FINITE ELEMENT ANALYSIS OF RESIDUAL STRESS IN WELDED THICK PLATES OF A SINGLE V BUTT WELD WITH DIFFERENT INCLUDED BEVEL ANGLES

by

Mohd Zulkifly Bin Abu Hamin

Dissertation submitted in partial fulfillment of the requirements for the

Bachelor of Engineering (Hons)

(Mechanical Engineering)

SEPTEMBER 2013

Universiti Teknologi PETRONAS

Bandar Seri Iskandar

31750 Tronoh

Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Finite Element Analysis of Residual Stress in Welded Thick Plates of a Single V Butt Weld with Different Included Bevel Angles

by

Mohd Zulkifly Bin Abu Hamin

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
In partial fulfilment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(MECHANICAL ENGINEERING)

Approved by,

____

(Dr. Saravanan Karuppanan)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

September, 2013

CERTIFICATION OF ORIGINALITY

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

____

(MOHD ZULKIFLY BIN ABU HAMIN)

ABSTRACT

The main purpose of this project was to study the effect of included bevel angle of single V butt weld on residual stresses in welded thick plates. It is known that the welding of thick plates will generate residual stress which will cause failure to the material performances in future. Thus, in order to minimize the residual stress on the welded thick plates, the included bevel angle of a single V butt weld was studied as it will determine the volume of the filler material required to weld the thick plates. Theoretically, the more the volume of filler material required for welding, the more the residual stress induced as more weld penetration is required to fill the groove.

In this project, a finite element method was used to determine the value of residual stresses on the plates. The different values of included bevel angle were considered while the other parameters were kept constant (e.g. heat input, root face, thickness, and root gap). ANSYS software was used to model and analyze the problem as it was proven to be an effective tool for the investigation of trends affected by the change welding parameters.

40 degree of included bevel angle was found to be the optimum bevel angle for a single V butt weld as it had the smallest distortion and required less volume of filler metal. In contrast, it experienced the largest value of residual stresses and this may be due to the less exposed area of deposited weld to the surrounding which caused the rate of weld cooling metal to decrease. But then, the value of residual stresses in 40 degree model was approximately same as the 50 and 60 degree model. Thus, 40 degree included bevel angle was most the desirable single V butt weld when considering the distortion and fabrication cost. Hopefully, the successful outcome of this project will be a great helping guidance for industrial application towards assessing the integrity of welded thick plates.

ACKNOWLEDGENT

Here, I would like to express my greatest gratitude to God the Almighty for his blessing that allows me to complete my Final Year Project (FYP). My deepest appreciation goes to my project supervisor Dr. Saravanan Karuppanan for his technical and moral support which made this final year project possible. His continuous feedback and support throughout the semester helping the student by giving encouragement, suggestions and ideas are invaluable. I appreciate the time he spent on reviewing my writing and improving my technical writing skills.

Thanks to Mr. Santosh Patel for his ideas, guidance and technical assistance throughout the course of the research work. I also would like to show my appreciation to my FYP project examiners who are Dr. Setyamartana Parman and Dr. Varaha Venkata Lakshmi Narasimha Rao Tadimalla. Thanks also to my colleagues who are doing the same FYP project discipline as me. Their help and support mean a lot to me to complete this project.

Lastly, I would like to express my thankfulness to my parents, my family and all the lectures in UTP for making me a better person today. I would like to thank Jabatan Perkhidmatan Awam (JPA) for giving me the opportunity to study in Universiti Teknologi PETRONAS (UTP).

Thank You.

TABLE OF CONTENT **PAGE**

LIST OF FIGURES **PAGES**

LIST OF TABLES **PAGES**

CHAPTER 1

INTRODUCTION

1.1 Background of Study

Residual stress is known as one of the defect factors to the material and as were creep, brittle fracture, fatigue, structural buckling and stress corrosion cracking (SCC) [1]. Sometimes, the effect is very damaging. Therefore it is very important to be able to monitor and control the residual stress. Almost no technical material, components or structure are available completely free of residual stress.

It goes similarly when welding thick plates. Currently there are many constructions that require welding of thick metals such as ship building, bridges and oil platform that require fabricators to take precautions on the welding parameter that may effect in the generation of residual stress in metal plates.

The fabricator need to decide on the type of welding process, type of joint and position when performing welding of thick plates to ensure that the integrity of the joint is unquestionable the fabrication cost is low.

Many investigators have developed analytical and experimental methods to predict welding residual stresses. Due to the complexity of the problem, the analytical methods are normally used for simple geometries, as the experimental methods are costly to use. However, by advancement in computer technology, it is now possible to use numerical techniques, like finite element methods to reasonably calculate the residual stress in welded structure. Thus, supposedly there is less chances for the fabricators to produce low quality of weld when welding thick plates.

1.2 Problem statement

When welding is performed on 2 thick plates, the geometry of the metal to be jointed is necessary to be considered especially when selecting single V butt weld as the type of joint. Single V butt weld is known as the type of joint that use minimum volume of weld metal to fill the joint aside from ability to produce complete penetration. This desire to keep cost low is certainly understandable. A fabricator supposedly would not want to deposit any more weld metal than is absolutely necessary so that profitability can be assured. Although the cost can be minimized, the single V butt weld is capable to produce high residual stress which can affect the integrity of the weld when certain parameters are not considered. The parameter that will be contributing to the residual stress in single V butt weld is included angle which is a parameter that determines the volume of the weld metal required to fill the joint especially when welding thick plates. Thus, this study will be focusing on optimization of the included bevel angle in single V butt weld on welded thick plates and its effect to the residual stresses.

1.3 Objective of this project

The main objectives of this project are:

- To develop a finite element model of a welding process on thick plates with different included angle of single V butt weld.
- To study the effect of included bevel angle in single V butt weld for welding thick plates.
- To study the stress and temperature distribution in the plate using thermal and structural analysis.

1.4 Scope of Study

This study will concentrate on simple welding configuration which was single V butt joint. The included angle of the single V Butt joint will be classified to 80, 70, 60, 50 and 40 degree. The type of the welding applied in this study was Shielded Metal Arc Welding (SMAW) because of the versatility of the process and the simplicity of its equipment and operation. ASTM A36 Low Carbon Steel was used for both metal plate and filler metal. The simulation was performed using ANSYS 14 software.
With ANSYS, the study will be focusing on:

- The deformation phenomena on the plate after the welding process
- The stress distribution along welded plate
- The temperature distribution along the welded plate

CHAPTER 2

LITERATURE REVIEW

2.1 Welding

Welding is the one of the most commonly used permanent joining technologies in various industries including shipbuilding, aerospace, automobile, construction and power generation. It is a process of joining two pieces of metal by creating a strong metallurgical bond between them by heating or by applying pressure or combination of both. It is distinguished from other forms of mechanical connections, such as bolting or riveting. It is one of the oldest and reliable methods of joining.

There are different methods and standards adopted and there is still a continuous search for new and improved methods of welding. As the demand for welding new materials and larger thickness component increases, mere gas flame welding which was first known to the welding engineers is no longer satisfactory and improved methods such as Metal Inert Gas Welding, Tungsten Inert Gas Welding, Shielded Metal Arc Welding and electron and beam welding have been developed.

In order to produce the high integrity of welding, the fabricator needs to decide the most suitable type of welding process. The welding process is selected depending on certain parameters which can be divided into two, structural and material factor and welding parameters. The structural factors include the type of geometry and sizes, the type of weld joint and weld groove shape. Among the material factors, the mechanical and physical properties of the parent and filler materials are important to be considered. Welding process parameters include among other things, type of the process employed, welding current, welding voltage and arc travelling speed [2].

Welding offers many advantages over other types of joining. It offers an ability to enable direct transfer of stress between members, eliminating gusset and splice plates necessary for bolted structures. Thus, it can minimize the weight of the joint. In the case of tension members, the absence of holes improves the efficiency of the section. It involves less fabrication cost compared to other methods due to handling of fewer parts and eliminating operations like drilling, punching etc. and consequently less labor cost required, thus is economically efficient.

Welding also is ideal for oil storage tanks and ships as it offers air tight and water tight joining. Besides, welded structures is smooth in appearance, therefore it looks pleasing in the eyes. The rigidity of welded structures is higher compared to structures with riveted and bolted connections and it is in line with the modern trend of providing rigid frames. A truly continuous structure is formed by the process of fusing the members together. Generally a welded joint has a great strength and even more strength than the parent metal itself, thereby placing no restriction on the joints [3].

Some of the disadvantages of welding are that it requires skilled manpower for welding as well as inspection. Also, non-destructive evaluation may have to be carried out to detect defects in welds. Welding in the field may be difficult due to the location or environment. Welded joints are highly prone to cracking under fatigue loading and not forgotten, the nemesis of welding, residual stresses can be developed in welded structures if some parameters in welding process are not considered when performing welding [4].

2.2 SMAW Welding

SMAW process is a very common type of welding method which has been practiced in connecting metals together. This type of welding process has very good capability and is simple. By applying this process, metals are welded together with fine quality in terms of strength and durability. In oil and gas industry, most of the pipelines are connected using this process. The concept of SMAW process is to use the heated arc that was created due to huge potential difference between the electrode tip and the metal surface. When this happened, the arc is capable to melt the metal and join them together [5]. By far, the connection quality resulting from this welding process can withstand high load and force that the metals need to withstand during its useful life [6]. Figure 2.1 shows how the arc penetrates the metal to form a pool of molten. This is where the two metals are jointed.

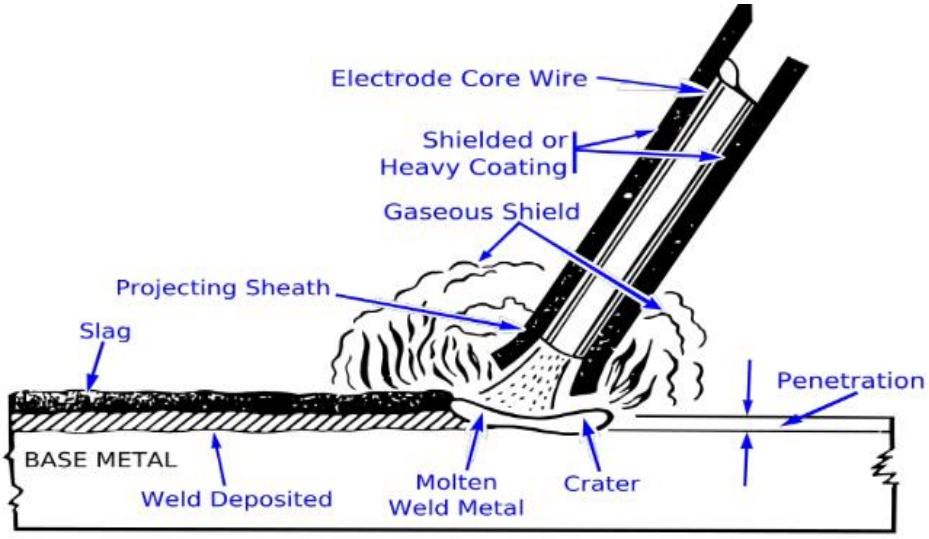


Figure 2.1: Typical SMAW welding in progress [5]

2.3 Residual Stress

Residual stress is a stress that remains in the base metal after the original cause of the stress has been removed. Residual stresses, especially under conditions of multi-axial stress, high loading rate and low temperature, may contribute to the brittle fracture of the welded structures. These stresses also influence the strength and fatigue life, the corrosion resistance and dimensional stability of the welded joints [7]. The knowledge of the residual stresses introduced during the welding operation is essential in the study of the structural integrity of welded structure.

Considerable effort has been devoted to study the mechanism of stress formation and to the development of numerical solutions to predict the residual stress fields in the weld. Stamenkovic [8] used the finite element analysis to perform welding simulation and predict the residual stress in butt welding of two similar carbon steel plates. The results obtained from his research showed that the axial residual stress from the finite element method is approximately close to the experimental result as shown in the Figure 2.2.

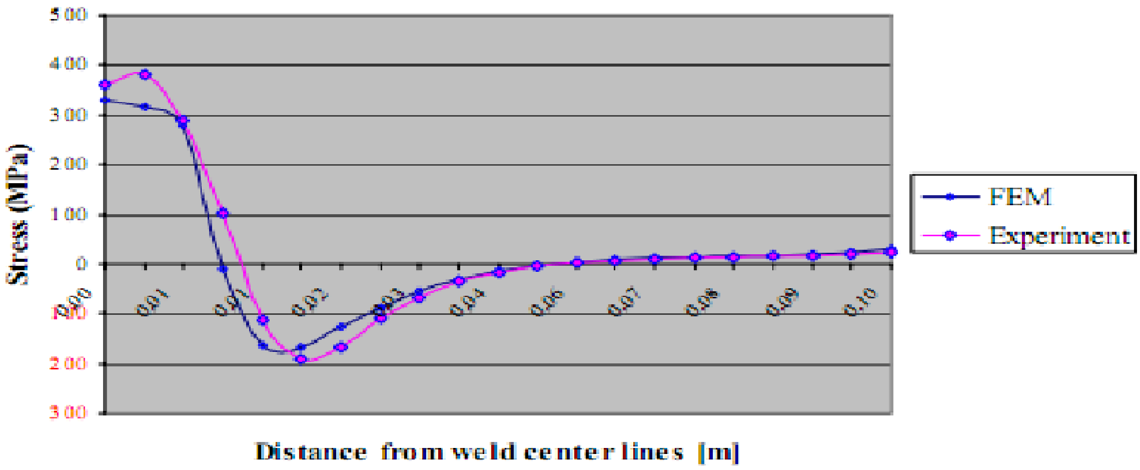


Figure 2.2: Comparison of the FEM with experimental result for axial residual stress [6]

When steel structures are welded, a localized fusion zone is generated in the weld joint because of the high heat input from the arc. Then, non-uniform temperature distribution is induced due to the heat conduction. Therefore, non-uniform heat deformation and thermal stresses are included in the as-welded parts. As a result, plastic deformation is retained within the weldment and non-linear plastic deformation and residual stresses exist after cooling of the welded joint.

Different parameters determine the amount of the residual stresses and its distribution pattern in welded joints. The major parameters according to Leggatt [9] 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, resulting from the processes used to manufacture the components and fabrication operation prior to welding
- Residual stresses generated or relaxed by manufacturing operations after welding or by thermal or mechanical loading during service life

2.4 Weld Geometry

Fatigue behavior of welded joints and welded structures is also greatly affected by the geometry of welded joints [10]. However in design practice, the effect of weld geometry parameters is simply ignored or considered to be non-significant to fatigue behavior of welded joint. This conservative attitude is no longer acceptable due to recent research progress concerning the effect of weld geometry on the fatigue of welded joints. It has been proven in the recent research which emphasized the importance of the size effect of weld geometry e.g. plate thickness and attachment length on the fatigue strength of welded joints. For example, Berge [11] used an experimental approach to investigate the effect of plate thickness on the fatigue strength of transverse fillet welds in axial loading. Meanwhile Nguyen and Wahab [12] developed an analytical model using Linear Elastic Fracture Mechanism (LEFM), finite element analysis (FEA), dimensional analysis and superposition approaches to determine the effect of residual stresses, weld geometry and undercut on the fatigue life of butt weld joints [12]. However, not many researchers have focused on groove parameters like included angle in single V butt weld [13]. One of the researches that have close relationship with this project was conducted by G. Mahendramani [13]. He stated that included angle in single V butt weld has a major effect in single V butt weld on shrinkages of a metal and it is consequently contributing for the residual stress to occur in the weld. His research can be concluded that as the included angle in single V butt weld is increased the shrinkages will increase.

2.5 Butt Weld

Full penetration butt welds are formed when the parts are connected together within the thickness of the parent metal. For thin parts, it is possible to achieve full penetration of the weld. For thicker parts, edge preparation may have to be done to achieve the welding. There are nine different types of butt joints: square, single V, double V, single U, double U, single J, double J, single bevel and double bevel. They are shown in Figure 2.3. In order to qualify for a full penetration weld; there are certain conditions to be satisfied while making the welds.

Welds are also classified into flat, horizontal, vertical and overhead according to their position. Flat welds are the most economical to make while overhead welds are the most difficult and expensive.

