Exelon Generation

200 Exelon Way
Kennett Square, PA 19348
www.exeloncorp.com

PROPRIETARY INFORMATION – WITHHOLD UNDER 10 CFR 2.390

10 CFR 50.55a

TMI-13-038
March 11, 2013

U.S. Nuclear Regulatory Commission
Attn: Document Control Desk
Washington, DC 20555-0001

Three Mile Island Nuclear Station, Unit 1
Renewed Facility Operating License No. DPR-50
NRC Docket No. 50-289

Subject: Relief Request RR-12-02 Concerning the Installation of a Full Structural Weld Overlay on the Lower Cold Leg Letdown Nozzle Dissimilar Metal Welds and Alloy 600 Safe-End

References:

1) Letter from M. Jesse (Exelon Generation Company, LLC) to U.S. Nuclear Regulatory Commission, "Submittal of Relief Request RR-12-02 Concerning the Installation of a Full Structural Weld Overlay on the Lower Cold Leg Letdown Nozzle Dissimilar Metal Welds and Alloy 600 Safe-End," dated October 18, 2012

2) Letter from P. Bamford (U.S. Nuclear Regulatory Commission) to M. Pacilio (Exelon Generation Company, LLC), "Three Mile Island Nuclear Station, Unit 1 – Request for Additional Information Regarding Relief Request RR-12-02, Relief Request Concerning Full Structural Weld Overlay of Dissimilar Metal Welds on the Lower Cold Leg Letdown Nozzle and Safe-End (TAC No. ME9818)," dated December 14, 2012

3) Letter from M. Jesse (Exelon Generation Company, LLC) to U.S. Nuclear Regulatory Commission, "Response to Request for Additional Information - Relief Request RR-12-02 Concerning the Installation of a Full Structural Weld Overlay on the Lower Cold Leg Letdown Nozzle Dissimilar Metal Welds and Alloy 600 Safe-End," dated January 17, 2013

In the Reference 1 letter, Exelon Generation Company, LLC proposed an alternative to the requirements contained in the American Society of Mechanical Engineers (ASME) Boiler and Pressure Vessel (B&PV) Code associated with the fourth Inservice Inspection (ISI) interval for Three Mile Island Nuclear Station (TMI), Unit 1. TMI, Unit 1 is proposing to perform a weld overlay of the lower cold leg letdown nozzle dissimilar metal welds (DMWs) and Alloy 600 safe-end. In the Reference 2 letter, the U.S. Nuclear Regulatory Commission Staff requested additional information. Reference 3 contained our response.

Attachment 1 transmitted herewith contains Proprietary Information.
When separated from Attachment 1, this document is decontrolled.

Upon further review, three (3) calculations submitted in Reference 3 are being resubmitted with updated proprietary markings. We note that one calculation ("TMI Unit 1 CL Letdown Nozzle DMW and Safe End Crack Growth Analysis" (proprietary (32-9186194-001) and non-proprietary (32-9196234-000) versions)) has not changed.

Attachment 1 contains the AREVA NP Inc. (AREVA) proprietary calculations. AREVA requests that the calculations be withheld from public disclosure in accordance with 10 CFR 2.390. Attachment 2 contains a non-proprietary version of the calculations. An affidavit supporting this request is contained in Attachment 3.

If you have any questions concerning this letter, please contact Tom Loomis at (610) 765-5510.

Respectfully,

D. P. Helker

David P. Helker
Manager - Licensing and Regulatory Affairs
Exelon Generation Company, LLC

Attachments: 1) Proprietary Version of Calculations
2) Non-Proprietary Version of Calculations
3) Affidavit

cc: Regional Administrator, Region I, USNRC
USNRC Senior Resident Inspector, TMI
USNRC Project Manager, [TMI] USNRC

Attachment 2

Non-Proprietary Version of Calculations

TMI Unit 1 Weld Residual Stress Analysis for CL Letdown Nozzle Weld Overlay
TMI-1 Letdown Nozzle Weld Overlay Sizing Calculation
TMI-1 Letdown Nozzle Weld Overlay Section III Analysis

AREVA

CALCULATION SUMMARY SHEET (CSS)

Document No. 32 - 9196236 - 001 Safety Related: ☒ Yes ☐ No

Title TMI Unit 1 Weld Residual Stress Analysis for CL Letdown Nozzle Weld Overlay – Non Proprietary

PURPOSE AND SUMMARY OF RESULTS:

AREVA NP Inc. proprietary information in the document are removed and their locations are indicated by pairs of braces “[ ]”. This document is the non-proprietary version of AREVA Document 32-9186192-002.

The purpose of this report is to document the results of weld residual stress finite element analysis of the Cold leg (CL) Letdown Nozzle Dissimilar Metal Welds (DMW) and Structural Weld Overlay (SWOL) at the Three Mile Island Unit 1 (TMI-1) Nuclear Power Plant. The analysis includes simulation of the existing DM Weld attaching the Letdown Nozzle to Safe end, DMW attaching Safe end to Elbow and the proposed Weld Overlay mitigation of the DM welds. The analysis also includes simulation of worst case repair welds of DMW that would have been performed. The state of stress after welding and operating (heat up/cool down) cycles as predicted by the ANSYS Version 13.0 finite element analysis, are summarized to support flaw evaluations of the DM welds.

The purpose of Revision 001 is to mark two additional instances of material descriptions as proprietary.

[redacted]

THE FOLLOWING COMPUTER CODES HAVE BEEN USED IN THIS DOCUMENT:

CODE/VERSION/REV	CODE/VERSION/REV
ANSYS 13.0 SP2	

THE DOCUMENT CONTAINS ASSUMPTIONS THAT SHALL BE VERIFIED PRIOR TO USE

☐ YES

☒ NO

TMI Unit 1 Weld Residual Stress Analysis for CL Letdown Nozzle Weld Overlay – Non Proprietary

Review Method: ☒ Design Review (Detailed Check)
☐ Alternate Calculation

Signature Block

Name and Title (printed or typed)	Signature	P/R/A and LP/LR	Date	Pages/Sections Prepared/Reviewed/Approved
Silvester Noronha Engineer IV	SJNoronha	P	2/19/13	All
Doug Killian Technical Consultant	DKillian	R	2/19/13	All
Tim Wiger Unit Manger	T.M. Wiger	A	2/20/13	All

Note: P/R/A designates Preparer (P), Reviewer (R), Approver (A);
LP/LR designates Lead Preparer (LP), Lead Reviewer (LR)

Project Manager Approval of Customer References (N/A if not applicable)

Name (printed or typed)	Title (printed or typed)	Signature	Date
N/A			

Mentoring Information (not required per 0402-01)

Name (printed or typed)	Title (printed or typed)	Mentor to: (P/R)	Signature	Date
N/A				

Record of Revision

Revision No.	Pages/Sections/Paragraphs Changed	Brief Description / Change Authorization
000	All	Original release
001	CSS, pages 1-3	Updated
	Sec 3.2.3, page 16	Marked two material descriptions as proprietary

Table of Contents

Page

Table of Contents
(continued)

List of Tables

List of Figures

1.0 INTRODUCTION

Primary water stress corrosion cracking (PWSCC) of Alloy 600/82/182 materials is a well-documented phenomenon in the nuclear power industry. Components have risk for PWSCC at the dissimilar metal welds (DMWs). The risk due to PWSCC increases with service time.

AREVA plans to mitigate the Three Mile Island (TMI-1) cold leg (CL) letdown nozzle Alloy 600/82/182 safe end and DMWs with a full structural weld overlay (FSWOL) during the T1R20 refueling outage in the fall of 2013. The planned modification using a FSWOL is a preemptive measure to reduce susceptibility of the DMW to PWSCC and to enhance the configuration such that improved coverage using ultrasonic examination of the nozzle to safe end DMW and the adjacent elbow to safe end weld DMW is accomplished.

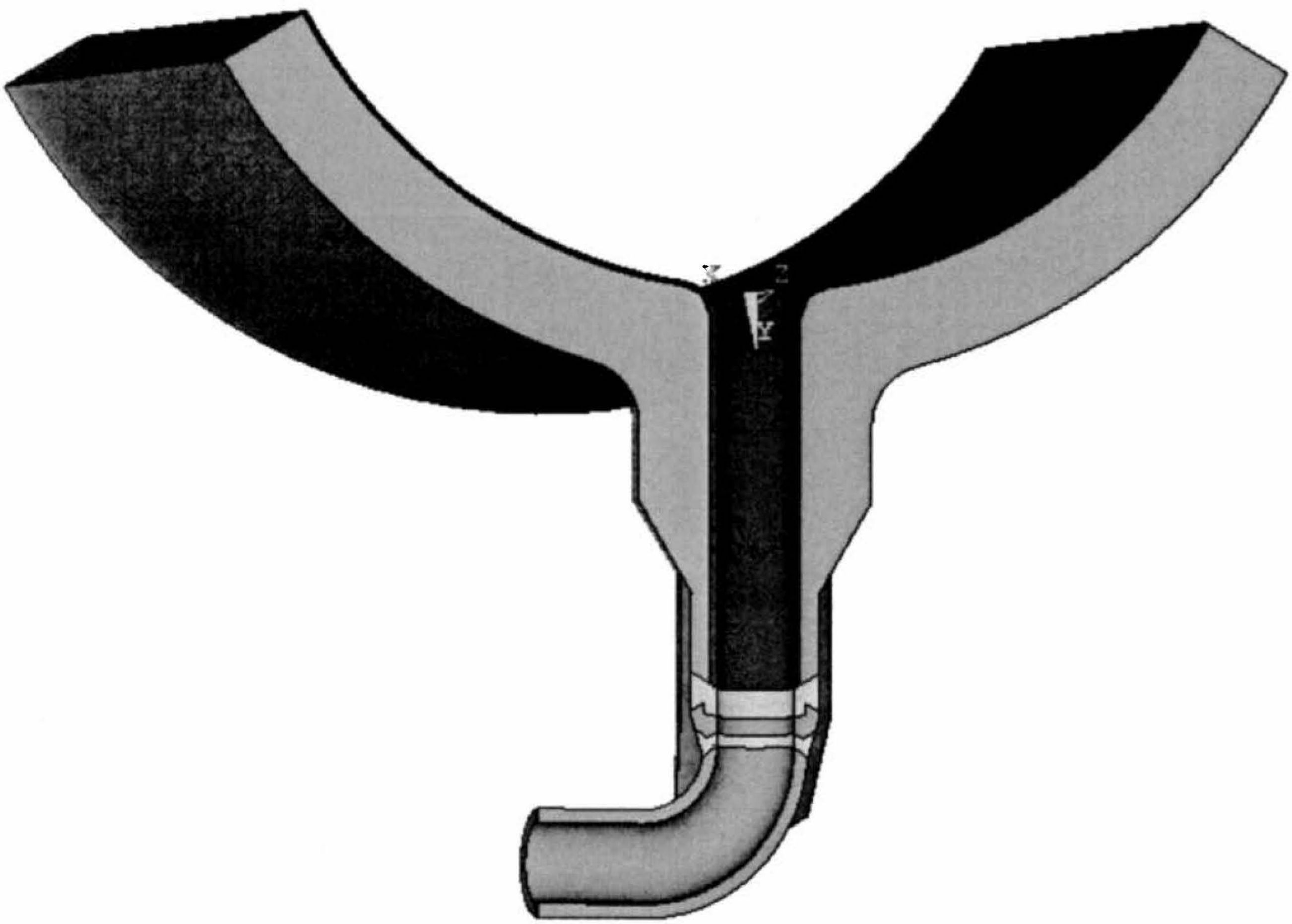
2.0 PURPOSE AND SCOPE

The purpose of this document is to report results of the weld residual stress finite element analysis of the Cold leg (CL) Letdown Nozzle Dissimilar Metal Welds (DMW) and Weld OverLay (WOL) at the Three Mile Island Unit 1 (TMI-1) Nuclear Power Plant. This analysis includes simulation of the existing DMWs attaching the safe end to the Letdown nozzle and the pipe-elbow as well as repairs to these DMWs. The proposed FSWOL is also simulated. The state of stress after welding and operating (heat up/cool down) cycles as predicted by the ANSYS Version 13.0 finite element analysis, are provided in this report to support fracture mechanics evaluation of postulated flaws in degraded DMWs and safe end.

3.0 ANALYTICAL METHODOLOGY

The analytical methodology used to predict the weld induced residual stresses in the DMWs and WOL involves three-dimensional finite element analysis. Due to the symmetric nature of the cold leg, pipe/elbow and WOL, a half symmetric model is used to represent the geometry of interest. The half symmetric model used to represent the letdown nozzle with FSWOL is shown in Figure 3-1. The following subsections discusses the modeling and methodology used in the welding simulations performed in this document.

Figure 3-1: CL Letdown Nozzle FSWOL Model



3.1 Welding Analysis Methodology

The WRS (Weld Residual Stress) finite element analysis is carried out per the WRS analysis procedure [1]. Due the symmetric nature of the model, a half-symmetric model was used in the analysis. The various stages of the welding processes for the structural components, including the Alloy 82/182 butt-welds and repair welds; and Alloy 52M Structural Weld Overlay (SWOL) are simulated using a 3-dimensional finite element model with the following sequential steps:

1. Simulate the Dissimilar Metal butt-weld joining the safe end to the letdown nozzle using Alloy 82/182 weld metal by activating the elbow and sequentially adding the weld passes.

2. Simulate the ID repair of the above weld by removing material and adding passes sequentially.

3. Simulate the Dissimilar Metal butt-weld joining the Alloy 600 safe end to the stainless steel elbow.

4. Simulate the repair weld by deactivating the repair weld volume and adding repair weld passes sequentially.

5. Simulate hydro-static testing by applying a static load step of [ ] psig and [ ] °F at the wetted surface with corresponding endcap pressures at the pipe ends.

6. Simulate three operating condition cycles by applying the steady state temperature and pressure ([ ] °F and [ ] psig [2]) as a static load step. Each operating cycle starts from ambient conditions (zero pressure and room temperature), applies steady state pressure and temperature conditions, and then returns to ambient conditions.

7. Simulate the Alloy 52M weld overlay by sequentially adding weld passes layer by layer.

8. Simulate three operating condition cycles by applying the steady state temperature and pressure ([ ] °F and [ ] psig [2]) as a static load step. Each operating cycle starts from ambient conditions (zero pressure and room temperature), applies steady state pressure and temperature conditions, and then returns to ambient conditions.

As explained above this simulation follow the sequential steps that consist of building the original geometry of the Letdown down nozzle DMWs including the original repairs and the SWOL buildup. The key steps of the welding simulations, illustrated with the finite element model, are shown in Figure 3-2 through Figure 3-4.

Figure 3-2: Welding of the DM Welds Attaching Nozzle to Safe end and Safe end to Elbow

