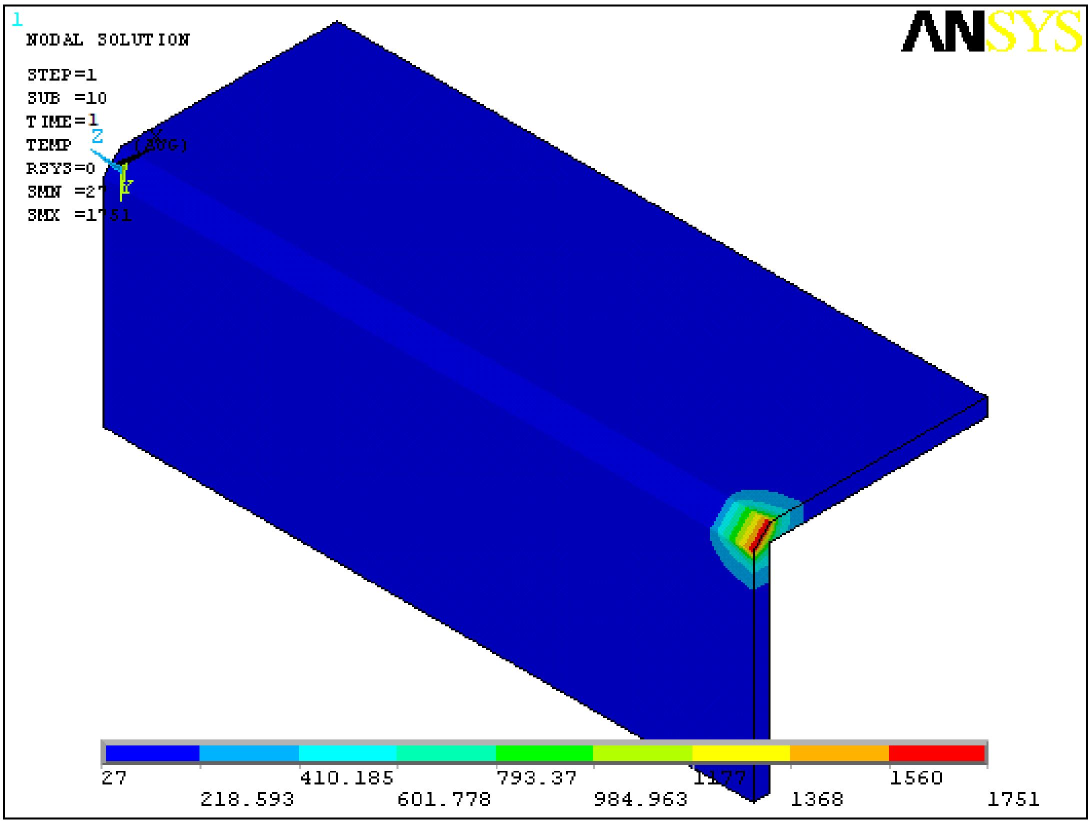


Figure 11a): Temperature distribution of the plate at t=1s

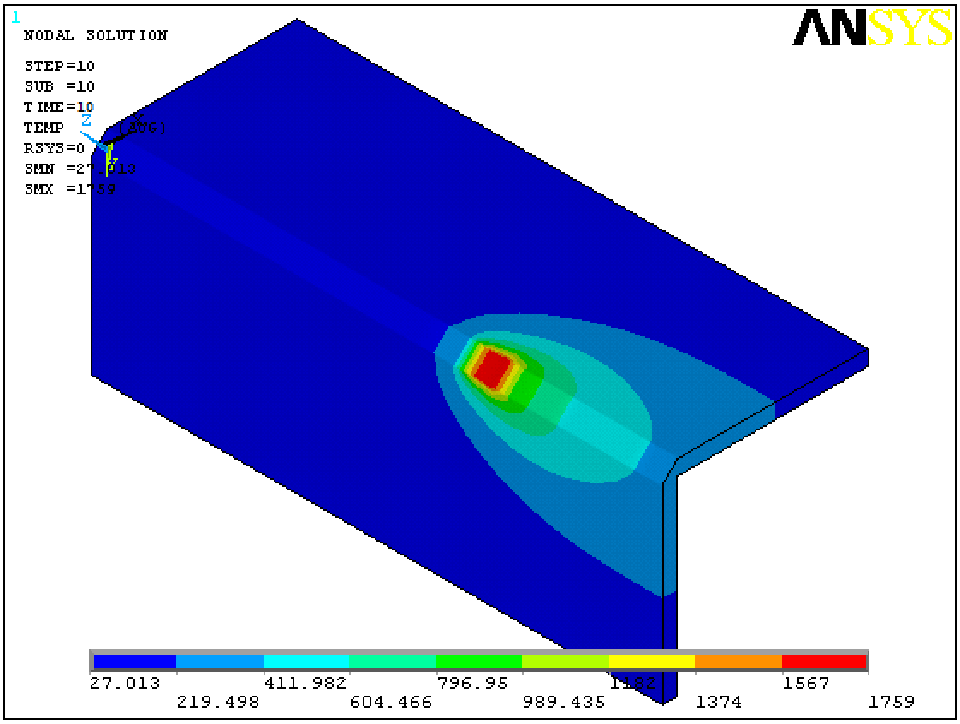


Figure 11b): Temperature distribution at t=10s

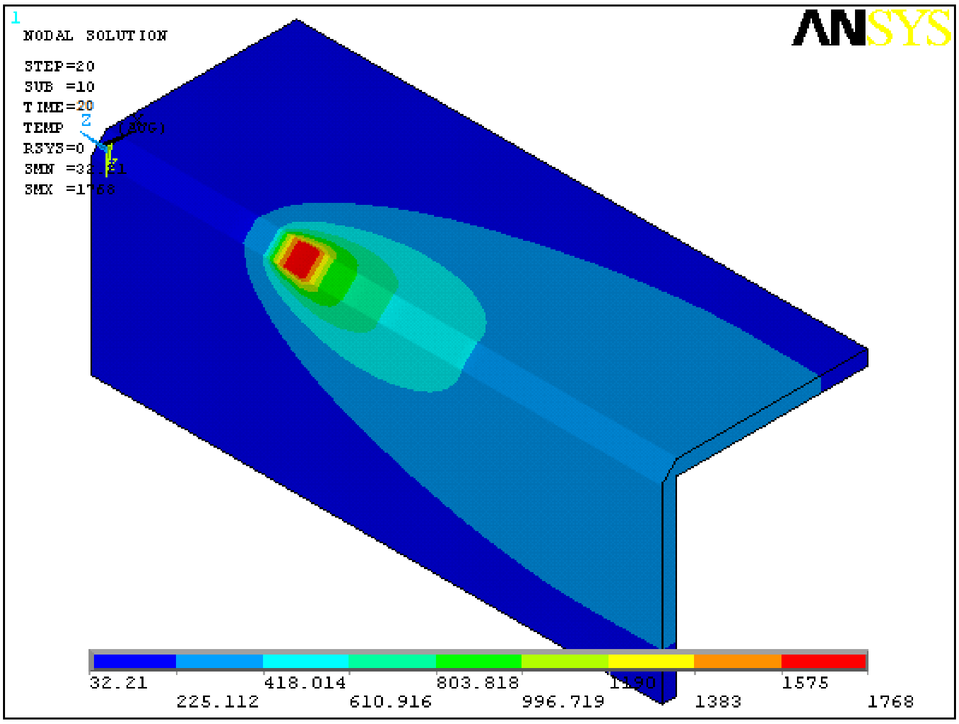


Figure 11c): Temperature distribution at t=20s

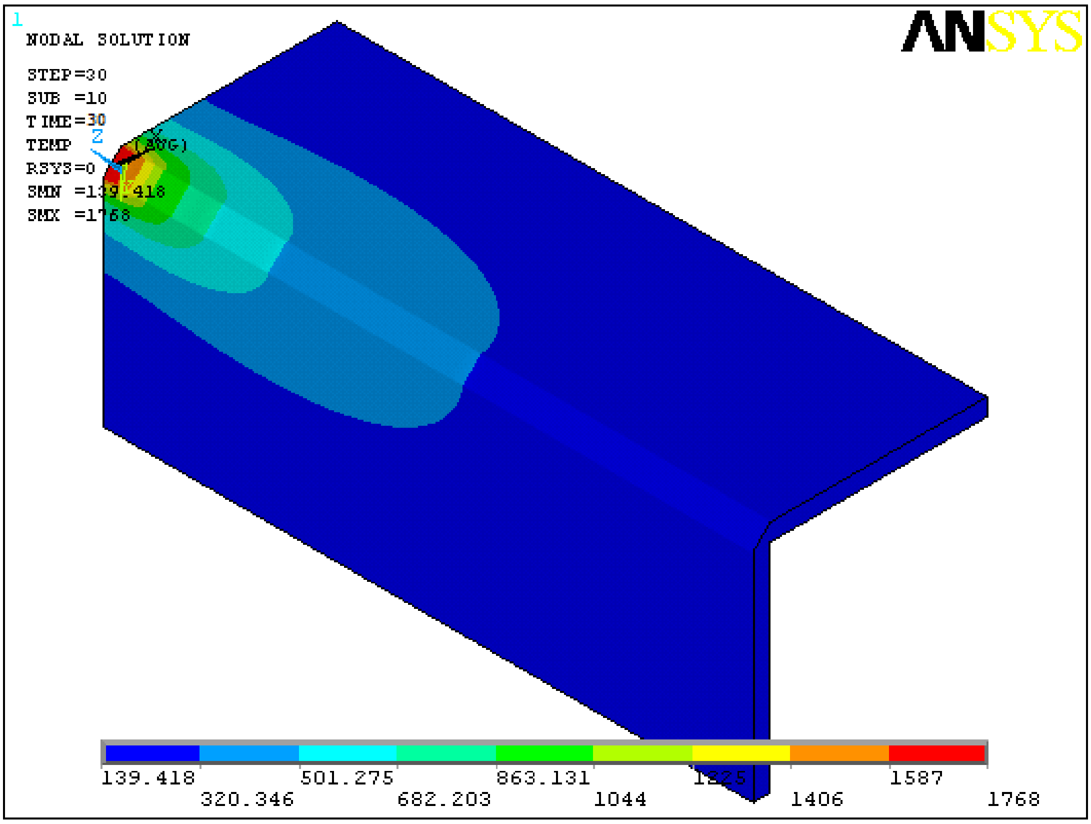


Figure 11d): Temperature distribution at t=30s

From the result, maximum temperature is 1768°C which beyond melting temperature of carbon steel, 1500°C. For workpiece, temperature has been increase from room temperature, 27°C to 139.418°C due to conduction from welded metal. There is also heat loss through surface of the workpiece due to convection.

4.4 Temperature Profile

Every node on the workpiece material experience change in temperature from the beginning of the welding process until it finished. Increase/decrease in temperature at any node is due to conduction and convection of heat. Figure 12 show two chosen nodes while Figure 13a) and 13b) show temperature profile at first node and second node respectively. Figure 13c) shows temperature profile from filler to distance away from weld line.

