Prediction of Heating Induced Temperature Fields and Distortions in Steel Plates

Bai-Qiao Chen

Dissertation to obtain the degree of Master in

Naval Architecture and Marine Engineering

Jury

President: Prof. Yordan Ivanov Garbatov
Supervisor: Prof. Carlos Antonio Pancada Guedes Soares
Member: Prof. Jose Manuel Antunes Mendes Gordo

December 2011
This page is intentionally left blank.
Acknowledgement

The deepest gratitude goes first and foremost to Prof. Carlos Guedes Soares for his invitation to come to Portugal to join this powerful research group at CENTEC as well as his constant encouragement, guidance and illuminating instruction.

High tribute shall also be paid to Dr. Malabika Adak, who has walked through all the stages of the calculation and writing of the thesis.

To all Professors of the CENTEC, the author is also deeply indebted to them for their direct and indirect help. Special thanks are due to Prof. Yordan Garbatov for all his help and guidance in research.

The author also owes sincere gratitude to the friends and colleagues, Eng. Xueqian Zhou, Eng. S. Saad Eldeen, Eng. Shan Wang, to name a few. Without their listening, discussions and suggestions, this thesis could not have reached its present form.

Finally, the author would like to thank his family for their patient, understanding and support.

Lisbon, July, 2011

Baiqiao Chen
This page is intentionally left blank.
Abstract

This work has developed models and techniques for predicting the temperature distributions and the distortions induced in steel plates by the line heating as well as the welding process. A Gaussian mathematical model of transient thermal process in line heating has been established to simulate the transient thermal analysis with moving heat source model by using finite element method. Finite element models have been developed. Furthermore, a series of load temperature curves for a bead-on-plate weld steel plate have been calculated with suitable convection boundary conditions and non-linear material properties. The time variant temperature fields obtained have been applied as thermal load into mechanical analyses to predict the plate distortions. Results have been compared to numerical and experimental results obtained from previous research. The effect of different parameters on the line heating response, including the torch speeds, heat inputs, thicknesses of plates has been studied.

The consideration of the permanent heating induced effects will allow the appropriate information of shape and amplitude of distortions and residual stresses to conduct the ultimate strength assessment of ship panels.

Keywords: Line heating, Temperature-dependent material properties, Thermal analysis, Temperature distribution, Welding distortion
This page is intentionally left blank.
Resumo

Este trabalho desenvolveu modelos e técnicas para prever as distribuições de temperatura e as distorções induzidas em chapas navais pelo processo de linhas de aquecimento e de soldadura. Um modelo matemático do processo térmico transiente de linhas de aquecimento foi criado para simular a análise térmica transiente com o modelo de fonte de calor em movimento usando o Método dos Elementos Finitos. Os modelos de elementos finitos foram desenvolvidos. Além disso, uma série de curvas de carga vs. temperatura para uma chapa de aço com soldadura ‘bead-on-plate’ foram calculadas com condições de fronteira de convecção adequadas e propriedades não lineares do material. Os campos de temperatura variante no tempo obtidos, foram aplicados como carga térmica nas análises mecânicas de forma a prever as distorções das chapas. Os resultados foram comparados com resultados numéricos e experimentais obtidos em estudos anteriores. O efeito dos diferentes parâmetros sobre a resposta da chapa a linhas de aquecimento, incluindo as velocidades de avanço dos maçaricos, entradas de calor e espessuras das chapas foi estudado.

A consideração dos efeitos permanentes induzidos por aquecimento, permitirá avaliar os valores da resistência última a partir da correcta informação sobre a forma e amplitude das distorções e tensões residuais, respectivas.

**Palavras-chave:** Linhas de aquecimento, Propriedades do material dependente da temperatura, Análise térmica, Distribuição de temperatura, Distorção de soldadura.
# Table of Contents

Acknowledgement ........................................................................................................................... I

Abstract ........................................................................................................................................III

Resumo .......................................................................................................................................... V

Table of Contents ............................................................................................................................ VII

List of Figures ................................................................................................................................ XI

List of Tables .................................................................................................................................. XV

Nomenclatures ................................................................................................................................. XVII

Abbreviations ................................................................................................................................. XIX

1. Introduction ................................................................................................................................................... 1

   1.1 Introduction to Line Heating and Welding ......................................................................................... 1

       1.1.1 Line Heating ................................................................................................................................. 1

       1.1.2 Arc Welding ................................................................................................................................. 2

   1.2 Problem Statement ............................................................................................................................. 3

   1.3 Objectives ........................................................................................................................................... 4

   1.4 Scope of the Work .............................................................................................................................. 4

   1.5 Structure of the Thesis ...................................................................................................................... 4

2. Literature Review ..................................................................................................................................... 7

   2.1 Brief Review ....................................................................................................................................... 7

   2.2 Types of Arc Welding .......................................................................................................................... 9

   2.3 Types of Welding Joints ...................................................................................................................... 10

   2.4 Types of Welding Deformations ....................................................................................................... 11

3. Heat Source Models .............................................................................................................................. 13

   3.1 Rosenthal’s Analytical Model ............................................................................................................ 13

   3.2 Gaussian Distributed Model ............................................................................................................. 14

   3.3 Goldak’s Distributed Model .............................................................................................................. 15

       3.3.1 Semi-Ellipsoidal Model ............................................................................................................... 15

       3.3.2 Double–Ellipsoidal Model ........................................................................................................... 16
6.3 Finite Element Analysis of Distortions after Heating ...................................................55
  6.3.1 Element Type ..................................................................................................56
  6.3.2 “Birth and death” Feature .........................................................................................57
  6.3.3 Mechanical Properties and Boundary Conditions ...................................................57
6.4 Results Discussion ........................................................................................................59
6.5 Parametric Study ..........................................................................................................62
  6.5.1 Effects of Heat Input .................................................................................................62
  6.5.2 Effect of Welding Speed ...........................................................................................63
  6.5.3 Effect of Plate Thickness ..........................................................................................64
  6.5.4 Effect of Boundary Condition ..................................................................................65
7. Conclusions and Recommendations ...............................................................................70
  7.1 Conclusions ..................................................................................................................71
  7.2 Recommendations ........................................................................................................71
Bibliography .......................................................................................................................73
This page is intentionally left blank.
List of Figures

Figure 1.1: Illustration of a fusion weld ................................................................. 3
Figure 2.1: Butt Joint Arrangement ........................................................................ 10
Figure 2.2: Corner Joint Arrangement ................................................................... 10
Figure 2.3: Edge Joint Arrangement ...................................................................... 11
Figure 2.4: Lap Joint Arrangement ......................................................................... 11
Figure 2.5: T Joint Arrangement ............................................................................. 11
Figure 2.6: Types of welding deformations ............................................................. 12
Figure 3.1: Gaussian distributed heat source ......................................................... 14
Figure 3.2: Semi-ellipsoidal distributed heat source .............................................. 15
Figure 3.3: Double-ellipsoidal distributed heat source .......................................... 16
Figure 3.4: Heat flux distributions at top layer (y=0) .............................................. 18
Figure 3.5: Temperature distribution of welded plate (FEM) ............................... 19
Figure 3.6: Temperature history of a reference plane in the welded plate (FEM) .... 19
Figure 3.7: Temperature distributions of the top surface of the welded plate. (a) along transverse direction and (b) along longitudinal direction (FEM) .............. 20
Figure 4.1: Thermal properties of Inconel 600 used by Friedman (1975) .............. 23
Figure 4.2: Mechanical properties of Inconel 600 used by Friedman (1975) ........ 23
Figure 4.3: Dimensions of work piece ................................................................. 24
Figure 4.4: Element mesh .................................................................................... 25
Figure 4.5: Finer element mesh .......................................................................... 25
Figure 4.6: Simulation results in different meshes .................................................. 26
Figure 4.7: Temperature distribution during the welding process ......................... 27
Figure 4.8: Temperature distribution during the cooling process ......................... 28
Figure 4.9: Nodes in the center line (Group 1) ...................................................... 28
Figure 4.10: Temperature results of nodes group 1 .............................................. 29
Figure 4.11: Nodes Group 2 (parallel to Group 1) .................................................. 29
Figure 4.12: Temperature results of nodes group 2 .............................................. 30
Figure 4.13: Nodes Group 3 (in z direction) .......................................................... 30
Figure 4.14: Temperature results of nodes group 3 .................................................................31
Figure 4.15: Nodes Group 4 (in y direction) .............................................................................31
Figure 4.16: Temperature results of nodes group 4 .................................................................31
Figure 4.17: Temperature-distance curve ................................................................................32
Figure 4.18: The effect of heating speed ..................................................................................32
Figure 4.19: The effect of heat input .........................................................................................33
Figure 4.20: The effect of heating speed ..................................................................................33
Figure 4.21: Comparison with Biswas’s result ..........................................................................34
Figure 5.1: Dimensions of work piece ......................................................................................35
Figure 5.2: Element mesh ........................................................................................................37
Figure 5.3: Finer element mesh ................................................................................................37
Figure 5.4: Elastoplastic stress-strain curve .............................................................................38
Figure 5.5: Bauschinger effect ..................................................................................................39
Figure 5.6: Kinematic hardening ...............................................................................................39
Figure 5.7: Nonlinear isotropic hardening stress-strain curve ..................................................41
Figure 5.8: Temperature distribution in time=1s .......................................................................42
Figure 5.9: Temperature distribution in time=5s .......................................................................42
Figure 5.10: Temperature distribution in time=10s .................................................................43
Figure 5.11: Temperature distribution in time=15s .................................................................43
Figure 5.12: Temperature distribution in time=20s .................................................................43
Figure 5.13: Temperature distribution in time=60s .................................................................43
Figure 5.14: Temperature distribution in time=1200s ...............................................................44
Figure 5.15: Temperature distribution in time=6000s ...............................................................44
Figure 5.16: Simulation results in different meshes .................................................................44
Figure 5.17: Temperature results of nodes group 1 .................................................................45
Figure 5.18: Temperature results of nodes group 2 .................................................................45
Figure 5.19: Temperature results of nodes group 3 .................................................................46
Figure 5.20: Temperature results of nodes group 4 .................................................................46
Figure 5.21: Temperature-distance curve ................................................................................47
Figure 5.22: The effect of welding speed ..................................................................................47
Figure 5.23: The effect of heat input.........................................................................................48
Figure 5.24: The effect of welding speed .................................................................................48
Figure 6.1: Deformation of a steel plate during and after welding.............................................49
Figure 6.2: Temperature and Stresses Changes During Welding (Weisman, 1976)..............51
Figure 6.3: Newton-Raphson iterative solution ........................................................................55
Figure 6.4: Mesh of the 3D model ............................................................................................57
Figure 6.5: Boundary condition I...............................................................................................58
Figure 6.6: Flow diagram of transient thermo-mechanical analysis model .........................58
Figure 6.7: Deformation of the plate during welding process ..................................................59
Figure 6.8: Deformation of the plate at the end of welding process ........................................60
Figure 6.9: Deformation of the plate during cooling process ...................................................60
Figure 6.10: Final deformation Style.........................................................................................61
Figure 6.11: Distortion pattern, perpendicular to the welding line (I).................................61
Figure 6.12: Distortion pattern, along the welding line (I).......................................................62
Figure 6.13: Effect of heat input on deformation with respect to time .......................................63
Figure 6.14: Effect of heat input on deformation with respect to X coordinates.......................63
Figure 6.15: Effect of welding speed on deformation with respect to time ...............................64
Figure 6.16: Effect of welding speed on deformation with respect to X coordinates ..........64
Figure 6.17: Effect of plate thickness on deformation with respect to time ............................64
Figure 6.18: Effect of plate thickness on deformation with respect to X coordinates.................65
Figure 6.19: Schematic boundary condition (II)........................................................................66
Figure 6.20: Distortion pattern, perpendicular to the welding line (II&III) ..................66
Figure 6.21: Distortion pattern, along the welding line (II&III) .................................................67
Figure 6.22: Schematic boundary condition (III) ......................................................................67
Figure 6.23: Schematic boundary condition (IV) .....................................................................68
Figure 6.24: Distortion pattern, perpendicular to the welding line (IV&V) .........................68
Figure 6.25: Distortion pattern, along the welding line (IV&V) .................................................69
This page is intentionally left blank.
List of Tables

Table 4.1: Material Properties of C-Mn steel (Biswas et al. 2007) ...........................................24
Table 5.1: Material Properties for Steel 25 (Lu et al. 2001) ......................................................36
## Nomenclatures