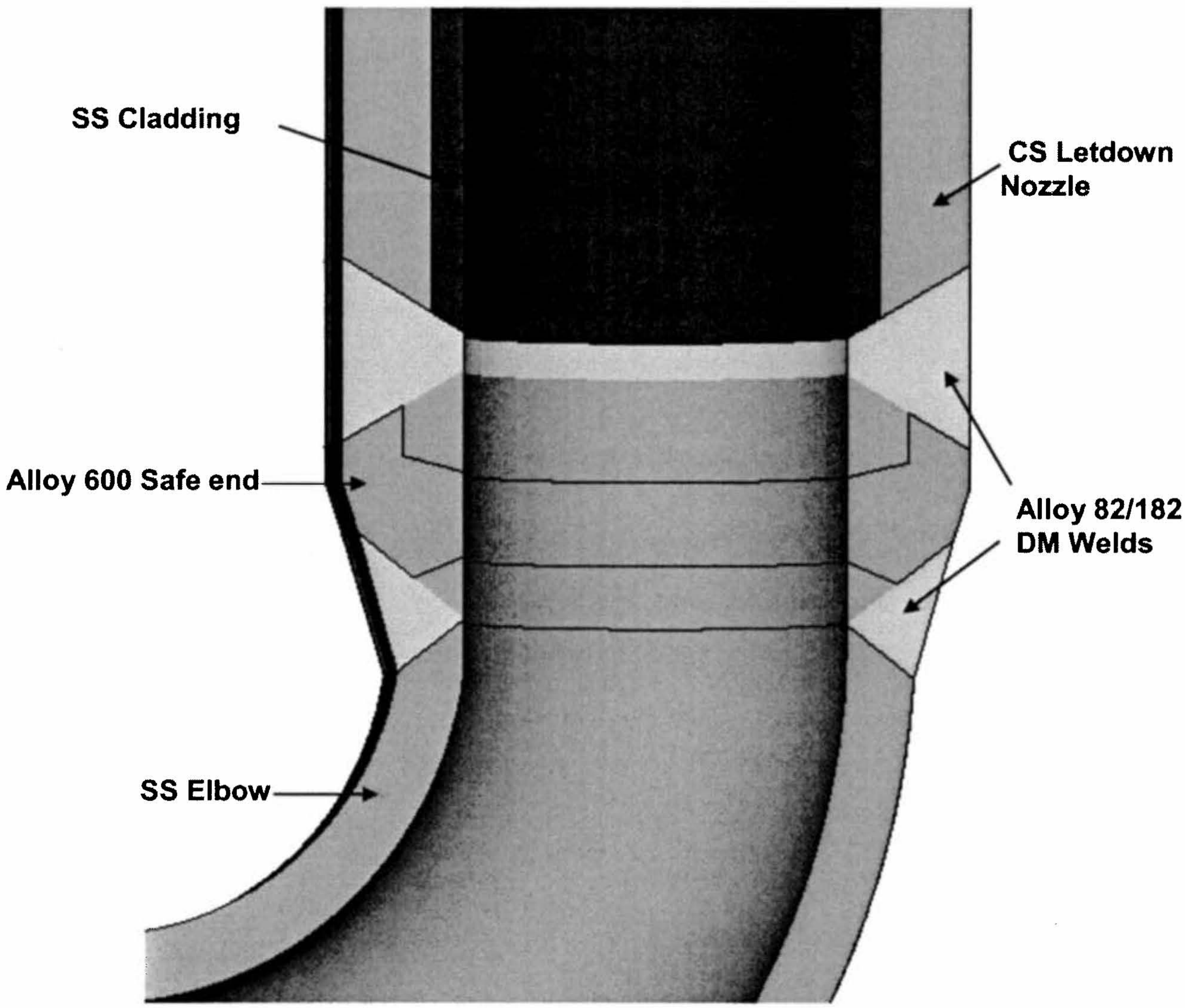


Figure 3-3: ID Repair welds extending 50% of the original DM Weld

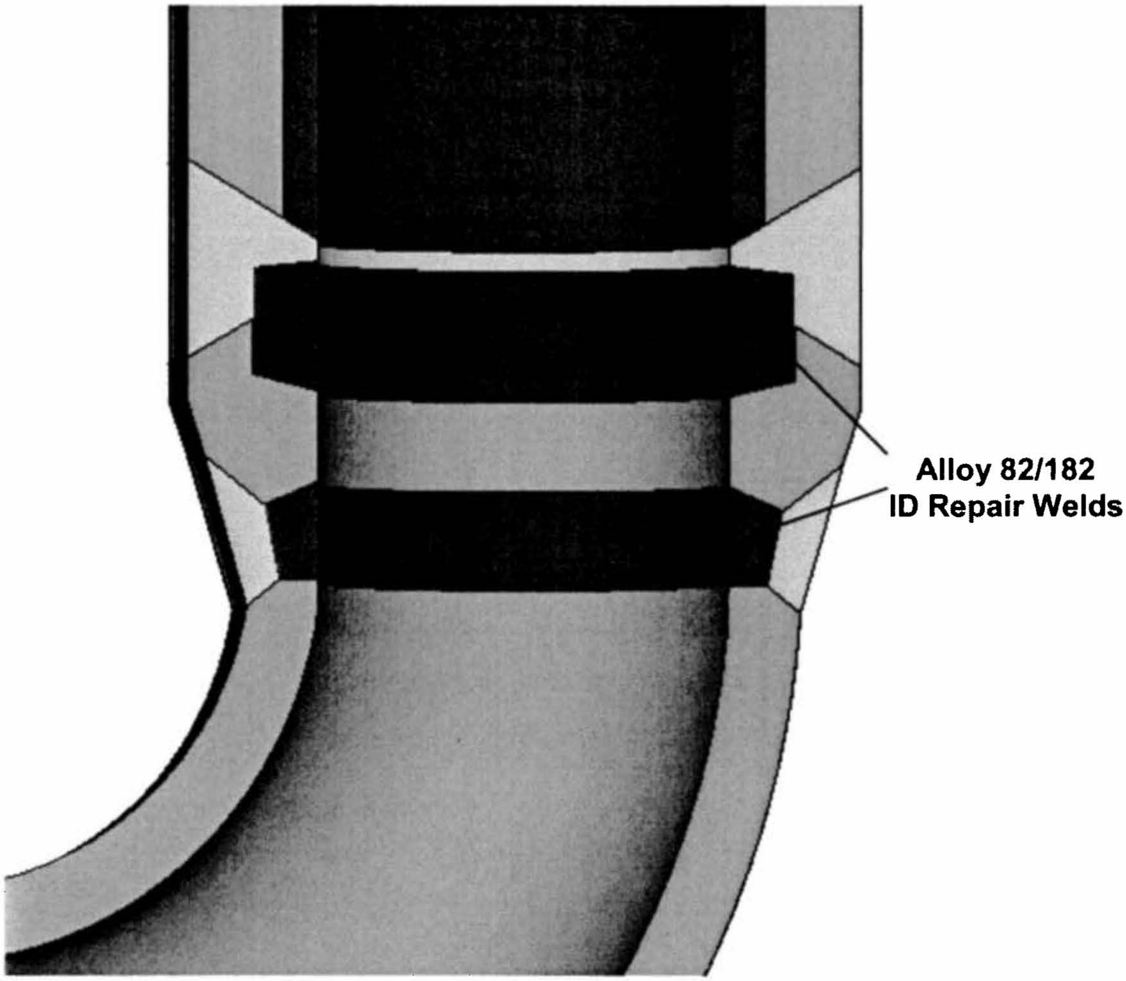
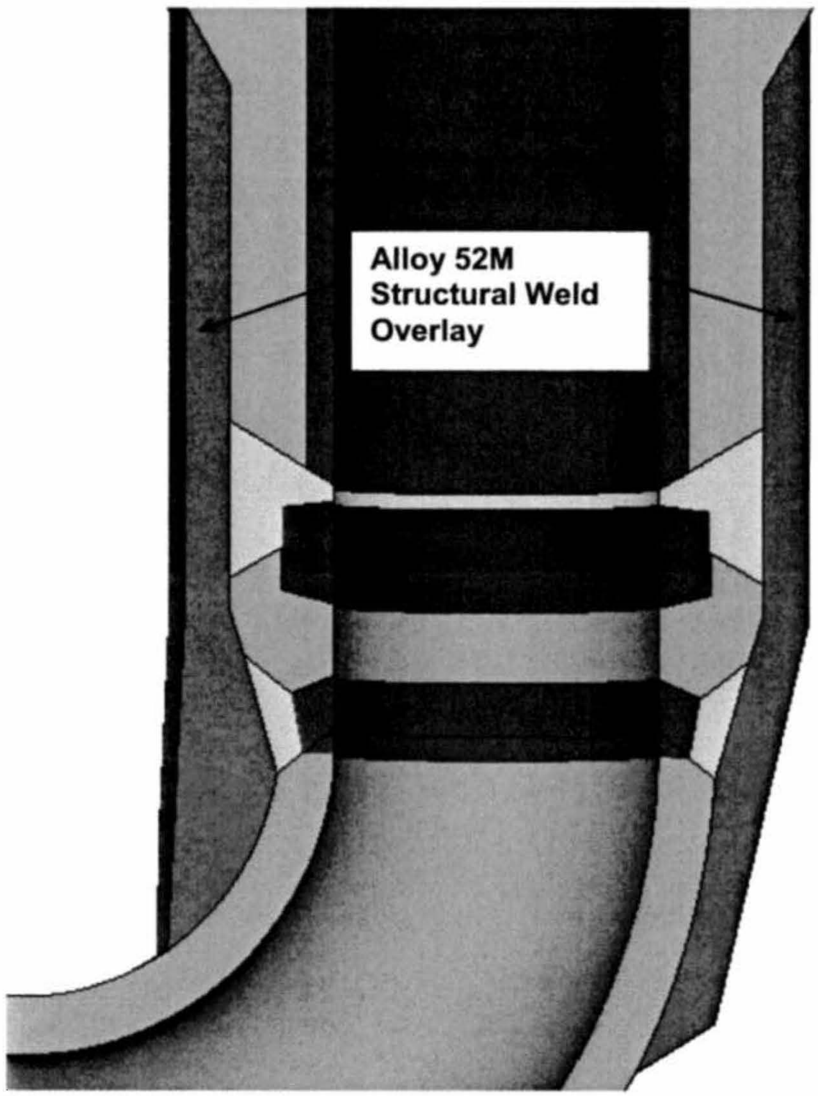


Figure 3-4: Full Structural Weld Overlay covering the DM Welds and the Safe end



The general purpose finite element code ANSYS [3] is used to perform the WRS finite element analysis. The finite element analysis is based on a 3-dimensional half-symmetric model. The basic steps comprising the multi-pass welding simulation of the DM welds, Repair welds and the Structural Weld overlay are as follows:

1. Develop the finite element model with the features necessary to accommodate weld pass deposition of the DM welds, repair welds and SWOL.

2. Define the temperature range for melting (solidus and liquidus temperatures).

3. Define thermal and mechanical temperature dependent material properties from ambient conditions (70°F) up to and including the melting region.

4. Define thermal and structural boundary conditions.

5. Define volumetric heat sources from welding procedure specifications, if available.

6. Simulate the thermal phase of the welding process using the ANSYS "birth and death" feature

   - Deactivate finite elements in all weld passes.

   - Activate finite elements in one weld pass at a time and perform transient thermal analysis to develop the history of the temperature field for subsequent structural analysis.

7. Simulate the structural phase of the welding process using the ANSYS "birth and death" feature

   - Deactivate finite elements in all weld passes.

   - Activate finite elements in one weld pass at a time and perform static structural elastic-plastic analysis using the temperature history from the thermal phase.

Static load steps are applied to simulate hydrostatic testing after the simulation of the DM welds and repair welding. Also, load steps are applied to simulate steady state operating conditions.

On completing the structural weld overlay simulation, static load steps to simulate the steady state operating conditions are applied again.

3.2 Design inputs

3.2.1 Geometry

The detailed dimensions of the CL Letdown nozzle and SWOL modeled in the WRS finite element analysis are obtained from References [4] and [5] . The key dimensions are shown in Table 3-1.

Table 3-1: Letdown Nozzle / Cold leg Dimensions

Dimension	Value
Letdown Nozzle ID	
Letdown Nozzle OD at DMW	
Cladding Thickness (Nominal) at Nozzle	
Cold leg ID	
Cold leg OD	
Cladding Thickness (Minimum) at Cold leg	

3.2.2 Finite Element Model

The finite element model is a three-dimensional half-symmetric model, as shown in Figure 3-5. The finite element mesh consists of ANSYS 8-noded thermal (SOLID70) and structural (SOLID185) elements. The weld pass depositions for the DM welds, repair welds and the SWOL are simulated using ANSYS's element "birth and death" feature. The thermal finite element model is documented in File "Thermal_Model.db" and the stress finite element model is documented in File "Stress_Model.db". Both files are archived as listed in Table 5-1.

The finite element mesh for the letdown nozzle and weld overlay are shown in Figure 3-6 through Figure 3-10. The dimensions of the letdown nozzle weld overlay finite element model are developed per References [4] and [5]. The weld passes employed in the dissimilar metal weld and repair weld simulations are based on the information in Reference [6]. The SWOL weld passes are based on information in References [7] and [8].

3.2.3 Material

Reference [9] provides the material designation of the components modeled in the WRS analysis.

Table 3-2: Component Material Designation

Component	Material Designation
Cold leg	
Letdown Nozzle	
Cladding	
DM Welds and Repairs	
Safe end	
Elbow	
First layer of WOL over SS Elbow until within 3/16 in. of outboard edge of DMW†	
Layer at the interface of SS Elbow and DMW attaching Elbow to Safe end†	
SWOL	

†Structural credit is not taken for this layer

The analysis herein uses the physical properties (thermal conductivity, specific heat, mean coefficient of thermal expansion, density, Young's modulus, and Poisson's ratio) and the stress-strain curves from Reference [10] that are representative of the materials listed in Table 3-2. For the letdown nozzle material [ ] that is not directly available in Reference [10], the material properties of [ ] are used, since the material properties for both materials were comparable [11] . All of the physical and mechanical properties, except the Poisson's ratio, are temperature dependent.

The multi-linear kinematic hardening model in ANSYS [3] is employed in this elastic-plastic structural analysis. Temperature dependent, true stress-strain material properties are used with the multi-linear kinematic hardening model for simulating the structural phase of the welding procedure and the operating transients.

Figure 3-5: Symmetry Planes

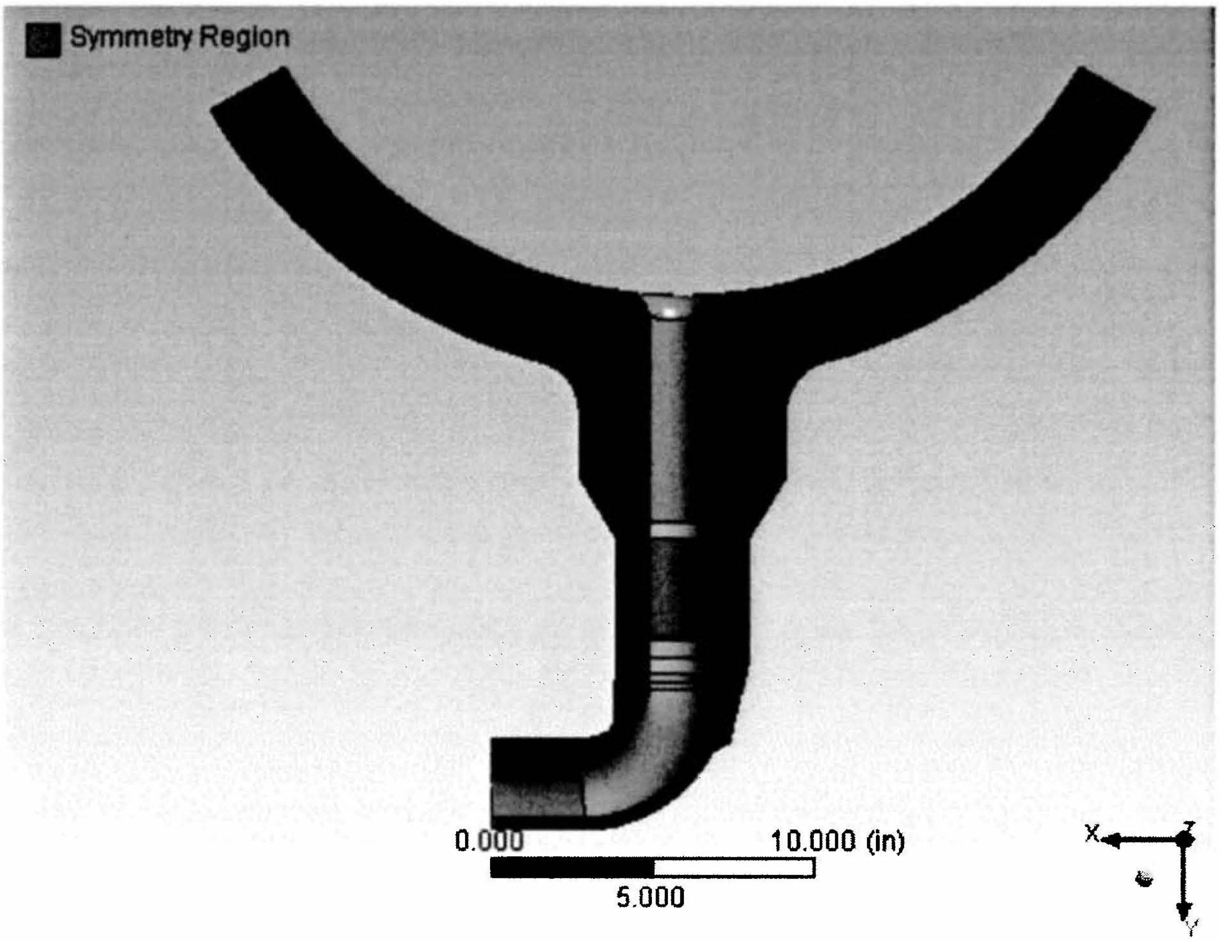


Figure 3-6: Finite Element Mesh

(a) Overall Mesh for Half-Symmetric 3D Model

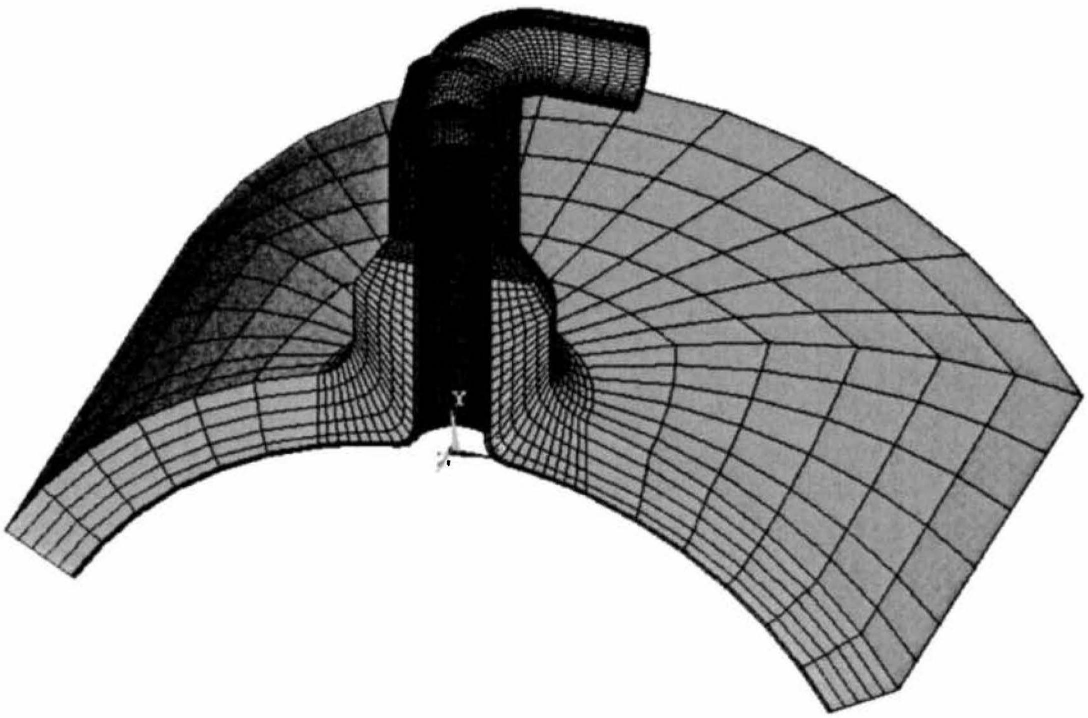


Figure 3-7: Detailed mesh showing DM Welds

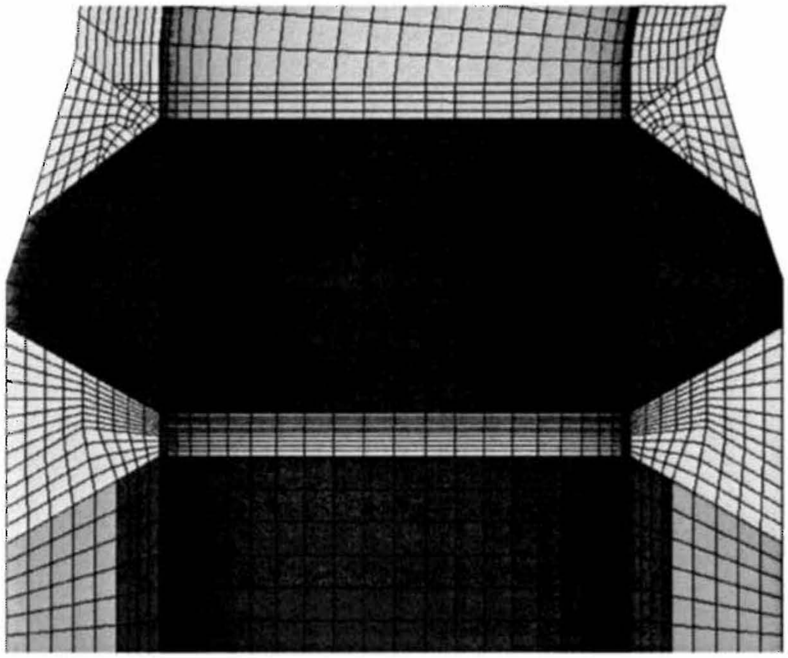


Figure 3-8: Detailed mesh showing Repair Welds

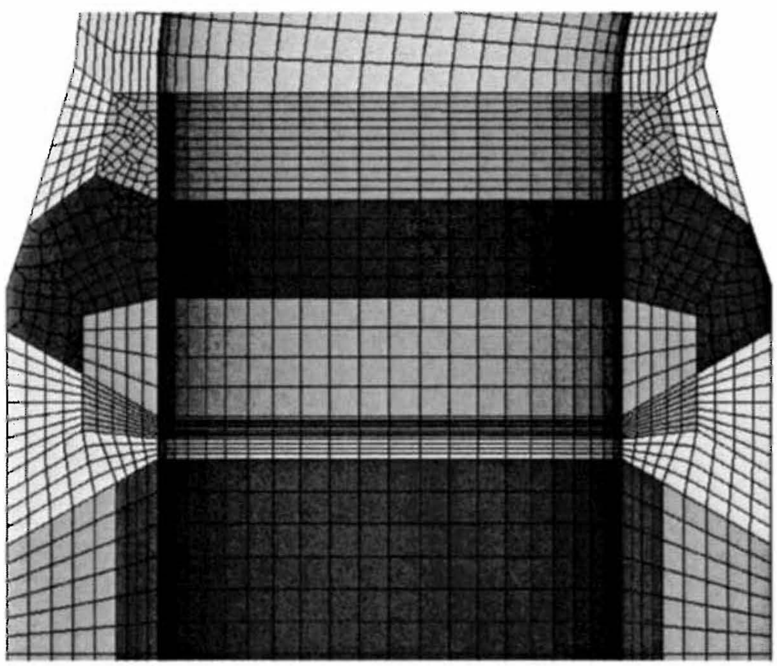


Figure 3-9: Detailed mesh showing Weld Overlay

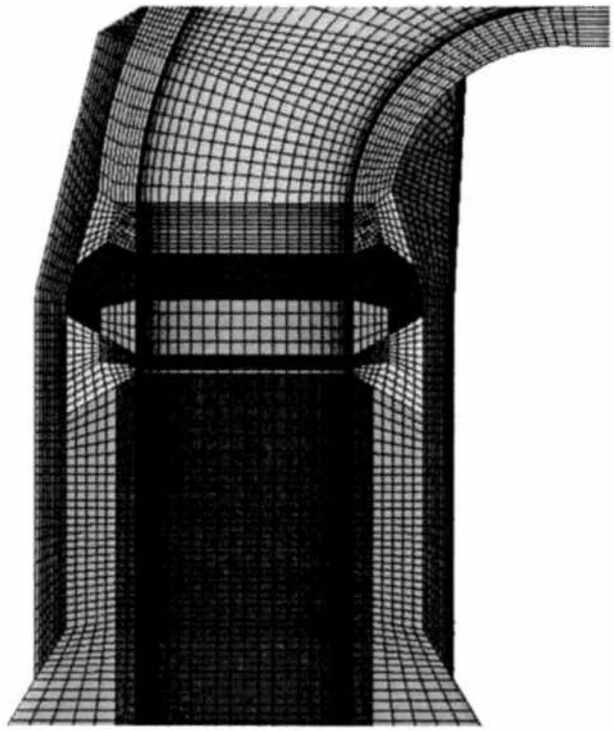
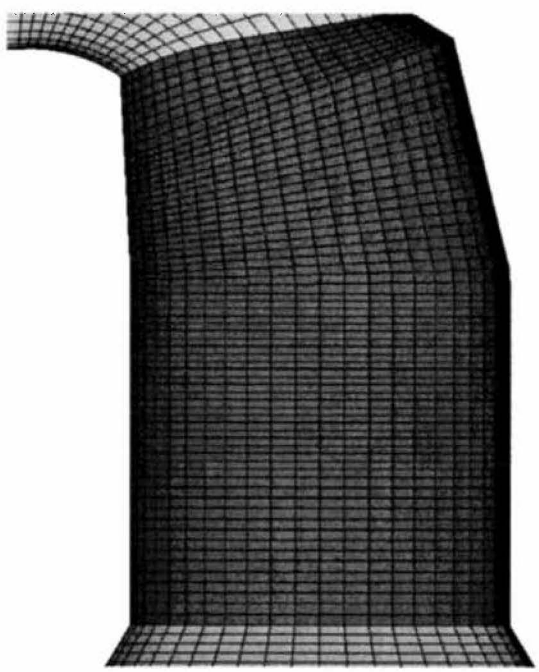


Figure 3-10: Detailed mesh showing Weld Overlay



3.2.4 Welding Parameters

References [6, 7 and 8] provide a set of welding procedure or parameters that are used in the present welding simulations to establish required parameters for the DM welds, repair welds and the SWOL. The welding parameters used in the modeling of the welding processes are shown in Table 3-3.

