Residual stress analysis of pipeline girth weld joints

Djarot B. Darmadi
University of Wollongong

Recommended Citation
UNIVERSITY OF WOLLONGONG

COPYRIGHT WARNING

You may print or download ONE copy of this document for the purpose of your own research or study. The University does not authorise you to copy, communicate or otherwise make available electronically to any other person any copyright material contained on this site. You are reminded of the following:

Copyright owners are entitled to take legal action against persons who infringe their copyright. A reproduction of material that is protected by copyright may be a copyright infringement. A court may impose penalties and award damages in relation to offences and infringements relating to copyright material. Higher penalties may apply, and higher damages may be awarded, for offences and infringements involving the conversion of material into digital or electronic form.
RESIDUAL STRESS ANALYSIS
OF
PIPELINE GIRTH WELD JOINTS

A thesis submitted in fulfillment of the requirements for the award of the degree

DOCTOR OF PHILOSOPHY

from

University of Wollongong

by

DJAROT B. DARMADI
Ir., MT.

Mechanical Engineering
Faculty of Engineering
2014
CERTIFICATION

I, Djarot B. Darmadi, declare that this thesis, submitted in fulfillment of the requirements for the award of Doctor of Philosophy, in the Mechanical Engineering, Faculty of Engineering at the University of Wollongong, Australia, is wholly my own work unless otherwise referenced or acknowledged. The document has not been submitted for qualification at any other academic institution.

Djarot B. Darmadi
22 July 2014
The author would like to express his gratitude to his supervisors, Professor John Norrish and Professor Anh Kiet Tieu for their excellent guidance and continuous encouragement because only under their patient supervisions could this thesis have been completed. Many thanks also to Dr Murugananth Marimuthu for his discussion, training, and permission to carry out Finite Element Analysis in his Weld Simulation and Modeling Laboratory. Special appreciation is also expressed to Dr Dominic Cuiuri for his comprehensive training, helpful supervisions and willingness to become involved in the experimental work required to validate the model. The author very much appreciates the friendly and conducive environment created by Joe Abbot in the Manufacturing and Research Laboratory (MRL) building, including the use of his skilful hands. Fruitful discussions that further improved this thesis were held with Prof. Huijun Li, Dr. Buyung Kosasih, Dr Mark Callaghan, Dr Lenka Kusmikova, Dr Ajit Godbole and Dr Stephen Pan.

Many thanks are also expressed to the Director General of High Education of the Indonesian Government for the scholarship to support the author while taking this PhD study. Thanks are also expressed to Prof. John Norrish on behalf of CRC pipelines for their support and willingness to extend the period of study. An International Postgraduate Tuition Award (IPTA) given by University of Wollongong is also appreciated.

The author is deeply grateful to his wife Jendra and son Bima for their continuous support and encouragement through the good and bad times. Thank you all indeed.
Residual stress is one of the important factors that should be considered when assessing the integrity of welded structures because it is well known that residual stress may lead to failure in the weld joint. Residual stress may lead to stress corrosion cracking in a corrosive environment and has resulted in failures in energy pipelines.

Residual stress and stress corrosion cracking in the region of the girth welded joint in pipelines was investigated in the current work. The emphasis was placed on residual stress modeling in pipeline girth welds since it was judged to be an essential prerequisite for stress corrosion cracking.

In this thesis, the residual stress around girth welds was studied with FEM using ANSYS (APDL) to give a deep insight into the formation of residual stress. A coupled Thermo-Metallurgical-Mechanical analysis was carried out because the metallurgical analysis of the high strength steel used extensively in pipelines cannot be ignored. The final residual stress prediction using ANSYS was verified with experimentally to test the validity of the FEM model.

The FEM analysis is presented step by step which commenced with a simple thermal analysis, followed by a thermo-mechanical analysis and finally a coupled thermo-metallurgical-mechanical analysis. The weld configuration also started from a simple bead-on-plate leading on to multi-pass weld on plate and multi-pass girth weld respectively. Various moving heat source models were used, commencing with a simple point heat source, and then a uniform surface heat source and finally Goldak’s volumetric heat source.

The FEM results for the early simple model were verified with an analytic solution. It should be noted that verification (comparing FEM results with analytic solution) can be done only on a simple model since it is difficult to obtain an analytic solution (not FEM) for a complex model. The simple model was represented by moving point heat source in an infinite solid. To analyse transient state a new analytic solution using a non integration technique was developed.
For the complex model the FEM results were validated with the experimental method to ensure that the model was correct. A new heat source model that combined Goldak’s volumetric heat source and uniform temperature load was used and it provided good results. The Goldak’s heat source model was used to represent the heat transferred to the base metal and previous bead (for multi-pass welding), whilst the uniform temperature load represents melted filler metal (droplets) that form the weld bead. A new programming technique using a database which was composed using the ANSYS standard files was also developed. It was found that using the database provided at least three advantages: greater flexibility, faster computing time and the ability to interchange data.

The role of solid state phase transformation (SSPT) was also studied in terms of volumetric change due to the atomic packaging factor (APF), alteration of the mechanical properties or transformation plasticity. To assess the contribution of each aspect: volume, mechanical properties and transformation plasticity, FEM with volumetric change (mechanical properties and transformation plasticity excluded), FEM with mechanical properties change (volumetric change and transformation plasticity excluded) and FEM with transformation plasticity (volumetric and mechanical properties excluded) were carried out. These contributions were studied by comparing the results of the FEM with and without SSPT consideration, SSPT with volumetric change only, SSPT with alterations to the mechanical properties only and SSPT with transformation plasticity only. The martensitic SSPT was divided into two classes, namely prime martensite and aged martensite and these prime and aged martensite modes were included in the FEM model which is a new feature. The concept of prime and aged martensite is not new one, but the literature did not indicate that this degree of detail had previously been included in FEM programs. Some papers do use SSPT but only prime martensite is modeled. It was found that the results which considered SSPT were closer to the experimental results for ferritic steel welding.

In the earlier chapters, all above aspects were simulated in plate welding and validated using secondary data. Using the same logic an FEM model was programmed on a multi pass girth weld joint. Phase transformations that were included in the model were austenite and martensite (both: prime and aged martensite) SSPT. A reduction factor (Kd) was used to model the yield stress of filler metal at elevated temperature. Using Kd
in the FEM model of welding may be considered as a new feature. All of the modeled aspects have been validated or verified in the plate welding moreover the temperature history and residual stress distribution obtained from FEM prediction in the multi pass girth welding were validated by the experimental results.

DC-LSND (Direct Cooling – Low Stress No Distortion) control is one of suggested stress mitigation techniques for welding. The DC-LSND typically applied on thin plate welding and the main goal is to reduce distortions. There are no published papers found that quantify the effect of DC-LSND on the residual stresses of multi pass thick pipe girth welds. In this work the effect of DC-LSND on the residual stress in a multi pass girth weld joint of pipe with 8mm thickness was studied through FEM analysis. From FEM predictions the application of DC-LSND reduced maximum residual stress by around 35%. From this result, it is suggested that DC-LSND could be applied to the multi pass butt girth welding of thick pipe although the practicality of this procedure would need to be assessed against its potential benefits.

Overall it can be said that the FEM model in this thesis provides an accurate prediction of residual stress and also a deeper understanding of its development particularly in pipeline girth welds in ferritic steels. It is in line with one of the recent research trends in the welding area: improving the basic understanding of the welding process and the resulting residual stresses.
COMPUTER MODELLING

“The process of providing to a computer, usually in the form of mathematical equation, a precise and unambiguous description of the system under study, including the relationships between system inputs and outputs, and using this description to simulate or model the described system”

(Academic Press Dictionary of Science Technology)

SIMULATION

“The technique of imitating the behavior of some situation of process (whether economic, military, mechanical, etc.) by means of a suitably analogous situation or apparatus, especially for the purpose of study of personnel training”

(The Oxford English Dictionary)

“The representation of a physical system usually by a computer or physical model that imitates some aspects of the system for a given purpose, eventually making approximations excluding some behaviors of the physical system.”

(J.L. Hansen – PhD Thesis T.U. Denmark)

Author’s own words: “Modeling and simulation are a pair of powerful tools to understand a certain physical phenomenon through tiered validation steps”

(Djarot B. Darmadi)
# Table of Contents

Certification .................................................................................................................. i  
Acknowledgements ...................................................................................................... ii  
Abstract ....................................................................................................................... iii  
Table of content ........................................................................................................... viii  
List of figures ............................................................................................................... x  
List of tables ............................................................................................................... xvii  
List of publications .................................................................................................... xviii  
List of symbols and abbreviations ............................................................................. xix  

1. Introduction .............................................................................................................. 1  
   1.1. Background ...................................................................................................... 2  
   1.2. Research objective ......................................................................................... 5  
   1.3. Research approach ......................................................................................... 5  
   1.4. Thesis outlines ............................................................................................. 7  
   References ............................................................................................................. 9  

## PART I BASIC CONCEPTS

2. A Review .................................................................................................................. 11  
   2.1. FEM simulation of welding processes ......................................................... 12  
   2.2. Thermal analysis of welding FEM model ...................................................... 13  
      2.2.1. Thermal properties of a material ............................................................ 14  
      2.2.2. Heat transfer and heat conservation ....................................................... 15  
      2.2.3. Parametric study via FEM .................................................................... 18  
   2.3. Development of stress in welding ................................................................. 22  
      2.3.1. A review of the three bars model ............................................................ 23  
   2.4. Solid state phase transformation in the stress development ....................... 31  
   References ............................................................................................................. 34  

## PART II FEM vs ANALYTIC SOLUTION

3. Moving Heat Source Simulation ............................................................................. 37  
   3.1. Solution by Rosenthal .................................................................................... 39  
   3.2. Moving point heat source model .................................................................. 40  
   3.3. Solution using non-dimensional integral to compensate for transient  
       conditions ..................................................................................................... 41  
   3.4. Element model (SOLID 70) ......................................................................... 47  
   3.5. Finite Element model for moving point heat-source ..................................... 47  
   3.6. Analytical model vs FE model – Quasi steady state .................................... 49  
   3.7. Analytical model, Rosenthal solution and FE model –  
       quasi steady state .......................................................................................... 52  
   3.8. Analytical models vs FE model – transient conditions ............................... 53  
   3.9. Conclusion .................................................................................................... 55  
   References ............................................................................................................. 57
# PART III MODELING EXPLORATION BASED ON EXISTING DATA

4. Thermal Analysis of NeT TG1 Specimen ................................................................. 60  
   4.1. Welding procedure provided by NeT .......................................................... 61  
   4.2. Previous works in weld modeling of NeT welding procedure ............... 64  
   4.3. Comparison of FE model to NeT data ....................................................... 66  
      4.3.1. Non linear material model ............................................................... 70  
      4.3.2. Inputted parameters ........................................................................... 72  
      4.3.3. Standard technique ............................................................................ 73  
      4.3.4. Birth and Death technique ................................................................. 75  
      4.3.5. Results and discussion ...................................................................... 79  
   4.4. Conclusion .................................................................................................. 87  

References .......................................................................................................... 88

5. Mechanical Analysis of NeT TG1 Specimen ..................................................... 91  
   5.1. Thermo mechanical analysis ............................................................ 93  
   5.2. Element model (SOLID45) ................................................................. 93  
   5.3. Mechanical properties ............................................................. 94  
   5.4. Exploring the birth and death technique ............................................. 97  
      5.5.1. Accommodating a growing weld bead .......................................... 101  
      5.5.2. Accommodating the melting phenomenon ................................... 101  
   5.6. Results and discussion .......................................................................... 102  
      5.6.1. Using isotropic hardening model .................................................. 109  
      5.6.2. Comparing to Masubuchi’s ............................................................. 110  
   5.7. Conclusions .............................................................................................. 111  

References .......................................................................................................... 112

6. Finite Element Model of Residual Stress Involving Metallurgical Considerations .................................................................................................................. 115  
   6.1. Welding procedures ................................................................................ 116  
   6.2. Material properties .............................................................................. 117  
   6.3. Considerations for phase transformation ........................................... 119  
   6.4. Finite element model ........................................................................... 122  
      6.4.1. Coupled thermo-metallurgical model .......................................... 123  
      6.4.2. Development of residual stress while considering phase transformation ................................................................. 125  
      6.4.3. Solutions and strategies ................................................................. 127  
      6.4.4. The ANSYS APDL modeling ....................................................... 132  
   6.5. Results and discussion .......................................................................... 138  
   6.6. Conclusion ................................................................................................ 144  

References .......................................................................................................... 146

# PART IV Pipeline Girth Welding

7. Residual Stress Analysis of The Girth Weld Joint ....................................... 149  
   7.1. Welding procedure and parameters .................................................. 149  
   7.2. Finite element model ........................................................................... 159  
      7.2.1. Coordinate transformation .......................................................... 161  
      7.2.2. Filtering peak temperature .......................................................... 165  
      7.2.3. Material properties for X-70 base metal and ER70S-6 filler metal ................................................................. 168
<table>
<thead>
<tr>
<th>Chapter</th>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.2.4.</td>
<td>Considering transformation plasticity</td>
<td>175</td>
</tr>
<tr>
<td>7.2.5.</td>
<td>Considering cooling rate</td>
<td>176</td>
</tr>
<tr>
<td>7.3.</td>
<td>Residual stress measurements</td>
<td>178</td>
</tr>
<tr>
<td>7.4.</td>
<td>Analysis and discussions</td>
<td>179</td>
</tr>
<tr>
<td>7.4.1.</td>
<td>Validation of the thermal results</td>
<td>183</td>
</tr>
<tr>
<td>7.4.2.</td>
<td>Residual stress validation</td>
<td>188</td>
</tr>
<tr>
<td>7.5.</td>
<td>Conclusions</td>
<td>202</td>
</tr>
<tr>
<td>References</td>
<td></td>
<td>205</td>
</tr>
<tr>
<td>8.</td>
<td>Simulating DC-LSND on the girth butt welding</td>
<td>209</td>
</tr>
<tr>
<td>8.1.</td>
<td>Heat sink modeling in ANSYS</td>
<td>213</td>
</tr>
<tr>
<td>8.2.</td>
<td>Applying DC-LSND on the girth weld</td>
<td>217</td>
</tr>
<tr>
<td>8.3.</td>
<td>Results and discussions</td>
<td>220</td>
</tr>
<tr>
<td>8.4.</td>
<td>Conclusions</td>
<td>227</td>
</tr>
<tr>
<td>References</td>
<td></td>
<td>228</td>
</tr>
<tr>
<td>PART V Discussions</td>
<td></td>
<td></td>
</tr>
<tr>
<td>9.</td>
<td>General Discussions</td>
<td>230</td>
</tr>
<tr>
<td>9.1.</td>
<td>Overview</td>
<td>230</td>
</tr>
<tr>
<td>9.2.</td>
<td>TMM in plate welding</td>
<td>232</td>
</tr>
<tr>
<td>9.2.1.</td>
<td>Change of material properties in SSPT</td>
<td>233</td>
</tr>
<tr>
<td>9.2.2.</td>
<td>Volumetric change in SSPT</td>
<td>235</td>
</tr>
<tr>
<td>9.2.3.</td>
<td>Aspects involved in SSPT</td>
<td>237</td>
</tr>
<tr>
<td>9.3.</td>
<td>Resetting plastic strain in the girth butt welding</td>
<td>238</td>
</tr>
<tr>
<td>9.4.</td>
<td>Considering lower annealing temperature</td>
<td>242</td>
</tr>
<tr>
<td>9.5.</td>
<td>Transformation plasticity</td>
<td>247</td>
</tr>
<tr>
<td>9.6.</td>
<td>Aged martensite considerations</td>
<td>250</td>
</tr>
<tr>
<td>References</td>
<td></td>
<td>254</td>
</tr>
<tr>
<td>PART VI Conclusions</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10.</td>
<td>General Conclusions</td>
<td>255</td>
</tr>
<tr>
<td>10.1.</td>
<td>Conclusions</td>
<td>255</td>
</tr>
<tr>
<td>10.2.</td>
<td>Future work</td>
<td>258</td>
</tr>
</tbody>
</table>
## List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Australia’s Pipelines</td>
<td>3</td>
</tr>
<tr>
<td>1.2</td>
<td>Research aspects</td>
<td>5</td>
</tr>
<tr>
<td>2.1</td>
<td>Development of Residual stress in welding</td>
<td>12</td>
</tr>
<tr>
<td>2.2</td>
<td>Finite element model for parametric study</td>
<td>19</td>
</tr>
<tr>
<td>2.3</td>
<td>Transient isothermal lines for moving heat source with varied heat; (a)10s, (b)50s, (c)100s and (d)200s</td>
<td>20</td>
</tr>
<tr>
<td>2.4</td>
<td>Transient isothermal lines for moving heat source with varied thermal conductivity; (a)10s, (b)50s, (c)100s and (d)200s</td>
<td>21</td>
</tr>
<tr>
<td>2.5</td>
<td>Transient isothermal lines for moving heat source with varied specific heat; (a)10s, (b)50s, (c)100s and (d)200s</td>
<td>21</td>
</tr>
<tr>
<td>2.6</td>
<td>Thermal stress in the three bars model (modified from Sindo Kou)</td>
<td>22</td>
</tr>
<tr>
<td>2.7</td>
<td>Typical distribution of residual stress in welding: (a) longitudinal residual stress, (b) transversal residual stress</td>
<td>23</td>
</tr>
<tr>
<td>2.8</td>
<td>Thermal residual stress formation</td>
<td>24</td>
</tr>
<tr>
<td>2.9</td>
<td>Three bars model</td>
<td>25</td>
</tr>
<tr>
<td>2.10</td>
<td>Strain-stress diagram of the material model by ANSYS</td>
<td>26</td>
</tr>
<tr>
<td>2.11</td>
<td>Sketch of the strain-stress diagram</td>
<td>26</td>
</tr>
<tr>
<td>2.12</td>
<td>Flow-chart for evaluating thermal stress in elastic strain rate</td>
<td>28</td>
</tr>
<tr>
<td>2.13</td>
<td>Distribution of stress when the middle bar is heated to 400°C (a) full model and (b) half model</td>
<td>28</td>
</tr>
<tr>
<td>2.14</td>
<td>Flow-chart for evaluating thermal stress in plastic strain rate</td>
<td>29</td>
</tr>
<tr>
<td>2.15</td>
<td>Distribution of stress at the three bars model heated to 1100°C (a) when the middle bar is heated, (b) when the temperature load is omitted</td>
<td>29</td>
</tr>
<tr>
<td>2.16</td>
<td>Flow-chart for evaluating residual stress</td>
<td>30</td>
</tr>
<tr>
<td>2.17</td>
<td>Solid state transformations in ferrite steels</td>
<td>32</td>
</tr>
<tr>
<td>3.1</td>
<td>FEM, analytic solution and experiment results</td>
<td>38</td>
</tr>
<tr>
<td>3.2</td>
<td>Fixed coordinate system and moving coordinate system</td>
<td>40</td>
</tr>
<tr>
<td>3.3</td>
<td>Plotted value for $e^{-x^2}$</td>
<td>42</td>
</tr>
<tr>
<td>3.4</td>
<td>Plotted value for $\int_0^\infty e^{-x^2} dx$</td>
<td>42</td>
</tr>
<tr>
<td>3.5</td>
<td>Point heat source in infinite media</td>
<td>44</td>
</tr>
<tr>
<td>3.6</td>
<td>Integral values for varied $\omega$ and $u$</td>
<td>45</td>
</tr>
<tr>
<td>3.7</td>
<td>Temperature profiles at a longitudinal path</td>
<td>46</td>
</tr>
<tr>
<td>3.8</td>
<td>Temperature profiles at a transversal path</td>
<td>46</td>
</tr>
<tr>
<td>3.9</td>
<td>Solid70 3D-thermal element (Release 12.0 Documentation for ANSYS)</td>
<td>47</td>
</tr>
<tr>
<td>3.10</td>
<td>FEM model for point heat source</td>
<td>48</td>
</tr>
<tr>
<td>3.11</td>
<td>Flow chart of the FEM simulation</td>
<td>49</td>
</tr>
<tr>
<td>3.12</td>
<td>Temperature profiles on longitudinal lines with moving point heat source model</td>
<td>50</td>
</tr>
<tr>
<td>3.13</td>
<td>Temperature profiles on transversal lines with moving point heat source model</td>
<td>51</td>
</tr>
</tbody>
</table>
Figure 3.14 Solutions from analytic, FEM and Rosenthal for temperature Profiles at longitudinal paths (left) and transversal paths (right) ... 52
Figure 3.15 Enlarged solutions for far field longitudinal paths ............... 53
Figure 3.16 Isothermal profiles at plate surface for transient state .......... 54
Figure 3.17 Isothermal profiles at plate cross section for transient state ...... 55

Figure 4.1 Sketch of welding process and coordinate system ................. 62
Figure 4.2 Thermocouple position on weld on bead welded plate .......... 62
Figure 4.3 Temperature dependence of thermal properties .................... 63
Figure 4.4 Flowchart for moving heat source model .......................... 66
Figure 4.5 FEM mesh design .............................................. 69
Figure 4.6 Direct method ..................................................... 71
Figure 4.7 Modified method .................................................. 71
Figure 4.8 Newton – Raphson iteration method .................................. 72
Figure 4.9 Standard technique ................................................ 74
Figure 4.10 Temperature distribution for (a) solid rectangle, (b) rectangle with a hole and (c) rectangle with deactivated elements inside the same hole as (b) .................................................. 76
Figure 4.11 Temperature flux for (a) solid rectangle, (b) rectangle with a hole and (c) rectangle with deactivated elements inside the same hole as (b) .................................................. 77
Figure 4.12 Birth and death technique ........................................ 78
Figure 4.13 Temperature histories using standard technique for location of thermocouple Tc1 & Tc4 (refer to figure 4.2) ......................... 80
Figure 4.14 Temperature histories using standard technique for location of thermocouple Tc2 & Tc5 (refer to figure 4.2) ......................... 80
Figure 4.15 Temperature histories using standard technique for location of thermocouple Tc3 & Tc6 (refer to figure 4.2) ......................... 81
Figure 4.16 Temperature histories using standard technique for location of thermocouple Tc7 (refer to figure 4.2) ........................ 81
Figure 4.17 Temperature histories using standard technique for location of thermocouple Tc9 (refer to figure 4.2) ........................ 82
Figure 4.18 Temperature histories using standard and birth & death technique for location of thermocouple Tc1 & Tc4 (refer to figure 4.2) ...... 82
Figure 4.19 Temperature histories using standard and birth & death technique for location of thermocouple Tc2 & Tc5 (refer to figure 4.2) ...... 83
Figure 4.20 Temperature histories using standard and birth & death technique for location of thermocouple Tc3 & Tc6 (refer to figure 4.2) ...... 83
Figure 4.21 Temperature histories using standard and birth & death technique for location of thermocouple Tc7 & Tc9 (refer to figure 4.2) ...... 84
Figure 4.22 (a) Weld pool of actual weld and (b) temperature contour from standard technique .................................................. 84
Figure 4.23 (a) Weld pool of actual weld and (b) temperature contour from birth and death technique .................................................. 85
Figure 4.24 Transverse cross section of weld pool at weld-start and weld-end points .................................................. 85
Figure 4.25 Comparison of temperature contour, (a) standard technique, (b) birth and death technique .................................................. 86
Figure 4.26 Heat source model .................................................. 87
<p>| Figure 5.1 | Pattern of longitudinal distribution of residual stress | 91 |
| Figure 5.2 | SOLID45 3D-structural elements (Release 12.0 Documentation for ANSYS) | 94 |
| Figure 5.3a | Temperature dependence stress-strain diagram for base metal | 95 |
| Figure 5.3b | Temperature dependence stress-strain diagram for weld metal | 95 |
| Figure 5.4 | Displacement distribution for (a) solid rectangle, (b) rectangle with a hole and (c) rectangle with deactivated elements inside the same hole as (b) | 97 |
| Figure 5.5 | Vector plot of displacement for (a) solid rectangle, (b) rectangle with a hole and (c) a rectangle with deactivated elements inside the same hole as (b) | 98 |
| Figure 5.6 | FEM procedures to predict residual stress distribution | 100 |
| Figure 5.7 | FEM longitudinal residual stress (Sz) prediction of bead on plate welding | 103 |
| Figure 5.8 | FEM transverse residual stress (Sx) prediction of bead on plate welding | 103 |
| Figure 5.9 | FEM longitudinal residual stress (Sz) prediction of bead on plate welding at the mid cross section area | 104 |
| Figure 5.10 | FEM transverse residual stress (Sx) prediction of bead on plate welding at the mid cross section area | 104 |
| Figure 5.11 | Residual stresses along the 2mm path in the mid cross section area | 105 |
| Figure 5.12 | FEM longitudinal residual stresses (Sz) prediction at symmetry area in a normal view | 106 |
| Figure 5.13 | FEM transverse residual stresses (Sx) prediction at symmetry area in a normal view | 106 |
| Figure 5.14 | Residual stresses on D2 line | 107 |
| Figure 5.15 | Residual stresses on D3 line | 108 |
| Figure 5.16 | Longitudinal residual stresses (Sz) on BD line | 108 |
| Figure 5.17 | Transverse residual stresses (Sx) on BD line | 109 |
| Figure 5.18 | Kinematic hardening vs Isotropic hardening model | 109 |
| Figure 5.19 | FEM vs Masubuchi’s | 110 |
| Figure 6.1 | CCT diagram for high strength steel (X-70) | 116 |
| Figure 6.2 | Geometry and pass sequences of the welding process | 117 |
| Figure 6.3 | Temperature dependent thermal properties of materials | 118 |
| Figure 6.4 | Temperature dependent mechanical properties of materials | 118 |
| Figure 6.5 | Effect of martensite on thermal expansion coefficient | 120 |
| Figure 6.6 | Effect of austenite to coefficient of thermal expansion | 121 |
| Figure 6.7 | Thermo-Mechanical-Metallurgical analysis | 123 |
| Figure 6.8 | Coupled thermo – metallurgical analysis | 125 |
| Figure 6.9 | Effects of prime and aged martensite to yield stress of base metal | 126 |
| Figure 6.10 | Effects of prime and aged martensite to yield stress of weld metal | 127 |
| Figure 6.11 | Procedure in TMM analysis | 130 |
| Figure 6.12 | Parameter transferred in multi-pass TMM | 131 |
| Figure 6.13 | ANSYS APDL basic pattern | 133 |
| Figure 6.14 | TMM ANSYS program | 135 |
| Figure 6.15 | Saved parameters | 136 |</p>
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.16</td>
<td>Predicted microstructure configurations</td>
<td>139</td>
</tr>
<tr>
<td>6.17</td>
<td>Longitudinal residual stress distributed on the whole model</td>
<td>140</td>
</tr>
<tr>
<td>6.18</td>
<td>Longitudinal residual stress distributed on mid cross section area</td>
<td>141</td>
</tr>
<tr>
<td>6.19</td>
<td>Longitudinal residual stress distributed on the top path of the mid cross section area</td>
<td>141</td>
</tr>
<tr>
<td>6.20</td>
<td>Longitudinal residual stresses without SSPT considerations</td>
<td>142</td>
</tr>
<tr>
<td>6.21</td>
<td>Peak temperatures on the top path at mid cross section area</td>
<td>143</td>
</tr>
<tr>
<td>6.22</td>
<td>Peak temperature distributions on the path close to the weld line</td>
<td>144</td>
</tr>
<tr>
<td>6.23</td>
<td>Residual stress distributions on the path close to the weld line</td>
<td>144</td>
</tr>
<tr>
<td>7.1</td>
<td>The rig for girth welding</td>
<td>150</td>
</tr>
<tr>
<td>7.2</td>
<td>Main menu of mechanised girth weld program</td>
<td>151</td>
</tr>
<tr>
<td>7.3</td>
<td>The pendant controller</td>
<td>151</td>
</tr>
<tr>
<td>7.4</td>
<td>System Setup menus</td>
<td>152</td>
</tr>
<tr>
<td>7.5</td>
<td>Centering the pipe: a) radial (left) and b) transverse position (right)</td>
<td>153</td>
</tr>
<tr>
<td>7.6</td>
<td>The pipe is tilted from it ideal position: a) ideal position, b) tilted horizontally, c) tilted vertically and d) tilted arbitrarily</td>
<td>153</td>
</tr>
<tr>
<td>7.7</td>
<td>The pipe is shifted from it ideal position: a) ideal position, b) shifted horizontally, c) shifted vertically and d) shifted arbitrarily</td>
<td>154</td>
</tr>
<tr>
<td>7.8</td>
<td>Presenting dial indicator measurements in a diagram: a) radial position (left side) and transversal position (right)</td>
<td>154</td>
</tr>
<tr>
<td>7.9</td>
<td>Using hydraulic jack to minimise hi-lo problem</td>
<td>155</td>
</tr>
<tr>
<td>7.10</td>
<td>End pipe preparation (gap) and passes design</td>
<td>155</td>
</tr>
<tr>
<td>7.11</td>
<td>Camera and its box for welding torch transversal positioning</td>
<td>156</td>
</tr>
<tr>
<td>7.12</td>
<td>Welding arc and gap as seen on camera: a) before welding and b) when welding is ongoing</td>
<td>156</td>
</tr>
<tr>
<td>7.13</td>
<td>AMV 4000</td>
<td>157</td>
</tr>
<tr>
<td>7.14</td>
<td>Current probe</td>
<td>157</td>
</tr>
<tr>
<td>7.15</td>
<td>STT Lincoln GMAW machine</td>
<td>158</td>
</tr>
<tr>
<td>7.16</td>
<td>The data logger and thermocouples attached on pipes surface</td>
<td>159</td>
</tr>
<tr>
<td>7.17</td>
<td>Experiment set up</td>
<td>159</td>
</tr>
<tr>
<td>7.18</td>
<td>Mesh configurations for the girth weld joint</td>
<td>160</td>
</tr>
<tr>
<td>7.19</td>
<td>Predicted cross section area (left) and mesh design on the cross section (right)</td>
<td>160</td>
</tr>
<tr>
<td>7.20</td>
<td>Sketch of the girth welding process</td>
<td>161</td>
</tr>
<tr>
<td>7.21</td>
<td>Cartesian axes with rotations</td>
<td>162</td>
</tr>
<tr>
<td>7.22a</td>
<td>A point at ((y,z)) observed from a coordinate (R_x = \theta)</td>
<td>162</td>
</tr>
<tr>
<td>7.22b</td>
<td>A point at ((x,z)) observed from a coordinate (R_y = \theta)</td>
<td>163</td>
</tr>
<tr>
<td>7.22c</td>
<td>A point at ((x,y)) observed from a coordinate (R_z = \theta)</td>
<td>163</td>
</tr>
<tr>
<td>7.23</td>
<td>Goldak’s heat source model in rotated coordinate system</td>
<td>164</td>
</tr>
<tr>
<td>7.24</td>
<td>Elements configurations of the girth weld joint</td>
<td>164</td>
</tr>
<tr>
<td>7.25</td>
<td>Distributions of temperature when heat source is on moving</td>
<td>165</td>
</tr>
<tr>
<td>7.26</td>
<td>Temperature history of an element at 6 o’clock</td>
<td>166</td>
</tr>
<tr>
<td>7.27</td>
<td>Filtering identification of peak temperature</td>
<td>167</td>
</tr>
<tr>
<td>7.28</td>
<td>Microstructure configurations at area close to 6 o’clock position for first pass with (right side) and without (left side) filtering peak temperature</td>
<td>167</td>
</tr>
</tbody>
</table>
Figure 7.29  Coefficient of thermal expansion for (a) base metal and (b) filler metal .......................................................... 169
Figure 7.30  Yield strength at elevate temperature .......................................................... 170
Figure 7.31  Normalised yield stress at elevated temperature ........................................ 170
Figure 7.32  Yield stresses of base metal and filler metal ............................................ 171
Figure 7.33  Effects of prime and aged martensite on yield stress of base metal .............. 172
Figure 7.34  Effects of prime and aged martensite on yield stress of filler metal .............. 172
Figure 7.35  Young’s modulus (E) models for base metal and weld metal .................. 173
Figure 7.36  Thermal conductivity models for base metal and weld metal .................. 173
Figure 7.37  Specific heat models for base metal and weld metal ................................. 174
Figure 7.38  Temperature histories of nodes at varied transverse distances .................. 176
Figure 7.39  Final phase as t₈/₅ function ................................................................. 178
Figure 7.40  Welding parameters for girth weld for (a) first pass, (b) second pass, (c) third pass, (d) fourth pass and (e) fifth pass ................... 180
Figure 7.41  TMM analysis of the girth weld joint ....................................................... 182
Figure 7.42  Sketch of measuring transversal position of thermocouples (left) and nodal numbers with transversal position close to the thermocouples (right) ................................................................. 183
Figure 7.43  Validation of temperature history at node 47055 for all passes ................. 185
Figure 7.44  Validation of temperature history at node 47056 for all passes ................. 185
Figure 7.45  Validation of temperature history at node 46397 for all passes ................. 186
Figure 7.46  Validation of temperature history at node 46398 for all passes ................. 186
Figure 7.47  Mistakes due to thermocouple attachments ............................................ 187
Figure 7.48  X70A – 3 o’clock – residual stress chart ................................................ 189
Figure 7.49  X70B – 3 o’clock – residual stress chart ................................................ 189
Figure 7.50  X70A – 9 o’clock – residual stress chart ................................................ 190
Figure 7.51  X70B – 9 o’clock – residual stress chart ................................................ 190
Figure 7.52  X70 – residual stress – unwelded chart .................................................. 191
Figure 7.53  Welding residual stress at 3 o’clock of sample A .................................... 192
Figure 7.54  Welding residual stress at 3 o’clock of sample B .................................... 192
Figure 7.55  Welding residual stress at 9 o’clock of sample A .................................... 193
Figure 7.56  Welding residual stress at 9 o’clock of sample B .................................... 193
Figure 7.57  Mean value of welding residual stress at 3 o’clock ............................... 194
Figure 7.58  Mean value of welding residual stress at 9 o’clock ............................... 194
Figure 7.59  Component of stresses in local coordinate system ............................... 196
Figure 7.60  Longitudinal residual stress profiles on full and half model ................. 197
Figure 7.61  Longitudinal residual stress on enlarge portion of half model ............... 197
Figure 7.62  Transversal residual stress profiles on full and half model .................. 197
Figure 7.63  Transversal residual stress on enlarge portion of half model ............... 198
Figure 7.64  Normal residual stress profiles on full and half model ....................... 198
Figure 7.65  Normal residual stress on enlarge portion of half model ................... 198
Figure 7.66  Residual stress on 3 o’clock path ....................................................... 199
Figure 7.67  Residual stress on 9 o’clock path ....................................................... 199

Figure 8.1  DC-LSNDI of a thin plate welding ....................................................... 210
Figure 8.2  hₘₙₜ at ξₘₚ = 0 ................................................................. 212
Figure 8.3  hₘₙₜ at ξₘₚ = 84.904 mm ................................................................. 212
Figure 8.4  Temperature and hₘₙₜ distribution at ξₘₚ = 0 ..................................... 212
<table>
<thead>
<tr>
<th>Figure Reference</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 9.18</td>
<td>Normal residual stress at 3 o’clock (right) and 9 o’clock (left) paths for a model with varied annealing temperature</td>
</tr>
<tr>
<td>Figure 9.19</td>
<td>Temperature history of nodes at layer 3 and layer 4 close to the evaluated path</td>
</tr>
<tr>
<td>Figure 9.20</td>
<td>Longitudinal residual stress at 3 o’clock (right) and 9 o’clock (left) areas</td>
</tr>
<tr>
<td>Figure 9.21</td>
<td>Transversal residual stress at 3 o’clock (right) and 9 o’clock (left) areas</td>
</tr>
<tr>
<td>Figure 9.22</td>
<td>Normal residual stress at 3 o’clock (right) and 9 o’clock (left) areas</td>
</tr>
<tr>
<td>Figure 9.23</td>
<td>Longitudinal residual stress at 3 o’clock (right) and 9 o’clock (left) paths</td>
</tr>
<tr>
<td>Figure 9.24</td>
<td>Transversal residual stress at 3 o’clock (right) and 9 o’clock (left) paths</td>
</tr>
<tr>
<td>Figure 9.25</td>
<td>Normal residual stress at 3 o’clock (right) and 9 o’clock (left) paths</td>
</tr>
<tr>
<td>Figure 9.26</td>
<td>Residual stress at 9 o’clock (left) and 3 o’clock (right) for model with and without aged martensite SSPT</td>
</tr>
<tr>
<td>Figure 9.27</td>
<td>Stress strain diagram for material model used in Figure 2.15 (left) and Figure 9.27 (right)</td>
</tr>
<tr>
<td>Figure 9.28</td>
<td>Three bars model with higher yield stress of middle bar; when the middle bar is heated (left) and temperature load is omitted (right)</td>
</tr>
<tr>
<td>Figure 9.29</td>
<td>Stress and strain for yield stress of middle bar equal to 150MPa, when middle bar is heated (left) and when temperature load is omitted (right)</td>
</tr>
<tr>
<td>Figure 9.30</td>
<td>Stress and strain for yield stress of middle bar equal to 160MPa, when middle bar heated (left) and when temperature load omitted (right)</td>
</tr>
</tbody>
</table>
List of Tables

Table 1.1  Next Simdex pipeline project  ..................................................  3
Table 4.1  Summary of welding procedure ..................................................  61
Table 4.2  Thermocouple mean-position ....................................................  62
Table 4.3  Properties of materials as a function of temperature ..................  63
Table 5.1a Base metal properties .............................................................  96
Table 5.1b Weld metal properties .............................................................  96
Table 5.2  Instantaneous thermal expansion $\alpha (x \times 10^{-6}/^{\circ}\text{C})$ ........ 96
Table 6.1  Welding parameters for each passes ........................................... 117
Table 6.2  Materials content of base metal and filler metal ........................ 118
Table 6.3  Latent heat of base metal and weld metal ................................ 122
Table 6.4  Change in material properties due to SSPT ............................... 129
Table 7.1  Suggested parameters for high strength carbon steels ............... 158
Table 7.2  Chemical composition of weld metal (wire) and base metal ........ 169
Table 7.3  Mechanical properties of AWS ER706S-6 ................................ 171
Table 7.4  Latent heat of the base metal and weld metal ............................ 174
Table 7.5  Cooling rate of nodes ............................................................ 177
Table 7.6  Parameters for moving heat source model .................................. 181
Table 7.7  Thermocouples and nodes transversal position .......................... 184
Table 7.8  Thermocouples and corresponding FEM nodes ......................... 184
Table 8.1  Mathematic formulas for $\xi_L$ .................................................. 218
List of Publications

PUBLISHED PAPERS:


SUBMITTED PAPERS:

