GENE-523-A101-0995
DRF # 137-0010-8

Residual Stress Analysis of the Peach Bottom Unit 3 Core Spray Pipe to Sleeve Fillet Weld

September 1995

Prepared By: Randy Stark
Randy Stark, Engineer
Engineering & Licensing Consulting Services

Approved By: Marcos T Herrera
Marcos Herrera, Principal Engineer
Engineering & Licensing Consulting Services

GE Nuclear Energy
175 Curtner Ave.
San Jose, CA 95125

IMPORTANT NOTICE REGARDING CONTENTS OF THIS REPORT

The only undertakings of the General Electric Company (GE) respecting information in this document are contained in the contract between PECO Energy Company and GE, and nothing contained in this document shall be construed as changing the contract. The use of this information by anyone other than PECO, or for any purpose other than that for which it is intended under such contract is not authorized; and with respect to any unauthorized use, GE makes no representation or warranty, and assumes no liability as to the completeness, accuracy, or usefulness of the information contained in this document, or that its use may not infringe privately owned rights.

TABLE OF CONTENTS

1. INTRODUCTION

An indication was observed in the Peach Bottom Unit 3 core spray pipe sleeve in the vicinity of the fillet weld. This fillet weld will induce weld residual stress and an area of sensitized material. The residual stress due to the fillet weld is a key parameter in determining the potential for crack initiation in the core spray pipe. Due to the uniqueness of the weld geometry, a finite element model was developed to simulate the residual stress behavior on the outside wall of the core spray pipe.

The finite element model was used to estimate the residual stresses generated in the weld material and the base metal as a result of the welding process. The numerical evaluation of weld residual stress is difficult due to numerous welding parameters and physical characteristics which affect the solution. Some of these characteristics include non-linearities such as temperature dependent material properties for both the weld metal and the base metal, highly complex heat transfer mechanisms which occur in the welding process, elastic-plastic behavior, and path dependence of the solution. Thus, this analytical evaluation should be useful as a qualitative tool to evaluate the residual stress behavior at the location of interest.

The use of analytical methods similar to those applied in this study have been used previously by GE (References 1 through 5). The results of these analytical methods have also been used in comparison with test data to demonstrate the effectiveness of the methods. In these cases, despite the complexity in modeling weld residual stresses, the analytical methods have been reasonably consistent with the results of the test measurements and observations (References 1 through 6).

2. ANALYSIS

The analytical determination of weld residual stress is comprised of three steps. These steps are:

1. Finite Element Model Development
2. Thermal Evaluation of Welding Simulation
3. Elastic-Plastic Stress Evaluation

The finite element model is developed to approximate the actual geometry of the welded joint between the core spray pipe and the connecting sleeve. The thermal evaluation provides the transient temperature history at each node in the model for the simulated weld pass. The elastic-plastic stress evaluation then uses the results of the thermal analysis to determine the residual stresses. Each of these steps are discussed further in the following sections.

2.1 Finite Element Model

A two-dimensional finite element model of the cross-section of the welded joint between the core spray pipe and the connecting sleeve was developed for this analysis using the ANSYS finite element analysis program (Reference 7). The model is composed of 1522 axisymmetric elements configured to closely approximate the actual weld joint geometry. The model is shown in Figures 2-1 and 2-2. The simulation of the weld pass requires both a thermal solution and an elastic-plastic stress calculation. This procedure is described in detail in Sections 2.2 and 2.3 of this report.

All material was stainless steel. Temperature dependent material properties are used for both materials ranging from 70 to 2700°F. A constant convection coefficient is used for all outer surfaces of the model. The top and bottom surfaces of the model (see Figure 2-

1) are extended a distance of approximately $3\sqrt{(rt)}$, where r is the pipe radius and t is the pipe thickness, to insure end effects do not affect the solution in the area of interest. The thickness of the core spray pipe is 0.280 in, and the thickness of the connecting sleeve is 0.219 in. Elements at the bottom surface in the model are restrained from motion in the area of interest, and elements at the top surface are coupled in the y-direction. No other restraints are applied to the model, and thus it is free to deflect during the weld simulation process. The weights of the pipe and sleeve are not considered in the analysis. Therefore, the predicted stress profiles are comprised solely of weld residual stresses.

