Three Dimensional Finite Element Analysis of Weld Overlay Application On a Plastically Formed Feeder Tube

F. H. Ku[1], P. C. Riccardella[1], M. S. Lashley[1], Y. Chen[2], R. K. Yee[3]
[1] Structural Integrity Associates Inc., California, USA
[2] Structural Integrity Associates Inc., Ontario, Canada
[3] San Jose State University, California, USA

Abstract

This paper presents a finite element analysis (FEA) model to predict the residual stresses in a tight-radius warm bend feeder tube in a CANDU nuclear reactor coolant system throughout the various stages of the manufacturing and welding processes, including feeder tube forming, Grayloc hub weld, and weld overlay application. The FEA employs 3-D elastic-plastic technology with large deformation capability to predict the residual stresses due to the feeder tube forming and various welding processes. The results demonstrate that the FEA method captures the residual stress trends resulted from warm bending and weld overlay with acceptable accuracy.

1. Introduction

Canadian CANDU nuclear reactors are pressurized heavy water reactors based on fuel channel design [1]. Each reactor design contains hundreds of feeder tubes to channel heavy water coolant through the reactor core [1, 2]. A feeder tube is formed by bending small diameter (1.5” to 3.5” nominal pipe size “NPS”) carbon steel pipes into elbow shapes and then welded onto an attachment Grayloc hub end fitting [1]. Cracking and loss of wall thickness have been observed [1], and subsequent root-cause analyses indicate that residual stresses from plastic deformation and welding, as well as flow accelerated corrosion are significant contributors to the feeder failures [2].

A proposed remedy to the excessive tensile residual stress issue is to install a weld overlay build up on the outside surface of the feeder tube-to-flange attachment weld to completely cover the weld, as shown in Figure 1. The weld overlay is expected to induce beneficial compressive residual stresses on the original feeder tube. In addition, the weld overlay essentially increases the total wall thickness of the feeder tube and, hence, lengthens the service life of the component against flow accelerated corrosion.

The feeder tube forming and weld overlay residual stress simulations by finite element methods have been successfully performed separately in the past [3, 4, 5]. However, a weld overlay residual stress analysis by finite element method that continues from the deformed shape due to bending and integrates its stress state in the simulation has not been well documented.

The unique FEA approach performed in this paper combines the feeder tube forming and weld overlay residual stress analyses into a single, continuous numerical simulation. That is, it combines multiple non-linear, elastic-plastic material behaviors into one FEA simulation, which includes large strain/large displacement, contact behavior, and residual stress from feeder tube forming, as well as element addition and removal, and residual stress from welding. The simulation presents a FEA methodology to predict the weld overlay (WOL) residual stresses in a 2.5” NPS with a 3.74” tight-radius warm bend feeder pipe. The analysis was conducted in sequence using the ANSYS finite element software package.

a) Post-Butt Weld Feeder Tube

b) Post-Weld Overlay

Figure 1: 2.5” NPS Tight-Radius Warm Bend Feeder Tube Specimen

2. Technical Approach

As prescribed previously, the objective of performing FEA is to simulate the processes of elbow forming, butt weld application, and WOL application in the correct sequence using one inclusive finite element (FE) model. Therefore, the FE model includes all components in the final WOL design, while utilizing elements with “birth and death” feature in ANSYS to activate and deactivate appropriate components during specific steps of the analysis.

Furthermore, the FEA for predicting residual stresses performed as a continuous analysis so that the loads and stress histories from different load steps are taken into account. In the analysis, the residual stresses and strains from the previous load step are input as initial conditions for the next load step computation.

Several major steps involved in the FEA simulation are illustrated in Figure 2. Essentially, these steps are those involved in bending a straight piece of feeder tube piping to form a 73° elbow using the warm-bending technique, trimming off a section of the elbow for the Grayloc hub attachment, performing the Grayloc hub attachment butt weld, deleting one inside surface elements to simulate wall thinning, and finally performing the weld overlay application, all in one continuous finite element analysis.

Due to the complexity of the many steps need to be simulated, the methodology implemented in this paper employs 3-D elastic-plastic FEA technology in ANSYS, and elements with large deformation capability, to predict the residual stresses due to the feeder tube forming and various welding processes.

