Comparison of Linear and Nonlinear FE Analysis of a Typical Vessel Nozzle

Michael A. Porter
Dynamic Analysis
Lenexa, Kansas

Dennis H. Martens
Black & Veatch Pritchard Corporation
Overland Park, Kansas

ABSTRACT

In an earlier paper by Porter & Martens, 1996 (1), the authors demonstrated that five different FEA software codes produced comparable results in the analysis of a typical thin wall nozzle-to-shell junction where the indicated stresses remained below the material yield point. Where the indicated stresses were above yield, considerable divergence was noted. In order to explore the stress redistribution patterns that may have caused the divergences, this paper presents a nonlinear (elastic-plastic, material nonlinearly only) analysis of the same nozzle. The results are compared with the results from the previous linear analysis. The results are discussed with respect to an evaluation procedure for Shell/Plate element Finite Element investigations presented in a paper by Porter, et al, 1999 (2).

INTRODUCTION

The stress distribution in a typical vessel nozzle-to-shell junction under combined loading of pressure and applied moment may include areas where the stresses indicated by linear Finite Element (FE) analysis exceed the material yield strength. This has been demonstrated in several previous papers, Porter and Martens, 1996 (1), 1998 (3); Porter, et al, 1997 (4), 1999 (2). Typically, the stresses indicated by the linear elastic methodology exceed the material's yield at the junction of the nozzle and shell. ASME Section Division 2 (1998) (5) provides stress evaluation and acceptance criteria based on linear elastic methodology. The linear elastic stress indicated by FE analysis may be evaluated using procedures reported by the Pressure Vessel Research Committee 3D Stress Criteria: Guidelines for Application by Hechmer and Hollinger 1997 (6) and Porter, et al 1999 (2).

The use of nonlinear FE analysis to investigate the nozzle-to-shell junction can increase the understanding of the actual stress distribution and associated strains. By comparing the linear and nonlinear analyses results, the authors expected to confirm that the elastic methodology provided reasonable and conservative stress results. The authors expected to confirm that low permanent strain values would be observed and confined to very localized areas. In addition, the authors wanted to compare the results of plate and shell elements with and without shear deflection included in the solution to the nonlinear FE results.

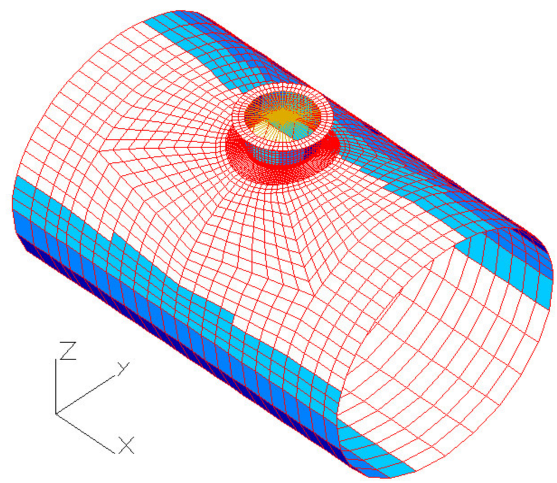

Figure 1 – Nonlinear model geometry

CURRENT NONLINEAR MODEL

The model used for this analysis consisted of approximately 2,670 nodes that defined approximately 2,720 nonlinear shell elements. The configuration of the model is depicted in Figure 1. The object being modeled is a 96" OD shell with a ½" wall thickness. A 24" OD nozzle, also with a ½" wall thickness, intersects the shell at a 90 degree angle. Additionally, a re-enforcing pad with a thickness of ½" and an OD of 42" is included around the nozzle. The shell, as modeled, is 144" long while the nozzle is 12" long. The left end of the shell is fully restrained while the right end is restrained in all but the X (axial) direction.

The loading on the nozzle is comprised of:

Internal Pressure:	165 psi
Nozzle Loads:	
Fz (up):	-6,480 LB
Mx (in plane):	33,160 FT-LB
My (out of plane):	38,250 FT-LB
Mz (vertical):	25,500 FT-LB

The material modeled was SA516-70 at 500 degrees F. For the purposes of the analysis, the following properties were assumed:

Modulus of Elasticity:	27.3×10^6 psi
Poisson's Ratio:	0.3
Yield Strength:	30,700 psi
Strain Hardening Modulus:	263,979 psi

The elements employed for the analysis used a von Mises with Isotropic Hardening material model. The stress-strain relationship used was a straight line from the yield point through the ultimate strength considering an elongation of 15% at failure. The analysis was conducted using the Algor APAK software package employing their Mechanical Event Simulation (MES) with Nonlinear Material Model option. The solution was obtained using a dynamic time integration technique with an implicit time stepping method, i.e., a time integration of the equations of motion. A solution step of 0.001 was selected with the load increasing in a linear manner from time 0 to time 0.1. The total solution required 100 steps. A displacement convergence tolerance of 0.001 was required for each step.

