COMPARISON OF LIMIT LOAD, LINEAR AND NONLINEAR FE ANALYSES OF STRESSES IN A LARGE NOZZLE-TO-SHELL DIAMETER RATIO APPLICATION

Michael A. Porter
Dynamic Analysis
2815 Stratford Road
Lawrence, Kansas 66049
785-843-3558
mike@dynamicanalysis.com

Steven R. Massey
Black & Veatch Pritchard Corporation
11401 Lamar
Overland Park, Kansas 66211
913-458-6000
masseysr@bv.com

Dennis H. Martens
Black & Veatch Pritchard Corporation
11401 Lamar
Overland Park, Kansas 66211
913-458-6000
martensdh@bv.com

ABSTRACT

The analyses address a nominal 62-inch diameter nozzle in a nominal 124-inch diameter shell with a reinforcement pad. The nozzle is in a channel of a heat exchanger. This results in stiffening of the shell (adjacent to the nozzle) by the tube sheet and the channel head. The results of a WRC 297 analysis, linear elastic analysis, limit load analysis and plastic analysis are compared. The finite element analyses were accomplished utilizing commercial software and typical modeling techniques. As there is significant variance in the results derived with the different methodologies, the authors discuss the comparison of the results.

INTRODUCTION

An investigation of the stresses at the junction area of a nozzle in a thin-walled vessel with imposed connected pipe moments is typically necessary for confirming that the application is safe and functional. The design of nozzles and required reinforcement is addressed by ASME Section VIII Div 1 [1]. The design-by-rule provisions utilized for a Div 1-designed vessel do not include provisions for the imposed pipe loads typically encountered. A WRC 297 [2] analysis of these loads is considered adequate for most applications. However, the design engineer is responsible for the confirmation that the WRC 297 usage is appropriate. If the nozzle is of an unusual configuration, is at the maximum usage a limit of WRC 297, or WRC 297 indicates high stress, the design engineer will need to conduct additional analyses. Finite element analysis is typically utilized.

The heat exchanger channel nozzle addressed in this paper has a diameter-of-nozzle to diameter-of-vessel ratio at the upper limits of WRC 297 application. In addition, the configuration of the nozzle reinforcement and the stiffening of the region by the adjacent tube sheet and channel head are not fully addressed by WRC 297. This is not an unusual problem associated with a thin walled heat exchanger application.

The design engineer must address the various design temperatures, pressures and imposed loading to which the nozzle is subjected. This includes the start-up and shutdown conditions. For this exchanger, these include both a design pressure at temperature with corresponding pipe imposed loading case and second case of no pressure but with the temperature with corresponding pipe loading imposed. The exchanger pressures and temperatures cycle only occasionally and do not accumulate sufficient cycles to require a cyclic stress analysis.

The nozzle reinforcement requirements of Section VIII Div 1 were met by the configuration.

NOZZLE INFORMATION

The nozzle and the shell material is SA 516-70, the related information is found in Table 1.

Table 1 - Nozzle Parameters

Design Internal Pressure	25 psig
Design Temperature	600˚ F
Moment Circumferential	3,000,000 in-lb
Moment Longitudinal	1,500,000 in-lb
Moment Torsional	0 in-lb
Axial Force (into shell)	-10,000 lb*
Nozzle Inside Diameter	62.83 inches
Nozzle Wall Thickness	1.42 inches
Shell Inside Diameter	123.58 inches
Shell Wall Thickness	.551 inches
Reinforcement Pad OD	68.1258 inches
Reinforcement Pad Thickness	0.512 inches

* This force does not include the end force on the nozzle due to pressure. Since these piping programs do not typically include the pressure end force, the end force must be added to the piping analysis results when conducting an FE analysis.

Figure 1 indicates the relative position of the nozzle to the tube sheet and channel head.

