COMPARISON OF LIMIT LOAD, LINEAR AND NONLINEAR FE ANALYSIS OF A TYPICAL VESSEL NOZZLE

Michael A. Porter
Dynamic Analysis
20706 Mill Road
Lenexa, Kansas 66220
913-422-5998
mike@dynamicanalysis.com

Wolf D. Reinhardt
Babcock & Wilcox Canada
581 Coronation Boulevard
Cambridge, Ontario
Canada N1R 5V3
519-621-2130
reinharw@pgg.mcdermott.com

Dennis H. Martens
Black & Veatch Pritchard Corporation
10950 Grandview Drive
Overland Park, Kansas 66210
913-458-6000
martensdh@bv.com

ABSTRACT

A limit load analysis of a vessel nozzle with pressure and external force loading was conducted. The limit load was defined by increasing the pressure and the nozzle external loads proportionally until collapse occurred. The evaluation of the limit load analysis was conducted in accordance with ASME BPVC Section VIII Division 2 [1].

The limit load analysis provides insight into the collapse load and failure mode. The results of the limit load analysis and a plastic analysis are compared to the results obtained by linear and nonlinear shell and plate element analyses of the same nozzle. The authors discuss the comparison of the results as there is some variance between the different methodologies.

INTRODUCTION

A stress investigation of the junction area of a typical nozzle in a thin wall vessel with imposed connected piping forces is necessary to assure that the design is adequate for the intended service. The design of the nozzle, shell and necessary reinforcement for the intended design pressure is addressed by the ASME Code Section VIII Division 1 and 2 [1] provisions. An investigation of the stresses imposed by connected piping is necessary to complete the design evaluation.

The present paper addresses the specific design initially used by Porter and Martens, 1996 [4]. Figure 1 shows geometry. The dimensions are given in Appendix A. The vessel shell, nozzle neck, and reinforcement design is appropriate per Section VIII Division 1 [1]. The nozzle reinforcement was selected to be in excess of that required by the rules of Section VIII Division [1]. Use of Welding Research Council Publication 107 [2] to investigate the connected piping forces and moments has been a standard practice for many years. The acceptability of the nozzle design was confirmed using this procedure.

Stress investigation methodology now includes several types of Finite Element Analysis (FE). Since the Design Engineer will want, or be required, to use the best available methodology for design; the use of FE may be appropriate. Appropriate FE use and interpretation has been the subject of many publications, such as Pressure Vessel Research Committee 3D Stress Criteria Guidelines for Application by Hechmer and Hollinger 1997 [3] and technical papers presented at PVP. ASME Section VIII Division 2 Appendix 4 [1] addresses the methodology for Limit Analysis preformed by FE. The use of FE for thin shell nozzle investigation has been addressed by Porter and Martens in previous papers presented at PVP 1996 [5] 1997 [6] 1998 [7] 1999 [8] 2000 [9]. Limit Load analysis has been addressed by Kalnins [10,11].

The thin shell and nozzle dimensions, material properties and loads are listed in Appendix A. The same shell, nozzle and piping loads were utilized for the Limit Analysis, as for the 1996 Linear and Nonlinear investigations. Only minor necessary adjustments were made to the FE model to accommodate the software used for the investigation.

LIMIT ANALYSIS

<u>Modeling</u>

A three-dimensional model of the nozzle and shell was built in ANSYS 5.6 [12]. The model was meshed with 8-noded shell elements. (SHELL93). This ensures an adequate representation of the curvature of the shell. Figure 1 shows that the mesh was refined close to the nozzle and on the reinforcement pad, but fairly large elements were used in those regions where a uniform stress is expected. Note that the ends of the cylindrical shell were closed to create the correct axial shell stress. Since these closures are farther away from the nozzle than $2.5\sqrt{Rt}$, no effect by the constraint on the limit load is expected.

Boundary conditions were applied at one of the shell end closures to prevent rigid body motion. The nozzle blow-off and external nozzle loads were applied at the nozzle-flange intersection.

