CHAPTER 3

FINITE ELEMENT SIMULATION OF WELDING

3.1 INTRODUCTION

Though welding process has many distinct advantages over other joining processes, it suffers from some major disadvantages like the evolution of residual stresses and distortions in the weldments. With the help of simulation tools like finite element method, the extent of the effects of welding input conditions can easily be estimated with minimum cost of experiment. As a test study, a 3D finite element model was developed to predict the complete transient temperature fields in a butt-joint of stainless steel plates. The predictions of the temperature histories at three different locations in the welded plate were compared with the corresponding experimental results of Ravichandran (1998), involving the same PAW input conditions. Weld bead shapes for all the welding input conditions were also predicted using the same thermal model. Then transient thermo elasto plastic simulation was carried out to determine the distribution of longitudinal residual stresses in the welded plate. With the knowledge gained in that model, a 3D finite element model to predict the longitudinal residual stress fields produced in a T-joint of carbon steel plates was developed.

3.2 WELDING SIMULATION OF A BUTT JOINT

3.2.1 Transient Thermal Simulation

In transient thermal analysis, the transient temperature field (T) in a plate during welding is a function of time (t) and spatial coordinates (x, y, z),
and is determined by the non-linear heat transfer equation (2.1) provided in section 2.3.1. Since the plate experiences surface heat losses in the form of convection and radiation, the boundary conditions as specified in equations (2.2) and (2.3) are required to be imposed in the plate regions open to the ambience.

The regions covered by the moving heat source are applied heat densities according to Goldak’s double ellipsoidal heat source model as described in equations (2.4) and (2.5).

### 3.2.1.1 Model geometry

A square butt joint of AISI 304 grade austenitic stainless steel plates of 175 mm x 150 mm x 6.7 mm size was modeled. One half of the weld plate along its width was meshed to make use of the symmetry about the weld line as shown in Figure 3.1. In order to strike a balance between accuracy and computational time of the FE model, a fine and uniform mesh using 8 node brick element was generated in the welding region. In other regions, a coarse mesh consisting of 4 node tetrahedron elements was generated. The effects of convection and radiation were accounted for by overlaying the surfaces of the plate with the surface elements as specified in the commercial popular finite element program ANSYS. The convective heat transfer coefficient and the emissivity were to be 11.14 W/m\(^2\) K and 0.4 respectively (Ravichandran 1998). The temperature dependent thermal properties (Ravichandran 1998) such as thermal conductivity and specific heat at various temperatures as given in Table 3.1 were considered. A 2D FE model, at the mid-cross section of the plate was used to determine approximate values of the heat source parameters on a trial and error basis to match the predicted peak temperature at specified location with the experimental peak value. The obtained heat source parameters are given in Table 3.2.
Table 3.1 Temperature dependent thermal properties

<table>
<thead>
<tr>
<th>Temperature (°C)</th>
<th>-173</th>
<th>177</th>
<th>1627</th>
<th>1777</th>
<th>3227</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermal Conductivity (W/m °C)</td>
<td>7</td>
<td>14</td>
<td>35</td>
<td>55</td>
<td>60</td>
</tr>
<tr>
<td>Specific Heat Capacity (J/kg °C)</td>
<td>250</td>
<td>475</td>
<td>700</td>
<td>700</td>
<td>700</td>
</tr>
</tbody>
</table>

In order to simulate the welding torch movement, the calculated volumetric heat densities have to be applied to specific nodes around the welding zone in the FE model. For this purpose, a local coordinate frame was created at the point from where the welding process was to be initiated. The origin of the local coordinate frame thus created was treated to be the centre of the heat source. The spatial coordinates of the nodes which came under the influence of the heat source, were extracted from the mesh and were given as inputs to a sub routine which calculated the respective heat densities to be

Figure 3.1 Meshed model of the half of the plate to be btt joined
applied to the nodes. Then the thermal model was solved for the unknown nodal temperatures and this was continued by moving the local coordinate frame in small increments of distance with the corresponding time step, in the direction of the welding till the other end of the plate was reached. Thereafter, heat densities were no longer applied to the thermal model and still the thermal model was solved until the welded plate was in thermal equilibrium with the ambience.

