OPTIMIZATION OF WELD OVERLAY DIMENSIONS FOR BUTT-WELDED PIPES

Aparna Chintapalli
Structural Integrity Associates, Inc.
3315 Almaden Expressway, Suite 24
San Jose, CA 95118
408-978-8200, achintapalli@structint.com

G. Angah Miessi
Structural Integrity Associates, Inc.
3315 Almaden Expressway, Suite 24
San Jose, CA 95118
408-978-8200, amiessi@structint.com

Francis H. Ku
Structural Integrity Associates, Inc.
3315 Almaden Expressway, Suite 24
San Jose, CA 95118
408-978-8200, fku@structint.com

Raymond K. Yee
San Jose State University
1 Washington Square
San Jose, CA 95192
408-924-3935, Raymond.Yee@sjsu.edu

ABSTRACT

Weld overlay technique can be used on a welded pipe with a flaw in the butt weld to prevent it from cracking further. Due to the application of weld overlay on top of the weld, compressive stresses are developed in the pipe wall and the weld. These stresses counteract the effect of the residual stresses from the butt weld and tensile stresses produced in the pipe during normal operation.

Existing guidelines in the nuclear industry specify minimum dimensions (length & thickness) of the weld overlay. However, there is no guideline regarding the optimum repair dimensions that should be used to obtain minimum residual stresses induced by the weld overlay technique. The optimum dimensions in this study refer to the minimum material that can be used for the weld overlay. This results in reduced cost, time and exposure to radiation. Hence a size sensitivity study is performed by varying three parameters, the width and thickness of the weld overlay, and the size of the pipe being repaired. The repaired pipe is assumed to be subjected to typical pressurizer water reactor (PWR) operating conditions. The weld overlay process is simulated using an axisymmetric finite element model.

The axial and hoop stresses in the region of the butt weld after the weld overlay are compared. The results from this study will be analyzed to establish optimum dimensions of the weld overlay for various pipe sizes to mitigate axial and circumferential crack initiation at the butt weld.

INTRODUCTION

Weld overlay (WOL) technique has been used on a flawed (cracked) pipe to prevent it from cracking further. Due to the application of weld overlay on a pipe, compressive stresses are developed in the pipe wall. These stresses counteract the effect of tensile stresses being produced in the pipe during normal operation.

An axisymmetric finite element model as shown in Figure 6 is commonly used to perform the weld overlay residual stress analysis. However, no study has been performed on a pipe with weld overlay in comparing the residual stresses distribution from a full 3-D model with a 2-D axisymmetric model. Hence, as the first part of this study, residual stress analysis using both 3-D finite element model and 2-D axisymmetric model were performed and compared to establish the validity of the 2-D model.

In addition, existing guidelines in the nuclear industry specify minimum dimensions (length & thickness) of the weld overlay. However, there is no guideline regarding the optimum repair dimensions that should be used to minimize the tensile residual stresses induced by the weld overlay technique. The optimum dimensions in this study refer to the minimum material that can be used for the weld overlay. This results in reduced cost, time and exposure to radiation for the welders applying the overlay on the pipes. Hence a size sensitivity study was performed in this study by varying three parameters, the width and thickness of the weld overlay, and the size of the pipe. The post-WOL pipe is assumed to be subjected to typical pressurizer water reactor (PWR) operating conditions. Subsequent to the 2-D model validation, the weld overlay process is simulated using an axisymmetric finite element model. The results from this study were analyzed to establish optimum dimensions of the weld overlay for various pipe sizes to mitigate axial and circumferential crack initiation at the butt weld.

2D AXISYMMETRIC MODEL AND 3D SOLID MODEL

The 3D finite element model consists of a pipe and butt weld. Figure 1 shows the applied boundary conditions with coupling on the left hand side of the 3D finite element model to simulate plane strain conditions. Figure 2 depicts the material identification of the components. Figure 3 shows the mesh comparison between 2D axisymmetric and 3D models. The progression of the welding is from the inside surface of the pipe to the outside surface of the pipe.

The analyses are performed using the ANSYS finite element software [1]. Axisymmetric PLANE55 elements are used in the thermal analysis, while axisymmetric PLANE182 elements are used in the stress analysis for the 2D axisymmetric model. SOLID70 elements are used in the thermal analysis, while SOLID185 elements are used in the stress analysis for the 3D model. A total of 840 elements were used to build the 2D axisymmetric model, whereas 9600 elements were used to build the 3D model. The weld bead depositions are simulated using the element "birth and death" feature in ANSYS.

The "birth and death" feature in ANSYS allows for the deactivation (death) and reactivation (birth) of the elements' stiffness contribution when necessary. It is used such that elements that have no contribution to a particular phase of the weld simulation process are deactivated (temporarily disengaged via EKILL command) because they have not been deposited. The deactivated elements have near-zero conductivity and stiffness contribution to the structure. When those elements are required at a later phase, they are then reactivated (via EALIVE command).

The analyses consist of a thermal pass to determine the temperature distribution due to the welding process, and an elastic-plastic stress pass to calculate the residual stresses through the thermal history. Appropriate weld heat efficiency shown in Table 4 along with sufficient cooling time are utilized in the thermal pass to ensure that the temperature between weld layer nuggets meets the required interpass temperature as well as obtain acceptable overall temperature distribution within the FEM (i.e., peak temperature, sufficient resolution of results, etc.).

