International Journal of Pressure Vessels and Piping 87 (2010) 26–32

Contents lists available at ScienceDirect

International Journal of Pressure Vessels and Piping

journal homepage: www.elsevier.com/locate/ijpvp

Welding overlay analysis of dissimilar metal weld cracking of feedwater nozzle

Y.L. Tsai [a,b,*], Li. H. Wang [b], T.W. Fan [b,c], Sam Ranganath [b], C.K. Wang [d], C.P. Chou [a]

[a] National Chiao Tung University, Mechanical Engineering Department, 1001 TaHsueh Road, HsinChu, Taiwan 30010, ROC
[b] Industrial Technology Research Institute (ITRI), 195 Chung Hsing Rd., Sec.4 Chu Tung, HsinChu, Taiwan 310, ROC
[c] Chung Hua University, Department of Civil Engineering and Engineering Informatics, 707, Sec.2, WuFu Rd., HsinChu, Taiwan 300, ROC
[d] Taiwan Power Company (TPC), No.242, Sec. 3, Roosevelt Rd., Zhongzheng District, Taipei City 100, Taiwan, ROC

ARTICLE INFO

Article history:
Received 9 September 2008
Accepted 15 February 2009

Keywords:
Feedwater nozzle
Alloy 182
Weld
ASME
FEM

ABSTRACT

Inspection of the weld between the feedwater nozzle and the safe end at one Taiwan BWR showed axial indications in the Alloy 182 weld. The indication was sufficiently deep that continued operation could not be justified considering the crack growth for one cycle. A weld overlay was decided to implement for restoring the structural margin. This study reviews the cracking cases of feedwater nozzle welds in other nuclear plants, and reports the lesson learned in the engineering project of this weld overlay repair. The overlay design, the FCG calculation and the stress analysis by FEM are presented to confirm that the Code Case structural margins are met. The evaluations of the effect of weld shrinkage on the attached feedwater piping are also included. A number of challenges encountered in the engineering and analysis period are proposed for future study.

1. Introduction

Inspection of the weld between the feedwater nozzle and the safe end at Kuo Sheng Unit-1 showed axial indications in the Alloy 182 weld as shown in Fig. 1. The indication appears to be in the weld as well as the adjacent Alloy 182 weld butter. The indication was sufficiently deep that continued operation could not be justified considering the crack growth for one cycle. Taiwan Power Company decided to implement a weld overlay to restore the structural margin and assure that there is no possibility for continued crack growth and potential through wall cracking. The weld overlay was based on ASME Code Case N-504-2 [1] and used Alloy 52 weld material. Alloy 52 is weld metal highly resistant to stress corrosion cracking and has been successfully used in BWR nozzle to safe end welds [2,3]. Fig. 2 shows a schematic of the weld overlay design. It is seen that the overlay covers the weld and the weld butter and extends all the way to the nozzle. This report describes the background on Code Case 504-2 on which the overlay design is based, the fatigue crack growth analysis, the finite element analysis to confirm that the Code Case structural margins are met and the results of the analysis to evaluate the effect of weld shrinkage on the attached feedwater piping.

2. Fatigue crack growth analysis

The crack growth relationship shown in Fig. 5 can be used to determine crack growth increment for the life of the overlay. The first step in the crack growth analysis is the determination of the stress intensity factor range, ΔK. The stress intensity factor is determined using the equations recommended in Section XI, ASME Code [4]. Both even though the actual flaw is axial, both axial and circumferential cracks are considered for the crack growth analysis.

2.1. Axial crack analysis

A semi-elliptic flaw with depth 1.1 in = 0.02794 m and length 3.75 in = 0.09525 m (Fig. 3) is used for the K calculation. The effective thickness for the purpose of determining the ΔK value is 1.1 + 0.43 = 1.53 in = 0.0389 m. The stress intensity factor range is given by:

$$\Delta K = (\Delta\sigma_h + p)M_m\sqrt{(\pi a/Q)} \tag{1}$$

where a is the crack depth, l is the crack length, σ is the membrane stress due to internal pressure, p is the crack face pressure, M_m is the membrane stress correction factor = G_0 in Table A-3320-1 of Appendix A, Section XI, ASME Code and Q is the flaw shape parameter given by:

$$Q = 1 + 4.593(a/l)^{1.65} - q_y \tag{2}$$

and

$$q_y = [\sigma M_m/\sigma_{ys}]^2/6 \tag{3}$$

where σ_{ys} = yield strength.

* Corresponding author.
E-mail address: YLTsai@itri.org.tw (Y.L. Tsai).

0308-0161/$ – see front matter
doi:10.1016/j.ijpvp.2009.11.008

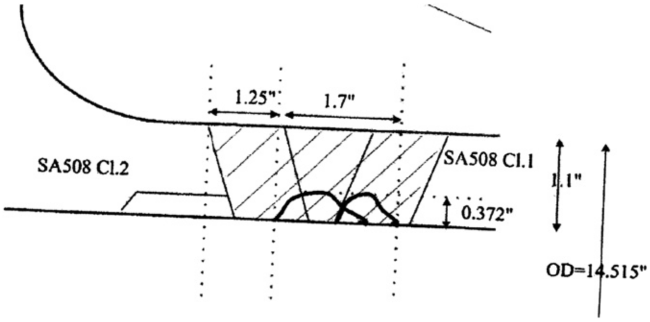


Fig. 1. Schematic of the indication.

