Residual Deflections and Stresses in a Thick T-Joint Plate Structure

Adak M1* and Soares CG2

1Department of Applied Mathematics, Yeshwantrao Chavan College of Engineering, Nagpur-441110, Maharashtra, India
2Centre for Marine Technology and Engineering (CENTEC), Technical University of Lisbon, Instituto Superior Técnico, Lisbon, Portugal

Abstract

In this paper transient thermo-elastic-plastic problem of a thick rectangular T-joint fillet weld plates is solved numerically in which we need to determine the temperature distribution, thermal deflections and stresses when the boundary conditions are known. At the T-joint section (weld pool area) high constant temperature is considered initially and convection boundary conditions are applied at other three edges as well as top and bottom surface of the plate. T-joint section and vertical plate is fixed (case 1) for plate stability, only T-joint section is fixed (case 2) as a boundary constraint for stress analysis. Numerical technique (Finite element method, Ansys software) is used to obtain the solution of the boundary value problem.

Keywords: Thermo-elastic-plastic analysis; Transient temperature distribution; Residual deflection; Residual stress; Finite element method

Introduction

When a material is heated with initial high temperature in a weld joint (T-joint section), the highly localized transient heat and strongly nonlinear temperature fields in both heating and cooling processes cause non-uniform thermal expansion and contraction and thus result in plastic deformation in the weld and surrounding areas. Here T-joint corner section (initially temperature is applied) of the plate can be assumed to be fixed to reduce the deformation and induces stress due to fixed constraint. For the stress analysis different boundary constraints are considered in the present study.

T-joint fillet welds are extensively used in construction, shipbuilding, steel bridges and supporting frames for pressure vessels and piping, etc. Residual deformation can occur near the T-joint and base metal due to localize heating and subsequent rapid cooling by welding process. The accurate prediction of welding distortions is extremely difficult because of thermal and mechanical conditions created during the welding process including locally high temperatures, temperature dependent material properties and heat source model.

In this study, two thick rectangular plates with same dimension are considered for welding process at T joint section. Due to simplifying heat source model and time consuming, initially high temperature is considered at T-joint section. To analyze the thermo-mechanical problem convection boundary conditions at all free edges of both plates and top, bottom surface and temperature dependent material properties, Tables 1-5 are considered. Numerical technique Finite element method through ANSYS software [1] is used to obtain the temperature distribution, residual deformation and stresses after heat-transfer at cooling stage.

The research activity in welding simulation started decades ago. Rosenthal [2] was among the first researchers to develop an analytical solution of heat flow during welding based on conduction heat transfer for predicting the shape of the weld pool for two and three dimensional welds. Some researcher have also developed the thermal finite element simulation to investigate the temperature distribution of a metal such as Kamala and Goldak [3], Nguyen et al. [4], Tekriwal and Muzumder [5], Yeung, and Thornton [6], and Bonifaz [7].

Over the past few years, finite element methods are used to predict distortion and residual stresses due to welding operations such as the studies by Friedman [8], Brown and Song [9], Mechaleris and Debicciari [10].

The effect of welding parameters, welding sequence, and welding joint geometry like butt joint, T-joint, corner joint is very important. Tsai and Cheng [11] investigated the distortion mechanism and the effects of welding sequence on thin panel distortions using a finite element analysis approach. Teng et al. [12] investigated the residual stresses and distortion of T-joint fillet welds using 2D finite element

<table>
<thead>
<tr>
<th>°C</th>
<th>Kg/m³</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>7800</td>
</tr>
<tr>
<td>1450</td>
<td>7438</td>
</tr>
<tr>
<td>3000</td>
<td>7438</td>
</tr>
</tbody>
</table>

Table 1: Steel density versus temperature.

<table>
<thead>
<tr>
<th>°C</th>
<th>W/m°C</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>55</td>
</tr>
<tr>
<td>750</td>
<td>29</td>
</tr>
<tr>
<td>800</td>
<td>25.4</td>
</tr>
<tr>
<td>1450</td>
<td>31</td>
</tr>
<tr>
<td>3000</td>
<td>31</td>
</tr>
</tbody>
</table>

Table 2: Steel thermal conductivity versus temperature.

