Westinghouse Non-Proprietary Class 3

LTR-PAFM-08-158

Salem Unit 1 Reactor Vessel Outlet Nozzle Dissimilar Metal Weld Flaw Evaluation (Fall 2008 Outage)

December 2008

Author: C. Ng*, Piping Analysis and Fracture Mechanics

Verifier: A. Udyawar*, Piping Analysis and Fracture Mechanics

Approved: S. A. Swamy*, Manager, Piping Analysis and Fracture Mechanics

Electronically approved records are authenticated in the Electronic Document Management System

Westinghouse

1.0 Introduction

An inside surface connected circumferential indication was detected in a Salem Unit 1 reactor vessel outlet nozzle dissimilar metal (Alloy 182) butt weld during the pre Mechanical Stress Improvement Process (MSIP®) inspection in the Fall 2008 Refueling Outage. The detected indication exceeded the acceptance standards of ASME Boiler & Pressure Vessel Code Section XI, Subsection IWB Article IWB-3500 (Reference 1). MSIP was subsequently performed for the reactor vessel outlet nozzle of interest during the Fall 2008 outage. This process induces compressive residual stresses on the inside surface and inner wall of the nozzle where the indication was detected. The post-MSIP residual stresses, when combined with the stresses due to normal operating piping reaction loads would mitigate any future crack propagation due to Primary Water Stress Corrosion Cracking (PWSCC).

To demonstrate that such a mitigation is technically sound, a flaw evaluation was performed for the nozzle of interest, using the rules of ASME Code Section XI, Article IWB 3600 for the as-found indication. Fatigue and PWSCC crack growth analyses were also performed to demonstrate the effectiveness of MSIP to mitigate any future crack propagation.

2.0 Geometry, Piping Loads and Material Properties

Nozzle dimensions at the weld location of interest (Reference 2) and the relevant design parameters used in the flaw evaluation are summarized in Table 2-1. Table 2-2 provides the as-found flaw dimensions as reported in Reference 2. The piping reaction loads at the reactor vessel outlet nozzle dissimilar metal weld are summarized in Table 2-3. It should be noted that the loads tabulated in Table 2-3 are bounding Salem Units 1 and 2 loads and therefore are conservative and acceptable to use for Salem Unit 1.

Table 2-1: Geometry and Design Parameters

Outside diameter	34.1 inch
Wall thickness	2.62 inch
Bounding Normal Operating Temperature	609 °F
Normal Operating Pressure	2235 psig

Table 2-2: As-found Flaw Parameters

Flaw Depth	0.634 inch
ID Flaw Length	2.06 inch
Flaw Depth / ID Flaw Length	0.31
Flaw orientation	Circumferential

Table 2-3: Reactor Vessel Outlet Nozzle Piping Reaction Loads

Loading	Forces (kips)	Moments (in-kips)		
	F_x (Axial)	M_x (Torsion)	M_y (Bending)	M_z (Bending)
Deadweight	-0.27	-83.5	-25.9	-1031
Thermal	-48.4	289	-320	-17737
OBE	670.8	455	3324	7463
SSE	870	691	4625	14893
LOCA	938	2090	22011	8930

Note:

OBE – Operating Basis Earthquake
SSE – Safe Shutdown Earthquake
LOCA – Loss of Coolant Accident

3.0 Flaw Evaluation

Since the as-found circumferential indication exceeded the acceptance standards of ASME Section XI, Subsection IWB Article IWB-3500, a flaw evaluation in accordance with the analysis procedures and acceptance criteria as specified in Subsection IWB-3600 was performed. The limit load approach in the ASME Code Section XI, Division 1 Appendix C was used in the flaw evaluation.

Using the equations in the ASME Code Appendix C, the maximum allowable end-of-evaluation period flaw depth to wall thickness ratios were calculated as a function of flaw depth to flaw length ratio for normal, upset, emergency and faulted conditions. The results from the limiting service condition are plotted in Figure 1 as well as the as-found flaw parameters. Since the as-found flaw depth is below the maximum allowable end-of-evaluation period flaw depth shown in Figure 1, the as-found indication was acceptable in accordance with the applicable ASME Section XI acceptance criteria prior to the MSIP application at Salem Unit 1, thus demonstrating that the plant was in an operable condition with the detected circumferential indication in the reactor vessel outlet nozzle dissimilar metal weld.

Figure 2 shows that the post-MSIP axial through-wall stress is compressive up to about 40% of the wall thickness from the inside surface after superposing the post-MSIP residual stress with the normal operating axial stress due to piping reaction loads at the reactor vessel outlet nozzle dissimilar metal weld. The post-MSIP residual stress shown in Figure 2 is obtained from Reference 3. This compressive stress region far exceeds the as-found flaw depth to wall thickness ratio of 0.24, thereby indicating that the MSIP application will arrest any future crack growth due to PWSCC for the as-found indication. This is illustrated further by determining the crack tip stress intensity factor (SIF) for a circumferential indication based on a linearized post-MSIP axial through-wall stress distribution profile and the influence coefficients obtained from Reference 4. The crack tip stress intensity factor calculated for a circumferential indication with a conservative aspect ratio (length/depth) of 6 and a conservative flaw depth to wall thickness ratio of 0.25, which bounds the as-found indication, is shown in Table 3-1. The crack tip stress intensity factor calculated is clearly negative and reinforces the conclusion that MSIP will mitigate any future crack growth due to PWSCC for the as-found indication.

Table 3-1: Post-MSIP Crack Tip Stress Intensity Factor

Flaw Length to Flaw Depth Ratio	6.0
Flaw Depth to Wall Thickness Ratio	0.25
Through-wall Membrane Stress	14.5 ksi
Through-wall Bending Stress (Compressive at the Inside Surface)	57.2 ksi
Crack Tip Stress Intensity Factor (K_I)	-32.6 ksi $\sqrt{in}$

In order to demonstrate that the as-found indication would remain in the compressive stress region in the post-MSIP configuration, fatigue crack growth for a circumferential indication with a conservative aspect ratio (length/depth) of 6 and a conservative flaw depth to wall thickness ratio of 0.25 was evaluated in accordance with the fatigue crack growth rate for Alloy 182 weld material from NUREG/CR-6907 (Reference 5) based on the applicable Salem Unit 1 design thermal transients at the reactor vessel outlet nozzle dissimilar metal weld. The results of the fatigue crack growth analysis are shown in Table 3-2 and there is no fatigue crack growth. This is primarily due to the fact that the post-MSIP axial through-wall stress is compressive up to about 40% of the wall thickness from the inside surface and there are no severe thermal transients at the reactor vessel outlet nozzle dissimilar metal weld. The results of the fatigue crack growth analysis demonstrated that MSIP will also mitigate any future fatigue crack growth and the as-found circumferential indication would remain in the compressive stress region. Therefore, PWSCC is no longer a credible crack growth mechanism for the as-found indication.

Table 3-2: Post-MSIP Fatigue Crack Growth Results

	Flaw Depth to Wall Thickness Ratio After			
Initial Flaw Depth to Wall Thickness Ratio	10 years	20 years	30 years	40 years
0.25	0.25	0.25	0.25	0.25

4.0 Summary and Conclusions

The circumferential indication detected in the Salem Unit 1 reactor vessel outlet nozzle dissimilar metal butt weld was found to meet the allowable limits of the ASME Section XI Article IWB-3600 and Appendix C requirements under normal, upset, emergency and faulted service conditions before MSIP application was performed during the Fall 2008 outage, thus demonstrating that Salem Unit 1 was in an operable condition at all times with the detected circumferential indication in the reactor vessel outlet nozzle dissimilar metal weld.

