Attachment 7

"Limerick Unit 2 Instrument Nozzle N-16D Repair Weld Residual Stress Analysis (Non-Proprietary)," AREVA Document No. 32-9277502-000, Non-Proprietary Version

AREVA

CALCULATION SUMMARY SHEET (CSS)

Document No. 32 - 9277502 - 000 Safety Related: ☒Yes ☐ No

Title Limerick Unit 2 Instrument Nozzle N-16D Repair Weld Residual Stress Analysis (Non-Proprietary)

PURPOSE AND SUMMARY OF RESULTS:

The purpose of this report is to document the results of the weld residual stress finite element analysis of the Reactor Vessel (RV) instrument nozzle N-16D penetration as-left J-groove weld at Limerick Unit 2 Nuclear Power Plant. This analysis includes weld simulation of the original butter and J-groove weld (including the inner diameter overlay) attaching the remnant instrument nozzle to the reactor vessel shell. The analysis also includes simulation of the recent weld repair involving outer diameter weld pad and J-groove weld of the replacement nozzle. The state of stress at the end of final welding, as predicted by the ANSYS Version 16.0 finite element analysis, is summarized to support flaw evaluations of the as-left (original) J-groove weld. The ANSYS computer model files including residual stress results are available for use in the subsequent fracture mechanics analysis.

Rev 000: The proprietary version of this document is 32-9274303-001.

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

If the computer software used herein is not the latest version per the EASI list, AP 0402-01 requires that justification be provided.

THE FOLLOWING COMPUTER CODES HAVE BEEN USED IN THIS DOCUMENT:

CODE/VERSION/REV	CODE/VERSION/REV
ANSYS 16.0	

THE DOCUMENT CONTAINS ASSUMPTIONS THAT SHALL BE VERIFIED PRIOR TO USE

☐Yes

☒ No

Review Method: ☒ Design Review (Detailed Check)

☐ Alternate Calculation

Does this document establish design or technical requirements? ☐ YES ☒ NO

Does this document contain Customer Required Format? ☐ YES ☒ NO

Signature Block

Name and Title (printed or typed)	Signature	P/R/A/M and LP/LR	Date	Pages/Sections Prepared/Reviewed/Approved
Tom Riordan Engineer IV	[signature]	P	04OCT2017	All
Ashok D Nana Supervisory Engineer	[signature]	R	10/4/17	All
David R Cofflin Engineering Manager	[signature]	A	10/4/17	All

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

In preparing, reviewing and approving revisions, the lead preparer/reviewer/approver shall use 'All' or 'All except ___' in the pages/sections reviewed/approved. 'All' or 'All except ___' means that the changes and the effect of the changes on the entire document have been prepared/reviewed/approved. It does not mean that the lead preparer/reviewer/approver has prepared/reviewed/approved all the pages of the document.

Project Manager Approval of Customer References and/or Customer Formatting (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 Release
	All	The proprietary version of this document is 32-9274303-001.

Table of Contents

Page

List of Tables

Page

List of Figures

1.0 INTRODUCTION

The purpose of this report is to document the results of the weld residual stress finite element analysis of the RV instrument nozzle N-16D penetration as-left J-groove weld at Limerick Unit 2 Nuclear Power Plant. This analysis includes weld simulation of the original butter and J-groove weld (including the inner diameter (ID) overlay) attaching the instrument nozzle to the reactor vessel (RV) shell. The analysis also includes simulation of the recent weld repair involving outer diameter weld pad and repair J-groove weld attaching the replacement nozzle. As shown in Reference [3], the repair process involved removing the outer portion of the original nozzle, welding a weld pad at the outer diameter and welding of replacement nozzle to the weld pad. The state of stress as predicted by the ANSYS Version 16.0 finite element analysis at the end of welding steps is provided in this report to support fracture mechanics evaluation of a postulated flaw in the as-left J-groove weld.