<table>
<thead>
<tr>
<th>°C</th>
<th>J/kg (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>388</td>
</tr>
<tr>
<td>700</td>
<td>894</td>
</tr>
<tr>
<td>740</td>
<td>1234</td>
</tr>
<tr>
<td>860</td>
<td>610</td>
</tr>
<tr>
<td>1450</td>
<td>814</td>
</tr>
<tr>
<td>3000</td>
<td>814</td>
</tr>
</tbody>
</table>

Table 3: Steel specific heat versus temperature.

© 2016 Adak M, et al. This is an open-access article distributed under the terms of the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original author and source are credited.

Received September 12, 2016; Accepted October 03, 2016; Published October 06, 2016


Copyright: © 2016 Adak M, et al. This is an open-access article distributed under the terms of the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original author and source are credited.
The results show that the temperature histories at various locations from the weld line are in reasonable agreement with the result obtained by Long [24]. The peak temperatures for both models are almost identical. The distances from the weld line of the verification model are slightly different from Long’s possibly due to mesh size. Figures 1 and 2 show the temperature distribution curves from Long’s and verification models, respectively.

In the present study T-joint plate structure type of model and same procedure have been followed which are described in below.

Problem Formulation

Thermal analysis

The conduction heating of the solid with constant temperature source at the fixed end is considered to simulate the welding process for 2-D problem reported by Sunar et al. [25]. Here, the 3D transient heat conduction equation for a thick rectangular T-joint plate heated at T-joint section is written as:

\[
\rho C \frac{\partial T}{\partial t} = \frac{\partial}{\partial x} \left( k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( k \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left( k \frac{\partial T}{\partial z} \right)
\]  

(1)

where \( \rho \), \( C \), and \( k \) are density, specific heat and thermal conductivity depends on temperature respectively.

Subject to the initial and boundary conditions in both horizontal and vertical plates:

Verifies the butt-welding analysis using finite element analysis

In this study, no experimental data are available to verify the simulation results and thus the results of the simulation are compared to numerical results obtained in a previous research. After validating the simulation results with the numerical results, the same finite element modeling is used to simulate the welding process through out this study.

A finite element simulation of a simple butt-joint welding for 3D model is verified comparing with the work of Long et al. [24]. To do this, a simple butt joint welding whose welding parameters are consistent to those of Long’s model was simulated by using the ANSYS finite element code. The model then is verified by comparison of its temperature distribution at some chosen nodes with the Long’s model. Plate geometry of the butt-joint welding is similar to the Long’s model.
\[ T(x, y, z, 0) = T_0 \text{ (ambient temperature).} \]  
(2)

At whole plate except T-joint section.

\[ T(x, y, z, 0) = T_1 \text{ (fixed high temperature for weld pool at T joint)} \]  
(3)

At the free surface except in the weld pool of the rectangular plate (Figure 3), convection boundary conditions are considered at horizontal and vertical plate. Therefore,

\[ \frac{\partial T}{\partial z} = \frac{h}{\rho c_p} \left[ T(x, y, z, t) - T_{amb} \right] \]  
(4)

On the surface along the x axis and parallel to x axis,

\[ \frac{\partial T}{\partial y} = \frac{h}{\rho c_p} \left[ T(x, y, z, t) - T_{amb} \right] \]  
(5)

On the surface top and bottom surface,

\[ \frac{\partial T}{\partial y} = \frac{h}{\rho c_p} \left[ T(x, y, z, t) - T_{amb} \right] \]  
(6)

where \( h \) is the convection coefficient.

The heat exchange between the welded plate and its surroundings during welding and subsequently, cooling takes place by both convection and radiation. In the thermal model the convection coefficient \( h = 15 \text{ W m}^{-2} \), emissivity \( \varepsilon = 0.9 \) and the Stefan-Boltzmann constant \( \sigma = 5.669 \times 10^{-8} \text{ W m}^{-2} \text{ C}^{-4} \) are considered by Long et al. [15].

**Mechanical Analysis**

Due to highly localized temperature at T-joint section as a weld pool, a very large temperature variation occurs over a small region. Consequently, considerable thermal stresses are generated due to the thermal gradient.

The thermal elasto-plastic material model based on the von Mises yield criterion and the isotropic strain hardening rule is considered. Stress-strain relations can be written as

\[ [\sigma] = [D] [\varepsilon'] \]  
(7)

\[ [D] = [D]^e + [D]^p \]  
(8)

and

\[ [\varepsilon'] = [\varepsilon] - [\varepsilon'] \]

Where \([D]^e\) is the elastic stiffness matrix, \([D]^p\) is the plastic stiffness matrix and \([\varepsilon], [\varepsilon']\) are the total and thermal strain vectors respectively.

