IN-PLANE SHRINKAGE STRAINS AND THEIR EFFECTS ON WELDING DISTORTION IN THIN-WALL STRUCTURES

DISSERTATION

Presented in Partial Fulfillment of the Requirements for
The Degree Doctor of Philosophy in the Graduate School of The Ohio State University

By
Wentao Cheng, B.S., M.S.

* * * * *

The Ohio State University

2005

Dissertation Committee:

Professor Chon L. Tsai, Adviser
Professor Avraham Benatar
Professor David W. Dickinson

Approved by

Adviser
Welding Engineering
Graduate Program
ABSTRACT

It is difficult to predict welding distortion in large welded structures and even more difficult to achieve both fast and accurate distortion predictions in such structures. Distortion modeling of large welded structures is always associated with enormous computational cost in terms of computer memory requirement and CPU time. This research work was conducted to obtain better understanding and characterization of the plastic deformation that leads to in-plane shrinkage in thin-wall structures. It also aimed to develop a fast and accurate approach for predicting the welding distortion in large thin-wall structures. Some examples of this type of structures are vehicle underbody frame and motor compartment frame.

The engineering approach was developed based upon the principle that plastic deformation is the root source of welding distortion. A hypothesis was proposed regarding such development and it is stated as the following: welding distortion can be accurately predicted if the plastic deformation the same as or similar to that induced by welding is introduced into the finite element analysis (FEA) model. On one hand, the research was intended to validate the hypothesis; on the other hand, the hypothesis served as a guideline to the
research. Specifically, there were two important aspects of the research work as implied in the hypothesis: 1) characterizing the plastic deformation produced by welding, and 2) generating the plastic deformation equivalent to the actual one in the FEA model without resorting to detailed transient analysis.

The FEA was the tool used in this work to study welding plastic deformation. Before FE simulations were carried out for this purpose, a proper FEA model was formulated and verified with experiments so that it was capable of rendering accurate distortion computation as well as capturing all the data necessary to investigate the plastic deformation. The model simulated the physical phenomena that have significant influence on plastic deformation and distortion, such as temperature-dependent material properties, continuous filler metal deposition, and plastic strain relaxation. The analysis results including both thermal and mechanical results were validated against experimental measurements.

After being verified by the experimental results, the FEA model was utilized to perform the welding simulations of three types of simple joints including butt-welded plates, Tee joint, and plate with slot weld. These weld joints had the same structural and process characteristics as those of large thin-wall automotive body structures. The analysis results were extracted and analyzed to study the in-plane shrinkage and plastic behaviors. The plastic strain distributions were examined in combination with the peak temperatures experienced during welding and the material softening at elevated temperatures. The in-plane shrinkage was correlated to the distribution
characteristics of plastic strains, and furthermore the relationships between the plastic strains and their influencing factors were established. It was concluded from this study that the in-plane shrinkage plastic strains are determined by the peak temperature and material’s softening temperature range, and that Peak temperature is the thermal parameter that controls the plastic strains.

The engineering approach was developed based on the findings obtained from the study described above. The research issue of central importance in developing the engineering approach was how to introduce the plastic deformation equivalent to the actual one into the FEA model. Temperature load was used in this study and then the central issue became formulating the temperature load, namely its distribution including area of application and magnitude. The distribution was determined based upon the findings obtained in the study of in-plane shrinkage and plastic behaviors mentioned above. The engineering approach was established by applying the temperature load to the FEA model. The engineering approach was then used to predict the welding distortions of automotive body structures, including front rail assembly and engine cradle. The predicted distortions were compared with the measurements and excellent agreements existed between them. Successful development and application of the engineering approach validated the hypothesis about distortion prediction proposed in this research.
To Dongmei and Emma
ACKNOWLEDGEMENTS

I wish to express my sincere appreciation to Professor Chon L. Tsai for his guidance, encouragement and insight throughout the duration of this research. I would also express my gratitude to Professor Avraham Benatar and Professor David W. Dickinson for serving as dissertation committee members and providing positive suggestions and comments.

I am very grateful to Dr. Zhili Feng for his help on this research. Thanks are also extended to Dr. Robert Frutiger, Dr. Jun Huang, and Dr. Pei-Chung Wang for their assistance.

I wish to thank my parents, Guanghu Cheng and Runxiang He, for consistently encouraging me to complete my doctorate degree and reminding me it is a major milestone of my life.

Finally, I want to offer sincere appreciation to my wife, Dongmei, for her unshakable faith in me and her willingness to endure with me during this long but fruitful journey.
VITA

November 13, 1968....................... Born – Hunan, People’s Republic of China

1992................................. B.E., Welding Engineering, 
Tsinghua University, Beijing, China

1996................................. M.S., Welding Engineering, 
The Ohio State University

1993 – 1997......................... Graduate Research Associate, 
The Ohio State University

1997 – 2000......................... Research Engineer, Edison Welding 
Institute, Columbus, Ohio

2000 – 2002......................... CAE Engineer, Technical Engineering 
Consultants Inc., Detroit, Michigan

2002 – Present....................... Senior Research Engineer, Engineering 
Mechanics Corporation of Columbus, 
Columbus, Ohio

PUBLICATIONS

Research Publication


FIELDS OF STUDY

Major Field: Welding Engineering

   Computational Welding Mechanics, Weld Structural Design, Finite Element Analysis
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Abstract</td>
<td>ii</td>
</tr>
<tr>
<td>Dedication</td>
<td>v</td>
</tr>
<tr>
<td>Acknowledgements</td>
<td>vi</td>
</tr>
<tr>
<td>Vita</td>
<td>vii</td>
</tr>
<tr>
<td>List of Tables</td>
<td>xiii</td>
</tr>
<tr>
<td>List of Figures</td>
<td>xiv</td>
</tr>
<tr>
<td><strong>1</strong> INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>1.1 Background</td>
<td>1</td>
</tr>
<tr>
<td>1.2 Welding Distortion</td>
<td>3</td>
</tr>
<tr>
<td>1.3 Control of Distortion</td>
<td>7</td>
</tr>
<tr>
<td>1.4 Prediction of Welding Distortion</td>
<td>10</td>
</tr>
<tr>
<td>1.4.1 Empirical Method</td>
<td>10</td>
</tr>
<tr>
<td>1.4.2 Analytical Solution</td>
<td>11</td>
</tr>
<tr>
<td>1.4.3 Numerical Modeling</td>
<td>12</td>
</tr>
<tr>
<td>1.5 Dissertation Objectives</td>
<td>21</td>
</tr>
<tr>
<td>1.6 Dissertation Organization</td>
<td>22</td>
</tr>
<tr>
<td><strong>2</strong> WELDING DISTORTION AND PLASTIC DEFORMATION</td>
<td>27</td>
</tr>
<tr>
<td>2.1 General Description</td>
<td>27</td>
</tr>
<tr>
<td>2.2 Three-Bar Analogy</td>
<td>29</td>
</tr>
<tr>
<td>2.3 Analysis of Tee Joint</td>
<td>44</td>
</tr>
<tr>
<td>2.4 Theoretical Description of Inherent Plastic Strains</td>
<td>51</td>
</tr>
<tr>
<td><strong>3</strong> FINITE ELEMENT MODEL AND VERIFICATION</td>
<td>53</td>
</tr>
<tr>
<td>3.1 Finite Element Analysis Model Formulation</td>
<td>55</td>
</tr>
<tr>
<td>3.1.1 Consideration of Physics of Welding</td>
<td>55</td>
</tr>
<tr>
<td>3.1.2 Thermo-mechanical Analysis of Weld Distortion</td>
<td>61</td>
</tr>
<tr>
<td>3.2 Experimental Setup</td>
<td>64</td>
</tr>
<tr>
<td>3.3 Finite Element Meshes</td>
<td>66</td>
</tr>
</tbody>
</table>
3.4 Thermal Analysis and Experimental Verification ................................................. 75
  3.4.1 Mathematical Formulation of Heat Input and Heat Transfer Computation ................................................................. 75
  3.4.2 Thermophysical Properties ....................................................................... 81
  3.4.3 Finite Element Implementation and Experimental Verification .................. 84
3.5 Mechanical Analysis and Experimental Verification .......................................... 89
  3.5.1 Effects of High-Temperature Material Behavior ......................................... 91
      Plastic Strain Relaxation .............................................................................. 91
      Filler Metal Deposition ............................................................................... 96
  3.5.2 Material Properties .................................................................................. 101
  3.5.3 Finite Element Implementation .................................................................. 102
  3.5.4 Experimental Verification ....................................................................... 108
3.6 Summary ........................................................................................................ 119

4 CHARACTERIZATION OF IN-PLANE SHRINKAGE PLASTIC STRAINS .............. 121
  4.1 FE Model and Parametric Study Matrix of Butt-Welded Plates .......... 122
  4.2 In-Plane Shrinkage and Plastic Strains in the Butt-Welded Plates .... 132
      4.2.1 A Transient State of Temperature and Plastic Strains ....................... 132
      4.2.2 Final State of Plastic Strains and Other Variables ......................... 137
          Peak Temperatures .................................................................................. 137
          Transverse Shrinkage .............................................................................. 142
          Longitudinal Shrinkage .......................................................................... 154
          Plastic Strains Contours ........................................................................ 156
          Plastic Strain Distributions Across the Plate .......................................... 159
          Plastic Temperature Range ..................................................................... 164
          Transverse Plastic Strains in and near Weld ......................................... 166
      4.2.3 Evolution of Plastic Strains ................................................................. 169
          Plastic Strains in the Weld ...................................................................... 171
          Plastic Strains near the Fusion Boundary .............................................. 172
          Plastic Strains in the Base Metal ............................................................ 183
      4.2.4 Effects of Heat Input on In-Plane Shrinkage and Plastic Strains .. 186
  4.3 Plate with Slot Weld and Tee Joint ............................................................... 194
      4.3.1 Geometry and FE Mesh of Plate with Slot Weld ............................. 194
      4.3.2 Geometry and FE Mesh of Tee Joint ............................................... 197
      4.3.3 In-Plane Shrinkage and Plastic Strains in the Two Joints ............... 200
  4.4 Summary ..................................................................................................... 203

5 DEVELOPMENT OF ENGINEERING APPROACH ............................................ 205
  5.1 Detailed and Simplified Approaches to Distortion Prediction ....................... 206
  5.2 Formulation of the Simplified Approach Based on Welding Plastic Deformation ............................................................ 207
      5.2.1 Theoretical Basis and Principle .......................................................... 207
      5.2.2 Formulation of Equivalent Temperature Load .................................... 212
  5.3 Methodology ............................................................................................... 218
5.4 Case Study of Tee Joint ........................................................................ 220
5.5 Case Study of Plate with Slot Weld ..................................................... 226
5.6 Conclusions ........................................................................................ 229

6 APPLICATION OF ENGINEERING APPROACH TO AUTOMOTIVE BODY
STRUCTURES .......................................................................................... 234
6.1 Problem Definition ............................................................................ 235
6.2 Welding Distortion of Front Rail Assembly ....................................... 238
6.3 Welding Distortion of Engine Cradle ................................................ 247

7 CONCLUSIONS AND DISCUSSIONS ..................................................... 251

REFERENCES .......................................................................................... 256
LIST OF TABLES

Table 3.1: Characteristic parameters of the finite element meshes .................. 74
Table 3.2: Temperature-independent properties used in the analysis ............... 82
Table 3.3: Critical parameters of the double ellipsoidal heat source model realized through DFLUX subroutine ................................................................. 86
Table 4.1: Welding parameters used in the parametric study ......................... 131
Table 5.1: CPU time consumed by the analyses of Tee joint performed on HP J5000 workstation ....................................................................................... 226
Table 5.2: CPU time consumed by the analyses of Butt joint performed on HP J5000 workstation ....................................................................................... 228
Table 6.1: Welding parameters used in the engineering approach development, front rail assembly, and engine cradle ...................................................... 238
Table 6.2: Statistical data for measured and predicted displacements ............ 247
Table 6.3: Measured and predicted distortions of the critical dimensions ........ 248
## LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 1.1: Classification of welding distortion [2]</td>
<td>5</td>
</tr>
<tr>
<td>Figure 2.1: Three-bar analogy analysis model</td>
<td>30</td>
</tr>
<tr>
<td>Figure 2.2: Material properties of the steel used in three-bar analogy analysis. (a) Young’s Modulus and yield stress; (b) stress-strain curve at room temperature</td>
<td>31</td>
</tr>
<tr>
<td>Figure 2.3: Heating and cooling history of the middle bar in three-bar analysis</td>
<td>33</td>
</tr>
<tr>
<td>Figure 2.4: Plastic strain history of the middle bar in three-bar analysis</td>
<td>34</td>
</tr>
<tr>
<td>Figure 2.5: Case study results showing the effect of structural rigidity on distortion and plastics deformation. (a) Plastic strain history in the middle bar; (b) Shrinkage and final plastic strain in the middle bar.</td>
<td>37</td>
</tr>
<tr>
<td>Figure 2.6: Case study results showing the effect of heat input on distortion and plastics deformation. (a) Plastic strain history in the middle bar; (b) Shrinkage in the middle bar</td>
<td>41</td>
</tr>
<tr>
<td>Figure 2.7: Three-bar analogy analysis model with external clamping constraint</td>
<td>42</td>
</tr>
<tr>
<td>Figure 2.8: Case study results showing the effect of welding fixture on distortion and plastics deformation. (a) Plastic strain history in the middle bar; (b) Shrinkage in the middle bar</td>
<td>43</td>
</tr>
<tr>
<td>Figure 2.9: Simple Tee joint and its dimensions</td>
<td>46</td>
</tr>
<tr>
<td>Figure 2.10: Comparison of plastic strain fields from transient and simplified analyses. (a) Transient analysis, $\varepsilon_{\text{p max}}^p = 4.84E-3$, $\varepsilon_{\text{p min}}^p = -6.21E-2$; (b) Simplified analysis, $\varepsilon_{\text{p max}}^p = 3.85E-3$, $\varepsilon_{\text{p min}}^p = -6.82E-2$</td>
<td>48</td>
</tr>
<tr>
<td>Figure 2.11: Comparison of distorted base plates from transient and simplified analyses. (a) Transient analysis; (b) Simplified analysis</td>
<td>49</td>
</tr>
</tbody>
</table>
Figure 2.12: Comparison of displacement fields from transient and simplified analyses. (a) Transient analysis, $U_{\min} = -1.47E-1$ mm, $U_{\max} = 1.18E-1$ mm; (b) Simplified analysis, $U_{\min} = -1.30E-1$ mm, $U_{\max} = 1.03E-1$ mm.

Figure 3.1: Comprehensive numerical procedure of computing weld distortion

Figure 3.2: Finite element analysis procedure for thermal-mechanical analysis of weld distortion

Figure 3.3: Schematic of the Butt joint. (a) Geometry of the workpiece and coordinate system used in the finite element analysis; (b) Groove geometry of the Butt joint and welding sequence of the four passes. Dimensions are in mm.

Figure 3.4: Schematic of the Tee joint and experimental set-up including double torches, positioning scheme, temperature and displacement measurement. (a) Geometry of the joint and coordinate system used in the finite element analysis; (b) Fillet weld size (*TC stands for Thermo-Couple). Dimensions are in mm.

Figure 3.5: Location of displacement and temperature measurements. (a) Location of LVDT's; (b) Location of thermo-couples (showing half model). Dimensions are in mm.

Figure 3.6: Finite element mesh of the Butt joint. (a) Overall mesh; (b) Mesh of the groove including the four weld passes.

Figure 3.7: Finite element mesh of the Tee joint. (a) Overall mesh; (b) Mesh of the weld and its surrounding area.

Figure 3.8: Goldak's double ellipsoid heat source model [120]

Figure 3.9: Temperature-dependent thermophysical properties used in the thermal analysis. (a) Thermal conductivity; (b) Specific heat.

Figure 3.10: Bead shape of the DH36 Tee joint. (a) Experiment; (b) FEA.

Figure 3.11: Comparison of thermal histories in the Tee joint between experimental and finite element results. The experimental data are labeled as “thermocouple X” and the predicted are designated as “FEM-TCX”, where X represents the ID number of a thermocouple.

Figure 3.12: A bar with both ends fixed between two rigid walls

Figure 3.13: Temperature-dependent stress/strain curve.
Figure 3.14: Heating and cooling history applied to the bar.................................95

Figure 3.15: Plastic strain histories in the bar..........................................................96

Figure 3.16: Computed in-plane distortion in a Butt joint. (a) Filler metal deposition not simulated in the analysis; (b) Filler metal deposition simulated in the analysis. ..............................................................100

Figure 3.17: Temperature-dependent elastic modulus and Poisson's ratio. (a) Steel-A; (b) Steel-B. .................................................................103

Figure 3.18: Temperature-dependent yield stress and linear thermal expansion. (a) Steel-A; (b) Steel-B. ...............................................................104

Figure 3.19: Stress-strain curves at different temperatures. (a) Steel-A; (b) Steel-B..................................................................................105

Figure 3.20: Distorted Butt joint after pass 1. The distortion is magnified by 100. .........................................................................................110

Figure 3.21: Location of the dowel pins for measuring transverse shrinkage. $A_1$, $A_2$, $A_3$, and $A_4$ are the embedded dowel pins on the top side of the plate; $B_1$, $B_2$, $B_3$, and $B_4$ are the embedded dowel pins on the bottom side of the plate. Dimensions are in mm.........................................................111

Figure 3.22: Comparison of cumulative average in-plane shrinkage for 4 passes FE analysis and experiment .........................................................112

Figure 3.23: Distorted shape of the middle section of the Tee joint. The distortion is magnified by 60.................................................................115

Figure 3.24: Comparison of predicted and measured out-of-plane displacements at LVDT1. The experimental data were acquired through LVDT1 in two welding trials represented by Specimens A and B.................................................................116

Figure 3.25: Comparison of predicted and measured out-of-plane displacements at LVDT2. The experimental data were acquired through LVDT2 in two welding trials represented by Specimens A and B.................................................................117

Figure 3.26: Comparison of predicted and measured out-of-plane displacements at LVDT3. The experimental data were acquired through LVDT3 in two welding trials represented by Specimens A and B.................................................................118
Figure 3.27: Comparison of the final out-of-plane distortion magnitudes from the FE analysis and experiments ........................................................ 119

Figure 4.1: One pass square-butt joint on 2-mm thick plates. (a) Joint dimensions; (b) Weld bead configuration. ............................................. 124

Figure 4.2: General features of the 3D finite element mesh for numerical simulation. (a) Overall mesh; (b) Enlarged view of the mesh near the fusion zone and heat affected zone. The weld starts at the right end of the plate. Only half of the joint is modeled due to symmetry about the weld centerline. .......................................................... 128

Figure 4.3: Predicted temperature contours surrounding welding arc at 24.75 seconds after arc initiation. Temperature is in Kelvin. The GMAW parameters are 24-V arc voltage, 140-A welding current, and 8.47 mm/s travel speed. Note that temperature is uniform through the thickness. ............................................................................. 129

Figure 4.4: Boundary conditions in the mechanical analysis ....................... 131

Figure 4.5: Evaluated cross sections in the plate. (a) For assessing a transient state of temperature and plastic strains; (b) For the final state of the variables.......................................................... 134

Figure 4.6: Temperature distributions at different cross sections during welding. The arc is 209.55 mm away from the weld start ......................... 135

Figure 4.7: Plastic strain distributions at different cross sections during welding. The arc is 209.55 mm away from the weld start. (a) Transverse plastic strain; (b) Longitudinal plastic strain. ......................... 136

Figure 4.8: Peak temperature isotherms in the plate. (a) Overall distribution; (b) Distribution near weld ......................................................... 139

Figure 4.9: Peak temperature distributions at different cross sections .......... 140

Figure 4.10: Predicted distortions in the plate. (a) Filler metal deposition simulated in the analysis; (b) Filler metal deposition not simulated in the analysis. ...................................................................... 141

Figure 4.11: Transverse displacement distributions at different cross sections ...................................................................................................... 143

Figure 4.12: Effective shrinkage and rigidity in the transverse direction. (a) Effective rigidity area; (b) Effective shrinkage area; (c) Effective
shrinkage area near weld start at instant \( t \); (d) Effective shrinkage area near weld start at instant \( t + \Delta t \).

Figure 4.13: Variations of the effective shrinkage and effective rigidity along the weld.

Figure 4.14: Transverse displacement distributions at different cross sections in the non-linear shrinkage zone.

Figure 4.15: Longitudinal displacement distributions at different cross sections.

Figure 4.16: Plastic strain contours in the plate. (a) Transverse plastic strains; (b) Longitudinal plastic strains.

Figure 4.17: Plastic strain distributions at different cross sections. (a) Transverse plastic strains; (b) Longitudinal plastic strains.

Figure 4.18: Plastic strain and peak temperature distributions along the direction transverse to weld. (a) Transverse plastic strains and peak temperatures; (b) Longitudinal plastic strains and peak temperatures.

Figure 4.19: Variations of elastic modulus and yield stress with temperature.

Figure 4.20: Plastic strain and peak temperature distributions in and near weld.

Figure 4.21: Plastic strain and temperature histories of different locations at the middle cross section. (a) \( x = 0 \) mm; (b) \( x = 1.98 \) mm; (c) \( x = 2.38 \) mm; (d) \( x = 3.93 \) mm; (e) \( x = 5.34 \) mm; (f) \( x = 11.29 \) mm. \( x \) is the distance to weld centerline.

Figure 4.22: Effects of heat input on peak temperature and transverse displacement. (a) Peak temperature distributions at the middle cross section; (b) Transverse displacement distributions at the middle cross section.

Figure 4.23: Effect of heat input on transverse shrinkage. (a) Shrinkage magnitude along the plate edge; (b) Minimum, maximum, and average shrinkage magnitude.

Figure 4.24: Effect of heat input on transverse plastic strains.

Figure 4.25: Effects of heat input on various critical zone widths.
Figure 4.26: Correlation of transverse shrinkage and plastic strain distribution characteristics. (a) Plastic zone width; (b) Width of compressive transverse plastic strains................................................................. 193

Figure 4.27: Geometry and dimensions of the plate with slot weld.............. 195

Figure 4.28: FE mesh of the plate with slot weld. (a) Overall mesh; (b) Mesh in the weld and its surrounding areas........................................................... 196

Figure 4.29: FE mesh of the Tee joint. (a) Overall mesh (top view); (b) Mesh in the weld and its surrounding areas........................................................... 199

Figure 4.30: Overlay plot of the transverse plastic strain field and peak temperature field in the plate with slot weld................................. 201

Figure 4.31: Overlay plot of the transverse plastic strain field and peak temperature field in the base plate of Tee joint......................... 202

Figure 5.1: Transverse plastic strain and peak temperature distributions at the middle cross section in the 175-A case. .............................................. 215

Figure 5.2: Distribution of the equivalent temperature load ...................... 216

Figure 5.3: Loading and unloading of the temperature load in the simplified analysis ........................................................................................................... 217

Figure 5.4: Comparison of the transverse plastic strain fields from transient and simplified analyses. (a) Transient analysis, $E_p^{\text{min}} = -6.21\times10^{-2}$, $E_p^{\text{max}} = 4.84\times10^{-3}$; (b) Simplified analysis, $E_p^{\text{min}} = -3.54\times10^{-2}$, $E_p^{\text{max}} = 7.68\times10^{-3}$.
................................................................................................................... 222

Figure 5.5: Comparison of the transverse displacement fields from transient and simplified analyses. (a) Transient analysis, $U_{\text{min}} = -1.47\times10^{-1}$, $U_{\text{max}} = 1.18\times10^{-1}$; (b) Simplified analysis, $U_{\text{min}} = -6.97\times10^{-2}$, $U_{\text{max}} = 5.02\times10^{-2}$. 223

Figure 5.6: Comparison of the distorted base plates from transient and simplified analyses. (a) Transient analysis; (b) Simplified analysis.
................................................................................................................... 224

Figure 5.7: Comparison of the residual (Von Mises) stress fields from transient and simplified analyses. (a) Transient analysis; (b) Simplified analysis. ........................................................................ 225

Figure 5.8: Transverse plastic strain fields from transient and simplified analyses. (a) Transient analysis; (b) Simplified analysis. .......................... 230
Figure 5.9: Comparison of the transverse displacement fields from transient and simplified analyses. (a) Transient analysis; (b) Simplified analysis. ................................................................. 231

Figure 5.10: Comparison of the deformed plate shapes from transient and simplified analyses. (a) Transient analysis; (b) Simplified analysis. ................................................................. 232

Figure 5.11: Comparison of the longitudinal stress fields from transient and simplified analyses. (a) Transient analysis; (b) Simplified analysis 233

Figure 6.1: Front rail assembly. (a) A view from top; (b) A view from bottom. 242

Figure 6.2: Distortion measurement locations in front rail assembly.............. 243

Figure 6.3: Finite element mesh of front rail assembly ............................... 244

Figure 6.4: Distorted front rail assembly in x-y (1-2) plane after welding and fixture release. Left: After welding but before fixture release; Right: After fixture release. ......................................................... 244

Figure 6.5: Distorted front rail assembly in x-z (1-3) plane after welding and fixture release. Left: After welding but before fixture release; Right: After fixture release. ......................................................... 245

Figure 6.6: Measured and predicted displacement in front rail assembly in different directions. (a) x-axis direction; (b) y-axis direction; (c) z-axis direction......................................................... 245

Figure 6.7: Finite element mesh of engine cradle ........................................ 249

Figure 6.8: Distorted shape of engine cradle ............................................. 250
CHAPTER 1

INTRODUCTION

1.1 Background

It is difficult to predict welding distortion in large welded structures and even more difficult to achieve both fast and accurate distortion predictions in such structures. Distortion modeling of large welded structures is always associated with enormous computational cost in terms of computer memory requirement and CPU time. This Ph.D. dissertation work is conducted to develop a fast and accurate approach for predicting the welding distortion in large thin-wall structures, such as automotive body structures. Some examples of this type of structures are vehicle underbody frame and motor compartment frame. These welded structures have the following structural and process characteristics:

- Steel structures made of low-carbon or high-strength low-alloy (HSLA) steels,
- Thin-wall structures with thickness of 1 to 2 mm,
- Gas Metal Arc Welding (GMAW) processes with high speed,
- Short welds with length up to 50 mm, and
- Full penetration in GMA welds.
There is usually a large number of GMA welds in these structures. For example, there are more than three hundreds welds in a typical car underbody frame and the total weld length can reach eight meters. Even though in-plane shrinkage is the primary distortion mode of these short and full-penetration GMA welds, the in-plane shrinkage induces global bending distortion in these structures. For example, auto space frames are manufactured with many connections that are made of weld joints. Globally, the centroid of a connection is different from that of the frame; locally, the centroid of an individual weld joint is also different from that of the connection. The differences in these centroids are the very reason why global bending is frequently observed in the automotive body structures. Excessive amount of welding distortion in the body structures not only hinders the vehicle assembly process but also reduces the vehicle reliability and durability. In order to control welding distortion in the structures, it is necessary to optimize the structural design and welding process. The design and process optimization demands the capability of fast and accurate distortion prediction.

The theoretical basis of this work is that the in-plane shrinkage especially transverse shrinkage is the root source of the welding distortion in the large thin-wall structures with the structural and process characteristics described above. This research indeed strives to prove the above statement. The overall weld distortion can be accurately predicted as long as the in-plane shrinkage can be accurately calculated in the individual weld joints in these structures.
1.2 Welding Distortion

One of the most troublesome problems a welding engineer usually encounters is that of welding distortion. The non-uniform heating and cooling cycle, which occurs in the weld and the adjacent base metal, is a distinctive feature of welding heat transfer. It produces complex strains during welding, including thermal strains and most importantly plastic strains that constitute the source of welding distortion. Although such description may be too simplistic to cover the complicated thermal and mechanical activities that occur in the structure during the welding process, it can serve as a concise and indeed an accurate statement mirroring the nature of thermomechanics of welding. A brief recital of development of distortion is given in the following context.

