Research Article
Prediction of Welding Deformation and Residual Stresses in Fillet Welds Using Indirect Couple Field FE Method

Abstract: Fillet welds are extensively used in shipbuilding, automobile and other industries. Heat concentrated at a small area during welding induces distortions and residual stresses, affecting the structural strength. In this study, indirect coupled-field method is used to predict welding residual stresses and deformation in a fillet joint due to welding on both sides. 3-D nonlinear thermal finite element analysis is performed in ANSYS software followed by a structural analysis. Symmetrical boundary conditions are applied on half of the model for simplification. Results of FE structure analysis predict residual stresses in the specimen. A comparison of simulation results with experimental values proves the authenticity of the technique. The present study can be extended for complex structures and welding techniques.

Keywords: Couple field, finite element method, fillet weld, residual stresses, symmetrical boundary condition

INTRODUCTION
Fillet joints are widely used in bridges and ship structures. Fillet welded joints usually suffer various welding deformation patterns such as longitudinal shrinkage, transverse shrinkage, angular distortion and bending. The concentrated thermal gradient followed by cooling during the welding process induces residual stresses and distortions. Excessive distortions of welded components have negative effects on fabrication accuracy, external appearance and various strengths of the structures. Various corrective measures like post weld heat treatment, flame straightening, vibratory stress relief, induction heat treatment and cold bending can be used to lower the distortion level. However, these methods are costly and time consuming. Welding induced residual stresses may cause early yielding and reduce buckling strength. Therefore, prediction and control of welding deformation and residual stresses is critical to improve the quality and reliability of the structure. Withers and Bhadeshia (2001) defined residual stresses and summarized their measurement techniques. Experimental methods for the prediction of residual stress include stress relaxation, x-ray diffraction, ultrasonic and cracking (Teng et al., 2001). All these methods are either destructive or expensive, which drive the requirement of simulation techniques.

A weld simulation model involves geometrical constraints, material nonlinearities, all physical phenomena and welding parameters such as welding speed, current, voltage, efficiency. Improved and complex simulation models also include number and sequence of passes and filler material. Researchers have been working in the field of computational welding mechanics in order to accurately predict welding residual stresses and deformations (Goldak 2005; Lindgren and Karlsson, 1988; Lindgren, 2001).

Welding process is treated as a transient nonlinear problem in finite element thermo-elastic-plastic analysis. Camilleri et al. (2003, 2005) computed welding temperature field by FE methods and validated the results by experiments. Lee et al. (2008), Ueda et al. (1988), Ueda and Yuan (1993) and Barroso et al. (2010) predicted the effect of different shapes and material properties on welding residual stresses and distortions. Mollicone et al. (2006) described modeling strategies to simulate the thermo-elastic-plastic stages of the welding process and compared FE model with experiments. Iranmanesh and Darvazi (2008) presented a FE based calculation process to study temperature field and residual stresses using 2 and 3-dimensional models in ANSYS 9.0.

Gao and Zhang (2011) addressed moving heat source, latent heat of phase change and characteristic parameters of materials in the simulation model. Moraitis and Labeas (2009) developed a 3D FE model to predict keyhole formation and thermo-mechanical response during laser beam welding of steel and
aluminum pressure vessel or pipe butt-joints. Xu et al. (2008) presented the FE method based on the inherent strain theory to simulate welding distortion in multi-pass girth butt welded pipes of different wall thickness. Sulaiman et al. (2011) investigated the capability of linear thermal elastic numerical analysis to predict the welding distortion due to GMAW by FEM software WELDPLANNER. Mrvar et al. (2011) simulated welding of pipe with finite element program SYSWELD.

In this study, temperature distribution due to fillet welding on both sides of the web is calculated at each load step followed by structure analysis using the temperature field data. It is assumed that the structural results do not affect the thermal analysis. Therefore, only unidirectional coupling is carried out. Experiments are performed to validate the simulation results. The computed deformations are compared with experimental results measured at several points and residual stresses are predicted.

SIMULATION METHOD

FE modeling: Model geometry used in this study is shown in Fig. 1a. Material of both flange and web is low carbon steel. For FE analysis the half of the model is considered and symmetric boundary conditions are applied. The temperature gradient is considerably lower in the regions away from the weld location. Therefore, bigger element size is used to reduce the number of degrees of freedom and the computation time (Fig. 1b).

Thermal analysis: Non-linear thermal analysis is conducted using solid 70, eight node brick elements. Welding arc is considered as a moving surface heat source. Temperature history of the plate is evaluated using three-dimensional transient thermal analysis.

Table 1: Welding parameters

<table>
<thead>
<tr>
<th>Welding speed (mm/sec)</th>
<th>Efficiency (%)</th>
<th>Current (A)</th>
<th>Voltage (V)</th>
<th>Torch angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>80</td>
<td>300</td>
<td>30</td>
<td>45°</td>
</tr>
</tbody>
</table>

Heat source model: In this study, at any time t, the heat of the welding arc is modeled by a surface heat source with a Gaussian distribution (Gao and Zhang, 2011). Thus, points lying on the surface of the work piece within the arc beam radius r receive distributed heat fluxes q as follows:

$$q_r = \frac{3Q}{\pi r_e^2} \exp \left( -\frac{r_e}{r_r} \right)$$

where, r_e is the radial distance measured from the instantaneous arc center on the surface of the work piece and Q is the heat input from the welding arc. Where Q = ηVI is the energy of the welding arc, η is the arc efficiency, V is voltage and I is the welding current respectively. The value of welding parameters are given in Table 1.