Table 3-3: Welding Parameters

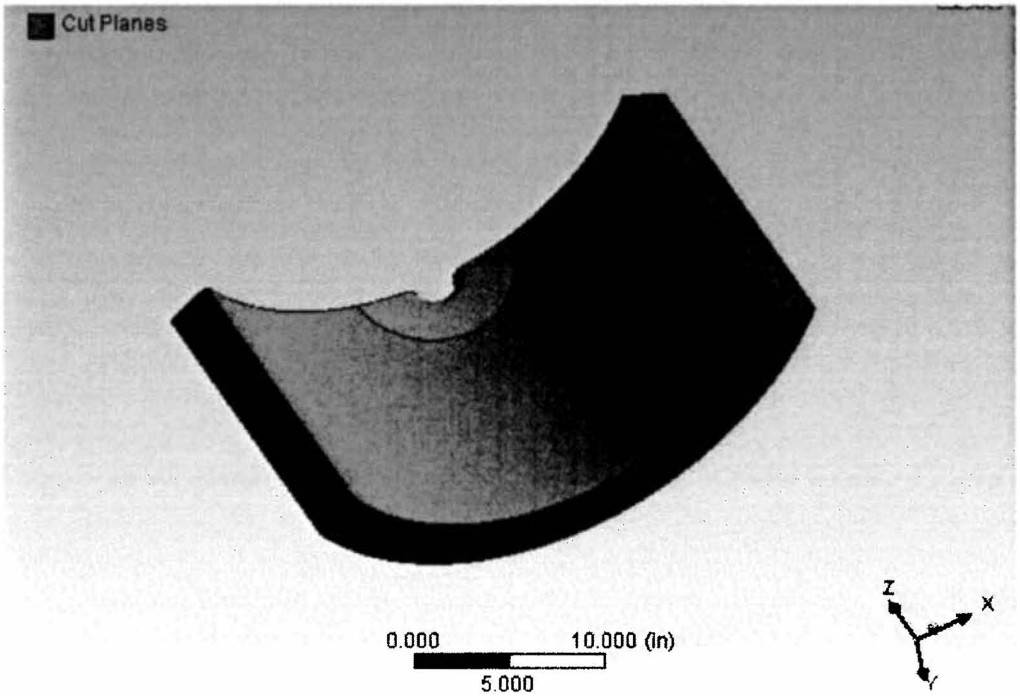
Welding Parameter	Value
DMW / RW Passes: Groove weld heat input calculated from typical welding parameters for a manual metal arc or manual gas shielded tungsten arc weld	
Rod Diameter	[
Current	
Voltage	
Travel Speed	
Arc Efficiency	
Maximum Interpass Temperature	]
Overlay Weld Passes	
Heat Input for the first layer [8]	[
Heat Input for 2nd layer onwards [8]:	
Maximum Interpass Temperature	]

3.3 Boundary Conditions for Welding Simulation

3.3.1 Thermal Analysis - Welding Simulation

The thermal model is loaded by a volumetric heat source applied to each weld pass. To enforce thermal continuity with adjacent components, adiabatic boundary conditions are applied at the symmetry planes (Figure 3-5) and the CL cutting planes (Figure 3-11). Thus no heat transfer occurs through the symmetry plane of the model as shown Figure 3-5 and the three cutting planes shown in Figure 3-11. Heat loss at the inner and outer surfaces is simulated using a heat transfer coefficient of [ ] btu/hr-ft^2-°F per the Reference [1] WRS procedure to model natural convection to an air environment. Radiative boundary conditions are not considered since radiation losses from the molten weld pool are included in the weld efficiency.

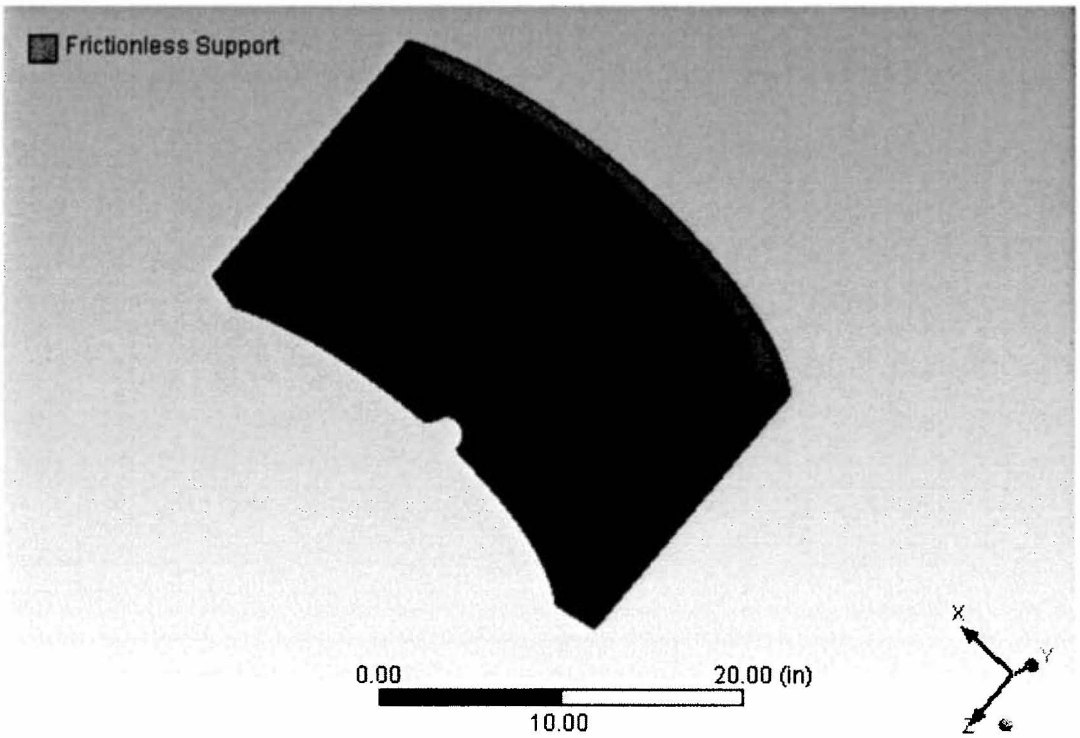
Figure 3-11: Insulated Surfaces at Cut Planes



3.3.2 Structural Analysis - Welding Simulation

The temperature history from the thermal analysis is used as the thermal load in the structural analysis. Frictionless support boundary conditions are maintained on all external "cut" surfaces of the finite element mode as shown in Figure 3-12.

Figure 3-12: Structural Model Constraints



4.0 ASSUMPTIONS

4.1 Assumptions Requiring Verifications

This calculation contains no major assumptions that must be verified prior to use on safety-related work.

4.2 Modeling Simplifications

The following is a list of modeling simplification that were used in this document to simplify the mesh:

1) A half-symmetric geometric model is considered appropriate to represent geometry for the welding simulation.

2) Weld passes are assumed to be deposited as full 360° weld passes.

3) No external piping loads are considered in this analysis. If external piping loads are suspected to impact the stresses used in subsequent flaw evaluations then the flaw evaluations should account for the external piping loads directly.

4.3 Engineering Approximations

The following is a list of engineering approximations that have no significant effect on the accuracy of the results calculated in this document.

1) Post-Weld Heat treatment (PWHT) if any is not simulated. This is a conservative assumption since any PWHT relieves the stresses in the weld.

2) Part of the Alloy 82 portion of the sulfur mitigation layer was modeled using Alloy 52M material as first specified for this project. Since the relevant thermal and mechanical properties of Alloy 82 and 52M are comparable, this approximation will have no significant effect on the results.

3) During the thermal analysis, thermal properties of low alloy steel were used instead of carbon steel thermal properties for the cold leg. This inadvertent error is determined to have virtually no effect on the results primarily because the cold leg is sufficiently removed from the welds that there is hardly any heating of the cold leg during the welding process.

5.0 COMPUTER USAGE

5.1 Software and Hardware

ANSYS Version 13.0 SP2 [3] was used in this calculation. Verification test cases were performed and documented herein.

- Computer program tested: ANSYS Version 13.0, verification tests vm32mod2D.vrt, vm32mod3D.vrt, vm38mod2D.vrt, and vm38mod3D.vrt.
- Error notices for ANSYS Version 13.0 SP2 were reviewed and none apply for this analysis.
- Computer hardware used: The computer hardware used for the stress runs is DELL (Service Tag # 600003). The hardware platform is Intel® Xeon® CPU E5645 at 2.4 GHz, 24 GB RAM and operating system is Microsoft Windows 7 Enterprise x64 Edition, Service Pack 1.
- Name of person running the test: Silvester Noronha
- Date of test: 11-05-2012
- Acceptability: For ANSYS 13.0 SP2, test cases vm32mod2D, vm32mod3D, vm38mod2D, vm38mod3D obtained from Reference [1] are run to verify that the answers are correct. The files vm32mod2D.vrt, vm32mod3D.vrt, vm38mod2D.vrt, and vm38mod3D.vrt contain output from the test cases. Review of the output shows that the answers are identical to those contained in Reference [1]. Appendix B lists the output from the test cases.

5.2 Computer Files

All ANSYS input files are collected and listed in Table 5-1. All computer runs and post processing data are documented in the ColdStor storage path [ ]. ANSYS verification input/output files are also listed.

Table 5-1: Listing of Computer Files

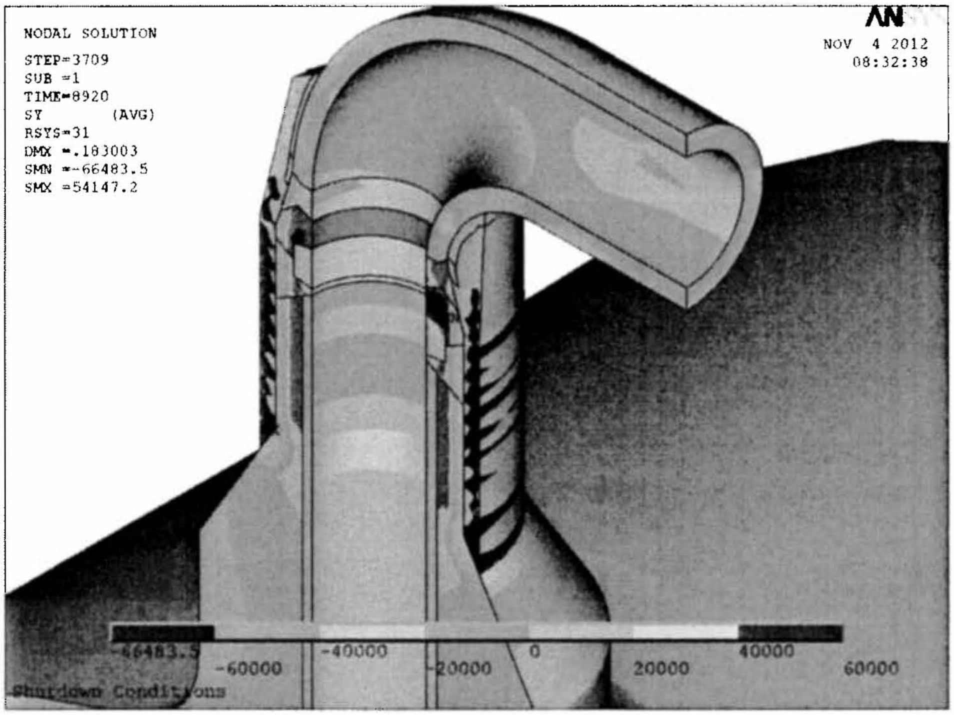
[ ]

6.0 CALCULATIONS/RESULTS

As discussed in Section 3.0, following the completion of the two DM welds, repair welds and the SWOL simulation, three steady state loading cycles were applied to the finite element model to obtain a stable state of stress after shakedown. This stress state is referred to as the residual stresses at cold conditions. The hoop and axial stress contours are shown in Figure 6-1 and Figure 6-2, respectively for shutdown conditions. Figure 6-3 and Figure 6-4 show hoop and axial stress contours, respectively for the operating conditions. The results are presented in a cylindrical coordinate system aligned with the axis of the nozzle.

Figure 6-5 shows the six path lines at the symmetric planes along which hoop and axial stresses are obtained. Hoop and axial stress distributions at shutdown conditions (70°F) are shown in Figure 6-6 and Figure 6-7 respectively. Figure 6-8 shows the hoop and Figure 6-9 shows the axial stress at steady state operating conditions (**[** **]** °F). The values of stresses plotted in Figure 6-6 through Figure 6-9 are also tabulated in Appendix A.

Figure 6-1: Hoop stress contours at shutdown (70°F). Obtained by applying three steady state loading cycles following the completion of the SWOL



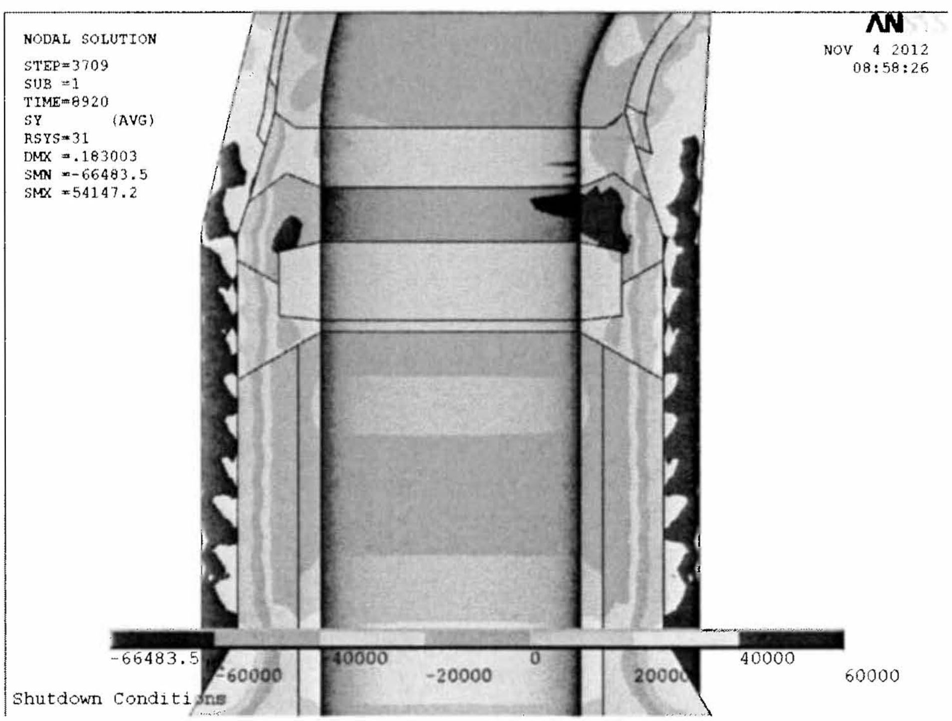
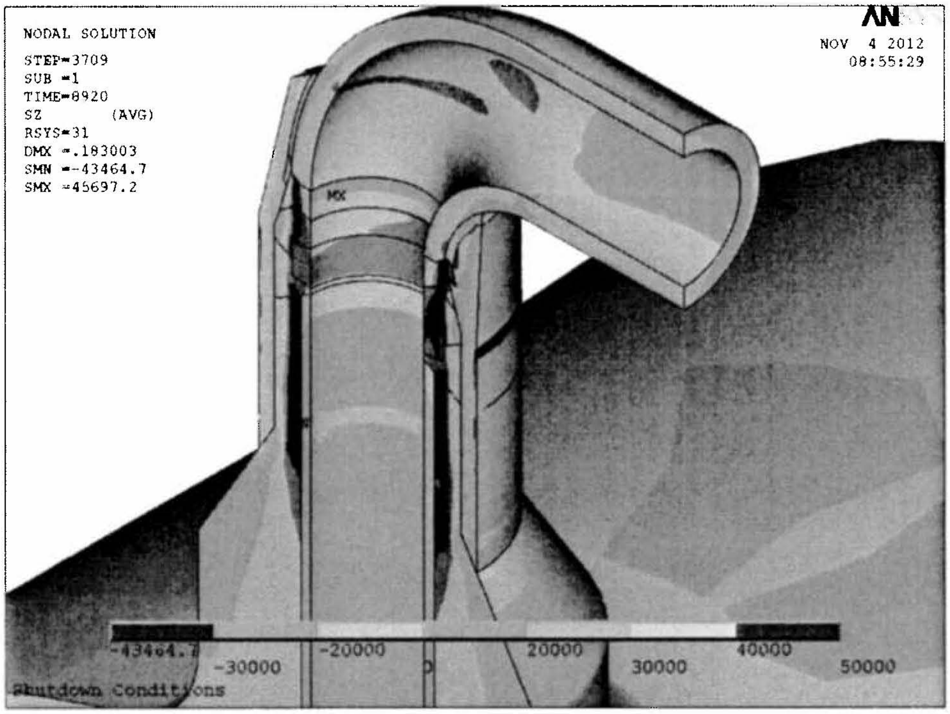


Figure 6-2: Axial stress contours at shutdown (70°F). Obtained by applying three steady state loading cycles following the completion of the SWOL



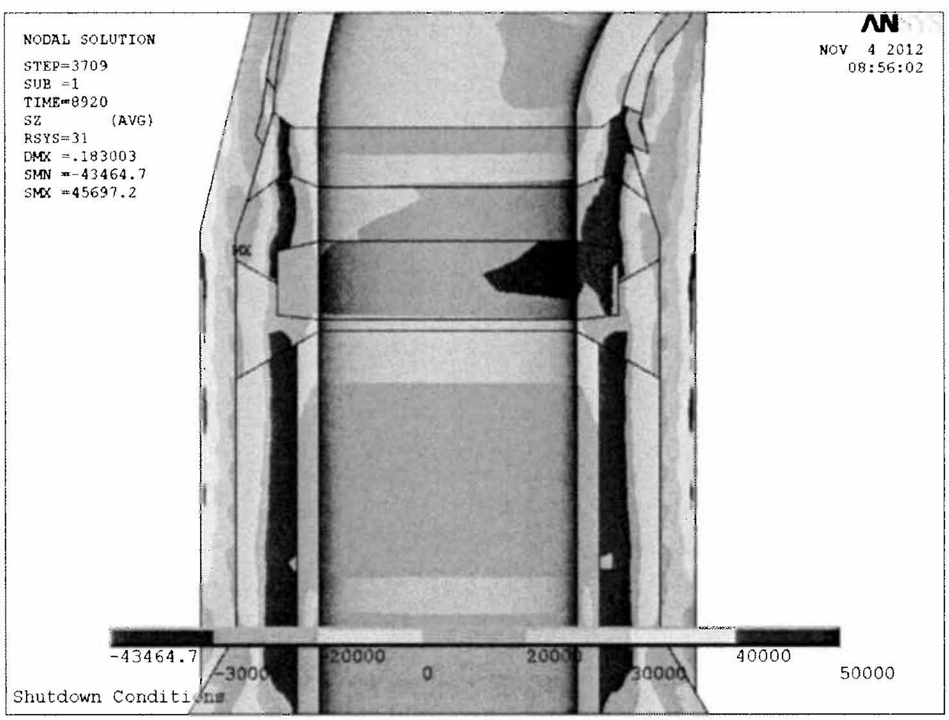


Figure 6-3: Hoop stress contours at steady state ([ ] °F). Obtained by applying two and a half steady state loading cycles following the completion of the SWOL