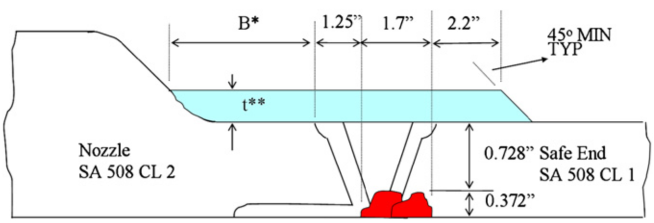


Fig. 2. Weld overlay design.

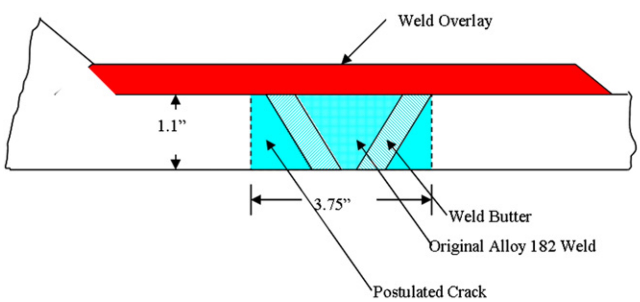


Fig. 3. Postulated axial flaw.

As is normal in fracture mechanics analysis, the applied stress is calculated based on the uncracked thickness (including the overlay). The hoop stress is given by: $\sigma_h = PD/2t$ where D = outside diameter of the overlay = 15.375 in = 0.391 m, t is the total thickness (including the overlay) = 1.53 in = 0.0389 m and P = internal pressure = 1050 psi = 7.24 MPa. The hoop stress range is calculated to be 5.18 ksi = 35.72 MPa. This is used in the K calculation.

Fig. 6 shows the predicted crack depth as a function of the number of cycles. The original stress report for the FW nozzle safe end considers a total of 120 startup shutdown cycles. There is also a pressure test prior to each startup, so there will be potential 120 more cycles of pressure cycling. If one accounted for license renewal to 60 years, a conservative estimate for the total number of cycles is (120 + 120) 60/40 = 360 cycles. As shown in Fig. 6, the crack depth after 360 cycles is 1.100,191 in = 0.0279 m and the incremental crack growth is very small = 0.000191 in = 4.85×10^{-6} m. This has to be added to the weld overlay thickness.

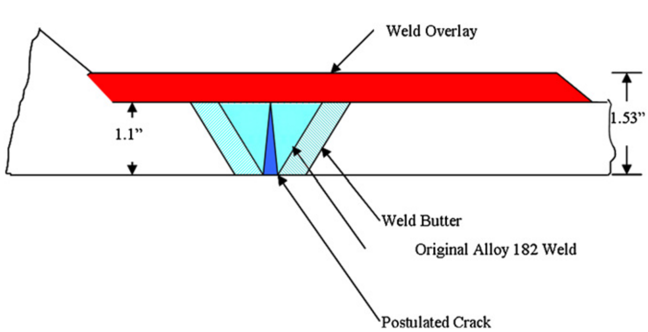


Fig. 4. Postulated circumferential flaw.

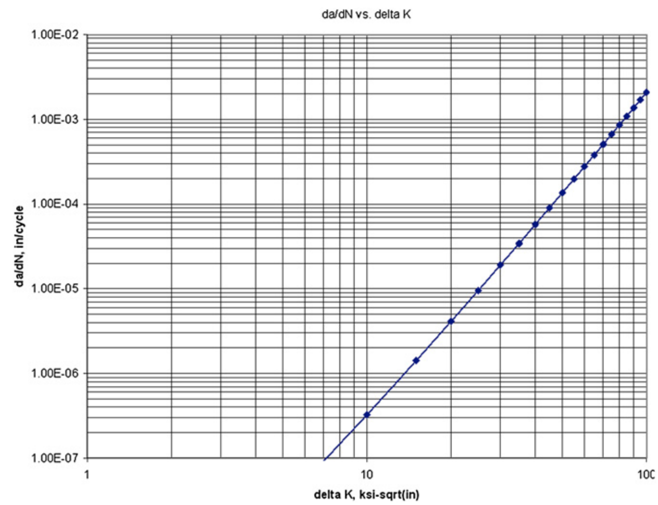


Fig. 5. Recommended fatigue CGR.

2.2. Circumferential crack analysis

For this case, the stress intensity solution an axially loaded pipe with a circumferential crack from ref. [5] is used. The stress intensity factor range is given by:

$$\Delta K = F_1 \Delta\sigma_{axial} \sqrt{(\pi a)}$$

where F_1 = 1.1259 + 0.2344(a/t)+2.2018(a/t)^2–0.2083(a/t)^3, a = crack depth (initial value = 1.1 in = 0.02794 m), t = total thickness (including the overlay) = 1.1 + 0.43 = 1.53 in = 0.0389 m (Fig. 4) $\Delta\sigma_{axial}$ = axial stress in the uncracked thickness (including the overlay) = $p\,D/4t = 1.05 \times 15.375/4 \times 1.53 = 2.6$ ksi = 17.93 MPa.

