Proceedings of the
International Conference on Mechanical Engineering 2005
(ICME2005) 28- 30 December 2005, Dhaka, Bangladesh

ICME05-AM-27

ASSESSMENT OF RESIDUAL STRESS AND DISTORTION IN WELDING BY FINITE ELEMENT METHOD

M Sundar[1], G Nandi[2], A Bandyopadhyay[2] and S C Roy[2]

[1] School of Laser Science & Engineering, Jadavpur University, Kolkata-700032
[2] Department of Mechanical Engineering, Jadavpur University, Kolkata-700032

ABSTRACT

Welding is a reliable and effective metal fabrication process which is widely used in industries. Localized heating during welding, followed by rapid cooling generates residual stress and distortion in the weld and base metal. In the last few decades various research efforts have been directed towards the control of welding process parameters aiming at reducing the residual stress and distortion. In the present work a study on a single pass butt-welding specimen by using Finite Element Method has been conducted to illustrate the temperature distribution, distortion and residual stress field developed in the weldment. Thermo elasto-plastic analysis has been used to find the residual stress and distortion. Element birth and death technique is used to formulate weld deposition. The relationships between the parameters and response have been drawn based on the simulation result, which are found to be in good agreement.

Keywords: Weldment, FEA, Distortion, Residual Stress, Temperature distribution.

1. INTRODUCTION

During welding, the weldment is locally heated by the welding heat source. Due to the non-uniform temperature distribution during the thermal cycle, incompatible strains lead to thermal stresses. These incompatible strains due to dimensional changes associated with solidification of the weld metal, metallurgical transformations and plastic deformation, are the sources of residual stresses and distortion.

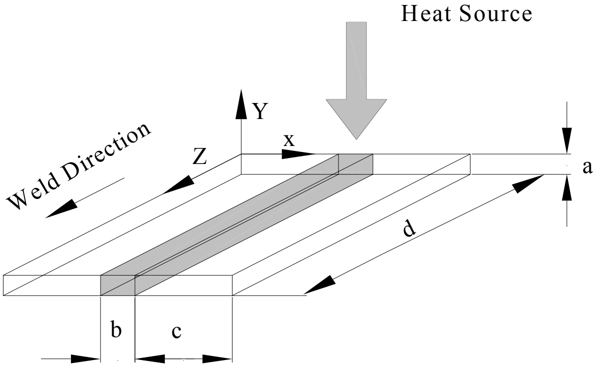
Generally, the finite element method has already been proven to be a successful tool to simulate the complex welding process as performed by Friedman [1]. In this paper a three dimensional welding problem has been solved using Finite Element Software ANSYS and the resultant residual stress and distortion is computed.

Residual stress distribution and distortion in a welded plate are strongly affected by many parameters, like structural, material and welding parameters. The structural parameters include geometry of the plates, thickness and width and joint type. Among the material parameters, mechanical and physical properties and type of filler-metal are considered. Welding process parameters include type of process employed, current, voltage, arc travel speed, and arc efficiency. To understand the formation of residual stress, the node temperature history during the welding process must be calculated. In particular, during the weld thermal cycle, mechanical properties of the material change drastically, especially when the material approaches melting temperature.

As phase change starts deviatoric stress [2] becomes zero and considerable plastic deformation occurs in the weld metal and the base metal. As temperature decreases during the cooling phase, the stress in the solidifying material increases, and becomes tensile due to the positive temperature gradient. The region placed away from the weld line, will therefore, be in compression. In some cases, residual stress may equal or exceed the yield stress of the parent plate material. The plastic strains resulting from the heating causes stress, which inturn produce internal forces that may cause buckling, bending and rotation. The residual stress combined with distortion and degradation of the material mechanical properties influence the buckling strength and fatigue life of welded structure.