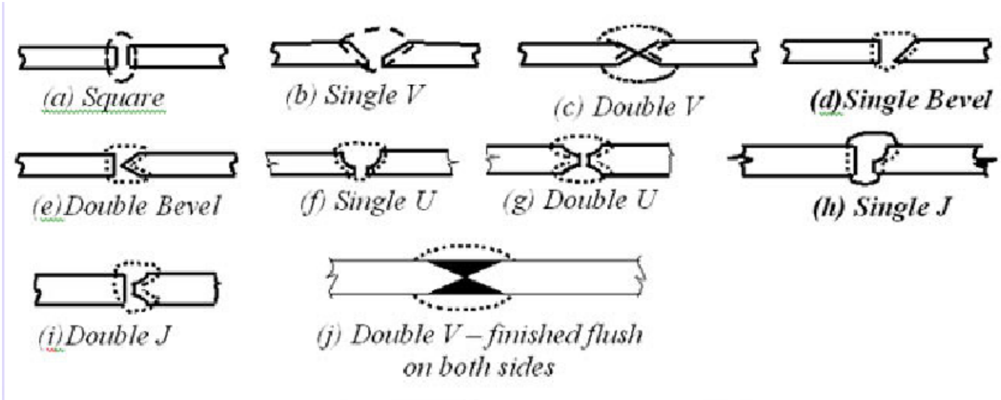


Figure 2.3: Different types of butt welds [10]

The main use of butt welds is to connect structural members, which are in the same plane. A few of the many different butt welds are shown in Figure 2.4. There are many variations of butt welds and each is classified according to its particular shape.

Each type of butt weld requires a specific edge preparation and is named accordingly. The proper selection of a particular type depends upon: Size of the plate to be joined; welding is by hand or automatic; type of welding equipment, whether both sides are accessible and the position of the weld [10].

Butt welds have high strength, high resistance to impact and cyclic stress. They are the most direct joints and introduce least eccentricity in the joint. But their major disadvantages are: high residual stresses, necessity of edge preparation and proper aligning of the members in the field.

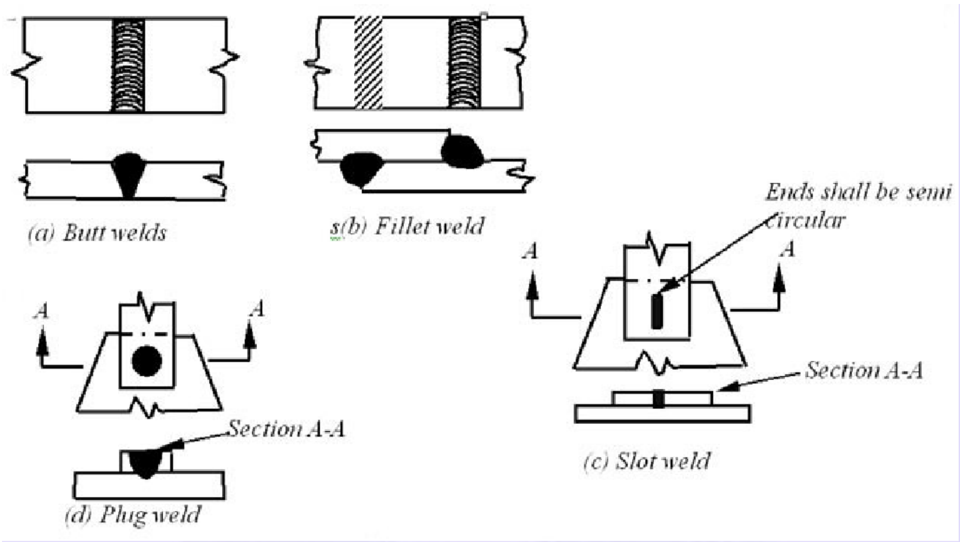


Figure 2.4: Common types of welds [10]

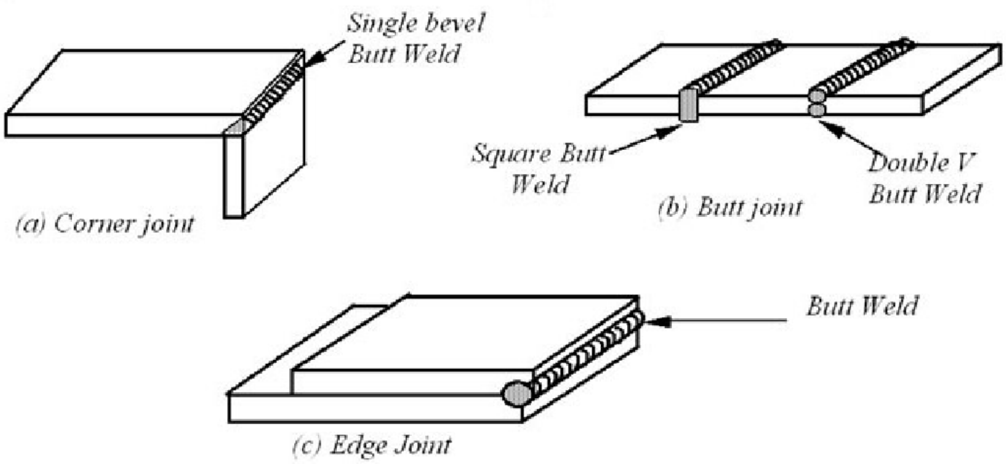


Figure 2.5: Typical connections with butt weld [10]

To minimize weld distortions and residual stresses, the heat input is minimized and hence the welding volume is minimized. This reduction in the volume of weld also reduces cost. For a butt weld, the root gap, *R*, is the separation of the pieces being joined and is provided for the electrode to access the base of a joint. The smaller the root gap the greater the angle of the bevel. The depth by which the arc melts into the plate is called the depth of penetration as shown in Figure 2.6(a). Roughly, the penetration is about 1 mm per 100 A and in manual welding the current is usually 150 – 200 A. Therefore, the mating edges of the plates must be cut back if through-thickness continuity is to be established. This groove is filled with the molten metal from the electrode. The first run that is deposited in the bottom of a groove is termed as the root run as illustrated in Figure 2.6 (c). For good penetration, the root faces must be melted.

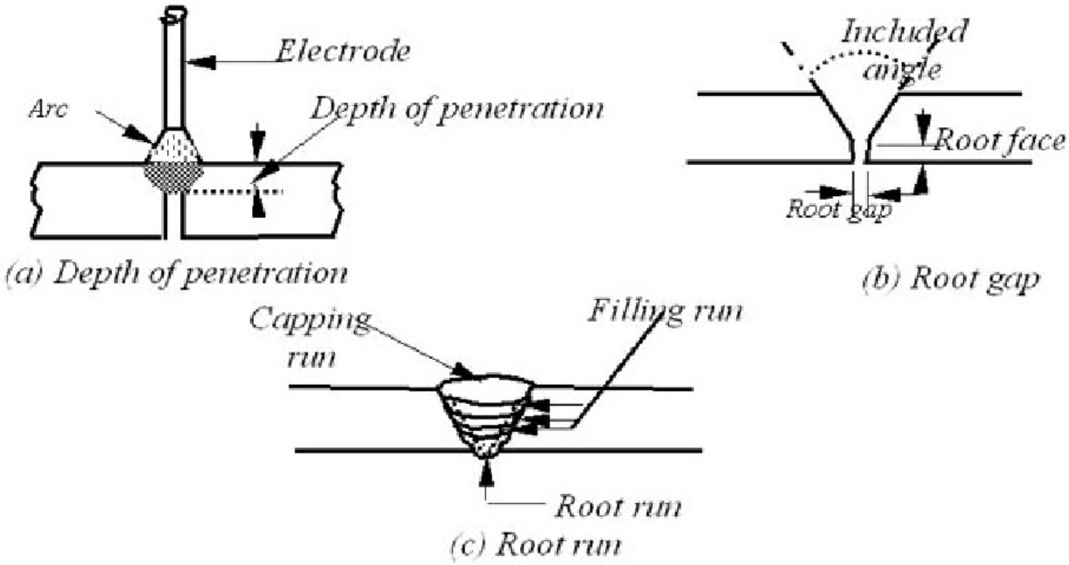
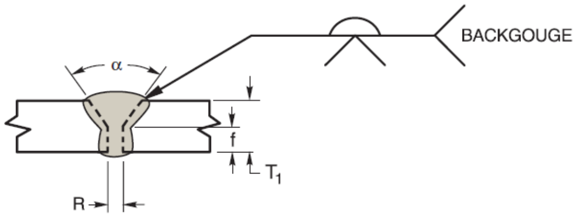


Figure 2.6: Butt weld details [10]

In order to have a better result, this project was referred to the current standards of welding. The standard used throughout the project was American Welding Society (AWS). AWS provide a standard requirement to performing welding which was classified according to the welding process and the geometry of the material. For further details, it can be referred to Table 2.1. SMAW was chosen as welding process in this project, and the geometry of the material including the most suitable root opening, root face and even included angle is shown. However, the effect of the included angle for single V butt weld was not discussed in AWS.

Table 2.1: Standard welding preparation for single V butt weld according to American Welding Society (AWS) [14]

Single-V-groove weld (2)
Butt joint (B)



Welding Process	Joint Designation	Base Metal Thickness (U = unlimited) T_1	T_2	Groove Preparation: Root Opening Root Face Groove Angle	Tolerances: As Detailed (see 2.12.1)	Tolerances: As Fit-Up (see 3.3.4)	Allowed Welding Positions	Gas Shielding for FCAW	Notes
SMAW	B-U2	U	—	R = 0 to 3 f = 0 to 3 α = 60°	+2, –0 +2, –0 +10°, –0°	+2, –3 Not limited +10°, –5°	All	—	1, 3, 9
GMAW FCAW	B-U2-GF	U	—	R = 0 to 3 f = 0 to 3 α = 60°	+2, –0 +2, –0 +10°, –0°	+2, –3 Not limited +10°, –5°	All	Not required	3, 9
SAW	B-L2c-S	Over 12 to 25	—	R = 0 f = 6 min α = 60°	R = ±0 f = +6, –0 α = +10°, –0°	+2, –0 Not limited +10°, –5°	F	—	3, 9
		Over 25 to 38	—	R = 0 f = 10 min α = 60°					
		Over 38 to 50	—	R = 0 f = 12 min α = 60°					

.

CHAPTER 3

METHODOLOGY

3.1 Research Methodology

Several steps were followed in order to obtain correct and relevent results. These steps are important to ensure that the method being used is correct and the results obtained are not misinterpreted. There were four major steps for this project. The first step was to understand the problems and the purposes of this project. Secondly was to understand the project by interpreting the literature review. During this step, the understanding of literature review was very essential. Both steps was to prevent misunderstanding of the project. The third step was the modelling and simulation part. All the data for the SMAW welding and the material properties for ASTM A36 low carbon steel metal plates were gathered to be put in the ANSYS software for the simulation. There were two analysis that need to be completed, namely structural and thermal analysis. The last part was about data interpretation. For better understanding, process flow of this project can be referred to Figure 3.1.

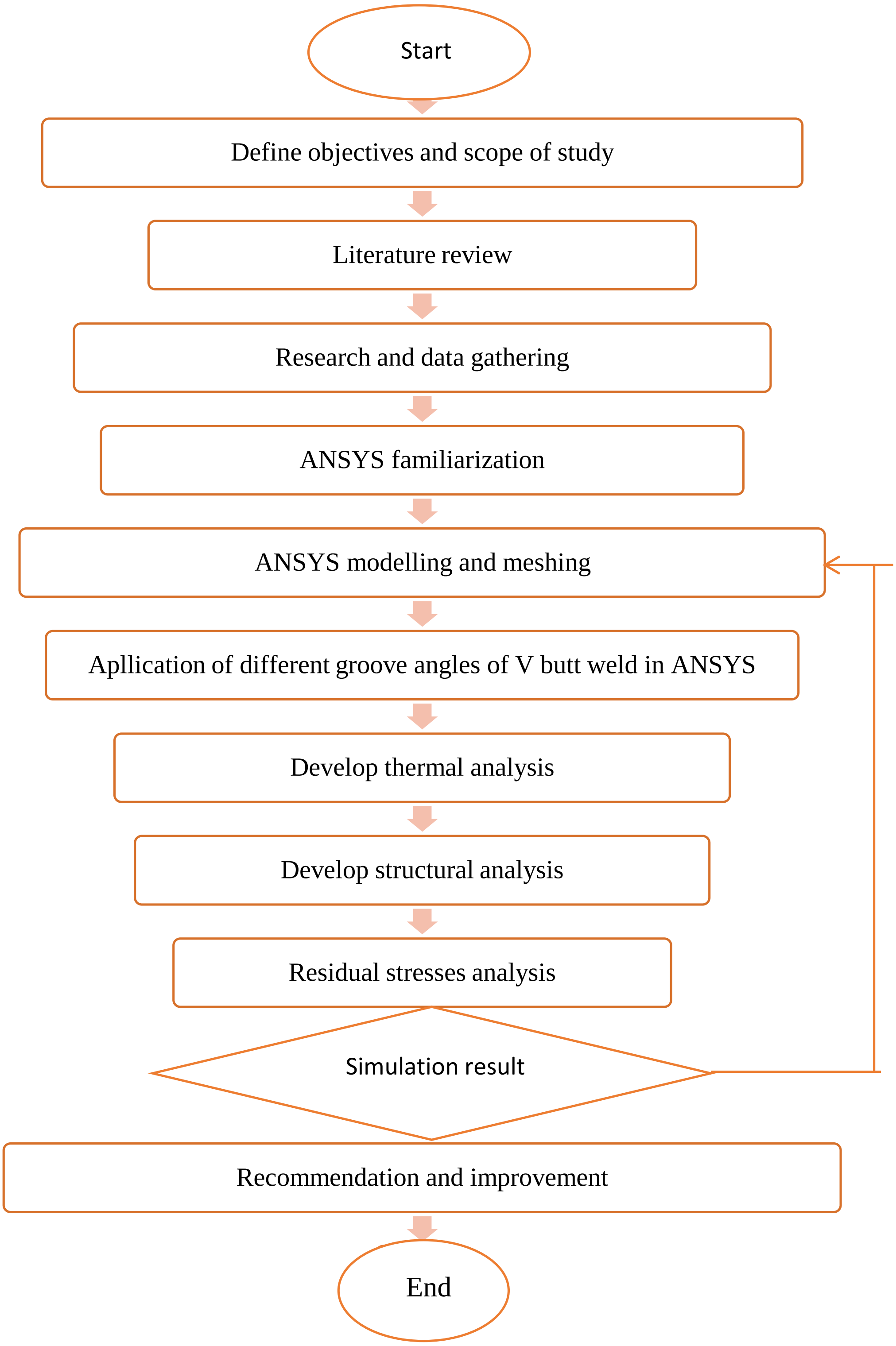


Figure 3.1: Flow chart of overall steps taken in completing the project

3.2 Steps involved in Finite Element Analysis

3.2.1 Pre - processing Phase

The steps involved were:

1. Define element types and options
2. Define material properties
3. Create model geometry
4. Define meshing controls
5. Mesh of the object created

3.2.2 Applying Boundary Conditions, Loads, and the Solutions

The method used to set the boundary conditions and loading of the model was by applying the conditions to the solid model (keypoints, lines, and areas). This method was used because if there were any changes in the meshing, there was no need to reapply the boundary conditions and the loads to the new model [15].

Required boundary conditions:

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

3.2.3 Results of Finite Element Model: Post Processing

The post processing was the part where the result of the analysis was obtained. The results displayed were:

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

3.3 Gantt Chart and Key Milestones

All activities involved in the flow to accomplish the analysis have been put in an appropriate schedule or Gantt chart. This is very important in order to ensure that every planned activity is completed according to the schedule. Gantt chart and key milestone for this project is shown in Table 3.1

Table 3.1: Gantt chart and Key Milestone of the project

Project Activities	Weeks (FYP1) 1	2	3	4	5	6	7	Mid –semester Break	8	9	10	11	12	13	14
Selection of project topic: FEA of welded thick plates with different included bevel angle using ANSYS • Knowing what kind of analysis is involved in FEA • Knowing roughly about residual stresses on welded thick plates and the effect of included bevel angle of single V butt weld on welded structure.	■	■													
Research on literatures related to the topic • Background of the project • Knowing the significance of this project in life and industry • Decide the objectives and scope of study of project		■	■	■	■	■	■								
Submission of Extended Proposal						▲									
Proposal defense • Make a presentation to defend this project as an FYP									■						
ANSYS Familiarization									■	■					
Study on the methods involved in the ANSYS simulation • Standard methodology (steps) of modeling, meshing and analysis							■		■	■	■	■	■	■	■
Prepare model for ANSYS simulation • Creating the metal plates with selected included bevel angle in single V butt weld.									■	■	■				
Study on simulations results												■	■	■	■
Submission of Interim Draft Report														▲	
Submission of Interim Report															▲

Table 3.2: Gantt chart and Key Milestone of the project for FYP2

Project Activities	Weeks														
	FYP2														
	1	2	3	4	5	6	7	Mid –semester Break	8	9	10	11	12	13	14
Study on Thermal Analysis • Apply Boundary Conditions • Steps involved in Thermal analysis	■	■	■	■	■										
ANSYS Thermal Modelling • Analyze the thermal profile		■	■	■	■	■	■		■	■					
Submission of Progress Report 2									▲						
Study on Mechanical Analysis • Apply Boundary Conditions • Steps involved in Mechanical Analysis			■	■	■	■	■		■	■	■				
ANSYS Mechanical Modeling • Analyze the residual stresses and Distortion			■	■	■	■	■		■	■	■	■			
Seminar									■						
Pre-SEDEX Presentation											■	■			
SEDEX Presentation												■	■		
Submission Draft of Dissertation													▲		
Submission of Soft Bound Dissertation Report														▲	
Submission of Technical Report															▲
Oral Presentation (VIVA)															▲
Submission of Hard Bound Dissertation															▲

3.4 Introduction to ANSYS Software

ANSYS is an engineering simulation software created by ANSYS Inc. ANSYS is a general purpose finite element modeling package used to numerically solve a wide variety of mechanical problem. It is widely used 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 with specific boundary conditions.