In the stress pass, symmetric boundary conditions are applied to the symmetry planes of the model and the weld centerline, and coupling boundary condition in the axial direction are applied on the free end of the pipe to ensure uniform axial displacement (shown in Figure 1).

MATERIAL PROPERTIES

Pipe material is Type 304 stainless steel. The weld material is Alloy 600. The WOL material is Alloy 690.

The thermal and structural properties (E, α, k, c_p) are obtained from Reference 2 for Type 304 stainless steel and Alloy 600. Stress-strain properties (yield stresses and tangent moduli) required for residual stress analysis are listed in the Table 3. Note that properties at high temperatures are necessary for simulating the behavior of molten metal in the residual stress analysis.

The yield stresses and tangent moduli for Alloy 600 (the base metal for Alloy 82/182) and Type 304 stainless steel are obtained from Reference 3.

The yield strength (YS) as a function of temperature for Alloy 690 (assumed base metal for Alloy 52M) are obtained from Reference 11. The tangent moduli (TM) for Alloy 690 are based on the tangent moduli of Alloy 600. To obtain temperature dependent YS and TM data for the material of interest, ASME Code [12] ultimate tensile stress (UTS) values at 70°F were obtained for both materials. The values from Reference 12 are, Alloy 600: UTS = 80 ksi; Alloy 690: UTS = 85 ksi. A UTS ratio for both materials is calculated and applied to Alloy 600 values (whose original values are obtained from Reference 3), to obtain the Alloy 690 TM temperature dependent values.

This analysis applies the bilinear kinematic hardening material behavior available within the ANSYS finite element program [1].

DIMENSIONS FOR 2D AXISYMMETRIC AND 3D MODELS

The dimensions and geometry of the 2D axisymmetric and 3D models are given in Table 1.

Table 1: Pipe Dimensions Used in the 2D And 3D Model Study

Pipe ID	3.82”
Pipe OD	4.5”
Pipe Thickness	0.337”
Height of weld	0.47”
Width of Weld	0.5”
Area of Nugget 1	0.0148536 in^2
Area of Nugget 2	0.0145236 in^2
Area of Nugget 3	0.0177641 in^2
Area of Nugget 4	0.0255527 in^2

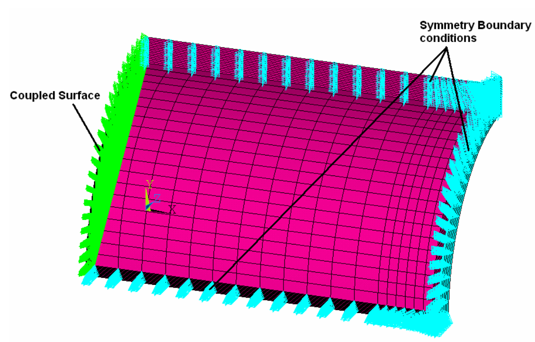

Figure 1: Boundary Conditions for 3D Model

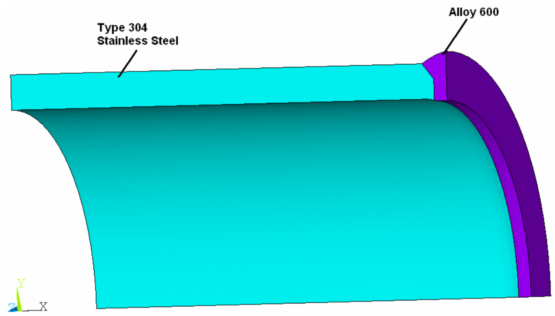

Figure 2: **Material Designation for the 3D Model (Quarter model of the pipe)**

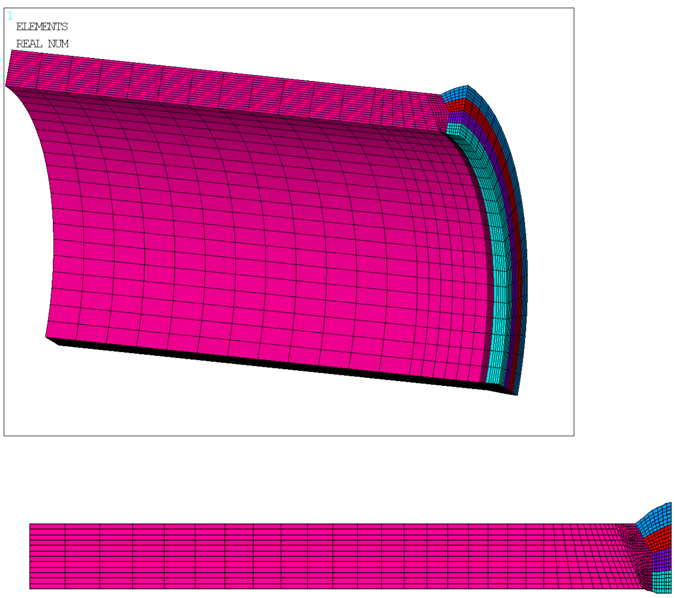

Figure 3: **2D Axisymmetric And 3D Solid Model**

2D AXISYMMETRIC MODEL FOR THE WOL PARAMETRIC STUDY

The 2D axisymmetric finite element model used in the WOL parametric study consists of a pipe, a pipe to pipe butt weld, and a WOL application. Figure 4 shows symmetry boundary conditions on the right hand side of the pipe and coupling in the axial direction on the other side of the pipe to simulate plane strain conditions. Figure 5 depicts the material identification for the components. Figure 6 shows the finite element mesh of the axisymmetric model. The progression of the welding is from the left to the right side of the pipe.