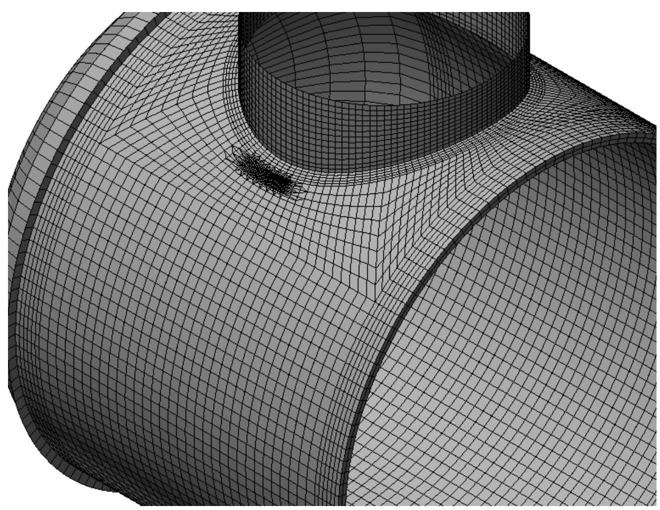

FIGURE 1 Finite Element Model of Nozzle

The open end of the model is actually the tube sheet end. The tube sheet and channel head have the effect of stiffening the shell and resisting the bending from the longitudinal moment loading. The circumferential moment is resisted by the channel shell and is not significantly resisted by the stiffening effect of the tube sheet. With the loads that were applied, these effects were minimal on the resulting maximum stresses.

WRC 297 ANALYSIS

The nozzle was investigated utilizing the WRC 297 methodology. The stress results indicated by WRC 297 are located at the crotch of the nozzle and do not represent the stresses at the outside diameter of the reinforcement pad included on the shell. The results of the WRC 297 analysis appear in Table 2 below.

TABLE 2 - Stress Intensity for Combined Loads, PSI

Part	Plane	Surface	Membrane	Bending	Combined
Vessel	Long.	Outer	S=709	S1=2,707	S2=4,453
		Inner	S=709	S1=2,707	S2=5,951
	Trans	Outer	S4=3,497	S5=47,199	S6=49,662
		Inner	S4=3,497	S5=47,199	S6=44,735
Nozzle	Long.	Outer	S8=709	Sbt=1,324	S9=1,629
		Inner	S8=709	Sbt=1,324	S10=1,727
	Trans	Outer	S11=3,497	Sbt1=22,403	S12=23,050
		Inner	S11=3,497	Sbt1=22,403	S13=25,253

The maximum membrane plus bending stress indicated in Table 2, 49,662 psi, is approximately twice that shown at that location, as indicated in Figure 2. However, note that most of the curves that must be read to determine factors in Figures 3–60 of WRC297 will give only approximate values because factor λ=5.4823 is beyond the end of the curves shown on most of the figures. The text in WRC 297 advises that the curves should not be extrapolated beyond the λ values at the end of the curves. In common practice, however, a designer who does not (or did not in the past) have access to FEA methods would extrapolate the values to arrive at the above results.

The maximum membrane plus bending stress of 49,662 psi would be compared to the allowable maximum membrane plus bending stress of 3Sm (58,200 psi) per ASME Section VIII, Div.2, and Appendix 4. This design would be declared adequate to sustain the applied loadings.

LINEAR ANALYSIS

An Elastic analysis of the nozzle was conducted using conventional Finite Element techniques.

Modeling

The model for the linear finite element analysis utilized 4-node linear plate elements. Approximately 132 elements (as suggested by Primm and Stoneking [3]) were used around the circumference of the nozzle, as illustrated in Figure 1. Since the major load was expected to be radial bending in the shell, the initial element size in this direction was selected to be somewhat less than in the circumferential direction. For the Finite Element model, the tubesheet was assumed to be rigid. Therefore, all of the connection nodes were fully restrained in both displacement and rotation.

Loading

The operating load consisted of an internal pressure, two moments and an axial force on the nozzle. The end load on the nozzle, due to the 25-psi internal pressure, is approximately 77,500 LB. The resulting axial load is then approximately 67,500 LB acting outward. Without the internal pressure, the 10,000 LB axial load acts inward. Since a loss of pressure could occur without an immediate reduction in the external forces and moments, cases with and without the internal pressure were examined.

Results

The results of the initial linear analysis for the operating condition indicated a maximum stress intensity of 39,000 psi. This occurred at the OD of the reinforcement pad in the location indicated Figure 2. Without the internal pressure, and hence the pressure end load, the maximum indicated stress was approximately 41,000 psi. Thus, the controlling case is that without the internal pressure applied.

