Improvement in Numerical Accuracy and Stability of 3-D FEM Analysis in Welding

Welding deformations and residual stresses are simulated using a 3-D finite element method model

BY J. WANG, Y. UEDA, H. MURAKAWA, M. G. YUAN, AND H. Q. YANG

ABSTRACT. The finite element method (FEM) of modeling and the rapid development of computer architecture provide the possibility of analyzing complex problems such as those encountered in welding processes where structures undergo a large temperature cycle. Due to the temperature dependence of material properties and the large deformation in welding, both material and geometrical nonlinearity must be taken into account. It is also important to ensure numerical accuracy and stability, especially at high temperature stages. Previous investigations in thermal elastic-plastic analysis showed that the stages with drastic change (such as the transition stage from elastic to plastic and unloading stage) suffered from numerical error. It has also been shown that the mesh size and temperature incremental used in the nonlinear computation has a significant influence on the accuracy of the solution.

To improve the accuracy of the numerical solution, weighting factors are introduced to take into account the transition from elastic stage to plastic stage as well as the temperature dependency of material properties. In order to prevent the numerical locking phenomena, a reduced integration method is presented in the 3-D solid elements with a local coordinate system. The effectiveness of the proposed methods is demonstrated through a 3-D analysis of a compressor assembled with plug welds.

J. Wang is with the Shanghai Jiao Tong University, Shanghai, China. Y. Ueda and H. Murakawa are with the Welding Research Institute, Osaka University, Osaka, Japan. M. G. Yuan is with the Daikin Industries Ltd., Japan. H. Q. Yang is with CFD Research Corp., Huntsville, Ala.

Introduction

It has been more than 20 years since the finite element method (FEM) was first applied to the thermal elastic-plastic analysis of welding in the 1970s (Ref. 1). Past studies were almost all limited to 2-D analysis, and those dealing with 3-D problems (Refs. 2-4) were relatively rare and were reported only recently. There are two major problems associated with the 3-D analysis of welding. One is the requirement of a large memory capacity of computers and long CPU time. The other is the difficulty in controlling the numerical accuracy and stability at very high temperature stages.

Compared with traditional FEM analysis, the thermal and mechanical behaviors in welding have the following characteristics:

1) Local high temperature (over the melting point of materials) and rapid changes of material properties with temperature.

2) High spatial and temporal gradients of temperature, stress, and strain (heating and cooling, or loading and unloading in the weldment were often observed at the same time).

3) Phase transformation and creep phenomena.

4) Complex geometries such as grooves, filled metal, and multipass welds.

5) Large deformation for thin structures.

KEY WORDS

3-D Analysis
Welding Deformation
Residual Stresses
Numerical Accuracy
Stability
FEM
Locking Phenomena
Reduced Integration
Temperature
Stress Fields

It is due to the above characteristics that the modeling of the welding phenomena must include both material and geometrical nonlinearity, and must be able to resolve the high gradients in time and space. To this end, one has to ensure the accuracy of the solution in all stages of the computational processes, otherwise the numerical error may accumulate with time and deteriorate the final solution.

The objective of this paper is to investigate factors that influence the accuracy of 3-D thermal elastic-plastic analysis. Based on the present finding, several techniques are proposed to improve the numerical accuracy and stability of the solution. The success of the proposed methods is demonstrated through a 3-D analysis of a compressor assembled with plug welds.

Factors Influencing Accuracy and

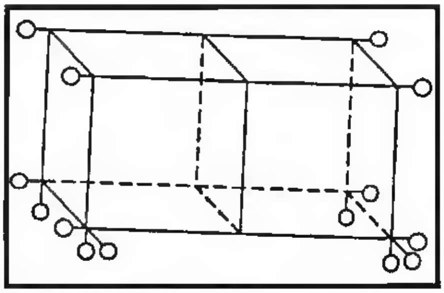

Fig. 1 — Model for analysis.

Convergence of Solution

Error and Convergence

In the nonlinear analysis, it is necessary to use the incremental forms of an equilibrium equation. According to the virtual work theorem, the equilibrium equation at the (n + 1) step can be written as follows:

$$[K]\{\Delta u\} = \{\Delta F\} + (F_n - f_n) \qquad (1)$$

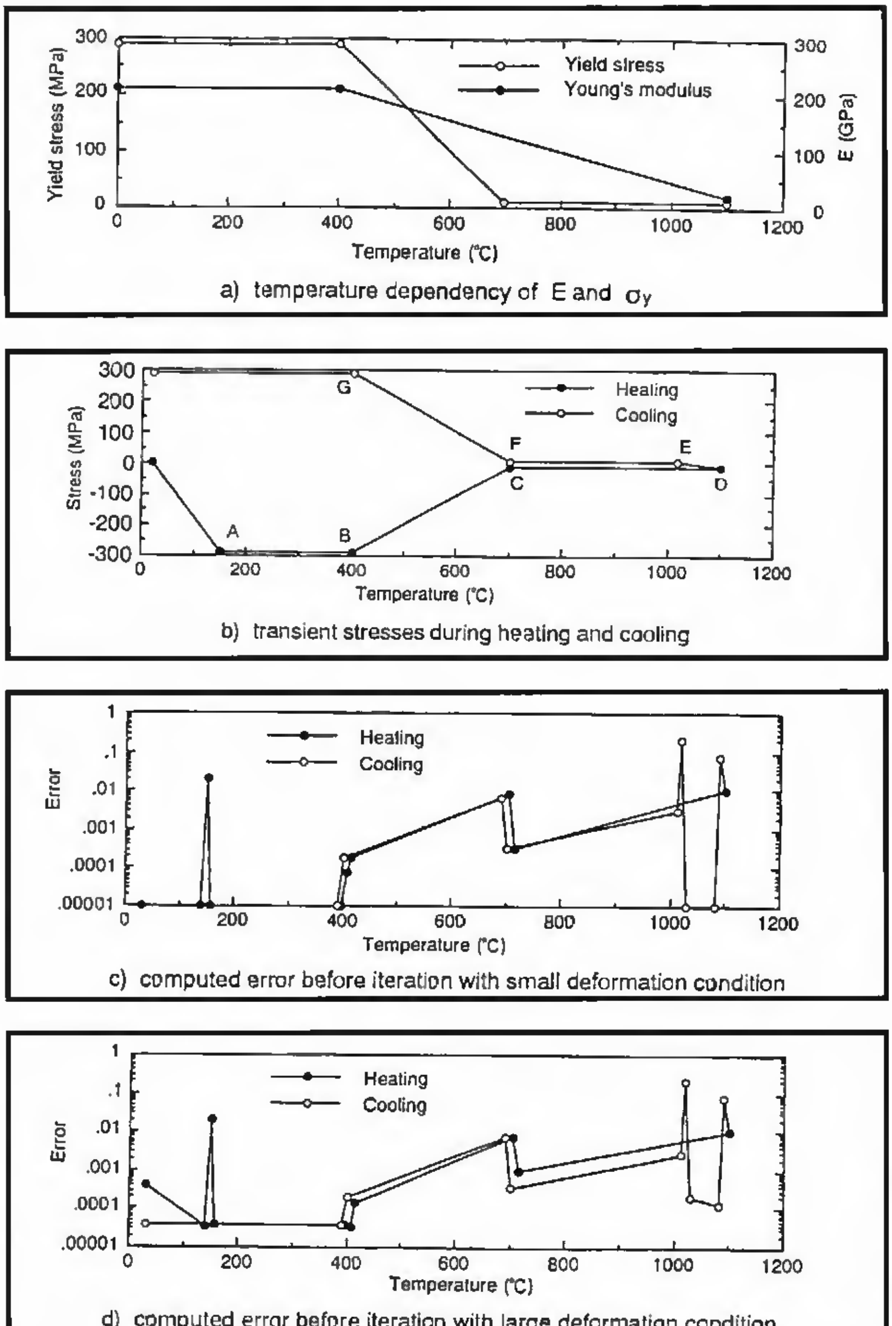

Fig. 2 — Computed results of transient stresses and error.

where [K] = the tangent stiffness matrix; {ΔF} = the external load increment; F_n = the external load at step n; f_n = the internal load at step n.

By solving the above equation, the increment of nodal displacement (Δu) can be obtained. In Equation 1, F_{er} (= $F_n - f_n$) is the nonequilibrium load induced by various nonlinear factors, and it should be reduced to less than a given tolerance after iterations. If F_{er} is too large, the convergence would not be achieved, and it is impossible to obtain an accurate result. In order to evaluate the accuracy of the computed result, the following parameter is introduced as a measure of error:

$$Err = \frac{\sqrt{\Sigma F_i^2}}{\sqrt{\Sigma F_j^2}} \qquad (2)$$

where F_i = the force at the free node i due to the nonequilibrium; F_j = the reaction force at the fixed node j.

The criterion of convergence is set up by using a small value ε. After a number of iterations within one step, if the following condition is satisfied,

$$E_{rr} < \varepsilon \qquad (3)$$

then the solution can be considered as converged, and the computation can be moved to the next step. Obviously, the less the value of E_{rr} the better the stability of the solution.