Fig. 7 shows the predicted crack depth as a function of the number of cycles. As shown in Fig. 7, the crack depth after 360

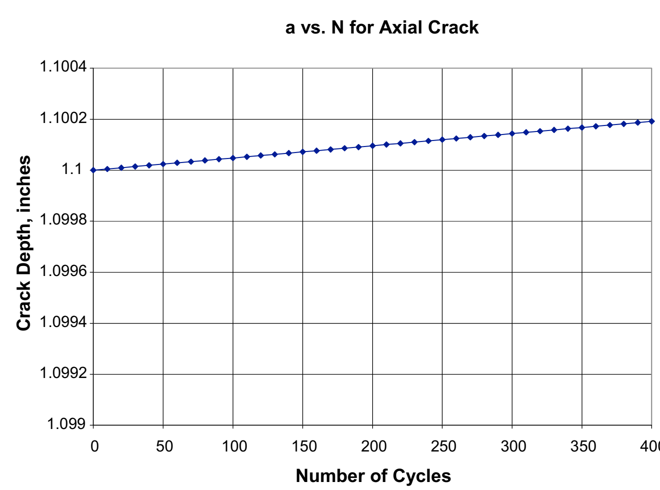


Fig. 6. Axial crack depth as a function of the number of cycles.

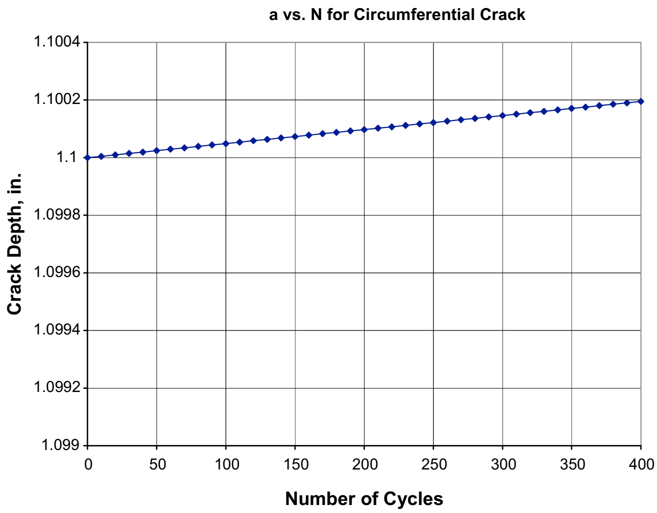


Fig. 7. Circumferential crack depth as a function of the number of cycles.

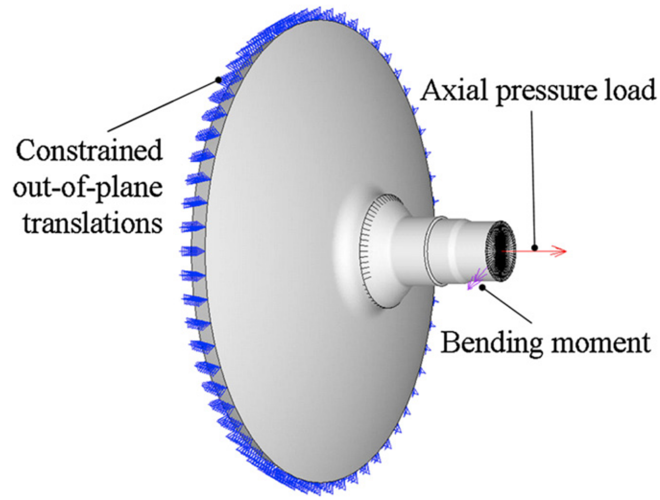


Fig. 9. Boundary conditions.

cycles is 1.10019 in = 0.0279 m and the incremental crack growth is very small = 4.83×10^{-6} m in, essentially the same as that for the axial crack. This has to be added to the weld overlay thickness.

Fatigue crack growth analyses were performed to determine the amount of potential future crack growth. The additional weld overlay thickness of 5.08×10^{-4} m for fatigue crack growth has been added in the present designed overlay thickness. Since the design margin is much larger than the predicted incremental fatigue crack growth, this overlay design is acceptable.

3. Weld overlay stress analysis

As stated in the Code Case, the primary stress limits of Section III are met as long as the 0.75 $\sqrt{Rt}$ length limit on each side is met. An alternate approach is to demonstrate by finite element stress analysis that the primary stress limits of Section III are met. For the proposed Kuo Sheng overlay, finite element analysis is necessary since the width B (as shown in Fig. 2) on one side of the crack is adjusted such that the weld overlay intersects the tapered region of the nozzle.

In order to determine the primary stresses in the region of the crack a finite element model of the nozzle and safe end is developed. Fig. 8 shows the proposed ANSYS finite element analysis model. A three-dimensional model is used to allow the application of safe end moments as well as loads. The model uses three-dimensional solid elements. This allows the postulation of both axial and circumferential cracks. No special crack tip elements are needed since the intent is to determine the primary membrane and bending stresses (as required by the Code Case) in the region of the crack. At the end of the safe end part of the model rigid beam elements were used to allow the application of moments and forces in addition to internal pressure. The design mechanical loads from the safe end stress report were used in the analysis. Fig. 9 shows the boundary conditions used in the model. The nodes at the end of the nozzle were constrained in the direction normal to the surface. Analysis was done for both axial and circumferential cracks. The differences in the axial crack and circumferential crack models were only in the region of the postulated crack. For the axial crack case, a rectangular crack with depth equal to the weld thickness and 3.75 in = 0.09525 m length was used. For the circumferential crack case, a 360° crack with depth equal to the weld thickness was used. This section describes the results of the weld overlay stress