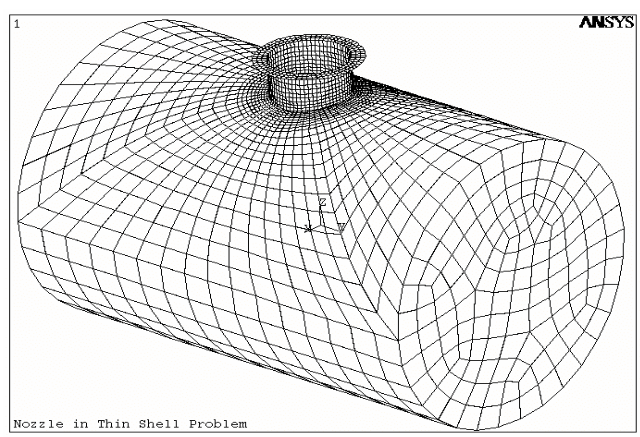

Figure 1- FE Model for Limit Analysis

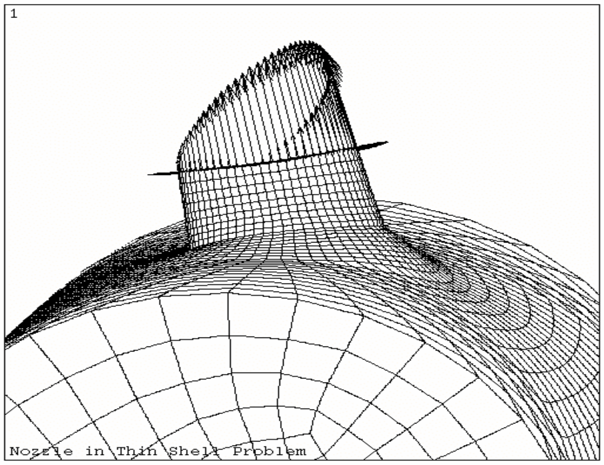

Figure 2 - Limit analysis failure mode under combined loads

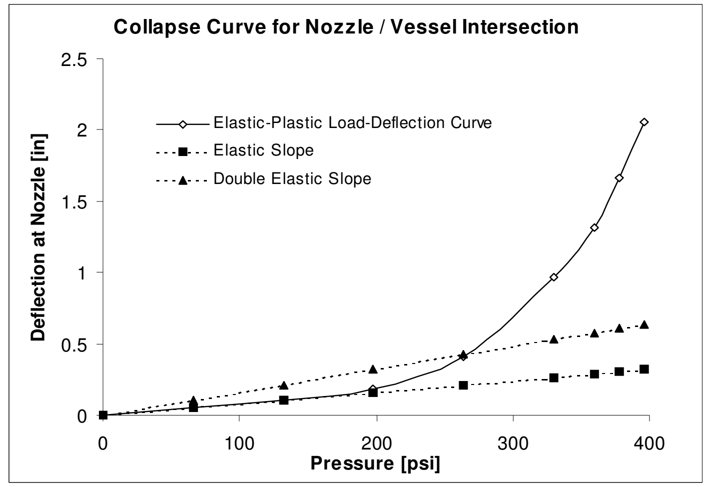

Figure 3: Plastic Analysis of the Shell / Nozzle Structure

All parts of the model, with the exception of the shell end closures, were given elastic-perfectly plastic material properties with yield stress Sy = 1.5 Sm = 30000 psi. This is in agreement with the requirements stated in Section VIII, Div. 2, Appendix 4-136.3 [1]. Since the only purpose of the end closures was to create the shell axial stress, they were kept linear-elastic. Note that the limit analysis is run as a “small deflection” analysis, although the calculated deflections at the limit state can be quite large. This means that no credit is being taken for geometric strengthening.

The von Mises yield criterion is used for the limit analysis. Accordingly, the Limit Loads obtained from the model are multiplied by 1/1.15 to obtain results consistent with the maximum shear stress criterion that are Code-acceptable.

Loading

In a non-linear limit analysis, the principle of superposition of loads does not apply. Therefore, if there are loads from different unrelated sources (like pressure and nozzle external loads), the result depends on the mode in which the loads are applied. All that is known at the onset is that at the limit state, the structure has to sustain at least 1.5 times the Design pressure and at the same time 1.5 times the external nozzle loads.

The first possible method of applying the loads is to increase both of them simultaneously and proportionally until the structure fails. This is convenient because, at the limit state, the same factor is applied to all design loads. It could be said that proportional application gives the averaged design margin of the structure and the corresponding limit failure mode. If the structure does not satisfy the limit criterion (i.e., if the ratio of limit load over design load is less than 1.5), the limit failure mode can be used to indicate where the structure needs to be strengthened. In a finite element limit analysis, very large deflections and plastic strains identify the failure location(s) at the limit state.