In order to analyze the effect of welding process parameters on the temperature distribution and its variations, three different sets of welding process inputs as considered by Ravichandran (1998) were applied. The details of welding input conditions are given in Table 3.2. The value for the arc efficiency was taken to be 0.64. Temperatures were measured at locations A, B and C, as shown in Figure 3.2, using thermocouples.

![Figure 3.2 Temperature measurement locations](image)
3.2.2 Non-linear Transient Stress Simulation

In the stress analysis, the finite element model is represented by a system of equilibrium equations obtained from the discretization of the virtual work equation (Tekriwal and Mazumder 1991). As welding simulation is highly non-linear and path dependent, the system of equilibrium equations is solved in small time steps.

### Table 3.2 Welding process and heat source parameters

<table>
<thead>
<tr>
<th>SL. No.</th>
<th>PTAW Process Parameters</th>
<th>Heat input $Q$ (kJ/mm)</th>
<th>Heat source parameters (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Voltage (V)</td>
<td>Current (Amps)</td>
<td>Welding speed (mm/min)</td>
</tr>
<tr>
<td>1</td>
<td>31</td>
<td>175</td>
<td>170</td>
</tr>
<tr>
<td>2</td>
<td>31</td>
<td>192.5</td>
<td>170</td>
</tr>
<tr>
<td>3</td>
<td>30</td>
<td>210</td>
<td>150</td>
</tr>
</tbody>
</table>

Each time step involves the calculation of incremental nodal displacements by Newton’s method of iteration. At the end of iteration, the state variables of strain and stress are updated and residual force is calculated and compared with the specified force tolerance (ANSYS, 2002).

3.2.2.1 Model geometry

The same finite element mesh was used with the thermal elements replaced by the structural elements. Table 3.3 shows the temperature dependent mechanical properties (Ravichandran 2002) of the weld plate used in the simulation. Since symmetry was assumed along the welding line during transient thermal analysis, the normal displacement of the nodes contained in the symmetry plane was arrested. Other minimum boundary conditions were
imposed to the plate to prevent its rigid body motion. In the elasto-plastic analysis, the plasticity of the material was modeled by von Mises’ yield criterion, associated flow rule and isotropic hardening.

**Table 3.3 Temperature dependent mechanical properties**

<table>
<thead>
<tr>
<th>Temperature (ºC)</th>
<th>30</th>
<th>400</th>
<th>810</th>
<th>1000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s Modulus (GPa)</td>
<td>275</td>
<td>211</td>
<td>140</td>
<td>60</td>
</tr>
<tr>
<td>Coefficient of Thermal Expansion (/ºC)</td>
<td>18E-06</td>
<td>18.8E-06</td>
<td>19.6E-06</td>
<td>20E-06</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.255</td>
<td>0.325</td>
<td>0.2683</td>
<td>0.24</td>
</tr>
<tr>
<td>Yield stress (MPa)</td>
<td>280</td>
<td>188</td>
<td>87</td>
<td>40</td>
</tr>
</tbody>
</table>

The temperature histories solved in the transient thermal simulation of the entire period of welding up to the complete cooling of the welded plate to the room temperature were given as inputs to the non-linear transient stress simulation, in the same sequence of time steps as used in thermal solution. During the stress simulation, a cut-off temperature of 800 ºC was used. The use of cut-off temperature in the non-linear transient stress simulation of welding is made inevitable due to the reason that the reliable material property values are not available especially at high temperatures. The temperatures obtained at various points of time in the previous transient thermal simulation were given as input to the structural model to solve for the unknown nodal displacements.

**3.3 WELDING SIMULATION OF A T-JOINT**

Based on the experience gained in the welding simulation of the butt-joint, an attempt was made to carry out the simulation of a T-joint. The
welding simulation of a T-joint involves the same sequence of steps that was followed in the simulation of welding of the butt-joint.