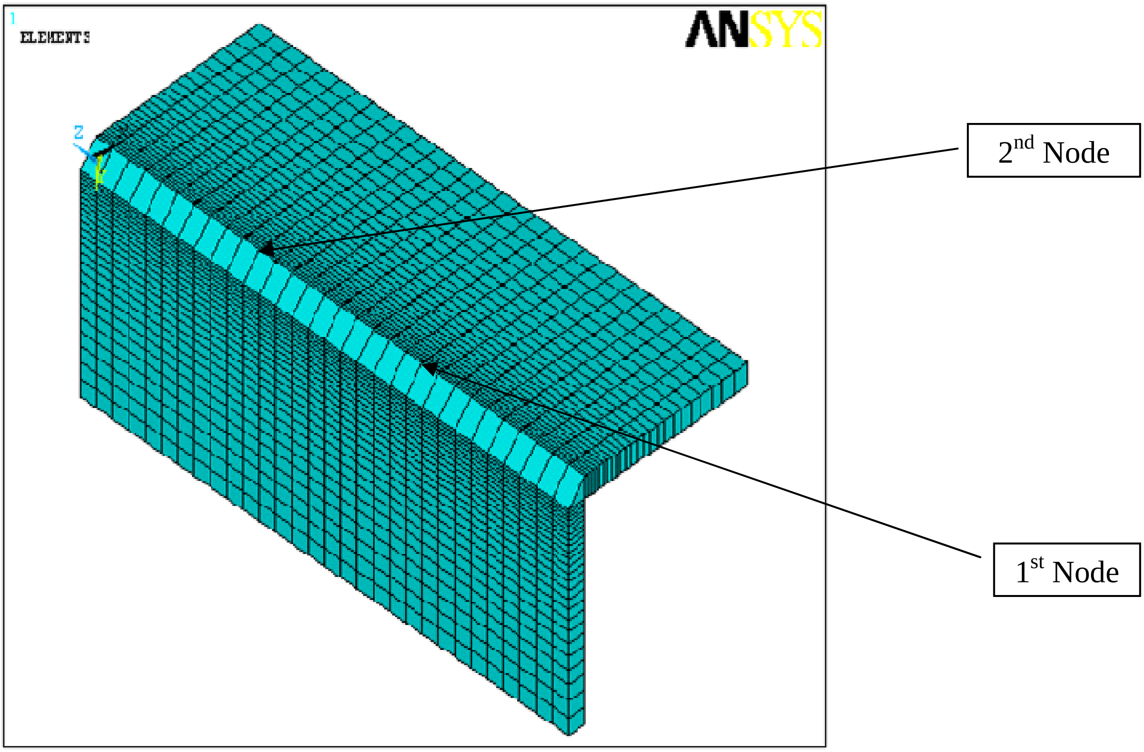


Figure 12: Three chosen nodes for temperature profile plot

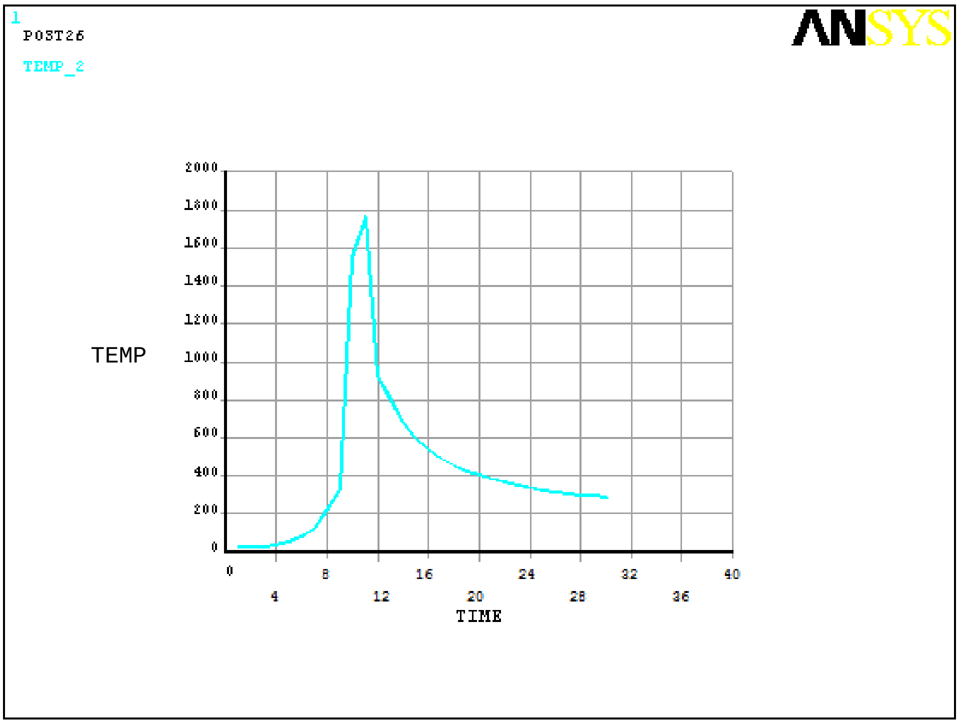


Figure 13a): Temperature profile at first node

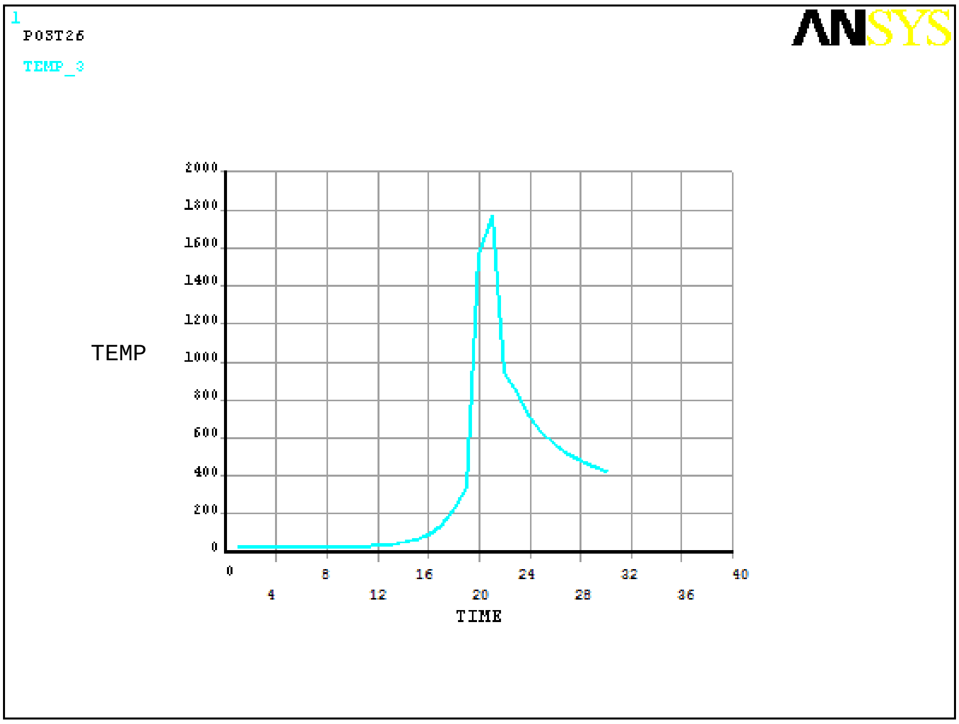


Figure 13b): Temperature profile at second node

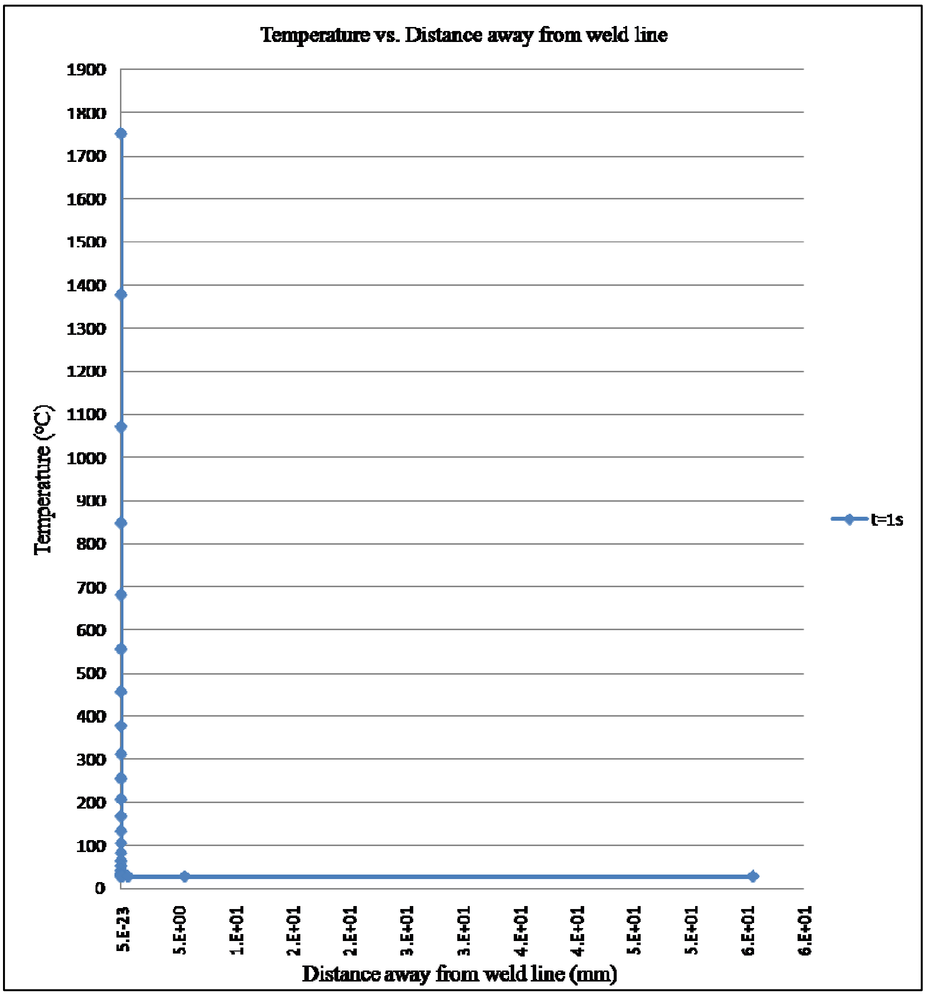


Figure 13c): Temperature vs. Distance away from weld line

At Node 1, highest temperature achieve at time, t=10s because essentially, Node 1 is part of 10th element. From time, t=1s until t=10s, the temperature keep increase because the welding torch approaching Node 1 and from time, t=11s until end of the process at time, t=30s, temperature decreasing since the welding torch move away from Node 1.

From figure 13c), it shows that temperature at sequence of node away from weld pool is decreasing significantly. Besides, this temperature is taken at time, t=1s where the heat is not yet reach node at the end of the plate.

4.5 Structural Analysis

After finished with thermal analysis, the simulation continues with structural analysis. The element type is change from SOLID70 to SOLID45 where the element type is capable of three D.O.F. Load apply for structural analysis is taken from previous thermal analysis result and solve for current load step.

