THERMO – MECHANICAL MODELING OF FRICTION STIR WELDED PLATE

by

MOHD HANIS B MOHD DAUD

8511

Dissertation submitted in partial fulfilment of

the requirements for the

Bachelor of Engineering (Hons)

(Mechanical Engineering)

JANUARY 2010

UNIVERSITI TEKNOLOGI PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

THERMO-MECHANICAL MODELING OF FRICTION STIR WELDED PLATE

by

MOHD HANIS B MOHD DAUD
8511

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

JANUARY 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.

____

MOHD HANIS BIN MOHD DAUD

ABSTRACT

In 1991, a new welding method that is potential to overtake conventional welding invented in The Welding Institute (TWI). It was called Friction Stir Welding (FSW), a solid state joining process where the metal is not melted during the process and is used for applications where original metal characteristics must remain unchanged as far as possible. The process is most suitable for components which are flat and long (plates and sheets) but can be adapted for pipes, hollow sections and positional welding. FSW has received a worldwide attention since it invention. The FSW process has been used in many fields such as aerospace, railway, automobile and shipbuilding industries. This present work is aimed to simulate the Friction Stir Welding process as a two dimensional model. A finite element technique is employed, within the context of a general purpose Finite Element Method (FEM) framework, to provide the temperature distributions involved in the welded joints. The modeling is required to optimize the welding process and condition as the actual process of the FSW is rather costly and we cannot measure the exact temperature of the process. Thermal modeling for the FSW is needed to predict the temperature distribution and field of the work piece. The scope of study in this project include to define boundary condition of the material and tool of FSW process, data gathering of the material and tool and analysis of the result of the simulation. The methodology in doing this study is derived in a flow chart and the steps are data collection, data analysis, designing, develop model, simulation and temperature analysis. The finding and data of this project may be use to predict the microstructure, properties, distortion and residual stress of the FSW

ACKNOWLEDGEMENT

First and foremost, all praise to Allah S.W.T for granting me the opportunity to complete this final year project, which has proven to be a very enriching experience.

Throughout the completion of this Final Year Project (FYP), many people have provided a great deal of support, guidance, advice, assistance and idea to me. Without their help this FYP would not be that meaningful and successful. I would like to take this opportunity to appreciate all who had contributed in my learning and success.

First of all I would like to express my gratefulness to Universiti Teknologi PETRONAS and Mechanical Engineering Department for providing us a wide range of facilities and giving us the opportunity to learn and broaden our knowledge by utilizing these facilities. My highest and heartiest appreciation is for my supervisor, Dr. Mokhtar B Awang for providing me with guideline and helped me to achieve the project objectives. His contribution on this project is invaluable. Certainly this project wouldn't be a success without his priceless advices and sincere attention

I would also like to thanks all the Post Graduate students of Mechanical Engineering Department especially Mr. Omar and Mr. Julendra who have been very cooperative in helping and supportive throughout the simulation work. They have spent numerous hours helping me from the starting phase till the testing phase of this project. Without their contribution, the completion of this project seems to be impossible.

Here I would also like to thanks to my parents for their support, their encouragement and understanding until the completion of this project.

TABLE OF CONTENTS

ABSTRACT			**iv**
ACKNOWLEDGEMENT			**v**
CHAPTER 1:	**INTRODUCTION**		1
	1.1	Background of study	1
	1.2	Problem statement	4
	1.3	Objectives and scope of study	4
	1.4	Feasibility of the project.	5
CHAPTER 2:	**LITERATURE REVIEW**		**6**
	2.1	Basics of numerical modelling	6
	2.2	Heat generation in friction stir welding	7
	2.3	Simulation model	10
CHAPTER 3:	**METHODOLOGY**		**13**
	3.1	Procedure identification	13
	3.2	Finite element modeling	14
	3.3	Gantt Chart	19
	3.4	Tool/equipment/software required	19
CHAPTER 4:	**RESULT AND DISCUSSION**		**20**
	4.1	Meshing of the geometry	20
	4.2	Simulation result	21

CHAPTER 5:	**CONCLUSIONS . . . .**	**28**
	5.1 Conclusions	28
	5.2 Recommendation . . .	28
REFERENCES	. . . .	30
APPENDICES	. . . .	33

LIST OF FIGURES

Figure 1: Schematic drawing of the friction stir welding 2

Figure 2: Methodology for completion of the project 13

Figure 3: PLANE55 Geometry 16

Figure 4: PLANE182 geometry 17

Figure 5: Meshed model of work piece 20

Figure 6: Temperature distribution across the work piece 22

Figure 7**:** Comparison of temperature distribution produced with Song and Kovacevic[12] model 24

Figure 8: Temperature in different zones of the FSW joint in as weld condition 25

Figure 9: Temperatures in different zones of the FSW 25

Figure 10: Von Mises stress distribution 26

Figure 11: Stress versus distance 27

LIST OF TABLES

Table 1: Properties of AA 6061-T6 14

Table 2: Properties of H13 tool steel 15

Table 3: Dimensions of the tool and the plates 15

Table 4: Operating condition of the model 18

CHAPTER 1
INTRODUCTION

1.1 Background of study

Friction Stir Welding (FSW) was invented and patented by Thomas et al of The Welding Institute in Cambridge, UK in Dec 1991[1]. The concept of FSW is a nonconsumable rotating tool (shoulder with profiled pin) is rotated and slowly plunged into the joint line between two pieces of sheet or plate material. The workpiece is clamped onto backing plate to prevent the abutting joint faces from being force apart. The rotating of tool will create advancing and retreating side along the seam that is formed and the softened and heated materials flow around pin to it backside where the material is consolidate to create high quality solid state weld. The shoulder of tool prevent the material from being expulsed from the pieces to welded thus preventing the formation of voids or other defect in the welded zone.

The process of FSW is most suitable for the components which are flat and long (plates or sheets), but it also can be adapted for pipes, hollow sections and positioning welding. The process of FSW is typically solid-state, meaning that the process operates below the solidus temperature of the metals being joined and no melting occurs during the process. According to Badheshia [2] the maximum temperature reached is of the order of 0.8 to 0.9 of melting temperature.

The result of the operated below melting temperature yields fine microstructures, absence of cracking, low residual distortion and no loss of alloying elements that are the main advantages of this solid phase process[3]. The heat generated between tool and material causes to reach a viscoplastic state that allow traversing of the tool along the weld line.

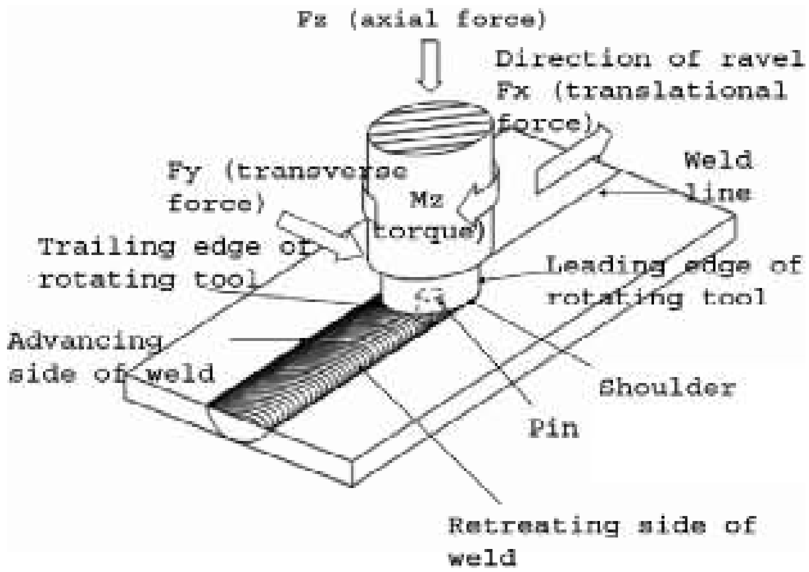

Figure 1: Schematic drawing of the friction stir welding [4]

Friction stir welding has been used to weld all wrought aluminium alloys, across the 2xxx, 5xxx, 6xxx and 7xxx series of alloys, some of which are bordering on being classed as virtually unweldable by fusion welding techniques. The process can also weld dissimilar aluminium alloys, whereas fusion welding may result in the alloying elements from the different alloys interacting to form deleterious intermetallics through precipitation during solidification from the molten weld pool.

