Transactions of the **13th International Conference on Structural Mechanics in Reactor Technology** (SMiRT 13), Escola de Engenharia - Universidade Federal do Rio Grande do Sul, Porto Alegre, Brazil, August 13-18, 1995

Stress analysis of a reinforced horizontal cylindrical pressure vessel resting on saddles

Miranda, C.A.J.
IPEN-CNEN/SP - Instituto de Pesquisas Energéticas e Nucleares, São Paulo, SP, Brazil
Aquino, C.T.E.
COPESP - Coordenadoria para Projetos Especiais, São Paulo, SP, Brazil

ABSTRACT. This work presents the results obtained from frequency and stress analyses performed in a horizontal cylindrical pressure vessel. The global behavior of the pressure vessel is studied, not considering local discontinuities caused by flanges, inlets, manways, etc. Initially, the lower frequencies of the pressure vessel are computed and, in sequence, the stresses due to the loadings are calculated. For the seismic loading, and for conservatism, the maximum acceleration of a typical response spectrum is used in 3 directions, and the SRSS method is adopted for their combination. Finally, for the stress verification, the procedures described in the ASME Code are followed.

1 INTRODUCTION

The structure is, basically, a large steel cylinder, with circumferential stiffeners, and torispherical heads on both ends. The vessel is divided in three different compartments named A, B and C, by membrane-type structural elements, shown in Figure 1. The equipment installed inside the vessel is considered to give no structural contribution, only its loading contribution is considered in the analyses. The analyzed structure is supported by four saddles which are plate-and-shell welded type structures. One of the saddles is a fixed type support with its three translations restrained. The other saddles allow longitudinal translations to accommodate the longitudinal thermal expansion of the vessel. To verify the stress results the limits of the ASME Boiler and Pressure Vessel Code, Section III, Division 1, Subsection NB, Class 1 Components (1989) were used

2 MODEL AND LOADS

The loads acting on the structure are the dead weight (including the internal equipment), postulated pressures, and seismic loads. From the specifications, the pressures can act in the compartments as indicated in Table 1. The adopted material properties are those of the ASTM A543-72a, Type B, Class 1 steel.

From a previous modal analysis, where the natural frequencies of the structure were calculated, the first frequency of the structure is greater than 33 Hz. For this reason the seismic loads were applied as equivalent static accelerations, with their values taken as 0.5g, where g is the gravity acceleration (9.81 m/s^2). This value is an upper limit of the seismic acceleration response spectra at the base and was used for each direction.

A half of the structure was modeled taking credit of the geometry symmetry and the symmetry or anti-symmetry of the loads. The finite element model, see Figure 1, consists of about 2300 nodes, with 6 dof per node, with shell and 3-D beam type elements. The ANSYS program (De Salvo & Gorman, 1992) was used to perform the analyses.

The check of the membrane-type structural element stresses is out of the scope of this analysis and, in consequence, they are modeled in an approximate way. Their stress state will be verified by specific analyses, considering their actual geometry, with their discontinuities and penetrations. There is one support under each pair of these structural elements. Figure 1 outlines the geometry and shows the developed model.