Numerical Results of a Simple Model

In order to determine the factors that may influence the accuracy and convergence, a model which is fixed at two ends, as shown in Fig. 1, is considered. The thermal cycle (20°-1100°-200°C-68°-2030°-392°F) is applied with uniform temperature across the model, and the material properties are taken as: thermal expansion coefficient $\alpha = 1.1 \times 10^{-5}$ °C^{-1}; Poisson's ratio $\mu = 0.3$; Young's modulus E_0 = 210 GPa, E_{1100} = 20 GPa; yield stress σ_{y0} = 290 MPa, σ_{700} = 10 MPa.

It should be noted that as with most previous studies, the present model was derived from decoupled thermal and mechanical energy balances, and the effects of mechanical deformation and the internal energy dissipation on the temperature fields is assumed to be negligible.

Figure 2A shows the temperature dependency of Young's modulus E and yield stress σ_y. The elastic-perfect-plastic constitutive relations were used in the analysis.

Computed results, with temperature step $\Delta T = 10°C$, for transient stresses and error in the whole cycle are shown in Fig. 2B, C and D, respectively. It can be seen that under the following conditions, the error can become relatively large: 1) transition from elastic to plastic stage (A, E); 2) unloading (D); 3) rapid changes of material properties with temperature (B, G); and 4) at very high-temperature stage (C–F).

If the above situations appear at the same time, such as unloading with reyielding at very high temperature, it would cause accuracy to decrease further, and even fail to reach convergence. Therefore, one should pay special attention to these situations and handle them with great care.

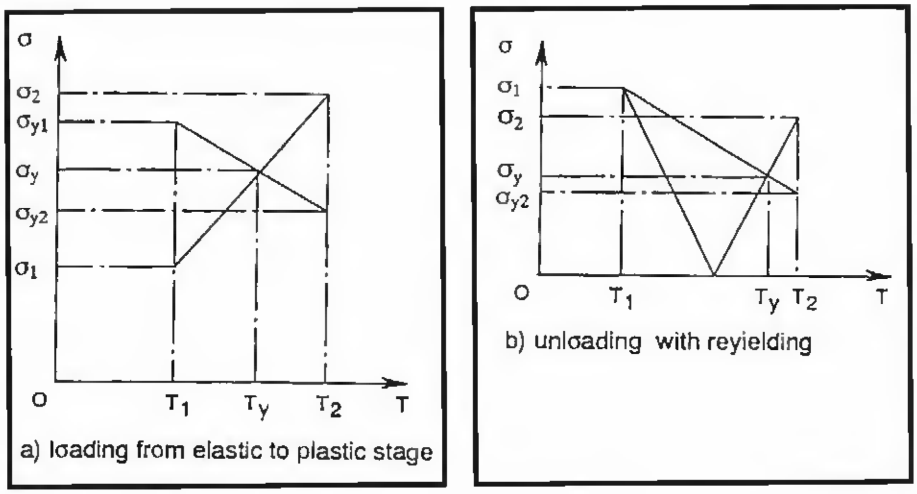

Fig. 3 — Relationship between stress and temperature in a step.

Other Influential Factors

Other factors that influence the accuracy in welding analysis include the oscillation in temperature distribution during thermal analysis by FEM, temperature increment, mesh division, large deformation and the locking phenomena in thermal deformation analysis of thin plate, etc.

Methods to Improve Accuracy and Convergence

3-D Thermal Analysis of Welding by FEM

Welding thermal analysis is the essential prerequisite for solving thermal elastic-plastic problems. It is important to ensure the high accuracy of the thermal analysis. If there is any serious oscillation in temperature distribution due to numerical error, it will influence the sequential analysis of mechanical behavior.

Weighting Factor Method for Elastic-Plastic Transition

When an element is in transition from elastic stage to plastic stage, as in the present step, it is necessary to introduce the concept of weighting factor ω ($1>\omega>0$), and divide this step into two parts. The first part is associated with elastic deformation and the second part is associated with the plastic deformation. The relative proportion of the two stages is $\omega:(1-\omega)$. The accurate determination of the weighting factor ω is important for improving the accuracy and convergence of the solution and for reducing the number of iterations. It is proposed that the stress is calculated by the following formula:

$$\Delta\sigma = D^e(\omega\Delta\varepsilon) + \Sigma[D^{ep}(1-\omega)\Delta\varepsilon/N] \quad (4)$$

where $\Delta\varepsilon$ = strain increment; $\Delta\sigma$ = stress increment; D^e = elastic matrix; D^{ep} = elastic-plastic matrix; and N = subdivision of plastic increment.

