Effect of Bending on Overpressurized Isolated Piping During Postulated Accident Conditions

John C. Minichiello [(1)], Dr. Taha D. Al-Shawaf [(1)], Evan D. Katzke [(2)]

3) Technical Consultant, Duke Engineering and Services, Naperville, IL
4) Engineer, Duke Engineering and Services, Charlotte, NC

ABSTRACT

The primary objective of this paper is to show that the change in cross sectional area due to bending in overpressurized isolated piping is negligible. It also shows that an increase in internal pressure will result in a decrease in the deformed area change due to bending. These results were obtained using 3-D thin shell non-linear finite element analyses. The results also show that for low hoop strain levels (< 2%), the axial stress due to internal pressure plus bending is very close to the elastic closed form solution.

INTRODUCTION/BACKGROUND

In Generic Letter 96-06 [1], the US Nuclear Regulatory Commission (NRC) expressed concerns that the containment penetration areas of nuclear power plants could become overpressurized during a Loss of Coolant Accident (LOCA) event. Their concerns arose because the isolation valves would close (or already be closed) and the piping might be filled with water. The water, which may be originally cool, would slowly heat up and increase in pressure to a level potentially exceeding the capacity of the piping, penetration, or fittings between the inboard and outboard containment isolation valves. Thus, the NRC requested licensees review their penetration arrangements to ensure overpressurization would not occur (e.g., by providing relief devices), or, if no relief devices existed, that the piping, penetrations, and fittings could withstand the pressure without loss of function.

In an effort to develop alternative acceptance criteria, EPRI performed analysis and testing in 1997 on closed end piping filled with water and then heated [2]. The EPRI results were promising, and indicated that acceptance criteria based on strain limits might be a viable approach. Strain, rather than stress, seemed more appropriate for this "overpressure," since the pressure is controlled by the volumetric expansion of the water, which the pipe prevents.

During their review of the EPRI work, the NRC raised a number of concerns about the use of strain criteria. One significant concern was the use of strain criteria without considering all concurrent loads (deadweight, thermal expansion), since the presence of piping bending loads may increase the strain to unacceptable levels. EPRI evaluated this concern using a simple closed form solution, and concluded that, for axial membrane stresses up to 10 ksi (considered to be an upper bound for weight stresses), the effect on overall strain was minimal [2]. The purpose of this paper is to present a finite element analysis which will address the following:

- At low levels of plastic strain (on the order of 5%), the stress in the pipe due to bending moment is reasonably well predicted by classical elastic stress equations
- The total growth in cross-sectional area of the pipe is essentially the same with a bending moment and pressure compared to pressure alone. This will ensure that the volume increase due to pressure alone can be used to determine the thermodynamic pressure due to the entrapped, heated water volume expansion.

By addressing these issues, and controlling the bending stress from the standard piping analysis to a level no more than that used in the studies, one can use simplified methods to predict the pressure rise and the resulting stresses in the piping.

AXISYMMETRIC LOADING

In order to ensure the problem was well understood, the ANSYS program was first used to generate a 3-D finite element model of a thin shell cylinder. This model utilized elastic-plastic analysis including large deflection and large strain effects. This model was used to run two axisymmetric loading conditions: pressure alone and pressure with axial compression. The closed form solution from [3] for plastic hoop strain under internal pressure with and without a sustained axial stress was used to benchmark the finite element model. Plastic hoop strains were determined at incremental pressures and compared to the closed form solution at the same pressures.

The ANSYS 3-D model utilizes elastic-plastic analysis including large deflection and large strain effects using plastic shell element Shell43 [4]. The elastic shell element Shell63 is also used in the 3-D shell model, but not in the plastic region of the model. When using the large strain capability, all stress-strain input and results are in terms of true stress and true strain [4].