2.2 Thermal Analysis

A thermal analysis of the weld pass was performed on the finite element model described in Section 2.1. Isoparametric thermal elements (STIF55 in the ANSYS Library) were used to determine the transient temperature response for the modeled weld pass.

The physics of the heat transfer near the weld pool is highly complex and requires a computationally intensive and highly specialized analysis to model accurately. The complete simulation of the heating process should account for conduction and convection within the liquid weld pool; convection, radiation, and evaporative losses at the weld pool surface; as well as heat conduction into the solid surrounding the weld pool. In order to simplify the modeling process of these complex heat transfer mechanisms, GE has employed a modeling technique which can be used to calculate the trends and approximate magnitudes of residual stresses generated by a multipass weld. This technique, called the nugget area heating method (NAH), simulates the weld process by heating the nodes in the model which lie inside the region of each weld pass to above the material melting temperature, holding this temperature for a specified period of time, then releasing the imposed temperature constraint and allowing the material to cool naturally. Each successive weld pass is simulated in this manner. The area of the heated weld nugget, the time required to ramp up the temperature, and the time at which the nugget is held at its maximum temperature are determined by the welding parameters. Figure 2-3 shows the NAH method schematically.

To simulate the weld pass, the material is heated from 70°F to 2700°F. The amount of time required to ramp up the temperature to 2700°F and the amount of time the weld nugget area is held at its maximum temperature are based on weld travel speed parameters.