Figure 3A and B shows the relationship between stress and temperature during loading and unloading with reyielding in a computational step, respectively. In the present analysis, the effect of latent heat due to phase change has been taken into account. Considering the dependency of yield stress on temperature from Fig. 3, the weighting factor for the transition between elastic and plastic stages can be defined as follows:

for loading

$$\omega = (\sigma_{y1} - \sigma_1)/(\sigma_{y1} - \sigma_1 + \sigma_2 - \sigma_{y2}) \quad (5)$$

for unloading with reyielding

$$\omega = 2\sigma_1/(2\sigma_1 + \sigma_2 - \sigma_{y2}) \quad (6)$$

where T_1, σ_1, σ_{y1} = temperature, equivalent stress, yield stress at previous step; T_1, σ_2, σ_{y2} = temperature, equivalent stress, yield stress at present step; $T_2 \sim T_y$ = elastic stage; $T_y \sim T_2$ = plastic stage.

The element reaches yielding at temperature T_y and the yield stress at that moment is given by

$$\sigma_y = \sigma_{y1}(1-\omega) + \sigma_{y2}\omega \quad (7)$$

It should be noted that the formula $\omega = (\sigma_y - \sigma_1)/(\sigma_2 - \sigma_1)$ can be used in sim-

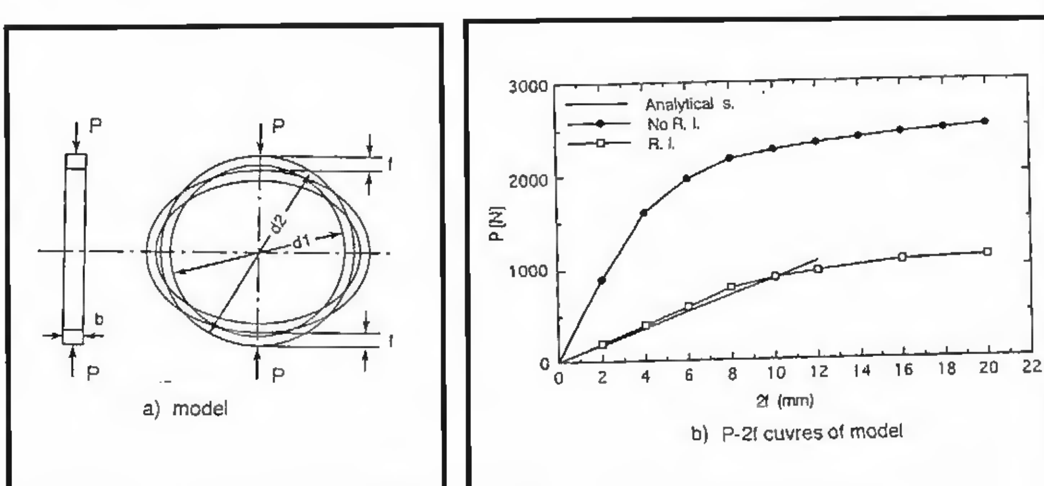

Fig. 4 — Simplified models for checking the locking phenomena.