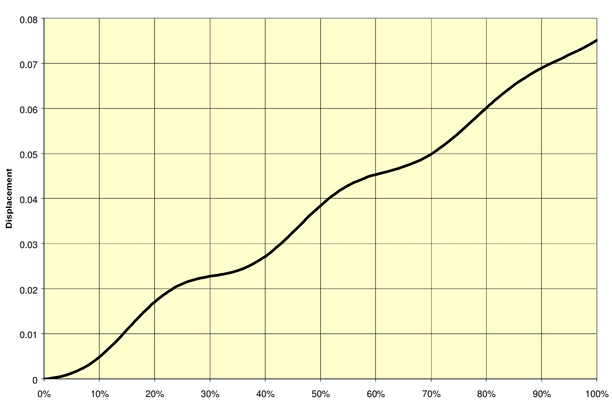

Figure 2 – Displacement of node at top of nozzle

NONLINEAR RESULTS

Figure 2 illustrates the displacement of a node on the nozzle near the top of the nozzle (far away from the plastic region) as a function of the applied load. Some oscillation of the displacement is evident. This oscillation is due to two factors: the processor used is simulating a real time event, and the rate of load application is of the same order of magnitude as the first modal frequency of the nozzle. If one draws a straight line from the origin to the end displacement, it can be seen that the displacement is oscillating about a straight line.

While this oscillation does not materially affect the results, it does make the identification of the onset of nonlinear displacement in the nozzle somewhat difficult.

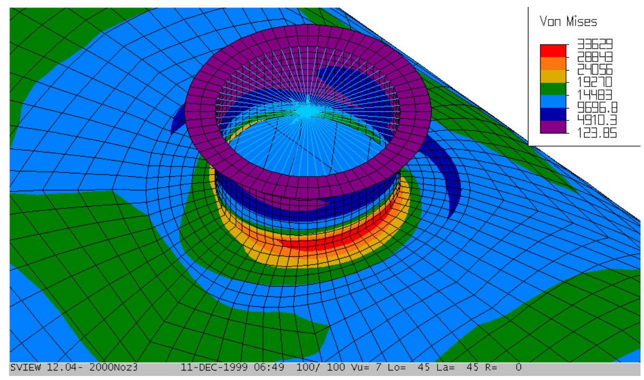

Figure 3 – Computed von Mises stress pattern

In the case of this nozzle and loading, the overall displacement remains linear, indicating that the nozzle can sustain the load applied without causing large-scale plasticity. Figure 3 illustrates the stress pattern in the fully loaded nozzle.

The maximum indicated stress is approximately 33,600 psi, some 3,000 psi above the material yield strength. Using this value and the strain hardening modulus, we can compute the magnitude of the strain in the model as indicated in Figure 4.

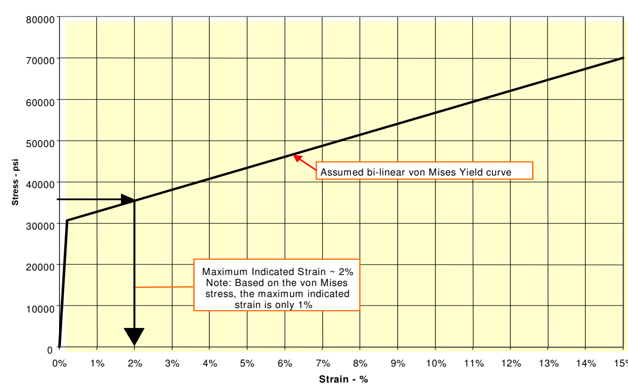

Figure 4 – Stress-strain with maximum strain

From Figure 4, we can see that the magnitude of the indicated strain is approximately 2%, which is a reasonably low amount of permanent strain for this material.

PRIOR FE LINEAR FE ANALYSIS

In several recent papers by Porter and Martens, 1996 (1); Porter, et al, 1997 (3), 1999 (2), the same nozzle geometry and loading were evaluated using linear FE with plate and shell elements. In reporting the results, the stresses were described in terms of the Stress Intensity. This was done so that interpretation of the results in accordance with the ASME Code (1998) (5) would be possible. Additionally, the stresses have been plotted as a function of the distance away from the nozzle-to-shell junction.

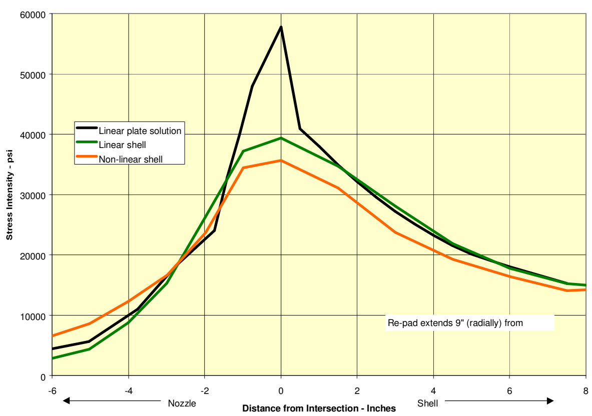

Figure 5 – Computed Stress Intensities

COMPARISON OF LINEAR AND NONLINEAR RESULTS

Figure 5 illustrates the indicated outside surface Stress Intensities in the models as a function of distance away from the junction.