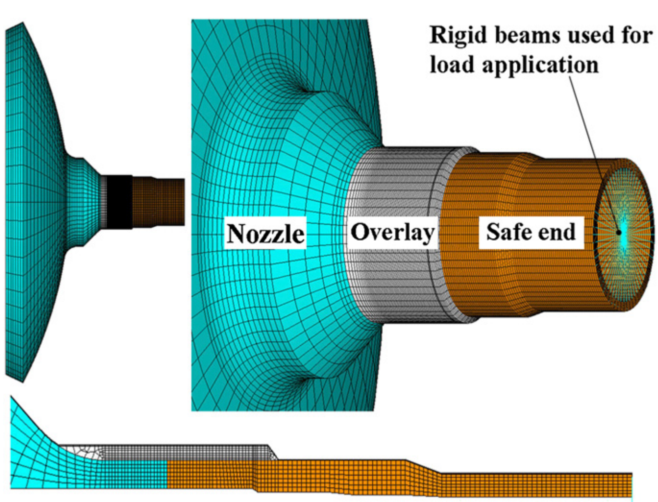


Fig. 8. Finite element model.

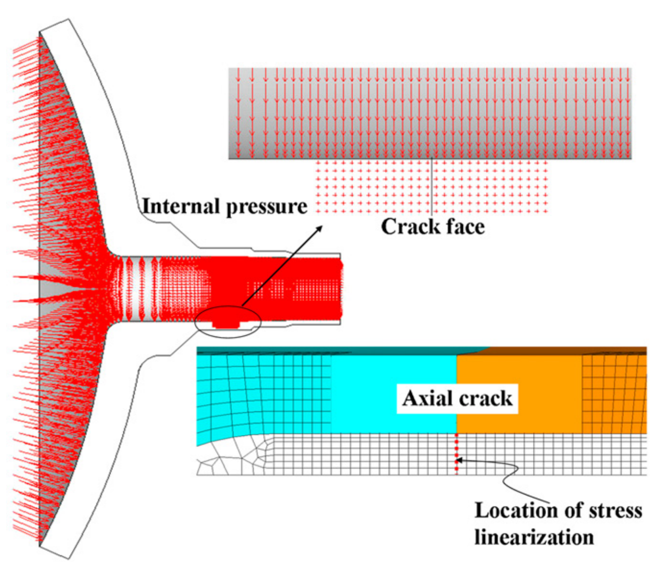


Fig. 10. Detail of the finite element model showing the postulated axial crack.

Table 1
Axial Crack – Linearized stresses through overlay section.

Load case 1	Stress intensity, psi	Axial stress, psi	Hoop stress, psi
Pm	20 000	140	3513
Pm + Pb	23 140	807	5258
Load case 2			
Pm	20 120	66	4100
Pm + Pb	23 380	1116	6694
Load case 3			
Pm	19 890	6302	3768
Pm + Pb	22 770	7447	6416
Load case 4			
Pm	20 130	1986	3964
Pm + Pb	23 180	3130	6571

analysis for both axial and circumferential cracks. The detailed analysis described in the next sections for axial and circumferential cracks (3.1 and 3.2) are for design mechanical loads. The loads for design conditions bound the values for Levels A and B conditions. However, for Levels C and D, the primary loads (e.g. pressure) can be higher than those for design conditions. Therefore, primary stress evaluations are performed for Design and Levels C/D conditions. Section 3.3 summarizes the primary stress results for Design and Levels C/D conditions.

3.1. Axial cracks (design conditions)

Fig. 10 shows the details of the model for the axial crack case. The postulated crack was axial covering the weld thickness (1.1 in = 0.02794 m) and extending equally into the safe end and nozzle. The internal pressure was 1300 psi (8.965 MPa) for the primary stress analysis. The primary stress assessment for the axial crack is confined to evaluating the stress in the overlay in the region of the crack. A case can be made that the stress is a peak stress or at worst, a primary local stress. Nevertheless, a conservative approach based on comparing the average stress in the overlay section (in the region of the crack) with the allowable value (S_m) for primary membrane stress will be used.

Several cases were evaluated for the axial crack case:

i) Load case 1 – Internal pressure (No pressure on crack surface; No axial end load from pressure)

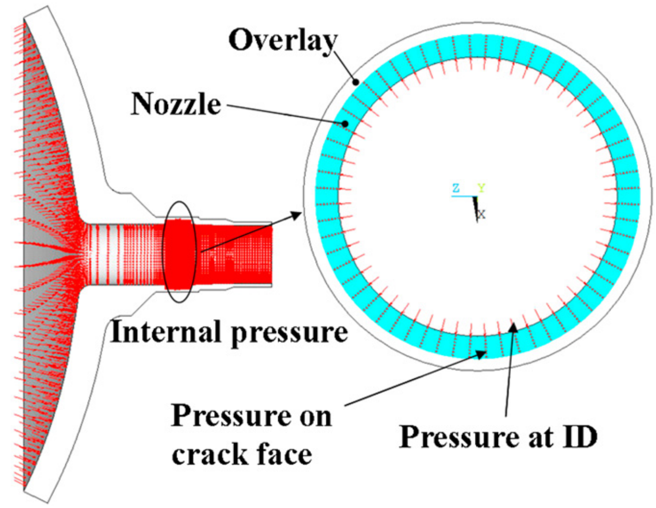


Fig. 12. Detail of the finite element model showing the circumferential crack.