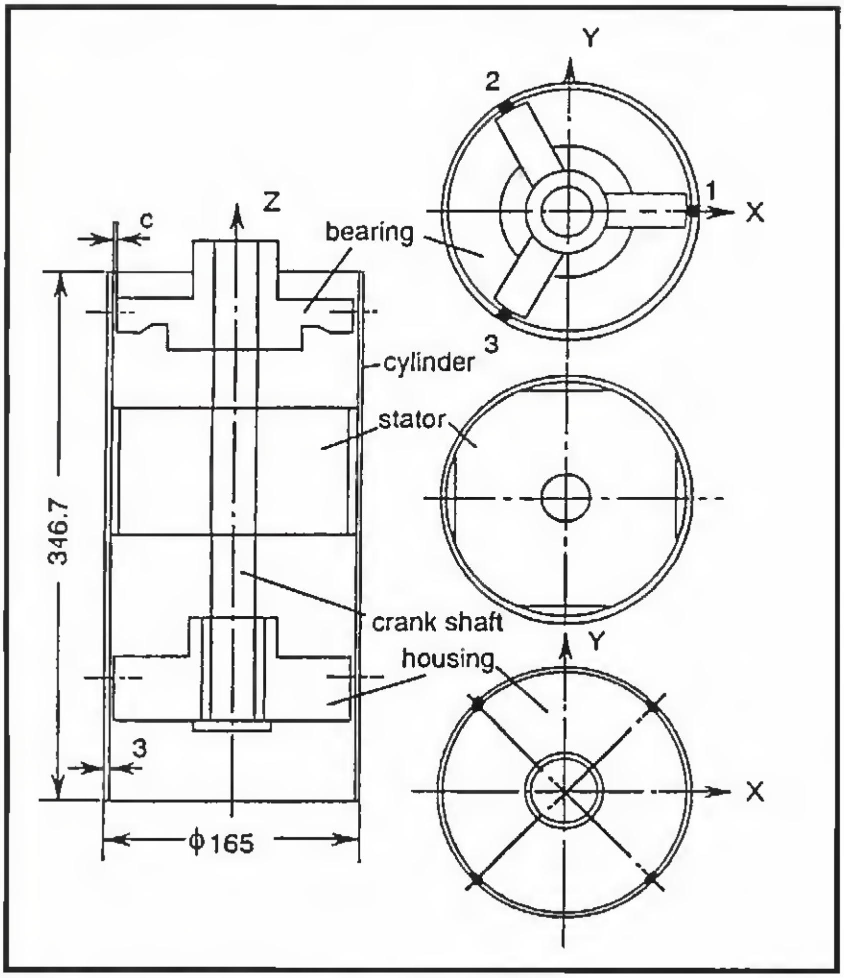

Fig. 5 — Diagram of the compressor.

ple elastic-plastic problems, since the yield stress σ_y is assumed to be constant. However, it is not accurate for thermal elastic-plastic problems if the current yield stress σ_{y2} is used instead of the true yield stress σ_y. Because of the temperature dependency of the yield stress, the following three cases may occur:

a) $\sigma_2 > \sigma_{y2} > \sigma_1 : 0 < \omega < 1$

b) $\sigma_2 > \sigma_1 > \sigma_{y2} : \omega < 0$

c) $\sigma_1 > \sigma_2 > \sigma_{y2} : \omega > 1$

If the factor ω becomes less than 0.0 or greater than 1.0, such as in cases b) and c), the iterative solution may become unstable. The unreasonable phenomena can be avoided by using Equations 5 and 6 as suggested above.

Assumption of Material Properties at a Very High Temperature Stage

Experimental data of E and σ_y taken at a very high temperature are seldom available, and they are usually taken as reasonably small values. These values, however, must satisfy certain conditions, or an unreasonably large stress increment may occur during unloading. In order to avoid the above phenomenon, variations of stress due to a change of Young's modulus should not be larger than that due to the change of thermal expansion. From this assumption, the following condition can be derived:

$$\Delta E^{\sigma_y}/_{E} < E\alpha\Delta T \tag{8}$$

It can be rewritten as

$$dE/dT < \alpha E^2/\sigma_y \tag{9}$$

Step Size and Mesh Division

The temperature step of ΔT can be large during the first stage when the deformation is elastic. It must be reduced when the nonlinear effects set in at high temperature. Our experience shows that ΔT should be controlled to be less than about 10°C. To determine the size of the mesh division, both accuracy of the computation and the capacity of the computer must be considered.

Locking Phenomena and Reduced Integration

The finite element method has been recognized as a versatile tool for structural analysis, including thermal elastic-plastic problems. In general, 3-D solid elements give the most accurate results in a mechanical simulation. However, it sometimes exhibits abnormal behavior that is often called locking phenomenon (Refs. 5, 6). Such phenomena can be observed as an overestimation of stiffness in plate bending analysis with shear deformation and in shell analysis. Since the

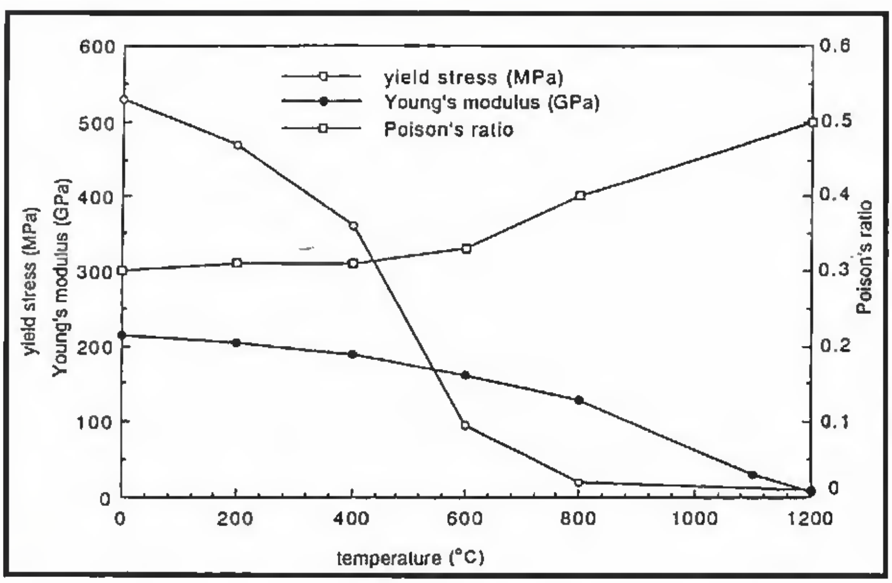

Fig. 6 — Temperature dependency of yield stress, Young's modulus and Poisson's ratio.