List of symbols and abbreviations

In some cases, a symbol is used to express different description. Explanation in the text where the symbol is used will make the meaning of all symbols unambiguous.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>units</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>area</td>
<td>m²</td>
</tr>
<tr>
<td>$A_{trsink}$</td>
<td>area of trailing heat sink</td>
<td>m²</td>
</tr>
<tr>
<td>$b$</td>
<td>width</td>
<td>m</td>
</tr>
<tr>
<td>$c_p$</td>
<td>specific heat</td>
<td>J.kg⁻¹.K⁻¹</td>
</tr>
<tr>
<td>$d$</td>
<td>distance between two lattice</td>
<td>m</td>
</tr>
<tr>
<td>$E$</td>
<td>young modulus</td>
<td>Pa</td>
</tr>
<tr>
<td>$g_t$</td>
<td>longitudinal parameters for trailing heat sink</td>
<td>m</td>
</tr>
<tr>
<td>$G$</td>
<td>irradiation</td>
<td>W</td>
</tr>
<tr>
<td>$h$</td>
<td>convection heat transfer coefficient</td>
<td>W.m⁻².K⁻¹</td>
</tr>
<tr>
<td>$h_{max}$</td>
<td>maximum coefficient of convection of the trailing heat sink</td>
<td>W.m⁻².K⁻¹</td>
</tr>
<tr>
<td>$h_{min}$</td>
<td>minimum coefficient of convection of the trailing heat sink</td>
<td>W.m⁻².K⁻¹</td>
</tr>
<tr>
<td>$h_r$</td>
<td>overall radiation heat transfer coefficient</td>
<td>W.m⁻².K⁻¹</td>
</tr>
<tr>
<td>$h_{sink}$</td>
<td>convection coefficient of the trailing heat sink</td>
<td>W.m⁻².K⁻¹</td>
</tr>
<tr>
<td>$I_0$</td>
<td>Bessel function first kind, order zero</td>
<td>-</td>
</tr>
<tr>
<td>$K_d$</td>
<td>reduction factor</td>
<td>-</td>
</tr>
<tr>
<td>$k_{media}$</td>
<td>constant parameters of heat sink</td>
<td>-</td>
</tr>
<tr>
<td>$l$</td>
<td>length</td>
<td>m</td>
</tr>
<tr>
<td>$l_0$</td>
<td>initial length</td>
<td>m</td>
</tr>
<tr>
<td>$m, n, p$</td>
<td>constant parameters of heat sink</td>
<td>-</td>
</tr>
<tr>
<td>$q$</td>
<td>heat</td>
<td>Joule</td>
</tr>
<tr>
<td>$q'$</td>
<td>heat rate</td>
<td>Joule/s</td>
</tr>
<tr>
<td>$q''$</td>
<td>heat generated for a certain area</td>
<td>Joule.m⁻²</td>
</tr>
<tr>
<td>$q'''$</td>
<td>heat generated for a certain volume</td>
<td>Joule.m⁻³</td>
</tr>
<tr>
<td>$q''''$</td>
<td>heat flux</td>
<td>W.m⁻¹</td>
</tr>
<tr>
<td>$q''''''$</td>
<td>volumetric heat rate</td>
<td>W.m⁻³</td>
</tr>
<tr>
<td>$q_{gen}$</td>
<td>heat generated in the system</td>
<td>Joule</td>
</tr>
<tr>
<td>$q_{in}$</td>
<td>inputted heat to the system</td>
<td>Joule</td>
</tr>
<tr>
<td>$q_{out}$</td>
<td>heat come out from the system</td>
<td>Joule</td>
</tr>
<tr>
<td>$q_{str}$</td>
<td>heat stored in the system</td>
<td>Joule</td>
</tr>
<tr>
<td>$q_{conv}$</td>
<td>convection heat transfer rate</td>
<td>W</td>
</tr>
<tr>
<td>$q_{max}$</td>
<td>maximum volumetric heat rate (at heat source center)</td>
<td>W.m⁻³</td>
</tr>
<tr>
<td>$q_{rad}$</td>
<td>heat transfer rate due to radiation</td>
<td>W</td>
</tr>
<tr>
<td>$q_x$</td>
<td>conduction heat transfer rate in x direction</td>
<td>W</td>
</tr>
<tr>
<td>$q_y$</td>
<td>conduction heat transfer rate in y direction</td>
<td>W</td>
</tr>
<tr>
<td>$q_z$</td>
<td>conduction heat transfer rate in z direction</td>
<td>W</td>
</tr>
<tr>
<td>$R_x$</td>
<td>relative distance to heat source center</td>
<td>m</td>
</tr>
<tr>
<td>$S_r$</td>
<td>tangential and radial component of stresses</td>
<td>N</td>
</tr>
<tr>
<td>$S_r$</td>
<td>tangential and radial component of stresses in cylindrical coordinate</td>
<td>N</td>
</tr>
<tr>
<td>$S_x, S_y, S_z$</td>
<td>component of stresses in x,y and z directions respectively</td>
<td>N</td>
</tr>
<tr>
<td>$r_e$</td>
<td>equivalent radius</td>
<td>M</td>
</tr>
<tr>
<td>$R_0$</td>
<td>a parameter in non dimensional integral</td>
<td>M</td>
</tr>
<tr>
<td>$R_x, R_y, R_z$</td>
<td>Coordinate rotation</td>
<td>deg, rad</td>
</tr>
<tr>
<td>$r_x, r_y, r_z$</td>
<td>parameters of heat source model</td>
<td>M</td>
</tr>
<tr>
<td>$t, \tau$</td>
<td>time</td>
<td>S</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
<td>Units</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
<td>-------</td>
</tr>
<tr>
<td>$T$</td>
<td>temperature</td>
<td>K ; °C</td>
</tr>
<tr>
<td>$T_0$</td>
<td>initial temperature</td>
<td>K ; °C</td>
</tr>
<tr>
<td>$T_{CO2}$</td>
<td>temperature of CO$_2$ snow in solid state</td>
<td>K ; °C</td>
</tr>
<tr>
<td>$T_{media}$</td>
<td>temperature of cooling media</td>
<td>K ; °C</td>
</tr>
<tr>
<td>$T_s$</td>
<td>surface temperature</td>
<td>K ; °C</td>
</tr>
<tr>
<td>$T_{w,sur}$</td>
<td>temperature at position far away from observed position; ambient/surrounding temperature.</td>
<td>K ; °C</td>
</tr>
<tr>
<td>$u$</td>
<td>a parameter in non dimensional integral</td>
<td>-</td>
</tr>
<tr>
<td>$V$</td>
<td>volume</td>
<td>m$^3$</td>
</tr>
<tr>
<td>$v$</td>
<td>a parameter in non dimensional integral</td>
<td>m$^3$</td>
</tr>
<tr>
<td>$w$</td>
<td>welding speed</td>
<td>m.s$^{-1}$ ; cm.min$^{-1}$</td>
</tr>
<tr>
<td>$w$</td>
<td>transversal length of heat sink</td>
<td>M</td>
</tr>
<tr>
<td>$z_{0,DC}$</td>
<td>z global coordinate for DC-LSND artificial center</td>
<td>m, mm</td>
</tr>
<tr>
<td>$z_j$</td>
<td>z global coordinate for an element</td>
<td>m, mm</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha$</td>
<td>thermal diffusivity</td>
<td>m$^2$.s$^{-1}$</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>coefficient of thermal expansion</td>
<td>K$^{-1}$ ; °C$^{-1}$</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>absorptivity</td>
<td>-</td>
</tr>
<tr>
<td>$\lambda$</td>
<td>wave length</td>
<td>nm, m</td>
</tr>
<tr>
<td>$\lambda$</td>
<td>thermal conductivity</td>
<td>W.m$^{-1}$.K$^{-1}$</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>emissivity</td>
<td>-</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>strain</td>
<td>m/m</td>
</tr>
<tr>
<td>$\eta_{eff}$</td>
<td>thermal efficiency</td>
<td>-</td>
</tr>
<tr>
<td>$\eta_{pre}$</td>
<td>preliminary thermal efficiency</td>
<td>-</td>
</tr>
<tr>
<td>$\nu$</td>
<td>poisson ratio</td>
<td>-</td>
</tr>
<tr>
<td>$\theta$</td>
<td>angle</td>
<td>° ; rad</td>
</tr>
<tr>
<td>$\theta_{DC}$</td>
<td>angle position of DC-LSND artificial center</td>
<td>° ; rad</td>
</tr>
<tr>
<td>$\theta_{j}$</td>
<td>angle position of elements</td>
<td>° ; rad</td>
</tr>
<tr>
<td>$\rho$</td>
<td>density</td>
<td>kg.m$^{-3}$</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Stefan – Boltzmann constant</td>
<td>W.m$^{-2}$.K$^{-4}$</td>
</tr>
<tr>
<td>$\sigma_{max}$</td>
<td>maximum stress</td>
<td>Pa</td>
</tr>
<tr>
<td>$\sigma_x, \sigma_y, \sigma_z$</td>
<td>stresses at x, y, z directions</td>
<td>Pa</td>
</tr>
<tr>
<td>$\omega$</td>
<td>a parameter in non dimensional integral</td>
<td>-</td>
</tr>
<tr>
<td>$\xi$ or $\xi_j$</td>
<td>relative position to heat source centre in a longitudinal direction</td>
<td>m</td>
</tr>
<tr>
<td>$\xi_f$</td>
<td>relative position to heat source centre in a transverse direction</td>
<td>m</td>
</tr>
<tr>
<td>Abbreviation</td>
<td>Meanings</td>
<td></td>
</tr>
<tr>
<td>--------------</td>
<td>----------</td>
<td></td>
</tr>
<tr>
<td>APDL</td>
<td>Ansys Parametric Design Language</td>
<td></td>
</tr>
<tr>
<td>APF</td>
<td>Atomic Packaging Factor</td>
<td></td>
</tr>
<tr>
<td>BCC</td>
<td>Body Centred Cubic</td>
<td></td>
</tr>
<tr>
<td>BCT</td>
<td>Body Centred Tetragonal</td>
<td></td>
</tr>
<tr>
<td>BM</td>
<td>Base Metal</td>
<td></td>
</tr>
<tr>
<td>CCT</td>
<td>Continues Cooling Transformations</td>
<td></td>
</tr>
<tr>
<td>CHT</td>
<td>Convective Heat Transfer</td>
<td></td>
</tr>
<tr>
<td>CTWD</td>
<td>Contact Tip to Work Distance</td>
<td></td>
</tr>
<tr>
<td>DC-LSND</td>
<td>Direct Cooling – Low Stress No Distortion</td>
<td></td>
</tr>
<tr>
<td>ERW</td>
<td>Electric Resistance Welding</td>
<td></td>
</tr>
<tr>
<td>FCAW</td>
<td>Flux Cored Arc Welding</td>
<td></td>
</tr>
<tr>
<td>FCC</td>
<td>Face Centred Cubic</td>
<td></td>
</tr>
<tr>
<td>FEM</td>
<td>Finite Element Method</td>
<td></td>
</tr>
<tr>
<td>GCS</td>
<td>Global Coordinate System</td>
<td></td>
</tr>
<tr>
<td>GIGO</td>
<td>Garbage In Garbage Out</td>
<td></td>
</tr>
<tr>
<td>GMAW</td>
<td>Gas Metal Arc Welding</td>
<td></td>
</tr>
<tr>
<td>GTAW</td>
<td>Gas Tungsten Arc Welding</td>
<td></td>
</tr>
<tr>
<td>GUI</td>
<td>Graphics Users Interface</td>
<td></td>
</tr>
<tr>
<td>HAZ</td>
<td>Heat Affected Zone</td>
<td></td>
</tr>
<tr>
<td>LBW</td>
<td>Laser Beam Welding</td>
<td></td>
</tr>
<tr>
<td>Mf</td>
<td>Martensite finish</td>
<td></td>
</tr>
<tr>
<td>MR</td>
<td>Melting Rate</td>
<td></td>
</tr>
<tr>
<td>Ms</td>
<td>Martensite start</td>
<td></td>
</tr>
<tr>
<td>PWHT</td>
<td>Post Weld Heat Treatment</td>
<td></td>
</tr>
<tr>
<td>SAW</td>
<td>Submerged Arc Welding</td>
<td></td>
</tr>
<tr>
<td>SCC</td>
<td>Stress Corrosion Cracking</td>
<td></td>
</tr>
<tr>
<td>SSPT</td>
<td>Solid State Phase Transformation</td>
<td></td>
</tr>
<tr>
<td>STT</td>
<td>Surface Tension Transfer</td>
<td></td>
</tr>
<tr>
<td>TIG</td>
<td>Tungsten Inert Gas</td>
<td></td>
</tr>
<tr>
<td>TM</td>
<td>Thermo Mechanical</td>
<td></td>
</tr>
<tr>
<td>TMM</td>
<td>Thermo Metallurgical Mechanical</td>
<td></td>
</tr>
<tr>
<td>UCS</td>
<td>User’s Coordinate System</td>
<td></td>
</tr>
<tr>
<td>WFS</td>
<td>Wire Feed Speed</td>
<td></td>
</tr>
<tr>
<td>WM</td>
<td>Weld Metal</td>
<td></td>
</tr>
<tr>
<td>WPS</td>
<td>Welding Procedure Specifications</td>
<td></td>
</tr>
<tr>
<td>XRD</td>
<td>X-Ray Diffraction</td>
<td></td>
</tr>
</tbody>
</table>
Chapter 1
Introduction

Withers \cite{1} discussed the role of residual stress on failure at various loads. Residual stress can be induced over a range starting from below grain size scale to the complete structure; it may affect failure mechanism at material level up to structural level. In the failure criteria for plastic collapse, residual stress is considered as secondary stress and has no significant effects on the failure. Residual stress has significant effect in fracture, i.e., catastrophic propagation of a crack. Residual stress generally leads to fracture in both regimes: linearly elastic fracture mechanics or linearly plastic fracture mechanic, by altering the stress intensity factor ($K_1$) and the total of integral $J$ respectively. However it should be noted that the increasing length of crack also alters the distribution of residual stress.

In fatigue load (mechanical or thermal) the effect of residual stress is more detrimental in high cycle fatigue (HCF) than low cycle fatigue (LCF). LCF involves fatigue at stress above yield point and the structure usually fails at less than 10,000 cycles. HCF involves a larger number of cycles at stress levels insufficient to induce plastic strain. The role of residual stress in HCF is in shifting stress intensity factor threshold ($K_{th}$).

At low stress levels over long periods creep mode dominates the failure mechanism. Creep strain can be driven by residual stress for instance in the case of thermal relaxation of weld residual stress when the residual stress is induced in area of poor material creep ductility at the operating temperature. This mechanism is found in nuclear reactor structures.

For oil, petrochemical and power generation industries and pipelines, stress corrosion cracking (SCC) may dominate the failure mechanism. This mode involves stress, corrosive environment and material susceptibility to the corrosion. Residual stress has significant effects by changing the stress factor in SCC mode which in turn determines the failure.
1.1 Background

Welding is a process that is almost universally applied for joining structures, and the welding of almost all materials can be carried out with recently developed technology. Controlling the integrity of a structure with a welded joint is extremely important because the welding process may reduce resistance to failure. One of the most important phenomena recognised in the welding process that reduces the resistance to failure is residual stress.

A study of various failed structures indicates the importance of residual stress, indeed the documented observation of failures since the 1990s clearly indicates that the majority of cracks that lead to failure were found near the welds \[^{[2]}\]. The initiation and growth rate of cracks are determined by the level of stress and the environmental conditions. A higher level of stress and a corrosive environment increase the susceptibility of welded structures to Stress Corrosion Cracking (SCC). Fundamentally stress can be working stress, residual stress, or a combination of both stresses. Vega and Hallen claimed that the tendency of SCC to occur adjacent to a weld bead is caused by high residual stress and the corrosion susceptibility of a HAZ microstructure \[^{[3]}\]. The tensile residual stress close to a welded joint generally reaches the tensile yield stress of the base metal. The corrosion rate of the HAZ in a pipeline weld made by submerged arc welding (SAW) is almost twice that of the parent pipe and is even higher if electric resistance welding (ERW) is used\[^{[2]}\]. Stress in the girth welded joints of pipelines will be increased further if displacement loads such as soil movement exist \[^{[4]}\]. The blowout of three pipelines in Argentina involved residual stress that triggered SCC failure \[^{[5]}\], and an explosion in the pipelines that transport natural gas to Sydney from central Australia, and the Trans-Canada pipelines that ruptured at Winnipeg \[^{[6]}\] occurred for similar reasons.

According to the British R6 or API 579 standards, regarding the significant effects of residual stress, it is compulsory to evaluate residual stress when assessing the integrity of welds \[^{[7]}\]. Xu et.al. emphasised the importance of reducing the residual stress because it leads to premature damage from fatigue as well as stress corrosion cracking and fracture \[^{[8]}\].
As materials transported through pipelines can reduce the long-term cost of transportation, they continue to be extensively installed \[9\]. In most instances, pipelines are used to transport liquids and gases. On the African continent, pipelines now cover North Africa, East Africa, Nigeria, South Africa and West Africa. While on the Asian continent, pipelines spread across Southeast Asia, South Asia and at East Asia. Middle Eastern countries also use pipelines to transport their oil. Pipelines also cover Europe and Russia and former Soviet Union. On the American continent pipelines, cover North America, Central America, South America and Caribbean. The Enbridge Pipeline
System is the longest oil pipeline in the world; it exceeds 5,000 km in length and stretches from Redwater (Canada) to Wisconsin (USA), and it delivers 1.4 million barrels of crude oil per day\[^{10}\]. Papua New Guinea, New Zealand and Australia also used pipelines. Pipelines in Australia are mostly used to transport gas (solid red lines) and oil (green lines), as shown in Figure 1.1. Despite the huge number of existing pipelines there are plans for even longer pipelines to be installed, as shown in Table 1.1\[^{11}\]. The plans for future pipelines in Australia are shown as dashed red lines in Figure 1.1.

Two types of welded joints are typically used in the construction of pipelines: longitudinal welds and girth welded joints. A longitudinal welded joint is a „shop“ fabricated joint where the welding conditions are well-controlled, whereas girth welding is carried out \textit{in situ}. As a perfect welding condition in the field is almost impossible to achieve, it means that the quality of a girth weld is inferior to that of a longitudinal weld under controlled condition. Furthermore, residual stress in the longitudinal weld joint of an individual pipe is easily mitigated using post weld heat treatment (PWHT) but it is quite difficult to perform PWHT on \textit{in situ} girth welds.

Since conventional welding depends almost entirely on the skill of the operator and is therefore difficult to reproduce, welding has traditionally been regarded as a craft rather than a technology. Recent research trends have been devoted to improving the basic understanding of the welding process, the efficiency of equipment, consumable, control and automation\[^{12}\]. This thesis aims to improve the knowledge on girth welding and the development of residual stress in a girth weld joint.

Due on the nature of pipeline welding, the girth weld is more susceptible to failure from the effect of residual stress. FEM was used to determine the residual stress in the girth weld, and the results have been validated experimentally. The assumptions adopted in the FEM analysis seem to correctly describe the real welding process, particularly of the girth weld joint. The model can reveal in some detail how the residual stress develops and the factors controlling it this can in turn will improve our basic understanding and lead to better control of the process.
1.2 Research objective

The objective of this research is:

to develop a FEM model that can accurately predict residual stress in a girth welded joint.

This primary objective can be divided into the following sub-tasks:
- Developing an accurate FEM model based on simple existing data of a welding process.
- FEM residual stress predictions of X-70 (a ferritic steel) girth weld.
- Experimental work to validate the FEM model.

1.3 Research approach

The girth welding process was investigated using a combination of FEM and experimental method because parameters from the experiment are required as input into FEM, and the experiment can also validate the accuracy of FEM predictions. In Figure 1.2, dashed arrows show that residual stress is obtained from both experiment or FEM model and some input data in the FEM model are obtained from the experiment.

Figure 1.2. Research aspects
In the FEM model, three aspects which are interrelated, namely thermal analysis (temperature field, temperature histories and shape of the weld pool), metallurgical analysis (CCT, dilation, phase transformation), and mechanical analysis (residual stress) are investigated. The fully-coupled Thermo-Metallurgical-Mechanical (TMM) analysis is shown in the Figure 1.2 and how these aspects affect each other will be discussed in Chapter 2.

The experimental method provides realistic values to validate the FEM model, but the experiment only provides limited results and is subject to experimental errors. For example, although an appropriate array of thermocouple based on knowledge of geometry and material properties may be able to provide sufficient information, but the temperature history using thermocouples can only be obtained for the point where the thermocouples are attached, it does not measure the whole temperature field. In another example, the distribution of residual stress is localized and not distributed over the whole area. Sometimes the experiments cannot provide the value of a parameter because of the difficulties of attaching the sensors at desired positions. While FEM modeling can add more data to the experimental method it is only a prediction, and the results still depend on assumptions made during modeling.

The FEM analysis was carried out using ANSYS, a general purpose FEM software package which provides more control for the users. In this research project, the ANSYS Parametric Design Language (APDL) mode was used because the user can input a program from the initial stage (geometry and material properties modeling) whilst another mode (Graphics Users Interface – GUI) can utilise the ANSYS tools and menus which are already available.

Since all of steps and material model must be programmed instead of using available tools, menus and data, the APDL model is more suited to the objectives of this research and allows a better understanding of stress development in the girth weld joint. And also using APDL more flexible logic can be done to accommodate all the programmer ideas.

However, this type of software with more control features poses problems in that the user can easily make mistakes without realising that the results may not be reliable. Being aware of this condition, in this thesis the program was continually tested from its inception. A comparison to the developed analytical method was carried out for a
simple moving point-heat-source in a “semi-infinite” solid. The results of moving the point-heat-source model were also compared to the uniformly distributed surface heat source model [13]. A verification of the stress developed by a thermal load was also carried out for a simple elasto-plastic three bars model [14] and a thermal analysis for simple bead-on-plate has also been carried out [15,16]. Results of FEM prediction were also compared to the existing data provided by the European Network on Neutron Techniques Standardization for Structural Integrity (NeT). A prediction of residual stress for a bead-on-plate weld was obtained through thermo-mechanical FEM analysis, which was also validated using existing NeT’s data [17]. FEM analysis of plate butt joint welding involving metallurgical analysis was carried out and is presented in Chapter 6. The FEM prediction was validated against the experiment results by Lee and Chang [18].

Based on the aforementioned experiences, a prediction of residual stress on the girth weld was carried out and the results were compared to the experimental work to validate and judge the accuracy of the model.

Furthermore, an application of the available stress mitigation technique, i.e. DC-LSND, on the girth weld was also studied through FEM simulation. From the FEM results it can be predicted that the DC-LSND can potentially be applied to advantage on pipeline girth welds.

1.4 Thesis outlines

This thesis is sub-divided into five main discussions: a general review (Chapter 2), a modeling exploration based on existing data (Chapters 3 to 6), the pipeline girth weld (Chapters 7 and 8), and a general discussion and conclusions (Chapter 9 and Chapter 10). At the end of each chapter, there is a list of the references that were used. By following these listed references readers can trace the researcher’s steps and gain a deeper insight into the topic discussed.

In Chapter 2 the basic ideas are discussed briefly, and from chapters 3 to 6 the verified or validated FEM models are discussed. The experience obtained from those chapters has provided the author confidence and insight to develop the correct FEM ANSYS model. While the knowledge and experience gained from Chapter 3 to 6 have led to the
prediction of the residual stress at the pipe girth welded joint which is presented in Chapter 7, where the experiment carried out on a “real” girth weld is described. The welding parameters were tested to ensure that the conditions were similar to the FEM simulation. Chapter 8 predicts the stress mitigation when DC-LSND is applied on the multi pass girt butt weld joint. Chapter 9 presents an overall discussion and finally, in Chapter 10 conclusions and recommendation for future work are discussed.
References


PART I

BASIC CONCEPTS
Chapter 2
A Review

Due to its high joint efficiency, lower weight and air (gas) or water tightness, welding is one of the most effective methods of joining materials, even though it does have several drawbacks such as distortion defects and residual stresses. Residual stress is regarded as the key factor to be considered in the structural integrity [1] of a weld as it plays an important role in crack initiation [2], increases the susceptibility to crack propagation [3, 4] and reduces component fatigue life [3, 5]. Residual stress increases the rate of stress-corrosion-cracking and can cause brittle fracture [5]. Moreover, it is also blamed for causing distortion which is detrimental to assembly tolerances and product quality [6].

Welding is perhaps the most non-linear problem encountered in structural mechanics because it involves many disciplines such as heat-transfer, mechanics and metallurgical science [7]. FEM is a versatile tool to gain a good understanding of the welding process in relation to the above disciplines.

A literature study showed that many important conclusions can be obtained by using an FEM to model the welding process. Thermal properties especially thermal conductivity, contribute significantly to the distribution of temperature, and it is this variation in temperature distribution that causes residual stress in a welded joint [8]. A study on different welding processes showed that the lower heat imbedded by laser beam welding (LBW) produces a lower peak temperature than gas tungsten arc welding (GTAW) especially in positions further away from the centre of the heat source. However, the peak temperature of LBW at the centre of the weld is higher due to a high concentration of heat [9]. By using FEM, the influence of the mechanical properties, especially the yield strength of the filler metal to the distribution of residual stress, can be observed [10]. FEM can also give an insight into how a solid-state phase transformation contributes to the residual stress of weldments [11]. Thus FEM simulations can

---

1 The materials in this chapter has been published in a paper:
complement welding experiments in providing the necessary data needed for the welding procedure specification (WPS)\textsuperscript{[12]}.

2.1 FEM simulation of welding processes

Since the 1970s, the scope of welding simulation has been to obtain residual stresses and the corresponding deformation\textsuperscript{[12]}. Not only does temperature contribute to the final residual stress, so too does the microstructure. How all these aspects affect the residual stress can be seen in Figure 2.1 and will be discussed in more detailed below.

![Figure 2.1 Development of residual stress in welding](image)

The first process to consider in fusion welding is thermal phenomenon i.e. the heat input from a welding heat source. This heat is concentrated in a small volume of base metal which causes fusion in this region. In the early stage, this concentrated heat is released to the cooler region mainly by conduction, although this release of heat from the base metal to its surroundings also occurs by convection and radiation. The heat is also absorbed by the base metal when phase transformation occurs (due to latent heat). The thermal properties of the base metal determine the rate and value of this heat transfer phenomenon (arrow 1 in Figure 2.1). By considering the law of energy conservation and boundary conditions, the transient temperature of certain positions can be obtained. If the distribution of temperature is the final goal, the analysis is called a thermal analysis of the welding process.

Further analysis should be carried-out to obtain the distribution of residual stress. Based on the distribution of temperature in the thermal analysis and considering the coefficient
of thermal expansion, thermal deformation can be calculated. Since the temperature is not uniformly distributed the elongation is also not uniform and as a result thermal stresses will be produced (arrow 2). The deformation that occurs may also change the thermal boundary condition, such as the contact body, to the welded base metal. Also, the plastic and elastic-strain produces heat that affects the temperature of the welded base metal (arrow 3). Analysis up to this stage is called thermo-mechanical (TM) analysis.

For base metals that experience a phase transformation, the welding process produces temperatures high enough to initiate a phase transformation. For instance, when the base metal is ferritic-steel the ferrite is transformed into austenite when heated and since the cooling rate of welding is high enough, when the base metal has cooled down martensitic structures may exist in the final weldments. Thermo-metallurgical-mechanical (TMM) analysis considers this solid state phase transformation phenomenon. The developed microstructure depends on the temperature history (arrow 4). Different microstructures have specific thermal properties which in turn will affect the heat transferred to the cooler region or influence the transient temperature of certain positions (arrow 5). The varied microstructure has a certain thermal expansion coefficient that determines the deformation. Also, significant expansion or shrinkage occurs due to the solid state phase transformation. This expansion or shrinkage in turn affect the developed stress (arrow 6), while Lindgren\textsuperscript{12} reported that the developed stress also influences the microstructure (arrow 7).

\section{2.2 Thermal analysis of welding FEM model}

Thermal analysis is a fundamental step in modeling welding phenomenon and any errors in thermal modeling can invalidate subsequent analysis. The welding process has been modeled as a moving heat source over a solid. The early analysis used the moving point heat source\textsuperscript{13}. Some paper on FEM modeling also used the point heat source model\textsuperscript{14-16} to represent the heat input from a welding torch. The heat source may be modeled as a surface heat source $\dot{q}''$ (J/m$^2$s)$\textsuperscript{11, 16-18}$ which can be distributed uniformly or according to a Gaussian distribution. The heat source can also be represented as a volumetric heat source $\dot{q}'''$ (J/m$^3$s)$\textsuperscript{19-21}$ which is a body heat load applied for certain
volumes. As with the surface heat source, the volumetric heat source can be distributed uniformly or according a certain pattern.

The heat input to the base metal by the welding heat source is represented by the aforementioned moving heat source model. The thermal analysis in Figure 2.1 involves heat transferred to the cooler regions of base metal while the heat is simultaneously transferred to the surroundings by convection and radiation.

2.2.1 Thermal properties of a material

Conductivity. Incropera and DeWitt\textsuperscript{[22]} defined conductivity as a material property that describes the transport of thermal energy. If heat is only transferred in the x-direction then conductivity can be expressed as: \( \lambda_x \equiv \frac{q_x}{\left( \frac{\partial T}{\partial x} \right)} \). For a certain temperature gradient a higher conductivity will increase the heat flux for each unit of time. The other two axes can be treated similarly. For isotropic material, the conductivity along the three axes is equal. The actual conductivity for certain materials such as metal depends on the temperatures. Typical temperature dependent conductivity will be described in Chapters 4, 6 and 7 where FEM material models are discussed.

Specific heat. Specific heat is defined as the energy required to change the temperature of a unit mass of material by one degree. The higher the specific heat is, the harder it is to increase its temperature. Cengel and Boles\textsuperscript{[23]} described specific heat as the capacity of materials to store energy. The value of specific heat for certain materials depends on the temperature and the typical temperatures dependent specific heat are shown in Chapters 4, 6 and 7.

Enthalpy. Enthalpy (\( h \)) was first called the heat content and was proposed by Mollier. It represents a combination of properties: \( u + PV \), where \( u \) is the internal energy stored in a system and \( PV \) is the work done by a system. For phase transformation stages, enthalpy is also known as latent heat. Enthalpy became important in the welding process since it involve phase transformation from solid to liquid and from liquid to solid.

Coefficient of convection. The coefficient of convection \( h \), describes the conditions of the boundary layer on the interface between fluid and solid. A higher \( h \) means more heat is transferred from higher to lower temperatures. Two factors that have a significant influence on \( h \) are the surface geometry and the nature of fluid motion.
Emissivity. Emissivity is a radiated property of a material surface. The higher the emissivity the more is the energy emitted by the radiation. An ideal radiator, that is a perfect black body, has an emissivity value equal to 1. Typically, a surface has a value of emissivity below unity.

2.2.2 Heat transfer and heat conservation

As discussed previously, there are three modes of heat transfer: conduction, convection and radiation; and all three exist simultaneously in the welding process.

Conduction is heat transfer within a material due to differences in temperature. The energy (heat) is transferred from a higher temperature to a lower temperature. Fourier’s law expressed this heat transfer phenomenon by the following expressions:

\[ q_x = -\lambda A \frac{\partial T}{\partial x} \]  \hspace{1cm} (2.1)

\[ q_y = -\lambda A \frac{\partial T}{\partial y} \]  \hspace{1cm} (2.2)

\[ q_z = -\lambda A \frac{\partial T}{\partial z} \]  \hspace{1cm} (2.3)

where \( q_x, q_y, q_z \) = conduction heat transfer rate in the x, y and z directions respectively (W)

\( \lambda = \) thermal conductivity \( \left( \frac{W}{m \cdot K} \right) \)

\( A = \) perpendicular area \( (m^2) \)

\( \frac{\partial T}{\partial x}, \frac{\partial T}{\partial y}, \frac{\partial T}{\partial z} = \) temperature gradient in the x, y and z directions respectively \( (^\circ C/m) \)

Convection is heat transferred by the movement of a fluid. As in conduction, the difference in temperature between solid and fluid is a prerequisite for convection where the energy (heat) is transferred from a higher temperature to a lower one. This energy is transferred by random molecular motion (which is usually called diffusion) and the relative motion of the bulk fluid. The convection heat transfer can be expressed by the following expression:

\[ q_{conv} = hA(T_s - T_w) \]  \hspace{1cm} (2.4)
where \( q_{\text{conv}} \) = heat transfer rate due to convection (W)

\[ h = \text{convection heat transfer coefficient} \left( \frac{\text{watt}}{m^2 \cdot K} \right) \]

\( A \) = area of contact (m\(^2\))

\( T_s \) = temperature of solid surface (°C)

\( T_\infty \) = temperature of fluid far enough from the surface (°C)

Radiation is heat transport by electromagnetic waves. The radiation heat transfer rate can be expressed by the Stefan–Boltzmann law:

\[
q_{\text{rad}} = \varepsilon \sigma T_s^4
\]  \( (2.5) \)

where \( q_{\text{rad}} \) = heat transfer rate due to radiation (W)

\[ \varepsilon = \text{emissivity} \ (0 < \varepsilon < 1) \]

\[ \sigma = \text{Stefan–Boltzmann constant} \left( 5.67 \times 10^{-8} \frac{\text{watt}}{m^2 \cdot K^4} \right) \]

\( A \) = surface area (m\(^2\))

\( T_s \) = surface temperature (°C)

Generally a surface emits and receives radiation; the received radiation is usually called irradiation, which in turn is reflected, transmitted, and absorbed. The absorbed irradiation is expressed as:

\[
G_{\text{abs}} = \alpha G
\]  \( (2.6) \)

where \( \alpha \) = absorptivity of surface \( (0 < \alpha < 1) \)

\( G \) = irradiation from others (W)

The value of \( \alpha \) depends on the surface properties and the nature of irradiation.

The net radiated heat is the difference between radiation and irradiation which is expressed as:

\[
q_{\text{rad}} = \varepsilon \sigma A(T_s^4 - T_\infty^4)
\]  \( (2.7) \)

where \( T_{\text{sur}} \) is the surrounding temperature of the evaluated surface.
In welding both convection and radiation occur, so it is convenient to express radiation in a linear form, similar to convection.

\[
q_{\text{rad}} = \varepsilon \sigma A (T_s^4 - T_\infty^4)
\]

\[
= \varepsilon \sigma A [(T_s + T_\infty)(T_s^2 + T_\infty^2)](T_s - T_\infty)
\]

\[
= \{\varepsilon \sigma [(T_s + T_\infty)(T_s^2 + T_\infty^2)]\} A(T_s - T_\infty)
\]

\[
= h_r A(T_s - T_\infty)
\]  

(2.8)

\(h_r\) is usually called the overall radiation heat transfer coefficient. It should be noted that \(h_r\) depends on the thermal properties of the materials and also on the temperature. The FEM tool (ANSYS) which will be used to analyse residual stress in this thesis has facility to model non linear radiative heat loss, but this linear model is useful in analytical stage. Some researchers (see chapter 4) have used this overall heat transfer coefficient and the above discussion gives some background on this parameter.

In welding, the transient conduction heat transfer problem is connected with a convection boundary condition on the surfaces of the base metal. If radiation is also taken into account, the heat transfer by radiation is added to that by convection and it is preferable to use the overall radiation heat transfer coefficient \(h_r\) (equation 2.8) since the heat transfer rate can be expressed as a linear function (instead of power 4) of temperature difference between the solid surface and the surrounding air.

Whatever the form of heat transfer (conduction, convection, and radiation, or a combination of these), it always follows the energy conservation law. For a certain system, the energy conservation law is expressed as:

\[
q_{\text{in}} + q_{\text{gen}} - q_{\text{out}} - q_{\text{str}} = 0
\]

which for convenience is expressed as:

\[
q_{\text{in}} - q_{\text{out}} + q_{\text{gen}} = q_{\text{str}}
\]

(2.9)

where
- \(q_{\text{in}}\) = input heat to the system (Joule)
- \(q_{\text{out}}\) = heat coming out from the system (Joule)
- \(q_{\text{gen}}\) = heat generated in the system (Joule)
- \(q_{\text{str}}\) = heat stored in the system (Joule)
Equation 2.9 is usually called a conservation energy equation. Heat stored in the system can be written in the following expression:

\[ q_{str} = \rho c_p dT dV \tag{2.10} \]

where  
\[ \rho = \text{density (kg/m}^3\text{)} \]
\[ c_p = \text{specific heat at constant pressure (J/kg.K)} \]
\[ dV = \text{volume increments (m}^3\text{)} \]

Applying energy conservation to the incremental volume with sides: dx, dy and dz results in the following expression:

\[ \{q_x + q_y + q_z\} - \{q_x + dx + q_y + dy + q_z + dz\} + q''''dxdydz = \rho c_p dT dx dy dz \]

where \( q'''' \) is heat generated for a certain volume. Substitute \( q_x + dx = \frac{\partial q_x}{\partial \xi} dx \) in the x-direction and similarly for the two other axes and expand conduction heat transfer equation with a constant thermal conductivity can be written as:

\[ \frac{\partial^2 T}{\partial \xi^2} + \frac{\partial^2 T}{\partial \sigma^2} + \frac{\partial^2 T}{\partial 

\[ \text{z}^2 \} + \dot{q}'''' = \frac{1}{\alpha} \frac{dT}{dt} \tag{2.11} \]

where \( \alpha = \frac{\dot{q}}{c_p \rho} \) is known as thermal diffusivity (m\(^2\)/s).

The temperature in a solid can be determined from equation 2.11 subject to the boundary conditions of natural or forced convection and radiation.

### 2.2.3 Parametric study via FEM

In this chapter the simplest moving heat source model (point heat source) is used. How parameters contribute to the distribution of temperature is demonstrated using an FEM model. Typical welding parameters and material properties are: heat load rate \( \dot{q} = 4200\text{J/s}, \) welding speed \( v = 6\text{cm/minute}, \) thermal conductivity \( \lambda = 42 \text{ W.m/}^\circ\text{C}, \) specific heat \( c_p = 525 \text{ (J/kg.}^\circ\text{C)} \) and density \( \rho = 8000 \text{ kg/m}^3. \) These parameters were applied on thin plate (thickness equal to 1 mm) of 175mm long and 120mm width. At this stage, adiabatic boundary condition was applied on surfaces to simplify and focus the problem
on the effect of parameters. SOLID70 brick elements with 1mm edge length were used in FEM modeling as shown in Figure 2.2. Isothermal temperatures were evaluated on top surface of the model and the results are studied and presented in Figures 2.3 – 2.5.

![Finite element model for parametric study.](image)

The first parameter considered is the intensity of the heat source at two levels 3150J/s and 4200 J/s. The origin describes the instantaneous position of the heat source. All the positions are considered to be relative from the position of the heat source. The heat source at this stage is modeled as a point heat source and moves from left to the origin. Isothermal lines for 1200°C, 600°C, 400°C, 150°C and 100°C are plotted after a certain time, as shown in Figure 2.3. Using a lower heat input results in the isothermal lines covering a narrower area than for a higher heat input. The longer time means that the difference between the same isothermal lines is wider. This phenomenon is found in all time spans up to 200s where the condition reaches a quasi steady state. Once the heat source had traversed far enough, the shape of the isotherms are steady. This condition is known as quasi steady state. The quasi steady state will be discussed in more detail in Chapter 3.
Figure 2.3. Transient isothermal lines for moving heat source with varied heat; (a) 10s, (b) 50s, (c) 100s and (d) 200s

Figure 2.4 shows the distribution of temperature for different thermal conductivity. The lines show an isothermal line at 1200°C, 600°C, 400°C, 150°C and 100°C respectively. The lower thermal conductivity (30 W.m/°C) is used to compare with the previous value of 42 W.m/°C. It can be seen here that the lower thermal conductivity is significantly different to the typical parameter in the longitudinal direction, but not in the transverse direction. A lower thermal conductivity keeps the high temperature close to the weld line longer than the higher value and as a result Figure 2.4 shows isothermal lines stretching in the longitudinal direction for certain time.

Figure 2.5 shows the temperature field for different specific heats. For a higher specific heat 700 (J/kg°C), the isothermal lines differ significantly in the transverse direction (y axis) whilst in the longitudinal direction the isothermal lines cover almost the same length. With a higher specific heat, a higher heat is needed to increase the temperature in the transverse direction, but since the heat is kept constant, the isothermal lines for the higher heat, covers a narrower width than the lower value.
Figure 2.4. Transient isothermal lines for moving heat source with varied thermal conductivity; (a) 10s, (b)50s, (c)100s and (d)200s

Figure 2.5. Transient isothermal lines for moving heat source with varied specific heat; (a) 10s, (b)50s, (c)100s and (d)200s
2.3 Development of stress in welding

The distribution of temperature in the welding process is not uniform, which causes a non-uniform thermal strain in the welded structure. This strain in turn causes thermal stress, as a result of misfit which grows until a uniform temperature at the welded structure is attained. The stress which still exists when the temperature of the welded structure is already uniform is called residual stress. The simplest explanation of how residual stress has developed in the welded structure is the three bars model, which will be presented in the next discussion.

Consider the three bars which are clamped together at their ends with a rigid body, as shown in Figure 2.6. First, as described in Figure 2.6a, the middle bar is heated and elongates due to thermal expansion. A compressive stress is exerted because its elongation is constrained by the side bars. Typically in a welding process the compressive stress exceeds the yield stress and as a result a plastic compressive strain is developed at the middle bar.

Next, as shown in Figure 2.6b, the middle bar is left to cool by transferring the heat to the environment. While it is cooling the middle bar shrinks according to its coefficient

![Figure 2.6 Thermal stress in the three bars model (modified from Sindo Kou[24])](image)
of thermal expansion but because compressive plastic strain has already developed in the middle bar, when it cools down to its initial temperature it is now shorter than its initial length. The final tensile stress is developed in the middle bar as a result of misfit.

In a welded structure, the weld metal and adjacent base metal can be considered as the middle bar in the three bars model, and areas further away from the weld metal that are not significantly affected by heat from the welding torch, can be considered as side bars. The final residual stress in the area close to the weld metal is in tension whilst in the area further from weld metal is in compression. Typical residual stress of butt joints obtained from experiments is shown in Figure 2.7.

![Figure 2.7. Typical distribution of residual stress in butt joint welding: (a) longitudinal residual stress, (b) transversal residual stress.](image)

The mechanical properties that contribute to the residual stress are thermal expansion, yield stress, and the stress-strain diagram for the base metal. The residual stress determined by both FEM and an analytical calculation for the very simple three bars model will be discussed in the next section.

### 2.3.1 A review of the three bars model

Many theories have been proposed to explain the development of thermal stress in the welding process. The simplest mechanism of the formation of residual stress is
proposed by Wells \[25\] and quoted by Lin and Chou \[26\]. The schematic diagram in Figure 2.8, modified from \[26\], shows a typical thermal cycle of the region close to the fusion line in the welding. As the region is heated from room temperature (point O) the volume expands, but this expansion is restrained by the surrounding bulk of cold material which first causes elastic compressive stress (point A). Further heating will develop plastic compressive stress (point B). It should be noted that the compressive stress at point B is lower than point A as a result of the higher temperature. When this region begins to cool, this plastic compressive stress still exists and because of misfit an elastic tensile stress is developed (point C). Further cooling to room temperature will result in the plastic tensile stress (point D) known as residual stress (also called thermal residual stress).

\[\text{Figure 2.8. Thermal residual stress formation}\]

The coefficient of volumetric thermal expansion as a material property is expressed by equation 2.12.

\[\alpha_v = \frac{1}{V} \left( \frac{\partial V}{\partial T} \right) \quad (2.12)\]

V is denoting volume and T is temperature.

If only a simple one dimensional case is studied, equation 2.12 can be expressed as the following expression:

\[\alpha = \frac{1}{l} \left( \frac{\partial l}{\partial T} \right) \quad (2.13)\]

where \(\alpha\) is coefficient of thermal expansion, \(l\) is length of heated material. For isotropic material \(\alpha = \alpha_v/3\). If the coefficient of thermal expansion is temperature independence, equation 2.13 can be simply expressed as the following expression:
\[ l_T = l_0 (1 + \alpha \Delta T) \]  \hspace{1cm} (2.14)

where \( l_T \) is final length as a result of temperature rise, \( l_0 \) is initial length and \( \Delta T \) is temperature rise.