2. WELDING SIMULATION

The finite element simulation of welding process consists of two analyses [3, 4]; transient thermal analysis and elasto-plastic analysis. To simplify the simulation procedure, uncoupled numerical simulations are used. In such uncoupled analyses, the results of the transient thermal analysis which include the temperature distribution, will be used for the second analysis together with the temperature dependent mechanical properties of the material i.e., thermal expansion coefficient, modulus of elasticity, Poisson's ratio, etc. Density in the elasto -plastic analysis is assumed to be constant.



a=12 mm, b=4 mm, c=40mm and d=150mm

Fig 1. Conceptual diagram of Arc Welding

In the present work Finite Element Analysis of single-pass butt-welding has been carried out. For this, a simple butt-joint welding whose welding parameters are consistent to those of Friedman's model with heat input Q=1200 W is considered and has been simulated using ANSYS.

3. MODEL DESCRIPTION

To accurately model temperature distribution through thickness and across the plate with time it is necessary to build a 3D model using Brick elements (fig 2). Once calculated, the temperature history for the plate is used for calculating the residual stress and distortions. The out of plane distortion is of a particular interest because if large enough it may affect buckling behavior of the plate. Applying heat flux [5] on element faces simulates the moving [6] heat source. The accuracy of the finite element method depends upon the density of the mesh used in the analysis. Sensitivity analysis of mesh density is performed and a satisfactory mesh is adopted for further studies, the higher is the heat input the higher is the number of nodes necessary to accurately interpolate high temperature gradient.

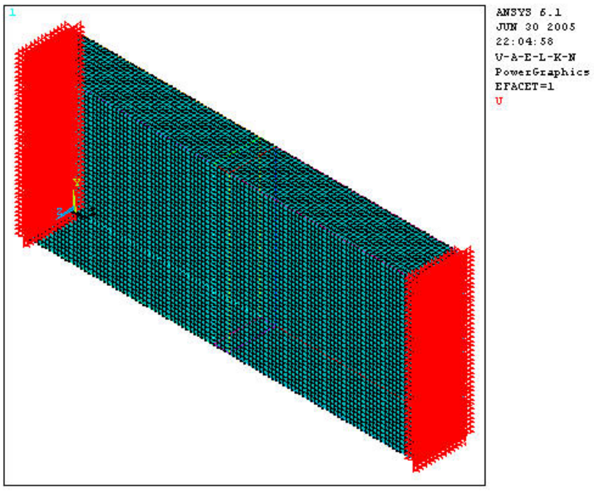


Fig 2. Mesh Model used for analysis

Several different materials may be used in structures where welding is involved, with low carbon steel being the most common. The weldment material employed in this work is mild steel and the properties of the material are at par with [7]. A temperature dependent material property has been used for the heat transfer and transient elasto-plastic analyses. For simplicity, the deposited weld material is also assumed to be low carbon steel. Two sets of temperature dependent material properties are needed in the analyses.
The following assumptions are made for analysis sake

- Thermal properties, i.e. conductivity, specific heat are temperature dependent
- Effects arising from phase change are taken into account, enthalpy changes during the phase change
- A combined convection and radiation boundary condition is used on the remaining of the top surface.
- The analysis is based on quasi-steady state, i.e., the heat source is moving at a constant velocity.
- Effective depth of penetration is shallow, thus, Gaussian heat flux distribution is used on Heat Affected Zone (HAZ).
- Deformation process is rate independent, and a elastic-plastic constitutive model with isotropic hardening is assumed for the material
- Mechanical properties depend on temperature, which means plasticization area is temperature-dependent.