locking phenomena seriously reduce the accuracy of the analysis, a reduced integration method and a mixed method have been proposed to prevent it. In this study, eight-node brick elements are used with reduced integration applied to the terms involving transverse shear terms of γ_{xy} γ_{xz} (x is direction of the thickness) in the local coordinates. To analyze the models with axisymmetric geometry such as cylinders or pipes, it is necessary to employ the cylindrical coordinates instead of rectangular Cartesian coordinates through the local coordinate transformation. The use of local coordinates makes it easy to introduce the reduced integration for particular components of strain.

In order to investigate the effects of such locking phenomenon, a model of a ring (d_1 = 150 mm, d_2 = 155.3 mm, b = 10 mm) applied by a pair of force P is used, as shown in Fig. 4A. Figure 4B shows the P-2f curve of the model computed under the different conditions. The analytical solution is as follows:

$$2f = Pr^3(\pi/4 - 2/\pi)/EJ \qquad (10)$$

where 2f = total displacement in loading direction; r = average radius of the ring; E = Young's modulus; J = moment of inertia of the cross-section.

From Fig. 4 it can be seen that the locking phenomenon has a significant effect when the solid elements are used without reduced integration. The use of reduced integration can prevent the locking phenomenon and give closer values to the analytical solution.

3-D Deformation Analysis of a Compressor with Plug Welds

A thin-walled cylinder is joined with an inner bearing by plug welds. Before this process, the cylinder is already connected with a stator by shrinkage fit and joined with a housing with plug welds. Figure 5 shows the diagram of the cylinder with the inner components. Three holes of 8-mm (0.32-in.) diameter are located with equal distance in the circumferential direction of the cylinder and filled with melted metal through the thickness of the wall by simultaneous gas tungsten arc (GTA) welding.

After plug welding, the axis of the bearing is deflected to cause eccentricity, and the radial deformations are also induced, especially at the ends of the cylinder. If such deformations go beyond the allowable variations, it would effect the following assembly process or even result in waste product.