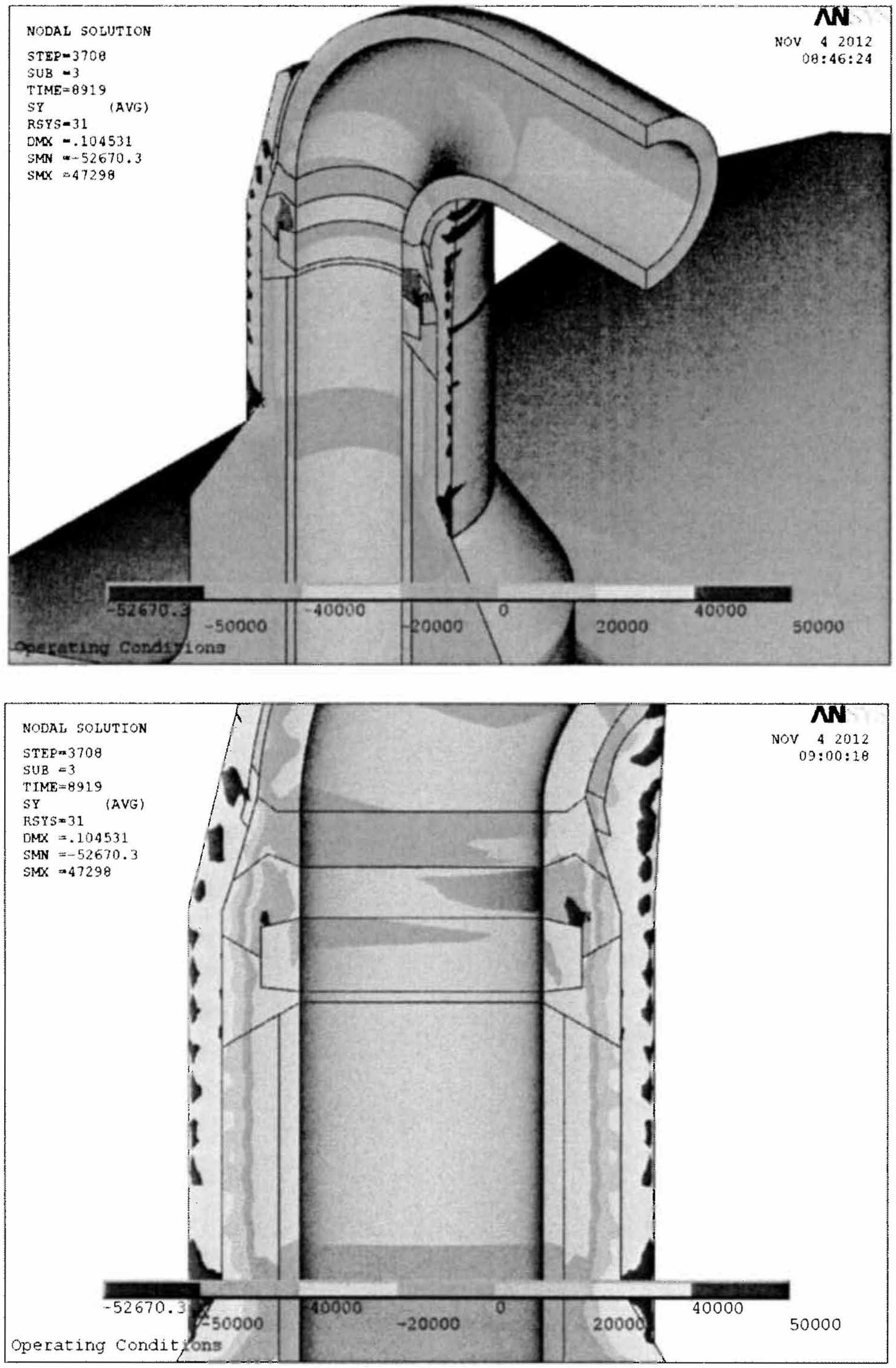
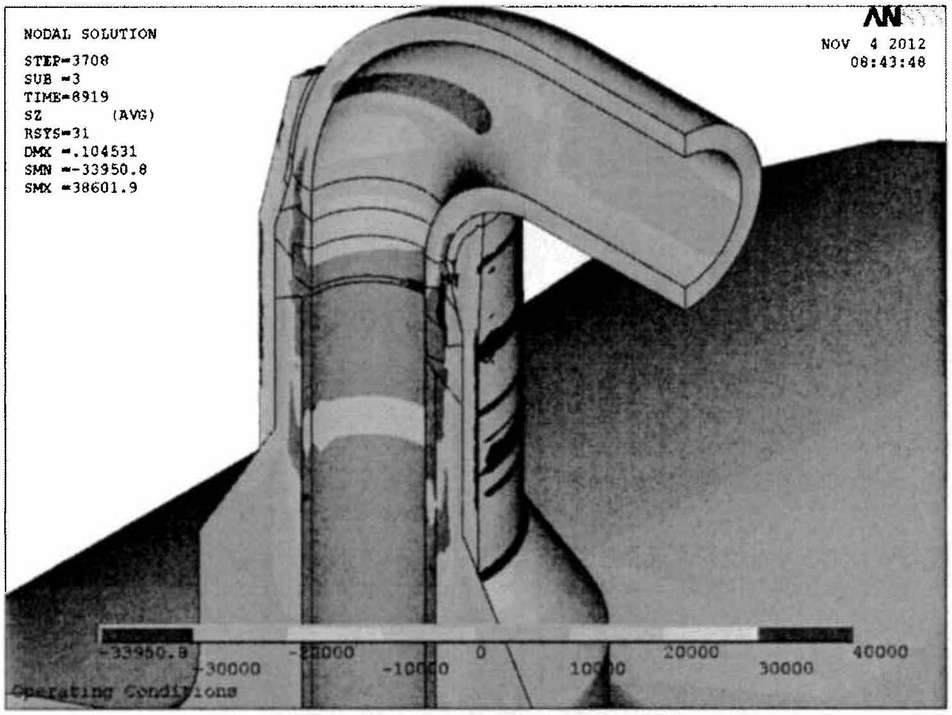


Figure 6-4: Axial stress contours at steady state ([ ] °F). Obtained by applying two and a half steady state loading cycles following the completion of the SWOL



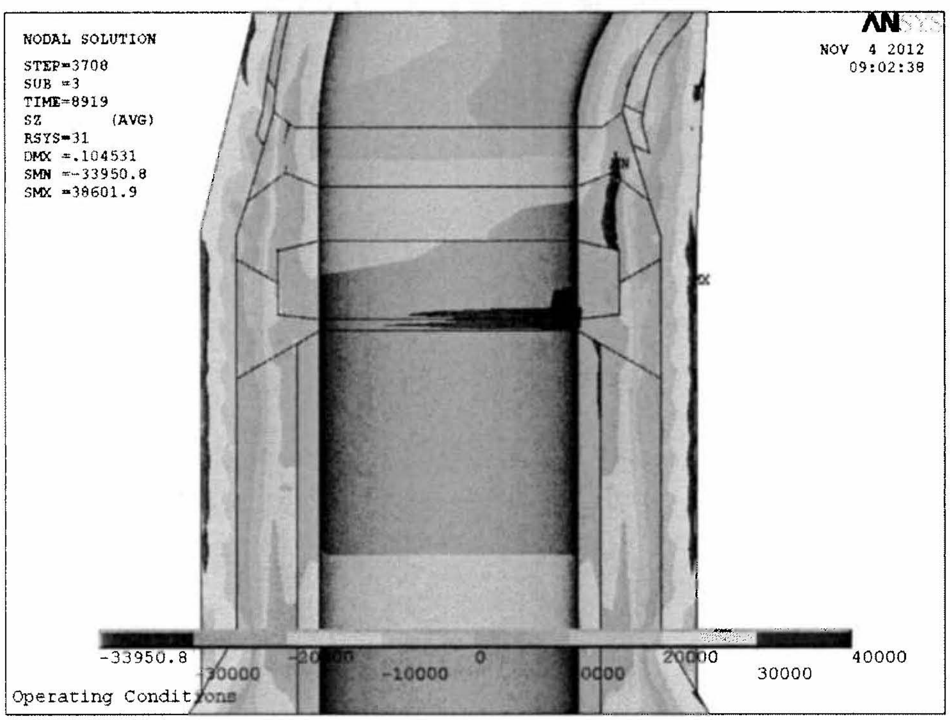
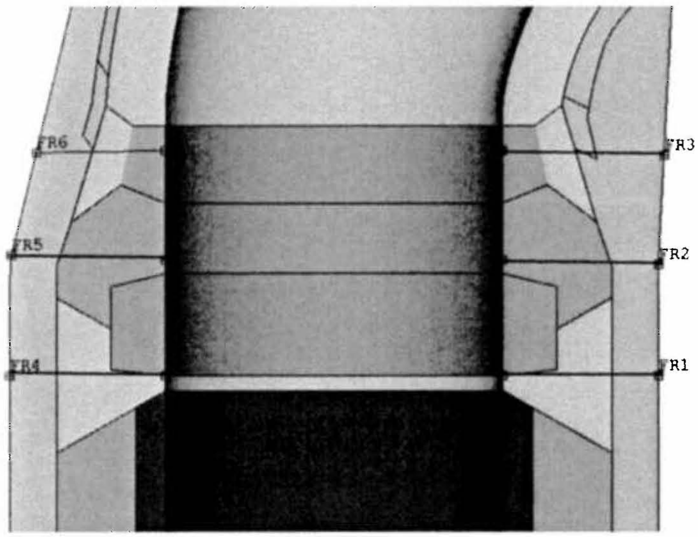


Figure 6-5: Path lines for hoop and axial stress distribution in the DM welds and safe end region



The node numbers corresponding to each path line are as follows:

Path line	ID Node	OD Node
FR1	7954	25230
FR2	28714	39031
FR3	26021	38991
FR4	7024	23498
FR5	28580	39138
FR6	26087	39128

Figure 6-6: Hoop stress distributions at shutdown (70°F). Obtained by applying three steady state loading cycles following the completion of the SWOL

[ ]

Figure 6-7: Axial stress distributions at shutdown (70°F). Obtained by applying three steady state loading cycles following the completion of the SWOL

[ ]

Figure 6-8: Hoop stress distributions at steady state ([ ] °F). Obtained by applying two-and-a-half steady state loading cycles following the completion of the SWOL

[ ]

Figure 6-9: Axial stress distributions at steady state ([ ] °F). Obtained by applying two-and-a-half steady state loading cycles following the completion of the SWOL

[ ]

7.0 REFERENCES

1. AREVA NP Document 32-2500013-001, "Technical Basis for Numerical Simulation of Welding Residual Stresses."
2. AREVA NP Document 18-1173549-006, "Functional Specification for RCS for Three Mile Island Unit One"
3. ANSYS Finite Element Computer Code, Version 13.0 SP2, ANSYS Inc., Canonsburg, PA.
4. AREVA NP Drawing 02-9185282C-000, "TMI Letdown Nozzle Existing Configuration"
5. AREVA NP Drawing 02-8059673D-003, "TMI Letdown Nozzle Weld Overlay Design"
6. AREVA NP Document 38-9194834-000, "Customer Supplied Documents – Three Mile Island Cold Leg Letdown Nozzle Weld Overlay"
7. AREVA NP Document 55-WP8/8/F6AW3-008, "Metallic Gas Tungsten Arc Welding - Welding Procedure Specification WP8/8/F6AW3-008"
8. AREVA NP Document 55-WP1/8/43/F43OLTBSCa3-003, Welding Procedure Specification WP1/8/43/F43OLTBSCa3"
9. AREVA Document 08-9182964-002, "TMI 'C' Cold leg Letdown Nozzle Weld Overlay"
10. AREVA NP Document 32-2500012-002, "Materials Database for Weld Residual Stress Finite Element Analysis"
11. ASME Boiler and Pressure Vessel Code, Section II, 2004 Edition with No Addenda

APPENDIX A: HOOP AND AXIAL STRESS TABLES

Figure 6-5 shows the path lines along which the stress results are obtained. The hoop and stress distribution at shutdown (70 °F), obtained by applying three steady state loading cycles subsequent to SWOL, and at steady state operating conditions ([ ] °F), obtained by applying two-and-a-half steady state loading cycles subsequent to SWOL, are listed in Table A-1 and Table A-2, respectively.

Table A-1: Hoop and Axial Stress Distributions at Shutdown Condition (70°F)

Along Path Line "FR1"			Along Path Line "FR2"			Along Path Line "FR3"		
Distance Along Path Line Measured from the ID (inches)	Hoop stress (ksi)	Axial stress (ksi)	Distance Along Path Line Measured from the ID (inches)	Hoop stress (ksi)	Axial stress (ksi)	Distance Along Path Line Measured from the ID (inches)	Hoop stress (ksi)	Axial stress (ksi)

Along Path Line "FR4"			Along Path Line "FR5"			Along Path Line "FR6"		
Distance Along Path Line Measured from the ID (inches)	Hoop stress (ksi)	Axial stress (ksi)	Distance Along Path Line Measured from the ID (inches)	Hoop stress (ksi)	Axial stress (ksi)	Distance Along Path Line Measured from the ID (inches)	Hoop stress (ksi)	Axial stress (ksi)
0	-31.143	-13.442	0	-54.6991	-18.3825	0	-16.329	4.6086
0.046786	-32.887	-17.447	0.0465	-55.8979	-19.988	0.038838	-19.188	-1.1848
0.093572	-33.771	-20.463	0.0929	-57.3099	-21.9476	0.077675	-21.93	-7.0771
0.14036	-34.16	-22.483	0.1394	-59.0877	-24.9729	0.11651	-24.401	-13.095
0.18714	-35.606	-24.506	0.1859	-61.1558	-28.2533	0.15535	-23.978	-17.71
0.23393	-38.958	-26.715	0.2323	-62.9025	-31.6811	0.19419	-22.56	-21.127
0.28072	-41.584	-27.798	0.2788	-63.0579	-33.4961	0.23303	-24.982	-27.817
0.3275	-41.905	-26.929	0.3252	-61.0751	-30.8738	0.27186	-24.94	-33.674
0.37429	-35.113	-26.069	0.3717	-57.6363	-26.0525	0.3107	-19.903	-34.414
0.42107	-29.228	-24.323	0.4182	-48.956	-18.5816	0.34954	-8.209	-27.302
0.46786	-14.472	-16.234	0.4646	-31.6707	-7.0996	0.38838	7.4075	-14.464
0.51465	-0.3548	-8.1741	0.5111	-15.9354	-0.0885	0.42721	17.326	-5.176
0.56143	8.7728	-7.6578	0.5576	-2.1951	2.3821	0.46605	21.382	0.15002
0.60822	16.974	-5.3343	0.604	10.6836	4.3071	0.50489	24.298	2.0168
0.655	30.542	4.8381	0.6505	25.6805	8.7005	0.54373	24.578	2.2591
0.70179	41.581	15.862	0.697	36.0855	15.5763	0.58256	30.22	9.1368
0.74858	43.044	23.358	0.7434	39.8134	19.5704	0.6214	35.055	16.13
0.79536	42.949	24.909	0.7899	41.2526	23.0089	0.66024	39.19	23.042
0.84215	42.797	26.025	0.8364	42.3151	26.9181	0.69908	40.983	27.213
0.88894	42.466	31.265	0.8828	42.7946	30.3797	0.73791	41.505	29.806
0.93572	42.856	39.392	0.9293	41.6909	32.2504	0.77675	41.608	31.025
0.98251	34.454	31.882	0.9757	36.7747	32.1422	0.81559	40.508	28.822

Table A-2: Hoop and Axial Stress Distributions at Steady State Operating Condition ([ ] °F)

Along Path Line "FR1"			**Along Path Line "FR2"**			**Along Path Line "FR3"**		
Distance Along Path Line Measured from the ID (inches)	Hoop stress (ksi)	Axial stress (ksi)	Distance Along Path Line Measured from the ID (inches)	Hoop stress (ksi)	Axial stress (ksi)	Distance Along Path Line Measured from the ID (inches)	Hoop stress (ksi)	Axial stress (ksi)
[								]

Along Path Line "FR4"			**Along Path Line "FR5"**			**Along Path Line "FR6"**		
Distance Along Path Line Measured from the ID (inches)	Hoop stress (ksi)	Axial stress (ksi)	Distance Along Path Line Measured from the ID (inches)	Hoop stress (ksi)	Axial stress (ksi)	Distance Along Path Line Measured from the ID (inches)	Hoop stress (ksi)	Axial stress (ksi)

APPENDIX B: VERIFICATION OF ANSYS COMPUTER CODE

Four verification problems were selected to test key features of the ANSYS finite element computer program [3] used in the current numerical welding simulations, the development of thermal stress in a cylinder and the elastic-plastic response of a cylinder under pressure loading.

The standard ANSYS verification manual test case VM32 exercises thermal and elastic stress analysis features of the axisymmetric two-dimensional 4-node PLANE55 and PLANE42 elements, respectively, using a long thick-walled cylinder subjected to a linear through-wall temperature gradient. This test case was been modified (vm32mod2D) by increasing the mesh refinement and changing the structural element type from PLANE42 to the 4-node PLANE182, which is used to verify 2D models. A companion three-dimensional test case (vm32mod3D) was created which utilizes the SOLID 70 thermal element and the SOLID185 structural element, which are used in the current welding simulations.

ANSYS verification manual test case VM38 determines stresses in a long thick-walled cylinder subjected to internal pressure using the PLANE42 axisymmetric structural element and an elastic-perfectly plastic material. Two pressure loads are considered; the first pressure of 12,990 psi loads the cylinder elastically to just below the yield strength of the material (30,000 psi), and the second puts the entire cylinder into a state of plastic flow (von Mises equivalent stress = 30,000 psi) at an ultimate pressure load of 24,011 psi (Pult). Test case VM38 was modified (vm38mod2D) to use the PLANE182 element. The stress-strain hardening model was changed from bilinear kinematic (BKIN) to multilinear kinematic (KINH) to better represent the current welding simulations. A companion three-dimensional test case (vm38mod3D) exercises the SOLID185 structural element. The error measure for the modified VM38 test cases is the ratio of the applied pressure to the theoretical value (24011 psi) of Pult such that the entire cylinder experiences an equivalent, or effective, stress of 30,000 psi.

All test cases executed properly, as demonstrated on the following pages.

Verification Problem VM32MOD

Thermal Stresses in a Long Cylinder

Two-Dimensional Analysis **File: vm32mod2D.vrt**

```
--------------- VM32MOD2D RESULTS COMPARISON ----------------

                  |   TARGET   |   ANSYS    |   RATIO

                    PLANE55 THERMAL ANALYSIS:

  T (C) X=.1875 in     -1.00000     -1.00000    1.000
  T (C) X=.2788 in     -0.67037     -0.67039    1.000
  T (C) X=0.625 in      0.00000      0.00000    0.000

                    PLANE182 STATIC ANALYSIS:

  A_STS psi X=.187      420.42       429.99    1.023
  T_STS psi X=.187      420.42       429.61    1.022
  A_STS psi X=.625     -194.58      -205.15    1.054
  T_STS psi X=.625     -194.58      -205.08    1.054
```

Three-Dimensional Analysis **File: vm32mod3D.vrt**

```
--------------- VM32MOD3D RESULTS COMPARISON ----------------

                  |   TARGET   |   ANSYS    |   RATIO

                    SOLID70 THERMAL ANALYSIS:

  T (C) X=.1875 in     -1.00000     -1.00000    1.000
  T (C) X=.2788 in     -0.67037     -0.67039    1.000
  T (C) X=0.625 in      0.00000      0.00000    0.000

                    SOLID185 STATIC ANALYSIS:

  A_STS psi X=.187      420.42       429.67    1.022
  T_STS psi X=.187      420.42       430.04    1.023
  A_STS psi X=.625     -194.58      -205.11    1.054
  T_STS psi X=.625     -194.58      -205.17    1.054

-----------------------------------------------------------
```

Verification Problem VM38MOD

Plastic loading of a Thick-Walled Cylinder

Two-Dimensional Analysis **File: vm38mod2D.vrt**

```
--------------- VM38MOD2D RESULTS COMPARISON ----------------

                      |   TARGET   |   ANSYS    |   RATIO

      PLANE182 FULLY ELASTIC ANALYSIS (psi):

 SIGR    LEFT END        -9984.     -10103.  1.012
 SIGT    LEFT END        18645.      18763.  1.006
 SIGR   RIGHT END         -468.       -481.  1.028
 SIGT   RIGHT END         9128.       9141.  1.001

      PLANE182 FULLY PLASTIC ANALYSIS (psi):

 SIGEFF  LEFT END        30000.      30000.  1.000
 SIGEFF RIGHT END        30000.      30000.  1.000
 Pult                    24011.      23350.  0.972

-----------------------------------------------------------
```

Three-Dimensional Analysis **File: vm38mod3D.vrt**

```
--------------- VM38MOD3D RESULTS COMPARISON ----------------

                      |   TARGET   |   ANSYS    |   RATIO

      SOLID185 FULLY ELASTIC ANALYSIS (psi):

 SIGR    LEFT END        -9984.     -10066.  1.008
 SIGT    LEFT END        18645.      18776.  1.007
 SIGR   RIGHT END         -468.       -475.  1.014
 SIGT   RIGHT END         9128.       9128.  1.000

      SOLID185 FULLY PLASTIC ANALYSIS (psi):

 SIGEFF  LEFT END        30000.      30000.  1.000
 SIGEFF RIGHT END        30000.      30000.  1.000
 Pult                    24011.      23360.  0.973

-----------------------------------------------------------
```

AREVA

CALCULATION SUMMARY SHEET (CSS)

Document No. 32 - 9196161 - 002

Safety Related: ☒ Yes ☐ No

Title TMI-1 Letdown Nozzle Weld Overlay Sizing Calculation (Non-Proprietary)

PURPOSE AND SUMMARY OF RESULTS:

AREVA NP Inc. Proprietary information in the document is indicated by pairs of braces " [ ] ".

Purpose:

The purpose of this report is to calculate the weld overlay size (thickness and length) at the two weld locations for the letdown nozzle on the cold leg at TMI Unit 1 per ASME B&PV Code, Section XI, Division 1 (References [2]) and Code Case N-740-2 (Reference [3]).

Rev. 001: Revised what information in the document is marked with pairs of square braces "[ ]". All references are updated to the latest revisions.

Rev. 002: Revised to remove proprietary statement and markings.

Summary:

The minimum full structural weld overlay thickness is determined to be [ ] for both welds. The weld overlay length is [ ] for both welds. The overlay length is measured from the intersection of the weld material with the adjacent base material on the outside surface.

Rev. 001: Results from Rev. 000 remain valid.

Rev. 002: Results from Rev. 000 and Rev. 001 remain valid.

[redacted]

THE FOLLOWING COMPUTER CODES HAVE BEEN USED IN THIS DOCUMENT:		THE DOCUMENT CONTAINS ASSUMPTIONS THAT SHALL BE VERIFIED PRIOR TO USE
CODE/VERSION/REV	CODE/VERSION/REV	☐ YES
		☒ NO

TMI-1 Letdown Nozzle Weld Overlay Sizing Calculation (Non-Proprietary)

Review Method: ☒ Design Review (Detailed Check)
☐ Alternate Calculation

Signature Block

Name and Title (printed or typed)	Signature	P/R/A and LP/LR	Date	Pages/Sections Prepared/Reviewed/Approved
Kristine Barnes, Engineer IV	Kristi Barnes	P	2/21/13	All.
Kaihong Wang, Principal Engineer	KWang	R	2/21/13	All, detailed review.
Tim Wiger, Manager	T.M. Wiger	A	2/21/13	All.