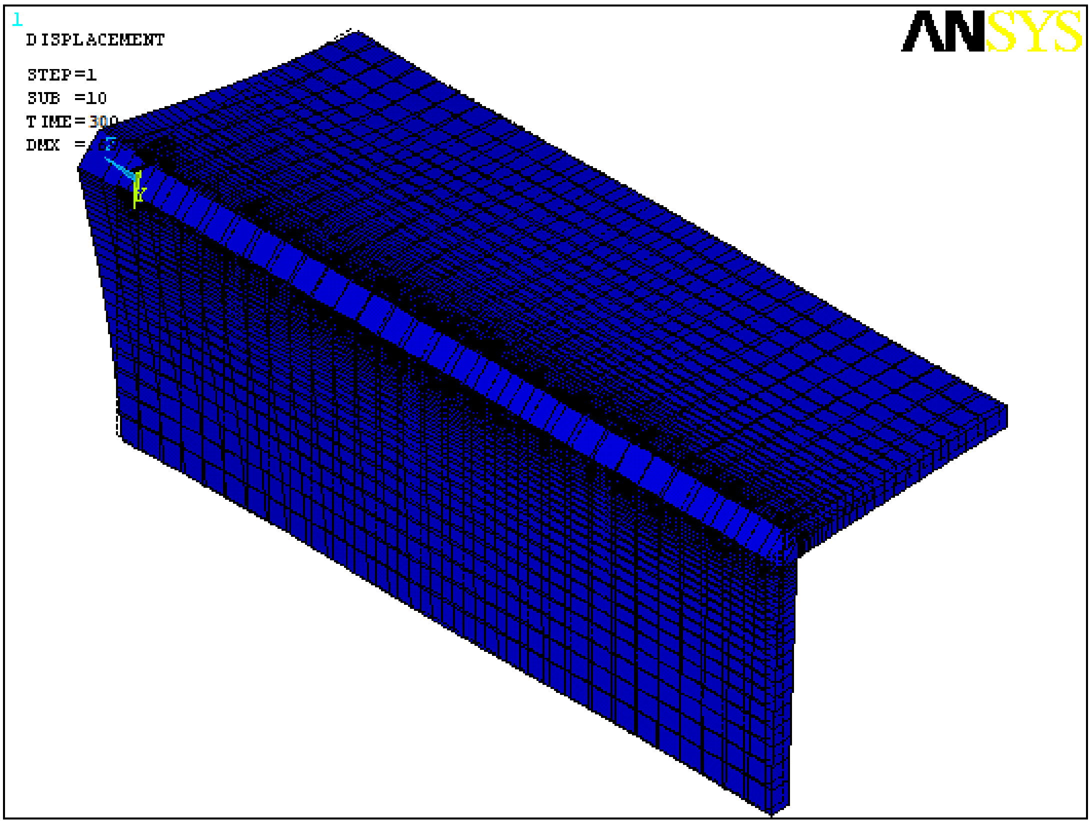


Figure 14: Deformation of welded plate

Workpiece metals are subjected to constraint at four (4) points which represent clamping during welding process. Figure 14 shows the deformation that occurs after 270s the plate is being cold. The welded plate experiences a transverse contraction at the x and y-direction. Solid line shows the deformed shape while dotted line represent undeformed shape of the plate.

During welding, the heats applied expand the workpiece in all direction, but due to constraint of the cooler element on the work piece that far from the welding direction, the workpiece undergoes a non-uniform expansion along z-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 and y-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.

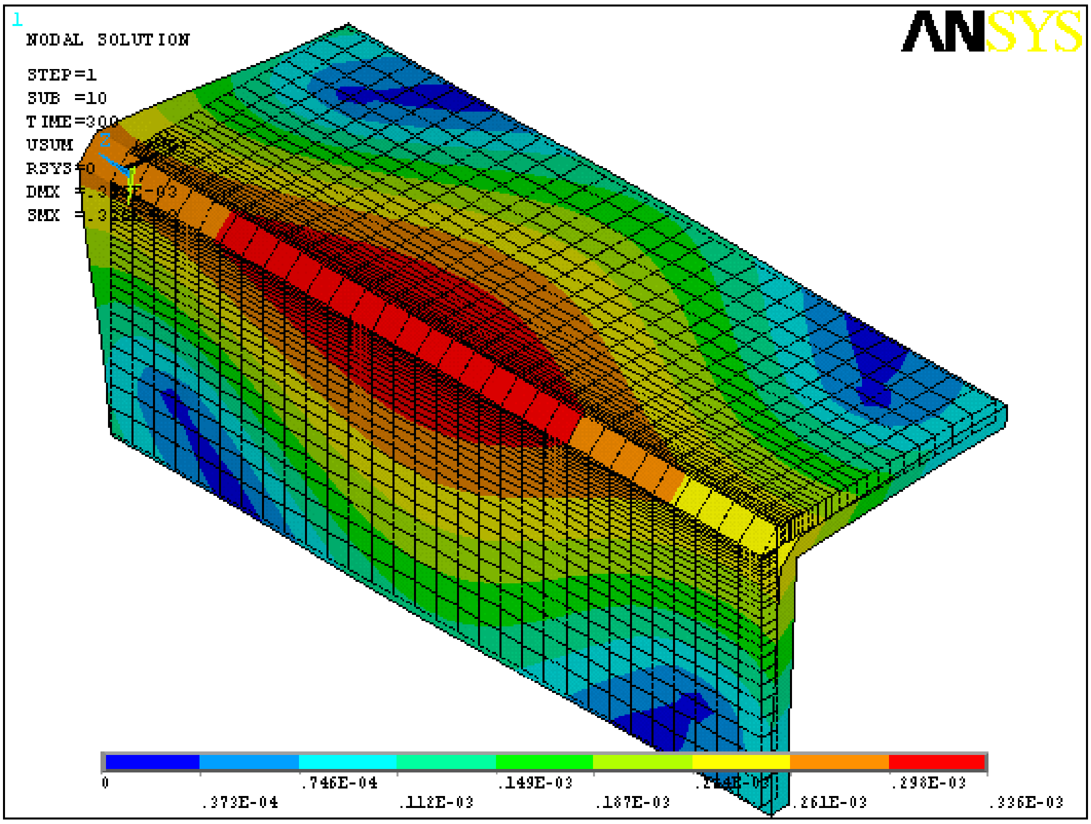


Figure 15: Displacement of workpiece at time, t=300s

Figure 15 above shows displacement that take place on the welded plate after 270s being cold. Maximum displacement occurs along weld pool which expands about 3mm. Area where there is no displacement is due to clamping of the workpiece assigned before welding process start.

Due to deformation of the welded plate, there is arising in stress which remains inside the metal called residual stress. Figure 16 below shows the stress concentration that remain after welding process finished.

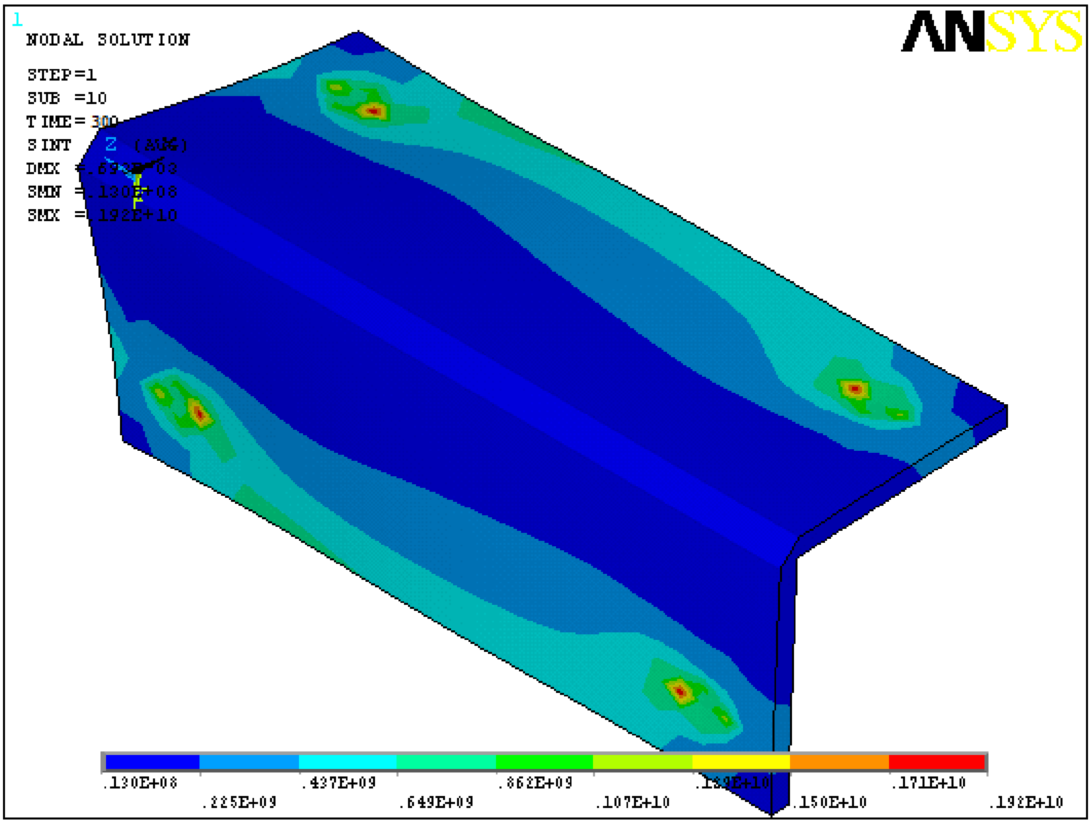


Figure 16: Residual stress concentration

Maximum residual stress located at the clamping area, because contraction of the plate is constraint at that point which leads to increase in stress concentration. Residual stress is significantly high with average of 103MPa.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

In the current research, finite element schemes for simulation of welding residual stresses have been developed. Detailed thermal and residual stresses analyses for corner joint are performed using ANSYS. The FEA method is an efficient technique in analyzing residual stresses in welding process. A three-dimensional finite element welding simulation was carried out on corner joint plate structure. The welding simulation was considered as a sequential coupled thermo-mechanical analysis and the element birth and death technique was employed. This method implemented in welding simulation can be used for other analyses. It could consider different process parameters. Moreover, various geometrical constraints and material non-linearities can be included in the analysis.