CLOSED FORM SOLUTION

The general expression for true hoop strain is [3]:

$$\delta_1 = \left[\frac{\sigma_1'}{k}\right]^{\frac{1}{n}} \left[\alpha^2 + \beta^2 + 1 - \alpha\beta - \alpha - \beta\right]^{\frac{(1-n)}{2n}} \left[1 - \frac{\alpha}{2} - \frac{\beta}{2}\right] \tag{1}$$

Where:

P = pressure

F = axial force

k = strength coefficient

n = strain-hardening exponent

$\alpha = \sigma_2'/\sigma_1' = (PD/4t + F/\pi Dt)/(PD/2t) = ½ + 2F/\pi PD^2$

$\beta = \sigma_3'/\sigma_1' = 0$

$\sigma_1' = PD/2t$ (hoop stress)

Assume:

$D = D_o e^{\delta 1}$, where D_o is the original mean diameter

$t = t_o e^{-\delta 1}$, where t_o is the original thickness

Note that α has been modified to include the axial stress term $F/\pi Dt$. By substituting the above statements into the general equation for hoop strain, the following equation is obtained.

$$\delta_1 = \left[\frac{3}{4} + \left(\frac{2F}{\pi P D_o^2 e^{2\delta_1}}\right)^2\right]^{\frac{(1-n)}{2n}} \left[\frac{P D_o e^{2\delta_1}}{2 k t_o}\right]^{\frac{1}{n}} \left[\frac{3}{4} - \frac{F}{\pi P D_o^2 e^{2\delta_1}}\right] \tag{2}$$

For F = 0 (no axial force), the equation simplifies to:

$$\delta_1 = \left[\frac{3}{4}\right]^{\frac{(1+n)}{2n}} \left[\frac{P D_o e^{2\delta_1}}{2 k t_o}\right]^{\frac{1}{n}} \tag{3}$$

The above equations were used to obtain the hoop strain for a given pressure. Iteration is required in order for the strain value to converge to the actual solution. This is done by selecting an initial value for the hoop strain, and solving the equation for the new strain value. Repeating this process a few times will cause the strain value to converge to the actual solution. Once the calculated strain value is approximately the same as the inserted value, the correct strain solution has been found.

The closed form solution for hoop strain has been calculated with and without an axial stress of 10 ksi (68948 kPa) compression and tension for various pressures. The axial force was determined by multiplying the axial stress by the cross sectional area of 3" Std. pipe (OD = 8.89 cm, thickness = 0.55 cm) which is calculated to be 10 ksi x 2.228 in^2 = 22280 lbf (99.106 kN). The 3" Std. pipe and 10 ksi (68948 kPa) stress were chosen because they were used in the original testing and analysis performed by EPRI [2], [5]. Typical values for k and n for stainless steel were taken from Ref [2], page 2-9. The final strain values are shown in Table 1. This table was developed using the following: k = 191521, n = 0.427, D_o = 3.284 in. (83.4 mm), t_o = 0.216 in. (5.49 mm).

Table 1 Closed Form Solution for Hoop Strain Corresponding to Various Axial Stress

Pressure		Without Axial Stress	10 ksi Axial Compression (68948 kPa)	10 ksi Axial Tension (68948 kPa)
(psi)	(kPa)	Strain (final)	Strain (final)	Strain (final)
1000	6894.8	0.0003	0.0014	0.0001
2000	13789.5	0.0017	0.0032	0.0013
3000	20684.3	0.0043	0.0065	0.0036
4000	27579.0	0.0087	0.0116	0.0074
5000	34473.8	0.0150	0.0189	0.0131
6000	41368.5	0.0240	0.0291	0.0212
7000	48263.3	0.0366	0.0432	0.0325
8000	55158.1	0.0543	0.0633	0.0486
9000	62052.8	0.0812	0.0946	0.0723
9600	68947.6	0.1061	0.1259	0.0935

FINITE ELEMENT RESULTS FOR AXIAL LOADING

An ANSYS 3-D finite element model has been used to benchmark the two closed form solutions for hoop strain in a thin walled cylinder. Plots of the model are shown in Figure 1. The ANSYS 3-D model was created using elastic and plastic shell elements (shell63 and shell43 respectively). A schematic showing boundary conditions and length of elastic and plastic region is shown in Figure 2.