Following the aforementioned theory, the development of residual stresses via analytical and FEM solutions will be demonstrated here. It is first modeled as a three bar model, which is shown in Figure 2.9. The three bars are connected at their ends with two other rigid bars. First, the middle bar is heated to a certain temperature and the longitudinal thermal stresses that have developed are then analysed. In the next step the previous temperature load is omitted and the final residual stresses are determined. The three bars have a 1mm x 1mm cross section and are 10mm long. Their typical properties are: density \( \rho = 8000 \) (kg/m\(^3\)), thermal conductivity \( \lambda = 20 \) (W/m.\(^\circ\)C), specific heat \( c_p = 500 \) (J/kg.\(^\circ\)C), Young’s modulus \( E = 15 \) GPa, Poisson’s ratio \( \nu = 0.3 \), thermal expansion \( \alpha = 20 \times 10^{-6} /{\circ}C \). The two bars that clamp the ends of the three bars have the same properties, except that \( \lambda = 20 \times 10^{-6} \) (W/m.\(^\circ\)C), \( \alpha = 20 \times 10^{-20}/\circ\)C and \( E = 10 \times 10^{10} \) GPa. With a much lower \( k \) it is expected the elevated temperature will only be localised in the middle bar. The very low \( \alpha \) means that no thermal expansion exists and a high \( E \) is to model the perfect toughness of the clamps.

Figure 2.9. Three bars model
An elasto-plastic material model was used and to avoid the problem of convergence in the plastic state, a very low E was applied as shown in Figure 2.10 which is represented in Figure 2.11. Subscript “pl” refers to the plastic region and subscript “y” refers to yield. The first 400°C of temperature applied on the middle bar will give a strain under the elastic limit. When the temperature load is omitted there should be no stresses left on the system. The analytical solution which will give an understanding of the thermal phenomenon follows the flow chart in Figure 2.12. The analytic solution only considers force and elongation parallel to the three bars and one dimensional analysis was taken.

![Figure 2.11. Sketch of the stress-strain diagram.](image)
for simplicity. The flow chart follows the equilibrium force law, and since the cross-section of the three bars are the same, the equilibrium force also means equilibrium stress. Also the flow chart considers the aforementioned residual stress mechanism theory. Firstly, the tentative stress at the side bars is taken arbitrarily, and by using the aforementioned mechanism of residual stress, the stress in the middle bar can be calculated. The stress in the middle bar is checked and should be twice the stress at the side bars. If this condition is not fulfilled, the tentative stresses in the side bars are corrected. These steps are repeated iteratively until the condition is met. It should be noted that the stress in the side bars should be doubled since there are two bars ($2\sigma_{sd} = \sigma_{md}$). Subscript “$sd$” refer to side bar and “$md$” refer to middle bar. The analysis for thermal stress is done when the middle bar is heated and when the temperature load is omitted. Following the flow chart at Figure 2.12, the stresses when the mid bar is heated to $400^\circ C$ are $\sigma_{md} = -79.58$ MPa (compression) and $\sigma_{sd} = 39.79$ (tension). The distribution of stress that resulted from the FEM simulation is shown in Figure 2.13. The bottom half of the model is shown in Figure 2.13b. The thermal stress in the middle bar is $-79.49$ MPa and $39.88$ MPa in the side bar. The difference with the analytical solution may be caused by excluding expansion in the lateral direction as a result of the temperature load and stress load. Also, the analytical solution does not accommodate the distribution of stress at the ends of the bars which causes localised high strain in that region. Although the localised stress may have no effect on the stress at the middle cross section area (since it is further away) but localised strain will give a different total length, which is used to determine the theoretical developed stress as shown by Figure 2.12. From the FEM side, the rigid clamping bar is modeled by a high Young’s modulus which is still not perfectly rigid. However, the difference between the analytical solutions with FEM simulation is low (0.11%). When the temperature load is omitted, there are no stresses left on the three bars since no plastic strain existed when the system was heated initially.
Figure 2.12. Flow-chart for evaluating thermal stress in elastic strain state.

Figure 2.13. Distribution of stress when the middle bar is heated to 400°C (a) full model and (b) half model.
Figure 2.14. Flow-chart for evaluating thermal stress in plastic strain state.

\[
\begin{align*}
\sigma_{th} & \quad \rightarrow \\
\varepsilon_{th} & = \frac{\sigma_{th}}{E} \\
\Delta l_{th} & = \varepsilon_{th} \times l_0 \\
\Delta l_{med} & = l_0 \alpha_0 \Delta T - \Delta l_{th} \\
\varepsilon_{med} & = \frac{\Delta l_{med}}{l_0 (1 + \alpha_0 \Delta T)} \\
\sigma_{med} & = \varepsilon_{med} E_{pl} + (\varepsilon_{med} - \varepsilon_0) E_{pl} \\
\sigma_{med} & = 2\sigma_{th}^2
\end{align*}
\]

Figure 2.15. Distribution of stress at the three bars model heated to 1100°C. (a) when the middle bar is heated, (b) when the temperature load is omitted.
The following analysis demonstrates the three bar model where plastic deformation has occurred, when the middle bar is heated to 1100°C. The analytical solution for thermal stress when the middle bar is heated follows the flow chart in Figure 2.14. Following the flow-chart, when the three bars are heated the stress at the middle bar and side bars is $\sigma_{md} = -150.36$ MPa and $\sigma_{sd} = 75.18$ MPa respectively. The FEM simulation produces a distribution of stress as shown in Figure 2.15a, with a stress at the middle bar of $\sigma_{md} = -150.12$ MPa, which shows an insignificant difference with the analytical model (0.16%).

When the temperature load is removed, there should be a distribution of stress as a result of previous compressive plastic strain when the system is heated. The FEM simulation for the remaining stress (residual stress) shows that the stresses are $\sigma_{md} = 68.22$ MPa and $\sigma_{sd} = -34.10$ MPa for the middle bar and side bars respectively. The analytical solution for the residual stress follows the flow chart in Figure 2.16. Following the flow chart, the residual stress at the middle bar is 67.73 MPa and -33.87 MPa at the side bars, which again shows a small difference with the FEM simulation (0.72%).
Overall, the FEM simulation and analytical solution show good agreement. The residual tensile stress at the middle bar and side bars can be determined by both methods. The analytical solution followed assumptions made by Wells [25] and the equilibrium stress theory. The confirmations from the FEM results have demonstrated the proposed mechanism of residual stress formation.

2.4. Solid state phase transformation and stress development

As demonstrated in section 2.3 the stress arises as a result of various strains. In section 2.3 only strains resulting from elevated temperature is discussed. When this strain is constrained it causes stress. It should be noted that since in welding the temperature distribution is not uniform the thermal strain will also not be uniform and the elongation in the heated area is constraint by far bulk material which is not significantly elongated by the thermal stress. Thus it no needs the presence of physical constraint to produce stress in welding process although the presence of physical constraint alters the distribution of residual stress. The stress induced by this thermal load can be classified as elastic and plastic strains. The plastic strain which is irreversible significantly contributes to the final residual stress since it causes misfit when the welded structure cooled to the room temperature.

When welding is applied on ferritic steels, it does not only produce thermal, elastic and plastic strains but also creep, volume change and also transformation plasticity strains as result of phase transformation. Comprehensive discussion can be found in literatures [27 – 34] which is partially discussed below.

Creep strain is not significant in the welding residual stress analysis and usually ignored [29] and the total strain can be expressed as in equation 2.15.

\[ \varepsilon_{\text{Tot}} = \varepsilon^{E} + \varepsilon^{P} + \varepsilon^{Th} + \varepsilon^{TrV} + \varepsilon^{TrP} \]  

(2.15)

\( \varepsilon^{\text{Tot}} \) is total strain, \( \varepsilon^{E} \) is elastic strain, \( \varepsilon^{P} \) is plastic strain, \( \varepsilon^{Th} \) is thermal strain, \( \varepsilon^{TrV} \) is strain due to volumetric change of phase transformation and \( \varepsilon^{TrP} \) is transformation plasticity. Concerning strain as result of phase transformation, volumetric strain and transformation plasticity will be discussed briefly in this section.
Since the welding process exposes a welded material to such a high temperature (above A1), a solid state phase transformation exists for parent materials such as feritic-steels. When a ferritic-steel is heated to a critical temperature A1, it is transformed into an austenite, a transformation that is completed at a critical temperature A3. The atomic structure of feritic-steels is body-centred-cubic (BCC) whilst the atomic-structure of austenitic-steels is face-centred-cubic (FCC). Since the total amount of atoms is constant, this atomic arrangement affects the volume of the transformed steels. A non dimensional parameter called the atomic packaging factor (APF) describes the volume fraction occupied by atoms. Mathematically, the APF is expressed as the atomic volume per total occupied volume. For simplicity, the atoms are described as a solid ball. The higher the APF the denser the atom which means the volume is less for a certain number of atoms. The APF of BCC is 0.68 whilst for FCC it is 0.74 which means when the ferrite transforms to austenite the transformed steel shrinks. It should be noted that 0.74 is the highest possible APF. This transformation is described with the solid line in Figure 2.17.

![Figure 2.17 Solid state transformations in ferritic steels.](image)

When austenite is cooled to the Ms temperature, it begins to transform to martensite, and this transformation is finished when the temperature is equal to Mf. Martensite has a body-centred-tetragonal (BCT) structure with APF equal to 0.67. With this lower APF, the transformed steel have a higher volume, as described in Figure 2.17. Since the shrinkage and expansion of the phase transformation affects the thermal stress it should be considered in the analysis. A more detailed analysis of phase transformation and its effect on the models will be discussed in Chapter 6.
Transformation plasticity is anomalous plastic behavior of steels when a phase transformation takes place. The plastic strain depends on proportion of phases and stress. According to Greenwood and Johnson [35] the microscopic plasticity is generated in the phase with lower yield stress in the presence of stress. The stress permits macroscopic plastic flow even if the load itself is insufficient to produce plasticity. Another mechanism is proposed by Magee [36] which is only applicable for the martensitic transformation. According to Magee steels under stress during martensitic transformations will form martensite plates and this affects the overall shape of the continuum. The stress causes transformation strains which are at the microscopic scale and do not average out to zero. No matter which mechanism is followed (Greewood-Johnson or Magee), it can be seen that the presence of residual stress in welding process while phase transformation takes place causes additional plastic strain which is called transformation plasticity. This transformation plasticity will be discussed in more detailed in chapter 7.
References


PART II

FEM
Vs
ANALYTICAL SOLUTION
Chapter 3
Moving Heat Source Simulations

Basically, the finite element method (FEM) considers that a structure is constructed from simple elements which are connected at their nodes, and fulfill equilibrium and compatibility conditions. Based on this definition, the first step in any FEM analysis is to divide an observed structure into elements.

Dividing a structure into discrete elements was pioneered by Alexander Hrennikoff \textsuperscript{[1]}. O. Zienkiewicz (1947) established a mathematical formulation for discrete elements \textsuperscript{[2]}.

Since a typical structure consists of a huge number of elements, a large number of equations must be solved and so involvement with a computer cannot be avoided. The first software to use the finite element method was NASTRAN, which was established by NASA in 1965. Today, many software packages are available and in this thesis ANSYS was used to carry out the finite element analysis.

One of the first applications of the finite element methods to welding can be found in a 1970’s in a published paper by Tall \textsuperscript{[3]}. In his paper the thermal analysis was modeled in a two dimensions whilst the mechanical analysis was essentially a one dimensional model. Friedman \textsuperscript{[4]} calculated the temperature, stress, and distortion, in a two-dimensional analysis. A constant temperature profile was used as a thermal load which moves along the weld line. Muraki T. \textit{et al.} \textsuperscript{[5]} developed a thermo-mechanical finite element computer program by using an elasto-plastic material model. In the 1970’s other papers were published where a two dimensional model was used \textsuperscript{[6 - 9]}. The possibility of a three dimensional model was restricted by the availability of the required computing power. Under these limited conditions, plane strain \textsuperscript{[10-14]}, plane stress\textsuperscript{[15-19]} and axis symmetry\textsuperscript{[20,21]} considerations were made in the approach to three dimensional modelling.

\textsuperscript{2} Material in this chapter has been published in a paper: Djarot B. Darmadi, John Norrish and Anh Kiet Tieu, \textit{Analytic and finite element solutions for temperatures profiles in welding using varied heat source models}, World Academy of Science, Engineering and Technology (WASET) 81, Singapore, 2011.
The most important aspect of modeling is accuracy which required an integrated approach which combines the FEM, analytical and experimental method. How the FEM and the analytical and experiment methods are linked to each other is shown in Figure 3.1.

![Figure 3.1 FEM, analytical solution and experimental results.](image)

From Figure 3.1 it can be seen that, FEM is verified by analytical solutions and validated with experimental measurements. Most published papers that are concerned with FEM modeling involved validation to judge the correctness of those models. A few papers, especially on welding simulation used analytical solutions to verify the results obtained from FEM analysis.

Rahman and Brust evaluated the propagation of cracks in welded pipes. They compared their proposed solution, which is based on the energy release rate theory, with the FEM model \[22\]. Anca et al.\[23\] compared the distribution of temperature obtained from the analytical method and FEM to predict the liquid region, the mushy region, and the solid region in the welding processes. Lin et al. evaluated the distribution of stress surrounding spot weld regions using a mathematical model and FEM \[24\]. Analytical solutions for temperature profiles using the point heat source and uniform disc surface heat source model obtained from the mathematical approach were verified with FEM results \[25\].

The analytical solution typically solves simple conditions with an exact solution which is considered to be correct, while FEM gives approximate solutions but can solve complex conditions (especially geometry). Both may be subject to possible errors in the assumptions made concerning material properties. Thus, verification can only be done on simple conditions where analytical solutions can be obtained. Validation compares the FEM results with experiment measurements. In the FEM model assumptions are
made to simplify the real conditions which are complex. A good FEM model can provide results that are close to both analytical solution and experimental measurements. This chapter presents verifications that were carried out to ensure a reliable approach was developed in the early stages of FEM modeling.

3.1 Solutions by Rosenthal

Rosenthal\(^{[26]}\) proposed a solution for the moving point heat source which is expressed in equation (3.1), where \(\Delta T\) is the rise in temperature, \(v\) is the moving speed of the heat source, and \(\alpha\) describes the thermal diffusivity of the solid. This proposed solution is based on the shape of the weld-pool. Since equation (3.1) is not a time (t) function, it is a solution for the quasi steady state. A quasi steady state is a steady state if the parameters are considered from moving coordinates \((\xi, y'', z'')\). The fixed coordinate system is expressed as \((x, y, z)\), which follows the right hand rule. The coordinates are described in Figure 3.2. Since the heat source is moving parallel to the x axis, the value of \(y''\) is equal to \(y\) and the value of \(z''\) is equal to \(z\).

\[
\Delta T = \exp\left(-\frac{v\xi}{2\alpha}\right) \cdot f(\xi, y', z') \tag{3.1}
\]

Equation (3.1) consists of the asymmetric function: \(\exp\left(-\frac{v\xi}{2\alpha}\right)\) and symmetric function: \(f(\xi, y', z')\). The asymmetric function is found along the direction parallel to \(\xi\) and the symmetric function is found along lines that are parallel to \(y\) and parallel to \(z\). If the welding speed equals zero, the asymmetric function will be a unity and only the symmetric function is left. Zero welding speed means a case of heat liberated by a stationary point which is much easier to obtain a solution for than the moving point heat source. A final solution proposed by Rosenthal for the moving point heat source on a semi-infinite solid is expressed in equation (3.2) where \(\lambda\) is the conductivity, and \(R\) is the distance from the heat source. \(T_0\) is added to account for the initial temperature of the welded plate. The symmetric function of equation (3.2) is \(\frac{1}{2\pi\lambda R} \exp\left(-\frac{vR}{2\alpha}\right)\).
Equation (3.2) is a solution for a quasi steady state condition since it is time independent.

3.2 Moving point heat source model

The moving point heat source is the simplest way to model welding phenomena. Two analytical solutions are used in this thesis; the first is Rosenthal’s equation which is already available, and new analytical solutions based on a solution to the conservation of energy proposed by H.S. Carslaw and J.C. Jaeger[27]. The solution by Carslaw and Jaeger in this thesis was developed using a non-dimensional integral techniques to obtain the solution for a moving point heat source. The non-dimensional integral follows the method used by R. Komanduri and Z.B. Hou[28].

Analytical solutions to welding heat flow problems were obtained by solving the partial differential equation of energy conservation (equation 3.3), where $T$ is the temperature, $x$, $y$, and $z$ are the three mutually orthogonal directions, $\alpha$ is the diffusivity, and $t$ is the time. Steady state solutions can be obtained by allowing time $t \rightarrow \infty$.

$$\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} = \frac{1}{\alpha} \frac{dT}{dt}$$

(3.3)

As previously discussed, D. Rosenthal[26] developed quasi steady state solutions for a
moving heat source by observing the distribution of temperature around coordinates which coincided with the moving heat source. Theoretically, after a certain time has elapsed, the temperature at a given position relative to the moving coordinate is steady and therefore the condition is called „quasi steady state”. R. Komanduri and Z.B. Hou[28] developed a non-dimensional integral to carry out thermal analysis in welding. They used a Gaussian distributed moving disc heat source to represent the heat load embedded by the welding torch. A solution for the non-dimensional integration is obtained through the numerical approach.

3.3 Solutions using non-dimensional integral to compensate for transient conditions

Carslaw and Jaeger[27] have proposed a solution for an instantaneous point heat source in infinite solid that liberates heat at \((x_0,y_0,z_0)\) as expressed in (3.4). \(\phi\) is a function which depends on the heat load and material thermal properties.

\[
\Delta T = \frac{4\pi \alpha t^{3/2}}{4 \sigma_\pi x_0^3} \exp \left\{ -\frac{(x-x_0)^2 + (y-y_0)^2 + (z-z_0)^2}{4\alpha t} \right\} \tag{3.4}
\]

Komanduri and Hou[28] obtained solutions for the distribution of temperature around the Gaussian distributed disc heat source as it is expressed in (3.5).

\[
T - T_0 = \frac{3.1576\pi \sigma}{4k\pi x_0^2} \exp(-\xi^2 \theta) \int_{r=0}^{r_0} \exp\left(-3\left(\frac{r}{r_0}\right)^2\right) r dr \]

\[
\cdot \int_{\theta=0}^{\theta_0} I_0\left(\frac{r^2}{4\alpha t}\right) \sqrt{(\xi^2 + 2\alpha/V)^2 + \xi^2} \exp\left(-\left(\omega + \frac{u^2}{4\alpha t}\right)\right) \frac{d\omega}{\omega^{3/2}} \tag{3.5}
\]

where \(\omega = v^2\tau/(4\alpha), R_h^2 = \xi^2 + y^2 + z^2 + r^2, V = v/(2\alpha), u = R_hV, \nu = 2\omega/V\) and \(I_0(p)\) is a modified Bessel function first kind, order zero. Since \(\omega\) is a non-dimensional term, equation (3.5) is called a non-dimensional integral.

The solution for the moving point heat source is based on (3.4), but if the heat source is liberated at the origin, equation (3.4) will be simpler as expressed in (3.6).

\[
\Delta T = \frac{\phi}{8\pi \sigma x_0^3} \exp \left\{ -\frac{x^2 + y^2 + z^2}{4\alpha t} \right\} \tag{3.6}
\]

The total heat for an infinite solid liberated by the heat source can be expressed as in (3.7), where \(\rho\) represents density and \(c_p\) is the specific heat.
\[
\dot{q}dt = \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} \rho \Delta T \, dx \, dy \, dz
\]  

Equation (3.8) is obtained by substituting \( \Delta T \) in equation (3.6) to equation (3.7) and considering that for isotropic materials \( \alpha \) and \( \phi \) are not a function of \( x, y \) and \( z \). Before solving equation (3.8), the solution for \( \int_0^{\infty} e^{-x^2} \, dx \) is analysed first. Figure 3.3 describes the nature of \( e^{-x^2} \). \( \int_0^{\infty} e^{-x^2} \, dx \) is obtained by calculating the total area under the curve in Figure 3.3, which is plotted in Figure 3.4. The integral value rapidly converges to \( 0.886227 \approx \frac{1}{2} \sqrt{\pi} \). Equation (3.8) is expressing equation (3.7) in a different form.

\[
\dot{q}dt = \frac{\phi \pi}{8(\pi \alpha)^{3/2}} \int_{-\infty}^{\infty} \exp\left(-x^2/4\alpha t\right) \, dx \int_{-\infty}^{\infty} \exp\left(-y^2/4\alpha t\right) \, dy \int_{-\infty}^{\infty} \exp\left(-z/4\alpha t\right) \, dz
\]  

Figure 3.3 Plotted value for \( e^{-x^2} \)

Figure 3.4 Plotted value for \( \int_0^{\infty} e^{-x^2} \, dx \)
Considering $\int_0^{\infty} e^{-x^2} \, dx = \frac{1}{\sqrt{\pi}}$, an integral term in equation (3.8) can be simplify as follows:

$$\int_0^{\infty} e^{-x^2/4\alpha t} \, dx = 4\sqrt{\alpha t} \int_0^{\infty} e^{-\left(\frac{x}{2\sqrt{\alpha t}}\right)^2} \, \frac{d}{2\sqrt{\alpha t}}$$

$$= 2\sqrt{\pi \alpha t}$$

The same values are obtained for $\int_0^{\infty} e^{-y^2/4\alpha t} \, dy$ and $\int_0^{\infty} e^{-z^2/4\alpha t} \, dz$. The total heat stored in an infinite solid can then be simplified as in equation (3.9).

$$\int_0^{\infty} \int_0^{\infty} \rho c \Delta T \, dx \, dy \, dz = \phi \rho c$$  (3.9)

Considering the total stored heat is equal to the inputted heat $q = \frac{\partial q}{\partial t}$, the value of $\phi$ now can be determined as $\phi = \frac{q \partial t}{\rho c}$ and substituting this value to equation (3.6) yields equation (3.10).

$$\Delta T = \frac{q \partial t}{\theta \rho c \kappa \alpha t^{3/2}} \exp\left\{-\frac{x^2+y^2+z^2}{4\alpha t}\right\}$$  (3.10)

The rise in temperature resulting from a moving point heat source initially at origin (0,0,0), and moving parallel to the x axis at a constant speed (Figure 3.5), can be obtained based on equation (3.10). Equation (3.10) can be interpreted as the rise in temperature in an infinite solid due to instantaneous heat liberated when $t = 0$ at the origin. When $t \to 0$ the temperatures rise at all points are zero except at the origin, which is infinite. (Note: for small values of $t$ at origin – $x = y = z = 0$ – the value of $\exp\left\{-\frac{x^2+y^2+z^2}{4\alpha t}\right\}$ will equal to 1. When $t$ is small the values of $\frac{q \partial t}{\theta \rho c \kappa \alpha t^{3/2}}$ will be very big and when $t \to 0$ it will be infinite and then the combined results will be infinite. For different place (not at origin) and small $t$ the value of $\exp\left\{-\frac{x^2+y^2+z^2}{4\alpha t}\right\}$ will be very small and when $t \to 0$ it will be equal to zero. Anything times zero is equal to zero and thus the combined result is equal to zero. It is one of the shortcomings of point load, even in FEM it is suggested to transformed point load to surface or volumetric load).
Equation (3.11) is obtained by adjusting equation (3.10) to the moving coordinates which coincides with the moving heat source, and where the temperature rise in a small time increment is expressed as $dT$.

$$dT = \frac{\dot{q} dt}{8\rho c \pi a t^{3/2}} \exp \left\{ -\frac{(\xi+vt)^2 + y^2 + z^2}{4a t} \right\}$$  \hspace{1cm} (3.11)

The total temperature rise at any points can be obtained by integrating equation (3.11). Considering the initial temperature as $T_0$, the temperature rise is expressed as in (3.12).

$$T - T_0 = \frac{\dot{q}}{8\rho c \pi a t^{3/2}} \int_{\tau=0}^{\tau} \frac{\exp \left\{ -\frac{(\xi+vt)^2 + y^2 + z^2}{4a t} \right\}}{\rho^{3/2}} d\tau$$  \hspace{1cm} (3.12)

This integration can be expressed in a non-dimensional form by substituting the following expressions: $\omega = v^2 \tau/(4\alpha)$, $R_h^2 = \xi^2 + y^2 + z^2 + r^2$, $V = v/(2\alpha)$, $u = R_h V$, $v \tau = 2\omega/V$ and equation (3.13) is obtained.

$$T - T_0 = \frac{\dot{q} v \exp(-\omega V)}{16\rho c \pi a^2 \tau^{3/2}} \int_{\omega=0}^{\infty} \frac{\exp \left\{ -\frac{u^2}{4\omega} \right\}}{\omega^{3/2}} d\omega$$  \hspace{1cm} (3.13)

For a semi-infinite solid case as in welding, the temperature obtained by equation (3.13) should be doubled since the heated volume is half that of an infinite solid. (The semi infinite case applies for fusion welding since the heat source is always applied on a surface of large material. In case of a large thick, the plate may consider as an infinite solid. Of course this case study neglect convection + radiation and adiabatic surfaces are considered as a boundary condition as modeled in section 3.5). For convenience $\rho c$ is replaced by $\lambda/\alpha$ and the final temperatures for any points in the welding model can be represented by equation (3.14).
The values for the definite integral in equation (3.14) depend on the value of the upper limit, which is a time function. Clearly, equation (3.14) is the solution for a transient state and the results for a quasi steady state can be found by replacing the value of \( t \to \infty \), thus the upper integration limit is equal to \( \infty \). Since a singularity is found when \( \omega = 0 \), the solution cannot be solved analytically but numerically by setting the lower limit close to 0 (e.g. 0.001). The integral expression in (14) is plotted and presented graphically in Figure 3.6. From this figure it can be concluded that when \( u \geq 3 \) the values of the integral are numerically closer to zero which means there is no significant elevation in temperature. It can also be drawn from Figure 3.6 that at a higher \( u \) value, the definite integration value converges at a higher value of \( \omega \) (which also means a longer time). For lower \( u \) values, it was found that when \( \omega \geq 5 \) the value of the integral converged, thus it is reasonable if the upper limit of integration of 5 is adopted, which makes the numerical solution much easier to obtain.

Instead of using \( \omega \) when the steady state condition is achieved, as shown by the temperature profile in Figures 3.7 and 3.8, the graphics in Figure 3.7 for a longitudinal path and Figure 3.8 for a transverse path were composed using the analytical solution expressed in equation (3.14). The legends t10s, t50s, t100s, t200s, t1000s in the legend box mean temperature profiles when elapsed time equal to 10s, 50s, 100s, 200s, 1000s respectively that coincide with non-dimensional parameters \( \omega = 0.25, 1.25, 2.5, 5 \) and

\[
T = T_0 + \frac{q_v \exp(-\omega)}{8\lambda \alpha^{1/2}} \int_{\omega=0}^{\omega} \frac{\exp(-\omega - \frac{u^2}{4\omega})}{\omega^{1/2}} d\omega
\]  

(3.14)
250 respectively. A longitudinal path with $y = 0$ and $z = 10\text{mm}$ was chosen to describe the temperature profiles in Figure 3.7.

![Figure 3.7 Temperature profiles along a longitudinal path.](image)

From Figure 3.7 it can be seen that the temperature profile for 200s coincides with that for 10000s. A transverse path with $\xi = 0$ and $z = 10\text{mm}$ was chosen to describe the temperature profiles in Figure 3.8. The temperature profile at the transversal path, from Figure 3.8, show that the temperature profiles were already superimposed to each other when $t = 100\text{s}$. These facts confirm the previous assumption that at $\omega = 5$ the temperature profile at the moving point heat source has already converged, and it is quite reasonable to adopt 5 as the upper limit of integration of equation (3.14).
3.4 Element model (SOLID70)

Basically, the element model describes the relationship between the geometry of a model with its degree of freedom. The SOLID70 element is suitable for a 3D thermal analysis model. Typically, the element is comprised of eight nodes with the temperature as the degree of freedom at each node. However, if a node is formed as a coalescence of nodes, the element can consist of four, five, or six nodes and a tetrahedral element, a pyramid element, or a prismatic element are formed as shown in Figure 3.9.

![Figure 3.9. SOLID70 3D-thermal element (Release 12.0 Documentation for ANSYS)](image)

3.5 Finite element model for a moving point heat source

Finite element analysis was carried out using ANSYS software. There are two options which can be used: ANSYS Parametric Design Language (APDL) and the Graphic User Interface (GUI). In this thesis the APDL mode was chosen because it provides flexibility and a greater ease of modification.

The model consists of 23328 SOLID70 thermal elements with 26011 nodes to represent a 250mm x 180mm x 90mm block. The heat source moves along the x abscissa with a certain velocity. In APDL it is programmed by changing the position of point heat source along the line (x,0,0) with changes in the x value according to time (Figure 3.10).

Typical welding parameters were chosen as following: the heat rate \( \dot{q} = 4200 \, J/s \), thermal conductivity \( \lambda = 42 \, W/(m.C^o) \), thermal diffusivity \( \alpha = 10 \, mm^2/s \), and welding
speed \( v = 1\text{mm/s} \). In this chapter adiabatic boundary condition was applied on surfaces means no convection or radiation was involved.

The moving point heat source was modeled by a heat source at a certain point, and heat is liberated for a certain time, depending on the welding speed and distance between two consecutive nodes. This distance is determined by the mesh size of the FEM model, and then after a chosen duration, the heat source is omitted and relocated to the next position where it liberates heat for the chosen duration. This procedure is repeated to the end of the weld length. The temperature profiles were observed after the heat source travels 200mm length (200s). The flowchart of the FEM model is shown in Figure 3.11. Since \( \theta \) in this position is equal to 5, the temperature profiles can be considered as quasi steady state for comparison with the analytical solution.
3.6 Analytical model vs FE model – quasi steady state

The results from using a point heat source model are presented in Figure 3.12 for the longitudinal lines and in Figure 3.13 for the transverse lines. The temperature profiles are evaluated at $z = 0\text{mm}$ (at the surface of the plate), $z = 10\text{mm}$, $z = 20\text{mm}$ and $z =$...
30mm (below the surface). The temperatures were evaluated on these lines by both the analytical solution and FEM, and the results are shown graphically in Figure 3.12 and 3.13. The analytical solution referred to here, is the solution provided by the non-dimensional integral as it was represented in equation (3.14) while the temperatures were obtained by making the upper limit of the definite integral equal to 5. The solid lines describe solutions provided by the analytical solution, while the markers describe the solution provided by FEM. From Figures 3.12 and 3.13 it can be seen that FEM results are in good agreement with the analytic solutions, especially for those points which are far enough away from the point heat source.

Figure 3.12 represents the temperature profiles at \( y = 0 \) with varied \( z \) values (see Figure 3.2 for the axis nomenclature). The analytical solutions and FEM are in better agreement for temperature data behind the heat source model. Both the analytical solutions and FEM can describe the peak temperature which lags for higher values of \( z \) (the peak temperature is at negative \( \xi \) values), as it is found in practice, since a longer
time is needed to transfer heat from the heat source to higher \( z \) values (due to the longer distance). Meanwhile for a certain negative value of \( \xi \) the heat that is liberated from previous positions has reached a point on the observed line with the same negative \( \xi \) value. The transverse line shown in Figure 3.13 are lines where \( \xi = 0 \) with varied \( z \) values. Symmetrical graphs are shown in Figure 3.13. This symmetry has emerged because of equal \( \xi \) values. From equation (3.14) an unsymmetrical function is provided by the \( e^{-\xi V} \) term and it only has an effect when \( \xi \) is varied because the value of \( V \) \((V=\nu/(2\alpha))\) depends on the welding parameters: welding speed \( (\nu) \) and thermal diffusivity, \( \alpha \) which are considered constant.

The peak temperatures in Figure 3.13 should have the same values with temperatures at \( \xi = 0 \) in Figure 3.12 since they are in same location. It should however be noted that the peak temperatures in Figure 3.13 are not the peak temperatures in Figure 3.12 (except for \( z = 0 \)) as a result of the lagging effect discussed previously and peak temperatures in Figure 3.12 are in different \( \xi \) depend on distance to the point heat source.
3.7 Analytical models, Rosenthal solution and FE model – quasi steady state

The Rosenthal solution provides a temperature field for the quasi steady state. The quasi steady state for analytical models (equation 3.14) can be obtained by setting the upper limit $\omega$ of the definite integral greater than 5. The solutions obtained from those three models were compared to each other and are presented in Figure 3.14. The observed paths are the same as in Section 3.6: a path on the surface, and then 10mm, 20mm, and 30mm below the surface. From Figure 3.14 it can be seen that all those models give relatively good agreements. However Figure 3.14 for longitudinal paths shows some deviation, especially for the position furthest behind the centre of the weld torch (further than -0.12m). Figure 3.14 is enlarged in Figure 3.15 for $\xi$ further than -0.12m. From Figure 3.15 it can be said that the solution from the analytical model developed in this chapter (equation 3.14) gives an even closer solution to the FEM model than the solution obtained from Rosenthal’s equation at the far field behind the centre of weld torch for the quasi steady state.

Figure 3.14 Solutions from analytic, FEM and Rosenthal for temperature profiles at longitudinal paths (left) and transversal paths (right).
3.8 Analytical models vs FE model – transient conditions

The obvious advantage of a non-dimensional integral (equation 3.14) over the Rosenthal solution (equation 3.2) is the possibility of providing a solution for the transient state. The transient state had solutions before a steady state was established (before $\omega \geq 5$). The data used are the same as in Section 3.6. The temperature distribution at the surface of the plate at 10s, 50s, 100s and 200s is presented in Figure 3.16, whilst that through the cross sectional area is presented in Figure 3.17. The distribution of temperature is plotted in the isothermal lines of 1200°C, 600°C, 400°C, 150°C and 100°C. Considering the data used, these elapsed times are equivalent to $\omega = 0.25$, 1.25, 2.5 and 5 respectively. The data obtained using FEM (red dashed lines) were compared to the analytical results (solid blue lines).
Overall the finite element results show a good agreement with the non-dimensional integral method. In Figure 3.16, both methods demonstrate that higher elapsed times will cause higher asymmetric curves for both results. The asymmetric function of equation (3.14) \( \frac{q_v \exp(-\frac{\beta}{\alpha})}{8 \lambda \sigma a^{3/2}} \) can be expressed as \( \frac{q_v \exp(-\frac{\beta}{\omega})(2 \omega)}{8 \lambda \sigma a^{3/2}} \). This asymmetric expression is magnified by \( \int_{\omega=0}^{\frac{\beta}{\omega}^{1/2}} \exp(-\omega \frac{\beta}{\omega}) d\omega \) which depends on the value of the upper limits of the definite integral. By evaluating the transient condition of Figure 3.16 (before the quasi steady state is achieved), it can be concluded that a longer time \( t \) results in higher values of the definite integral, which in turn produces a more asymmetric curve.

The temperature profiles on the transverse section (\( y'z' \) plane, refer to Figure 3.2) for the transient state are shown in Figure 3.17. The results obtained from the finite element simulation and analytical solution show a symmetrical plot over the origin. From the perspective of an analytical solution, this result reflects the distribution of temperature which is expressed by equation 3.14. For \( \zeta = 0 \) the temperature because of heat liberated by the heat source is expressed as \( T = T_0 + \frac{q_v}{8 \lambda \sigma a^{3/2}} \int_{\omega=0}^{\frac{\beta}{\omega}^{1/2}} \exp(-\omega \frac{\beta}{\omega}) d\omega \) which can also be
expressed as \( \frac{qv}{0.4\pi\alpha^3/2} \int_{\omega=0}^{\omega/\alpha} e^{\left(-\frac{\omega^2 - y^2}{2\alpha^2}\right)} d\omega \). Only \( y \) and \( z \) at the last expression describe the position and function as symmetrical for \( y \) and \( z \), thus it is a symmetrical function. These symmetrical results are also confirmed by the FEM results that have shown good agreement with the analytical solutions.

The temperature profiles at time \( t = 100 \) or \( \omega = 2.5 \) are almost the same as those at time \( t = 200 \) or \( \omega = 5 \) which indicates that the temperature have already converged before \( \omega = 5 \). Thus the temperature profiles for the transverse direction converged earlier than those in a longitudinal direction. This phenomenon is also implicitly described in Figure 3.16 and 3.17.

![Figure 3.17 Isothermal profiles at plate cross section for transient state](image)

3.9 Conclusions

In this chapter a simple FEM model is compared with an analytic solution. The transient state and steady state of the temperature profile in welding can be described very well by the expression \( T = T_0 + \frac{qv}{0.4\pi\alpha^3/2} \int_{\omega=0}^{\omega/\alpha} e^{\left(-\frac{\omega^2 - y^2}{2\alpha^2}\right)} d\omega \). This new solution is developed based on the energy conservation law for a moving point heat source in infinite solids and follows non-dimensional integration technique of Komanduri [28].
improvement of this solution over the current Rosenthal equation is that the equation can describe the profile of the transient state temperature. This transient or steady state can be determined by the upper limit \( \omega \) of the integral term. When the limit \( \omega \) is close to 5 the quasi steady state is reached, and below this value the condition is considered as transient. For a steady state evaluation, the solution also shows a better agreement to the finite element simulation than Rosenthal’s for longitudinal paths, especially at a distance further away from the moving point heat source.

The good agreement between FEM and analytic solutions means the developed code according to the flow-chart in Figure 3.11 is able to represent a moving heat source phenomenon especially the point heat source. This code can be used as a basis for a more complex moving heat source in the future.
References


PART III

MODELING EXPLORATION
BASED ON EXISTING DATA
Welding applications subject materials to cycles of non-uniform temperature that in turn causes problems which may lead to premature fatigue damage, stress corrosion and fracture [1-4]. Studies of the distributions of transient temperature in weldments are important for quality control and they can be used as an aid to the development of welding procedures [5].

Rosenthal [6] proposed an analytical solution for a moving point heat source that can be applied to the welding process. Following this, the analytical solutions for moving heat sources attracted much attention [7-10]. More recently the high speed computational resources needed for numerical analysis have become more readily available and FEM has become a popular tool for weld modeling. A clear advantage of numerical analysis over the analytical method is its ability to obtain a solution for complex geometry and boundary conditions which may be difficult to achieve using analytical solutions.

A major issue in modeling is accuracy and validity [11-14]. To this end the European Network on Neutron Techniques Standardization for Structural Integrity (NeT) has published experimental data and procedures which can be accessed at https://odin.jrc.ec.europa.eu [15,16]. Experimental work was carried out using bead-on-plate welding where nine thermocouples were attached at different measured points.

Using the NeT’s experimental data, a FEM thermal analysis is carried out. At this stage, the logic and parameters just follow some previous published papers with some modification when necessary. Experience on how to compose a FEM model in ANSYS

---

3 Material in this chapter has been published in papers:
APDL code is believed to be valuable contribution to the development of the model of multi pass girth welds in ferritic steel pipe lines.

A new heat source model using a combination of Goldak’s double ellipsoid and uniform temperature was used. The uniform temperature represents the melting filler metal which forms the weld bead and the Goldak’s heat source model represents the heat that was transferred to the base metal.

4.1 Welding procedure provided by NeT

A complete welding procedure was obtained at https://odin.jrc.ec.europa.eu[16]. These welding procedures are summarised on Table 4.1. Four identical plate specimens (called A11, A12, A21, and A22) were produced. Following the NeT procedures, a sketch diagram for the welding set up and thermocouple positions are presented in Figures 4.1 and 4.2 respectively. The coordinate system here is following that used by NeT. It should be noted that the origin of the coordinate system is where the weld stops. The positions of the thermocouples based on the coordinate system chosen, are tabulated in Table 4.2.

<table>
<thead>
<tr>
<th>Table 4.1. Summary of welding procedure.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Parent material:</strong></td>
</tr>
<tr>
<td>Size: 120 x 180 x 17mm</td>
</tr>
<tr>
<td>AISI Type 316L Austenitic Stainless Steels</td>
</tr>
<tr>
<td>Residual stress is relieved using heat treatment</td>
</tr>
<tr>
<td><strong>Filler:</strong></td>
</tr>
<tr>
<td>Ø0.8mm</td>
</tr>
<tr>
<td>Type: 316S96</td>
</tr>
<tr>
<td>Spec: A5.9.93 (ASME); ER316H</td>
</tr>
<tr>
<td><strong>Welding parameters:</strong></td>
</tr>
<tr>
<td>Welding process: Tungsten Inert Gas (TIG)</td>
</tr>
<tr>
<td>Type: Bead-on-plate</td>
</tr>
<tr>
<td>Weld length: 60mm</td>
</tr>
<tr>
<td>Heat Input: 633 J/mm</td>
</tr>
<tr>
<td>Welding speed: 2.27 mm/s</td>
</tr>
</tbody>
</table>

Four sets of temperature histories for the four specimens were obtained from nine thermocouples. As thermocouple Tc8 was not pushed far enough into the hole and there is no clear reference to where the data was obtained, the Tc8 thermocouple was not considered. For thermocouple Tc9, the data from specimen A21 were also excluded.
since they showed a lower temperature and were inconsistent with data from other specimens \([17]\).

![Figure 4.1. Sketch of welding process and coordinate system.](image)

![Figure 4.2. Thermocouple position on weld on bead welded plate.](image)

<table>
<thead>
<tr>
<th>Thermocouple</th>
<th>Mean position (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>X</td>
</tr>
<tr>
<td>T1</td>
<td>8.50</td>
</tr>
<tr>
<td>T2</td>
<td>8.00</td>
</tr>
<tr>
<td>T3</td>
<td>7.50</td>
</tr>
<tr>
<td>T4</td>
<td>11.50</td>
</tr>
<tr>
<td>T5</td>
<td>11.50</td>
</tr>
<tr>
<td>T6</td>
<td>12.00</td>
</tr>
<tr>
<td>T7</td>
<td>0</td>
</tr>
<tr>
<td>T8</td>
<td>0</td>
</tr>
<tr>
<td>T9</td>
<td>0</td>
</tr>
</tbody>
</table>
The temperature dependent properties of the base metal and welding filler were obtained from NeT \[^{[16]}\]. The material properties for the base metal and weld metal are shown in Table 4.3. The density and Poisson’s ratio for both materials are 7966 kg/m\(^3\) and 0.294 respectively. In Figure 4.3 the thermal properties for both metals are shown graphically.