A review of the post-MSIP through-wall axial stress distribution shows that the indication of interest in the reactor vessel outlet nozzle dissimilar metal weld is well within the compressive stress region of the nozzle and that the resulting crack tip stress intensity factor calculated is negative. Fatigue crack growth evaluation was also performed for the post-MSIP configuration to demonstrate that there is no fatigue crack growth and that the as-found indication would remain in the compressive stress region of the nozzle. Therefore, it can be concluded that the MSIP application implemented at the Salem Unit 1 reactor vessel outlet nozzle of interest is effective and will mitigate any future crack growth due to both PWSCC and fatigue crack growth mechanisms.

5.0 References

1. Rules for Inservice Inspection of Nuclear Power Plant Components, ASME Boiler & Pressure Vessel Code, Section XI, 1998 Edition, including 2000 Addenda.
2. WesDyne Letter WDI-PJF-1303981-TR-010, Rev. 0, "Salem Unit 1 – ASME Code Section XI IWB-3500 Evaluation of Flaw Indication #11 in Outlet Nozzle 29-RC-1140-1", dated October 26, 2008.
3. NuVision Engineering Report 4418-6-001, "MSIP® Finite Element Stress Analysis Procedure - Salem Unit 1 Outlet Nozzle To Safe End Weld 29-RC-1140-1 Flaw Evaluation", November 2008.
4. S. Chapuliot, M.H. Lacire and P. Le Delliou, "Stress Intensity Factors for Internal Circumferential Cracks in Tubes over a Wide Range of Radius over Thickness Ratios", ASME PVP Volume 365, 1998.
5. NUREG/CR-6907, ANL-04/3, "Crack Growth Rates of Nickel Alloy Welds in a PWR Environment," May 2006.

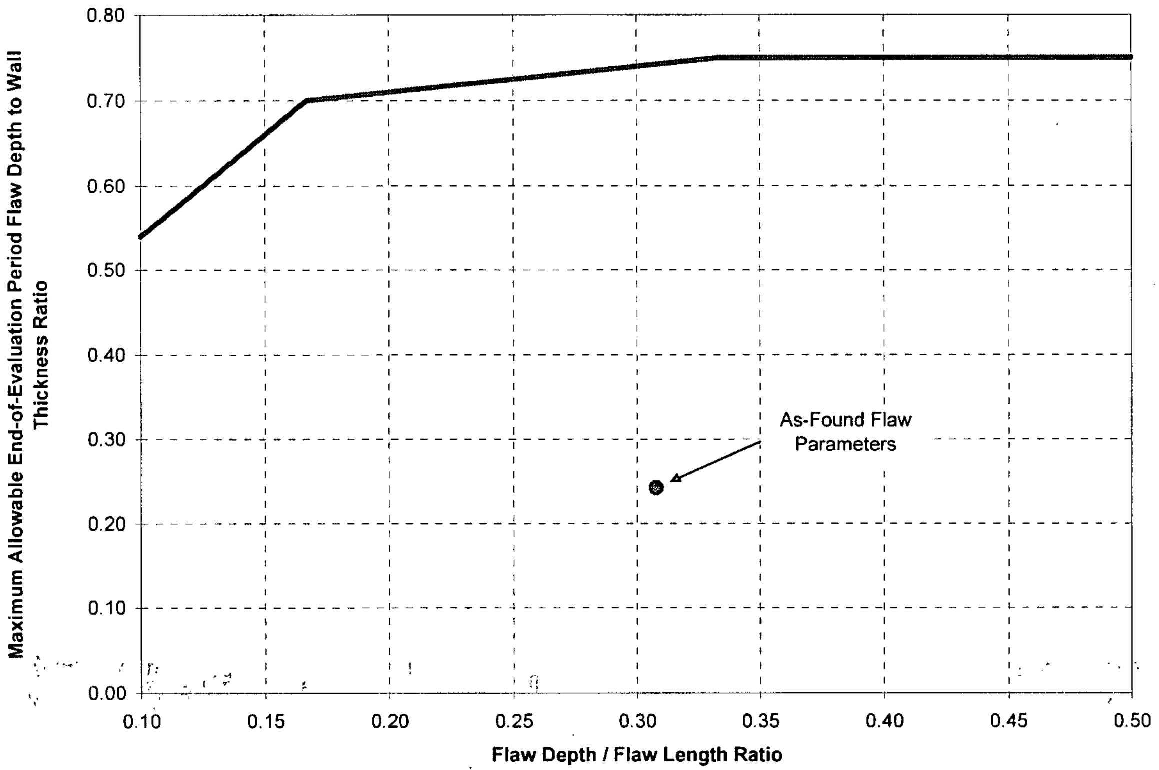

Figure 1: Comparison of As-Found Flaw Size with Maximum Allowable End-of-Evaluation Period Flaw Size

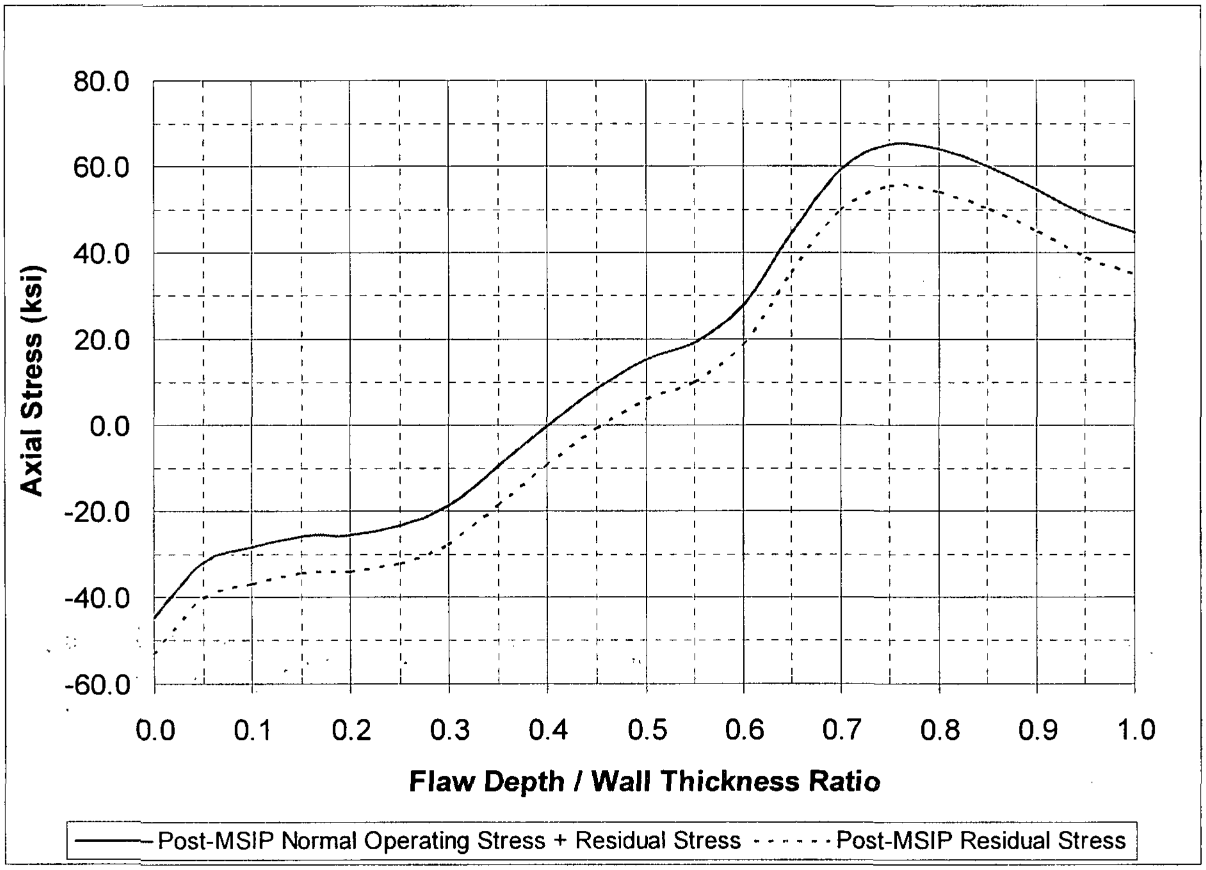

Figure 2: Reactor Vessel Outlet Nozzle Dissimilar Metal Weld Post-MSIP Through-wall Axial Stress Distribution