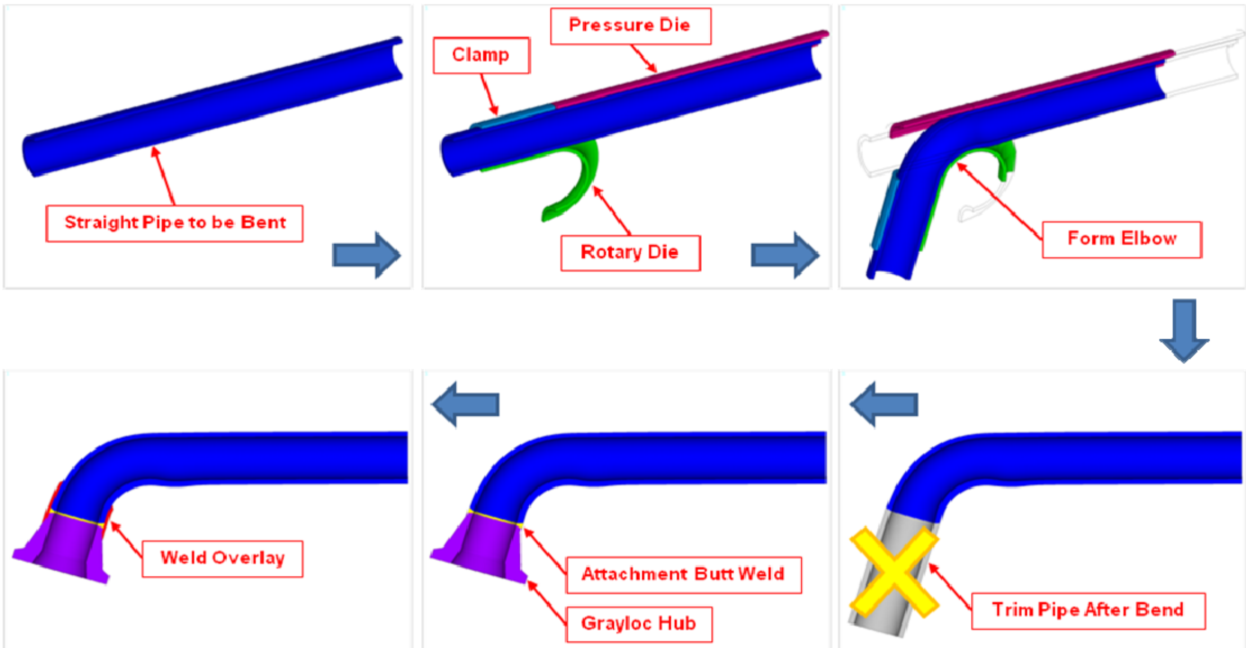


Figure 2: Major Steps Simulated Using FEA

2.1 Finite element model

The model geometry is a 2.5" Schedule 80 feeder pipe, designated as "L-Type", with a tight bend radius of 3.74" (95 mm) and a bend angle of 73. In addition, the fixture tool used to create the tight radius bend is also modeled. The tool is comprised of a clamp, a pressure die, and a rotary die.

A 3-D FE model is constructed and primarily meshed with about 40,000 solid brick elements, as shown in Figure 3. The model includes all components in the various steps of the entire fabrication process up to the completion of the weld overlay. Namely, the model includes all components prescribed above that may or may not be present at a particular step of the fabrication process. Appropriate analytical techniques are used in the analysis to only include the applicable components and material at each step of fabrication. Utilizing the "birth and death" feature in ANSYS to activate and deactivate appropriate components during specific steps of the analysis, along with appropriate material changes, the model can be used to represent the actual material and component configurations at any particular step of the analysis. Therefore, the model shown in the figure does not necessarily represent the actual material and component configurations existed at a particular step of the analysis.

Taking advantage of geometric symmetry, the resultant 3-D FE model is a 180° half model, as if the assembly is cut in half through the vertical mid-plane. Although the half model is compatible with the bending simulation, due to the nature of the half model, the welding simulation is treated as symmetrical across the symmetry plane, neglecting the actual start/stop location of the actual welding process. The developed FE model, shown in Figure 3, includes the straight feeder pipe prior to bending/forming, clamp, pressure die, rotary die, Grayloc hub, Grayloc hub-to-feeder pipe butt weld, and weld overlay.