The conducted project has successfully simulated electrode travelling with filler and achieves expected result. Temperature distribution is representing by temperature contour. The thermal result then is used as a load for structural analysis in order to get the stress distribution on the welded plate. The result obtained from the structural analysis is the residual stress distribution that remains on the welded plate. The deformation of the plate also obtained from this analysis.

5.2 Recomendation

In order to get more accurate result, the heat flux can be treated as non-constant value, which is different from this project. The heat input can be model by using Rosenthal equation. Besides, the filler material can be treated to not having the same material with the work piece. This project is conducted by assuming the metal is undergo elastic deformation, therefore to model better simulation, plastic deformation properties need to take into consideration.

REFERENCES

1. JF Lancaster *"The Metallurgy of Welding, Brazing and Soldering"* George Allen and Unwin Ltd, 1965.

2. Zuheir Barsoum *"Residual Stress Analysis and Fatigue Assessment of Welded Steel Structure"* Stockholm, Sweden 2008.

3. Z. Barsoum, 2008 *"Residual Stress Analysis and Fatigue of Multi-Pass Welded Tubular Structures"* Engineering Failure Analysis Vol. 15, pg 863-874.

4. Martinsson J. *"Comparison Between Different Contemporary FCG Programs On Welded Component"* 2003.

5. Finch D, Burdekin FM. *"Effect of Welding Residual Stresses on Significance of Defects in Various Types of Welded Joint"* Eng Fract Mech 1992.

6. E. Armentani, R. Esposito, R. Sepe, 2007 *"The Effect of Thermal Properties and Weld Efficiency on Residual Stresses in Welding"* Achievement in Materials and Manufacturing Engineering Vol. 20, pg 319-322.

7. Ueda Y., Fukuda K., and Kim Y.C. *"New Measuring Method of Axisymmetric Three-Dimensional Residual Stresses Using Inherent Strains as Parameters"* Journal of Engineering Materials and Technology 1986.

8. Rybicki E.F., Schmueser D.W., Stonesifer R.W., Groom J.J., Mishler H.W., *"A Finite Element Model for Residual Stresses and Deflections in Girth-Butt Welded Pipes"* Journal of Pressure Vessel Technology, 1978.

9. Wilkening W.W., and Snow J.L. *"Analysis of Welding-Induced Residual Stresses with the ADINA System"* Conputers & Structures Volume 47, 1993.

10. Karlsson C.T. *"Finite Element Analysis of Temperature and Stresses In A Single-Pass Butt-Welded Pipe – Influence of Mesh Density and Material Modeling"* Engineering Computational Volume 6, 1989.

11. Dong P., Ghadiali N., and Brust F.W *"Residual Stress Analysis of A Multi-Pass Girth Weld"* Fatigue, Fracture and Residual Stresses, 1998.

12. Shack W.J., Ellingson W.A., and Pahis L.E. *"Measurement of Residual Stresses in Type 304 Stainless Steel Piping Butt Weldments"* EPRI NP-1413, 1980.

13. Karlsson C.T., and Josefson B.L. *"Three-Dimensional Finite Element Analysis of Temperature and Stresses in Single-Pass Butt-Welded Pipe"* Journal of Pressure Vessel Technology Volume 112, 1990.

14. Leggatt R.H. *"Residual Stresses In Welded Structures"* Journal of Pressure Vessels Piping Volume 85, 2008.

15. Xiangyang Lu and Tasnim Hassan *"Residual Stresses in Butt and Socket Welded Joints"* Washington DC, August 2001.

16. Dragi Stamenkovíc, Ivana Vasovíc *"Finite Element Analysis of Residual Stress in Butt Welding Two Similar Plates"* Scientific Technical Review, Vol. LIX, No. 1, 2009.

17. O.C. Zienkiewicz and R.L Taylor *"The Finite Element Method: Its Basis and Fundamental"* Sitxh Edition, 2005.

18. F. Vakili Tahami, A. H. Daei Sorkhabi, M. A Saeimis, A. Homayounfar, 2009 *"3D Finite Element Analysis of The Residual Stress In Butt-Welded Plates With Modeling of The Electrode Movement"* Journal of Zhejing University, Vol. 10 pg 37-43.

19. N.R Mandal *"Welding and Distortion Control"* Journal of Alpha Sciene International Ltd.,2004.

20. Panagiotis Michaleris and Andrew Debiccari *"Pridiction of Welding Distortion"* Journal of Edison Welding Institute, Columbus, Ohio., April 1996.

APPENDICES

Appendix A

Gantt Chart for FYP I and FYP II

	FYP I														FYP II													
Details/Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Selection of Project Title	■	■																										
Preliminary Research Work		■	■	■																								
Familiarization to ANSYS				■	■	■	■	■	■	■																		
Submission of Preliminary Report				■																								
Study on Type of Welding					■	■																						
Study on Properties of Material							■	■																				
Submission of Progress Report 1								■																				
Seminar								■																				
Study on Thermal Analysis									■	■	■	■	■	■	■	■	■											
ANSYS Thermal Modeling									■	■	■	■	■	■	■	■	■											
Submission on Interim Report														■														
Oral Presentation														■														
Submission of Progress Report 2																		■										