ATTACHMENT 2

Westinghouse

To: R. Walters
cc: S. A. Swamy
R. L. Brice-Nash
C. K. Ng

Date: December 03, 2008

From: M. Dowdell
Ext: 724-722-6102
Fax: 724-722-5597

Your ref:
Our ref: LTR-PAFM-08-166

Subject: Salem Units 1 and 2 - RCL Piping Stresses at the RPV Nozzles

References:

1. Westinghouse Calculation: CN-PAFM-06-1, Rev. 1, "Salem Unit 2 RCL Piping and Nozzle Qualifications for the Replacement Steam Generator," B. Carpenter. [Westinghouse Proprietary Document]
2. ASME Boiler and Pressure Vessel Code, Section III, 1986 Edition.
3. ASME Boiler and Pressure Vessel Code, Section III, 1974 Edition.
4. NuVision Report: 4418-4-001-00, "Analytical Verification of MSIP© for RV Cold Leg Nozzle to Safe End Weld Salem Unit 1," October 2007. [NuVision Proprietary Document]
5. NuVision Report: 4418-4-002-00, "Analytical Verification of MSIP© for RV Hot Leg Nozzle to Safe End Weld Salem Unit 1 & 2," October 2007. [NuVision Proprietary Document]
6. NuVision Report: 4418-4-003-00, "Analytical Verification of MSIP© for RV Cold Leg Nozzle to Safe End Weld Salem Unit 2," September 2008. [NuVision Proprietary Document]
7. Westinghouse Calculation: CN-SMT-02-54, Rev. 1, "Salem 1 RCL SG Snubber Elimination Results," L. M. Valasek. [Westinghouse Proprietary Document]
8. Westinghouse Calculation: CN-PAFM-05-91, Rev. 1, "Salem 2 RSG Deadweight, Thermal and LOCA WESTDYN Runs," B. Carpenter. [Westinghouse Proprietary Document]

R. Walters,

Please transmit the following to the Salem customer.

The normal operating condition stresses at the reactor pressure vessel (RPV) inlet and outlet nozzles for Salem Units 1 and 2 have been evaluated against the stress allowable, S_m. Based on the following evaluations, the normal operating condition stresses at the RPV inlet and outlet nozzles are less than 1 S_m, as consistent with NUREG-0313.

The evaluation was performed using the following methodology. The normal operating condition stresses consisted of deadweight, 100% power thermal, and operating pressure stresses. The stresses were evaluated at the location of minimum wall thickness for the area between the RPV nozzle and the reactor coolant loop (RCL) piping. The normal operating condition stress was then evaluated against the minimum stress allowable, S_m, for the materials used.

The minimum wall thickness for the section from the RPV nozzle to the RCL piping was determined from a review of Figures 2.1-1 found on pages 19 of References 4, 5, and 6. From the review, the minimum wall thickness occurs at the location between the safe end and the safe end weld to the RCL piping. For the Salem Unit 1 and 2 hot legs, the minimum wall thickness was determined to be 2.4 inches; calculated as ((34.0" – 29.2") / 2), see Figure 1. For the Salem Unit 1 and 2 cold legs, the minimum wall thickness was also determined to be 2.4 inches; calculated as ((32.5" – 27.7") / 2), see Figure 1.

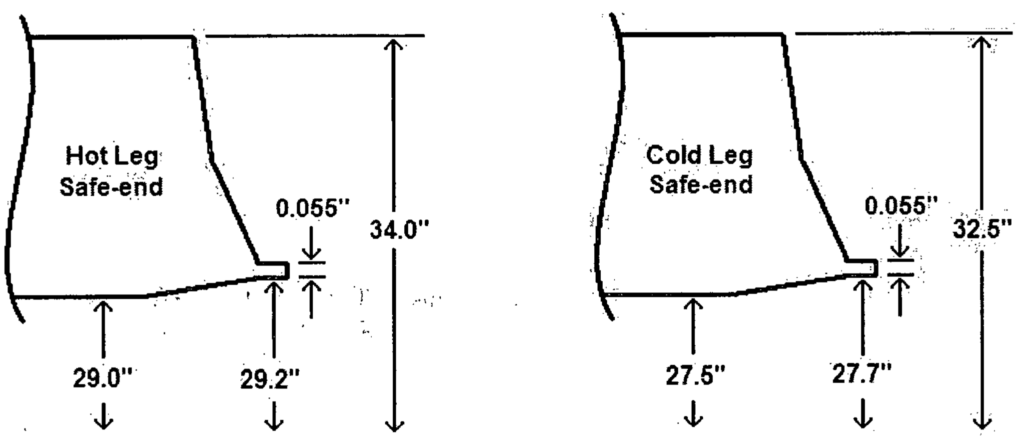

Figure 1: Hot Leg and Cold Leg Safe End Dimensions

The following equations were used to calculate the deadweight (DW), thermal (TH), and pressure (P) stresses, as well as the section modulus (Z). The outer diameter (D_o) and the inner diameter (D_i) were taken from the location of the minimum wall thickness, as seen in Figure 1. For the following equations, a normal operation pressure of 2235 psi was used. The stress indices of 1.0 at the outlet nozzle and 1.62 at the inlet nozzle (References 1 and 7) were also used.

$$\sigma_{DW} = \frac{iM_{DW}}{Z} \qquad \sigma_{TH} = \frac{iM_{TH}}{Z} \qquad \sigma_P = \frac{PD_i^2}{D_o^2 - D_i^2} \qquad Z = \frac{\pi}{32}\frac{D_o^4 - D_i^4}{D_o}$$

The resultant moment loads (M) at the RPV nozzles for Salem Units 1 and 2 were taken from References 7 and 8. The deadweight and thermal resultant moment loads are summarized in Table 1 for the RPV inlet and outlet nozzles.

Table 1: Resultant Moment Loads at the RPV Nozzles

Unit	Nozzle	Case	Load (in-k)	Reference
Unit 1	Outlet	DW	802	Ref. 7
Unit 1	Outlet	TH	17739	Ref. 7
Unit 1	Inlet	DW	344	Ref. 7
Unit 1	Inlet	TH	2552	Ref. 7
Unit 2	Outlet	DW	1034.26	Ref. 8
Unit 2	Outlet	TH	10813.4	Ref. 8
Unit 2	Inlet	DW	350.23	Ref. 8
Unit 2	Inlet	TH	2136.12	Ref. 8

The deadweight, thermal, and pressure stresses are calculated below in Table 2. The total stress listed in Table 2 is the summation of the deadweight, thermal, and pressure stress to represent the stress at the normal operating condition.

Table 2: Deadweight, Thermal, and Pressure Stresses

Unit	Nozzle	Pressure Stress (ksi)	Deadweight Stress (ksi)	Thermal Stress (ksi)	Total Stress (ksi)
Unit 1	Outlet	6.282	0.456	10.082	16.820
Unit 1	Inlet	5.935	0.350	2.597	8.882
Unit 2	Outlet	6.282	0.588	6.146	13.015
Unit 2	Inlet	5.935	0.356	2.174	8.465

The stress allowable (S_m) was determined for the materials of the RPV nozzle, safe end, and RCL piping. The stress allowable for the weld materials is not listed as it is not the limiting allowable. The RCL piping for Salem Units 1 and 2 are qualified according to the B31.1 piping code, which does not contain the stress allowable, S_m. However for this evaluation, the stress allowable, S_m, was determined using the ASME Code. Comparisons were made between the stress allowables in the 1974 and 1986 Editions (References 2 and 3) and there were no differences in value for the materials considered. The stress allowables are listed in Table 3. The stress allowables at 600°F and 650°F were determined from the ASME Code and the allowable at 620°F was interpolated.