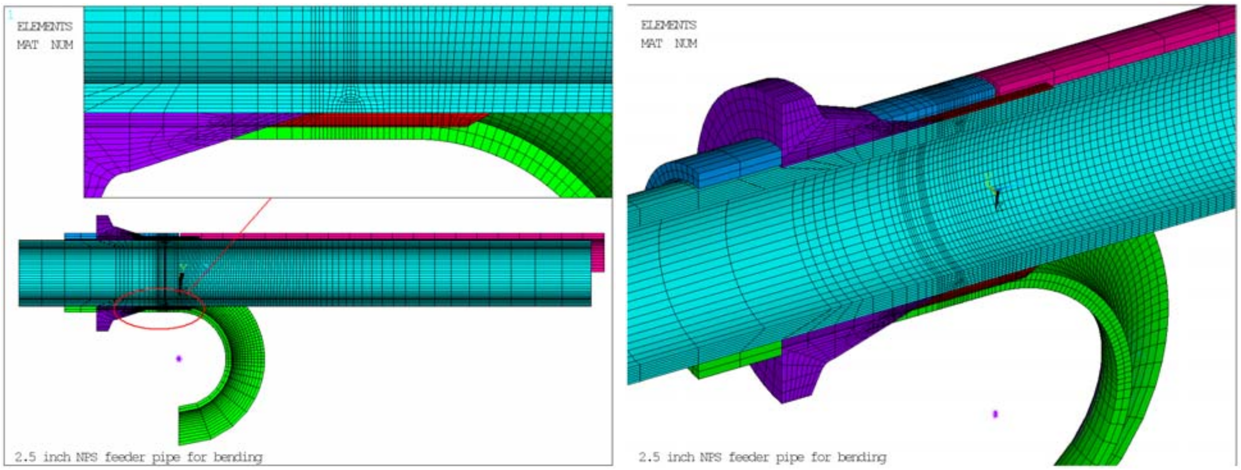


Figure 3: Finite Element Model Used in Performing the Continuous FEA

2.2 Feeder tube bending analysis

The feeder tube is formed by bending a straight pipe in a fixture consisting of the pressure die to keep the tube in position during bending, the rotary die to create the bend radius, and the clamp to grip onto the tube and rotating along with the rotary die [1]. A warm bended feeder tube is locally torch heated around the elbow intrados prior to bending [1]. Since detailed information on the local torch heating is unavailable, for the purpose of this analysis, the maximum torch heat temperature is assumed to be 1500°F(816°C), and the temperature gradient during the bending process is assumed to be unchanged; that is, no heat transfer during the bending process. This assumption is validated in the baseline case 850°C intrados temperature assumption used in a similar analysis [4], which yielded comparable FEA predictions to experimental data.

The feeder tube elbow forming simulation is a contact analysis involving large strain and large displacement. The mating surfaces between the fixture tool and unbent feeder pipe are modeled with about 10,000 surface contact surface and target surface elements, respectively. The contact behaviors during the bending process are simulated as follows:

1. The fixture tool is simulated as rigid bodies with stiffness one order of magnitude higher than the stiffness of the feeder tube.
2. As shown in Figure 4, a high coefficient of friction of 0.8 is assigned for the clamp contact pair because it is expected to fully grip onto the feeder pipe without slippage
3. The pressure die moves at the same tangential speed with the outer radius of the rotary die so that friction is irrelevant and, hence, the pressure die contact pair is assigned a frictionless contact

4. The feeder tube is expected to be in tight-slipping contact with the rotary die. Thus, a coefficient of friction of 0.4 is assigned for this contact pair, which is a typical value for rough steel surface contact.

After the bend, the fixture tool is moved away from the feeder tube to allow for spring back, and then the model is allowed to cool to room temperature. The residual stresses developed within the feeder bend are calculated using non-linear, elastic-plastic load/unload stress reversal relations. A section of the pipe is then trimmed in preparation for the Grayloc hub attachment, as illustrated in Figure 2 above.

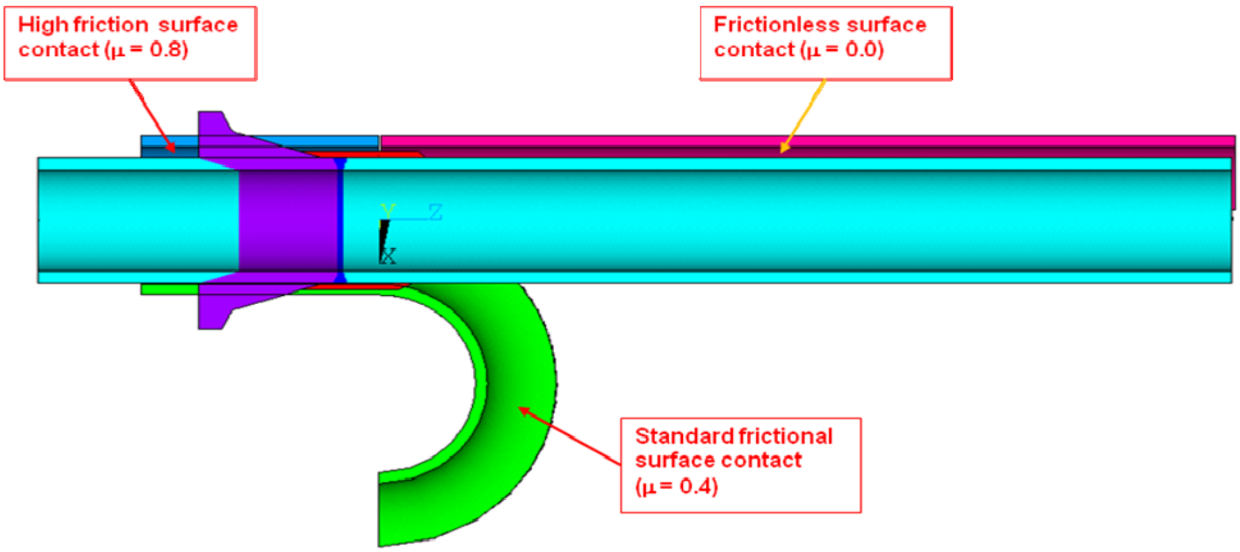


Figure 4: Surface Contact Behavior Configurations for Bending Simulation

2.3 Grayloc hub-to-feeder tube attachment butt weld

This welding process is performed to simulate the weld thermal cycle history and the resultant influence on the residual stresses on the overall geometry. Weld residual stress analyses are temperature controlled, nonlinear, and path-dependent problems as a result of the cumulative stress-strain cycling history inherent with each simulated welding process. The FEA utilizes a decoupled multi-physics simulation process, which consists of a thermal pass to determine the temperature distribution history due to the welding process, and a stress pass to calculate the residual stresses throughout the thermal transient history.