While the above method is normally the preferred one, there are occasions where a slightly different point of view is desired. For example, it may be known that the design pressure is not likely to be exceeded, but the external loads are known less precisely. In this case, the question may be: “Given the design pressure, how much increase in external load can the structure tolerate?” To answer this question with a limit analysis, the pressure would be fixed at 1.5 times the Design value (the minimum value required when the structure collapses). The external loads would then be increased until the limit state is reached. The ratio of (external load at limit state)/(design external load) equals 1.5 times the design margin SF. The factor SF for the external load that is obtained here would be larger than that obtained with the first method because, in the present method, the pressure is not scaled.

A third possibility is to ask, “Given the design nozzle loads, what is the maximum pressure that can be applied?” In this case, the external loads would be fixed at 1.5 of their design values and the pressure is increased until plastic collapse occurs. Otherwise, this case resembles the previously discussed second one.

The conclusion from the discussed loading methods is that a limit analysis can give useful information beyond just showing that a design is acceptable. All three methods will lead to the same result concerning the question whether the design is acceptable or not. On the other hand, using a specific method of load application enables the designer to optimize the structure or explore its sensitivity to variations in a specific load.

Results

To perform the limit analysis, a minimum load step of about 0.5 psi and a starting load step of about 80 psi were specified. During the analysis, the FE software loaded the structure and kept bisecting the load step until non-convergence occurred with the minimum load step. The last converged solution is, therefore, expected to give the limit load to an accuracy of about 0.5 psi, assuming that the accuracy of the equilibrium iterations is consistent with that of load stepping.

Table 1 shows limit loads that were obtained for the present nozzle-shell structure. Note that the loads were applied proportionally, so the factor SF = (load at limit state) / (design load) is common for all loads in a specific load case.

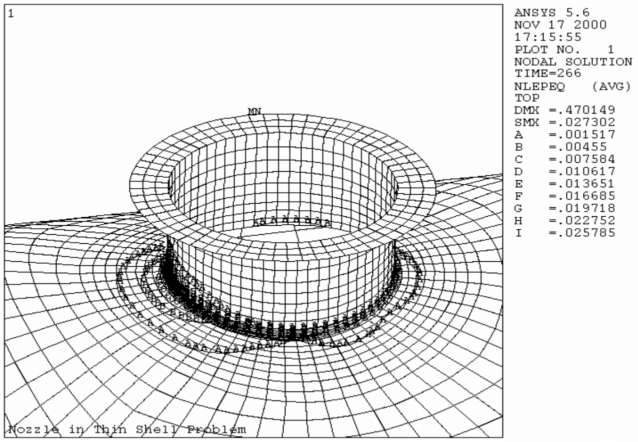

Figure 4 - Equivalent Plastic Strain at Collapse

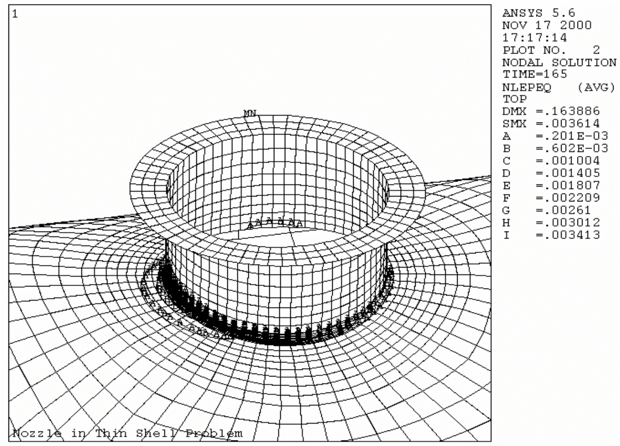

Figure 5 - Equivalent Plastic Strain under Design Loads

Table 1: Limit Loads of the Nozzle / Shell Structure

Load Case	Limit Pressure	SF
Unperforated shell / pressure	311 psi	1.88
Pressure, no nozzle blow-off	285 psi	1.73
Pressure and nozzle blow-off	254 psi	1.54
Pressure and all nozzle loads	219 psi	1.33

The limit pressure for the shell alone was obtained from the formula

$$p = Sy \ln\left(1 + \frac{t}{R}\right) \qquad [2] \qquad (1)$$

Where **t** is the shell thickness and R is the shell radius. All other values were obtained from an FE limit analysis with a reduction factor of 1.15 applied to the result. This reduction factor was confirmed both at the design load and at collapse by forming the ratio of the largest von Mises equivalent stress in the plastic region to the largest stress intensity. The ratio was found to deviate less than 1% from 1.15.