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$c$</td>
<td>Specific heat</td>
</tr>
<tr>
<td>$e_i$</td>
<td>Thermal expansion</td>
</tr>
<tr>
<td>$h_f$</td>
<td>Convection heat transfer coefficient</td>
</tr>
<tr>
<td>$k$</td>
<td>Thermal conductivity</td>
</tr>
<tr>
<td>$q$</td>
<td>Heat flux</td>
</tr>
<tr>
<td>$q_{\text{max}}$</td>
<td>Maximum heat flux</td>
</tr>
<tr>
<td>$r$</td>
<td>Distance from heat source center</td>
</tr>
<tr>
<td>$E$</td>
<td>Modulus of elasticity</td>
</tr>
<tr>
<td>$I$</td>
<td>Current</td>
</tr>
<tr>
<td>$[K]$</td>
<td>Stiffness matrix</td>
</tr>
<tr>
<td>$Q$</td>
<td>Heat input</td>
</tr>
<tr>
<td>$R$</td>
<td>Radius of heating spot</td>
</tr>
<tr>
<td>$T$</td>
<td>Body Temperature</td>
</tr>
<tr>
<td>$T_\infty$</td>
<td>Surrounding temperature</td>
</tr>
<tr>
<td>$U$</td>
<td>Voltage</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>Thermal diffusivity</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>Strain / Thermal emissivity</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Stress / Stefan-Boltmann constant</td>
</tr>
<tr>
<td>$\sigma_y$</td>
<td>Yield stress</td>
</tr>
<tr>
<td>$\eta$</td>
<td>Arc efficiency</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density of the material</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>Poisson’s ratio</td>
</tr>
<tr>
<td>$\nabla$</td>
<td>Derivator ($\partial / \partial x, \partial / \partial y, \partial / \partial z$)</td>
</tr>
</tbody>
</table>
This page is intentionally left blank.
<table>
<thead>
<tr>
<th>Abbreviations</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>AC</td>
<td>Alternating electric current</td>
</tr>
<tr>
<td>APDL</td>
<td>ANSYS Parametric Design Language</td>
</tr>
<tr>
<td>BISO</td>
<td>Bilinear isotropic hardening</td>
</tr>
<tr>
<td>BKIN</td>
<td>Bilinear kinematic hardening</td>
</tr>
<tr>
<td>CCT</td>
<td>Continuous cooling transformation</td>
</tr>
<tr>
<td>CHABOCHE</td>
<td>Nonlinear kinematic hardening</td>
</tr>
<tr>
<td>DC</td>
<td>Direct electric current</td>
</tr>
<tr>
<td>FE</td>
<td>Finite element</td>
</tr>
<tr>
<td>FEA</td>
<td>Finite element analysis</td>
</tr>
<tr>
<td>GMA</td>
<td>Gas metal arc</td>
</tr>
<tr>
<td>GMAW</td>
<td>Gas metal arc welding</td>
</tr>
<tr>
<td>GTAW</td>
<td>Gas tungsten arc welding</td>
</tr>
<tr>
<td>HAZ</td>
<td>Heat affected zone</td>
</tr>
<tr>
<td>KINH</td>
<td>Kinematic hardening</td>
</tr>
<tr>
<td>MIG</td>
<td>Metal inert gas</td>
</tr>
<tr>
<td>MISO</td>
<td>Multilinear isotropic hardening</td>
</tr>
<tr>
<td>MKIN</td>
<td>Multilinear kinematic hardening</td>
</tr>
<tr>
<td>MMA</td>
<td>Manual metal arc (welding)</td>
</tr>
<tr>
<td>PDE</td>
<td>Partial differential equation</td>
</tr>
<tr>
<td>SAW</td>
<td>Submerged arc welding</td>
</tr>
<tr>
<td>SMAW</td>
<td>Shielded metal arc welding</td>
</tr>
<tr>
<td>TIG</td>
<td>Tungsten inert gas</td>
</tr>
</tbody>
</table>
This page is intentionally left blank.
1. Introduction

1.1 Introduction to Line Heating and Welding

Line heating / welding is the fundamental process in manufacturing marine structures. Line heating / welding deformation is, however, inevitable due to non-uniform distribution of temperature and plastic yielding during welding / line heating. It depends on various factors such as plate thicknesses, welding speed, heat input, material properties etc. The deformation due to line heating / welding is one of the principal obstacles in enhancing the productivity in manufacturing procedure of marine structures. This should be much more seriously considered in the case of steel plates with thickness found in ship and offshore structures. It is most important to know how much deformation causes due to line heating / welding with different welding parameters.

In the present work a computationally efficient solution strategy for three-dimensional transient analysis of thermal and subsequent mechanical response due to submerged arc welding as well as line heating is presented. The effect of parameters (plate thicknesses, welding speed, heat input, and material properties) during welding and line heating is studied.

Non-uniform thermo-elastic-plastic problem is solved by the Finite Element Method to predict the temperature distribution and deformation due to line heating and welding with temperature dependent material properties, Newton’s convection boundary conditions and moving heat input (Gaussian heat source model). The results compared very well with the previous research result.

1.1.1 Line Heating

The line heating method is a popular technique used in shipyards to form ship hull pieces. Even recently, skilled workers have performed the forming process by their empirical intuition and so no systematic way to effective formation has been found. The automation of it can reduce working time, rework costs, and inferior hull pieces; this is so attractive to shipbuilders who wish higher productivity of ships.

Line heating is proved to be usually quicker and more accurate than methods using heavy machinery. One shipyard found that the costs associated with furnacing and pressing amounted to 60 man-hours per plate to build plate forming jigs and 17 man-hours of actual forming time per plate. This dropped to just ten hours of forming time with no time for heavy jigs required at all. In addition, the improved accuracy made other assembly tasks easier and reduced rework. Line heating also gives small steel boat builders the tools to build much more complicated shapes with only minor investment in new equipment - a couple of hundred dollars at the most. As a result of these benefits, many shipyards, including some of the most advanced Japanese and European yards, use line heating almost exclusively for forming shell plate.
The idea is to use heat distortion, normally a problem in metal boat construction, to the builder's advantage. Heat causes steel to expand. If the expansion is restrained in one direction, it will expand more in the other directions.

When a small spot on a steel plate is heated to a temperature above about 1100 degrees F, it expands and also becomes soft enough to be readily distorted. By cooling the steel surrounding it, the hot spot is prevented from expanding sideways. It will only be able to expand upwards, and will do so readily because it is softened, thereby making a lump. When the steel subsequently cools and hardens, this lump will remain, so that the surrounding steel will be pulled in to fill the space the lump has left. As a result, the steel plate will distort. If a line on a plate has been heated and cooled, there will be a line of such lumps and the plate will bend. The edges will rise towards the lump. This is the basic principle of line heating.

Line heating is also used for removing distortions. The bends can be removed the same way it being produced, just by heating the opposite side. Another common use is removing buckles or other out-of-plane distortion in flat panels, especially after welding. The welding heat around the edges, or along stiffeners, has shrunk the panel at those points and the extra metal left over in the middle is bulging. Removing this extra metal by line heating patterns in the middle of the panel. One common pattern is "pine leaf" consisting of scattered short lines like pine needles scattered on the panel. The "hungry horse" distortion produced by welding stiffeners can also be removed by lines parallel to the stiffener.

The ability to form steel by simple heating also means that it is possible to correct minor inaccuracies. This means that exact fits between premade construction modules can be obtained by line heating. This in turn reduces the risk in modular construction, so it is more feasible. Since modular construction has significant benefits for reducing labor, and especially for home builders, reducing facility costs, this is potentially an important benefit.

In a word, line heating is an important tool for builders working in steel that increases the range of shapes they can produce and reduces labor.

### 1.1.2 Arc Welding

Arc welding, which is heat-type welding, becomes one of the most important manufacturing operations for the joining of structural elements for a wide range of applications, including guide way for trains, ships, bridges, building structures, automobiles, and nuclear reactors, to name a few.

Arc welding is a type of welding that uses a welding power supply to create an electric arc between an electrode and the base material to melt the metals at the welding point. It requires a continuous supply of either direct (DC) or alternating (AC) electric current, which create an electric arc to generate enough heat to melt the metal and form a weld, and consumable or non-consumable electrodes. The welding region is sometimes protected by some type of inert or semi-inert gas, known as a shielding gas, and/or an evaporating filler material. The process of arc welding is widely used because of its low capital and running costs. Getting the arc
started is called striking the arc. An arc may be struck by either lightly tapping the electrode against the metal or scratching the electrode against the metal at high speed.

To sum up, the arc welding process is a remarkably complex operation involving extremely high temperatures, which produce severe distortions and high levels of residual stresses. These extreme phenomena tend to reduce the strength of a structure, which becomes vulnerable to fracture, buckling, corrosion and other type of failures.

During the welding process, a liquid weld pool is created through the interaction of an intense heat source and the substrates being joined (see in Figure 1.1). Melting on the front side of the weld pool eliminates the interface between the materials, while solidification on the back side of the weld pool fuses the substrates together to create a solid joined part. Surrounding the fusion zone is a heat-affected zone, where the substrate is heated to temperatures up to the melting point of the metal being joined. Solidification in the fusion zone and solid-state phase transformations in the heat-affected zone are responsible for dramatic changes in the microstructure and properties of the welding joint.

Figure 1.1: Illustration of a fusion weld

1.2 Problem Statement

The problem of welding distortion during large steel fabrications leads to dimensional inaccuracies and misalignments of structural members, which can result in corrective tasks or rework when tolerance limits are exceeded. This in turn, increases the cost of production and leads to delays. In fabrication industries, for example, expenses for rework such as straightening could cost a lot. Therefore, the problems of distortion and residual stresses are always of great concern in welding industry.

In order to deal with this problem, it is necessary to predict the amount of distortion resulting from the welding operations. One way to predict the distortion and shrinkage of steel welding is through numerical analysis such as finite element analysis (FEA). Once the techniques to
predict the distortion and shrinkage are identified, then the problems can be controlled accordingly.

Within the welding procedures, there are many factors such as welding process type, welding process parameters, welding sequence, preheat patterns, level of constraint and joint details that contribute to the distortion of the welded structure. Knowing which parameters have an effect on the quality of the weld and which parameters give the most significant effect on the weld quality are the main issues in welding industry.

1.3 Objectives

The objective of this work is to be able to predict numerically the temperature distribution and weld induced distortions in simple components of steel ship structures.

Numerical models in finite elements are developed and validated with available data of temperature distribution in steel plates as a result of heating or welding procedures. The weld induced distortions are calculated and compared with published results.

Parametric studies are made to check the effect of welding conditions and boundary conditions on the resulting distortions.

1.4 Scope of the Work

The actual problem is that in shipbuilding, inaccuracies occur during the heating or welding process of the production and increase partly from the thermal distortions and partly in the form of dimensional variations due to human factors.

Furthermore, with the increasing use of automation, it is supposed to be attractive to be able to quantify and calculate the thermal distortions by means of mathematical models, so that the required tolerances of the automation process can be achieved as efficient as possible.

In this thesis, the responses of single pass butt-joint of arc welding are evaluated through the finite element code ANSYS®. Since this is a first attempt on the thermal distortion and the residual stresses, several simplifications have been considered, such as, the phase change are not taken into consideration, to assume that the welding rod is of the same material as the substrate, which have to be correct in the future work to make more accurate result closer to the reality. The heat source used in the calculation is calculated with the Gaussian model, while one result of the ellipsoidal source model is presented. The double ellipsoidal source model is supposed to be utilized in the continuous work.

The expected achievement from this research are, to improve planning and work scheduling by reducing the rework, to reduce the production cost significantly by reducing the measurements and rework, and to improve the quality of the ship production.

1.5 Structure of the Thesis
Introduction to the subject is included in Chapter 1.

Chapter 2 includes the brief literature reviews of the work of previous researchers, descriptions of the types of arc welding, welding joints, and types of welding deformations.

A brief description of different heat source models proposed from 70 years ago until recently 20 years is given in Chapter 3.

The beginning of Chapter 4 outlines the theory of the heat transfer, then the highly temperature dependent thermal properties are introduced. Based on the finite element method, numerical results are achieved and compared with the numerical and experimental results of previous researchers. At the end of this Chapter, a parametric study is performed. An example of the comparison between the Gaussian distributed model and the ellipsoidal model is presented as well.

Chapter 5 is also about finite element analysis but it concentrates on the butt-joint welding process. The mesh style, temperature dependent mechanical properties and meshing are discussed. And parametric studies based on numerical results are carried out for different parameters including welding speed, plate thickness and heat input.

Furthermore, the thermal distortion is discussed in Chapter 6. Finite element analyses are conducted to predict the welding induced distortion. Parametric study is included at the end of this chapter.

Finally, discussions and conclusions are presented in Chapter 7.
This page is intentionally left blank.
2. Literature Review

2.1 Brief Review

The research activity in welding simulation started decades ago. Rosenthal (1946) was among the first researchers to develop an analytical solution of heat flow during welding based on conduction heat transfer for predicting the shape of the weld pool for two and three-dimensional welds. Using the Fourier partial differential equation (PDE) of heat conduction, he introduced the moving coordinate system to develop solutions for the point and line heat sources and applied this successfully to address a wide range of welding problems. His analytical solutions of the heat flow made possible for the first time the analysis of the process from a consideration of the welding parameters namely the current, voltage, welding speed, and weld geometry.