Friction stir welding can also make hybrid components by joining dissimilar materials such as aluminium and magnesium alloys. The thicknesses of 6082-T6 that have so far been weld have ranged from 1.2mm to 50mm in a single pass, to more than 75mm when welding from both sides. Welds have also been made in pressure die cast aluminium material without any problems from pockets of entrapped high pressure gas, which would violently disrupt a molten weld pool encountering them [5].

The process advantages result from the fact that the FSW process (as all friction welding of metals) takes place in the solid phase below the melting point of the materials to be joined. The benefits therefore include the ability to join materials which are difficult to fusion weld, for example 2000 and 7000 aluminium alloys. Friction stir welding can use purpose-designed equipment or modified existing machine tool technology. The process is also suitable for automation and adaptable for robot use. Other advantages are as follows [6]:

- Low distortion, even in long welds
- Excellent mechanical properties as proven by fatigue, tensile and bend tests
- No arc
- No fume
- No porosity
- No spatter
- Low shrinkage
- Can operate in all positions
- Energy efficient
- Non-consumable tool
- One tool can typically be used for up to 1000m of weld length in 6000 series aluminium alloys
- No filler wire
- No gas shielding for welding aluminium
- No welder certification required
- Some tolerance to imperfect weld preparations - thin oxide layers can be accepted
- No grinding, brushing or pickling required in mass production
- Can weld aluminium and copper of >50mm thickness in one pass.

The limitations of the FSW process are being reduced by intensive research and development. However, the main limitations of the FSW process are at present [6]:

- Workpieces must be rigidly clamped
- Backing bar required (except where self-reacting tool or directly opposed tools are used)
- Keyhole at the end of each weld
- Cannot make joints which required metal deposition (e.g. fillet welds)

1.2 Problem Statement

Thermal modeling for the FSW is needed to predict the temperature distribution and field of the workpiece. The modeling is required to optimize the welding process and condition as the actual process of the FSW is rather costly and we cannot measure the exact temperature of the process. The thermal modeling also has been applied to work as an input to predict the microstructure, properties, distortion, and residual stress of the FSW process.

A computational two dimensional finite element model of the FSW process is a good way to do thermal modeling of process. It will describe the main aspects of the process and to show and evaluate the computational requirements needed for appropriate capture of the main phenomenon involved during PSW process.

1.3 Objectives and Scope of Study

1.3.1 Objectives

The project is aimed to simulate the Friction Stir Welding process as two dimensional problems. A Finite Element technique is employed to provide temperature distribution and patterns of flow for the material and involved in the welded joints. This project also is aimed to study on the residual stress of the friction stir welded plates.

1.3.2 Scope of Study

i. The research focus on a butt joint configuration between two identical plates
ii. The research focus on 2-D analysis of the FSW
iii. Steady state simulation
iv. The finite element analysis of friction stir welding is done using ANSYS software

1.4 Feasibility of the project

The project is feasible as it utilizes as an analysis of three dimensional finite element model of FSW is a best method to do thermal modeling of the process itself. The proposed project will be an improvement to the history of the FSW, as it will describe the main aspect of the process. The computational tool that presented in the project may be of the great relevance for technologist seeking to set the process control variable and to obtain suitable material properties that yields adequate on response of the structural components.

CHAPTER 2
LITERATURE REVIEW

2.1 Basics of Numerical Modeling

There are three primary approaches to numerical modeling. The Finite Element, Finite Difference, and Finite Volume approaches all present modeling strengths and weaknesses.

The Finite Element (FE) approach is a widely popular choice due to its generic formulation, a technique that lends itself to commercial code production. The nodal points and elemental volumes are generically formulated to accommodate a wide range of problems. Integration of the governing differential equations is usually accomplished by a fully algebraic approach called Gaussian Quadrature. This integration approach, coupled with generic nodal and elemental volumes lends itself to generalized code production. The FE approach can be used for irrotational material advection, thermal diffusion, small and large displacements of solid materials, electricity and magnetism, and wave propagation. The FE approach is usually not the method of choice for analysts numerically modeling fluid flow problems, a regime usually suited for the Finite Difference or Finite Volume approaches.

The Finite Difference (FD) approach is frequently included in analyses that involve time dependent results, and also in numerical solutions that require problem-specific attention. The FD approach, unlike the FE approach, is one that is usually specifically formulated for a distinct family of numerical problems. This requires the analyst to intimately understand the finite difference approximation of the governing differential equations utilized in his or her approach. This FD method has been further refined to include a class of fluid dynamic problems, commonly referred to as the Finite Volume method.

The Finite Volume (FC) approach is a popular choice of formulation for the advection and diffusion of heat and material. The FV approach usually considered a subset of the Finite Difference method. Scientists studying flow phenomenon such as aerodynamics and hydrodynamics may choose this approach. Compressible and incompressible fluid flow regimes are examples of problems that might be modeled in a FV approach.

The selection of one or more of these modeling approaches largely depends of the problem. Each of these numerical techniques have been exhaustively studied and characterized. One can read from decades of findings published on the results of various numerical methods, or combinations of methods for specific applications

2.2 Heat generation in Friction Stir Welding

According to Schneider [7], the heat in Friction Stir Welding (FSW) is generated by combination of friction and plastic dissipation during deformation of the metal. The heat generation is often assumed to occur predominantly under the shoulder, due to its greater surface area, and to be equal to the power required to overcome the contact forces between the tool and the work piece.

But later, various experiments conducted show that the assumption is partially declined and gave some explanation on the influence of all contact surfaces in heat generating of the FSW [3].

From the aspect of welded joint heat treatment during welding, friction stir welding can be described in four phases [8]:

1. Dwelling: the material is preheated by a stationary, rotating tool in order to achieve a sufficient temperature ahead of the tool to allow the traverse movement. This period includes the plunging of the tool into the work pieces at one point of the joint line.

2. Transient heating: when the welding tool begins traversal movement along joint line there is a transient period where the heat production and temperature around the tool rises until pseudo steady-state is reached.
3. Pseudo steady – state. Although fluctuations in heat generation will occur the thermal field and temperature around the tool remain effectively constant, at least on the macroscopic scale. Microscopic transformations are present on a high level
4. Post steady – state. Near the end of the weld heat may “reflect” from the end of the weld pieces and backing plate leading to additional heating around the tool.

For any welding process it is, in general, desirable to increase the travel speed and minimize the heat input as this will increase productivity and possibly reduce the impact of welding on the mechanical properties of the weld. At the same time it is necessary to ensure that the temperature around the tool is sufficiently high to permit adequate material flow and prevent flaws or tool fracture. When the traverse speed is increased, for a given heat input, there is less time for heat to conduct ahead of the tool and the thermal gradients are larger. At some point the speed will be so high that the material ahead of the tool will be too cold and the flow stress too high, to permit adequate material movement, resulting in flaws or tool fracture. If the ‘hot zone’ is too large then there is scope to increase the traverse speed and hence productivity.

Heat generation during FSW arises from two main sources; friction at the surface of the tool and the deformation of the material around the tool [9]. The heat generation is often assumed predominantly under the shoulder, due to its greater surface area, and to be equal to the power required to overcome the contact forces between the tool and workpiece. The contact condition under the shoulder can be described by sliding friction, using a friction coefficient μ and interfacial pressure P, or sticking friction, based on the interfacial shear strength at an appropriate temperature and strain rate. Mathematical approximations for the total heat generated by tool shoulder Q_{total} have been developed using both sliding and sticking friction model [10];

$$Q_{total} = \frac{2}{3}\pi P \mu \omega \left(R^3_{shoulder} - R^3_{pin} \right) \text{ for sliding} \tag{1}$$

$$Q_{total} = \frac{2}{3}\pi \tau \omega \left(R^3_{shoulder} - R^3_{pin} \right) \text{ for sticking} \tag{2}$$

Where

Ω is the angular velocity of the tool

$R_{shoulder}$ is the radius of the tool shoulder

R_{pin} is the radius of the tool pin

Several other equations have been proposed to account for factors such as the pin but the general approach still remains same.

A major difficulty in applying these equations is determining suitable values for the friction coefficient or the internal shear stress. The conditions under tool are both extreme and very difficult to measure. To date, these parameters have been used as fitting parameters where the model works back from measured thermal data to obtain a reasonable simulated thermal field. While this approach is useful for creating process models to predict, for example residual stress it is less useful for providing insights into the process itself.