As can be seen, there is considerable divergence between the linear plate results and the nonlinear shell results, especially at the nozzle-to-shell junction. Away from the junction, the divergence is considerably less, especially in the nozzle. In general, the linear results are somewhat conservative, as would be expected.

The reported stress intensities from the use of linear plate elements are significantly higher at the junction than those reported by either the linear shell (with shear deflection) or the nonlinear shell elements. Compared to the nonlinear shell results, the linear shell element results are generally somewhat conservative.

Considering the very significant increase in computational effort, the use of nonlinear FE analysis for similar thin wall vessel geometries is probably not justified. The use of linear thin plate or shell elements yields results which are demonstratively conservative and are appreciably less expensive.

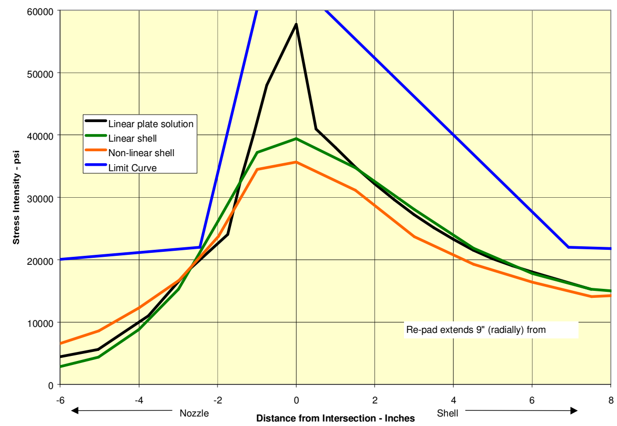

Figure 6 – Stress Intensity evaluation

INTERPRETATION RELATIVE TO ASME CODE

In a previous paper by Porter, et al, 1999 (2), the authors presented a suggested evaluation procedure for linear Shell/Plate FE nozzle models. This procedure involved plotting the stress intensity as a function of distance from the nozzle-to-shell junction and comparing these values with a set of criteria curves, as illustrated in Figure 6. While this procedure is believed to be valid for linear FE models, it is not appropriate for evaluating nonlinear models.

If the stresses in the nonlinear model are allowed to approach the 3.0 Sm levels indicated in the ASME Code, the strains would become unacceptable. Rather than a specific limit on the stress, a limit on the local strain and/or deformation would seem appropriate. Unfortunately, there is no guidance in ASME Section VIII 1998 (5) regarding such a limit. Additionally, the established evaluation criteria in ASME Section VIII (5) are based on Stress Intensity rather than the von Mises stress used by most, if not all of the commercial nonlinear FE software codes.

CONCLUSIONS

Nonlinear FE may not be necessary for thin wall vessels. The results using linear FE appear to be suitably conservative, at least for the subject example.

Shell elements with shear deflection develop stresses very close to the nonlinear elements for this model. Thus, the results using this type of element are less conservative than the linear plate element. This raises the question of the applicability of the current ASME Code criteria when evaluating shell elements with shear deflection.

Clearly, the 1.5Sm and 3Sm criteria cannot be applied to nonlinear FE results. A more applicable criterion would be based on a limit for local deformation and/or strain. Both the Limit Analysis and the Plastic Analysis methods given in Appendix 4 of the ASME Code require the determination of a limit load which is then decreased by a factor of 2/3 to determine the allowable load. Currently, however, there is no specific guidance in Section VIII, Division 2 for an evaluation based on deformation/strain alone.

RECOMMENDATIONS

If nonlinear FE without the computation of a limit or collapse load is used to evaluate a thin wall nozzle, prudent engineering judgment, with little guidance from the ASME Code, must be used to evaluate the results. The development of deformation/strain criteria for evaluating such analyses would be beneficial.

REFERENCES

1. Porter, M. A. and Martens, D. H., 1996, "A Comparison of the Stress Results from Several Commercial Finite Element Codes with ASME Section VIII, Division 2 Requirements," *PVP Vol. 336*, ASME, New York, NY., pp. 341-346.

2. Porter, M. A., Martens, D. H., and Caldwell, S. M., 1999, “A suggested Evaluation Procedure For Shell/Plate Element Finite Element Nozzle Models," *PVP Vol. 388*, ASME, New York, NY., pp. 227-232.

3. Porter, M. A. and Martens, D. H., 1998, Stress Evaluation of a Typical Vessel Nozzle Using PVRC 3D Stress Criteria: Guidelines for Application," *PVP Vol. 368*, ASME, New York, NY., pp. 297-301.

4. Porter, M. A., Martens, D. H., and Hsieh, C. S., 1997, "A Comparison Finite Element Codes and Recommended Investigation Methodology," *PVP Vol. 359*, ASME, New York, NY., pp. 241-246.

5. ASME (1998), "ASME Boiler and Pressure Vessel Code", Section VIII, Division 2, The American Society of Mechanical Engineers, New York, NY.

6. Hechmer, J. L. and Hollinger, G. L., 1997, “3D Stress Criteria: Guidelines for Application,” PVRC Grant 91-14 Final Report, ASME, New York, NY.