The pipe length is determined based on a minimum of 5 diameters in order to avoid boundary condition effects. In order to minimize computer run time, the center half of the pipe utilizes plastic shell elements, while the two ends use elastic shell elements. This will also simulate the actual deformation configuration for capped end conditions since the end caps were not modeled. This elastic/plastic shell element arrangement is acceptable because only the middle portion of the plastic section is of interest.

A standard Poisson's ratio (0.3) and a very stiff elastic modulus (3 x E_{normal} = 87.76E6 psi (6.051E8 kPa)) were chosen for the elastic element material properties. The empirical equation $S = k\delta^n$, taken from Ref [3], was used to determine the non-linear stress-strain relationship and to calculate fifteen data points that were input into the ANSYS model. The elastic modulus for the plastic elements was chosen as 10.028E6 psi (6.914E7 kPa) based on the first stress-strain data point of .001 and 10028 psi (69140.6 kPa) respectively. Poisson's ratio was taken as 0.3. The strength coefficient (k) and strain-hardening exponent (n) were chosen as 191521 and 0.427 respectively, which are reasonable values for stainless steel, [2], and match the closed form solution values.

The boundary conditions for the ANSYS model are free at one end, and fixed in all directions at the other end except for the radial displacement, which is free to expand.

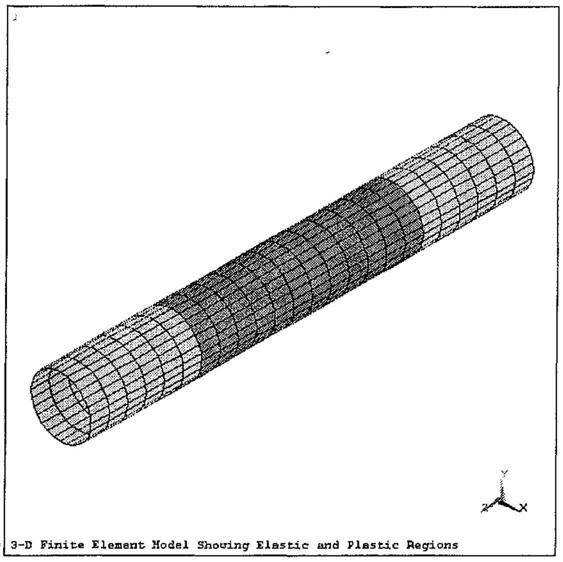


Figure 1: Finite Element Model Showing Elastic and Plastic Regions

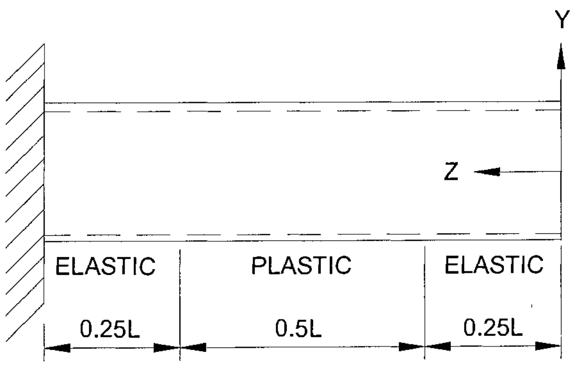


Figure 2: Schematic Showing Boundary Conditions and Regions

The ANSYS model was run under two loading conditions:

1. Internal pressure, and
2. Internal pressure with sustained axial compression.

The closed form solutions and ANSYS solutions for hoop strain with and without axial compression are shown together in Table 2.

Table 2 Closed Form (CF) and ANSYS Solutions for Hoop Strain

Pressure		CF Strain (No Compression)	ANSYS Strain (No Compression)	CF Strain (10 ksi (68948 kPa) Compression)	ANSYS Strain (10 ksi (68948 kPa) Compression)
(psi)	(kPa)				
2000	13789.5	0.0017	0.0018	0.0032	0.0033
3000	20684.3	0.0043	0.0044	0.0065	0.0065
4000	27579.0	0.0087	0.0086	0.0116	0.0115
5000	34473.8	0.0150	0.0150	0.0189	0.0185
6000	41368.5	0.0240	0.0250	0.0291	0.0300
7000	48263.3	0.0366	0.0370	0.0432	0.0426
8000	55158.1	0.0543	0.0550	0.0633	0.0623

The pressure-strain data points shown in Table 2 show very good correlation between the closed form solution and the ANSYS benchmark solution for hoop strain with and without an axial compressive stress. Thus, the ANSYS model is now used to evaluate the condition of pressure with bending.