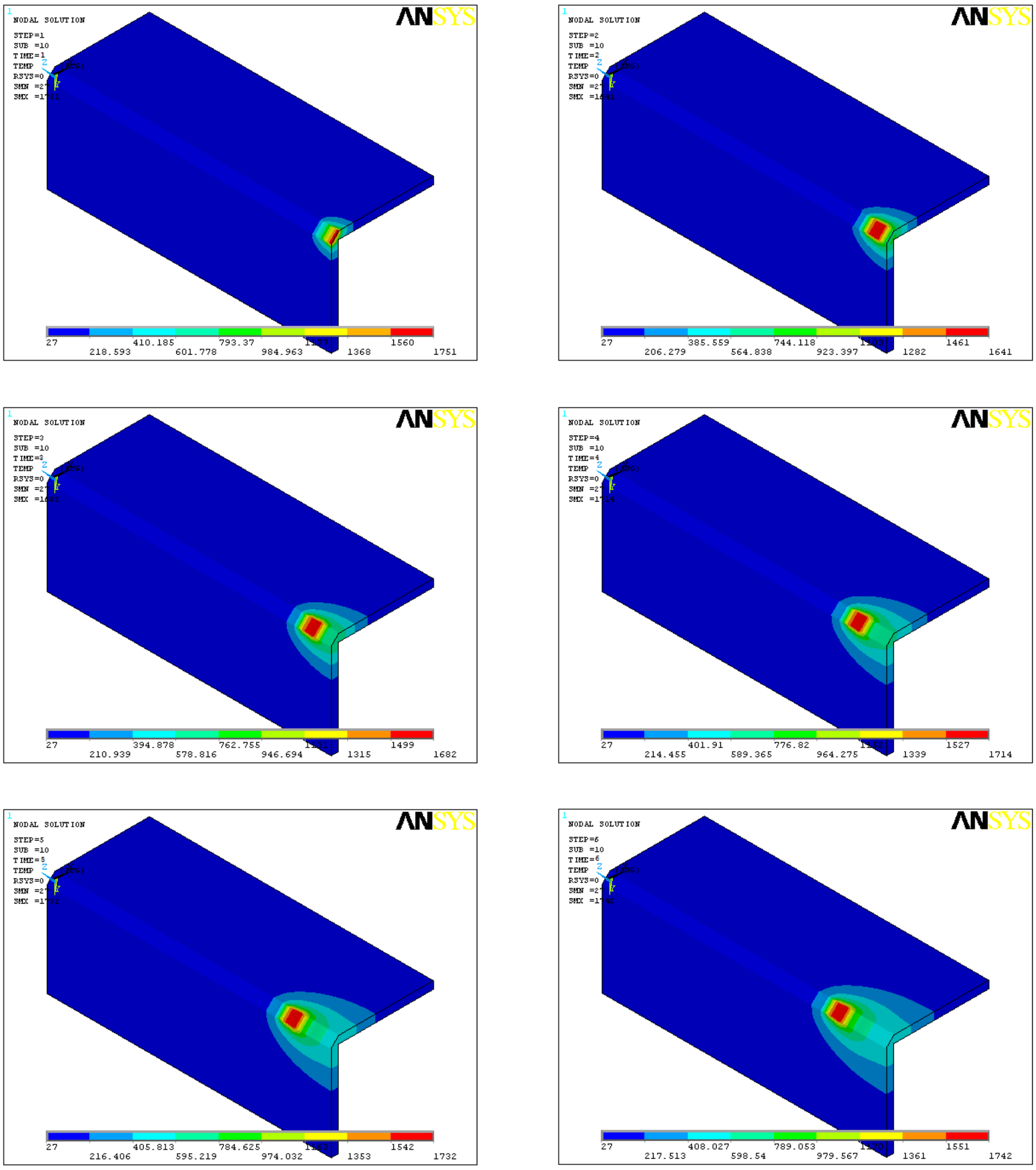
	FYP I														FYP II													
Details/Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Study on Mechanical Analysis																												
ANSYS Mechanical Modeling																												
Submission of Progress Report 3																												
Seminar																												
Poster Exhibition																												
Submission of Dissertation Report																												
Oral Presentation																												

Appendix B

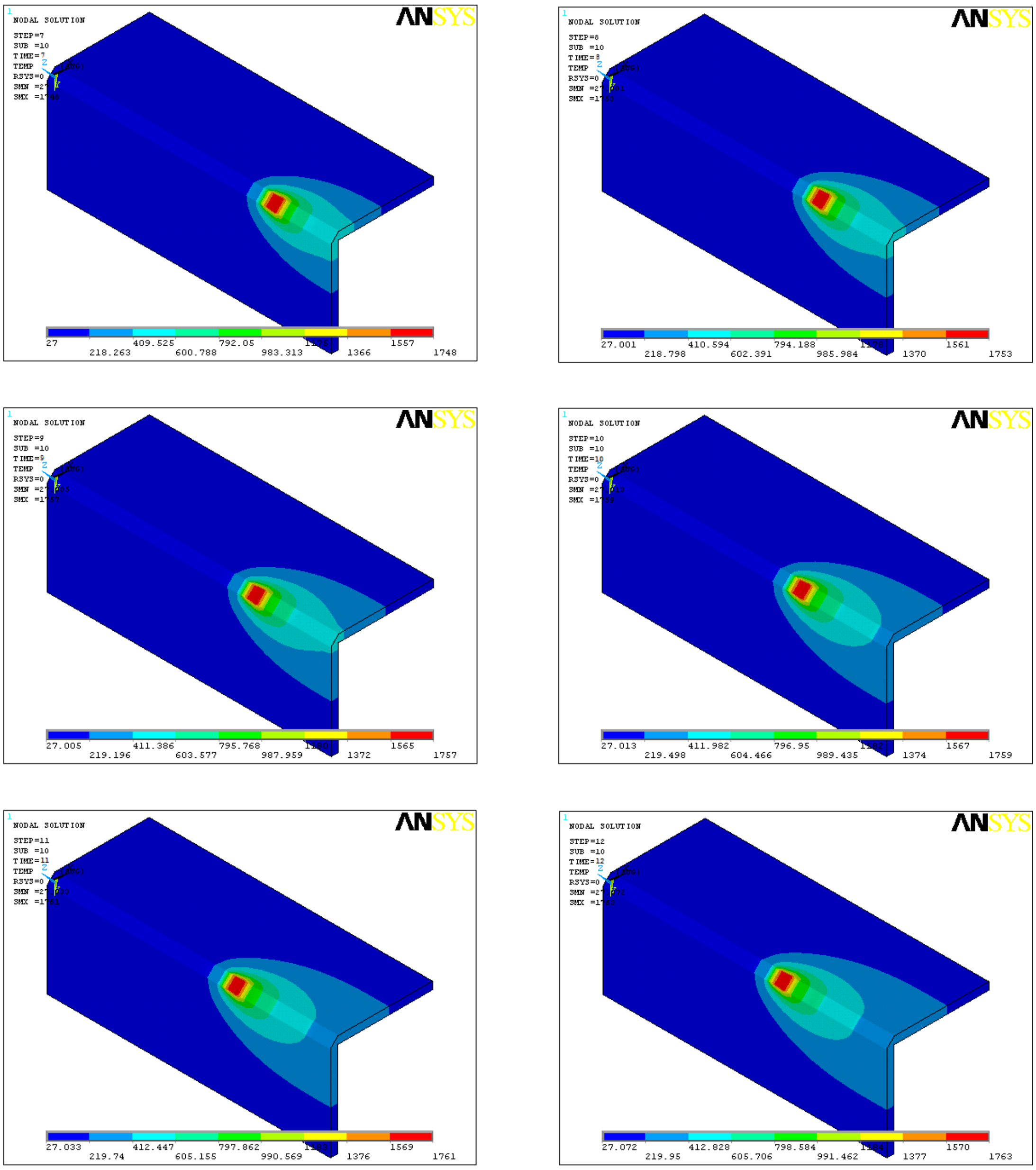
Project Milestone

	FYP I														FYP II													
Details/Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Completion of Literature Review				●																								
Completion of Familiarization to ANSYS										●																		
Completion of Thermal Analysis																	●											
Completion of Mechanical Analysis																										●		
Completion of Dissertation																												●