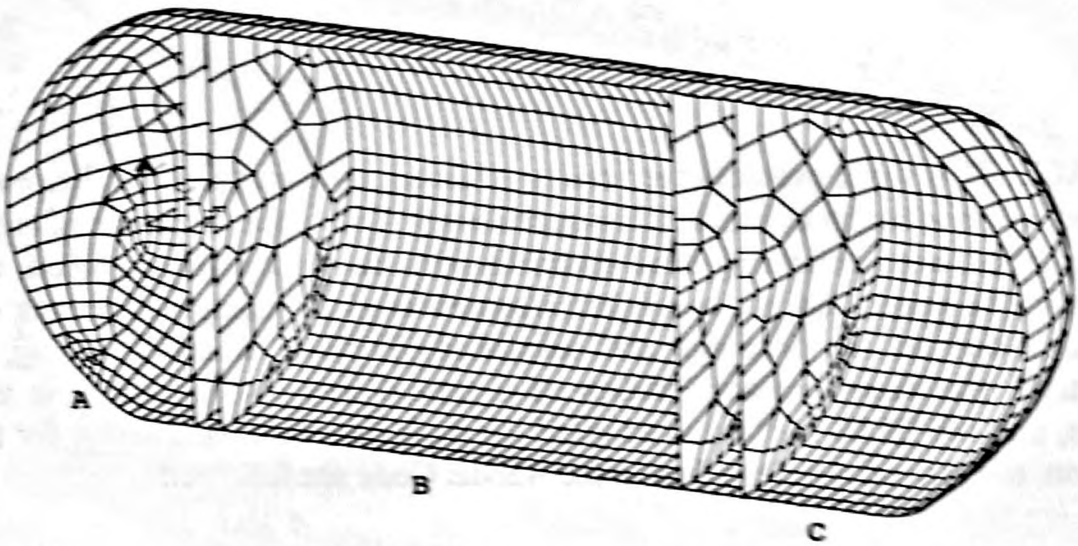

Figure 1 - Finite Element Model - Half geometry and the compartments A, B and C

2.1 *Boundary Conditions*

Considering the axes shown in Figures 1 and 2, it can be noticed that: a) on the nodes of the plane X=0 - due to the adopted model symmetry and for loads acting in the Y (vertical) and Z (axial) directions, symmetry conditions were defined. Anti-symmetry conditions were defined for the loads acting in the X (transversal) direction; b) For the nodes associated with the saddle supports - the respective constraints of the support type were imposed directly on the nodes of the supported structure. The support saddles are considered much more rigid than the cylinder, therefore they are not modeled. This may be considered a conservative assumption. The boundary conditions associated with the saddle supports are presented in the Figure 2.

2.2 *Loads and Load Combinations*

For the analyses the loads were defined in different load steps to permit individual load evaluations and a proper combination of them. For each compartment, isolated or combined pressures were postulated as indicated in Table 1. The considered load steps are then: a) L. step 1 - dead weight, including the vessel itself - cylinder, stiffeners, and heads, and the internal equipment (applied as an 1g acceleration in the Y direction and some distributed forces, in the same direction, associated with the equipment); b) L. steps 2, 3 and 4 - pressure distribution in the compartments, as indicated in Table 1; c) L. step 5 and

L. step 6 - a static 0.5g acceleration value applied respectively in the axial (Z) direction and in the transversal (X) direction, representing the seismic components in the Z and X directions, with the respective distributed forces due to the equipment. For the Y seismic component it was taken a half of the results of the L. step 1. To combine the results of the spatial seismic components the SRSS (Square Root of the Sum of the Squares) method was adopted. The obtained seismic results were combined with the others always in two steps: using the positive sign at first and the negative sign at the second time.

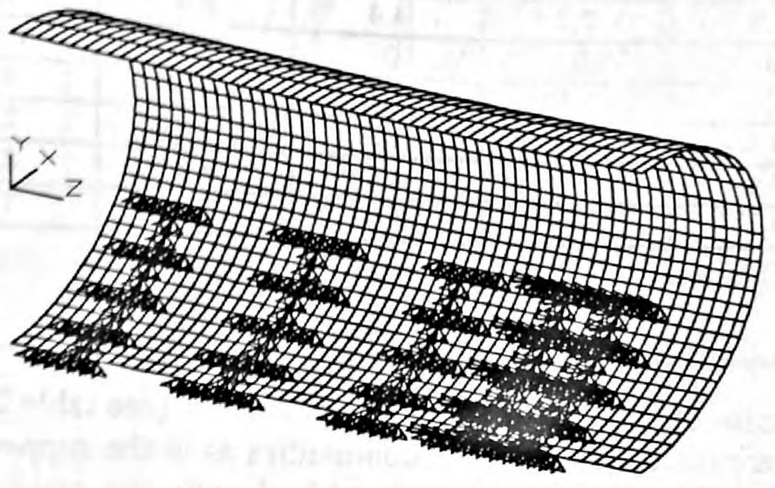

Figure 2 - Boundary Conditions Associated with the Saddle Supports

Table 1: Pressure distributions in the compartments A, B and C

Load Step	Compartment A	Compartment B	Compartment C
#2	External pressure	Internal pressure	External pressure
#3	Internal pressure	Internal pressure	External pressure
#4	No pressure	No pressure	External pressure

Load Combinations. To verify the stresses in the analyzed structure five load combinations were considered, covering the specifications and the Code requirements. Combination #1: only dead load (L. step 1); Combination #2: dead load + seismic load; Combination #3: as in Combination #2 + pressure distribution in L Step 2; Combination #4: as in Combination #2 + pressure distribution in L. Step #3; Combination #5: as in Combination #2 + pressure distribution in L. Step #4.