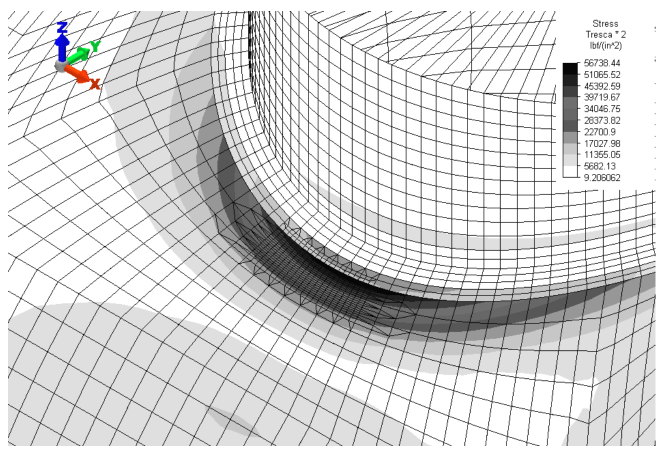

Figure 2 Linear FE Stress Profile

Examination of the stress results indicated a high stress gradient in the elements nearest the region of highest stress. This is an indication that results of the analysis are not fully converged. To evaluate the solution, a stress convergence analysis was conducted.

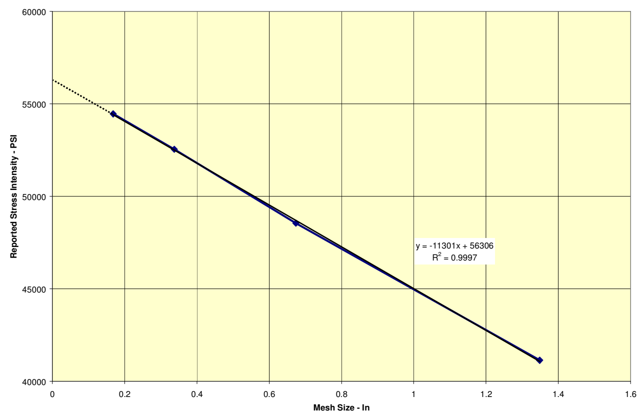

Figure 3 – Model Convergence Analysis

Figure 3 illustrates the highest indicated stress as a function of the mesh dimensions in the area of highest stress. The size of the elements were cut in half three times during the analysis and the highest stress plotted. Plotting the stress as a function of mesh size results in a straight line. This allows an easier estimation of the converged stress value that the function resulting from a mesh density (1/mesh size) plot. From these data, an estimate of the stress at an element size of 0 (converged) was developed. The value of the converged stress at the edge of the re-pad was 56,300 psi.

The results of the linear analysis are indicated graphically in Figure 4. The criteria line in Figure 4 was established as indicated in a paper by Porter et al [4].

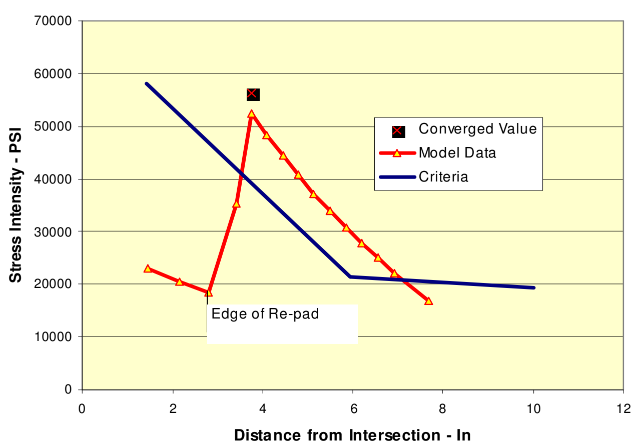

Figure 4 Linear Stress Results

It should be noted that the indicated stress in the outer surface of the shell at the edge of the reinforcement pad OD exceeds the Code criteria established in the paper by Porter et al. However, if we consider this point to be a structural discontinuity and apply the PL + PB + Q criteria as is done with WRC 297, the maximum indicated stress intensity of 56,300 psi is slightly lower than the allowable of 3Sm (58,200 psi) per ASME Section VIII, Div.2, Appendix 4.

LIMIT LOAD ANALYSIS

A Limit Load analysis was conducted for the nozzle assembly using the techniques outlined by Kalnins and Reinhardt [5] and Porter et al [6].

Modeling

The model was identical to the linear model with 2 exceptions:

1. 8-node quadratic shell elements (shell93) were used.
2. No mesh refinement was conducted in the high stress region.

Based on an initial linear analysis using these elements, it was concluded that the mesh refinement used for the 4-node elements was not necessary

Loading

Since the controlling loads were the moments and axial force without the internal pressure, the internal pressure was not applied. The material was assumed to become perfectly plastic (tangent modulus = 0) at 1.5Sm (29,100 psi).

Results

The displacement versus applied load curve is illustrates in Figure 5. The model failed to converge at a load of approximately 3 times the design load. Using the 2/3 reduction required by Code and the factor of 1.15 to convert from von Mises to Stress Intensity, the analysis indicates that the nozzle is acceptable by a factor of approximately (3 x 2/3 / 1.15) = 1.743.