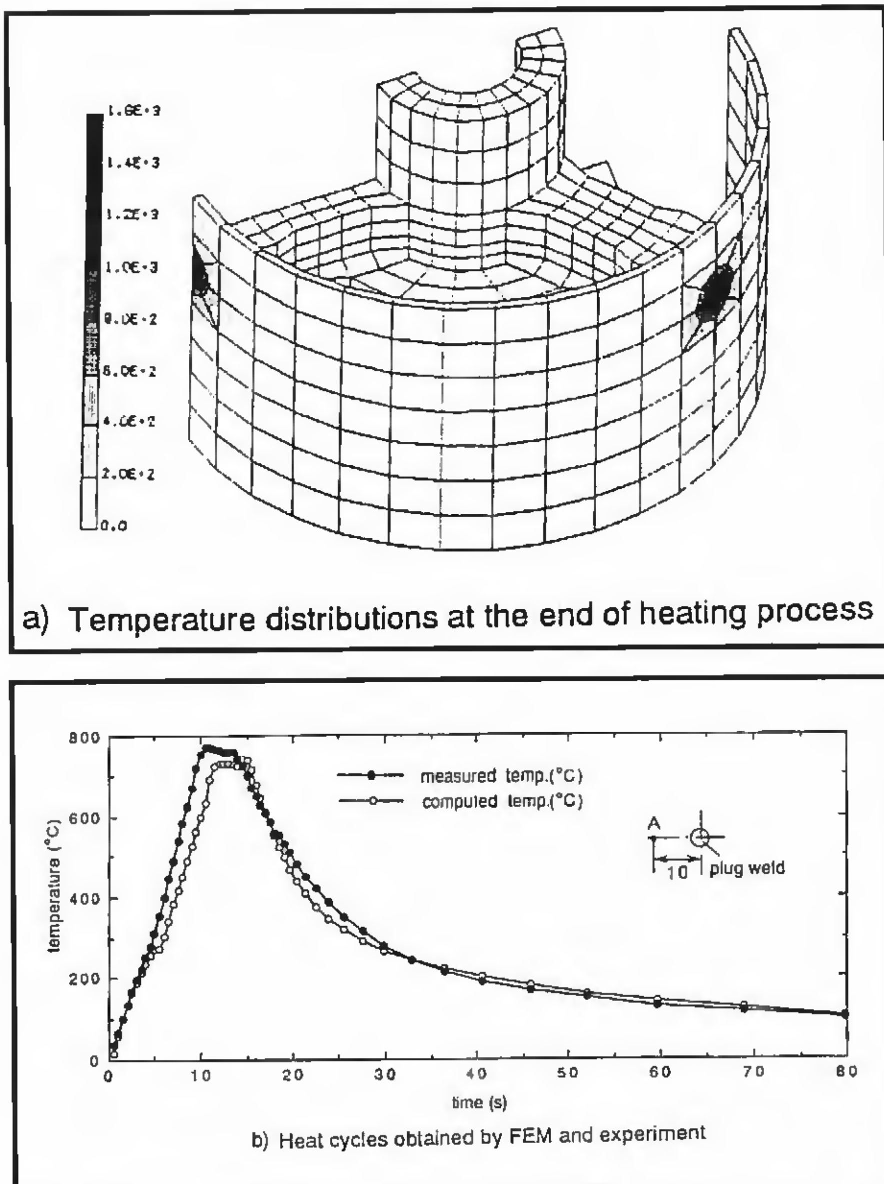

Fig. 7 — 3-D welding thermal analysis.

Simulation Model

In order to simplify the simulation model, the part of the cylinder containing only the bearing is analyzed. The fixed boundary conditions are used where the cylinder is in contact with the stator, which has a great stiffness. The eight-node brick elements with local coordinate system are used and only half of the structure is considered. The reduced integration method is applied to the cylinder to prevent the locking phenomena.

The material used is SM41 steel. The temperature dependency of the material properties used in the simulation are given based on experimental data, and are shown in Fig. 6B.

The thermal physical properties of the material are given in Table 1.

3-D Welding Thermal Analysis by FEM

Table 1 — Thermal Physical Properties of the Material Use in the Investigation

Heat conductivity	$\lambda = (54.43 - 0.000042 \cdot T^2) \cdot 0.001$	J/(mm · s · °C)
Thermal capacity	$c = 0.41 + 0.00063 \cdot T$	J/g · °C
Density	$\rho = (7.82 - 2.625 \cdot 0.0001 \cdot T) \cdot 0.001$	g/mm³
Heat transfer coefficient	$\beta = 33.5 \cdot 10^{-6}$	J/(mm² · s · °C)
Thermal expansion coefficient	$\alpha = 1.3 \cdot 10^{-5}$	°C⁻¹

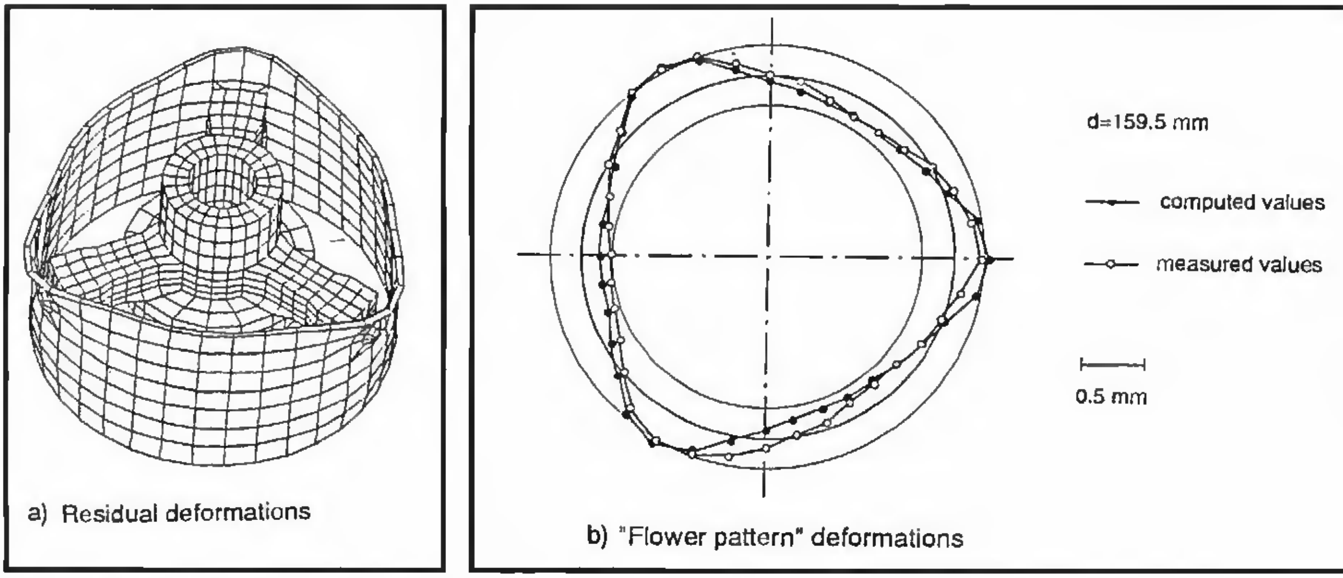

Fig. 8 — Deformation analysis of the compressor.