During the heating cycle, localized heating induces non-uniform expansion, which is concentrated in the part in and near the welding pass and is constrained by the surrounding parts. During the cooling cycle, shrinkage of the heated part occurs when heating has stopped. Plastic deformation takes place due to the unevenness of the temperature fields, the restraining effects of the structure, and variations in the material properties that occur during the heating and cooling cycle. Plastic deformation remains after welding is completed. Dimensional changes evolve through the welding process and they are manifested in forms of shrinkage, bending, rotation, and buckling after the process is over. These dimensional changes are collectively known as welding distortion [1].
According to their appearance, welding distortions can be classified into the following categories: transverse shrinkage, longitudinal shrinkage, angular distortion, longitudinal bending, in-plane rotation and buckling [2], as illustrated in Transverse shrinkage, which is the in-plane distortion transverse to weld, plays significant role in the global distortion of welded structures, such as large trusses and sheet metal space frames. Longitudinal shrinkage and longitudinal bending can be significant in fabricating a long beam such as the T-beams and I-beams, which are commonly used in shipbuilding and construction and manufactured by fillet welding a web to a flange or flanges. It is a commonly accepted concept that buckling distortion is inevitable in a thin-walled structure where fusion welding is applied. Buckling distortion is not rare in ship hull structures and upper deck structures. There are numerous industrial applications bearing distortion problems in welded products. To enumerate a few, the following welded structures come up:

- Automotive structures such as truck underbody frames and car engine cradles;
- Construction equipment, for example, cranes and bulldozers;
- Oil and gas pipeline components;
- Marine vessels such as hull and upper deck structures in shipbuilding, hovercraft and surface effect craft fabrication;
- Bridges and buildings;
- Pressure vessels;
- Nuclear containment apparatus.
A large number of differing and interacting factors affect the type and extent of welding distortion. The magnitude and distribution of distortion in a welded structure are dependent on the parameters pertaining to welding process and joint design utilized when applying the welding procedure. These parameters fall into four categories. The first category contains geometric parameters, such as dimensions of the structure. The second category comprises material parameters, for example, temperature dependent thermophysical and mechanical properties, temperature range of material softening and plastic temperature range. The third category can be defined as those associated with welding parameters such as heat input, welding
sequence and application of preheating and post heating. The fourth category
includes the parameters indicating the degree of restraint against thermal
expansion and contraction, for instance, welding fixture and the type and size of
welded joint.

In addition to the above-mentioned influential factors, initial imperfections
in welded structures also contribute to welding distortion. Initial imperfections,
which usually exist in form of initial stress, initial distortion and joint mismatch,
are caused by the manufacturing processes prior to welding such as stamping,
hydroforming, casting and extrusion. The reason why all these factors affect
welding distortion is that they determine the formation and final state of plastic
deformation, which in turn determine the distortion in a welded structure.

Welding distortion is a major concern of the welding community. Distortions in weldments caused by conventional fusion welding processes
have presented welding engineers with problems ever since fusion welding was
introduced. Welding distortion causes complex consequences, most of which
are detrimental during fabrication and service. From the point of view of
structural integrity, welding distortion acts as initial imperfection of welded
components. Performance of welded structures in service can be highly
dependent on distortion. For example, it has certain influence on the buckling
behavior of a panel structure and often reduces the ultimate loading capacity of
the structure [3]. From the perspective of rationality of manufacturing
technology, the consequences of welding distortions are out-of-tolerance
geometry and variable quality of products. Preventive and corrective actions
have to be taken in order to avoid the degraded dimensional quality induced by distortion. These actions are time-consuming and associated with significant costs. In summary, welding distortion has imposed limitations on the design of welded structures and has restricted the application of welding techniques in metal fabrication. It has been long recognized that effective control of distortion is necessary to eliminate or reduce distortion.

1.3 Control of Distortion

Control of distortion ensures dimensionally consistent fabrication of metal constructions and provides means to meet more strict dimensional requirements, either for better structural integrity or for improved product performance. Control of distortion is often governed by codes, specifications, or regulations based on rational assessments of dimensional quality, safety and economics. Distortion control techniques can be categorized into two groups: distortion correction methods and design and process optimization techniques. Of the two the former is more conventional than the latter which heavily relies on computer aided engineering (CAE).

Distortion correction methods are adopted to eliminate or reduce welding distortion before and/or after welding. According to the timing of their application, distortion correction methods can be further classified as prior welding treatment and post-weld treatment. Prior welding treatment includes various methods applied before welding such as elastic pre-straining and
elastic-plastic pre-straining, whereby the welding distortions are compensated by the counter-deformations formed in the structural elements prior to welding. Post-weld treatment incorporates the measures taken after welding. For example, the distortions in welded panels of a ship hull are annihilated using flame straightening or other special flattening processes. These methods, applied either before or after welding, are arranged as special operations in the manufacturing procedure. Both need special installations, resulting in increasing cost and variable quality of welded structures. Additionally, owing to their complexity, reduced efficiency and reduced flexibility in practical execution, applications of distortion correction methods are limited.

Manufacture of low distortion or even distortion-free structures can be achieved through design and process optimization, which leads to better control of distortion than distortion correction methods. The optimization ensures that the distortion is minimized by choosing the optimum design and process measures prior to welding. The optimization enables design and manufacturing engineers to use in-process distortion control techniques. In-process control refers to the measures applied during welding for controlling distortions, for example, using the thermal absorbing effect of water mist or other volatile materials sprayed onto the weldments to intensively reduce the temperature in the side zones adjacent to welds and the welding distortions [4]. Another example is described in Reference 5 as low stress non-distortion (LSND) welding, in which an active in-process control of distortion during welding is realized through a stretching effect provided by a preset temperature
distribution. Other in-process techniques include synchronous deferential heating and commonly used techniques such as sequence optimization and fixture usage [6]. In-process control that is aided by design and process optimization allows the fabricators to reduce or eliminate distortions during welding, without having to undertake costly correction operations. It is the most effective way of reducing distortion because it controls the formation of plastic strains produced in regions near the weld. The design and process optimization before welding is implemented is the prerequisite of successful in-process control of distortion. Nowadays, the design and process optimization for welded structures almost solely depends on computer-aided numerical analyses, especially those covering prediction of welding distortion.

The capability of predicting distortion provides the possibility of reducing distortion through optimizing both weld design and process. Such capability can be specifically directed at that of predicting how the weldment being studied deforms and how to perform proper controls to change the distortion being considered. Prediction of distortion can be used to test various part designs and welding procedures mathematically without physical builds, in other words, a diverse number of factors influencing distortion can be properly taken into account by calculating their effects on distortion. Under a given set of conditions, prediction of distortion allows one to choose those factors yielding a minimum amount of distortion in the final structures. Prediction of distortion will lead to better understanding of the distortion mechanisms and better control of welding distortion.
1.4 Prediction of Welding Distortion

As the name implies, prediction of welding means the quantification or math-based evaluation of the level of distortion that can be expected after welding. There have been many investigations on this subject. Numerous predictive models available can be classified into three categories: Empirical Method, Analytical Solution and Numerical Modeling.

1.4.1 Empirical Method

Much knowledge about welding distortion was obtained through experiments in the early years due to lack of alternative means such as fast computers. A large amount of empirical information was gathered on various kinds of welding distortion encountered during fabrication of welded structures. In fact, various empirical formulae have been developed during the past 70 years. Most formulae are found in the literature published before 1980 [4,7,8,9,10,11,12,13,14,15,16,17,18,19]. Very few recent researches on empirical method have been reported. The most recent work that can be found in open literature was conducted by Rybicki et al on an automotive structure [20]. They proposed empirical formulae to predict the transverse distortion of square box beams. In their formula, the distortion is the function of the welding heat input and the section properties of the beam.

In general, empirical methods are based on empirical information. They need extensive experimental results to validate the suggested prediction formulae. None of the published empirical formulas are universally applicable.
This is not unreasonable since welding distortion depends on too many factors to account for all of them within a simple empirical formula. Most of the empirical methods are applicable to a very specific welding situation and not applicable to complex structures.

1.4.2 Analytical Solution

Most of the analytical solutions have been formulated to compute the heat transfer of welding [21,22,23,24,25,26,27,28,29,30,31,32,33,34,35,36]. Among these solutions, Rosenthal's solution is the most important one, which has been the foundation of many analytical and numerical analyses of distortion during the past 50 years. Rosenthal's solution first time mathematically described the heat distribution of a moving heat source [21,27,28]. Okerblom introduced plastic deformation into his distortion calculation that was calculus and Strength-of-Materials type analysis [26]. Based on a theory similar to the bending-beam theory and the concept of shrinkage force, Sasayama et al [11] developed an analytical solution to determine the longitudinal bending distortion in fillet welded T-beams and I-beams. Moshaiov and Song formulated a solution to compute the deformation of a ring stiffener under welding [37].

The only advantage of analytical solutions to distortion is that all the relations can be expressed explicitly by mathematical formulation. However, the limitation of their use is considerable and mainly lies in their inability to handle complex geometry and temperature-dependent material behavior. Analytical solutions can only handle very simple geometry and cannot be
applied to practical industrial structures. The complexity of the problem of welding distortion in industrial structures makes the application of complete analytical solutions almost impossible. Analytical solutions are usually based on the assumption that none of the material constants change with temperature, which is far from truth. Difficulty in analytical method increases when the material properties change from constant to nonlinear with temperature. In most cases, it is very difficult to obtain analytical solutions. In most studies, close form solutions were given only for the cases of constant material properties. If more accurate results, especially the information on high-temperature behavior, are required, numerical modeling such as finite element method must be adopted to give the necessary information.

1.4.3 Numerical Modeling

Finite Element Analysis (FEA) is the most powerful numerical modeling technique. FEA has become one of the most popular engineering tools in recent decades due to its ability to analyze a wide variety of physical problems and advancement in computer technology. Application of FEA to the inelastic analysis of structures and continua has received considerable attention. To a large extent, this effort has been motivated by the need to estimate the quality, reliability and durability and predict the material response under extreme conditions of thermal and mechanical loading, which are frequently observed in welded structures.
FEA has been increasingly used in the investigation of thermomechanical phenomena occurred in welding during the past 30 years. Indeed, compared with empirical and analytical methods, FEA has made it possible to gain further and better understanding of the thermomechanics of the welding process. Advanced finite element techniques are under development to achieve simulating reality in welding. FEA has proved to be an efficient tool for simulating welding and subsequent loading of structures [38].

In comparison to experimental methods, numerical modeling methods including FEA are flexible and inexpensive. Numerical modeling can be used to conduct numerical experiments on computer without the use of expensive and arduous laboratory experiments. Besides furnishing quantitative assessment of various variables, for instance, temperature, stress, strain and displacement, numerical modeling can be applied to a variety of different welding processes and different trims of the same process. It can also provide information that could not otherwise be obtained from laboratory experiments. For example, it is extremely difficult, if not impossible to experimentally measure and record the evolution of plastic strains during welding. But it is not difficult to calculate and export plastic strains in a computer-simulated experiment through numerical simulation of welding.

One of the attractive features of numerical modeling is its ability to adapt researcher's interests, for example, different modeling techniques can be chosen to investigate different technical issues. It is, however, not without disadvantages. Meaningful and accurate numerical modeling has to be based
on thorough comprehension of underpinning physics reflecting the nature of processes. A colossal obstacle to numerical modeling of welding is lack of understanding of the physics behind welding, which makes some welding problems very difficult to be analyzed numerically.

A major advantage for numerical modeling is that a numerical answer can be obtained even when a problem has no analytical solution. However, it is important to realize that a numerical solution is always numerical and an approximation. Analytical methods usually give a result in terms of mathematical functions that can then be evaluated for specific instances, for example, explicit expression of the relationship between distortion and heat input. This is not the case for numerical modeling. Nonetheless, numerical results can be further analyzed to obtain trend in data and show the behavior of the solution.

The most important part of this dissertation work is to analyze FEA results to gain the insight of distortion behavior such as plastic deformation. This work is not intended to pursue and perfect numerical modeling techniques for welding distortion problems. Although accurate numerical modeling is necessary to generate better understanding the problems, it is only a tool to accomplish the objective of this dissertation.
Historical Review and Research Issues in Numerical Modeling of Welding Distortion

In the late 1960's and early 1970's, most of the research work on the structural behavior during welding was concentrated on experiments. In those years, the effort was made to develop computer programs for the analysis of heat flow in weldments. Before 1970's, there were only few series of solutions and classical finite difference solutions for the nonlinear analysis of transient welding heat transfer. And they were not enough to be used in engineering practice. Makhnenko et al reported on some numerical analysis and experimental measurements of distortion and residual stresses in welded pipes [39]. When digital computers became widely used, and the FEA method became highly developed in 1970's, research in this area shifted to the numerical simulation of the welding process using FEA, which increased our understanding of the process. The trend is reflected in the literature [40,41,42,43,44,45,46,47,48] which are all pertaining to two-dimensional (2D) FEA. All these papers reported the use of 2D finite element (FE) models for analyzing welding processes in plates, pipes and other simple structures. The results of thermal analyses were in fairly good agreement with experiments, and stress/strain results agreed qualitatively with experiments.

In the 1980's, most of the analyses were still confined to two-dimensions [49,50,51,52,53,54,55,56,57,58,59]. 2D analysis also has many applications in other engineering fields. For example, it is widely used for the analysis of temperature distribution and thermomechanical behavior in steel bridges
[60,61,62]. In a 2D analysis, the structure is considered satisfying plane strain or axisymmetric condition. Computation domains lie in sections normal to the weld direction for weldments under quasi-stationary thermal conditions.

Because of its simplicity and facility for modeling, the plane strain boundary condition along the longitudinal direction in weldment has been widely used for analyzing the thermal and residual stresses and distortions of welding processes. However, the model using the plane strain boundary condition cannot satisfactorily describe the thermomechanical behavior in real situations, where a weldment is unconstrained in the welding direction, or where the restraint caused by run-off tabs, tack welds, fixturing, and the solidified portions of the weld is expected to significantly affect the development of distortion. In the 2D axisymmetric analysis, the loading conditions or heating of the weld pass are considered axisymmetric. The assumption implies that the travel speed of the heat source is so fast that the nonsymmetrical deformation might be negligible. Such an assumption of axisymmetry may provide reasonably good results regarding the overall deformation. However, the predicted local deformations are different from measured ones since the actual welding process cannot take place all at once across the whole circumference.

2D FEA may be inadequate for analyzing welding distortion. Three-dimensional (3D) FEA has to be conducted to calculate the distortion of certain welded structures, especially large and complex structures. Full 3D FE models are required both since complex structures normally involve elastic and plastic responses in all three dimensions and since the action of welding generates
out-of-plane distortions and stresses. FE simulations have been performed using 2D models for more than thirty years. Fully 3D simulations of welding have been conducted for more than twenty years [63,64,65,66,67,68,69,70,71,72,73,74,75,76,77]. The issue of 2D versus 3D analysis will be revisited and discussed in detail in Chapter III.

There are two major schools of FEA developed and applied to welding distortion: detailed transient approach [2,55,57,67,72,73,74,75,78,79,80,81,82,83,84,85,86,87] and simplified approach [88,89,90,91,92,93,94,95,96]. A detailed approach is usually a straightforward transient analysis of moving heat source. A simplified approach is typically a static analysis simulating weld shrinkage.

Transient analysis using moving source model can be carried out with many known welding heat sources, e.g. gas flames, electric arcs, plasma arcs, high energy density beams such as laser beam and electron beam. There exist two types of transient analysis: coupled and uncoupled thermomechanical analysis. They differ in the way of treating the thermal and mechanical processes of welding. Heat transfer and mechanical behavior are simultaneously computed in the coupled analysis. The mechanical field is coupled to the temperature field through the temperature dependent constitutive properties and the thermal strain. The uncoupled analysis is formulated upon the consideration that the temperature field and mechanical behavior of a welded structure are generally uncoupled.
The welding process is one of the most nonlinear phenomena in structural mechanics. Not only the mechanical properties change nonlinearly with temperature, but also the thermal behavior itself. However, the thermal and mechanical processes can be separated in the analysis in the following sense. The stress and strain fields do not affect the thermal activities [97,98,99], for example, heat generated due to the plastic deformation in the welding process is negligible in comparison with the heat from the arc [67]. The uncoupled thermomechanical analysis is carried in two parts: thermal analysis and mechanical analysis. The first part performs a transient heat transfer analysis and computes the thermal history of the entire weldment, which will be used as part of the input information for the second part. The second part uses the results of the first part and performs a transient thermo-elasto-plastic analysis in order to compute the distortions, strains and stresses in the weld.

The coupled analysis is more difficult to implement than the uncoupled one. The advantage of uncoupling the thermal and mechanical analyses is to reduce the computational cost. They generally have different convergence rates and sometimes the difference can be of several orders of magnitude. In the uncoupled calculations where the convergence can be satisfied separately, larger time increment can be used for the one with faster convergent rate to reduce its computation time. In a coupled approach, however, both the thermal and mechanical analyses have to use the same small time increment determined by the slower convergence rate. The computation for long simulation times can
become prohibitively expensive. Therefore, the uncoupled analysis is much more widely used than the coupled one.

The thermomechanical model usually incorporates all the thermophysical and mechanical properties of the material as functions of temperature. Boundary conditions used in the numerical simulation can be quite general and match those used in the experiment. Besides temperature-dependent nonlinear material properties, filler metal deposition, solidification shrinkage, phase transformation, plastic strain relaxation and other physical phenomena are considered and formulated into the transient analysis. The transient approach proves useful in understanding the more general case of distortion problem, for example, it can provide insightful information on the formation mechanism of distortion and plastic strain. The major obstacle in the detailed transient approach is the enormous requirement for computing time and processing memory [53,63,69]. A transient analysis of welding distortion, which usually uses an exact model of weld and requires a 3D formulation, would be costly for a single analysis run and thus is too costly to run for the numerous cases required to study and optimize the welding process. The CPU requirements practically prohibit the use of the transient approach for prediction of welding distortion in large structures. Therefore, an alternative approach is need for efficient and precise computation of distortion in large structures.

Efforts have been made to study the means for reducing computation cost, especially computational time, in predicting distortion. Methods to simplify the FE welding simulation can be classified into two categories. The methods
in the first category are devised to simplify the complexity of FE models through using the modeling techniques to reduce model size, such as hybrid solid-shell modeling [63], adaptive meshing and sub-structuring [69]. Specific details of these modeling techniques are described in the corresponding references. The methods in the second category are focused on developing simplified and rapid computing algorithms for distortion prediction.

The methods in the second category are also identified as simplified or engineering approaches in this dissertation. Instead of conducting lengthy transient analyses, static analyses are executed within simplified approaches. Note that the term “static” is in contrast to “transient” as in a transient analysis with time dependency. It does not refer to the opposite of “dynamic” as in a dynamic analysis associated with eigenmodes or inertial effects. The nature of the simplified approaches is to simulate the weld deformation through applying the equivalent shrinkage in the welds. Such idealization and simplification are based on the physical observation of welding mechanics that weld shrinkage is the driving force of distortion. The equivalent shrinkage is in the form of equivalent temperature load [93,94], equivalent mechanical load [91,92] and inherent strain [88,89].

During the last fifteen years, efforts have been made at the Welding Engineering Program at the Ohio State University for developing numerical technologies capable of achieving fast and reasonably accurate prediction of distortion. Several numerical programs have been created for this purpose, including Inherent Shrinkage (IS) method [87,93,100], spring shrinkage model
[91,92], a special program for predicting buckling distortion [94], and plasticity-based distortion analysis (PDA) [95,96]. Jung and Tsai developed the procedure of PDA for predicting angular distortion in fillet welds. In their research, the characteristic relationship between cumulative plastic strains and angular distortion was established. The fast and accurate distortion prediction was achieved by directly mapping cumulative plastic strains into elastic models using equivalent thermal strains. The research presented in this dissertation is different from Jung and Tsai’s in the following perspectives.

In Jung and Tsai’s work, the plate thickness of the Tee joint was 3.2 mm, whereas the plate thickness in this work is about 2 mm and joined with full-penetration welds. The fillet welds in their investigation had partial penetration and therefore the plastic strains were not evenly distributed through the thickness. Their work was focused on local weld deformation, i.e. angular distortion, whereas this work is intended to investigate how in-plane shrinkage affects distortion in thin-wall structures such as automotive space frames.

1.5 Dissertation Objectives

This research is conducted to obtain better understanding and characterization of the plastic deformation that leads to in-plane shrinkage in thin-wall structures. This research is also performed to develop a fast and accurate engineering approach for predicting the welding distortion in large thin-wall structures, such as automotive body structures. These research efforts are
made to prove that in-plane shrinkage is the root cause of welding distortion in thin-wall structures.

On understanding and characterization of the plastic deformation, the research efforts are directed towards investigating the distribution characteristics of plastic strains and their relationships with the influential factors that control the plastic deformation. The engineering approach is developed based upon the principle that plastic deformation is the root source of welding distortion. A hypothesis is proposed regarding such development and stated as the following: welding distortion can be accurately predicted if the plastic deformation same as or similar to that induced by welding is introduced into the finite element analysis (FEA) model. On one hand, the research is intended to validate the hypothesis; on the other hand, the hypothesis serves as a guideline to the research. Successful implementation and application of the engineering approach demonstrates the validity of the hypothesis.

1.6 Dissertation Organization

This dissertation is divided into seven chapters. The first chapter provides fundamental information on welding distortion such as its definition, classification and control of distortion. Explicitly stated in Chapter One is the concept that plastic deformation determines distortion. Prediction of welding is introduced as a means of distortion control. Predictive methods are grouped under three heads: Empirical, Analytical and Numerical. They are described in
terms of their history, characteristics, advantages and disadvantages. Numerical methods are presented in much more details. Prior research on numerical modeling of welding distortion is review and discussed. The historical development is reviewed first, followed by examination of the research issues including 2D versus 3D modeling, uncoupled versus coupled thermomechanical analysis and detailed transient versus simplified static approach. This extensive review is given to accomplish an insightful comprehension of the problem so that the research efforts of this dissertation can be justified and appreciated. The objectives of this dissertation are defined in the section of Dissertation Objectives.

Essential information on welding plastic deformation and distortion is supplied in Chapter Two. The formation mechanism of plastic deformation and distortion is narrated qualitatively and from the theoretical perspective. The mechanism is also demonstrated through the three-bar analogy, which is a one-dimensional thermal elastic-plastic analysis performed on a three-bar model and utilized to simulate the welding process for bead-on-plate. Following the three-bar analogy is the recounting of effect of various influencing factors on plastic deformation. What are looked at next include some important issues associated with numerical modeling of plastic deformation such as strain relaxation upon melting. Also reiterated in Chapter Two are the principle that plastic deformation determines welding distortion and its implications on constructing fast and accurate distortion prediction methods.
In Chapter Three, the FEA model is developed and verified. FEA is the tool used in this work to study welding plastic deformation. Before FE simulations are carried out for this purpose, a proper FEA model is formulated and verified with experiments so that it is capable of rendering accurate distortion computation as well as capturing all the data necessary to investigate the plastic deformation. The model simulates the physical phenomena that have significant influence on plastic deformation and distortion, such as temperature-dependent material properties, continuous filler metal deposition, and plastic strain relaxation. In order to validate the accuracy of the FEA model, laboratory experiments are carried out to show the agreement of the numerical results with measured data. Specifically, the ability of the model to represent thermal and mechanical behavior is examined by comparing the computed and measured temperature and distortion. As soon as the FEA model is verified by the experiments, it is used as the substitute of experiments in the rest of the dissertation. The plastic deformation obtained from such FEA is deemed as the actual plastic deformation induced by welding.

In Chapter Four, the FEA model is utilized to perform the welding simulations of three types of simple joints including butt-welded plates, Tee joint, and plate with slot weld. These weld joints have the same structural and process characteristics as those of large thin-wall automotive body structures, as described in the section of Background. The analysis results are extracted and analyzed to study the in-plane shrinkage and plastic behaviors. The plastic strain distributions are examined in combination with the peak temperatures
accumulated during welding and the material softening at elevated temperatures. The in-plane shrinkage is correlated to the distribution characteristics of plastic strains, and furthermore the relationships between the plastic strains and their influencing factors are established. It is concluded from this study that the in-plane shrinkage plastic strains are determined by the peak temperature and material’s softening temperature range, and that peak temperature is the thermal parameter that controls the plastic strains.

Chapter Five proposes and formulates a fast and accurate approach to distortion prediction for thin-wall structures, which is also referred to as the engineering approach in this dissertation. The engineering approach is developed based on the findings obtained from the study of in-plane shrinkage and plastic behaviors mentioned above (Chapter Four). The research issue of central importance in developing the engineering approach is how to introduce the plastic deformation equivalent to the actual one into the FEA model. Temperature load is used in this study and then the central issue becomes formulating the temperature load, namely its distribution including area of application and magnitude. The engineering approach is established by applying the temperature load to the FEA model.

In Chapter Six, the engineering approach is applied to two industrial structures, and the computed results are compared with those obtained from the laboratory experiments. Successful development and application of the engineering approach validate the hypothesis about distortion prediction
proposed in this research. Presented in the last chapter, Chapter Seven, are the conclusions of this dissertation and recommendations for future research.
CHAPTER 2

WELDING DISTORTION AND PLASTIC DEFORMATION

2.1 General Description

Plastic deformation is the root source of welding distortion. Specifically, the cumulative shrinkage plastic strains and their distributions in the weldment after completion of the welding process determine welding-induced distortion. It is a commonly accepted concept that distortions in welded structures due to conventional fusion welding processes are inevitable because of the localized non-uniform heating by the welding heat source.

In a welded joint, the expansion and contraction forces act on the weld metal and adjacent base metal. During heating, temperatures are higher in these regions and they expand more than the regions away from the weld. The thermal expansions in and near the weld are curbed by the surrounding base metal at lower temperatures. The restrained expansions induce compressive longitudinal stresses in the regions near the welds. Compressive longitudinal plastic strains are also produced in these regions because of their high temperatures and the corresponding low yield stresses of the material. When the weld metal melts, the material's virgin state is recovered and plastic strain relaxation occurs. Upon cooling, the weld metal attempts to contract to the
volume it would normally occupy at the lower temperature, but it is restrained from doing so by the neighboring cooler base metal. Tensile longitudinal stresses develop within the weld, finally reaching the yield strength of the weld metal. At this point, tensile longitudinal plastic strains start to accumulate in the weld. By the time the weld reaches room temperature, the weld contains locked-in tensile residual stresses of yield magnitude and the base metal away from the weld is usually in compression with smaller magnitude. Tensile plastic strains are present in the fusion zone and compressive plastic strains exist outside the fusion zone but within certain width range. The tensile plastic strains in the fusion zone may be regarded as a product from reverse yielding, since this zone shrinks from a strain-free state. The compressive plastic strains are generated primarily due to suppression of thermal strains by the cooler surrounding materials during the heating cycle. The internal tensile and compressive forces are in equilibrium with the joint deforming to comply with the strain compatibility. The amount and distribution of weld distortion and residual stresses depend on the final state of the plastic strain distribution and their compatibility in the joint.

In the following sections, a case study is conducted including the three-bar analogy and analysis of a Tee joint. The three-bar analogy is a one-dimensional thermal elastic-plastic analysis performed on a three-bar model and utilized to simulate the welding process for a butt joint. The case studies are employed to illustrate the formation mechanism of plastic deformation and how a fast and accurate distortion prediction can be accomplished. Even
though these subjects are researched and presented in much more details in the subsequent chapters, they are discussed here through the case studies for the purpose of providing the overall understanding the research issues.

2.2 Three-Bar Analogy

To illustrate the welding plastic behavior, a simple example of heating and cooling a three-bar model is examined. The model is illustrated in Figure 2.1. It consists of a middle bar, two side bars and two end-constraining blocks that connect the three bars. $A_m$ and $A_s$ are the cross section areas of the middle bar and side bars respectively. $A_s/A_m$ ratio is 2 in this case.

Figure 2.2 shows the material properties of a mild steel constituted to the model, including the temperature-dependent yield stress and Young's Modulus as well as the room-temperature stress-strain curve indicating the elastic-plastic behavior with an isotropic hardening. As temperature increases, both yield stress and Young's Modulus decrease. They start to dramatically decrease at $T_1$ (473°K) and are assumed to be very small at temperatures greater than $T_2$ (1173°K). It is also assumed that the stress-strain relationship remains the same for temperatures lower than the melting point $T_m$ (1700°K). The most significant effect of melting on the material behavior is plastic strain relaxation. When melting occurs, the material returns to its nascent strain-free state. The temperature for plastic strain relaxation is set at $T_m$ so that the melting effect can be taken into account.
Figure 2.1: Three-bar analogy analysis model

$A_m$: Cross section area of middle bar

$A_s$: Cross section area of side bars
Figure 2.2: Material properties of the steel used in three-bar analogy analysis. 
(a) Young’s Modulus and yield stress; (b) stress-strain curve at room temperature
In the analysis, only the middle bar experiences the linearized heating-cooling cycle. The heating and cooling history is described in Figure 2.3. The bar is linearly heated up from room temperature \(300^0\text{K}\) to a peak point \(T_{\text{peak}}\) \(2000^0\text{K}\), and then cooled back to room temperature linearly. The model is in a free state since the applied boundary condition is for the sole purpose of rigid-body motion prevention. Therefore, the thermal expansion and contraction of the middle bar are only restricted by the internal constraints from the side bars and end-constraining blocks. Without losing generality, the three-bar model can be treated as an analogy to a butt joint provided that the middle bar represents the weld metal and side bars stand for the base metal. It should be aware that \(\text{As} / \text{Am}\) ratio is usually greater than 10 in a real welded plate.