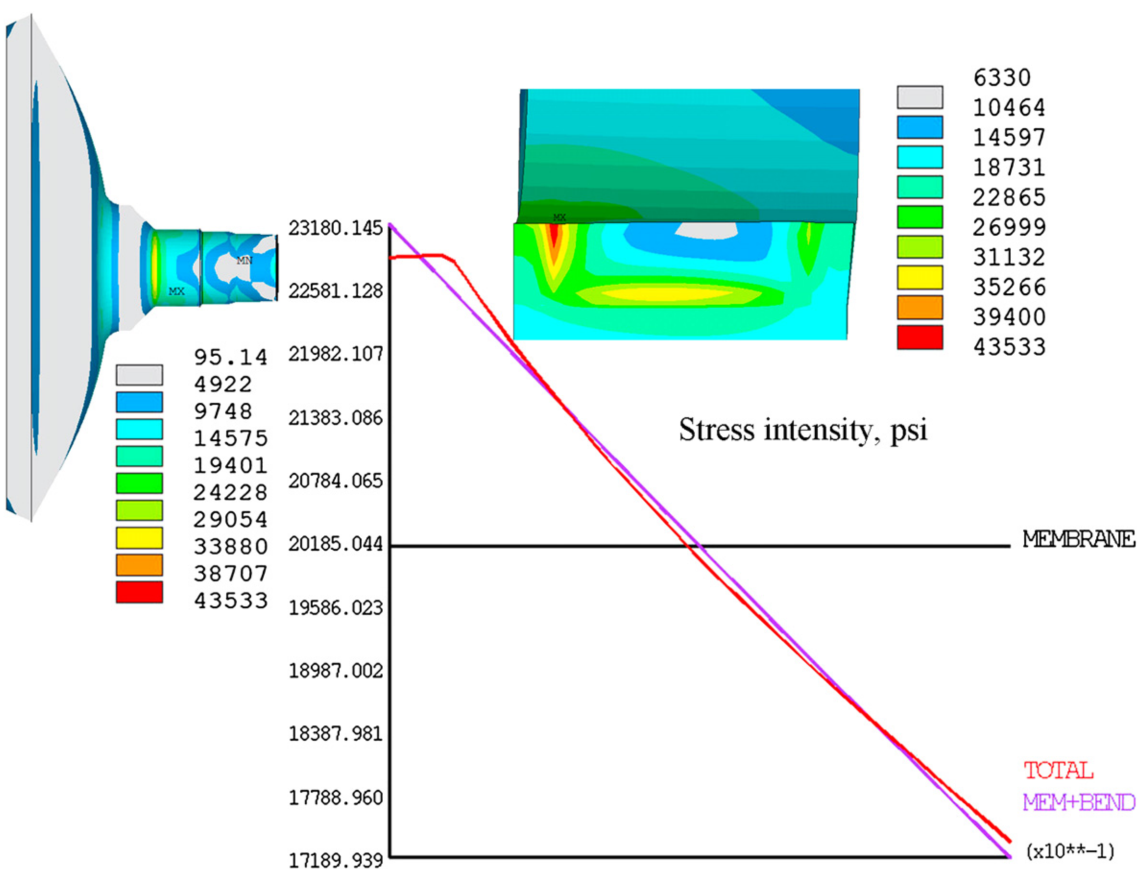


Fig. 11. Stress results from load Case 4 for axial crack.