The above stress-strain relations can be written in Cartesian coordinates of \((x, y, z)\) as,

\[ \varepsilon_x = \frac{1}{E} [\sigma_x - \nu \sigma_y] + \alpha \Delta T(x, y, z, t) \]  
(9)

\[ \varepsilon_y = \frac{1}{E} [\sigma_y - \nu \sigma_x] + \alpha \Delta T(x, y, z, t) \]  
(10)

\[ \varepsilon_z = \frac{1}{E} [\sigma_z - \nu \sigma_x] + \alpha \Delta T(x, y, z, t) \]  
(11)

\[ \gamma_{xy} = 2(1+\nu) \frac{\tau_{xy}}{E} \]  
(12)

\[ \gamma_{yx} = 2(1+\nu) \frac{\tau_{yx}}{E} \]  
(13)

\[ \gamma_{zx} = 2(1+\nu) \frac{\tau_{zx}}{E} \]  
(14)

Where \(E, \nu\) and \(\alpha\) are the modulus of elasticity, Poisson’s ratio and coefficient of thermal expansion, respectively. In the above equation, \(\Delta T(x, y, z, t)\) represents the temperature rise at a point \((x,y,z)\) at time \(t\) with respect to that at \(t = 0\) corresponding to a stress free condition.

Boundary conditions: T-joint section and vertical plate are fixed as a boundary constrain during heat transfer (case 1). Only T-joint section is fixed as a boundary constrain during heat transfer (case 2).

Due to above boundary conditions with different temperature at different point \((x,y,z)\), thermal deflection and stress field are investigated using FEM approach.

**Solution of the Problem**

Two rectangular plates with same dimension 100 mm × 250 mm × 6 mm (Figure 3) are consider for finite element modeling.

For thermal analysis, assume,

\[ T \approx T = \sum N_i T_i = [N][T] \]  
(15)

where \([N]\) is the shape function.

Applying Galerkin criterion, the heat conduction equation (1) can be written to each element as follows:

\[ \int \{N_i \frac{\partial^2 T_i}{\partial x^2} + N_j \frac{\partial^2 T_i}{\partial y^2} + N_k \frac{\partial^2 T_i}{\partial z^2}\} dxdydz = \int \{N_i \rho c_p \frac{\partial T_i}{\partial t}\} dxdydz \]  
(16)

Using by parts,

\[ \int \{N_i \frac{\partial T_i}{\partial x} + N_j \frac{\partial T_i}{\partial y} + N_k \frac{\partial T_i}{\partial z}\} dS = \int \{[N_i \frac{\partial N_i}{\partial x} + N_j \frac{\partial N_i}{\partial y} + N_k \frac{\partial N_i}{\partial z}] T_i\} dxdydz = 0 \]

Using convection boundary conditions (2) - (6), final equation takes form as

\[ [C] \frac{d[T]}{dt} = [K][T] \]  
(17)

Where, \([C]\) is specific heat matrix, \([K]\) is conductivity matrix, \([T]\) is vector of nodal temperature, \(d[T]/dt\) is vector of time derivative of \([T]\).
Convergence and Stability Scheme

The terms $\frac{dT}{dt}$ and $T$ are written in various forms in different time stepping schemes.

$$[C]\left\{\frac{T(t+\Delta t) - T(t)}{\Delta t}\right\} = [K]\{\partial (T(t + \Delta t) + (1 - \partial)T(t))\}$$

This analysis requires an integration of the heat conduction equation with respect to time. The Crank-Nicholson theta integration ($\theta = 1/2$) scheme is tested for convergence and stability.

Material Properties

In thermal-elastic-plastic analysis, temperature dependent material properties are used.

Steel grade AH36 (high strength shipbuilding steel) is implemented with temperature dependent properties as specified in Tables 1-5 and Figure 4.