Figure 2.4 shows the plastic strain history in the middle bar during the heating and cooling cycle. Since the material softens as the temperature increases, the plastic strain builds up quickly. It is in compression because the thermal expansion of the middle bar is impeded by the side bars and end-constraining blocks. The plastic strain is relaxed when melting occurs. On cooling, the middle bar contracts and its thermal contraction is suppressed by the internal constraints that also restrain its thermal expansion during heating. Consequently, the tensile plastic strain emerges as the temperature drops blow the melting point. In the early stage of cooling, the bar is still at high temperatures and has little strength or rigidity, and therefore, the plastic strain accumulates rapidly. Its growth slows down and finally flattens as the temperature approaches room temperature. As a result of the reverse yielding
process described above, the final plastic strain is in tension. The middle bar is shorter than its original length after cooling, or in other words, the bar exhibits shrinkage.

Figure 2.3: Heating and cooling history of the middle bar in three-bar analysis
Welding distortion is primarily influenced by the interaction between the final state of the cumulative plastic strains in the welded structure and its structural rigidity. The plastic deformation is the driving force of distortion and the structural rigidity acts as the resistance to distortion. Distortion can vary...
significantly in kind and magnitude due to its dependency on the process and structural variables.

The process variables include welding heat input, sequence and fixture. The structural variables comprise geometry and size of the welded assembly. Since these factors affect the formation and final state of the plastic strains, they heavily impact distortion. Their effect can be investigated through the three-bar analogy analysis. Although the three-bar model is simplistic compared to a welded joint, their inherent similarity makes it good enough to demonstrate the effect of the above factors on welding distortion and plastic deformation.

Three case studies are conducted and directed towards illustrating the effects of structural rigidity, heat input and fixture. The analysis case discussed above is used as the baseline model in the case studies. The other models are compared with it in the following context. For the purpose of unambiguous comparison, the premises of the baseline model are reiterated here:

- Internal constraint: \( \frac{A_s}{A_m} = 2 \)
- Loading condition: \( T_{peak} = 2000 \, ^{0}\text{K} \)
- Boundary condition: free state (no external constraint)

The effect of structural rigidity is demonstrated in the first case study, which consists of five models including the baseline model. The five models have the same loading and boundary conditions. However, they differ in the structural rigidity as they have different \( \frac{A_s}{A_m} \) ratios of 2, 4, 6, 8 and 10 respectively.
The results of the case study are plotted in Figure 2.5. Each curve in Figure 2.5a represents the plastic strain history in the middle bar of the corresponding model. The plastic strains in different models follow a similar path of evolution. They are in compression during heating and experience plastic strain relaxation when melting occurs. They also exhibit reverse yielding during cooling and end in tension. The most visible difference among the plastic strains is that they deviate from one another in the late stage of cooling. The maximum magnitudes in compression they reach during heating are different too. Comparisons of the final plastic strains and shrinkage magnitudes are given in Figure 2.5b. When the $A_s/A_m$ ratio increases, the plastic deformation increases and distortion decreases because of the increased internal constraint caused by the augmented side bar area. Once the $A_s/A_m$ ratio exceeds 6, the final plastic strain starts to saturate and the shrinkage magnitude begins to level out. The most important phenomenon revealed in Figure 2.5b is that the distortion synchronously varies with the plastic deformation.
Figure 2.5: Case study results showing the effect of structural rigidity on distortion and plastics deformation. (a) Plastic strain history in the middle bar; (b) Shrinkage and final plastic strain in the middle bar.
In addition to the baseline model, two models are analyzed to simulate the effect of heat input in the second case study. All three models have the same structural rigidity ($A_s/A_m = 2$) and boundary condition but different loading conditions, which prescribe the peak loading temperatures of 1000, 1200 and 2000$^o$K respectively for the three models.

Figure 2.6a shows the plastic strain histories of the three models. The plastic strain in the baseline model ($T_{\text{peak}} = 2000^o$K) undergoes compression, relaxation and reverse yielding and end up with tension. The plastic strain relaxation is not observed in the other two models whereas their peak temperatures are lower than the melting temperature. The model with $T_{\text{peak}} = 1000^o$K witnesses its compressive plastic strain building up during heating and reaching its maximum magnitude at the end of heating. The compressive plastic strain remains unchanged with its extreme value throughout cooling. Compared with the 1000$^o$K model, the model with $T_{\text{peak}} = 1200^o$K displays the accumulated compressive plastic strain of higher magnitude by the end of heating due to its higher temperature load or simulated heat input. Because of the same reason, the 1200$^o$K model endures stronger shrinkage than the 1000$^o$K model and experiences reverse yielding during cooling, which is absent in the 1000$^o$K model.

Different plastic strain histories lead to different final plastic strains in the three models, which consequently induce distinctive distortion behaviors. It can be seen in Figure 2.6b that the larger shrinkage distortions are associated with the higher peak loading temperatures. It is concluded from this case study that
heat input affects welding distortion in that it alters the final plastic strain
distribution from one weldment to another.

The third case study is intended to evince the impact of welding fixture on
the plastic strain and distortion. It consists of two models. One is the baseline
model. The other one differs from the baseline model only in boundary
condition. The baseline model is assigned the free state boundary condition,
which only inhibits rigid body motion and exerts no external constraint on the
model. The baseline model is dubbed as the free model henceforth. In the
other model, the end-constraining blocks are clamped during the thermal
loading process. The boundary condition is depicted in Figure 2.7. Such
boundary condition imposes very rigid external constraint on the model. In fact,
it substantially restricts the thermal expansion and contraction of the middle bar.
The boundary condition can be ideally deemed as the clamping fixture. In order
to obtain the final distortion, the clamping fixture is removed and the free state
boundary condition is imposed on the model after cooling. The model is
referred to as the clamped model in the following context.

The results of the study are portrayed in Figure 2.8. The plastic strain
histories of the two models are compared in Figure 2.8a. The plastic strains
develop along a similar course in terms of compressive yielding, relaxation and
reverse tensile yielding. Owing to its external constraints, the clamped model
yields in more compression during heating and more tension during cooling
than the free model. The final tensile plastic strain of the clamped model has a
magnitude 38.2% higher than that of the free model. The difference in the
plastic strains causes the difference in the distortions, as shown in Figure 2.8b. The shrinkage magnitude of the clamped model is 73.8% of that of the free model.

There exists a prevailing trend in the results of the case studies using the three-bar analogy analysis. Any change in the welding distortion can be traced back to the variation of the plastic deformation. Various factors affect the welding distortions because they change the final plastic strains. The dependency of the distortion on the plastic deformation, which is demonstrated in the above case studies, reflects the principle that plastic deformation is the root source of welding distortion.
Figure 2.6: Case study results showing the effect of heat input on distortion and plastics deformation. (a) Plastic strain history in the middle bar; (b) Shrinkage in the middle bar.
As: Cross section area of middle bar
As: Cross section area of side bars

Figure 2.7: Three-bar analogy analysis model with external clamping constraint
Figure 2.8: Case study results showing the effect of welding fixture on distortion and plastics deformation. (a) Plastic strain history in the middle bar; (b) Shrinkage in the middle bar.
2.3 Analysis of Tee Joint

In Chapter One, a hypothesis is proposed regarding prediction of welding distortion. It is stated in the hypothesis that the welding distortion can be accurately computed if the plastic deformation the same as or similar to the actual one induced by welding is introduced into the FEA model. The case study presented below is used to illustrate the validity of the hypothesis. The case study is part of the investigation in Chapter Four and all the detailed information about the case are covered in that chapter, and therefore only necessary data are given out in this section for the purpose of illustration.

The welded structure analyzed in this study is a Tee joint. The dimensions of the joint are specified in Figure 2.9. The material of the joint is the same mild steel as that used in three-bar analogy analysis. GMAW is employed to build the Tee joint. The central portion of this study is to infuse the actual welding plastic deformation into the FEA model. Two types of FE analyses are conducted in the study.

As narrated in Chapter One, accurate distortion prediction can be obtained from the transient thermomechanical FEA of welding processes. In the following context, this type of analysis is referred to as the *transient* analysis. Based on the confidence in its high accuracy, the transient analysis is used as numerical experiment to substitute the laboratory trial with the benefits of cost and time saving. The plastic strains and distortion extracted from the transient analysis are treated as the actual ones here.
The other type of analysis is referred to as the simplified analysis. Compared with the transient analysis, the simplification indicates that it is a static analysis ignoring the transient effect of welding, which includes the plastic strain and structural rigidity evolution during welding. In the simplified analysis, the equivalent temperature load is applied to the welded structure to simulate the weld shrinkage and generate the plastic deformation. The accuracy of the simplified analysis mainly depends on how the equivalent temperature load is applied to the structure. The formulation of equivalent temperature load is presented in Chapter Five. The proof of the hypothesis can be acquired if the distortion predicted by the simplified analysis is close to that by the transient analysis while the two analyses produce similar plastic strain fields.

Both simplified and transient analyses are performed on the simple Tee joint in this study. Compared in this paragraph are the results from the two analyses, including the plastic strain fields, distorted shapes, displacement fields, and residual stress fields in the base plate. Figure 2.10 manifests the transverse plastic strain (PE11) fields calculated by the two analyses. The plastic strain fields resemble each other with certain degree of discrepancy. In both analysis results, both compressive and tensile plastic strains are present with the compressive strains concentrating in the weld and the tensile strains surrounding the weld. The maximum magnitudes of the plastic strains from the two analyses are comparable: 6.21E-2 from the transient analysis and 6.82E-2 from the simplified analysis. Although the shapes of the compressive plastic zones are similar to each other, the shapes of the tensile plastic zones exhibit
apparent difference. The difference is caused by the transient effect of welding heat transfer, which is not captured in the simplified analysis.

Figure 2.9: Simple Tee joint and its dimensions
A comparison of the distorted shapes obtained from the two analyses can be found in Figure 2.11. They are very comparable as both exhibit obvious transverse shrinkage and slight longitudinal shrinkage. As shown in Figure 2.12, the two displacement fields have almost identical distribution pattern. Their extreme values are close: $U_{\text{min}} = -1.47\times10^{-1}$ mm and $U_{\text{max}} = 1.18\times10^{-1}$ mm from the transient analysis; $U_{\text{min}} = -1.30\times10^{-1}$ mm and $U_{\text{max}} = 1.03\times10^{-1}$ mm from the simplified analysis.

Even though the plastic strain field from the simplified analysis does not perfectly match that from the transient analysis, it closely approximates the latter in terms of distribution pattern and maximum magnitude. The displacement fields computed by the two analyses are very similar. So are the distorted shapes of the base plate. The unequivocal similarities are attributed to the agreement between the plastic fields of the two analyses. On the other hand, the difference between the two plastic fields results in the subtle differences between the displacement fields. The close match between the simplified and transient analysis presented here serves as the proof of the hypothesis that the distortion can be accurately predicted if the plastic deformation the same as or similar to that formed during welding can be introduced to the weldment.
Figure 2.10: Comparison of plastic strain fields from transient and simplified analyses. (a) Transient analysis, $\varepsilon_{p_{\text{max}}} = 4.84\times10^{-3}$, $\varepsilon_{p_{\text{min}}} = -6.21\times10^{-2}$; (b) Simplified analysis, $\varepsilon_{p_{\text{max}}} = 3.85\times10^{-3}$, $\varepsilon_{p_{\text{min}}} = -6.82\times10^{-2}$. 
Figure 2.11: Comparison of distorted base plates from transient and simplified analyses. (a) Transient analysis; (b) Simplified analysis.
Figure 2.12: Comparison of displacement fields from transient and simplified analyses. (a) Transient analysis, $U_{\text{min}} = -1.47\times10^{-1}$ mm, $U_{\text{max}} = 1.18\times10^{-1}$ mm; (b) Simplified analysis, $U_{\text{min}} = -1.30\times10^{-1}$ mm, $U_{\text{max}} = 1.03\times10^{-1}$ mm.
2.4 Theoretical Description of Inherent Plastic Strains

The welding-induced cumulative plastic strains are also called inherent plastic strains. Welding-induced, incompatible plastic strains (assuming a 2-D plane-strain condition for illustration purpose) at each heating or cooling time increment may be written as follows:

\[
\nabla^2 (\sigma_x + \sigma_y) = -\frac{E}{1-\nu} \nabla^2 (\alpha \theta) - [g(x, y) + \Delta g(x, y)]
\]

Equation 2.1

where \( \nabla^2 \) is the Laplacian operator, \( \sigma_x \) and \( \sigma_y \) are thermal stress components in the respective x and y directions, \( E \) is Young’s modulus, \( \nu \) is Poisson’s ratio, \( \alpha \) is thermal expansion coefficient, \( \theta \) is a temperature function, \( g(x, y) \) is a cumulative plastic strain function, and \( \Delta g(x, y) \) is the plastic strain increment function over each thermal loading step. The plastic strain functions may be written as follows:

\[
g(x, y) = \frac{E}{1-\nu^2} \left( \frac{\partial^2 \varepsilon_x^p}{\partial y^2} + \frac{\partial^2 \varepsilon_y^p}{\partial x^2} - 2 \frac{\partial^2 \varepsilon_{xy}^p}{\partial x \partial y} \right) - \frac{\nu E}{1-\nu^2} \nabla^2 (\varepsilon_x^p + \varepsilon_y^p)
\]

Equation 2.2

\[
\Delta g(x, y) = \frac{E}{1-\nu^2} \left( \frac{\partial^2 (\Delta \varepsilon_x^p)}{\partial y^2} + \frac{\partial^2 (\Delta \varepsilon_y^p)}{\partial x^2} - 2 \frac{\partial^2 (\Delta \varepsilon_{xy}^p)}{\partial x \partial y} \right) - \frac{\nu E}{1-\nu^2} \nabla^2 (\Delta \varepsilon_x^p + \Delta \varepsilon_y^p)
\]

Equation 2.3

The Laplacian thermal strains are governed by the rate of enthalpy change in the weldment. Upon completion of the heating and cooling cycles the

51
cumulative plastic strains in the weldment are usually compressive and become the inherent shrinkage strains, $g_I(x, y)$, that interact with the structural rigidity to result in residual stresses ($\sigma_x^R$ and $\sigma_y^R$) and distortion ($\varepsilon_x^D$ and $\varepsilon_y^D$). Residual stresses may be written with a possible reverse yielding, $g^R(x, y)$ as follows:

$$\nabla^2 (\sigma_x^R + \sigma_y^R) + g^R(x, y) = -g^I(x, y) \quad \text{Equation 2.4}$$

Distortion may be written in the form of final total strains as follows:

$$\varepsilon_x^D - \varepsilon_x^I = \frac{1}{E} \left[ \sigma_x^R - \nu (\sigma_y^R + \sigma_z^R) \right] + \varepsilon_x^{PR} \quad \text{Equation 2.5}$$

$$\varepsilon_y^D - \varepsilon_y^I = \frac{1}{E} \left[ \sigma_y^R - \nu (\sigma_x^R + \sigma_z^R) \right] + \varepsilon_y^{PR} \quad \text{Equation 2.6}$$

$$\varepsilon_z^D - (\varepsilon_x^I + \varepsilon_y^I) = \frac{1}{E} \left[ \sigma_z^R - \nu (\sigma_x^R + \sigma_y^R) \right] - \varepsilon_x^{PR} - \varepsilon_y^{PR} = \text{const} \tan t \quad \text{Equation 2.7}$$

$$\gamma_{xy}^D - \gamma_{xy}^I = \frac{2(1+\nu)}{E} \tau_{xy}^R + \gamma_{xy}^{PR} \quad \text{Equation 2.8}$$

where the superscripts: D represents distortion strains, I is the inherent (cumulative) shrinkage strains, R is residual stresses, and PR is plastic strains due to reverse yielding. Equations 2.1 through 2.8 demonstrate that the cumulative plastic strains govern the final state of residual stresses and weldment distortion.
CHAPTER 3
FINITE ELEMENT ANALYSIS MODEL AND VERIFICATION

In this chapter, a FEA model – moving heat source model is formulated and verified by experimental results. The moving source model will be used as a numerical tool in the subsequent studies of this dissertation. In this model, the movement of the welding heat source is simulated. Therefore transient thermal and mechanical behaviors of weldments are the domains of calculation. Up to date it is probably the most accurate numerical model that can be used to predict welding distortion if the correct model formulation is chosen.

The requirements of the moving source model adopted in this research are:

- It can predict welding distortion accurately, and
- It can provide a basis to investigate the welding plastic deformation.

To fulfill the above requirements, formulation of the model becomes critical. The finite element model formulation discussed here is not the mathematical formulation or computer implementation of the finite element method (FEM) itself. The formulation and implementation of the finite element method are carried out using a commercial finite element code ABAQUS. They are not the interest and out of the scope of this dissertation. The model formulation recited
here is the numerical procedure to approximate the physical phenomena in welding as close as possible.

The moving source model in this study is used to calculate the transient temperature field in a weldment and its mechanical response. The most important model formulation issues are how to compute heat flux distribution in the welding arc and how to establish proper thermo-elasto-plastic material behavior in the model. They will be addressed in detail in the following sections.

Generic verification of the FEA model is performed using two types of weld joints: a constrained joint and an unconstrained joint. The constrained joint is a four-pass double-V Butt joint. It is heavily constrained with restraining block so that there is virtually no out-of-plane distortion and in-plane shrinkage is the dominant distortion mode. The unconstrained joint is a double side welded Tee joint. It is welded without any restraining fixture except those used to prevent rigid body motion. The out-of-plane distortion is apparent in the Tee joint. It should be noted that in-plane shrinkage is the focus of this dissertation work. If the FEA model passes the generic verification on both constrained and unconstrained joints, then it is good enough to be used as the numerical tool to investigate in-plane shrinkage. The details of the experimental setup and FEA procedure are presented in the following sections.
3.1 Finite Element Analysis Model Formulation

3.1.1 Consideration of Physics of Welding

Weld distortion is only one aspect of mechanical effects of welding. Although weld distortion can be classified into several simple forms, the physical processes, such as electromagnetic, thermal, metallurgical and mechanical phenomena that lead to distortion cannot be simply described using one or several mathematical equations. During welding the welding arc melts the electrode and base metal with the filler metal deposited into the groove. How the arc distributes thermal energy into the workpiece and exerts electromagnetic force on the weld pool is the domain of arc physics. As the arc starts and travels, the weld pool is formed traveling with the arc. Besides heat convection and conduction in the weld pool, fluid flow exists in the pool. It is the investigation subject of fluid mechanics of weld pool. The weld metal and heat affected zone experience complex thermal cycles, which determine how the weld metal solidifies and how the microstructure evolves in the weld metal and heat affected zone. It is the interest of continuum mechanics to study transient thermal, stress and strain fields. To study solidification and phase transformation is within the realm of metallurgists. These physical processes affect weld distortion more or less. Theoretically, they all should be formulated into the finite element model to calculate distortion.

In practice, all the aspects of welding mentioned above have made the analysis of weld distortion challenging. However, every single one of them can
be computed separately. The transient temperature field in and near the weld pool was most rigorously computed by modeling the arc physics and fluid mechanics of the weld pool. Models for the physics of welding arcs could predict the velocity, temperature and pressure in a welding arc [101]. Most models for the weld pool assumed that the pressure, thermal flux, current density and thermal efficiency of the arc, the temperature-dependent viscosity, surface tension, thermal conductivity, and specific heat of the liquid were known [102,103,104]. They computed the temperature, velocity, pressure, and geometry of the weld pool. They also computed the geometry of liquid surface and the solid-liquid interface in the weld pool. This was achieved by applying the laws of conservation of momentum and energy in the form of the coupled Navier-Stokes and heat equation.

Some researchers believe that the stress and strain in welds cannot be predicted without simultaneously predicting the evolution of microstructure in welds [105,106]. There has been significant advancement in the last decade in microstructure modeling of steels based on the fundamental phase transformation thermodynamics and kinetics theories. These models were often formulated using chemical thermodynamics and classical nucleation and growth theory. They were used to simulate various metallurgical phenomena such as austenite grain growth, carbide coarsening and dissolution, austenite decomposition, and carbide precipitation. For a given material and welding procedure, the microstructures were predicted as a function of welding thermal
history and chemical composition. Also studied was the effect of evolution of microstructure on the thermal stress analysis of welds.

There are other issues relating microstructure to the residual stress and strain analysis of welds. They include predicting the thermal and mechanical constitutive properties for a given microstructure, the effect of the evolution of stress on the evolution of microstructure, performance and failure analyses of a weld with its microstructure and residual stress, and transformation plasticity. The volume change in a phase transformation, for example martensitic transformation, produces an additional component to plastic strain increment, which is defined as transformation plasticity. The anomalous stress strain curves are usually associated with phase transformations [107]. In classical plasticity the strain rate is proportional to the deviatoric stress rate. However, the transformation plastic strain rate is proportional to the deviatoric stress, not the deviatoric stress rate. It is also proportional to the rate of transformation and the fraction volume change in the transformation. It is inversely proportional to the yield strength of the weaker phase [108,109]. The major research issue of this dissertation is how the plasticity in the weld affects distortion. It seems that transformation plasticity should be investigated. Nevertheless transformation plasticity is not a major concern here because it only dominates the residual stresses in high strength steels with high hardenability, which are not the materials engaged in this study. It is should be pointed that microstructure modeling cannot be carried out solely using FEA and it is not the subject of this study.
If the welding arc, weld pool and microstructure evolution are incorporated into the finite element modeling of weld distortion, then computing distortions in welds begins with specifying the geometry, material properties and welding procedure. The transient temperature field is computed next. It drives the microstructural changes that determine the constitutive parameters, latent heats and volume changes. Finally, a thermo-elasto-plastic stress analysis uses the temperature and microstructural data to compute the transient and residual stress and strain fields. This nonlinear transient problem is solved with iterative methods. The procedure is illustrated in Figure 3.1.

The mechanical behavior of welds such as distortion is sensitive to the close couplings between the hydro-magneto-dynamics of welding arc, the fluid mechanics of weld pool, heat transfer, microstructure evolution and thermal stress analysis. In Figure 3.1, the dominant couplings in welding are shown with bold lines. The secondary couplings are shown with dashed lines. It is ideal that a comprehensive computational model can be formulated to predict distortion. In reality, it is very difficult to realize this algorithm since the cost of computing will be a deterrent and the greatest limitation has always been the knowledge of the physics, mathematics and computer science needed to model welds. Interactions between thermal, mechanical, metallurgical, chemical, and fluid process in the molten pool are complex. It is essential and efficient that the most influencing coupling on weld distortion be formulated into the FEA model.
As shown in Figure 3.1, the numerical modeling procedure can be divided into two categories: microscopic and macroscopic modeling. In the microscopic modeling, the stress and strain in welds are considered to be a metal physics problem. The phase transformations including grain growth, dissolution and precipitation are computed. In the macroscopic modeling, weld distortion is
considered to be a thermo-mechanical problem. Transient fields of temperature, displacement, stress and strain are computed. In principle, this can be done by solving the equations of continuum solid mechanics. The couplings at the microscopic level have secondary effects on weld distortion prediction. For example, the effects of microstructure on heat transfer and distortion are very small in low-carbon steels and low-alloy steels. Therefore, they are not included in the FEA model in this study.

The couplings at the macroscopic level have dominant effects on distortion and should be formulated into the FEA model. As mentioned in Chapter One, the changes in the stress-strain field do not play a noticeable role on the temperature field during welding. The secondary coupling in the macroscopic modeling is insignificant to distortion computation and the coupling effect between the thermal and mechanical analysis is virtually one-way. The thermo-mechanical behavior of the weldment during welding can be simulated using uncoupled formulation, because the dimensional changes are negligible and the mechanical work done is insignificant compared to the thermal energy from the welding arc [110]. A study using both coupled and uncoupled models for the same GTAW (Gas Tungsten Arc Welding) weld demonstrated that there is virtually no difference between the stress-strain results from the two models [111].

The thermal analysis and mechanical analysis are uncoupled in this study. Some researchers also called the analysis procedure a decoupled or sequentially coupled approach [99,110,111]. To calculate distortion, the
thermal analysis is conducted first, and then the temperature history obtained is used as input in the subsequent mechanical analysis. The analysis procedure is usually dubbed as transient thermo-mechanical analysis.

3.1.2 Thermo-mechanical Analysis of Weld Distortion

The most effective and important finite element analysis of distortion is a transient thermo-mechanical analysis. In the uncoupled analysis formulation described in the last section, the heat transfer analysis is decoupled from the mechanical analysis. However, the formulation considers the contribution of the transient temperature field to mechanical analysis through thermal expansion and volumetric change, as well as temperature-dependency of thermo-physical and mechanical properties. The heat transfer or thermal analysis is conducted first to obtain transient temperature field. Then the mechanical or stress-strain analysis is performed to calculate distortion. The temperature history from the thermal analysis is used as thermal loading in the mechanical analysis. Since the temperature field computed from the thermal analysis is considered to drive the mechanics of the welding process, the first step is to solve the energy equation with FEA. The research issue is decoupling the physics of the arc and weld pool from the energy equation. This is done by modeling the heating effect of the arc. This issue will be discussed in detail later in this chapter (section 3.4.1) about the thermal analysis.

The mechanical analysis involves large strain and large rotations. An elasto-plastic material model with temperature-dependent material properties is
adopted in this study. In addition to the elasto-plastic constitutive material behavior, the deposition of filler metal and plastic strain relaxation are also modeled in the mechanical analysis to accurately simulate the plastic behavior in the weldment. The detailed FEA procedure will be discussed in the subsequent sections about the mechanical analysis (sections 3.5.1 and 3.5.3).

A general-purpose finite element code ABAQUS® is utilized in this research. ABAQUS has the features necessary to solve large variety of problems and to converge well and easily. It is capable of computing the material and geometric non-linearity due to temperature-dependent thermo-elasto-plastic properties, various boundary conditions, and path-dependent plasticity that are involved in the thermo-mechanical analysis of distortion. ABAQUS provides the interface for user-subroutines which allow the special numerical treatment of computational welding mechanics be implemented through ABAQUS. Figure 3.2 shows the finite element analysis procedure for the thermo-mechanical analysis of weld distortion. In this figure, DFLUX is the ABAQUS user subroutine used to model the arc as a heat flux distribution that is a function of time and space; UMAT is the ABAQUS user subroutine employed to constitute material behavior.
Geometry

Mesh Generation

- Heat input model – Goldak’s Model
- Temperature-dependent thermophysical properties
- Initial conditions
- Boundary conditions

Non-linear Transient Heat Transfer Analysis Using ABAQUS with DFLUX user subroutine

Transient Temperature Fields

Thermal Analysis

- Elasto-plastic constitutive equations
- Temperature-dependent material properties
- Numerical treatment of filler metal addition
- Initial conditions
- Boundary conditions

Non-linear Transient Thermal Stress-Strain Analysis Using ABAQUS with UMAT user subroutine

Transient and Residual Stress/Strain Fields

Mechanical Analysis

Figure 3.2: Finite element analysis procedure for thermal-mechanical analysis of weld distortion

63
3.2 Experimental Setup

The geometry of the Butt joint is designated in Figure 3.3. Special considerations were made to minimize out-of-plane distortion. First, the joint was made of 69.85-mm thick plates. Such thick plates have good resistance to transverse angular distortion. Second, the double-V groove geometry combined with the welding sequence was designed to confine out-of-plane motion. Third, the restraining members could effectively block out-of-plane distortion. The coordinate system in the finite element analysis is fixed on the workpiece. The origin of the coordinate system is located at the geometrical center of the workpiece, with the x-axis normal to the welding direction, y-axis along the thickness direction and z-axis along the centerline of the workpiece. The welding arc started at \( z = z_0 \) (when \( t = 0; z_0 = 152.4 \) mm) and moved in the negative direction of along the z-axis. The experimental procedure is elaborated on in Reference 74. The welding parameters of the GMAW process are:

- Arc voltage: 27.5 Volts,
- Welding current: 250 Amperes,
- Travel speed: 3.81 mm/second,
- Filler wire: E70S-6, 1.143 mm in diameter, and
- Wire feed speed: 118.53 mm/second.
The double-fillet Tee Joint was made with Flux Cored Arc Welding (FCAW). The experimental set-up and geometry of the joint are illustrated in Figure 3.4. The base plate measures 508 x 508 mm and the height of the stiffener is 203.2 mm. During welding two torches traveled simultaneously on both sides of the stiffener to maintain symmetry about the stiffener. The nominal fillet size is 6.35 mm. The plate and stiffener were tack welded together before welding. As shown in Figure 3.4, the two positioning brackets mounted on the stiffener were used to support the joint and prevent any rigid body motion. To obtain the thermal response during welding, the thermocouples were placed both in the interior and on the surface of the plate. The interior thermocouples (TC1 & TC2) were located beneath the weld bead in order to record the largest temperature change. The surface thermocouples (TC3 & TC4) were primarily used to help validate the finite element model. Figure 3.5 displays the placement of the thermo-couples.

Linear voltage displacement transducers (LVDT) were used to record the real-time out-of-plane displacement of the plate. The LVDT’s were positioned beneath the plate, as indicated in Figure 3.4. The location of the LVDT’s is shown in Figure 3.5. Further details of the experimental procedure can be found in References 75 and 112.