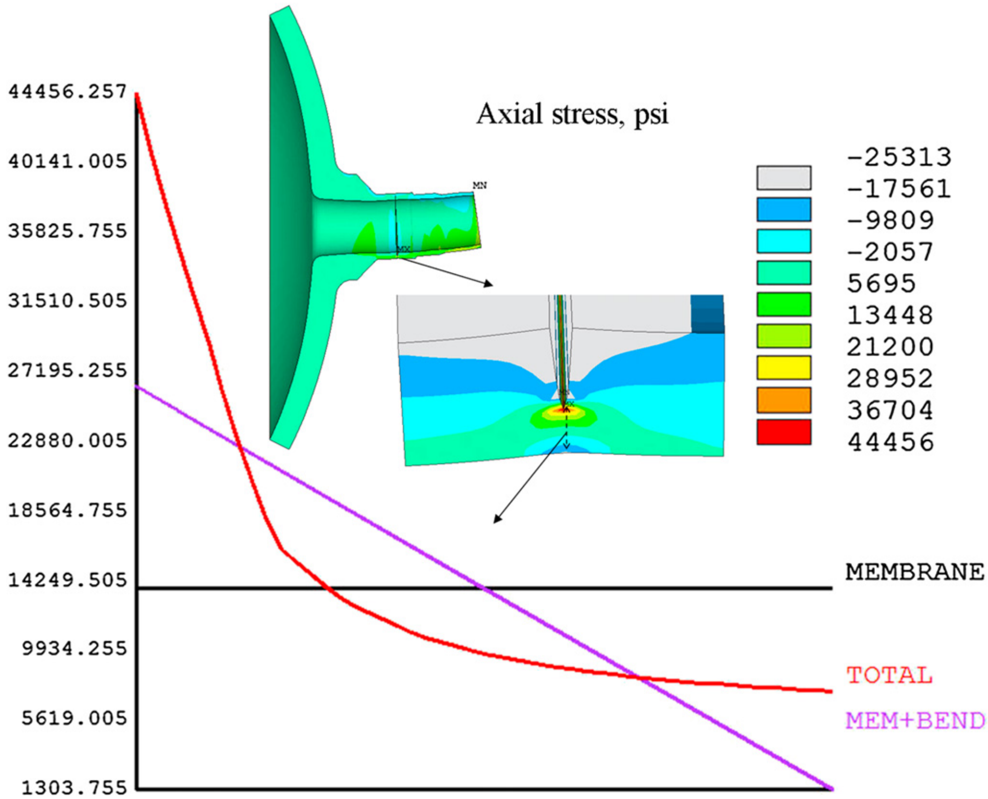


Fig. 13. Axial stress in the overlay for the postulated circumferential crack.

ii) Load case 2 – Internal pressure (Pressure on crack surface; No axial end load from pressure)
iii Load case 3 – Internal pressure (Pressure on crack surface; Axial end load from pressure) and end moment (In plane of crack)
iv) Load case 4 – Internal pressure (Pressure on crack surface axial end load from pressure) and end moment (In plane normal to crack)

Table 1 shows the results of the stress analysis for the four cases described above. Linearized axial and hoop stresses as well as stress intensity values are shown in Table 1. The differences between the four cases are not all that significant. What is interesting is that the stress intensity values are much higher than the hoop stresses. The larger value for the stress intensity results from the shear stress required to equilibrate the hoop stress near the crack. Only the results of Load Case 4 are discussed in detail here. It is seen that the shear stress is high for equilibrium reasons.

Fig. 11 show the detailed stresses in the ligament and the stress linearization for the overlay ligament. In the cases shown in Fig. 11, the Pm + Pb linearized stress is slightly higher than the peak stress. This is due to the fact that the elements are three-dimensional solids and there is a singularity at the end of the crack. The differences are minor and do not have any effect on the acceptability of the stresses. It is seen that the membrane stress Pm, is well below the allowable value of $S_m = 23.3$ ksi = 160.68 MPa. Similarly, the membrane + bending stress Pm + Pb is also below the allowable value of 1.5 $S_m = 35$ ksi = 241 MPa. Thus, the primary stress limits are met even though the stress in the region of the overlay is more like a peak stress, not a membrane stress.

Table 2
Circumferential Crack – Linearized stresses through overlay section at the crack.

	Stress intensity, psi	Axial stress, psi
Pm	14 560	13 820
Pm + Pb	25 350	26 330

3.2. Circumferential cracks (design conditions)

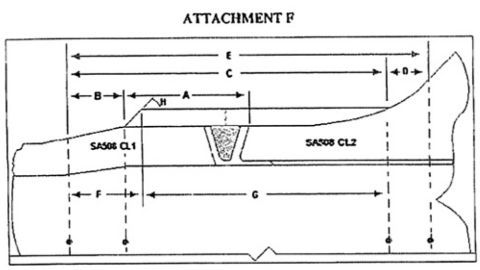
In this case, a 360° crack with depth equal to the weld thickness is postulated. Even though the crack that led to the application of the overlay was axial, since the design basis was a through thickness circumferential crack, the postulated crack was circumferential. Fig. 12 shows the details of the model for the circumferential crack case. Fig. 13 shows the results of the axial stress analysis for the circumferential crack. The applied loading for the design conditions was 1300 psi (8.965 MPa) pressure and 980 in-kip = safe end moment. The axial stress distribution and its linearization in the overlay ligament are also shown in Fig. 13. It is seen that the membrane stress intensity is 14.56 ksi = 100.41 MPa (less than $S_m = 23.3$ ksi = 160.68 MPa) and the membrane + bending stress is 25.35 ksi = 174.81 MPa (less than 1.5 $S_m = 1.5 \times 23.3 = 35$ ksi = 241.36 MPa). The primary stress limits are met. Table 2 shows the results of the linearization for both the axial stress and the stress intensity. The membrane stress and membrane + bending stress are determined by equilibrating the force and moment respectively. ANSYS provides this as a post-processor option. The stress limits are met for both the axial stress and stress intensity.

3.3. Primary stress results for Design and Levels C/D conditions

The previous sections (3.1 and 3.2) describe the evaluation of primary stress for design conditions. This section describes the evaluation of primary stresses for Design and Levels C/D (emergency and faulted) conditions. The pressure and moment loads are 1300 psi = 8.9648 MPa and 980 in-kip for the Design conditions

Table 3
Primary Stresses at the crack section.