3.5 Steps Involved in Developing Simulation Using ANSYS Software

3.5.1 Thermal Analysis

1. Define the element type

The element type set for the model in thermal analysis was SOLID70 as it has a 3-D thermal conduction capability. SOLID70 is an element defined by eight nodes having three degree of freedom at each node. This element type is applicable to a 3-D steady state and transient thermal analysis [16]. The element also can compensate for mass transport heat flow from a constant velocity field. Since this analysis will be followed by structural analysis, SOLID70 element can be changed to structural element.

GUI Method: **ANSYS Main Menu > Preprocessor > Element Type > Add/Edit/Delete**

2. Define material properties

In thermal analysis, there were several thermal properties required to be applied on the model. These properties were based on the metal plate properties, ASTM A36 low carbon steel. The properties needed were thermal conductivity, specific heat and density. This property does change with temperature. The ASTM A36 thermal properties are as in Table 3.2.

Table 3.3: Material properties of A36 low carbon steel [17]

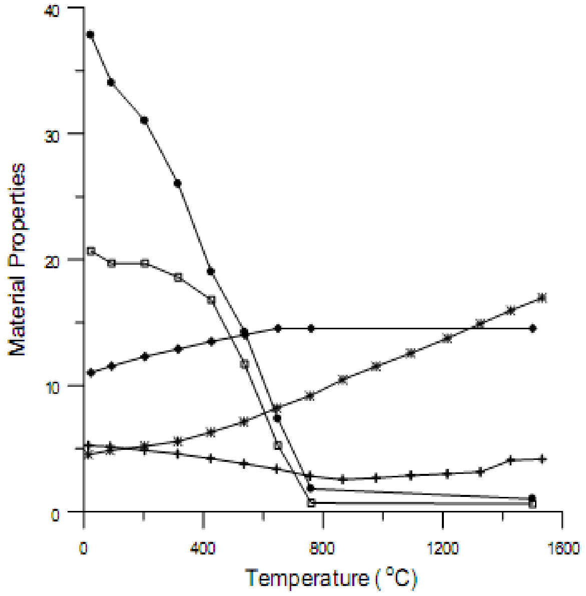
No.	Temperature (K)	Specific heat (J/kg.K)	Conductivity (W/m.K)	Yield Stress (MPa)	Thermal Expansion Coefficient (10^{-5}/K)	Young's Modulus (GPa)
1	273	480	60	380	1.1	210
2	373	500	50	340	1.15	200
3	473	520	45	315	1.2	200
4	673	650	38	230	1.3	170
5	873	750	30	110	1.42	80
6	1073	1000	25	30	1.45	35
7	1273	1200	26	25	1.45	20
8	1473	1400	28	20	1.45	15
9	1673	1600	37	18	1.45	10
10	1823	1700	37	15	1.45	10

GUI Method: **ANSYS Main menu > Preprocessor > Material Props > Material Models**

3. Assumptions

Several assumptions have been considered to perform the simulation. These assumptions were considered for the ease of the related analysis and calculation. Also, some of the assumptions were made so that the simulation will be smooth in ANSYS working plane. Assumptions made were as follows:

- Material properties of the filler and electrode are the same as the material properties of the base metal
- The mechanical properties are time dependent. Chang and Teng [18] assumed that when temperature increase, the modulus of elasticity, yield stress and thermal conductivity for ASTM A36 carbon steel decrease as illustrated in Figure 3.2.



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	×10 W/m °C
*—c	Specific heat	$\times 10^2$ J/Kg °C

Figure 3.2: Temperature dependent material properties of ASTM A36 carbon steel [18]

- The analysis was based on quasy-steady state where the heat source was moving at a constant velocity.
- Temperature distribution along the element was constant at all surfaces of the element.
- Heat input was moving at a constant speed of 5 mm/sec.
- Convection boundary condition was applied on the model
- Radiation heat transfer was neglected.
- Initial temperature was set to 27℃
- Electrode diameter was 4√2 mm

4. Boundary Conditions

To simulate this project, a few boundary conditions have been considered which were:

- The bulk air temperature was set to be 27℃
- Estimated convection heat transfer coefficient, *h* for carbon steel was set to 15 W/(m^2℃)
- Both ends of the plate were assumed to be clamped and the displacement is constant at the both end of the plate.

5. Modeling

Create geometry of the model. There were 3 volumes for the model. First and third volume were the work piece (metal plates) while the second volume was the filler material. The dimension of the model is as per Table 3.3, Figure 3.3 and 3.4. The thickness and the width of the plates were constant which were 20 mm and 70 mm, respectively. The variable parameter was the width of the second volume which varies for different included angle of the single V butt weld.

Table 3.4: The variable values of included bevel angle

Included bevel angle of V butt weld (degree)	Root face *f* (mm)	Root Gap, *R* (mm)
80	**3**	**2**
70	**3**	**2**
60	**3**	**2**
50	**3**	**2**
40	**3**	**2**

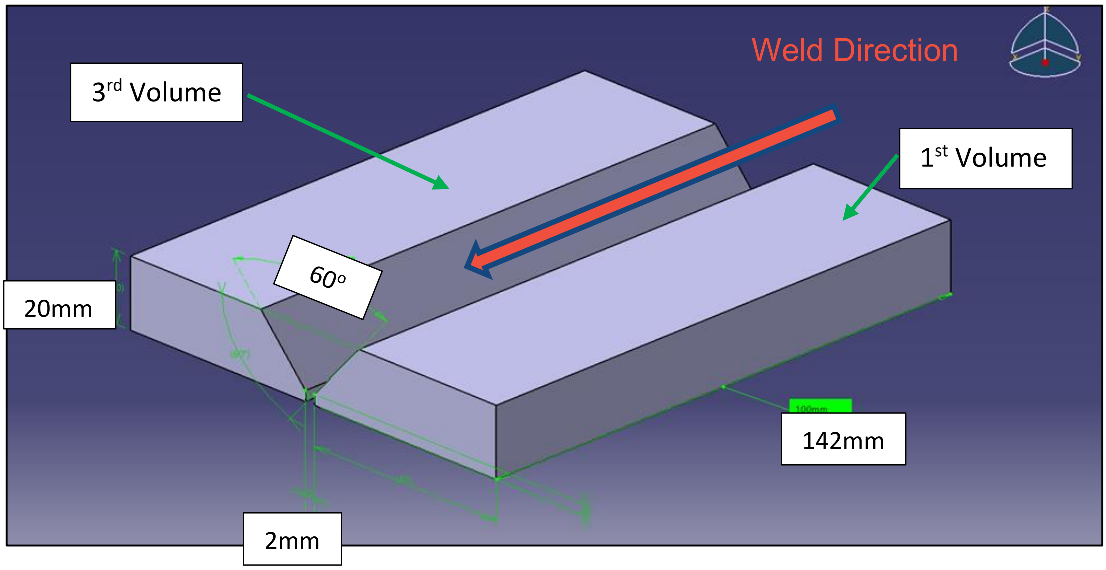


Figure 3.3: Isometric view of the model

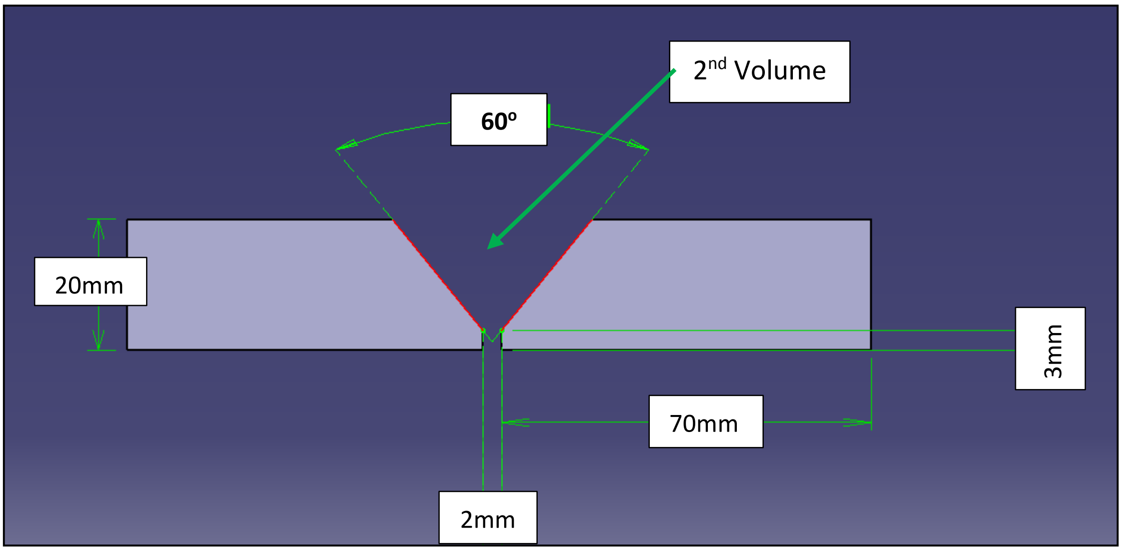


Figure 3.4: Front view of the model

Geometry Modeling

There were five models used for this research and they can be classified based on the included bevel angles. All of the models have been constructed using ANSYS Software. Figure 3.5 to Figure 3.9 show the geometry model used in this analysis.

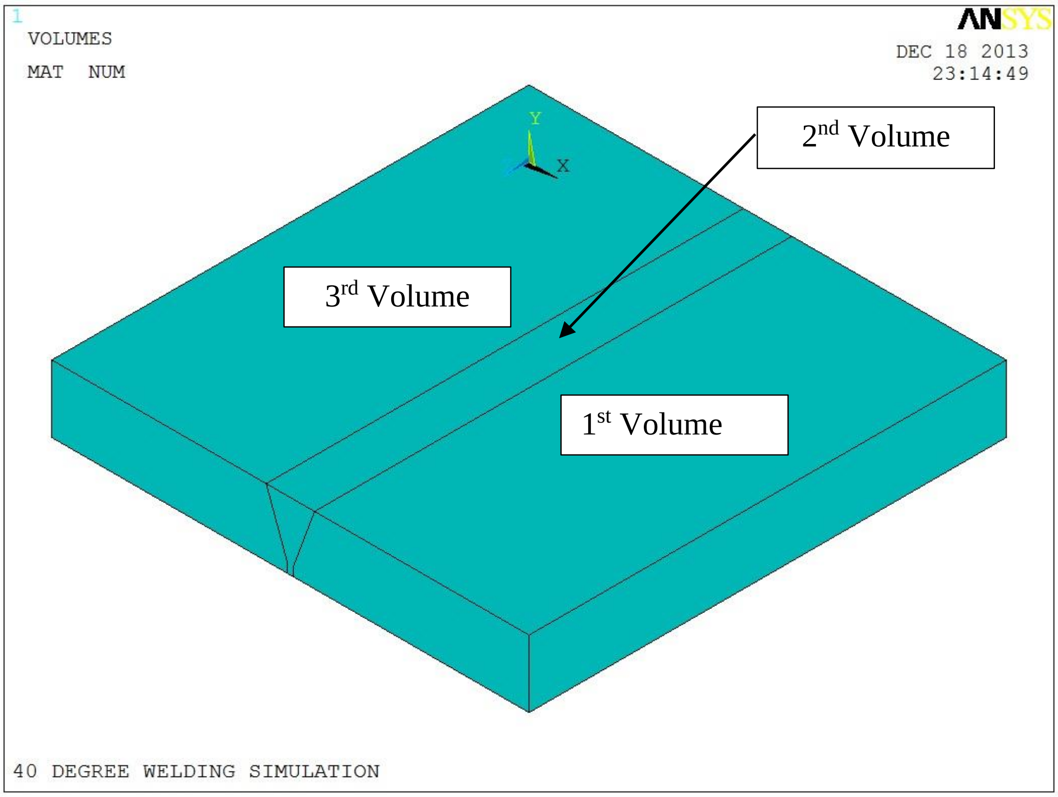


Figure 3.5: Geometry model in ANSYS for 40 Degree of Included Bevel Angle

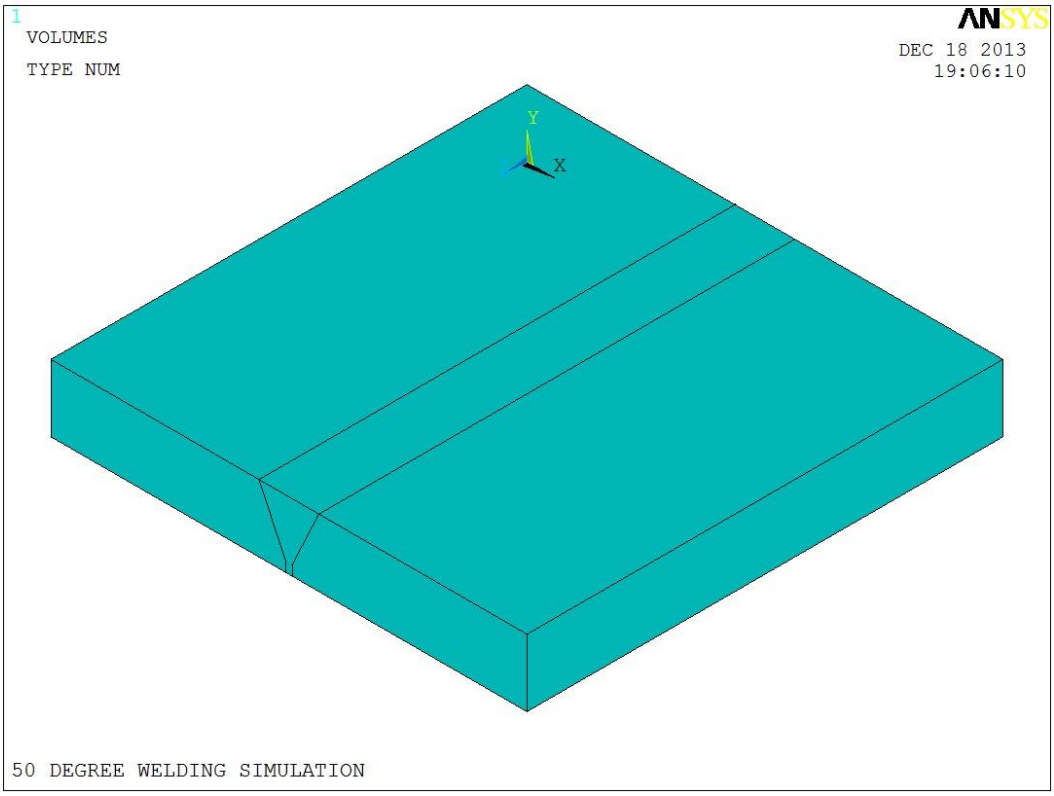


Figure 3.6: Geometry model for 50 Degree of Included Bevel Angle

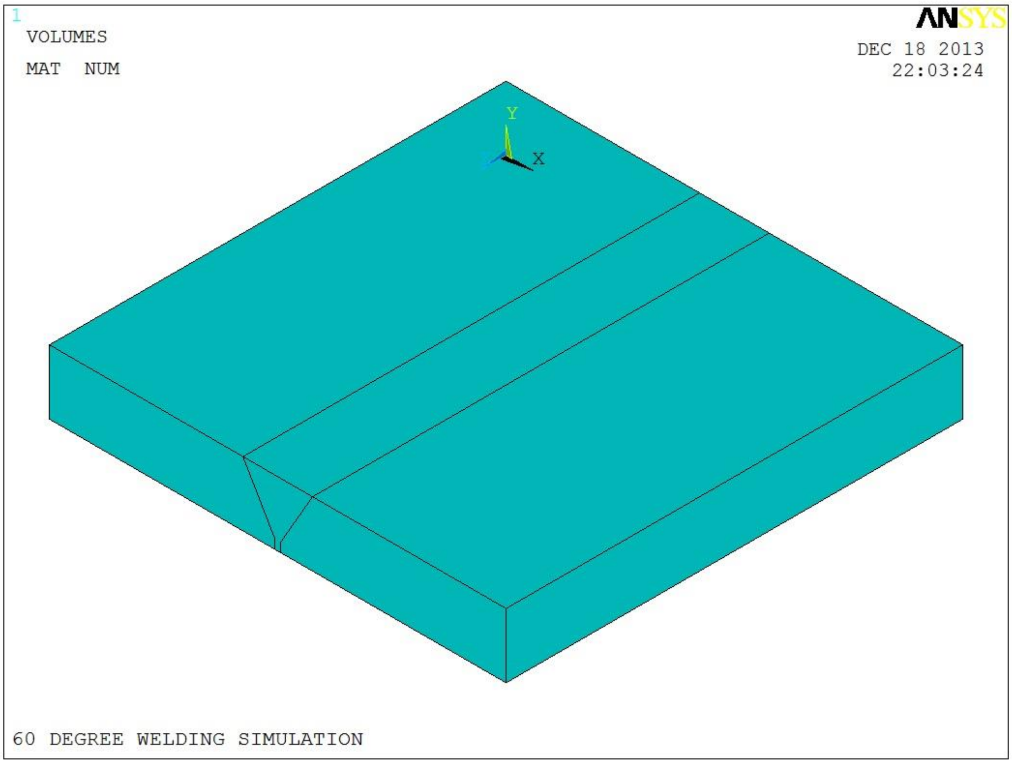


Figure 3.7: Geometry model for 60 Degree of Include bevel angle of a single V butt weld