The weld overlay repair simulation is performed using the ANSYS finite element software [1]. Axisymmetric PLANE55 elements are used in the thermal analysis, while axisymmetric PLANE182 elements are used in the stress analysis of the simulation. The weld bead depositions are simulated using the element "birth and death" feature in ANSYS.

The analyses consist of a thermal pass to determine the temperature distribution due to the welding process, and an elastic-plastic stress pass to calculate the residual stresses due to the thermal history.

In the stress pass, symmetric boundary conditions are applied on one end of the pipe, and couples in the axial direction are applied on the other end of the pipe to ensure uniform axial displacement.

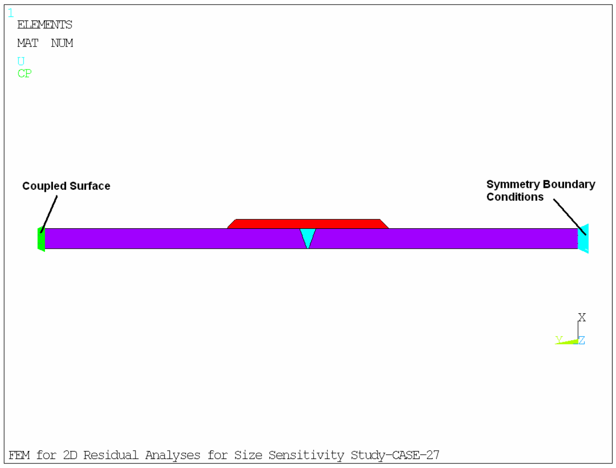

Figure 4: Applied Boundary Conditions

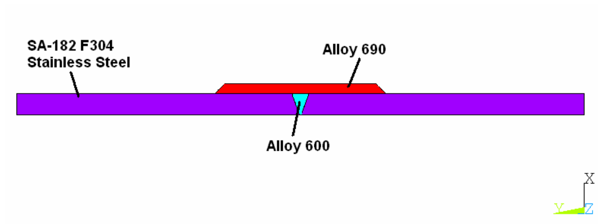

Figure 5: Material Designation

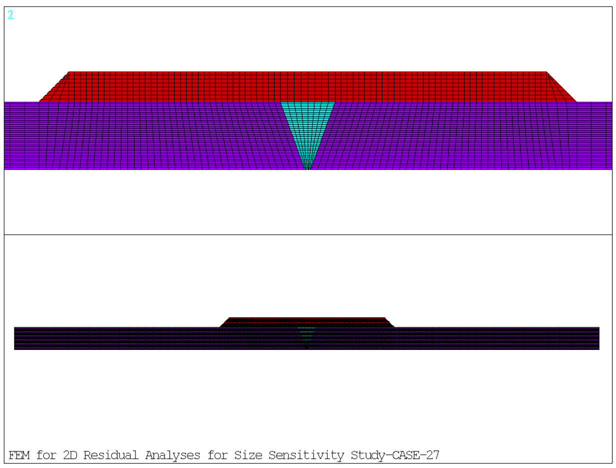

Figure 6: Mesh Pattern of the Axisymmetric Model

WOL PARAMETRIC STUDY CASES

Three variables are considered in this parametric study, namely, the WOL width, WOL thickness and the pipe size. As a result, 27 cases of residual stress analysis are considered in this study with the above varying parameters. The dimensions of the finite element models for the 27 cases are shown in Table 2.

Table 2: Model Dimensions for WOL Parametric Study

Case No.	WOL Width (in)	WOL Thickness (in)	Pipe Thickness (in)	Pipe ID (in)
1	1.706	0.3	0.438	2.624
2	2.493			
3	3.279			
4	1.706	0.4		
5	2.493			
6	3.279			
7	1.706	0.5		
8	2.493			
9	3.279			
10	4.066	0.3	0.719	7.187
11	4.853			
12	5.640			
13	4.066	0.4		
14	4.853			
15	5.640			
16	4.066	0.5		
17	4.853			
18	5.640			
19	6.426	0.3	1.125	10.5
20	7.213			
21	8.000			
22	6.426	0.4		
23	7.213			
24	8.000			
25	6.426	0.5		
26	7.213			
27	8.000			

Table 3: **Yield Strength (YS) and Tangent Modulus (TM)**

Temp. (°F)	Alloy 82/182/600[3]		TP 304[3]		Alloy 52/52M/690	
	YS (ksi)	TM (ksi)	YS (ksi)	TM (ksi)	YS [14] (ksi)	TM*** (ksi)
70	53.9	531.1	35.8	531.1	49.2	564.3
550	46.0*	361.5*	26.5	361.5	36.4	384.1
1000	45.7*	216.1*	19.1	216.1	32.7	229.6
1300	41.6	138.6	15.5	138.6	30.5	147.3
1600**	24.7	80.5	10.5	80.5	27.0	85.5
>=2500**	2.0	5.0	2.0	5.0	2.0	5.0

Notes:

* Linearly Interpolated.