2.2 Simulation Model

Most problems governed by differential equations can be solved by approximating the problem with numerical method and formulating a solution based on that method. Numerical modeling is the diversion of a geometrical domain into finite number of nodal points and elemental volumes, the approximation of the governing boundary physics affecting each nodal point and its neighboring point. The solution of the system of the equation resulting from this approximation.

If a problem can be quantified mathematically, it also has numerically modeled a solution. However, natural error in the solution, the selection of simplifying assumptions and the implementation of boundary conditions require a foundation of experience and knowledge. Numerical modeling applications might include the modeling of fluid motion, heat transfer, deformation of solids, coupled mechanical with chemical or thermal, as well as macro or microscopic modeling. Each of these problem areas also has multiple solution methods.

In the last years the scientific community has been very active on topics related to FSW processes. Anyway it should be observed that not so many publications are referred to the numerical modeling and simulation of the process due to the complex flow and difficulties in modeling the boundary conditions [11].

FSW is a quite complex process both for the heat generation aspect and for the deformation one, influenced by many parameters, technological (tool rotational speed, feed rate, etc.), and related to tool and workpiece materials (thermal properties, friction couple, etc.).

In order to develop a proper computer aided engineering of the FSW process, two main approaches have been followed: first of all thermal models, taking into account the heat generated by both friction forces work and the material deformation work, have been proposed by Song et al. [12], Schmidt et al. [13] and Chao et al. [14], trying to highlight the temperature distributions nearby the rotating pin. Second, finite element thermo-mechanical models have been presented: important contributions in 2D modeling include that of Xu and Deng [15] who developed a 2D finite element procedure to simulate the FSW process with focus on velocity field, material flow characteristics and the equivalent plastic strain distribution.

The commercial FEM code ABAQUS was employed and an Arbitrary Eulerian-Lagrangian finite element formulation with adaptive meshing was utilized, that considered large elastic-plastic deformation and temperature-dependent material properties. The researchers compared their predicted results to experimental data available and observed reasonable correlation between the equivalent plastic strain distribution and the distribution of the microstructure zones in the weld. However their FEM analysis was not a thermo-mechanically coupled procedure. The temperature data obtained from experiment was superimposed as a prescribed temperature field for the deformation analysis. This severely affected the welding force and stress prediction as the material properties (flow stress model) used by the author were actually temperature-dependent.

Ulysse [16] presented a 3D FEM visco-plastic model for FSW of thick aluminum plates using a commercial FEM code FIDAP. The author investigated the effect of tool speeds on the process parameters. It was found that the higher translational speed leads to higher welding force, while increasing the rotational speed has the opposite effect, that of force reduction. Reasonable agreement between the predicted and the measured temperature was obtained and the discrepancies were explained as an inadequate representation of the constitutive behavior of the material for the wide ranges of strain-rate, temperatures and strains typically found during FSW.

Chen and Kovacevic [17] developed a 3D FEM model to study the thermal history and thermo-mechanical phenomena in the butt-welding of aluminum alloy 6061-T6 using a commercial FEM code ANSYS. Their model incorporated the mechanical reaction between the tool and the weld material. Experiments were conducted and X-ray diffraction technique used to measure the residual stress in the welded plate. The welding tool (i.e. the shoulder and pin) in the FEM model was modeled as heat source, with the nodes moved forward at each computational time step. This simple model severely limited the accuracy of the mechanical stress and force predictions.

Colegrove and Shercliff used CFD commercial software for a 2D and 3D numerical investigation on the influence of pin geometry during FSW [18], comparing different pin shapes in terms of material flow and welding forces on the basis of both a stick and a slip boundary condition at the tool-workpiece interface. In spite of the good obtained results, the accuracy of the analysis is limited by the assumption of isothermal conditions.

The research group has developed a FEM model of the FSW process through the commercial FEA software DEFORM-3D ™, Lagrangian implicit code designed for metal forming processes. The work piece was modelled as a rigid visco-plastic material, and the welding tool assumed rigid [19-21]. In particular a "single block approach" was followed avoiding numerical instabilities due to the self contact of the blanks to be joined.

Despite significant recent advances in numerical modeling of the FSW process, the previous models have severe limitations in either the representation of geometry, or the material behavior, or the boundary conditions. The objective of this research is to develop a numerical model that can be use for optimal design of FSW process. In this paper, a fully 2D FEM model for the FSW process is proposed, that is thermo-mechanically coupled and with rigid-viscoplastic material behavior. Shercliff et. al [18] states that the validity of this approach steams from large inelastic strain, with hot metal flow corresponding to viscoplastic behavior at a very low Reynolds number. Finally the numerical model was used to develop a temperature profile along the work piece.

CHAPTER 3

METHODOLOGY

3.1 Procedure Identification

Before proceeding with the project itself, steps were drawn out diagrammatically to clear out on the flow to ensure the study can be completed in the given time. The methodology is summarize in Figure 2

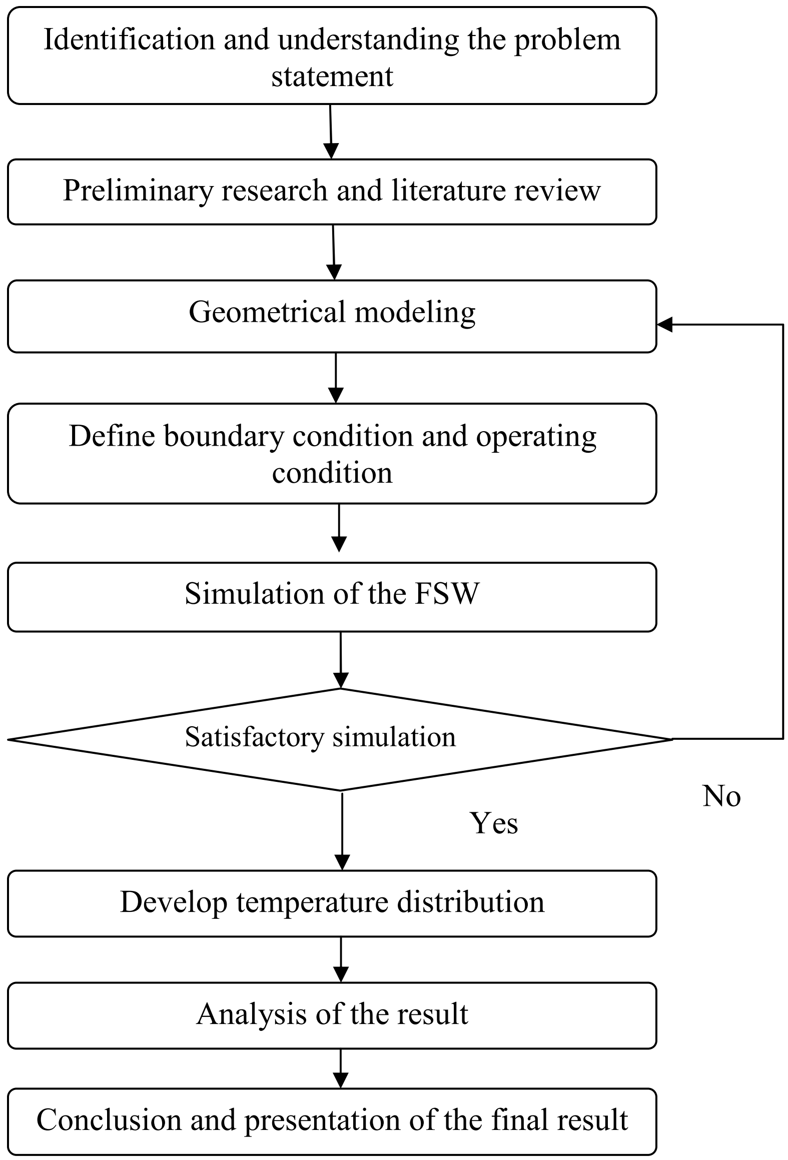

Figure 2: Methodology for completion of the project

For this project, after the title selection was done, literature review process was conducted. Literature review is very essential in doing this project since all relevant information regarding this project was gathered and studied during this period. The internet is the biggest and easiest source of information. Nevertheless, some literatures were also gathered from books that can be found in the library. Then moved to geometrical modeling where the work piece and tool is model and meshed for the simulation. After boundary condition and operating condition is applied, simulation is conducted and analyzed to get temperature distribution and stresses of the model.