### 3.3.1 Transient Thermal Simulation

The details of T-joint geometry are given in Figure 3.3. The length of the model was fixed as 210 mm so that the length of the fillet weld on each side of the specimen could be divided into three step welds for the purpose of optimizing the welding sequence to be considered later in chapter 6. Single pass weld on each side was considered as multipass would require enormous computational time especially in the analysis of effects of various welding sequences. Figure 3.4 shows the finite element meshed model of the T-joint. A fine mesh was provided in the fillet weld region as it would involve very high temperature gradients, whereas a relatively coarse mesh was provided in regions where the temperature gradients would be less. The solid brick element with 8 nodes (Solid70) was used throughout the entire mesh. In order to account for surface heat losses by convection and radiation, surface elements were overlaid on the outer surfaces of the main mesh representing the specimen. The total number of nodes and elements including surface elements are 14229 and 11832 respectively.

Goldak’s double ellipsoidal model of heat source was used to obtain the transient temperature histories. A local coordinate frame was created at the start of the fillet and was moved gradually in the direction of welding in order to ensure smooth change of temperatures in the model. The arc voltage was considered as 30 V, the welding current was 210 Amps and the welding speed was 4.467 mm/min. Temperature dependent thermal properties (Pankaj Biswas 2007) given in Table 3.4 were used for the transient thermal analysis. Latent heat effects were accounted in the analysis by increasing the specific heat at the corresponding temperatures of phase change.
Figure 3.3 T - Joint geometry

Figure 3.4 Meshed model of T-joint
Table 3.4 Temperature dependent thermal properties

<table>
<thead>
<tr>
<th>Temperature (ºC)</th>
<th>0</th>
<th>400</th>
<th>700</th>
<th>800</th>
<th>810</th>
<th>1500</th>
<th>1550</th>
<th>1600</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermal Conductivity (W/m ºC)</td>
<td>52</td>
<td>39</td>
<td>29</td>
<td>26</td>
<td>26.1</td>
<td>34</td>
<td>42</td>
<td>50</td>
</tr>
<tr>
<td>Specific Heat Capacity (J/kg ºC)</td>
<td>400</td>
<td>300</td>
<td>800</td>
<td>1400</td>
<td>900</td>
<td>900</td>
<td>1600</td>
<td>400</td>
</tr>
</tbody>
</table>

The filler metal addition was modeled by using the element birth-death feature available in the finite element software package, ANSYS. Initially, the elements which constituted the fillet weld were killed (deactivated) and they were made alive (activated), when they were under the influence of the moving heat source. The transient thermal simulation was carried out till the welded plates were cooled to room temperature.

3.3.2 Non-linear Transient Stress Simulation

After the transient thermal simulation, the same finite element mesh without the surface elements was used for the stress simulation. Suitable displacement boundary conditions were given to the structural model so as to prevent the rigid body motion of the model. Temperature dependent mechanical properties (Pankaj Biswas 2007) given in Table 3.5 were used for the stress simulation. Since the non-linear stress simulation is path dependent, the temperature histories obtained at various time steps in the previous thermal simulation, were given as inputs at the corresponding time steps to the structural model.
Table 3.5 Temperature dependent mechanical properties

<table>
<thead>
<tr>
<th>Temperature (ºC)</th>
<th>30</th>
<th>200</th>
<th>315</th>
<th>650</th>
<th>800</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s modulus (GPa)</td>
<td>206</td>
<td>196</td>
<td>183</td>
<td>48</td>
<td>14</td>
</tr>
<tr>
<td>Coefficient of thermal expansion (ºC)</td>
<td>5E-06</td>
<td>12.45E-06</td>
<td>13.27E-06</td>
<td>14.72E-06</td>
<td>15E-06</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.31</td>
<td>0.31</td>
<td>0.31</td>
<td>0.31</td>
<td>0.31</td>
</tr>
<tr>
<td>Yield stress (MPa)</td>
<td>248</td>
<td>227</td>
<td>200</td>
<td>69</td>
<td>21</td>
</tr>
</tbody>
</table>

For modeling plasticity, von Mises’ yield criterion, associated flow rule and isotropic hardening were considered. A cut-off temperature of 800 ºC was used in the modeling. The computational time taken for both the thermal and structural analyses was around 30 hours in a 1 GB RAM 3 GHz P IV personal computer.