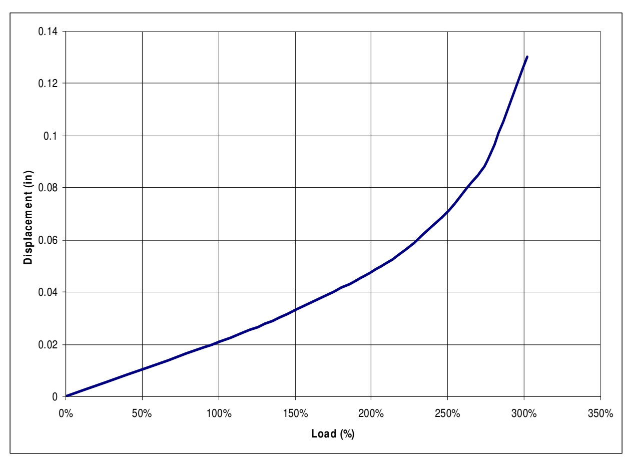

Figure 5 – Displacement vs. Load for Limit Load Model

The collapse occurs at the edge of the re-pad at the same point indicated in the elastic analysis. The failure mode would seem to be buckling in the shell of the exchanger. Without conducting an n arc-length analysis, it is not certain that the Limit Load analysis is conservative.

PLASTIC ANALYSIS

A plastic analysis was conducted for the nozzle assembly using the techniques outlined by Kalnins and Reinhardt [c] and Porter et al [d].

Modeling

The model was identical to the linear model with 2 exceptions:

1. 8-node quadratic shell elements (shell93) were used.
2. No mesh refinement was conducted in the high stress region.

Based on an initial linear analysis using these elements, it was concluded that the mesh refinement used for the 4-node elements was not necessary.

Loading

Since the controlling loads were the moments and axial force without the internal pressure, the internal pressure was not applied. The material was assumed to have a yield of 38,000 psi and a tangent modulus of 202,000 psi) for the analysis.

Results

The results for the plastic analysis using strain as the criteria are illustrated in Figure 6. Using the double slope method detailed in Section VIII, Div. 2, Appendix 4-136.5, the collapse load is approximately 1.42 times the design load. Thus, using the 2/3 value from the same source, the allowable load is approximately 0.95 times the design load.

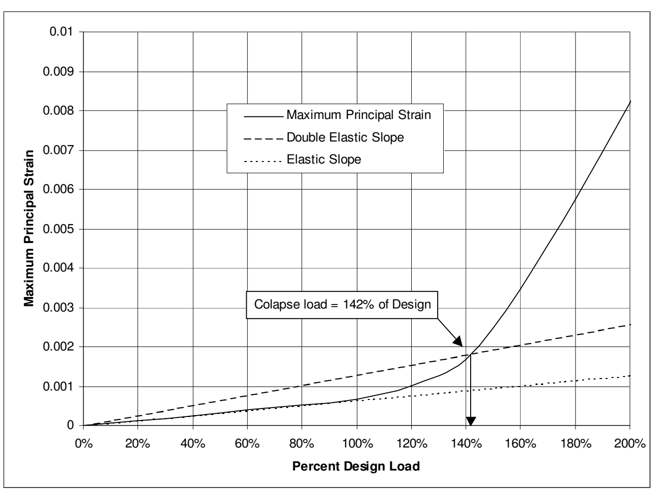

Figure 6 – Strain vs. Load for Plastic Analysis of Model

If, however, displacement is used as the criteria (both are allowed in Appendix 4-136.5), we get a substantially different result. These displacement-based results are illustrated in Figure 7.

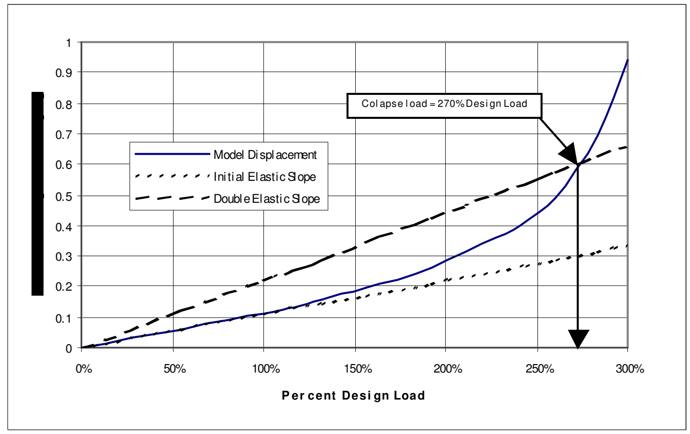

Figure 6 – Displacement vs. Load for Plastic Analysis of Model

Using displacement as the criteria, the collapse load is approximately 270% of the design load. Using the 2/3 criteria, the allowable is approximately 1.8 times the design load. The collapse point was at the same location as indicated in the Elastic and Limit Load analyses. The same precaution concerning a buckling collapse would apply.