Numerically, multiple weld beads are lumped together into bigger lump passes to reduce computation time. This technique is a common practice in the FEA industry for residual stress analyses, such as those illustrated in various papers [6, 7]. The deposition of the weld metal is simulated by imposing a heat generation function on the elements representing the active lump pass for each weld. Appropriate weld heat input, heat efficiency, and appropriate cooling time, are input into the thermal pass to ensure that sufficient heat penetration is achieved, the temperature between weld passes meets the required interpass temperature, as well as obtain a reasonable overall temperature distribution is obtained within the finite element model. Then, the temperature history is imported into the stress pass to calculate the residual stresses due to the thermal cycle from the heating and cooling of the weld elements by using nonlinear, elastic-plastic load/unload stress reversal relations.

The Grayloc hub attachment butt weld is deposited via discrete volumetric weld nuggets in a step-by-step process from the root, around the circumference, and then through the pipe thickness. Each weld nugget and layer will experience the welding thermal cycle as defined by the selected heat source computation. Appropriate heat transfer effects are included with the analysis.

Specifically, the Grayloc hub attachment butt weld is simulated in three layers, with one weld bead ring in the first layer and two weld bead rings in the second and third layer each. Each bead ring is further divided into 15 lumped weld nuggets to simulate the sequential weld nugget deposition during the realistic welding process. That is, each weld nugget is deposited sequentially one after another, as illustrated in Figure 5. A total of 75 weld nuggets are defined for the Grayloc hub attachment butt weld simulation.