The plate thickness of the Tee joint is 12.7 mm. It is intended that the selection of plate thickness allows apparent transverse angular distortion be experienced by the joint. For steel plates, the maximum angular changes caused by fillet welds appear when the plate thickness is around 9.5 ~ 12.7 mm.
When the plate is thinner than this range, the angular change decreases because the thin plate is heated more evenly in the thickness direction and the out-of-plane bending is reduced. When the plate is thicker, the angular change is also reduced because the rigidity of the plate increases.

Figure 3.4 and Figure 3.5 also display the origin and orientation of the finite element coordinate system, which is fixed on the joint. The x-axis is along the edge of the base plate, the y-axis is in the thickness direction of the base plate and the z-axis is parallel to the longitudinal direction of the joint or welding direction. The two fillet welds were welded synchronously. The purpose of concurrent welding on both sides of the joint is to obtain symmetric angular changes on the two sides so that the finite element model can be simplified to a symmetric half model to represent the joint. The welding parameters of the FCAW process are:

- Arc voltage: 24 Volts,
- Welding current: 245 Amperes,
- Travel speed: 5.93 mm/second, and
- Wire feed speed: 126.15 mm/second.

3.3 Finite Element Meshes

The finite element meshes for the Butt and Tee joints are presented in Figure 3.6 and Figure 3.7 respectively. Among the difficulties associated with finite element analysis of weld behavior, one of the frequently encountered
problems is how to construct the meshes adequately describing the essential geometric features of welded structures using coarse meshes while at the same time obtaining sufficient details around the joints with refined meshes. To describe weld and heat affected zone (HAZ) in any detail requires a resolution of at least one millimeter [113]. Typical welded structures may have dimensions of the order of meters. Typical weld length may also be in meters. Obviously, fine meshes cannot be built over a whole welded structure. The mesh in each domain of analysis should be coarse enough to make the computations feasible. It was reported that increasingly finer meshes ultimately led to convergence problems [114].

The size of the mesh should be selected in such a fashion that the balance between the computing cost and prediction accuracy could be achieved. In other words, the finite element analysis can yield the temperature and stress-strain fields with acceptable accuracy while having the fewest number of elements in the model. The meshes in this study are also opted in accordance with the characteristics of the moving heat source simulation. In the moving heat source model, the weld and HAZ experience the highest temperature gradients and thus small elements are used in the regions.
Figure 3.3: Schematic of the Butt joint. (a) Geometry of the workpiece and coordinate system used in the finite element analysis; (b) Groove geometry of the Butt joint and welding sequence of the four passes. Dimensions are in mm.
Figure 3.4: Schematic of the Tee joint and experimental set-up including double torches, positioning scheme, temperature and displacement measurement. (a) Geometry of the joint and coordinate system used in the finite element analysis; (b) Fillet weld size (*TC stands for Thermo-Couple). Dimensions are in mm.
Figure 3.5: Location of displacement and temperature measurements. (a) Location of LVDT’s; (b) Location of thermo-couples (showing half model). Dimensions are in mm.
Figure 3.6: Finite element mesh of the Butt joint. (a) Overall mesh; (b) Mesh of the groove including the four weld passes.
Figure 3.7: Finite element mesh of the Tee joint. (a) Overall mesh; (b) Mesh of the weld and its surrounding area.
Away from the heat source, the temperature and stress-strain fields incline to change slowly and distribute linearly. Therefore, the mesh size is increased gradually farther away from the weld. This meshing algorithm is clearly demonstrated in Figure 3.6 and Figure 3.7. The element size in the weld affects the computational error of temperature and displacement. More elements in this area provide more integration points for the model to calculate the heat flux distribution and thus more accurate temperature distribution can be obtained. It has been found there should be at least 6 elements spreading over bead width or heat source application width so that reasonably accurate temperature prediction can be achieved [87]. Similar observations were also reported by other researchers. In a study by Paley and Hibbert, accurate temperature distribution was resulted from spanning 2 to 10 elements over the heat source width [115]. There are 8 elements scattered over the bead width of each pass in the Butt joint as shown in Figure 3.6; 6 elements over the fillet weld bead width in the Tee joint as shown in Figure 3.7.

The mesh of the Butt joint is created as a full model since the position and sequence of the weld passes are not symmetric. However, a half model is generated for the Tee joint since the two fillet welds are symmetric about the neutral plane (yz plane shown in Figure 3.4) of the stiffener plate and deposited simultaneously. This reduces the number of elements by 50 percent and CPU time more than 50 percent.

Tetrahedron and triangular prism elements are avoided in the meshes because they are numerically unstable at high temperatures during transient
heat transfer analysis. A special numerical technique is implemented using ABAQUS command *CONTACT PAIR [116], which bonds mismatched large and small elements through a multiple point constraint (MPC) interface. This technique eliminates the need for tetrahedron, triangular prism and severely distorted brick elements. The contact-pair interfaces created by this numerical treatment are plotted in Figure 3.6 and Figure 3.7. Figure 3.7 displays the model partition in the fillet weld. The partition is also used to avoid triangular prism elements. The number of nodes, number of elements, and element types in the meshes are summarized in Table 3.1. ABAQUS 8-node linear brick heat transfer element DC3D8 and 8-node linear brick, reduced integration stress/displacement element C3D8R are employed in the thermal and mechanical analysis respectively.

<table>
<thead>
<tr>
<th>Model</th>
<th>Element Type</th>
<th>Number of Nodes</th>
<th>Number of Elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Butt Joint</td>
<td>DC3D8, C3D8R</td>
<td>19603</td>
<td>15580</td>
</tr>
<tr>
<td>Tee Joint</td>
<td>DC3D8, C3D8R</td>
<td>26487</td>
<td>21600</td>
</tr>
</tbody>
</table>

Table 3.1: Characteristic parameters of the finite element meshes
3.4 Thermal Analysis and Experimental Verification

The purpose of the thermal analysis is to capture the transient temperature field during welding. Thermal history will be interpolated into the mechanical analysis model to calculate the transient and final stress and strain. The mathematical nature of the thermal analysis is solving the heat transfer governing equations which are the equations of continuum mechanics. To solve the equations requires the geometry, initial conditions, boundary conditions and constitutive parameters to be specified. The dimensions of the finite element models are given in the previous section. The other relevant parameters and their numerical treatment in the thermal analysis are discussed in this section.

3.4.1 Mathematical Formulation of Heat Input and Heat Transfer Computation

Accurate prediction of weld distortion requires an accurate analysis of welding thermal cycle. Heat transfer in a weldment includes heat generation by the welding arc, heat conduction in the weld metal, HAZ and base metal, and heat loss by convection and radiation at the surfaces. Modeling of the energy distribution in the welding arc is the core of the analysis using the moving source model.

The interaction of the welding arc with the weld pool is a complex physical phenomenon that still cannot be completely understood and rigorously modeled. Therefore, modeling of arc physics and fluid flow phenomena is not attempted in this dissertation. The heat transferred from the arc to the workpiece includes: 1) convection by the plasma jet flow, and 2) radiation out of
the arc [101]. The heat generated by the arc should be computed as the convective and radiative boundary conditions in the heat transfer calculation. However, it is infeasible to formulate the computation in such manner because there are no mathematical formulae capable of describing the phenomena adequately and precisely. Practically, the heat input is treated either as body heat flux distributed throughout a volume in the workpiece or as surface heat flux applied on a surface of the workpiece. Net heat input from the arc to the workpiece can be expressed by the following equation:

$$q_0 = \eta EI$$  \hspace{1cm} \text{Equation 3.1}$$

where \( q_0 \), \( \eta \), \( E \), and \( I \) are net heat input, arc efficiency, arc voltage, and current respectively. The spatial distribution of thermal energy in the arc is the key to the heat input model. It has been under intensive investigation for about 60 years and many models have been introduced into welding heat flow analysis.

In the model proposed by Rosenthal and Rykalin [27,28,117], the arc heat input was assumed to be an infinitesimally concentrated point or line heat flux, which is often referred to as point or line source. Moving coordinates were used with Eulerian formulation to solve the steady state heat equation. In order to obtain analytical solutions, several significant assumptions had to be made: 1) quasi-stationary state, 2) the geometry must be prismatic and the arc must travel along the axis of the prism, and 3) temperature-independent material properties. If the assumptions were satisfied, the solution could be exact. However, the assumptions were not true in most welds and therefore the
solutions were usually in error. The accuracy of an analytical solution was considerably low in or near the fusion zone and HAZ. In the regions not too close to the arc, Rosenthal's solution could give reasonably accurate results for simple weldment such as bead-on-plate.

It is now common practice to discard Rosenthal's assumptions and turn to numerical methods. By adopting a coordinate system fixed on the workpiece and solving the nonlinear transient heat equations, most of the assumptions can be dismissed and better simulation of the temperature distribution near the arc can be obtained. The numerical models replacing Rosenthal's analytical model treat the arc as a finite heat source. In these models, the heating effect of the arc is modeled as applied heat flux or distributed power density in the weld pool to reflect more accurately the digging action of the arc.

Pavelic et al. proposed a Gaussian distribution of flux deposited on the surface of the workpiece [29]. In Pavelic's model, the thermal flux was distributed in a circular disc and was described by the following equation:

\[
q(r) = q(0) \cdot e^{-Cr^2}
\]  \hspace{1cm} \text{Equation 3.2}

where \(q(r)\) is the heat flux at radius \(r\); \(q(0)\) is the maximum heat flux at the center of the arc; \(C\) is the concentration coefficient; and \(r\) is the radial distance from the center of the arc to the point of calculation.

Equation 3.2 is valid for a stationary arc. The concentration coefficient \(C\) is related to the arc width. Tsai modified Pavelic's model with a skewed Gaussian distribution of flux in which the arc column was distorted backwards to
reflect the effect of welding speed [118]. Andersson used a piecewise constant flux over the weld pool [48]. Westby distributed the power density as a constant throughout the weld pool [119].

Goldak et al. proposed a non-axisymmetric three-dimensional heat source model in which the power density was dispensed with a Gaussian distribution in a double ellipsoid representing the weld pool [120]. Figure 3.8 displays the power density distribution of the arc prescribed by Goldak’s model. The front half of the source is the quadrant of one ellipsoidal source and the rear half is the quadrant of another ellipsoid. The reason of combining two ellipsoidal sources is that the temperature gradient in front of the heat source is steeper than that at the trailing edge of the weld pool. If the arc is moving along the positive direction of the Z-axis as illustrated in Figure 3.8, the power density inside the front quadrant takes the following form:

\[
q_{arc}(x,y,z,t) = \frac{6\sqrt{3f}q_0}{abc\pi^{\frac{1}{2}}} \exp \left\{ -3 \left[ \frac{x^2}{a^2} + \frac{y^2}{b^2} + \frac{(z-z_0-\nu t)^2}{c^2} \right] \right\} \text{ Equation 3.3}
\]

The power density distribution inside the rear quadrant becomes:

\[
q_{arc}(x,y,z,t) = \frac{6\sqrt{3f}q_0}{abc\pi^{\frac{1}{2}}} \exp \left\{ -3 \left[ \frac{x^2}{a^2} + \frac{y^2}{b^2} + \frac{(z-z_0-\nu t)^2}{c^2} \right] \right\} \text{ Equation 3.4}
\]

where \( q_{arc} \) is the power density, \( q_0 \) the energy input rate or heat input as described in Equation 3.1, and \( \nu \) the travel speed. \( a \) is the characteristic dimensional distribution parameter that defines the width (in x direction) in
Flux $q(x,y,z,t)$

which 95 percent of the heat flux is deposited. Similarly, $b$ and $c$ are the characteristic parameters in $y$ and $z$ direction respectively. $a$ and $b$ can be related to half bead width and bead depth (penetration). $f_f$ and $f_r$ are the fractions of the heat deposited in the front and rear quadrant, and $f_f + f_r = 2$. Goldak’s Gaussian distribution is virtually an empirical curve fit to the experimental results. However, experiments have shown that the double
ellipsoid model, which spreads the thermal loads throughout the weld pool, is more accurate than the other heat input models [121,122]. On one hand, the distribution parameters in this model need to be determined experimentally; On the other hand, the introduction of these parameters into the model provides the capability of obtaining good approximations to the heat flux distributions for the thermal analysis. Therefore, Goldak's heat source model is used in this dissertation.

The fundamental governing equation of heat conduction in a three-dimensional model as illustrated in Figure 3.8 can be written as:

\[
\frac{\partial}{\partial t} \left( \rho \cfrac{\partial \theta}{\partial t} \right) + \lambda \left( \cfrac{\partial \theta}{\partial x} \right) + \lambda \left( \cfrac{\partial \theta}{\partial y} \right) + \lambda \left( \cfrac{\partial \theta}{\partial z} \right) + Q = \rho C_p \frac{\partial \theta}{\partial t} \tag{Equation 3.5}
\]

where \(\theta, \lambda, Q, \rho,\) and \(C_p\) are the temperature, thermal conductivity, rate of internal heat generation, density, and specific heat respectively. As mentioned in the beginning of this section, the welding arc is treated as a heat flux distribution instead of thermal boundary conditions. The convective and radiative heat loss on the exposed surfaces of a weldment can also be regarded as volumetric heat generation or loss. By using such prescription, the rate of internal heat generation \(Q\) in Equation 3.5 can be composed as:

\[
Q = q_{arc} + q_i - \frac{\int q_{con} dA}{\int dV} - \frac{\int q_{rad} dA}{\int dV} = q_{arc} + q_i - \frac{\int h(\theta - \theta_e) dA}{\int dV} - \frac{\int \sigma \varepsilon (\theta^4 - \theta_e^4) dA}{\int dV} \tag{Equation 3.6}
\]
where $q_{\text{arc}}$ is the heat flux from the arc as described in Equations 3.3 and 3.4, $q_i$ the latent heat of solidification, $q_{\text{con}}$ the surface heat loss by convection, $q_{\text{rad}}$ the surface heat loss by radiation. Other variables are explained in the NOMENCLATURE. By substituting Equations 3.1, 3.3, 3.4 and 3.6 into Equation 3.5, the governing equation of heat conduction in this study can be rewritten as:

$$
\frac{\partial}{\partial x} \left( \lambda \frac{\partial \theta}{\partial x} \right) + \frac{\partial}{\partial y} \left( \lambda \frac{\partial \theta}{\partial y} \right) + \frac{\partial}{\partial z} \left( \lambda \frac{\partial \theta}{\partial z} \right) + 6\sqrt{3f\pi \rho E_l \eta}\exp\left\{ -3 \left[ \frac{x^2}{a^2} + \frac{y^2}{b^2} + \frac{(z-z_0-\nu t)^2}{c^2} \right] \right\} + q_i \quad \text{Equation 3.7}
$$

$$
\int h(\theta - \theta_r) \, dA - \int \sigma_e (\theta^4 - \theta_r^4) \, dA = \int \int (\rho C_p \frac{\partial \theta}{\partial t}) \, dV
$$

3.4.2 Thermophysical Properties

Thermal analysis requires some basic thermophysical properties including density, thermal conductivity, specific heat and latent heat. Latent heat with the liquidus and solidus temperatures need to be specified if the effect of heat fusion is modeled. In this study, the effect of weld dilution on the property changes is ignored. The materials are assumed to be homogeneous and isotropic. Material properties such as thermal conductivity and specific heat are temperature-dependent since a temperature range from room temperature to melting point and above is experienced during welding. If the material
properties are prescribed as temperature-independent, it causes substantial error.

Note that the material of Butt joint is designated as Steel-A and Tee joint Steel-B. Figure 3.9 shows the temperature-dependent properties and Table 3.2 lists the temperature-independent properties for the two steels. The properties are collected from various sources [123,124,125,126]. The thermal conductivities of both steels above their respective melting temperatures are artificially increased to compensate for the convective heat transfer effect caused by the fluid flow in the weld pool. As shown in Figure 3.9, the thermal conductivity of the steel is enhanced linearly from the solidus to liquidus and remains constant above the liquidus. It is suggested by many researchers that the conductivity at the liquidus be 2~5 times the value at the solidus [99,127,128]. An enhancement factor of 3 is adopted in this study.

<table>
<thead>
<tr>
<th></th>
<th>Density (g/mm³)</th>
<th>Latent Heat (J/g)</th>
<th>Solidus (°K)</th>
<th>Liquidus (°K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel-A</td>
<td>7.82 x 10⁻³</td>
<td>275</td>
<td>1750</td>
<td>1800</td>
</tr>
<tr>
<td>Steel-B</td>
<td>7.82 x 10⁻³</td>
<td>247</td>
<td>1728</td>
<td>1827</td>
</tr>
</tbody>
</table>

Table 3.2: Temperature-independent properties used in the analysis
Figure 3.9: Temperature-dependent thermophysical properties used in the thermal analysis. (a) Thermal conductivity; (b) Specific heat.
3.4.3 Finite Element Implementation and Experimental Verification

Some essential modeling issues for the finite element analysis, such as discretization of the geometry and specification of the material properties have been elaborated on in the previous sections. Current discussion of the finite element implementation is focused on the numerical analysis procedure itself, especially the implementation issues related to the analysis code ABAQUS utilized in this study.

The core of the modeling effort is in the user subroutine DFLUX for ABAQUS code, which allows the simulation of welding arc as a heat flux moving along the joint. The double ellipsoidal heat flux distribution equations (Equations 3.3 and 3.4) are programmed into DFLUX subroutine. The arc position is calculated according to the elapsed welding time and the heat flux is computed at each integration point according to its location with respect to the arc position. The heating effect of the arc is integrated over time and space to yield the total heat input. The arc voltage, current, travel speed and arc efficiency are the input parameters of the subroutine. They are given in section 3.2 for the Butt and Tee joints investigated in this chapter. It is also required that the geometry of the double ellipsoidal heat source be specified in the subroutine.

The heat source geometry is defined by the characteristic distribution dimensions a, b and c as illustrated in Figure 3.8. Strictly, these parameters are not known before the solution is rendered. However, imagine that the exact temperature field is known. Prescribe the temperature field to the heat equation
and compute the residuals, which are effectively the flux and power density satisfying the heat equation. This is the flux and power density model that can produce the exact temperature field. Since the exact temperature field is not known, the best available approximation should be used. This could be based on experimental data. The approach is followed to determine $a$, $b$ and $c$ values. Since $a$ and $b$ are interpreted as half bead width and penetration respectively, they are determined from weld cross sections or bead shapes.

One cross section of the Tee joint is depicted in Figure 3.10a. The measurement of $a$ and $b$ are labeled in this figure. Parameter $c$ is assigned equal to $a$ in the front quadrant and $4a$ in the rear quadrant. A preliminary parametric study, in which $c$ varies from $a$ to $10a$, reveals that $c$ has insignificant effect on transient temperature field as long as the travel speed is not abnormally low. The travel speeds of the welding processes for the Butt and Tee joints are 3.81 and 5.93 mm/second respectively. Because they are high enough, the influence of travel speed on heat transfer is dominant over the influence of $c$. Figure 3.10b shows the computed bead shape when measured $a$ and $b$ values are the input in the DFLUX subroutine. In this work, the arc efficiency is initially assumed to be 80% and applied to the FE model. The efficacy is then adjusted to 90% to obtain good agreement between analysis results and thermocouple measurements. Some important input parameters of the heat source model are listed in Table 3.3.

The thermal boundary conditions are stipulated in the ABAQUS input file via the commands `*FILM` and `*RADIATE`. In theory, the convection heat
transfer coefficient should be a function of the surface temperature and the distance from the electrode because the shielding gas exists in the GMA welding process. However, the effect of convection and radiation is secondary compared with the heat input from the arc. Therefore, the convection heat transfer coefficient has a constant value of 84 W/m²-K [129]. The surface radiative emissivity is 0.3 [130].

Comparison of the experimental and computed thermal histories in the Tee joint is given in Figure 3.11. The experimental data are acquired from the embedded thermocouples of which locations are demonstrated in Figure 3.5. The predicted temperatures are extracted from the nodal points in the model,

<table>
<thead>
<tr>
<th>Joint</th>
<th>a (mm)</th>
<th>b (mm)</th>
<th>η (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tee Joint</td>
<td>4.50</td>
<td>6.00</td>
<td>90</td>
</tr>
<tr>
<td>Butt Joint</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>First pass</td>
<td>5.56</td>
<td>6.35</td>
<td>90</td>
</tr>
<tr>
<td>Second pass</td>
<td>5.56</td>
<td>3.18</td>
<td>90</td>
</tr>
<tr>
<td>Third pass</td>
<td>7.65</td>
<td>3.62</td>
<td>90</td>
</tr>
<tr>
<td>Forth pass</td>
<td>7.65</td>
<td>3.62</td>
<td>90</td>
</tr>
</tbody>
</table>

Table 3.3: Critical parameters of the double ellipsoidal heat source model realized through DFLUX subroutine
Figure 3.10: Bead shape of the DH36 Tee joint. (a) Experiment; (b) FEA.
Figure 3.11: Comparison of thermal histories in the Tee joint between experimental and finite element results. The experimental data are labeled as “thermocouple X” and the predicted are designated as “FEM-TCX”, where X represents the ID number of a thermocouple.
which are placed in the same positions as the thermocouples. The maximum difference between the measured and predicted temperature profiles is about 50°C, which is considered reasonably small error for weld thermal analysis. The accuracy of the thermal analysis is also reflected in the similitude between the predicted and actual bead shapes, as shown in Figure 3.10. In summary, the finite element results are well correlated to the measurements.

3.5 Mechanical Analysis and Experimental Verification

As described in Section 3.1.1 and 3.1.2, decoupled thermo-mechanical analysis is adopted in this study and carried in two parts. The first part performs a transient heat transfer analysis and computes the thermal history of the entire weldment. The second part then uses the thermal history as part of the input information and performs a transient thermo-elasto-plastic analysis in order to compute the distortions, strains and stresses in the weld. The second part is referred to as mechanical analysis, in which the mechanical field is coupled to the temperature field through the temperature dependent constitutive properties and the thermal strain.

The mechanical analysis computing the stress and strain fields is a very important aspect of the computational weld mechanics. Localized heating of the welding arc causes the nonlinear distribution of temperature field in a weldment. The uneven temperature filed generates the transient thermal stress and deformation, which turn into the residual stress and distortion after welding.
is completed. The thermal strain field drives the stress field developed in welding. The computational weld mechanics deals with welding thermal stress/strain problems using the principle of continuum solid mechanics. An exact solution to the thermal stress/strain problem satisfies three basic laws of the continuum solid mechanics: 1) the conservation of linear momentum and angular momentum, or in terms of mathematical description, the equilibrium equation; 2) the constitutive relationship between stress and strain, or the constitutive equation; 3) the relationship between strain and displacement, or the compatibility equation (also called continuity equation). The solution also satisfies the initial and boundary conditions.

The solution of the mechanical analysis is executed through the finite element analysis code ABAQUS. ABAQUS is a general-purpose finite element program that has employed displacement formulation for solving structural problems. In displacement-based FEA, nodal displacements are primary unknowns and strains/stresses are secondary unknowns. The solution procedure conforms to the basic laws of solid mechanics stated above and can be concisely described as the following: the governing equations are expressed in terms of nodal displacements using the equations of equilibrium and an applicable law relating forces to displacements. The displacements are solved first, and then the strains are obtained using the strain/displacement relationship, and finally the stresses are solved based on the stress/strain relationship.
The mechanical behavior of weldment is greatly affected by the high-temperature material behavior since the weld and HAZ undergo heating and cooling over a large temperature range spreading from room temperature to melting point and beyond. The constitutive relationship between welding stresses and strains are highly temperature-dependent. The strong temperature dependence can be attributed to two major factors: 1) the high-temperature material behavior associated with melting and solidification; and 2) the variation of material properties with temperature either at high temperatures or relatively low temperatures. It is critical to apply the suitable constitutive model to the mechanical analysis in order to precisely simulate the weld mechanical behavior.

3.5.1 Effects of High-Temperature Material Behavior

The high-temperature material behavior, which greatly affects weld distortion and is investigated in this study, includes the phenomena related to weld melting and solidification, namely plastic strain relaxation and filler metal deposition. They dictate the plastic behavior and rigidity upon melting and solidification. Their effects on weld mechanical behavior and their numerical treatments in the mechanical analysis are presented in the following context.

Plastic Strain Relaxation

Upon melting, the material’s virgin strain-free state is recovered in that all the strains including the plastic strain formed in the weld metal are “annihilated"
(relaxed). If plastic strain relaxation is not properly accounted in the FEA, false plastic deformation will be generated in the FEA model. The predicted weld distortion by such FEA has been found to be much smaller than the magnitude of the experimentally measured distortion [76]. This is because of the existence of compressive plastic strains in the weld metal created during the heating cycle of the welding process in the FEA model. In reality, these heating-induced plastic strains are relaxed when the weld metal becomes molten liquid. The physically, non-existing compressive plastic strains in the FEA model results in the prediction of smaller weld distortions.

To illustrate the plastic strain relaxation phenomenon, it is informative to examine a one-dimensional problem as illustrated in Figure 3.12. It is a simple example of heating and cooling a bar while it is rigidly restrained. Assuming linear temperature dependence of material properties and a linear elastic-perfectly plastic behavior with an isotropic hardening, as shown Figure 3.13, the strain and stress history can be readily computed. The material properties are linearly interpolated between the two prescribed temperatures, \( \theta_1 \) (1293\( ^{0}\)K) and \( \theta_2 \) (1593\( ^{0}\)K). Both yield stress and Young’s Modulus are assumed to be very small at temperatures greater than \( \theta_2 \). It is also assumed that the stress-strain relationship remains the same for temperatures lower than \( \theta_1 \).

Figure 3.14 displays the heating and cooling history. The bar is heated up linearly from room temperature (293\( ^{0}\)K) to \( T_{PSR} \), which is the temperature for the plastic strain relaxation. The plastic strain relaxation occurs at \( T_{PSR} \), which is
set artificially at 1793°K in this example. The bar is then cooled back to room temperature linearly. Since both ends of the bar are rigidly fixed, the total strain \( \varepsilon^{\text{total}} \), which consists of three components: elastic \( \varepsilon^e \), plastic \( \varepsilon^p \) and thermal \( \varepsilon^T \), is kept zero during heating and cooling, as described in the following equation.

\[
\varepsilon^{\text{total}} = \varepsilon^e + \varepsilon^p + \varepsilon^T
\]

Equation 3.8

From Equation 3.8, the plastic strain component can be expressed by the following equations:

\[
\varepsilon^p = 0 \text{ and } \varepsilon^e = -\varepsilon^T \text{ when } \varepsilon^e \leq \text{yield strain} \quad \text{Equation 3.9}
\]

\[
\varepsilon^p = -(\varepsilon^e + \varepsilon^T) \text{ when } \varepsilon^e > \text{yield strain} \quad \text{Equation 3.10}
\]

\[
\varepsilon^p \approx -\varepsilon^T \text{ and } \varepsilon^e \approx 0 \text{ when temperature reaches } \theta_2 \quad \text{Equation 3.11}
\]

\[
\varepsilon^p \approx \varepsilon^e = 0 \text{ when plastic strain relaxation occurs} \quad \text{Equation 3.12}
\]
Figure 3.12: A bar with both ends fixed between two rigid walls

Figure 3.13: Temperature-dependent stress/strain curve

θ < θ₁
θ > θ₂
The comparison of plastic strain histories is given in Figure 3.15. A very different plastic strain history is obtained when plastic strain relaxation is taken into account and the material’s virgin strain-free state is recovered. Plastic strain relaxation leads to a tensile residual plastic strain whereas a compressive residual plastic strain is produced without considering the plastic strain relaxation phenomenon.
Filler Metal Deposition

The filler metal is continuously deposited into the weld joint in the welding processes that consume filler metal, such as GMAW. Continuous filler metal deposition induces the accumulated rigidity as the solidification front advances with the weld pool. It needs to be properly simulated in the mechanical analysis, otherwise false plastic deformation and weld distortion will be generated. Figure 3.16 presents the predicted in-plane distortion in a Butt joint. The details of the analyses are given in the next chapter. In the analysis with
filler metal deposition simulated (Figure 3.16b), the in-plane rotation is an apparent distortion mode in addition to the transverse shrinkage, which is frequently observed in actual weld joints. The in-plane rotation is primarily caused by the varying rigidity of the solidified weld as a result of continuous filler metal deposition. However, the analysis ignoring the effect of filler metal deposition (Figure 3.16a) cannot predict in-plane rotation. In fact, filler metal deposition was not simulated in some reported studies for the purpose of reducing the complexity of computation and the computational cost. In these studies, filler metal deposition was simply ignored by leaving the finite elements representing the filler metal in the FEA solution domain all the time. Under such treatment, the filler metal exists in the weld joint even before welding. The artificially high stiffness of the filler metal prior to its deposition disturbs the deformation patterns in the adjacent region and consequently causes erroneous plastic deformation and distortion prediction [131].