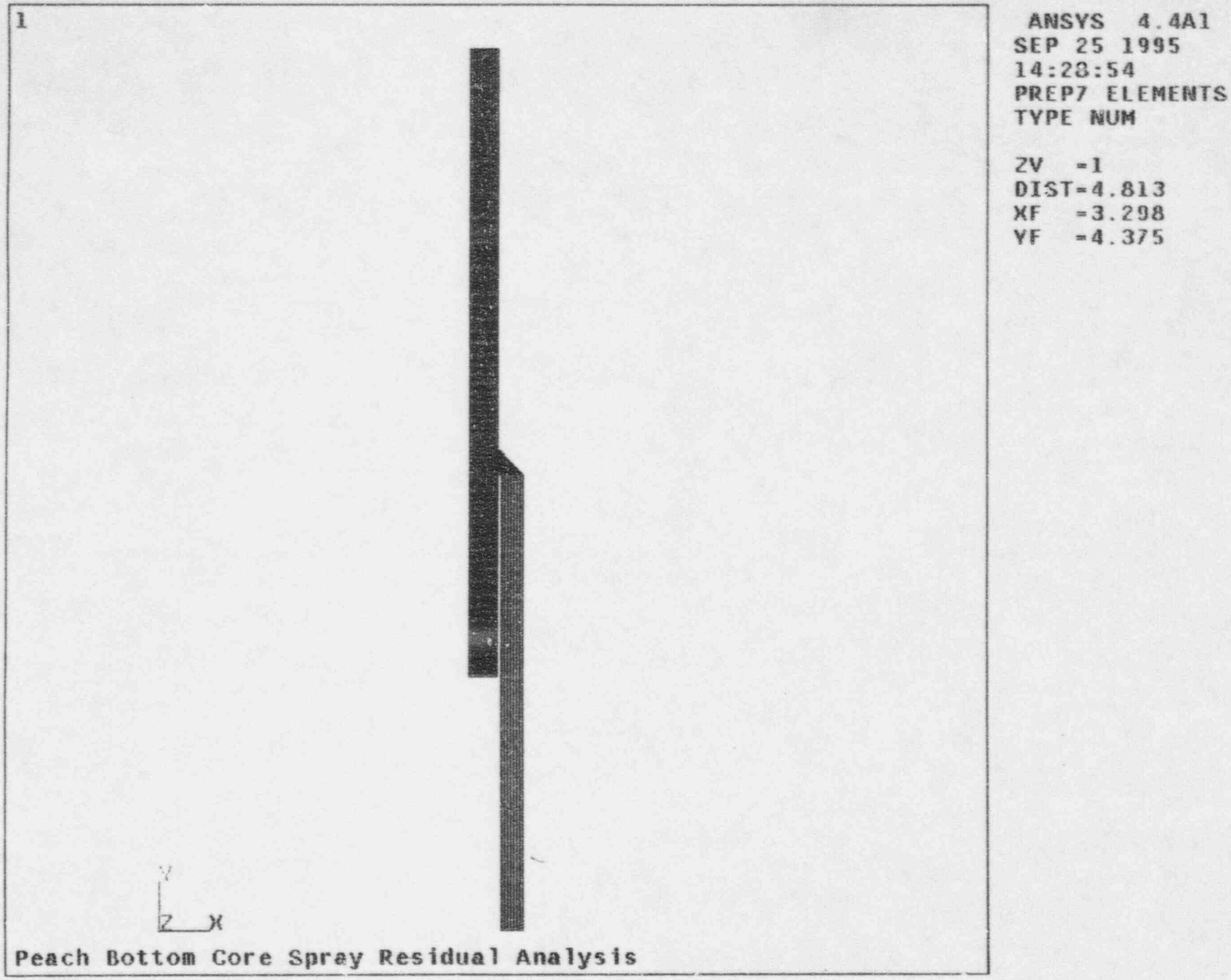

Figure 2-1: Finite Element Model

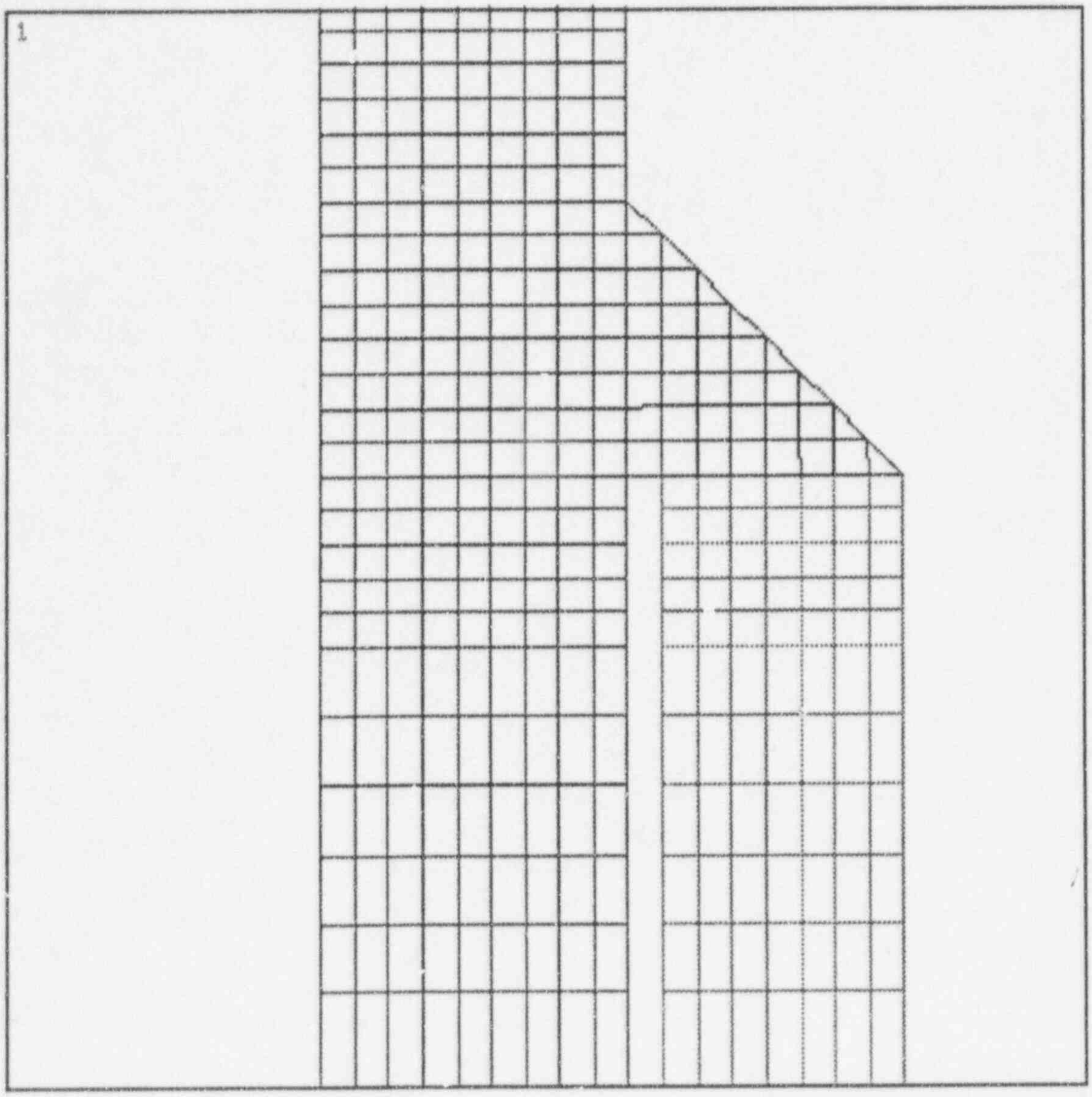

Figure 2-2: Finite Element Model - Close-up of Weld Joint

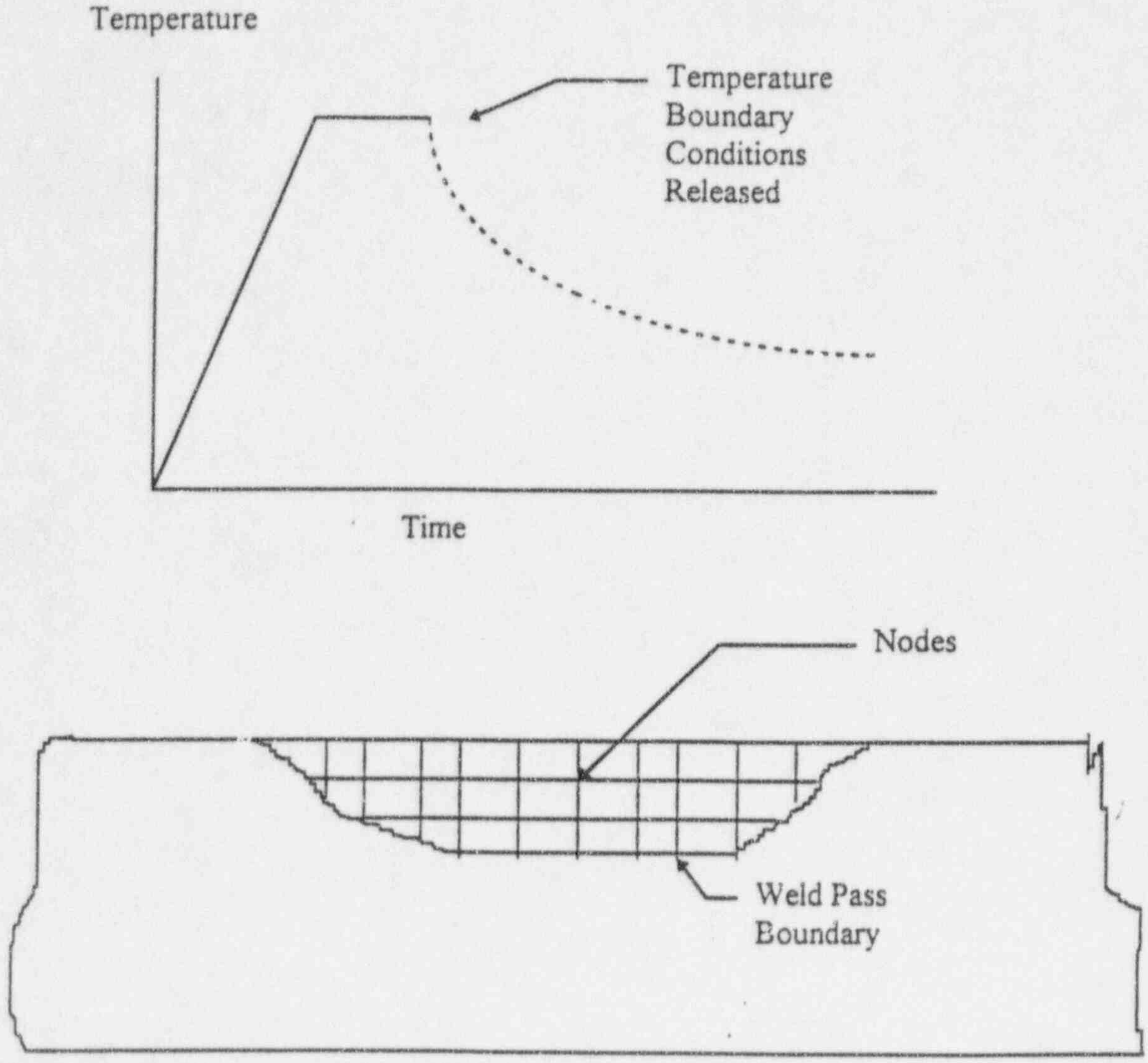

Figure 2-3: Schematic Illustrating Nugget Area Heating Technique

2.2 Elastic-Plastic Stress Analysis

The elastic-plastic mechanical analysis uses the results obtained from the thermal solution to calculate residual stresses.

Since the heat generation due to plastic deformation is negligible compared to the heat flux imposed by the welding process itself, the thermal and mechanical analyses are performed uncoupled in this evaluation. This implies that the thermal and mechanical analyses are performed in series, as discussed previously, rather than simultaneously.

The mechanical analysis was performed using the same finite element model developed for the thermal analysis. For the stress analysis, the isoparametric stress element was used (STIF42 in the ANSYS Library). The model assumes that there is no azimuthal variation in the stress due to the weld. Although in actual welds there is variation