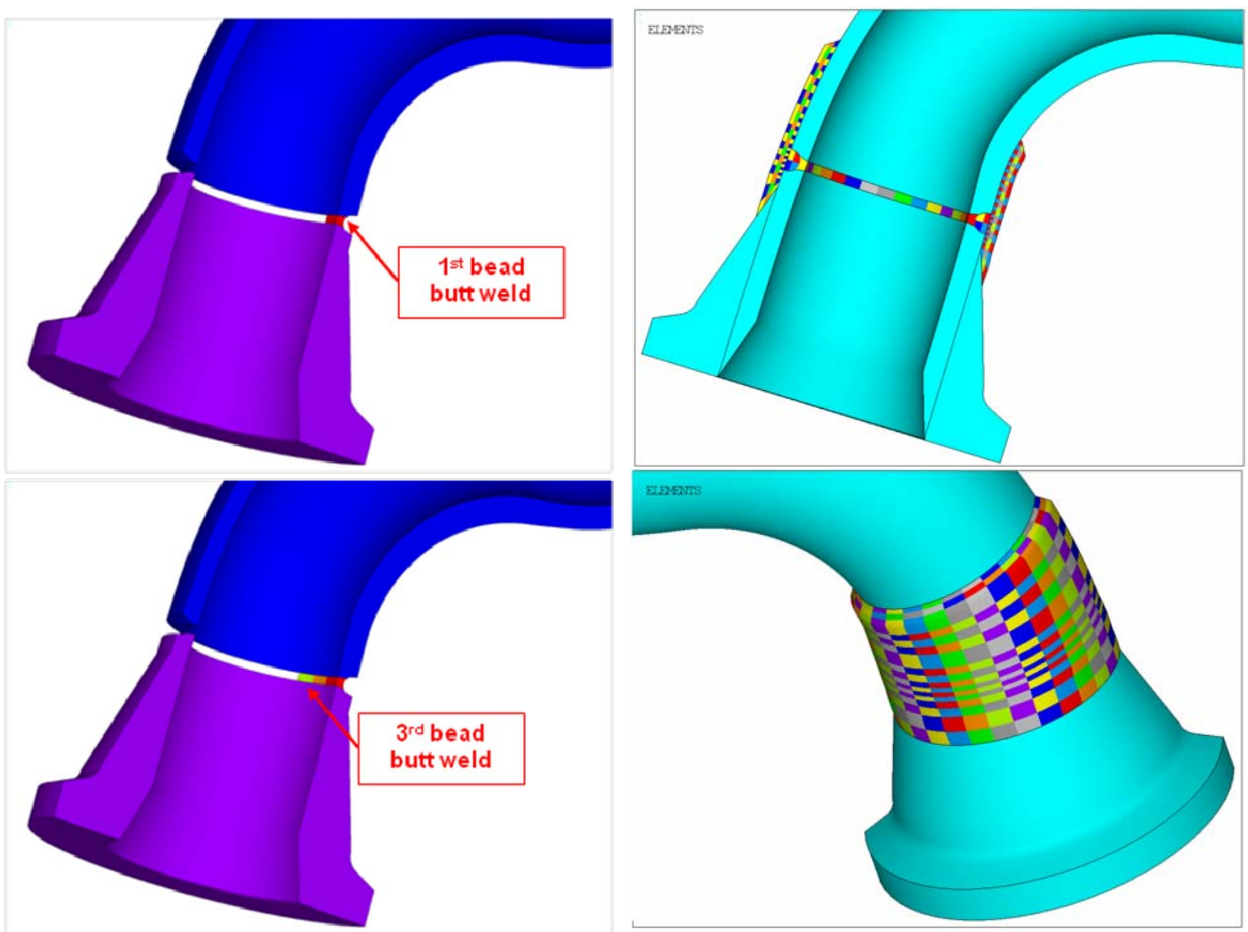


Figure 5: Sequential Deposition of Weld Nuggets and Nugget Definitions

2.4 General thinning

General wall thinning is simulated by "deletion" of the inner layer of finite elements prior to application of the weld overlay. This step is performed after the Grayloc hub attachment butt weld and is accomplished using the element "birth and death" feature of ANSYS to deactivate/remove the first inner layer of elements within the feeder tube, which is equivalent to approximately 1.0 mm of general thinning throughout the model. Residual stresses are reported before and after this step for comparison to experimental measurements, designated as "Pre-Thin" and "Post-Thin".

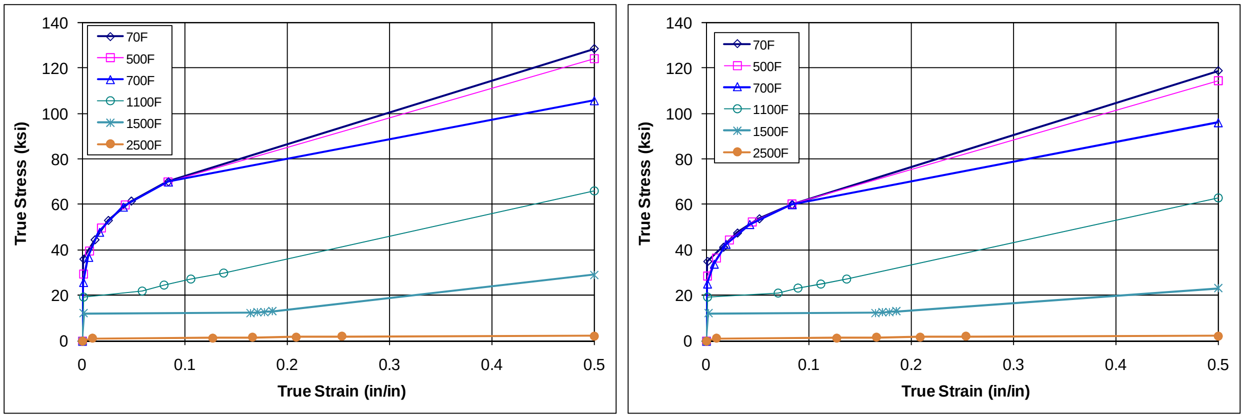
2.5 Weld overlay application

The weld overlay design is applied as the final step in the FEA. The FEA simulation process of the WOL application is similar to that described for the Grayloc hub attachment butt weld. The WOL is composed of three layers. Each layer of the WOL is applied in a step-by-step process from the first layer to the last layer, beginning at the prescribed toe location and finishing at the final toe location. The overlay encompasses the prescribed arc length within the elbow bend. Similar to the weld nugget definition for the butt weld, each weld bead ring for the WOL is also divided into 15 weld nuggets. As a result, a total of 1,020 weld nuggets are defined for the WOL simulation.

After the completion of the WOL application, the outer most element layer of the WOL is deleted to simulation surface grinding. Residual stresses are recorded after the grinding step, designated as "Post-WOL".

3. Material Properties

The materials of the components are of carbon steel variants. Temperature dependent material properties, including elastic modulus, coefficient of thermal expansion, thermal conductivity, thermal diffusivity, specific heat of capacity, yield strength, ultimate strength, and total elongation, are used in the FEA. The nonlinear stress and strain properties are based on isotropic hardening rule, which is governed by the Ramberg-Osgood stress-strain law. The derived material stress-strain curves are plotted in Figure 6.



a) SA-105 for Grayloc Hub Material b) SA-106 Grade B for Feeder Tube Material

Figure 6: Material Stress-Strain Curves

4. Results and Discussions

Two sets of results are discussed in this section: the warm-bend simulation, and the butt weld and WOL applications.

For the warm-bend simulation, the post-bending von Mises stress contour around the bend (after spring back and cooling) is plotted in Figure 7. The figure shows that the maximum stress of 99 ksi, or 683 MPa, occurs at the hump near the end of the bend.