Postulated crack	Pressure (psi) and moment (in-kip)	Conditions	P_M		$P_L + P_B$	
			Calculated value, ksi	Allowable value, ksi	Calculated value, ksi	Allowable value, ksi
Circumferential crack	1300 psi 980 in-kips	Design	14.56	23.3 (Sm)	25.35	35.0 (1.5 Sm)
	1460 psi 1341 in-kips	Levels C and D	19.92	28.0 (1.2S m)	34.7	42.0 (1.8S m)
Axial crack	1300 psi 980 in-kips	Design	20.13	23.3 (Sm)	23.18	35.0 (1.5 Sm)
	1460 psi 1341 in-kips	Levels C and D	22.6	28.0 (1.2 Sm)	26.0	42.0 (1.8 Sm)



Final WOL Dimensions						
Location ID	Measurement (inch)* 45	135	225	315	Purpose for measurement and/or punch marks	
"A+B"	7.345	7.384	7.461	7.462	Taken from Diagram "A-1"	
"F"	1.807	1.808	2.301	2.240	Actual final measurement from completed WOL	
A+B-F=	5.538	5.576	5.160	5.222	Design requirement: 5.15" (Measure by subtracting A+B-F≧5.15")	QC A/R ACC
"G_d" Design	7.400	7.400	7.400	7.400	WOL Overlay length-Design	
"G_a" Actual	7.873	7.922	7.778	7.744	Actual final measurement of WOL	
$G_a \geqq G_d$	ACC	ACC	ACC	ACC	Design requirement (G_d): 7.400	QC A/R ACC
"H"	<45	<45	<45	<45	Required to equal to or less than 45 degrees	QC A/R ACC

Fig. 14. As built weld overlay configuration.

and 1460 psi = 10.068 MPa and 1341 in-kips = 151.55 m N Levels C/D conditions. The calculated stresses and the allowable values are shown in Table 3 for Design and Levels C/D (emergency and faulted) conditions.

3.4. Summary of the weld overlay stress analysis

The evaluation of the primary stresses confirms that the primary stress limits are met for both the postulated axial and circumferential cracks under the overlay. The thickness of the overlay used in the analysis was the minimum value of 0.43 in = 0.0109 m. The actual thickness is probably at least 0.2 in = 5.08×10^{-3} m higher. Thus the primary stress values reported here are conservative.

4. Weld shrinkage analysis

The Code Case requires the evaluation of the effects of weld shrinkage on the associated piping and pipe supports. Shrinkage stress is like a fabrication stress and is in itself not a concern from a Code viewpoint. The significant concern for the shrinkage stress in the piping is mainly due to the potential for SCC initiation or crack growth in existing cracks. Fig. 14 shows the as-built configuration of the overlay. Fig. 15 shows the shrinkage measurements at four azimuth locations. The shrinkage varies around the circumference. The maximum value, 0.016 in (4.064×10^{-4} m) over a 10-in length is used conservatively in this analysis. The details of the piping analysis are described here. Note that the piping lengths and other dimensions are in foot units in the model, so the deflection and stress are in ft and lb/ft^2 units.

ATTACHMENT G					
Shrinkage Results					
Location ID	Measurement (inch)* 45	135	225	315	Purpose for measurement and/or punch marks
Initial "E"	10.760	10.771	10.812	10.838	From initial measurement prior to welding (Diagram A-1)
Final "E"	10.756	10.766	10.796	10.825	Measurement after WOL completed
Result	0.004	0.005	0.016	0.013	Final shrinkage value: "C"Initial – "C" Final

Fig. 15. Weld shrinkage data conditions.

4.1. Analysis model

The analysis model (Fig. 16) is composed of ANSYS PIPE16 (straight pipe) and PIPE18 (curved pipe) elements. It is based on the piping analysis model obtained from PECL piping stress report and its computer input listing [6]. The model was modified to remove pressure and temperature loads and to facilitate application of the measured overlay shrinkage (0.016-in in 10-in overlay length). The shrinkage was simulated by applying a displacement of 0.016 in ($=4.064 \times 10^{-4}$ m) at nodes 2 and 191. This means that the shrinkage was conservatively applied on both nozzles N4A and N4B.

Boundary conditions are shown in Fig. 16. All degrees of freedom were fixed at the two FW nozzles, far end of the FW pipe, and the support ends of various snubbers and spring supports. In the first case, mechanical anchors and rigid restraints are built into the model, as appropriate, for the piping support system. The stiffness

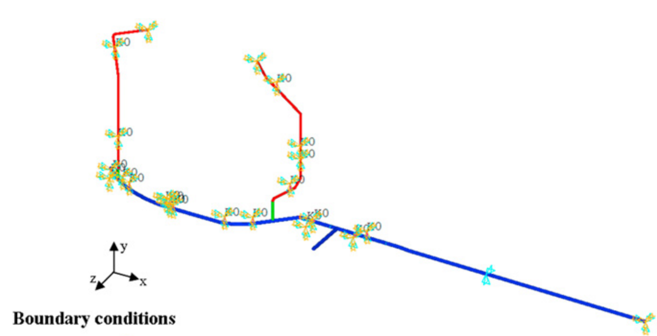


Fig. 16. Analysis model and boundary conditions.