of residual stress around the circumference of the shroud, analytical methods have been found to give reasonable estimates of a typical location (References 1, 4, and 5).

Temperature dependent material properties for the elastic modulus, thermal expansion coefficient, and Poison's Ratio were used in the stress evaluation. In addition, the non-linear properties of yield strength and tangent modulus were also input as a function of temperature. The material density, for both the base metal and the weld metal, was assumed to be constant in this analysis.

The required iterative solution process ends when the user specified convergence criterion is satisfied. Plastic analysis is performed based on the Von Mises yield criterion and the Prandtl-Reuss equation. Subsequent yielding is evaluated using a bilinear kinematic hardening model. The final stress state of the welded joint is obtained when the solution for the weld pass is calculated as the temperature in the model cools to its original steady state temperature.

3. RESULTS

Stress contour plots for the axial stresses, σ_y, for two cases are shown in Figures 3-1 and 3-2 respectively. Figure 3-1 shows a sleeve-to-pipe gap of 0.0625 in, and Figure 3-2 shows a sleeve-to-pipe gap of 0.031 in. These plots show the stress state in the welded joint after the weld pass has been completed. The scale on both figures gives stress values in psi.

Figure 3-1 shows compressive residual stresses on the outside surface of the core spray pipe in the vicinity of the Heat Affected Zone for the 0.0625 in. gap case. Significant tensile stresses are predicted on the inside surface of the connecting sleeve, which is consistent with the observed crack which is postulated to have initiated on the inside surface of the sleeve at Peach Bottom Unit 3.

Figure 3-2 also shows compressive stresses on the outside surface of the core spray pipe in the vicinity of the Heat Affected Zone for the 0.031 gap case. Again, as in the previous case, tensile stresses on the inside surface of the connecting sleeve are predicted.

Figures 3-3 shows the hoop residual stresses for the 0.031 in. case. These results indicate compressive hoop stress on the outside surface of the pipe in the vicinity of the crevice.