Note: P/R/A designates Preparer (P), Reviewer (R), Approver (A);
LP/LR designates Lead Preparer (LP), Lead Reviewer (LR)

Project Manager Approval of Customer References (N/A if not applicable)

Name (printed or typed)	Title (printed or typed)	Signature	Date
N/A	N/A	N/A	N/A

Mentoring Information (not required per 0402-01)

Name (printed or typed)	Title (printed or typed)	Mentor to: (P/R)	Signature	Date
N/A	N/A	N/A	N/A	N/A

Record of Revision

Revision No.	Pages/Sections/Paragraphs Changed	Brief Description / Change Authorization
000	All	Initial release.
001	Pages 1-3 Page 4 Pages 6-12	Updated to Rev. 001 Updated TOC Added Rev. 001 purpose and revised what information is marked with pairs of square braces "[ ]".
002	Pages 1-3 Page 6	Updated to Rev. 002 and latest CSS form. Added Rev. 002 purpose.

Table of Contents

List of Tables

1.0 INTRODUCTION

Primary water stress corrosion cracking (PWSCC) of Alloy 82/182 materials is a well recognized phenomenon in the nuclear power industry. High temperature components such as those nozzles connected to the cold leg have higher risk to PWSCC at these dissimilar metal (DM) welds, i.e., the Alloy 82/182 welds.

Three Mile Island Unit 1 (TMI-1) plans to mitigate the cold leg letdown nozzle Alloy 82/182 DM welds with full structural weld overlays (FSWOL) during the T1R20 outage in the fall of 2013. The corresponding DM welds to be overlaid are located at the [ ] letdown nozzle. Since the weld between the nozzle safe end and the pipe elbow is connected to the weld between the nozzle and safe end, the weld overlay has to extend onto both welds that are Alloy 82 (Reference [1]). The letdown nozzle material is [ ], the piping is [ ], and the nozzle safe end material is [ ] (Reference [1]). In the following, nozzle weld refers to the weld between the nozzle and safe end, and safe end weld refers to the weld between the safe end and the piping elbow.

2.0 PURPOSE AND SCOPE

The purpose of this calculation is to determine the minimum structural weld overlay length and thickness required for the repair of the letdown nozzle at TMI-1, in accordance with the Design Specification (Reference [1]) and References [2] and [3] criteria related to the weld overlay sizing.

Specifically, this calculation determines the minimum structural requirements for the weld overlay size (thickness and length) for the two weld locations (nozzle weld and safe end weld) at the cold leg letdown nozzle per References [2] and [3].

The minimum structural requirement on the thickness calculated herein does not include any allowance for possible crack growth, weld material dilution layers or surface machining.

Rev. 001: Revised what information in the document is marked with pairs of square braces "[ ]". All references are updated to the latest revisions. Rev. 002: Revised to remove proprietary statement and markings.

3.0 ANALYTICAL METHODOLOGY

According to Reference [3], paragraph 2(b)(4), the combined wall thickness at the weld overlay with the flaw size assumptions given in paragraph 2(b)(3) shall be evaluated as well as meet the requirements set forth in IWB-3640 of Reference [2]. For the assumed circumferential and axial flaws, IWB-3640 of Reference [2] instructs to use formula given in Appendix C of the same reference to calculate the corresponding stresses with a flaw present either circumferentially or axially.

3.1 FSWOL Thickness by Circumferential Flaw Criteria

As above mentioned, the required thickness of weld overlay repair with the assumption of the circumferential flaw (paragraph 2(b)(3)(a) of Reference [3]) can then be determined by formulas given in Appendix C of Reference [2]. The criterion is based on net section plastic collapse, which predicts adequate load capacity of flawed pipes repaired by weld overlays for given applied stresses, σ_m and σ_b. Here σ_m is the pipe primary membrane stress and σ_b is the pipe primary bending stress. Note that only the applied loads such as pressure, deadweight and seismic loads are needed in evaluating σ_m and σ_b. Stresses due to temperature gradients and thermal expansion need not be considered since these loads cause stresses that are self limiting, and therefore do not affect the net section plastic collapse.

For a circumferentially flawed pipe, the relation between the applied loads in term of bending stress and the flaw depth at incipient plastic collapse per Reference [2], Section C-5321 is given by:

$$\sigma_b^c = \frac{2 \cdot \sigma_f}{\pi} \left(2 - \frac{a}{t} \right) \cdot \sin \beta \tag{1}$$

where t is the pipe thickness including the overlay, β is the angle that defines the location of the neutral axis (for details see Figure C-4310-1 of Reference [2]), a is the flaw depth. The flow stress σ_f is the average of the material ultimate tensile strength (S_u) and yield strength (S_y) (Section C-8200 of Reference [2]). The value for S_u (=80.0 ksi) and S_y (=27.5 ksi) both at 650°F are taken from Reference [4] for the overlay material Alloy 690, and therefore σ_f = (80.0+27.5)/2 = 53.75 ksi.

The assumed circumferential through-wall flaw penetrates the compressive bending region such that $(\theta + \beta) > \pi$, where θ is one-half of the flaw angle (180 degrees), and therefore the angle β per Reference [2] is given by:

$$\beta = \frac{\pi}{2 - \frac{a}{t}} \cdot \left(1 - \frac{a}{t} - \frac{\sigma_m}{\sigma_f} \right) \tag{2}$$

where σ_m is the pipe primary membrane stress in the axial direction in the unflawed section of the pipe. The allowable bending stress S_c is given by:

$$S_c = \frac{\sigma_b^c}{SF_b} - \sigma_m \left(1 - \frac{1}{SF_m} \right) \tag{3}$$

where SF_b and SF_m are specified in C-2621 of Reference [2] for service levels A to D.

For a circumferentially flawed pipe, the relation between the applied membrane stress and the flaw depth at incipient plastic collapse bending stress per Reference [2], Section C-5322 is given by:

$$\sigma_m^c = \sigma_f \left(1 - \frac{a}{t} \cdot \frac{\theta}{\pi} - \frac{2\varphi}{\pi} \right) \tag{4}$$

where,

$$\varphi = \arcsin\left(0.5 \cdot \frac{a}{t} \cdot \sin \theta \right) \tag{5}$$

The allowable membrane stress S_t for each service level is given by:

$$S_t = \sigma_m^c / SF_m \tag{6}$$

Additionally, Section C-5300 of Reference [2] also states that in no case shall the resulting flaw depth be greater than $a = 0.75t$, and the weld overlay thickness should be adjusted to satisfy this criterion if necessary.

3.2 FSWOL Thickness by Axial Flaw Criteria

Similarly, when the flaw is assumed to be in the axial direction (paragraph 2(b)(3)(b) of Reference [3]), the pipe hoop stresses shall be evaluated according to C-5400 of Reference [2]. The flawed pipe in this case is the original weld combined with the weld overlay, considering now the flaw depth as the original weld thickness (i.e., the original weld is completely cracked in the axial direction). The required thickness of the weld overlay repairs with the presence of the axial flaw in the assumed size can then be determined by formula given in Appendix C-5420 of Reference [2]. The allowable hoop stress σ_{ha} is given by:

$$\sigma_{ha} = \frac{\sigma_f}{SF_m}\left(\frac{1-\left(\frac{a}{t}\right)}{1-\left(\frac{a}{t}\right)\Big/M_2}\right) \tag{7}$$

where

$$M_2 = \left(1+\frac{1.61}{4R_m t}\cdot l^2\right)^{0.5} \tag{8}$$

with R_m the mean radius of the overlaid pipe and t the pipe thickness including the overlay, and l is the assumed axial length of the flaw. The safety factor SF_m is specified in C-2622 of Reference [2] for service levels A to D. The applied hoop stress under internal pressure of P is calculated by:

$$\sigma_h = \frac{PR_m}{t} \tag{9}$$

Again, Section C-5400 of Reference [2] also states that in no case shall the resulting flaw depth be greater than $a = 0.75t$, and the weld overlay thickness should be adjusted to satisfy this criterion if necessary.

3.3 FSWOL Length

Per Reference [3], paragraph 2(b)(1), to provide for load redistribution from the item into the weld overlay and back into the item without violating applicable stress limits of NB-3200, the length of the weld overlay should extend at least $0.75(Rt_n)^{1/2}$ beyond each end of the observed flaw where R and t_n are the outside radius and the nominal wall thickness of the pipe prior to depositing the weld overlay.

4.0 ASSUMPTIONS

This calculation contains no assumptions that must be verified prior to use on safety-related work. Simplifications in modeling and simulation used in the calculation are due to the acceptability requirement specified in Reference [1] for the following two assumptions stated in Reference [3]:

Circumferential Flaw – 100% through wall (original weld) for the entire circumference.

Axial Flaw –100% through wall (original weld) for a length of 1.5″, or the combined width of the weld plus buttering, whichever is greater. Since the two Alloy 82 welds (letdown nozzle to safe end weld and elbow to safe end weld) are close to each other (Reference [5]), the combined length of the two welds (including the short safe end in between) is [ ] at maximum. To be conservative, the total length of [ ] is considered as the axial flaw length in the following calculation.

5.0 DESIGN INPUTS

5.1 Geometry

Based on the geometry given in Reference [6], the nozzle weld has an inside diameter of [ ] and the outside diameter is [ ] at the nozzle end; the safe end weld has an inside diameter of [ ] and the outside diameter is [ ] at the piping connection for the [ ] pipe. As illustrated in Reference [5], the two [ ] welds (letdown nozzle to safe end weld and elbow to safe end weld) are close to each other with a short safe end in between; the total length of the combined welds is [ ] at maximum.

5.2 Materials and Properties

Per Reference [1], the nozzle is welded to the cold leg and fabricated from [ ]. The safe end material is [ ]. The letdown nozzle to safe end and elbow to safe end welds are both [ ]. The weld overlay material is Alloy 52M with material properties equivalent to Alloy 690, SB-166.

The ultimate strength and yield strength of the overlay material Alloy 690 is S_u = 80.0 ksi and S_y = 27.5 ksi, both at 650°F are taken from Reference [4] for the overlay material.

5.3 Applied Loads

As identified in Reference [2], Appendix C, only primary stresses (σ_m – maximum applied pipe primary membrane stress, σ_b – maximum applied pipe primary bending stress and σ_h – maximum applied hoop stress) are needed to determine the acceptability of a flawed pipe for continued service. The primary stresses considered in this application result from internal pressure, dead weight (DW), seismic loads (OBE or SSE). Section C-2620 of the same reference also specifies safety factors SF_m and SF_b applied individually to membrane and bending stresses respectively, for each Service Level: A (Normal), B (Upset), C (Emergency), and D (Faulted). Below are the required safety factors as specified in Reference [2], Sections C-2621 and C-2622:

Service Level A:	$SF_m = 2.7$	$SF_b = 2.3$
Service Level B:	$SF_m = 2.4$	$SF_b = 2.0$
Service Level C:	$SF_m = 1.8$	$SF_b = 1.6$
Service Level D:	$SF_m = 1.3$	$SF_b = 1.4$

The limiting load combinations for the ASME Code Service Level conditions are as follows:

Service Level A:	Maximum Normal Pressure + DW + OBE
Service Level B:	Maximum Upset Pressure + DW + OBE
Service Level C:	Maximum Emergency Pressure + DW + OBE
Service Level D:	Maximum Faulted Pressure + DW + SSE

Note that DW+OBE combination is conservatively used for both Normal and Emergency conditions.

The piping loads at the safe end are taken from Reference [7]. The axial membrane stress due to the internal pressure is determined by *PD/4t*, where *P* is the maximum specified service level pressure presented in Reference [8] and *D* is the outside diameter of the pipe. The SRSS (square root of the sum of squares) moment is conservatively defined as $\sqrt{Torsion^2 + M_y^2 + M_z^2}$. The maximum pressures approximated from the thermal design transients documented in Reference [8] are collected as follows:

Normal – [ ] psi [ ]
Upset – [ ] psi [ ]
Emergency – [ ] psi [ ]
Faulted – [ ] psi [ ]

Table 5-1 lists the total loads to be used for both the nozzle and safe end weld overlays.

Table 5-1 Loading Conditions at the Nozzle/Safe End

Load Case	Moments (in-lbf)			
	Torsion	M_y	M_z	SRSS
DW+OBE	[			
DW+SSE				
Total for Normal, Upset and Emergency				
Total for Faulted				]

6.0 COMPUTER USAGE

No engineering software is used in the sizing calculation.

7.0 CALCULATIONS

7.1 FSWOL Thickness

As above mentioned in Section 3.0, the weld overlay thickness with the flaw size assumptions given in paragraph 2(b)(3) shall be evaluated as well as meet the requirements set forth in IWB-3640 of Reference [2], which directs to use formula given in Appendix C of the same reference to calculate the corresponding stresses with a flaw present either circumferentially or axially.

7.1.1 Circumferential Flaw Evaluation

The weld overlay thickness is determined through an iterative approach. The outside diameter at the weld overlay location is obtained by postulating an overlay thickness; the primary stresses σ_m and σ_b are then calculated for the applied loads; the allowable stresses S_c and S_t obtained by Equations (3) and (6) should be equal to or greater than respective applied stresses when an allowable flaw depth is reached. Sections C-5300 and C-5400 (Reference [2]) also state that in no case shall the resulting flaw depth be greater than $a = 0.75t$, and the weld overlay thickness should be adjusted to satisfy this criterion if necessary. The results from the iteration along with parameters used in the calculation are listed in Table 7-1.

Table 7-1 FSWOL Thickness at the Nozzle End (Circumferential Flaw)

Param.		Normal	Upset	Emergency	Faulted
d_o, inch	Weld outside diameter, Reference [6]				
d_i, inch	Weld inside diameter, Reference [6]				
a, inch	Assumed crack depth, $= (d_o - d_i)/2$				
P, psi	Maximum service pressure, Section 5.3				
M, in-lbf	SRSS moment, Table 5-1				
t_{wol},	Weld overlay thickness (rounded value)				
t, inch	Pipe thickness including weld, $= a + t_{wol}$				
A, inch^2	Sectional area, $= (\pi/4)((d_o + 2t_{wol})^2 - d_i^2)$				
Z, inch^3	Section modulus, $= (\pi/64)((d_o + 2t_{wol})^4 - d_i^4)/(d_o/2 + t_{wol})$				
σ_f, psi	Flow stress, $= (S_y + S_u)/2$, Section 3.1				
SF_m	Membrane stress safety factor, Section 5.3				
SF_b	Bending stress safety factor, Section 5.3				
β, rad	Angle β to neutral axis by Eq. (2)				
φ, rad	Angle φ by Eq. (5)				
σ^c_b, psi	Bending stress at failure by Eq. (1)				
S_c, psi	Allowable bending stress with required safety factor be Eq. (3)				
σ_b, psi	Applied bending stress, $= M/Z$				
σ^c_m, psi	Membrane stress at failure by Eq. (4)				
S_t, psi	Allowable membrane stress with required safety factor by Eq. (6)				
σ_m, psi	Applied membrane stress, $= P(d_o+2t_{wol})/4t$				
r	Flaw depth ratio, a/t				

The piping loads listed in Table 5-1 are identical for both welds. Since the two welds with the same ID are directly connected from the outside surface while the inside surfaces are separated by the safe end, the combined weld needs to be considered by using the maximum weld thickness (which is the weld between the nozzle and safe end) in the calculation of the overlay thickness.

7.1.2 Axial Flaw Evaluation

The previously determined weld overlay thickness shown in Table 7-1 is used in Equations (7) through (9) to ensure acceptability of the WOL for axial flaw criteria. The results along with the parameters used in the calculation are listed in Table 7-2 for the nozzle weld, which remains bounding for the safe end weld.

Table 7-2 FSWOL Thickness at the Nozzle End (Axial Flaw)

Param.	Description	Normal	Upset	Emergency	Faulted
d_o, inch	Weld outside diameter, Reference [6]				
d_i, inch	Weld inside diameter, Reference [6]				
t_{wol},	Weld overlay thickness (rounded value)				
a, inch	Assumed flaw depth, $= (d_o - d_i)/2$				
l, inch	Assumed axial flaw length, Section 4.0				
t, inch	Pipe thickness including weld, $= a + t_{wol}$				
P, psi	Maximum service pressure, Section 5.3				
R_m, inch	Mean radius, $= (d_i + (d_o + 2t_{wol}))/4$				
M_2	By Eq. (8)				
σ_f, psi	Flow stress, $= (S_y + S_u)/2$, Section 3.1				
SF_m	Membrane stress safety factor, Section 5.3				
σ_{ha}, psi	Allowable hoop stress with required safety factor by Eq. (7)				
σ_h psi	Applied hoop stress, $= PR_m/t$				
r	Flaw depth ratio, a/t				

As shown in Table 7-1 to Table 7-2, the minimum overlay thickness is [ ] for both welds, controlled by the thickness ratio criterion.

7.2 FSWOL Length

To meet the requirement of Reference [3], paragraph 2(b)(1), with R = [ ] and t_n = [ ] for the nozzle weld, the full thickness weld overlay length shall be at least:

$$0.75(Rt_n)^{1/2} = 0.75\ [\quad]^{1/2} = [\quad]$$

Note that the weld overlay length is to be conservatively measured in full thickness from the intersection of the weld material with the adjacent base material (nozzle or elbow) on the outside surfaces.

8.0 RESULTS, SUMMARY/CONCLUSIONS

In accordance with References [1], [2] and [3], the minimum weld overlay size is calculated as follows:

Thickness – [ ] for both the nozzle and safe end welds

Length – [ ] for both the nozzle and safe end welds

The length is all measured in full thickness on each side of the weld intersection, as noted in Section 7.2.

Note that the weld overlay thickness is the minimum required by the applicable acceptance criteria for primary sources of loading. The final weld overlay thickness should additionally consider fatigue crack growth, weld material dilution, and machining allowance in the final overlay design.

9.0 REFERENCES

[1] AREVA NP Inc. Design Specification 08-9182964-002, "TMI-1 'C' Cold Leg Letdown Nozzle Weld Overlay."
[2] ASME Boiler and Pressure Vessel Code, 2004 Edition with no Addenda, Section XI, Division I.
[3] Code Case N-740-2, "Full Structural Dissimilar Metal Weld Overlay for Repair or Mitigation of Class 1, 2, and 3 Items, Section XI, Division I."
[4] ASME Boiler and Pressure Vessel Code, 2004 Edition with no Addenda, Section II, Part D – Properties.
[5] AREVA NP Inc. Drawing 02-9185282C-000, "TMI-1 Letdown Nozzle, Existing Configuration."
[6] AREVA NP Inc. Drawing 02-131964E-06, "Assembly and Details for 1 ½" TEMP. CONN's, 1" Drain Nozzle and 2 ½" Drain Nozzle."
[7] AREVA NP Inc. Drawing 02-163313E-03, "Reactor Coolant System Nozzle Loadings."
[8] AREVE NP Inc. Functional Specification 18-1173549-006, "Functional Specification for Reactor Coolant System for TMI-1."

AREVA

CALCULATION SUMMARY SHEET (CSS)

Document No. 32 - 9196160 - 002 Safety Related: ☒ Yes ☐ No

Title TMI-1 Letdown Nozzle Weld Overlay Section III Analysis (Non-Proprietary)

PURPOSE AND SUMMARY OF RESULTS:

AREVA NP Inc. Proprietary information in the document is indicated by pairs of braces " [ ] ".

PURPOSE:

This document presents the thermal and structural analyses of the TMI-1 cold leg letdown nozzle with a weld overlay. The purpose of this calculation is to qualify the weld overlay design to the requirements of the ASME B&PV Code Section III, Division 1, 2004 Edition with no addenda (Reference [2]).

Rev. 001: Revised what information in the document is marked with pairs of square braces "[ ]".

Rev. 002: Revised to remove proprietary statement and markings.

SUMMARY:

The thermal and structural analyses demonstrate that the cold leg letdown nozzle weld overlay design satisfies the ASME Code (Reference [2]) primary and primary plus secondary stress requirements as well as criteria against fatigue failure. Based on the loads and cycles specified in References [1], [11], and [12], the fatigue analyses performed in this document indicate that the maximum fatigue usage factor for the cold leg letdown nozzle weld overlay design is [ ] .

This document contains 60 pages including pages 1 - 56, Appendix A (3 Pages), and Appendix B (1 Pages).

Rev. 001: Results made in Rev. 000 remain unchanged.

Rev. 002: Results from Rev. 000 and Rev. 001 remain unchanged.