Understanding of the theory of heat flow is essential in order to study the welding process analytically, numerically or experimentally. Since the pioneering work of Rosenthal, considerable interest in the thermal aspects of welding was expressed by many researchers such as Kamala and Goldak (1993), Nguyen et al. (1999), and Komanduri and Hou (2000).

The most critical input data required for welding thermal analysis are the parameters necessary to describe the heat input to the weldment from the arc. Goldak et al. (1984) derived a mathematical model for welding heat sources based on a Gaussian distribution of power density. They proposed a double ellipsoidal distribution in order to capture the size and shape of the heat source of shallow and deeper penetrations. Some researchers have also developed the thermal finite element simulation to investigate the temperature distribution of a metal such as Kraus (1986), Tekriwal and Muzumder (1988), Yeung and Thorthon (1999), and Bonifaz (2000).

Over the past forty years, finite element methods have been used extensively in an attempt to predict distortion and residual stresses due to welding operations such as the studies by Friedman (1975), Michaleris and Debbicari (1997) and Taylor et al. (1999). Generally, the finite element method has already been proven to be a successful tool to simulate the complex welding process as performed by Friedman (1975). His 2-D finite element analysis work was then used by Taylor et al. (1999) to verify their 3-D computational modeling of welding phenomena. The results of finite element analysis done by Taylor et al. (1999) were in reasonable agreement with the result obtained by Friedman (1975). The Freidman’s work has also been used in this thesis as a verification finite element model.

Most of the welding research in the past was conducted to investigate the distribution of residual stress and distortion of welded metal. The work performed by Mandal and Sundar (1997) for example, estimates the welding shrinkage in a welded butt joint by applying a mathematical model approach. Michaleris and Debbicari, and Okumoto (1997, 1998)
conducted thermo-elasto-plastic finite element analysis for welding simulation to predict the welding distortion. They have claimed that their approaches have been proven consistent to experimental and empirical data. Furthermore, Puchaicela (1998) in his article reviewed and analyzed several formulas and figures in an attempt to provide a practical guide for the control and reduction of distortion.

Not only the welding residual stress and distortion have been studied by welding researchers, but the effects of welding parameter, welding sequence, welding joint geometry, and root opening has also been investigated by several researchers in the past.

Harwig et al. (1999) for instance, studied the effect of welding parameters and electrode classification on the diffusible hydrogen content of gas shielded flux cored arc welds. In 1999, Tsai et al. studied the effect of welding sequence on buckling and warping behavior of a thin-plate panel structure. Tsai et al. (2001) have also investigated the effects of welding parameters and joint geometry on the magnitude and distribution of residual stresses on thick-section butt joints. The effect of the root opening on mechanical properties, deformation and residual stress has been reported by Jang et al. (2001).

As shown by the above researchers, residual stress distribution and distortion in a welded plate are strongly affected by many parameters and by their interaction. Yet, certain aspects of the welding phenomenon are still subject to further research specifically the effects of heat input, welding speed, restraint, plate curvature, and gap on arc welding responses as applied to curved steel plate welding.

In the recent ten years, there is growing concern about the numerical research on line heating and welding process. Murakawa et al. (2005) predicted the hot cracking of a weld using temperature-dependent interface elements. Mahapatra et al. (2007) modelled the effect of the position of tack weld constraints on the angular distortions created in one-sided fillet welding created by SAW. It is clear from this literature review that the thermal history and the resulting distortions in a welded joint are strongly affected by various parameters and their interactions. A number of finite-element (FE) models aimed at illustrating the effect of using different modelling strategies for the simulation of the thermo-elasto-plastic stages of the welding process are presented by Mollicone et al. (2006).

Adak and Mandal (2010) studied the heat sinking as a method of distortion mitigation and used the pseudo-linear equivalent constant rigidity concept for thermo-mechanical analysis of plates undergoing welding with simultaneous heat sinking. The proposed concept was found to be computationally more efficient and simpler to model compared to FEM for solving similar thermo-elasto-plastic nonlinear problems.

In 2011, Heinze et al. investigated a single-layer gas metal arc (GMA) weld of 5 mm thick structural steel is experimentally and numerically. The numerical modeling begun with a mesh analysis based on modal analyses. The sensitivity of welding-induced distortion is examined regarding different continuous cooling transformation (CCT) diagrams.
### 2.2 Types of Arc Welding

One of the most common types of arc welding is shielded metal arc welding (SMAW), which is also known as manual metal arc welding (MMA) or stick welding. An electric current is used to strike an arc between the base material and a consumable electrode rod. The electrode rod is made of a material that is compatible with the base material being welded and is covered with a flux that protects the weld area from oxidation and contamination by producing CO2 gas during the welding process. The electrode core itself acts as filler material, making the separate filler unnecessary. The process is very versatile, requiring little operator training and inexpensive equipment. The versatility of the method makes it popular in a number of applications including repair work and construction.

Gas metal arc welding (GMAW), commonly called MIG (Metal Inert Gas), is a semi-automatic or automatic welding process with a continuously fed consumable wire acting as both electrode and filler metal, along with an inert or semi-inert shielding gas flowed around the wire to prevent the weld site from contamination. Constant voltage, direct current power source is most commonly used with GMAW, but constant current and alternating current are used as well. With continuously fed filler electrodes, GMAW offers relatively high welding speeds. MIG welding has been used successfully in industries like aircraft, automobile, pressure vessel, and ship building. However, the more complicated equipment reduces convenience and versatility in comparison to the SMAW process.

Submerged arc welding (SAW) is a high-productivity automatic welding method in which the arc is struck beneath a covering layer of flux. This increases arc quality, since contaminants in the atmosphere are blocked by the flux. The slag that forms on the weld generally comes off by itself and, combined with the use of a continuous wire feed, the weld deposition rate is high. Working conditions are much improved over other arc welding processes since the flux hides the arc and no smoke is produced. The process is commonly used in industry, especially for large products. As the arc is not visible, it requires full automatization. In-position welding is not possible with SAW.

Gas tungsten arc welding (GTAW), or tungsten inert gas (TIG) welding, is a manual welding process that uses a non-consumable electrode made of tungsten, an inert or semi-inert gas mixture, and a separate filler material. Especially useful for welding thin materials, this method is characterized by a stable arc and high quality welds, but it requires significant operator skill and can only be accomplished at relatively low speeds. It can be used on nearly all weldable metals, though it is most often applied to stainless steel and light metals. It is often used when quality welds are extremely important, such as in bicycle, aircraft and naval applications.

A related process, plasma arc welding, also uses a tungsten electrode but uses plasma gas to make the arc. The arc is more concentrated than the GTAW arc, making transverse control more critical and thus generally restricting the technique to a mechanized process. Because of its stable current, the method can be used on a wider range of material thicknesses than the
GTAW process and is much faster. A variation of the process is plasma cutting, an efficient steel cutting process.

Other arc welding processes include atomic hydrogen welding, carbon arc welding, electroslag welding, electrogas welding, and stud arc welding.

2.3 Types of Welding Joints

Welds are made at the junction of the various pieces that make up the weldment. The junctions of parts, or joints, are defined as the location where two or more members are to be joined. Parts being joined to produce the weldment may be in the form of rolled plate, sheet, pipes, castings, forgings, or billets. The five basic types of welding joints for plates are listed below.

Butt Joint: A joint between two members lying approximately in the same plane as shown in Figure 2.1.

![Figure 2.1: Butt Joint Arrangement](image)

Corner Joint: A joint between two members located approximately at right angles to each other in the form of an angle as shown in Figure 2.2.

![Figure 2.2: Corner Joint Arrangement](image)

Edge Joint: A joint between the edges of two or more parallel or mainly parallel members as shown in Figure 2.3.
Lap Joint: A joint between two overlapping members as shown in Figure 2.4.

T Joint: A joint between two members located approximately at right angles with respect to each other in the form of a T as shown in Figure 2.5.

2.4 Types of Welding Deformations

During welding, there are non-uniform heating and cooling cycles in the weld and adjacent base metal, which causes complex thermal strains. The stresses resulting from the strains produce internal forces causing shrinkage of the material. The stresses that would exist in a weldment after all external loads are removed are called residual stresses. Depending on the shrinkage pattern and the shape of the structure welded, various deformations such as bending, buckling and rotation take place.

Generally, welding deformations can be classified into in-plane deformation such as transverse shrinkage, longitudinal shrinkage, rational distortion, and out-of-plane deformation
including angular distortion, buckling distortion and longitudinal bending distortion, as shown in Figure 2.6. In principle, there are four fundamental types of deformations, namely the longitudinal shrinkage, the transverse shrinkage and the angular distortions in two directions and various forms of distortions are produced as their combination. These four types of deformation can be regarded as fundamental components of the deformations due to welding.

Figure 2.6: Types of welding deformations
3. Heat Source Models

3.1 Rosenthal's Analytical Model

The research activity in heating and welding simulation started decades ago. Rosenthal (1941) was among the first researchers to develop an analytical solution of heat flow during welding based on conduction heat transfer for predicting the shape of the weld pool for two and three-dimensional welds. Using the Fourier partial differential equation (PDE) of heat conduction, he introduced the moving coordinate system to develop solutions for the point and line heat sources and applied this successfully to address a wide range of welding problems. His analytical solutions of the heat flow made possible for the first time the analysis of the process from a consideration of the welding parameters namely the current, voltage, welding speed, and weld geometry. According to the thickness, shape and dimensions of the weldment and the pattern of heat transfer during welding process, Rosenthal simplified the welding heat source into three types: point type, line type and area type.

In the case of surfacing bead welding in thick weldment, where the heat transfers in all three directions, the heat source can be considered as a point source. The temperature field of this instantaneous concentrated point source can be obtained from the following formula.

\[
T = \frac{2Q}{c\rho(4\pi\alpha)t^{1/2}} \exp\left(-\frac{D^2}{4\alpha}\right)
\]

(3-1)

Where \( Q \) is the instantaneous heat energy offered by the heat source, \( Q = \eta IU \), \( \eta \) is the efficiency of the heat source, \( I \) is the welding current and \( U \) is the welding voltage, \( \alpha \) is the thermal diffusivity, \( D \) is the distance away from the center of the heat source, \( Q = (x^2 + y^2 + z^2)^{1/2} \).

In the case of welding in infinite and thin plate, the temperature deviation in the direction along the plate thickness can be ignored, so the heat propagates to two directions. The heat source can be regarded as a line along the plate thickness. The temperature field in this case can be calculated from the following equation.

\[
T = \frac{Q}{4\pi \alpha \eta t} \exp\left(-\frac{d^2}{4\alpha}\right)
\]

(3-2)

where \( d \) is the distance away from the line source, \( d = (x^2 + y^2)^{1/2} \).

In the case of butt-welding in bars and welding robs, the temperature distributes uniformly in the section of the bar which is equivalent to heat propagation of a small area with constant temperature. The heat source can be defined as area source, and its temperatures field can be expressed as follows.
\[ T = \frac{Q}{c\rho A(4\pi\alpha\tau)^{3/2}} \exp\left(-\frac{d^2}{4\alpha\tau}\right) \]  
\[ (3-3) \]

where \( A \) is the area of the section and \( \tau \) is the distance away from the area source.

His analytical solution (Rosenthal 1946) is based on the concentration heat source, but it did not consider the changes of material properties with temperature, the phase change, and latent heat. Hence, the result deviation in the heat affect zone is relatively big. Nevertheless, owing to its acceptable accuracy in the low temperature zone and its simplicity, it has been widely used in engineering applications.

### 3.2 Gaussian Distributed Model

Since the pioneering work of Rosenthal, Friedman (1975) proposed to apply the Gaussian Distributed heat source to approximately express the heat flux in heating spot, as shown in Figure 3.1.

The heat flux \( q(r) \), with a distance \( r \) from the heat source center, can be expressed as:

\[ q(r) = q_{\text{max}} \exp\left(-\frac{3r^2}{R^2}\right) = \frac{3Q}{\pi R^2} \exp\left(-\frac{3r^2}{R^2}\right) \]

\[ (3-4) \]

where \( q_{\text{max}} \) is the maximum heat flux in the center of the heat source; \( Q \) is the heat input energy of the arc; and \( R \) is the radius of the heating spot.

For the normal welding process, such as GTAW and SMAW, the Gaussian distributed heat source modal is proved to provide precise enough results.

![Figure 3.1: Gaussian distributed heat source](image-url)
3.3 Goldak’s Distributed Model

3.3.1 Semi-Ellipsoidal Model

However, for the welding process in which the momentum effect of arc is considerably large, the disadvantage of the Gaussian distribution model in the resulting accuracy appears due to neglecting the effect of the arc stiffness.

To solve this problem, Goldak et al. (1985) initially proposed a semi-ellipsoidal heat source in which heat flux is distributed in a Gaussian manner throughout the heat source’s volume.

As can be seen in Figure 3.2, the semi-axes of the ellipsoid are $a_h, b_h, c_h$. Heat flux reaches the maximum value in the center of heat source, whose distribution is given by the following equation.

\[
q(x, y, z) = q_{max} \exp(-Ax^2 - By^2 - Cz^2)
\]  

(3-5)

where A, B, C are the heat flow distribution coefficient.