<table>
<thead>
<tr>
<th>Temperature</th>
<th>Specific Heat, (c_p) kJ/(kg°C)</th>
<th>Conductivity, (\lambda) W/(m°C)</th>
<th>Thermal expansion, (\alpha) (\times 10^6) (°C(^{-1}))</th>
<th>Young’s Modulus (Gpa)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Weld &amp; Parent</td>
<td>Weld &amp; Parent</td>
<td>Weld &amp; Parent</td>
<td>Parent &amp; Weld</td>
</tr>
<tr>
<td>20</td>
<td>0.488</td>
<td>0.492</td>
<td>14.12</td>
<td>14.56</td>
</tr>
<tr>
<td>100</td>
<td>0.502</td>
<td>0.502</td>
<td>15.26</td>
<td>15.39</td>
</tr>
<tr>
<td>200</td>
<td>0.520</td>
<td>0.514</td>
<td>16.69</td>
<td>16.21</td>
</tr>
<tr>
<td>300</td>
<td>0.537</td>
<td>0.526</td>
<td>18.11</td>
<td>16.86</td>
</tr>
<tr>
<td>400</td>
<td>0.555</td>
<td>0.538</td>
<td>19.54</td>
<td>17.37</td>
</tr>
<tr>
<td>500</td>
<td>0.572</td>
<td>0.550</td>
<td>20.96</td>
<td>17.78</td>
</tr>
<tr>
<td>600</td>
<td>0.589</td>
<td>0.562</td>
<td>22.38</td>
<td>18.12</td>
</tr>
<tr>
<td>700</td>
<td>0.589</td>
<td>0.575</td>
<td>23.81</td>
<td>18.43</td>
</tr>
<tr>
<td>800</td>
<td>0.589</td>
<td>0.587</td>
<td>25.23</td>
<td>18.72</td>
</tr>
<tr>
<td>900</td>
<td>0.589</td>
<td>0.599</td>
<td>26.66</td>
<td>18.99</td>
</tr>
<tr>
<td>1000</td>
<td>0.589</td>
<td>0.611</td>
<td>28.08</td>
<td>19.27</td>
</tr>
<tr>
<td>1100</td>
<td>0.589</td>
<td>0.623</td>
<td>29.50</td>
<td>19.53</td>
</tr>
<tr>
<td>1200</td>
<td>0.589</td>
<td>0.635</td>
<td>30.93</td>
<td>19.79</td>
</tr>
<tr>
<td>1300</td>
<td>0.589</td>
<td>0.647</td>
<td>32.35</td>
<td>20.02</td>
</tr>
<tr>
<td>1400</td>
<td>0.589</td>
<td>0.659</td>
<td>33.78</td>
<td>20.21</td>
</tr>
</tbody>
</table>

Figure 4.3. Temperature dependence of thermal properties \[^{[16]}\].
4.2 Previous work in modeling NeT welding procedures

There are two main techniques used in weld modeling, the standard technique and the birth-and-death technique. The standard technique models basically deal with the weld bead in situ, whilst the element birth-and-death technique considers the growing weld bead. Further details can be found in Sections 4.3.3 and 4.3.4.

There are at least eight participants involved in the NeT weld modeling exercise: The British Energy-Frazer-Nash Consultancy, UK \cite{18} modeled the welding process using ABAQUS. The moving heat source is modeled as Goldak’s ellipsoidal distributed heat source. Two different sets of parameters for the heat source were used: \( r_x = 0.7\text{mm}, r_y = 2.8\text{mm}, r_z = 2.8\text{mm} \) and \( r_x = 0.4\text{mm}, r_y = 1.6\text{mm}, r_z = 1.6\text{mm} \) with the heat efficiency set to 75% and 60% respectively. Different dwell times were used for each model at the starting point of the weld; with 1.5s dwell for the first model and 0.7s dwell for the second model. Convection was represented by \( h_{\text{conv}} = 4.2\text{W/m}^2\text{K} \) at 100°C and \( h_{\text{conv}} = 11.8\text{W/m}^2\text{K} \) at 1000°C. Radiation was modeled using an emissivity of 0.4. The element birth and death technique were used for both models.

Imperial College, UK also used the ABAQUS code for the welding simulation \cite{19}. The standard and the element birth and death techniques were used in conjunction with a moving Goldak’s ellipsoidal heat source model. For the standard technique the heat source parameters were: \( r_x = 2.8\text{mm}, r_y = 3.2\text{mm}, r_z = 2.9\text{mm} \) while for the element birth and death techniques \( r_x = 1.4\text{mm}, r_y = 3.4\text{mm}, r_z = 2.9\text{mm} \). The dwell at the starting point of the weld start point was set to 1s. The convective heat transfer (CHT) coefficient is 5.7 W/m²K and radiation emissivity was set to 0.75. The heat efficiency is 100%.

INSA-Framatone, France used the SYSWELD code with three different moving heat source models: Gaussian surface heat source, conical heat source and prismatic heat source \cite{20}. The heat efficiency was 75%, the CHT coefficient 15W/m²°C and the radiation emissivity 0.7.

The Joint Research Centre, Institute for Energy, Petten, Netherlands \cite{21} used the ANSYS code. The moving heat source was modeled with two different models: the melting temperature field and uniform heat generated field. The melting temperature
was loaded at a prescribed melted zone while the heat rate for the uniform heat generated field was applied at a prescribed melting zone. The heat efficiency was set at 70%. The CHT coefficient was set to 10 W/m$^2$C and no radiation was considered.

Serco, a Technical and Assurance Service, UK [18] used the SYSWELD code with Goldak’s double ellipsoidal moving heat source model $r_{x1} = 1.6$mm and $r_{x2} = 3.2$mm, $r_y = 1.6$mm, $r_z = 3.2$mm. (Serco made 2 models with 2 different heat source model; the first is Goldak’s heat source with $r_x = 1.6$mm, $r_y = 1.6$mm , $r_z = 3.2$mm and the second is Goldak’s heat source with $r_x = 3.2$mm, $r_y = 1.6$mm, $r_z = 3.2$mm). The heat efficiency was set to 80% while the overall convective coefficient was 13.57 W/m$^2$K at 100°C.

Three other participants also made contributions: The Institute for Nuclear Research – Romania, Korea Power Engineering Company and University of the West of England, UK [17]. The Institute for Nuclear Research used ANSYS code. Uniform heat was generated at nodes inside the melted zone. Convection was modeled with a convective coefficient of 10W/m$^2$K. The Korea Power Engineering Company used the ABAQUS code with the heat source modeled as a combination of heat flux and body heat load. The heat flux was set to 60% and the body heat load set to 40% of the total heat. The element birth and death technique was used. The CHT coefficient was 10 W/m$^2$K$^o$ and the heat efficiency 75%. The University of the West of England also used the ABAQUS code with a uniform moving heat source in the melted zone. The overall convective coefficient at 1000°C was 89.2 W/m$^2$K$^o$ while the CHT coefficient was 10 W/m$^2$K$^o$ and the radiation emissivity 0.4. The heat efficiency was 80%.

From the above, it can be seen that the assumed welding efficiency varied from 60% to 100%. The participants used various welding efficiencies to describe the heat absorbed by the plate. Three participants used Goldak’s heat source model, but the parameters ($r_x$, $r_y$ and $r_z$) differed, depending on the researcher’s assumptions. At the starting point of the weld, the participants used dwell times that varied, from 0 s (no dwell) to 1.5 s. The cooling mode due to convection and radiation were modeled using a combination of the following: CHT coefficient, radiation emissivity, or the overall CHT coefficient; however the values chosen varied widely. In summary it can be seen that the welding models gave different results depending on the basic assumptions made.
4.3 Comparison of FE model to NeT data

From Figure 3.1 it can be seen how FEM is related to the other methods (Analytical method and experimental method). Whilst verification compares the FEM results with the analytical solutions, validation compares the FEM results with experimental measurements.

Figure 4.4 Flowchart for moving heat source model

In this chapter secondary data are used to validate the FEM model, and some important theoretical considerations that were used in the FEM program are discussed. These considerations are non-linearity, birth&death technique and the heat source model.

Welding is typically modeled by a moving heat source over the solid. The flowchart in Figure 4.4 describes the general idea of how welding is simulated in FEM. First comes the modeling, where a geometric model and thermal properties of material, such as...
conductivity, density, and specific heat, are defined. Generally, a non-linear material model is involved since the thermal properties of the material depend on the temperature which is unknown. The last step in modeling is element formation (building mesh) based on the type of element chosen that describes the relationship between the geometry and degree of freedom of the element.

The second step is finding a solution which may also be called simulation. In this stage the initial condition, such as the initial temperature of the plate, and boundary conditions such as convection and radiation at the surfaces, as well as the load, are defined. For the welding model, the load is a moving heat source. There is a varied heat source model that can be used, and it is usually divided into three classes: the point heat source, the surface heat source and the volumetric heat source. For a transient analysis the solution is obtained at each time step based on the initial condition, boundary condition and load. For a non-linear material model, an iteration is needed to obtain the final solution.

The last step is interpreting the solution. Generally the default outputs for thermal analysis are the distribution of temperature and temperature history at the nodes.

The three main configurations of moving heat source used are: the point heat source, the surface heat source and the volumetric heat source. In this thesis, the volumetric heat source is based on Goldak’s ellipsoidal distributed heat source since it is the most general form of heat source models. Also from the point of view of FEM simulation, the volumetric heat source as a body load to model welding processes has at least two advantages over a surface load: a better accuracy and flexibility. The model in this thesis is based on a moving heat source over an infinite solid. Following the coordinate system set by NeT, welding process moves parallel to the z axis and if $\xi$ represents a moving abscissa parallel to the z axis, the heat load at a certain small increment volume can be expressed by equation (4.1).

$$\dot{q}'''_{(x,y,\xi)} = \dot{q}_0 e^{-Ax^2-By^2-C\xi^2} \quad (4.1)$$

A, B and C are constants. Considering the energy conservation and that welding is applied onto a plate, i.e. a semi-infinite solid, equation (4.2) is obtained.

$$2\dot{q} = 2n_{eff}VI = 8 \int_0^{\infty} \int_0^{\infty} \int_0^{\infty} \dot{q}_0 e^{-Ax^2-By^2-C\xi^2} \, dx \, dy \, d\xi \quad (4.2)$$
It is mathematically known that $\int e^{-t^2} dt = \text{erf} (t)$, and at the limits $\int_0^\infty e^{-t^2} dt = \frac{1}{2} \sqrt{\pi}$.

As a result, equation (4.2) can be written as in (4.3).

$$2q = \frac{q_o \sqrt{\pi}}{\sqrt{ABC}} \quad (4.3)$$

For the Gaussian distributed surface disc heat source, the heat varies according to distance from the centre which is expressed as $q'' = Ce^{-3(r_i/r_o)^2}$ [22]. $r_i$ is the distance from the centre point, and $r_o$ is the outside radius of the disc. For $r_i = 0$ the heat flux will have the maximum value of $q_{\text{max}}'' = C$ whilst for $r_i = r_o$ the minimum heat flux will be $q_{\text{min}}'' = Ce^{-3}$. The ratio between the minimum heat flux at the edge of the disc and maximum heat flux at the centre of the disc is about 5%. Similarly, the heat rate for a Goldak heat source model that is generated at the edge elements was considered to be equal to 5% of the heat rate applied at the centre element. For elements at $(r_x,0,0)$, $(0,r_y,0)$, and $(0,0,r_z)$, $q'''_{(x,y,z)} = 5\% q_0''$; and using equation (4.1) and (4.3), equation (4.4) can be obtained for the Goldak heat source model.

$$q'''_{(x,y,z)} = \frac{6\sqrt{3}q}{\pi \sqrt{\pi r_x r_y r_z}} e^{-\frac{3x^2}{r_x^2}} e^{-\frac{3y^2}{r_y^2}} e^{-\frac{3z^2}{r_z^2}} \quad (4.4)$$

The maximum value of equation (4.4) is $q_{\text{max}}''' = \frac{6\sqrt{3}q}{\pi \sqrt{\pi r_x r_y r_z}}$ at position (0,0,0). By considering this condition and substituting $r_e^2 = \frac{3x^2}{r_x^2} + \frac{3y^2}{r_y^2} + \frac{3z^2}{r_z^2}$, equation (4.4) can be simplified to (4.5).

$$q'''_{(x,y,z)} = q_{\text{max}}''' e^{-r_e^2} \quad (4.5)$$

To obtain a good model of the heat source, a very fine mesh should be used in the area where the heat source will pass through. Equation (4.5) can be used to judge how fine the meshes need to be, but they depend on the $r_x$, $r_y$ and $r_z$ values; the higher these values are the coarser the mesh needs to be. Brick elements of 0.3mm brick mesh have been used. The adequacy of the mesh size will be analysed after the $r_x$, $r_y$ and $r_z$ values have been obtained. Further details can be found in Section 4.3.3.
In this work the finite element model and simulation have been carried out using ANSYS Parametric Design Language (APDL) mode because it is more flexible than the Graphics User Interface (GUI) mode. A one-half model was used due to symmetry. The model consists of 43,639 nodes and 71,128 elements of SOLID70. Denser meshes close to the weld line are needed due to the high temperature gradient at this position. Very fine meshes in the weld area are needed to match the moving heat source closely. Dense meshes are also applied at positions surrounding a thermocouple to locate its position accurately. The final FEM mesh design is shown in Figure 4.5.

The general material properties obtained from https://odin.jrc.ec.europa.eu are given in Table 4.3 and the thermal properties are presented graphically in Figure 4.3. Since a non-linear material model is used, iteration is performed using the Newton-Raphson iteration method.

This study used two approaches; a standard approach and an element birth-and-death technique approach. In the standard approach the deposited weld bead is modeled in-situ with the welded plate, while in the birth and death techniques the weld deposit metal is developed in accordance with the position of the moving heat source model. The standard approach can rapidly provide temperature history data, and although the
element birth and death technique is closer to the real conditions, a longer computing time is needed.

### 4.3.1 Non-linear material model

Many physical situations present nonlinearities too large to be ignored. The sources of nonlinearities can be divided into three classes: a changing status (contact element), geometry and nonlinear material properties. The last is the one most frequently involved in the practical world.

In thermal analysis, the temperature is mostly a unit which should be obtained. For one dimensional conduction problems, the relationship between temperature and heat rate, material thermal properties and geometric condition can be expressed by Fourier law:

\[ q_x = -kA \frac{dT}{dx} \]

In a linear material model \( k \) is considered independent of \( T \), whilst in a non-linear material model \( k \) depends on \( T \). Since \( T \) is not known a-priori the problem becomes more complicated and an iterative method is normally used. Three important methods will be presented in this chapter: the direct method, the modified method, and the Newton-Raphson method.

To present nonlinearity in general terms, it can be expressed mathematically as in equation 4.6 \[^{23}\].

\[
y = (k - f(x))x
\]  

(4.6)

Where \( x \) is the unknown which should be obtained and \( k \) is a constant. For a certain value of \( y \) the degree of freedom \( x \), should be found.

Figure 4.6 represent the direct method graphically. For a certain value of \( y_A \) the value of \( x \) should be \( x_A \). In the direct method, the first \( f(x) \) is considered equal to zero. The tentative first \( x \) values, which is symbolised as \( x_1 \) can be found easily. Applying this \( x_1 \) value to \( f(x) \), the tentative second \( x \) value, \( x_2 \), can be obtained. This method is applied iteratively until the tentative \( x \) values are converged. Red lines describe each iteration slope. It can be seen from Figure 4.6, that the lines start from the same point and its values depend on previous \( x \) values.
The second is a modified method that alters equation (4.6) in the form of \( x = \frac{y + xf(x)}{k} \).

First, \( f(x) \) is considered as zero and an \( x_1 \) value can be obtained. By applying this \( x_1 \) value to the term \( xf(x) \), the tentative \( x_2 \) value can be found. This method is applied iteratively until the \( x \) values are converged. Figure 4.7 represents the direct method graphically. It can be seen from Figure 4.7 that the same slope is used at each iteration, and although values of \( x_1, x_2, .. \) obtained by the modified method may differ from those obtained by the direct method, when it has converged the value will be same at \( x_A \).

![Figure 4.6 Direct method](image)

![Figure 4.7 Modified method](image)

The Newton-Raphson iteration method is depicted in Figure 4.8. At an initiation point A, the values of \( y \) and \( x \) are \( x_A \) and \( y_A \) respectively. The \( x \) value should be obtained when \( y \) is \( y_B \).
When developing equation (4.6), the slope at any point (usually called the tangent stiffness) can be obtained using equation (4.7).

\[
\frac{dy}{dx} = k - \frac{d(xf(x))}{dx}
\]  

(4.7)

Using the initial point \((x_A, y_A)\), \(x_1 = x_A + \frac{y_B - y_A}{(dy/dx)_A}\). The coincidence \(y_1\) can be obtained by substituting \(x_1\) to the equation (4.7). Now \((x_1, y_1)\) as an initial point and \(x_2\) can be found iteratively as \(x_2 = x_1 + \frac{y_B - y_1}{(dy/dx)_1}\). The procedure can be repeated until \(x\) is converged or until \(y\) is converged to \(y_B\).

Figure 4.8 Newton – Raphson iteration method

### 4.3.2 Input parameters

Basically the inputted parameters in this thesis follow previous published papers. It should be noted that the goal of this chapter is to develop an ANSYS code that can provide good results. The obtained experiences proved valuable when composing the more complex girth-weld model.

The first parameter which should be determined is welding efficiency. It was recommended by NeT to use 75% efficiency which fit with normal welding efficiency of TIG ranges between 65% and 88\%\(^{[24]}\).

Other parameters \((r_x, r_y, r_z)\) were determined by fitting mid length thermocouples with a peak temperature (Tc2, Tc5 and Tc7). It should be noted that, the APDL code can accommodate adjustable \(r_x\), \(r_y\) and \(r_z\). Mid-length thermocouples are insensitive to arc
dwell and initiation mode at the starting point of the weld, and are also insensitive to the arc extinction model at the end point of the weld.

The next step is modeling the arc dwell and initiation at the starting point of the weld. The arc dwell time just follows previous published paper while considering NeT recommendations and 1 second dwell time was used. Two choices of arc initiation mode can be used, either a stepped load mode or a ramped load mode and ramped load mode was chosen. The arc extinction was also modeled by a ramp load which is removed after 0.001s. The 0.001s extinction time is practically a step load and the ramp load mode is easier to be converge.

The parameters for convection \( (h) \) and radiation \( (\varepsilon) \) were determined from the mean values of previous published paper. Since convection heat transfer coefficient from previous papers varied from 4.2 – 15 (W/m\(^2\).\(^\circ\)C), a value of CHT 10 (W/m\(^2\). \(^\circ\)C) was adopted while emissivity was assumed to be 0.4 considering variation of 0 (no radiation) – 0.75.

### 4.3.3 Standard technique

In the standard technique the weld bead is modeled before the moving heat source load is simulated. The moving heat source is modeled by applying the Goldak’s ellipsoidal model at a certain point and heat is liberated for a given time duration depending on the welding speed and distance between one point and the next consecutive point. This distance is determined by the mesh size of the FEM model. Since the mesh size on the heat source path is 0.3 mm and the welding speed is 2.27 mm/s, the time duration is 0.13s, after which the heat source is removed and relocated to the next position, which then liberates the heat for the same time duration. This procedure is repeated up to the end of the weld length. The welding is simulated using the standard technique with the flow-chart shown in Figure 4.4, with a detail block for simulating the moving heat source shown in Figure 4.9.

Following the NeT procedures, it should be noted that the data were obtained from four tests (A11, A12, A21 and A22) and each test comprised nine initial temperature data (Tc1 – Tc9 Figure 4.2). The initial temperatures varied from 21°C to 29°C. From the NeT temperature histories, the initial temperature assigned to the base metal was 25°C.
Figure 4.9 Standard technique

The parameters $r_x = 3\text{mm}$, $r_y = 5\text{mm}$, $r_z = 2\text{mm}$ to describe heat source $q''_{(x,y,z)}$ (equation 4.4) were obtained after evaluating the peak temperatures at the mid-length position. Considering the arc dwell in some published paper varied from 0.7s to 1.5s $^{[17]}$, the dwell time at the start of the weld is taken as 1s. The mid-length macrograph was also helpful in giving tentative values of the heat source parameters especially $r_x$ and $r_y$ whilst the bi-lobed transverse boundaries described tentative values of $r_z$.

The brick mesh sizes along the heat source trajectory is 0.3mm. The values of the body heat source depend on the distance from the centroid of the elements to the centre of the
heat source. Since the brick mesh sizes is 0.3mm, the closest $r_e^2$ is 0.03645mm$^2$ which when applied to equation (4.5) results in $\dot{q}_{(x,y)} = 0.96\dot{q}_{\text{max}}$. The maximum body heat will be represented by 96.42% of its value, and it can be assumed that the mesh size along the weld path is sufficiently fine. (It would be good to study what mesh size is optimum for convergence but this is likely to involve a large number of models with varying mesh size and needs careful and time consuming study, however the most important factor in the welding is probably how the heat source is modeled. The values of the volumetric heat rate is really dependant on the mesh size since it determine the $r_e^2$).

4.3.4 Birth and Death technique

The birth and death technique is very useful when modeling welding phenomenon, and ANSYS uses the EKILL and EALIVE commands. When an element is killed it will be deactivated although it remains in the model. Deactivated elements have a conductivity close to zero (for mechanical analysis, stiffness is set close to zero). Stress, plastic strain, creep strain, and other dependent variables are set to zero. The next discussion will explore the birth and death technique in thermal analysis, from a very simple model.

How a structure with deactivated elements behaves in thermal analysis is simply shown at Figure 4.10. Figure 4.10 shows temperature distribution over a plate. In Figure 4.10a, a solid square is modeled with its left side maintained at 200\(^\circ\)C and its right side at 0\(^\circ\)C. Steady state analysis was carried out and the distribution of temperature is shown. In Figure 4.10b a solid square with a circular hole in the middle is modeled. The left side of the square was loaded with a temperature of 200\(^\circ\)C and its right side was maintained at 0\(^\circ\)C. The temperature distribution was plotted after a steady state condition had been achieved.

In Figure 4.10c a solid square is modeled. Elements inside a circle are deactivated. The circle is at the centre of the square with a diameter that is exactly the same as the hole in Figure 4.10b model. It can be seen by observing the active element, that the distribution of temperature shown is exactly the same as the model of a square with a hole (Figure
4.10b). If the birth and death technique is applied in single pass weld it behaves like there is isolation for the weld bead ahead of torch.

In Figure 4.11, instead of temperature distributions, the thermal flux is plotted for a solid rectangular model, a solid rectangle with a hole in the centre, and a solid rectangle with deactivated elements inside the circle. By observing the active elements of the rectangle with deactivated elements inside the hole, the conclusion is that the thermal flux of the active elements shows exactly the same vectors as the rectangle with a hole.

Overall it can be concluded that, when the elements have been deactivated, it looks like they have been omitted, even though they still remain in the model. This condition can be used to model a growing weld bead, a real geometrical model for which in FEM is not easy.
In an FEM model of bead on plate welding using the element birth and death technique, the metal bead grows with the moving heat source. How the welding is simulated using the birth-and-death technique follows the flow-chart shown in Figure 4.4 with a detail block for simulating the moving heat source shown in Figure 4.12. First, all of the weld bead elements were “omitted” using the EKILL command in ANSYS and then the growth was modeled using the EALIVE command. The position of weld bead elements was retrieved using *GET command and compared to the position of instantaneous heat source centre. Born elements are elements of weld bead which lay behind the moving heat source.
When the elements are born, their temperature should be at the superheated temperature $2400^\circ C$ of the melting embedded filler metal. Instead of applying a temperature node load, the body heat load rate ($\dot{q}_{body}$) at the elements was applied. The body heat load value is such that it can produce a temperature of $2400^\circ C$ at the growing weld bead. The heat needed to elevate the weld pool from the initial temperature of $25^\circ C$ to $2400^\circ C$ was
evaluated using \( q = mc\Delta T \). It should be noted that the specific heat \( c_p \), is temperature-dependent as expressed in Table 4.3 and in Figure 4.3. The specific heat at a certain temperature range was taken as the mean value between these ranges. The heat rate can be obtain using \( \dot{q} = mc\Delta T \left( \frac{x}{T} \right) \) expression. Finally, the body heat load which should be applied at the born weld bead can be obtained using equation (4.8).

\[
\dot{q}'''' = \rho c \Delta T \left( \frac{x}{T} \right) \quad (4.8)
\]

4.3.5 Results and discussion

The temperature histories for the standard technique are shown in Figure 4.13 to 4.17. From these figures it can be claimed that this simulation has shown a good agreement with the experimental results, but apart from showing the general trend, the transient temperatures also matched the experimental data. For the close field thermocouples, especially at mid-length and the weld end (Tc2 and Tc3), the FEM model predicted a lower temperature than the measured peak temperature. The lower prediction may be caused by radiation from the heat source which was not modeled. The „below bead” thermocouple (Tc7), showed a higher value in the model than the measured value. The „below bead” thermocouple was inserted 6.5mm deep in a hole with a diameter of 1.2mm. The possibility that the thermocouple was not fully located at the tip of the hole is high. Incomplete insertion could result in a larger distance from the heat source and a lower peak temperature.

The predicted temperature histories at the thermocouple positions resulting from the application of both techniques are shown in Figure 4.18 to 4.21. It can be seen that the temperature profiles using both techniques do not significantly differ. The conclusion is that welding thermal solution of structures like the bead on plate, with a minimal weld bead, are not sensitive to either the presence of the bead ahead of the torch (standard technique), or any forward conduction suppression made to account for its absence. Certainly this is not the case for the multi-pass girth weld joint where the weld beads are too large to be ignored, and the using of birth and death technique become compulsory.
Figure 4.13. Temperature histories using standard technique for location of thermocouple Tc1 & Tc4 (refer to figure 4.2).

Figure 4.14. Temperature histories using the standard technique for the location of thermocouples Tc2 & Tc5 (refer to figure 4.2).
**Figure 4.15.** Temperature histories using standard technique for the location of thermocouples Tc3 & Tc6 (refer to figure 4.2).

**Figure 4.16.** Temperature history using standard technique for the location of thermocouple Tc7 (refer to figure 4.2).
Figure 4.17. Temperature history using standard technique for the location of thermocouple Tc9 (refer to figure 4.2).

Figure 4.18. Temperature histories using standard and birth & death techniques for the location of thermocouples Tc1 & Tc4 (refer to Figure 4.2).
**Figure 4.19.** Temperature histories using standard and birth & death techniques for the location of thermocouples Tc2 & Tc5 (refer to Figure 4.2).

**Figure 4.20.** Temperature histories using standard and birth & death techniques for the location of thermocouples Tc3 & Tc6 (refer to Figure 4.2).
Figure 4.21. Temperature histories using standard and birth & death techniques for the location of thermocouples Tc7 & Tc9 (refer to Figure 4.2).

Figure 4.22 (a) actual weld pool\textsuperscript{[16]} and (b) temperature contour from standard technique
Figure 4.23 (a) actual weld pool and (b) temperature contour from birth and death technique

Figure 4.24. Transverse cross section of weld pool at weld-start and weld-end points

The shape of the weld pool at the mid-length cross section was compared with actual shape of the weld pool in Figure 4.22 and 4.23, for both standard and birth&death techniques. The actual cross section of the weld pool was obtained by cutting the welded plate[17]. The shape of the weld pool predicted by the FEM model is the region where the temperature was above 1400°C (melting point of AISI 316 stainless steel). The weld pool here can be modeled quite well using both techniques, which follow the procedure shown in Figures 4.9 and 4.12.
Figure 4.24 shows the transverse cross section of weld start and weld end and the actual cross section obtained from NeT\cite{16, 17}. The differences are caused by the mass transfer mode that was not modeled in the developed FEM model. This mass transfer mode is not sensitive at the mid-length cross section that makes the actual weld-pool fit with the FEM model.

In Figure 4.25 the sections of weld pool predicted by both techniques was compared. Although the distribution of temperature, especially for high temperatures above 1900°C shows a different distribution for both techniques, the shape of the weld pool is the same.

The heat source in this model was differentiated into two types, uniform heat load in the weld bulge and the distributed heat load following Goldak’s double ellipsoid function with the sketch shown in Figure 4.26. Due to the high concentrated heat load of Goldak’s along the weld line the isothermal temperature “develops” from the weld line which is seen as a dot in the cross section area. Since in the standard technique there is a cold weld bead in front of the heat source part of the heat should be transferred to the cold weld bead. On the other hand, in the birth and death technique this cold weld bead is deactivated. Based on this description it is possibly why in standard technique the high temperature (1900°C) occupied a narrower area and was deeper to the centre as a result of a lower heat since a part of the heat was transferred to the cold weld bead which was not deactivated.
4.4 Conclusion

A comprehensive model of the transient thermal behaviour of a weld was established and validated using the experimental results from the NeT consortium.

The shape of the weld pool predicted by the standard technique and birth-and-death technique show well matched results, and an evaluation of the predicted shape of the weld pool reinforces the validity of the welding model.

The validated APDL code and obtained experiences while solving the thermal analysis in this chapter are a good start for the next more complicated analysis.
References


Chapter 5

Mechanical Analysis of NeT TG1 Specimen

The temperature in the welding process is not distributed uniformly and causes a non-uniform thermal strain in a welded structure, which in turn, causes thermal stress. This thermal stress continues to grow until the welded structure reaches a uniform temperature. Any stress remaining while the temperature of the welded structure is already uniform is called residual stress and is known as a major cause of failure in welded structures especially in pipelines \([1-3]\).

![Figure 5.1 Pattern of longitudinal distribution of residual stress.](image)

Longitudinal residual stress close to the weld line (Figure 5.1) can be roughly predicted using Masubuchi’s equation (equation 5.1) \([4]\). This equation generally is only valid for \(x/b \leq 0.6\). \(\sigma_z(x)\) describes the distribution of longitudinal residual stress which depends on the transverse distance from the centre of the weld \((x)\); \(\sigma_{\text{max}}\) is the maximum stress.

---

\[4\] Material in this chapter has been published in a paper: Djarot B. Darmadi, Muruganant Marimuthu, Anh Kiet Tieu and John Norrish, *Numerical prediction of residual stress in 316L bead-on-plate welds*, 15\textsuperscript{th} International Conference on Advances in Material & Processing Technology (AMPT), Wollongong – Australia, 23-26 September 2012, paper id: 11355.
tensile residual stress, and $b$ is the width of the region of tensile stress. In many cases the maximum tensile residual stress can be represented by the $\sigma_{\text{yield}}$ of the base metal whilst the value of $b$ remains unknown, based on the thermal analysis and welding parameters. Sindo Kou modified Masubuchi’s equation using an exponential function, as shown in equation 5.2 \[5\].

\[
\sigma_z(x) = \sigma_{\text{max}} \left\{ 1 - \left( \frac{x}{b} \right)^2 \right\} 
\] \hspace{1cm} (5.1)

\[
\sigma_z(x) = \sigma_{\text{max}} \left\{ 1 - \left( \frac{x}{b} \right)^2 \right\} \exp \left[ -0.5 \left( \frac{x}{b} \right)^2 \right] 
\] \hspace{1cm} (5.2)

Actually, the development of residual stress in the welding phenomenon is very complex because the pattern of residual stress depends on the type of welding process (heat input model), and the geometry of the welded joint and the base metal \[6\]. As there is no single mathematical equation available to predict the residual stress pattern, a numerical prediction such as FEM is more suited to deal with this complexity.

The basic mechanism of the development of residual stress was described by Wells \[7\] and is presented in Chapter 2. To understand the development of residual stress, Sindo Kou \[5\] used the three bars model. The analytical solution regarding the elasto-plastic material model compared to the FEM provides a clearer idea on the development of residual stress \[8\]. First, heating and cooling during welding always causes elongation/shrinkage in the base metal and the weld metal, but this elongation/shrinkage depends on the elevation of temperature and coefficient of thermal expansion, and this elongation/shrinkage causes distortion and stresses. Some FEM model papers focused on this distortion \[9 - 13\]. If the distortion is constrained it will cause residual stress. These constraints can be the rigid surrounding materials, or the cool region away from the weld line. Some FEM model papers focused only on this residual stress \[14 - 17\] and others on both phenomenon (distortion and residual stress) \[18 - 23\], since they co-exist in a welded structure.

This chapter discusses the FEM model of the development of residual stress in a bead-on-plate welding of 316L Austenitic Stainless Steel. The thermal analysis of the model has been presented in Chapter 4. The welding procedure follows the experimental data provided by the NeT consortium \[24\] and the result (residual stress) is validated using the
NeT consortium member data which is published in international journals\(^{[25-29]}\). Details of the welding procedure have also been described in chapter 4.

## 5.1. Thermo mechanical analysis

Thermo-mechanical analysis involves thermal analysis and mechanical analysis. From thermal analysis the distribution of temperature at certain time can be predicted, whereas in mechanical analysis, the distribution of temperature from thermal analysis is applied as a thermal load. This thermal load in turn causes the elements to expand and since the temperature is not distributed uniformly, a non-uniform elongation exerted to the welded structures will cause thermal stresses. When all of the temperature in a welded structure is uniform (achieved after certain time duration), no further stress is added and the remaining stress is called residual stress.

In a thermo-mechanical analysis a mechanical analysis is applied sequentially after thermal analysis has been completed. This uncoupled model is reasonable since the distribution of temperature does affect the development of residual stress, even though the residual stress has an insignificant effect on the distribution of temperature in a welded structure as it has been discussed and shown in Figure 2.1.

This chapter discusses the residual stress exerted on the bead-on-plate welding discussed in Chapter 4 where the thermal analysis was carried out.

## 5.2. Element model (SOLID45)

The element model used in the thermo-mechanical analysis is SOLID45. These elements accommodate structural analysis. They consist of eight nodes and three degrees of freedoms (dof) at each node. The degrees of freedom are translation in three perpendicular directions: \(x\), \(y\) and \(z\) respectively. The nodes at the element can coincide with each other and the element can be in a prism or tetrahedral forms, as shown in Figure 5.2. Since SOLID70 and SOLID45 have basically the same geometry, they are often used as a pair in a thermo-mechanical model, i.e. SOLID70 is used in thermal analysis while SOLID45 is used in mechanical analysis. It should be noted that the
SOLID45 element cannot act in the pyramid form like the SOLID70 element, and since the elements that compose the geometric model in thermal analysis should be exactly the same as in mechanical analysis, the pyramid form of SOLID70 should be avoided [8,17].

Figure 5.2. SOLID45 3D-structural element (Release 12.0 Documentation for ANSYS)

5.3. Mechanical properties

The mechanical properties depend on temperature and the mechanical properties experienced in the welding process vary significantly with the wide range of temperatures, from ambient to the material melting point. Stress-strain diagrams of the weld metal and base metal material model for various temperatures are shown in Figure 5.3a and 5.3b respectively. In FEM modeling, a multi-linear kinematic hardening material model was used to accommodate those temperature dependent stress-strain relationships. The multi-linear model consists of linear lines which represent a non-linear plasticity. In APDL mode for a database input of certain properties at certain variables and between consecutive defined variables the defined property is considered linear. In case of Figure 5.3a, as an example to describe non linear stress-strain curve at 20°C the “properties” input were $\sigma$, i.e. 210MPa, 238MPa, 292MPa, … for the variables $\varepsilon$, i.e. 0.107%, 0.3%, 1.1%, 2.1% ….. And between those defined points ANSYS applies linear interpolation. If a nonlinear model is defined by enough points in a linear model it can be considered the multi-linear model represent the non-linear model. The multi-linear kinematic hardening model in ANSYS is used to represent non-linear kinematic hardening in the real model.
The other mechanical properties have been discussed in Chapter 4. Since in the ANSYS APDL mode the material properties should be defined in a consistent range of temperatures, the material properties for the defined temperatures are obtained using linear interpolation. Those material properties, after being interpolated, are presented in Table 5.1a and 5.1b, for base metal and weld metal respectively.

![Figure 5.3a. Temperature dependent stress-strain diagram for base metal][9]

![Figure 5.3b. Temperature dependence stress-strain diagram for weld metal][9]
It should be noted that, the thermal expansions ($\alpha$) provided by NeT in Tables 5.1 are means between zero and the relevant temperature. The instantaneous thermal expansion which was input into ANSYS APDL is shown in Table 5.2.

**Table 5.1a. Base metal properties** [9].

<table>
<thead>
<tr>
<th>$T$ (°C)</th>
<th>$\lambda$ (W/m°C)</th>
<th>$c_p$ (J/kg°C)</th>
<th>$\alpha$ ($x 10^{-6}$/°C)</th>
<th>$E$ (GPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>14.12</td>
<td>492</td>
<td>14.56</td>
<td>195.60</td>
</tr>
<tr>
<td>275</td>
<td>17.76</td>
<td>523</td>
<td>16.70</td>
<td>181.13</td>
</tr>
<tr>
<td>550</td>
<td>21.67</td>
<td>556</td>
<td>17.95</td>
<td>159.75</td>
</tr>
<tr>
<td>750</td>
<td>24.52</td>
<td>581</td>
<td>18.58</td>
<td>137.75</td>
</tr>
<tr>
<td>800</td>
<td>25.23</td>
<td>587</td>
<td>18.72</td>
<td>131.40</td>
</tr>
<tr>
<td>900</td>
<td>26.66</td>
<td>599</td>
<td>18.99</td>
<td>116.80</td>
</tr>
<tr>
<td>1100</td>
<td>29.50</td>
<td>623</td>
<td>19.53</td>
<td>80</td>
</tr>
<tr>
<td>1400</td>
<td>33.78</td>
<td>659</td>
<td>20.21</td>
<td>2</td>
</tr>
</tbody>
</table>

**Table 5.1b. Weld metal properties** [9].

<table>
<thead>
<tr>
<th>$T$ (°C)</th>
<th>$\lambda$ (W/m°C)</th>
<th>$c_p$ (J/kg°C)</th>
<th>$\alpha$ ($x 10^{-6}$/°C)</th>
<th>$E$ (GPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>14.12</td>
<td>488</td>
<td>14.56</td>
<td>171</td>
</tr>
<tr>
<td>275</td>
<td>17.76</td>
<td>532.75</td>
<td>16.70</td>
<td>151.88</td>
</tr>
<tr>
<td>525</td>
<td>21.32</td>
<td>576.25</td>
<td>17.87</td>
<td>133.13</td>
</tr>
<tr>
<td>700</td>
<td>23.81</td>
<td>589</td>
<td>18.43</td>
<td>120</td>
</tr>
<tr>
<td>850</td>
<td>25.95</td>
<td>589</td>
<td>18.86</td>
<td>102.95</td>
</tr>
<tr>
<td>1000</td>
<td>28.08</td>
<td>589</td>
<td>19.27</td>
<td>83</td>
</tr>
<tr>
<td>1400</td>
<td>33.78</td>
<td>589</td>
<td>20.21</td>
<td>1.70</td>
</tr>
</tbody>
</table>

**Table 5.2. Instantaneous thermal expansion $\alpha$ ($x 10^{-6}$/°C)**

<table>
<thead>
<tr>
<th>$T$ (°C)</th>
<th>Base Metal</th>
<th>$T$ (°C)</th>
<th>Weld Metal</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>14.56</td>
<td>20</td>
<td>14.56</td>
</tr>
<tr>
<td>275</td>
<td>19.18</td>
<td>275</td>
<td>19.18</td>
</tr>
<tr>
<td>550</td>
<td>21.53</td>
<td>525</td>
<td>21.45</td>
</tr>
<tr>
<td>750</td>
<td>21.43</td>
<td>700</td>
<td>21.06</td>
</tr>
<tr>
<td>800</td>
<td>21.33</td>
<td>850</td>
<td>21.62</td>
</tr>
<tr>
<td>900</td>
<td>21.48</td>
<td>1000</td>
<td>22.32</td>
</tr>
<tr>
<td>1100</td>
<td>22.77</td>
<td>1400</td>
<td>23.53</td>
</tr>
<tr>
<td>1400</td>
<td>23.45</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

It is reported that varied hardening models (elasto-plastic, kinematic and isotropic) were used as material model by some investigators of weld simulation [27]. Regarding the stress-strain diagram in Figures 5.3 the elasto-plastic mode was dropped as the hardening model in this chapter. In this Chapter kinematic hardening is simulated first. The results using isotropic hardening are then compared with the results of the kinematic hardening model and discussed.
Others properties such as density and Poisson’s ratio for base metal and weld metal are not considered to be affected by temperature. The values for density and Poisson’s ratio are 7966 kg/m³ and 0.294 respectively.

5.4. Exploring the birth and death technique

How the birth and death technique behaves in thermal analysis was discussed in Section 4.3.4. The element model in thermal analysis (SOLID70) considers temperature as a degree of freedom whilst displacement is considered in mechanical analysis using SOLID45 element model. In mechanical analysis, when the element model is dead (deactivated) the EKILL command has been applied, the stiffness of the element is set close to zero. How the birth and death technique behaves in mechanical analysis is shown simply in Figure 5.4 that describes displacement distribution.