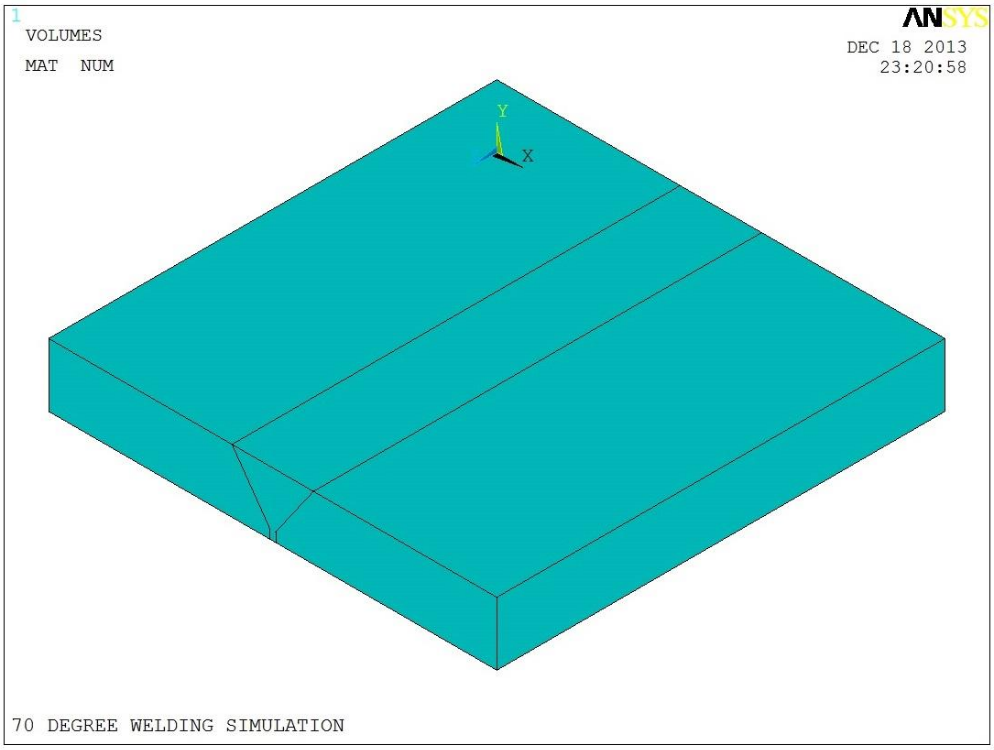


Figure 3.8: Geometry model for 70 Degree of Included bevel angle of a Single V butt weld

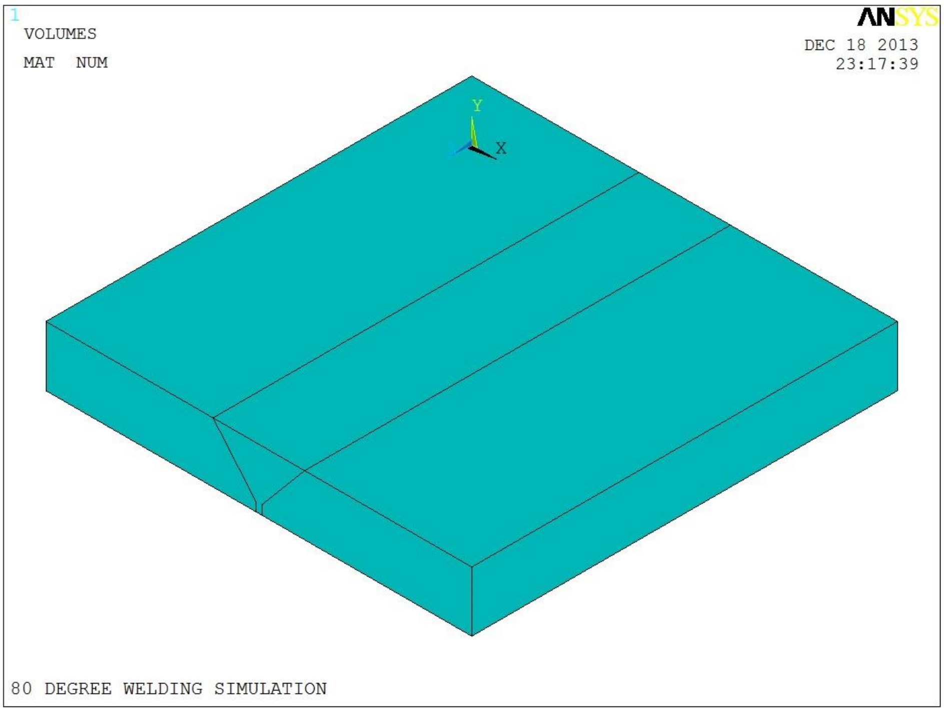


Figure 3.9: Geometry model for 80 Degree of Included bevel angle of a single V butt weld

6. Meshing

Meshing is a method of representing field variables such as displacement by polynomial function that produces a displacement field compatible with applied boundary condition. The size of the finite element mesh has a significant effect on the accuracy of the results and computational cost. Thus, the element size was kept constant for every model. The size of the mesh is depending on the type of analysis. For this analysis, the element size of 0.005 m in y-direction and 0.01 m for z-direction was selected. Bias command was used to set the meshed for x-direction.

GUI method: **ANSYS main Menu > Preprocessor > Meshing > Mesh > Mesh Size**

Meshed Model

To increase the accuracy of analysis, bias mashed has been applied along the filler work piece contact because deformation and stress were critical at that area. Meshed models are shown in Figure 3.10 to Figure 3.14. The number of elements and nodes created from the selected element size were 15900 elements and 5364 nodes in each model.

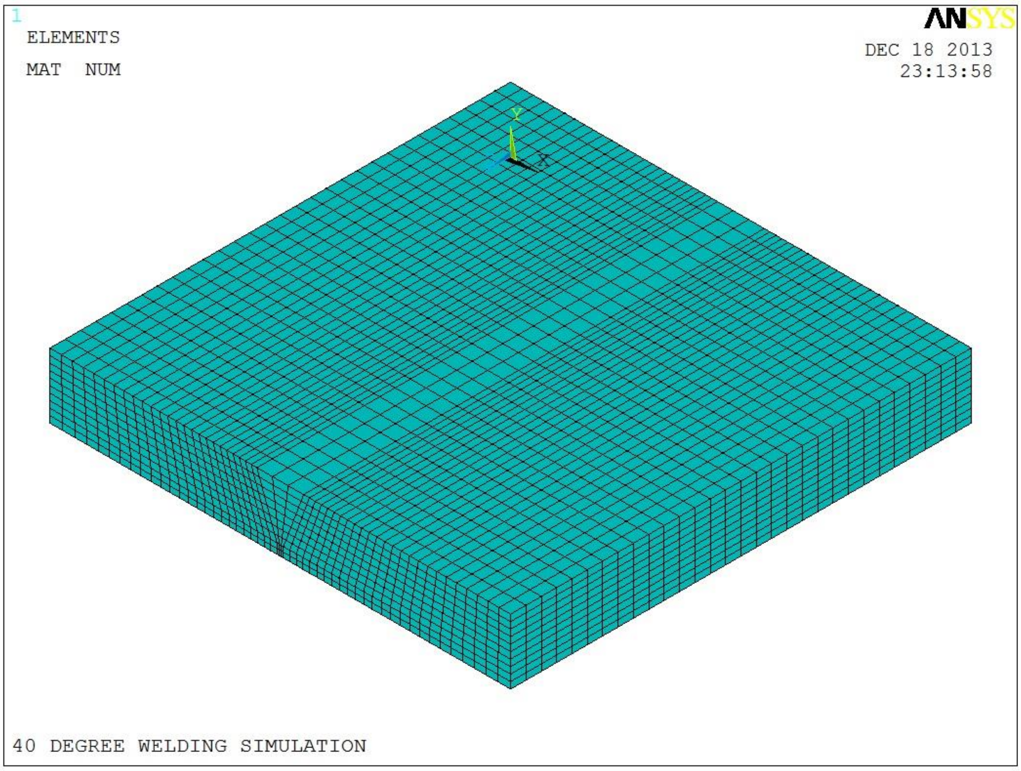


Figure 3.10: Meshed model for 40 Degree of Included bevel angle of a Single V butt weld

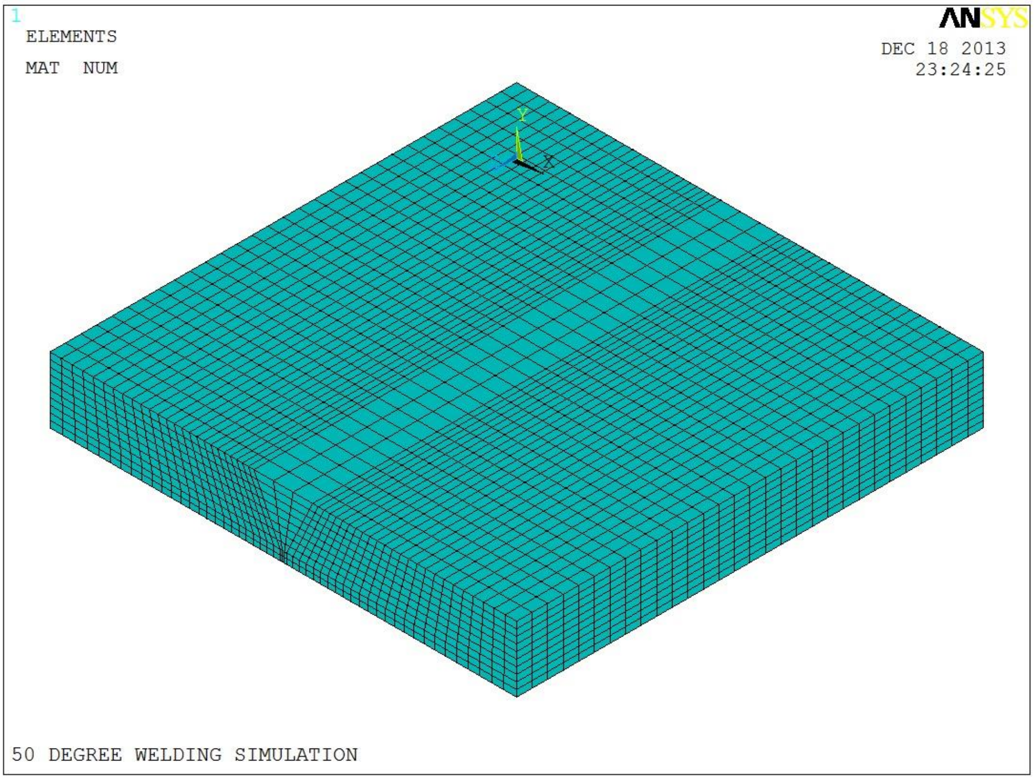


Figure 3.11: Meshed model for 50 Degree of Included bevel angle of a Single V butt weld

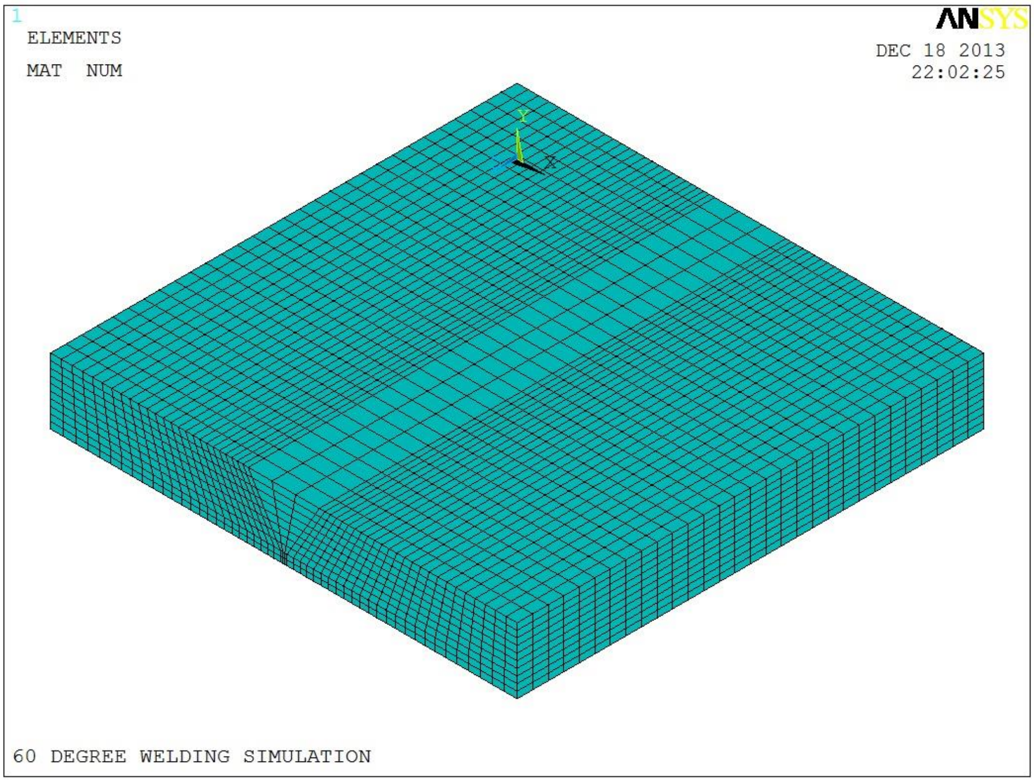


Figure 3.12: Meshed model for 60 Degree of Included bevel angle of a Single V butt weld

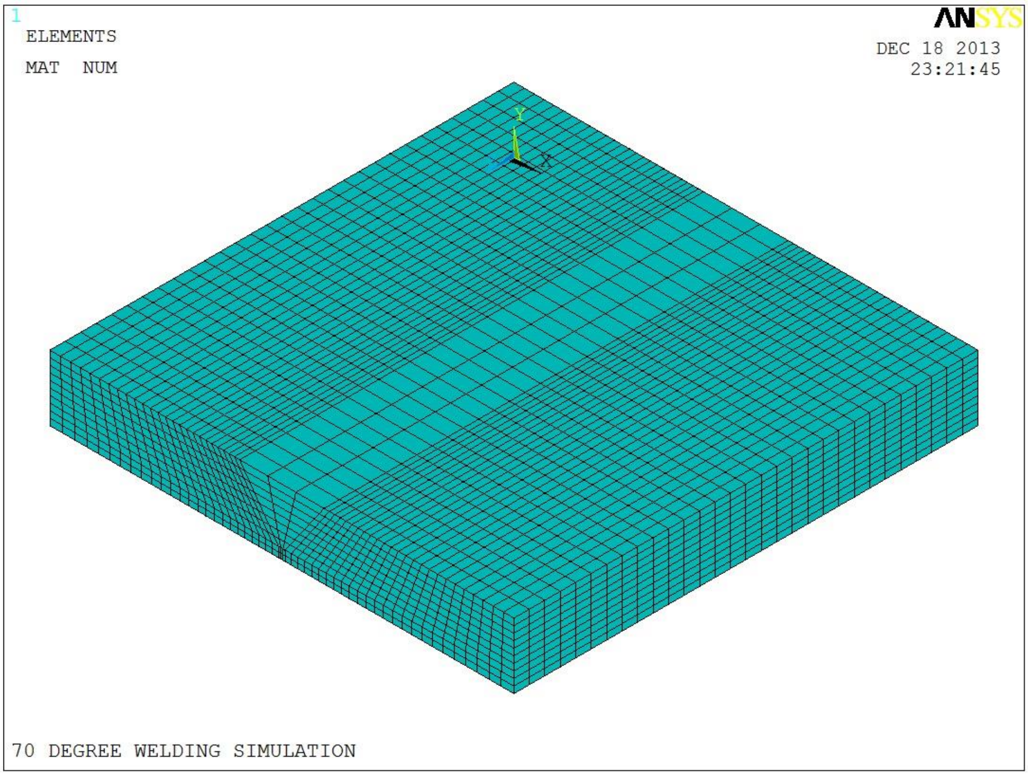


Figure 3.13: Meshed model for 70 Degree of Included bevel angle of a Single V butt weld