Figure 3.2: Semi-ellipsoidal distributed heat source

Since heat flow is distributing inside the semi-ellipsoid in the weldment surface, the heat flux can be expressed as:

\[
Q = \eta IU = 4 \iiint_{0}^{\infty} q(x, y, z) dx dy dz
\]

\[
= 4q_{max} \int_{0}^{\infty} \exp(-Ax^2) dx \int_{0}^{\infty} \exp(-By^2) dy \int_{0}^{\infty} \exp(-Cz^2) dz
\]

\[
= 4q_{max} \left( \frac{1}{\sqrt{A} \sqrt{\pi}} \right) \left( \frac{1}{\sqrt{B} \sqrt{\pi}} \right) \left( \frac{1}{\sqrt{C} \sqrt{\pi}} \right)
\]

\[
= \frac{q_{max} \pi \sqrt{\pi}}{2 \sqrt{ABC}}
\]

(3-6)
where

\[ q_m = \frac{2Q\sqrt{ABC}}{\pi \sqrt{\pi}} \]  \hspace{1cm} (3-7)

Assume that 95% of the heat energy produces inside the semi-ellipsoid, as

\[ q(a_h, 0, 0) = q_m \exp(-Aa_h^2) = 0.05q_m \]  \hspace{1cm} (3-8)

Thus,

\[ A = \frac{3}{a_h^2} \]  \hspace{1cm} (3-9)

Also,

\[ B = \frac{3}{b_h^2}, \quad C = \frac{3}{c_h^2} \]  \hspace{1cm} (3-10)

Substitute Eq.(3-6) and Eq.(3-7) into Eq.(3-5), the heat flux \( Q(x, y, z) \) at a point \((x, y, z)\) within the semi-ellipsoid is given by the following equation:

\[ q(x, y, z) = \frac{6\sqrt{3Q}}{a_h b_h c_h \pi \sqrt{\pi}} \exp\left(-\frac{3x^2}{a_h^2} - \frac{3y^2}{b_h^2} - \frac{3z^2}{c_h^2}\right) \]  \hspace{1cm} (3-11)

3.3.2 Double–Ellipsoidal Model

The experience with this heat source showed that the predicted temperature gradients in front of the arc were less steep than experimentally observed ones and gradients behind the arc were steeper than those measured. To overcome this, they combined two semi-ellipsoids and proposed a new heat source called double ellipsoidal heat source, as shown in Figure 3.3.

Since two different semi-ellipsoids are combined to give the new heat source, the heat flux within each semi-ellipsoid are described by different equations.
For a point within the first semi-ellipsoid located in front of the welding arc, the heat flux equation is described as:

\[ q_f(x, y, z) = \frac{6\sqrt{3}(f_f Q)}{abc \sqrt{\pi}} \exp \left( -\frac{3x^2}{a^2} - \frac{3y^2}{b^2} - \frac{3z^2}{c_f^2} \right), \quad x \geq 0 \]  

(3-12)

where \( f_f \) is the heat input proportion in the front part.

The heat input is equal to:

\[
2\iiint_{0}^{\infty} q_f(x, y, z) dx dy dz \\
= \frac{6\sqrt{3} f_f Q}{abc \pi \sqrt{\pi}} \int_{0}^{\infty} \exp \left( -\frac{3x^2}{a^2} \right) dx \int_{0}^{\infty} \exp \left( -\frac{3y^2}{b^2} \right) dy \int_{0}^{\infty} \exp \left( -\frac{3z^2}{c_f^2} \right) dz \\
= 2 \frac{6\sqrt{3} f_f Q}{abc \pi \sqrt{\pi}} \frac{\sqrt{\pi}}{2} \frac{\sqrt{\pi}}{2} \frac{\sqrt{\pi}}{2} \frac{\sqrt{\pi}}{2} = \frac{1}{2} (f_f Q) 
\]

(3-13)

For points \((x, y, z)\) within the second semi-ellipsoid, covering the rear section of the arc, the heat flux equation is described as:

\[ q_r(x, y, z) = \frac{6\sqrt{3}(f_r Q)}{abc \sqrt{\pi}} \exp \left( -\frac{3x^2}{a^2} - \frac{3y^2}{b^2} - \frac{3z^2}{c_r^2} \right), \quad x < 0 \]  

(3-14)

where \( f_r \) is the heat input proportion in the rear part.

Also,

\[
2\int_{0}^{\infty} \int_{0}^{\infty} \int_{0}^{\infty} q_r(x, y, z) dx dy dz = \frac{1}{2} (f_r Q) 
\]

(3-15)

Since

\[ \eta I U = Q = \frac{1}{2} (f_f Q) + \frac{1}{2} (f_r Q) = \frac{1}{2} Q (f_f + f_r) \]  

(3-16)

It is obviously that,

\[ f_f + f_r = 2 \]  

(3-17)

Then in the past twenty years, considerable interest in the thermal aspects of welding was expressed by many researchers, such as Michaleris and DeBicacci (1997), Wahab et al. (1998), and Gery et al. (2005).

### 3.4 Modelling Heat Source using FEM

The modelling of a moving heat source is a typical transient process. During the moving of the heat source, the heat energy keeps constant, but the location of the center of the heat source changes from time to time.
Once the finite element model established, the selected heat source model can be applied in the model as a function discussed before. The principle is that the parameters in the function which stand for the locations of the heat source are supposed to be changed with time. All the outer surface of the finite element model are subjected to heat convection, while the elements inside geometric heat source model defined by the function are subjected to heat flux instead.

The idea of time loop is adopted, which means the calculated result of the present time step is used as the initial conditions of the next time step. The time interval for each time step can be calculated as the dimension of the plate in welding direction divided by the welding speed.

Therefore, at each time step, the parameters of the heat source function and the heat source center change with time, the moving of the heat source can be well simulated.

In the present work, the finite element (FE) package ANSYS® has been used to simulate the line heating and welding process. The simulation of the heat source is accomplished by a code editing with ANSYS Parametric Design Language (APDL).

Figure 3.4 gives an example of the double ellipsoidal heat source model. It plots the heat distributions at the top layer of a welded plate, in which the welding parameters are 288 A current, 25 V voltage, 90% efficiency, 1 m/min speed and the heat source parameters are $a=b=4 \text{ mm}$, $c_r=4 \text{ mm}$, $c_f=16 \text{ mm}$, $f_f=0.6$, $f_r=1.4$ (Gery et al. 2005).

Figure 3.5 shows the temperature distribution of the welded plate when the welding torch passed the position at the coordinate of 300 mm in the welding direction. A symmetrical finite element mesh of a half plate is considered. The size of the plate is 335 mm * 120mm and the thickness of the plate is 3 mm. The filler material is considered as part of the plate. A fine mesh is used in the welding area in order to apply more accurate heat input when the moving heat source passes the area at specific time steps.
A detailed temperature distribution in a reference plane is shown in Figure 3.6, when the heat source passes the plane at time $t_0$ and the subsequent time steps. It is clear that heat input in the welding pool is transferred quickly first in the thickness direction and then in the width direction to reach uniform distributions. This group of graphs can be used to predict the boundary of the fusion zone (HZ) and the heat affected zone (HAZ).
Figure 3.7 shows the temperature distributions at top surface of the plate of the reference plane along the transverse direction, as well as that along the longitudinal direction at top surface of the plate. It is concluded that the temperature in the fusion zone (FZ) is high when the welding torch passed the plane but then it decreased rapidly with time. It also displays a sharp temperature drop close to the fusion zone in Figure 3.7(b). However, in the areas far away from the heating center, the effect of the heat decreases very fast. The effect disappears when the distance reaches 15.5 mm.

![Temperature distributions](image)

The study shows that the temperature distributions highly depend on the welding parameters as well as the heat source parameters. More detailed sensitivity analyses are performed in Chapter 4 and 5.
4. Temperature Distribution Field

4.1 Theory of Heat Transfer

Welding is a process with sharply local heating to high temperature and rapid cooling afterwards, during which the temperature highly depends on the time and location, while the mechanical properties of the material varies with the temperature, and melting, phase changing and latent heat exist. To sum up, the weld induced temperature field analysis is a typical nonlinear transient heat transfer issue.

During the welding process, the temperatures of different parts of the weldment vary immensely due to the local heating. Consequently, heat transfer occurs both inside the weldment and between the weldment and surrounding.

Heat transfer mechanisms can be grouped into 3 broad categories:

4.1.1 Conduction

Regions with greater molecular kinetic energy will pass their thermal energy to regions with less molecular energy through direct molecular collisions, a process known as conduction. In metals, a significant portion of the transported thermal energy is also carried by conduction-band electrons.

In general, the rate of the heat transfer through conduction is given by the Fourier equation.

\[ q^* = -k \nabla T \]  \hspace{1cm} (4-1)

where \( q^* \) is the heat flux, \( k \) is the thermal conductivity of the material, \( T \) is temperature and \( \nabla = (\frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z}) \).

The conservation of energy in a differential form can be written as:

\[ \rho c \frac{\nabla T}{\nabla t} = \frac{\partial}{\partial x} (k \frac{\partial T}{\partial x}) + \frac{\partial}{\partial y} (k \frac{\partial T}{\partial y}) + \frac{\partial}{\partial z} (k \frac{\partial T}{\partial z}) + \dot{Q} \] \hspace{1cm} (4-2)

where, \( \rho \) is the density of the material, \( c \) is specific heat, \( \dot{Q} \) is heat generated inside the element and \( t \) is time.

If the thermal conductivity of the material, \( k \) is assumed constant, then Eq.(4-2) becomes:

\[ \frac{\nabla T}{\nabla t} = \alpha (\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}) + \dot{Q} \] \hspace{1cm} (4-3)

where \( \alpha \) is thermal diffusivity \( (\alpha = \frac{k}{c\rho}) \).
Eq. (4-3) is the differential equation of heat conduction for a stationary, homogeneous, isotropic solid with constant thermal conductivity, $k$. The solution to this equation is strongly dependent on the boundary conditions and initial conditions.

### 4.1.2 Convection

When heat conducts into a static fluid it leads to a local volumetric expansion. As a result of gravity-induced pressure gradients, the expanded fluid parcel becomes buoyant and displaces, thereby transporting heat by fluid motion (i.e. convection) in addition to conduction. Such heat-induced fluid motion in initially static fluids is known as free convection.

The boundary conditions for the heat transfer coefficient are divided into radiation and convection. The later is given as,

$$ q_{\text{con}}^* = h_f \cdot (T - T_\infty) \tag{4-4} $$

where, $h_f$ is convection heat transfer coefficient, $T$ is body temperature, $T_\infty$ is surrounding temperature.

### 4.1.3 Radiation

All materials radiate thermal energy in amounts determined by their temperature, where the energy is carried by photons of light in the infrared and visible portions of the electromagnetic spectrum. When temperatures are uniform, the radioactive flux between objects is in equilibrium and no net thermal energy is exchanged. The balance is upset when temperatures are not uniform, and thermal energy is transported from surfaces of higher to surfaces of lower temperature.

The radiation heat transfer formula is

$$ q_{\text{rad}}^* = \sigma \varepsilon \cdot (T^4 - T_\infty^4) \tag{4-5} $$

where, $\varepsilon$ is the emissivity, $\sigma$ is the Stefan-Boltmann constant, $T$ is body temperature, $T_\infty$ is surrounding temperature.

Understanding of the theory of heat flow is essential in order to study the welding process analytically, numerically or experimentally.

### 4.2 Thermal Material Properties

The material properties of metal, for instance, the specific heat, thermal conductivity, modulus of elasticity and yield stress, vary with the change of temperature. The average values can be used in calculations if the temperature does not vary too much. While in the welding process, the temperature of the weldment varies shapely. In this case, to neglect the material properties differences by temperature will result in big deviations. Consequently, the temperature-dependent material properties are supposed to be taken into consideration in welding process simulation.
Quite a few researchers have addressed this topic, Tsai and Eagar (1983), Guedes Soares et al. (1998) and Adak and Mandal (2010), to name a few. For instance, the temperature-dependent material properties used in Friedman (1975)'s analyses are plotted in Figure 4.1 and 4.2, respectively.