SUMMARY

The WRC 297 analysis indicated that the design was adequate for the loads specified by a small margin. The linear stress analysis indicated that there was a potential problem at the edge of the re-pad. However, considering the edge of the re-pad to be a structural discontinuity, it did meet the 3Sm criteria by a slightly smaller margin than did the WRC 297 analysis. According to the Limit Load analysis, the design had a margin of safety of 1.74 for the given load, while the Plastic analysis indicated a margin of safety of approximately 0.95 or 1.8 depending on whether strain or displacement is used for the evaluation. These results are summarized in Table 6

Table 3 – Summary of Analysis Results

Methodology	Allowable/Indicated Stress at Design Load
WRC 297	1.17
Elastic FE	1.03
Limit Load FE	1.74
Plastic FE (Strain)	0.95
Plastic FE (Displacement)	1.80

DISCUSSION

The indicated maximum stress intensity (calculated by WRC 297 methodology) due to the circumferential loading was 49,662 psi. This stress is normally considered to occur at the crotch of the nozzle-to-shell junction. The re-pad is not included in the WRC

297 analysis. Thus, if the re-pad is considered as a thick nozzle section, we can compare the WRC 297 results to the FE results.

The Elastic analysis indicated a maximum stress at the edge of the repad that would exceed the criteria recommended by Porter teal. [4], but would allowable if judged by the 3Sm criteria alone.

The Limit load analysis indicated the configuration was acceptable with a design safety margin of 1.74 times the design loadings. This seems reasonable in that the limit load procedure is more involved and would be expected to be somewhat less conservative. However, since the collapse mode is buckling, further analysis is indicated.

With a design safety margin of 0.95 times the design loadings, the Plastic analysis, using strain as the criteria, indicated the configuration was not acceptable. This result was not expected. Using displacement as a criterion, the plastic analysis results compare well with the Limit Load analysis.

CONCLUSIONS

The geometry of this nozzle was such that the WRC 297 procedure could not be applied in strict adherence to the procedure limitations. However, it is common practice in industry to use it anyway. Based on comparison with the other procedures, this seems to be not particularly problematic.

The elastic analysis procedure is relative simple to conduct, although the need for a convergence analysis is indicated. The use of recommended "rules of thumb" for mesh density is not safe for all cases, as was demonstrated in this model. More problematic is the interpretation of the results. According to the criteria suggested by Porter et al, the nozzle would not be acceptable. Using the WRC 297 criteria, however, it would be acceptable. The evaluation in this particular case is not clear cut.

The Limit Load procedure seems to indicate an even higher margin of safety for this nozzle. This was expected, although the magnitude of the margin of safety increase was somewhat surprising.

The Plastic analysis results were not expected. The indicated margin of safety, 0.95 using strain is less than the 1.0 required for an acceptable design. The discrepancy between the Limit Load and Plastic results would seem to raise questions about the use of these procedures for actual design work.

REFERENCES

1. ASME (2002 with addenda), "ASME Boiler and Pressure Vessel Code," Section VIII - Divisions 1 and 2, The American Society of Mechanical Engineers, New York, NY
2. Revised WRC 297, September 1987, Mershon, Mokhtarian, Ranjan, & Rodabaugh Welding Research Council, New York, NY.
3. Primm, A. H., and Stoneking, J. E., 1989, "Accuracy for the Finite Element Method for Pressure Vessel/Nozzle Design", PVP-175, 1989, The American Society of Mechanical Engineers, New York, NY. pp 3-9
4. 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
5. Kalnins, A., and Reinhardt, W., (2002) "Do's and Don't in Using Elastic-Plastic Finite Element Analysis for Design by Section III and Section IV Rules," PVPD-40, ASME, New York, NY
6. Porter, M., Reinhardt, W., and Martens, D., (2001), "Comparison of Limit Load, Linear and Nonlinear FE Analysis of a Typical Vessel Nozzle," PVP Vol. 430, (<u>Pressure Vessel and Piping Design and Analysis</u>), ASME, 2001