Heat transfer model: Equation (2) is the governing Eq. of 3D transient heat transfer in such methods while Eq. (3) represents the heat loss due to convection and radiation.

$$k \frac{\delta^2 T}{\delta x^2} + k \frac{\delta^2 T}{\delta y^2} + k \frac{\delta^2 T}{\delta z^2} + \frac{\delta Q}{\delta t} = \rho C \frac{\delta T}{\delta t}$$

$$h(T - T_0) + \beta \in (T^4 - T_0^4) = q_s$$

where Q is the internal heat energy released or consumed per unit volume (J/mm³), q_s is the heat loss, T is temperature, T_0 is ambient temperature, t is time, k
is thermal conductivity (W/mm °C), ρ is density, C is specific heat (J/g °C), h is a convection coefficient, β is the Stefan-Boltzman constant and ε is emissivity. Considering a quasi-steady state situation, Eq. (2) can be rewritten in the form of Eq. (4), where u (mm/s) is the velocity in the x-direction.

$$k \frac{\delta^2 T}{\delta x^2} + k \frac{\delta^2 T}{\delta y^2} + k \frac{\delta^2 T}{\delta z^2} - u \frac{\delta Q}{\delta x} = -u \rho C \frac{\delta T}{\delta x}$$  (4)

**Material model:** Figure 2 shows temperature dependent thermal properties of the material (Khurram et al., 2011). For interpretation of heat transfer by convection in the weld pool, an exaggerated value of the thermal conductivity is considered for temperatures above the melting point. The latent heat of fusion is combined in the material model by increasing the thermal conductivity for temperatures above the melting point. The latent heat of fusion is given in Table 2 (Iranmanesh and Darvazi, 2008). For interpretation of heat transfer by convection, the velocity in the x-direction.

The elastic strain is modeled using the isotropic Hook’s Law with temperature-dependent Young’s module and Poisson’s ratio. For the plastic strain of the model with yield level of von misses, temperature dependent mechanical properties and hardening linear kinematic model is obtained. Heat strain is calculated using coefficient of thermal expansion given in Table 2.

$$\varepsilon = \varepsilon^e + \varepsilon^p + \varepsilon^{th}$$  (5)

**RESULTS**

**Welding deformations:** Transverse deformation are normal to the weld bead as shown in Fig. 3 and are a result of thermal strains produced during welding. Expansion and contraction during welding in the direction parallel to the welding line causes longitudinal shrinkage in Fig. 4 and 5 represents out of plane deformation which is the basis for angular distortion.
Fig. 3: Deformation in transverse direction

Fig. 4: Deformation in longitudinal direction

Fig. 5: Out of plane Deformation
The measured and simulated transverse shrinkage at various points at the mid section of the plate is shown in Fig. 6. Longitudinal deformation at the two extreme ends of the plate are depicted in Fig. 7. The out of plane deformation is more prominent than the other two deformations.

Fig. 6: Comparison of experimental and simulation transverse deformation

Fig. 7: Comparison of experimental and simulation longitudinal deformation

Fig. 8: Comparison of experimental and simulation out of plane deformation

Fig. 9: Stress history of a point at mid thickness

Fig. 10: Transverse stresses at the middle of the plate

Fig. 11: Longitudinal stress at the middle of the plate

Measured and simulated deformation at various points at mid section of the plate are plotted in Fig. 8. It is evident that the out of plane deformation is more prominent in comparison with the other two deformations.
Welding residual stresses: Residual stress distribution is not uniform across the thickness of the plate with maximum at the top surface and decreases gradually to minimum at the bottom (Khurram et al., 2011). Therefore all stresses are computed at the mid thickness of the plate. Every point within the plate experiences variable stresses during and after welding. Figure 9 shows the stress history of a point at mid thickness of the flange. Results demonstrate that stress is maximum in transverse direction.

Simulation results of transverse and longitudinal residual stresses ($\sigma_x$, $\sigma_z$) at the flange mid-center are shown in Fig. 10 and 11, respectively. The residual stresses perpendicular to the flange or out of plane are due to non-uniform expansion and contraction with in the thickness as shown in Fig. 12.

CONCLUSION

This research provides basic theory and instruction to simulate welding residual stresses and deformations in fillet weld joint. Due to symmetry, only half model is considered for analysis. A non-linear transient thermal analysis is performed using a Gaussian distribution based moving heat source. Temperature distribution is computed at each time step independently. Using the results of thermal analysis and applying symmetric boundary conditions, a transient coupled 3D finite element structural analysis is performed. Experiments are also conducted to validate simulation results. Conclusions of this study are summarized as following:

- Transverse stresses values dominate other stresses through out the welding cycle.
- Transverse and longitudinal stresses are compressive in nature near the weld line. Their value gradually decreases as the distance from the weld line increases and eventually, become tensile near the edges of the plate.
- Out of plane stresses near the weld are are tensile, which gradually decrease and become compressive. However the values are much lower in contrast to other stresses.
- The current method can be used to simulate complex geometries and various welding technologies.

REFERENCES