![Figure 4.1: Thermal properties of Inconel 600 used by Friedman (1975)](image)

![Figure 4.2: Mechanical properties of Inconel 600 used by Friedman (1975)](image)

Table 4.1 lists both thermal and mechanical properties of C-Mn steel which was used in the study of Biswas et al. (2007).
Table 4.1: Material Properties of C-Mn steel (Biswas et al. 2007)

<table>
<thead>
<tr>
<th>Temperature</th>
<th>°C</th>
<th>0</th>
<th>100</th>
<th>300</th>
<th>450</th>
<th>550</th>
<th>600</th>
<th>720</th>
<th>800</th>
<th>1450</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermal conductivity</td>
<td>W m⁻¹ °C⁻¹</td>
<td>51.9</td>
<td>51.1</td>
<td>46.1</td>
<td>41.1</td>
<td>37.5</td>
<td>35.6</td>
<td>30.6</td>
<td>26.0</td>
<td>29.5</td>
</tr>
<tr>
<td>Specific heat capacity</td>
<td>J kg⁻¹ °C⁻¹</td>
<td>450</td>
<td>499</td>
<td>566</td>
<td>631</td>
<td>706</td>
<td>773</td>
<td>1080</td>
<td>931</td>
<td>438</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>-</td>
<td>0.28</td>
<td>0.31</td>
<td>0.33</td>
<td>0.34</td>
<td>0.36</td>
<td>0.37</td>
<td>0.37</td>
<td>0.42</td>
<td>0.47</td>
</tr>
<tr>
<td>Thermal expansion coefficient</td>
<td>10⁻⁶ °C⁻¹</td>
<td>10</td>
<td>11</td>
<td>12</td>
<td>13</td>
<td>14</td>
<td>14</td>
<td>14</td>
<td>14</td>
<td>15</td>
</tr>
<tr>
<td>Modulus of elasticity</td>
<td>GPa</td>
<td>200</td>
<td>200</td>
<td>200</td>
<td>150</td>
<td>110</td>
<td>88</td>
<td>20</td>
<td>20</td>
<td>2</td>
</tr>
<tr>
<td>Film coefficient</td>
<td>W m⁻² °C⁻¹</td>
<td>1.0</td>
<td>6.5</td>
<td>7.5</td>
<td>7.3</td>
<td>7.2</td>
<td>7.2</td>
<td>7.1</td>
<td>7.1</td>
<td>7.0</td>
</tr>
</tbody>
</table>

4.3 Verification of Line Heating Process

4.3.1 Experimental and Numerical Work done by Biswas et al.

In the past twenty years, considerable interest in the thermal aspects of welding was expressed by many researchers, for example, Biswas, Mandal and Sha from Indian Institute of Technology. They carried out 3D transient FE thermo-mechanical analyses using temperature-dependent material properties, Newton’s convection, and Gaussian distribution of heat. As can be seen here, they did some experimental work and compared the thermal transients and residual deformation results with numerical ones.

As presented by several researchers, finite element method (FEM) is successfully used to evaluate thermal response of the complex welding process. In the present work, the finite element (FE) package ANSYS® has been used to simulate the arc welding process.

Figure 4.3: Dimensions of work piece
4.3.2 Finite element model

In this work, the line heating process is simulated in ANSYS®. The dimensions of the work piece are $300^\times 260^\times 6$ mm$^3$, as shown in Figure 4.3 in which the heat center is moving along the $x+$ direction in line KI, with constant heating speed $V$.

Furthermore, for the sake of saving computational time and reducing computer configurations requirement, a half model has been taken in the simulation, with symmetric boundary condition setting in the symmetry plane, since the model is symmetric along the X-Z plane.

Figure 4.4 displays the element mesh of the geometric model to be solved. An element size of 2 mm is used in the heating zone, while 10 mm element size is used in the area far away from the heat affected zone (HAZ). Free mesh is performed in the middle zone.

![Figure 4.4: Element mesh](image)

Finer meshes are adopted to study the effect of the mesh style on the result. In this case, four different element sizes, 1.25 mm, 2.5 mm, 5 mm, and 10 mm are modeled. All the elements are hexahedrons (see Figure 4.5).

![Figure 4.5: Finer element mesh](image)

FE results demonstrate that the temperature distribution of the node located in the center of the welding line of both models are almost identical since the maximum deviation in all time
steps is only 0.21%, which means the relatively coarser model is satisfied to be utilized in the simple line heating process.

Figure 4.6 demonstrates the peak temperatures of one node located in the center of the welding line, with respect to the Gaussian and ellipsoidal source model, in the condition of 5350 W heat input and 6 mm/s heating speed. Both models reach a similar peak temperature but the former one leads a relatively slow cooling rate.

![Figure 4.6: Simulation results in different meshes](image)

### 4.3.3 Finite element Analysis

Figure 4.7 displays the temperature field distributions during the heating process, in the finer mesh case. It is obvious that the temperatures increase rapidly in the beginning, from room temperature to around 500°C within ten seconds. After that, the temperature field tends to be stable, which means the temperature of a certain point is varied by time, but the certain temperature is moving with the heat source, meanwhile, the peak temperature is retaining of a value of approximately 547 °C.

During this period, the contour of the temperature distribution in the front of the heat source is quite dense. On the contrary, in the rear part, the temperature distribution gradient is comparatively small. Generally speaking, the shapes of the contours are close to ellipsoids.

The cooling process in the finer mesh case is described in Figure 4.8. After removing the heat source, the peak temperatures drop to about 350°C in 100 seconds. However, the changing rate is relatively lower comparing with the heating process. In the cooling stage, the contours enlarge by time (see Figure 4.8). After 6000 seconds, the peak temperature becomes less than 70°C and eventually the temperatures in all nodes are tending to the room temperature.
4.3.4 Results discussion

To obtain more detailed information of the temperature field, several groups of nodes are selected. Firstly 7 nodes in the center line with uniform distance are chosen (see Figure 4.9), and the temperatures of these 7 points are plotted in Figure 4.10, which indicates that, after reaching a stable state, the curve of the temperature of each node is of the same tendency, and for a certain node, the temperature shapely increases when subjected to heat input, and then fall down with a relatively lower speed after achieving the peak temperature. The first and
last curves are with lower temperature than the other nodes, which is owing to less heat affected time.

Figure 4.8: Temperature distribution during the cooling process

Figure 4.9: Nodes in the center line (Group 1)
Another 7 points are selected as Figure 4.11, whose temperature distributions are similar to the previous ones, except the peak temperatures are a little lower since it is a bit far from the heat source (see Figure 4.12).

In the Z direction, 3 more nodes are taken as Figure 4.13. Figure 4.14 shows their temperatures distributions. The temperature reduces with the increment of distance from the heating surface but the effect is not so significant.
In the Y direction, 7 nodes are marked as Figure 4.15, with the distances away from the center line of 2, 4, 6, 8, 14 and 26 mm respectively. The more distance away from the heating line, the lower temperatures produced (see Figure 4.16).

Figure 4.17 illustrates the relationship between the peak temperatures and the distances from the center line.
Figure 4.14: Temperature results of nodes group 3

Figure 4.15: Nodes Group 4 (in y direction)

Figure 4.16: Temperature results of nodes group 4
4.3.5 Parametric Study

For the sake of finding out the effects of heating process parameters (heating speed and heat input) and geometric parameter (thickness of the metal plate), a series of calculations with different parameters have been performed.

Figure 4.18 explains the temperature distributions of the node located in the middle of the center line, in 5 cases of variant heating speeds, 4 mm/s, 6 mm/s, 8 mm/s, 10 mm/s and 12 mm/s. It can be concluded that the lower the speed is, the higher temperature becomes in the result.
The heat input in the previous calculation is 5350 W. Keep the fixed heating speed 6mm/s, change the heat input to 3000 W, 6500 W and 7950 W, then the corresponding temperature distributions are plotted in Figure 4.19. There is not doubt that the higher heat input results in higher temperature.

To study the effect of the plate thickness on the temperature field result, 5 cases are conducted in which the heating speeds are fixed to 6mm/s, the heat inputs are set as 5350 W, but the plate thickness changes as 4 mm, 6 mm, 8 mm, 10 mm and 12 mm. It can be proved in Figure 4.20 that the thinner the plate, the higher temperature is obtained.
4.3.6 Comparison

In Figure 4.21, a finite element simulation of a line heating process was verified using the work of Biswas et al. (2007). To do this, the same geometry and material properties as Biswas’s model were used, the heating process with total heat input 5350 Watts and heating speed 6 mm/sec was simulated using ANSYS® codes. It shows that the temperature history result of present work is in a good agreement with the experimental result obtained by Biswas et al. (2007).

![Figure 4.21: Comparison with Biswas’s result](image)
5. Finite Element Analysis of Bead-on-plate Welding Process

5.1 Finite element model

Due to the development of the computer science and engineering techniques, the finite element method (FEM), as an efficient method in engineering analysis, has been improved with critical breakthroughs in theoretical study, computer program development as well as in application aspects during the last fifty years. A series of qualified software have been developed corresponding to FEM, ANSYS, ABAQUS, ADINA, NASTRAN, MARC, SYSWELD, to name a few.

In this work, the welding process has been simulated in ANSYS®. The dimensions of the work piece are 120*120*6 mm³, as shown in Figure 5.1 in which the arc center is moving along the x+ direction in line KI.

Generally speaking, severe physical and chemical reactions occur during the welding process, between the base metal and the melted pool, which includes the thermodynamics and heat transfer, the interaction between the heat source and metal, solidification in the welding line, the phase transformations in the welding joint, and so on. To focus on the temperature field of the welding structure, some factors have a low effect on it are supposed to deserve low weight in consideration or sometimes even be neglected. Hence, several simplifications have been considered during the simulation.

First of all, the chemical reactions, agitation and convection phenomena, and the phase change are not taken into consideration in the model.
Secondly, the heat transfer between the base metal and the experiment table are neglected; it is assumed that there is only convection heat transfer between the edges of the work piece and the air, without radiation heat transfer.

Thirdly, it is assumed that the welding rod is of the same material as the substrate.

Finally, it is assumed that the welding process is conducted with constant heating speed \( V \); the heat source is represented by the Gaussian model.

The welding parameters performed in this work are: voltage \( U = 25 \text{V} \), current \( I = 180 \text{A} \), heat efficiency \( \eta = 0.75 \), effective heating radius \( R = 6 \text{mm} \), welding speed \( V = 10 \text{mm/s} \). The initial temperature is set to 25 °C. Furthermore, for the sake of saving computational time and reducing computer configurations requirement, a half model has been taken in the simulation, with the symmetric boundary condition setting in the symmetry plane, since the model is symmetric along the X-Z plane.

In this analysis, eight-node three-dimensional brick thermal element, Solid 70, is used. In the mean time, Steel 25 (GB/T 699-1988) is used whose temperature-dependent material properties are listed in Table 5.1.

| Table 5.1: Material Properties for Steel 25 (Lu et al. 2001) |
|-----------------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|
| Temperature (°C) | 20        | 250       | 500       | 750       | 1000      | 1500      | 1700      | 2500      |
| Thermal conductivity (W/m²°C) | 50        | 47        | 40        | 27        | 30        | 35        | 140       | 142       |
| Density (Kg/m³)   | 7820      | 7700      | 7610      | 7550      | 7490      | 7350      | 7300      | 7090      |
| Specific heat capacity (J/kg/°C) | 460       | 480       | 530       | 675       | 670       | 660       | 780       | 820       |
| Poisson’s ratio   | 0.28      | 0.29      | 0.31      | 0.35      | 0.4       | 0.45      | 0.48      | 0.5       |
| Thermal expansion coefficient (×10⁻⁵/°C) | 1.10      | 1.10      | 1.10      | 1.10      | 1.10      | 1.10      | 1.10      | 1.10      |
| Modulus of elasticity (GPa) | 205       | 187       | 150       | 70        | 20        | 19        | 18        | 12        |
| Yield stress (MPa) | 220       | 175       | 130       | 40        | 25        | 2         | 1         | 0.1       |
| Tangent modulus (GPa) | 20.5      | 18.7      | 15.0      | 7.0       | 2.0       | 1.9       | 1.8       | 1.2       |

Figure 5.2 displays the element mesh of the geometric model to be solved. An element size 2mm is used in the welding zone, while 8 mm element size is used in the area far away from the heat affected zone (FAZ). Free mesh is performed in the middle zone.
Meanwhile, a finer mesh is used to study the effect of the mesh configuration on the result. In this case, four different element sizes, 1mm, 2mm, 4mm, and 8mm are modeled. And all the elements are hexahedrons (see Figure 5.3).

\section*{5.2 Plastic Material Options}

During the course of a nonlinear material analysis, a number of material-related factors can cause the structure’s stiffness to change. Nonlinear stress-strain relationships of plastic, multilinear elastic, and hyperelastic materials will cause a structure’s stiffness to change at different load levels (and, typically, at different temperatures). Creep, viscoplasticity, and viscoelasticity will give rise to nonlinearities that can be time-, rate-, temperature-, and stress-related. Swelling will induce strains that can be a function of temperature, time, neutron flux level (or some analogous quantity), and stress. Any of these kinds of material properties can be incorporated into an ANSYS analysis if using appropriate element types.
Most common engineering materials exhibit a linear stress-strain relationship up to a stress level known as the proportional limit. Beyond this limit, the stress-strain relationship will become nonlinear, but will not necessarily become inelastic. Plastic behavior, characterized by non-recoverable strain, begins when stresses exceed the material's yield point. Because there is usually little difference between the yield point and the proportional limit, the ANSYS program assumes that these two points are coincident in plasticity analyses (see Figure 5.4).