4. FINITE ELEMENT FORMULATION

The governing equation for heat conduction in a solid is

$$\frac{\partial}{\partial x}\left(K\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(K\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(K\frac{\partial T}{\partial z}\right)+Q=\rho C_s\frac{\partial T}{\partial t} \qquad (1)$$

Since welding processes undergo a high temperature cycle and exhibit material properties that are temperature dependent, the transient temperature can be calculated by an extrapolation method with a two- time interval as

$$T(\tau)=T(t-\Delta t)+\frac{\tau}{\Delta t}[T(t-\Delta t)-T(t-2\Delta t)] \qquad (2)$$

Residual stresses are calculated by using the principle of virtual work. In this method, one considers infinitesimal nodes displacements $\{\delta\}$ imposed onto the body. This causes an external total virtual work and it is equal to the total internal virtual work and it is defined by stresses $\{\sigma\}$ and strains $\{\varepsilon\}$. By using FEM, strain–displacement can be expressed briefly as follows:

$$\{\varepsilon\}=[B]\{\delta\}^e \qquad (3)$$

where [B] is the strain–displacement interpolation matrix and $\{\delta\}^e$ is the displacement vector for an element. The stress–strain relation is defined as follows:

$$\{\sigma\} = [C]\{\varepsilon\} \tag{4}$$

where [C] is the elasticity matrix.
The node displacement is obtained from

$$\{\delta\} = [K]^{-1}(\{R\}_T \{R\}_{P1}) \tag{5}$$

From this equation the pure elastic stress can be obtained as follows:

$$\{\sigma\} = [C][B]\{\delta\} - [C]\alpha_m \{\Delta T\} - \{\sigma\}_{P1} \tag{6}$$

4.1 Boundary Conditions

For top Surface, heat flux input with heat loss due to convection and radiation is

$$K\frac{\partial T}{\partial n} = Q - h_c\,(T\text{-}T_\infty) \tag{7}$$

For all other surfaces (except the top surface), heat loss due to convection and radiation is

$$K\frac{\partial T}{\partial n} = h_c\,(T\text{-}T_\infty) \tag{8}$$

where $h_c = 24.1 \text{ X } 10^{-4}\ \alpha\ T^{1.61}$ (9)

Movement of the heat flux :

$$\xi = x\text{-}Ut \tag{10}$$

At $\xi = 0$; $x = Ut$

In this analysis temperature-dependent thermal properties are assumed, therefore non-linear equations are solved, with all complexity related to their solutions. To determine temperatures and other thermal quantities that vary over time there is the need to perform a transient thermal analysis. Implicit method of time discretisation is employed which allows for larger time steps. Using the finite element analysis the thermal and stress analysis are uncoupled while in reality thermal effect and mechanical deformation occur at the same time. The de-coupling of the analyses becomes acceptable if one assumes that dimensional changes (mechanical deformation) during welding process are negligible because thermal energy change is predominant over mechanical work done during welding, and the internal energy dissipation effect on the temperature distribution is negligible.

In ANSYS the heat transfer analysis is conducted using element type SOLID70. This element type has a three-dimensional thermal conduction capability and eight nodes with a single degree of freedom (temperature) at each node. The element is applicable to a three-dimensional, steady-state or transient thermal analysis. The element can also compensate for mass transport heat flow from a constant velocity field.

The heat input from the welding electrode is modeled by using heat flux as the input for the heat transfer from the rod to the work piece. This heat flux is based on the welder setting and the efficiency of the arc. The heat loss is modeled using convection and radiation heat transfer. The convective coefficient is assumed to vary with temperature. The result of the heat transfer analysis is the time dependent temperature distribution at each node of the element.

To evaluate distortion and residual stress distribution the thermal analysis is performed first to find nodal temperatures as a function of time. Once defined temperature history for each node, temperature nodal temperature loads are applied to the structural model. The plate absorbs a part of the heat generated. There are losses from the surfaces in the form of convection and radiation.

The material deposition is modeled using an element “birth and death” technique [8]. To achieve the “death element” effect, the 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 then reactivates the “death element” by allowing its stiffness, mass, element loads, etc. return to their original values.

4.2 Thermal Analysis

The thermal analysis has been carried out for two cases, case-1 welding speed=10mm/sec and case-II welding speed=5mm/sec with constant time step of 0.5 sec. All other parameters are kept constant. After the welding phase is finished, the time step is progressively increased up to 1000 sec to allow the plate to cool down to the room temperature. Since during the welding process phase change occurs, to account the effect of latent heat, i.e. heat energy which, is released or stored by the material during a phase change enthalpy is specified.