2.0 ANALYTICAL METHODOLOGY

The analytical methodology used to predict the weld induced residual stresses in the as-left J-groove weld involves using a three-dimensional finite element model. [

] The [ ] model used to represent the instrument nozzle penetration is shown in Figure 2-1.

Figure 2-1: The Solid Model of Instrument Nozzle N16D used in the Analysis

[

]

Figure 2-2: The Detailed View of Welds in the Solid Model

[ ]

The parts shown by different colors in Figure 2-2 are the reactor vessel shell, the cladding of the RV shell, the remnant nozzle, the butter and J-groove weld, the ID overlay, the existing weld buildup, the repair weld pad and repair J-groove weld attaching replacement nozzle to the weld pad.

2.1 Welding Analysis Methodology

The WRS (Weld Residual Stress) finite element analysis is carried out per the WRS analysis procedure [1]. [ ] The various stages of the welding processes for the structural components including the existing/as-left J-groove weld and repair J-groove weld are simulated and consist of the following sequential steps:

1. Welding of the weld butter (for as-left J-groove) to the RV shell ([ ]) using [ ]
2. Post-Weld Heat Treatment (PWHT) is simulated for the J-groove butter, RV shell, outer surface original weld buildup and RV cladding. Since PWHT was performed the original Weld Buildup ([ ]) on the RV shell is modeled as part of the original shell structure (rather than a weld). The ID cladding which was also welded prior to the PWHT processes is also not simulated as a weld.
3. Welding of the as-left J-groove weld attaching remnant N-16D nozzle ([ ]) to the RV shell using [ ]

4. Welding of [ ] on the as-left J-groove weld and butter at the ID of the RV around the nozzle N-16D.
5. Two preservice hydrotests are simulated, followed by three operating cycles. Operational loads at steady state condition are simulated by applying the corresponding temperature and pressure as a static load step.
6. Simulated welding of the new weld pad ([ ]).
7. Simulated welding of the new J-groove weld ([ ]) attaching repair nozzle to the new weld pad. It is noted that of the three possible configurations provided on the design drawings (References [3] and [5]), the as-implemented configuration (based on repair traveler [6]) is modeled in the analysis.

As explained above this simulation follows the sequential steps that consist of building the original geometry of the instrument nozzle configuration including the butter, as-left J-groove weld and ID weld overlay. The model includes the original geometry which consists of the RV shell and cladding, the as-left J-groove weld, and the remnant nozzle as well as the repair components including weld pad, replacement nozzle and new J-groove weld. The key steps of the welding simulations, illustrated with the finite element model, are shown in Figure 2-3 through Figure 2-7.

Figure 2-3: Butter Welding

[ ]

Figure 2-4: J-Groove Weld Attaching Nozzle to RV Shell

[ ]

Figure 2-5: ID Weld Overlay covering at J-Groove weld and Butter

[ ]

Figure 2-6: Weld pad

[ ]

Figure 2-7: Weld Pad and J-Groove weld attaching Replacement Nozzle to the Weld Pad

[ ]

The general purpose finite element code ANSYS (Ref. [2]) is used to perform the WRS finite element analysis.

The basic steps comprising the multi-pass welding simulation of each weld simulated are as follows:

1. Develop the finite element model with the features necessary to accommodate the weld beads for each weld.
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.
6. Thermal phase using the ANSYS "birth and death" feature
    - Deactivate finite elements in all weld passes.
    - Activate one weld pass at a time and perform transient thermal analysis to develop the history of the temperature field for subsequent structural analysis.
7. Structural phase using the ANSYS "birth and death" feature
    - Deactivate finite elements in all weld passes.
    - Activate one weld pass at a time and perform static structural elastic-plastic analysis using the temperature history from the thermal phase.

3.0 ASSUMPTIONS

3.1 Unverified Assumptions

There are no unverified assumptions used in this analysis

3.2 Justified Assumptions

There are no justified assumptions used in this analysis.