Plasticity is a non-conservative, path-dependent phenomenon. In other words, the sequence in which loads are applied and in which plastic responses occur affects the final solution results. If plastic response is anticipated in an analysis, the loads should be applied as a series of small incremental load steps or time steps, so that the model will follow the load-response path as closely as possible. The maximum plastic strain is printed with the substep summary information in the output.

The automatic time stepping feature will respond to plasticity after the fact, by reducing the load step size after a load step in which a large number of equilibrium iterations was performed or in which a plastic strain increment greater than 15% was encountered. If too large a step was taken, the program will bisect and resolve using a smaller step size.

Other kinds of nonlinear behavior might also occur along with plasticity. In particular, large deflection and large strain geometric nonlinearities will often be associated with plastic material response. If the large deformations are expected in the structure, these effects must be activated in the analysis with the corresponding command. For large strain analyses, material stress-strain properties must be input in terms of true stress and logarithmic strain.

The available options for describing plasticity behavior in ANSYS are described as follows.

1. Bilinear Kinematic Hardening.
The Bilinear Kinematic Hardening (BKin) option assumes the total stress range is equal to twice the yield stress, so that the Bauschinger effect is included (see Figure 5.5). This option is recommended for general small-strain use for materials that obey von Mises yield criteria (which includes most metals). It is not recommended for large-strain applications. It can be combined with creep and Hill anisotropy options to simulate more complex material behaviors. Figure 5.6(a) illustrates a typical display of bilinear kinematic hardening properties.

The Multilinear Kinematic Hardening (KINH and Mkin) options use the Besseling model, also called the sublayer or overlay model, so that the Bauschinger effect is included. KINH is preferred for use over Mkin because it uses Rice’s model where the total plastic strains remain constant by scaling the sublayers. To define more stress-strain curves and more points per curve is available. If more than one stress-strain curve for temperature dependent properties is defined, then each curve should contain the same number of points. The assumption is that the corresponding points on the different stress-strain curves represent the
temperature dependent yield behavior of a particular sublayer. These options are not recommended for large-strain analyses. It can be combined with the Hill anisotropy option to simulate more complex material behaviors. Figure 5.6(b) illustrates typical stress-strain curves for the MKIN option.


The Nonlinear Kinematic Hardening (CHABOCHE) option uses the Chaboche model, which is a multi-component nonlinear kinematic hardening model that allows user to superpose several kinematic models. Like the BKin and MKIN options, it can be used to simulate monotonic hardening and the Bauschinger effect. This option also allows people to simulate the ratcheting and shakedown effect of materials. By combining the CHABOCHE option with isotropic hardening model options BISO, MISO, and NLISO, further capability of simulating cyclic hardening or softening is achievable. It can also be combined with the Hill anisotropy option to simulate more complex material behaviors. The model has $1+2n$ constants, where $n$ is the number of kinematic models, and is defined by NPTS in the TB command. This model is suitable for large strain analysis.


The Bilinear Isotropic Hardening (BISO) option uses the von Mises yield criteria coupled with an isotropic work hardening assumption. This option is often preferred for large strain analyses. It can be combined with Chaboche, creep, viscoplastic, and Hill anisotropy options to simulate more complex material behaviors.

5. Multilinear Isotropic Hardening.

The Multilinear Isotropic Hardening (MISO) option is like the bilinear isotropic hardening option, except that a multilinear curve is used instead of a bilinear curve. This option is not recommended for cyclic or highly non-proportional load histories in small-strain analyses. It is, however, recommended for large strain analyses. The MISO option can contain up to 20 different temperature curves, with up to 100 different stress-strain points allowed per curve. Strain points can differ from curve to curve. It can be combined with nonlinear kinematic hardening (CHABOCHE) for simulating cyclic hardening or softening. It can also be combined with creep, viscoplastic, and Hill anisotropy options to simulate more complex material behaviors.


The advantage of this model is that the material behavior is defined as a specified function which has four material constants. The material constants can be obtained by fitting material tension stress-strain curves. Unlike MISO, there is no need to be concerned about how to appropriately define the pairs of the material stress-strain points. However, this model is only applicable to the tensile curve like the one shown in Figure 5.7. This option is suitable for large strain analyses. It can be combined with Chaboche, creep, viscoplastic, and Hill anisotropy options to simulate more complex material behaviors.
Others options, including Anisotropoc, Hill Anisotropoc, Drucker-Prager, Extended Drucker Prager, Gurson Plasticity and Cast Iron, are available as well.

![Figure 5.7: Nonlinear isotropic hardening stress-strain curve](image)

In the present study, the Bilinear Isotropic Hardening (BISO) option is adopted.

## 5.3 Thermal load applying

In the thermal analysis in ANSYS®, the loads can be applied either on the solid model (keypoints, lines, and areas) or on the finite element model (nodes and elements). The loads can be specified using the conventional method of applying a single load individually to the appropriate entity, or as complex tabular boundary conditions, or as function boundary conditions. There are five types of thermal load which can be specified in Ansys®:

1. **Constant temperatures**

   There are DOF constraints usually specified at model boundaries to impose a known, fixed temperature.

2. **Heat flow rate**

   There are concentrated nodal loads. Use them mainly in line-element models (conducting bars, convection links, etc.) where the convections and heat fluxes can not be specified. A positive value of heat flow rate indicates heat flowing into the node (that is, the element gains heat), and vise versa. If both the constant temperature and the heat flow rate are specified at a node, the temperature constraint prevails.

3. **Convections**

   Convections are surface loads applied on exterior surfaces of the model to account for heat list to (or gain from) a surrounding fluid medium. They are available only for solids and shells.

4. **Heat fluxes**
Heat fluxes are also surface loads and it can be used when the amount of heat transfer across a surface (heat flow rate per area) is known or is calculated through a FLORTRAN CFD analysis. A positive value of heat flux indicates heat flowing into the element, and vice versa. Note that heat flux is used only with solids and shells. An element face may have either convections or heat flux specified as a surface load, but not both of them. The one specified last will be used if both of them are specified on the same element face.

5. Heat generation rates

Heat generation rates can be applied as ‘body loads’ to represent heat generated within an element, for example by a chemical reaction or an electric current. Heat generation rates have units of heat flow rate per unit volume.

For the surfacing bead welding in a thin plate where the effect of the molten metal is neglected, relatively satisfied results are achievable when the heat source is simulated as heat flux. While in the fillet welding and the groove welding, the application of heat generation rate on the three dimensional element will result in more precise predictions.

5.4 Three Dimensional Analysis

Figures 5.8 to 5.11 display the temperature field distributions in the heating process, in the finer mesh case. It is obvious that the temperatures increase rapidly in the beginning, from room temperature to over 1000°C within one second. After that, the temperature field tends to be stable, which means the temperature of a certain point is varied by time, but the certain temperature is moving with the heat source, meanwhile, the peak temperature is retaining of a value of approximately 1126°C.
During this period, the contour in the front of the heat source is quite dense. On the contrary, in the rear part, the temperature distribution gradient is comparatively small. Generally speaking, the shapes of the contours are close to ellipsoids.
The cooling process in the finer mesh case is described in the following four figures, from Figure 5.12 to Figure 5.15. After removing the heat source, the temperatures drop very quickly, with the decrement of over 1000 °C in 5 seconds. However, the changing rate is relatively lower comparing with the heating process. In the cooling stage, the contours enlarge by time (see Figure 5.14), and eventually the temperatures in all nodes are tending to the room temperature (see Figure 5.15).

Figure 5.16 demonstrates the peak temperatures of the node located in the center of the welding line, with respect to the coarser and finer mesh, are 1308 °C and 1126 °C respectively. More than ten percent deviation is observed.
5.5 Results Discussion

To obtain more detailed information of the temperature field, several groups of nodes are selected. Firstly 7 nodes in the welding line with uniform distance are chosen (see Figure 4.9), and the temperatures of these 7 points title nodes group 1 are plotted in Figure 5.17, which indicates that, after reaching a stable state, the curve of the temperature of each node is of the same tendency, and for a certain node, the temperature shapely increases when it subject to heat input, and then fall down with a relatively lower speed after achieve the peak temperature. The first node and the last one are with lower/higher temperature than the other nodes, which is owing to the less/more heat affected time.

Another 7 points are selected as Figure 4.11, whose temperature distributions are similar to the previous ones, expect the peak temperatures are much lower since it is a little far from the heat source, and the decrements of temperatures are slower (see Figure 5.18).
In the Z direction, 3 more nodes are taken as Figure 4.13. Figure 5.19 implies their temperatures distributions. The temperature reduces with the increment of distance from the heating surface and it changes faster than the distance changes.

![Temperature results of nodes group 3](image)

In the Y direction, 7 nodes are marked as Figure 4.15. The more distances away from the heating line, the lower temperatures produced (see Figure 5.20).

Figure 5.21 illustrates the relationship between the peak temperatures and the distances from the welding line.

![Temperature results of nodes group 4](image)
5.6 Parametric Study of Welding Process

5.6.1 Effects of Welding Speed

For the sake of finding out the effects of welding process parameters (welding speed and heat input) and geometric parameter (thickness of the metal plate), series of calculations with different parameters have been performed.

Figure 5.22 explains the temperature distributions of the node located in the middle of the welding line, in 5 cases of variant welding speeds, 4 mm/s, 6 mm/s, 8 mm/s, 10 mm/s and 12 mm/s. It can be concluded that the lower the speed is, the higher temperature got in the result.
5.6.2 Effects of Heat Input

The heat input in the previous calculation is \( Q = \eta \cdot I \cdot U = 0.75 \cdot 25 \cdot 180 = 3375 \) W. Keep the fixed welding speed 8mm/s, change the heat input to 1500 W, 2500 W, 4500 W and 5500 W, then the corresponding temperature distributions are plotted in Figure 5.23. There is not doubt that the higher heat input results in higher temperature. It is also notable that all the deviations of temperature by 1000 W changing in heat input are more or less 380 °C.

![Figure 5.23: The effect of heat input](image)

5.6.3 Effects of Plate Thickness

To study the effect of the plate thickness on the temperature field result, 5 cases are conducted in which the welding speeds are fixed to 8 mm/s, the heat inputs are set as 3375 W, but the plate thickness changes as 4 mm, 6 mm, 8 mm, 10 mm and 12 mm. It can be proved in Figure 5.24 that the thinner the plate, the higher temperature will be obtained.

![Figure 5.24: The effect of welding speed](image)
6. Welding Distortions

6.1 Introduction

Since the temperatures are highest in the region near the welding torch, this region expands more than regions further away. During the heating, the stresses in the region near the weld are compressive plastically because the thermal expansion in this region is restrained by surrounding metal with lower temperature and higher yield stress. When the welding has been completed and the plate starts to cool, it deforms in the opposite direction. If the material was completely elastic during the entire period of the heating and cooling cycle, the plate would return to its initial shape with no residual distortion. However, for metals like steel and aluminium plastic deformations occur. As a result of the compressive plastic strains produced in the regions near the welding zone, the plate continues to deform after passing its initial shape, which results in a negative final distortion when the plate cools down to its initial temperature, as illustrated in Figure 6.1. The deformations may be so large that the object cannot fulfill its intended function or fit its intended location.

![Figure 6.1: Deformation of a steel plate during and after welding](image)

Welding-induced distortion arises from localized heating and subsequent non-uniform cooling process during welding. Therefore, complex strains occur in the weld and adjacent regions producing stresses, which cause deformation of the welded component. Thus, any welding process is able to result in a certain distortion of the fabricated product.

Because welding induced distortion directly affects the product quality of the joined components it is necessary to mitigate distortion to meet product requirements. Mitigation techniques such as specific weld preparation, optimized welding sequencing, pre-bending, heat sinking, thermal straightening, thermal tensioning could be applied. Adak and Mandal
(2010) investigated experimentally and numerically the influence of heat sinking on deflection resulting in a model suitable for parameter optimization and distortion prediction.

At this point, the numerical simulation of welding-induced distortion represents a useful tool because it enables prediction of distortion, which leads to a prior description of distortion development and allows well directed optimization of welding induced distortion evolution in practice.

Over the past 30 years, the finite-element method has been used in an attempt to predict distortion due to welding.

6.2 Theory of Welding Deformations

During the heating and cooling cycles of a welding process, many factors affect shrinkage of the metal, making accurate predictions of distortion complex and difficult.

The physical and mechanical properties of the metal that affect the degree of distortion change with the application of the heat. As the temperature of the weld increases, the yield strength, the modulus of elasticity and the thermal conductivity of the steel decrease, whereas, the specific heat and the coefficient of thermal expansion increase.

The changes in temperature and stresses during welding process have been reported by Weisman (1976). To illustrate physically how residual stresses are formed during the welding process, a bead fillet on a plate is described as shown schematically in Figure 6.2(a). The figure also shows the arc, which is moving at a speed \( v \), located at the origin \( O \). The hatched area, \( M-M' \), in Figure 6.2(a) shows the region where plastic deformation occurs during the welding. The egg-shaped region near the origin \( O \) indicates the region where the metal is melted. The metal outside the hatched area remains elastic during the entire welding cycles.