** Assumed based on data.

*** Values of tangent moduli for Alloy 690 are based on the corresponding values for Alloy 600. To obtain temperature dependent YS and TM data for the material of interest, ASME Code ultimate tensile stress values at 70°F were obtained for both materials. Alloy 600: UTS = 80 ksi; Alloy 690: UTS = 85 ksi. A UTS ratio for both materials is calculated and applied to Alloy 600 values to obtain the Alloy 690 TM temperature dependent values.

WELD BEAD SIMULATION

In order to reduce computational time, individual weld beads or passes are lumped together into weld nuggets. This methodology is based on the approaches used in References 7 through 10.

The number of equivalent bead passes is estimated by dividing each nugget area by the area of an individual bead. The resulting number of equivalent bead passes per nugget is used as a multiplier to the heat generation rate.

The butt weld for the 2D axisymmetric model and 3D model is performed in four layers, with one nugget per layer. As a result, a total of four nuggets are defined for the butt weld.

The number of nuggets for the butt weld of the axisymmetric model for the parametric study varies with the pipe size. For the 2.5" nominal pipe, the butt weld is performed in four layers, with one nugget per layer for the first two layers and two nuggets per layer for the last two layers. As a result, a total of six nuggets are defined for the butt weld.

For the 7" nominal pipe, the butt weld is performed in six layers, with one nugget per layer for the first three layers and two nuggets per layer for the last three layers. As a result, a total of nine nuggets are defined for the butt weld.

For the 10.5" nominal pipe, the butt weld is performed in ten layers, with one nugget per layer for the first five layers and two nuggets per layer for the last five layers. As a result, a total of fifteen nuggets are defined for the butt weld.

Similarly for the parametric study, the number of nuggets for the weld overlay varies with WOL width as well as the thickness.

HEAT INPUT SIMULATION

Analytically, the deposition of the weld metal is simulated by imposing a heat generation function on the elements representing the active weld nugget for each weld. In ANSYS, the heat generation (*HGEN*) applied to the weld nuggets is a volumetric energy rate, as energy per volume per time.

Algebraically, the heat generation rate (*HGEN*) to be input in ANSYS is the heat efficiency (e) times the calculated total heat input (Q) divided by the volume of the corresponding weld nugget (V_{nugget}) and the ramp time (t_{ramp}):

Q_L = Heat input per unit length

e = Heat input efficiency

t_{ramp} = Heat input ramp time

A_{nugget} = Nominal cross-sectional area of a weld nugget

A_{bead} = Cross-sectional area of a typical weld bead

$N = A_{nugget} / A_{bead}$ = Number of beads per weld nugget

L_{circ} = Nominal circumference of a weld nugget

$V_{nugget} = L_{circ} A_{nugget}$ = Approximate volume of a weld nugget

$Q = NQ_L L_{circ}$ = Total heat input for a full 360° circumferential weld nugget

$$HGEN = \frac{eQ}{V_{nugget} t_{ramp}} = \frac{eNQ_L L_{circ}}{L_{circ} A_{nugget} t_{ramp}} = \frac{eQ_L}{A_{bead} t_{ramp}}$$

Eq. (1)

The assumed heat input parameters for the weld repair are presented in Table 4. The assumed heat input parameters for the weld overlay are summarized in Table 5.

Table 4: Weld Repair Heat Input Parameters

Parameter	Value
Weld bead width(1)	0.25 in
Weld bead thickness(1)	0.08 in
Weld bead area (A_{bead})	0.02 in^2
Heat input 1st nugget (Q_L)(1)	22 kJ/in
Heat input remaining nuggets (Q_L)(1)	30 kJ/in
Heat efficiency (e)(1)	0.70

(1) Assumed values.

Table 5: Weld Overlay Heat Input Parameters

Parameter	Value
Weld bead width(1)	0.25 in
Weld bead thickness(1)	0.10 in
Weld bead area (A_{bead})	0.025 in^2
Heat input 1st nugget (Q_L)(1)	30 kJ/in
Heat input remaining nuggets (Q_L)(1)	32 kJ/in
Heat efficiency (e)(1)	0.70

(1) Assumed values.

WELDING SIMULATION

The residual stress analysis conducted herein is a temperature controlled weld residual stress simulation. It is also a nonlinear, path-dependent problem as a result of the cumulative stress-strain cycling history inherent with the simulated welding process.

Since multiple weld beads are lumped into one lump nugget in the finite element analysis, the heat input for each lumped nugget is the total heat generated by the number of theoretical weld beads that it includes. The calculated total heat input is imposed onto the active nugget simultaneously, which is conservative and will result in extremely high temperatures (above 3,000°F) at or near the peak of the heat generation period. However, the melting point of the affected material is slightly below 3,000°F. This, in turn, implies that a weld element temperature of 3,000°F and above represents molten material, and no additional heat is necessary.