3.2 Finite element modeling

3.2.1 Selection of welding work piece and tool

The data, information and materials properties related to welding work piece and tool of the FSW were obtained. The chosen material for the work piece is Aluminum Alloy 6061-T6 (AA6061-T6). This alloy has excellent joining characteristic and good acceptance of applied coated. It also has high strength, good workability and high resistance to corrosion. Besides this alloy is widely use and available in the industry. The mechanical property of the alloy is in Table 1[23]. For the tool material the material is tool steel H13. Tool steels typically have excess carbides (carbon alloys) which make them hard and wear-resistant. Most tool steels are used in a heat-treated state, generally hardened and tempered. The properties a of the H13 [24] is shown in Table 2

Table 1: Properties of AA 6061-T6

Properties	**Values**
Density	2700 kg/m^3
Specific Heat	896 J/Kg-K
Thermal Conductivity	167 W/m-K
Melting Point	925 K
Modulus of elasticity	80GPa

Possion's ratio	0.33
Thermal expansion	0.000023/K

Table 2: Properties of H13 tool steel

Properties	**Values**
Density	7800 kg/m^3
Thermal Conductivity	28.6 W/m-K
Thermal Expansion	10.4 ($10^{-6}/^0C$)

3.2.2 Geometrical modeling

After all the properties data is gathered, the work piece plates and tool is designed. The dimension of the tool and plates are shown in the table below.

Table 3: Dimensions of the tool and the plates

Tool	Dimension (mm)	Plates	Dimension (mm)
Pin diameter	6.0	Long	100
Pin height	3.5	Width	40
Shoulder diameter	12.0	Thick	10
Shoulder height	10.0		

3.2.3 Meshing geometry

Before meshing the geometry, the element type of the work piece should be assigned first. In this model the assigned thermal element type of the work piece is PLANE55 type. PLANE55 can be used as a plane element or as an axisymmetric ring element with a 2-D thermal conduction capability. The element has four nodes with a single degree of freedom, temperature, at each node.

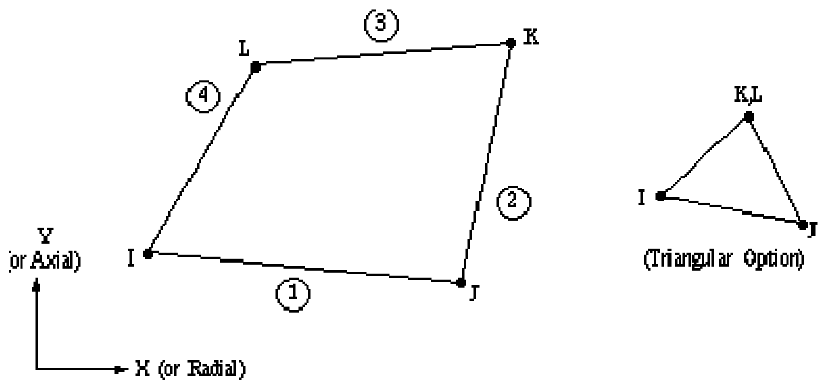

Figure 3: PLANE55 Geometry

The element is applicable to a 2-D, steady-state or transient thermal analysis. The element can also compensate for mass transport heat flow from a constant velocity field. The geometry, node locations, and the coordinate system for this element are shown in Figure 3. The element is defined by four nodes and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions.

The corresponding plane for the structural analysis of the model is PLANE182. PLANE182 is used for 2-D modeling of solid structures. The element can be used as either a plane element (plane stress, plane strain or generalized plane strain) or an axisymmetric element. It is defined by four nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element has plasticity, hyperelasticity, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

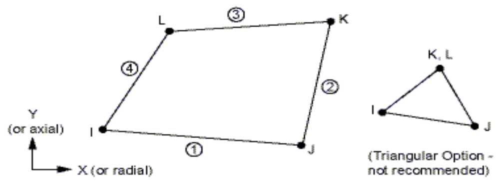

Figure 4: PLANE182 geometry

The geometry and node locations for this element are shown in Figure 4. The element input data includes four nodes, a thickness (for the plane stress option only), and the orthotropic material properties. The default element coordinate system is along global directions. You may define an element coordinate system using **ESYS**, which forms the basis for orthotropic material directions.

After element types of model are defined, the model is meshed with quadrilateral type of mesh with 0.001 element edge length.

3.2.4 Define boundary condition and operating condition

Heat generated

The heat source in FSW is considered to be friction between the tool shoulder and work piece surface. The local heat generated over the interface is assumed the frictional work in this model where it can be calculated by the following expression [25];

$$Q = 2\pi\mu F_n N \mathrm{R_S}$$

The coefficient of friction μ varies with temperature. The detail of the variation is not clear so an effective coefficient of friction of 0.4 is assumed in this model [26]

Boundary conditions

The boundary and initial conditions that are applied in this model are summarized as

I. Convection boundary conditions for all the work piece surfaces that exposed to the air

$$k\frac{\partial T}{\partial N} = h(T - T_o)$$

The surface of the work piece in contact with the back plate is approximated to the convection condition with an effective coefficient of convection 3000 W/m^2K [29]. The mode of heat transfer between work piece and other surfaces is modeled with coefficient of $30W/m^2K$.

II. The initial boundary condition for the calculation is

$$T(x, y, z, t) = T_o$$

The initial temperature of the work piece is assumed to be atmospheric temperature which is 27^oC (300 K)

Operating condition

Table 4: Operating condition of the model

Properties	Value
Tool rotation speed	2000 rpm
Welding speed	1.6 mm/s
Applied force	25 kN
Ambient temperature	300 K

3.2.5 Assumptions

Several assumptions had been made in developing this model which is:

- Heat generated at tool shoulder and work piece interface is frictional heat only
- Friction or contact between tool and work piece is not model, only frictional heat is account in this model
- Heat generated during penetration of tool to work piece and extraction of tool from work piece is not considered
- The tool pin is cylinder and thread of the pin is neglected
- Heat transfer by radiation is neglected
- Material properties are uniform
- The plates of work piece is assumed as one piece to simplified the simulation
- The heat transfer between bottom of the work piece and backing plate is 2250 $W/m^2.K$

3.3 Gantt Chart

Attached in Appendix 1.

3.4 Tool/Equipment/software required

ANSYS is engineering simulation software. It develops general purpose finite element analysis and computational fluid dynamics software. ANSYS has developed a range of computer aided engineering (CAE) product.

These general purpose finite element modeling packages for numerically solving mechanical problems, including static/dynamic structural analysis (both linear and non linear), heat transfer and fluid problems, as well as acoustic and electromagnetic problems. ANSYS software includes solvers for thermal, structural, CFD, electromagnetic, acoustic and also can coupled these separate physics together in order to address multidisciplinary application. ANSYS software also can be used in civil engineering, electrical engineering, physics and chemistry.

CHAPTER 4
RESULT AND DISCUSSION

4.1 Meshing of the Geometry

In this investigation, a two-dimensional model was developed to predict the thermal cycles of friction stir butt weld joint using the finite element method. In the numerical simulation of the FSW of aluminum alloy, it was assumed that two identical plates were welded symmetrically. The cross section of the butt joint between two plates is modeled using commercial finite element package ANSYS. The meshed plate is shown in Figure 6 which has 891 nodes.