Table 3: Material Stress Allowable (S_m)

Component	Material	Sm @ 600°F (ksi)	Sm @ 620°F (ksi)	Sm @ 650°F (ksi)
RPV Nozzle	SA-508 Cl. 2	26.7	26.7	26.7
Safe End	SA-182 F316	17.0	16.9	16.7
RCL Piping	SA-376 TP316	17.0	16.9	16.7

Based on Table 3, the lowest allowable at 620°F (bounding temperature of the hot leg during normal operation) is 16.9 ksi. From Table 2, the maximum total normal operating condition stress is at the outlet nozzle and is 16.82 ksi. Therefore, the stresses at the RPV inlet and outlet nozzles for Salem Units 1 and 2 are less than 1 S_m.

Should you have any questions, please contact the undersigned.

Regards,

Author: M. Dowdell*
Piping Analysis & Fracture Mechanics

Verifier: B. Carpenter*
Piping Analysis & Fracture Mechanics

Electronically approved records are authenticated in the Electronic Document Management System.

Attachment 3

Response to Item 4

The description of the MSIP® analysis procedure is given in report Number 4418-6-001, Outlet Nozzle to Safe End Weld 29-RC-1140-1 Flaw Evaluation. The tool location and amount of squeeze is established such that the post- MSIP® residual stresses are compressive in the inner weld region through about 50% of the wall thickness.

The limitations on the application of MSIP® regarding size of the indications that may be found in the pre- MSIP® inspection are 10% circumferential length and 30% through-wall depth for circumferential flaws. With the qualified NDE inspections, the 30% through-wall depth limitation is also used for axial flaws. These are consistent with what the NRC has approved in the past.

If an indication is found in the pre- MSIP® inspection, then depending on its orientation, the appropriate through-wall stress distribution (axial or hoop) is determined for the actual squeeze at the location of the indication. This through-wall stress distribution includes the compressive residual stress generated by MSIP®, plus the stresses due to the operating loads and is used in the fracture mechanics evaluation to confirm there will be no flaw growth.

MSIP® FINITE ELEMENT STRESS ANALYSIS PROCEDURE

SALEM UNIT 1

OUTLET NOZZLE TO SAFE END WELD 29-RC-1140-1 FLAW EVALUATION

PROTECTED BY U.S. PATENT
Nos. 4,683,014 & 4,612,071

Prepared for

WESTINGHOUSE ELECTRIC COMPANY, LLC
PITTSBURGH, PENNSYLVANIA

November 2008

Prepared by	**Date** 11-19-08	**Quality Assurance**	**Date** 11-21-08
Verified by	**Date** 11/19/08	**Approved by**	**Date** 11-21-08
Project Secretary	Date 11/19/08		

NuVision Engineering, Inc.
River Park Commons
2403 Sidney Street, Suite 700
Pittsburgh, PA 15203 - 2181
(412) 586 – 1810
FAX: (412) 586 – 1811

Rev.				
Date				
Appr. QA				

MSIP® is a registered trademark of AEA Technology Engineering Services Inc. MSIP® has been registered as a trademark in the US Patent and Trade Mark Office

MSIP® FINITE ELEMENT STRESS ANALYSIS PROCEDURE

SALEM UNIT 1

OUTLET NOZZLE TO SAFE END WELD 29-RC-1140-1 FLAW EVALUATION

Table of Contents

MSIP® FINITE ELEMENT STRESS ANALYSIS PROCEDURE
SALEM UNIT 1
OUTLET NOZZLE TO SAFE END WELD 29-RC-1140-1 FLAW EVALUATION

1.0 INTRODUCTION

MSIP® is a mechanical process that replaces the large tensile as-welded residual stresses along the inner region of piping in the vicinity of a weld with a zone of compressive residual stress. In general terms, MSIP® consists of squeezing a pipe plastically near a weld using a specifically designed set of rings that grip a short length of pipe. The squeezing is continued until the tensile residual stresses along the inner region of the weld are replaced by compressive stresses.

A finite element stress analysis is performed that provides the analytical basis for MSIP® and verifies that the process does accomplish the desired stress improvement. The general objective of the evaluation is to analytically simulate MSIP® on the pipe-to-safe end-to-nozzle assembly and to predict the resulting redistribution of residual stresses. This is accomplished by performing a nonlinear finite element analysis of the process. Residual stresses are determined in the weldment for the as-welded condition with a local ID weld repair, post-hydrostatic test and post- MSIP® process as well as steady state plant operating conditions both before and after the application of MSIP®. A comparison of the analytical residual stress patterns shows that MSIP® is successful in alleviating the substantial weld-induced tensile stresses in both the axial and circumferential (hoop) directions along the inner region of the nozzle to safe end weld.

A key objective of the evaluation is to determine the acceptable range of MSIP® tool location and radial contractions necessary to achieve the desired stress redistribution. Ranges for the desired radial contraction and tool locations are used in development of the MSIP® Parameters for field application of MSIP®.

2.0 ANALYSIS METHODOLOGY

The ANSYS finite element program is used for the MSIP® analysis. The analysis of MSIP® on the nozzle-to-safe end weldment is performed using an axisymmetric finite element computer model of the weldment assembly. The model represents dimensional details of the Nozzle, safe end, attached pipe and the circumferential welds that connect the pipe to safe end and the safe end to the nozzle.

Temperature dependent inelastic material properties are used in the evaluation to ensure that the nonlinear behavior of the materials is represented. The inelastic behavior of the material is simulated using a bilinear kinematic hardening (BKIN) option within ANSYS.

The computer model is first used to simulate the residual stresses induced by the welding process in the weld regions. This is accomplished by imposing a temperature gradient in the weld region and then removing it, which results in residual stresses distributions similar in magnitude to the weld residual stresses (including ID weld repair) reported in the literature.

Next pressure due to the hydrostatic test is applied to the model and then removed. Normal operating temperature and pressure are then applied and removed.

The application of MSIP® is simulated in the model by first applying pressure along the outer surface of the safe end to simulate loading applied by the MSIP® tool clamp ring assembly and then removing it. The maximum pressure generated by the MSIP® tool is increased until acceptable residual stress redistribution is achieved in the nozzle to safe end weld region upon removal of the tool.

Steady state plant operating loading, represented by normal operating temperature and internal pressure, is then applied to the post- MSIP® residual stress pattern. A uniform temperature equivalent to the plant normal operating condition (T_{hot}) is applied throughout the material, and a uniform pressure (P_{op}) equivalent to the plant normal operating condition is also applied to all inside surfaces of the nozzle, safe end, weld and pipe including blow-off pressure. The results of primary importance are the final axial and hoop stresses in the inner region of the nozzle-to-safe end weld.

Finally a series of finite element analyses are performed varying the amounts of radial contractions and MSIP® tool locations to determine acceptable ranges for field implementation of MSIP®.

2.1 FINITE ELEMENT MODEL GEOMETRY AND BOUNDARY CONDITIONS

A 2-D axisymmetric finite element computer model of the RV hot leg nozzle assembly was used in the analysis. The dimensions of the weldment assembly were based on drawings and walk-down information and were used as the foundation of the finite element model.

The model was constructed using the ANSYS PLANE42 axisymmetric isoparametric elements. The overall finite element model and its detail in the weld region are shown in Figures 1 and 2 respectively.

The overall length of the model on the pipe side of the assembly is extended to be equal to or greater than $4(rt)^{(0.5)}$ where "r" is the average radius and "t" is the wall thickness. The nodes at the end of the pipe are coupled in the axial direction to simulate the remaining piping not modeled. To ensure stability of the finite element solution, the nozzle end of the model is constrained normal to the nozzle cut plane. The model boundary conditions are shown in Figure 3.

The loading applied by the contour ring of the MSIP® tool clamp ring assembly is simulated by a radial pressure along the outer surface of the pipe. The applied tool pressure is also illustrated in Figure 3.