3.3 Modeling Simplifications

The following simplifications are used in the model:

[ ]

4.0 DESIGN INPUTS

4.1 RV and N16D Nozzle Geometric Data

The detailed dimension of the instrument nozzle N-16D modeled in the WRS finite element analysis are obtained from References [3], [4] and [5]. The key dimensions are in Table 4-1.

Table 4-1: RV and Instrument Nozzle Dimensions

Dimension	Value	Reference(s)
Shell radius to base metal ID		[3]
Cladding thickness (nominal)		[4], [ ]
Shell thickness		[3]
ID weld overlay thickness (nominal)		[4],
Original Weld buildup thickness		[4],
Original Instrument Nozzle ID (towards the ID of the shell)		[4],
Original Instrument Nozzle OD (towards the ID of the shell)		[4],
Original Instrument Nozzle Bore ID (towards the ID of the shell)		[4],
Repair Weld pad thickness		[3]
Replacement Instrument Nozzle ID		[6]
Replacement Instrument Nozzle OD		[6]
Replacement Instrument Nozzle Bore ID		[6]

4.2 RV and N16D Nozzle Material Information

References [3], [4] and [5] provide the material designation of the components modeled in the WRS analysis.

Table 4-2: Component Material Designation

Component	Material Designation	Reference(s)
RV Shell	[	[3]
Cladding		[3]
Original Weld Buildup		[3]
Original J-groove butter		[3]
Original J-groove Weld		[3]
Original Instrument Nozzle		[3]
ID weld overlay		[3, 4]
New Weld Pad		[3]
Replacement Nozzle		[5]
J-groove Weld attaching Replacement nozzle to weld pad	]	[3]

Physical properties for all materials including stress-strain curves are taken from reference [11]. [

]

4.3 Welding Parameters

Reference [7] provides welding parameters used in the welding attachment of nozzle N16D to the RV.

Table 4-3: Welding Parameters for Butter Weld

Welding Parameter	Value	Reference(s)
Rod Diameter	[	[7] [
Current		[7]
Voltage		[7]]
Travel Speed		Typical, [1]
Maximum Interpass Temperature	]	[7] [ ]

Table 4-4: Welding Parameters for J-Groove Weld

Welding Parameter	Value	Reference(s)
Current (Root Pass)	[	[7] [ ]
Voltage (Root Pass)		[7] [ ]
Travel Speed (Root Pass)		Typical, [1]
Rod Diameter (Remainder layers)		[7] [
Current (Remainder layers)		[7]
Voltage (Remainder layers)		[7]]
Travel Speed		Typical, [1]
Maximum Interpass Temperature	]	[7] [ ]

The weld parameters for the J-groove weld remainder passes are used for the ID overlay as well.

Table 4-5: Welding Parameters for the New (Repair) Weld Pad

Welding Parameter	Value	Reference(s)
Current (Layer 1)	[	[8]
Voltage (Layer 1)		[8]
Travel Speed (Layer 1)		[8]
Current (Remaining Layers)		[8]
Voltage (Remaining Layers)		[8]
Travel Speed (Remaining Layers)		[8]
Maximum Interpass Temperature	]	[8]

Table 4-6: Welding Parameters for New J-Groove Weld

Welding Parameter	Value	Reference/(s)
Current	[	[9 , 7]
Voltage		[7]
Travel Speed		Typical, [1]
Maximum Interpass Temperature	]	[9]

*WPS [9] has no voltage values listed, hence typical values from [7] are used.

Table 4-7: PWHT Parameters

PWHT Parameter	Value	Reference/(s)
Heat up / Cool down rate†	[	[4, 10]
Hold Temperature		[4]
Hold Time	]	[4]

†According to [ ] [7] PWHT Specification for Low Alloy Steel Plates, the heatup rate shall not exceed [ ]. Per the ASME Construction Code [10], the PWHT heat uprate shall be not more than 400 degrees Fahrenheit per hour divided by maximum thickness (in inches). [ ]

4.4 The Finite Element Model

[

] Figure 4-2 shows the flow chart of the overall analysis sequence with corresponding input filenames.