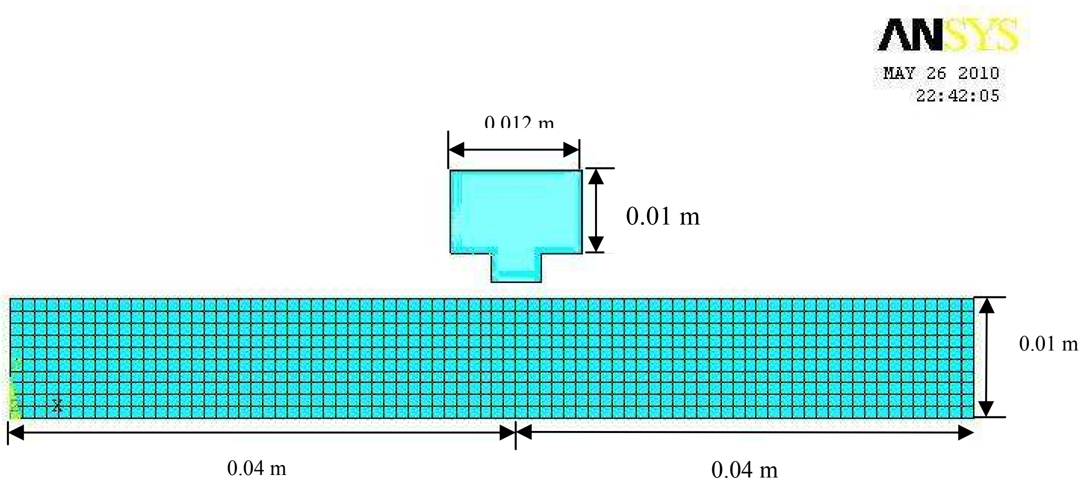

Figure 5: Meshed model of work piece

4.2 Simulation Result

4.2.1 Thermal Analysis

A heat source Q having a linear distribution of heat from the center to the outside diameter of the tool is applied to the top surface of the work piece to simulate the heat generation due to the friction. Tool rotating speed is manipulate and varies to see different temperature profile in each rotating speed. The selected tool rotating speed are 537 rpm, 637 rpm, and 737 rpm. These values are inserted to the heat generated equation, Q. In this numerical model, the vertical, top and bottom surfaces of the work pieces are assumed to transfer heat by convection to the ambient. At all the surfaces except bottom a convective heat transfer of 30 W/m^2K is used which is for natural convection between aluminum and air. In friction stir welding, the work pieces are placed over backup plates and clamped. The bottom surface is modeled with convective heat transfer of 2250 W/m^2K to account for the heat flowing through the contact interface between backing plate and bottom surface of the work piece

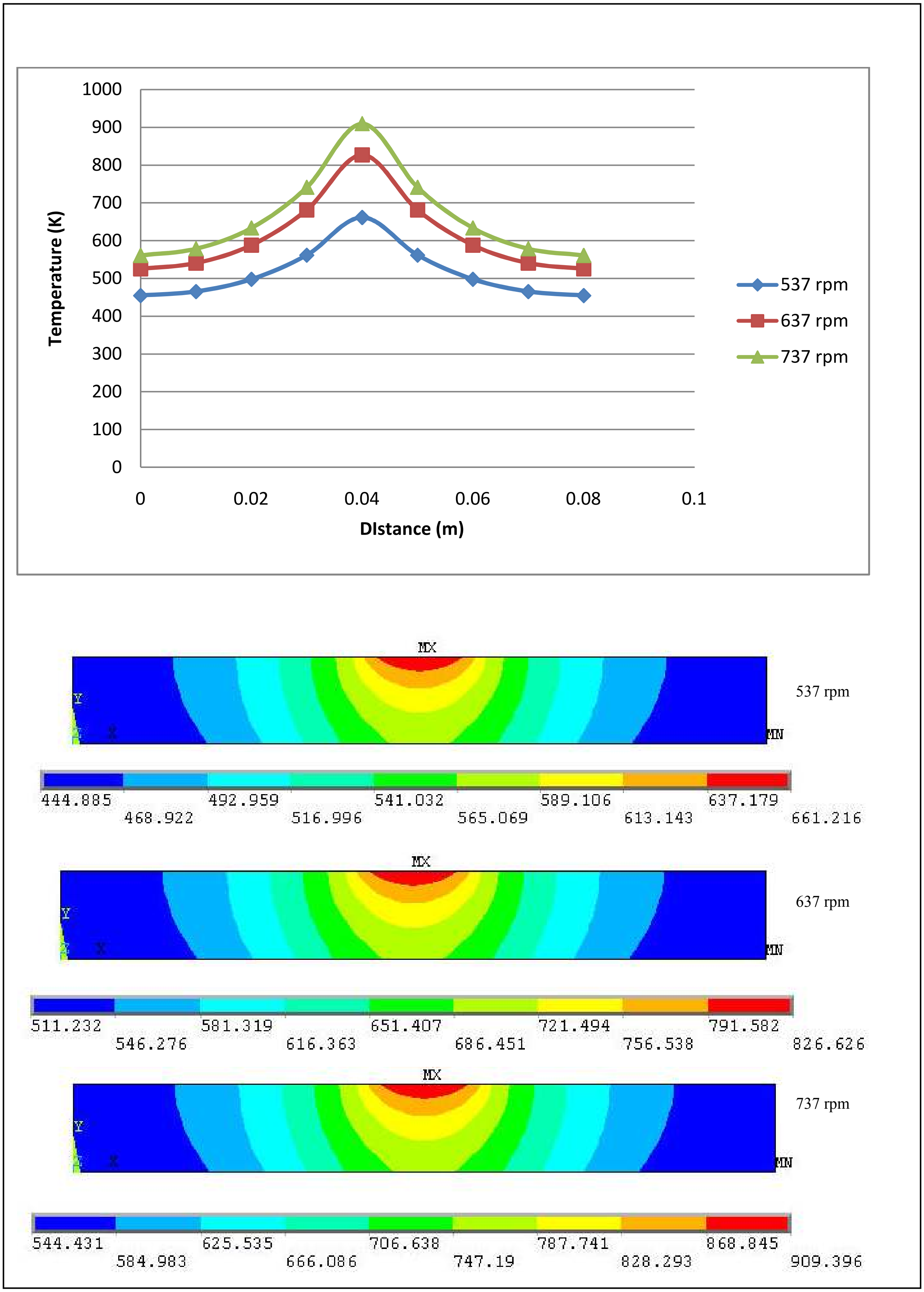

Figure 6: Temperature distribution across the work piece

Figure 6 shows different temperature variation at a particular point in the center line during welding operation. The temperature was initially at room temperature and slowly increased and reached the maximum when the tool reaches the point. Figure 6 show graph of temperature distribution for each tool rotation speed. The peak temperature calculated from 737 rpm tool rotation speed is 909 K. The peak temperature calculated from 637 tool rpm rotation is 826 K and the peak temperature calculated from 537 rpm tool rotation speed is 661 K. All peak temperatures of the models occurred at the top surfaces of the work piece. This is because the top surface, particularly in the tool shoulder area is the source of heat generated, Q for the welding process.

According to Badheshia [2], the maximum temperature developed during FSW process ranges from 80% to 90% of the melting temperature of the welding material. Here the melting point of aluminum alloy AA 6061-T is 925 K (652°C) so the range of temperature developed during FSW is between 740 K to 833 K.

From the result, it is concluded that 737 rpm tool rotation speed is not suitable for the FSW process as the peak temperature produced is 909 K or 98% of the melting temperature. As the peak temperature is beyond the range of melting temperature, the tool will stick with the work piece thus the welding process is disturb as the tool is not moving because it has fuse with the work piece.

537 rpm tool rotation speed also is not suitable for the FSW process because the peak temperature produced is about 661 K or 71% of the melting temperature. Low temperature will result in low bonding between the two plates thus the welded joint will easily fracture.

The peak temperature produced when 637 rpm tool rotation speed in this simulation is 826 K or 89% of the melting point of alloy which is in the range of the maximum temperature developed. Song and Kovacevic [12] had modeled the heat transfer in FSW of aluminum alloy 6061-T6. Figure 7 (left side) shows the result of the temperature distribution modeled by Song and Kovacevic [12] and compared with result of this model. They found that the maximum temperature developed during welding was 820K which also in range of the maximum temperature developed.

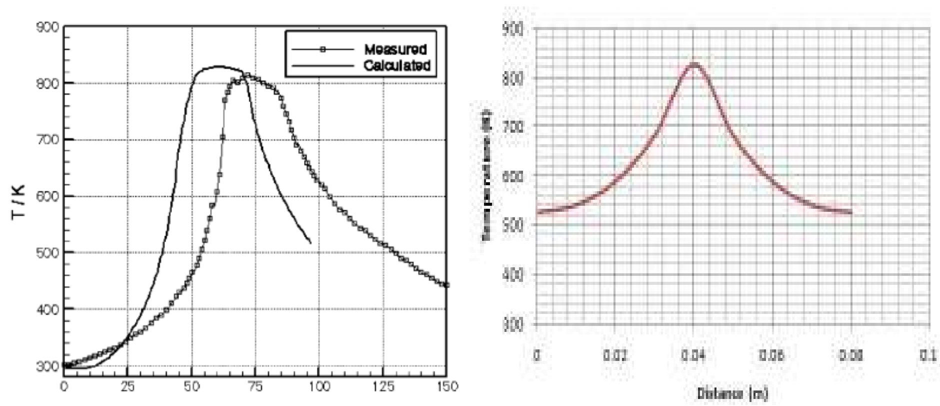

Figure 7: Comparison of temperature distribution produced with Song and Kovacevic [12] model

4.2.2 Temperature in different zones of FSW

The microstructure of a friction-stir weld depends in detail on the tool design, the rotation and translation speeds, the applied pressure and the characteristics of the material being joined. There are a number of zones. The heat-affected zone (HAZ) is as in conventional welds. The central nugget region containing the onion-ring flow-pattern is the most severely deformed region, although it frequently seems to dynamically recrystallise, so that the detailed microstructure may consist of equiaxed grains. The layered (onion-ring) structure is a consequence of the way in which a threaded tool deposits material from the front to the back of the weld. It seems that cylindrical sheets of material are extruded during each rotation of the tool, which on a weld cross section gives the characteristic onion-rings

The thermomechanically-affected zone (TMAZ) lies between the HAZ and nugget; the grains of the original microstructure are retained in this region, but in a deformed state. The top surface of the weld has a different microstructure, a consequence of the shearing induced by the rotating tool-shoulder. Figure 8 below show the maximum temperature range of different zones in FSW.