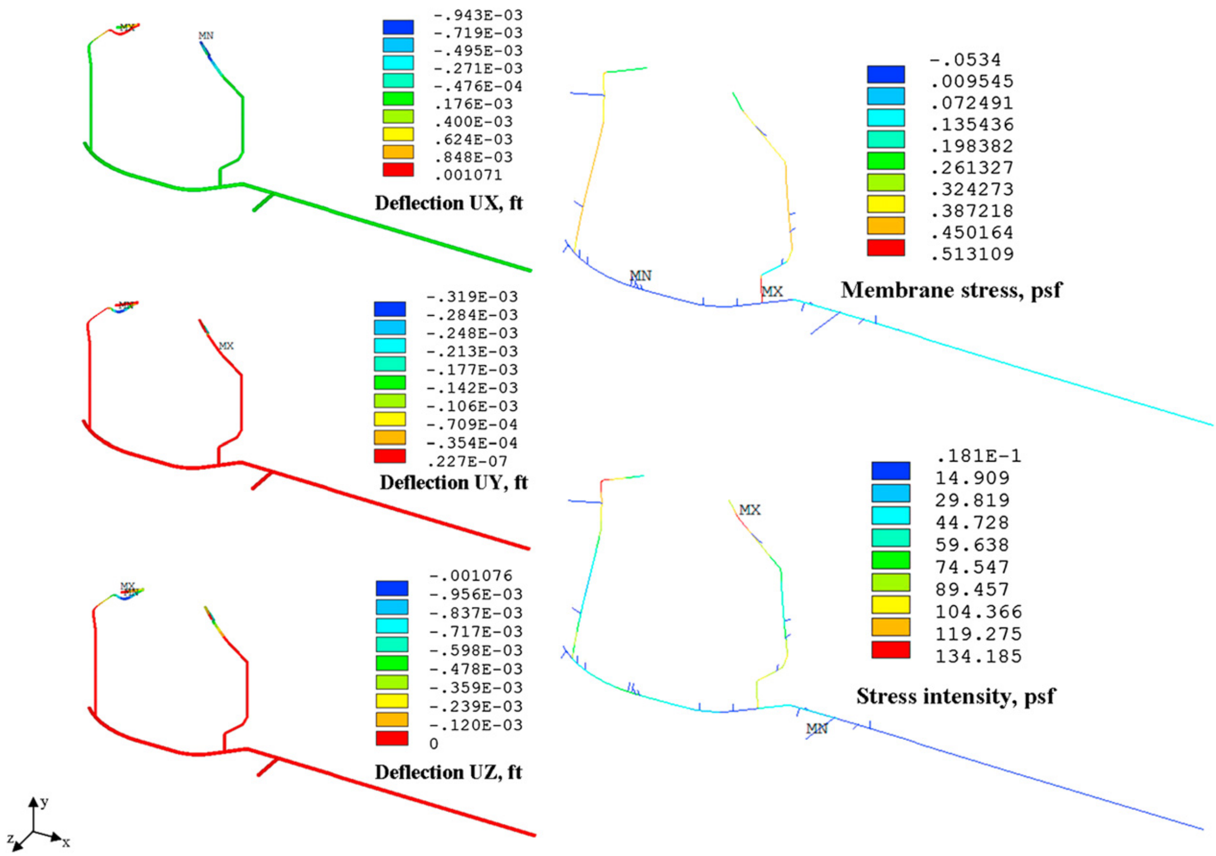


Fig. 17. Deflections and piping stresses – Overlay shrinkage at nozzle/safe end welds.

at each snubbers, hangers and restraints are taken from reference [6]. The support stiffness at nodes 17 and 179 was assumed to be 1 lb/ft which is very low and results in virtually no restraint. This maximizes the displacement, but not the reactions.

Load was applied as a displacement of 0.016 in ($=4.064 \times 10^{-4}$ m) at nodes 2 and 191. Essentially, the overlay shrinkage was applied at the nozzle weld.

4.2. Reference analysis results

Calculated deflections and stresses are shown in Fig. 17, respectively. This applies for the case where the support stiffness is very low. Note that the plotted deflections are in ft and stresses are in lb/ft^2 units. Calculated stresses are negligible (<1 psi $= 6.896$ Pa). This is a result of assumed negligible stiffness of the support nearest to the nozzle. This is consistent with expectations since the shrinkage is small and there is very little constraint.

5. Conclusions

1. The analysis described here considers the fatigue crack growth and primary stresses in the weld overlay. It is seen that all primary stresses are within the Code limits. This confirms that the overlay is sufficiently wide to assure that there is stress redistribution.
2. The measured shrinkage is small and the resulting stresses and reaction forces are very small. Thus, the welded overlay meets the requirements of Code Case N-504-2.

Acknowledgements

The task of welding overlay analysis enhance direct funding of the work by Taiwan Power Company is gratefully acknowledged. This paper is published with the permission of Taiwan Power Company.

References

[1] Code Case N-504-2. Alternative rules for repair of classes 1, 2, and 3 austenitic stainless steel piping section XI, Division 1.
[2] NUREG – 0313. Rev. 2 Technical report on material selection and processing guidelines for BWR coolant pressure boundary piping' US Nuclear Regulatory Commission; January 1988.
[3] NUREG/CR – 6921. Crack growth rates of nickel alloys in a PWR environment from the Davis-Besse and V.C. Summer Power Plants.
[4] ASME boiler and pressure vessel code, section XI, Rules for in-service inspection of nuclear power plants. New York, NY: American Society of Mechanical engineers; 2004 [Edition].
[5] Buchalet C, Bamford W. Stress Intensity solutions for continuous surface flaws in reactor pressure vessels. ASTM STP 590, American Society for Testing and Materials.
[6] PECL. " Feedwater lines inside containment stress Re-analysis to include new loads". Calc No 22A.01/0; March 1987.