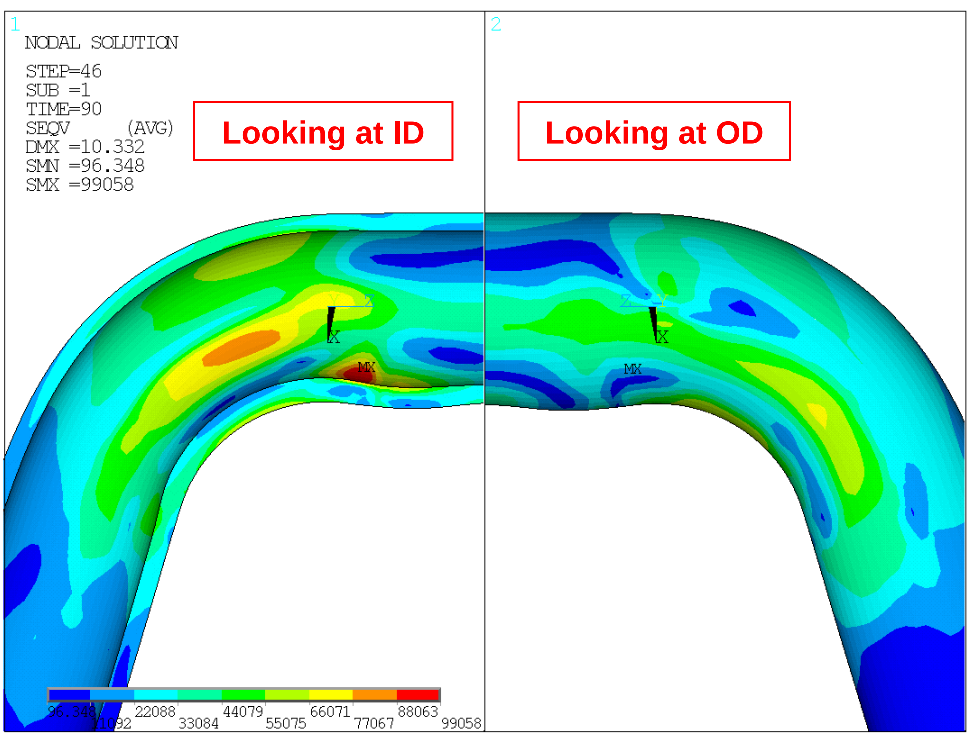


Figure 7: Post-Bend Von Mises Stress Contour Plot (psi)

For complete assessment of the analytical residual stress predictions, axial and hoop stress results have been extracted along various sections around the circumference of the model, from 0° to 180° starting at the elbow intrados towards the extrados.

For the butt weld and WOL simulation, Figures 8 and 9 depict the hoop and axial residual stress distributions after the weld overlay application. The patterns illustrated in these contour plots are typical of weld overlays – compressive stresses throughout the overlay region on the inside surface and throughout most of the underlying feeder material (blue and blue-green zones in the figures). These are counterbalanced by relatively high tension in the WOL material itself, plus in a small portion of the feeder material near the outside surface (red and orange zones).

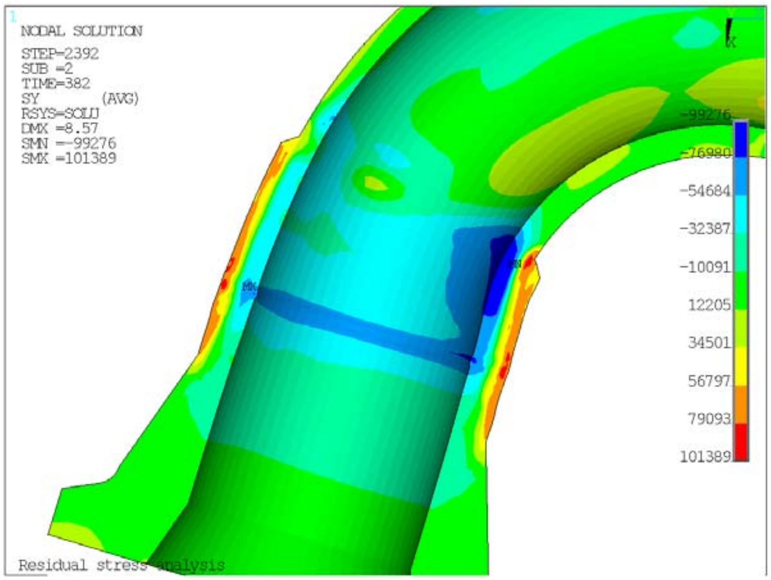


Figure 8: Post-WOL Hoop Residual Stress Contour Plot (psi)

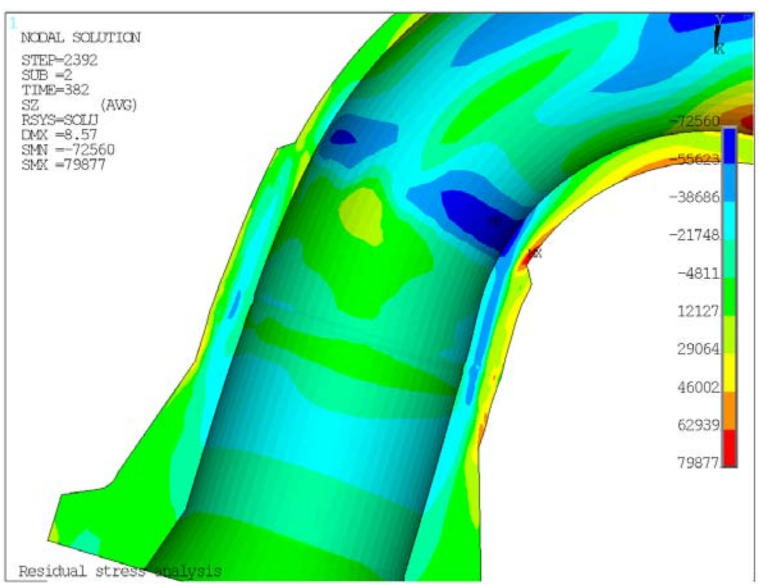


Figure 9: Post-WOL Axial Residual Stress Contour Plot (psi)

Similar to the warm-bend analysis, "Pre-Thin", "Post-Thin", and "Post-WOL" stress results in three locations along the elbow bend are extracted from the FEA at the corresponding fabrication steps for stress comparison.