	Nugget, TMAZ and HAZ Near Nugget	HAZ Near Unaffected Zone	Unaffected Base Metal Zone
Max. temperature range	300°C ~ 475°C	300°C ~ 200°C	<200°C
Yield strength during cooling	60% of σ_s	80% of σ_s	100% of σ_s

Figure 8: Temperature in different zones of the FSW joint in as weld condition [27]

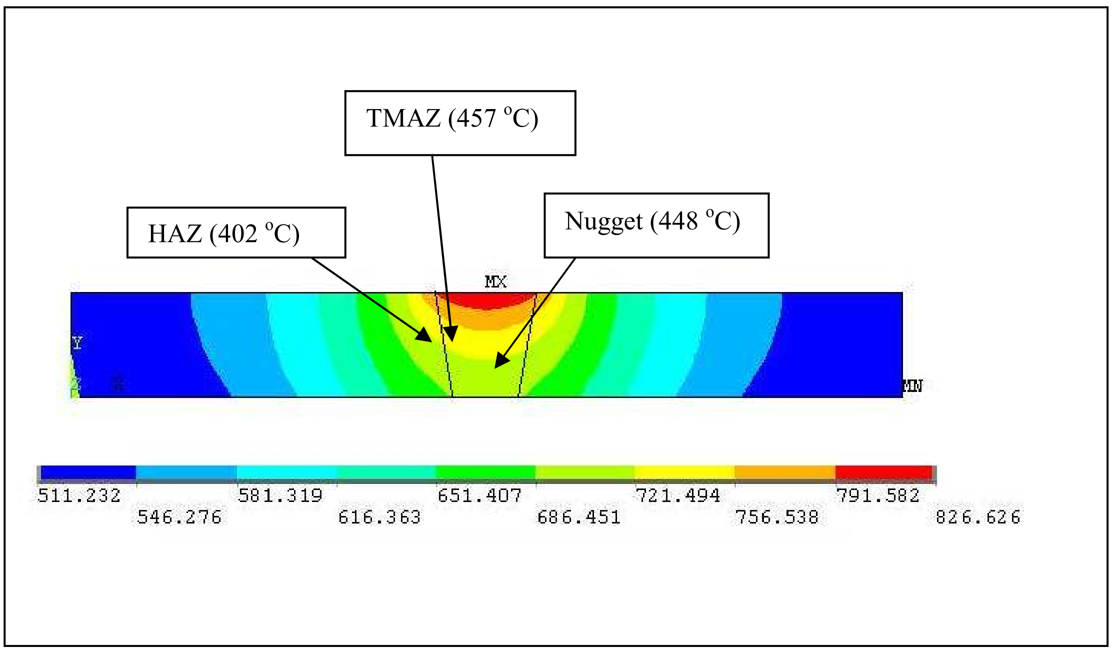

Figure 9: Temperatures in different zones of the FSW

Figure 9 show the temperatures in TMAZ, HAZ and nugget zones of the FSW in this simulation. From the Figure 9 it is concluded that the temperatures produced in this simulation is within the range of the maximum temperature range shows in Figure 8.

4.2.3 Structural Analysis

Temperature distribution on the work piece is used as a load for the structural analysis in determine the Von Mises stress of the work piece during FSW and displacement vector of the work piece.

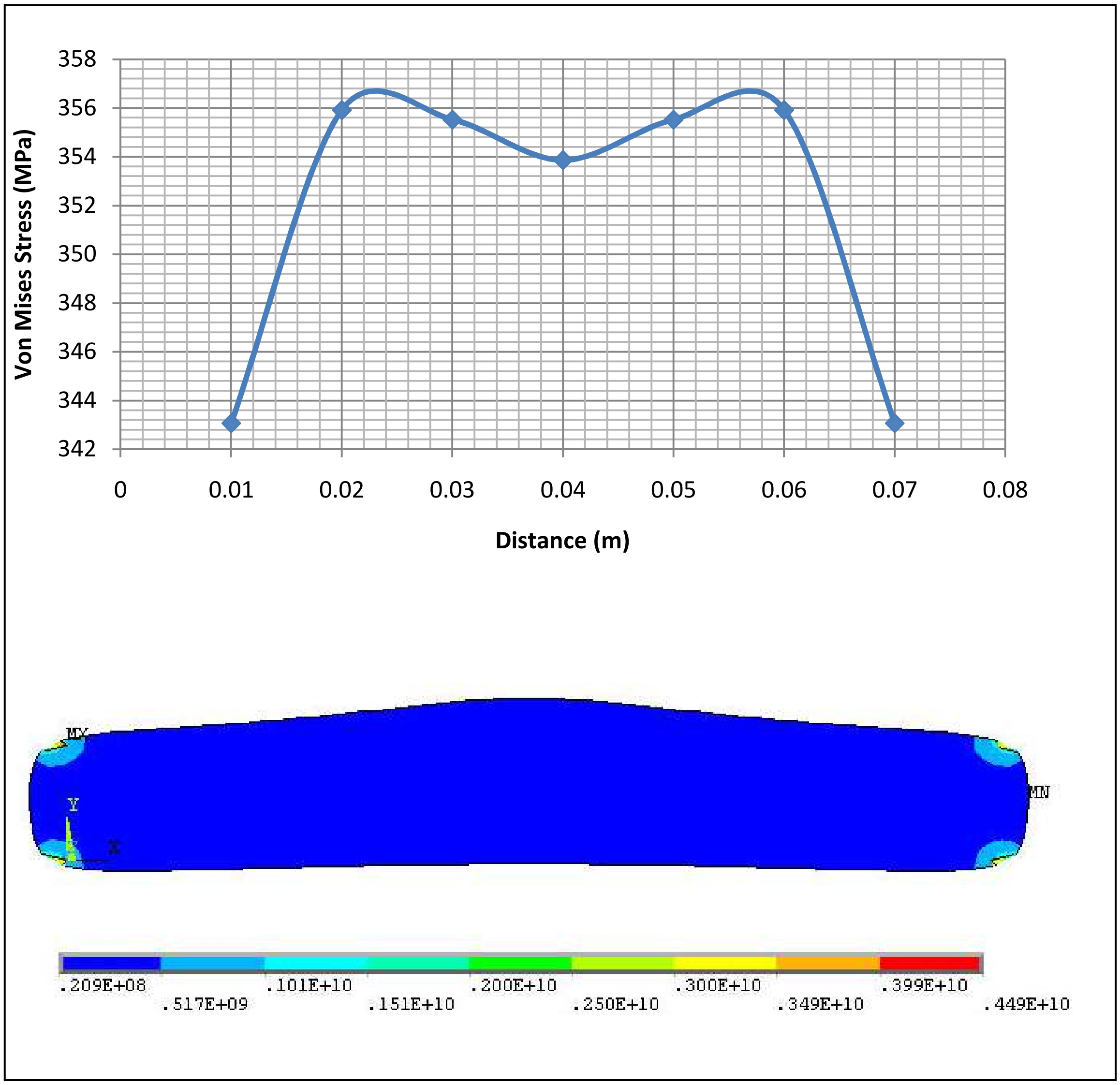

Figure 10: Von Mises stress distribution

Figure 10 show the Von Mises stress of the work piece when FSW process happened. From the graph the stress is near to zero in the high temperature area. It can be seen that the stress distribution in tool area is very less as it has not affected by thermal stress or by structural loading. In the other area, it is observed that the stress starts to increase due to the mechanical force in the horizontal direction and reaches maximum at edge. Due to the thermal expansion and constraint on the sides by the fixture results in compressive stress in this area.