THE FOLLOWING COMPUTER CODES HAVE BEEN USED IN THIS DOCUMENT:		THE DOCUMENT CONTAINS ASSUMPTIONS THAT SHALL BE VERIFIED PRIOR TO USE
CODE/VERSION/REV	CODE/VERSION/REV	☐ YES
ANSYS R13.0 SP2		☒ NO

TMI-1 Letdown Nozzle Weld Overlay Section III Analysis (Non-Proprietary)

Review Method: ☒ Design Review (Detailed Check)
☐ Alternate Calculation

Signature Block

Name and Title (printed or typed)	Signature	P/R/A and LP/LR	Date	Pages/Sections Prepared/Reviewed/Approved
Kristine Barnes Engineer VI	Kristi Barns	P	2/21/13	All
Kaihong Wang Principal Engineer	KWang	R	2/21/13	All
Tim Wiger Manager	T.M. Wiger	A	2/21/13	All

Note: P/R/A designates Preparer (P), Reviewer (R), Approver (A);
LP/LR designates Lead Preparer (LP), Lead Reviewer (LR)

Project Manager Approval of Customer References (N/A if not applicable)

Name (printed or typed)	Title (printed or typed)	Signature	Date
N/A			

Record of Revision

Revision No.	Pages/Sections/Paragraphs Changed	Brief Description / Change Authorization
000	All	Initial Issue
001	Pages 1-3 Page 9 Pages 11-13, 19, 21, 24, 28, 29, 52, A-2	Updated to Rev. 001. Added Rev. 001 purpose and revised pairs of square braces "[ ]" Revised pairs of square braces "[ ]".
002	Pages 1-3 Page 9	Updated to Rev. 002 and latest CSS form. Added Rev. 002 purpose.

Table of Contents

Page

Table of Contents
(continued)

List of Tables

List of Tables
(continued)

List of Figures

1.0 INTRODUCTION

It is well recognized that the Alloy 600/82/182 dissimilar metal welds (DMWs) are susceptible to the primary water stress corrosion cracking (PWSCC), especially those in high temperature components such as cold leg nozzles. The possibility of PWSCC at the DMWs escalates with increased plant service time. Three Mile Island Unit 1 (TMI-1) plans to mitigate the PWSCC in the cold leg letdown nozzle Alloy 82/182 dissimilar metal (DM) welds with full structural weld overlays (FSWOLs).

The weld overlay is designed to cover the Alloy 82/182 welds between the nozzle safe end and the elbow. Application of the weld overlays alters the local stress distribution. A detailed finite element analysis (FEA) is performed to investigate stress conditions under various operational transients. The results are summarized to certify that criteria per ASME Code Section III for Class 1 components are satisfied for the letdown nozzle with overlay. The analysis is focused on the overlaid region for requirements on both stress distribution and fatigue failure criteria.

1.1 Purpose and Scope

As required by the Design Specification (Reference [1]), the purpose of this calculation is to perform a structural assessment of the TMI-1 letdown nozzle repaired by weld overlay, following the requirements of the ASME Code Section III. The results of the calculation is documented in this report to certify that the repair meets the stress criteria and fatigue requirements of the ASME Code Section III 2004 Edition (Reference [2]).

The analysis is focused on the weld overlaid region for requirements on both stress distribution and fatigue failure criteria. The scope of the analysis includes the weld overlay, letdown elbow, weld between the elbow and the safe end, safe end, dissimilar metal weld between the safe end and the nozzle, letdown nozzle, and a portion of the cold leg.

A detailed finite element analysis (FEA) is performed to determine stress conditions under various operational transients. Two models are created due to the variations of the pipe wall thickness of the existing bent pipe configuration as documented in Reference [5]. One side is measured as [ ] thick (intrados) and the other side is measured as [ ] (extrados). Because of the difference in thickness, a test case is run to determine if a model with uniform pipe thickness of [ ] or [ ] gives the highest stress results. The more critical model is used for stress and fatigue evaluations.

Rev. 001: Revised what information in the document is marked with pairs of square braces "[ ]".

Rev. 002: Revised to remove proprietary statement and markings.

2.0 ANALYTICAL METHODOLOGY

The general methodology of the stress analysis consists of following steps:

1. Develop two 3D finite element model of the nozzle (with uniform pipe thicknesses of [ ] and [ ], respectively) with a welded curved pipe from the weld overlay drawings. Only the minimum overlay design configuration is used to develop the model in performing the analysis required by Reference [1] (see Section 3.0 for more details). The model incorporates the geometry of the CL letdown nozzle (including adjacent cold leg, nozzle, safe end, welds, weld overlay and the curved pipe), appropriate materials and boundary conditions. There are two finite element models consisting of thermal and structural elements, respectively so as to enable the thermal and structural analysis using ANSYS R13.0 (Reference [3]).

2. Run a test case using the heat up and cool down transients to determine which model (thick or thin pipe thickness) is most critical, see Appendix A.

3. Apply the design conditions (pressure and temperature) to the structural finite element model and obtain the deformation and stresses in the model. The deformation field is used to verify the correct behavior of the model and correct modeling of the boundary and load conditions.

4. Apply the thermal loads pertaining to the service level transients in the form of transient temperatures and corresponding heat transfer coefficients versus time. Each of the major service level transient requires a separate run on the thermal finite element model.

5. Review the results of the thermal analysis by examining the magnitude of temperature difference between critical locations in the model at all time points. Determine the critical time points for stress analysis.

6. Apply the corresponding mechanical (pressure) and thermal (nodal temperature) loads at each time point identified in Step 5 to the structural finite element model. Since the weld overlay configuration contains layers of different material having different coefficients of expansion, it is possible that one material is in compression and the other is in tension due to thermal expansion.

7. Define paths and linearize the stresses along the path to compute membrane and membrane+ bending stresses. The standard method in defining a path is to go from a free surface to a free surface. However, using this method ANSYS may average the stresses at the boundary of two material to compute the membrane and membrane + bending stresses. In addition to the free surface to free surface path, two partial paths (one in each material) are defined at the same location to ensure maximum stress intensities are captured. These paths will be used to check the $3S_m$ criteria and to obtain maximum K_e factor. It is recognized that no continuous and progressive displacement can occur in one of the materials without the other material restraining that displacement.

8. Calculate stresses due to nozzle external loads by manual computation to add to the stress results due to pressure and temperature effect.

9. Compare the primary + secondary stresses to the ASME Code criteria for acceptability. Because weld overlay adds material to the original structure, the primary stresses in the original design will bound those in the weld overlay design. Therefore, the primary stresses in the overlay design need not be checked against the Code allowables (see discussion in Section 6.4.1).

10. Perform the fatigue evaluation for each material.

11. Document the stresses and temperature for the fracture mechanics analysis of the letdown nozzle weld overlay design, see Appendix B.

3.0 ASSUMPTIONS

3.1 Unverified Assumptions

This analysis contains no assumptions that must be verified prior to use on safety-related work.

3.2 Justified Assumptions

Justified assumptions used for the analysis are listed as follows:

1. Since the pipe thickness around the elbow is varying, two finite element models (thick or thin pipe thickness) are built and tested under the transient conditions of heat up and cool down. The results are documented in Appendix A. It is concluded that the results from the thin pipe are bounding for most critical locations, and therefore, the model with thin pipe is used to complete the analysis.

2. The FSWOL design consists of two configurations in terms of the overlay thickness: the minimum and maximum conditions. Based on similar Section III analyses as well as studies on the nozzle DMW repair with Alloy 690, the stress intensity ranges and cumulative fatigue usage factors at most critical locations remain bounding when the analysis is performed using the minimum overlay thickness. Therefore, the minimum FSWOL configuration is then used in this analysis.

3.3 Modeling Simplifications

Simplifications in modeling and simulation used for the analysis are listed as follows:

1. The outside surface of the letdown nozzle and part of the cold leg modeled is insulated. However, a small heat transfer coefficient of [ ] Btu/hr-in^2-^{o}F is used in this calculation to account for imperfect insulation.

2. The weld between the safe end and elbow is considered symmetric and the actual pipe bend starts at the top of the weld.

3. The smaller overlay radius of [ ] taken from the minimum FSWOL thickness (Reference [6]) is modeled in the finite element analysis as it yields higher stress concentration. Element sizes around these transition regions are refined to ensure the convergence in an effort to capture the accurate peak stresses.

4. Based on the evaluation of temperature and pressure fluctuation, some transients are enveloped (see Table 4-8) by the bounding transient and the corresponding numbers of cycles are summed up in the fatigue assessment.

4.0 DESIGN INPUTS

4.1 Geometry

The detailed dimensions of the cold leg letdown nozzle existing configuration are shown in Reference [4]. Major dimensions used for building the finite element model include: cold leg inside radius ([ ] to base metal), cold leg thickness ([ ]), nozzle inside diameter ([ ]), and outside diameter ([ ] at nozzle end).

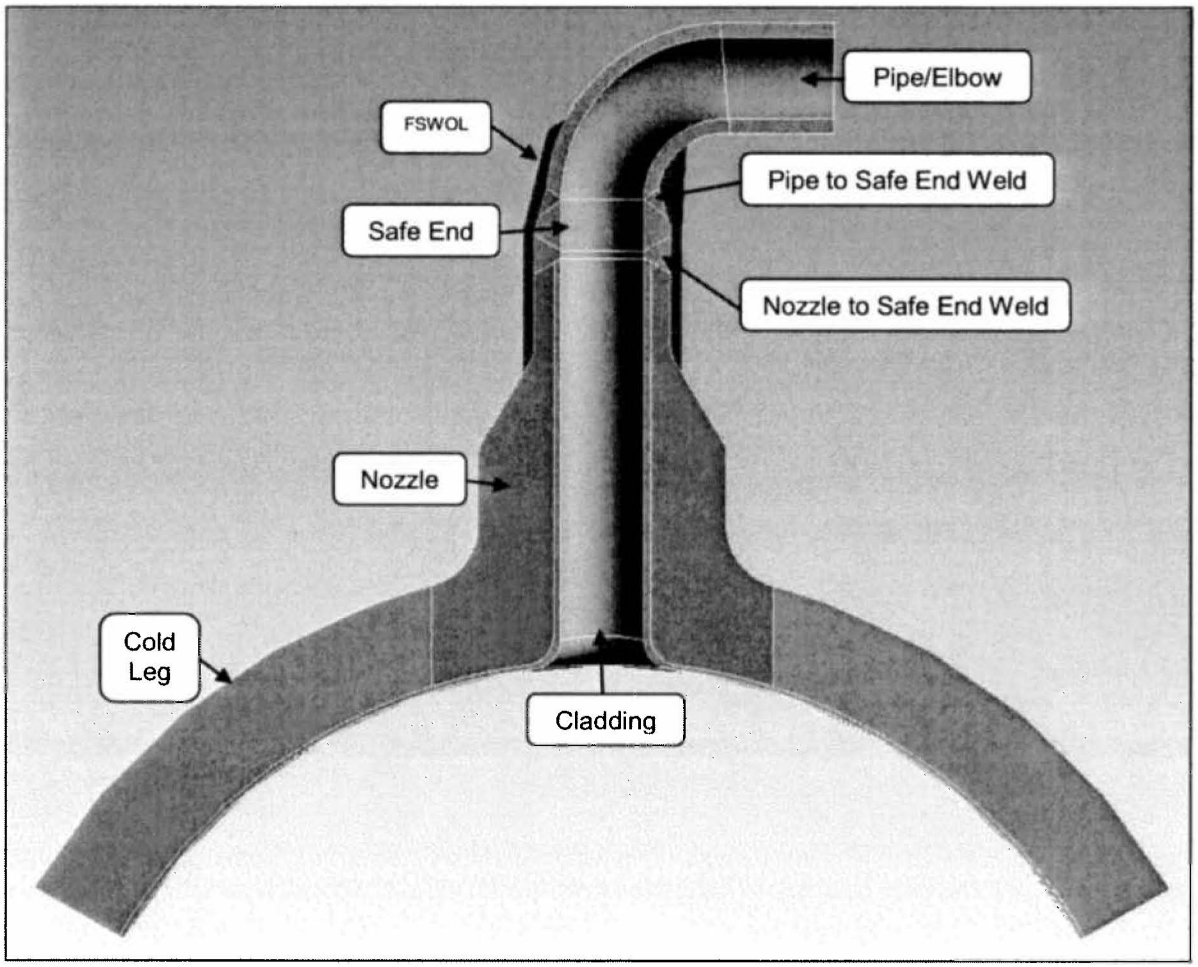


Figure 4-1: Nozzle-FSWOL Geometry

The pipe elbow is considered to be a [ ] Elbow with a thickness of [ ] . However, NDE results show that the thickness of the elbow ranges from [ ] to [ ] (Reference [5]). Two models are created, one with each thickness, and the resulting stresses for the [ ] transients ([ ]) are compared in Appendix A, which demonstrates that the thin pipe with the nominal thickness of [ ] bounds the model with the thicker pipe.

The weld overlay configurations are shown in Reference [6] for the minimum and maximum weld overlay. The minimum thickness of the weld overlay is [ ] . The weld overlay is tapered to the elbow on the extrados side and is angled out on the intrados side. At a distance of [ ] from the safe end to pipe weld, the thickness of the weld overlay is [ ] on the intrados and extrados side.

4.2 Finite Element Model

The finite element model is built based on the weld overlay design with the minimum weld overlay size. The model is developed in ANSYS R13.0 Workbench and the geometry file (nodes, elements, and components) created is found in [ ] .

The 3D model is meshed with SOLID187/186 (10 node / 20 node) elements for the structural analysis and SOLID87/90 (10 node / 20 node) elements for thermal analysis. The meshed model with the minimum weld overlay is shown in Figure 4-2. The meshed model with material properties (Section 4.3) is documented by [ ].

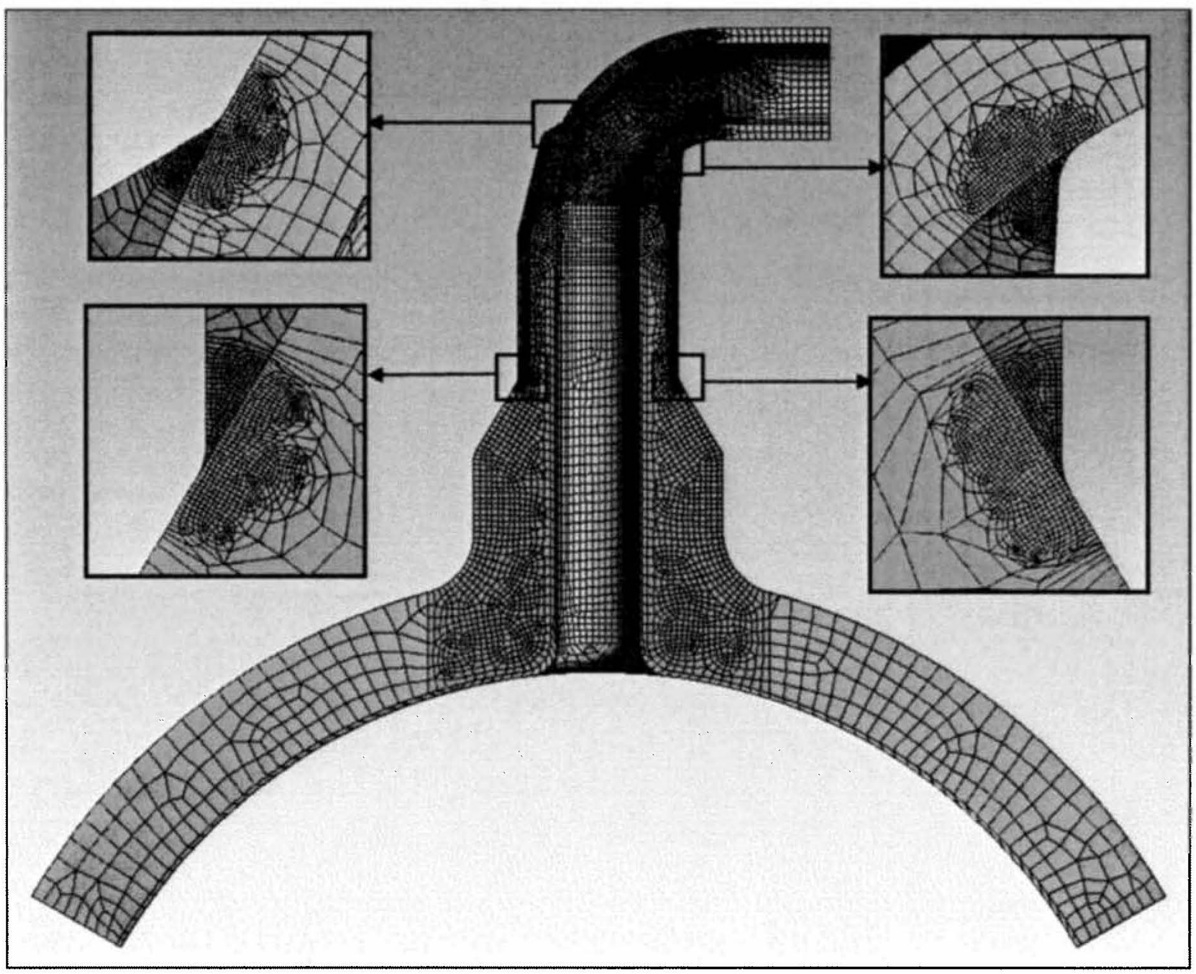


Figure 4-2: Finite Element Model (FEM)

4.3 Materials

Reference [1] provides the material designations for each component of the cold leg letdown nozzle. Material properties are found in References [7] (new material), [8] (existing material), and [9]. Since thermal Conductivity (k) values for original materials are not provided in Reference [8], they are taken from the next code year, Reference [10]. In addition, the piping material [ ] is also not included in Reference [8] and has material properties from the next code year, Reference [10]. The Weld Overlay is the only new material, all others are existing.

Cold Leg Piping
Letdown Nozzle
Nozzle to Safe End Weld
Safe End
Safe End to Letdown Piping Weld
Letdown Piping
Weld Overlay

[ ]

The following tables provide the material physical properties – mean coefficient of thermal expansion (α), specific heat (C), thermal conductivity (k), density (ρ), and the mechanical properties – modulus of elasticity (E), Poisson's ratio (μ). The units of data listed are:

Temperature	*Temp*	°F
Young's Modulus	E	10^6 psi
Poisson's Ratio	μ	unitless
Density	ρ	lb/in^3
Mean Coefficient of Thermal Expansion	α	10^{-6} in/in-°F
Thermal Conductivity	k	Btu/hr-in-°F
Specific Heat	C	Btu/lb-°F
Design Stress	S_m	ksi
Yield Strength	S_y	ksi
Ultimate Strength	S_u	ksi

Note that specific heat (C) is a calculated value: $C = k / (\rho * \textit{thermal diffusivity})$ where thermal diffusivity is taken from the same source as thermal conductivity k.

The material properties are read into the FEM from the ANSYS files: [ ].

Table 4-1: Cold Leg Piping

[ ]									
Temp, °F	E	μ	ρ	α	k	C	S_m	S_y	S_u
70									
100									
200									
300									
400									
500									
600									
650									
700									
Reference	[8]	typical	[9]	[8]	[10]	calculated	[8]	[8]	[8]

Table 4-2: Letdown Nozzle

[ ]									
Temp, °F	E	μ	ρ	α	k	C	S_m	S_y	S_u
70									
100									
200									
300									
400									
500									
600									
650									
700									
Reference	[8]	typical	[9]	[8]	[10]	Calculated	[8]	[8]	[8]

Table 4-3: Nozzle to Safe End Weld / Safe End / Safe End to Pipe Weld

[ ]									
Temp, °F	*E*	*μ*	*ρ*	*α*	*k*	*C*	S_m	S_y	S_u
70									
100									
200									
300									
400									
500									
600									
650									
700									
Reference	[8]	typical	[9]	[8]	[10]	Calculated	[8]	[8]	[8]

Table 4-4: Letdown Piping

[ ]									
Temp, °F	*E*	*μ*	*ρ*	*α*	*k*	*C*	S_m	S_y*	S_u
70									
100									
200									
300									
400									
500									
600									
650									
700									
Reference	[8]	typical	[9]	[8]	[10]	Calculated	[10]	[10]	[10]

* [ ] is not included in the S_y tables in Reference [10], therefore, the S_y values for the type [ ] material is used. This is reasonable as the S_y value listed in the S_m table for [ ], [ ] is [ ] ksi.

Table 4-5: Cladding

			[		]	
Temp, °F	*E*	μ	ρ	α	*k*	*C*
0						
100						
200						
300						
400						
500						
600						
650						
700						
Reference	[8]	typical	[9]	[8]	[10]	Calculated

Table 4-6: Weld Overlay

				[	]				
Temp, °F	*E*	μ	ρ	α	*k*	*C*	S_m	S_y	S_u
70									
100									
200									
300									
400									
500									
600									
650									
700									
Reference	[7]	typical	[9]	[7]	[7]	Calculated	[7]	[7]	[7]

4.4 Boundary Conditions

4.4.1 Thermal Analysis

During operation, the inside surfaces of the cold leg, letdown nozzle, nozzle to safe end weld, safe end, safe end to pipe weld, and pipe are in contact with the reactor coolant water coming out of the steam generator on its way back to the reactor. Appropriate heat transfer coefficients (HTCs) for the inside of the cold leg and inside of the letdown nozzle and piping are provided in Reference [11].

The outside surface of the cold leg, letdown nozzle, pipe, and weld overlay are exposed to ambient temperatures. A small HTC of [ ] Btu/ hr-in^2-°F is applied during all transient events to account for imperfect insulation.