Table 1 shows the reduction in limit load due to the presence of the hole, the further reduction when the nozzle blow-off load is applied, and finally the effect of the other external nozzle loads. Evidently, the structure with all loads applied cannot be justified with a limit analysis that satisfies the Code requirements (SF=1.5). Except for the first case (unperforated shell), failure consistently occurred in the reinforcement pad beside the nozzle (at the intersection of the nozzle neck to the reinforcement pad, Figure 2). As noted previously, the nozzle design and external applied loads were verified to be acceptable using the design rules of Section VIII Division 1 [1] and WRC 107 [2].

The factor SF shown in Table 1 is close enough to 1.5 to justify a few "what if" scenarios. If the Code would allow the use of the von Mises criterion, the value of SF with all loads applied would be 1.53 and the design would be acceptable. For a thin-walled structure like the present one, geometric strengthening is expected to play an important role. A limit-type analysis with the "large deflection" option turned on resulted in a limit pressure in excess of 287 psi (SF > 1.74). The additional margin in the last case comes at the expense of fairly high plastic strains in the shell-to-nozzle intersection. Of course, the plastic strains from limit analysis are not real, since material hardening tends to reduce the strain level significantly compared to a perfectly plastic analysis. However, this lies outside the domain of limit analysis yet it raises the question of the application of small deformation versus large deformation solutions for this investigation. The objective of a Code-type limit analysis is to prevent large permanent deformations, and this includes the deformations necessary to produce geometric strengthening. However this may be excessively conservative based on the successful application history of Section VIII Division 1 [1] design rules and closed form stress investigation methods such as WRC 107 [2].

Since the limit analysis was not sufficient to show that the present structure is acceptable, a plastic analysis was run for comparison.

PLASTIC ANALYSIS

Using the limit analysis model, a plastic analysis was run following Section VIII, Div. 2, Appendix 4-136.5 [1]. In this analysis, non-linear geometric effects were included as well as strain hardening. A von Mises bilinear isotropic hardening model was used with a tangent modulus (strain hardening slope) of 263,979 psi. The model was loaded up to 2.4 times the design loads, with pressure and external loads applied proportionally. According to the "double elastic slope" method of 6-153 [1], the point of largest axial deflection on the nozzle was used to monitor the deformation of the structure. A plot of the axial deflection (z-direction in Figure 1) of this point over the applied pressure was prepared. Since the first load step resulted in a purely elastic loading of the nozzle, this point was taken to compute the elastic slope as

$$S = \frac{p_1}{u_1} \qquad (2)$$

Where p_1 is the pressure and u_1 is the displacement of the reference point at the end of the first load step.

When a line with slope 2S is drawn, its intersection with the actual load-deflection curve marks the collapse point of the structure, which is found to be at 266 psi and the corresponding external nozzle load (Figure 3). The SF is then calculated to be 266/165 = 1.61. The Code does not explicitly state whether any correction for maximum shear stress theory is required. In AD-140 (a) of Section VIII Div. 2 [1], the statement "The theory of failure used in this Division is the maximum