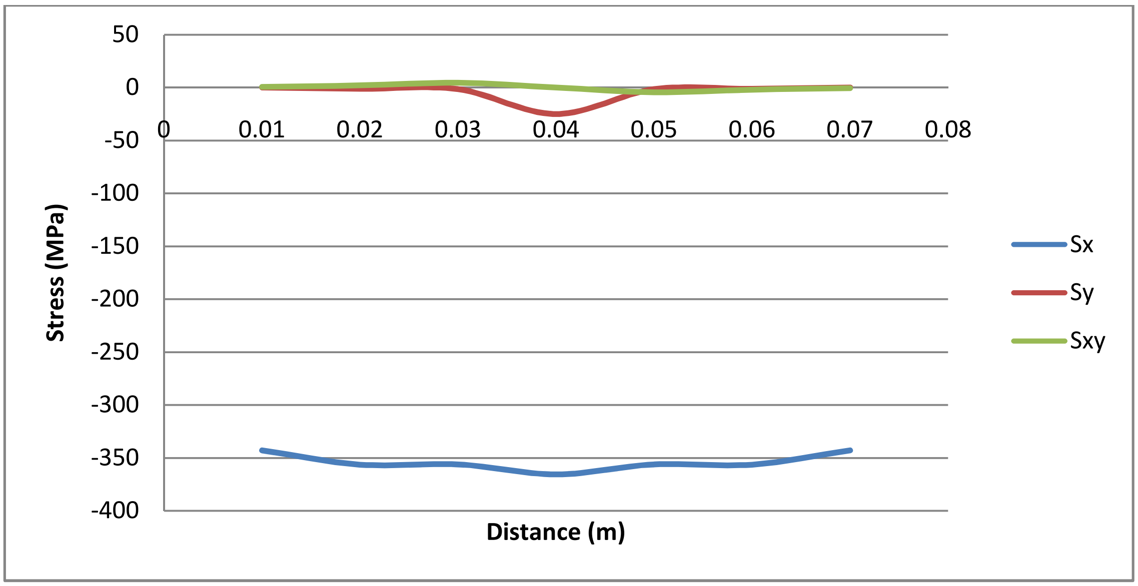

Figure 11: Stress versus distance

Figure 11 show a graph of stresses (Sx, Sy, and Sxy) versus distance when tool is plunged into the work piece during the welding process. It can be seen that stress distribution in the outer edge of work piece is very less as it has not affected by thermal stress and structural loading. The stresses start to increase as we move to the middle area of the work piece. This is due to the mechanical force in the horizontal direction and reaches maximum at tool shoulder area. The area behind the tool shoulder produce large compressive stress value because of thermal expansion and constraint on the both sides of the work piece.

CHAPTER 5
CONCLUSION

5.1 Conclusion

A two dimensional steady state model of work piece was developed in order to build qualitative frame work to understand the process of FSW. In doing this analysis, the preparation is done by gathering previous journals and papers focusing on the development of the FSW. The maximum peak temperature produced in this simulation is around 826 K (553^{o}C). This value is within the range of the maximum temperature range suggested by Badheshia [2], where he stated that the maximum temperature in the FSW process is in the range of 80% to 90% (740 K to 833 K) of the melting temperature of the work piece. The temperature values produced in different zones such as TMAZ, HAZ and nugget area of the welded plates which are 730K (457^{o}C), 675K (402^{o}C) and 721K (448^{o}C) also within the range as proposed by Chao and Xin [31]. The proposed project title can help in improvement of Friction stir Welding, a computational finite element model of the FSW process is presented so as to describe the main aspects of the process and to show and evaluate the computational requirement needed for the appropriate capture of the main phenomena involved. The calculated results were compared with experimental data published by researchers. The prediction showed that the maximum temperature gradients in longitudinal and lateral directions were located just beyond the shoulder edge. The data from the thermal model of FSW can be used to predict the microstructure, distortion, properties and residual stress of the FSW that will be useful in improving the process of the FSW itself. The objective of the project can be fulfilled successfully through the hard work and good engineering judgment.

5.2 Recommendation

Future efforts must be oriented towards improving the simulation of the friction stir welding by doing three dimensional model to capture the effect of heat around tool area to the work piece. A more accurate characterization of the frictional condition between the tool and the work piece also should be considered. Besides that, Research focus in the near future will be on the transient model of the friction stir welding as well as the limit of weldability in terms of sheet thickness.

REFERENCES

1. Thomas, M., Nicholas, J., Needham, J., Murch, M., Templesmith, P., and Dawes, C., Friction Stir Welding, GB Patent Application No. 9125978.8 Dec.1991, US Patent No. 5460317, 1991.
2. Friction Stir Welding. Retrieved August, 13, 2009 from http://www.msm.cam.ac.uk
3. Zhu, X., Chao, Y., 2003, "Numerical simulation of transient temperature and residualstresses in friction stir welding of 304L stainless steel", Journal of Materials Processing Technology 146 (2004) 263–272
4. Crawford, R., Cook, G.E., Strauss, A.M.and Hartman, D.A. (2006) 'Modelling of friction stir welding for robotic implementation', *Int.J. Modelling, Identification and Control*, Vol. 1, No. 2, pp.101–106
5. Friction Stir Welding. Retrieved October, 24, 2009 from http://www.azom.com/details.asp?ArticleID=1170
6. Friction Stir Welding - Process advantages. Retrieved October, 25, 2009 from http://www.twi.co.uk/content/fswproc.html
7. Schneider, J.A (2007) 'Temperature Distribution and Resulting Metal Flow', *Friction Stir Welding and Processing,* (pp. 36-49)
8. ĐURĐANOVIĆ, M., MIJAJLOVIĆ, M., MILČIĆ, D., STAMENKOVIĆ, D., 'Heat Generation During Friction Stir Welding Process', *Tribology in industry*, Volume 31, No. 1&2, 2009
9. Qi, X, Chao, Y J (1999). "Heat transfer and Thermo-Mechanical analysis of FSW joining of 6061-T6 plates". 1st International Symposium on FSW (CD ROM). Thousand Oaks, USA: TWI
10. Frigaard, O; Grong, O; Midling, O T (2001). "A process model for friction-stir welding of age hardening aluminium alloys". *Metallurgical and Material Transactions* **32A** (5): 1189–1200

11. Soundararajan, V., Zekovic, S., Kovacevic, R., 'Thermo-mechanical model with adaptive boundary conditions for friction stir welding of Al 6061', *Int. J. Machine Tools & Manufacture*, 45 (2005) 1577-1587
12. Song, M., Kovacevic, R., (2003) "Thermal modeling of friction stir welding in a moving coordinate system and its validation", *Int. J. of Machine Tools & Manufacture*, Vol. 43, pp. 605-615.
13. Schmidt, H., Hattel, J., Wert, J., (2004), 'An analytical model for the heat generation in friction stir welding'*, Modeling and Simulation in Materials Science and Engineering*, Vol. 12, pp. 143-157.
14. Chao, Y., Qi, X., Tang, W., (2003), Heat transfer in friction stir welding – Experimental and numerical studies, Transaction of the ASME, Vol. 125, pp. 138-145.
15. Deng, X. and Xu, S., 'Two-Dimensional Finite Element Simulation of Material Flow in the Friction Stir Welding Process', *Journal of Manufacturing Processes* Vol. 6/No. 2 2004
16. Ulysse, P., Three-dimensional modeling of friction stir-welding process, Machine Tools & Manufacture, 42 (2002) 1549-1557
17. Chen, C., Kovacevic, R., Finite element modeling of friction stir welding – thermal and thermo- mechanical analysis, *Machine Tools & Manufacture*, 43 (2003) 1319-1326
18. Shercliff, H., Colegrove, P. (2007), 'Process Modeling', *Friction Stir Welding and Processing* (PP. 186-217)
19. Buffa, G., Hua, J., Shivpuri, R., Fratini, L., "A continuum based FEM model for friction stir welding - model development", Material Science and Engineering A, Vol. 419/ 1-2, pp. 389-396 (2006)
20. Buffa, G., Hua, J., Shivpuri, R., Fratini, L., "Design of the friction stir welding tool using the continuum based FEM model", Material Science and Engineering A, Vol. 419/1-2 pp. 381-388. (2006)
21. Fratini, L., Beccari, S., Buffa, G., Friction stir welding FEM model improvement through inverse thermal characterization, Transaction of NAMRI/SME, 33 (2005), pp. 259-266