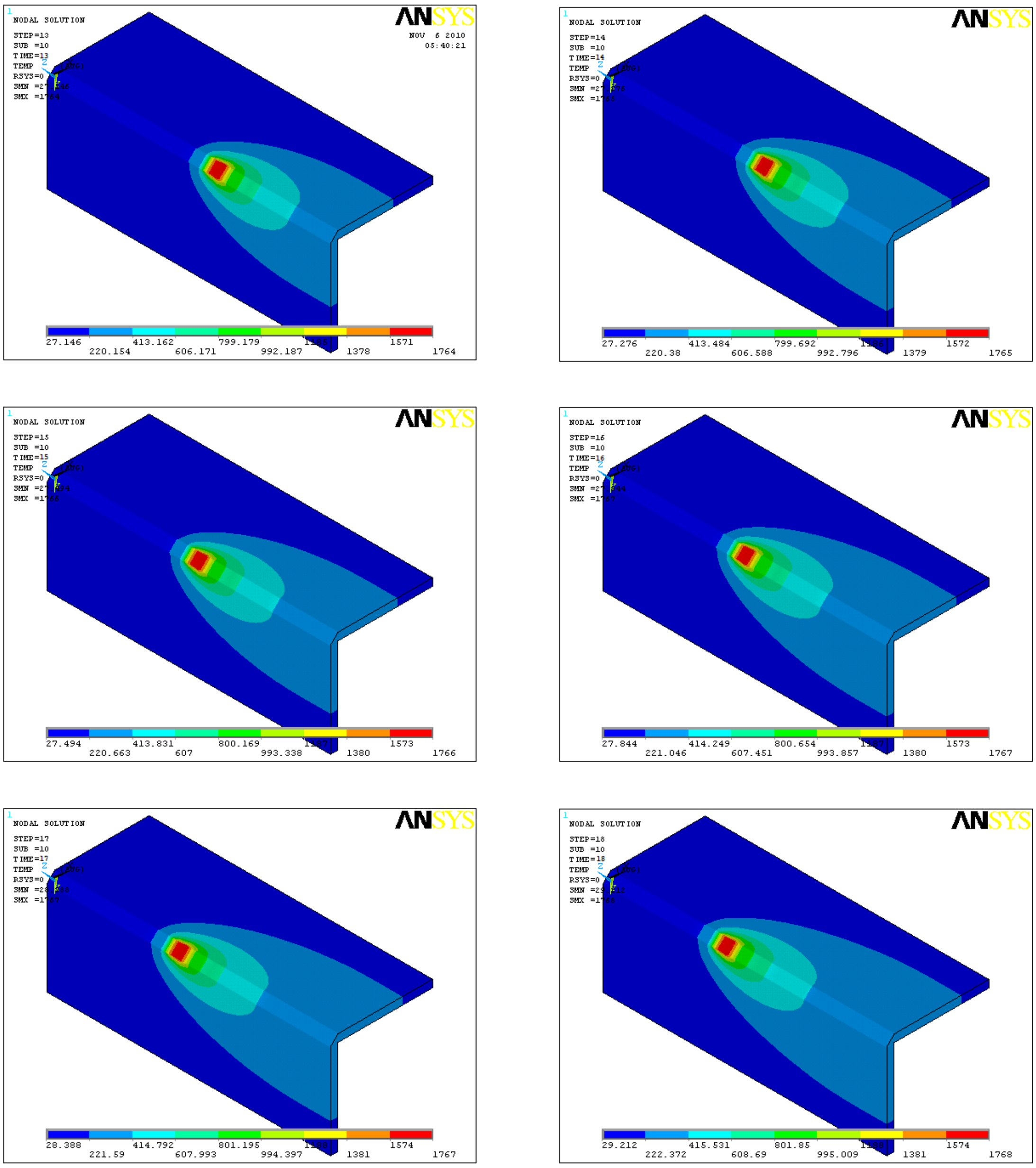
APPENDIX C



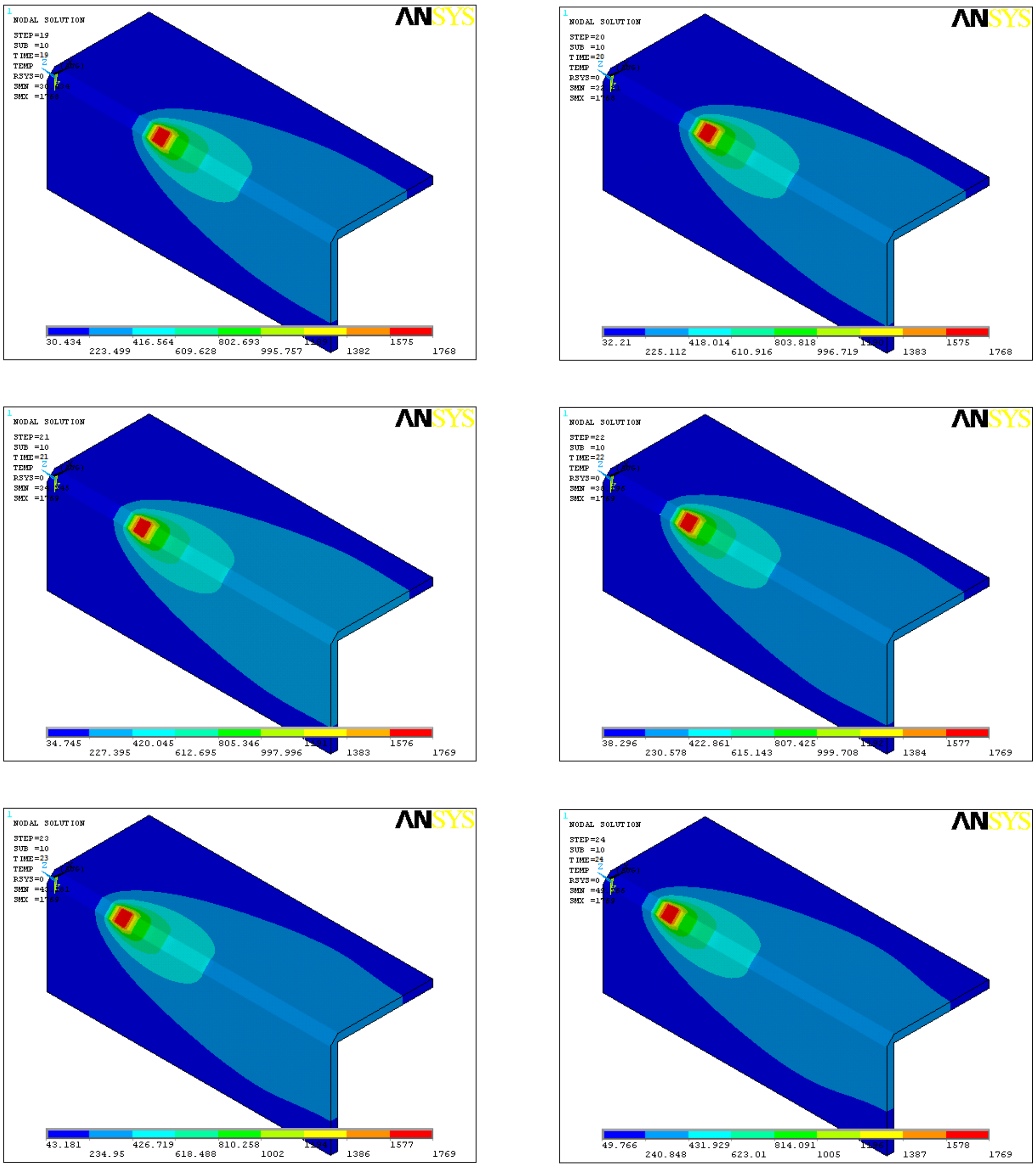
Temperature distribution from t=1s to t=6s