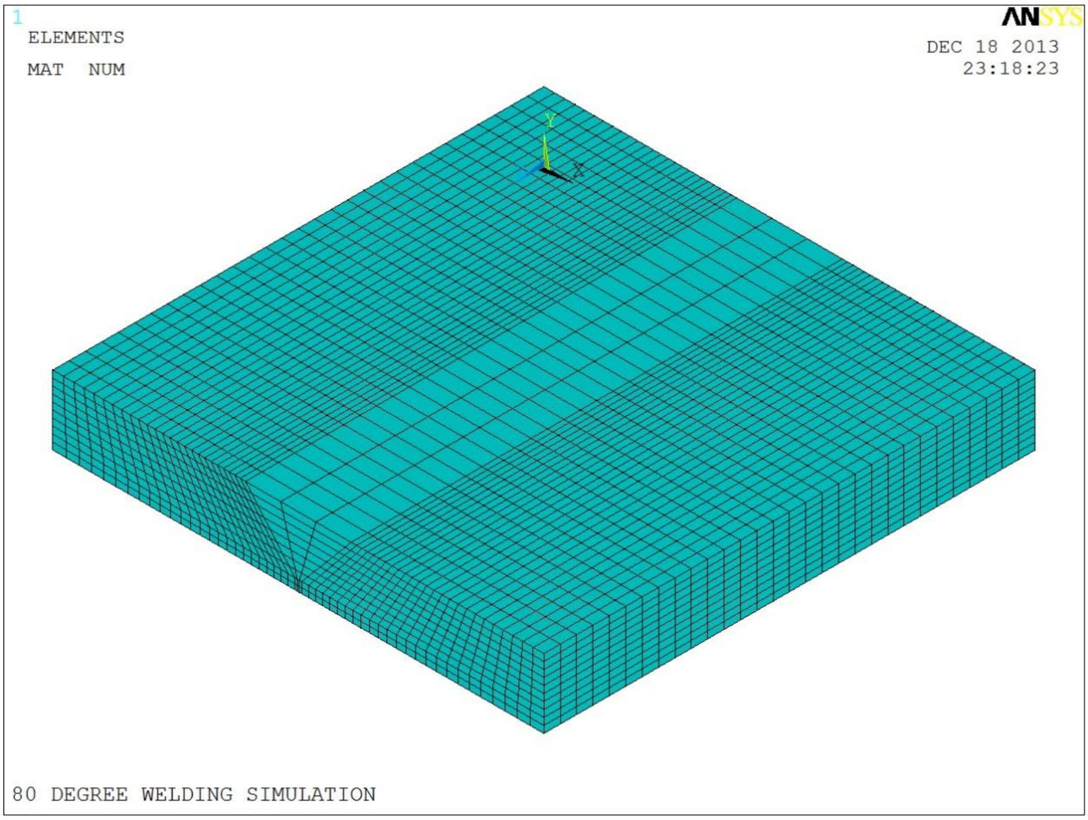


Figure 3.14: Meshed model for 80 Degree of Included bevel angle of a Single V butt weld

7. Analysis Type

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

GUI method: **ANSYS Main Menu** > **Solution** > **Analysis Type** > **New Analysis** > **Transient** > **Full**

8. Turn on the Newton-Rahpson Solver.

Type NROPT, FULL into the command line to apply Newton-Raphson Solver. This step was essential as element killing can only be done when the N-R solver was turned on.

9. Set the solution control

Set the time at the end of the load step to 1 s, and turn off the automatic time stepping. Set the number of load step to 1 in order to ensure the solution was done every 1 second at the end of each load step. Thus the electrode travelling speed can be controlled.

10. Applying boundary condition

The boundary conditions are very important for ANSYS to solve the problems without any error which will affect the results [15]. There were two boundary conditions, namely bulk air temperature and the thermal convection. The bulk air temperature was set to be 27 ^{o}C and the convection heat transfer, was applied on the model [19]. The heat convection was calculated based on Equation (3.1).

$$q = h_c A(T - T_\infty) \tag{3.1}$$

where $h_c = 15 \frac{W}{m^2}$ °C

GUI Method:

Convection:

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

Uniform temperature:

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

11. Define the load input to the model

Heat input during welding simulation was modeled in ANSYS by a distributed heat flux applied on individual elements. The heat was applied through birth and death technique. The birth and death technique was used to deactivate and reactivate selected elements in the analysis in order to simulate the filler material deposition in welding process except for the first element. The amount of heat input was calculated using Equation (3.2).

$$Q = \mu \left(\frac{UI}{vx} \right) \tag{3.2}$$

where:

μ – Arc Efficiency

v – Travel speed of welding

U – Voltage

I - Current

x – Electrode diameter used

For SMAW commonly used were, μ=0.85, v= 5mm/s, U= 24V, I= 150A, respectively while electrode diameter used was $4\sqrt{2}$ mm. Therefore the heat input will be:

$$Q = \frac{\mu(UI)}{vx} = \frac{0.85(24V\ X\ 150A)}{\frac{5mm}{s}\ X\ 4\sqrt{2}mm} = 108e6 \left(\frac{W}{m^2} \right)$$

GUI Method: **ANSYS main Menu** > **Solution** > **Define Loads** > **Apply** > **Thermal** > **Heat Flux**

To control the electrode travel speed, time of load step need to be set in solution control. Load step was set to 1s to ensure the solution was done every 1 second at the end of each load step. On top of that, the initial temperature or ambient temperature was set 27 ^{o}C. The temperature was set to the room temperature. For heat convection, it was applied on the surface of the plate. The value of heat convection was $h_c = 15\frac{W}{m^2}$°C. Figure 3.15 shows the initial condition and heat convection that have been applied to each model of included bevel angle of a single V butt weld.

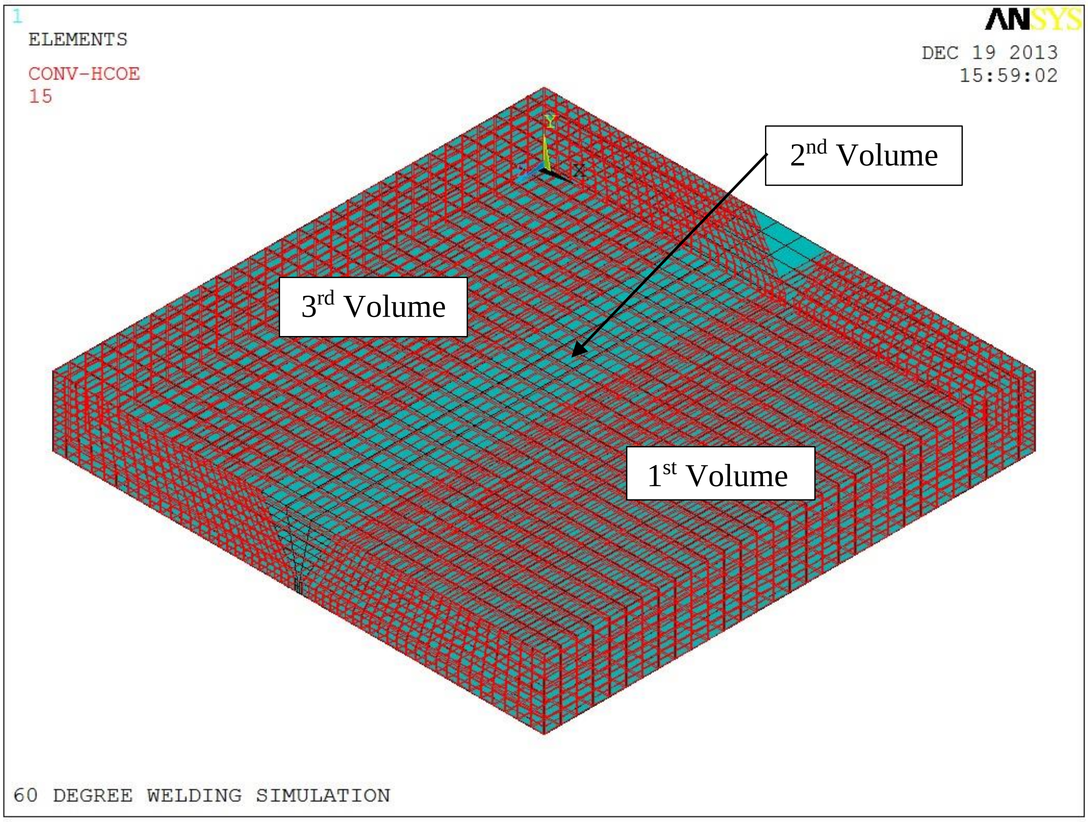


Figure 3.15: Applying the Initial condition and Heat Convection to the model

12. Birth and death techniques

Restart the analysis from previous load step. Set the time at the end of load step 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. When an element was reactivated, its stiffness, mass, element loads, etc. return to their full original values [20]. Solve this load step and repeat this step until the last element on volume 2. Figure 3.16 shows the application of Birth and death technique on the element in volume 2.

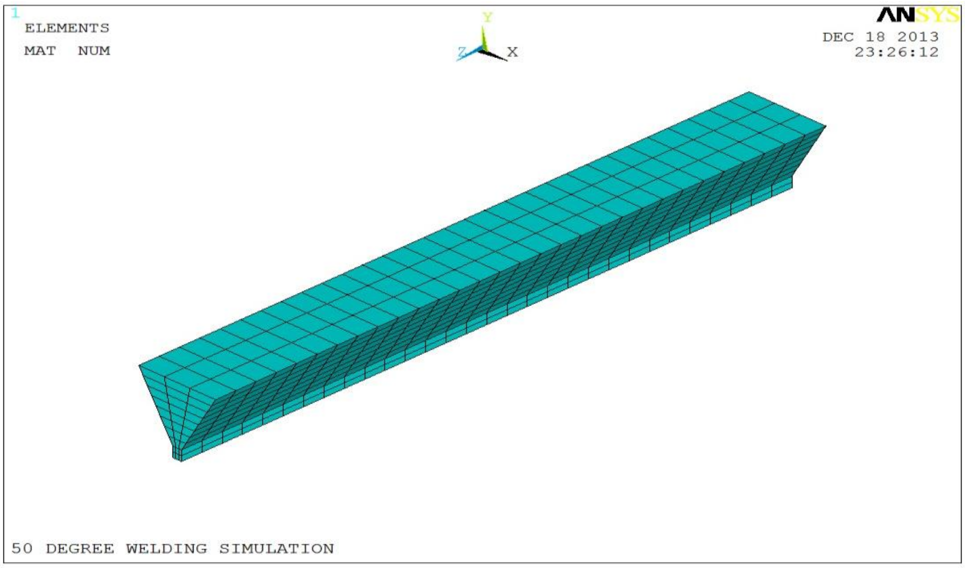


Figure 3.16: Applying Birth and Death technique on the selected element

13. Solve

Solve command was initiated to solve after all the pre-processor command required for the thermal analysis was completed. This analysis was solved by two type of SOLVE command which were solved by Current LS and Increment LS. The total number of step was 30. This indicates 30 steps that need to be solved. Figure 3.17 shows the computation of the thermal analysis simulation.

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

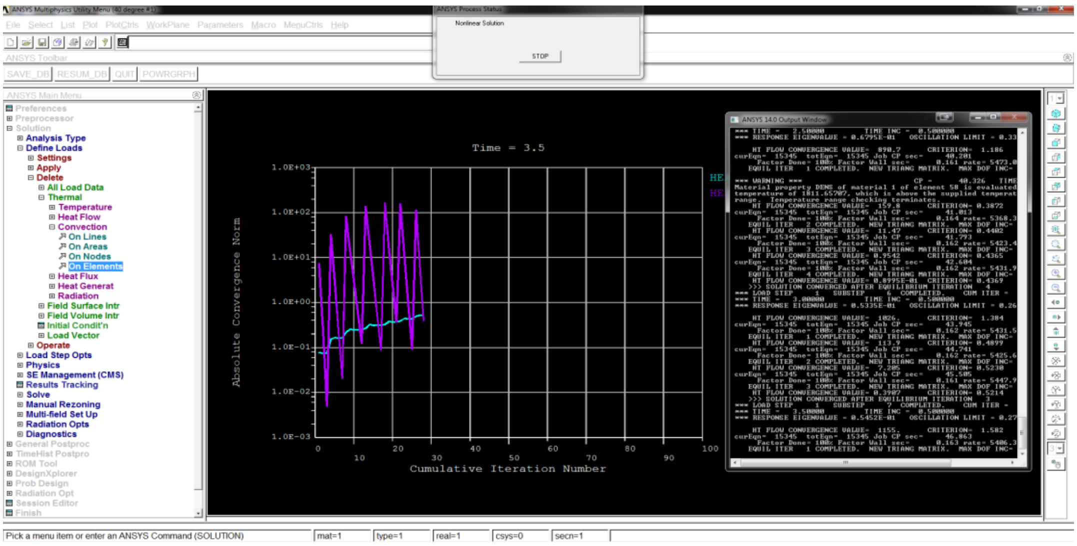


Figure 3.17: Solve the current LS

14. Read Results

The results from the welding simulation will show the temperature distribution along the welding line for different value of included bevel angle. The contour plot will show the temperature distribution and the maximum temperature that can be seen from the contour plot.

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

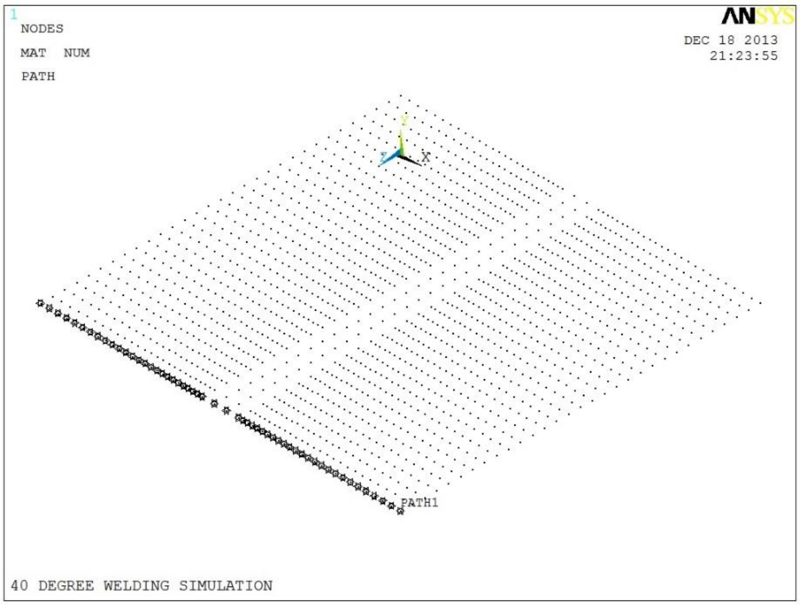


Figure 3.18: Selecting nodes to create graph

There were 54 nodes were selected to get the von Mises stress distribution after welding as shown in Figure 3.18. The value of the stress for each model will be recorded and tabulated in table using Microsoft Excel to help analyzing the trend of the von Mises stress across the weld line.

15. Data Storage

The data from the path created along the welding line was plotted into graph and the data will be saved to be used in the structural analysis. The data will be saved in .rth file. Figure 3.18 shows the selected nodes that will be used to obtain the temperature distribution and von Mises stress. The data will be recorded and plotted in graph.

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

16. Finish

FINISH Command will be issued after all the process in thermal analysis was done. This is crucial in ANSYS before structural analysis.

3.5.2 Structural Analysis

The data from the thermal analysis such as temperature will be used for the structural analysis. Structural analysis was conducted to determine the effects of thermal loads on physical structures of the metal [20]. The results from the analysis were used to verify the structure fitness for use. Some structural properties such as yield strength, tensile strength, Poisson Ratio, Young's modulus and the melting point of the material were required in order to perform the structural analysis.