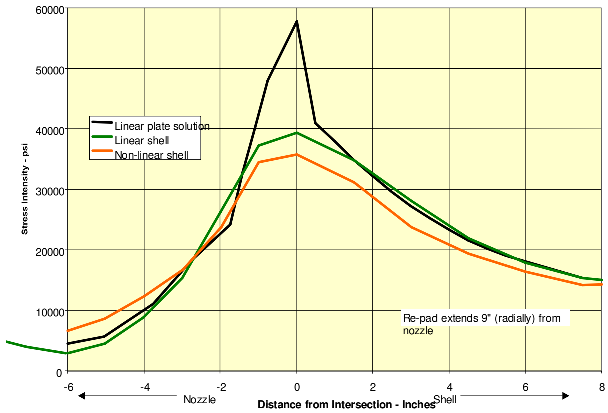

Figure 6 – Computed Stress Intensities

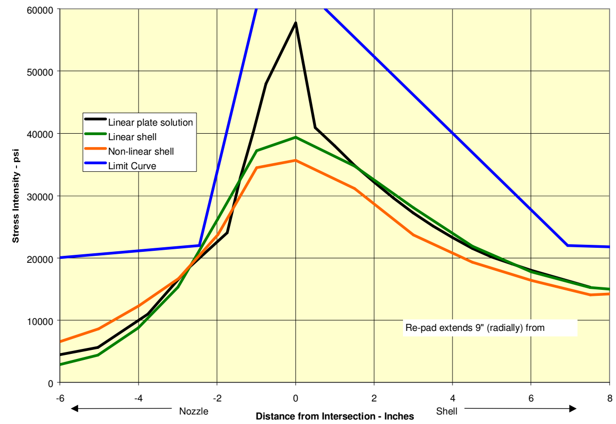

Figure 7 – Stress Intensity evaluation

shear stress theory…" can be found. However, it can be argued that no correction is necessary for plastic analysis since the von Mises theory is not strictly used as a failure theory here, failure being indicated by the double elastic slope method. This would make the structure acceptable according to Plastic Analysis.

It is instructive to look at the expected strains when hardening is included. At the collapse point, the largest equivalent plastic strain is 2.6% (Figure 4), while the maximum equivalent plastic strain under design conditions is about 0.4% (Figure 5). These values are small, indicating that the permanent deformation in operation is probably acceptable. Figure 2 confirms that the pressure / deflection curve deviates negligibly from the elastic line under design conditions. Due to the thin walls, the deflection of the nozzle is nonetheless very noticeable. Whether this deflection is acceptable depends on the characteristics of the attached piping and would have to be determined on a case-to-case basis.

PRIOR LINEAR AND NONLINEAR FE

In several recent papers by Porter and Martens, 1996 [4]; Porter, et al, 1997 [6], 1999 [8], 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) [1] would be possible. Additionally, the stresses were plotted as a function of the distance away from the nozzle-to-shell junction.

In a 2000 paper by Porter and Martens [9], the same nozzle geometry and loading were evaluated using nonlinear FE. The elements employed for the analysis used a von Mises material model with Isotropic Hardening. 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 [13] software package employing their Mechanical Event Simulation (MES) with Nonlinear Material Model option.

Discussion of previous results

Figure 6 illustrates the indicated outside surface Stress Intensities from the linear and nonlinear 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.

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.

Under the imposed DESIGN loading, the stresses in the immediate vicinity of the nozzle exceed the Code limit of 1.5 Sm on primary local membrane stress. However, the FE stresses at the juncture of the shell and nozzle elements are known to be unreliable. Since the shell elements represent the midplane of the modeled geometry, it is not appropriate to evaluate stresses in regions less than half a wall thickness away from the juncture. The real geometry will have a filleted transition between nozzle and shell which is also not represented in the model and which might suggest that the evaluation point be moved even further from the point of intersection in the model. As reported by Heckmer and Hollinger [3], the stress analysis in the area of this junction involves transition element evaluation which may not be necessary except for fatigue evaluation. The methodology of stress classification lines and transitions elements is discussed in the Heckmer and Hollinger PVRC report [3] and in Porter and Martens 1998 [7].

Finally, the applied nozzle bending moment causes a stress "hot spot" in the shell which does not extend around the circumference of the nozzle. To address these issues, a previous paper by Porter, et al, 1999 [8], suggested an 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 7. While this procedure is believed to be valid for linear FE models, it is not appropriate for evaluating nonlinear models. Figure 7 presents the proposed acceptance criteria profile overlaid on Figure 6. It should be noted that the maximum strains reported by the nonlinear analysis were fairly small, in the order of 1% on a von Mises basis. This is in agreement with the Plastic Analysis results reported earlier.

In the evaluation of the results from the nonlinear model, one could envision a method that is more direct than the inelastic methods that are presently permitted by the Code. Rather than a specific limit on the stress, a limit on the local strain and/or deformation would seem appropriate, since the final objective is to avoid excessive plastic deformations of the structure. Unfortunately, there is no guidance in ASME Section VIII 1998 [1] regarding such a limit. Additionally, the established evaluation criteria in ASME Section VIII [1] are based on Stress Intensity rather than the von Mises stress used by most, if not all of the commercial nonlinear FE software codes. As reported by Porter and Martens 2000 [9], FE indicated stresses developed using linear plate and shell element solutions that include shear deflection may not be suitably conservative for direct comparison to ASME stress allowables.