22. Final Year Project Guidelines for Mechanical Engineering
23. Aluminum 6061-T6; 6061-T651.Retrieved August, 30, 2009 from http://asm.matweb.com/search/SpecificMaterial.asp?bassnum=MA6061T6
24. Tool Steel AISI H13. Retrieved August, 30, 2009 from http://www.efunda.com/Materials/alloys/tool_steels/show_tool.cfm?ID=AISI_H13&prop=all&Page_Title=AISI%20H13
25. X. K. Zhu, and Y. J. Chao: Numerical Simulation of Transient Temperature and residual Stresses in Friction Stir Welding of 304L Stainless Steel. *Journal of Materials Processing Technology*, 2004, volume 146, pp, 263-272.
26. Vijay Soundararajan, Srdja Zekovic and Radovan Kovacevic:Thermo-mechanical Model with Adaptive Boundary Conditions for Friction Stir Welding of Al 6061. *International Journal Machine Tools and Manufacture*, 2005, pp, 1-11.
27. Chao, H. Xin, “Thermal-thermo mechanical modeling of friction stir welding of Aluminum Alloy”

APPENDICES

APPENDIX A: Project Gantt chart

Semester 1 (FYP I)

No	Progress Work	Week														
		1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
1	Selection of Project Topic															
2	Preliminary Research Work															
3	Submission of Preliminary Report					X										
4	Selection of tool and workpiece															
5	Designing of the tool and workpiece															
6	Submission Of Progress Report								X							
7	Seminar								X	X						
8	Geometrical modelling using gambit															
9	Submission of interim														X	
10	Oral presentation															X

Semester 2 (FYP II)

No	Progress Work	Week													
		1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Define boundary condition														
2	Selection of operating condition														
5	Progress report I submission				X										
6	Simulation on														
7	Progress report II							X							
8	Seminar							X							
9	Develop temperature distribution														
10	Analysis of the result														
11	Dissertation draft														X
12	Oral presentation									During study week					
13	Hard bound dissertation									Seven (7) after oral presentation					

APPENDIX B: ANSYS coding for thermal analysis

```
/BATCH
! /COM,ANSYS RELEASE 12.0.1  UP20090224       04:49:28    05/26/2010
/input,menust,tmp,'',,,,,,,,,,,,,,,,1
! /GRA,POWER
! /GST,ON
! /PLO,INFO,3
! /GRO,CURL,ON
! /CPLANE,1
! /REPLOT,RESIZE
WPSTYLE,,,,,,,,0
/CWD,'E:\New Folder'
/FILNAME,new3,0
!*
/NOPR
/PMETH,OFF,0
KEYW,PR_SET,1
KEYW,PR_STRUC,0
KEYW,PR_THERM,1
KEYW,PR_FLUID,0
KEYW,PR_ELMAG,0
KEYW,MAGNOD,0
KEYW,MAGEDG,0
KEYW,MAGHFE,0
KEYW,MAGELC,0
KEYW,PR_MULTI,0
KEYW,PR_CFD,0
/GO
!*
! /COM,
! /COM,Preferences for GUI filtering have been set to display:
! /COM,  Thermal
!*
/PREP7
!*
ET,1,PLANE55
!*
!*
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,KXX,1,,167
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,C,1,,896
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,DENS,1,,2700
FINISH
/SOL
FINISH
/PREP7
K,1,0,0,0,
K,2,0.08,0,0,
K,3,0.08,0.01,0,
K,4,0.46,0.01,0,
FLST,2,4,3,ORDE,2
FITEM,2,1
FITEM,2,-4
KDELE,P51X
K,1,0,0,0,
K,2,0.08,0,0,
K,3,0.08,0.01,0,
K,4,0.046,0.01,0,
K,5,0.034,0.01,0,
K,6,0.034,0.01,0,
KDELE,       6
K,6,0,0.01,0,
LSTR,       1,       6
```

```
LSTR,       6,       5
LSTR,       5,       4
LSTR,       4,       3
LSTR,       3,       2
LSTR,       2,       1
FLST,2,6,4
FITEM,2,1
FITEM,2,2
FITEM,2,3
FITEM,2,4
FITEM,2,5
FITEM,2,6
AL,P51X
AESIZE,ALL,0.001,
MSHKEY,0
CM,_Y,AREA
ASEL, , , ,       1
CM,_Y1,AREA
CHKMSH,'AREA'
CMSEL,S,_Y
!*
AMESH,_Y1
!*
CMDELE,_Y
CMDELE,_Y1
CMDELE,_Y2
!*
PHYSICS,WRITE,thermal, , ,
PHYSICS,CLEAR
ETCHG,TTS
!*
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,8e10
MPDATA,PRXY,1,,0.33
MPTEMP,,,,,,,,
MPTEMP,1,0
UIMP,1,REFT,,,
MPDATA,ALPX,1,,23e-6
PHYSICS,WRITE,struct, , ,
FINISH
/SOL
!*
ANTYPE,0
!*
!*
PHYSICS,READ,THERMAL
FINISH
/PREP7
FINISH
/SOL
FLST,2,4,4,ORDE,4
FITEM,2,1
FITEM,2,-2
FITEM,2,4
FITEM,2,-5
/GO
!*
SFL,P51X,CONV,30, ,300,
FLST,2,1,4,ORDE,1
FITEM,2,6
/GO
!*
SFL,P51X,CONV,2250, ,300,
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
```

```
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
FLST,2,13,1,ORDE,3
FITEM,2,12
FITEM,2,46
FITEM,2,-57
!*
/GO
F,P51X,HEAT,4002
! /STATUS,SOLU
SOLVE
FINISH
/SOLU
!*
PHYSICS,READ,STRUCT
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
```

```
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,1.08222638492,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
! /DIST,1,0.924021086472,1
! /REP,FAST
FLST,2,4,1,ORDE,4
FITEM,2,1
FITEM,2,-2
FITEM,2,58
FITEM,2,92
!*
/GO
D,P51X, , , , , ,ALL, , , , ,
LDREAD,TEMP,,, , ,'new3','rth',' '
FINISH
/PREP7
TREF,300,
FINISH
/SOL
! /STATUS,SOLU
SOLVE
FINISH
/POST1
!*
! /EFACET,1
! PLNSOL, U,X, 0,1.0
!*
! /EFACET,1
! PLNSOL, U,Y, 0,1.0
!*
! /EFACET,1
! PLNSOL, U,SUM, 0,1.0
!*
! /EFACET,1
! PLNSOL, S,X, 0,1.0
!*
! /EFACET,1
! PLNSOL, S,Y, 0,1.0
!*
! /EFACET,1
```

```
! PLNSOL, S,XY, 0,1.0
!*
! /EFACET,1
! PLNSOL, S,Z, 0,1.0
!*
! /EFACET,1
! PLNSOL, S,EQV, 0,1.0
!*
! /EFACET,1
! PLNSOL, S,1, 0,1.0
!*
! /EFACET,1
! PLNSOL, S,3, 0,1.0
!*
! /EFACET,1
! PLNSOL, S,Z, 0,1.0
!*
! /EFACET,1
! PLNSOL, S,XZ, 0,1.0
!*
! /EFACET,1
! PLNSOL, S,XY, 0,1.0
!*
! /EFACET,1
! PLNSOL, S,YZ, 0,1.0
!*
! PRNSOL,S,COMP
!*
! PRNSOL,S,COMP
!*
!*
! PRNSOL,U,X
!*
! PRNSOL,S,PRIN
!*
! /EFACET,1
! PLNSOL, S,EQV, 0,1.0
!*
! /EFACET,1
! PLNSOL, S,X, 0,1.0
!*
! /EFACET,1
! PLNSOL, U,SUM, 0,1.0
FINISH
! /EXIT,ALL
! /COM,ANSYS RELEASE 12.0.1  UP20090224       00:56:35    05/27/2010
/POST1
!*
! /EFACET,1
! PLNSOL, S,EQV, 0,1.0
!*
! PRNSOL,S,PRIN
!*
!*
!*
! PRNSOL,S,COMP
! /REPLOT,RESIZE
! LGWRITE,'db','txt','E:\NEWFOL~1\',COMMENT
```