All thermal boundary conditions are shown in Figure 4-3.

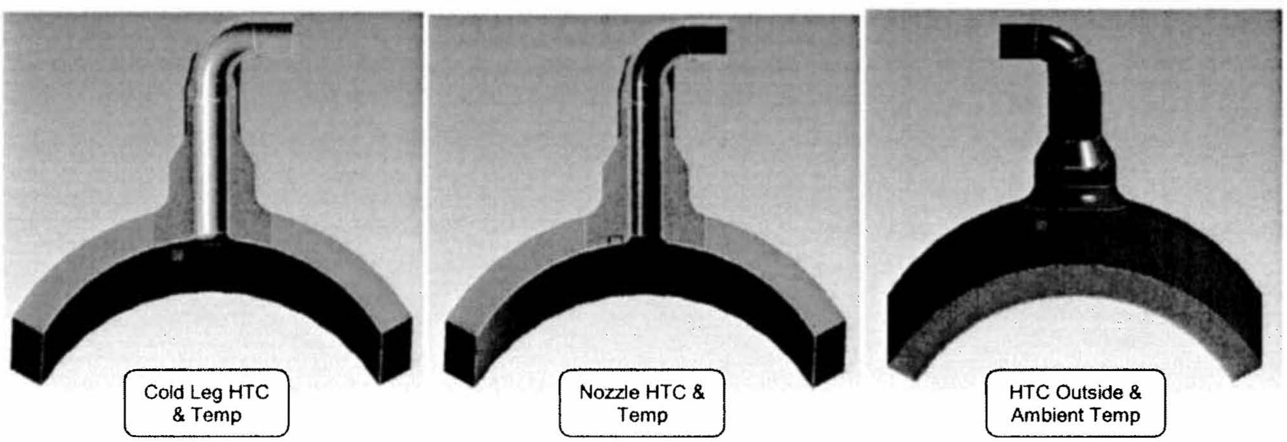


Figure 4-3: Thermal Boundary Conditions

4.4.2 Structural Analysis

The reactor coolant system pressure is applied to all interior surfaces of the cold leg, letdown nozzle, nozzle to safe end weld, safe end, safe end to pipe weld, and pipe. The upper end of the pipe has a pressure $p_{Pendcap}$ and the back section of the cold leg has a pressure $p_{CLendcap}$ applied to represent the hydrostatic end load.

The pressures are calculated (for the design condition) as:

$$p_{Pendcap} = \frac{-p \cdot r_{pipe}^2}{\left(R_{pipe}^2 - r_{pipe}^2\right)} = [\qquad]$$

$$p_{CLendcap} = \frac{-p \cdot r_{coldleg}^2}{\left(R_{coldleg}^2 - r_{coldleg}^2\right)} = [\qquad]$$

Where: p = pressure applied = [ ] psi* for design condition (Reference [12])
R_{pipe} = OD of the pipe = [ ] in for thickness [ ] (Reference [4])
r_{pipe} = ID of the pipe = [ ] in (Reference [4])
$R_{coldleg}$ = OD of cold leg = [ ] in (Reference [4])
$r_{coldleg}$ = ID of cold leg = [ ] in (Reference [4])

*While the design pressure is [ ] psig, the calculated difference in endcap pressures is negligible.

For other transients, the pressure at each time point is used to calculate the corresponding end cap pressures.

The boundary conditions for the structural analysis are set to have no displacement in the circumferential direction along the cylindrical planes of the cold leg and the symmetry plane.

All structural boundaries are shown in Figure 4-4.

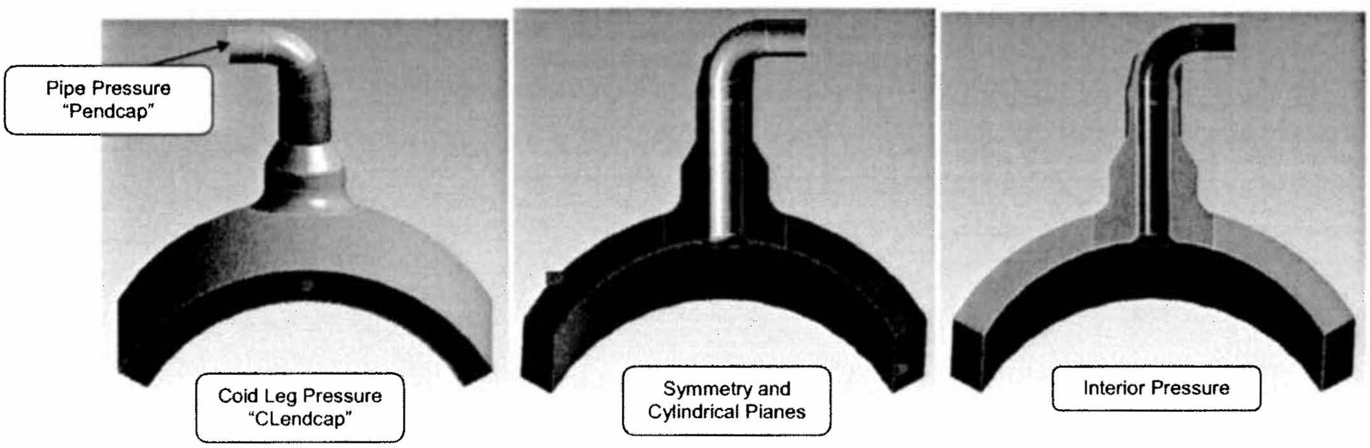


Figure 4-4: Structural Boundary Conditions

4.5 Loads

4.5.1 External Loads

Loads applied to the mode include temperatures and heat transfer coefficients for the thermal analysis and internal pressure for the structural analysis. External loads are shown in Table 4-7 per Reference [13] and are applied at the location between the nozzle and piping as shown in Figure 4-5.

Table 4-7: External Loads

Loading Condition	[ ]
Dead Load + OBE	
Dead Load + SSE	
Max Thermal	
Max Thermal + OBE	
Thermal Range	
OBE Range	
Bounding Case	

Note 1: [ ]

[ ]

Figure 4-5: External Load Orientation (Reference [13])

4.5.2 Design Conditions

The design pressure and temperature of the TMI-1 reactor coolant system is [ ] psig and [ ] °F (Reference [12]). These design conditions are simulated on the model by applying a uniform and reference temperature of [ ] °F throughout the model (the temperature is used to determine the material properties, not for thermal expansion) and a uniform pressure of [ ] psig on all inside surfaces of the model. The equivalent endcap pressures on the letdown piping and cold leg are also applied as described in Section 4.4.2.

4.5.3 Operational Transient Loads

The letdown nozzle is located on the cold leg. The inside surfaces are subjected to the cold leg temperatures and pressures as defined in Reference [12]. Temperatures and pressures along with the appropriate heat transfer coefficients (HTCs) are documented in Reference [11]. The applicable transients and number of cycles are listed in Table 4-8. Several transients are enveloped ([ ]) since their temperature and pressure fluctuation is bounded by the transient used.

Table 4-8: Transients and Number of Cycles

Transient	Description	# Cycles
[		]

5.0 COMPUTER USAGE

All computer files generated for this analysis and the Installation Test files have been uploaded to AREVA NP ColdStor found in the following directory: [ ].
All files listed in Table 5-1and Table 5-2 have been uploaded to ColdStor in October 2012.

5.1 Hardware

ANSYS R13.0 Service Pack 2 (Reference [3]) was run on computer "KBARNES3" with Windows 7 Enterprise Service Pack 1, 64 bit Operating System with 8 GB of RAM available.

5.2 Software Verification

The following EASI List computer program is used in the calculation:

ANSYS Release 13.0 SP2 (Reference [3])

ANSYS was tested on computer KBARNES3 by Kristine Barnes on 10/26/2012 and the results of the test were acceptable. Details of the verification tests have been included in the files listed in Table 5-1.

Table 5-1: Software Verification Runs

[ ]

5.3 Analysis Generated Files

The following table lists the computer files associated with the analysis and qualification of the TMI-1 Cold Leg Letdown Weld Overlay analysis.

Table 5-2: Analysis Computer Files

[ ]

6.0 CALCULATION

6.1 Design Condition

Stress analysis of the model under the design pressure and temperature provides a basis for verification of the expected behavior of the model, the boundary and load conditions. It also verifies attenuation of stress effects at regions away from the nozzle.

The ANSYS output for the design condition is documented in the following file:

[ ]

Figure 6-1 shows the deformed shape of the FSWOL model under design pressure. The stress intensity contour plot is shown in Figure 6-2.

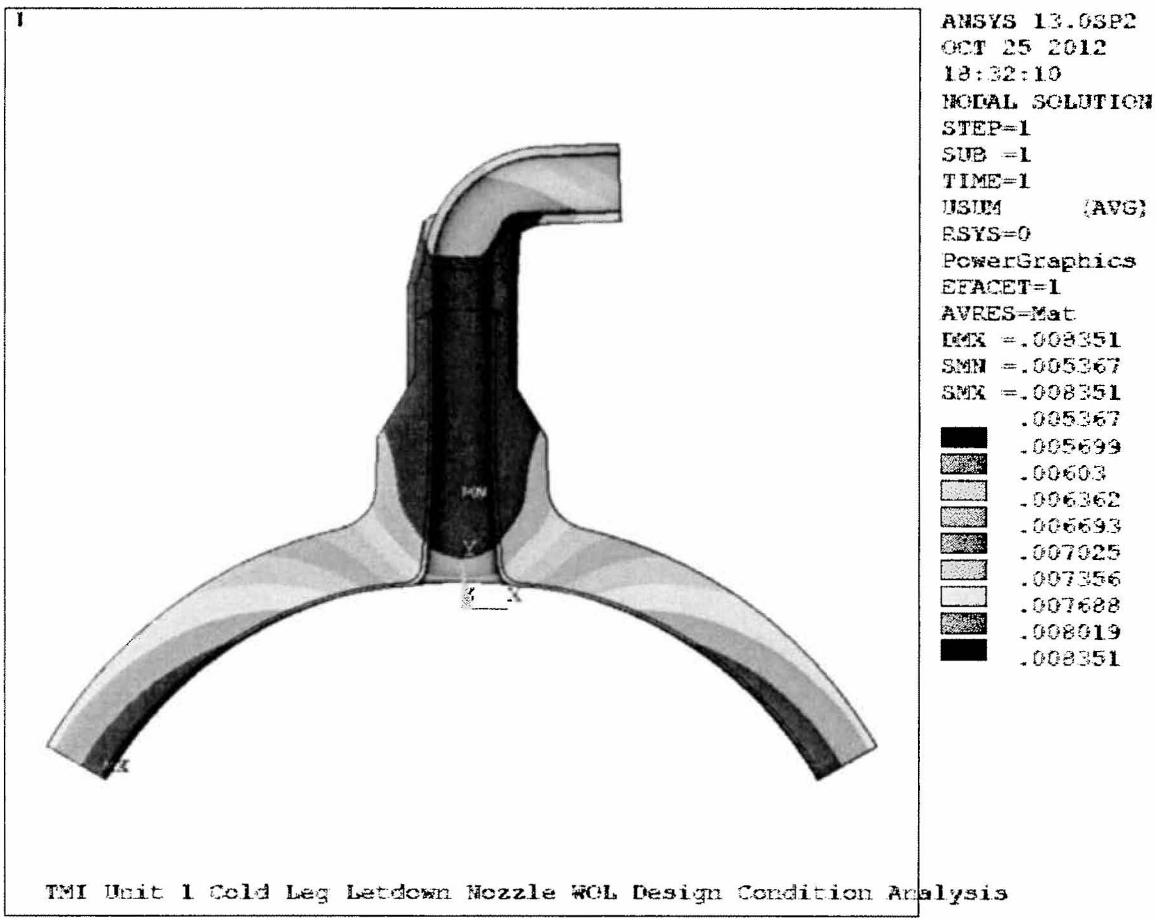


Figure 6-1: Deformed Shape for Design Condition

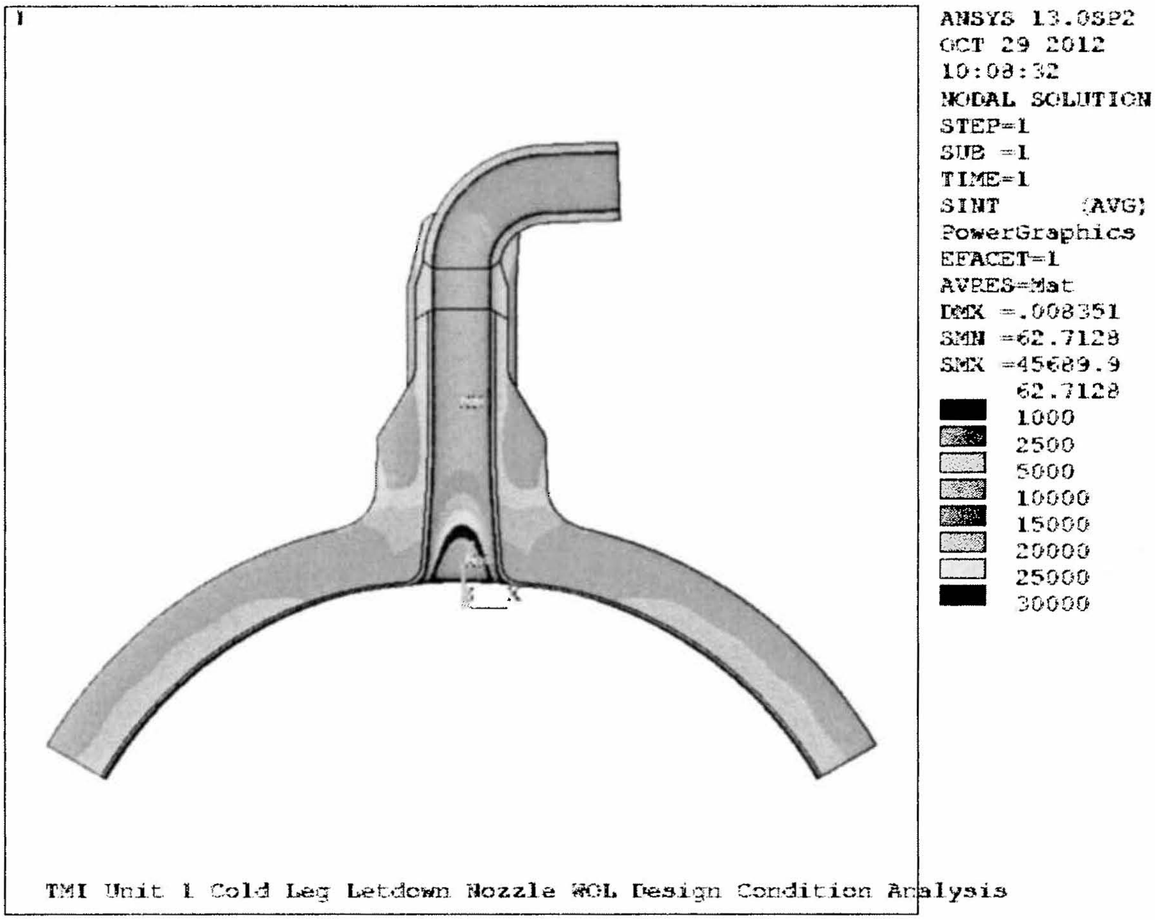


Figure 6-2: Stress Intensity Contours for Design Condition

6.2 Thermal Analysis

The ANSYS input files containing the transient definitions tabulated in Reference [11] are:

Table 6-1: Transient Temperature Files

[ ]

The thermal analysis output files including the temperature gradient output are as follows:

Table 6-2: Thermal Analysis Output Files

[ ]

The results of the thermal analysis are evaluated to identify the maximum and minimum temperature gradients between critical locations in the model and the corresponding time points. These temperature gradients generate maximum and minimum thermal stresses, which in in turn contribute to the maximum range of stress intensities in the model.

The locations for the evaluation of temperature gradients are listed by coordinates in Table 6-3. The locations are shown in Figure 6-3.

Table 6-3: Locations for Temperature Gradients

Path	Inside Coordinate			Outside Coordinate		
	X	Y	Z	X	Y	Z
A						
B						
C						
D						
E						
F						

[ ]

Figure 6-3: Approximate Locations for Temperature Gradient Evaluation

The temperature of selected nodes versus transient time as well as the temperature gradient are shown in Figure 6-4 to Figure 6-13. These figures are provided to show the trend and visual aid only. Specific data is taken from computer output files.

[ ]

Figure 6-4: Thermal Gradients at Selected Locations ([ ])

[ ]

Figure 6-5: Thermal Gradients at Selected Locations ([ ])

[ ]

Figure 6-6: Thermal Gradients at Selected Locations ([ ])

[ ]

Figure 6-7: Thermal Gradients at Selected Locations ([ ])

[ ]

Figure 6-8: Thermal Gradients at Selected Locations ([ ])

[ ]

Figure 6-9: Thermal Gradients at Selected Locations ([ ])

[ ]

Figure 6-10: Thermal Gradients at Selected Locations ([ ])

[ ]

Figure 6-11: Thermal Gradients at Selected Locations ([ ])

Figure 6-12: Thermal Gradients at Selected Locations ([ ])

Figure 6-13: Thermal Gradients at Selected Locations ([ ])

6.3 Structural Analysis

Nodal temperatures from the thermal analysis are input into the structural model within ANSYS. The time points selected for stress analyses are based on criteria such as pressure extremes, temperature gradient extremes as well as those of analytical interest. The time points of interest for the transient models are listed in Table 6-6 to Table 6-15. Stress analysis is performed for each of the listed time points. The ANSYS output files from the stress analysis are listed in Table 6-5.

The ANSYS input files containing the transient pressure definitions tabulated in Reference [11] are:

Table 6-4: Transient Pressure and Time Point Files

[ ]

Table 6-5: Structural Analysis Output Files

[ ]

Table 6-6: Time Points of Interest for [ ]

[ ]

Table 6-7: Time Points of Interest for [ ]

[ ]

Table 6-8: Time Points of Interest for [ ]

[ ]

Table 6-9: Time Points of Interest for [ ]

[ ]

Table 6-10: Time Points of Interest for [ ]

[ ]

Table 6-11: Time Points of Interest for [ ]

[ ]

Table 6-12: Time Points of Interest for [ ]

[ ]

Table 6-13: Time Points of Interest for [ ]

[ ]

*Duplicate time points resulted due to rounding. While this affects the time stamp for each step, the correct thermal and pressure load is applied together at the required intervals.

Table 6-14: Time Points of Interest for [ ]

Table 6-15: Time Points of Interest for [ ]

[ ]

*Duplicate time points resulted due to rounding. While this affects the time stamp for each step, the correct thermal and pressure load is applied together at the required intervals.

6.4 ASME Code Criteria

The ASME Code qualification (Reference [2]) involves two basic sets of criteria:

1. Assure that failure does not occur due to the application of the design loads.
2. Assure that failure does not occur due to repetitive loading.

In general, the primary stress intensity criteria of the ASME Code assure that the design is adequate for application of the design loads. The ASME Code criteria for cumulative fatigue usage factor assures that the design is adequate for repetitive loadings.

6.4.1 ASME Code Primary Stress Intensity Criteria

Per NB-3213.8 of Reference [2], the primary stresses are those normal or shear stresses developed by an imposed loading such as internal pressure and external loadings. A thermal stress is not classified as a primary stress. The classification as well as the limit of primary stress intensity is specified in NB-3221 of Reference [2] for design condition. The limit of primary stress intensity of Level B (Upset), Level C (Emergency), Level D (Faulted), and Test Conditions are specified in NB-3223, NB-3224, NB-3224, and NB-3225 of Reference [2] respectively.

The primary stress intensity criteria are the basic requirements in calculating the weld overlay size which is under the assumption that a 360° circumferential flaw has grown through the original weld. Loading conditions in each service level have been considered in the weld overlay sizing calculation. The nozzle to pipe region has been reinforced by the weld overlay since adding material to the nozzle outside region reduces primary stresses resulting from internal pressure and external loads. The overlay further reduces stress concentrations by eliminating the outside surface discontinuity. Therefore, the primary stress requirement for the nozzle, welds with overlay, safe end, and pipe have been satisfied for all service level loadings without the need for further evaluation.

Other related criteria include the minimum required thickness (NB-3324 of Reference [2]), and reinforcement area (NB-3330 of Reference [2]), which were addressed in the original nozzle/cold leg designs. Adding weld overlay will increase the nozzle wall thickness, and therefore, these requirements are satisfied.

6.4.2 ASME Code Primary + Secondary Stress Intensity Range and Fatigue Usage Criteria

The stress analysis for transient conditions is required for a component to satisfy the requirements for repetitive loadings.

Computer runs for each transient time point selected for stress analysis are contained in the computer output files listed in Section 5.3. The overall stress profile is then reviewed to determine the critical locations that require detailed stress/fatigue analysis. The objective to assure that (1) the most severely stressed locations are evaluated and (2) the specified region is quantitatively qualified.

Once the specific locations for detailed stress evaluation are established, the related path lines can be defined for input to ANSYS. ANSYS post-processor POST1 is used to linearize stresses along the path lines.