CONCLUSIONS

The results of the investigation of the present design problem with various analysis methods are shown in Table 2.

Table 2 - Margin of Structure against Various Design Limits

Analysis Type	(Analysis Result) / (Design Limit) (> 1 fails)
WRC 107 and Section VIII Stress Index	0.97
Elastic Analysis, maximum PL	1.18
Averaged PL Elastic Analysis [8]	0.87
Limit Analysis	1.12
Plastic Analysis / von Mises	0.93

The maximum deviation between the results is about 30%. The original nozzle design is suitable based on Section VIII Division 1 [1] rules and the applied external loading is acceptable per WRC 107 [2] verification. The nozzle design fails under the applied loadings if the maximum local membrane stress from the elastic analysis is used [8]. However, the design is confirmed acceptable by elastic and inelastic methodology as reported by Porter et al [8], [9]. Plastic analysis as reported in this paper confirmed the suitability of the nozzle design and applied loadings with minimal permanent strain or deformation indicated. The plastic analysis was performed by using the (von Mises based) FE results directly. It is argued that this is acceptable because failure of the structure is not based on the use of the yield criterion, but on the stress-deflection curve. Interestingly, the margin indicated by plastic analysis is about the same as the margin from the elastic methodology proposed in Porter et al [8].

The use of Limit Load analysis, based on small deformation methodology, indicated that the nozzle design is not acceptable under the given loads. The analysis fails by about the same margin as the elastic analysis based on the maximum local membrane stress, so these two methods are found to give consistent results for the present thin-walled structure.

Compared to the results of the WRC107 analysis, it is apparent that the use of Limit Load small deformation methodology is overly conservative for this particular nozzle application. The present results suggest that this conservatism could be addressed in two ways by suitable extensions of the existing analysis rules. One possibility is to allow geometric strengthening (using large deformation analysis) to be credited in a limit analysis. Another option is to allow the use of the von Mises failure theory. Theses issues will be discussed further in the next section.

FURTHER DISCUSSION

With the increasing availability of powerful numerical tools like FE, structural details can be modeled and analyzed to an unprecedented degree. This level of detail was not available and, therefore, was not intended when the rules of the ASME Code were established. Use of detailed finite element models makes a lot of information available to the designer, but not all of the information is relevant for Code design. Especially for elastic design methods, suitable evaluation methods need to be established to guide the designer in selecting the most relevant information. Examples of this are the classline method for elastic solid FE models and the suggestions by Porter et al [8] for elastic shell models.

As additional methods to analyze a component become practical and available, problems can arise when the various methods give different answers. In many cases, a component could be designed either "by rule," by classical interaction analysis, with an elastic FE shell model, by elastic FE using a solid model, by FE limit analysis, or by FE plastic analysis. Most designers would feel that the above mentioned methods are listed in an order of increasing sophistication, i.e. revealing an increasing amount of information about the true behavior of the structure. Since a higher degree of uncertainty should be balanced by a higher factor of safety, one would expect the more traditional methods to give more conservative results. If the results in specific cases do not support this expectation, however, the designer faces a dilemma. Should he use the traditional method, knowing that it is, or could be, less conservative than a supposedly more "exact" one? Or should he use the new method and strengthen the design even though traditionally designed vessels have performed satisfactorily in practice?

The Code does not offer any guidance here, since it offers its several design approaches in parallel and equally acceptable. Normally, there is no reason to assume that the design resulting from any one method is inferior to another, especially when this method has been widely applied and the designs have performed satisfactorily. Occasionally, there may be reasons to prefer or reject a method on technical grounds. For example, if serious strain concentrations are expected in a structure, an elastic-plastic analysis is advisable. Also, in case of a totally new design for which no previous experience exists, diligence may suggest getting as much information as possible, maybe by exploring several design methods.

While it will probably be impossible to resolve all conflict between different acceptable design approaches, an attempt should be made to ensure consistency as far as possible. To this end, practical examples such as the present thin-walled nozzle / shell intersection can be very useful. If several examples exist which suggest that a certain method is too conservative or unacceptably lenient compared to other methods, a proper adjustment should be made. Possible adjustments could be a change in the evaluation methodology, or even a change in the factor of safety.