Figure 4-1: Finite Element Mesh

[ ]

Figure 4-2: Simulation Flow Chart

[ ]

4.5 Boundary Conditions for Welding Simulation

4.5.1 Thermal Analysis – Welding Simulation

The thermal model is loaded by a volumetric heat source applied to each weld pass. [ ] per the Reference [1] WRS procedure to model natural convection to an air environment. [ ]

4.5.2 Structural Analysis - Welding Simulation

The temperature history from the thermal analysis is used as the thermal load in the structural analysis. [ ]

Following the butter weld, PWHT is simulated. Per Table 4-7, PWHT was performed at a nominal temperature of [ ] and heating and cooling rates of [ ]. Heat treatment starts and ends at a temperature of 70°F. [ ]

Once ID weld overlay is completed two hydro tests were simulated by applying cycles of 1.25 × design pressure at a temperature of 100°F. Pressure is applied to all wetted surfaces, and end cap pressures are applied at the original nozzle end and at the boundary plane representing horizontal cross-section of the vessel away from the nozzle. Following the hydro test operating cycles are also simulated. The cold condition (70°F, 0 psig) and operating condition [ ] [4]) are repeated 3 times.

5.0 RESULTS

The welding process steps are discussed in Section 2.0. On the completion of ID weld overlay two hydrostatic tests followed by three operating cycles are simulated. The axial and hoop stress contours at cold conditions following these operating cycles are shown in Figure 5-1. The stress-contours are presented in cylindrical coordinate systems that are aligned with the axis of the nozzle, where z is axial and y is hoop; the unit of stresses is in psi. The coordinate triad shown in the contour plots represents the global coordinate system in which "X" represent the hoop, "Y" the axial direction of the RV Shell.

Following the completion of the final weld, the repair/new J-groove weld simulation, the stress state is obtained. The axial and hoop stress contours are shown in Figure 5-2 in units of psi. The results are presented in cylindrical coordinate system aligned with the axis of the nozzle, where z is axial and y is hoop.

The ANSYS computer model files (see Table 6-1) including residual stress results are available for use in the subsequent fracture mechanics analysis.

Figure 5-1: Distribution of Residual Axial (top) and Hoop (bottom) Stresses (psi) after Operating Cycles Step – at cold condition

Figure 5-2: Distribution of Residual Axial (top) and Hoop (bottom) Stresses (psi) at the end of new J-Groove weld (final step) – at cold condition

6.0 COMPUTER USAGE

The computer files pertinent to revision 000 of this document are located in ColdStor directory: '\cold\General-Access\32\32-9000000\32-9274303-000\official'

6.1 Hardware / Software

ANSYS Version 16.0., Reference [2], was used for all FE runs documented herein. Use of this version of ANSYS is acceptable since error notices were reviewed and none was found applicable to this analysis. The installation verification results are found to be acceptable. The installation verification files can be found under directory: '..\Verification'

Computer program tested: ANSYS Version 16.0, verification tests vm32mod2D.vrt, vm32mod3D.vrt, vm38mod2D.vrt, and vm38mod3D.vrt.

The computer used for this analysis is a multi-node server (HPCv2), the computing node used to run this analysis was determined by PBS que scripts. The node "auslynchpc34" was used by the que for the runs. The verification runs were submitted in the same queue and the analysis results are reported in Table 6-1.

The hardware platform of auslynchpc34: Intel(R) Xeon(R) CPU E5-4650L 0 @ 2.60GHz; 96 GB RAM; Operating system: Red Hat Enterprise Server v6.4, kernel: 2.6.32-358.el6.x86_64.

The queue was initiated by Martin Kolar on behalf of the preparer, Silvester Noronha.

6.2 Computer Files

The complete list of computer files associated with this analysis is listed in Table 6-1.