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

1. Change Element

In order to conduct the structural analysis, the element type must be reliable. The previously used element was SOLID70. This element was only used for thermal analysis. SOLID70 will be replaced with SOLID185. SOLID185 is used for 3-D modeling of solid structures. It is defined by eight nodes having three degrees of freedom at each node, translation in the nodal x, y, and z directions. Figure 3.19 shows the step involved to switch the element type of the model from thermal to structural analysis.

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

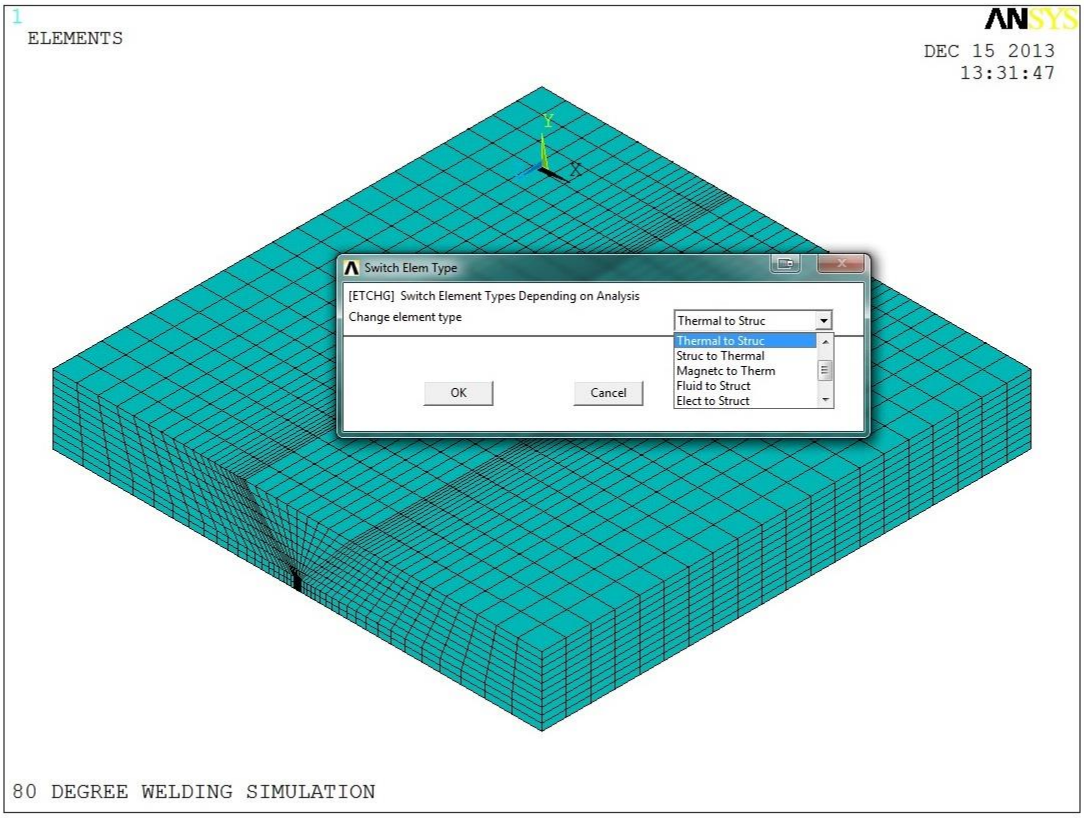


Figure 3.19: Switching the element type of the model

2. Define Properties

Some additional properties were defined in order to perform the structural analysis. The properties are as shown in Table 3.2.

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

3. Specify Initial Conditions

Initial conditions were required in order to obtain better results in the analysis. Both ends of the plate were assumed to be clamped and the displacement was fixed to be zero at the both ends of the plate.

GUI Method:

Displacement:

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

Reference Temperature:

ANSYS Main Menu > Solution > Define Loads > Settings > Reference Temp

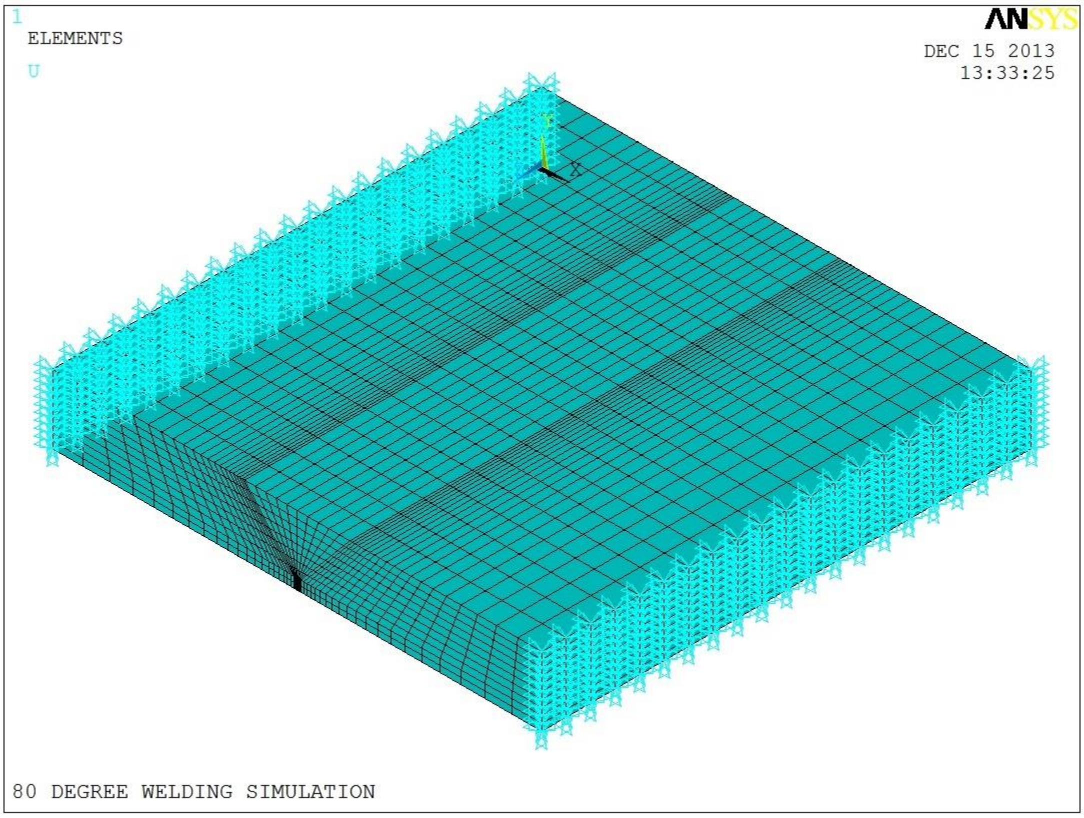


Figure 3.20: Applying zero displacement on the selected nodes

Work piece metals were subjected to constraint at the left and right side area which represented clamping during welding process. Figure 3.20 shows the nodes selected on the right and left side of the plate that were subjected to displacement constraint.

4. Solve

The solve command was initiated after all the temperature profile, parameters and boundary conditions for structural analysis was set up. The temperature from the thermal analysis will be loaded into the structural analysis. The effect on structural properties of the plate affected by the thermal load from the welding process will be evaluated. Figure 3.21 shows the computation for structural analysis.

GUI Method:

Load Temperature:

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

Solve:

ANSYS Main Menu > General Postproc > Path Operations > Archive Path > Retrieve > Path from file

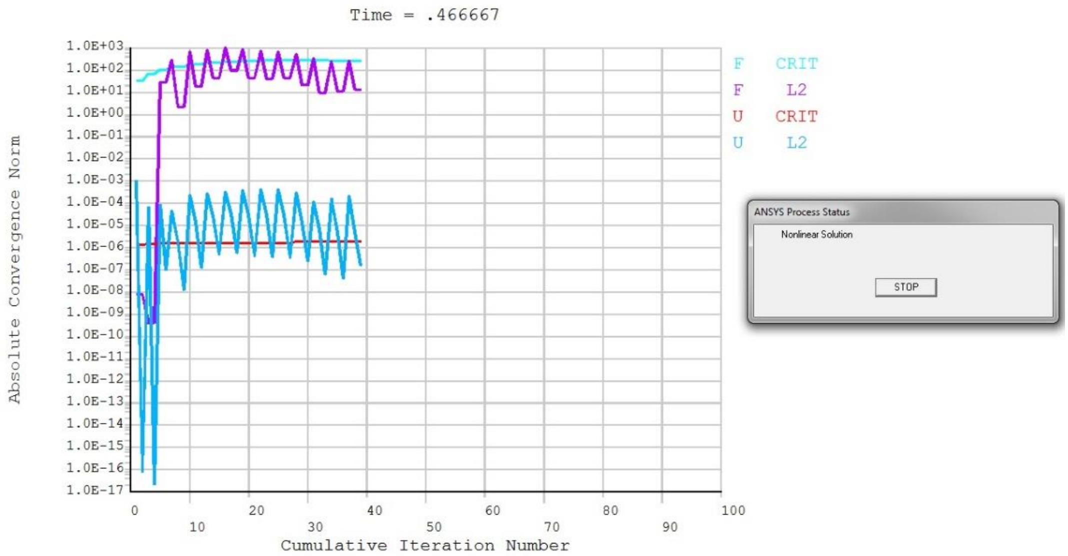


Figure 3.21: Solve current LS

CHAPTER 4

RESULTS AND DISCUSSION

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

4.1 Temperature Distribution

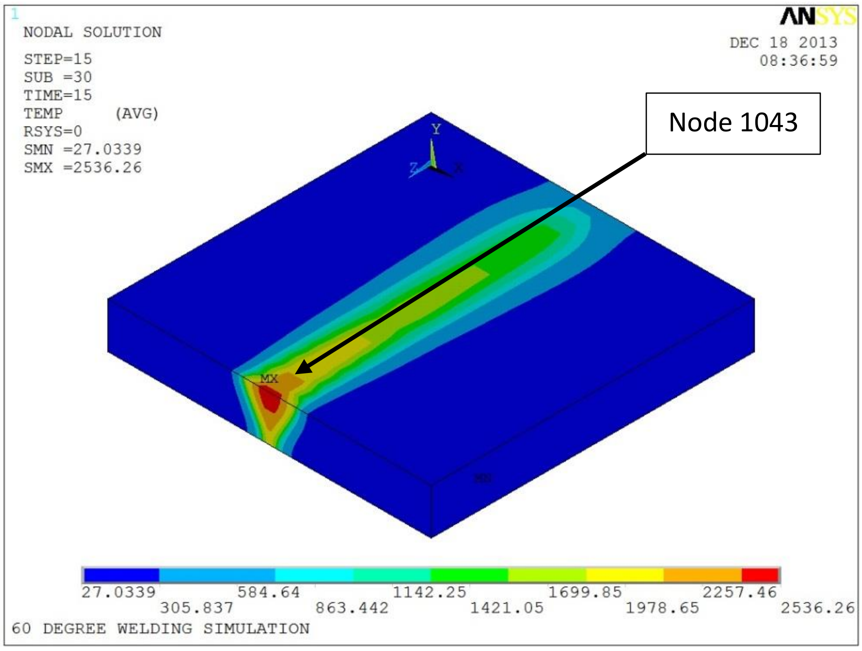


Figure 4.1: Temperature distribution along the welding line for 60 degree model of included bevel angle.

The contour plot in Figure 4.1 shows the temperature distribution on the plate and the maximum temperature can be seen from the contour plot. The heat flux was set to 108E+06 W/m^2 as the heat input to one element per unit area per time. The initial temperature of the plate was set to 27^{o}C and the temperature of the surrounding air was set to 27 ^{o}C with convection coefficient of 15 W/m^2.^{o}C. The maximum temperature shown was 2536 ^{o}C at node 1043 and it exceeded the melting point of the ASTM A36 which was 1500 ^{o}C. The temperature distribution was taken at the last step of welding simulation.

4. 2 Sum of Displacement/Distortion

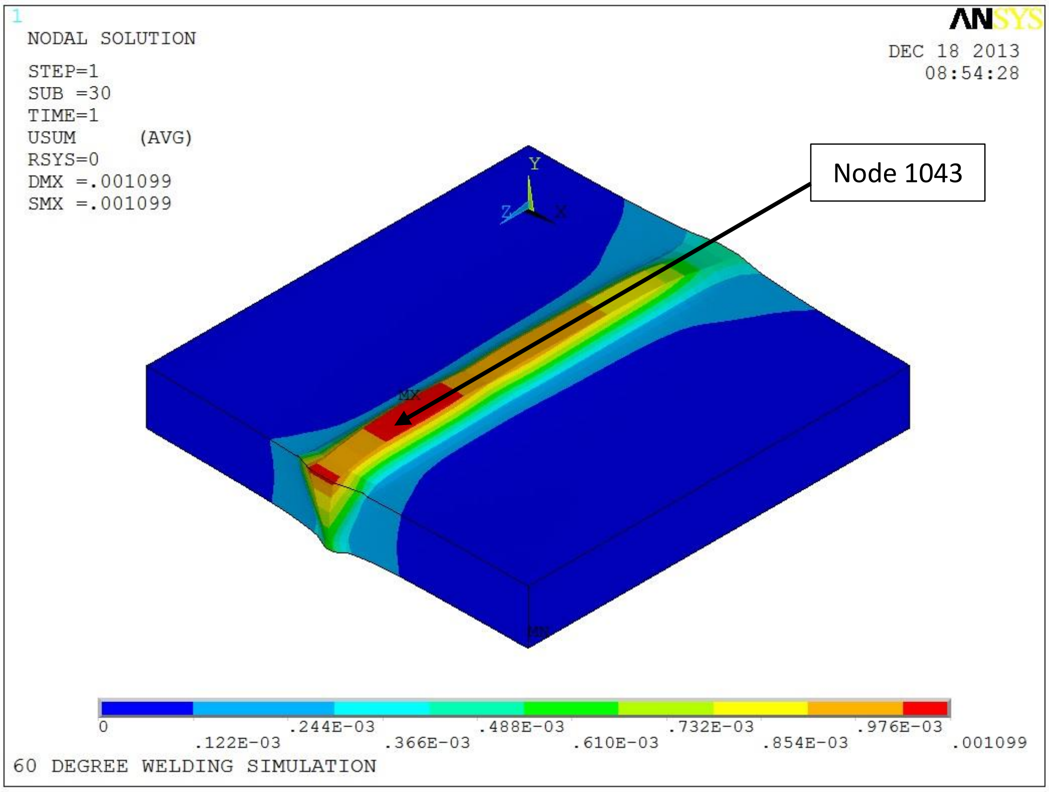


Figure 4.2: Sum of Displacement/Distortion for 60 Degree of Included Bevel Angle of a Single V Butt Weld

From Figure 4.2, maximum displacement occurred at node 1043. The maximum magnitude of the displacement was 1.099 mm. This showed clearly how maximum temperature affects the displacement of the metal. Since higher temperature creates more energy, the metal structure would gain much energy to distort weaker structure around. That was why the distortion was the most at the maximum temperature. Also, the distortion magnitude was affected by the Thermal Coefficient Expansion parameters of the corresponding materials.