4.3 Distortion and Residual Stress Analysis

The distortion and residual stress analysis has been carried out for case-I. Then in the second part of the analyses, a non-linear structural analysis is carried using the temperature distributions, which is obtained from the heat transfer analysis. In this analysis, element SOLID70 is replaced by a three-dimensional (3-D) structural element SOLID45. The element is defined by eight nodes having three degrees of freedom at each node (translations in the nodal x, y, and z directions). The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities ANSYS Users Manual, 2001).

An important problem in the analysis of residual stress during welding is how stress develops in regions near the welding pool. When structural members are joined by fusion welding the material of the plates has to be heated to its melting point and then cooled again rapidly under restraint conditions imposed by the geometry of the joint. As a result of this severe thermal cycle the original microstructure and properties of the metal in a region close to the weld are changed. This part of the metal, or zone, is usually referred to as heat-affected zone (HAZ). The changes in the HAZ are also dependent upon the thermal and mechanical history of the metal. Therefore, after the welding process there will be different zones with different mechanical properties. In particular, there is a softening of the material in the HAZ, and there is a decrease of the mechanical properties of the material, i.e. yield strength, ultimate strength of the material, but the elastic modulus remains unaffected by the welding process.

The analysis is performed for the time period between the start of welding and the end of cooling phase. Within each time increment, the solution of elastic-plastic problem is found by linearizing the non-linear stress-strain relation in an incremental way. The analysis is performed and stresses and displacements are calculated by Newton-Raphson iterative process. The iterations are repeated until convergence is achieved. Boundary conditions are imposed to prevent any rigid body motion of the plate.

5. RESULTS AND DISCUSSION

5.1 Thermal Analysis

The temperature distributions are shown through Figure 3 to 8. Figure 3 & 6 show the nodal temperature along the weld direction while Figure 4 & 7 show the nodal temperature perpendicular to the welding direction. Figure 5 & 8 show the surface temperature of the weld at 5 sec. The isothermal enclosures are moving with the arc along the welding direction. It has been demonstrated that, due to motion of the workpiece relative to the arc, causing advection heat flow to occur in the moving direction, the peak temperatures at the workpiece surface are near the trailing edge of the arc rather than at the arc centre.

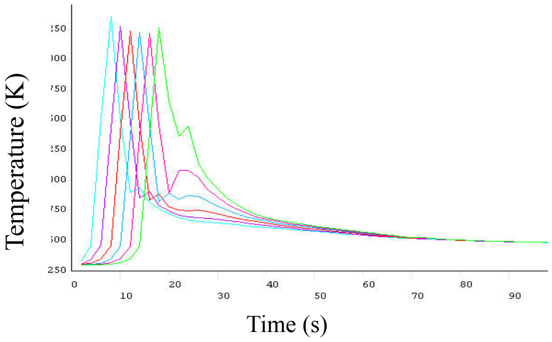


Fig 3. Temperature distribution along welding direction for welding speed =5 mm/s

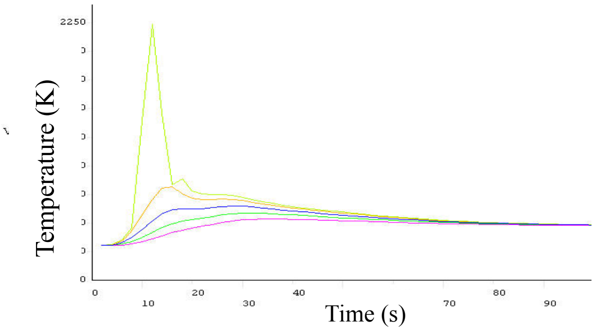


Fig 4. Temperature distribution perpendicular to welding direction along the center for welding speed =5 mm/s