Figure 5.4. Displacement distribution for (a) solid rectangle, (b) rectangle with a hole and (c) rectangle with deactivated elements inside the same hole as (b)
In Figure 5.4a, a solid rectangle is constrained in all directions on the left surface. On the right surface the rectangle is loaded by a small displacement to the positive x-direction, but constrained in the other two directions (y and z). The x-displacement distribution of steady state in the rectangle is then plotted. In Figure 5.4b the rectangle has a hole at the centre and the same displacement load and constraint as in Figure 5.4a are applied in Figure 5.4b. The x-displacement distribution of steady state in the rectangle with a hole is then plotted. In Figure 5.4c the solid rectangle has dead (deactivated) elements inside a hole which has a position and size equal to the hole in Figure 5.4b. The same displacement load and constraint are applied and the x-displacement distribution of steady state in the rectangle with the dead elements is plotted. It can be seen that the x-displacement distribution in the active elements of Figure 5.4c has exactly the same pattern as in Figure 5.4b.

![Figure 5.5](image)

Figure 5.5. Vector plot of displacement for (a) solid rectangle, (b) rectangle with a hole and (c) rectangle with deactivated elements inside the same hole as (b)

In Figure 5.5, instead of the distributions of displacement, the displacement vectors are plotted for a solid rectangular model as a solid rectangle with a hole in its centre, and a
solid rectangle with deactivated elements inside the circle. By observing the active
elements on the rectangle with deactivated elements inside the hole, a conclusion was
reached that the displacement vector of active elements showed the same vectors as the
rectangle with a hole.

Overall it can be concluded that deactivated elements appear to have been omitted,
although they remain in the model. In mechanical analysis, this condition can be used to
model a growing weld bead because a real growing geometric model is not easily
created in ANSYS-APDL. First, the base metal and weld bead were modeled, then all
the weld bead elements were deactivated, and then parts of the weld bead elements were
activated after the relative position of those elements to the heat source centre had been
evaluated.

5.5. FEM for the thermo-mechanical model

The flow-chart for the comprehensive procedure to obtain the residual stress in the
thermo-mechanical analysis is shown in Figure 5.6. The following will discuss each step
in more details.

Considering the equality of element geometry and the degree of freedom as discussed in
Section 5.2, the SOLID45 element was used in the mechanical analysis. Since the nodal
temperature in thermal analysis will be used as a thermal load in the mechanical
analysis, the meshes in the mechanical analysis should be exactly same as thermal
analysis to accommodate temperature load mapping. This can be achieved by resuming
the geometric model in the thermal analysis and replacing the element (SOLID70) with
the structural elements model (SOLID45).

At this stage, a proper geometric model has been built, so the next steps are to define the
mechanical properties of the element and define the boundary condition and initial
condition of the welded structure. The boundary condition is determined by how the
welded element is constrained in real conditions. The initial condition is the initial
temperature that is considered when no thermal stress has been exerted. Since the
geometry of the welding process was symmetric, to save computing time only a half
model was built. For the simulated bead-on-plate welding which follows the NeT welding procedure (described in Chapter 4), the boundary conditions were defined for the clamping vice and plane of symmetry. The vice was considered as a light clamping, thus no constraint was applied, while the symmetry plane was modeled as an area with no displacement in a normal direction (parallel to the x axis).

Figure 5.6. FEM procedures to predict the distribution of residual stress.
The temperature history of each node in thermal analysis is saved in the *.rth file. For a certain time the temperature at each load can be retrieved using the LDREAD command and applied as a thermal load. Applying this temperature as a thermal load produces strain and thermal stress that can be calculated by considering the stress-strain relationship. This application of thermal load is repeated until no more significant thermal stress has developed. The remaining thermal stress is considered to be residual stress.

5.5.1. Accommodating a growing weld bead

As in thermal analysis, a growing weld bead is accommodated using a combination of the EKILL command and EALIVE command. This technique can also be applied in mechanical analysis. First, all the weld bead elements are deactivated using the EKILL command and then the position of the weld bead elements is checked relative to the position of the welding torch. When the elements lie behind the welding torch, the EALIVE command is applied and the element activated. The temperature distributions for a certain time from the thermal analysis are applied to this geometric model with the growing weld bead.

5.5.2. Accommodating the melting phenomenon

While a material is melting, its stiffness and plastic strain are significantly decreased. Shan et.al [25] put zero values on the stiffness and plastic strain. To identify which elements are melting, the temperature of the elements should be checked in the thermo-mechanical analysis.

Temperature data in the thermal analysis are saved in the file *.rth. It is time consuming to retrieve those files to be used as parameters, so a data-base which can provide parameters that give the temperature of each element at a certain time has been constructed. It was constructed during the thermal analysis stage, and it should be fast and easy to open in the thermo-mechanical analysis stage. In this FEM procedure, a *.txt database file was made. Based on the parameters retrieved from the data-base, the elements which melted in the thermo-mechanical analysis can be determined. To save
computing time, only those elements close enough to the weld-line were checked. With the melted elements, the stiffness was set close to zero before a temperature load was applied, and the plastic strain was set to zero after the temperature load was applied.

Deactivating an element using the birth and death technique causes near zero stiffness (10^{-6} N by ANSYS default), and the element is deactivated before a thermal load is applied. Plastic strain can be set to zero by applying the DESOL post processor command. This command is used to manipulate the ANSYS output file, i.e. plastic strain of the elements.

5.6. Results and discussion

As discussed in Chapter 4 and shown in Figure 4.4, ANSYS solved the FEM problem in three steps: modeling, simulation, and post processing. NeT provided experimental data for residual stress measurement. Those residual stresses are for longitudinal and transverse directions and no normal stress is available [27]. Following the available residual stresses data, Figures 5.7 and 5.8 show pictures of the distribution of longitudinal and transverse stress respectively, where it can be seen that the high tensile longitudinal or transverse residual stress in the region close to the weld bead are balanced by compressive stress in the field away from the weld bead. It should be noted especially in Figure 5.8 that those stresses are varied along the weld bead. When the welding starts, the entire region in the plate is set equal to its initial condition. For the next position the plate temperature changes due to the previous welding and this condition develops further with ongoing welding. Moreover the geometry also changes due to the embedded weld bulge on the plate surface.

In Figure 5.9 the distribution of longitudinal residual stress for the cross sectional area in the middle of the weld line (at z = 30mm refer to Figure 4.1 and 4.2) is shown. In the a similar manner the distribution of transverse residual stress is shown in Figure 5.10. Experiment data are obtained from the white dashed B2 line which is superimposed in the Figures 5.9 and 5.10. The line is lied 2mm below the surface of base metal, and along the line the numerical values of stresses (longitudinal and transversal) were retrieved from the ANSYS *.rst file and will be compared to the measured distribution of residual stress.
Figure 5.7 FEM longitudinal residual stress ($S_z$) prediction of the bead on plate welding

Figure 5.8 FEM transverse residual stress ($S_x$) prediction of the bead on plate welding
Experimental results for residual stress were provided by four participants in the NeT consortium: 1) Joint Research Centre, Institute for Energy, The Netherlands – JRC \cite{26}; 2) Nuclear Physics Institute, Czech Republic – NPI \cite{27}; 3) Materials Engineering, The Open University, England – OU \cite{28} and 4) Heinz Maier-Leibnitz Institute – HMI \cite{29}. All these participants used neutron diffraction to measure the residual stress and the results were used to validate the FEM prediction discussed in this paper.
In Figure 5.11 the data from all four participants was compared to the FEM data that predicted the distribution of longitudinal and transverse residual stresses along the 2mm path shown in Figures 5.9 and 5.10. The trend of the predicted residual stress can be seen to match those of the measured data. Predicting the longitudinal residual stress in the area close to the weld line is slightly lower compared to the general measured data, but it is close to the data provided by NPI.

The predicted longitudinal residual stress close to the weld line is higher than transverse residual stress unlike the area further away from the weld. This prediction is confirmed by the experiment results. The higher longitudinal residual stress is qualitatively understandable considering the discussion on the three bar elasto-plastic model in Chapter 2.31 which suggests that the residual stress is mainly formed in the longitudinal direction. Considering the forces balance, the higher residual stress in the area close to the weld line result in lower residual stress for the further area.
Figure 5.12 shows a normal view of the distribution of longitudinal residual stress in the area of symmetry. It also shows two selected paths 2mm and 3mm below the top surface. The 2mm path is shown as a red dashed line and the 3mm path is shown as a white dash-dot line (D2 and D3 lines). The blue dot line is also shown which is mid length through thickness of the base metal (BD line). The numerical data for those paths can be retrieved from the ANSYS *.rst file and compared to the NeT consortium experimental measurement. In the same manner Figure 5.13 shows predicted transverse residual stress.

A comparison of the predicted longitudinal and transverse residual stresses with the experimental result at the 2mm path (D2 line) is shown in Figures 5.14. The HMI did not report any result for the 2mm path. It should be noted that the end of the weld is laid on the origin of the coordinate system and the weld line spans to z = 60mm (Figure 4.2) where the weld started. By comparing the predicted and measured transverse residual stress.

Figure 5.12 FEM longitudinal residual stresses (Sz) prediction at symmetry area in a normal view.

Figure 5.13 FEM transverse residual stresses (Sx) prediction at symmetry area in a normal view.
stress at the 2mm path, as shown in Figure 5.14, it can be concluded that the FEM prediction agrees with the measured results quite well.

The other evaluation was done on the residual stresses at the 3mm path (D3 line) as shown in Figure 5.15. Only the HMI and JRC reported the distribution of residual stress at the 3mm path. When the predicted distribution of residual stress and the measured data were evaluated, the FEM prediction again gave well matched results.

The last evaluation is a residual stress validation on BD line. The longitudinal residual stress validation is shown in Figure 5.16 while Figure 5.17 is for transverse residual stress. All participants provide the experiment data. Zero y shows top surface position of the base metal and negative y shows the weld metal position of bop. The prediction on this line is also validated with the experiment.

Figure 5.14 Residual stresses on the D2 line
Figure 5.15 Residual stresses on the D3 line

Figure 5.16 Longitudinal residual stress (Sz) on BD line
5.6.1 Using isotropic hardening model

Figure 5.17 Transverse residual stress (Sx) on BD line

Figure 5.18 Kinematic hardening vs Isotropic hardening model
In the previous NeT simulation discussed in this chapter the kinematic hardening model was used. Figure 5.18 compares the results obtained from kinematic hardening and isotropic hardening model. Instead of using MKIN command for kinematic hardening, ANSYS APDL uses MISO command to represent isotropic hardening. From Figure 5.18 no significant differences were found when the isotropic hardening model was used. However using isotropic hardening gave a slightly higher result compared to the kinematic hardening models.

5.6.2 Comparing to Masubuchi’s

Figure 5.18 compares the results predicted by FEM and Masubuchi’s equation. Based on the FEM results the maximum residual stress of 250 MPa and 20mm width of the tensile area are used and applied to Masubuchi’s equation. Results from both Masubuchi’s initial (equation 5.1) and modified (equation 5.2) equation are plotted together with FEM prediction.

![Figure 5.18 FEM vs Masubuchi’s](image)

Figure 5.18 shows that modified Masubuchi’s equation can describe residual stress levels close to those derived by FEM. However it should be noted that, the maximum residual stress and width of tensile area were obtained from FEM prediction; which means the modified Masubuchi’s technique alone cannot provide the residual stress results.
5.7. Conclusion

A validated residual stress FEM prediction for bead on plate can be provided through the developed FEM model. This validated result confirmed the correctness of assumptions made and also the appropriateness of the developed code and techniques in ANSYS APDL. The melting phenomenon was approached by low stiffness and the plastic stress on the melted materials was omitted. A certain database contains an array of parameters which describe the temperature history created in the developed FEM model, and which can accelerate computer time. This database was proved useful in the Thermo-Mechanical-Metallurgical analysis which will be discussed in the next chapters.
References


[22] The Thao Doan, Thien Phuc Tran, Tuong Quan Vo and Ill Soo Kim (2010), *Finite element prediction of residual stresses and distortion in T joint fillet welds*, Proc. APIEMS, pp. 1 – 8.


Chapter 6

Finite Element Model of Residual Stress Involving Metallurgical Considerations

There is an increasing need for light steel structures involving the use of high strength ferritic-steels. During welding, these ferritic steels undergo a solid state phase transformation (SSPT) which affects the development of residual stress in the welded structures. Taljat et al. [1] defined residual stress as the stress that remains in a weld as a result of the transformation from a liquid to solid phase associated with the solidification and non-uniform cooling on the welded plate altered by SSPT. From that definition, it can be seen that for ferritic-steels welding SSPT plays an important role in the development of residual stress. From previously published papers [2 – 14] it can be concluded that phase transformation alters the residual stress not only due to changes in the material properties, but also by volumetric change.

SSPT alters the properties of materials and induces volumetric changes as a result of different atomic arrangements (see Figure 2.17). When a ferritic steel is heated above its transformation temperature (A1) its ferrite/pearlite structure (BCC) is starting to transform to Austenite (FCC). This transformation finishes at a certain temperature (A3). Due to the high Atomic Packaging Factor (APF) of austenite, when ferrite/pearlite is transformed to austenite a volumetric shrinkage is induced while the structure is heated. These existing austenitic structures, when cooled down to room temperature may be transformed into martensite structures (BCT) or bainite, depend on the cooling rate [15, 16]. For very low cooling rates, the austenite may transform to ferrite/pearlite (see Figure 6.1). A CCT diagram for X70 pipeline steel is represented in Figure 6.1. Unlike the other, microstructural components bainite has an acicular microstructure (not a phase) found by E.S. Davenport and Edgar Bain [17]. Since austenitic structures (FCC) have the highest possible Atomic Packaging Factor (APF) the phase transformation will expand the volume of the structures as it cools down to room temperature. Since this
transformation significantly affects the development of residual stress, it should be included in the FEM model.

![CCT diagram for high strength X-70 steel](image)

**Figure 6.1. CCT diagram for high strength X-70 steel** [16]

This chapter is a preliminary study which intended to develop ANSYS code that was able to model SSPT phenomenon especially the volumetric and material property changes. Transformation plasticity was ignored at this stage and but is accommodated in the next girth weld model (chapter 7). The developed logic and experimental procedure and results mainly follow a paper by Lee and Chang [18] with some modification elsewhere. From this paper; full martensite SSPT is implied for elements which are heated above A1. Also the cooling rate was considered high that only martensite was formed. In this thesis, martensite which is reheated above A1 will experience aged martensite SSPT. A kinematic hardening model was used following Lee and Chang.

### 6.1. Welding procedures

This section describes welding procedures carried out by Lee and Chang [18]. The base material was a 30mm thick, high strength carbon steel (POSTEN80) plate. The electrode for the Flux-Cored Arc Welding (FCAW) process was a MGS-80 type, 1.2 mm in diameter. The joint was a butt-joint type with a double V surface preparation, as shown in Figure 6.2. Five passes of FCAW were carried out to join the plate. First, three welding passes were laid on the top V groove, and then the plate was turned over and
the next two passes were laid. The welding parameters for each pass are listed in Table 6.1. There is no information about the cross sectional shape of the weld bead for each pass, so the shape of the weld bead was approached using the melting rate equation \[ MR = k + \alpha l + \frac{\beta l^2}{A} \] for FCAW (equation 6.1).

![Figure 6.2. Geometry and pass sequences of the welding process.](image)

<table>
<thead>
<tr>
<th>Pass</th>
<th>Current (A)</th>
<th>Voltage (E)</th>
<th>Welding Speed (mm/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>250</td>
<td>30</td>
<td>2.7</td>
</tr>
<tr>
<td>2</td>
<td>260</td>
<td>32</td>
<td>2</td>
</tr>
<tr>
<td>3</td>
<td>260</td>
<td>35</td>
<td>1.9</td>
</tr>
<tr>
<td>4</td>
<td>250</td>
<td>30</td>
<td>4.1</td>
</tr>
<tr>
<td>5</td>
<td>250</td>
<td>30</td>
<td>3.3</td>
</tr>
</tbody>
</table>

The melting rate \( MR \) (gr/min) is the mass of filler materials melted into the welding joint. \( K, \alpha, \) and \( \beta \) are the constants; \( l \) is the stick-out length and \( A \) is the cross section area of the filler materials. The relative area of the cross section of the weld-bead for each pass can be evaluated by dividing \( MR \) by the welding speed.

A preheating temperature of 110°C for the first pass was applied and the inter-pass temperature was 200°C ± 10°C.

### 6.2. Material properties

The chemical composition of both metals, plate and electrode, is shown in Table 6.2, and the thermal properties and mechanical properties for both metals, which are temperature dependent, are shown in Figures 6.3 and 6.4 respectively. There is no difference in the thermal properties of both metals and Figure 6.3 describes the thermal properties for both the base metal and weld metal. The temperature dependent Young’s
The modulus of both metals is also superimposed over each other. The differences are found in the temperature dependence of the yield stress and ultimate stress, as shown in Figure 6.4. The linear expansion coefficient and Poisson’s ratio are temperature independent with values equal to $1.2 \times 10^{-5}$ (1/K) and 0.3 respectively.

**Table 6.2.** Materials content of base metal and filler metal

<table>
<thead>
<tr>
<th>Material</th>
<th>POSTEN80 (mass,%)</th>
<th>MGS-80 (mass,%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>C</td>
<td>0.07</td>
<td>0.05</td>
</tr>
<tr>
<td>Si</td>
<td>0.3</td>
<td>0.44</td>
</tr>
<tr>
<td>Mn</td>
<td>0.91</td>
<td>1.35</td>
</tr>
<tr>
<td>P</td>
<td>0.015</td>
<td>0.006</td>
</tr>
<tr>
<td>S</td>
<td>0.004</td>
<td>0.001</td>
</tr>
<tr>
<td>Cr</td>
<td>0.45</td>
<td>0.6</td>
</tr>
<tr>
<td>Ni</td>
<td>0.97</td>
<td>2.3</td>
</tr>
<tr>
<td>Cu</td>
<td>0.02</td>
<td></td>
</tr>
<tr>
<td>V</td>
<td>0.038</td>
<td></td>
</tr>
<tr>
<td>Mo</td>
<td>0.45</td>
<td>0.25</td>
</tr>
<tr>
<td>B</td>
<td>0.0016</td>
<td></td>
</tr>
</tbody>
</table>

Figure 6.3. Temperature dependent thermal properties of materials

Figure 6.4. Temperature dependent mechanical properties of materials
6.3. Considerations for phase transformation

Lee & Chang \[18\] observed the microstructure of the weld region and HAZ, and found martensite in both regions, which confirmed that the weld region and HAZ experienced a martensitic transformation. The values needed to determine the volumetric change during martensitic transformation are the martensite start temperature, the martensite finish temperature, and fraction volume of the martensite.

The martensite start temperature can be approached using a formula proposed by Andrew\[20\], as shown in equation (6.2), which depends on the chemical content of the materials.

\[
Ms = 539 – 423C – 30.4Mn – 12.1Cr – 17.7Ni – 7.5Mo \quad (6.2)
\]

Using the chemical content at Table 6.2, the martensite start for HAZ is 456°C and 427°C for the weld metal.

The martensite finish for both metals is 200°C. The martensite fraction is obtained using Koistinen – Marburger’s\[21\] equation for carbon steels, as shown in equation (6.3).

\[
f_m = 1 - e^{0.011(T-M_f)} \quad (6.3)
\]

For a small temperature increment, the martensite fraction differential can be obtained from equation (6.4).

\[
df_m = -0.011e^{0.011(T-M_f)}dT \quad (6.4)
\]

The volumetric change due to martensite phase transformation can be obtained using equation (6.5).

\[
\Delta \varepsilon^{spt} = df_m \times \Delta \varepsilon^{mar} \quad (6.5)
\]

Where \(\Delta \varepsilon^{spt}\) expresses the volumetric change increment and \(\Delta \varepsilon^{mar}\) equal to \(1.5 \times 10^{-3}\)\[18, 22\] is the volumetric change of full martensite due to a solid state phase transformation.

The total volumetric change increment is a summation of volumetric change due to the martensite phase transformation and to the difference in temperature as follows.

\[
\Delta \varepsilon^T + \Delta \varepsilon^{spt} = \alpha dT + \Delta \varepsilon^{mar} df_m
\]
By evaluating equation (6.6), it is clear that the total volumetric change depends on the $M_s$ values which are different for the base metal (456°C) and the weld metal (427°C). Using these values, Figure 6.5 shows the effect of the martensite solid state phase transformation on the coefficient of thermal expansion for the base metal and weld metal. The coefficient of thermal expansion, without considering SSPT, is also plotted on the figure for a comparison.

\[
\Delta E_{\text{SSPT}} = 1.2 \times 10^{-5} dT + 1.5 \times 10^{-3} \left( -0.011 e^{0.011(T - M_s)} dT \right) \\
= \left\{ (1.2 - 1.65 e^{0.011(T - M_s)}) 10^{-5} \right\} dT
\]  
(6.6)

SSPT is also found when ferrite/pearlite transforms to austenite. The transformation started at $A_1$ (760°C) and finished at $A_3$ (920°C). The total austenitic transformation strain is $2.288 \times 10^{-3}$ [18, 20]. For simplification, the volumetric change is considered linear, and the volumetric change for any temperature at an austenitic phase transformation range is approached using linear interpolation, as shown in equation (6.7).

\[
\Delta E_{\text{SSPT}} = \frac{\Delta T}{A_3 - A_1} \Delta E^{\text{aus}} = \frac{-2.288 \times 10^{-3}}{920 \degree C - 760 \degree C} \Delta T
\]  
(6.7)

The total volumetric change is the sum of solid state phase transformation and elongation due to the temperature load, as shown below.
\[ \Delta \varepsilon_T + \Delta \varepsilon_{SSPT} = \alpha \Delta T + \frac{-2.288 \times 10^{-3}}{920 \, ^\circ C - 760 \, ^\circ C} \Delta T \]

\[ = 1.2 \times 10^{-5} \Delta T - 1.43 \times 10^{-5} \Delta T \]

\[ = -0.23 \times 10^{-5} \Delta T \]  

(6.8)

The effect of austenitic solid state phase transformation to the coefficient of thermal expansion is shown in Figure 6.6.

Figure 6.6. Effect of austenite on the coefficient of thermal expansion

The solid – liquid phase transformation was also found and based on previous FEM simulation (chapter 5), the effects should be considered. The solid – liquid phase transformation in thermal analysis is typically modeled by accommodating the latent heat. A latent heat of 270 kJ/kg at a temperature transformation range of 1450\(^\circ\)C - 1500\(^\circ\)C was applied \[22\]. When a metal becomes a liquid, the thermal conductivity linearly increases three times, between 1450\(^\circ\)C and 1727\(^\circ\)C \[1\]. This increased thermal conductivity is due to the effects of convective stirring.

Latent heat is also found when SSPT takes place. The latent heat of martensite SSPT can be obtained by evaluating the enthalpy difference between \(M_s\) and \(M_f\). The enthalpy of martensite SSPT depends on the \(M_s\) and \(M_f\) temperatures which are determined by the composition of the materials as shown in equation (6.2). Enthalpy at a certain
temperature can be evaluated using an equation proposed by Lee [23], as shown in equation (6.9).

\[ \Delta H_M^{\text{func}} \left( \frac{L}{L_{\text{mol}}} \right) = 0.041T^2 - 0.078T - 5079.047 \]  \hspace{1cm} (6.9)

Using equations (6.9) and (6.2), and considering the material compositions for weld metal and base metal as shown in Table 6.2, the latent heat for martensitic SSTP of the base metal is 75.07 kJ/kg and for the weld metal 66.51 kJ/kg.

The latent heat for austenite SSPT is 4200 J/mol [24]. Considering the materials composition, the austenite SSPT latent heat for base metal and weld metal is 25.35 kJ/kg and 26.27 kJ/kg, respectively. Table 6.3 shows the values of this latent heat and the temperature range of the suitable SSPT.

Table 6.3. Latent heat of the base metal and weld metal.

<table>
<thead>
<tr>
<th></th>
<th>Latent Heat (kJ/kg)</th>
<th>Temperature range (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Base Metal</td>
<td>Weld Metal</td>
</tr>
<tr>
<td>Martensite SSPT</td>
<td>75.07</td>
<td>66.51</td>
</tr>
<tr>
<td>Austenite SSPT</td>
<td>25.35</td>
<td>26.27</td>
</tr>
<tr>
<td>Melting</td>
<td>270</td>
<td></td>
</tr>
</tbody>
</table>

6.4. Finite element model

As discussed in Chapter 5 the thermo-mechanical uncoupled analysis is acceptable because the mechanical effects had an insignificant effect on the thermal analysis. Uncoupled analysis cannot be applied to thermo-metallurgical effects because the phase transformation alters the thermal properties of the materials whilst the temperature history of certain elements determines the developed phase at the element. In mechanical-metallurgical analysis phase transformation does affect mechanical analysis (in this case it is residual stress) due to an alteration of the mechanical properties and volumetric change, but the mechanical results have an insignificant effect on metallurgical analysis. Deaconu [25] described how those analyses affect each other in the Thermo-Mechanical-Metallurgical analysis shown in Figure 6.7.
From the above considerations thermo-mechanical and mechanical-metallurgical uncoupled analysis were applied to the FEM model whilst coupled analysis was applied to the thermo-metallurgical analysis.

### 6.4.1. Coupled thermo-metallurgical model

As shown in Figure 6.7, the temperature histories of a certain element determined the phase transformation of an observed element. When a ferrite/pearlite element is heated above A1 the element transforms to austenite. This transformation is completed at temperature A3. When cooled down to room temperature, this austenite transforms to prime martensite, it started at an $M_r$ temperature and finished at an $M_f$ temperature following Koistinen – Marburger’s law.

In multi-pass welding in this study, a prime martensite heated above A1 transforms to austenite and when cooled down to room temperature transforms to aged martensite in the $M_r$-$M_f$ temperature range.

When a phase transformation takes place, a certain element alters its thermal properties, which in turn affects the heat flow in the elements. The thermal property which is altered significantly when phase transformation occurs is latent heat, which is needed to transform one phase to another phase. The latent heats applied in this developed model are shown in Table 6.3.
The ANSYS procedure for thermo-mechanical analysis, is described as a flow-chart in Figure 6.8. First, the results from the thermal analysis without involving phase transformation were obtained for the initial analysis. From the thermal analysis, the temperature history for each pass can be retrieved from the ANSYS *.rth file and the peak temperature for each element can be determined. If the peak temperature exceeds A1 (760°C for both materials: weld metal and base metal), a solid state transformation occurs on the observed elements. The element will experience pearlite/ferrite – austenite – prime martensite in the first welding pass, and for the next passes the elements with peak temperature exceeding A1 will experience pearlite/ferrite – austenite – prime martensite, prime martensite – austenite – aged martensite or aged martensite – austenite – aged martensite depending on its initial phase.

The peak temperature can be obtained by comparing three consecutive temperature histories for a certain element, namely $T_{n-1}$, $T_n$ and $T_{n+1}$. $T_{\text{peak}}$ which equals $T_n$, is obtained when the condition $T_{n-1} \leq T_n \geq T_{n+1}$ is fulfilled. In the austenitic transformation range, the transformation takes place when the observed element is heated, whilst at the martensitic transformation range, the transformation takes place when the element is cooling down. By comparing $T_n$ to the previous temperature ($T_{n-1}$) it can be determined if an element is heated or cooled, i.e., when $T_n \geq T_{n-1}$ the element is heated whilst if $T_n \leq T_{n-1}$ the observed element is cooled.

Since the initial peak temperature was observed from thermal analysis without considering phase transformation, it should be reformulated when phase transformation takes place. The peak temperature of elements from previous analysis was replaced using the last analysis with the phase transformation considered. This process was iterated until a convergent solution was obtained. The convergence was evaluated by evaluating the microstructure configured at a certain cross sectional area. From the coupled thermo-metallurgy analysis, the initial phase, developed phase, and final phase for certain passes can be determined. Those phases will be used in the thermo-metallurgy-mechanical analysis to determine the residual stress.
6.4.2. Development of residual stress while considering the phase transformation

Thermal stress is basically the result of strain in an observed element. This strain, regarding the stress-strain diagram, determines the thermal stress. When a temperature load is applied to an element, due to the coefficient of thermal expansion and elevation
in temperature, the element will either expand or contract. When phase transformation takes place, expansion/contraction due to volumetric change as a result of APF difference occurs. The simple technique to incorporate the expansion/contraction of the phase transformation is use of a modified coefficient of thermal expansion, as described in equations (6.6) and (6.8).

Figure 6.9. Effects of prime and aged martensite on the yield stress of base metal

The phase transformation also alters the material properties of the observed element. The most important material properties that affect thermal stress is yield stress. When martensite is formed, the yield stress of the element increases, but an escalation of the yield stress depends on the fraction of martensite which is calculated using equation (6.3). The prime martensitic yield stress is 1050 MPa whilst the aged martensitic yield stress is 1120 MPa\textsuperscript{[26]}. The combined yield stress is obtained by weighting the initial yield stress and martensitic yield stress according the existing martensitic fraction. Figures 6.9 and 6.10 show the alteration in the yield stresses due to the existence of martensite, for the base metal and weld metal respectively.
6.4.3. Solutions and strategies

Some important phenomena obtained from the above discussion are summarised as follows:

a) From coupled thermo – metallurgical analysis the phases of each elements for a certain welding pass can be predicted.

b) The existing phase is determined by three important factors: the initial phase, the temperature gradient (heated or cooled), and the peak temperature of an element.

c) The initial phase for the first pass is ferrite/pearlite for all elements, and for the next pass depends on the previous thermo-metallurgical analysis.

d) The predicted phase transformation in the coupled thermo – metallurgical analysis determined material properties of the materials which will be used in uncoupled thermo – mechanical and metallurgical – mechanical analysis.

e) Three most important material properties that are affected by solid state phase transformation are: the coefficient of thermal expansion ($\alpha$), the yield stress ($\sigma_y$), and the existence of latent heat when phase transformation takes place.
There are three possible initial phases, depending on which pass the analysis is carried out on: ferrite-pearlite, prime martensite, or aged martensite. The temperature gradient has only two possibilities, heated or cooled, whilst the peak temperature may be divided into two classes: below $A_1$ and equal to or above $A_1$, in which a determination will be made as to whether the state transformation exists in a certain pass or not. Thus, there will be $3 \times 2 \times 2 = 12$ possible conditions where certain elements at certain times should be modeled.

Table 6.4 shows all of the possible material models which should be accommodated in the TMM analysis. As has been noted in item e) above, in mechanical analysis the change due to SSPT is coefficient of thermal expansion $\alpha$ and yield stress $\sigma_y$. Latent heat is also altered because of SSPT but this is in the thermal analysis stage. The subscripts in Table 6.4 express respective phases: $\gamma$-$\alpha$, $M''$ and $M''$ for ferrite/pearlite, prime martensite and aged martensite. Since the yield stress of martensite (both prime and aged martensite) as a function of temperature differs when it is cooled or heated subscripts “c” and “h” are added, i.e., $\sigma_y(M''_c)$ is yield stress temperature dependence which follows “Prime Cooled” curve in Figures 6.9 or 6.10 whilst $\sigma_y(M''_h)$ follows “Prime Heated”. Aged martensite follows the same manner. These materials models interchange with each other depending on the previous three factors already mentioned in item b): the initial condition, the temperature gradient and the peak temperature. The initial condition depends on the predicted phase of previous passes, except for the first pass where the initial condition is always pearlite-ferrite.

When an element is heated, the material model is equal to initial model, despite the peak temperature experienced by certain observed elements. When an element is cooled and its peak temperature is below $A_1$, the material behaves in its initial phase, but if its peak temperature is above $A_1$, the behaviour of the element is altered due to martensitic SSPT. Thus, for certain initial phases, only two element models are needed: the first is for heated elements and cooled elements with peak temperature below $A_1$, and the
Table 6.4. Change in material properties due to SSPT

<table>
<thead>
<tr>
<th>Initial Phase</th>
<th>Heated</th>
<th>Cooled</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Ferrite/Pearlite</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Below A1, without austenite SSPT</td>
<td>$\sigma_y (\gamma - \alpha)$</td>
<td>$\sigma_y (\gamma - \alpha)$</td>
</tr>
<tr>
<td>Above A1, austenite SSPT takes places</td>
<td>$\sigma_y (M'c)$</td>
<td>$\sigma_y (M'c)$</td>
</tr>
<tr>
<td><strong>Prime martensite</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Below A1, without austenite SSPT</td>
<td>$\sigma_y (M'h)$</td>
<td>$\sigma_y (M'h)$</td>
</tr>
<tr>
<td>Above A1, austenite SSPT takes places</td>
<td>$\sigma_y (M'h)$</td>
<td>$\sigma_y (M'h)$</td>
</tr>
<tr>
<td><strong>Aged martensite</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Below A1, without austenite SSPT</td>
<td>$\sigma_y (M''h)$</td>
<td>$\sigma_y (M''h)$</td>
</tr>
<tr>
<td>Above A1, austenite SSPT takes places</td>
<td>$\sigma_y (M''h)$</td>
<td>$\sigma_y (M''h)$</td>
</tr>
</tbody>
</table>

second is for cooled elements that experienced a peak temperature above A1. Since there will be three possible initial phases, the total number of element models required
for certain materials are six instead of twelve which will simplify the ANSYS simulation. The material model for the base metal and weld metal is different so the ANSYS program should model $6 \times 2 = 12$ models.

In the mechanical stage, the growing weld bead should also be accommodated using the birth and death techniques, as discussed in Chapter 5. For a certain thermal load, a certain grown weld bead is modeled and the rest are deactivated. Another modeling phenomenon is melting. When an element has melted its plastic strain is removed. The melting condition can be obtained from the temperature data of each element for a certain time step in thermal analysis. To remove the plastic strain, the DESOL command is used, as discussed in Chapter 5.

![Figure 6.11. Procedure in the TMM analysis](image)

The thermal load can be obtained directly from the `thermal.rth` file which is produced by the thermal analysis in ANSYS. Unfortunately extracting data from this file is time consuming when accessed for parameters, so a shadow database was created to save
computing time (Chapter 5). This database is in txt format and a certain parameter can be quickly and easily retrieved from it.

The flow chart in Figure 6.11 describes how all those aspects are carried out in the developed ANSYS procedure.

In multi-pass welding, the exchange of data for each step should be tightly controlled. This data includes element configuration, the temperature histories of each element, the time step and other miscellaneous data. The other important parameters are: the initial phase while the element is heated, the predicted phase for each element when the welded plate is cooling down to room temperature, and the peak temperature of each
element which is in the form of an array. How those data are interchanged between passes is described in Figure 6.12. It should be noted that only the thermal load is retrieved directly from the ANSYS result files, whilst the other data is obtained from the created database.

Element meshes and configurations have been resumed from the first data base used in first pass of the coupled thermo-metallurgical analysis. Since a mechanical analysis was carried out sequentially after the coupled thermo-metallurgy analysis was completely finished, the data and mesh configuration cannot be retrieved directly after each pass has finished. After each pass has been completed the data are duplicated and used in appropriate time step in the mechanical analysis. It should be noted that each pass consists of many time steps, and in each time step a certain temperature for a certain element is recorded in the ANSYS *.rth files. Since the data base was created outside the normal ANSYS procedure, the data should be correctly flagged to guarantee the appropriate temperature load data mapping for certain time steps and certain elements. The element configuration and parameter transfer for the next pass (3, 4, and 5) follow the same pattern as in pass 2.

6.4.4. The ANSYS APDL modeling

All of the above considerations need to be written in ANSYS program, but it the standard ANSYS system needs some adjustment to accommodate them. ANSYS APDL basically follows the pattern shown in Figure 6.13. Four types of consecutive blocks should be defined: 1) ANSYS setting, 2) Pre-Processing, 3) Solution/Processing and 4) Post-Processing. ANSYS setting defines what kind of analysis will be carried out such as: transient or steady state analysis, thermal or mechanical analysis, etc. The pre-processing block consists of the material model, the geometrical model and the load definition. In the material model the type of element, material properties, etc., are defined. The geometrical model defines the geometry of the specimen, meshing configuration, the relationship between geometries, and initial configuration of the microstructure etc. Load definition block describes loads, constraints, initial and boundary conditions. In the thermal analysis, the load was defined as a moving heat source. The heat source was represented by a combination of Goldak’s distributed volumetric heat source model and the uniform heat source model, as discussed in
Chapter 4. Goldak’s distributed volumetric heat source represents the heat embedded by a welding torch. The heat source parameters $r_x$, $r_y$ and $r_z$ are equal to those used previously in Chapter 4, i.e. 5mm, 2mm and 3mm respectively. The value of heat for certain relative position depends on the voltage, current and welding efficiency for each pass. The voltage and current are tabulated in Table 6.1 and the welding efficiency was considered equal to 0.85$^{[9,18]}$. The efficiency used here is different from that used in the previous chapter since the welding process is different, in this case FCAW is used whereas previously TIG was considered. Even in the same welding process the efficiency may differ depending on various parameters. In the current analysis the data used in references [9, 18] is employed since it aims to conduct a modeling exploration based on existing data. The heat intensity load for each element was calculated using equation 4.4. Following data by Lee$^{[9]}$ and Lee & Chang$^{[18]}$, convection and radiation were represented by convection coefficient of 15W.m$^{-2}$K and 0.2 emissivity respectively. It should be noted that this value differs from values in chapters 4 and 5 which may be caused by the different environment. The initial temperature was equal to the preheat temperature of 110°C. The uniform temperature load of 2400°C represents the heat from the melted filler metal that formed the weld bead for each pass. The heat of Goldak’s heat source should be reduced by the heat to melt the filler metal.

Figure 6.13. ANSYS APDL basic pattern
The solution/process block typically executes the calculation based on FEM analysis solely by the ANSYS program. The results are retrieved and interpreted in the post-process block. To obtain an effective process and results, the program should be built by following the basic structure shown in Figure 6.13.

Constants or values (called parameters) can be defined in any process (Pre-processing, processing or post processing). Another benefit of using parameters is that they can be implemented more quickly than using the results files obtained from ANSYS post-processing approach discussed previously.

From Figure 6.7 which described the residual stress prediction involving the metallurgical consideration, three different analyses were carried out: thermal analysis, metallurgical analysis and mechanical analysis. Microstructures are defined in the material model and as mentioned in Section 6.4.3, twelve different material models are defined to represent all possible microstructures.

Microstructures are defined as a material which follows the properties described in Section 6.3, especially those shown in Figures 6.5 and 6.6, and Figures 6.9 and 6.10 in Section 6.4.2. The microstructures were represented by material models and this requires 12 models (see section 6.4.3). When transformation takes places in an element in a certain time the element is switched to the proper element model as explained previously (see Table 6.4). The existing microstructure depends on the initial microstructure and the peak temperature reached in a certain pass and its temperature gradient in a certain time step. The flow chart of the thermo-metallurgy-mechanical analysis in ANSYS APDL is shown in Figure 6.14. To ensure a consistent geometrical model in each analysis, the RESUME command was used to copy the preprocessing stage from each analysis.

In the metallurgical analysis, there is no finite element calculation performed by ANSYS. The geometry model especially the configuration of the microstructure was duplicated from the thermal analysis. This configuration of the microstructure was recalculated when welding was ongoing. All the predicted microstructures configuration were saved in PRED1(i), PRED2(i), PRED3(i), PRED4(i) and PRED5(i) for element i at pass 1st, pass 2nd, pass 3rd, pass 4th and pass 5th respectively whilst for initial
microstructure configurations are saved in \text{INIT1}(i), \text{INIT2}(i), \text{INIT3}(i), \text{INIT4}(i) \text{ and } \text{INIT5}(i). The initial microstructure for all elements in the first pass was austenite.

Some of the parameters were obtained from the post processing stage in the thermal analysis using the *GET command. Parameters are in an array form. Typical parameter configurations are shown in Figure 6.15. By default the dimension of parameters array
is 3, but the real dimension is defined in *DIM command. The TIME(i) parameters are in one dimensional array and are described in real time (in seconds) for certain i<sup>th</sup> time steps. The second and third values in definition for the one dimension array are kept as 1. TEMP(i, j) defines the temperature (in °C) at i<sup>th</sup> time step of node j. TPEAK(i) shows the peak temperature of elements i at a certain pass. TMAX1(i) defines real time (second) when the peak temperature of element i was reached for the 1<sup>st</sup> pass whilst TMAX2(i), TMAX3(i), TMAX4(i) and TMAX5(i) are for the 2<sup>nd</sup> pass, 3<sup>rd</sup> pass, 4<sup>th</sup> pass and 5<sup>th</sup> pass respectively. This database was formed from ANSYS post processing files in the thermal analysis stage, however when it is coupled with metallurgical analysis, parameters in the database can be revised. The data is shown in Figure 6.15 in its „raw” form.

```
*DIM, TEMP    , ARRAY,   147,       65402,       1,
  
  *SET, TEMP    (     126,   65402,       1),  200.5419023837
  *SET, TEMP    (     127,   65402,       1),  199.8736012935
  *SET, TEMP    (     128,   65402,       1),  199.2146155278
  *SET, TEMP    (     129,   65402,       1),  198.4845373333
  *SET, TEMP    (     130,   65402,       1),  197.7606519666
  *SET, TEMP    (     131,   65402,       1),  196.0865018539
  *SET, TEMP    (     132,   65402,       1),  194.4669872133
  *SET, TEMP    (     133,   65402,       1),  192.9016331259
  *SET, TEMP    (     134,   65402,       1),  191.3881535053
  *SET, TEMP    (     135,   65402,       1),  189.9233473592
  *SET, TEMP    (     136,   65402,       1),  187.300439905
  *SET, TEMP    (     137,   65402,       1),  184.825443043
  *SET, TEMP    (     138,   65402,       1),  182.481782146
  *SET, TEMP    (     139,   65402,       1),  180.256770373
  *SET, TEMP    (     140,   65402,       1),  177.490127784
  *SET, TEMP    (     141,   65402,       1),  174.8348604693
  *SET, TEMP    (     142,   65402,       1),  172.368655197
  *SET, TEMP    (     143,   65402,       1),  170.029848041
  *SET, TEMP    (     144,   65402,       1),  167.720786258
  *SET, TEMP    (     145,   65402,       1),  165.949137780
  *SET, TEMP    (     146,   65402,       1),  163.5953366859
  
  *DIM, TMAX1   , ARRAY,   59016,       1,
  
  *SET, TMAX1   (       1,       1,       1),  11.851848000000
  *SET, TMAX1   (       2,       1,       1),  11.851848000000
  *SET, TMAX1   (       3,       1,       1),  11.851848000000
  *SET, TMAX1   (       4,       1,       1),  10.370367000000
  *SET, TMAX1   (       5,       1,       1),  8.888860000000
  *SET, TMAX1   (       6,       1,       1),  11.851848000000
  *SET, TMAX1   (       7,       1,       1),  10.370367000000
  *SET, TMAX1   (       8,       1,       1),  10.370367000000
  *SET, TMAX1   (       9,       1,       1),  11.851848000000
  *SET, TMAX1   (      10,       1,       1),  11.851848000000
  *SET, TMAX1   (      11,       1,       1),  13.333329000000
  *SET, TMAX1   (      12,       1,       1),  13.333329000000
  
```

Figure 6.15. Saved in parameters.