As a result, a temperature control algorithm is implemented into the heat generation application such that it is turned off when the temperatures of all active weld elements have reached 3,000°F during the HGEN period.

Moreover, since temperature results that are beyond 3,000°F are unrealistic, a maximum temperature of 3,000°F is the upper limit for the residual stress analysis. This temperature control algorithm essentially restricts the maximum achievable temperature to 3,000°F when they are imported for the stress calculation.

After the butt weld is completed, the model is cooled down to a uniform ambient temperature of 70°F. The weld overlay simulation is then subsequently applied.

ASSUMPTIONS AND JUSTIFICATIONS

In the finite element model, the following assumptions are used in the residual stress evaluation:

1. Homogeneous, isotropic material behavior.
2. No latent heat of weld beads is assumed nor phase transformation effects included in the model.
3. Heat loss due to radiation is neglected.
4. Due to the nature of the axisymmetric model, the torch point heat source is treated as a full circumferential 360° volumetric heat source in the analytical model. Therefore, effects due to heat source travel circumferentially and local effects at the start/stop of the welding are neglected.

5. In order to reduce computational time, simplification of the individual weld beads is done by not explicitly modeling each one; rather a lumped pass or nugget (i.e., composite number of weld bead passes) that represents the calculated number of individual weld bead passes is used.

6. Weld bead size of the butt weld is assumed to have a bead width of 0.25” and a bead thickness of 0.08”. The area of a weld bead is calculated as 0.02 in^2. The assumed bead area is used to calculate the equivalent number of bead passes for a nugget.

7. Weld bead size of the weld overlay is assumed to have a bead width of 0.25” and a bead thickness of 0.10”. The area of a weld bead is calculated as 0.025 in^2. The assumed bead area is used to calculate the equivalent number of bead passes for a nugget.

8. Surface convection is assumed to be constant in the analysis. A convection heat transfer coefficient boundary condition of 5.0 Btu/hr-ft^2-°F at 70°F bulk ambient temperature is applied to simulate the air condition at the inside surface of the pipe during the application of the butt weld. A value of 5.0 Btu/hr-ft^2-°F is considered a reasonable and representative value for free (non-forced) convective heat transfer in air. This is the condition under which the butt weld would have been made. Convective heat transfer is assumed for the ID of the pipe as the fabrication of the component would have typically been done in the shop and the inside surface of the pipe would have been open to atmospheric air or purge gas. An insulated boundary condition would not be appropriate.

9. During the weld overlay process, the applied heat transfer boundary condition of 5.0 Btu/hr-ft^2-°F at 70°F bulk temperature was prescribed at the inside surface of the pipe to simulate air inside the pipe .

10. A heat transfer coefficient of 5.0 Btu/hr-ft^2-°F at 70°F was prescribed at the outside surface of the pipe during the application of the WOL process (exclusive of the mutual surface between the weld overlay material and original pipe material). This represents an open air environment. A value of 5.0 Btu/hr-ft^2-°F is considered a reasonable and representative value for free (non-forced) convective heat transfer in air (natural convection condition).

11. As shown in Tables 4 and 5, simplified heat inputs are assumed during the various weld layers and welding processes.

12. Heat efficiency (η) values are obtained from references 4 and 5, to account for the specified welding process (i.e., GTAW, GMAW, SAW, etc.), which can vary significantly. For the GTAW process, the heat efficiency can be ranged from 0.60 – 0.80 as cited in reference [4] and 0.21 – 0.48 as cited in reference [5]. A η value of 0.70 during the heat generation period is used for all welding processes. Use of these parameters is not intended to model the specific welding process. Rather the combined heat input (which in ANSYS equals the heat efficiency value times the User set heat generation) is selected so that the time necessary to heat the selected nugget to molten temperatures is reasonable (a few seconds).

13. A ramp time of 6 seconds (for both ramp up and ramp down) is assumed for this analysis. Ramp time is the period of application of the heat in the analysis, whether inputting (ramp up) or removing (ramp down) the heat source.

14. A maximum temperature of 350°F between the depositions of weld nuggets is assumed for all welding processes as per Reference 6.

ACCEPTANCE CRITERIA

The analytical procedure described in previous sections has provided reasonable results as seen in previous similar analyses and comparable results to test data. This can be demonstrated by observing the fusion boundary prediction of the welds. Figure 7 shows the predicted fusion boundary for the three welding processes as generated by ANSYS for this specific overlay. The fusion boundaries represent the predicted maximum temperature contour mapping that the weld nugget elements will reach during each welding process. Note that the figure is a composite showing the maximum temperature among all nuggets of each weld.

The figure shows that all weld elements have reached temperatures between 2,677°F and 3,000°F. It also shows that the heat penetration depth, where temperatures are above 1,300°F, is similar in size to the heat affected zone (HAZ) of roughly between 1/8” and 1/4”.