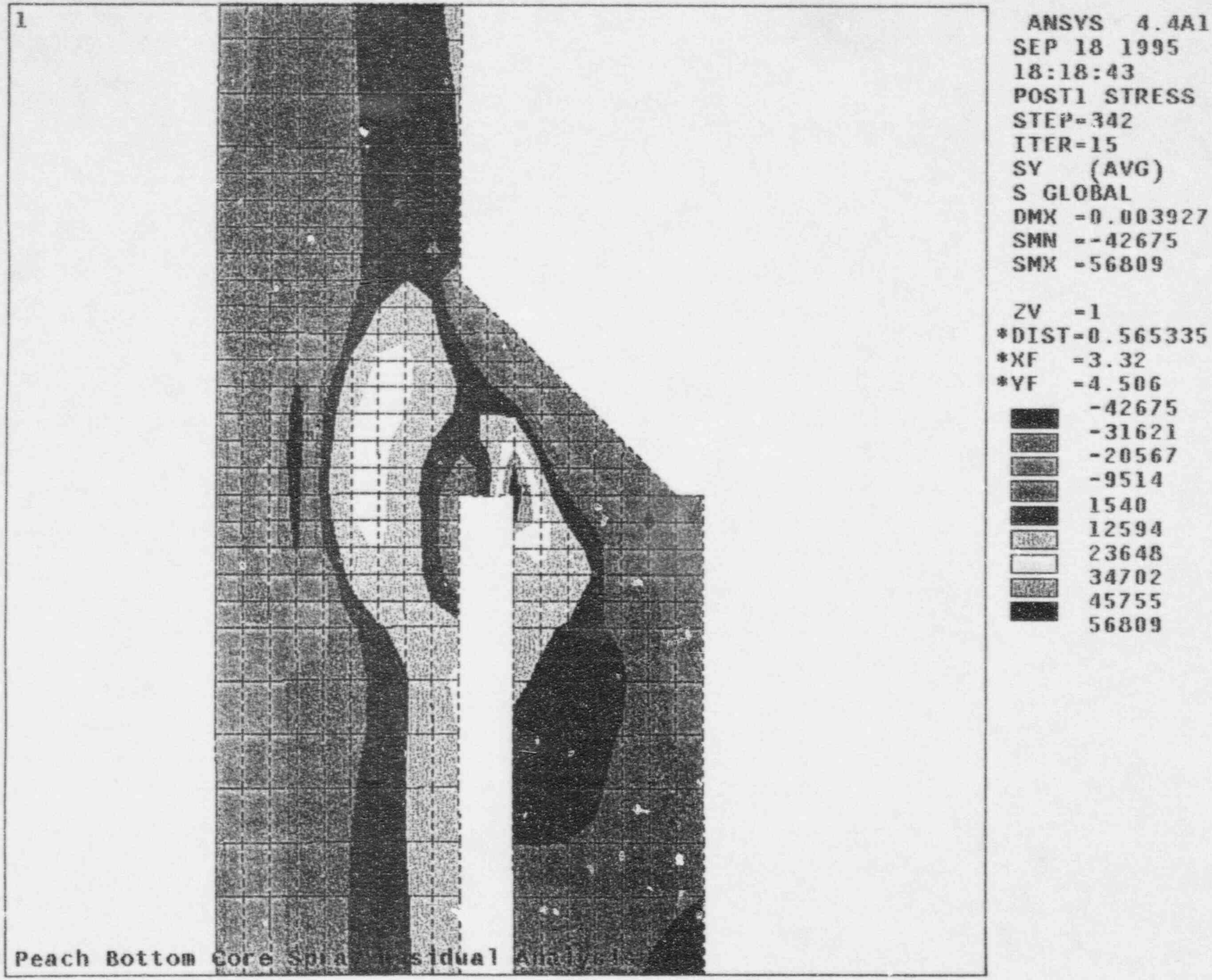

Figure 3-1: Residual Axial (σ_y) Stresses for Case 1

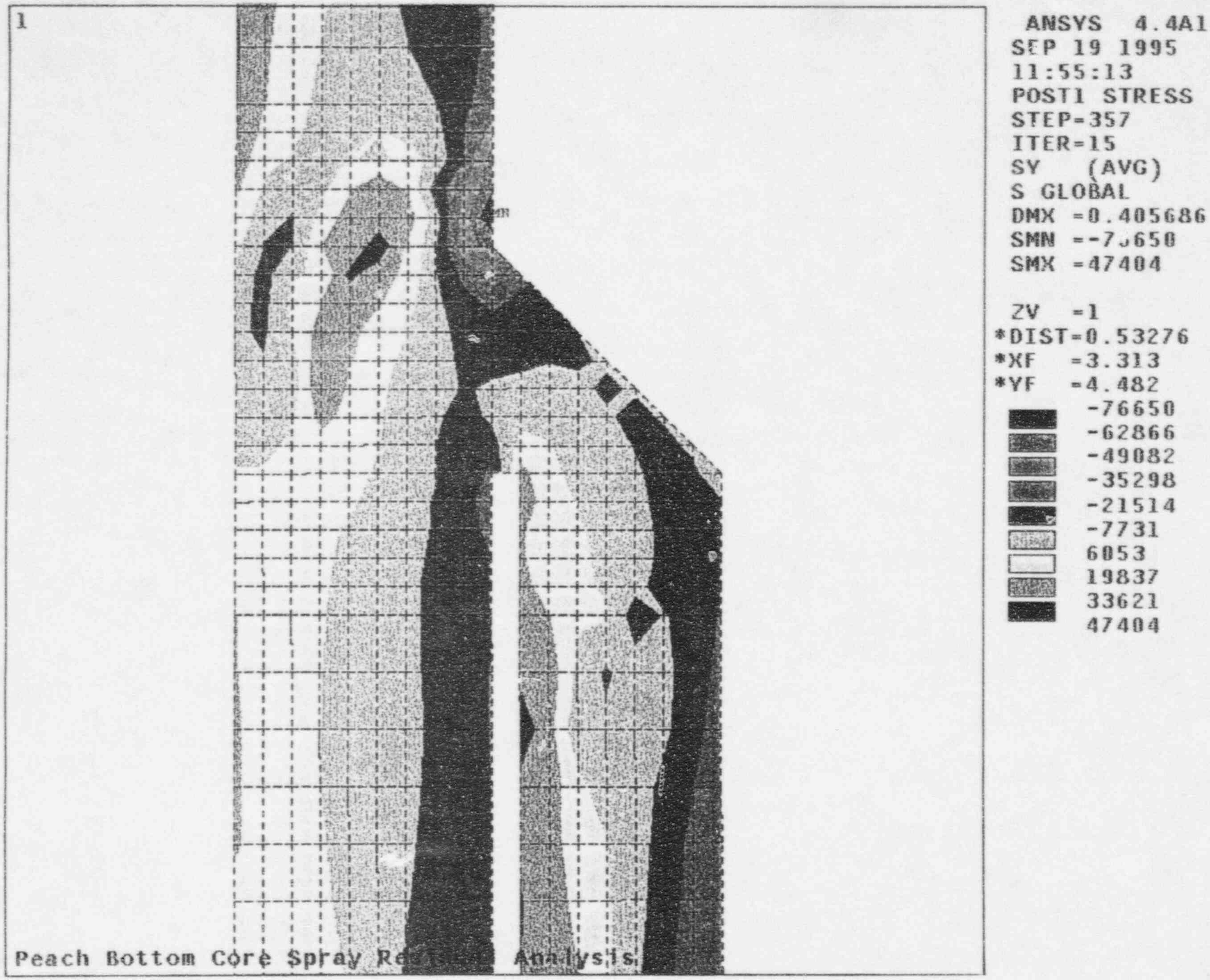

Figure 3-2: Residual Axial (σ_y) Stresses for Case 2

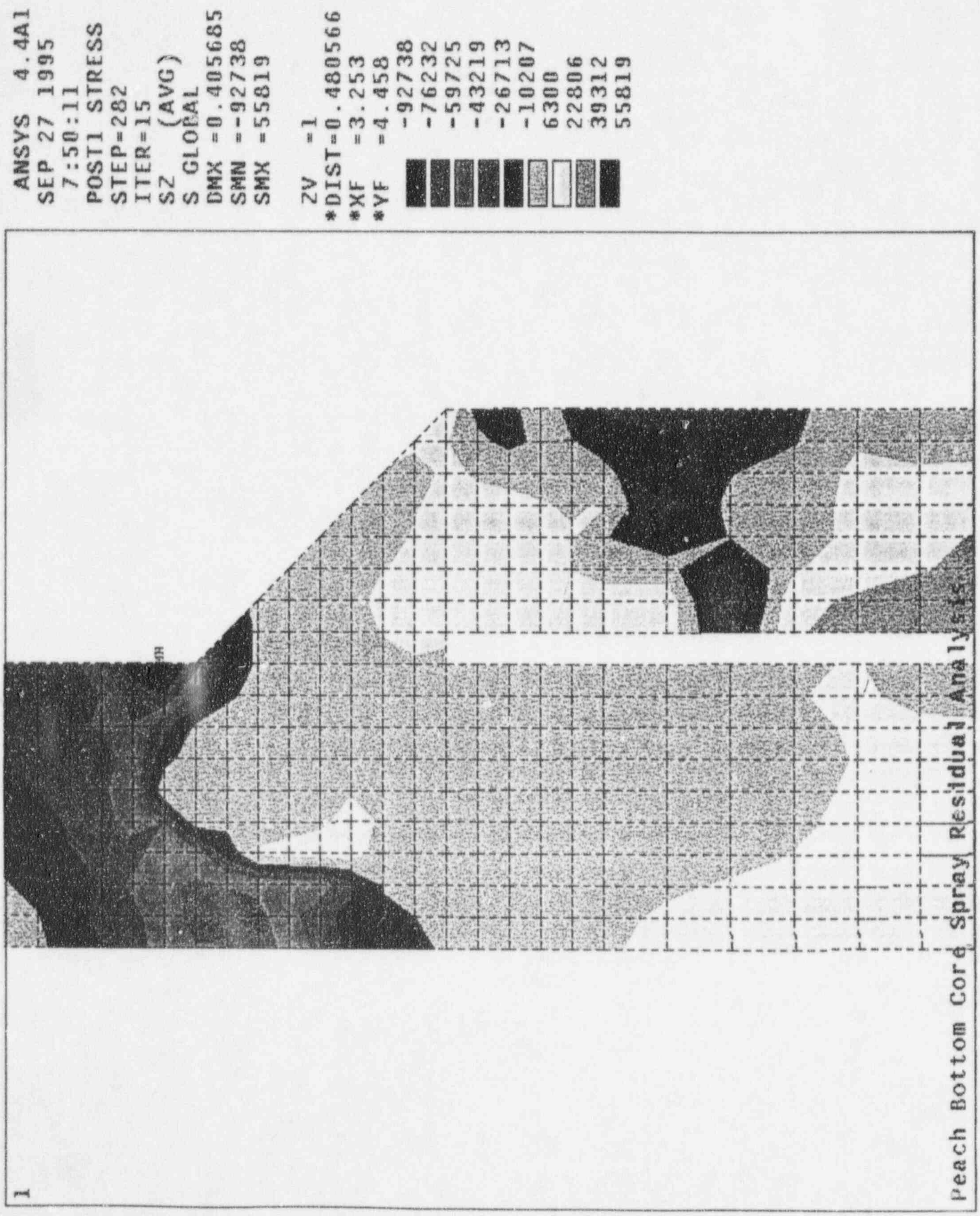

Figure 3-3: Residual Hoop (σ_z) Stresses for Case 2

4. CONCLUSIONS

Results of this evaluation indicate that fillet weld induces compressive stress on the outside surface of the pipe in the vicinity of the crevice. Tensile stress is calculated on the inside surface of the sleeve. This tensile stress is consistent with the observed crack initiation in the sleeve which is postulated as initiating on the inner surface of the sleeve and progressing through-wall to the outer surface. The compressive stress found in the pipe will inhibit the initiation of cracking in the pipe.