The numerical techniques that have been utilized to simulate continuous filler metal deposition include element rebirth [66,67,132,133,134] and assigning an appropriate material behavior to the filler metal [72,111,135]. Element deactivation and reactivation are utilized in the element rebirth technique. Selected elements are deactivated before certain instant and then reactivated thereafter. Two main numerical schemes have been attempted to implement the element rebirth: instantaneous and gradual element reactivation. In the instantaneous element reactivation scheme, all the elements in one weld pass are reactivated at once immediately before the pass is deposited. The
instantaneous reactivation scheme is straightforward and can be executed with relative ease. It is frequently adopted in two-dimensional analyses. In the gradual element reactivation scheme, the elements representing the filler metal are individually reactivated either at a predetermined temporal sequence or upon reaching certain critical temperature threshold such as the melting point. The gradual reactivation scheme is physically more appropriate than the instantaneous scheme in that the gradual scheme can be utilized to simulate the continuous filler metal addition as the welding arc travels. It is mainly used in three-dimensional analyses.

There are implementation and accuracy issues associated with the element rebirth technique. From implementation standpoint, it is very tedious and difficult to apply the rebirth procedure in three-dimensional models [67], especially when the gradual reactivation scheme is used. From the perspective of accuracy, numerical difficulties may be encountered upon element reactivation. For example, numerical divergence arises due to the strain mismatch between the deformed elements representing the already-deposited filler metal and the strain-free elements that are being reactivated and represents the metal just added to the weld.

Constituting the material behavior of filler metal is a better alternative to the element rebirth technique in terms of accuracy and ease of implementation. The elements strand for the filler metal are present in the FEA solution domain throughout the mechanical analysis. However, the elements are given very low stiffness such as that of the air before the filler metal is deposited. The
actual stiffness is assigned to the elements when the moving weld pool encompasses them. The algorithm of changing material behavior is employed to simulate continuous fill metal deposition in this dissertation. The algorithm is realized through coding the material property user subroutine, which will be described in details in the subsequent section 3.5.3 about the FE implementation.
Figure 3.16: Computed in-plane distortion in a Butt joint. (a) Filler metal deposition not simulated in the analysis; (b) Filler metal deposition simulated in the analysis.
3.5.2 Material Properties

Note that the material of Butt joint is designated as Steel-A and Tee joint Steel-B. The material properties used for the mechanical analysis include elastic modulus, Poisson's ratio, stress versus plastic strain curves, and thermal expansion coefficient. These mechanical properties are temperature-dependent. They were collected from various sources in the open literature [74,123,124,125,126]. The temperature dependency of elastic modulus and Poisson's ratio is presented in Figure 3.17. The elastic moduli of the two steels decrease as the temperature increases. There are critical temperatures at which the moduli decrease to a very low value (10 GPa). The critical temperature for Steel-A is $1173^0\text{K}$ and $1273^0\text{K}$ for Steel-B. The elastic moduli are kept at the low value throughout the temperature range above the critical temperatures. The low magnitude of the moduli is set to reflect the low strengths of the steels at high temperatures, including the near-zero strength of the molten weld pool. The non-zero value of the moduli is utilized to avoid numerical convergence difficulties, i.e. singularity in assembling the stiffness matrix. The phenomenon can be found in the yield stresses plotted in Figure 3.18, which shows the variations of yield stress and thermal expansion with temperature. The yield stress of Steel-A diminishes to a low magnitude of 10 MPa at $1173^0\text{K}$ and remains unchanged above the critical temperature. The same tendency appears in the yield stress curve of Steel-B with the critical temperature of $1273^0\text{K}$. 

101
An important ingredient of the mechanical analysis is the constitutive equations governing the mechanical behavior of weldment. In this study, the mechanical constitutive behavior is assumed to be rate-independent elasto-plastic, obeying the Von Mises yield criteria and associated flow rules. The constitutive relations are defined by the stress versus plastic strain curves at different temperatures, as shown in Figure 3.19.

3.5.3 Finite Element Implementation

The mechanical analyses on the Butt and Tee joints are undertaken using the general purpose code ABAQUS. The mechanical analysis is transient and nonlinear and the implementation procedure in ABAQUS is established and followed to satisfy the requirements of this type of analysis. The main components of the analysis procedure include setup of initial and boundary conditions, specification of loading conditions, treatment of high-temperature material behavior via user subroutines. The initial conditions of the model are specified using the *INITIAL CONDITIONS command in ABAQUS.

The initial temperature of the entire model is set at room temperature and the model is stress-free at this temperature. Free boundary conditions that prevent rigid body motion are applied to the model of Butt joint. In addition to the free boundary conditions, symmetry boundary conditions are applied at the symmetry plane of the Tee joint model, indicated in Figure 3.7a.
Figure 3.17: Temperature-dependent elastic modulus and Poisson’s ratio. (a) Steel-A; (b) Steel-B.
Figure 3.18: Temperature-dependent yield stress and linear thermal expansion. (a) Steel-A; (b) Steel-B.
Figure 3.19: Stress-strain curves at different temperatures. (a) Steel-A; (b) Steel-B.
As stated earlier in this chapter, the mechanical analysis is the sequel to the thermal analysis as the welding process simulation is accomplished using the decoupled thermo-mechanical analysis procedure. The critical link between the thermal and mechanical analyses is the use of the thermal history from the thermal analysis as the load in the mechanical analysis. The link is established through assigning the loading conditions in the mechanical analysis. In the thermal analysis, the computed thermal histories at all nodes are recorded and stored in a temperature results file. In the mechanical analysis, the *TEMPERATURE command is used to read the temperature results file and map the thermal histories onto the nodes in the mechanical model. The mapped nodal temperature fields at different times serve as the transient loading conditions in the mechanical analysis.

ABAQUS user subroutine UMAT is used to compute the constitutive material behavior, namely defining the temperature-dependent but rate-independent elasto-plastic mechanical properties, as described in section 3.5.2. At each time increment during the entire welding cycle, the UMAT subroutine is called at each element integration point of the model to determine the temporal evolution of the material properties in the entire solution domain. The user subroutine is also used to deal with the issues associated with welding stress analysis, which cannot be readily handled by the general purpose commercial FE programs such as ABAQUS. These issues include continuous filler metal deposition and annihilation of plastic strain upon melting.
At each time increment, the algorithms for continuous filler metal deposition and plastic strain relaxation are executed at each element integration point in the FE model. The elements that represent the weld bead always exist during the mechanical analysis. However, among these elements, the ones standing for unmolten filler metal are given a set of artificially very soft properties. An element in the weld bead is judged belonging to unmolten filler metal if its experienced peak temperature is lower than the melting temperature. The element is switched back to the actual properties as soon as its temperature is reaching the melting temperature, or in other words, when it is on the verge of the approaching weld pool. In order to avoid numerical convergence problems, the switching is done based on the temperature at integration point, not on the location relative to the center of the welding arc. The reason is that the size of the weld pool varied during welding because of the initial and terminal transient temperature stages at the weld ends. The algorithm of switching properties enables the simulation of continuous filler metal deposition.

The algorithm of plastic strain relaxation is applied to all the elements in the model and executed in the following manner: whenever the temperature within an element is higher than the melting point, the corresponding strain components including elastic, thermal, and plastic components are set to zero. As to the element in the weld bead, the unrealistic plastics strains accumulated prior to the property switching are eliminated within the increment in which the switching takes place.
3.5.4 Experimental Verification

The accuracy of the mechanical analysis is verified by comparing the FEA and experimental results. In addition to the comparison made between the final distortions from the analyses and experiments, the corresponding distortion histories are examined side by side. It is obvious that more useful data such as plastic strain history can be exacted from the FE model if it is capable of accurately predicting the whole weld distortion process. It should be noted that strain components including plastic strains are among the analysis results although they are not presented here.

Both experimental measurements and FE analysis reveal that the dominant distortion mode of the Butt joint is in-plane transverse shrinkage. The out-of-plane distortion is trivial due to the double-V groove geometry and the strong constraint imposed by the restraining block. Figure 3.20 displays the distorted shape of the Butt joint after weld pass 1 is deposited. The computed shrinkage at the end of the weld is more severe than the weld start. The analysis also predicts that the restraining bars bow outwards, as indicated in Figure 3.20. Both phenomena were observed in the experiments [74].

Eight dowel pins were embedded on both top and bottom sides of the plate for measuring the transverse shrinkage. The location of the dowel pins is shown in Figure 3.21. The gauge pins were positioned 25.4 mm from the start and end of the weld. Before and after each weld pass, digital calipers were instrumented to gauge the distances between the pins across the weld, including four measurements between pins $A_1$ and $A_2$, $A_3$ and $A_4$, $B_1$ and $B_2$, 

108
and $B_3$ and $B_4$. The measured distances were then used to calculate the pin separations after each pass. Obviously, the pin separations were in contraction. The in-plane shrinkage was obtained by averaging the measured pin separations.

The shrinkage measurements for four passes are plotted in Figure 3.22 along with the predicted distortion history from the FE analysis. The horizontal axis of the chart in Figure 3.22 is labeled “Pass Number”. It denotes the progression of the weld passes and is also a normalized scale of time. One unit on the axis is equivalent to the 800-second heating and cooling period experienced by each pass in the analysis. Note that there are two “dips” on the shrinkage history curve at the starts of pass 3 and 4 respectively. They correspond to the thermal expansions during heating while the passes are being deposited. There is a very good agreement between the predicted and measured shrinkage values at the end of each pass. The prediction errors are 7.8%, 3.1%, -0.4%, and 3.7% for pass 1 to 4 respectively. The negative error for pass 3 indicates the under-prediction. The FE model is apparently capable of capturing the cumulative shrinkage as the weld passes are added to the groove.
Figure 3.20: Distorted Butt joint after pass 1. The distortion is magnified by 100.
Figure 3.21: Location of the dowel pins for measuring transverse shrinkage. $A_1$, $A_2$, $A_3$, and $A_4$ are the embedded dowel pins on the top side of the plate; $B_1$, $B_2$, $B_3$, and $B_4$ are the embedded dowel pins on the bottom side of the plate. Dimensions are in mm.
The primary distortion mode of the Tee joint is angular distortion, as evidenced in the experimental data reported in References 75 and 112. Angular distortion is also clearly manifested in the FE prediction, as exhibited in Figure 3.23. The figure displays the distorted shape of the section that is perpendicular to the weld and through the middle point of weld length. The
angular distortion in Figure 3.23 is reflected in the out-of-plane displacements of the base plate.

The out-of-plane displacements were measured using LVDT’s during experiments. The computed and gauged out-of-plane displacement histories are compared at three LVDT locations designated as LVDT1, LVDT2, and LVDT3 in Figure 3.5a. In order to monitor and log the most amount of out-of-plane displacement, the three LVDT’s were positioned as far away from the weld as possible. They were placed about 171.5 mm away from the weld root and exactly 177.8 mm away from the middle-thickness plane of the stiffener. As to the position along the welding direction, LVDT1 and LVDT3 are close to the end and start of the weld respectively, and LVDT2 is pointing to the middle of weld length.

Figure 3.24 gives the comparison of the calculated and measured displacement histories at LVDT1. In the analysis, the displacement history is recorded at the node where LVDT1 is placed. The recorded displacement transients are written to an ABAQUS output file that is undergone data processing after analysis. The “FEA” curve in Figure 3.24 is generated on such data. In the experiments, two welding trials of the Tee joint were conducted for the same welding conditions stated in the section describing the experimental procedure (Section 3.2). The welded specimens from the two laboratory trials are designated “Specimen A” and “Specimen B”. The experimental data collected from the two specimens are labeled correspondingly in Figure 3.24.
The horizontal axis in Figure 3.24 stands for the actual welding and cooling time. The vertical axis is the scale of out-of-plane displacement. A positive value on the vertical axis means the base plate moves upwards as illustrated in Figure 3.23; a negative value on the axis indicates a downward movement. The same trend is reflected in both analysis and experimental results that are plotted in Figure 3.24. As welding starts, the base plate sinks down, and then bends up as cooling begins. When the cooling continues, the upward out-of-plane displacements rapidly accumulate and rise to their saturation levels. The saturation occurs at about 100 seconds after welding starts. The agreement between the analysis and experiments is apparent in terms of moving trend of data. Such agreement can also be found in the comparisons of the computed and measured displacement transients at LVDT2 and LVDT3. The comparisons at LVDT2 and LVDT3 are presented in Figure 3.25 and Figure 3.26 respectively.

In addition to the same moving trend of data, the FE analysis is verified by the comparable distortion magnitudes from the analysis and experiments. Note that the “FEA” curve falls in between the two curves of the measured data when the distortion stabilizes in terms of magnitude, as observed in Figure 3.24, Figure 3.25, and Figure 3.26. The comparison of the final distortion magnitudes is given in Figure 3.27, in which the final out-of-plane displacements are plotted against the three LVDT locations. The calculated distortion magnitudes are very close to the average values of the measurements.
Figure 3.23: Distorted shape of the middle section of the Tee joint. The distortion is magnified by 60.
Figure 3.24: Comparison of predicted and measured out-of-plane displacements at LVDT1. The experimental data were acquired through LVDT1 in two welding trials represented by Specimens A and B.
Figure 3.25: Comparison of predicted and measured out-of-plane displacements at LVDT2. The experimental data were acquired through LVDT2 in two welding trials represented by Specimens A and B.
Figure 3.26: Comparison of predicted and measured out-of-plane displacements at LVDT3. The experimental data were acquired through LVDT3 in two welding trials represented by Specimens A and B.
3.6 Summary

In this study, a FE model has been established and used to compute welding-induced thermal profiles as well as weld distortion during and after welding. The FE model, namely the moving source model or decoupled
thermo-mechanical analysis has been applied to both Butt and Tee joints. The applicability and accuracy of the FE model have been validated against the experimental measurements in temperature and distortion histories. It has been proved that the FE model can accurately predict both in-plane shrinkage and out-of-plane angular distortion. Since the primary distortion mode investigated in this dissertation is in-plane shrinkage, the study of the Tee joint or prediction of angular distortion is only for the purpose of verifying the FE model.

Since the FE model is verified by the experiments, it can be used as the substitute of laboratory experiments, especially when evolution of the welding plastic strains is the research subject. The plastic deformation obtained from such FEA is deemed as the actual plastic deformation induced by welding. Indeed, the FE model developed in this chapter is utilized to conduct numerical welding experiments in the following chapters.
In this chapter, in-plane shrinkage — the simplest and most fundamental type of welding distortion is studied. The aim of this chapter is to investigate the welding-induced plastic strains and its relationship with in-plane shrinkage. The analyses are conducted to address several important aspects of the plastic deformation as related to the shrinkage distortion and its prediction, including

1) estimating the magnitude and distribution of the plastic strains,
2) investigating how the plastic strains determine the in-plane shrinkage, and
3) finding the critical factors that control the plastic strains.

Three types of simple weld joints are utilized here to study the characteristics of in-plane shrinkage plastic strains, namely butt-welded plates, Tee joint, and plate with slot weld. The fact that in-plane shrinkage and plastic strains are obviously affected by joint constraint led to the concern if the plastic strains in the simple joints are the same as or similar to those in large thin-wall structures. The constraint in the butt-welded plate arises from the accumulated rigidity of solidifying weld since continuous filler metal deposition is simulated in the analysis. In other words, its constraint is internal and self-induced by weld solidification. As to the other two types of simple joints, the weld lengths are
much smaller than the joint dimensions and therefore the joint constraints are mainly geometrically induced and similar to those in large structures.

The FEA model formulated and verified in Chapter Three is utilized here to conduct numerical welding experiments in lieu of laboratory ones. The plastic strains obtained from FE simulations are deemed as the actual plastic strains induced by welding. All three simple joints are assigned the material properties of Steel-A, which has also been presented in Chapter Three.

4.1 FE Model and Parametric Study Matrix of Butt-Welded Plates

Figure 4.1 shows the dimensions of the joint. Both groove geometry and weld bead configuration are also illustrated in the figure. The joint length is 304.8 mm. The plate thickness is 2 mm. The weld groove has the shape of rectangular gap and the weld bead also has a rectangular shape under the assumption of perfectly full penetration in the FE model. The assumption that the fusion zone distributes evenly in the thickness direction eliminates any angular distortion caused by a usual nugget-shape weld bead and excludes the influence of angular distortion on in-plane shrinkage. The assumption makes it possible to concentrate this investigation solely on in-plane shrinkage.

It should be pointed out that the welding experiments referred to in this chapter are numerically simulated “experiments” in computer rather than laboratory welding trials. Since it is extremely difficult, if not impossible to quantify the plastic behavior experimentally, a numerical simulation stands out
as a good alternative. The numerical welding simulations are constructed to approximate the practical welding experiments as closely as possible. As shown in Figure 4.1b, the width of weld groove is designated by $W_g$. $W_g$ varies with different welding conditions and it is 3.175 mm when the following welding conditions are used:

- Arc voltage: 24 Volts,
- Welding current: 280 Amperes, and
- Travel speed: 8.47 mm/second.

Figure 4.2 illustrates the general features of the three-dimensional FE mesh used for the numerical simulations. Since the butt joint is symmetric about the weld centerline, only one half the joint is modeled with symmetry boundary conditions. This allows a reduction of one half the number of elements and more than one half the computing time. The symmetric half model can allow errors in the analysis of multi-pass welds, especially in the weld area where the sequence and position of weld passes are not exactly symmetric about the weld centerline. Nevertheless, half model is accurate enough for a single-pass butt joint [136]. The 3D FE model consists of 5022 elements and 10412 nodes. Diffusive heat transfer elements, which are 8-node linear brick elements and designated as DC3D8 in ABAQUS, are used in the thermal analysis; stress/displacement elements, which are 8-node linear brick elements with reduced integration and designated as C3D8R in ABAQUS, are employed in the mechanical analysis.
Figure 4.1: One pass square-butt joint on 2-mm thick plates. (a) Joint dimensions; (b) Weld bead configuration.
Note that, fine elements are used in the fusion zone and HAZ to capture the drastic temperature changes in these areas. In order to avoid any out-of-plane distortion and obtain perfectly full penetration, the FE model is formulated in a particular fashion so that even distributions of the temperature, stress and strain in the thickness direction are obtained. Such formulation, which will be explained later, rationalizes the use of just one layer of elements through thickness in the model, as illustrated in Figure 4.2b. The darker shaded elements represent the weld/filler metal in the figure.

In the thermal analysis, the heat input from the welding arc is applied as a volumetric heat source, where the heat generated by welding is simulated with a power density moving along with the torch. The power density has a Gaussian distribution, which is similar to the double ellipsoidal heat source model by Goldak et al. [120,137]. Assuming two ellipsoidal functions for arc heat distribution ahead of and behind the arc respectively, the heat input that is calculated from products of arc efficiency, arc voltage, and welding current is applied to the plate. Another assumption is that the power density along the boundary of the ellipsoids equals to 5 percent of the maximum value at the arc center. If the arc is moving along the y-axis, then the ellipsoidal heat function in the plate takes the following form:

$$ q(x,y,t) = \frac{3f\eta El}{\pi a(\beta a)H} \exp\left\{-3\left[\frac{(x-x_0)^2}{a^2} + \frac{(y-y_0-vt)^2}{(\beta a)^2}\right]\right\} $$  \hspace{1cm} \text{Equation 4.1}$$

The physical meaning of the terms in Equation 4.1 are provided below:
• $q(x, y, t)$ is the power density in W/mm$^3$,
• $x$ and $y$ are the global coordinates in mm. Note that the directions of $x$, $y$, and $z$ correspond to 1-, 2-, and 3-axis directions in Figure 4.2a. $x$ is in the direction transverse to weld and $y$ is along the weld axis,
• $t$ is the time elapsed after arc initiation and its unit is second,
• $f$ is the weight factor and equals to 1.4 behind the arc and 0.6 ahead of the arc,
• $\eta$ is arc efficiency and 90 percent is used in this study,
• $E$ is the arc voltage in Volt,
• $I$ is the welding current in Ampere,
• $H$ is the plate thickness in mm,
• $x_0$ and $y_0$ are the coordinates of the arc starting point and their unit is mm, and
• $v$ is the arc travel speed in mm/s.

$a$ and $\beta a$ are the characteristic distribution parameters that defines the region in which 95 percent of the heat flux is deposited. The value of $a$ is 4.8 mm in this study. $\beta$ is the lag factor that determines the slope of the heat distribution function in y-direction (welding direction). It takes the values of 4.0 behind the arc and 1.0 ahead of the arc respectively. The lag and weight factors are calibrated with published experimental data [77].

Although 3D FE model is used in this study, the heat flux exerted on the plate has a 2D distribution. The heat function in Equation 4.1 gives a 2D
distribution in the x-y plane and keep the energy constant through the thickness direction, which is the 3-axis direction in Figure 4.2a. In fact, the heat function defined by Equation 4.1 is just a derivation of that defined by Equations 3.3 and 3.4 in Chapter Three. The heat distribution in Chapter Three is 3D while it is uniform through the plate thickness in this Chapter. Therefore, the heat energy distribution can be more accurately described as a double elliptical disk, which disperses heat flux uniformly in the thickness direction. In Equation 4.1, \( a \) is the length of disk semi-axis in the direction transverse to weld and \( \beta a \) is the dimension of disk semi-axis along the welding direction. As a result of such treatment of the heat energy from the welding arc, the predicted temperature does not change through the thickness, as demonstrated in Figure 4.3. Filler metal addition is not considered in the thermal analysis because the filler metal ahead of the weld pool has negligible effect on heat transfer.
Figure 4.2: General features of the 3D finite element mesh for numerical simulation. (a) Overall mesh; (b) Enlarged view of the mesh near the fusion zone and heat affected zone. The weld starts at the right end of the plate. Only half of the joint is modeled due to symmetry about the weld centerline.
Figure 4.3: Predicted temperature contours surrounding welding arc at 24.75 seconds after arc initiation. Temperature is in Kelvin. The GMAW parameters are 24-V arc voltage, 140-A welding current, and 8.47 mm/s travel speed. Note that temperature is uniform through the thickness.

In the mechanical analysis, symmetry boundary conditions and those preventing rigid body motion are imposed on the model, as plotted in Figure 4.4. The same FE mesh used in the thermal analysis is employed in the mechanical analysis. The analysis is based on the temperature gradients calculated in the thermal analysis, which represent the input information or loading. An elasto-plastic constitutive law is adopted. To simulate the plastic strain relaxation and continuous filler metal deposition, several enhancements are added to the default isotropic elasto-plastic model of ABAQUS, as
discussed in more detail in Chapter Three. They are simply reiterated here for the purpose of clarification. In the simulation, the filler metal is added in the model prior to the analysis STEP. To avoid generating any false plastic deformation, a low value of Young's modulus and low yield strength are assigned to the filler metal. As the torch passes by, the actual material data set is imposed. Furthermore, for peak temperatures exceeding the critical temperature threshold, all strains including the plastic strains are eliminated to simulate annealing. These features are coded on a UMAT user subroutine in ABAQUS.

Welding current is the variable of the parametric study conducted in this chapter. Welding current controls welding heat input that is one of the most influential factor affecting welding distortion and plastic behavior. It is intended to investigate how welding current and heat input affect in-plane shrinkage and plastic strains. The welding parameters included in this study are listed in Table 4.1. The arc voltage and travel speed are kept at constant values of 24 Volts and 8.47 mm/second respectively in all the cases. Both heat input and normalized heat input are given in Table 4.1. Note that the base for normalization is the highest heat input. Since the welding current is the sole variable in this study, a case will be referred to using its current number in the following context. For example, Case 1 will be referenced as “105-A case” or “case of 105 A”.
Table 4.1: Welding parameters used in the parametric study

<table>
<thead>
<tr>
<th>Case Number</th>
<th>E (V)</th>
<th>Travel Speed (mm/s)</th>
<th>I (A)</th>
<th>Heat Input (J/mm)</th>
<th>Normalized Heat Input</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>105</td>
<td>297.64</td>
<td>0.38</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>122.5</td>
<td>347.24</td>
<td>0.44</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>140</td>
<td>396.85</td>
<td>0.50</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>175</td>
<td>496.06</td>
<td>0.63</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>210</td>
<td>595.28</td>
<td>0.75</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>245</td>
<td>694.49</td>
<td>0.88</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>280</td>
<td>793.70</td>
<td>1.00</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 4.4: Boundary conditions in the mechanical analysis
4.2 In-Plane Shrinkage and Plastic Strains in the Butt-Welded Plates

Presented here are the analysis results of single case, the 140-A case. The results of the other cases in the parametric study will be given in next section (Section 4.2.4). In this section, the temperature, displacements, and plastic strains in the butt-welded plate are examined. A transient state of these variables is investigated first, followed by the analysis of the final state of the variables. “Transient state” means the conditions during welding, in other words, it is a snapshot of the variable during welding. “Final state” refers to the conditions at a moment when the joint is completely cooled down. The evolution of the plastic strains is studied through analyzing the plastic strain history.

4.2.1 A Transient State of Temperature and Plastic Strains

A transient state of the temperature and plastic strains is captured 24.75 seconds after arc initiation. Note that the total welding time is 36 seconds. At this instant, the arc location is 209.55 mm from the weld start. Figure 4.3 exhibits the predicted temperature contours surrounding the welding arc. Figure 4.5a shows the eight cross sections where the temperature and plastic strains in the plate are collected. The cross sections are perpendicular to the weld. They are marked with their distances with respect to the arc location. Positive sign indicates the cross section ahead of the arc and negative signs denote the ones behind the arc. The cross sections passing through the arc is labeled “0”.

132
Figure 4.6 displays the temperature distributions in the eight cross sections described above. A quasi-steady state of heating and cooling process in the plate is evident. The statement is based on the observation that the temperature curves at different locations can also be deemed as the temperature profiles experienced by any single cross section at different instants. Figure 4.7a and Figure 4.7b show the transverse and longitudinal plastic strain components respectively. The curves in the eight cross sections are plotted in the same figures. The transverse and longitudinal stress/strain components mentioned in this dissertation are in the direction transverse and parallel to weld respectively. The plastic strains are concentrated in the weld and HAZ. In fact, there are no plastic strains if the distance to weld centerline is greater than 25 mm. Note that the cut-off distance in Figure 4.7 is 10 mm. The plastic strain distributions in Figure 4.7 bear a trend that resembles the quasi-steady state reflected in the temperature profiles in Figure 4.6. Both transverse and longitudinal components are in the deepest compression at the arc. The cross section is in the heating stage and its expansions in both directions are confined by the surrounding materials. The plastic strain relaxation occurs as the arc passes by, as demonstrated in the cross section 19.05 mm behind the arc. The strain components have been annihilated in the fusion zone which is about 2-mm wide from weld centerline. As the arc travels further away, the cross sections left behind are in the cooling stage. The longitudinal strains turn into tension in the weld due to reverse yielding. The transverse strains in the weld develop into compression again.
Figure 4.5: Evaluated cross sections in the plate. (a) For assessing a transient state of temperature and plastic strains; (b) For the final state of the variables.
Figure 4.6: Temperature distributions at different cross sections during welding. The arc is 209.55 mm away from the weld start.
Figure 4.7: Plastic strain distributions at different cross sections during welding. The arc is 209.55 mm away from the weld start. (a) Transverse plastic strain; (b) Longitudinal plastic strain.
4.2.2 Final State of Plastic Strains and Other Variables

From the standpoint of accumulated peak temperature, welding heat transfer reaches the stationary or quasi-stationary state in the middle portion of a long weld with notable end effects at the start and end of the weld. The accumulated peak temperature is the maximum temperature experienced by a material point in the weldment. It will be referred to as peak temperature from this point forward in this dissertation. Does the plastic strain evolution also experience similar quasi-steady state to that of the heat transfer during welding? In order to answer such question, it is necessary to obtain and examine the plastic strain data at several cross sections along the weld.

As shown in Figure 4.5b, the data of the final state are gathered at the seven cross sections with the first cross section 38.1 mm apart from the weld start and a distance interval of 38.1 mm to one another. The seventh cross section is 266.7 mm from the weld start and also 38.1 mm to the weld end. The collected data include strain components and temperature.

Peak Temperatures

Among the data obtained from the analysis, the peak temperature is recorded at each node in the FE model. The peak temperature is very meaningful when considering how far away the welding heat energy can dissipate into the base metal. The isotherms of peak temperature are plotted in Figure 4.8. The isotherms are straight lines throughout most portion of the weld except at both ends, indicating that the quasi-steady state of heat transfer
prevails as the welding arc moves along the groove. It is noticed that the average distance from the 473°C isotherm to weld centerline is 24.5 mm, which is 13.6 percent of the plate half width.

The peak temperature distributions at seven cross sections are sketched together in Figure 4.9. Refer to Figure 4.5b for the location of the cross sections. All curves almost coincide with one another, demonstrating the quasi-steady state of the moving heat source. The peak temperature curves also represent the maximum temperature boundary of all temperature curves experienced during the heating and cooling cycles. The half width of the fusion zone can be measured from the peak temperature curves. The half width is 2.12 mm as the melting temperature is 1700°C.