FINITE ELEMENT RESULTS WITH BENDING

Internal pressures were applied to the cylinder face starting with 2000 psi and were increased in increments of 1000 psi up to 8000 psi. The bending stress was applied to the ANSYS model by use of nodal forces along the edge nodes. The nodal forces were determined in ANSYS based on a sinusoidal function of the maximum nodal force as shown below.

$$F_i = F_{max} \operatorname{Sin}(\theta_i) \tag{4}$$

where: θ_I = 0°, 12.86°, 25.71°, 38.57°, ...360°
F_{max} = 796 lbs (3540.78 N) is the tributary nodal force for top or bottom node.

The angle (θ_i) is the central angle between side node and the node which the force is to be determined.

Table 3 ANSYS Solution for Hoop Strain With 10 ksi (68948 kPa) Bending Stress

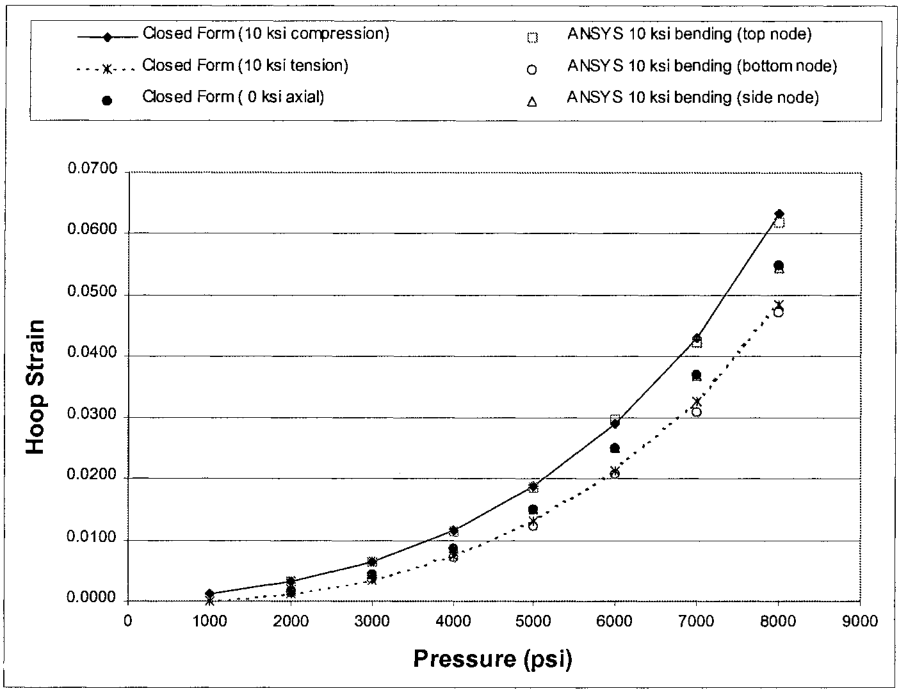
Pressure		ANSYS Strain (at side node)	ANSYS Strain (at top node)	ANSYS Strain (at bottom node)
(psi)	(kPa)			
2000	13789.5	0.0018	0.0032	0.0016
3000	20684.3	0.0044	0.0065	0.0037
4000	27579.0	0.0087	0.0115	0.0072
5000	34473.8	0.0152	0.0185	0.0123
6000	41368.5	0.0250	0.0298	0.0209
7000	48263.3	0.0369	0.0424	0.0310
8000	55158.1	0.0545	0.0619	0.0472

Figure 3 is presented to show the effect of a 10 ksi (68948 kPa) bending stress on the hoop strain. Since the ANSYS model has been shown to match the closed form solutions with and without axial stress, the ANSYS results with an applied bending stress (Table 3) will be compared to the closed form solution results with axial compression and tension (Table 1). From Figure 3, it is evident that the hoop strain at the top node under 10 ksi (68948 kPa) bending matches closely the hoop strain under 10 ksi (68948 kPa) axial compression. Similarly, at the bottom node the hoop strain matches closely the hoop strain under 10 ksi (68948 kPa) axial tension. It is therefore logical that at the side node along the neutral bending axis, the hoop strain matches the closed form solution with only internal pressure (no axial stress). This finding is important because it verifies that pure compression or tension can be applied to obtain the same result as if bending were applied.