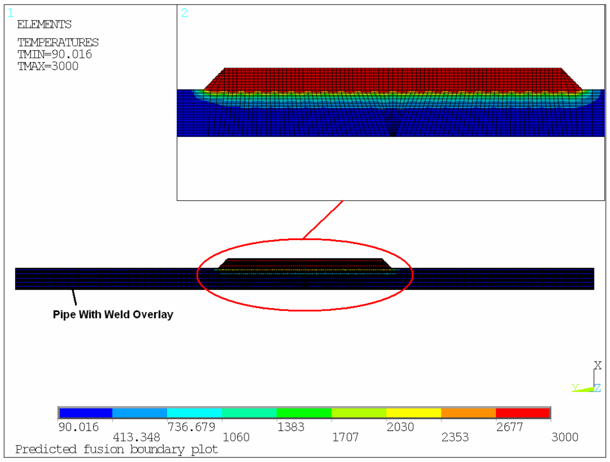

Figure 7: Fusion Boundary Plot for the Weld Overlay

RESULTS AND DISCUSSIONS

2D Axisymmetric Model And 3D Solid Model

Figures 8 and 9 show the comparison between the axial and hoop residual stress distributions for the as-welded condition at 70°F, respectively. The stresses are plotted along the weld centerline from the inside diameter of the pipe to the outside diameter. The axial direction and the hoop direction are with respect to the global coordinate system, axial is (SY) and hoop is (SZ). It can be seen from the figures that the trend for axial and hoop stress plots for 3D model and 2D axisymmetric model are similar. Furthermore, the stresses on the inside surface of the pipe are more relevant to the purpose of this study, and these stresses are in good agreement. Therefore, the 2D axisymmetric model is a valid approach and can be used in the parametric study.

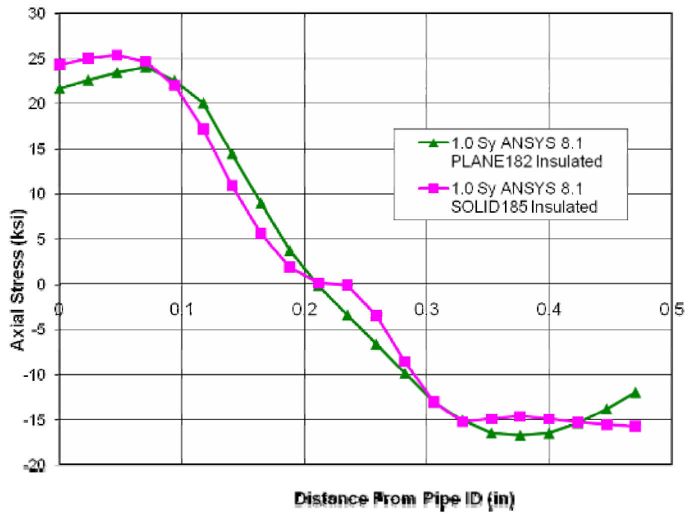

Figure 8: **Axial Residual Stress Comparison for 2D And 3D models**

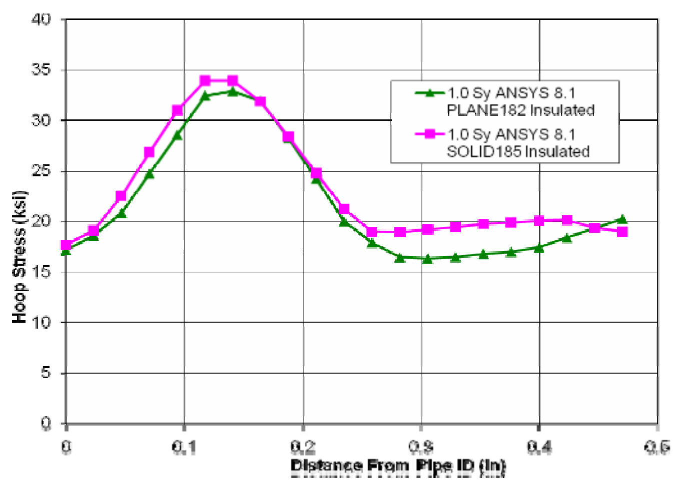

Figure 9: Hoop Residual Stress Comparison for 2D And 3D models

Axisymmetric Model Parametric Study

Figures 10 and 11 show the axial and hoop point value stress results for the 2.5” pipe on the inside surface from the parametric study. These are 3D plots, showing the changes in the stresses with varying WOL width and thickness. The axial and hoop stresses for the plots are obtained from the location at 0.5” away from the weld centerline as shown in Figure 12. A distance of up to 0.5” on either side of the weld centerline forms the area of interest in this study. Compressive stresses resulting from the application of the WOL are expected in this area.

Figures 13 and 14 show the axial and hoop point value stress results for the 7” pipe on the inside surface respectively. Similarly, Figures 15 and 16 show the axial and hoop point value stress results for the 10.5” pipe on the inside surface with respect to varying WOL width and thickness.