Beside any previous array parameters which were derived from the post-processing of the thermal analysis, others were generated based on considered conditions. The
generated parameters are the initial and predicted microstructure configurations. The initial microstructure for element $i$ are labeled in INIT1($i$), INIT2($i$), INIT3($i$), INIT4($i$) and INIT5($i$) for the 1$^{st}$ pass, 2$^{nd}$ pass, 3$^{th}$ pass, 4$^{th}$ pass and 5$^{th}$ pass respectively, whilst PRED1($i$), PRED2($i$), PRED3($i$), PRED4($i$) and PRED5($i$) described the predicted microstructure. For the next pass the initial microstructure was set directly from microstructure predicted from the previous pass as shown graphically in Table 6.4.

The microstructure configuration and temperature at the ends of each pass were evaluated. If the microstructure and temperature did not converge, the thermal analysis was repeated. In this thermal analysis the microstructure configuration was redefined based on parameters modified in the metallurgical analysis. The initial microstructure configuration was defined using INIT1($i$), INIT2($i$), INIT3($i$), INIT4($i$) and INIT5($i$) parameters. When the time step was higher than TMAX1($i$), TMAX2($i$), TMAX3($i$), TMAX4($i$) and TMAX5($i$) for pass 1, 2, 3, 4 and 5 respectively this meant that the element($i$) had cooled, the element($i$) had swapped to PRED1($i$), PRED2($i$), PRED3($i$), PRED4($i$) and PRED5($i$). This coupled thermo-metallurgical analysis was iterated until the microstructure and temperature converged.

After the coupled thermo-metallurgical analysis was finished the last step was Mechanical Analysis (Figure 6.14). As in the metallurgical analysis, the pre-process step from thermal analysis was resumed to ensure that equal geometry and data mapping can be carried out correctly. The microstructure for each pass for a certain time step was generated again using parameters in the database. For a certain pass the first microstructure configuration was determined by INIT1($i$), INIT2($i$), INIT3($i$), INIT4($i$) and INIT5($i$). As in thermal analysis which considered microstructure, when a certain element had cooled down (time step was equal or greater than TMAX1, TMAX2, TMAX3, TMAX4 and TMAX5 for the 1$^{st}$, 2$^{nd}$, 3$^{th}$, 4$^{th}$ and 5$^{th}$ passes, respectively) the element microstructure was swapped to PRED1($i$), PRED2($i$), PRED3($i$), PRED4($i$) and PRED5($i$).

In mechanical analysis the distribution of temperature in thermal analysis was applied as a thermal load using the LDREAD command. The FEM method which was solely calculated by ANSYS was performed in the solution/process block and the solution can be evaluated in the post-processing stage.
6.5. Results and discussion

It should be noted that some simplification in this TMM model have been made. The main goals of this chapter were to check and evaluate ANSYS APDL program to determine whether it can manifest the logic discussed in this chapter. The correctness of the logic will be evaluated later after the results have been obtained.

As shown in Figure 6.1, at a slower cooling rate, not only martensite will be developed in the weldment, bainite and ferrite may also be developed and mixed with martensite. Also, not all austenite when cooled down to the room temperature at a fast cooling rate transformed to martensite, some austenite may have also retained.

The first simplification considered that only martensite would be developed when a certain element was heated above A1 and the cooling rate was neglected. This assumption is acceptable since phase transformation has a significant effect on the residual stress at areas close to the weld line (this will be discussed in more detail later in Chapter 9). In this region the cooling rate is typically high and as a consequence martensite will be formed. Martensitic volumetric change has high value of SSPT and thus affects significantly the residual stress. The other reason for the simplification is that a different X-70 material with a different CCT diagram will be obtained. Since this CCT diagram in Figure 6.1 is not a CCT diagram for POSTEN80 and MGS-80, sticking with the CCT diagram will not be useful.

The second simplification considered that all of austenite was transformed to martensite. In previous discussed TMM theory, the residual stress can alter due to phase transformation and the corresponding volumetric change in the material properties, but since martensite has the lowest possible APF and the highest volume expansion, neglecting any retained austenite is an extreme condition. A consideration of any retained austenite in the residual stress prediction will provide values between the TMM analysis with 100% martensite and TMM without any martensite formation, i.e. no SSPT. Another reason is that no data is available on what percentage of retained austenite is available for certain cooling rate for POSTEN80 and MGS-80.
The microstructure after convergence for each pass is presented in Figure 6.16. In Figure 6.16a, i.e. with the microstructure at the end of the 1\textsuperscript{st} pass, the elements close to the weld bead which have a peak temperature above A1 were transformed into martensite (shown in yellow). The same observation can be found in Figure 6.16b with the microstructure at the end of the 2\textsuperscript{nd} pass, except for the elements which were already in a martensitic phase, the peak temperature above A1 caused the element to transform...
to aged martensite (shown in red). The microstructure configured at the end of the 3\textsuperscript{rd}, 4\textsuperscript{th}, and 5\textsuperscript{th} pass follow the same logic as at the end of the 2\textsuperscript{nd} pass. Figure 6.16 demonstrates that the ANSYS code can follow the developed logic. However, this microstructure configuration is not validated with experiment data which is beyond the scope of this thesis.

The distribution of residual stress by considering SSPT is presented in Figure 6.17. Longitudinal residual stress was measured on the surface in the mid section of the plate. It should be noted that only experimental longitudinal residual stress is provided by Lee & Chang\textsuperscript{[18]} which is considered enough for the objective of this chapter; development of ANSYS code that can model the SSPT phenomenon. The FEM prediction of the distribution of longitudinal residual stress in the mid cross section is presented in Figure 6.18. An inspection of a path on the top surface of Figure 6.18 indicates a longitudinal residual stress which is comparable to the experimental results.

Figure 6.17 Longitudinal residual stress distributed on the whole model
Figure 6.18 Longitudinal residual stress distributed on mid cross section area

Figure 6.19 Longitudinal residual stress distributed on the top path of the mid cross section area.

Figure 6.19 shows a comparison between FEM predictions with and without SSPT and the experimental results to obtain a general insight into the effects of SSPT on residual stress. It can be argued from Figure 6.19 that the martensitic SSPT decreased the tensile residual stress in the area close to the weld beads due to an expansion of martensitic SSPT that occurred while the elements were cooled to room temperature. Since the expansion was constrained by the surrounding area away from the weld line, it caused
compressive stress which reduced the predicted residual stress without SSPT. Figure 6.19 shows that the SSPT led to a prediction that was closer to the experimental results, thus indicating that SSPT should be used to predict residual stress of high strength steels.

The prediction of longitudinal residual stress on the same cross sectional area without SSPT is presented in Figure 6.20. When Figures 6.18 and 6.20 are compared it shows that the residual stress using SSPT caused residual stress that was not distributed smoothly. The distribution of residual stress from the FEM prediction without SSPT was redistributed similar to the predicted phase shown in Figure 6.16.

The peak temperature in the top surface of both models (with and without SSPT) was evaluated and as shown in Figure 6.21, there are no significant differences. As discussed with the developed model, SSPT does not change the thermal properties, apart from the latent heat, when austenitic SSPT and martensitic SSPT are exhibited. Moreover, this latent heat is much lower than the latent heat of melting (270 kJ/kg). As a result of all the assumptions mentioned above, the peak temperature obtained from a thermal analysis of the evaluated path has almost the same for both cases.

On the other hand the mechanical properties that do change due to SSPT are the coefficient of thermal expansion and the yield stress which created residual stress in the
model with SSPT and was different to the case without SSPT. As a result, the residual tensile stress in the model with SSPT was reduced near the weld line and increased away from the weld line. It is interesting to relate the distribution of peak temperature (Figure 6.21) with the prediction of residual stress (Figure 6.19) and these figures are represented again in Figures 6.22 and 6.23 to evaluate the area where the peak temperature was above A1. Figure 6.22 shows that the peak temperature is equal to A1 when the distance from centre of the weld was between 15.63mm and 18.75mm. Residual stress in the model involving SSPT decreased close to the weld line and increased away from it, and coincidently this change from decreased residual stress and increased residual stress also lie between 15.63mm and 18.75mm (Figure 6.23). If the peak temperature of an element is above A1, the element experience SSPT and as a consequences it will expand when cooling from $M_s$ to $M_f$ temperatures. This expansion that is constrained by the surrounding materials causes compressive stress in the area where the peak temperature is above A1. That is why, in the area where the peak temperatures were above A1 (close to the weld line) the high tensile residual stress was compensated by the compressive stress developed due to martensitic SSPT. Furthermore, the reduction in residual stress close to the weld line caused an increase in residual stress away from the weld line. This phenomenon was caused by the force equilibrium principal.

![Figure 6.21 Peak temperatures on the top path at mid cross section area.](image)
6.6. Conclusion

The prediction of residual stress using the logic described in this chapter can provide results that match with previous experiment measurements. Using ANSYS TMM analysis was undertaken following the logic shown in Figure 6.14. For the APDL mode,
this logic must be written in ANSYS code which is understood by the ANSYS program. Much effort was spent to develop this code. It can be claimed that the developed ANSYS code can follow very well the logic shown in Figure 6.14.

Principally, SSPT affects residual stress through three phenomenons: volumetric change due to atomic arrangement, material properties alterations and transformation plasticity. The first two have been discussed in this chapter and the transformation plasticity will be discussed in the model of pipeline girth weld joints (Chapters 7 and 9). The omission of transformation plasticity may have caused the residual stress over-estimation by the FEM model.

Involving martensitic SSPT in the prediction of residual stress reduced the residual tensile stress near the centre of the weld. The martensite SSPT causes expansion while the transformed elements are cooling to room temperature, and this expansion is constrained by the surrounding area that causes compressive stress. The elements near the weld line experienced SSPT because their peak temperature was higher than A1, which is why the compressive stress that developed near the weld line compensated for the residual tensile stress as a result of thermal load.

A new feature in this thesis is the consideration of both prime martensite and aged martensite in a multi-pass welding model. Aged martensite was also considered for elements which were in previous phase had prime martensite SSPT.

The use of the parametric database is also a new feature. As shown in Figure 6.14, these parameters play an important role in the development of the ANSYS code. There are at least three advantages in using a parametric database: 1) Faster simulation time because it is easy to use and fast to retrieve compared to the ANSYS post processing files (*.rth, *.rts files), and 2) the results can be accessed easily, and 3) it makes available the required parameters that are not provided by ANSYS default. An evaluation of peak temperature on the top path, which was presented in Figure 6.21, was obtained directly from this parametric database and it may be difficult to obtain it directly from the ANSYS post processing files. Furthermore the database can be accessed by another program, since the database is in *.txt format. This fact makes it possible to combine the ANSYS results with another program.
References


PART IV

THE PIPELINE

GIRTH WELD
Chapter 7
Residual Stress Analysis of The Girth Weld Joint

This chapter discusses the modeling of the girth weld joint and the experimental work used as a validation. The first thing to be discussed is the welding procedure and parameters that were used in the girth welding. Weld modeling is then being carried out using approach previously described. The third part of the investigation is the measurement of residual stress and the last part of this chapter is a comparison between the FEM model and experimental results. Comparison of the actual temperature histories from certain nodes which represent the position of the thermocouples are made with those predicted using the coupled thermo-metallurgical analysis. A second comparison is made between the measured and the FEM prediction of residual stress.

7.1. Welding procedure and parameters

Two X70 pipes 42cm diameter, 21cm length and 8mm thickness were girth welded. The girth weld was carried out using an automated rig (Figure 7.1) which provided constant parameters. Three control motors with attached welding torches provided motions relative to the welded pipe. The first motor moves the welding torch parallel to the axial centerline of the pipe, and the second motor moves the welding torch in a radial direction. Both motors were used to place the welding torch in the desired weld line and to adjust the CTWD (Contact Tip to Work Distance) respectively. The third motor moves the welding torch around the pipe at a speed that determines the welding velocity.
It may have been easier to let the pipe revolve around its own axis while keeping the torch stationary. However moving the welding torch is closer to the real situation since it is impossible to turn the pipe when girth welding of pipelines in the field. As in actual girth weld line of pipelines, no heat treatment was applied (such as stress relieving or annealing) to the specimen.

Figure 7.2 shows the main interface seen on the computer screen when the girth weld controller is operating. The welding speed, CTWD, and the length of the stringer bead are shown on the screen. Zero point positioning in the circumferential direction can be applied by clicking the JOG tab bar and the rotational motor can be moved manually using the pendant controller shown in Figure 7.3. Two buttons are available, i.e. the red and the green button, which move the welding torch clockwise or anticlockwise respectively.

Figure 7.1.
The rig for girth welding
Choosing the setup menu of the main menu in Figure 7.2 will display the System Setup window shown in Figure 7.4, which is basically used to define the “zero point”. After entry of input data (Pipe Thickness, Root Face, Gas Shield Base to Contact Tip, Gas Shield Base to Pipe measured, Pendant Sensitivity and Welding/Motion Delay), the Calibrate tab bar is chosen to set the CTWD real distance using the red and green
buttons on the pendant which control the movement of the torch away from and towards the pipe surface respectively. The Centre Torch tab bar is used to manually control the initial transverse position of the welding torch, again using the red and green buttons on the pendant. Pushing the OK tab bar will again display the main menu in Figure 7.2. Once the zero point has been set, the system is ready to perform girth weld by pushing the START task bar of the main menu.

![Image of System Setup menu](image)

Figure 7.4 System Setup menu

Before welding can commence, the pipe should be setup (centred) on the rig (Figure 7.5). There are two goals that must be achieved when centring the pipe, the first is to provide constant CTWD and the second is to provide an equal transverse distance to the weld bug. The transverse movement by pendant control is limited by the construction of the rig (± 3cm).
Figure 7.5. Centering the pipe: a) radial (left) and b) transverse position (right)

Figure 7.6. The pipe is tilted from its ideal position: a) ideal position, b) tilted horizontally, c) tilted vertically and d) tilted arbitrarily.
Figure 7.7. The pipe is shifted from its ideal position: a) ideal position, b) shifted horizontally, c) shifted vertically and d) shifted arbitrarily.

Figure 7.8. Presenting dial indicator measurements in a diagram: a) radial position (left side) and transversal position (right).
Positioning the pipe in a desired position was not easy because the pipe can be tilted or shifted horizontally, vertically, or arbitrarily as shown in Figures 7.6 and 7.7. The typical results from the dial indicator measurement are presented in graphics and shown in Figure 7.8. Using both graphics it can be determined what should be done to obtain the ideal pipe position.

Furthermore, the pipe was not always perfectly round and thus a surface on one side was not matched with its opposite side. This is known as a “hi-lo” problem, so to provide good joint, a hydraulic jack was used to push the pipe externally or internally to align the edges of the pipes (as shown in Figure 7.9).

![Figure 7.9. Using hydraulic jack to minimise the hi-lo problem](image1)

End preparations on the 8mm thick pipe wall followed AS 2885.2 as shown in Figure 7.10. First a machined surface preparation provided the trimmed 60° edge was made. This edge was refined using a manual grinder and a 1.6±0.8 mm land is formed with a hand file.
A perfectly consistent gap parallel to the welding jig is almost impossible to be achieved due to the sum of small errors in surface preparation and positioning the pipe in the welding rig. To accommodate the inconsistency a pendant and a camera to detect the relative position of the welding torch to the gap were used. Figure 7.11 shows the camera with a cover to protect the camera from spatter and also to attach the light filter. The pendant initiates one of the motors mentioned above which control the transverse movement of welding torch using the dial (Figure 7.3). The sensitivity of the dial can be adjusted in the System Setup menu (Figure 7.4). By monitor the welding arc and the gap ahead (as shown in Figure 7.12) where the welding torch should be positioned transversally by the pendant can be predicted.
While welding was ongoing, the welding parameters (current and voltage) were recorded using an AMV4000 data logger (Figure 7.13). A sensor probe 1001 (Figure 7.14) was used to detect the current whilst the difference in voltage was measured directly at the pipe and filler wire.

![Figure 7.13 AMV4000](image)

![Figure 7.14 Current probe](image)

The welding speed of 300 mm/min (Figure 7.2) is controlled by the motor on the rig. The speed of wire feed is controlled by the GMAW welding machine and different values are applied for the root pass and filling passes. A semi automatic STT (Surface Tension Transfer) Lincoln GMAW machine as shown in Figure 7.15 was used. The STT is a controlled short-circuit transfer mode patented by Lincoln Electric that makes the GMAW process more applicable to the welding of open root weld preparations. STT uses a closed loop control where the arc current and voltage were automatically adjusted.
to provide the desired weld bead. The arc current and the speed of the wire feed were controlled independently while the heat input was kept constant regardless of any alteration to the electrode extension \cite{1-4}.

![Figure 7.15 STT Lincoln GMAW machine](image)

Table 7.1. Suggested parameters for high strength carbon steels \cite{2}.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Specification</th>
</tr>
</thead>
<tbody>
<tr>
<td>Included angle</td>
<td>60°</td>
</tr>
<tr>
<td>Root opening</td>
<td>1.6 – 2.8 mm</td>
</tr>
<tr>
<td>Root face (land)</td>
<td>1.6mm</td>
</tr>
<tr>
<td>Welding position</td>
<td>5G vertical down</td>
</tr>
<tr>
<td>Shielding gas</td>
<td>100%CO₂</td>
</tr>
<tr>
<td>Backing gas</td>
<td>NA</td>
</tr>
<tr>
<td>Electrode</td>
<td>L-56 ER70S-6 (φ 1.2mm)</td>
</tr>
<tr>
<td>Peak current</td>
<td>360A</td>
</tr>
<tr>
<td>Background current</td>
<td>55-65A</td>
</tr>
<tr>
<td>Tail-out current</td>
<td>0</td>
</tr>
<tr>
<td>Wire feed speed</td>
<td>3m/min</td>
</tr>
</tbody>
</table>

The parameters controlled by this machine are the wire feed, peak current, background current and tail-out current. The wire feed controls the rate of deposition and the peak current controls the arc length. The background current and the tail-out current control the heat-input. Tail-out current serves as a coarse heat control and background current is a fine heat control. The peak current affects the shape of the root face whilst the background current and tail-out current affect the back bead. Suggested parameters for welding the root pass of high-strength steels are shown in Table 7.1. Following the suggested parameters especially welding position, it should be noted that for each pass,
two different runs were applied from the 12 o’clock position to the 6 o’clock position: either clock wise or anti clock wise. The filling pass, especially passes 2,3 and 4, used a higher wire feed speed (4 mm/min) and as consequence other parameters were changed to provide a good weld bead, i.e., the peak current was 415 Amp and the background current 90 amp.

![Figure 7.16 Data logger and thermocouples attached to the surface of the pipes](image)

A National Instruments USB 6211 with a seven port data logger was used to record the temperature histories at different position on the welded pipe (Figure 7.16). These temperature histories were used as validation of the thermal analysis. A sketch of the overall experimental set up is shown in Figure 7.17.

![Figure 7.17 Experiment set up](image)

7.2. Finite element model

The mesh design of the pipe is shown in Figure 7.18. A very fine mesh was applied close to the weld line because it was necessary to model the moving heat source and
also an abrupt thermal profile close to the weld. A symmetrical model to save computation could not be applied because the individual weld bead was not always symmetric to the centre of V groove especially for the 3rd and 4th passes (Figure 7.10). The model consists of 78380 elements (SOLID70 and SOLID45 for coupled thermo-metallurgical analysis and mechanical analysis respectively) and 94976 nodes.

A cross section of the weld bead was obtained following the pass design shown in Figure 7.10 and thus the cross sectional area for each pass can be calculated by considering the diameter of the wire (\( d_{\text{wire}} \)), wire feed speed (wfs) and welding speed (\( v \)) using equation 7.1. Using Figure 7.10 and equation 7.1, the configuration of the weld beads was predicted as shown by the ANSYS graphs in Figure 7.19.

Figure 7.18 Mesh configurations for the girth weld joint

Figure 7.19 Predicted cross sectional area (left) and mesh design on the cross section (right)
SSPT was taken into account when the FEM for the girth weld was carried out. The finite element model followed the methodology discussed in Chapter 6. First, a coupled thermo-metallurgical analysis was performed iteratively until the phase configuration and temperature at a chosen position converged. Using the results from the coupled thermo-metallurgical analysis, a sequential (uncoupled) mechanical analysis was carried out to obtain the residual stress.

Using the suggested parameter \[^2\], two runs were carried out for each pass: clock wise and anti clock wise as shown in Figure 7.20. The weld started at 12 o’clock and ended at 6 o’clock. The residual stress was measured at 3 and 9 o’clock paths which were far enough from the welding start and end, and the FEM prediction on these paths were compared to the measurement to evaluate the correctness of FEM model.

![Figure 7.20 Sketch of the girth welding process.](image)

Some important aspects which are different with plate welding are discussed in the next sections.

### 7.2.1. Coordinates transformation

The obvious difference between a girth weld and welding on a plate is the coordinate transformation required to define the moving heat source model. As was explained in
Section 4.3, Goldak’s distributed volumetric heat source model was expressed in a position relative to the centre of a moving heat source where the intensity of the volumetric heat source in a certain element was determined by its relative position to the centre of the heat source. Besides the relative position, in pipe welding the relative rotation is also needed. The right hand rule as in Figure 3.2 was also used with an additional rotation rule shown in Figure 7.21. Rotation around axis x, y, and z axes is known as $R_x$, $R_y$ and $R_z$ respectively. The $R_x$, $R_y$ and $R_z$ shown in Figure 7.21 are the positive values.

In Figure 7.22 the coordinate of a certain point is tabulated, evaluated from the initial coordinate and from a new coordinate which was rotated with an angle equal to $\theta$. The “new” position can be expressed as follows; in terms of its initial values and rotation angle by a simple trigonometry function, as shown in Figure 7.22

![Figure 7.21 Cartesian axes with rotations.](image)

![Figure 7.22a. A point at (y,z) observed from a coordinate Rx = $\theta$](image)
Figure 7.22b A point at \((x,z)\) observed from a coordinate \(R_y = \theta\)

Figure 7.22c A point at \((x,y)\) observed from a coordinate \(R_z = \theta\)

\[
\begin{align*}
\text{Rx}\theta : \\
\begin{bmatrix}
  x' \\
  y' \\
  z'
\end{bmatrix} &= \\
\begin{bmatrix}
  1 & 0 & 0 \\
  0 & \cos\theta & \sin\theta \\
  0 & -\sin\theta & \cos\theta
\end{bmatrix} \\
&\begin{bmatrix}
  x \\
  y \\
  z
\end{bmatrix}
\end{align*}
\]

\[
\begin{align*}
\text{Ry}\theta : \\
\begin{bmatrix}
  x' \\
  y' \\
  z'
\end{bmatrix} &= \\
\begin{bmatrix}
  \cos\theta & 0 & -\sin\theta \\
  0 & 1 & 0 \\
  \sin\theta & 0 & \cos\theta
\end{bmatrix} \\
&\begin{bmatrix}
  x \\
  y \\
  z
\end{bmatrix}
\end{align*}
\]

\[
\begin{align*}
\text{Rz}\theta : \\
\begin{bmatrix}
  x' \\
  y' \\
  z'
\end{bmatrix} &= \\
\begin{bmatrix}
  \cos\theta & 0 & -\sin\theta \\
  0 & 1 & 0 \\
  \sin\theta & 0 & \cos\theta
\end{bmatrix} \\
&\begin{bmatrix}
  x \\
  y \\
  z
\end{bmatrix}
\end{align*}
\]

The square matrix on the right side is called the transformation matrix and Goldak’s heat source model can be defined for the rotated coordinate system shown in Figure 7.23. For example, Figure 7.23 shows a coordinate system rotated around the z axis \(R_z\theta\). All other relative positions are evaluated from the new coordinate system \((x_r, y_r)\) with the origin at the centre of the heat source. This initial coordinate system is usually called a global coordinate system (GCS) whilst the new local coordinate system is
termed user’s coordinate system (UCS). The angle of rotation can be obtained using trigonometric expression shown in equation 7.2.

\[
\arcsin \theta = \frac{y_c}{\sqrt{x_c^2 + y_c^2}}
\]  

(7.2)

Figure 7.23. Goldak’s heat source model in rotated coordinate system

After the global coordinate of elements has been transformed to the local coordinate system with the origin at the centre of the heat source, the combined heat source models (Goldak’s volumetric heat source and uniform temperature load) as previously used in Chapter 6 can be applied. The relative angle was used in the girth weld model, instead of the relative longitudinal position (evaluating the x coordinate position in a logic shown by Figures 4.14 and 5.6) to apply the birth and death technique and uniform temperature load.

Figure 7.24 Elements configuration of the girth weld joint.
The element configuration close to the weld line with killed (deactivated) elements and temperature distribution at a certain time step after the first pass are shown in Figures 7.24 and 7.25 respectively. Figure 7.25 shows that the coordinate transformation and the evaluation of relative angle developed are able to describe the birth and death technique in the girth welding process.

![Figure 7.25 Distributions of temperature when the heat source is moving.](image)

### 7.2.2. Filtering peak temperature

As discussed in Chapter 6, the defined peak temperature of an element for a certain pass has an important role in determining the real time phase of an element. The peak temperature $T_2$ is obtained by evaluating temperatures of three consecutive time steps, i.e. $T_1$, $T_2$ and $T_3$ to satisfy the condition $T_1 < T_2 > T_3$.

Figure 7.26 shows the temperature profile for an element close to the 6 o’clock position. On the left side is the temperature history of the element for the first pass since welding started until it cooled down to room temperature, and on the right side the second “peak temperature” of the element has been observed.
It can be concluded from Figure 7.26 that there are two temperatures which fulfill the condition required to identify the peak temperature (T1 < T2 ≥ T3): 2400°C and 21.52°C. The database for the temperature histories and prediction of peak temperature as shown in Figure 6.14 were developed after thermal analysis was completed which means that the phase transformation was applied based on the predicted peak temperature and predicted time when the peak temperatures were reached. All this information is saved in a database of parameters. Unfortunately the 21.52°C occurred after reaching 2400°C, and after a completed iteration the peak temperature was identified as 21.52°C. Since 21.52°C is far below 760°C the observed element does not experience SSPT whilst for 2400°C the peak temperature of the element should experience SSPT.

The second peak temperature is caused by numerical oscillation and existed especially in elements close to 6 o’clock position. To avoid wrongly identifying the peak temperature, peak temperature filtering was programmed to follow the logic shown in Figure 7.27. First, all the initial peak temperatures were set to equal zero as a tentative peak temperature. Once the certain “peak temperature” has been found, it is saved in a dummy parameter which will be compared to the previous tentative peak temperature. If the dummy parameter is higher than the tentative peak temperature, the tentative peak temperature is swapped with the dummy parameter. This step is iterated until the end of time step for certain pass. The last tentative peak temperature that is the highest one (for a certain path) is assigned as the peak temperature for the evaluated element. In Figure 7.28 the microstructure predicted close to the 6 o’clock position was compared for the
first pass with and without the peak temperature filtering and element configuration is displayed for a half model. From Figure 7.28, it can clearly be seen that the logic in Figure 7.27 has handled the peak temperature quite well, and so a sound predicted phase could be provided.

Figure 7.27 Filtering identification of peak temperature

Figure 7.28 Microstructure configurations at area close to 6 o’clock position for first pass with (right side) and without (left side) filtering peak temperature.
7.2.3. Material properties for X-70 base metal and ER70S-6 filler metal.

The coefficient of thermal expansion has a significant effect on the residual stress as discussed in Section 2.3. In this section the properties of both yield stress and thermal expansion will be discussed below, although their roles will be discussed further in Chapter 9.

The first material property to consider is the coefficient of thermal expansion for base and filler metal when SSPT is considered. Based on previous literature study [5-7], the coefficient of thermal expansion at room temperature was considered equal to 14 μm/m°C and 13.24 μm/m°C for base metal and filler metal respectively. When an austenitic transformation takes place between 760°C (A1) and 920°C (A3), the total austenitic transformation strain for filler and base metal (2.29 x 10⁻³) [8-10] is assumed to be linear and can be represented by a coefficient of thermal expansion equal to -2.3 μm/m°C (equation 6.8). The negative thermal expansion means when temperature increases the heated material shrinks. In equation 2.14 for a coefficient of thermal expansion the volume is expanded linearly while due to the austenite transformation the volume is shrunk and the net result is that the material is shrunk. This result is confirmed by typical temperature-dilatation diagram of steels when austenite SSPT take place (Figure 2.17).

The coefficient of thermal expansion at atomic level depends on the distance between atoms, which is described as the APF in SSPT. When austenite is formed, the coefficient of thermal expansion was considered to have shifted in proportion to the APF (0.68 for the initial phase and 0.74 for austenitic phase).

The chemical composition for ER70S-6 filler metal and X70 base metal is shown in Table 7.2. Using Andrew’s equation (equation 6.2) and the chemical composition from the table, the martensite start (Ms) for base metal is 454°C and 455°C for the weld metal. The coefficient of thermal expansion can be obtained using equation 6.6 which is a differential form of Koistinen-Marburger’s law (equation 6.3). When a fully martensitic phase is achieved, the coefficient of thermal expansion is again shifted proportional to the APF of martensite (0.67). The coefficients of thermal expansion for base metal and filler metal are presented graphically in Figure 7.29.
Table 7.2 Chemical composition of weld metal (wire) and base metal [4,5]:

<table>
<thead>
<tr>
<th></th>
<th>%C</th>
<th>%Mn</th>
<th>%Si</th>
<th>%S</th>
<th>%P</th>
<th>%Cr</th>
<th>%Ni</th>
<th>%Mo</th>
</tr>
</thead>
<tbody>
<tr>
<td>wire</td>
<td>0.08-0.09</td>
<td>1.42-1.65</td>
<td>0.81-0.87</td>
<td>0.006-0.010</td>
<td>0.004-0.010</td>
<td>0.01-0.05</td>
<td>≤ 0.04</td>
<td>≤ 0.01</td>
</tr>
<tr>
<td>BM</td>
<td>0.053</td>
<td>1.9</td>
<td>0.19</td>
<td>0.0008</td>
<td>0.012</td>
<td>0.01</td>
<td>0.17</td>
<td>0.24</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>%V</th>
<th>%Cu</th>
<th>%B</th>
<th>Ca</th>
<th>N</th>
<th>Nb</th>
<th>Ti</th>
<th>Al</th>
</tr>
</thead>
<tbody>
<tr>
<td>wire</td>
<td>≤ 0.01</td>
<td>0.17-0.22</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>BM</td>
<td>0.044</td>
<td>0.01</td>
<td>≤ 0.0001</td>
<td>0.0033</td>
<td>0.0047</td>
<td>0.043</td>
<td>0.023</td>
<td>0.042</td>
</tr>
</tbody>
</table>

Figure 7.29 Coefficient of thermal expansion for base metal (left) and filler metal (right).

The next discussion will describe how the yield stress at elevated temperature was modeled for filler metal. Determination of the mechanical properties, especially the yield stress at an elevated temperature, is challenging and little published experimental data is available. Some researchers predicted the mechanical properties at elevated temperature using an artificial neuron network [11-14]. In this thesis the yield stress is expressed by an empirical equation based on some previous experimental results.

Theoretically, most of the strength of metal alloy at high temperature is divided into two regions which are called “normal” low temperature and the creep control regions. A typical graphic of yield strength at an elevated temperature is shown in Figure 7.30. The material switches between each region at a “switching temperature” which is strain rate dependent [15, 16].

To evaluate the effect of the high temperature on the yield strength, instead of using its absolute value, it is better to use the reduction factor Kd that denotes the ratio between the value of yield strength at an elevated temperature to its value at room temperature. The Kd values obtained experimentally from previous published papers [6, 11] are shown in Figure 7.31. The normalised yield stress at an elevated temperature from the weld and
base metal model developed in Chapter 6 is included in the figure. In the figure there is also the assumed Kd as an input parameters in the ANSYS APDL model. The equation for the Kd model in the normal low temperature region is shown in equation 7.3, whilst in the creep control region it is shown in equation 7.4. The switching temperature is assumed to be 650°C.

\[ K_d = -2 \times 10^{-6}T^2 - 7 \times 10^{-5}T + 1 \] (7.3)

\[ K_d = 2 \times 10^{-7}T^2 - 5.7 \times 10^{-4}T + 0.414 \] (7.4)

Figure 7.30 Yield strength at elevated temperature.

Figure 7.31. Normalised yield stress at elevated temperature.
Figure 7.32 shows the yield stress for base metal and filler metal. Yield stress at an elevated temperature for base metal (X70) was obtained from previously published paper\(^5\) and data by Zhixiong Zhu\(^{[17]}\) whilst for filler metal it was derived using the Kd predictions shown in Figure 7.31, equations 7.3 and 7.4. The yield stress of ER70S-6 at room temperature depends on the inert gas used when welding is carried out as shown in Table 7.3\(^{[18]}\) and since 90% Ar and 10% CO\(_2\) were used the yield stress was 470MPa.

![Figure 7.32 Yield stresses of base metal and filler metal.](image)

<table>
<thead>
<tr>
<th>Inert Gas</th>
<th>Yield Strength (Mpa)</th>
<th>Tensile Strength (MPa)</th>
<th>Elongation (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>100% CO(_2)</td>
<td>440</td>
<td>560</td>
<td>29</td>
</tr>
<tr>
<td>75%Ar, 25%CO(_2)</td>
<td>460</td>
<td>565</td>
<td>27</td>
</tr>
<tr>
<td>90%Ar, 10%CO(_2)</td>
<td>470</td>
<td>580</td>
<td>28</td>
</tr>
<tr>
<td>98%Ar, 2%O(_2)</td>
<td>455</td>
<td>565</td>
<td>27</td>
</tr>
</tbody>
</table>

Table 7.3. Mechanical properties of AWS ER706S-6\(^{[17]}\)

When martensitic SSPT takes place the yield stress was altered due to the contribution of yield stress of prime martensite and aged martensite. Considering the carbon content and the initial yield stress, the prime martensite yield stresses were 1027 MPa and 937 MPa for base metal and filler metal respectively \(^{[19]}\), whilst the aged martensite yield stresses were 1082 MPa and 1003 MPa. As discussed in Chapter 6 (Figures 6.9 and 6.10), the combined yield stress was obtained by weighting the initial and martensite yield stress according the existing martensitic fraction from the Koistinen – Marburger’s equation (6.3). The corrected yield stresses when SSPT occurs are shown in Figures 7.33 and 7.34 for base metal and filler metal respectively.
Figure 7.33. Effects of prime and aged martensite on the yield stress of base metal

Figure 7.34. Effects of prime and aged martensite on the yield stress of filler metal
The other properties required followed previously published papers [5-7] and they are shown graphically in Figures 7.35 – 7.37. The density of both filler and parent metal are considered to be 7850 kg/m$^3$. 

---

**Figure 7.35** Young’s modulus (E) models for base metal and weld metal.

**Figure 7.36** Thermal conductivity models for base metal and weld metal.
The latent heat when martensite SSPT takes place follows equation (6.9) whilst for austenite SSPT was considered to be 4200 J/mol\(^{[20]}\). By considering the composition, and equations 6.9 and 6.2 the martensitic SSPT latent heat for each metal can be obtained. Austenitic SSPT in kJ/kg unit can be calculated based on its composition and assumption that the austenite SSPT latent heat is 4200 J/mol. The latent heat during solid-liquid phase transformation was assumed to be 270 kJ/kg in the temperature range 1450\(^\circ\)C - 1500\(^\circ\)C\(^{[19]}\). The latent heats and temperatures when phase transformations occur are summarised in Table 7.4. In this chapter isotropic hardening model was used instead of kinematic hardening since in some papers it has been shown that kinematic hardening under predicts residual stress\(^{[21, 22]}\).

**Table 7.4.** Latent heat of the base metal and weld metal.

<table>
<thead>
<tr>
<th>Latent Heat (kJ/kg)</th>
<th>Temperature range ((^\circ)C)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Base Metal</td>
</tr>
<tr>
<td>Martensite SSPT</td>
<td>74.360</td>
</tr>
<tr>
<td>Austenite SSPT</td>
<td>25.031</td>
</tr>
<tr>
<td>Melting</td>
<td></td>
</tr>
</tbody>
</table>
7.2.4. Considering transformation plasticity

When SSPT takes place, anomalous plastic flow is reported \[23-25\]; while the stress or temperature is kept constant a plastic deformation is observed. This plastic deformation is called as transformation plasticity; that is plastic flow arising from SSPT. A comprehensive discussion of transformation plasticity can be obtained from the publications of J.B. Leblond \[26-29\], and this information was used as a material model in this thesis.

In terms of welding FEM model, the transformation plasticity occurs in element (grains) which develops SSPT under stresses resulted by an instantaneous thermal load. When transformation plasticity is included, it alters the residual stress distribution \[30-31\] and in this chapter, unlike the previous model, transformation plasticity is incorporated to refine the FEM estimates. To underlined the role of transformation plasticity, Oddy et. al.\[30\] even stated that there are two ways that SSPT alters residual stress which both are irreversible: first is volume change and the second is the transformation plasticity. Irreversible transformation plasticity means when the transformed phase back to the initial phase does not cancel the plastic deformation. Transformation plasticity does not depend on transformation rate, therefore is not a creep phenomenon and simply ceases when SSPT is completed.

Mathematically, the transformation plasticity can be expressed as a general solution shown in equation 7.5.

\[ \varepsilon^{TP} = K \sigma \phi(f_m) \]  

(7.5)

\( \varepsilon^{TP} \) is transformation plastic strain, \( K \) is a constant and \( \phi(f_m) \) is a normalized function of martensite fraction which when no martensite formed equal to zero and when transformed fully to martensite equal to unity. There are some normalized function have been proposed and in this thesis a function proposed by Desalos \[32\] as expressed in equation (7.6) is used. It should be noted that using equation 7.6 \( \phi(0) = 0 \) and \( \phi(1) = 1 \) which meet a prerequisite of a normalized functions.

\[ \phi(f_m) = f_m(2 - f_m) \]  

(7.6)

\( K \) based on experiment data for several steels varied between 4.5e-5 to 10e-5 (MPa\(^{-1}\)) \[25\] and the mid value of 7.25e-5 (MPa\(^{-1}\)) is taken in this thesis. Indeed, there are some
function proposed for $K$ as a function of relative difference between the two phases \cite{26-29} but because of the lack of data, a moderate constant value as mentioned above was taken.

For instantaneous temperature in the range of martensite SSPT, $f_m$ can be obtained following Koistinen – Marburger (equation 6.3) and considering equation 7.6 $\phi(f_m)$ can be determined. For a certain temperature then $\varepsilon^{sp}$ is linearly proportional to $\sigma$, which simplify the material model.

### 7.2.5. Considering cooling rate

Based on the CCT diagram for X-70 in Figure 6.1 page 113, bainite and ferrite SSPT can be developed depending on the cooling rate. Ferrite may not formed as indicated by the nature of temperature history in the girth weld. In the welding model, only elements close to the weld line will experience peak temperature above A1. The cooling rate in these elements is high which restricts the possibility of ferrite formation.

![Temperature histories of nodes at varied transverse distances](image)

**Figure 7.38.** Temperature histories of nodes at varied transverse distances

The composition of base and weld metals, as shown in Table 7.2 in page 167, is different from that indicated in Figure 6.1 but it was considered that this will not significantly alter the CCT diagram. The cooling rate of elements which have peak temperature higher than Ac$_1$ can be observed by evaluating temperature histories of elements closed to the ongoing weld bead. For simplicity, evaluation was done when the
first pass was applied. The temperature histories which are shown in Figure 7.38 are obtained from nodes at the inner surface of the pipe at 3 o’clock position. In Table 7.5 temperature histories of nodes with varied transverse distances are numerically tabulated. The cooling rate is expressed as time needed to cool from 800°C to 500°C ($t_{8/5}$).

<table>
<thead>
<tr>
<th>Nodes</th>
<th>Distance (mm)</th>
<th>Tpeak (°C)</th>
<th>$t_{8/5}$ (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>n3</td>
<td>0</td>
<td>2400.00</td>
<td>9.596</td>
</tr>
<tr>
<td>n4</td>
<td>1</td>
<td>2400.00</td>
<td>9.596</td>
</tr>
<tr>
<td>n47055</td>
<td>2.8</td>
<td>2070.28</td>
<td>10.396</td>
</tr>
<tr>
<td>n47056</td>
<td>4.6</td>
<td>1287.52</td>
<td>11.195</td>
</tr>
<tr>
<td>n47057</td>
<td>6.4</td>
<td>907.25</td>
<td>11.995</td>
</tr>
<tr>
<td>n47058</td>
<td>8.2</td>
<td>646.276</td>
<td>n/a</td>
</tr>
</tbody>
</table>

Table 7.5. Cooling rate of nodes.