It is also important to verify that the axial stresses in the shell are behaving as expected under internal pressure and bending loads. Since the cylinder is basically a simple beam, the axial stresses under low hoop strains (1-2%) should be Pr/2t + M/Z. The bending stress, M/Z, was applied as −10 ksi (68948 kPa) at the top node and 10 ksi (68948 kPa) at the bottom node. At the fourth time step (P = 5000 psi (34473.8 kPa)), the hoop strain is between 1% and 2% as shown in Figure 3. Therefore, the axial stresses at the top and bottom nodes should be approximately:

Fa = (5000 psi x 1.642")/(2*0.216") ± 10 ksi = 19 ksi ± 10 ksi
= 9 ksi (62052.8 kPa) (top node), 29 ksi (199948 kPa) (bottom node)

The ANSYS results show that the axial stresses are 9.54 ksi (65776 kPa) at the top node (Sz), and 29.6 ksi (204084.8 kPa) at the bottom node (Sz), which are close to the predicted values and are therefore acceptable.



Note: 1000 psi = 1 ksi = 6894.8 kPa

Figure 3: Effect of Bending Stress on Hoop Strain

EFFECT OF BENDING ON CROSS SECTIONAL AREA

The pipe deformed cross section is important for determining the equilibrium pressure between the cylinder and the enclosed water as the water is heated. In this section, the effect of an applied bending stress in addition to internal pressure on the cross sectional area will be determined.

The radial displacements at two pressure increments (5000 psi, 34473.8 kPa and 8000 psi, 55158.1 kPa) have been obtained at all nodes at the center of the cylinder from the ANSYS run with bending. Figure 4 shows the nodal displacement at the central of the cylinder under pressure and bending, while Figure 5 shows the nodal displacements for pressure only.

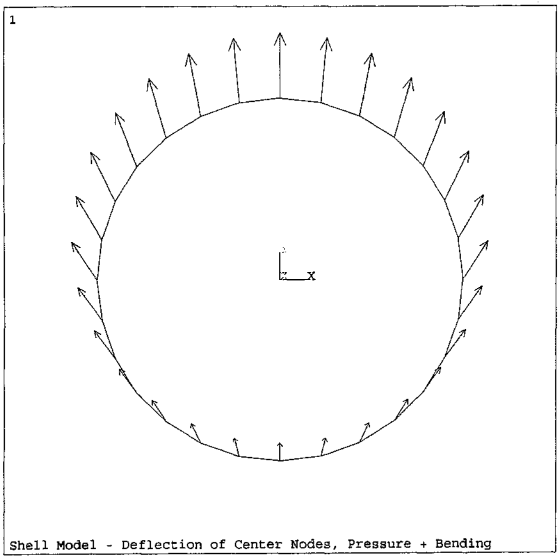


Figure 4: Vector Plots of Nodal Displacements Due to Pressure and Bending Load

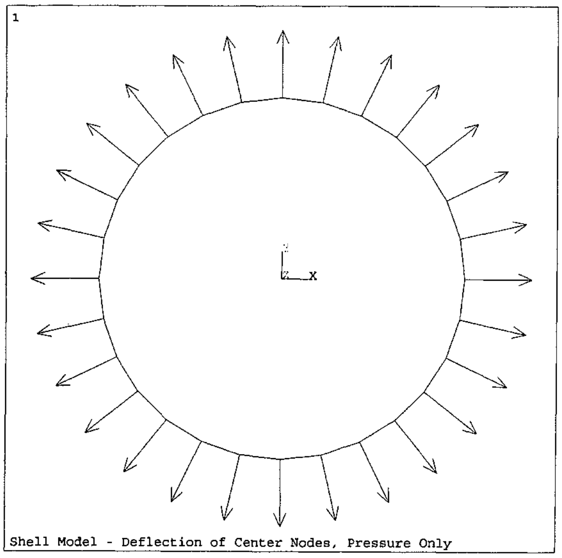


Figure 5: Vector Plot of Nodal Displacements Due to Pressure Load Only

The enclosed area of the displaced shape is calculated by summation of 28 circular sectors (pie shapes) using the following formula.