2.2 MATERIAL PROPERTIES

The different materials of construction for the RV hot leg nozzle assembly are shown in Figure 4 and tabulated in Table 1.

Table 1 – RV Nozzle Assembly Materials

Component Description	ANSYS Mat. Number in Model	Material Designation
Nozzle	1	SA-508 Cl. 2
Cladding	2	E308L
Nozzle to Safe End Weld	4	Inconel 182
Safe End	5	SA-182-F316
Pipe to Safe End Weld	6	E308L
Pipe	8	SA-376 TP316

The inelastic behavior of the material is simulated using the bilinear kinematic hardening (BKIN) option within ANSYS. The material properties used in the analysis are based on certified material test reports (CMTR) when available otherwise typical yield strength and ultimate tensile strength of the material are used.

2.3 SIMULATION OF AS-WELDED STRESSES

The residual stresses due to welding are simulated in the model by applying a temperature distribution in each of the weld regions. The temperature distribution is then removed to achieve a weld residual stress profile. The temperature distributions are applied in the same sequence as they were when the weldment was assembled. The nozzle to safe end as-designed weld was first applied followed by an ID weld repair to the nozzle to safe end weld. This was then followed by pipe to safe end weld.

2.4 PRE-MSIP® LOADING

Prior to the application of MSIP® the effect of the hydrostatic testing and normal operating conditions on the residual stress patterns is evaluated. This is accomplished by applying a uniform temperature to the entire model as well as a pressure loading to the inner surfaces of the nozzle assembly. The loading due to blow-off is applied to the end of the pipe furthest from the nozzle.

2.5 SIMULATION OF TOOL APPLICATION

After establishing the as-welded stresses with ID weld repair and the residual stress distribution after normal plant operations, the mechanical loading of MSIP® was simulated. This is accomplished by applying a pressure along the outer surface of the pipe as shown in Figure 3.

The pressure load is incrementally applied to ensure convergence of the nonlinear solution. After full loading is achieved, the pressure load is removed to simulate the removal of the tool. There is a permanent change to the outside diameter of the pipe when the pressure is applied and removed from the model. All post- MSIP® results are for the case when the tool is removed.

2.6 POST- MSIP® OPERATING LOAD

After the desired MSIP® stress redistribution is achieved, the effect of the plant normal operating conditions on the post- MSIP® residual stress patterns is evaluated. This is accomplished by applying a uniform temperature to the entire model as well as a pressure loading to the inner surfaces of the nozzle assembly and the loading due to blow-off, as described in Section 2.4.

3.0 ANALYSIS RESULTS

The effectiveness of MSIP® application on the nozzle assembly is determined through this analysis [Ref. 1]. As stated previously, a model with representative dimensions is generated followed by application of local ID weld repair residual stresses. Hydrostatic test pressure and operating conditions are then applied to the model. Once the residual stress condition in the nozzle to safe end weld is established, the MSIP® application is then modeled to evaluate the redistribution of stresses in the weld region. The analysis results show that an unfavorable residual tensile stress resulting from ID weld repair exists along the inner region of the nozzle to safe end weld and is converted to a compressive stress field through application of MSIP®.

Pre-MSIP® ISI of the Outlet Nozzle to Safe End Weld 29-RC-1140-1 (#14 Hot Leg nozzle/safe end weld) detected an indication at the buttering interface. The indication was characterized as an ID connected circumferential flaw that was about 2% in circumferential length and 24% through-wall in depth [Ref. 2].

These results also verify that the application of MSIP® would generate compressive stresses such that the flaw would remain in the compressive zone post- MSIP® during operation.

3.1 AS-WELDED STRESSES

The resulting residual stresses from an ID weld repair to the nozzle to safe end are illustrated in Figures 5 and 6. This stress pattern produces conservative high tensile residual stresses at the ID of the material. Figure 7 shows the path along which the through-wall stresses are plotted for the flaw evaluation. Though the indication is a circumferential flaw both the pre-MSIP® hoop and axial through-wall stresses are shown in Figure 8 and tabulated in Table 2.

3.2 POST-MSIP® STRESSES

The post-MSIP® stresses are shown in Figures 9 and 10. While these results correspond to the squeeze applied to the #14 hot leg nozzle/safe-end weld, they are typical for the range specified for the other Salem nozzle welds.

The most important observation to be gained from these figures is the general redistribution of stresses produced by MSIP®. These results clearly show that the large residual tensile stresses along the inner region of the Inconel weld, induced by the welding process, are replaced by compressive stresses. These compressive hoop and axial residual stresses generated by MSIP® will provide mitigation against PWSCC in the nozzle to safe end weld.

The post-MSIP® hoop and axial through-wall stresses including operating pressure and temperature are shown in Figure 11 and tabulated in Table 3. These results confirm that the identified flaw will remain in the compressive zone at operation.