The final distorted shape of the plate is given in Figure 4.10a. Three types of distortion are apparent in the weld joint: transverse shrinkage, longitudinal shrinkage, and in-plane rotation. They are commonly observed in butt joints. The capability of predicting in-plane rotation can be attributed to the proper simulation of continuous filler metal deposition in the FE analysis. The algorithm and implementation of simulating continuous filler metal deposition is presented in Chapter III. Figure 4.10b illustrates the distorted plate resulted from the FE analysis without simulating filler metal deposition. Transverse and longitudinal shrinkages are present in the plate but not in-plane rotation.
Figure 4.8: Peak temperature isotherms in the plate. (a) Overall distribution; (b) Distribution near weld.
Figure 4.9: Peak temperature distributions at different cross sections
Figure 4.10: Predicted distortions in the plate. (a) Filler metal deposition simulated in the analysis; (b) Filler metal deposition not simulated in the analysis.
Transverse Shrinkage

The final transverse displacements at the seven cross sections are plotted in Figure 4.11. In the chart, the horizontal (x) axis designates the distance to the weld centerline. The vertical (y) axis displays the transverse displacement in negative magnitude that represents shrinkage. Therefore, the data points standing for larger shrinkage are positioned further down along the y-axis. The displacement curves are distinguished from one another by distinctive legends, indicating the distances from the cross sections and weld start. In the following discussion, the displacement curves will be referred to using their distance numbers in the chart legend. For example, 38.1-mm curve is referred to the displacement curve plotted across the cross section that is 38.1 mm away from the weld start.

The displacement curves in Figure 4.11 share some common characteristics. There is no transverse displacement at the weld centerline (x = 0 mm) due to the symmetry boundary condition applied on it. The transverse shrinkage appears in the plate immediately outside the weld centerline. The shrinkage, or precisely, its magnitude abruptly increases in the area where is slightly away from the centerline. The increase in shrinkage magnitude is exhibited as a dip in a displacement curve. At 4.99 mm away from the centerline (x = 4.99 mm), the shrinkage reaches its maximum magnitude which is the bottom of the curve, and then it reverses its course. Beyond certain distance, the shrinkage manifests slow change that is very close to a linear change. Hence, across its width, the plate can be divided into two regions
according to the transverse shrinkage behavior: non-linear shrinkage zone and linear shrinkage zone. The two zones can be partitioned at $x = 24.5$ mm, as shown in Figure 4.11. It is found later in this section that the transverse plastic strain zone in the plate also ends at $x = 24.5$ mm.

Figure 4.11: Transverse displacement distributions at different cross sections
In the non-linear shrinkage zone, the shrinkage varies drastically with its magnitude going up and down. On the other hand, it changes smoothly in the linear shrinkage zone. If judged by shrinkage magnitude, the shrinkage behaviors in the two zones are also different. In the nonlinear zone, the displacement curves can be sorted by the descending order of the shrinkage magnitude as 114.3-, 152.4-, 76.2-, 190.5-, 38.1-, 228.6-, and 266.7-mm curve; in the linear zone, the order is 38.1-, 76.2-, 114.3-, 152.4-, 190.5-, 228.6-, and 266.7-mm curve.

Another important observation is the obvious end effect revealed in Figure 4.11. A similar trend exists in all the curves except the two close to the weld start and end. In summary, there are two distinct phenomena related to the transverse shrinkage:

1) Different shrinkage behaviors in the parted zones across the plate, and
2) End effect.

In order to explain the two phenomena, the concept of effective shrinkage is introduced here. The concept is formed as the result of considering the shrinkage behavior from the perspective of timing and location of its occurrence. In the following context, the effective shrinkage concept will be described in detail and utilized to interpret the transverse shrinkage in the plate.

The transverse shrinkage of a cross section is the result of the interaction between the effective shrinkage and effective rigidity. The effective shrinkage consists of the weld and base metal shrinkage. The effective rigidity is the accumulated joint rigidity during welding. Figure 4.12a and b illustrate the
effective rigidity and effective shrinkage imposed on a cross section respectively. Assuming that the welding arc is traveling to the cross section "Arc", the area A is experiencing the heating cycle, as shown in Figure 4.12b. The metal enclosed in the area is actually in the state of expansion with the rising temperature. However, the area B is under cooling while it is trailing behind the arc. The metal in the area is in contraction, or in other words, the effective shrinkage takes effect.

As the other part of the equation to determine the transverse shrinkage distortion at the cross section "Arc", the joint rigidity which interacts with or in more precise words resists the metal contractions in the area B comes from the metal behind the weld pool. The shaded area in Figure 4.12a is the part of the plate constituting the joint rigidity resisting the effective shrinkage over the cross section. In this area, the filler metal has been deposited into the weld groove and the two halves of the butt joint are connected, therefore this portion of the plate actively participate in the mechanical behavior in the transverse direction induced by welding. The portion of the plate that is in front of the weld pool has no filler metal attaching it to its counterpart in the joint and thus has negligible effect on the transverse shrinkage. Let the area behind the weld pool be prescribed as the effective rigidity area and the joint rigidity associated with the area is the effective rigidity when the welding arc is at the cross section “Arc”. Note that the effective rigidity is defined with respect to the transverse shrinkage.
Figure 4.12: Effective shrinkage and rigidity in the transverse direction. (a) Effective rigidity area; (b) Effective shrinkage area; (c) Effective shrinkage area near weld start at instant t; (d) Effective shrinkage area near weld start at instant t+\Delta t.
Figure 4.12 continued

(c)

(d)
As revealed in Figure 4.12a, the front edge of the effective rigidity area aligns flush with the forefront of the weld pool. Since the cross section "Arc" can be any section in the plate, the effective rigidity area increases linearly as the weld pool moves forward at a constant speed. It is apparent that the effective rigidity is also a transient parameter varying linearly with the welding time or the distance from a cross section to the weld start. The variation of effective rigidity along the weld demonstrates the effect of continuous filler metal deposition.

In Figure 4.13, the effective shrinkage and rigidity are schematically expressed as two function curves varying with the distance from a cross section to the weld start or equivalently varying with the elapsed welding time. The effective rigidity curve is a straight line reflecting the linear relationship between the effective rigidity and welding time. At the initial moments of welding, both effective shrinkage and rigidity have low values. As the weld pool moves forward, the effective shrinkage gradually develops behind the pool. However, the effective shrinkage is not proportional to the distance to the weld start because its area does not have a regular shape such as rectangle. The change of effective shrinkage with time is non-linear. Figure 4.12c and d show the effective shrinkage area B near the weld start at the instant t and t+\Delta t (\Delta t > 0) respectively. The increase of area B is obviously not linear due to the geometric non-linearity of the area’s outer boundary. Such boundary is produced by the nonlinear heat transfer of welding. The effective shrinkage curve in Figure 4.13 is sketched as a parabolic curve to display the non-linear...
Figure 4.13: Variations of the effective shrinkage and effective rigidity along the weld

The curve manifests another important trend that the effective shrinkage gains little growth or saturates itself after the welding arc travels far enough away from the weld start. The saturation point is related to the moment when the effective shrinkage is fully developed. After the saturation, the effective shrinkage keeps unchanged even though the weld pool...
continues moving. It can be explained graphically in stating that the area B in Figure 4.12b reaches its maximum size at certain moment during welding and retains the size from then on.

Figure 4.13 is utilized to interpret the transverse shrinkage behavior observed in the transverse displacement chart in Figure 4.11. Let the graph in Figure 4.13 be dubbed as the effective shrinkage-rigidity diagram. The shrinkage in the linear shrinkage zone (Figure 4.11) is clearly dictated by the effective rigidity in that the cross sections with higher rigidity have smaller shrinkage magnitudes. If elucidated in detail, the displacement curves can be sorted by the descending order of the maximum shrinkage magnitude near the plate edge as 38.1-, 76.2-, 114.3-, 152.4-, 190.5-, 228.6-, and 266.7-mm curves. It is the same order of the effective rigidity of the cross sections if arranged in the ascending order.

Figure 4.14 is the enlarged view of the non-linear shrinkage zone of the transverse displacement chart in Figure 4.11. The shrinkage behavior in the non-linear zone is not as simple and straightforward as that in the linear zone. However, the shrinkage behavior can be explained using the effective shrinkage-rigidity diagram. The displacement magnitudes at the boundary of the non-linear zone (x = 24.5 mm) are extracted and plotted in the diagram. They are used to represent the shrinkage severity at different cross sections based upon the observation that the displacement curves have like shapes in Figure 4.14.
The moving trend revealed by the displacement magnitudes possesses a bell shape, see Figure 4.13. The shrinkage distortion intensifies as the cross sections locate farther away from the weld start. It reaches the peak near the middle section of the plate and then it weakens as the cross sections find themselves closer to the weld end. Among the cross sections, the 38.1-mm cross section has the lowest effective rigidity. Contrary to the intuition, it does not have the maximum distortion magnitude. As described earlier, the effective shrinkage near the weld start is also very weak. The relatively small driving force of the distortion leads to relatively small distortion in spite of low rigidity.

The 76.2-mm cross section is exposed to larger distortion than the 38.1-mm section. Although both effective shrinkage and rigidity increase from the 38.1-mm to 76.2-mm, the augment of the effective shrinkage is more rapid than the linear increment of the rigidity due to the evolution of effective shrinkage. Different ascending paths of the effective shrinkage and rigidity produce larger distortion in the 76.2-mm section. The same scenario can be employed to account for the augmentation of the distortion magnitude from the 114.3-mm to 76.2-mm section. The augmentation is less than that from 76.2-mm to 38.1-mm section, which implies slower increase of the effective shrinkage. Its slower increase signals the effective shrinkage is approximating saturation. The saturation effect is further evidenced in the small difference between the distortion magnitudes of the 114.3-mm and 152.4-mm sections. It can be inferred that the effective shrinkage is fully developed and arrives at its saturation point near the 6-inch section. After the saturation, the transverse
shrinkage is controlled by the effective rigidity because the distortion driving force does not change any more. The distortion magnitudes in the 190.5-, 228.6- and 266.7-mm sections are smaller than that of the 152.4-mm section and decrease toward the weld end.

Figure 4.14: Transverse displacement distributions at different cross sections in the non-linear shrinkage zone
The transverse displacement chart in Figure 4.11 divulges clear end effect. The 38.1-mm and 266.7-mm curves, which are near the weld start and end respectively, have a similar shape that is distinctly different from the shape of the curves not so close to both ends of the weld. However, the driving force and resistance of distortion at the weld start are very different from those at the weld end, as depicted in the effective shrinkage-rigidity diagram. Therefore, it is not easy to instinctively understand the similarity between the two end curves. A reasonable explanation can be placed in the following arguments: the effective shrinkage is almost counteracted by the effective rigidity near the weld end, which is similar to the situation at the weld start. The magnitude difference between the two curves is caused by the effective rigidity difference.

In the above analysis of the transverse shrinkage in the butt joint, the effective shrinkage-rigidity diagram is utilized to interpret the shrinkage behavior. The diagram provides reasonably sound explanations for the two phenomena of the transverse shrinkage:

1) The peculiar ordering of the shrinkage magnitudes at different cross sections in the non-linear shrinkage zone, and

2) The end effect.

The first phenomenon cannot be understood by intuition and is otherwise very difficult to explain without the shrinkage-rigidity diagram. The concept of effective shrinkage enables the establishment of the diagram. The successful use of the diagram constitutes a phenomenological proof of the effective-shrinkage concept.
Longitudinal Shrinkage

The final longitudinal displacement curves at the multiple cross sections can be obtained by replacing the transverse displacements with longitudinal displacements in Figure 4.11, as displayed in Figure 4.15. The cross sections near the weld start have positive displacements, indicating their movement.

Figure 4.15: Longitudinal displacement distributions at different cross sections
toward the weld end. On the other hand, the cross sections near the end bear negative displacements, showing their motion toward the start. Consequently, a shrinkage distortion is perceived in the longitudinal direction. Note that the positive direction of the displacement is the same as the welding direction.

The partition of linear and non-linear shrinkage zone, which is based on the transverse shrinkage behavior, is also valid to gauge the longitudinal shrinkage distortion. Figure 4.15 shows an approximate linear distribution of the displacements in the linear shrinkage zone \((x > 24.5 \text{ mm})\). The non-linear zone encompasses the weld and its surrounding area and it coincides with the longitudinal plastic zone, which will be demonstrated later in this chapter. In this zone, the longitudinal displacements exhibit non-linear distributions. The non-linearity is the result of the complex physical phenomena that occur in and near the weld, such as highly localized heating action of the welding arc, continuous filler metal addition, melting and solidification of the weld metal, and plastic strain relaxation.

The magnitudes of the longitudinal displacements are smaller than those of the transverse displacements. For example, the maximum magnitude of the longitudinal displacements is about one half of that in the transverse direction. In the longitudinal direction, majority of the effective rigidity comes from the base metal with relatively small contribution from filler metal addition, which heavily influences the evolution and intensity of the effective rigidity in the transverse direction. Unlike the transverse rigidity, which is very small at the
weld start and linearly increases along the weld as the weld pool advances, the longitudinal rigidity is comparatively even along the weld and is higher than the transverse rigidity at least near the weld start. Higher rigidity induces smaller displacement magnitudes in the longitudinal direction.

Figure 4.15 demonstrates the end effect of a continuous weld on longitudinal shrinkage distortion, especially in the non-linear shrinkage zone. The middle section displacement curves, including the 114.3-, 152.4- and 190.5-mm curves, possess like shapes that are easily distinguished from those of the curves near both ends of the weld. In the non-linear shrinkage zone, the 38.1-mm curve exhibits more curvature than the 76.2-mm curve; the 76.2-mm curve has more curvature than the 114.3-mm curve. The phenomenon is resulted from the additional rigidity caused by continuous filler metal deposition. It is reasonable to state that the effect of filler metal addition cannot be ignored in the longitudinal direction even though it does not considerably impact the longitudinal rigidity.

Plastic Strains Contours

Figure 4.16 shows the contours of the transverse and longitudinal plastic strains. Both plastic strain components are uniform along the weld except at both weld start and end. The uniform distribution of the plastic strains in the middle portion of the weld is similar to that of the peak temperature plotted in Figure 4.8. Such similarity indicates the peak temperature is the thermal parameter governing the plastic strains. As displayed in Figure 4.16a, the
transverse plastic strains are compressive in the weld and its immediately adjacent area. The compressive zone is much wider than the fusion zone. The measurement of the compressive zone width will be given later in this section. The transverse plastic strains turn into tension outside the compressive zone. The tensile zone is wider than the compressive zone but with much lower magnitudes. As illustrated in Figure 4.16b, the longitudinal plastic strains are in tension in the weld. Further analysis in this section reveals that the tensile zone of the longitudinal plastic strains almost coincides with the fusion zone. The coincidence is attributed to the effect of melting and solidification of weld metal, i.e. plastic strain relaxation. The compressive zone is outside the fusion zone with a larger width and much lower magnitudes than the tensile zone.
Figure 4.16: Plastic strain contours in the plate. (a) Transverse plastic strains; (b) Longitudinal plastic strains.
The plastic strain distributions at seven cross sections are plotted together in Figure 4.17. Refer to Figure 4.5b for the location of the cross sections. Figure 4.17a and Figure 4.17b show the transverse and longitudinal plastic strain components respectively. It shows little independence of those strain components with respect to the welding direction. The differences between different cross sections are not significant and are mainly confined within the fusion zone. The above observations indicate that the plastic strain evolution also experience similar quasi-steady state to that of the heat transfer during welding. However, the quasi-steady state in the plastic strain distributions is not as strong as that demonstrated in the temperature distributions, which can be perceived by comparing Figure 4.17 and Figure 4.9.

Both transverse and longitudinal plastic strains diminish rapidly beyond 25 mm away from the weld centerline, as shown in Figure 4.17. In order to examine the correlation of plastic strains with peak temperatures, the middle cross section of the plate is taken out under scrutiny. The middle cross section is the 6.0-inch section labeled in Figure 4.5b. The transverse and longitudinal plastic strains across the middle section are charted in Figure 4.18a and Figure 4.18b respectively. The peak temperature distribution is plotted along with the plastic strain distributions. In these charts, the horizontal (x) axis designates the distance to the weld centerline in mm. The extreme value shown on the x-axis is 35 mm instead of the half width of the plate that is about 180 mm.
The transverse plastic strain completely fades out quickly beyond 25 mm. The vertical axis that is located at the left end of x-axis and labeled with "Plastic Strain" contains the scale for the plastic strains; the vertical axis that is placed at the right side and entitled "Temperature" ticks the peak temperatures in degree of Kevin. As shown in Figure 4.18a, non-trivial portion of the transverse plastic strain distribution is bounded within x = 24.5 mm where the corresponding peak temperature is 473\(^0\)K. The strain magnitude at x = 24.5 mm is only 0.9% of the maximum magnitude.

Both compressive and tensile strains are present in the transverse plastic strain distribution. The compressive plastic strains concentrate in and near the weld. The compressive zone ends at 5.5 mm away from the weld centerline where the corresponding peak temperature is 970\(^0\)K. Beyond the compressive zone, the plastic strains turn into tensile. The tensile plastic zone that ends at the outer boundary of the plastic zone (x = 24.5 mm) is much wider than the compressive zone. However, the compressive plastic strains have much higher magnitudes than the tensile plastic strains, implying that rigorous material softening takes place in the compressive zone while high temperatures are experienced. It is not justified to claim that the effective shrinkage only develops in the compressive plastic zone. The tensile plastic strains next to it are partially caused by the reverse yielding of the relatively tremendous compressive plastic strains. The material that is associated with these reverse-yielded tensile plastic strains does participate in the effective shrinkage. The
circumstance will be fully checked when the plastic strain history is looked at later in this chapter.

Similar to the transverse plastic strains, the longitudinal plastic strains are present in the area where the peak temperatures are higher than 473\textdegree{K}, as demonstrated in Figure 4.18b. The strain magnitude at $x = 24.5$ mm is only 2.8\% of the maximum magnitude. The tensile zone almost coincides with the fusion zone. The compressive zone is located outside the tensile zone with much larger width and lower stain magnitudes.
Figure 4.17: Plastic strain distributions at different cross sections. (a) Transverse plastic strains; (b) Longitudinal plastic strains.
Figure 4.18: Plastic strain and peak temperature distributions along the direction transverse to weld. (a) Transverse plastic strains and peak temperatures; (b) Longitudinal plastic strains and peak temperatures.
Plastic Temperature Range

The analysis of the plastic strain distributions in the previous section reveals that the plastic zone is confined within the $473^\circ$K isotherm of the peak temperature accumulated during welding. It will be demonstrated later in this chapter that the peak temperature isotherm is indeed the plastic zone boundary in all the cases investigated in the parametric study. To better understand the nature of the phenomenon and to identify the cause, the material behavior at elevated temperatures need to be revisited. Figure 4.19 charts the elastic modulus and yield stress against temperature. As disclosed in the chart, both material properties decrease as temperature rises. For the elastic modulus, its most drastic decrease occurs between $473^\circ$K and $1173^\circ$K.

The scope of temperature from $473^\circ$K to $1173^\circ$K is the material softening range or plastic temperature range. The temperature of $473^\circ$K is the lower softening temperature in that the rapid lessening of the elastic Modulus starts at this temperature. It is also the lower bound of plastic temperature range because plastic strains are developed as long as the material experiences the temperatures above the threshold. The temperature of $1173^\circ$K is the upper softening temperature or upper bound of plastic temperature range. When the temperature is higher than $1173^\circ$K, the elastic Modulus and yield stress become extremely small. This critical temperature is also the mechanical rigidity recovery temperature. Apparently, the plastic zone boundary can be determined through locating the lower bound of plastic temperature range in the peak temperature profile. Therefore, the weld plastic behavior is inherently
determined by material’s plastic temperature range, which itself can be deemed as a material property; the peak temperature is another important parameter that controls the plastic behavior.

The effects of plastic temperature range and peak temperature on the weld plastic behavior are not independent of each other. In fact, the two factors act together to govern the weld plastic behavior. Their effects can be described as the following, considering formation of plastic strains. Since the material intensively softens as the temperature increases beyond its plastic temperature range, plastic strains build up quickly and are usually in compression. These plastic strains are relaxed when melting occurs. Therefore, the thermal parameter that governs the cumulative plastic strains is the peak temperature the material can reach during welding. Although plastic strains may also be formed due to large incompatible thermal strains, the softening effect is usually so large that it overrides the incompatible thermal strains. The peak temperature and plastic temperature range define the plastic zones. Figure 4.8 and Figure 4.9 show that the peak temperature distribution is independent of the welding direction, except in the region of arc start and stop. This illustrates that, for structural welding, simultaneously laying weld bead along the entire joint may reach the same peak temperature results as the moving source model.
Transverse Plastic Strains in and near Weld

The plastic strain distributions at several cross sections have been evaluated in a previous section. However, the plastic strain relaxation phenomenon is not the focus of the previous analysis of plastic strain distributions. Since the relaxation phenomenon takes place in the fusion zone, the plastic strains in and near the weld need to be scrutinized.
Figure 4.20: Plastic strain and peak temperature distributions in and near weld
Figure 4.20 shows the distributions of transverse longitudinal plastic strains and peak temperatures in and near the weld at the middle cross section. The chart has the same abscissa and ordinate as those in Figure 4.18. Note the analysis results presented in the previous sections and here are for the single analysis case, 140-A case. In the analysis, high-temperature material behaviors are simulated including plastic strain relaxation and continuous filler metal deposition. For the purpose of investigating their effects, an alternative analysis is performed on the 140-A case without simulating the high-temperature behaviors aforementioned. The only difference between the alternative analysis and 140-A case is that plastic strain relaxation and filler metal deposition are not simulated in the alternative analysis.

In order to reveal the strain relaxation phenomenon, the curve of transverse plastic strains from the alternative analysis is also plotted in Figure 4.20 and labeled as “Transverse, without relaxation”. The plastic strain curves from the 140-A case are labeled with “Transverse” and “longitudinal”. The strain relaxation is evident by comparing the two transverse plastic strain distributions, namely the trends exhibited by the two curves in the weld and its immediate adjacent region. The two curves coincide with each other outside the compressive zone (x > 5.5 mm). They divert from each other inside the compressive zone and the most significant difference is in the fusion zone. Without the strain relaxation simulated, the plastic strain continues descending into more compression while moving from the fusion boundary to weld centerline, and reaches the maximum magnitude at the centerline; on the other
hand, the descending trend is arrested and reversed when the strain relaxation is simulated. The difference in the two curves can be attributed to the strain relaxation.

As stated above, a diversion exists between the two curves representing the transverse plastic strains with and without strain relaxation simulated, and furthermore, the turning point on the curve with relaxation (curve “Transverse”) is at the fusion boundary (x = 2.12 mm). Since there is no node at the fusion boundary, the curve seems to change its moving trend outside the fusion zone at x = 2.38 mm. However, the peak temperature at that location is 1616°K and below melting temperature (1700°K) which is the threshold of strain relaxation. In the subsequent section describing the plastic strain history, it is found that strain relaxation does not happen at this location (x = 2.38 mm) but takes place at the node (x = 1.98 mm) immediately inside the fusion boundary. It can be concluded that plastic strain relaxation occurs in the fusion zone.

4.2.3 Evolution of Plastic Strains

The final state of plastic strains and other variables have been examined in the previous context. Evolution of plastic strains is analyzed in this section. Analysis of plastic strain history aims to garner insights into the formation mechanism of plastic strains. The analysis also enables bettering understanding of the effects of plastic strain relaxation and filler metal deposition, and moreover provides the evidence verifying the effective shrinkage concept proposed earlier in this chapter.
The plastic strain histories are recorded at individual nodes during the FE analysis. The plastic strain histories of six nodes at the middle cross section are extracted and analyzed here. The middle cross section is the 6.0-inch section labeled in Figure 4.5b. The six nodes are located in and near the weld from the weld centerline towards the plate edge, and therefore, the nodes are distinguished by their distances \((x)\) to the weld centerline as follows: \(x = 0, 1.98, 2.38, 3.93, 5.34, \) and \(11.29\) mm. The plastic strain histories investigated are from two analysis cases: Case 140-A and its alternative analysis. Plastic strain relaxation and filler metal addition are simulated in the 140-A case, of which the analysis results are presented throughout section 4.2. The alternative analysis of 140-A case is performed without simulating the two high-temperature phenomena just mentioned. Part of the results from the alternative analysis is given in the preceding section titled “Transverse Plastic Strains in and Near Weld”.

Figure 4.21 shows the plastic strain histories at the seven locations mentioned above. Also plotted in Figure 4.21 are the temperature histories at the same locations. The plastic strains of 140-A case are labeled “Transverse” and “Longitudinal” for the corresponding strain components. The plastic strains obtained from the alternative analysis are labeled “Transverse, without relaxation or filler” and “Longitudinal, without relaxation or filler”. The legend “without relaxation or filler” is the abbreviation of “without simulating plastic strain relaxation or filler metal deposition”. The duration placed on the time scale is either from 16 to 22 second or from 16 to 26 second. Such time scale
is set up based on the following considerations: 1) the welding arc travels to the middle cross section at $t = 18$ s, 2) most drastic plastic strain change occurs when the arc is passing by the cross section, and 3) the final distributions of the plastic strains at the cross section have been elaborated on in the previous sections (see Figure 4.18 and Figure 4.20).

**Plastic Strains in the Weld**

The plastic strain and temperature histories at the weld centerline ($x = 0$ mm) are plotted in Figure 4.21a. As the welding arc approaches, the temperature rises rapidly. Melting starts when the melting temperature ($1700^0K$) is reached and the corresponding time lapsed after arc initiation is 18 seconds ($t = 18$ s). The temperature keeps going up and reaches its peak at $t = 18.2$ s, and then it decreases. There is an apparent slope change on the temperature curve during cooling shortly after the peak value is attained. The slope change is caused by latent heat that lowers the cooling rate. Solidification begins when the temperature drops to $1700^0K$ ($t = 19.4$ s).

The most noticeable event in the liquid state ($t = 18 \sim 19.4$ s) is the plastic strain relaxation in the 140-A case: both transverse and longitudinal plastic strains that are created before melting are annihilated. The strain relaxation is not an instantaneous event. It takes place over a span of 0.38 seconds. The longitudinal component is set to zero, but the transverse component never reaches zero. These characteristics of strain relaxation can be attributed to the mathematical treatment of solid mechanics in the FEA. For example, the
equilibrium condition has to be satisfied in the whole model at the end of each increment. The stresses and strains of a node have to be balanced with those of the other nodes. Even though the strains of a node are set to zero, the balancing effect may cause the strains to be at non-zero values. This is why the transverse plastic strain has non-zero values during the relaxation. However, its minimum magnitude in the liquid state is 1.39E-3 and trivial compared to the magnitude of 5.31E-2 immediately before melting, and hence its magnitude reduction is a significant 97.4 percent as a result of strain relaxation. The balancing effects in the longitudinal and transverse directions are obviously different. The difference may be caused by the different rigidity characteristics in the two directions, which will be elaborated on in the following context.

The importance of incorporating plastic strain relaxation in the FEA is demonstrated in comparing the plastic strains with and without strain relaxation simulated. Without strain relaxation, the plastic strains continue accumulated in the liquid state, which is not true in the physical reality. False plastic strains are generated if the strain relaxation phenomenon is ignored.

**Plastic Strains near the Fusion Boundary**

Figure 4.21b displays the plastic strain and temperature histories of the node just inside the fusion zone (x = 1.98 mm). Note that the fusion boundary is at x =2.12 mm. The peak temperature is above the melting point but lower than that experienced at the weld centerline. The duration of liquid state is also
shorter. The strain relaxation does happen. The extent of relaxation is not as full as that at the weld centerline. This is reasonable considering the following: compared to the node at the centerline, this node is much closer to the fusion boundary, and therefore the balancing effect from the neighboring nodes outside the fusion zone is more severe. Still, the effect of relaxation on the plastic strain evolution is significant in that there exist substantial discrepancies between the plastic strain histories with and without strain relaxation computed.

There are common features in the plastic strain histories at the two locations examined above, as illustrated in Figure 4.21a and Figure 4.21b. The strain rates before melting are much higher than those after solidification. The different strain rates are generated by the different temperature change rates during heating and cooling. In the fusion zone, the heating rates are much higher than the cooling rates as exhibited in the temperature curves: steep ascending slopes before melting and much less steep descending slopes after solidification. Higher temperature change rates produce higher strain rates.

Another common feature is the moving trends of plastic strains. Both transverse and longitudinal components are in compression during the heating cycle before melting. The compressive plastic strains are created due to the constrained expansion of weld metal. During the cooling cycle after solidification, the weld metal shrinks. The weld metal shrinkage is constrained by the adjacent cooler materials. The plastic strains turn into tension due to the constrained shrinkage. Such formation mechanism of plastic strains is reflected in the reverse yielding in the longitudinal direction as shown in Figure 4.21a and
Figure 4.21b. The reverse yielding refers to the trend change from compressive to tensile observed in the longitudinal plastic strains when the weld metal starts to solidify and cool down. The formation mechanism does not explain the trend that the transverse plastic strains accumulate into more compression during cooling. However, the trend can be easily justified using the effective shrinkage concept proposed earlier in this chapter. The compressive transverse plastic strains in the weld metal are caused by the effective shrinkage behind the weld pool that just solidifies. This formation mechanism is illustrated in Figure 4.12b. Imagine that the two nodes studied here are in the cross section “Arc”. The effective shrinkage in area B imposes compression upon the weld metal ahead of area B. The compression generated by effective shrinkage must dominate the tension induced by constrained shrinkage in the solidified weld metal.