To describe the temperature changes during welding, several cross-sections are analyzed as shown in Figure 6.2(b). In some distance ahead of the welding torch, which is section along \( A-A \), the temperature gradient, \( \Delta T \) due to the welding is almost zero. Along section \( B-B \), which crosses the welding arc, the temperature change is very high and the distribution is very uneven. Along section \( C-C \), which is some distance behind the welding arc, the temperature change becomes less steep and more even. Finally, along section \( D-D \), which is very far from the welding arc, the temperature change due to welding has returned to nearly zero.

The distribution of stresses \( \sigma_x \), in the X-direction at cross sections \( A-A \), \( B-B \), \( C-C \), and \( D-D \) are illustrated in Figure 6.2(c). Normal stresses in y-direction, \( \sigma_y \), shearing stresses, \( \tau_{xy} \), also exist, but are usually much smaller than \( \sigma_x \). Along section \( A-A \), the thermal stresses due to welding are almost zero. Stresses in region below the weld pool at section \( B-B \) are also almost zero because molten metal cannot support a load. Stresses in the heat-affected zone (HAZ) are compressive, because the expansion of these areas is restrained by surrounding metal where the temperature is lower. Since the metal temperature in these regions is high and the yield strength of the material is low, the stresses are as high as the yield strength of the material at
the corresponding temperature. The magnitude of compressive stress reaches a maximum with increasing distance from the weld or with decreasing temperature. Stresses in regions away from the weld line, however, are tensile to balance with the compressive stresses in areas near the weld.

![Diagram showing weldment of large plate, temperature distribution, and stress distribution during welding.](Figure 6.2: Temperature and Stresses Changes During Welding (Weisman, 1976))

At section C-C, where the weld metal and heat-affected zone have cooled, the result is tensile stresses in regions near the weld as they try to shrink and compressive stresses at greater distance. Finally, section D-D represents a cooled-down region, where high-tensile stresses are present in the HAZ zone and compressive stresses exist in base plate away from the weld.

### 6.2.1 Stress-strain relationship

In the elastic region, the total strain increment can be expressed as,

\[
\{ \delta e \} = \{ \delta e \}_r + \{ \delta e \}_t
\]

(6-1)
where \( \{de\}_e \) is the elastic strain increment due to the consistency condition, \( \{de\}_r \) is the thermal strain increment. The elastic matrix \( [D]_e \) (relevant to the modulus of elasticity and Poisson's ratio) varies with temperature, then,

\[
\{de\}_e = d\left[[D]_e^{-1}\{\sigma\}\right] = [D]_e^{-1}\{d\sigma\} + \frac{\partial[D]_e^{-1}}{\partial T}\{\sigma\}dT
\]  

(6-2)

\( \{d\sigma\}_T \) is the differential increment of \( \{\alpha_0T\} \) where \( \alpha_0 \) is the linear expansion coefficient of the initial temperature.

\[
\{de\}_r = \{\alpha_0dT + Td\alpha_o\} = \left\{\alpha_0 + \frac{\partial\alpha_0}{\partial T}T\right\}dT = \{\alpha\}dT
\]  

(6-3)

The linear expansion coefficient \( \alpha \) also varies with temperature. Its effective value is,

\[
\{\alpha\} = \left\{\alpha_0 + \frac{\partial\alpha_0}{\partial T}T\right\}
\]  

(6-4)

Substitute Eq.(6-2) and (6-3) into Eq. (6-1), then,

\[
\{d\sigma\} = [D]_e \{de\}_e - [D]_e \left\{\alpha + \frac{\partial[D]_e^{-1}}{\partial T}\{\sigma\}\right\}dT
\]  

or,

\[
\{d\sigma\} = [D]_e \{de\}_e - [C]dT
\]  

(6-5)

where \( [D] = [D]_e \), \( [C] = [D]_e \left\{\alpha + \frac{\partial[D]_e^{-1}}{\partial T}\{\sigma\}\right\} \) is relevant to temperature. Eq.(6-5) is the stress increment-strain increment relationship in the elastic region, taking into consideration the temperature dependent material properties.

While in the plastic region, assume that the material starts to yield when the value of the yield function \( f(\sigma, \sigma, \ldots) \) reaches \( f_0(\sigma, T, K) \), which is,

\[
f = f_0(\sigma, T, K(\varepsilon_p))
\]  

(6-7)

where T is the temperature and K is the strain hardening exponent.

Its differential form is,

\[
df = df_0
\]  

(6-8)

The total strain increment in the plastic region can be calculated by,

\[
\{de\} = \{de\}_p + \{de\}_e + \{de\}_r
\]  

(6-9)

where \( \{de\}_p \) is the plastic strain increment.

According to the flow rule,
\[
\{d\varepsilon\}_p = \xi \left[ \frac{\partial f}{\partial \sigma} \right]
\]  

(6-10)

From the equations above, \( \xi \) can be solved as,

\[
\xi = \left[ \frac{\partial f}{\partial \sigma} \right]^T [D]_p \{d\varepsilon\} - \left[ \frac{\partial f}{\partial \sigma} \right]^T [D]_p \left( \{\alpha\} + \frac{\partial[D]}{\partial T}^{-1} \{\sigma\} \right) + \frac{\partial f}{\partial \sigma} / s
\]

(6-11)

Then the stress-strain relationship will be,

\[
\{d\sigma\} = [D]_p \{d\varepsilon\} - \left( [D]_p \{\alpha\} + [D]_p \frac{\partial[D]}{\partial T}^{-1} \{\sigma\} \right) - [D]_p \left( \frac{\partial f}{\partial \sigma} \right) / S dT
\]

(6-12)

where \([D]_p\) is the elasto-plastic matrix,

\[
[D]_p = [D]_p - [D]_p \left( \frac{\partial f}{\partial \sigma} \right)^T [D]_p / S
\]

(6-13)

Let \([D]_p\), \(\{C\} = \{C\}_p = \left( [D]_p \{\alpha\} + [D]_p \frac{\partial[D]}{\partial T}^{-1} \{\sigma\} \right) - [D]_p \left( \frac{\partial f}{\partial \sigma} \right) / S \),

it becomes Eq.(6-6).

It can be concluded that the stress-strain relationship is,

\[
\{d\sigma\} = [D]_p \{d\varepsilon\} - \{C\} dT
\]

(6-6)

while in the elastic region, \([D] = [D]_p\), \(\{C\} = \{C\}_p\), and in the plastic region, \([D] = [D]_p\), \(\{C\} = [C]_p\).

Consider one element of the entire structure. It is with temperature \(T\) nodal force \(\{F\}_e\), nodal displacement \(\{\delta\}\), strain \(\{e\}\) and stress \(\{\sigma\}\) at time \(t\), and with \(\{F + dF\}_e\), \(\{\delta + d\delta\}\), \(\{e + d\varepsilon\}\) and \(\{\sigma + d\sigma\}\) at time \(T + dt\). The following formula can be obtained by applying the virtual displacement principle.

\[
\{d\sigma\}_T \{F + dF\}_e = \int_0^{\Delta V} \{d\sigma\}_T [B]^T (\{\sigma\} + [D]\{d\varepsilon\} - \{C\}dT) dV
\]

(6-14)

where \([B]\) is geometric matrix and is relevant to the geometry of the element.

Since the structure is in an equilibrium state at time \(t\),

\[
\{dF\}_e = \int_0^{\Delta V} [B]^T \{\sigma\} dT
\]

(6-15)
Consider the stress-strain relationship,

\[
\{dF\}^\gamma = \int_\Delta \left[ B^T \right] \left[ (D) \{de\} - \{C\}dT \right] dV
\]

(6-16)

or,

\[
\{dF\}^\gamma + \{dR\}^\gamma = \left[ K \right]^\gamma \{d\delta\}
\]

(6-17)

where the equivalent nodal force of the initial strain is \( \{dR\}^\gamma = \int_\Delta \left[ B^T \right] \{C\}dT dV \), the element stiffness matrix is \( \left[ K \right]^\gamma = \int_\Delta \left[ B^T \right] [D] B dV \).

Substitute \( \{C\} \) and \([D]\) from Eq.(6-6) into Eq.(6-17), the equivalent nodal load and stiffness matrix of the element can be obtained and be used in the global stiffness matrix and load column vector, to produce the following system of algebraic equations on order to solve the nodal displacement.

\[
\left[ K \right] \{d\delta\} = \{dF\}
\]

(6-18)

where \( \left[ K \right] = \sum \left[ K \right]^\gamma \), \( \{dF\} = \sum \left( \{dF\}^\gamma + \{dR\}^\gamma \right) \), normally \( \sum \{dF\}^\gamma \) is close to zero in the welding-related issues.

### 6.2.2 Thermo-elasto-plastic FEM in welding

The key to solve the thermo-elasto-plastic issues is to convert the nonlinear stress-strain relationship gradually into the linear one during the loading process. Since there is no external force during the welding process, the load is actually caused by the temperature variation. The solution can be obtained by dividing the temperature variation calculated from the thermal analysis into several load increments to apply to the structure gradually.

In general, there are three methods to solve the elasto-plastic issues, which are tangent stiffness matrix increment method, initial stress increment method and initial strain increment method. However, in the last two of these methods, a linear system of equations with the same stiffness matrix is required, which can not be guaranteed in each step of thermo-elasto-plastic solution owing to the mechanical properties of the material varies with the change of the temperature. And these two methods often result in non-convergence, so they are not adopted in the thermo-elasto-plastic solutions.

In the tangent stiffness matrix increment method, the stiffness matrix is corrected in each loading step according to the stress state of the element. When the element yields, the stiffness matrix \( \left[ K \right] \) in Eq.(6-18) is relevant to the cointantaneous stress level. So the equation is nonlinear. In order to linearize it, the load is gradually increased in a certain stress and strain level. If the load value is small enough, Eq.(6-18) can be approximately expressed as,
\[ [K]\Delta\delta = \{\Delta F\} \]  

where \( \{\Delta\delta\} \) is the displacement increment in the \( i \) th load step, \( \{\Delta F\} \) is the applied load the \( i \) th load step, \( \{\Delta F\} = \{F\}/n \), \( n \) is a positive integer.

In this method, the differentials of stress and strain are replaced by their increments, so the stiffness matrix \([K]\) is only related to the stress level before loading. Hence, the relationship between the load and the displacement increment is linear. Consequently the increments of displacement, stress and strain are easily obtained. The total displacement, strain and stress after the \( i \) th load step can be calculated by these increments plus the result after the \((i-1)\)th load step.

Nevertheless, the error if the pure increment is inevitable and it can be accumulated more and more until the result becomes equilibrium nonconvergent. To solve it, the Newton-Raphson iteration method can be used. Before each solution, the Newton-Raphson method evaluates the out-of-balance load vector, which is the difference between the restoring forces (the loads corresponding to the element stresses) and the applied loads. Then a linear solution will be performed, using the out-of-balance loads, and the convergence will be checked. If convergence criteria are not satisfied, the out-of-balance load vector is re-evaluated, the stiffness matrix is updated, and a new solution is obtained (see Figure 6.3). This iterative procedure continues until the problem converges.

![Figure 6.3: Newton-Raphson iterative solution](image)

6.3 Finite Element Analysis of Distortions after Heating
The finite element analysis of the welding process can be defined as a three-dimensional coupled thermo-mechanical analysis. In the present work, the three-dimensional finite element model was solved using ANSYS® code in which three methods are provided to simulate the welding process.

If the temperatures of all nodes are known quantities, then they can be directly defined by the corresponding commands, such as BE, BFE or BFK in the structural analysis. In this case, the node temperatures are supposed to be defined as body loads instead of degrees of freedom. However, it is difficult to obtain the temperatures of all nodes. Hence, this first method is seldom used.

Two frequently used methods are the direct method and indirect method.

The direct method is basically used in the bidirectional coupled applications, in which the result of thermal analysis affects the structural analysis; meanwhile the structural deformation has a certain effect on the thermal calculation as well. To use the direct method, the element selected in the simulation should include both thermal degree of freedom and structural degrees of freedom. The thermal and structural results can be obtained in the same time after the calculation.

However, in some cases, the effect of the structural result on the thermal analysis is very small and can be even neglected. The indirect method can be utilized for these single directional coupled issues. In this method, the thermal analysis and structural analysis are performed separately. The temperature distribution of all nodes can be obtained from the thermal analysis solved in the first step, and be used as body loads to applied in the some geometric model to do the mechanical analysis. Note that the thermal elements need to be converted into structural element in the second step.

The indirect method is adopted in the calculation in the present work due to its strength of less computational time and sufficient precise.

6.3.1 Element Type

In the present work, the three-dimensional finite element model is solved using ANSYS. For thermal analysis, eight-noded, three-dimensional brick thermal element Solid 70 is used. For the mechanical analysis, the element is converted to the corresponding structural element. By means of inputting command to change thermal element type to structural type, the element Solid 70 can be automatically replaced by the equivalent structural element Solid 185 which is also an eight-noded, three-dimensional element but has plasticity, hyper-elasticity, stress stiffening, creep, large deflection, and large strain capabilities. Note that in old version of ANSYS, Solid 70 is converted by the thermal to structural command to Solid 45 element.