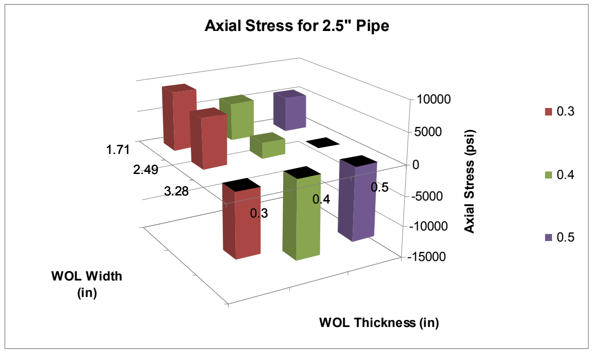

Figure 10: Axial Stress Distribution for the 2.5" Pipe

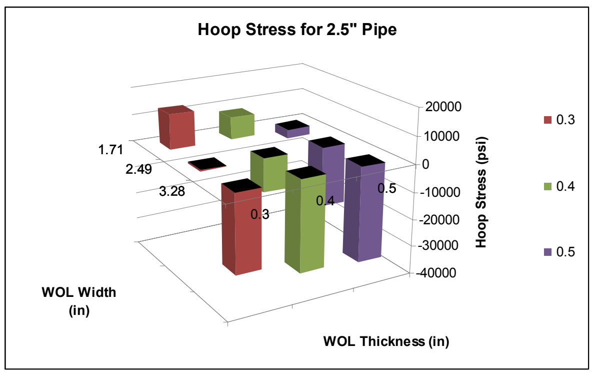

Figure 11: Hoop Stress Distribution for the 2.5" Pipe

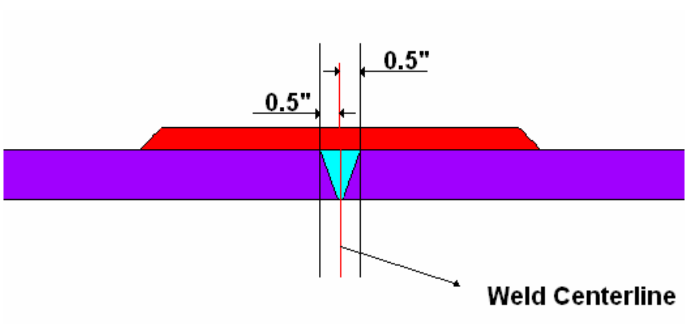

Figure 12: Area of Interest for the Parametric Study

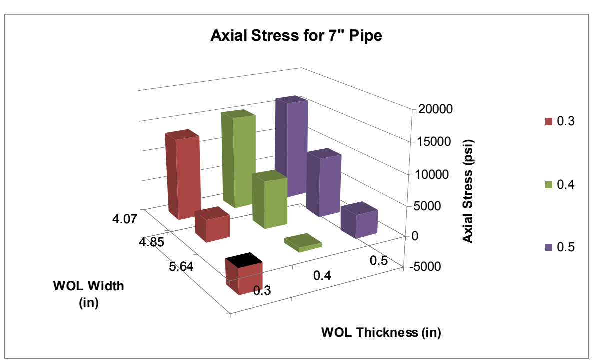

Figure 13: Axial Stress Distribution for the 7" Pipe

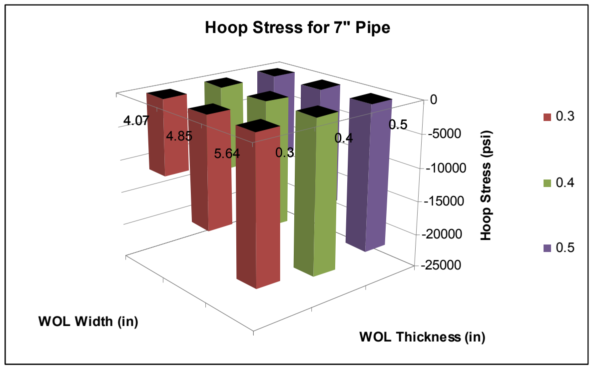

Figure 14: Hoop Stress Distribution for the 7" Pipe

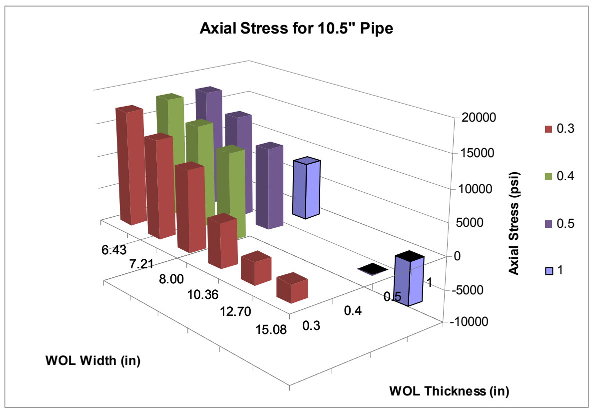

Figure 15: Axial Stress Distribution for the 10.5" Pipe

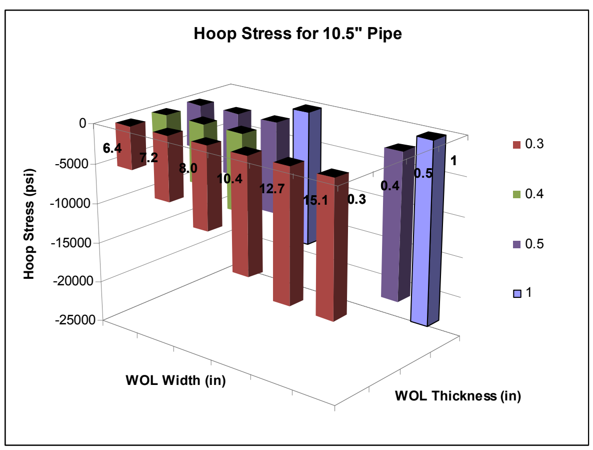

Figure 16: Hoop Stress Distribution for the 10.5" Pipe

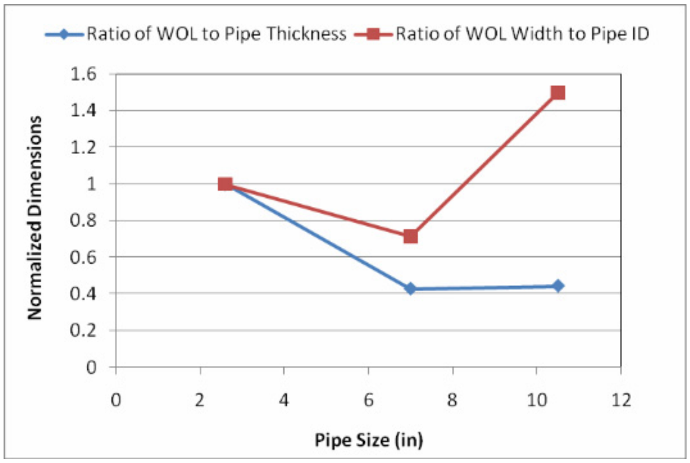

Figure 17: **Normalized Dimensions Plot With Pipe Size**

DISCUSSIONS AND CONCLUSIONS

The following conclusions are made from the parametric study on weld overlay size sensitivity on residual stresses in a pipe:

- Optimal dimensions for 2.5" pipe are found in Case 8 (thickness of WOL = 0.5", width of WOL = 2.49", area of WOL= 1.245 in^2) because compressive axial (-53 psi) and hoop stresses (-22542 psi) are obtained on the inside surface of the pipe with these dimensions utilizing minimum amount of material for applying the WOL.
- Similarly for 7" pipe, the optimal dimensions are found in Case 12 (thickness of WOL = 0.3", width of WOL = 5.64", area of WOL = 1.692 in^2).
- Axial stress plots for 10" pipe show that compressive axial stresses cannot be obtained at the Dissimilar Metal Weld (DMW) region from the existing cases. Hence more residual stress runs are performed with increasing WOL width and thickness, until compressive stresses are shown in the area of interest with the optimal WOL dimensions. This occurs when the width is increased to 15.08" and the thickness to 0.5" for the WOL. Hence the optimum area of the WOL for a 10.5" pipe is 7.54 in^2.
- From the trend shown in the stress comparison bar charts, it is observed that residual stress improvement is more sensitive to the WOL width rather than the WOL thickness.