Figures 10 through 12 also include the experimental data from the XRD measurements in addition to the FEA results for the purpose of stress comparison. The following observations are drawn from the comparison:

a) In general, the pre-WOL FEA stress predictions are in the same ballpark as the experimental measurements, as shown in the figures, except for the OD residual axial stress at the "Grayloc Weld" location (Figure 10) that the FEA residual axial stress tends to be over predicted by about 170 MPa at the 90° azimuth (cheek). However, the intrados (0°) and extrados (180°) FEA predictions are within the range of the experimental measurements.

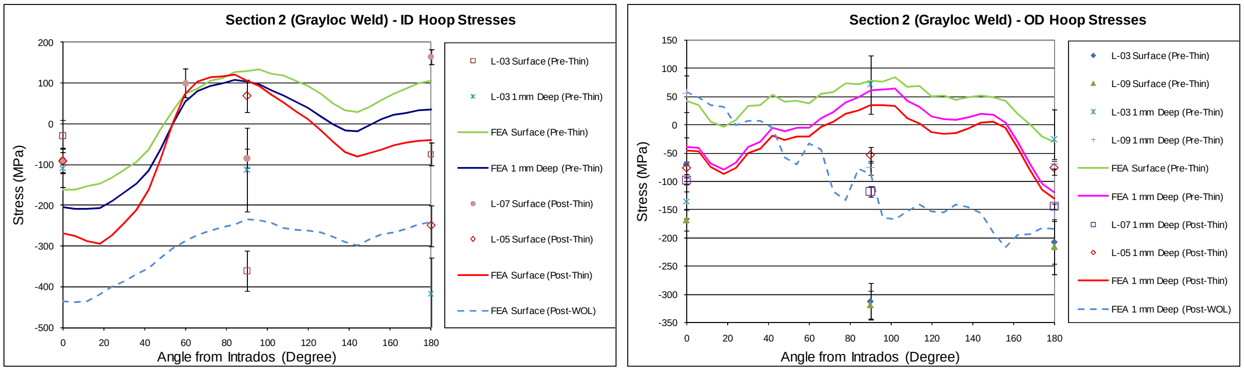
b) The Post-WOL FEA predictions at the cheek (90°) and extrados (180°), as shown in Figures 11 and 12, agree well with the experimental measurements. The FEA results tend to conservatively over predict the Post-WOL residual stresses at the intrados (0°); this could be due to the use of the half model in the analysis, where the intrados is essentially the start location of the weld bead progression.

c) Both analysis and measurements demonstrate that the WOL creates a favorable reversal of hoop stresses on the ID surface of the feeders under and in the close vicinity of the WOL. All measurements and analyses indicate that this region is placed in a compressive state of stress by the weld overlay process.

d) Analyses show that the residual stresses in the highly cold-worked regions at the Apex (mid-span of the elbow bend) of the bends in L-type feeders are not affected by the weld overlays. These regions are of sufficient distance from the ends of the overlays that the overlay welding has no effect on them.

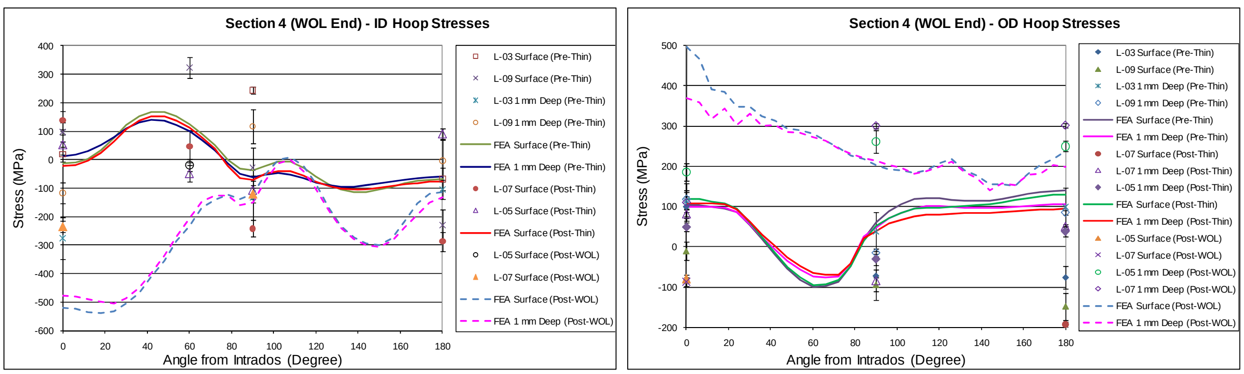
e) Both analysis and measurements show that the stresses at 1 mm depth near the OD of the overlays as well as in the original feeders at the End-of-WOL are tensile. Measurements of hoop stresses in this region are ~300 MPa, somewhat less than the predicted values by analysis, but are large enough to be a concern.

f) Measurements were performed on the original WOL (Sample L-05) as well as an overlay on an overlay in which a field repair was simulated (Sample L-07). These results, as well as analyses of the two cases, demonstrated essentially no effect of the repair. The measurement results on L-05 and L-07 are virtually the same.