Bilinear Isotropic Hardening option was adopted and it uses the von Misses yield criteria coupled with an isotropic work hardening assumption shown in Figure 5. This option is often preferred for large strain analyses. The material behaviour is described by a bilinear stress-strain curve starting at the origin with positive stress and strain values. The initial slope of the curve is taken as the elastic modulus of the material. At the specified yield stress, the curve continues along the second slope defined by the tangent modulus (having the same units as the elastic modulus). The tangent modulus can not be less than zero nor greater than the elastic modulus.

**Element Type**

For thermal analysis a 3D 8-node element (SOLID 70) with three degrees of freedom at each node (translations in the nodal x, y and z directions) is used, and for structural analysis the element type is SOLID 185. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capacities. This choice was arrived at after a parametric study using different element types shown in Figure 6, which led to the conclusion that choosing an element having both thermal and structural characteristics is according to experience not optimal.

**Mesh Generation**

In order to generate the system of equations, as well as to find the result at each point of the rectangular plate, a mesh is generated throughout the volume. In order to achieve this, the ANSYS program allows for control of the size and geometry of the mesh in order to obtain the most precise solution. However, a very large number of grid points can compromise the computer's capacity. Figure 7 shows the mesh in the symmetry half of the rectangular plate. Welding direction being z-axis, sharp temperature gradients are expected along both x and y-axes. Therefore very fine grid has been adopted in the weld bead region; finer mesh grid has been chosen away from weld region and course grid is chosen along welding direction (z-axis).

**Birth and Death Feature**

The material deposition is modeled using an element "birth and death" technique. To achieve the death element effect, ANSYS code does not actually remove the element from the model. Instead, the weld elements are first deactivated by multiplying their stiffness by a huge reduction factor. Meanwhile, to obtain the birth element effect, the ANSYS program reactivates the death element by allowing its stiffness, element load etc. return to their original values.

**Results and Discussion**

Numerical simulation is done for thermal analysis as well as mechanical analysis using equations (1)- (17). Temperature field at different heating periods are obtained numerically and corresponding deflection shape and stress fields are obtained using finite element method.
Figures 8 and 9 shows the temperature distribution in the rectangular T-joint plate for different heating periods. The decay of temperature profiles is sharp in the early heating period and as the heating period progresses, it becomes more gradual (Figure 2). As the time progress to 30 sec, temperature contours extend in the transverse direction (x axis), while temperature distribution in the longitudinal direction (y axis) does not change.

Figure 2 shows the temperature distribution along transverse direction (x axis) for different time periods and in heating zone, temperature is high, gradually temperature decrease.

Figure 10 shows the deformed shape after cooling process due to boundary condition (case 1). Significant bend-up residual distortion is observed. In this case, boundary conditions are applied in heating direction as well as in vertical plate, so residual distortion can be interpreted in horizontal plate.

As boundary constraint is applied along heating direction only (case 2), deformation shape is like Figure 11. In this case horizontal...
plate as well as vertical plate both is deformed. So, boundary constraint is most effectible in residual deformation.

According to the 1st boundary constraint end edge of the horizontal plate is free, so angular deformation is maximize at middle of the free edge of horizontal plate and very low at end points of the edge shown in Figure 12. Deformation at the control points perpendicular to the heating line at horizontal plate is observed in Figure 13. Here is also high deformation at end edge of plate (free edge).

Transverse stresses and longitudinal stresses are calculated by using finite element method shown in Figures 14 and 15. From Figure 14, it is possible to identify regions where the residual stresses and deformation reached maximum value and can lead to damages in plate structure. The longitudinal stress component is more in the heat affected region. The maximum magnitude of Von-Mises stress is more than the yield strength of material.

Overall Conclusions

A thick rectangular T-joint plate structure with same dimension is considered for welding process with fixed high temperature at T-joint section for simplifying heat source model. Results of the numerical analysis of temperature and stress field distribution in T-joint section, carried out by the finite element methods through ANSYS software. Two type of boundary conditions (cases 1 and 2) to reduce the deformation are applied in T-joint fillet welded structure and final deformations pattern are obtained in Figures 9 and 10 as well as stress analyses are predicted. Results show that deformation is not very high and transverse and longitudinal stresses are not much more than yield strength in rectangular T joint plate structure.

Acknowledgement

This work has been made under the plurianual funding of the Portuguese Foundation of Science and Technology (Fundação para Ciência e Tecnologia) to the Centre for Marine Technology and Engineering (CENTEC).

References

1. Ansys Inc Theory reference Ansys Inc.