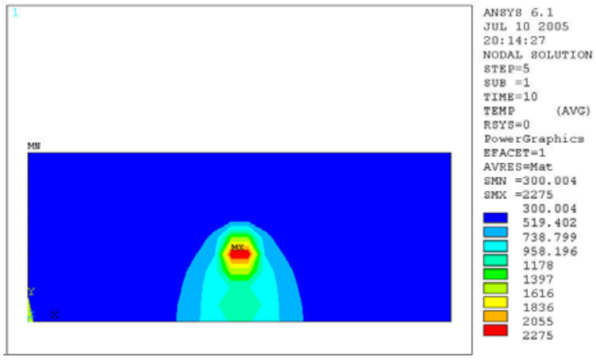


Fig 5. Temperature distribution at time =5 sec for welding speed =5 mm/s

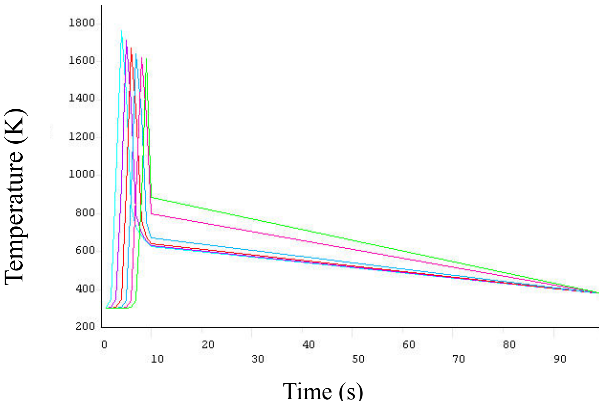


Fig 6. Temperature Distribution along Welding Direction for Welding speed =10 mm/s

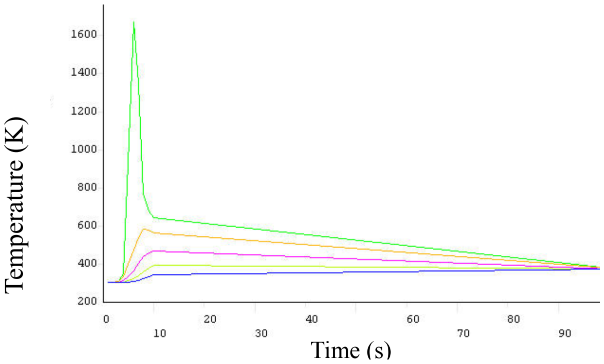


Fig 7. Temperature distribution perpendicular to welding direction along the center for welding speed =10 mm/s

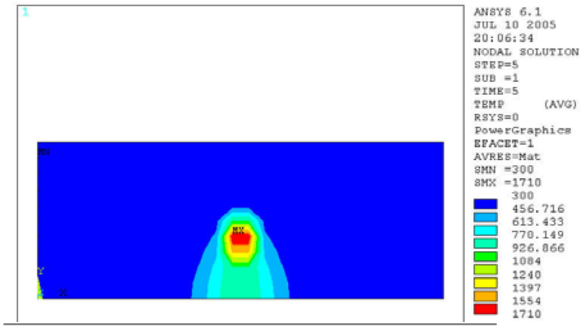


Fig 8. Temperature distribution at time =5 sec and welding speed =10 mm/s

From the figure it is seen that a slower moving velocity (Case- I) of the workpiece would lead to larger temperature gradients in the direction of movement of the workpiece, but lower temperature gradients in the directions vertical to the movement of the workpiece within the solid domain where the temperature is lower than the melting point, in addition to a high peak temperature on the molten surface. The maximum temperature for case-I and case-II are 2200 and 1750 K respectively.

5.2 Distortion and Residual Stress

It is found that magnitude and distribution of residual stress is strongly affected by temperature gradient, temperature distribution through the thickness and width of the plates, thermal expansion coefficients of the materials and mechanical properties of material at elevated temperatures.