Table 6-1: List of Computer Files

CRC Checksum	Size	Modified Date & Time	File Name
./ModelData:			
44166	6014	Sep 08 2017 15:11:27	
34553	6756001	Sep 03 2017 10:27:24	
36198	6694194	Sep 03 2017 10:25:55	
34837	5870	Sep 03 2017 09:47:27	
./ThermalAnalysis:			
39692	576	Sep 03 2017 09:41:45	
11344	493	Sep 03 2017 09:41:45	
59978	998	Sep 03 2017 09:41:45	
04138	598	Sep 03 2017 09:41:45	
64232	659	Sep 03 2017 09:41:45	
61602	10691	Sep 03 2017 09:42:22	
08825	223221	Sep 08 2017 22:13:03	
06847	9529	Sep 03 2017 09:42:22	

CRC Checksum	Size	Modified Date & Time	File Name
55346	23473	Sep 08 2017 22:54:49	
33448	10224	Sep 03 2017 09:42:22	
01378	24423	Sep 08 2017 22:45:59	
26056	9830	Sep 03 2017 09:42:22	
24642	23864	Sep 09 2017 00:38:59	
45700	10408	Sep 03 2017 09:42:22	
00958	24582	Sep 09 2017 00:22:38	
01259	55637	Sep 03 2017 09:41:45	
./StructuralAnalysis:			
49906	2156	Sep 07 2017 09:19:43	
09794	474	Sep 03 2017 22:08:02	
37885	11967	Sep 03 2017 22:08:02	
49614	3872	Sep 03 2017 22:08:15	
52775	260763	Sep 09 2017 05:09:12	
51121	2851	Sep 08 2017 15:04:21	
06084	12402	Sep 09 2017 14:56:41	
27707	3541	Sep 04 2017 06:53:19	
63127	13967	Sep 09 2017 14:35:39	
32308	3771	Sep 04 2017 06:52:27	
15188	14165	Sep 09 2017 12:40:24	
50989	33619968	Sep 10 2017 10:48:36	
06611	1.77E+08	Sep 10 2017 10:48:35	
34330	3377	Sep 04 2017 06:56:27	
28324	13766	Sep 10 2017 10:48:36	
58179	3022	Sep 03 2017 22:08:15	
42883	311104	Sep 09 2017 06:06:47	
12164	3367	Sep 04 2017 06:55:41	
22758	13750	Sep 10 2017 07:13:29	
./Verification:			
30336	3551	Sep 03 2017 12:10:55	vm32mod2D.inp
04564	55395	Sep 08 2017 21:50:01	vm32mod2D.out
48891	606	Sep 08 2017 21:50:01	vm32mod2D.vrt
42236	4940	Sep 03 2017 12:10:54	vm32mod3D.inp
00376	110067	Sep 08 2017 21:50:03	vm32mod3D.out
34040	606	Sep 08 2017 21:50:03	vm32mod3D.vrt
51869	2458	Sep 03 2017 12:10:54	vm38mod2D.inp
24568	15061	Sep 08 2017 21:50:02	vm38mod2D.out
20215	632	Sep 08 2017 21:50:02	vm38mod2D.vrt
47844	3112	Sep 03 2017 12:10:55	vm38mod3D.inp
29377	17491	Sep 08 2017 21:50:04	vm38mod3D.out
09779	632	Sep 08 2017 21:50:04	vm38mod3D.vrt

7.0 REFERENCES

1. [ ]
2. ANSYS Finite Element Computer Code, Version 16.0, ANSYS Inc., Canonsburg, PA
3. [ ]
4. [ ]
5. [ ]
6. [ ]
7. [ ]
8. [ ]
9. [ ]
10. ASME B&PV Code, Section III, Nuclear Vessels,1968
11. [ ]
12. [ ]

APPENDIX A: VERIFICATION OF ANSYS COMPUTER CODE

Four verification problems were selected to test key features of the ANSYS finite element computer program [2] 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 the 2D models. A companion three-dimensional test case (vm32mod3D) was created which utilizes the SOLID70 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
```