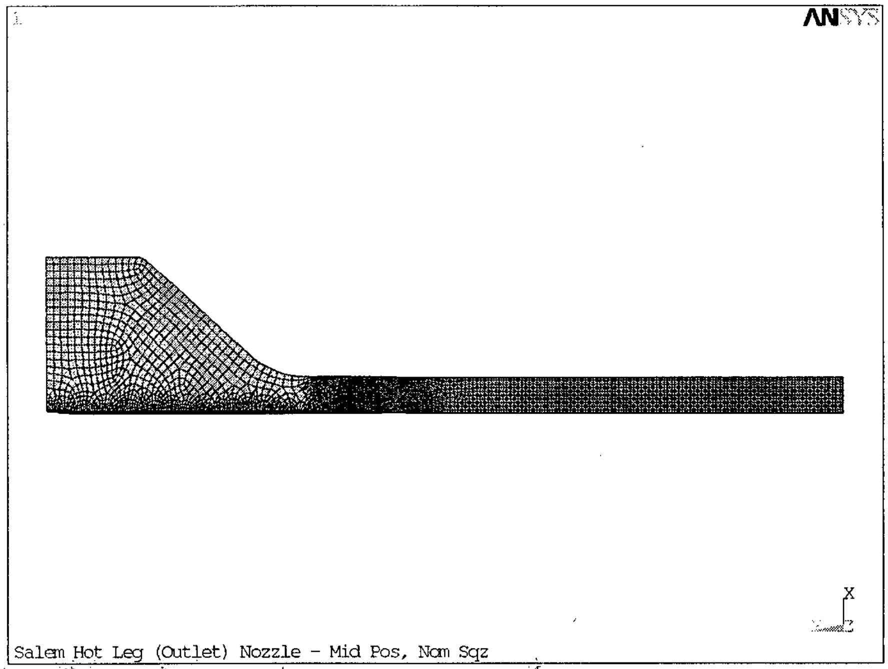

Figure 1 – Finite Element Model

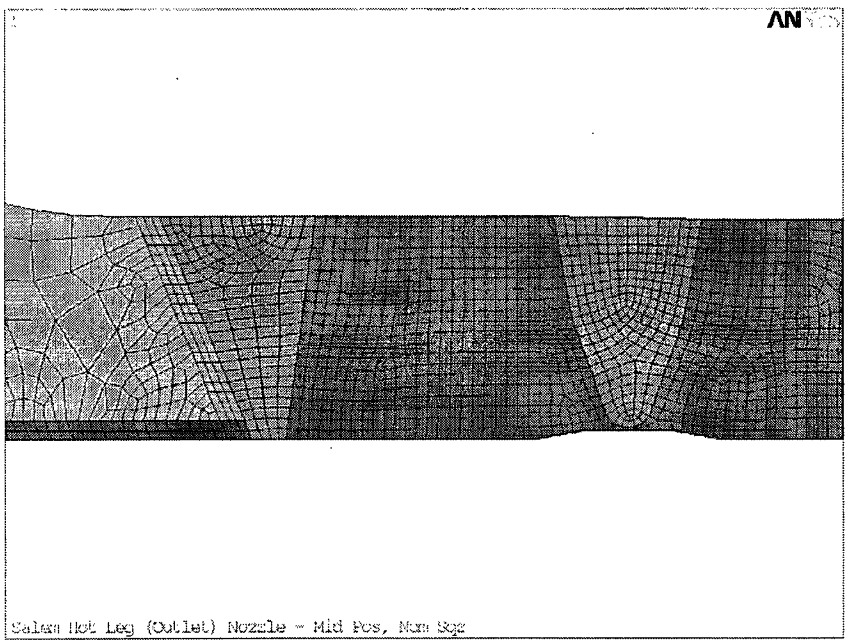

Figure 2 – Finite Element Model – Weld Detail

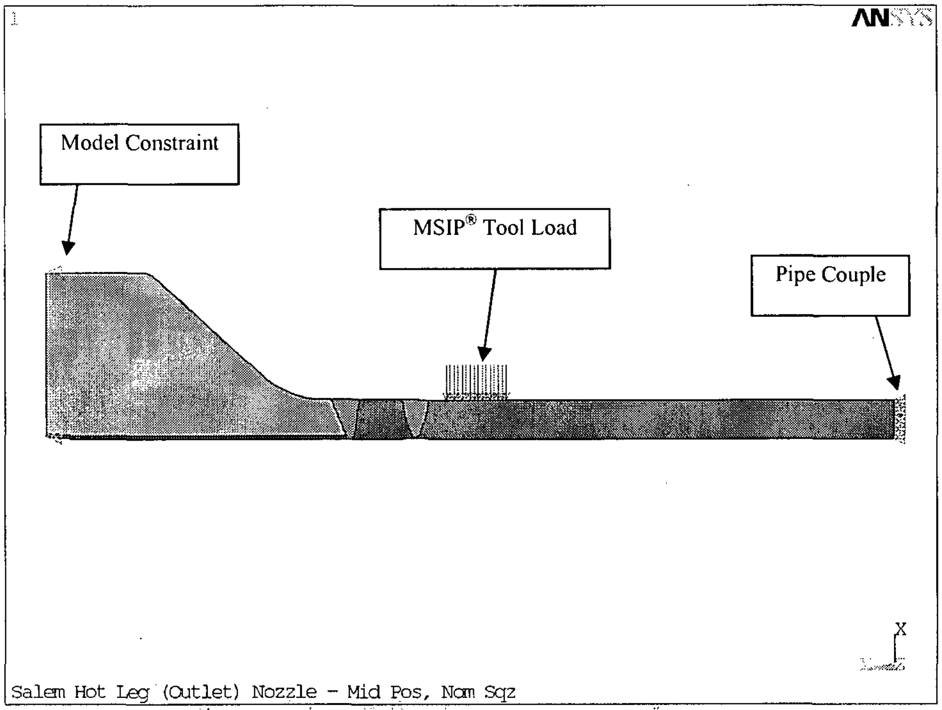

Figure 3 - Finite Element Model – Boundary Conditions w/ Tool Pressure Loading

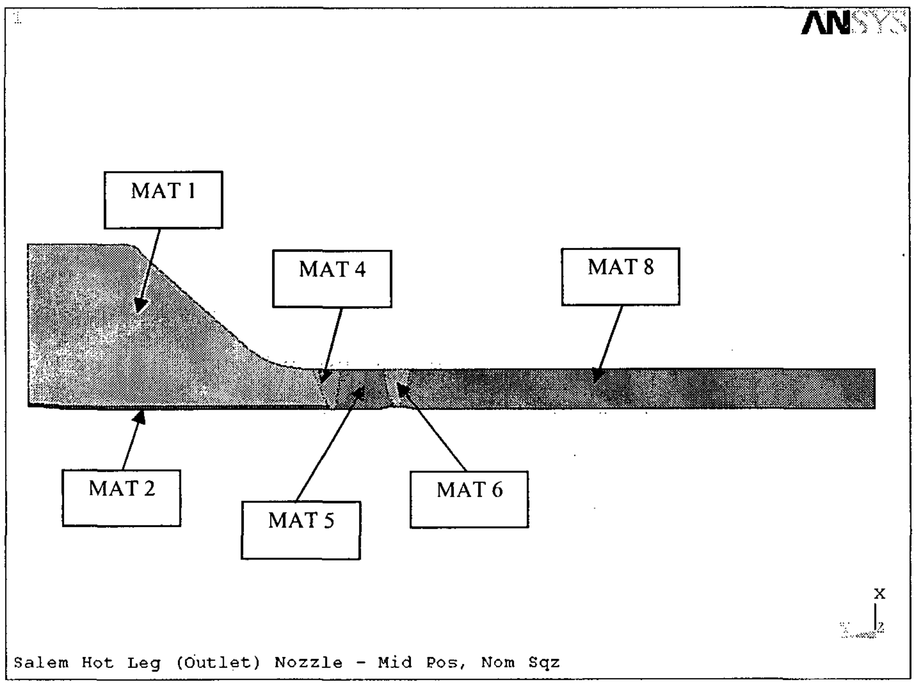

Figure 4 – Finite Element Model – ANSYS Material Number

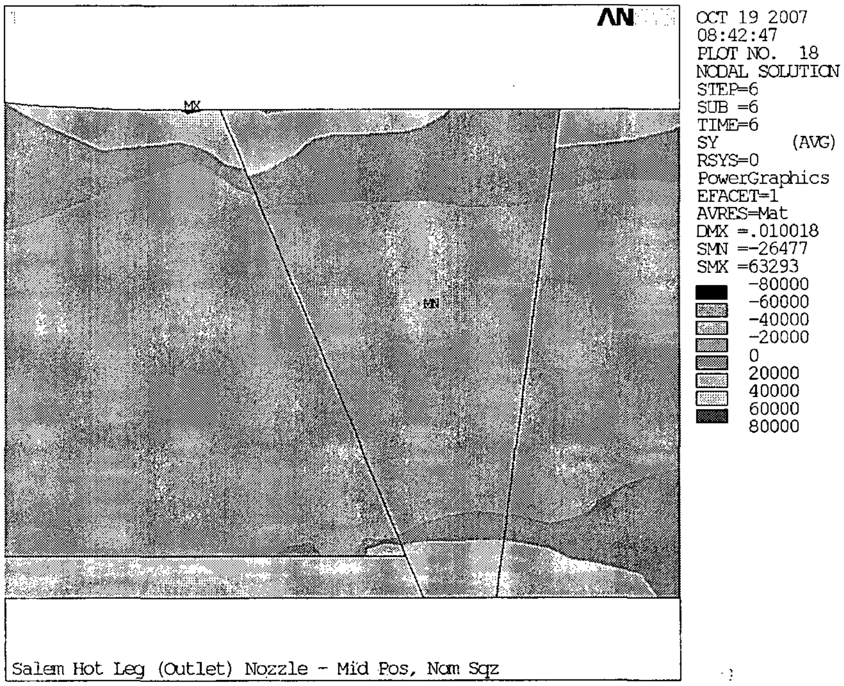

Figure 5 – NSE Weld – As-Designed Weld w/Local ID Weld Repair Residual Axial Stress