Figure 4.21c shows the plastic strain and temperature histories of the node just outside the fusion zone (x = 2.38 mm). Its peak temperature is below the melting temperature and its plastic strains do not go through strain relaxation. The plastic strains start to emerge at t = 17.7 s when the temperature reaches 473°K that is the lower bound of plastic temperature range, and they accumulate rapidly thereafter. The slopes of plastic strain curves change when cooling begins. The slope changes can be explained using the same scenarios described above for the nodes inside the fusion zone: for the longitudinal component, its change corresponds to the change from constrained expansion during heating to constrained shrinkage during cooling;
for the transverse component, from constrained expansion to effective shrinkage.

Strain relaxation is not present during the evolution of plastic strains at this node, and therefore, the difference between the plastic strains of the 140-A case and its alternative analysis is mainly produced by the effect of filler metal deposition, namely the additional rigidity from unmelted weld metal. Continuous filler metal deposition is simulated in the 140-A case: the unmelted weld metal ahead of the weld pool is assigned the material properties of air, so it adds virtually no rigidity to the weld joint. On the other hand, filler metal deposition is not incorporated in the alternative analysis. It treats the unmelted weld metal as the same solid material in the other part of the model. As a result of such treatment, the unmelted weld metal contributes to the weld joint additional rigidity that does not exist in reality and is false.

The additional rigidity of unmelted weld metal has different effects in the longitudinal and transverse directions. As demonstrated in Figure 4.21c, the longitudinal plastic strains of the two cases take almost the same evolution path, and the transverse components deviate from each other. The analysis results give rise to the conclusion that the effect of additional rigidity on plastic strain is not significant in the longitudinal direction. The conclusion is also supported by the fact that the weld metal only contributes a small portion of the joint rigidity compared to the base metal even if the whole weld is counted. On the other hand, the effect of additional rigidity is significant in the transverse direction. When filler metal deposition is simulated, the joint rigidity in the
transverse direction is proportional to the progression of solidification front, as illustrated in the effective shrinkage-rigidity diagram in Figure 4.13. If filler metal deposition is not simulated, the rigidity can be considered constant during welding since the whole weld including unmelted portion is modeled as solid.

The effect of filler metal deposition on transverse plastic strain, or equivalently, the effect of the additional rigidity generated by unmelted weld metal diminishes as the distance to the weld increases. The trend is easily seen by comparing the transverse plastic strains of the 140-A case and its alternative analysis at the locations farther away from the weld, as shown in Figure 4.21d, Figure 4.21e, and Figure 4.21f. The differences in the transverse plastic strains become smaller farther away from the weld and finally disappear as displayed in Figure 4.21f. Up to now, the differences between the 140-A case and its alternative analysis have been clearly examined in light of the effects of plastic strain relaxation and filler metal deposition. The analysis results mentioned in the remaining contexts of this section (section 4.2.3) only refer to those of the 140-A case.
Figure 4.21: Plastic strain and temperature histories of different locations at the middle cross section. (a) $x = 0$ mm; (b) $x = 1.98$ mm; (c) $x = 2.38$ mm; (d) $x = 3.93$ mm; (e) $x = 5.34$ mm; (f) $x = 11.29$ mm. $x$ is the distance to weld centerline.
Figure 4.21 continued

Transverse
Longitudinal
Transverse, without relaxation or filler
Longitudinal, without relaxation or filler
Temperature

Liquid state

T = 1700^0K

X = 1.98 mm

Time (second)
Figure 4.21 continued

---

Transverse
Longitudinal
Transverse, without relaxation or filler
Longitudinal, without relaxation or filler
Temperature

-0.02
0.00
0.01
16 17 18 19 20 21 22 23 24 25 26
Time (second)

200
600
1000
1400
1800
2200

Temperature (°K)

X = 3.93 mm
T = 473°K

1st stage
2nd stage
3rd stage

t = 17.8 s

Continued
Figure 4.21 continued

(e)
Figure 4.21 continued

- Transverse
- Longitudinal
- Transverse, without relaxation or filler
- Longitudinal, without relaxation or filler
- Temperature

Temperature (°K)

Plastic Strain

Time (second)

Transverse, without relaxation or filler
Longitudinal, without relaxation or filler

T = 473°K
X = 11.29 mm

T = 473°K

Time (second)

(f)
Plastic Strains in the Base Metal

The plastic strain evolution in and near the weld has been investigated in the above contexts. Farther away from the weld, the formation mechanisms of plastic strains are different from those in and near the weld, as demonstrated in Figure 4.21d, Figure 4.21e, and Figure 4.21f showing the plastic strain and temperature histories of the nodes at $x = 3.93$ mm, $x = 5.34$ mm, and $x = 11.29$ mm respectively.

As illustrated in Figure 4.18, the final plastic strains at $x = 3.93$ and 5.34 mm have common characteristics: they have the same signs, for example, the transverse components at both locations are compressive. The plastic strain evolutions at the two locations also have the following common features:

1) The plastic strains are produced when the temperature reaches the lower bound of plastic temperature range ($473^0K$),

2) The transverse component emerges in tension instead of compression as those do in the weld, and

3) The plastic strain evolution can be divided into three stages. The first feature is easy to understand in that the plastic temperature range is also the material softening temperature range. The second feature will be explained below in conjunction with the third one. The plastic strain histories of the two locations will be analyzed together due to their common features.

The three stages are labeled in Figure 4.21d and Figure 4.21e. They are defined according to the moving trend in the plastic strains. Different formation mechanisms dominate in different stages. The first stage is between the onset
of plastic strains and the instant when the temperature rises to the peak value. The longitudinal plastic strain appears in compression and accumulates into more compression as the temperature increases. The compressive strain is created by constrained expansion considering this stage is within the heating cycle. However, the mechanism of constrained expansion cannot be utilized to interpret the tensile transverse plastic strain developed in this stage. The tensile strain is the result of reverse yielding. This formation mechanism is described as the following: the transverse shrinkage in the adjacent materials that are closer to the weld produces reverse yielding at the locations farther away from the weld. In plain text, heavy contraction in and near the weld takes the neighboring outer materials moving toward the weld centerline. Consequently, the outer materials are stretched and tensile plastic strains are created.

In the second stage, both transverse and longitudinal plastic strains almost remain unchanged. This trend is more obvious at the location (x = 5.34 mm) farther away from the weld, as shown in Figure 4.21e. Notice that the start of second stage is also the inception of cooling cycle. The constrained shrinkage comes into effect in both transverse and longitudinal directions. In the longitudinal direction, the effects of constrained shrinkage and reverse yielding are balanced so that the plastic strain has little variation. The reverse yielding mechanism is that the longitudinal shrinkage in the materials closer to the weld imposes compression upon those farther away from the weld. The compressive strain from reverse yielding is cancelled out by the tensile strain
from constrained shrinkage. In the transverse direction, the constant plastic strain is the result of the balance between the compression from effective shrinkage and the tension from constrained shrinkage and reverse yielding. The mechanism of transverse reverse yielding is delineated in the first stage.

In the third stage, the balance between different formation mechanisms is broken, and therefore the plastic strains vary with time. Constrained shrinkage becomes the dominant mechanism and the plastic strain increases in the longitudinal direction. Effective shrinkage is the governing mechanism in the transverse direction and it causes the decreasing plastic strain.

When the materials are far enough away from the weld, the dominant plastic strain formation mechanism is the reverse yielding produced by the materials closer to the weld. The statement is supported by the plastic strain histories of the node 11.29 mm away from the weld centerline, as plotted in Figure 4.21f. Both transverse and longitudinal plastic strains emerge at an instant \((t = 18.9 \text{ s})\) before the temperature rises to \(473^\circ\text{K}\). They are clearly generated by the reverse yielding mechanism. The slopes of plastic strain curves increase as the temperature reaches \(473^\circ\text{K}\) \((t = 19.8 \text{ s})\) when the material enters plastic temperature range and starts to soften.
4.2.4 **Effects of Heat Input on In-Plane Shrinkage and Plastic Strains**

As found in the previous sections, the transverse shrinkage is the dominant distortion mode over the longitudinal shrinkage in the butt-welded plates. Since the transverse shrinkage is mainly determined by the transverse plastic strains, they will be focused on when evaluating the in-plane shrinkage and plastic behaviors in the rest of this dissertation.

The effects of heat input on in-plane shrinkage and plastic deformation are investigated through the parametric study of which the analysis details have been described in section 4.1. The analysis matrix is given in Table 4.1. Heat input is varied among different analysis cases by adjust the welding current that is the sole variable in the parametric study.

The peak temperatures at the middle cross section of all the cases are plotted together in Figure 4.22, along with the transverse displacement distributions. The magnitudes of both peak temperature and displacement increase with the increasing heat input. The peak temperature curve of higher heat input encompasses more area than that of lower heat input. More materials are heated as more heat are dissipated into the welded plate, for example, the fusion zone widens as shown in Figure 4.22a. The transverse displacement curves have very similar shapes in spite of their differences in magnitude. The similarity is understandable because the same types of welding process and welded structure are involved in different cases.

The distorted plate shape of the 140-A case is illustrated in Figure 4.10a. The other cases have the similar distortion pattern in the plate. The similarity is
demonstrated in Figure 4.23a, which charts the transverse shrinkage magnitudes along the plate edge. The abscissa is labeled “Distance to Weld Start”. The distance referred to here is the relative distance in the longitudinal direction. The shrinkage magnitudes near the weld start are higher than those near the weld end. There is indeed a slope in every curve in this chart. The slope is obviously the result of in-plane rotation of the deformed plate edge, as shown in Figure 4.10a. The effect of heat input on shrinkage magnitude is more clearly revealed in Figure 4.23b, which correlates the shrinkage magnitudes at plate edge with heat input levels.

Figure 4.24 displays the transverse plastic strain distributions at the middle cross section in different cases. Besides the differences in magnitude and distribution width, there are apparent resemblances among the plastic strain distributions produced under different heat input levels. The compressive zone is in and near the welds and the neighboring tensile zone is located farther away from the welds. As heat input increases, the compressive zone becomes larger and its magnitude decreases. Higher heat input also generates large plastic zone.

As discovered in the analysis of 140-A case, peak temperature is the thermal parameter that governs the plastic strain distributions. In the parametric study, it is also of central importance to establish the relationships between the peak temperature and distribution characteristics of plastic strains. Two relationships examined here are between the critical zone widths of peak temperatures and plastic strains, including the following pairs:
• Width of $473^0K$ isotherm and plastic zone width, and
• Width of $1173^0K$ isotherm and compressive zone width of transverse plastic strains (width of compressive strains)

Figure 4.25 reveals the relationships. The curves representing the widths of $473^0K$ peak temperature isotherm and the plastic zone coincided with each other, indicating the plastic zone was bounded by the $473^0K$ isotherm. Strong correlation exists between the widths of $1173^0K$ isotherm and compressive zone of transverse plastic strains especially when the heat input is high. It can be deemed that transverse plastic strains change signs from compressive to tensile while crossing the $1173^0K$ peak temperature isotherm.

The temperature values of $473^0K$ and $1173^0K$ are not randomly chosen the numbers. They are the lower and upper bound of the plastic temperature range of the material, as stated in the previous section titled Plastic Temperature Range. As described above, the peak temperature isotherms at the two critical values determine the distribution characteristics of plastic strains. If the characteristics control the transverse shrinkage, then the peak temperature also governs the transverse shrinkage. The correlation of the distribution characteristics and transverse shrinkage is revealed in Figure 4.26, namely the relationship between average shrinkage magnitude and plastic zone width, and that between the average shrinkage magnitude and compressive zone width of transverse plastic strains. As shown in Figure 4.26a, the shrinkage magnitude can be expressed as a linear function of plastic zone
width. The R-squared value, or coefficient of determination is 0.978 indicating a strong linear correlation. As illustrated in Figure 4.26b, the shrinkage magnitude increases with the increasing compressive zone width of transverse plastic strains. Their relationship can also be regressed as a linear function. The coefficient of determination of the curve fitting is 0.934.
Figure 4.22: Effects of heat input on peak temperature and transverse displacement. (a) Peak temperature distributions at the middle cross section; (b) Transverse displacement distributions at the middle cross section.
Figure 4.23: Effect of heat input on transverse shrinkage. (a) Shrinkage magnitude along the plate edge; (b) Minimum, maximum, and average shrinkage magnitude.
Figure 4.24: Effect of heat input on transverse plastic strains

Figure 4.25: Effects of heat input on various critical zone widths
Figure 4.26: Correlation of transverse shrinkage and plastic strain distribution characteristics. (a) Plastic zone width; (b) Width of compressive transverse plastic strains.
4.3 Plate with Slot Weld and Tee Joint

As mentioned in the beginning of this chapter, the material of the two joints in this section is Steel-A. Its material properties have been presented in Chapter Three. The FEA model formulated and verified in Chapter Three is utilized here to conduct numerical welding experiments on the weld joints. GMAW process is used to make the weld and the process parameters are the same as those of the 175-A case as given in Table 4.1.

4.3.1 Geometry and FE Mesh of Plate with Slot Weld

As shown in Figure 4.27, the plate with slot weld is indeed a plate with a rectangular slot into which a GMA weld was deposited. The joint can be viewed as a plate with a repair weld. Figure 4.27 indicates the geometry and dimensions of the plate and weld.

The FE mesh of the joint was generated using the same types of ABAQUS 3D solid elements as those in the previous study of butt-welded plates. Figure 4.28 illustrates the finite element mesh of the joint. Due to the large temperature gradients in and near the weld, refined mesh is constructed there in order to obtain computation accuracy and smooth numerical convergence. Coarse mesh is placed in the regions far away from the weld for the purpose of reducing computational cost. The numbers of elements and nodes in the model are 4914 and 9416 respectively.
Figure 4.27: Geometry and dimensions of the plate with slot weld
Figure 4.28: FE mesh of the plate with slot weld. (a) Overall mesh; (b) Mesh in the weld and its surrounding areas.
4.3.2 Geometry and FE Mesh of Tee Joint

The simple Tee joint was composed of a 500 x 300 mm base plate and a 100 x 50 mm web plate, as illustrated in Figure 2.9. Both plates had a thickness of 1.93 mm. Angular distortion is inevitable in a fillet-welded Tee joint. However, the angular distortion is insignificant compared to the in-plane shrinkage due to thin gauge of the joint.

The finite element mesh of the Tee joint is shown in Figure 4.29. Dense mesh is placed in the weld and its surrounding areas in order to accommodate the large temperature gradients and high stress/strain fields in the region as a result of welding. The mesh becomes coarser away from the weld. There are 3342 elements and 3341 nodes in the model. ABAQUS element types S4R and S3R are employed in the mechanical analysis. S4R is 4-node doubly curved general-purpose shell element with reduced integration, hourglass control and finite membrane strains. S3R is 3-node triangular general-purpose shell element with finite membrane strains. For the thermal analysis, ABAQUS element types DS4 and DS3 are used. DS4 is 4-node heat transfer quadrilateral shell element. DS3 is 3-node heat transfer triangular shell element.

Unlike the FE models in Chapter Three and those in the previous studies of this chapter, which are generated using 3D solid elements, the model of Tee joint is constructed with shell elements. The use of shell elements is motivated by the following facts and considerations: massive industrial applications, ease of use, low computational cost and independence of principle on
implementation. Shell elements are frequently used in many computer-aided engineering (CAE) applications, especially those dealing with large fabricated structures such as automotive structures, naval structures and pressure vessels. In the automotive industry, FE models composed of shell elements are extensively utilized in math-based evaluations such as manufacturing assembly analysis, vehicle crash analysis, and vehicle durability analysis. Major automotive companies even have internal guidelines that specify shell element types and sizes for certain CAE applications.

Compared with 3D solid elements, it is much easier to use shell elements to build finite element models. A shell-element model usually has less elements and nodes than a solid-element model for the same structure, which may lead to lower computational cost in the analyses using the former. Most important of all, the use of shell elements is driven by the consideration of independence of principle on implementation. The principle upon which this study is based is the hypothesis regarding distortion prediction. It is stated in the beginning of this dissertation. It is indeed an interpretation of the principle that plastic deformation is the root source of welding distortion. Independence of principle on implementation means that the principle prevails no matter which element type is used in FE implementation.
Figure 4.29: FE mesh of the Tee joint. (a) Overall mesh (top view); (b) Mesh in the weld and its surrounding areas.
4.3.3 **In-Plane Shrinkage and Plastic Strains in the Two Joints**

The analysis results of the two joints confirm the findings obtained in the weld simulation of butt-welded plates as delineated in the preceding section: peak temperature and material softening are the two dominant factors controlling the plastic strains and consequently the in-plane shrinkage. Figure 4.30 and Figure 4.31 are the overlay plots of the transverse plastic strain fields and peak temperature fields in the plate with slot weld and the base plate of Tee joint respectively. The plastic strain fields are plotted in shaded contours while the peak temperature fields in line contours. Both plastic strains and peak temperatures have relatively uniform distributions along the weld axis in most portion of the weld except at its ends where apparent end effects are demonstrated.

The most important revelation in two figures is that the peak temperatures govern the plastic strains. If the end effects are ignored, the plastic strain field is bordered by the $473^0K$ peak temperature isotherm, which is the lower bound of material's plastic temperature range. Furthermore, the compressive plastic zone was enclosed by the $1173^0K$ peak temperature isotherm, which is the upper bound of material's plastic temperature range.
Figure 4.30: Overlay plot of the transverse plastic strain field and peak temperature field in the plate with slot weld
Figure 4.31: Overlay plot of the transverse plastic strain field and peak temperature field in the base plate of Tee joint.
4.4 Summary

In this chapter, the in-plane shrinkage and plastic strains are studied for three types of simple weld joints. Welding distortion in form of transverse shrinkage is correlated to plastic strains and peak temperatures. It is found that peak temperature and material softening have dominant effect on the in-plane shrinkage plastic strains. The results of this study provide a theoretical basis for the engineering approach development in next chapter. Given below were the important findings of this study:

1) In-plane shrinkage plastic strains are determined by the peak temperatures and material’s plastic temperature range. Peak temperature is the thermal parameter that controls the plastic strain distributions,

2) The peak temperatures are independent of the welding direction, except in the regions of arc start and end,

3) The plastic strains have uniform distributions along the welding direction in most part of a weld except at its start and end, where end effects are demonstrated,

4) The plastic strains have non-uniform distributions in the direction perpendicular to the weld. Their magnitudes have maximum values in the weld and diminish when moving away from the weld into the base metal,

5) The plastic zone is bounded by the 473$^\circ$K peak temperature isotherm, the lower bound of plastic temperature of the material,
6) The compressive zone of transverse plastic strains can be defined with the $1173^0\text{K}$ peak temperature isotherm, the upper bound of plastic temperature of the material,

7) The transverse shrinkage magnitude can be expressed as a linear function of plastic zone width, and

8) The transverse shrinkage magnitude can also be expressed as a linear function of compressive zone width of transverse plastic strains.
CHAPTER 5
DEVELOPMENT OF ENGINEERING APPROACH

The study presented in this chapter aims to develop a fast and accurate engineering approach for predicting the welding distortion in large thin-wall structures. The term “engineering approach” is equivalent to the term “simplified approach” in this dissertation. The two terms will be used interchangeably in this chapter and subsequent chapters. In Chapter One, a hypothesis is proposed regarding prediction of welding distortion using numerical methods. It is stated in the hypothesis that the welding distortion can be accurately computed if the actual plastic deformation induced by welding is introduced into the FEA model. The engineering approach to distortion prediction is formulated by using the hypothesis and implemented through FEA. Successful development and application of the engineering approach will demonstrate the validity of the hypothesis. In addition to providing the proof of the hypothesis, the engineering approach will be proved very useful when it is applied to industrial structures such as those presented in Chapter Six. In fact, the development of efficient and accurate prediction methods is highly needed for industrial applications and has been vigorously pursued by many researchers.
5.1 Detailed and Simplified Approaches to Distortion Prediction

As delineated in Chapter One, detailed and simplified approaches are the two major schools of finite element methods of predicting welding distortion. A detailed approach is typically a straightforward transient analysis that simulates the thermal and mechanical effects of a moving heat source. The analyses in Chapters Three and Four fall into the category of the detailed approaches. Even though they can provide very accurate predictions, the detailed approaches consume enormous computing time and processing memory. The High computational cost practically prevents them being applied to industrial structures, especially large welded structures.

The motivation behind the development of the simplified approaches is to reduce the computational cost and make it possible to achieve quick and accurate distortion predictions for large welded structures. The terminology "simplified approaches" should be used with reference to the detailed approaches. The simplification of the simplified approaches refers to the faster and simpler computing algorithms compared to those of the detailed approaches. For example, static analyses are executed within the simplified approaches instead of transient analyses. The nature of the simplified approaches is to simulate the weld deformation through applying the equivalent shrinkage in the welds. Such idealization and simplification are based on the physical observation of welding mechanics that weld shrinkage is the driving
force of distortion. Past research work on the simplified approaches has been reviewed in Chapter One.

5.2 Formulation of the Simplified Approach Based on Welding Plastic Deformation

5.2.1 Theoretical Basis and Principle

It is reasonable to infer that obvious discrepancies may exist between the results from the simplified and detailed approaches applied to the same welded structure. The discrepancies originate from the static analyses adopted in simplified approaches because they ignore the transient effects of welding, which include the plastic strain and structural rigidity evolution during welding. In most reported investigations involving simplified approaches, significant differences are observed either between the predictions rendered by the simplified approaches and laboratory measurements or between the computational results from the simplified and detailed approaches. In these simplified approaches, the computational speed-ups are accomplished at the sacrifice of accuracy. None of them is based on welding plastic deformation.

In order to overcome the above-mentioned drawback of the simplified approaches, the welding-induced plastic deformation is utilized as criteria or reference to establish a new simplified approach to distortion prediction in this research. When devising the simplified approach, the research issue of central importance is how to create the plastic strains that are very similar to, if not
exactly same as the actual ones induced by welding. The equivalent temperature load is employed to generate the plastic strains in this study. "Equivalent" means that the plastic strains produced by the temperature load should be equivalent to the actual ones. The equivalent temperature load is provided through distributing a specific temperature gradient in the weld zone and its neighboring area, in which the temperature gradient is ramped up and down to generate plastic strains. The following considerations contribute to choosing the temperature load over other load types in this study.

From implementation point of view, it is easy to maneuver temperature load in a finite element analysis. If it takes great amount of time and efforts to implement a simplified approach, the approach itself is by no means simplified even though it may yield fast calculation. Ramping up and down of the temperature load resembles the thermal history of welding in the sense that welding plastic strains are caused by non-uniform heating and cooling. However, the application of the equivalent temperature load is quite different from the thermal history that is used as the thermal load in the detailed approaches such as the moving source model in Chapters Three and Four. The welding thermal history is transient in nature so that the accumulation of structural rigidity owing to filler metal addition and the evolution of plastic strains can be simulated in the detailed approach. The equivalent temperature load is static in nature in that its application is time-independent and there is no heat transfer occurred in the loading process.
The manner in which the temperature load is applied to a finite element model is the same as that of linearly loading and unloading a force to the model. The "static" nature of the temperature load is the very reason why the simplified approach is much faster than the detailed approach. Beyond the ease of implementation and computing speed, the more critical aspect of the simplified approach is how to produce the plastic strains approximating the actual values to the maximum extent. It depends on the proper distribution of the temperature load.

Determination of the distribution of the temperature load, namely its magnitude and area of application has to be linked to the welding plastic strains so that accurate distortion prediction can be achieved. In other words, the temperature load distribution has to be established based upon the characteristics of plastic strains and peak temperatures, such as those obtained in Chapter Four. For better understanding of the simplified approach developed in this research, some important findings from Chapter Four are recited here. They are the characteristics of plastic strains and peak temperatures accumulated during welding:

1) The longitudinal and transverse plastic strains have uniform distributions along the welding direction in most part of a long weld except at its start and end, where end effects are demonstrated,

2) Similar to the plastic strains, the peak temperatures are independent of weld axis, except in the regions of arc start and end,
3) The welding plastic strains are determined by peak temperature and material’s plastic temperature range. Peak temperature is the thermal parameter that controls the plastic strain distributions,

4) The longitudinal and transverse plastic strains have non-uniform distributions in the direction perpendicular to the weld. Their magnitudes have maximum values in the weld and diminish when moving away from the weld into the base metal,

5) The plastic zone is much wider than the fusion zone. Specifically, the plastic zone is the region where the peak temperatures are higher than 473 K that is the lower bound of plastic temperature range of the material. The transverse shrinkage magnitude can be expressed as a linear function of plastic zone width,

6) The compressive zone of transverse plastic strains can be defined with the 1173 K peak temperature isotherm, the upper bound of plastic temperature of the material. The transverse shrinkage magnitude can also be expressed as a linear function of compressive zone width of transverse plastic strains, and

7) Since only the fusion zone experiences the plastic strain relaxation, the plastic strains that are generated outside the fusion zone during the heating cycle remain after cooling. Their signs and magnitudes may change due to the reverse yielding during the cooling cycle. The influence of these plastic strains on distortion cannot be omitted.
These characteristics provide the guidelines to constitute the distribution of the temperature load. Characteristics 1, 2, and 3 illustrate that simultaneously laying weld bead along the entire joint may reach the same peak temperature and plastic strain results as the moving source model. In the simplified approach, "weld bead" is in the form of temperature load. Characteristics 4, 5, and 6 suggest that the temperature load be applied beyond the fusion zone. The plastic zone should be a reasonable choice for the area of application. Considering Characteristic 4, the magnitudes of the temperature load are not necessarily homogeneous within the area of application. The magnitudes in and near the weld should be higher than those closer to the boundary of the plastic zone. Characteristic 6 further suggests that the $1173^0K$ peak temperature isotherm can be used to divide the plastic zone where the temperature load is applied. In other words, the temperature load applied inside the $1173^0K$ isotherm should have higher magnitude than that outside the isotherm. This idea will be further explored and utilized in formulating the equivalent temperature load in the following section.

Characteristic 7 justifies the fashion in which the temperature load is applied, literally ramping up and down of the load. If the loading history is only composed of the descent of the load, which mimics the cooling cycle of welding, the plastic strains that are developed outside the fusion zone during the heating cycle may be overlooked. In summary, the simplified approach is constructed with the following features:
• The equivalent temperature load is laid simultaneously along the entire joint,
• The temperature load is applied to the plastic zone,
• The magnitudes of the load are not uniform with higher value in the weld and its immediate adjacent areas, and
• The loading history includes both linearly ascent and descent of the temperature load.

5.2.2 Formulation of Equivalent Temperature Load

This section presents the formulation of equivalent temperature load for the simplified approach. Specifically, the distribution of temperature load is determined using the principle described in the preceding section. The formulation of temperature load is based on a particular analysis case reported in Chapter Four, the 175-A case. Its welding parameters are as the following: 24-V arc voltage, 175-A welding current, and 8.47 mm/s travel speed. Figure 5.1 shows the distributions of the peak temperatures and transverse plastic strains at the middle cross section of the plate. The chart is very similar to that in Figure 4.18a for the 140-A case. Refer to Chapter Four for the details on how the chart is set up.

The overall application area of the temperature load is the plastic zone that is bounded by the 473\(^{\circ}\)K peak temperature isotherm. The overall area is divided into two portions using the 1173\(^{\circ}\)K peak temperature isotherm. Note that the transverse plastic strains change from being compressive to tensile...
near the 1173\(^0\)K isotherm. As stated in Characteristic 6 in the preceding section, the transverse shrinkage magnitude is proportional to the compressive zone width of transverse plastic strains. The load magnitude inside the 1173\(^0\)K isotherm should be higher than that outside. As shown in Figure 5.1, the average value of the peak temperatures inside the 1173\(^0\)K isotherm is 1700\(^0\)K. This value is used as the load magnitude inside the isotherm. The average peak temperature between the 1173\(^0\)K and 473\(^0\)K isotherms is 700\(^0\)K, which is the load magnitude outside the 1173\(^0\)K isotherm. Figure 5.2 illustrates the equivalent temperature load distribution when it is applied to a FE model.

What are instrumented in the simplified approach are loading and unloading the equivalent temperature load to the structure. The temperature load behaves similarly to mechanical load in that their loading and unloading processes are executed in the same manner. An initial temperature of 300\(^0\)K (room temperature) is assigned to the whole structure before applying the load. As illustrated in Figure 5.3, there are two steps in the analysis: the loading step that simulates the heating cycle of welding, and the unloading step that represents the cooling cycle. In this study, \(\theta_p\) and \(\theta'_p\) take the values of 1700\(^0\)K and 700\(^0\)K respectively.