4.3 Stress Intensity plot

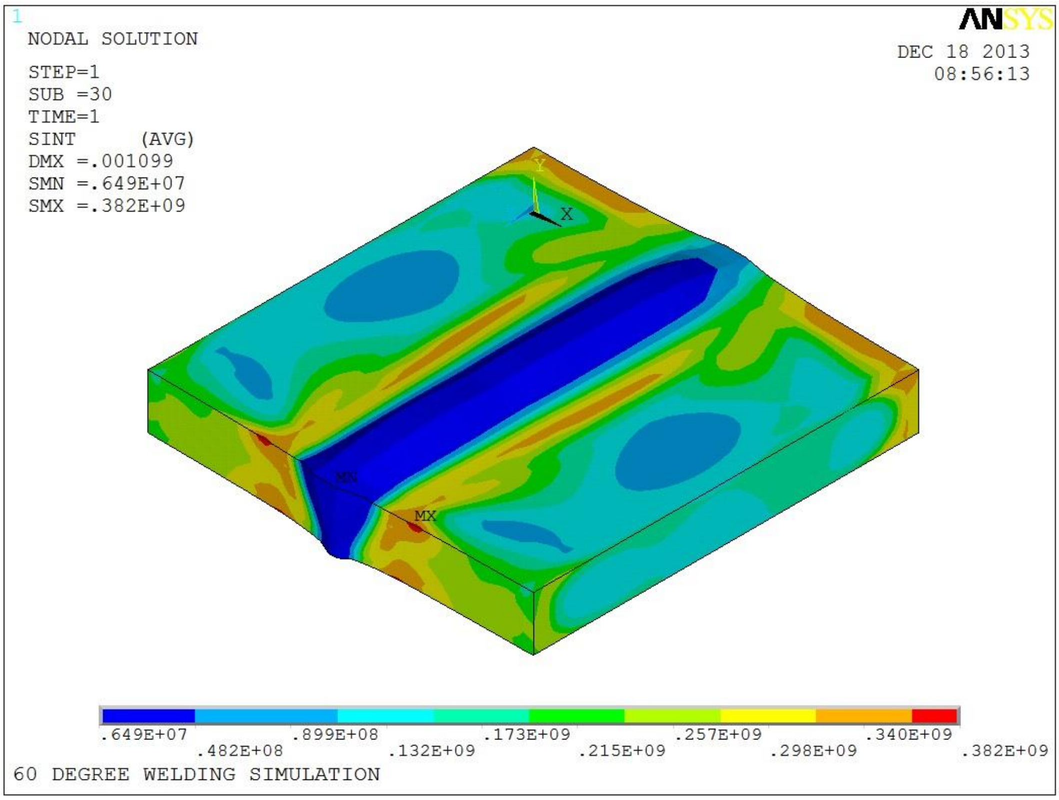


Figure 4.3: Stress Intensity for 60 Degree of Included Bevel Angle of a Single V Butt Weld

Stress intensity plot shown in Figure 4.3 indicated that accumulation of stresses was around the filler metal which was at HAZ region. This was where all the stresses were concentrated. By this plot, stress concentration point was the point where the metal was the weakest. Any micro crack can lead to crack propagation and also corrosion initiation. Thus, extra precautions such as heat treatment could be done on that area to reduce the severity of stress concentration.

4.4 Stress in Z- axis

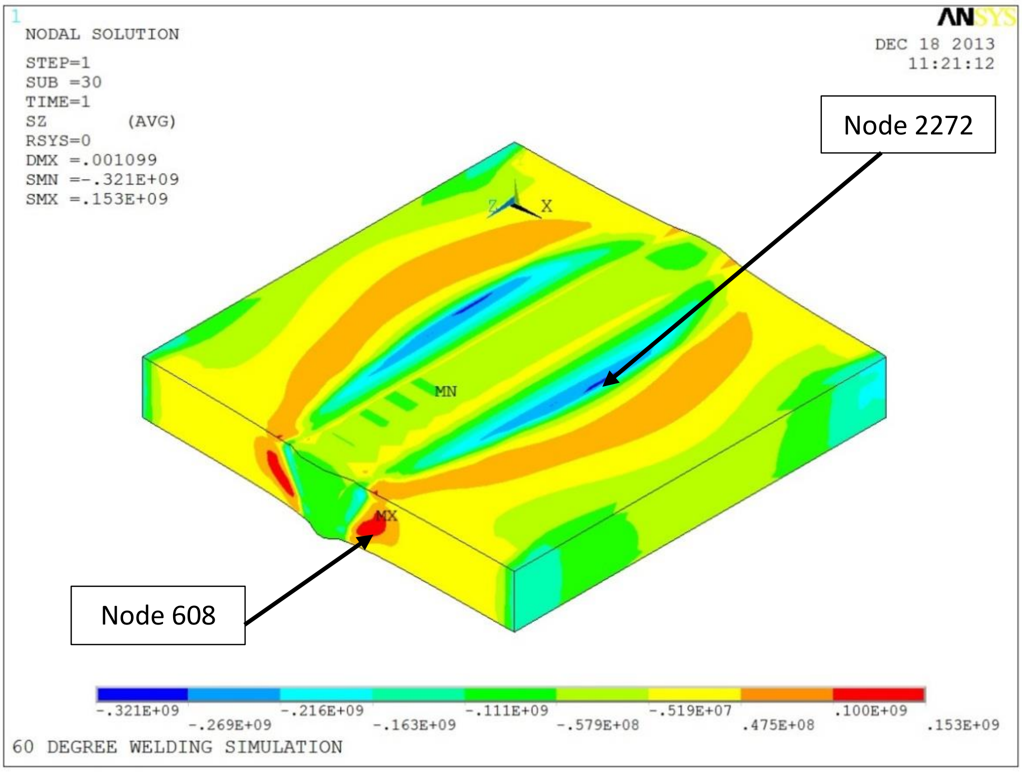


Figure 4.4: Stress in the Z-axis for 60 Degree of Included Bevel Angle of a Single V Butt Weld

Figure 4.4 shows the stresses in Z axis which were the axial stresses acting on the plate. The axial stresses initiated from the tensile stress from the welding line. The maximum axial stress was -321 MPa at node 2272, defined as the compressive stress. The maximum tensile stress plotted was 153 MPa, situated at the region along the welded line which was node 608. Distortion will occur at the point where the stress exceeded the tensile strength. However as the weld plate was going through the cooling process, the axial stress value will be decreased depending on the cooling method.

4.5 von Mises Stress Distribution

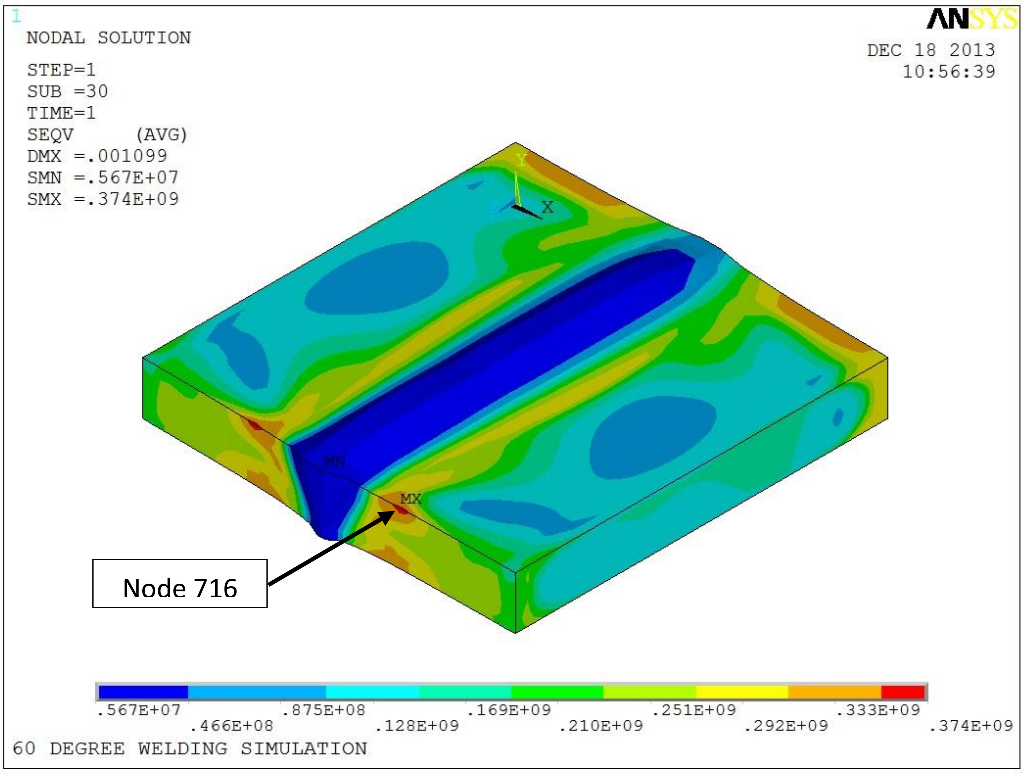


Figure 4.5: von Mises Stress Distribution for 60 Degree of Included Bevel Angle of a Single V Butt Weld

von Mises stress distribution shown in Figure 4.5 was used to predict the yielding or failure of the materials due to the load applied from the welding process. The material will undergo plastic deformation when the von Mises stress exceeded the yield strength which was the critical value. Based on the figure above, the maximum stress plotted was 374 MPa at node 716, while yield strength of the weld plate was 250 MPa. Hence, the area that experienced higher von Mises Stress than the Yield Strength will undergo plastic deformation. The plastic deformation occurred mostly at the HAZ region.

4.6 von Mises total mechanical Strain

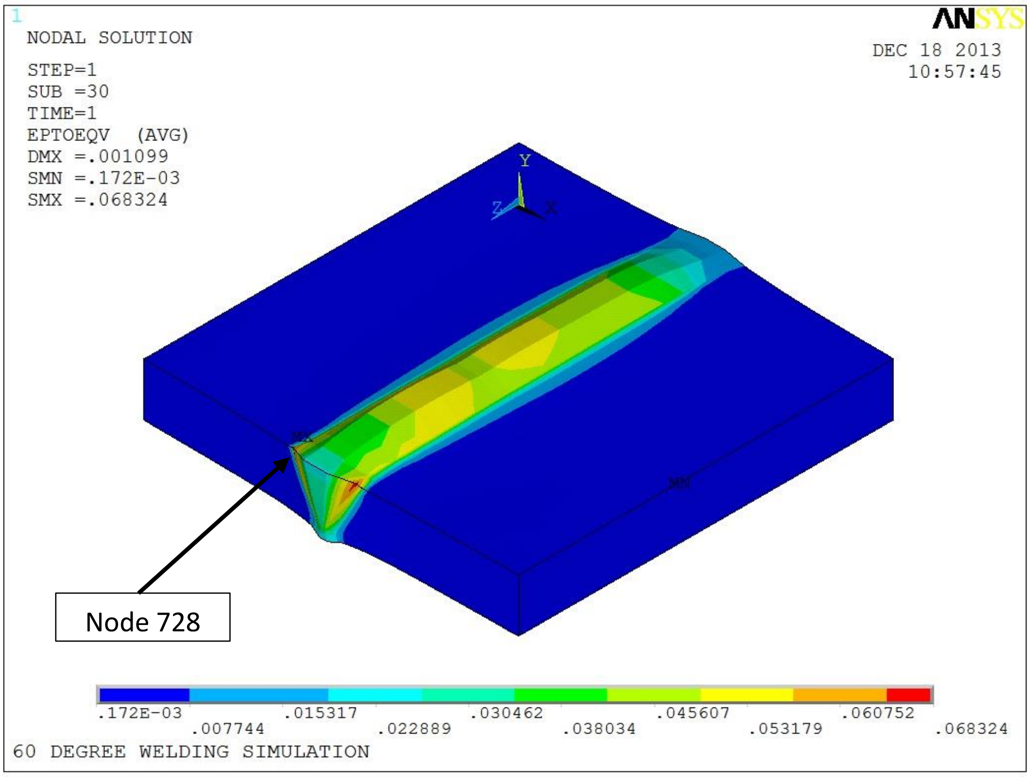


Figure 4.6: von Mises Total Mechanical Strain for 60 Degree of Included Bevel Angle of a Single V Butt Weld

The mechanical strain is shown in the Figure 4.6. This plot is to show the formation of HAZ along the welding line. The maximum strain was 0.068324 m/m which occurred at node 728. Strain indicated that the distortion occurred and based on the stress value, the stress will cause plastic deformation to the metal.

4.7 Deformation

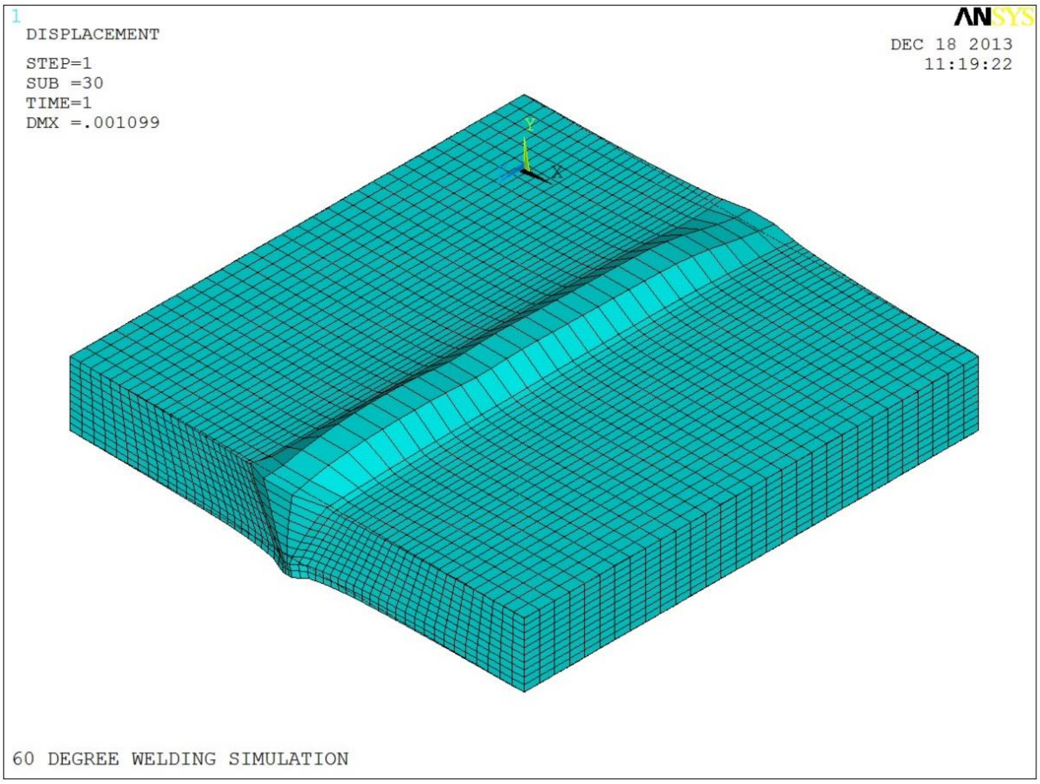


Figure 4.7: Deformation in 60 Degree of Included Bevel Angle model of a Single V Butt Weld

Figure 4.7 illustrates the deformation that occurred after welding process was completed. The welded plate experienced a transverse contraction at the Z and Y-direction. The material has undergone deformation due to high thermal load acting on the surface of the materials.

4.8 Data Results Comparison

Table 4.1: Summary and comparison of stresses from different included bevel angle of a single V butt weld.

Included Bevel Angle (Degree)	80	70	60	50	40
Max. tensile Stress in X Direction (MPa)	114.0	113.0	113.0	113.0	111.0
Max. tensile Stress in Y Direction (MPa)	312.0	270.0	230.0	192.0	158.0
Max. tensile Stress in z Direction (MPa)	154.0	160.0	153.0	150.0	181.0
Maximum Stress Intensity (MPa)	375.0	376.0	382.0	383.0	384.0
Von Mises Stress (MPa)	360.0	360.0	374.0	374.0	375.0
Von Mises Mechanical Strain (m/m)	0.0633	0.0663	0.0683	0.0687	0.0690
Maximum Temperature (Celcius)	2774	2652	2536	2429	2338
Maximum Distortion (mm)	1.209	1.143	1.099	1.074	1.024

Table 4.1 shows the value of residual stresses, maximum temperature profile and maximum distortion at the different included bevel angle model of a single V butt weld.