$$A = \sum_{i=1}^{28} \frac{\pi(\frac{R_i + R_{i+1}}{2})^2}{28} \tag{5}$$

In Table 4, the change in the displaced area due to two loading conditions (internal pressure, and internal pressure plus bending) is investigated for two applied internal pressures (5000 psi (34473.8 kPa) and 8000 psi (55158.1 kPa)). The original area of the pipe is calculated using the mean pipe radius (πr^2). The column labeled $\Delta A/A$ is the change in the displaced area due to the loading condition, or (loaded area – original area)/original area. The last column calculates the change in $\Delta A/A$ for the two loading conditions (internal pressure vs. internal pressure plus bending). Table 4 shows that the area enclosed by the displaced shape does not significantly change due to an applied bending stress. At both 5000 psi (34473.8 kPa) and 8000 psi (55158.1 kPa), the change in the displaced area is less than 3% (which is considered small). It should also be noted that as the pressure is increased from 5000 psi (34473.8 kPa) to 8000 psi (55158.1 kPa), the change in the displaced area due to the bending stress decreases. This is expected since the effect of a constant bending stress should decrease as the internal pressure is increased.

Table 4 Change in Displaced Area due to Bending

Pressure		Bending Stress		Original Area		Loaded Area		ΔA/A	% Change due to Bending
(psi)	(kPa)	(ksi)	(kPa)	(sq. in.)	(sq. cm)	(sq. in.)	(sq. cm)		
5000 psi	34473.8	0	0	4.235	27.32	4.368	28.18	0.0314	2.9
5000 psi	34473.8	10	68948	4.235	27.32	4.372	28.21	0.0323	
8000 psi	55158.1	0	0	4.235	27.32	4.735	30.55	0.1181	2.4
8000 psi	55158.1	10	68948	4.235	27.32	4.747	30.63	0.1209	

SUMMARY AND CONCLUSIONS

1. The effects of bending stresses on the maximum hoop strain are similar to those from a pure compression or tension force. In addition, at low hoop strain levels (< 2%), the ANSYS axial stress due to internal pressure plus bending is very close to the elastic closed form solution for the axial stress (Pr/2t + M/Z).
2. The change in cross sectional area due to adding a bending stress to the internal pressure is negligible. It was further shown that an increase in internal pressure will result in a decrease in deformed area change due to bending.

It should be noted that the maximum hoop strains calculated are between 4.5% and 6.5% as indicated in Tables 5 and 6. Therefore, the above conclusions are valid for hoop strains less than the maximum hoop strains determined in this calculation.

ACKNOWLEDGEMENTS

The authors would like to thank Dr. Edward A. Wais for his assistance and comments during the initial work on this effort.

REFERENCES

1. NRC Generic Letter 96-06, "Assurance of Equipment Operability and Containment Integrity During Design-Basis Accident Conditions", Including Supplement 1, Nov. 13, 1997.
2. EPRI report TR-108812, "Response of Isolated Piping to Thermally Induced Overpressurization during a Loss of Coolant Accident (GL96-06)," Stephen R. Gosselin, Dec. 1997.
3. Collins, Jack A., "Failure of Materials in Mechanical Design," Analysis Prediction Prevention, Second Edition, Chapter 5 – Relationships Between Stress and Strain, John Wiley & Sons.
4. ANSYS User's Manual, Volumes I – IV, Swanson Analysis Systems, Rev. 5.
5. EPRI Draft, "Response to NRC Comments Regarding EPRI White Paper Report TR-108812," Steve Gosselin of EPRI and Ed Wais of Wais and Associates, 5/14/98.