3.4 RESULTS AND DISCUSSIONS

3.4.1 Temperature Distributions in Butt-Joint

Figures 3.5 - 3.7 show the temperature distributions in the butt welded plate during welding at different instances under different heat input conditions. It is observed that temperature distributions around the source are typical of the fusion zone (FZ) and the heat affected zone (HAZ). It is also noticed that the heat source preheats a small region of weld plate ahead of the moving heat source. The peak temperature at the centre of fusion zone is in proportion to the heat input. The heat generated by the moving heat source along the weld is gradually propagated towards all directions in the plate through conduction, convection and radiation.
Figure 3.5 Temperature distributions in Butt-Joint at 47.65 s (1.225 kJ/mm)

Figure 3.6 Temperature distributions in Butt-Joint at 47.65 s (1.348 kJ/mm)
Figure 3.7 Temperature distributions in Butt-Joint at 54 s (1.613 kJ/mm)

3.4.2 Temperature Histories at Specified Locations in Butt-Joint

The predicted temperature histories at locations A, B and C and the corresponding measured temperatures reported in the literature are shown in Figures 3.8-3.10 for comparison. It is found that the predicted values are in good agreement with the experimental results.

Figure 3.8 Comparison of temperature histories at location A
Figure 3.9 Comparison of temperature histories at location B

Figure 3.10 Comparison of temperature histories at location C
It is found that the predicted peak temperatures, heating rates and cooling rates in the simulated model match reasonably well with the experimental results which bring out the efficacy of the developed finite element model.

The accuracy of the FE model can be further increased by assigning the thermal properties of plasma arc to the nodes very close to the heat source. This is very essential to precisely model keyhole effect of plasma arc welding. This was not considered in the present model for lack of reliable data.

### 3.4.3 Effects of Welding Process Parameters on Bead Geometry in Butt-joint

The primary advantage of a finite element model is that more information the model can be extracted from the simulated results than that from an experiment which requires to be carried out separately for different objectives (Lindgren 2007). As the shape of weld bead geometry comprising fusion and heat affected zones greatly influences the strength of the weld (Murugan and Gunaraj 2005), it is always important to predict the bead geometries from the finite element model. The predicted bead geometries for various heat input conditions are shown in Figures 3.14 -3.16. It is evident that the increase in heat input widens the fusion and heat affected zones and also increases the depth of penetration as well as the bead cross sectional area.
Figure 3.11 Predicted bead geometry at 1.225 kJ/ mm

Figure 3.12 Predicted bead geometry at 1.348 kJ/ mm
3.4.4 Effect of Heat Input on Residual Stress Field in Butt-Joint

Figure 3.17 shows the predicted longitudinal residual stress field at the mid cross-section of the welded plate in the transverse direction of welding for the three different heat input conditions. It is evident that tensile residual stresses are resulted in the weld zones, which are balanced by the compressive residual stresses at the opposite sides. It is also obvious that the tensile residual stress zone is enlarged in accordance with the increase in the HAZ at higher heat input.
3.4.5 Residual Stress Field in T-Joint

When the heat input was given to the T-joint in the manner described in the section 3.3.1, the temperature distributions were obtained as shown in Figure 3.18. It is evident from the figure that the temperature distributions obtained are typical of a fusion welding process. The temperature histories, thus obtained in the transient thermal analysis, were given as input in the stress simulation. Figure 3.19 shows the distributions of the longitudinal residual stresses at the mid-section of the horizontal plate, when the welded T-joint attained the room temperature on cooling. It is evident from the figure that the tensile stresses were developed in the weld zone. These tensile residual stresses gradually decrease in the transverse direction away from weld center line and become compressive residual stresses towards the edge of the plate. This is because during the cooling
phase when the temperature of the weld zone falls rapidly, the weld metal tends to contract. This contraction of the weld metal is constrained by neighborhood of the weld zone, resulting in the tensile residual stresses in the weld zone. Compressive residual stresses were resulted in order to balance the tensile residual stresses for the equilibrium of the T-joint.

Figure 3.15 Temperature distributions in T – Joint 20.4 s
Figure 3.16 Longitudinal residual stress distribution in T – Joint