Considering table 7.6 and Figures 6.1 and 7.38, the $t_{8/5}$ of elements with peak temperature exceeding $A_{c1}$ will not be longer than 13s and only bainite or martensite transformations will occur as it was expected.

The bainite transition was modeled using JMAK$^{[33]}$ (Johnson-Mehl-Avrami-Kolmogrov) equation. The JMAK equation shown in (7.7) describes the bainite fraction formed when SSPT takes place. $k$ and $n$ are material properties, and $t$ is cooling time.

$$f_b = 1 - \exp(-kt^n)$$  \hspace{1cm} (7.7)

Following equation (7.7), the values of $f_b$ varied between zero and unity. It should be noted from (7.7) that $f_b$ never reach zero or unity that is why 5% $f_b$ was considered to represent full martensite transformation and 95% $f_b$ represents full bainite transformation. From the CCT diagram in Figure 6.1 full martensite transformation is achieved when $t_{8/5}$ is less than or equal to 7s whilst full bainite transformation takes place when the $t_{8/5}$ is higher or equal to 100s. Fitting these two conditions into JMAK equation, $k$ is equal to 0.003 and $n = 1.5$ (more or less). Substituting those values into equation (7.7), the bainite fraction can be expressed as cooling time function as shown by equation (7.8). A correction factor 1.053 is applied to obtain full bainite transformation for cooling rate equal or higher than 100s.

$$f_b = 1.053(1 - \exp(-0.003t^{1.5}))$$  \hspace{1cm} (7.8)
When the bainite transformation finished (temperature of an evaluated element equal to martensite start – ($M_s$)) then martensite transformation is started and will be ended at martensite finish ($M_f$) temperature. The volume fraction of martensite was determined by Koistinen – Marburger law $^{[34]}$. Since the bainite is formed prior to martensite transformation, the Koistinen – Marburger law should be modified as shown in equation (7.9) with $T$ as instantaneous temperature. It should be noted that equation (7.9) expressed martensite fraction at temperature between $M_s$ and $M_f$:

$$f_m = (1 - 1.053(1-exp(-0.003t^{1.5}))) (1-exp(0.011(T - M_s)))$$  \hspace{1cm} (7.9)$$

The final portion of martensite can be obtained by equation (7.10). Using equations (7.8) and (7.10) the final fraction of bainite and martensite for varied $t_{8/5}$ can be obtained and shown graphically in Figure 7.39. The shaded area is the possible phase in the girth welding process. Based on Figure 7.39, it is believed to be acceptable if only full martensite transformation is considered which makes the coupled TMM analysis much simpler.

$$f_m = (1 - 1.053(1-exp(-0.003t^{1.5})))$$  \hspace{1cm} (7.10)$$

![Figure 7.39. Final phase as $t_{8/5}$ function](image)

**7.3. Residual stress measurements**

The residual stress measurement was carried out at ANSTO (Australian Nuclear Science and Technology Organization) using the X-ray diffraction technique (XRD).
Basically XRD uses Bragg’s law which describes the constructive interference of an X-ray beam \[^{[35-37]}\].

One disadvantage of X-ray diffraction is that it is only suitable for measuring residual stress on surfaces. Measurement depths of only 10-20\(\mu\)m are standard, but when coupled with electro polishing surface removal, depths of up to 1-1.5\(\text{mm}\) are achievable. The present study was prompted by a need to assess the effect of residual stress on stress corrosion cracking and these surface stress measurements were considered valuable. Nominal accuracy depends on the observed material, and for steel the nominal accuracy is \(\pm 30\text{MPa}^{[38]}\), an accuracy that is seriously affected by the grain size and texture.

### 7.4. Analysis and discussion

The important parameters for moving the heat source model are the heat input, which is obtained from the product of voltage and current divided by the welding speed, and heat source parameters \(r_x, r_y,\) and \(r_0\) which were equal to 5\(\text{mm},\) 3\(\text{mm}\) and 2\(\text{mm}\) respectively. Real voltage and current were measured in the experiment using an AMV4000 welding monitor where the welding speed in every pass was set to constantly equal 300\(\text{mm/min}\). Real time heat input can then be calculated based on real voltage and real current. Figure 7.40 shows the voltages, current and power. In this figure the mean power for each pass were 1950, 2722, 2500, 2500 and 1950\(\text{W}\) for pass 1, pass 2, pass 3, pass 4, and pass 5 respectively. As in previous chapters the mixed mode heat source model was used and the heat transferred by the welding torch was divided into the uniform heat load needed to increase the temperature of the filler metal to the melting temperature, and Goldak’s distributed heat source that represents the heat applied to the base metal. The heat needed to melt the filler metal was equal to \(\rho c\Delta T\) which was equal to 10.872\(\text{GJ/m}^3\) which can be transformed into \(\text{W}\) by multiplying it with the cross sectional area of each bead times the welding speed (300\(\text{mm/min} = 0.005\text{m/s}\)), or it can be transformed to \(q''''\) which is equal to 82.40\(\text{MW/m}^3\). The cross sectional area can be obtained by evaluating the FEM geometry model shown in Figure 7.19. It should be noted that there are small differences between the theoretical cross sections using equation 7.1 and the cross section of the ANSYS model, although an attempt was made to make the cross section as close to the theoretical approach as possible. The body heat
load in the centre of the heat source model can be obtained using equation 4.4. The input data into the ANSYS APDL program for each pass are tabulated in Table 7.6. All the parameters and material properties discussed in Section 7.2.3 were used in the FEM simulation. The logic for FEM was discussed in Chapter 6 and the complete methodology is represented on one page, as shown in Figure 7.41, where again the important role of the parameter database developed in this thesis can be seen.

Figure 7.40 Welding parameters for the girth weld for (a) first pass, (b) second pass, (c) third pass, (d) fourth pass and (e) fifth pass.
Table 7.6. Parameters for moving heat source model.

<table>
<thead>
<tr>
<th>Pass</th>
<th>Power (Volt. Amp)</th>
<th>$A_c$ (mm$^3$)</th>
<th>$H_{\text{uniform}}$ (W)</th>
<th>$H_{\text{goldak}}$ (W)</th>
<th>$q_{\text{max}}$ (GW/m$^3$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>pass1</td>
<td>1950</td>
<td>11.3</td>
<td>612.51</td>
<td>1337.49</td>
<td>83.21</td>
</tr>
<tr>
<td>pass2</td>
<td>2722</td>
<td>15.0</td>
<td>813.62</td>
<td>1908.38</td>
<td>118.72</td>
</tr>
<tr>
<td>pass3</td>
<td>2500</td>
<td>14.9</td>
<td>810.02</td>
<td>1689.98</td>
<td>105.13</td>
</tr>
<tr>
<td>pass4</td>
<td>2500</td>
<td>14.9</td>
<td>807.41</td>
<td>1292.59</td>
<td>105.30</td>
</tr>
<tr>
<td>pass5</td>
<td>1950</td>
<td>11.0</td>
<td>597.17</td>
<td>1352.83</td>
<td>84.16</td>
</tr>
</tbody>
</table>

The first run was thermal analysis without SSPT, followed by the first metallurgical analysis. The important output from this metallurgical analysis was the real time (second) for each time step, the temperature of each element in each time step, the peak temperature of each element, the time when the peak temperature was reached, and the predicted SSPT configuration. All these parameters were saved in a data base of parameters from where they could be retrieved quickly and easily.

The SSPT configuration predicted by metallurgical analysis was compared to the SSPT configuration from a previous thermal analysis and when a different configuration was found the thermal analysis considering SSPT was run again. The thermal analysis considering SSPT executes light blue blocks whilst the thermal analysis without SSPT does not. SSPT was carried out after each time step by evaluating the current time compared to the time when peak temperature for each element was reached. When the current time was higher than the time when peak temperature was reached this means that the observed element has cooled and a martensitic SSPT can be executed at a temperature between $M_s$ and $M_f$ based on the predicted phase for each element. After the second thermal analysis considering SSPT had finished a metallurgical analysis was run and the resulting SSPT configuration was again compared with the SSPT configuration from the previous thermal analysis. This sequence was repeated until the SSPT configuration used in the thermal analysis was equal to the SSPT configuration predicted by the metallurgical analysis. After the coupled thermo-metallurgical analysis was completed a sequential mechanical analysis (uncoupled analysis) considering SSPT was then conducted to obtain the residual stress. A thermal load was applied using the thermal analysis post processing file (thermal.rth) and SSPT was conducted using information obtained from parameters saved in the txt data base.
Figure 7.41 TMM analysis of the girth weld joint
7.4.1. Validation of the thermal results

Using all the material models and the logic discussed previously, the ANSYS APDL was programmed. The results of the welding simulation for thermal analysis, especially the temperature history for a certain node, can be retrieved from the *.rth file. The FEM predicted temperature histories are validated with temperature histories recorded by the thermocouples attached inside and outside the pipe, and laid on a path close to the 3 o’clock position in the transverse position shown in Figure 7.42. For practical reasons the transverse positions of the thermocouple were measured relative to the edges of the gap, and then the values were transferred into the global coordinate system, as shown in Table 7.7. Figure 7.42 shows the mesh with flagged node numbers which represent the thermocouples. The transverse positions of the FEM nodes in the global coordinate system and the thermocouples in close proximity are included in Table 7.7. Table 7.8 shows the thermocouples and corresponding FEM nodes. Since Tc4 was detached while welding was on-going (pass 4) and the temperature history of the thermocouple was incomplete, the temperature history of Tc4 was not used as a validation tool. The temperature histories of the FEM selected nodes and the measured results of the thermocouple close to the nodes are shown in Figures 7.43 – 7.46.

![Figure 7.42. Sketch of measuring transversal position of thermocouples (left) and nodal numbers with transversal position close to the thermocouples (right)](image)

From Figures 7.43 – 7.46 it can be argued that the model provided quite good temperature histories at the evaluated nodes. The minor differences between the
predictions and the experiment were due to some simplification in the FEM model and experimental inaccuracy. Those simplifications in FEM that may have caused the discrepancies are:

Table 7.7. Thermocouples and nodes transversal position

<table>
<thead>
<tr>
<th>Thermocouples</th>
<th>measured (mm)</th>
<th>GCS (mm)</th>
<th>Nodes</th>
<th>GCS (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tc1</td>
<td>1.86</td>
<td>2.86</td>
<td>4</td>
<td>1.000</td>
</tr>
<tr>
<td>Tc2</td>
<td>3.30</td>
<td>4.30</td>
<td>47055</td>
<td>2.800</td>
</tr>
<tr>
<td>Tc3</td>
<td>3.70</td>
<td>4.70</td>
<td>47056</td>
<td>4.600</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>47057</td>
<td>6.400</td>
</tr>
<tr>
<td>Tc4</td>
<td>1.50</td>
<td>6.54</td>
<td>27908</td>
<td>6.328</td>
</tr>
<tr>
<td>Tc5</td>
<td>2.20</td>
<td>7.24</td>
<td>46397</td>
<td>7.552</td>
</tr>
<tr>
<td>Tc6</td>
<td>3.20</td>
<td>8.24</td>
<td>46398</td>
<td>8.776</td>
</tr>
<tr>
<td>Tc7</td>
<td>3.60</td>
<td>8.64</td>
<td>46068</td>
<td>10.000</td>
</tr>
</tbody>
</table>

Table 7.8. Thermocouples and corresponding FEM nodes

<table>
<thead>
<tr>
<th>Experiment</th>
<th>ANSYS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tc1</td>
<td>N 47055</td>
</tr>
<tr>
<td>Tc2</td>
<td>N 47056</td>
</tr>
<tr>
<td>Tc3</td>
<td></td>
</tr>
<tr>
<td>Tc4</td>
<td>N 27908</td>
</tr>
<tr>
<td>Tc5</td>
<td>N 46397</td>
</tr>
<tr>
<td>Tc6</td>
<td></td>
</tr>
<tr>
<td>Tc7</td>
<td>N 46398</td>
</tr>
</tbody>
</table>

- Contact between the jig and pipe was not modeled. In fact part of the heat was transferred through this contact with the jig which produces a higher cooling rate than if the pipe is held with an adiabatic contact.

- Thermal properties were not directly measured on the welded pipe. As mentioned previously in Section 7.2.3, the thermal properties were obtained from a previous literature study. Although the materials were the same as that used in the literature (X70 and ER70S-6), since X70 has wide variations in composition, a small difference in the thermal properties may have existed.
Figure 7.43 Validation of temperature history at node 47055 for all passes.

Figure 7.44 Validation of temperature history at node 47056 for all passes.
Figure 7.45 Validation of temperature history at node 46397 for all passes.

Figure 7.46 Validation of temperature history at node 46398 for all passes.
The coefficient of convective heat transfer and emissivity for radiation were also based on literature and previous simulations (chapters 4 – 6).

The heat source power in the FEM model was represented by its mean value and as shown in Figure 7.40, the actual heat source power varied over the length of the weldment.

From the experimental side some shortcomings that might have contributed to the deviation between the experimental results and ANSYS prediction are listed below:

- Some errors in the prediction of the weld bead cross section. As described previously in Section 7.2, the weld beads were predicted using a theoretical approach (equation 7.1) so it may differ from the actual formed weld beads. On the other hand the weld beads varied at different positions in the weld line due to the pendant control used to accommodate the inconsistent gap.

- The small error in identification of thermocouple position and since the evaluated position is close to the weld line this small error might cause discrepancies in peak temperature prediction.

- Actual relative transverse distance to the centre of the weld bead may have shifted due to the pendant control used to accommodate the inconsistent gap. If the relative actual transverse distance was closer to the ideal position the temperature of the thermocouple would be higher, but it would be lower if the actual distance was further away.

Errors on aimed measured temperatures due to thermocouple attachments as described graphically in Figure 7.47. The temperature measured by the thermocouple detects temperature at a slightly different position than the assumed target position. This may also impose a different environment with different coefficients of convection heat transfer. The larger the diameter of the
thermocouple wire the greater the distance between the target aimed for and the measured temperature. The diameter of the thermocouple was 0.5mm and the combined junction size was varied between 1 – 1.5mm.

Despite all these shortcomings the FEM predicted the temperature histories quite well, as shown in Figures 7.43 – 7.46. These temperature histories indicate that the thermal model developed can be considered to be the correct model. When the developed phase configurations are incorporated with the correct thermal results, it is expected that residual stress prediction will be close to the measured results.

7.4.2. Residual stress validation

Residual stress measurement was carried out at ANSTO using XRD and a review was described in Section 7.3. The most important result from this measurement is the longitudinal residual stress, or in terms of the girth weld may known as tangential stress. The longitudinal stress was measured at the 3 o’clock and 9 o’clock transverse paths on the outer surface of the pipes. Normal and transverse stress are also obtained from the XRD measurement. It should be noted that the nature of XRD only provides residual stress on the surfaces. The two samples of girth weld were called sample A and sample B. The welding parameters for the samples were kept constant. (Note: A and B are different samples. Although it was attempted to maintain the welding parameters constant for both samples but welding itself is a „noisy” process and the result will always be slightly different as it is shown). Stress on an unwelded sample was measured to include initial stress as a correcting factor to the residual stress of the welded samples. All those measurement results are shown in Figures 7.48 – 7.52. Figures 7.48 and 7.49 show measured residual stress at 3 o’clock path for samples A and B respectively whilst Figures 7.50 and 7.51 are for 9 o’clock path. The data of sample A for 3 o’clock transverse residual stress at distance 45mm from weld centre line then was dropped since it is inconsistence with the others and considered as an outlayer data. Figure 7.52 shows initial stress on a path in a transverse direction to the weld line. To obtain residual stress as a result of the welding process alone, the measured residual stress should be decrease by the initial stress. Initial stress will vanish in the area close to the weld line since it will be melted when the welding is applied. On the other hand,
initial stress is close to zero when it is close the weld centre line. For this reason the measured residual stress is decreased by initial residual stress in area beyond ±20mm from the weld centre line.

Figure 7.48. X70A – 3 o’clock – residual stress chart

Figure 7.49. X70B – 3 o’clock – residual stress chart
Figure 7.50. X70A – 9 o’clock – residual stress chart

Figure 7.51. X70B – 9 o’clock – residual stress chart
Initial residual stress varied from one sample to another and considering Figure 7.52 and residual stress for far field area in Figures 7.48 - 7.51, it is quite fair if the initial longitudinal stress is considered equal to 100Mpa while initial normal and transverse stress are assumed to be 150Mpa. Dropping the outlier data in Figure 7.48 and excluding initial residual stress, Figures 7.53 – 7.56 shows the residual stress distribution as a result of the girth welding process. Compared with the ANSYS prediction, the mean data of samples A and B as shown in Figures 7.57 and 7.58 are produced. Using the mean data, as representative, will decrease the number of scattered data on a graph while keep the general trends. The ±30 Mpa error bars are added to show the nominal accuracy of the XRD measurement.
Figure 7.53. Welding residual stress at 3 o’clock of sample A

Figure 7.54. Welding residual stress at 3 o’clock of sample B
Figure 7.55. Welding residual stress at 9 o’clock of sample A

Figure 7.56. Welding residual stress at 9 o’clock of sample B
Figure 7.57. Mean value of welding residual stress at 3 o’clock

Figure 7.58. Mean value of welding residual stress at 9 o’clock
ANSYS provided results in the global coordinates that in term of stresses are those stresses in x, y, and z directions and which are denoted in the ANSYS post process results as Sx, Sy and Sz respectively. To obtain the longitudinal and normal stresses, all stress components in a longitudinal (tangential) direction and normal direction should be retrieved for a correct interpretation of the ANSYS results.

The tangential stress was obtained as follows: First, the position of element was identified to obtain its local coordinate system with regards to the ANSYS geometric model (Figure 7.18) and right hand rule for translation and rotation (Figure 7.21). It can be concluded from those figures that there is no component of Sx to the longitudinal and normal stresses because it is perpendicular to the longitudinal and normal directions.

Basically an element can be in tensile stress or compressive stress in all direction or it may be tensile in one direction and compressive in another. It should be noted that this stress is not really a vector unit and to add one stress to another is not possible using the vector additional rule. The components of tensile stress will always be a tensile stress despite their direction and the component of a compressive stress has the same rule. This means the components of a positive value of stress (tensile) will always be positive and a negative value of stress (compressive) will always be negative. Based on this argument, the transformation matrix in equation 7.11, is used absolute values instead of the real values, i.e. |sin x| not sin x.

In Figure 7.59 a component is studied which is in tensile stress with the yz Cartesian coordinate system. The yz coordinate system is a plane that if viewed from a positive x axis and using the right hand rule, the positive angle is anti-clock wise. The local coordinate consist of a tangential axis and a radial axis and the component of a stress on this direction can be obtained using simple trigonometric relations. The components of those stresses (Sy and Sz) on the tangential (longitudinal stress) and radial directions (normal stress) are

\[ St = Sy \left| \sin \theta \right| + Sz \left| \cos \theta \right| \] and \[ Sr = Sy \left| \cos \theta \right| + Sz \left| \sin \theta \right| \]

or when expressed in a matrix, as shown in equation 7.11. The angle \( \theta \) is obtained from equation 7.12.
Using equations 7.11 and 7.12 the distribution of longitudinal stress on the girth weld is shown in Figures 7.60 – 7.61. In Figure 7.60 the longitudinal residual stress was shown in a full specimen and in a half specimen by passing through a cutting plane. On this cutting plane lay the evaluated paths (3 and 9 o’clock paths) where the FEM predictions will be verified with the previous discussed experimental results. Figure 7.61 enlarges the 3 and 9 o’clock parts for the half model in Figure 7.60. With the equal manner of the longitudinal stress, Figures 7.62 and 7.63 describe transversal stress while Figures 7.64 and 7.65 present normal stress. Numerical residual stress, which has been shown graphically on Figures 7.60 – 7.65, on defined paths can be retrieved from ANSYS post process using a series of PATH commands. Those numerical results are compared with the experiment results as shown in Figure 7.66 for 3 o’clock path and Figure 7.67 for 9 o’clock path.
Figure 7.60 Longitudinal residual stress profiles on full and half model

Figure 7.61 Longitudinal residual stress on enlarge portion of the half model

Figure 7.62 Transversal residual stress profiles on full and half model
Figure 7.63 Transversal residual stress on enlarge portion of the half model

Figure 7.64 Normal residual stress profiles on full and half model

Figure 7.65 Normal residual stress on enlarge portion of the half model
Figure 7.66. Residual stress on 3 o’clock path

Figure 7.67. Residual stress on 9 o’clock path
Some asymmetry in the stress distribution is expected due to in process variations in the welding process. From Figures 7.66 and 7.67 it can be concluded the FEM prediction can provide results that have a good agreement with the experiment measurements. It should be noted that, although the final bead model look symmetric (Figure 7.19) the residual stress distribution by FEM was not symmetric over the weld centre line since the actual joint geometry varied. It is obvious the 3rd and 4th passes were not symmetric (see Figures 7.10 and 7.19 for the passes). When the 3rd pass was applied the constraint conditions were added by previous 1st and 2nd passes, and similarly with the 4th pass the previous 3 passes all contribute to the constraint. Beside the difference in constraints, the geometry and initial conditions were not symmetric when pass 3 and pass 4 were applied. Thus, it is understandable that the final residual stress prediction by FEM was not symmetric over the weld centre and this was also consistent with the experimental results.

Higher longitudinal residual stress (compared to the other two component) can be demonstrated by FEM prediction that validated by the experiment results. Principally, as it has been theoretically discussed and presented with three elasto-plastic bars model the residual stress is a result of plastic strain in the mid bar. The value of the plastic strain depends on the magnitude of compressive stress when the mid bar is heated. This compressive stress is determined by temperature difference between the mid bar and the side bars or in welding case by temperature difference between adjacent areas. From the temperature profile for a certain time, the temperature difference is higher in the transverse direction than in the longitudinal direction which would explain why the longitudinal residual stress is higher than transverse or normal stress. Certainly the other phenomenon previously discussed as volumetric and material properties change and also transformation plasticity due to SSPT, stiffness and plastic strain mitigation when melting takes place also contribute to the final residual stress as additional aspects.

All the residual stress (longitudinal, transversal and normal) on 9 o’clock path as shown by FEM prediction are generally higher than 3 o’clock path validated with the experiment results. The residual stress difference between 3 o’clock and 9 o’clock paths is caused by the constraint which is different when clock wise and counter clock wise welding, where 3 o’clock and 9 o’clock paths laid respectively, are applied. When counter clock wise welding is applied the previous clock wise welding added constraint
to the welded specimen and this additional constraint increases the final residual stress. The FEM prediction as shown in Figure 7.67 overestimates the longitudinal residual stress. In actual condition due to temperature elongation when the first clock wise run, where 3 o’clock path laid, was applied the gap in anti clock part was changed slightly. The gap change may cause lower measured longitudinal residual stress which was not modeled in FEM simulation.

The other minor discrepancies were found due to imperfections from both the FEM method and experimental work. From the FEM side the errors are likely to be caused by the phenomenon listed below:

- The additional mechanical load as a result of pipe setting on the rig was not modeled.
- In FEM when the elements had cooled down from temperatures above A1 only the martensitic transformation was modeled, which is the extreme condition.
- The properties of materials, especially at elevated temperatures, were based on theoretical approaches which only represented the general or mean value, regardless of the unique behavior of the X70 pipes.

The errors due to experimental work are listed below:

- Basically the welding itself is a “noisy” process as shown by Figures 7.48 and 7.49 and also Figures 7.50 and 7.51. From samples A and B with constant welding parameters, some differences in the distribution of residual stress were still exhibited.
- A perfectly uniform surface preparation was not provided and since a consistent gap was not provided a manual pendant control was needed and the weld beads are not perfectly straight. This condition differs from situation which was modeled in FEM where an ideal condition was assumed.
- The available interface program does not provide changeable welding parameters whilst the weld bead for constant welding parameters varied with different position due to force of gravity.
The accuracy of X-ray diffraction methods is quantitatively around ±30 MPa for steel materials.

Regardless all the above mentioned imperfections, quite a good prediction of residual stress has been obtained, as shown by Figures 7.66 and 7.67. From those validated results it can be said that the assumptions taken to develop the FEM model can be considered to be reasonable.

### 7.5. Conclusions

Based on the validation of temperature histories and residual stress of the girth weld, it can be argued that there is good agreement between the FEM predictions and experiments, or in other words the theoretical approaches developed in plate welding (Chapter 6) can be applied to girth welding. However, from the simulation point of view, several factors should be noted:

- A coordinate transformation is needed because the ANSYS default is the Cartesian coordinate system. This transformation is needed to express Goldak’s moving heat source in Cartesian coordinate relative to the centre of the heat source.
- To obtain the distribution of longitudinal and normal stresses, manipulations based on the available perpendicular “Cartesian stresses” of elements namely $S_x$, $S_y$, $S_z$ and the position of the elements, were made. This manipulation was made using a data base of parameters.
- Peak temperature filtering should be done for the half pipe welding method (two runs for each pass) to avoid mistakes in identifying peak temperature.

Some deviation between the FEM analysis and experimental results were caused by imperfections from both simulation and experiment. Despite all the shortcomings, a quite close prediction was obtained which means the FEM procedure for the girth weld joint developed in this thesis can be considered to represent the actual phenomenon, especially for the thermal analysis and residual stress (mechanical). Further improvements to the girth weld modeling and validation could be achieved in both the
experimental work and FEM modeling. For experimental work it is suggested that the activities listed below should be performed:

- Preheat treatment may be applied to the pipes to omit the previously imposed residual stress due to fabrication and surface preparation.
- Improved automatic tracking of the weld seam could be utilized eg. using laser seam tracking.
- To provide a more consistent heat input a closed loop controller that gives feedback to the welding machine could be developed. The inconsistent heat input may be accommodated by welding parameters (current or voltage) or CTWD adjustment.
- A program that can adjust the welding parameters to provide a consistent weld bead in all positions of the girth weld may be very useful. Certainly, a study of the shape of the weld bead with respect to its different positions is needed.
- A flexible and portable jig with all the above improvement could be produced to make it applicable to real girth welding process in pipelines.

From the FEM point of view the ideas listed below may be improve the accuracy of the available program:

- It is better to obtain all of the modeled properties by directly measuring the observed materials, especially their properties at elevated temperatures, instead of the theoretical approach used in this thesis. However a huge of amount of effort and the availability of extensive elevated temperature measuring instruments would be needed.
- Considering the effect of the cooling rate on the phase development would increase the accuracy of the program instead of considering its extreme condition by only regarding the martensitic phase transformation. Developing this kind of program needs a primary data of experimental results suggested in the first point.
- In real conditions not all the austenite was transformed to martensite, and certainly consideration of retained austenite will increase the accuracy of the developed program, but again it needs a primary data based on experimental observation of the materials under study.
Considering a full martensite transformation (neglecting retained austenite) is an extreme condition, so the results involving retained austenite should be between the model without SSPT and that with full martensite transformation. The model without SSPT will be compared with model with full martensite transformation in the Chapter 9.
References


Chapter 8
Simulating DC-LSND on Girth Butt Welding

A great deal of efforts has been spent to mitigate residual stress with the most prominent technique being the application of extra heating or cooling. This can be divided into three different classes: applying extra heating/cooling before welding, while welding is ongoing, and after welding. Applying heating or cooling at the same time as welding saves production time and has economic advantages.

One technique for applying heating or cooling while welding is ongoing is called as DC-LSND (Dynamically Controlled – Low Stress No Distortion) one version of which was patented by Q.Guan, C.X. Zhang and D.L. Guo \cite{1, 2}. This technique applies a cooling source behind the welding torch.

Many subsequent researchers have published details of their own experiment and analyses of DC-LSND \cite{3-12}. Typically DC-LSND is applied on single bead butt joint of thin plate with the main goal of reducing distortion. No published papers which discuss the effect of DC-LSND on multi pass girth welds and the resultant residual stress could be found. Experimental investigation of the technique was beyond the scope of the current work but based on the development of the FEM residual stress prediction work reported above the possible effect of DC-LSND on multi pass girth butt weld joint was studied using the FEM approach.

Applying a trailing heat sink increases the complexity of the welding phenomenon which is already complex. An analytical solution to this complex phenomenon is almost impossible to obtain and the only suitable approach may be a numerical analysis using FEM. Although one author has mentioned that an ideal trailing heat sink is not her main goal, some excellent data applicable to the current analysis was found in a series of writings by E.M. Van der Aa \cite{10-12}. This data was used as a basis for the model of a trailing heat sink in this chapter. Van der Aa used CO\textsubscript{2} snow as a cooling media for the
trailing heat sink and applied it to a single pass of thin plate welding. A trailing heat sink was applied onto the flat surface of a 100mm wide by 200mm long, and either 1.5mm or 2mm thick AISI 316L stainless steel plate with a welded joint in the middle and parallel to the 200mm long side. The sketch of the DC-LSND system used by E.M. Van der Aa is shown in Figure 8.1.

The cooling heat source model in this chapter utilises the CO$_2$ snow trailing heat sink that is used in the E.M. Van der Aa papers and it will be studied the effects of applying DC-LSND on the residual stress in the butt girth weld joint discussed in Chapter 7.

The heat which is transferred from a surface area where DC-LSND was applied was assumed to be by a convection mechanism. The area where convection takes place is a double ellipsoid that can be expressed in terms of a local coordinate with an origin at the artificial centre of the trailing heat sink, as shown in equation 8.1. $A_{Hsink}$ is an area where convection takes place, $\xi_T$ and $\xi_L$ are the relative transverse and longitudinal distances from artificial centre of the heat sink respectively, $w$ is the transverse parameter of the area considered equal to 0.025m and $g_l$ is longitudinal parameter of the area where it equal to 0.10m when $\xi_L$ has positive values and 0.02m when $\xi_L$ has negative values. 0.10m and 0.02m are denotes as $g_1$ and $g_2$ respectively.

$$A_{Hsink} = \sqrt{\left(\frac{\xi_T}{w}\right)^2 + \left(\frac{\xi_L}{g_l}\right)^2} \leq 1 \quad (8.1)$$

![Figure 8.1. DC-LSND of a thin plate welding \cite{10-12}](image)
The cooling strength on the cooling area depends on the convection coefficient and temperature of the cooling media. The Gaussian distributed coefficient of convection after having been modified to suit the APDL code, can be expressed as a relative coordinate to the heat sink centre, as shown in equation 8.2. It should be noted that equation 8.2 is a general form of double ellipsoid of Gaussian distribution model and constants are used to match the model with the experimental measurements.

\[ h_{\text{sink}} = h_{\text{max}} p \cdot \exp\left(-m \left(\frac{\xi}{w}\right)^2\right) \exp\left(-n \left(\frac{\xi}{g_1} - 1\right)^2\right) \quad (8.2) \]

\( h_{\text{max}} \) expresses the maximum trailing heat sink’s coefficient of convection which is equal to 2000 W.m\(^{-2}\).K\(^{-1}\); \( p, m \) and \( n \) are constants equal to 1.2, 1, and 8 respectively, based on experimental results. Two important notes should be considered when equation 8.2 is used, the first is the equation is only applied inside area which fulfills equation 8.1 and the second is that the maximum value of the equation is constrained to 2000 W.m\(^{-2}\).K\(^{-1}\) whilst the minimum value is 25 W.m\(^{-2}\).K\(^{-1}\). Observing \( h_{\text{sink}} \) along the longitudinal axis (assigning \( \xi_T \) equal to zero), the \( h_{\text{sink}} \) can be represented as shown in Figure 8.2.

From Figure 8.2 it can be seen that the value of \( h_{\text{sink}} \) for \( \xi_L \) from -20 to 24.47mm is equal to 25(W.m\(^{-2}\).K\(^{-1}\)) and equal to 2000(W.m\(^{-2}\).K\(^{-1}\)) for \( \xi_L \) greater than 84.90mm. The higher is the \( \xi_L \) value the higher the term \( \exp\left(-n \left(\frac{\xi}{g_1} - 1\right)^2\right) \) of equation 8.2. The maximum value is equal to 1 when \( \xi_L \) is 100mm but evaluating \( h_{\text{sink}} \) at this position has no meaning with regards to equation 8.1 where the value of \( \xi_T \) will be equal to zero. The full value of \( \xi_T \) (equal to \( w = 25\)mm) is found at \( \xi_L = 0 \) but at this position, from Figure 8.2 the value of \( h_{\text{sink}} \) achieves its allowed minimum value, so at this position \( h_{\text{sink}} \) will be uniformly distributed equal to 20 W.m\(^{-2}\).K\(^{-1}\). It may be beneficial to evaluate \( h_{\text{sink}} \) at a path with \( \xi_L = 84.904\)mm which lay inside the area defined by equation 8.1 and the \( \xi_T \) is spanned from -13.208 to 13.208mm. The value of \( h_{\text{sink}} \) along this path is shown in Figure 8.3.
Figure 8.2 $h_{\text{sink}}$ at $\xi_r = 0$

Figure 8.3 $h_{\text{sink}}$ at $\xi_L = 84.904\text{mm}$

Figure 8.4 Temperature and $h_{\text{sink}}$ distribution at $\xi_r = 0$
The temperature of the cooling media (CO₂ snow) which is symbolised as $T_{\text{media}}$ also varies in the cooling area depending on the longitudinal position relative to the artificial centre of the heat sink, as shown in equation 8.3. As in equation 8.2, equation 8.3 has been modified to fit the ANSYS APDL code. $k_{\text{media}}$ is a constant which is equal to 38 based on experimental results\textsuperscript{[10-12]} and $T_{\text{CO₂}}$ is the temperature of CO₂ snow in a solid state, which is -78°C. At a path where $\xi_T = 0$, the distribution of temperature is plotted as shown in Figure 8.4 with $h_{\text{sink}}$ also superimposed on the graph. Figure 8.4 is the same as that of Van der Aa which means the modifications made to suit ANSYS APDL code are consistent with previous work. The cooling strength (heat flux) of the heat sink which follows the convection phenomenon depends on the coefficient of convection $h_{\text{sink}}$ and temperature difference between the surface and the cooling media, as shown in equation 8.4. $T_{\text{plate}}$ is the temperature of the plate which has increased from room temperature due to welding.

\begin{align*}
T_{\text{media}} &= k_{\text{media}} \log(g_1 - \xi_L) + T_{\text{CO₂}} \quad (8.3) \\
\dot{q}^{\ast}_{\text{sink}} &= h_{\text{sink}}(T_{\text{plate}} - T_{\text{media}}) \quad (8.4)
\end{align*}

A high $h_{\text{sink}}$ and a low $T_{\text{media}}$ at high $\xi_L$ will cause a very high cooling effect, which also depends on the temperature of the plate from the results of FEM thermal analysis. Since the higher $\xi_L$ means it is closer to the welding torch which means a higher $T_{\text{plate}}$ and as a result the temperature difference has increased, and the cooling strength close behind the welding torch, is very high. The cooling effect expressed by equation 8.4 indicates that a higher $\xi_L$ but higher $h_{\text{sink}}$ (see Figure 8.4) means stronger cooling strength. Of course this is only defined at $A_{\text{Hsink}}$. In the $A_{\text{Hsink}}$ higher $\xi_L$ lower $T_{\text{media}}$ (see Figure 8.4) results in stronger cooling strength. The initial temperature of plate as a result of welding $T_{\text{plate}}$ will be higher if closer to the heat source centre (higher $\xi_L$ – inside $A_{\text{Hsink}}$), and in $A_{\text{Hsink}}$ the small ellipse of Figure 8.16 is when all the discussed values are in the conditions that caused the cooling strength maximum: high $h_{\text{sink}}$, low $T_{\text{media}}$ and high $T_{\text{plate}}$.

### 8.1. Heat sink modeling in ANSYS

There are published papers which attempt to model the heat sink in an FEM model: using a negative heat flux (distributed or uniform) and applying convection onto a
surface. As described implicitly, this research considers that the heat sink relies on convection phenomenon. There are two reasons for using this model: the first is that applying a negative heat flux resulted in a very low temperature, even below 0°C, which is unrealistic [4] and the second is that the comprehensive model used by Van der Aa also used the convection phenomenon.

Figure 8.5 Defines $h_{sink}$ in ANSYS APDL
How the heat sink was modeled in ANSYS APDL is described in the flow-chart shown in Figure 8.5. Here the position of an element represented by its relative position to a centre point was first evaluated using the *GET command. This initial position in

Figure 8.6 $h_{sink}$ at the defined area

Figure 8.7 Defines $T_{media}$ in ANSYS APDL
ANSYS was defined in Cartesian coordinates and a coordinate transformation is needed as discussed in Section 7.2.1. The transformed coordinate can be obtained from the element’s relative local coordinate \((\xi_L, \xi_T)\). The area where the convection caused by the trailing heat sink was applied by a double-ellipsoid evaluated by equation 8.1 depends on the value of \(\xi_L\). On the elements inside the ellipsoid the values of \(h_{sink}\) were evaluated based on equation 8.2 with the following constraints: the maximum value is 2000 (W.m\(^{-2}\).K\(^{-1}\)) and the minimum value is 20 (W.m\(^{-2}\).K\(^{-1}\)). By following the flow-chart in Figure 8.5, the distributed \(h_{sink}\) inside the double ellipsoid is determined and shown in Figure 8.6.

Figure 8.7 describes how ANSYS APDL obtains the distribution of temperature of the cooling media on the double ellipsoid area where the convection heat transfer is applied. Similar to the case of \(h_{sink}\), the relative local position of an observed element is evaluated first. Based on the relative local position \((\xi_T, \xi_L)\) it can be determined whether the element is laid inside the double ellipsoid and when the element is inside the double ellipsoid, the temperature of the cooling media follows equation 8.3. Applying the flow-chart shown in Figure 8.7 using ANSYS APDL, the temperature distribution of the cooling media is shown in Figure 8.8. The artificial centre of the trailing heat sink has lagged 133 mm (see Figure 8.1) behind the heat source centre which modeled the welding torch. Considering Rz equal to 3mm the DC-LSND have 30mm offset with the heat source model.

![Figure 8.8 T\(_{media}\) at the defined area](image-url)
The next thing to be studied through FEM simulation is the application of the DC-LSND model to the girth weld joint. As previously described, it was assumed that the cooling strength and temperature can be expressed solely in terms of the relative transverse position ($\xi_T$) and relative longitudinal position ($\xi_L$) to the artificial centre of the heat sink. Indeed equations 8.1 to 8.4 are based on flat surfaces, however this approach can be accepted as long as the distance from the surface to the nozzle does not exceed the core length where the drop in the pressure head is not less than 95% \cite{13}. For the case of impinging jet applied to a curved surface it has been shown that on a convex surface Nusselt’s number will increase on a convex surface and decrease on a concave surface, but the difference in Nusselt’s number is not really significant\cite{14}. Since the heat transfer coefficient of convection depends solely on the Reynold’s and Nusselt’s number, while for almost equal speed of impinging media the Reynold’s number will not change significantly so it is hoped that the convection heat transfer coefficient will not alter much due to the radius of curvature. However, a more precise approach needs an experimental investigation of an actual heat sink applied onto the surface of the pipe which is beyond the discussion of this thesis.

8.2. Applying DC-LSND on the girth weld

The most suitable available command that can be used for distributed convection used in DC-LSND simulation are the following two consecutive SFE commands:

\[
\begin{align*}
SFE, \text{noELM}, \text{noSURF}, \text{CONV}, 1, h_{\text{sink}} \\
SFE, \text{noELM}, \text{noSURF}, \text{CONV}, 2, T_{\text{media}}
\end{align*}
\]

noELM is the number of an evaluated element, noSURF is the number of relative surface of an observed element, $h_{\text{sink}}$ is the coefficient of convection obtained using the flow-chart in Figure 8.5 and $T_{\text{media}}$ is the temperature of the cooling media obtained by following a logic described as a flow-chart in Figure 8.7. For SOLID70 and SOLID45 elements there are six possible numbers of relative surfaces that should be carefully evaluated to obtain the correct surface where convection will be applied. Elements where convection due to DC-LSND are applied will also be altered when the birth and death technique is applied that should be carefully modeled.
As discussed in Section 7.2.1, it is easier in the girth weld model if the global Cartesian coordinate was transferred to the local cylindrical coordinate. To fit with the experimental work, it was assumed that 12 o’clock position where the welding was started $\theta = 0$ and considering the right hand rule (Figure 7.21) the counter clock wise directions, i.e. 12, 11, 10, 9 … o’clock are positive values that coincide with $0, \pi/6, \pi/3, \pi/2$ … radian respectively. The angle of certain elements or nodes can be obtained using equation 8.5.

$$\theta = \cos \left( \frac{y}{\sqrt{y^2 + z^2}} \right) ; \text{for } z \geq 0$$

$$\theta = 2\pi - \cos \left( \frac{y}{\sqrt{y^2 + z^2}} \right) ; \text{for } z < 0$$ (8.5)

It should be noted that welding procedures suggested for the STT Lincoln GMAW machine meant that the welding was done with 5G vertical down position; that is there were two welding stages for every pass, i.e., anti-clock wise and clock wise. The local transverse coordinate ($\xi_T$) did not change with the $R_x\theta$ (Figure 7.21) and the mathematical expression for the local longitudinal coordinate ($\xi_L$) depended on the direction of welding, i.e. clock wise or anti-clock wise, the position of the cooling heat source centre and the observed element position where $z > 0$ were equal to quadrants 1 and 2, and $z < 0$ was equal to quadrants 3 and 4. The mathematical formulae for $\xi_L$ regarding all of those prerequisites are tabulated in Table 8.1.