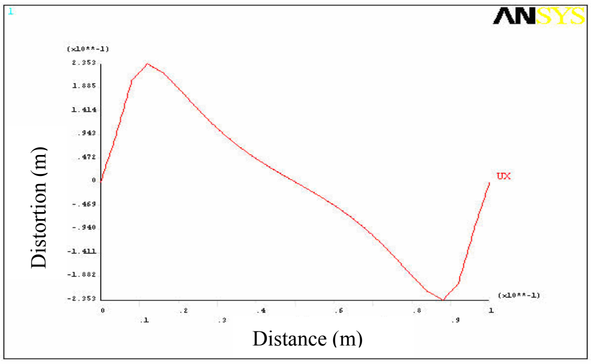


Fig 9. Distortion in X-direction for welding speed 10 mm/s

The out-of-plane displacement is shown in Fig. 9 and 10. The maximum displacement amplitude is 0.278 mm, this means that the out-of-plane displacement is 2 % of plate thickness. This displacement is very important for the buckling behavior of welded panel.

This result confirms our assumption that it is necessary to use three dimensional eight node brick elements. For a similar plate with same geometry, same boundary condition with higher thickness, residual stress would be higher but distortions would become smaller due to an increased stiffness of the plate.

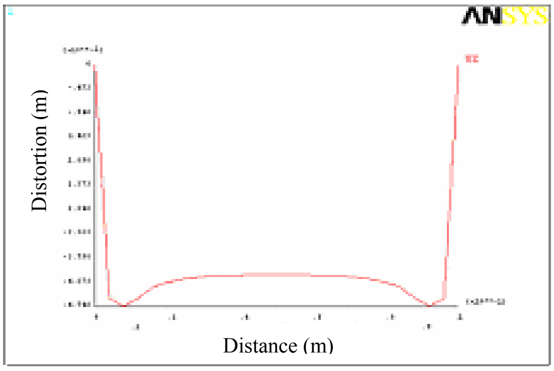


Fig 10. Distortion in Z-direction for welding speed 10 mm/s

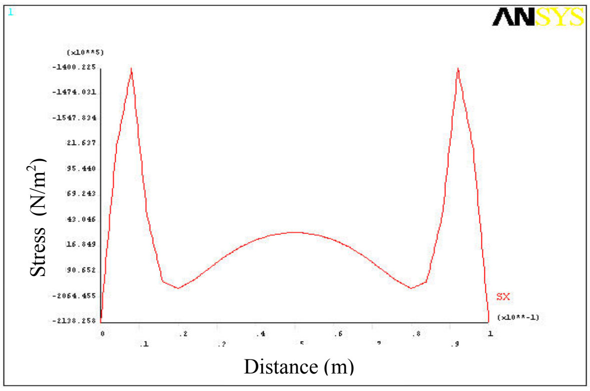


Fig 11. Stress in X-direction for welding speed 10 mm/s

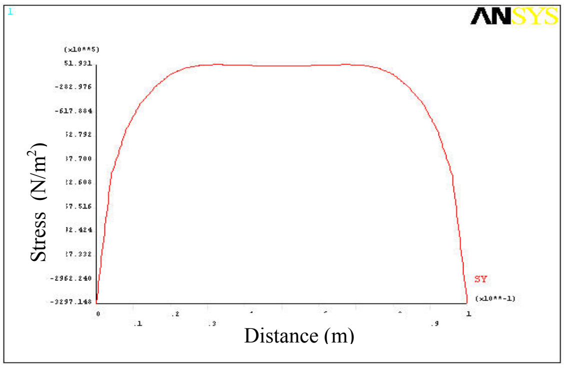


Fig 12. Stress in Y-direction for welding speed 10 mm/s

According to the geometry and theory, larger residual stress and distortions are expected parallel to the weld direction and closer to the weld zone. This behavior is well evidenced from the Fig. 9-13.

A stress acting normal to the direction of weld bead is known as a transverse residual stress. Fig. 11 shows the transverse residual stress. A very large tensile stress is produced near the surface of the plate. Owing to the locally concentrated heat source, the temperature near the weld bead and heat affected zone rapidly changes with distance from the heat source.