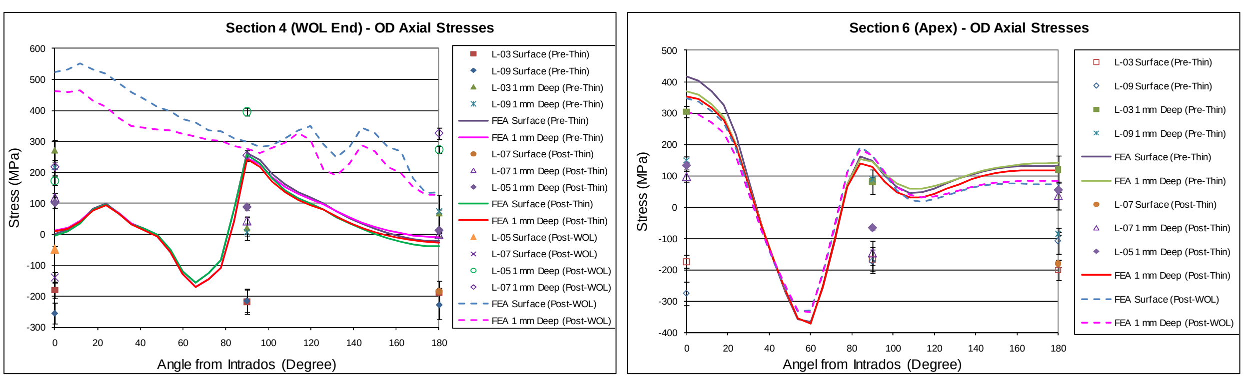
a) ID Region b) OD Region

Figure 10: Hoop Residual Stress Comparison Plot, Grayloc Weld Location



a) ID Region b) OD Region

Figure 11: Hoop Residual Stress Comparison Plot, End-of-WOL Location



a) End-of-WOL Location b) Apex Location

Figure 12: Axial Residual Stress Comparison Plot

5. Conclusions

A finite element analysis methodology to numerically predict the residual stresses on an elbow due to plastic formation and various welding processes has been developed and validated. The techniques and procedures presented in this paper have demonstrated the work performed to accomplish the finite element analysis and results validation process. Specifically, a finite element residual stress analysis of a CANDU nuclear reactor L-Type 2.5" NPS feeder tube elbow forming and weld overlay application has been performed. The results of the analysis have demonstrated that the FEA method reasonably captures the residual stress trends resulted from manufacturing warm bending, Grayloc hub attachment welding, weld overlay and WOL repair with acceptable accuracy. The FEA methodology developed in this work can be extended for use in other similar applications.

6. References

[1] M. Yetisir, et. al., PVP2005-71581, "The Effect of Manufacturing Process on Residual Stresses of Pipe Bends", Proceedings of the 2005 ASME Pressure Vessels and Piping Conference (PV2005), Denver, Colorado, USA, 2005 July 17-21.

[2] J.P. Slade, T.S. Gendron, "Flow Accelerated Corrosion and Cracking of Carbon Steel Piping in Primary Water-Operating Experience at the Point Lepreau Generating Station", *The Minerals*, Metals & Materials Society, 2005.

[3] Y. Ding, M. Yetisir, PVP2008-61113, "Residual Stress Modeling of Warm-Bent Tight-Radius CANDU Feeder", Proceedings of the 2008 ASME Pressure Vessels and Piping Conference (PV2008), Chicago, Illinois, USA, 2008 July 27-31.

[4] T. Zhang, et. al., "Weld-Overlay Analyses – An Investigation of the Effect of Weld Sequencing", Proceedings of the 2008 ASME Pressure Vessels and Piping Conference (PV2008), Chicago, Illinois, USA, 2008 July 27-31.

[5] A. Chintapalli, et. al., PVP2008-61634, "Weld Overlay Size Sensitivity on Residual Stresses in a Welded Pipe", Proceedings of the 2008 ASME Pressure Vessels and Piping Conference (PV2008), Chicago, Illinois, USA, 2008 July 27-31.

[6] Rybicki, E. F., et al., "Residual Stresses at Girth-Butt Welds in Pipes and Pressure Vessels", U.S. Nuclear Regulatory Commission Report NUREG-0376, R5, November 1977.

[7] Rybicki, E. F., and Stonesifer, R. B., "Computation of Residual Stresses Due to Multipass Welds in Piping Systems", *Journal of Pressure Vessel Technology*, Vol. 101, May 1979.