The heating processes include preheating and welding stages. The electric current during welding is about 200 A and the total heating time is about 15 s.

The heat of GTA welding is assumed to apply uniformly to the volume of the hole to be filled with plug welds. The rates of heat input used for the present model are 10.5 $J/(mm^3 \cdot s)$ and 21.0 $J/(mm^3 \cdot s)$ during the preheating stage and the welding stage, respectively.

Figure 7A shows the temperature distributions of the model at the moment when the heating is stopped. Figure 7B shows the heating cycles at point A on the surface of the cylinder, which is 10 mm (0.4 in.) away from the center of the plug weld. The measured curve is also given in Fig. 7B.

3-D Thermal Elastic-Plastic Analysis by FEM

During preheating, there are no materials in the holes and there is a clearance between the cylinder and the bearing. To model this clearance, dummy elements with a very small value of Young's modulus are used to fill the holes completely. Figure 8A shows the residual deformations of the model after plug welding. The shape of the end of the cylinder with the radial deformations looks like a "flower pattern," as shown in Fig. 8B, and the simulated curve is found to agree well with the experimental results. The following factors that affect the eccentricity and flower pattern are examined: 1) heat input of GTA welding; 2) clearances between the cylinder and the bearing (0.2 ~ 1 mm); 3) three plug welds deposited simultaneously(–3 ~ 3 s); 4) deviation in the position of the welds (–3 ~ 3 mm); and 5) use of the fixtures (bearing is fixed in radial and axial directions).

The precision of the compressor assembled with plug welds can be maintained by controlling the above parameters according to the information obtained by 3-D numerical simulations using this model. The accuracy of the result changes with the stages in the welding process. The error involved in the approximate solution at the given step after iterations is as follows: Elastic Stage < 10^{-12}; E1-P1 Stage (< 500°C) $10^{-5} \sim 10^{-7}$; E1-P1 Stage (500°–100°C, usually) 10^{-4}–10^{-6}; E1-P1 Stage (500°–1100°C, specially) 10^{-2}–10^{-3}.

The distributions of residual stresses in this model can also be obtained. The maximum principal stresses are observed in the HAZ around the plug welds, and they have reached a value that is important in determining the probability of crack initiation.

Conclusions

The factors that influence the accuracy and convergence of 3-D thermal elastic-plastic analysis are investigated. On the basis of the present investigation, some methods to improve the accuracy of the solution are proposed. These methods include the accurate analysis of welding heat transfer by FEM, introducing the new weighting factors in which the temperature dependency of the yield stress is considered, special consideration of E and σ_y at very high temperatures, proper selection of the temperature step and mesh size, and reduced integration to prevent the locking phenomena.

The welding deformations and stresses of a compressor have been simulated successfully by full 3-D thermal elastic-plastic analysis. The brick elements with reduced integration scheme and local coordinate system are used. The accuracy and convergence of the solution are shown to be satisfactory in all computational stages. The simulated results of the residual deformations are found to agree well with the experimental values.

References

1. Ueda, Y., and Yamakawa, T. 1971. Analysis of thermal elastic-plastic stress and strain during welding by finite element method. *Transactions of the Japan Welding Society* 2(2): 90–100.

2. Goldak, J., Oddy, A., Gu, M., Ma, W., Mashaie, A., and Huges, E. 1991. Coupling heat transfer, microstructure evolution and thermal stress analysis in weld mechanics. *Proceedings of IUTAM Symposium on the Mechanical Effects of Welding*, Lulea, Sweden.

3. Nasstrom, M., Wikander, L., Karlsson, L., Lindgren, L. E., and Goldak, J. 1991. Combined 3-D and shell modeling of welding. *Proc. of IUTAM Symposium on the Mechanical Effects of Welding*, Lulea, Sweden.

4. Brown, S., and Song, H. 1992. Implications of three-dimensional numerical simulations of welding of large structures. *Welding Journal* 71(2): 55-s to 62-s.

5. Carpenter, N., *et al.* 1986. Locking and shear scaling factors in C° bending elements, *Computers & Structures* 22(1): 39–52.

6. Ueda, Y., Murakawa, H., and Masuda, H. 1985. Locking phenomena in elastic-plastic buckling analysis by finite element method. *Proc. of 9th Symposium on Numerical Analysis of Structures*, Tokyo, Japan.