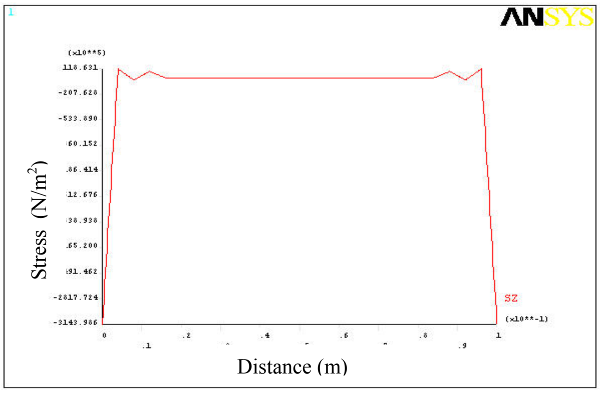


Fig 13. Stress in Z-direction for welding speed 10 mm/s

The longitudinal residual stress distribution is shown in Fig. 12. It is worth mentioning that the residual stress distribution is strongly affected by the boundary conditions assumed. The stress is tensile in nature near the weld line, while away from the weld line the stress is in compressive. The shape of longitudinal stress distribution does not change but the stress amplitude decreases towards the bottom surface. This is due to the assumption that heat flux is coming through the top weld surface. Fig. 13 reveals that the stress distribution is almost uniform along the thickness of the work piece.

6. CONCLUSIONS

- A complex welding process phenomenon has been simulated using a commercial finite element package, ANSYS.
- A special feature of "birth and death" element has been used to simulate the deposition of weld material.
- Based on the simulation results, distortion or shrinkage of the weldment can be predicted. Thus, the experimental analysis, which might be costly, can be avoided.
- The temperature near the weld bead and the HAZ decreases rapidly with the distance from the centre of the heat source.
- The transverse residual is high near the weld and reduces as it moves further. This is because yield force immediately after welding is very low and a part of the specimen near the bead deforms plastically under very low external load. The region of the large tensile residual stress can correlate approximately with the one of the large equivalent plastic strain.

7. REFERENCES

1. Friedman, E., 1975, "Thermomechanical analysis of the welding process using the Finite Element Method", Transaction of the ASME, pp 206-213.
2. Tsai, C., Kim, D., Jaeger, J., Shim, Y., Feng, Z., and Papritan, J., Feb 2001, "Design Analysis for Welding of Heavy W Shapes", Welding Journal, pp 35-41.
3. Rao, N.R.N., Tall.L., April 1961"Residual stressee in welded plates", Welding Journal, 40, pp 468s.
4. Hong, J.K, Tasi, C.L., Dong, P, 1998, "Assessment of numerical procedure for residual stress analysis of multipass welding", Welding Journal, 77, pp 372s.
5. Pathak, A.K., Datta, G.L., July 1999, "Three Dimensional Finite Element Analysis on Heat Flow in Arc Welding", Indian Welding Journal, Vol-3, pp 32-38.
6. Rosenthal, D., May 1941, "Mathematical theory of heat distribution during welding and cutting", Welding Journal, Vol-20, pp 220s-234s.
7. Brown, S., and Song, H., 1992, "Finite Element Simulation of welding of large structure", Journal of Engineering for Industry, vol. 144, pp 441-451.
8. *Basic Analysis Procedure Guide*, Release 7.

8. NOMENCLATURE

Symbol	Meaning	Unit
h_c	Combined convection and radiation heat transfer coefficient	(W/m^2)
T	Temperature	(K)
T_∞	Ambient temperature	(K)
V	Voltage	(V)
I	Current	(A)
C_s	Specific heat	(J/kg-K)
k	Thermal conductivity	(W/m-K)
Q	Heat Flux	(W/m^2)
t	Time	(s)
ρ	Density	(kg/m^3)
X,Y,Z	Coordinates	(m)
U	Scanning Speed	(m/s)