In each step, the temperature load is applied to the nodes in the application area through the ABAQUS commands *TEMPERATURE and *STATIC. The node numbers and load magnitudes are specified in the former. The latter one prescribes how the load is imposed upon the finite element model. In the loading step, the load is linearly ramped up to its peak magnitude
$\theta_p$ over a duration of 1 second; in the unloading step, it is linearly ramped down to room temperature from $\theta_p$ also over a duration of 1 second. Both loading and unloading durations are set to 1 second via the *STATIC command. However, the nature of the simplified analysis determines that the lengths of the durations have no effect on the computational results. First, the material constitutive behaviors are time (rate)-independent. Secondly, there is no transient effect caused by the temperature load such as heat transfer.
Figure 5.1: Transverse plastic strain and peak temperature distributions at the middle cross section in the 175-A case.
Figure 5.2: Distribution of the equivalent temperature load
Figure 5.3: Loading and unloading of the temperature load in the simplified analysis
5.3 Methodology

Both detailed and simplified approaches are utilized and investigated in this study. Correspondingly, two types of FE analyses are conducted in the study: transient analysis and simplified analysis. As reviewed in Chapter One, accurate distortion prediction can be obtained from the transient thermomechanical FE analysis of welding processes, which is a widely used detailed approach. A good example of this type of analysis is the moving source model that has been verified by laboratory experiments in Chapter Three. In the following context, this type of analysis is referred to as the transient analysis. Based on the confidence in its high accuracy, the transient analysis is used as numerical experiment to substitute laboratory trial with the benefits of cost and time saving. Typical measurement from a laboratory experiment is displacement. It takes a great deal of efforts to measure the full field of residual stresses, and if not impossible, it is extremely difficult to gauge the plastic strain fields. The plastic strains and distortion extracted from the transient analysis are treated as the actual ones here. The above considerations justify such treatment.

The other type of analysis is referred to as the simplified analysis. It is the finite element implementation of the simplified approach developed in this research. In the simplified analysis, the equivalent temperature load is applied to the welded structure to simulate the weld shrinkage and generate the plastic deformation. The accuracy of the simplified analysis mainly depends on how
the equivalent temperature load is applied to the structure. The principles of
constituting the temperature load have been reasoned and delineated in the
preceeding section.

Case studies are conducted on the two welded joints analyzed in the
preceeding chapter: the plate with slot weld and Tee joint. Both transient and
simplified approaches were performed on joints. The necessary input data
have been described in Chapter Four, including joint geometry, FE meshes,
material properties, and process conditions. They are not reiterated here
except that the two joint are made with the same GMA welding conditions of the
175-A case in Chapter Four, on which the equivalent temperature load is also
formulated.

From an implementation standpoint, the central portion of this study is to
compare the results of the simplified analysis with those of the transient
analysis in lieu of laboratory measurements. Among the analysis results, the
most significant and revealing data are the plastic strains and welding
distortions. They are scrutinized in the rest of this chapter. The predicted
distortions and plastic strains using the detailed approach are regarded as
actual ones. The objective of the case studies is to demonstrate that the
distortions predicted by the simplified approach match those of the simplified
approach only when the temperature load generates the plastic strains
approximating the actual ones.
5.4 Case Study of Tee Joint

As stated in Chapter One, the major type of distortion researched in this dissertation is in-plane shrinkage induced by GMAW. In this study, the transverse shrinkage of the base plate is the dominant distortion in the Tee joint. Another apparent form of distortion in the Tee joint, the angular distortion is indeed caused by the transverse shrinkage. However, the angular distortion is insignificant compared to the in-plane shrinkage because of thin gauge of the Tee joint. Therefore the analysis results reported here are related to the transverse shrinkage, including plastic strain fields, distorted shapes of the base plate, displacement fields and residual stress fields.

The transverse plastic strain field from the simplified analysis resembles the one from the detailed analysis with certain degree of discrepancy, as displayed in Figure 5.4. In both analysis results, both compressive and tensile plastic strains are present with the compressive strains concentrating in and near the weld and the tensile strains surrounding compressive zone. The maximum magnitudes of the plastic strains from the two analyses are comparable: 6.21E-2 from the detailed analysis and 6.82E-2 from the simplified analysis. Although the shapes of the compressive plastic zones are similar to each other, the shapes of the tensile plastic zones exhibit apparent difference. The difference is caused by the transient effect of welding heat transfer, which is not captured in the simplified approach. Noticeably, the widths of the
compressive zones from the two analyses are comparable. So are the widths of the tensile zones.

The transverse displacement field predicted by simplified analysis approximated that by the detailed analysis to very high degree, as shown in Figure 5.5. The two displacement fields have almost identical distribution pattern. Their extreme values are close: $U_{\text{min}} = -1.47E-1$ mm and $U_{\text{max}} = 1.18E-1$ mm from the detail analysis; $U_{\text{min}} = -1.30E-1$ mm and $U_{\text{max}} = 1.03E-1$ mm from the simplified analysis. Figure 5.6 presents a comparison of the distorted shapes of the base plate obtained from the two analyses. The distorted shapes computed by the two analyses are very comparable as both exhibit obvious transverse shrinkage and slight longitudinal shrinkage. Figure 5.7 demonstrates the residual stress (Von Mises stress) fields from the two analyses. The stress fields have twin-like distributions with the same maximum stress magnitude of 3.81E2 MPa.

In summary, the equivalent temperature load induces the correct distortion distribution since it generates the plastic deformation approximating the actual plastic deformation. Significant computational speed-up is garnered by using the simplified analyses. Table 5.1 lists the CPU time consumed by the two types of analyses in this case study as well as the normalized CPU time and computational speed-up factors while the detailed analysis is the reference case.
Figure 5.4: Comparison of the transverse plastic strain fields from transient and simplified analyses. (a) Transient analysis, $E_{p\text{min}}^p = -6.21\times10^{-2}$, $E_{p\text{max}}^p = 4.84\times10^{-3}$; (b) Simplified analysis, $E_{p\text{min}}^p = -3.54\times10^{-2}$, $E_{p\text{max}}^p = 7.68\times10^{-3}$. 
Figure 5.5: Comparison of the transverse displacement fields from transient and simplified analyses. (a) Transient analysis, $U_{\text{min}} = -1.47\times10^{-1}$, $U_{\text{max}} = 1.18\times10^{-1}$; (b) Simplified analysis, $U_{\text{min}} = -6.97\times10^{-2}$, $U_{\text{max}} = 5.02\times10^{-2}$. 

223
Figure 5.6: Comparison of the distorted base plates from transient and simplified analyses. (a) Transient analysis; (b) Simplified analysis.
Figure 5.7: Comparison of the residual (Von Mises) stress fields from transient and simplified analyses. (a) Transient analysis; (b) Simplified analysis.
<table>
<thead>
<tr>
<th>Analysis</th>
<th>CPU Time (seconds)</th>
<th>Normalized CPU Time</th>
<th>Speed-up Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Transient Analysis</td>
<td>7785</td>
<td>100%</td>
<td>1.0</td>
</tr>
<tr>
<td>Simplified Analysis with Type I load</td>
<td>395</td>
<td>5.1%</td>
<td>19.7</td>
</tr>
<tr>
<td>Simplified Analysis with Type II load</td>
<td>616</td>
<td>7.9%</td>
<td>12.6</td>
</tr>
<tr>
<td>Simplified Analysis with Type III load</td>
<td>742</td>
<td>9.5%</td>
<td>10.5</td>
</tr>
</tbody>
</table>

Table 5.1: CPU time consumed by the analyses of Tee joint performed on HP J5000 workstation

5.5 Case Study of Plate with Slot Weld

A comparison of the transverse plastic strain fields from the transient and simplified analyses is presented in Figure 5.8. In general, the resemblance between the two fields can be easily detected in spite of some degree of discrepancy. Both compressive and tensile zones are present in the two fields. The compressive zones are confined within the weld and its immediate neighboring area and are sandwiched by the tensile zones. The minimum \((E_{p_{\text{min}}}^{\text{p}})\) and maximum \((E_{p_{\text{max}}}^{\text{p}})\) values of the two fields are similar: \(E_{p_{\text{min}}}^{\text{p}} = -3.95E-2\) and \(E_{p_{\text{max}}}^{\text{p}} = 5.40E-3\) from the transient analysis; \(E_{p_{\text{min}}}^{\text{p}} = -3.88E-2\) and \(E_{p_{\text{max}}}^{\text{p}} = 3.01E-3\) from the simplified analysis. The widths of the compressive and tensile zones from the two analyses are also comparable.
The plastic strain fields differ in their shapes, especially the shapes of the tensile zones. The tensile zone of the simplified analysis is more uniform along the weld axis than that of the transient analysis. The phenomenon is understandably the consequence of simultaneously applying the temperature load along the weld in the simplified analysis. The loading fashion essentially excludes the transient effect of welding heat transfer. The phenomenon has also been noticed in the case study of the Tee joint. That study also reveals that the predicted distortions by the two approaches are very close, which is attributed to the similarity between the plastic fields from the transient analysis and simplified analysis. The difference between the two plastic fields has minimal influence on deviating the calculated distortions. The following evaluation of the analysis results will lead to the same conclusion.

Figure 5.9 shows the comparison of the transverse displacement fields from the transient and simplified analysis. The two displacement fields demonstrate twin-like distribution patterns. Their extreme values are close: $U_{\text{min}} = -1.32E-1$ mm and $U_{\text{max}} = 1.20E-1$ mm from the transient analysis; $U_{\text{min}} = -1.20E-1$ mm and $U_{\text{max}} = 1.06E-1$ mm from the simplified analysis. The deformed shapes of the plate are plotted in Figure 5.10. The deformed shapes computed by the two analyses are very similar. Both exhibit obvious transverse shrinkage and slight longitudinal shrinkage.

Compared in Figure 5.11 are the longitudinal residual stress fields from the two analyses. Their overall distribution patterns approximate each other very much. The difference can be found in the regions of weld start and end
where apparent end effect of transient heat transfer is present in the result of transient analysis. The two stress fields also have close magnitudes. The maximum tensile stress from the simplified analysis is about 93% of that of the transient analysis. The extreme value of compressive stress of the former is about 110% of the latter.

The computational time of the two analyses are listed in Table 5.2. Also included in Table 5.2 are the normalized CPU time and computational speed-up factor while the transient analysis is the reference case. Both analyses are executed on HP J5000 workstation. The speed-up factor of the simplified analysis is 16.2.

<table>
<thead>
<tr>
<th>Analysis</th>
<th>CPU Time (seconds)</th>
<th>Normalized CPU Time</th>
<th>Speed-up Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Transient Analysis</td>
<td>12168</td>
<td>100%</td>
<td>1.0</td>
</tr>
<tr>
<td>Simplified Analysis</td>
<td>753</td>
<td>6.2%</td>
<td>16.2</td>
</tr>
</tbody>
</table>

Table 5.2: CPU time consumed by the analyses of Butt joint performed on HP J5000 workstation
5.6 Conclusions

Even though the plastic strain field from the simplified approach does not perfectly match that from the detailed approach, it closely approximates the latter in terms of distribution pattern and maximum magnitude. The displacement fields computed by the two analyses are very similar. So are the distorted shapes of the plates. The unequivocal similarities are attributed to the agreement between the plastic fields of the two analyses. On the other hand, the difference between the two plastic fields results in the subtle differences between the displacement fields. The close match between the simplified and detailed analyses presented here serves as the proof of the hypothesis that the distortion can be accurately predicted if the plastic deformation the same as or similar to that formed during welding is introduced to the weldment. As expected, significant computational speed-up is garnered by using the simplified approach.
Figure 5.8: Transverse plastic strain fields from transient and simplified analyses. (a) Transient analysis; (b) Simplified analysis.
Figure 5.9: Comparison of the transverse displacement fields from transient and simplified analyses. (a) Transient analysis; (b) Simplified analysis.
Figure 5.10: Comparison of the deformed plate shapes from transient and simplified analyses. (a) Transient analysis; (b) Simplified analysis.
Figure 5.11: Comparison of the longitudinal stress fields from transient and simplified analyses. (a) Transient analysis; (b) Simplified analysis
CHAPTER 6
APPLICATION OF ENGINEERING APPROACH TO AUTOMOTIVE BODY STRUCTURES

The engineering approach developed in Chapter V is applied to two industrial structures in this chapter. The two structures, front rail assembly and engine cradle are automotive body structures. They belong to the very type of welded structures for which this Ph.D. dissertation work is conducted to develop the engineering approach to fast and accurate distortion prediction. They possess the same structural and process characteristics as those investigated in the two preceding chapters when the engineering approach is formulated. The structural and process characteristics make the welded structures suitable application objects of the engineering approach. The characteristics include:

- Steel structures made of low-carbon or high-strength low-alloy (HSLA) steels,
- Thin-wall structures with thickness of 1 to 2 mm,
- Gas Metal Arc Welding (GMAW) processes with high speed,
- Short welds with length up to 50 mm, and
- Full penetration in GMA welds.
6.1 Problem Definition

There are usually a large number of GMA welds in automotive body structures. For example, there are more than three hundreds welds in a typical car underbody frame and the total weld length can be up to eight meters. Like other auto body structures, the front rail assembly and engine cradle studied here are assembled with GMA welds. Welding distortion is always present in the structures. Even though in-plane shrinkage is the primary distortion mode of the short and full-penetration GMA welds used in the structures, the in-plane shrinkage induces bending distortion if the welds are deposited on space frames such as box beams. Hence, global bending is frequently observed in the structures.

Excessive amount of welding distortion in the body structures not only hinders the subsequent vehicle assembly process but also reduces the vehicle reliability and durability. In order to control welding distortion in the structures, it is necessary to optimize the structural design and welding process. The design and process optimization demands the capability of fast and accurate distortion prediction. Such optimization demand can be satisfied with the engineering approach but not the detailed transient approach utilized in the previous chapters. The high computational cost inherently associated with the transient approach prevents it from being applied to large welded structures. For example, tens of analysis cases need to be conducted in order to optimize the design and process parameters of the two structures mentioned above. If the
transient approach is employed, it may take months not days to complete the
design and process optimization, which neither fits any production schedule nor
is acceptable to any automobile manufacturers. On the other hand, the
engineering approach makes the optimization feasible due to its significantly
lower computational cost and reasonable accuracy.

The engineering approach utilized in this chapter has been developed in
Chapter Five. In general, the engineering approach is applicable to the front rail
assembly and engine cradle since they have the same structural and process
characteristics as the welded structures simulated in the two preceding
chapters, based on which the engineering approach is developed. Still, some
assumptions need to be made before it can be applied to the two body
structures. The engineering approach is formulated on single material and
certain welding parameters. There are multiple grades of steel in either one of
the two structures. Also the welding parameters used in their manufacture do
not fixate on one set of parameters but rather form a range or process window.
The following assumptions are made in order to apply the engineering
approach:

1) Both structures are made of the same steel as that involved in the
development of engineering approach in Chapter Five, namely Steel-A,
and

2) The GMA welds in both structures are deposited using the same set of
welding parameters as that simulated in Chapter Five.
The first assumption is made taking account of the following facts. Even though the steels made of the structures are not exactly same grade as that in Chapter Five, they are all HSLA steels and have similar material properties. The second assumption is given based on the following considerations. The set of welding parameters simulated in the formulation of engineering approach falls into the process windows of the welded structures, as demonstrated in Table 6.1. The above assumptions are meaningful from the perspective of robustness of the engineering approach, in other words, whether it can tolerate material and process variation of welded structures. It is not uncommon to have substantial variations in welding parameters for the nominal set of parameters specified. It is known that there is always scatter in material properties for any steel. The assumptions accommodate these variations and beyond. The robustness of engineering approach will be attested if it renders reasonably accurate distortion predictions under the assumptions.
<table>
<thead>
<tr>
<th>Development of Engineering Approach</th>
<th>Arc Voltage (V)</th>
<th>Welding Current (A)</th>
<th>Travel Speed (mm/s)</th>
<th>Heat Input (J/mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Front Rail Assembly</td>
<td>20~24</td>
<td>175~190</td>
<td>8.04~8.47</td>
<td>413~567</td>
</tr>
<tr>
<td>Engine Cradle</td>
<td>22~25</td>
<td>180~200</td>
<td>8.47~8.89</td>
<td>445~590</td>
</tr>
</tbody>
</table>

Table 6.1: Welding parameters used in the engineering approach development, front rail assembly, and engine cradle

6.2 Welding Distortion of Front Rail Assembly

The welded front rail assembly consists of five parts, identified by part number in Figure 6.1. There are 22 GMA welds in the assembly with a total weld length of 0.97 m. The weld locations are shown in Figure 6.1. The 22 welds are deposited sequentially during welding. The weld numbers shown in Figure 6.1 also designate the actual welding sequence that Weld 1 is deposited first and Weld 22 is the final weld deposited. The structure is clamped in the fixture and cooled down to room temperature before it is released from the fixture and taken to the measurement station. A total of 10 samples are welded using the same nominal welding parameters, as given in Table 6.1. Distortion measurements are performed for all the welded samples.
Distortion measurements are taken using filler gauges and a coordinate measurement machine (CMM) upon completion of welding. The CMM machine uses a probe to determine displacements to within ±0.05 mm. The measurements are taken at 66 locations in the front rail assembly illustrated in Figure 6.2. Among the 66 independent measurements, 14 are in the x-axis direction, 23 in the y-axis direction, and 29 in the z-axis direction. Although each measurement is referred to as a "location", some three measurements are actually taken at the same point. More details of the welding procedure and distortion measurements can be found in Reference 138.

Like the Tee joint in Chapter Five, the FE model of front rail assembly is constructed with shell elements. Figure 6.3 shows the element mesh. Four-node shell elements (ABAQUS element type S4R) and three-node shell elements (ABAQUS element type S3R) are used in the FE model. The model contains 6102 nodes and 6495 elements. As shown in Figure 6.3, the FE mesh is graded from fine to coarse as moving away from the welds which are marked as red elements. Boundary conditions are applied to the FE model in the analyses according to the actual locating and clamping schemes used in the welding experiments. The analysis procedure of the engineering approach is implemented in this study. Since the analysis procedure has been described in detail in Chapter Five, it is not recited here.

The results presented in this and next paragraph are obtained from the analysis using the engineering approach while the actual welding sequence and fixture are simulated. Figure 6.4 shows the distorted front rail assembly in x-y
plane (1-2 plane) after welding (but before fixture release) and after fixture release. The assembly is hold close to its original shape by the welding fixture before its release (after welding). At this stage, the global deformation is small although some local distortion is apparent. The assembly displays large global deformation after the fixture is removed. Even though moderate shrinkage in the x-axis direction is observed in the whole structure, large distortion occurs in the front rail in the y-axis direction. The distorted front rail has a bow shape with one end (left end in Figure 6.4) bending upward about 0.6 mm and its mid-section (closer to the other end) bending downward about 1 mm. The distorted bow shape in the front rail is caused by the extensive weld deposition on its top surface, including Weld No. 1, 2, 3, 5, 11, 12, 15, 16, 17 and 18, as labeled in Figure 6.1a. These welds constitute much more severe shrinkage on the top surface than that on the bottom surface. Figure 6.5 displays the distortion of the assembly in x-x plane (1-3 plane) after welding (but before fixture release) and after fixture release. Larger distortion in the z-axis direction is also observed after fixture release than that just after welding.

The FEA results are compared with the average measured displacements in Figure 6.6, in which a separate chart is plotted for each coordinate direction. The averaged measured values are plotted with the error bars representing the variations or range of the measurements over the 10 welded samples. The engineering approach predicts the correct sign of all the displacements except in two locations: point 25.5 in the y-axis direction and point 14 in the z-axis direction. The overall quantitative agreement between the FEA results and
experimental data is very good, which is confirmed by the statistical data listed in Table 6.2. If the data points in Figure 6.6 are plotted using continuous curves instead of discrete bars, two curves will be obtained standing for the measured and predicted displacement respectively. The correlation coefficient is used to indicate how the shapes of two curves are close or similar, with the value of 1.00 indicating a perfect match. The correlation coefficient between the FEA results and experimental results is 0.94. 71% of the predicted displacements fall within the range of the measured values. The average magnitude of the measured displacements is 0.25 mm and the predicted average displacement magnitude is 0.26 mm. Table 6.2 also lists the average difference (0.13 mm) between the mean measured displacements and predicted values at each of the 66 measurement points. The mean square error in Table 6.2 is computed using the following equation:

\[
\text{mean square error} = \frac{\sum_{i=1}^{n} (u_{i}^{\text{FEA}} - u_{i}^{\text{Experiment}})^2}{n}
\]

Equation 6.1

In the above equation, \(u_{i}^{\text{FEA}}\) is the predicted displacement and \(u_{i}^{\text{Experiment}}\) is the mean value of the measured displacement at each point and \(n\) is 66.
Figure 6.1: Front rail assembly. (a) A view from top; (b) A view from bottom.
Figure 6.2: Distortion measurement locations in front rail assembly
Figure 6.3: Finite element mesh of front rail assembly

Figure 6.4: Distorted front rail assembly in x-y (1-2) plane after welding and fixture release. Left: After welding but before fixture release; Right: After fixture release.
Figure 6.5: Distorted front rail assembly in x-z (1-3) plane after welding and fixture release. Left: After welding but before fixture release; Right: After fixture release.

Figure 6.6: Measured and predicted displacement in front rail assembly in different directions. (a) x-axis direction; (b) y-axis direction; (c) z-axis direction.
Figure 6.6 continued

(b)

(c)
<table>
<thead>
<tr>
<th>Statistical Data</th>
<th>Measured</th>
<th>Predicted</th>
</tr>
</thead>
<tbody>
<tr>
<td>Correlation Coefficient</td>
<td>---</td>
<td>0.94</td>
</tr>
<tr>
<td>Percentage within the Measurement Data Range</td>
<td>---</td>
<td>71%</td>
</tr>
<tr>
<td>Average Displacement Magnitude (mm)</td>
<td>0.25</td>
<td>0.26</td>
</tr>
<tr>
<td>Average Difference at Each Measurement Point (mm)</td>
<td>---</td>
<td>0.13</td>
</tr>
<tr>
<td>Mean Square Error (mm²)</td>
<td>---</td>
<td>0.025</td>
</tr>
<tr>
<td>Number of Points directionally off the measurement</td>
<td>---</td>
<td>2 out of 66</td>
</tr>
</tbody>
</table>

Table 6.2: Statistical data for measured and predicted displacements

6.3 Welding Distortion of Engine Cradle

The experimental procedures of engine cradle including welding and distortion measurement procedures are very similar to those of the front rail assembly and are elaborated on in Reference 138. There are a total of 45 GMA welds in the welded structure with a total weld length of 2.1 m. The FEA procedure for the engine cradle is the same as that for the front rail assembly described in the previous section, and therefore its details are not reiterated here. It is necessary to state that the engineering approach is utilized to predict the welding distortion in engine cradle, and that the actual welding procedure is simulated. The FEA model of engine cradle is shown in Figure 6.7. It consists of 9037 shell elements and 8017 nodes.
The distorted shape of cradle is illustrated in Figure 6.8. Predicted and measured distortions are compared for 3 critical dimensions shown in Figure 6.8, U, V, and W. The critical dimensions are in the x-y plane (1-2 plane). The results are listed in Table 6.3. $\delta_y^U$ is the distortion value of dimension U in the y-axis directions. $\delta_x^V$ is the distortion value of dimension V in the x-axis directions. $\delta_y^W$ is the distortion value of dimension W in the y-axis directions. A positive distortion value means that the corresponding dimension is enlarged after welding and a negative value indicates reduction in the dimension. The percentage difference between the measured and prediction distortions is calculated using the measured as base. A negative difference value denotes under-prediction of the FEA. The agreement between the prediction and measurement is excellent.

<table>
<thead>
<tr>
<th>Distortion</th>
<th>Measured (mm)</th>
<th>Predicted (mm)</th>
<th>Difference (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\delta_y^U$</td>
<td>2.10</td>
<td>1.93</td>
<td>-8.1%</td>
</tr>
<tr>
<td>$\delta_x^V$</td>
<td>-1.39</td>
<td>-1.60</td>
<td>15.1%</td>
</tr>
<tr>
<td>$\delta_y^W$</td>
<td>1.22</td>
<td>1.42</td>
<td>16.4%</td>
</tr>
</tbody>
</table>

Table 6.3: Measured and predicted distortions of the critical dimensions
Figure 6.7: Finite element mesh of engine cradle
Figure 6.8: Distorted shape of engine cradle
CHAPTER 7

CONCLUSIONS AND DISCUSSIONS

This Ph.D. dissertation work is conducted to obtain better understanding and characterization of the plastic deformation that leads to in-plane shrinkage in thin-wall structures. This work is also performed to develop a fast and accurate engineering approach for predicting the welding distortion in large thin-wall structures, such as automotive body structures. These research efforts are made to prove that in-plane shrinkage is the root cause of welding distortion in thin-wall structures.

In order to develop the engineering approach based upon the principle that plastic deformation is the root source of welding distortion, a hypothesis is proposed regarding such development. The hypothesis is stated as the following: welding distortion can be accurately predicted if the plastic deformation same as or similar to that induced by welding is introduced into the FEA model. The dissertation research is conducted to validate the hypothesis. On the other hand, the hypothesis serves as a guideline to the research of this dissertation. Specifically, there are two important aspects of the research work as implied in the hypothesis:

1) Characterizing the plastic deformation produced by welding, and
2) Generating the plastic deformation equivalent to the actual one in the FEA model without resorting to detailed transient analysis.

The FEA is the tool used in this dissertation to study welding plastic deformation. Before FE simulations are carried out for this purpose, a proper FEA model is formulated so that it is capable of rendering accurate distortion computation as well as capturing all the data necessary to investigate the plastic deformation. The FEA model is dubbed as Moving Source Model and its procedure is a decoupled or also-called sequentially-coupled thermomechanical analysis. The model simulates the physical phenomena that have significant influence on plastic deformation and distortion, such as temperature-dependent material properties, continuous filler metal deposition, and plastic strain relaxation. The analysis results of the FEA model including both thermal and mechanical results are validated against experimental measurements.

After being verified by the experimental results, the FEA model is utilized to perform the welding simulations on butt-welding of thin-wall plates, which have the same structural and process characteristics as those of large thin-wall automotive body structures. The analysis results are extracted and analyzed to study the in-plane shrinkage and plastic behaviors. The concept of effective shrinkage is proposed and employed to interpret the transverse shrinkage behavior in the plate. The plastic strain distributions are examined in combination with the peak temperatures accumulated during welding. Evolution of the plastic strains is investigated using the plastic strain history data, which provide many insights into some important phenomena such as plastic strain
relaxation and filler metal deposition. The most important findings garnered from this study are:

1) The accumulated plastic strains are determined by the peak temperature and material's plastic temperature range, and

2) Peak temperature is the thermal parameter that controls the plastic strains.

The two findings along with the others stated in the corresponding chapter (Chapter Four) constitute a theoretical basis for constructing the engineering approach.

The research issue of central importance in developing the engineering approach is how to introduce the plastic deformation equivalent to the actual one into the FEA model. Temperature load is used in this work and then the central issue becomes formulating the temperature load, namely its distribution including area of application and magnitude. The distribution is determined based upon the findings obtained in the study of in-plane shrinkage and plastic behaviors mentioned above. The engineering approach is established by applying the temperature load to the FEA model. Two simple joints are simulated using the Moving Source Model and the predicted distortions and plastic strains are regarded as actual ones. The engineering approach is also applied to the joints. Its distortion predictions match those of the Moving Source Model only when the temperature load generates the plastic strains approximating the actual ones.
The engineering approach is used to predict the welding distortions of two automotive body structures, including front rail assembly and engine cradle. The predicted distortions are compared with the measurements and excellent agreements exist between them. The engineering approach provides a very useful means to achieve fast and accurate distortion prediction for large welded thin-wall structures. Successful development and application of the engineering approach validate the hypothesis about distortion prediction proposed in this dissertation.

It is demonstrated that the engineering approach developed in this dissertation can be used for practical purpose. However, there are obvious limitations of the approach. It is developed for thin-wall structures assembled with short and full-penetration GMA welds. The engineering approach may not be applicable to the welded structures with different structural and/or process characteristics. Since it is formulated using one set of welding parameters and the material properties of one particular steel, the engineering approach may yield erroneous predictions if the simulated welding parameters are deviate too much from those used in the engineering approach development, and/or if a very different material is utilized, i.e. TMCP steels. The application scope of engineering approach can only be thoroughly defined with more investigations.

Welding plastic deformation is influenced by many factors. It is impossible to investigate the effects on plastic deformation of all the influencing factors within single dissertation research. The author considers the following factors should be incorporated in the future research on distortion prediction and
welding plastic deformation, including interaction between neighboring welds, welding sequence, and weld bead shape.
REFERENCES


submitted to Advanced Manufacturing Technology Development, Ford Motor Company.


124 Unterweiser, P. M. and Penzenik M. 1979. Worldwide Guide to Equivalent Irons and Steels. ASM.