5. REFERENCES

1. Herrera, M.L., Lange, C.H., Ranganath, R, Analytical Evaluation of Residual Stresses in Piping Subjected to Induction Heating Stress Improvement Process and Application to Operating Plants , ASME Technical Paper 81-PVP-19.

2. Herrera, M.L., Parsons, S.R., Chapman, T.L., Residual Stress Analysis of a Sweepolet Joint Subjected to the Induction Heating Stress Improvement (IHSI) Process, ASME Volume PVP-98-3

3. Herrera, M.L., Mehta, H.S., Ranganath, S., Residual Stress Analysis of Piping with Pre-Existing Cracks Subjected to the Induction Heating Stress Improvement Treatment, ASME Technical Paper 82-PVP-60

4. Herrera, M.L., Analytical Evaluation of Residual Stresses in Piping due to Last Pass Heat Sink Welding, ASME Technical Paper 84-PVP-36

5. Rybicki, E.F., Stonesifer, R.B., Computation of Residual Stresses due to Multipass Welds in Piping Systems, Journal of Pressure Vessel Technology, May 1979, Vol. 101.

6. Rybicki, E.F., McGuire, P.A., A Computational Model for Improving Weld Residual Stresses in Small Diameter Pipes by Induction Heating, Presented at the 1980 Pressure Vessel and Piping Technology Conference, August 11-15, 1980, San Francisco, California.

7. DeSalvo, G.J., Ph.D. and Swanson, J. A., Ph.D., ANSYS Engineering Analysis System User's Manual, Revision 4.1, Swanson Analysis Systems, Inc., Houston, PA, March 1, 1983.