The path lines selected for primary plus secondary stress range calculation and fatigue failure evaluation are listed in Table 6-16. The approximate location of the full paths and partial paths is shown in Figure 6-14.

Table 6-16: Path Lines for Linearized Stresses

	Inside Coordinate			Outside Coordinate			Material(1)	
	X	Y	Z	X	Y	Z	Inside	Outside
Path1(2)								
Path2								
Path3								
Path3a								
Path3b								
Path4								
Path4a								
Path4b								
Path5								
Path5a								
Path5b								
Path6								
Path6a								
Path6b								
Path7								
Path7a								
Path7b								
Path8(3)								
Path9								
Path9a								
Path9b								
Path10								
Path10a								
Path10b								
Path11								
Path11a								
Path11b								
Path12								
Path12a								
Path12b								
Path13								
Path13a								
Path13b								
Path14(3)								

(1) Materials: [ ]
(2) Node for the inside of Path1 is selected 1 element in excluding the cladding. The affect is negligible as this is not a critical location.
(3) Nodes on Path8 and Path14 are selected to ensure that the highest Total stress ranges are obtained.

[ ]

Figure 6-14: Approximate Locations of Path Lines for Stress Analysis

The definition of these path lines, linearized stress components, and stress intensity ranges (M+B and Total) for these paths are contained in the following output files:

Table 6-17: Stress Intensity Ranges

[ ]

6.4.2.1 Maximum Primary + Secondary Stress Intensity Range NB-3222.2

The enveloped external loads are listed in Table 4-7. These loads which cause periodic stress changes need to be included in calculating the maximum stress intensity ranges. Except for PATH1 where the stress variation due to external loads is negligible, the stress intensities due to enveloping external load is calculated at all other path locations. The geometric characteristics of each cross section at these path locations are listed in Table 6-18.

Where:

D	- Outside diameter (in)
d	- Inside diameter (in)
$I = \frac{\pi}{64}\left(D^4 - d^4\right)$	- Moment of inertia (in^4)
$S_{OD} = \frac{I}{D/2}$	- Section modulus of the nozzle: outside diameter (in^3)
$S_{ID} = \frac{I}{d/2}$	- Section modulus of the nozzle: inside diameter (in^3)

Table 6-18: Geometric Characteristics of Path Line Cross Section

	Node Coordinates (Inside)			Node Coordinates (Outside)							
	X	Y	Z	X	Y	Z	ID	OD	I	S_{OD}	S_{ID}
Path2	[										
Path3											
Path4/Path9											
Path5/Path10											
Path6											
Path7[1]											
Path8/Path14[2]											
Path11											
Path12[1]											
Path13[1]											]

Note 1: The ID of the path is the same as the ID of the pipe. The OD of the path is calculated as 2 * the distance between the inside and outside nodes ($2 \cdot \sqrt{(x_{in} - x_{out})^2 + (y_{in} - y_{out})^2}$) + ID.

Note 2: The ID and OD are the nominal dimensions of a [ ] .

The membrane + bending stress intensities due to external loads are calculated as follows:

$\sigma_{ax\_Mb} = \frac{M_b}{S}$	- Axial bending stress due to external bending moment (M_b) (ksi)
$\tau_{s\_Mt} = \frac{M_t}{2 \cdot S}$	- Shear stress due to external torsion moment (M_t) (ksi)
$S_{int} = \sqrt{\sigma^2_{ax\_Mb} + 4 \cdot \tau^2_{s\_Mt}}$	- Membrane + Bending stress intensity range (ksi)

Where $S = S_{ID}$ for the inside diameter and S_{OD} for the outside diameter.

Stress intensities at the inside and outside nodes of the selected path lines are computed and listed in Table 6-19.

Table 6-19: Stress Intensities due to External Loads (ksi)

	Inside Node			Outside Node		
	σ_{ax_Mb}	τ_{s_Mt}	S_{int}	σ_{ax_Mb}	τ_{s_Mt}	S_{int}
Path2						
Path3						
Path4/Path9						
Path5/Path10						
Path6						
Path7						
Path8/Path14						
Path11						
Path12						
Path13						

The summary of maximum stress intensity ranges including external loads is listed in Table 6-20. As shown in Table 6-20, the $3S_m$ Primary + Secondary stress intensity limit (NB-3222.2 of Reference [2]) has been met for all locations.

Table 6-20: Membrane + Bending Stress Intensity Range Summary (ksi)

	Inside Node				Outside Node			
	M+B SI Range	Ext. Load SI Range	Total M+B SI Range	$3S_m$ Limit @ 650 °F	M+B SI Range	Ext. Load SI Range	Total M+B SI Range	$3S_m$ Limit @ 650 °F
Path1								
Path2								
Path3								
Path3a								
Path3b								
Path4								
Path4a								
Path4b								
Path5								
Path5a								
Path5b								
Path6								
Path6a								
Path6b								
Path7								
Path7a								
Path7b								
Path8								
Path9								
Path9a								
Path9b								
Path10								
Path10a								
Path10b								
Path11								
Path11a								
Path11b								
Path12								
Path12a								
Path12b								
Path13								
Path13a								
Path13b								
Path14								

* Note: As documented in [ ], the maximum range is between [ ], time point 16 and [ ], time point 20. The maximum temperature of those two time points is [ ] °F and is used to find the $3S_m$ allowable limit of [ ] ksi (= [ ]).

6.4.2.2 Fatigue Usage Factor NB-3222.4

In order to calculate the fatigue usage factors per Section NB-3222.4 of the ASME Code (Reference [2]), the total stress intensity ranges are computed and documented in the following ANSYS files:

[ ]

Total stress is used to ensure peak stresses are accounted for that may not be depicted in membrane plus bending stress intensity ranges. The summary of total stress intensity ranges for all paths is shown in Table 6-21.

Since the external load stress intensity is calculated as the membrane + bending load a stress concentration factor is applied to them before being added to the transient total stress intensity range. Per Reference [14], a stress concentration factor for a bar with fillets between the transition is approximately [ ] for a member with an r/d ratio [ ]. A review of the total stress ratio to membrane + bending stress at critical time points in the 1A transient produces a stress concentration ratio closer to [ ]. Therefore, a factor of [ ] is conservatively applied to the external loads.

Table 6-21: Total Maximum Primary + Secondary Stress Intensity Range Summary (ksi)

	Inside Node			Outside Node		
	Total SI Range	Ext. Load SI Range	Total + Ext. SI Range	Total SI Range	Ext. Load SI Range	Total + Ext. SI Range
Path1						
Path2						
Path3						
Path3a						
Path3b						
Path4						
Path4a						
Path4b						
Path5						
Path5a						
Path5b						
Path6						
Path6a						
Path6b						
Path7						
Path7a						
Path7b						
Path8						
Path9						
Path9a						
Path9b						
Path10						
Path10a						
Path10b						
Path11						
Path11a						
Path11b						
Path12						
Path12a						
Path12b						
Path13						
Path13a						
Path13b						
Path14						

Based on a review of the Stress Intensity range results in Table 6-20 the following paths will produce the highest cumulative fatigue usage factors (CFUF):

1. Path3/3a Inside Node for Nozzle material [ ]
2. Path11/11a Inside Node for Safe End/Safe End Weld material [ ]
3. Path8 Outside Node for Pipe material [ ]
4. Path13b Inside Node for FSWOL material [ ]

The CFUF values for these critical locations will bound the CFUF at all other locations. All selected locations use maximum Total stress intensity ranges and have a K_e value of 1.0.

Note that only stress intensity ranges and corresponding transient extremes are taken from the output files listed previously in this section since stress intensities due to external loads are not included in the ANSYS output files. The maximum stress intensity due to external loads is conservatively added to every SI range except for the pipe and FSWOL locations. At the pipe and FSWOL locations (Table 6-24 and Table 6-25), the maximum SI range is added to the cycles from transients [ ] and [ ] [ ]). For all other transients, the thermal range external loads listed in Reference [13] are added to the transient SI range since Note B of Reference [13] states that the thermal range is applicable to thermal transient cycles that are not associated with heat up and cool down. Per Reference [13], the thermal range external loads are half of the maximum external loads.

Table 6-22 to Table 6-25 provide the calculation of the CFUFs based on the loads and cycles in Table 4-8. The values of E_{curve} and allowable cycles are taken from Figures I-9.1 and I-9.2.1 and Table I-9.1 of Reference [2].

Table 6-22: [ ] Material CFUF (Path3/3a)

ANSYS File: [ ]
Path: Path3/3a Inside — K_e = 1.0
Material: [ ]
UTS (Su) (psi) =
E_{curve} (psi) =
E_{matl} (psi @T=[ ]°F)= [ ]
$E_{ratio} = E_{curve}/E_{matl}$ = [ ]

Range Index	Transients Extreme	Req'd Cycles	SI_{Total}	$SI_{External}$	S_{alt}	E_{ratio} x S_{alt}	Allowable Cycles	Usage Factor
1	[							
2								
3								
								]

Table 6-23: [ ] Material CFUF (Path11/11a)

ANSYS File:[ ]
Path: Path11/11a Inside — K_e = 1.0
Material:[ ]
UTS (Su) (psi) =
E_{curve} (psi) =
E_{matl} (psi @T=[ ]°F) = [ ]
$E_{ratio} = E_{curve}/E_{matl}$ = [ ]

Range Index	Transients Extreme	Req'd Cycles	SI_{Total}	$SI_{External}$	S_{alt}	E_{ratio} x S_{alt}	Allowable Cycles	Usage Factor
1	[							
2								
3								
								]

Table 6-24: [ ] Material CFUF (Path8)

ANSYS File: [ ]
Path: Path8 Outside K_e = 1.0
Material: [ ]

UTS (Su) (psi) = [
E_{curve} (psi) =
E_{matl}(psi @T=[ ]°F) =] $E_{ratio} = E_{curve}/E_{matl}$ = [ ]

Range Index	Transients Extreme	Req'd Cycles	SI_{Total}	$SI_{External}$*	S_{alt}	E_{ratio} x S_{alt}	Allowable Cycles	Usage Factor
1	[							
2								
3								
4								
5								
6								
7								
8								
9								
								]

*Maximum external loads added to [ ] and [ ]. Thermal range external loads (half maximum) added to all other transients.

Table 6-25: [ ] Material CFUF (Path13b)

ANSYS File: [ ]
Path: Path13b Inside K_e = 1.0
Material: [ ]
UTS(Su)(psi@ T=[ ]°F) = [
E_{curve} (psi) =
E_{matl} (psi @ T=[ ]°F) =] $E_{ratio} = E_{curve}/E_{matl}$ = [ ]

Range Index	Transients Extreme	Req'd Cycles	SI_{Total}	$SI_{External}$*	S_{alt}	E_{ratio} x S_{alt}	Allowable Cycles	Usage Factor
1	[							
2								
3								
								]

*Maximum external loads added to [ ] and [ ]. Thermal range external loads (half maximum) added to all other transients.

7.0 RESULTS/CONCLUSIONS

Stress analysis and design qualification for the TMI-1 cold leg letdown nozzle with weld overlay is performed in this calculation following the requirements of the ASME Code (Reference [2]).

The TMI-1 letdown nozzle with weld overlay satisfies the ASME Code primary and primary plus secondary stress requirements as well as the criteria for fatigue. The primary stress criteria are satisfied as described in Section 6.4.1. The primary plus secondary stress criteria and fatigue requirements are evaluated in Section 6.4.2. The summary of the maximum primary plus secondary, membrane + bending stress intensity ranges and fatigue usage factors are listed in Table 7-1.

Table 7-1: M+B Stress Intensity Ranges and CFUFs

Component	Material	Maximum M+B Range (ksi)		CFUFs	
		Calculated	Allowable	Calculated	Allowable
Nozzle	[				
Safe End / Safe End Welds					
Pipe					
FSWOL					]

Based on the loads and cycles specified in References [1], [11], and [12], the fatigue analysis performed in this calculation indicates that the maximum fatigue usage factor for the cold leg letdown nozzle weld overlay design is [ ].

8.0 REFERENCES

1. AREVA NP Document 08-9182964-002, "TMI 'C' Cold Leg Letdown Nozzle Weld Overlay."
2. ASME Boiler and Pressure Vessel Code, Section III, Division I, 2004 Edition, No Addenda.
3. ANSYS Release 13.0 SP2, ANSYS Inc., Canonsburg, Pa.
4. AREVA NP Drawing 02-9185282C-000, "TMI Letdown Nozzle Existing Configuration."
5. AREVA NP Document 38-9187200-000, "Letdown Line Nozzle Field Data."
6. AREVA NP Drawing 02-8059673D-003, "TMI Letdown Nozzle Weld Overlay Design."
7. ASME Boiler and Pressure Vessel Code, Section II, Materials, 2004 Edition, No addenda.
8. ASME Boiler and Pressure Vessel Code, Section III, Division 1, 1965 Edition, including Addenda through Summer 1967.
9. AREVA NP Document NPGD-TM-500, Rev D, "NPGMAT-NPGD Material Properties Program User's Manual," March 1985.
10. ASME Boiler and Pressure Vessel Code, Section III, Division 1, 1971 Edition.
11. AREVA NP Document 51-9187446-001, "TMI Letdown Nozzle Weld Overlay Design Transients."
12. AREVA NP Document 18-1173549-006, "Reactor Coolant System for Three Mile Island Unit One."
13. AREVA NP Drawing 02-163313E-03, "Reactor Coolant System Nozzle Loadings."
14. Popov, E.P., "Mechanics of Materials," Second Edition, 1976.

APPENDIX A: PIPE THICKNESS COMPARISON

A.1 Model Comparison

Two models were created to compare the nominal pipe thickness of [ ] (thin model) with the maximum thickness measured by the NDE evaluation (Reference [5]) of [ ] (thick model). The only dimension changed between the two models was the pipe thickness. The rest of the dimensions around the Safe End to Pipe weld and the FSWOL were all allowed to land within the confines of the dimensions given in References [4] and [6].

[ ]

Figure A-1: Pipe Thickness Difference

The temperatures and pressures for transients [ ] and [ ] (Section 4.5.3) were applied to each model and the linearized stresses along the path lines shown in Section 6.4.2. The Membrane + Bending and Total stress intensities for each model are compared in Table A-2.

Table A-1 shows the summary of the maximum M+B and Total stress intensities for each material. It also list if the stress comes from the thick or thin model. As seen in Table A-1, most of maximum stresses are found in the thin model with the nominal pipe thickness. For the places where the maximum stress is found in the thick pipe mode, the maximum stresses in the thin pipe model are not significantly lower. Therefore, the nominal pipe thickness will be used for the full qualification as it bounds the thick model.

Table A-1: Summary of Thin vs. Thick Comparison

Material	Max M+B (ksi)	Path	Controlling Pipe Thickness	Material	Max Total (ksi)	Path	Controlling Pipe Thickness
Nozzle		Path 3	Thick	Nozzle		Path 3/3a	Thick
Safe End		Path 11a	Thin	Safe End		Path 11/11a	Thin
Pipe		Path 8	Thin	Pipe		Path 14	Thick
FSWOL		Path 13b	Thin	FSWOL		Path 13b	Thin

Table A-2: Thick vs. Thin Stress Intensities (ksi)

Path	M+B (ksi)			M+B (ksi)		
	Inside Nodes			Outside Nodes		
	Thin	Thick	Delta	Thin	Thick	Delta
Path1						
Path2						
Path3						
Path3a						
Path3b						
Path4						
Path4a						
Path4b						
Path5						
Path5a						
Path5b						
Path6						
Path6a						
Path6b						
Path7						
Path7a						
Path7b						
Path8						
Path9						
Path9a						
Path9b						
Path10						
Path10a						
Path10b						
Path11						
Path11a						
Path11b						
Path12						
Path12a						
Path12b						
Path13						
Path13a						
Path13b						
Path14						

Path	Total (ksi)			Total (ksi)		
	Inside Nodes			Outside Nodes		
	Thin	Thick	Delta	Thin	Thick	Delta
Path1						
Path2						
Path3						
Path3a						
Path3b						
Path4						
Path4a						
Path4b						
Path5						
Path5a						
Path5b						
Path6						
Path6a						
Path6b						
Path7						
Path7a						
Path7b						
Path8						
Path9						
Path9a						
Path9b						
Path10						
Path10a						
Path10b						
Path11						
Path11a						
Path11b						
Path12						
Path12a						
Path12b						
Path13						
Path13a						
Path13b						
Path14						

APPENDIX B: STRESSES FOR FRACTURE ANALYSIS

B.1 Component Stresses for Fracture Analysis

This section provides supplemental stress and thermal results of the transient analyses for the fracture mechanics analysis of the cold leg letdown weld overlay.

For stress and temperature evaluation, the paths are defined through the nozzle at the weld overlay region. The locations are the same as those used in the main body of this calculation for the structural analysis shown in Figure 6-14. Table B-1 lists the new fracture path names.

Table B-1: Paths for Fracture Mechanics Evaluation

Fracture Path Name	Structural Path Name
FR_1	Path4
FR_2	Path5
FR_3	Path6
FR_4	Path9
FR_5	Path10
FR_6	Path11

The stresses are evaluated in the global coordinate system with the Y axis oriented along the nozzle axis, X in the radial, and Z in the hoop direction. The axial (S_y) stresses, hoop (S_z) stresses and temperatures are listed at [ ] equidistant intervals ([ ] locations) along the path for all transients and time points. These equidistant intervals (distance along the path from the inside node) for all 6 paths are documented in "[ ]". Output files containing the stresses (S_y and S_z) and temperature (T) values for all transients are listed in Table 5-2. The units of the tabulated data in the files are provided in Table B-2.

Table B-2: File Names and Units of ANSYS Output

ANSYS Output	Unit	ANSYS Output File Name
[		]

Where * is the name of the transient.

Data listed in the stress and temperature output fields are ordered as follows:

First Column: Path Number

Second Column: Time

Third to Last Column: Stress or Temperature along the Path

Attachment 3

Affidavit

AFFIDAVIT

COMMONWEALTH OF VIRGINIA)
) ss.
CITY OF LYNCHBURG)

1. My name is Gayle F. Elliott. I am Manager, Product Licensing, for AREVA NP Inc. (AREVA NP) and as such I am authorized to execute this Affidavit.

2. I am familiar with the criteria applied by AREVA NP to determine whether certain AREVA NP information is proprietary. I am familiar with the policies established by AREVA NP to ensure the proper application of these criteria.

3. I am familiar with the AREVA NP information contained in the Calculation Summary Sheets (CSS) 32-9183944-003, entitled "TMI-1 Letdown Nozzle Weld Overlay Sizing Calculation," dated February 2013, 32-9185635-002, entitled "TMI-1 Letdown Nozzle Weld Overlay Section III Analysis," dated February 2013, and 32-9186192-002, entitled, "TMI Unit 1 Weld Residual Stress Analysis for CL Letdown Nozzle Weld Overlay," dated February 2013 and referred to herein as "Documents." Information contained in these Documents has been classified by AREVA NP as proprietary in accordance with the policies established by AREVA NP for the control and protection of proprietary and confidential information.

4. These Documents contain information of a proprietary and confidential nature and is of the type customarily held in confidence by AREVA NP and not made available to the public. Based on my experience, I am aware that other companies regard information of the kind contained in these Documents as proprietary and confidential.

5. These Documents have been made available to the U.S. Nuclear Regulatory Commission in confidence with the request that the information contained in these Documents

be withheld from public disclosure. The request for withholding of proprietary information is made in accordance with 10 CFR 2.390. The information for which withholding from disclosure is requested qualifies under 10 CFR 2.390(a)(4) "Trade secrets and commercial or financial information."

6. The following criteria are customarily applied by AREVA NP to determine whether information should be classified as proprietary:

(a) The information reveals details of AREVA NP's research and development plans and programs or their results.

(b) Use of the information by a competitor would permit the competitor to significantly reduce its expenditures, in time or resources, to design, produce, or market a similar product or service.

(c) The information includes test data or analytical techniques concerning a process, methodology, or component, the application of which results in a competitive advantage for AREVA NP.

(d) The information reveals certain distinguishing aspects of a process, methodology, or component, the exclusive use of which provides a competitive advantage for AREVA NP in product optimization or marketability.

(e) The information is vital to a competitive advantage held by AREVA NP, would be helpful to competitors to AREVA NP, and would likely cause substantial harm to the competitive position of AREVA NP.

The information in these Documents is considered proprietary for the reasons set forth in paragraphs 6(c) and 6(d) above.

7. In accordance with AREVA NP's policies governing the protection and control of information, proprietary information contained in these Documents have been made available, on a limited basis, to others outside AREVA NP only as required and under suitable agreement providing for nondisclosure and limited use of the information.

8. AREVA NP policy requires that proprietary information be kept in a secured file or area and distributed on a need-to-know basis.

9. The foregoing statements are true and correct to the best of my knowledge, information, and belief.

SUBSCRIBED before me this 1st

day of March, 2013.

Sherry L. McFaden
NOTARY PUBLIC, COMMONWEALTH OF VIRGINIA
MY COMMISSION EXPIRES: 10/31/14
Reg. # 7079129

SHERRY L. MCFADEN
Notary Public
Commonwealth of Virginia
7079129
My Commission Expires Oct 31, 2014