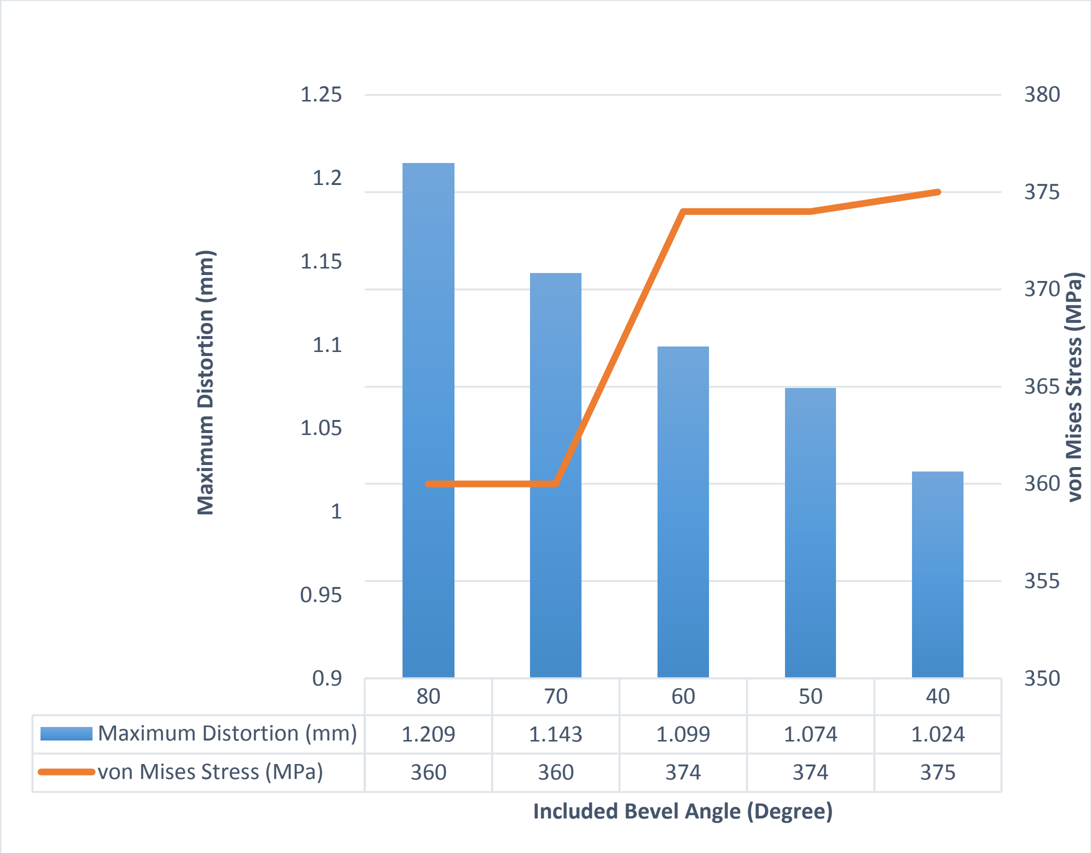


Figure 4.8: Maximum Distortion and Maximum von Mises stress versus Included Bevel Angle

Figure 4.8 shows the maximum distortion which occurred on each of included bevel angle and the maximum von Mises stress. Based on the tabulated data, it can be shown that the von Mises stress is inversely proportional to distortion and the larger the included bevel angle the smaller the value of von Mises stress occurred on the weld plate. The decrease in the value of von Mises stress may be due to the decrease in the exposed area of deposited weld metal to the surrounding. In other words, the decrease in the exposed area decrease the cooling rate of weld metal, which increased the value of von Mises stress indirectly. In contrast, the larger the included bevel angle the larger the distortion occurred on the welded plate. The 80 degree model of included bevel angle experienced the largest maximum distortion which was 1.209 mm. This was due to the high temperature that had been applied during the welding process as 80 degree model of included bevel angle had maximum temperature of 2774 ^{o}C.

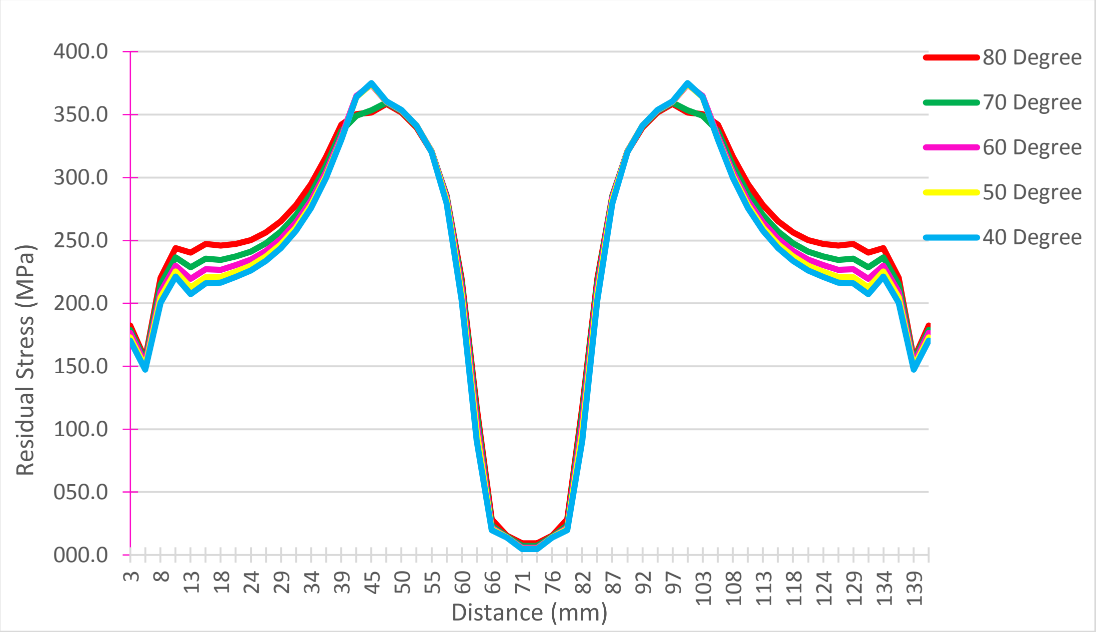


Figure 4.9: The line graph of the maximum von Mises Stress against the transverse distance along the weld plate.

Figure 4.9 shows the line graph of the von Mises stresses at selected nodes for each model of included bevel angle. The coordinates of the nodes were the same for each model and the stress values were taken right after the welding process was finished. Based on the graph, the von Mises Stress values increased as it's distance is nearer to the weld line. The model of 40 degree of included bevel angle experienced the maximum von Mises stress of 375 MPa at node 716. Meanwhile 80 and 70 degree model of included bevel angle experienced the smallest maximum von Mises Stress which was 360 MPa at the same node. Hence, it can be deduced that the larger the included angle the smaller the value of von Mises Stress.

4.9 Validation

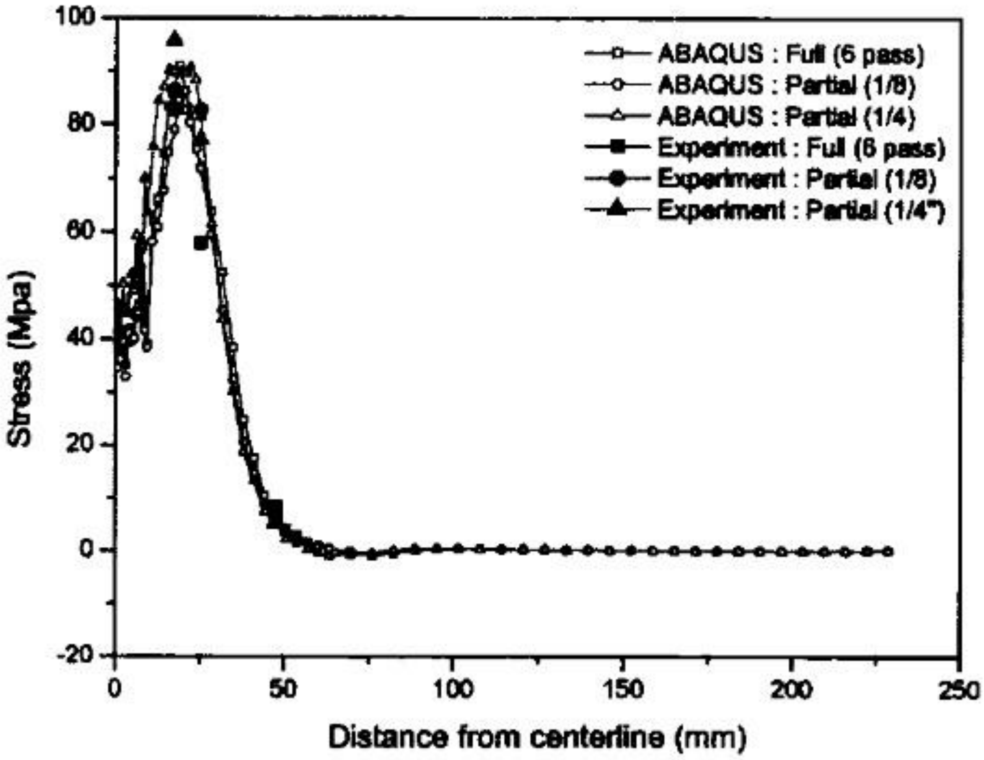


Figure 4.10: Transverse residual stress distribution at the top surface [21].

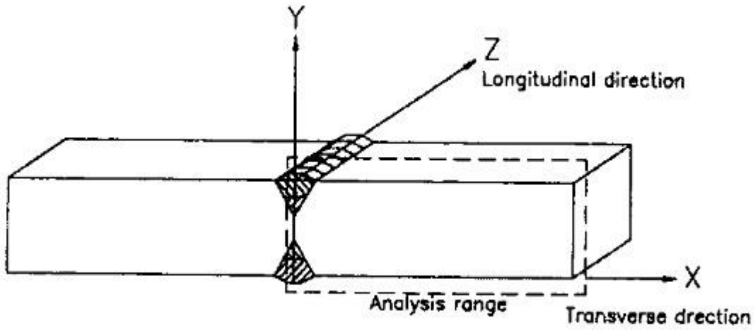


Figure 4.11: The analysis range of the weld plate [21].

Figure 4.10 shows the line graph of the transverse residual stress distribution at the top surface of the weld plate. The objective of this paper was to analyze the residual stress on the thick metal plates using the partial penetration welding method. This study used ABAQUS as the simulation software and its results were compared to the experimental results. The thickness of the plate was 25.4 mm and the material used was the same as the current project which was ASTM A36 low carbon steel. Figure

4.11 shows the analysis range of the weld plate for this project. But, the type and geometry of the butt weld joint was completely different. This project used double V butt joint and it did not apply the root face and root gap for the weld preparation as the current project. The welding method used was Gas Metal Arc Welding (GMAW), thus the heat input value will be different, leading to different values of transverse residual stress as the current project. However, based on the results obtained, its pattern for the transverse residual stress distribution was approximately the same as the von Mises stress distribution pattern for the current project. In addition, the thick plate was welded by welding base material at room temperature without preheating and cooling down to room temperature which made it to have the same procedure as the current project.

4.10 Discussion

4.10.1 Temperature Distribution

Based on maximum temperature distribution tabulated in Table 4.1, it can be seen that the 80 degree model of included bevel angle experienced the largest temperature which was 2774 ^{o}C. Meanwhile the 40 degree model of included bevel angle experienced the smallest temperature which was 2338 ^{o}C. Thus it can be concluded that the larger the included bevel angle of a single V butt weld the larger the temperature will be experienced by the model. Besides the maximum temperature was located at the centre of the heat flux, which in the real case was the arc produced by the electrode. The temperature far from the welded area was maintained at 27 ^{o}C.

The temperature distribution at HAZ area can explain the behavior changes of the microstructure and properties of the metal. The microstructure and properties changed because of the heat applied in the welding process. The minimum temperature value at the first node in volume 2 was increasing due the heat conduction in the metal itself that transfer the heat from the hotter area to cooler area.

4.10.2 Residual Stress and Distortion/ Displacement

Referring to Table 4.1, we can conclude that 80 degree model of included bevel angle experienced the smallest maximum stress intensity and von Mises stress compared to the other four models. Most of this stress was accumulated at the HAZ region. Besides, this model had undergone the highest maximum temperature, 2274 ^{o}C when the welding activity was performed which caused distortion and expansion to occur.

The distortion occurred most at the welded line. The distortion indicates a stress accumulation that exhibits a constant movement towards outer the surface of the metal plate. The distortions were most likely to form the weld beads that were accumulated at the top of the welded line. The distortion was not too severe with a value of 0.001209 m, which was around 1 mm only. A proper surface finishing and machining across the surface will remove the distorted metal at the upper layer of the welded line. However, when considering the required volume for filler metal, 80 degree model will be undesirable as it consumed more filler metal and it will become uneconomic to the fabricator.

On the other hand, 40 degree model of included bevel angle experienced the smallest distortion, 1.024 mm located at the welded line which was better for weld integrity. Meanwhile its residual stresses gave the highest value of maximum von Mises stress and stress intensity of 375 MPa and 384 MPa, respectively. Although, 40 degree model of included angle had smallest distortion and required volume for filler metal, it still produced highest residual stresses. It may be due to the less exposed area of deposited weld to the surrounding which will decrease the cooling rate of weld metal. This will cause the increase in the value of von Mises stress. However the values of residual stresses were approximately the same as 50 and 60 degree model of the included bevel angle. And when considering the distortion and the volume of required filler metal for both 50 and 60 degree model, the 40 degree model was found to be more superior. Thus, the most desirable included bevel angle was 40 degree model as it has smallest distortion and required volume for filler metal but residual stress was slightly higher. The 40 degree model was proven to be most desirable for producing high integrity single V butt joint on thick plates based on the data collected from the simulation.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

Finite element models of welding process on thick plates with different included bevel angle of a single V butt weld had been successfully developed. It had been shown that the included bevel angle for a single V butt weld do affect the integrity of the weld joint by influencing the residual stress, distortion and temperature profile when conducting welding.

From the results, it was shown that the value of residual stresses increased as the included bevel angle for a single V butt weld decreased. There were several stresses that remain on the metal plate that caused the changed in mechanical properties of the metal plate. On the other hand, the values of distortion increased as the included bevel angle for a single V butt weld increased. The values were obtained from the contour plot of the simulation. Thus, the most desirable included bevel angle was 40 degree model as it has smallest distortion and required volume for filler metal but had slight high of residual stress.

5.2 Recommendation

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

In order to save the time consumed for the simulation, the use of command script in ANSYS software is recommended.

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

Next, the filler metal can be treated to not having the same material with the work piece. In real welding, the filler metal or electrode has different material properties to the work piece and usually its melting temperature is lower than the work piece. Thus, a more accurate results can be yielded in thermal and structural analysis.

Lastly, in order to get more accurate result, the heat flux can be treated as non-constant input. In actual welding activity the electrode generated heat inconstantly. The heat input can be modelled by using Rosenthal equation. This equation can be employed in the ANSYS software, and the heat input will be applied as per equation used.

REFERENCES

[1] Krebs J, Kassner, “Influence of welding residual stresses on fatigue design of welded joints and components”, *Welding in the World*; 51:54–68. 2007.

[2] Tso-Liang T, Chih-Cheng L., “Effect of welding conditions on residual stresses due to butt welds”. *International Journal of Pressure Vessels and Piping*; 75:857–64. 1998.

[3] J.Farrell. (2007). “Advantages and Disadvantages of Welded”. 4mechtech [Online]. Available: http://4mechtech.blogspot.com/2013/01/advantages-and-disadvantages-of-welded.html

[4] Shim Y, Feng Z, Lee S, Kim D, Jaeger J, Papritan JC, Tsai CL., Determination of residual stresses in thick-section weldments”. *Welding Journal* pp.305–12. September 1992.

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

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

[7] Glinka G., “Residual stresses in fatigue and fracture: theoretical analyses and experiments”, *in Niku-Lari A, Residual Stresses Generation*, UK: Pergamon Press, 1986.

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

[9] Leggat, R.H., “Residual stress in welded structures”. *Int. J. Pressure Vessel Piping* vol.85 pp. 144-151, 2008.

[10] Sean J., (2012, Aug.). Shield Metal Arc Welding (SMAW), *Welding Process Weld Design*, Available:
http://www.corrosionist.com/Shielded_metal_arc-welding_(SMAW).htm

[11] Berge S., "On the effect of plate thickness in fatigue of welds", *Engineering Fracture Mechanic,* vol. 21(2), pp. 423–35, 1985.

[12] Nguyen TN, Wahab MA., "The effect of weld geometry and residual stresses on the fatigue of welded joints under combined loading", *J Mater Process Technology,* vol. 77, pp. 201–8, 1998.

[13] G. Mahendramani., "Effect of included angle in V-groove butt joints on shrinkages in Submerged Arc welding process", *Int. journal of Eng. Science and technology (IJEST),* vol. 14, April 2012.

[14] American Welding Society (AWS), standard Welding Preparation for Single V Butt Weld, AWS D1.1: Structural Welding Code- Steel [Online]. Available:
https://law.resource.org/pub/us/cfr/ibr/003/aws.d1.1.2000.pdf

[15] Ali Moarrefzadeh, "Finite Element Simulation for Thermal Profile in Shield Metal Arc Welding (SMAW) Process", *International Journal of Emerging Trends in Engineering and Development*, vol. 14, pp. 150-35, 2012

[16] David Roylance, "Finite Element Analysis", *Department of Materials Science and Engineering*, Massachusetts Institute of Technology, Cambridge, 2001.

[17] D. Gary, Temperature Dependent Thermal, Physical and Mechanical, *Carbon Steel Handbook*, 2007.

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

[19] A.Capriccioli, P. Frosi, “Multipurpose ANSYS FE Procedure for Welding Process Simulation”, *Fusion Engineering Design*, vol. 84, pp. 546-553, 2009.

[20] Saeed Moaveni., “Finite Element Analysis”, *Theory and Application with ANSYS*, Minnesota State University, Mankato, 1999.

[21] K. Seok, Y.L. Shim and J.I. Song, “Residual Stress and Fracture Analysis of Thick Plate for Partial Penetration Multi Pass Weldment”, *KSME International Journal*, vol. 16 No. 9, pp. 1039, 2002.