In this conversion, the element mesh which plays an essential role in the calculation remains the same as the thermal analysis geometric model.
6.3.2 “Birth and death” Feature

The material deposition is modeled using an element “birth and death” technique. To achieve the “death element” effect, the ANSYS® code does not actually remove the element from the model. Instead, the weld elements are first deactivated by multiplying their stiffness (or conductivity, or other analogous quantity) by a huge reduction factor. Meanwhile, to obtain the “birth element” effect, the ANSYS® program then reactivates the “death element” by allowing its stiffness, mass and other properties return to their original values.

6.3.3 Mechanical Properties and Boundary Conditions

Once the element type is converted to structural type, proper structural boundary condition is supposed to be applied in the finite element model so that the structural analysis is able to be performed. The mechanical properties of material are also temperature dependent, as shown before in Table 4.1 and Table 5.1.

In this work, the welding process is simulated in ANSYS®. The dimensions of the work piece are 300*260*6 mm³, as shown in Figure 4.3 in which the arc center is moving along the x+ direction in line KI. The welding parameters performed in this work were: heat input = 6550 W, effective heating radius R = 7 mm, welding speed V = 6 mm/s. The initial temperature is set to 32 °C. Half of the model is created due to the symmetric properties of the plate geometry (see Figure 6.4). Symmetric boundary condition is applied in the middle of the plate. Three nodes at the bottom of the model which locate inside the welding region are fixed in all degrees of freedom to avoid the rigid motion of the plate (see Figure 6.5). While three more nodes in the other end with the corresponding locations are fixed as well (see Figure 6.4).
The transient thermal results are read and applied as thermal load to the structural analysis. Note that in this case the type of this analysis must be set to static analysis (Type 0 in ANSYS). The flow diagram shown is Figure 6.6 describes the three-dimensional finite element thermo-mechanical analysis model for a homogenous, isotropic solid without heat generation in a rectangular coordinate system.
6.4 Results Discussion

The deformed shape of the plate during welding process is plotted in Figure 6.7. The dashes stand for the undeformed edge. It is observed that at the beginning of the welding the displacement is non-uniform and the workpiece starts to distort at one corner of the free side. The welding joint is expanding when subjected to heat input. Owing to the boundary conditions, displacement downward occurs near the welding joint, at the rear part of the heating source.

![Figure 6.7: Deformation of the plate during welding process](image)

The maximum displacement downward is moving with the heat source. A time delay of its occurrence compared with the heat source can be observed, which means the compressive deformation happens in the rear part of the heat source due to the decrease of the temperature (see Figure 6.8).

Moreover, at the end of the welding process, the distortion becomes more uniform and its distribution is ellipsoidal from the middle of welding joint to other parts of the workpiece. The other corner of the free edge starts to distort when subject to the heating effect.

The welding process lasts for 50 seconds, given the length of the plate is 300 mm and the welding speed in this calculation is set as 6 mm/s. The heat input will be deleted from the model after 50 mm, and consequently the temperature will start to reduce due to the air convection, which leads to the cooling process. The deformed shape of the plate during cooling process is shown in Figure 6.9. It is observed that the more than 0.7 mm displacements occur in the free edge, while the maximum vertical displacement locates in the middle portion of the free edge. The distortions in the welding joint are close to zero, except its two ends are tiny deformed.
The temperatures of all nodes of the plate return to the room temperature when the time comes to 5000 seconds. The final deformed shape is illustrated in Figure 6.10. A scale factor x10 has been used to make it more visible. The distribution of the contour is similar to that in Figure 6.9 while the peak value of the displacement is a little bigger. The deformation distribution is with the approximate shape of a half ellipsoid, from the middle of the free edge to other locations of the model. The maximum displacement is 2.11 mm. The two ends of the welding joint are of small deformation which can be explained as the effect of the fixed nodes' boundary condition.
Figure 6.10: Final deformation Style

Figure 6.11 shows the distortion pattern along the Y axis which is perpendicular to the welding line in which the angular deformations are plotted.

![Distortion pattern along Y-axis](chart1)

The present numerical result is of a relatively good agreement comparing with that obtained by Biswas et al. (2007) after the same procedure (see Figure 6.11). The differences happen in the location close to the welding zone and the peak value in the plate edge, which are owing to the boundary condition configuration.

The distortion shape along the X axis which is along the welding line is displayed in Figure 6.12. The corresponding result of Biswas et al. (2007) is also included. The numerical results
are satisfied, which further validate the efficiency of the present finite element method in predicting the welding distortions.

![Graph showing deformation along the welding line](image)

**Figure 6.12: Distortion pattern, along the welding line (l)**

### 6.5 Parametric Study

Welding simulation is a powerful, cost-efficient tool to predict welding induced distortion. Nevertheless, the effects on calculation result quality are often unknown, thus, sensitivity analyses should be performed to evaluate the influences of certain parameters on distortion development.

In the previous calculation, the heat input was 6550 W, the welding speed was 6 mm/s and the plate thickness was 6 mm. Based on these three welding and geometric parameters, sensitivity analyses are performed as follows.

#### 6.5.1 Effects of Heat Input

Keep the welding speed and plate thickness as 6 mm/s and 6 mm, three different cases of 5350 W, 6550 W, 7750 W heat input values have been studied. One node located in the middle of the welding joint at the bottom of the plate is selected. The displacement curves with respect to time in all cases are plotted in Figure 6.13.

It is observed that all three nodes are deformed of the same trend till around 500 seconds and the changing rate becomes close to zero. It can be concluded that the more heat input will result in bigger final displacement.
Figure 6.13: Effect of heat input on deformation with respect to time

Figure 6.14 plots the displacement curves along the plate length (the welding direction) in all cases. The plate is deformed with the same pattern when the input heat is high (6550 W and 7750 W). While the deformed pattern of the plate turns out to be more uniform along the x axis when the plate model is subjected to lower heat input (5350 W).

6.5.2 Effect of Welding Speed

To study the effect of the welding speed on the displacement of the plate, three different cases with 6 mm/s, 8 mm/s and 10mm/s welding speed have been calculated, in which the heat input is kept as 6550 W and the plate thickness is kept as 6 mm. One node located in the middle of the welding joint at the bottom of the plate is selected. The displacement curves with respect to time of this node in all cases are plotted in Figure 6.15.
Since the length of the plate is constant in these cases, the faster welding speed means less heat affected time on the plate. It can be concluded that the faster welding speed will result in bigger final displacement, with the same heat input. The deformed trends of the higher speed cases are not identical to the lower speed case. Figure 6.16 plots the displacement curves along the plate length (the welding direction) in all cases. The plate is deformed uniform along x axis when subjected to faster moving heat source (8 mm/s and 10 mm/s).

6.5.3 Effect of Plate Thickness

To study the effect of the plate thickness on the displacement of the plate, three different cases with 6 mm, 8 mm and 10mm thick plates have been calculated, in which the heat input is kept as 6550 W and the welding speed is kept as 6 mm/s. One node locates in the middle of the welding joint at the bottom of the plate is selected. The displacement curves with respect to time of this node in all cases are plotted in Figure 6.17.
It is observed that the final displacement changes from more than 2 mm to about 0.8 mm when the thickness of the plate varies from 6 mm to 8 mm. After that, when the thickness further increases to 10 mm, the final displacement reduced to less than 0.5 mm. Figure 6.18 plots the displacement curves along the plate length (the welding direction) in all cases. Thicker plate results in more uniform deformed pattern.

### 6.5.4 Effect of Boundary Condition

In the finite element simulation of the welding process, the temperature field is applied as the thermal load to the model when the analysis converted from thermal to structural type. Although few researchers included the detailed setting of boundary conditions in their publications, one common sense is that the boundary condition is supposed to be applied for the sake of avoiding the rigid motion of the model. Hence, another boundary condition has
been applied to the same model, and more analyses have been performed to study the effect of boundary condition on the displacement of welded plate.

Figure 6.19 shows the boundary condition setting (II), in which a symmetric boundary condition is applied in the area used to divide the full model into two halves. Three same nodes in the plate bottom near the welding line are constrained in all degrees of freedom, while three more nodes in the other side of the diagonal line where the first three nodes located are fixed in all degrees of freedom, instead of three nodes in the other end of the welding joint in the previous boundary condition setting. The resultant distortion patterns in both directions perpendicular to and along the welding line are plotted in Figure 6.20 and Figure 6.21, respectively.
Comparing with the results of boundary condition setting (I), the distortion patterns as well as the maximum distortion values vary a lot, which can prove that the boundary condition plays an essential role in the finite element simulation of the welding process.

One more group of calculation of different boundary conditions has been completed. The same symmetric area is considered, but all nodes in the plate bottom in the welding line are constraint in all degrees of freedom, while three more nodes in the other side of the diagonal line where the first three nodes located are fixed in all degrees of freedom (see Figure 6.22). The resultant distortion patterns in both directions perpendicular to and along the welding line are also plotted in Figure 6.20 and Figure 6.21, respectively.
In this case, the distortion perpendicular to the welding line is similar to setting (I), to some extent, but the one along the welding line is still different. However, both deformations in transverse and longitudinal directions are changing compared with the results of boundary condition (II), to a large extent. It proves again the significant effect of the boundary conditions on the final deformed shape of the welded plate. On the other hand, appropriate constraints are supposed to be applied during the plate fabrication process to meet different requirements on plate shape.

Figure 6.23: Schematic boundary condition (IV)

Nevertheless, simply supported boundary conditions have also been simulated. Figure 6.23 displays the schematic boundary condition IV, in which the UZ (along the plate thickness) and
UX (along the plate length / the welding direction) degrees of freedom are constrained in the edge at the bottom of the plate. The UX constraint is for the sake of preventing the rigid body motion of the plate. Meanwhile, a ‘slightly’ case has also been studied, in which only one of the previous selected nodes is fixed in the longitudinal direction. The resultant distortion patterns in both directions perpendicular to and along the welding line are plotted in Figure 6.24 and Figure 6.25, respectively. To some extent, the simply supported boundary conditions reach a good agreement of the reference results.

![Figure 6.25: Distortion pattern, along the welding line (IV&V)](image)

To sum up, the boundary conditions have a significant effect on the simulating result. More research work is going to be contributed in this interesting subject.
This page is intentionally left blank.
7. Conclusions and Recommendations

7.1 Conclusions

Computational line heating and welding simulations are not fully established as a science but almost sufficiently developed to be applied to various simplified problems in the shipbuilding industry.

After completed this research, several conclusions can be drawn from the results of this investigation.

A complex three dimensional line heating and welding process phenomenon was simulated using a commercial finite element package ANSYS®, in order to compute the distortion pattern and deformation of steel plates.

A mathematical model of moving heat source was established in ANSYS to simulate the transient thermal analysis. A series of load temperature curves of a bead-on-plate weld steel plate were calculated with suitable convection boundary conditions and non-linear material steel properties. Results were validated to numerical and experimental results obtained from previous research. The time variant temperature distributions obtained from previous analysis were applied as thermal load in mechanical analyses to predict the plate distortions.

Parametric studies were performed in both thermal and mechanical analyses. Heating speed, heat input and plate thickness were proved to have a significant impact on the line heating and welding response. Higher heat input, lower heating speed, thinner plate result in higher temperature and bigger deformation. Those parameters also have effect on the final deformed pattern of the plate.

Three models of plates in a bead-on-plate welding process with different restraints have been presented. The restraint on the model has a critical effect on the simulating result. However, for certain parameters, the mesh types and heat source model types turned out to have a relatively lower effect on the results.

7.2 Recommendations

The study has shown the advantage of using the finite element method for a line heating and welding simulation tool, but the objective of simulating welding distortions in the production line was ambitious, and large three dimensional problems have not been solved satisfactorily due to several simplifications made in the finite element analyses.

In the near future more research work is going to be included focusing on various types of welding joints, such as the T joint. Large stiffened panel and other more complex marine structure instead of single welded plate, and structures subjected to multi-pass welding are planned to be modelled in ANSYS code.
Hopefully experimental tests and measurements will be carried out and the corresponding finite element models and calculations are supposed to be completed. Much more will be seen comparing with both results.

Further work on the simulation of welding distortion could be done in order to be able to take into account the welding sequence. This involves more elements due to the need of more weld beads.

In the present work calculations by a finite element program have been performed. Subsequently, it should be possible to make corrective pre-adjustments of geometry and change the joining process parameters or the joining sequence in order to improve the accuracy in the production.

A more comprehensive study could be done More attention could be focused on the welding mechanics such as volume changes due to phase transformation, plasticity transformation, chemical composition and alloy-specific materials parameters.

At last it could be interesting to estimate the probability of defects and the risk of failure in a welding joining process.
**Bibliography**