- It can be seen from Figure 17, which plots the normalized optimum WOL thickness (to pipe thickness) and width (to pipe size), that the trend is not linear. A generalized trend cannot be determined from the available data points.

Due to the limited scope of this work, further studies are recommended to be performed to predict the optimum dimensions for a WOL for other pipe sizes.

ACKNOWLEDGMENTS

I would like to acknowledge Mr. Angah Miessi, Mr. Francis Ku, Mr. Richard Bax, and Dr. Raymond Yee for their guidance and support in completing this paper and to thank them for their advice and suggestions.

REFERENCES

1. ANSYS\Mechanical, Revision 8.1 (w/Service Pack 1), ANSYS Inc., June 2004.
2. ASME Boiler and Pressure Vessel Code, Section II, Part D, Material Properties, 2001 Edition with Addenda through 2003.
3. EPRI Report NP-7085-D, Project T303-1, "Inconel Weld-Overlay Repair for Low-Alloy Steel Nozzle to Safe-End Joint," January 1991.
4. Kou, S., "Welding Metallurgy," 2nd Edition, Wiley-Interscience, 2003.
5. Lancaster, J. F., "Metallurgy of Welding," 6th Edition, Abington Publishing, 1999 Page No. 139.
6. ASME Boiler and Pressure Vessel Code Case N-740, "Dissimilar Metal Weld Overlay for Repair of Class 1, 2, and 3 Items," Section XI, Division 1.
7. 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.
8. 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, Page No. 149.
9. Dong. P., "Residual Stress Analysis of a Multi-Pass Girth Weld: 3-D Special Shell Versus Axisymmetric Models," Journal of Pressure Vessel Technology, May 2001, Vol. 123, Page No. 207.
10. *Materials Reliability Program: Technical Basis for Preemptive Weld Overlays for Alloy 82/182 Butt Welds in PWRs (MRP-169),* EPRI, Palo Alto, CA, and Structural Integrity Associates, Inc., San Jose, CA: 2005. 1012843.
11. Special Metals (www.specialmetals.com) INCONEL Alloy 690 Information, Special Metals Corporation, October 2003.
12. ASME Boiler and Pressure Vessel Code, Section II, Part A, Ferrous Material Specifications, 2001 Edition with Addenda through 2003.