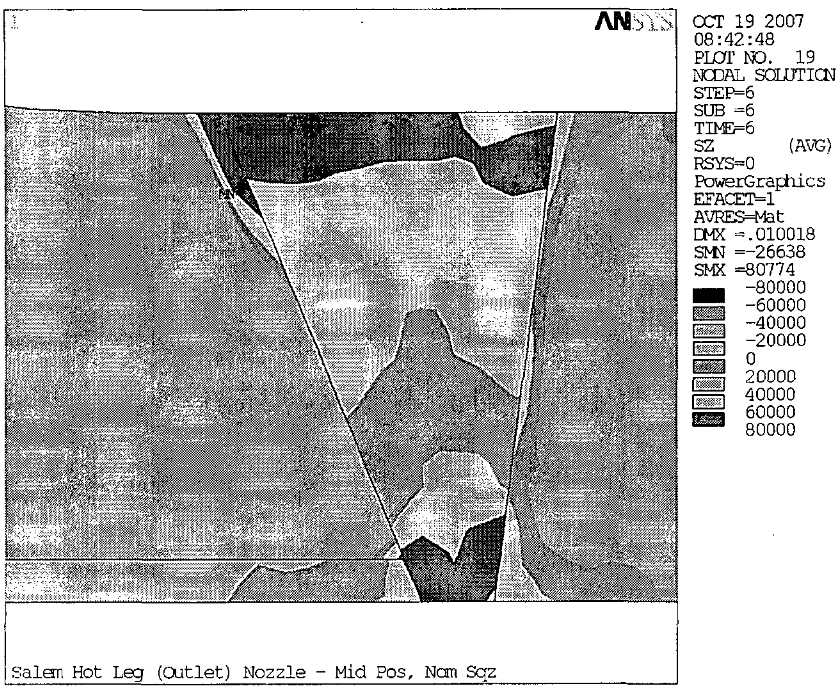

Figure 6 – NSE Weld – As-Designed Weld w/Local ID Weld Repair Residual Hoop Stress

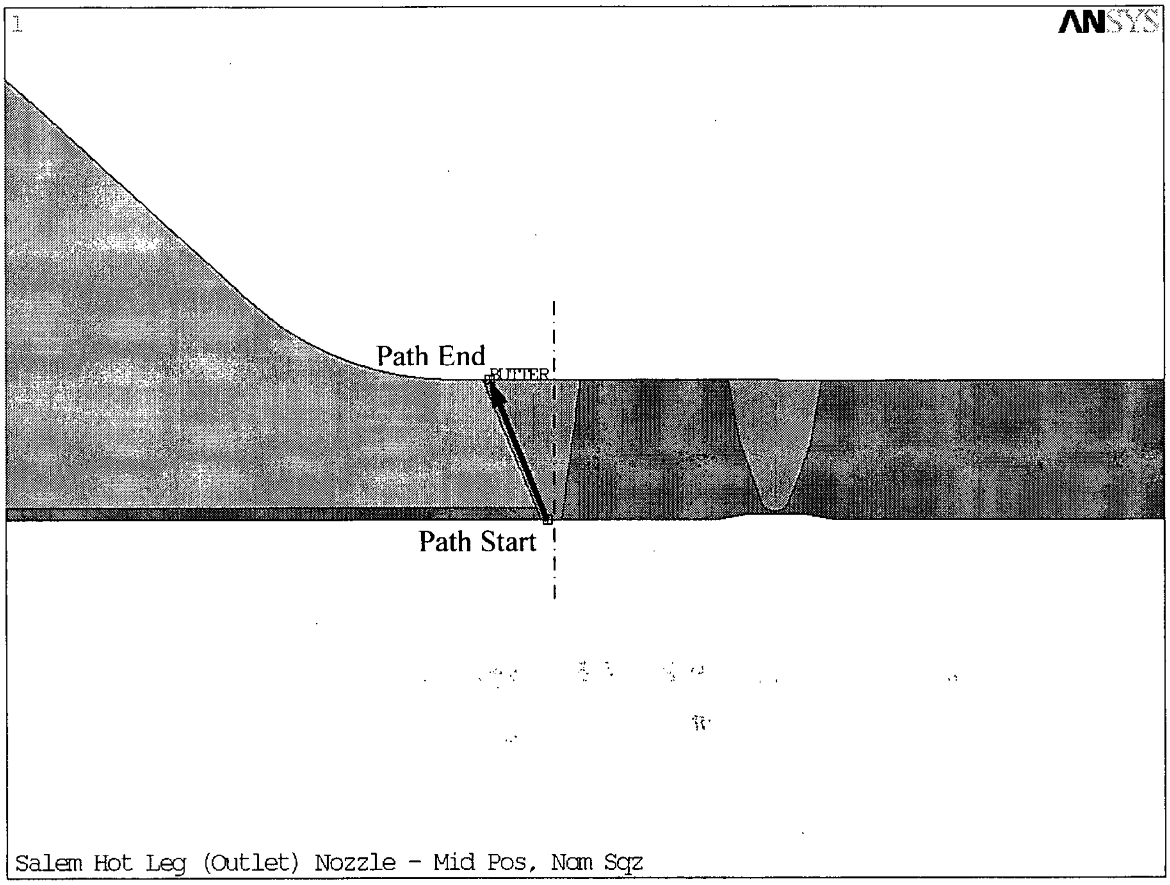

Figure 7 – Salem Hot Leg, Path Along Flaw Location

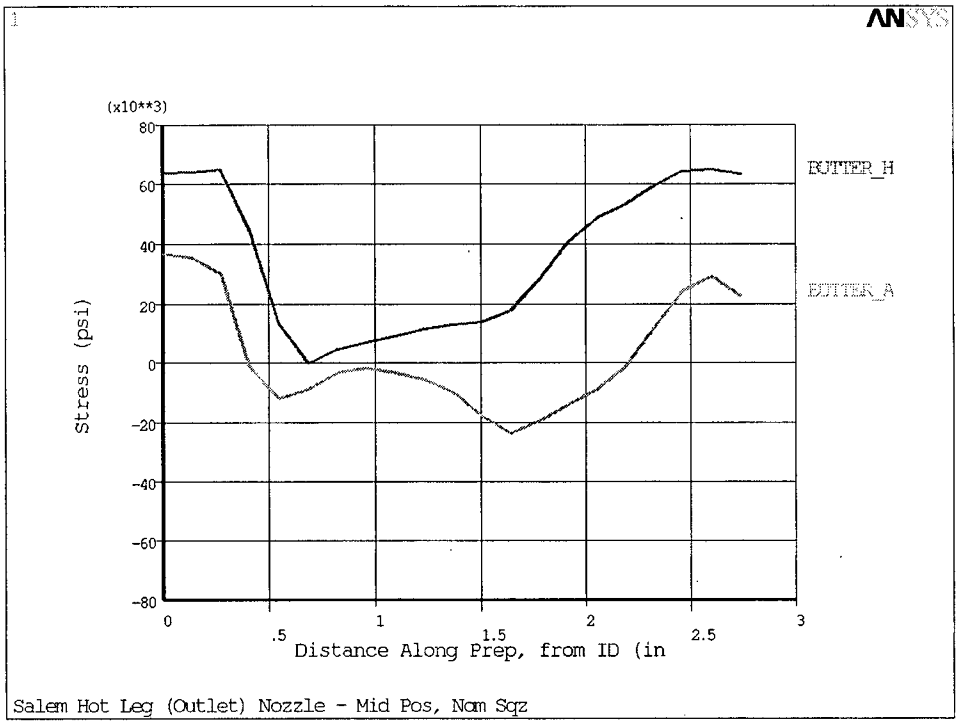

Figure 8 – Salem Hot Leg, Stress Along Flaw Path, PRE MSIP

Table 2 – Salem Hot Leg, Stress Along Flaw Path, PRE MSIP

Distance (in)	PRE MSIP Axial (psi)	PRE MSIP Hoop (psi)
0.000	36,840	63,680
0.137	35,507	63,979
0.274	30,249	64,860
0.411	-1,445	44,486
0.548	-11,735	13,586
0.685	-8,893	-100
0.822	-3,263	4,780
0.958	-1,772	6,932
1.095	-3,295	9,060
1.232	-5,512	11,412
1.369	-9,763	13,120
1.506	-17,675	13,978
1.643	-23,517	17,840
1.780	-19,326	28,342
1.917	-13,924	40,834
2.054	-8,911	48,676
2.191	-1,337	53,315
2.328	11,938	59,630
2.465	24,391	64,479
2.601	29,302	65,009
2.738	22,724	63,573