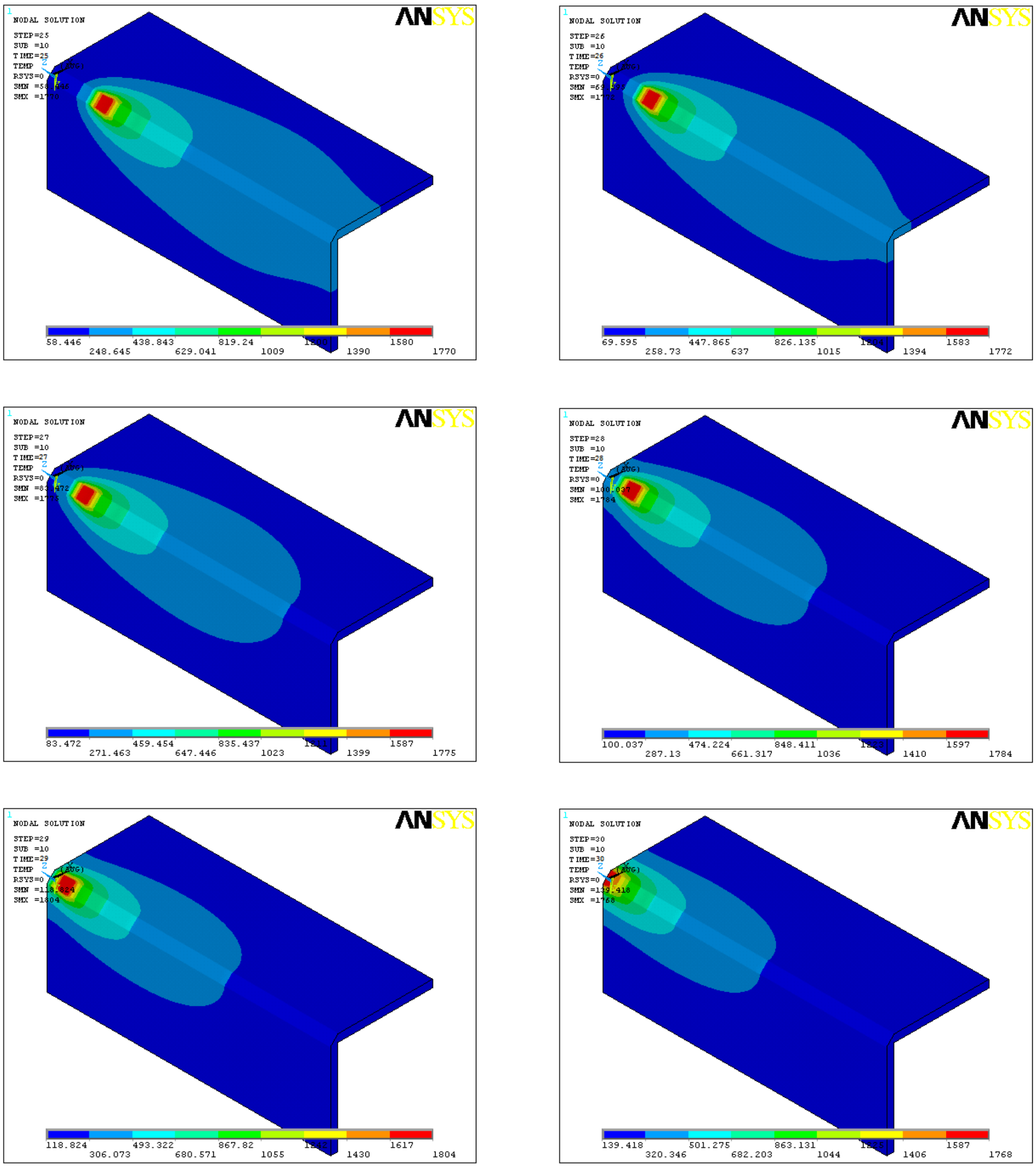
Temperature distribution from t=7s to t=12s



Temperature distribution from t=13s to t=18s



Temperature distribution from t=19s to t=24s



Temperature distribution from t=25s to t=30s

APPENDIX D

ANSYS Command for heat flux load step.

```
Time,1
NSUBT,10,10,10
ESEL,S,ELEM,,1530
EALIVE,ALL
SFE,1530,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,1
SFEDELE,1530,3,HFLUX

Time,2
NSUBT,10,10,10
ESEL,S,ELEM,,1529
EALIVE,ALL
SFE,1529,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,2
SFEDELE,1529,3,HFLUX

Time,3
NSUBT,10,10,10
ESEL,S,ELEM,,1528
EALIVE,ALL
SFE,1528,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,3
SFEDELE,1528,3,HFLUX

Time,4
NSUBT,10,10,10
ESEL,S,ELEM,,1527
EALIVE,ALL
SFE,1527,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,4
SFEDELE,1527,3,HFLUX

Time,5
NSUBT,10,10,10
ESEL,S,ELEM,,1526
EALIVE,ALL
SFE,1526,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,5
SFEDELE,1526,3,HFLUX

Time,6
NSUBT,10,10,10
ESEL,S,ELEM,,1525
EALIVE,ALL
SFE,1525,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,6
SFEDELE,1525,3,HFLUX

Time,7
NSUBT,10,10,10
ESEL,S,ELEM,,1524
EALIVE,ALL
SFE,1524,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,7
SFEDELE,1524,3,HFLUX
```

```
Time,8
NSUBT,10,10,10
ESEL,S,ELEM,,1523
EALIVE,ALL
SFE,1523,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,8
SFEDELE,1523,3,HFLUX

Time,9
NSUBT,10,10,10
ESEL,S,ELEM,,1522
EALIVE,ALL
SFE,1522,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,9
SFEDELE,1522,3,HFLUX

Time,10
NSUBT,10,10,10
ESEL,S,ELEM,,1521
EALIVE,ALL
SFE,1521,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,10
SFEDELE,1521,3,HFLUX

Time,11
NSUBT,10,10,10
ESEL,S,ELEM,,1520
EALIVE,ALL
SFE,1520,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,11
SFEDELE,1520,3,HFLUX

Time,12
NSUBT,10,10,10
ESEL,S,ELEM,,1519
EALIVE,ALL
SFE,1519,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,12
SFEDELE,1519,3,HFLUX

Time,13
NSUBT,10,10,10
ESEL,S,ELEM,,1518
EALIVE,ALL
SFE,1518,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,13
SFEDELE,1518,3,HFLUX

Time,14
NSUBT,10,10,10
ESEL,S,ELEM,,1517
EALIVE,ALL
SFE,1517,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,14
SFEDELE,1517,3,HFLUX

Time,15
NSUBT,10,10,10
ESEL,S,ELEM,,1516
EALIVE,ALL
SFE,1516,3,HFLUX, ,65e6, , ,
```

```
ESEL,ALL
LSWRITE,15
SFEDELE,1516,3,HFLUX

Time,16
NSUBT,10,10,10
ESEL,S,ELEM,,1515
EALIVE,ALL
SFE,1515,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,16
SFEDELE,1515,3,HFLUX

Time,17
NSUBT,10,10,10
ESEL,S,ELEM,,1514
EALIVE,ALL
SFE,1514,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,17
SFEDELE,1514,3,HFLUX

Time,18
NSUBT,10,10,10
ESEL,S,ELEM,,1513
EALIVE,ALL
SFE,1513,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,18
SFEDELE,1513,3,HFLUX

Time,19
NSUBT,10,10,10
ESEL,S,ELEM,,1512
EALIVE,ALL
SFE,1512,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,19
SFEDELE,1512,3,HFLUX

Time,20
NSUBT,10,10,10
ESEL,S,ELEM,,1511
EALIVE,ALL
SFE,1511,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,20
SFEDELE,1511,3,HFLUX

Time,21
NSUBT,10,10,10
ESEL,S,ELEM,,1510
EALIVE,ALL
SFE,1510,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,21
SFEDELE,1510,3,HFLUX

Time,22
NSUBT,10,10,10
ESEL,S,ELEM,,1509
EALIVE,ALL
SFE,1509,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,22
SFEDELE,1509,3,HFLUX

Time,23
```

```
NSUBT,10,10,10
ESEL,S,ELEM,,1508
EALIVE,ALL
SFE,1508,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,23
SFEDELE,1508,3,HFLUX

Time,24
NSUBT,10,10,10
ESEL,S,ELEM,,1507
EALIVE,ALL
SFE,1507,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,24
SFEDELE,1507,3,HFLUX

Time,25
NSUBT,10,10,10
ESEL,S,ELEM,,1506
EALIVE,ALL
SFE,1506,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,25
SFEDELE,1506,3,HFLUX

Time,26
NSUBT,10,10,10
ESEL,S,ELEM,,1505
EALIVE,ALL
SFE,1505,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,26
SFEDELE,1505,3,HFLUX

Time,27
NSUBT,10,10,10
ESEL,S,ELEM,,1504
EALIVE,ALL
SFE,1504,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,27
SFEDELE,1504,3,HFLUX

Time,28
NSUBT,10,10,10
ESEL,S,ELEM,,1503
EALIVE,ALL
SFE,1503,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,28
SFEDELE,1503,3,HFLUX

Time,28
NSUBT,10,10,10
ESEL,S,ELEM,,1502
EALIVE,ALL
SFE,1502,3,HFLUX, ,65e6, , ,
ESEL,ALL
LSWRITE,29
SFEDELE,1502,3,HFLUX

Time,30
NSUBT,10,10,10
ESEL,S,ELEM,,1501
EALIVE,ALL
SFE,1501,3,HFLUX, ,65e6, , ,
ESEL,ALL
```

```
LSWRITE,30
SFEDELE,1501,3,HFLUX

Time,300
NSUBT,10,10,10
ESEL,S,ELEM,,1501,1530
EALIVE,ALL
ESEL,ALL
LSWRITE,31

LSSOLVE,1,31,1
```