The analysis of the complex nozzle structure using a proposed evaluation method in conjunction with linear and nonlinear FE has confirmed a safe and practical design that is consistent with past practice utilizing design tools such as WRC 107. The Limit Load Analysis of the nozzle did not confirm the nozzle design would meet the requirements of ASME Section VIII Division 2 Appendix 4-136.3 [1]. The plastic analysis did confirm the nozzle would meet the requirements of ASME Section VIII Division 2 Appendix 4-136.5. Based on the results of the other analyses, it is difficult for the authors to accept the results of the Limit Load analysis as limiting for the nozzle. At present, the use of Limit Load analysis may not be appropriate for confirming ASME acceptability of the thin-walled nozzle design. This could be addressed by allowing the use of large deflection limit analysis, However, this approach raises some doubts whether it is compatible with the objective of a limit analysis. The limit analysis is expected to prevent large deformations. At small deformation levels, the effects of geometric strengthening are rather moderate. The geometric strengthening effect often occurs close to the collapse point of the structure when deflections are large, thus crediting strengthening under a condition that the method seeks to prevent. Therefore, the results suggest that it may rather be appropriate for the Code to allow the use of the von Mises failure theory for Limit Analysis. This would also be more convenient for obtaining the limit load, since commercial FE codes generally use the von Mises theory.

RECOMMENDATIONS

The authors recognize that the current ASME Code does not provide clear definitions for interpretation of FE indicated stresses and strains. There is a need for ASME to establish FE interpretation guidelines in addition to those presented in the PVRC 3D Stress Criteria Guidelines [3]. Changing the Code to explicitly allow the use of the von Mises yield criterion for limit analysis should be considered.

REFERENCES

1. ASME (1998), "ASME Boiler and Pressure Vessel Code," Section VIII, Division 2, The American Society of Mechanical Engineers, New York, NY.
2. Wichman, K. R., Hopper, A. G., and Mershon, J. L., 1977, "Welding Research Council Bulletin No. 107," (June 1977 reprint of 107) August, 1965, Welding Research Council, New York, NY.
3. Hechmer, J. L. and Hollinger, G. L., 1997, “3D Stress Criteria: Guidelines for Application,” PVRC Grant 91-14 Final Report, ASME, New York, NY.
4. 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.
5. Martens, D. H., Porter, M. A., and Hsieh, C. S., 1996, "Nozzle Stiffness and Stress Computation Using a Parametrically Controlled Finite Element Modeling Approach," PVP Vol. 336, ASME, New York, NY., PP 323-335
6. 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.
7. 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.
8. 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.
9. Porter, M. A. and Martens, D. H., 2000, “Comparison of Linear and Nonlinear FE Analysis of a Typical Vessel Nozzle,” PVP Vol. 399, ASME, New York, NY, PP 83-86
10. Kalnins, A.: “Guidelines on Plastic Analysis in Pressure Vessel Design: Application of Limit Analysis,” Final Report, Grant #99-PNV-04, Pressure Vessel Research Council, New York
11. Kalnins, A., and Updike, D., “Limit Pressures of Cylindrical and Sperical Shells,” ASME PVP Conference Vol. 407, pp. 69-75, 2000.
12. ANSYS, 1999, ANSYS 5.6 User Manual, 11th Ed., Canonsburg, PA.
13. Algor, 2000, Algor Release 12, Pittsburgh, PA.

APPENDIX A

Shell/Nozzle Data:

The model used for evaluation in this paper consists of a 96" diameter, ½" thick vessel with a 24" diameter, ½" thick nozzle attached perpendicular to the centerline of the vessel. The vessel was reinforced at the nozzle intersection with a 42" diameter, ½" thick pad. The nozzle reinforcement design is per ASME Section VIII, Division 2 [1]. The material is SA516-70. For this material at a temperature of 500 °F, the ASME Section VIII, Division 2 allowable properties are Sm= 20.5 Ksi and Sy= 30.7 Ksi.

Internal Pressure:	165 psi
Nozzle Loads:	
Fz (axial, up):	-6,480 LB *
Mx (circumferential bending):	33,160 FT-LB
My (longitudinal bending):	38,250 FT-LB
Mz (torsion):	25,500 FT-LB

* This force does not include the end force on the nozzle due to pressure. The end force must be added to the typical piping analysis program-reported results, as these programs typically do not include the pressure end force.