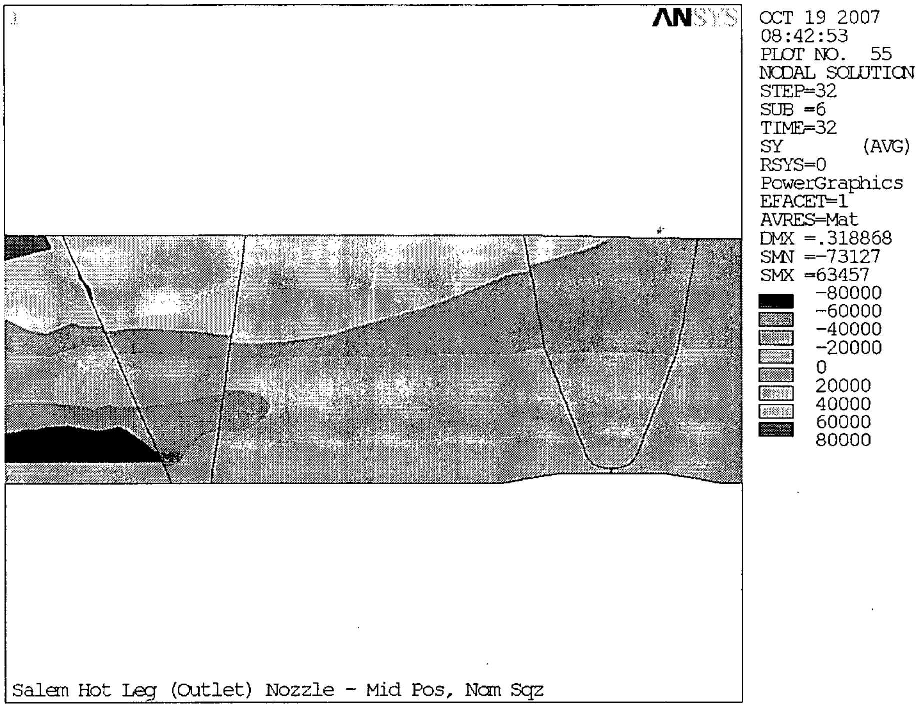

Figure 9 – Post MSIP® Residual Axial Stress – Nom. Position

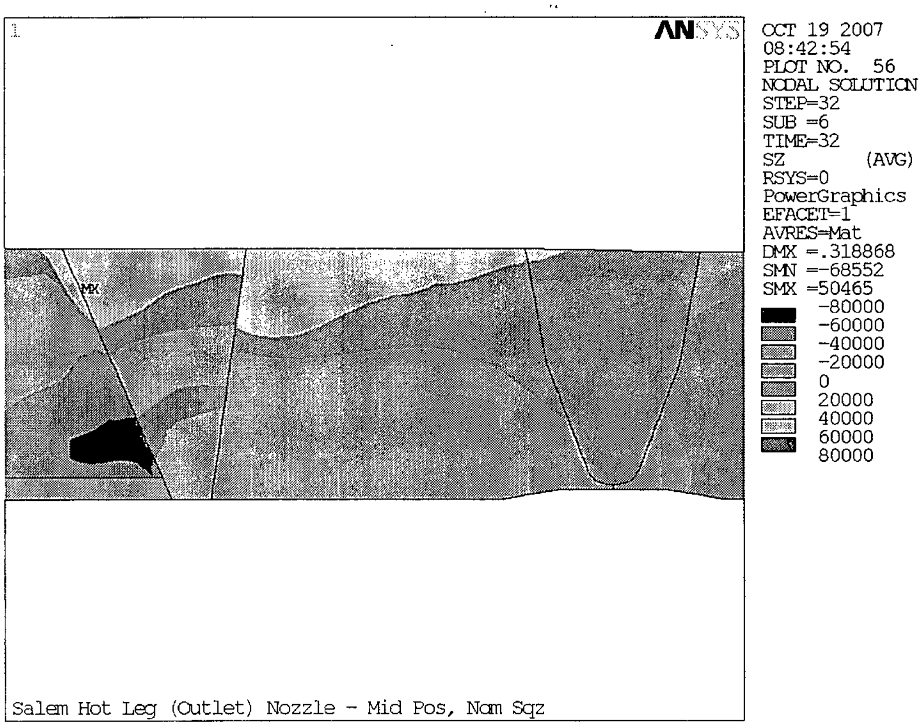

Figure 10 – Post MSIP® Residual Hoop Stress – Nom. Position

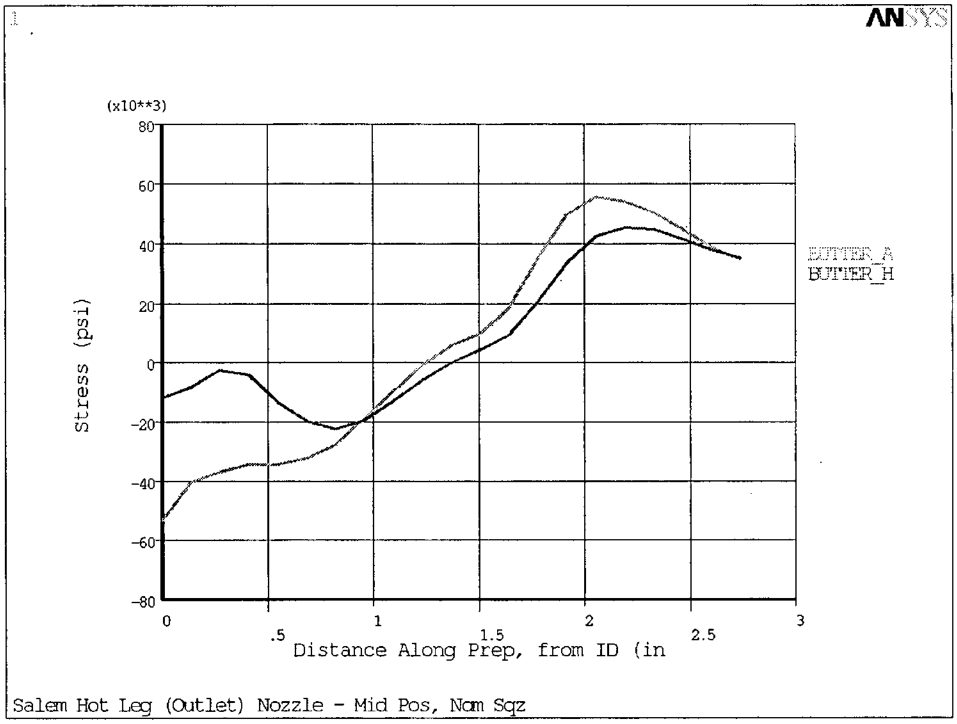

Figure 11 – Salem Hot Leg, Stress Along Flaw Path, POST MSIP w/ Operating Pressure and Temperature

Table 3 – Salem Hot Leg, Stress Along Flaw Path, POST MSIP w/ Operating Pressure and Temperature

Distance (in)	POST MSIP w/ Operating Axial (psi)	Hoop (psi)
0.000	-53,318	-11,915
0.137	-40,423	-8,181
0.274	-36,902	-2,443
0.411	-34,400	-4,026
0.548	-34,258	-13,268
0.685	-32,135	-19,569
0.822	-27,644	-22,323
0.958	-18,625	-19,284
1.095	-9,468	-12,683
1.232	-815	-5,942
1.369	5,843	20
1.506	9,730	4,405
1.643	18,503	9,269
1.780	35,023	20,609
1.917	49,771	33,688
2.054	55,587	42,425
2.191	54,204	45,436
2.328	50,385	44,880
2.465	44,842	41,465
2.601	38,896	37,931
2.738	34,774	35,286

4.0 REFERENCES

1. Analytical Verification of MSIP® for RV Hot Leg Nozzle to Safe End Weld, Salem Unit 1 & 2, NuVision Engineering, Inc. Report 4418-4-002-00

2. WESDYNE Letter, WDI-PJF-1303981-TR-010, Rev. 0, October 26, 2008