Table 8.1. Mathematical formulas for $\xi_L$

<table>
<thead>
<tr>
<th>Welding direction : Anti clock wise</th>
<th>Welding direction : Clock wise</th>
</tr>
</thead>
<tbody>
<tr>
<td>$z_{0,DC} \geq 0$</td>
<td>$z_{0,DC} \geq 0$</td>
</tr>
<tr>
<td>for $z_j \geq 0$</td>
<td>for $z_j \geq 0$</td>
</tr>
<tr>
<td>$\xi_L = (\theta_j - \theta_0)R_j$</td>
<td>$\xi_L = (\theta_j - \theta_0 + 2\pi)R_j$</td>
</tr>
<tr>
<td>$z_{0,DC} \leq 0$</td>
<td>$z_{0,DC} \leq 0$</td>
</tr>
<tr>
<td>for $z_j \geq 0$</td>
<td>for $z_j \geq 0$</td>
</tr>
<tr>
<td>$\xi_L = (\theta_j - \theta_0 + 2\pi)R_j$</td>
<td>$\xi_L = (\theta_j - \theta_0)R_j$</td>
</tr>
</tbody>
</table>

$z_{0,DC} = z$ global coordinate for DC-LSND artificial centre.
$z_j = z$ global coordinate for evaluated elements.
$\theta_0 = \angle$ position of the DC-LSND artificial centre.
$\theta_j = \angle$ position of evaluated elements.
Using the formulae in Table 8.1 and previously discussed logic, a typical convection coefficient and temperature of the cooling media are shown in Figures 8.9 and 8.10 respectively. These figures describe the trailing heat sink when applied at the first pass with clockwise welding direction. Note that the heat sink model can follow the contour of V-grove and the active elements, i.e. the bead top surface of the first pass. It can be concluded from the figures that ANSYS can defined correctly the DC-LSND model of Van der Aa, in the girth weld model. However unlike Van der Aa the current analysis is applied to a thicker material on a curved surface.

Figure 8.11 show the temperature distribution of a certain time step of the first pass when the heat source of girth weld with trailing heat sink model is applied. It can be seen that the trailing heat sink alters the temperature profiles (compared to Figure 7.25). This temperature profile will be applied as thermal load in TMM analysis and since the thermal load has changed, the final residual stress should be altered. How this new thermal load affects the residual stress will be discussed in more detail in section 8.3.

Figure 8.9. Distributed \( h_{\text{sink}} \) (W.m\(^{-2}\).K\(^{-1}\)) of DC-LSND model applied on the girth weld
8.3. Results and discussions

Figure 8.11 showed temperature contours which were altered due to the application of DC-LSND in which can be clearly seen at the isothermal lines especially for 200°C line.
How the DC-LSND affects the residual stress, can be studied by evaluating the final residual stress when DC-LSND is applied in a similar manner to that in Section 7 of the girth-weld model. Figure 8.12 shows residual stress profiles for a model with DC-LSND in the areas of 3 and 9 o’clock which are comparable to Figures 7.59, 7.61 and 7.63 for girth-weld model without DC-LSND. Figure 8.13 compares residual stress for the girth-weld model (Chapter 7) evaluated at 3 and 9 o’clock paths and the residual stress resulting from a girth-weld with DC-LSND application for the equal paths. From these figures especially Figure 8.13 it can clearly be seen that the application of DC-LSND embeds compression residual stresses for the area close to the weld line. When a heat source is applied such as the welding torch, the final residual stress is tensile residual stress in the area close to the weld centre as a result of plastic strain and misfit that has been discussed thoroughly in section 2.3. Using an equal analogy, when a “cooling source” is applied, i.e. the DC-LSND, simply the final residual stress is the opposite of tensile that is compressive residual stress. This compressive residual stress resulting from the DC-LSND compensates for the tensile residual stress due to the leading welding torch as shown in Figure 8.13.

Practically, the main concern for structure integrity from residual stress is the tensile residual stress which in turn decreases the lifetime of the welded structure through its effect on fatigue load or stress corrosion cracking. The analysis shows that the application of DC-LSND would be expected to decrease the residual tensile stress near the centre of weld line on both paths. The maximum values of longitudinal residual stress decrease by 39.16% and 31.16% at 3 and 9 o’clock paths respectively. Whilst for transversal residual stresses are 50.59% and 38.25% and for normal residual stress are 34.81% and 19.35% at 3 and 9 o’clock paths respectively. Taking the mean values, the residual stress decrease 35.55% by application of DC-LSND.
8.12 Residual stress profile at 3 and 9 o’clock areas when DC-LSND is applied
As discussed in Section 2.3, thermal stresses exist as a result of differences in temperature of the weld line with its adjacent transverse position which had been described as side bars. Based on this theory it may useful to compare temperature histories of nodes which have different transverse distance. Nodes 4, 47056, 47057, 45903, 74728, and 74729 on the pipe’s inner surface have transverse distance from weld centre equal to 1 mm, 4.6 mm, 6.4 mm, 10 mm, 13.82 mm, and 17.24 mm respectively were chosen. The temperature histories for these nodes when the first pass was applied
are presented in Figures 8.14 and 8.15 for girth weld with and without DC-LSND. It can be seen from Figure 8.14 that the temperature history of nodes 4, 47056 and 47057 are altered by DC-LSND application but not for the rest. This result can be evaluated using convection coefficient of DC-LSND model which is mathematically expressed by equation 8.2 and shown graphically in Figures 8.2 and 8.4. Considering equation 8.2 and the permitted maximum $h_{sink}$, the area inside the small ellipse in Figure 8.16 has $h_{sink}$ equal to 2000 W.m$^2$.K$^{-1}$ and will decrease exponentially to the surrounding area. Only the area inside $A_{Hsink}$ of the small ellipse actually experiences DC-LSND and this area is shown by the grey area in Figure 8.16. This area may called as $A_{Hmax}$ where the coefficient of convection equal to 2000 W.m$^2$.K$^{-1}$.
The centre of the small ellipse can be expressed by $\xi_L$ and $\xi_T$. $\xi_L$ is obtained when the longitudinal term of the equation 8.2: $\exp\left\{8 \left( \frac{\xi_L}{0.1} - 1 \right)^2 \right\}$ is equal to unity and $\xi_T$ equal to zero since the equation is a symmetric function to $\xi_T$; so the centre of ellipse is at $\xi_L = 100\text{mm}$ and $\xi_T = 0$. The major axis of the ellipse is in the longitudinal direction and equal to $0.1\sqrt{\frac{\ln(1.2)}{8}} = 0.01510\text{m} = 15.10\text{mm}$ whilst the minor axis in the transverse direction is equal to $0.025\sqrt{\ln(1.2)} = 0.01068\text{m} = 10.68\text{mm}$. But it should be noted that the $h_{\text{sink}}$ is only applied in the area which satisfies equation 8.1 which graphically is shown in Figure 8.16 as part of the large ellipse behind the small one. The intersection between the big and the small ellipses are at $\xi_L = 92.73\text{mm}$ and $\xi_T = \pm 9.36\text{mm}$. As only nodes 4, 47056 and 47057 have transverse distance below 9.36mm, their temperature histories are altered by the application of DC-LSND model behind the welding torch while the rest (nodes 45903, 74728 and 74729) are not as shown by temperature histories of those nodes in Figure 8.15.
It should be emphasised that the thermal stress is caused by temperature distribution which is not uniform, so if a material is uniformly heated and cooled no thermal stress is formed as long as no constraint is applied. Without DC-LSND the temperature of those selected nodes is uniform at around 130\(^\circ\)C and 48\(^\circ\)C whilst with DC-LSND at around 78\(^\circ\)C and 180\(^\circ\)C which means the thermal stress will not developed further. The 48\(^\circ\)C and 180\(^\circ\)C may be considered as the equilibrium temperature where temperature for varied transversal position especially the closest node (node 4) with its adjacent (node 47056) can be considered same.

Basically the magnitude of developed thermal stress depends on the temperature distribution and yield stress. When an element near the weld line reaches its peak temperature the temperature differs significantly with its adjacent transverse elements. However at this high temperature the stiffness of the element is very low and as a result the thermal stress may not high. Furthermore, the resetting plastic strain is mitigating the thermal stress in the near melted element (this will be discussed further in Section 9.3), which is why it is sometime said that basically the residual stress is formed when the weld line is cooled down to the room temperature. This thermal stresses is developed further until the element near the weld line is cooled down to 48\(^\circ\)C if DC-LSND is not applied. Put simply the thermal stress is developed from 855\(^\circ\)C to 48\(^\circ\)C for the elements near the weld line which is represented by node 4. At 48\(^\circ\)C the difference of temperatures may not be high but since the yield stress is high enough the thermal
stress may be quite high. When DC-LSND is applied, elements near to the weld line develop thermal stress from 855°C to 180°C which has a narrower temperature span from peak temperature to equilibrium temperature than the girth weld without DC-LSND and also the stiffness at 180°C is lower than that at 48°C. This discussion may explain qualitatively why the final residual stress for girth weld especially close to the weld line with DC-LSND is lower.

8.4. Conclusions

Based on FEM predictions, the application of DC-LSND on multi-pass girth butt weld joint of thick pipe has been demonstrated to reduce the residual stress significantly. The stress reduction is a result of increasing equilibrium temperature especially for area close to the weld centre. The change of temperature was caused by the high coefficient of convection in $A_{Hmax}$ ellipse (shown by grey area in Figure 8.16). Coincidently the ellipse is in front part of $A_{Hsink}$ where the temperature caused by the welding heat source ($T_{plate}$) is higher than the rear part. Furthermore $T_{media}$ in the $A_{Hmax}$ is lower and as a consequence the cooling strength of the $A_{Hmax}$ ellipse area is very high.
References


PART V

GENERAL DISCUSSIONS
Chapter 9
General Discussions

9.1. Overview

Specific discussions concerning sources of error and the approach adopted have been included at the end of each chapter but the following summarises the overall outcomes of the work.

The study reported here indicates that FEM is a powerful tool to model a technically complex phenomenon. However, the results depended on assumptions made by users. When using a FEM software package such as ANSYS it is important that the programmer sets out the problem in logic and syntax which are understood by the chosen package.

There are two ways to ensure the reliability of an FEM model, shown in Figure 3.1, the first is; comparing the results obtained from FEM with the analytic solution. Since the analytic solution provides a solution for a simple condition, only a simple FEM model can be verified. Comparing the FEM model with Rosenthal’s solution for a welding heat source indicated the validity of the logic and syntaxes that were made and discussed in Chapter 3. The FEM model in Chapter 3 represented the thermal analysis of a moving point heat source in an infinite solid and Rosenthal’s solution represents an analytic solution of the temperature field of a moving point heat source in an infinite solid.

A general analytical solution that covers the transient and steady state was developed based on Carslaw and Jager’s \cite{1} general solution for energy conservation in an infinite solid. This solution was obtained using a non dimensional integral technique proposed by Komanduri and Hou \cite{2}. The transient solution obtained from FEM showed a good agreement with the analytical solution. For a steady state, the solution obtained from Rosenthal, the non dimensional integral and FEM also showed a good agreement.
Another approach to validating a FEM model was undertaken by comparing the FEM prediction with the experimental results. Under practical conditions welding is not as simple as most analytical models suggest. The FEM model should accommodate many factors to make it comparable to the experiment. However, some reasonable simplifications still need to be taken to make sure the problem can be solved using FEM.

Figure 9.1 indicates the difference between real welding conditions and simplifications which need to be adopted taken in the FEM thermal analysis [3] even for the very simple TIG process. In a thermal analysis using the FEM model, and the energy conservation law, the welding phenomenon was modeled as a moving heat source, with heat transfer due to conduction to the surrounding cooler metal, heat loss to environment due to convection and radiation, and latent heat as a result of phase transformation.

The values of transient temperatures are determined by the response of the material’s thermal properties which are temperature dependent subject to boundary condition of the heat source, conduction, convection, radiation and latent heat.

The heat source can be classified into three different ways: point heat source, surface heat source and volumetric heat source. The most prominent moving heat source model was Goldak’s volumetric heat source where the intensity of heat at certain position is
determined by its position relative to the centre of the heat source. Goldak’s heat source model was discussed thoroughly in Chapter 4.

In this thesis, a new heat source model was developed. The heat source was modeled as a combination of Goldak’s heat source model and a uniform temperature volumetric heat source. Goldak’s heat source was used to represent heat embedded into the welded metal by the welding torch and the uniform temperature volumetric heat source model was used to represent the melted filler metal that forms the weld bead. The validity of the heat source model was cross-checked against the experimental data as discussed in Chapter 4.

By using the validated temperature distribution as the thermal load, the transient thermal stresses in the welded plate can be obtained. The residual stress is a “steady-state” thermal stress, i.e. it is a thermal stress when the plate has cooled down close to ambient temperature. Relating the temperature distribution to thermal stress is usually known as thermo-mechanical analysis, but since the distribution of temperature is insignificantly influenced by the stress state, sequential thermo-mechanical analysis was used in this thesis and has been discussed in Chapter 5.

Since in this thesis the girth weld joint of ferritic steels pipes was discussed, SSPT needed to be considered. The Thermo-Metallurgy-Mechanical (TMM) analysis considers SSPT in the development of residual stress. The method used to carry out TMM has been shown in Figure 6.7 and in more detail in Figures 6.14 and 7.39.

### 9.2. TMM in plate welding

The FEM model that considers the SSPT for plate welding has been discussed in Chapter 6. From Figure 6.11 it can be concluded that SSPT affects the residual development of stress in welding because of changes in the properties of the material and volumetric change. The most important material property affected by SSPT was yield stress. The yield stress at a certain condition depends on each phase fraction which is accounted for according to the Koistenen–Marburger equation (equation 6.3). The effect of SSPT on yield stresses is presented graphically in Figures 6.9 and 6.10 for base metal and weld metal respectively.
As mentioned above, another aspect of SSPT that influences the residual stress is the volumetric change that occurs as a result of an atomic arrangement of each phase (Sections 2.4 and 6.3). Following the Koistenen-Marburger equation when martensitic SSPT takes place the volumetric change was presented graphically, as shown in Figure 6.5 (equation 6.6), but when austenitic SSPT takes places the volumetric change was considered linear (equation 6.8) and was presented graphically in Figure 6.6.

How this SSPT influenced the development of residual stress was shown in Figures 6.19 and 6.23, by comparing to the residual stress prediction without involving SSPT. As discussed above, the SSPT influences residual stress due to alteration of material properties and volumetric change. How each aspect influences the residual stress and to what extent it alters the development of residual stress is an interesting question.

### 9.2.1. Change of material properties in SSPT

In this chapter the volumetric change due to SSPT will be excluded as was considered in the material model stage. To exclude volumetric change, the FEM model follows the logic shown in Figure 9.2, which is a modification of the flow-chart in Figure 6.11 where volumetric change due to SSPT was excluded. (Note: mechanical property which was altered is yield stress).

It should be noted that altering the thermal properties due to SSPT, especially latent heat, was still considered in the coupled thermo-metallurgical analysis to determine the predicted phase. Using a flow-chart as in Figure 9.2, the predicted residual stress was obtained, as shown in Figure 9.3. The predicted residual stress was compared with the predicted residual stress without considering SSPT.
It can be concluded from Figure 9.3, that if only the mechanical properties, i.e., the yield stress, was included in the residual stress prediction, the tensile residual stress close to the weld joint would increase. Since the initial mechanical response due to thermal load was elongation, simply increasing yield stress will raise the residual stress. When yield stress changes due to SSPT is considered without a volumetric change, the residual stress in the weld line increases by 27.30% and the maximum residual stress increase by 23.84% compared to residual stress without SSPT. (Note: SSPT results in a volumetric change, but not only that; the material properties especially yield stress are also altered. In the SSPT model both contributes to the final residual stress).
9.2.2. Volumetric change in SSPT

Whilst in Section 9.2.1 the volumetric change was neglected in subsequent mechanical analysis, in this chapter the change in mechanical properties was neglected. The goal was to separate the effect of volumetric change from the change in the mechanical properties. The FEM model followed the flowchart in Figure 9.4 and as in Section 9.2.1 in coupled thermo-metallurgical analysis the change in thermal properties due to SSPT should be included.

The prediction of residual stress by excluding mechanical properties of the material is presented in Figure 9.5. Residual stress prediction without SSPT was also plotted in the same figure to study the effect of the volumetric change due to SSPT.

Two volumetric changes occurred due to SSPT, when austenite and martensite transformation took place. Volumetric shrinkage occurred while the elements were heated due to Austenite transformation with increased the tensile stress whilst volumetric expansion while the elements were cooled due to martensite transformation produced a compressive stress close to the weld line. The Austenite transformation existed at high temperatures (760°C - 920°C) when the yield strength of the weld metal and base metal were low. On the other hand, Martensite transformation took place in the lower temperature range (456°C - 200°C for base metal and 427°C - 200°C for weld
metal) when the yield strength of the weld metal and base metal were higher than austenitic SSPT. Using the analogy used in Section 9.2.1: the lower yield strength altered the residual stress less; so it is understandable that the net effect of volumetric change due to SSPT reduced the tensile residual stress close to the weld line as shown in Figure 9.5. The volumetric change due to SSPT, excluding yield stress changes, reduce the residual stress in the weld line by 67.04% and the maximum residual stress reduce by 62.68% compare to residual stress without SSPT.

Figure 9.4. Excluding mechanical properties change in SSPT
9.2.3. Aspects involved in SSPT

Results obtained by FEM prediction, as described in Section 9.2.1 and Section 9.2.2 are compared to evaluate each parameter that was changed due to SSPT in residual stress development. Those results were plotted together with the residual stress prediction with SSPT and without SSPT, as shown in Figure 9.6.

From Figure 9.6 it can be seen that the prediction of residual stress with SSPT is a combined result of a model with a change in material properties only and a model that only accommodates volumetric change. Neglecting volumetric change due to SSPT will overestimate the residual stress close to the weld line whilst neglecting the change in material properties will give a lower prediction. Accommodating both phenomena gives results intermediate between those two cases. As discussed in Chapter 6, the consideration of residual stress with SSPT (by accommodating alterations in material properties and volumetric change) gave a residual stress prediction close to the experimental results compared to the prediction without SSPT.
9.3. Resetting plastic strain in the girth butt welding.

How SSPT affects residual-stress has been discussed thoroughly in Section 9.2 using the plate butt welding studied in Chapter 6. Another important aspect that needs to be discussed is how the melting phenomenon is considered in the APDL program developed in this thesis. The girth butt welding procedures discussed in Chapter 7 will be taken as a case study in this section.

As shown in Figure 7.39, when melting exists in a certain element the element loses its stiffness and plastic strain. Element stiffness at a high temperature is already low due to its low yield point. From the ANSYS manuals, when an element is deactivated (using the EKILL command) by default the stiffness is set close to zero. Deactivating the melting element was used to develop the APDL program in Chapters 6 and 7. But as mentioned above, since the material model has exhibited low stiffness at melting temperature the deactivating technique will not have any significant effect on the residual stress. Secondly, the melting phenomenon reduces plastic strain in the melting elements. Basically this can be accommodated by resetting the plastic strain of the melting element using the DESOL command. DESOL is an ANSYS post processing...
command and since melting can exist at any time step on certain elements, the program should toggle from the simulation block to the post processing block at each time step.

Results with plastic strain relieving when melting takes places have been presented in Chapter 7. To study the effect of this plastic strain relieving, in this section the FEM model without plastic strain relieving blocks is simulated and the result is compared to the previous model with plastic strain relieving. Figures 9.7 – 9.9 describe longitudinal, transversal and normal stress respectively for FEM model without plastic strain relieving in areas of 3 and 9 o’clock. The numerical residual stress then retrieved from ANSYS post process results at 3 and 9 o’clock paths and presented graphically in Figures 9.10 – 9.12. Figure 9.10 shows longitudinal stress distribution on 3 and 9 o’clock paths for a model with and without plastic strain resetting. The measurement result from both A and B samples are also plotted. Note that the mean values as in Chapter 7 (Figures 7.55, 7.56, 7.64 and 7.65) are not plotted but the longitudinal residual of both A and B specimens. Plotting both experiment data gives insights of the position of ANSYS results compared to the results obtained from the real “noisy” welding process.

It can be seen that without resetting the plastic strain the FEM prediction overestimates the residual stress as a result of the welding process. Indeed, the FEM longitudinal residual stress prediction on 9 o’clock path also overestimated the residual stress. The reason for the overestimation has been discussed in chapter 7 that is; a result of gap geometry which changed in the real welding process. This change is caused by distributed temperature when a previous pass in clock wise direction was applied. Moreover, the difference between the FEM model and measured longitudinal residual stress is not as high as predicted by model without accommodating the plastic strain reset and it may still acceptable considering the varied parameters in the real welding process.

Figures 9.11 and 9.12 describe transverse and normal residual stress on the 3 and 9 o’clock paths which the residual stress profiles on the pipe are shown in Figures 9.8 and 9.9 respectively. From those figures it can be concluded that as in longitudinal stress, the FEM model without plastic strain reset always overestimates the all component of residual stress (longitudinal, transversal and normal).
Figure 9.7. Longitudinal residual stress at 3 o’clock (right) and 9 o’clock (left) areas

Figure 9.8. Transversal residual stress at 3 o’clock (right) and 9 o’clock (left) areas

Figure 9.9. Normal residual stress at 3 o’clock (right) and 9 o’clock (left) areas
Figure 9.10. Longitudinal residual stress at 3 o’clock (right) and 9 o’clock (left) paths

Figure 9.11. Transversal residual stress at 3 o’clock (right) and 9 o’clock (left) paths

Figure 9.12. Normal residual stress at 3 o’clock (right) and 9 o’clock (left) paths
An explanation of the overestimated prediction can be obtained by following the mechanism of residual stress formation described in Section 2.3. When an element is heated, the melting elements which are laid very close to the weld line are typically in compression because its temperature is higher than the surrounding elements. The higher temperature means higher thermal expansion and because expansion is constrained by the bulk of cooler far field materials, those melting elements are in state of compression that caused elastic and plastic shrinkage. The elastic shrinkage will recover while the load (thermal load) is omitted whilst the plastic shrinkage is kept in the elements. The plastic shrinkage that exists, as a result of misfit, finally produces tensile residual stress when the welded materials are cooled to room temperature. By resetting the plastic strain in the melting element the final net tensile residual stress in the melted element will decrease, which is why if resetting the plastic strain is considered to be correct, without resetting the plastic strain the FEM predicts a higher tensile residual stress close to the centre of the weld joint where elements experience peak temperature equal to or exceeding the melting temperature. The correctness of mitigating the plastic strain when an element has melted was confirmed with predictions closer to the measurement of residual stresses.


In previous models when an element is at its melting temperature, it will lose its stiffness and plastic strain. As has been discussed in Chapter 5, deactivating an element by application of EKILL command caused the element to lose its stiffness and removing plastic strain can be performed by application of DESOL command. It was suggested that this phenomenon happens even at temperature below melting which is called as annealing temperature. Theoretically annealing temperature depends on the carbon content which is divided into two regimes, hypoeutectoid and hypereutectoid. In hypoeutectoid the annealing temperature is around 50°C above AC$_3$ temperature whilst 50°C above austenite – cementite line for hypereutectoid steels. Since the steel under this study is hypoeutectoid then the annealing temperature is 50°C above AC3 which is equal to 970°C. The Girth weld FEM model in Chapter 7 used this annealing temperature. To observe the effect of this lower annealing temperature, in this section annealing is considered to take place at melting temperature (1400°C). The result from
the model of this section then compared to the previous outcomes as shown in Figures 9.16 up to 9.18. By comparing both results the effect of a lower annealing temperature can be observed. Figures 9.13 – 9.15 show residual stress profile for areas close to the 3 and 9 o’clockpaths which are comparable to Figures 7.59, 7.61 and 7.63.

Figure 9.13 Longitudinal residual stress at 3 o’clock (right) and 9 o’clock (left) areas for a model with annealing temperature equal to 1400°C.

Figure 9.14. Transversal residual stress at 3 o’clock (right) and 9 o’clock (left) areas for a model a model with annealing temperature equal to 1400°C.

Figure 9.15. Normal residual stress at 3 o’clock (right) and 9 o’clock (left) areas for a model a model with annealing temperature equal to 1400°C.
Figure 9.16. Longitudinal residual stress at 3 o’clock (right) and 9 o’clock (left) paths for a model with varied annealing temperature

Figure 9.17. Transversal residual stress at 3 o’clock (right) and 9 o’clock (left) paths for a model with varied annealing temperature

Figure 9.18. Normal residual stress at 3 o’clock (right) and 9 o’clock (left) paths for a model with varied annealing temperature
From Figures 9.16 – 9.18 it can be concluded that the different residual stress prediction following those two assumptions of annealing temperature is clearly seen in the area close to the weld line. It is not surprising since the annealing takes place in this area, no matter what the annealing temperature is assumed (970°C or 1400°C). It should be noted that temperature slope is very high in the weld centre line and the covered area in transverse distance for element with temperature above 1400°C and 900°C will not significantly change. Indeed, a small difference can be seen in far field residual stress for varied annealing temperature but is just a consequence of maintaining forces balance in the observed area.

Transverse residual stress for 1400°C is almost equal to predicted residual stress with 970°C annealing temperature. The geometric constraint for transverse direction is different to other direction; in this direction the bulk solid low temperature which never experience annealing is laid whilst in the two other direction only material which experienced annealing is laid. While annealing takes place strain is revised by omitting plastic strain but in the transverse direction this strain correction is constrained by far field solid with low temperature. As has been mentioned before, the distance from the weld centre line in the transverse direction for element with peak temperature above 1400°C and 970°C will not be altered significantly due to the high temperature slope in this direction. This different condition of constraint finally may cause a different contribution of altered annealing temperature those results in a significant residual stress change in the longitudinal and normal direction but not in the transverse direction.

With the higher annealing temperature, residual stress should be higher since the residual stress is basically formed when a material is cooled down from this annealing to the room temperature. Higher annealing temperature means residual stress will be formed in the wider temperature span which in turn results in higher residual stress. But longitudinal residual stress distributions shown in Figure 9.16 exhibits reversed trend. It must be underlined that in the selected paths (3 and 9 o’clock surface paths) is laid layers 3 and 4 (pass 6 and 8 respectively), above this layer is layer 5 (pass 9 and 10 respectively for clock wise and anti clock wise down ward welding). To understand the lower longitudinal residual stress for 1400°C annealing temperature the temperature history of a certain node in those nodes must be observed (nodes 14541 and 27905) layers (layers 3 and 4) close to the selected paths. The typical temperature history is
shown in Figure 9.19. Node 14541 is laid at layer 3 whilst node 27905 at layer 4. When layer 3 is applied, node 14541 has a peak temperature equal to 2400°C which equal to applied uniform temperature load in the mixed mode of the heat source model. Node 27905 shows the same manner that when layer 4 is applied the peak temperature reaches 2400°C and below it in the other layers.

From Figures 9.19 it can be seen that, when layer 5 is applied the node under observation experienced temperature history with peak temperature equal to 1222°C and 1035°C. If annealing temperature is considered equal to 1400°C then when the 5th layer is applied no future annealing temperature is found instead when the 3rd or 4th layers are applied for node 14541 and node 27905 respectively. But if annealing temperature is considered equal to 970°C the second annealing temperature is exhibited that when the 5th is performed. From previous discussion if annealing phenomenon is not considered in the residual stress formation the residual stress is formed as a result of compressive plastic strain which is developed when the material in the centre line is heated while the expansion is constrained by cool far field bulk material. The developed plastic strain caused misfit which in turn is considered as a reason for developed tensile stress in the area close to the weld centre. All this mechanism has been discussed in chapter 2.3 when discussing the three bar elasto-plastic material model. When annealing takes place, residual stress is formed due to contraction of elements in the weld centre line when cooled down to room temperature which is constrained by far bulk cooler material. Since all nodes under observation when 5th layer is applied have peak
temperature above 970°C and below 1400°C which mechanism will be followed depends on the initial assumptions. When annealing temperature is considered equal to 970°C all those observed nodes develop residual stress when 5th layer is applied due to contraction which is constrained when those nodes cooled down to room temperature which in turn caused higher tensile residual stress result. On the other hand when annealing temperature is considered equal to 1400°C the developed residual stress produced by misfit as a result of plastic compressive stress when those nodes are heated which in turn caused lower tensile residual stress result.

9.5. Transformation plasticity.

When SSPT takes place not only volumetric and mechanical properties changes exist but also transformation plasticity that describes additional strain when SSPT takes place under instantaneous stress. A theoretical approach and description of how this transformation plasticity was included in FEM model has been discussed in section 7.2.4. In this section the contribution of the transformation plasticity to the developed residual stress by comparing the model to the model without transformation plasticity considerations will be discussed.

Figure 9.20. Longitudinal residual stress at 3 o’clock (right) and 9 o’clock (left) areas
The residual stress profiles in 3 and 9 o’clock areas, ignoring transformation plasticity, are presented in Figures 9.20 up to 9.22 for longitudinal, transversal and normal residual stress respectively which are comparable to Figures 7.59, 7.61 and 7.63 respectively. In Figure 9.23, the longitudinal residual stress without incorporating transformation plasticity is compared to the Chapter 7 model which included transformation plasticity phenomenon. Experimental results for both specimens (namely specimens A and B) are also plotted in Figure 9.23 to validate the FEM predictions. In the same way, Figures 9.24 and 9.25 show residual stress predictions as well as experimental results for transversal and normal residual stress respectively.

From these figures it can be understood that ignoring transformation plasticity cause the residual stress prediction to be “more tensile” especially in the area close to the weld line. It should be noted that basically the initial excitation in the mechanical analysis
stage of the welding FEM model is strain ($\varepsilon$). The strain is obtained from the elongation due to temperature load which is retrieved from FEM thermal analysis results. When the strain is in a proportional region the embedded stress equal to the strain times the slope of the stress-strain curve which equal to modulus Young $E$. While if the strain exceeds the yield strain the stress produced is equal to the yield stress plus the strain times the slope of the plastic lines. Since the stress-strain curves are modeled by multi linear isotropic lines there will be always slopes.

Figure 9.23. Longitudinal residual stress at 3 o’clock (right) and 9 o’clock (left) paths

Figure 9.24. Transversal residual stress at 3 o’clock (right) and 9 o’clock (left) paths

When transformation plasticity is modeled, a part of the strain is considered to be a transformation plasticity (or it can be said that apart of strain is due to the transformation plasticity strain) and the strain that produces stress is decreased. Although the FEM model of the girth weld is not as simple as discussed, but it can be qualitatively explain that the residual stress by involving transformation plasticity will have lower tensile stress results compare to the model that ignores the transformation
plasticity. If the model incorporating the transformation plasticity is considered to be correct, ignoring the transformation plasticity cause over prediction of the residual stress.

Figure 9.25. Normal residual stress at 3 o’clock (right) and 9 o’clock (left) paths

9.6. Aged martensite considerations.

As has been discussed in Chapter 7, aged martensite SSPT was considered in the girth weld model. Aged martensite is a phase which is formed at the cooling stage of a martensite when reheated above A1. Figure 9.26 shows comparison of a model involving aged martensite transformation with a model considers prime martensite only.

Figure 9.26. Residual stress at 9 o’clock (left) and 3 o’clock (right) for model with and without aged martensite SSPT.

From the figure it can be concluded that involving aged martensite lowers all final residual stresses. Considering the SSPT model of prime martensite and aged martensite presented in Figures 7.33 and 7.34, basically aged martensite represents a higher yield
stress material compared to prime martensite. How this higher yield stress produces lower final residual stress will be demonstrated below.

In the previous three bars model it has been demonstrated how residual stress developed as a result of thermal load. Figure 2.15 page 29 shows the stress when the middle bar is heated to 1100°C. It should be noted that in the model the yield stress of all bars are 150 MPa. It can be demonstrated using the three bars model that increasing the yield stress of middle bar will lower the final tensile stress in the mid bar. The yield stress of the middle bar in the next model is increased to 160 MPa. The stress-strain diagram for both model are shown in Figure 9.27. Figure 9.28 shows residual stress development for this increased yield stress of the middle bar.

Figure 9.27. Stress strain diagram for material model used in Figure 2.15 (left) and Figure 9.27 (right).

Figure 9.28. Three bars model with higher yield stress of middle bar; when the middle bar heated (left) and temperature load omitted (right)
From Figure 2.15 it can be seen that when the middle bar is heated the stress in the side bars is 75 Mpa and -150 for middle bar and when the temperature load is omitted the stresses in side bars and middle bar are -34.1 MPa and 68.2 MPa respectively. Figure 9.28 shows that when the middle bar is heated the stresses in side bars and middle bar are 80 MPa and -160 MPa respectively. When the temperature load is omitted (right side), the stress in side bars and middle bar are -33.9 MPa and 67.8 MPa respectively. This result is consistent with FEM model involving higher yield stress (Aged martensite).

How the lower residual stress is produced can be understood by evaluating not only stress in the 3 bars model but also the strain. Stress and strain for the model with yield stress of middle bar equal to 150 MPa are shown in Figure 9.29 whilst Figure 9.30 shows the result for 160 MPa. The first column of each graph which is green shaded describes the stress and the remaining three columns describe strain. All strains in the figures use mm/m units.

Figure 9.29. Stress and strain for yield stress of middle bar equal to 150 MPa when middle bar is heated (left) and when temperature load is omitted (right).

Figure 9.30. Stress and strain for yield stress of middle bar equal to 160 MPa when middle bar is heated (left) and when temperature load is omitted (right).
Indeed the higher yield stress in the middle bar (160 MPa) results in higher stress for those bars when the middle bar is heated (80.05 Mpa and -160.1 MPa) whilst for 150 MPa the stresses are lower (75.06 MPa and -150.12 MPa) but it does not mean the strains especially plastic strain are higher. The total strains to keep the force balance are 5.004 mm/m for side bars and -12.269 mm/m for middle bar when 150 MPa model is used. Whilst 5.336 mm/m and -11.667 mm/m for 160 MPa model. Since the elastic strain limited to 10 mm/m (or -10 mm/m) the plastic strains are -2.257 mm/m and -1.661 mm/m for 150 MPa and 160 MPa models respectively. The higher observed stress seems to result from the higher Young’s modulus of 160 MPa material. This lower plastic strain finally produced lower residual stress which is caused by misfit. This discussion may explain qualitatively why the model involving aged martensite in the multi-pass girth weld FEM model caused lower residual stress in the area close to the weld line. Since the results consider aged martensite SSPT close to the experiment results, this argument is considered to be valid.
References


PART VI

GENERAL

CONCLUSIONS
Chapter 10

General Conclusions

Following the discussions presented in Chapters 1 to 9 some conclusions and future work can be recommended. All the conclusions were obtained from the FEM results described in Chapters 2 - 9. All the FEM results were verified with the analytical results (Section 2.3 and Chapter 3) or validated with either secondary experiment results (Chapters 4, 5, and 6) or primary experimental results (Chapter 7). Chapter 8 evaluated the possibility to apply DC-LSND in multi pass butt girth weld joint of thick pipelines and Chapter 9 discussed effects of SSPT and melting phenomenon in more details.

10.1. Conclusions

1. Welding modeling for high strength steel must involve three coupled analyses: thermal, metallurgical, and mechanical analysis. Thermal and metallurgical analysis need to be fully coupled because they have a significant effect on each other whilst mechanical analysis can be done sequentially due to the weak effect of mechanical results on the thermal and metallurgical results.

2. The new heat source model that combined Goldak’s heat source model and uniform temperature load provided a good prediction of thermal and mechanical results.

3. The developed general database for parameters has accelerated computing time, and provided flexibility and inter-changeability between different analyses.

4. The birth and death technique enables realistic modeling of the growing weld bead. The main idea of the birth and death technique is to deactivate all the weld bead elements and then activate the proper weld bead elements by evaluating the relative position of those elements to the centre of the weld.

5. SSPT changes the prediction of residual stress especially in the area close to the weld line through thermal properties, mechanical properties, and volumetric change. The change in thermal properties had an insignificant effect whereas for
mechanical properties the elevated yield stress due to the formation of martensite and volumetric change from phase transformation had clear effects on the residual stress. The elevated yield stress developed a higher tensile residual stress whilst the SSPT volumetric alteration developed a compressive residual stress near the centre of the weld. When austenitic transformation takes place, the shrinkage due to SSPT caused increased tensile residual stress in the centre of the weld, however austenite SSPT was exhibited at high temperature where the yield stress of the material was low, and as a result, a relatively small tensile residual stress was formed. On the other hand, expansion due to a martensitic transformation which formed a residual compression stress near the weld line took place at a lower temperature and caused a relatively high residual compressive stresses to form.

6. Another important aspect concerning SSPT that should be incorporated in the FEM model is transformation plasticity. Ignoring transformation plasticity causes overprediction of the final residual stress since a part of strain which should be a portion of transformation plasticity is converted to stresses.

7. The maximum residual longitudinal stress for the girth weld obtained from the ANSYS prediction was 452 MPa for the 3 o’clock position and 552 MPa for the 9 o’clock position, whilst the measurements were (428 ± 30) MPa and (379 ± 30) MPa for the 3 o’clock and 9 o’clock paths respectively. These predicted and measured values were close to the yield stress of materials at room temperature (600 MPa and 470 MPa for base metal and filler metal respectively) this underlined the importance of considering residual stress when assessing the integrity of welded structures.

8. When SSPT takes place yield stress and “artificial coefficient of thermal expansion” are significantly changed. The artificial coefficient of thermal expansion was used to accommodate the volumetric change due to SSPT combined with the initial thermal expansion. From a case study in chapter 9 it can be concluded the effect of the volumetric change (64.86%) is more significant than yield stress change (25.57%).

9. When melting occurred in an element of the FEM model, it is important to remove its plastic strain otherwise FEM will overestimate the residual stress, especially close to the weld line. By resetting the plastic strain, the maximum
predicted residual stress at 3 o’clock was 452 MPa, 1.82% higher than the measured value, which is acceptable regarding some of the limitations mentioned in Section 7.5. Ignoring the resetting plastic strain when an element was melting gave a maximum predicted residual stress that was 194% higher than the measured value. At the 9 o’clock position, and considering plastic strain resetting, the maximum predicted residual stress of 552 MPa was 34.96% higher than the measured value (379.03±30 MPa). Note that the 34.96% is when compared to the mean value of the measured results, but if compared to the maximum results (sample A) it will be 20.26%. This is acceptable considering the imperfections in both sides (modeling and experiment work). However ignoring the resetting of plastic strain means that the maximum residual stress was 900 MPa, i.e. 137% higher than the measured value. The higher deviation at 9 o’clock (compared to the prediction at 3 o’clock) is understandable since the initial gap was changed further after welding at the opposite side had been carried-out due to thermal expansion. From those facts it can be considered that FEM without resetting plastic strain when melting takes place overestimated the residual stress.

10. The plastic strain and stiffness removal mentioned in point 9 above, exists even below melting temperature which called as annealing temperature.

11. A validated FEM model for girth weld was developed and the parametric study based on the validated FEM can be carried out. In this thesis the application of DC-LSND was modeled to study the possibilities of decreasing residual stress which in turn may increase the lifetime of the girth weld of pipelines.

12. From the girth weld with DC-LSND using the FEM parametric study, the residual stresses decreased when a CO₂ trailing heat sink was applied. The DC-LSND reduces the maximum tensile stress by 35.55%.

13. The DC-LSND reduces the residual stress by increasing the equilibrium temperature and decreasing the duration from peak temperature to the equilibrium temperature.
The contributions of this research can be summarised as follows:

1. A new analytical solution for the moving point heat source. This new solution for the transient state complements Rosenthal’s existing solution for the distribution of temperature of a moving point heat source at a steady state.
2. A new heat source model that is a combination of existing Goldak’s heat source model and uniform temperature load.
3. An FEM model that considers both prime martensite and aged martensite in multi-pass welding using a strength reduction factor (Kd) in the FEM model to represent the filler metal properties at elevated temperature.
4. A new database of parameters developed to interchange the parameters required between analysis (thermal, metallurgical and mechanical) and between blocks in certain analyses.
5. A validated FEM model for a girth-welded joint of X-70 pipes.
6. The predictions of residual stresses when DC-LSND is applied on the girth welding.

10.2. Future work

1. As shown in Chapters 6 and 7, this thesis only considered martensite SSPT which is the extreme condition. However, the discussion in this thesis pointed out that future development of FEM modeling should consider some further aspects that should be included in the modeling assumptions for specific materials. In real high strength steel, not only is martensite formed, so too is bainite depending on cooling rate. There are two kinds of bainite: upper bainite and lower bainite and the best way to include this bainite in the model is a mathematical model that describes the volumetric change as a result of SSPT such as Koistenen-Marburger equation for martensitic SSPT. The other aspect is the cooling rate that may be obtained by evaluating the temperature history of a certain element.
2. Combining the parameter data base with the other program that can optimise welding parameters. Since the data base in the general format (*.txt) the data should be able to be opened by another program.
3. Fully coupled TMM may also be a worthy of future work, but it needs a strong material science basis, especially in the coupled mechanical-metallurgical aspect.

4. The experiment work proved that improving infrastructure of the mechanized girth weld has practical benefits. These improvements cover two aspects: surface preparation and controlling the welding parameter. Controlling the welding parameters means improving control of automatic path and automatic welding parameters. Improving the automatic path can be achieved using surface sensors that can guide the welding torch on a correct path, and improving control of the welding parameter can be done with automatically adjustable welding parameter control. The control must depend on the position of the welding torch because different positions need different parameters. An expert learning system that can give optimised parameter for each position may be able to be developed. Overall it can be said that the application of single-sided girth weld is a challenging topic with a wide field of operating practice where many problems have not yet been solved.

5. Experimental work with DC-LSND on the girth weld should be carried out because from FEM study, the application of DC-LSND proved that it can decrease the residual stress in the girth weld.