3 RESULTS

Typical stress distributions are presented in Figures 3 to 8, in terms of Tresca stress intensities (SI) values associated with the MIDDLE or the external (TOP) shell element surfaces, plotted over the deformed structure. The values associated with the iso-stress intensity curves of Figures 3 to 8 are indicated in Table 2. Due to that stated before (§2) the SI values in the membrane-like structural elements are not presented . The maximum stress value in the heads, due to the dead weight (load step 1), is very low: about 2.3 MPa. The same applies for the load steps 5 and 6 (seismic Z and X acceleration), respectively: 8.1 MPa and 3.3 MPa. These stress distributions will not be presented.

Table 2 - Values of the SI curves in the figures - MPa

Figure	curve A	curve B	curve C	curve D	curve E	
3	3.2	9.5	15.8	22.1	26.4	L. Step 1
4.a	4.8	12.3	19.9	27.5	35.1	Load Step 2
4.b	4.6	---	19.9	---	35.1	
4.c	6.1	16.9	27.8	38.6	49.4	
5.a	7.7	11.3	14.9	18.5	22.1	Load Step 3
5.b	0.3	0.8	1.3	1.8	2.3	
5.c	0.9	2.7	4.4	6.2	7.9	
6.a	0.	0.	0.	0.01	0.02	Load Step 4
6.b	1.3	3.3	5.3	7.2	9.2	
6.c	0.3	0.8	1.3	1.8	2.3	
7	0.9	2.7	4.5	6.3	8.1	L. Step 5
8	0.4	1.1	1.9	2.6	3.3	L. Step 6

4 DISCUSSION

In general, the stresses obtained from the analyses are low (see table 2), with peak values at those points where there were gross discontinuities as in the supports. It's a hard task, sometimes an impossible one, to separate and classify the stresses in 3-D models according to ASME code. There is a discussion (Hechmer & Hollinger, 1991) on this subject suggesting new code requirements that contemplates largely used 3-D finite element stress analyses, instead of the actual code requirements derived from the shell theory with simple geometries and compatibility among them. In this paper, in general, to establish the membrane stress values in the heads, the stress values associated with the curves that were $\sqrt{Rt}$ far from any discontinuity or load application point were adopted. R and t are, respectively, the mean radius and the thickness of the shell. With this assumption the MIDDLE stresses are associated with the membrane values Pm, and the greater between the TOP and the BOT stresses are associated with the membrane + bending values Pl+Pb. Similar hypothesis will be considered for the cylinder where, due to the supports and the stiffeners, with an spacing less than $\sqrt{Rt}$, there is no Pl stress. The MIDDLE stress values, far from the supports and the load application points, will be considered as Pm values. The Pl+Pb stresses is taken from the greater TOP or BOT stress values.

In each structural part, for the sake of conservatism, and only when it is possible, the maximum nodal stress values found in the analyses were adopted.

Near the supports the shear stresses in the shell, calculated from the reactions were verified to prevent shell tearing.

This is a conservative verification because, in the modeling, the stiffening effect over the cylindrical shell, due to the equipment support bases in the vessel interior were not taken into account.

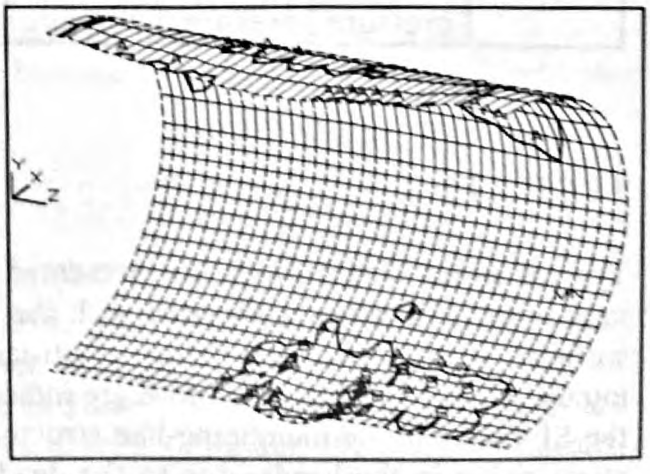

Figure 3: Load Step 1 - Dead Weight SI TOP - Cylinder

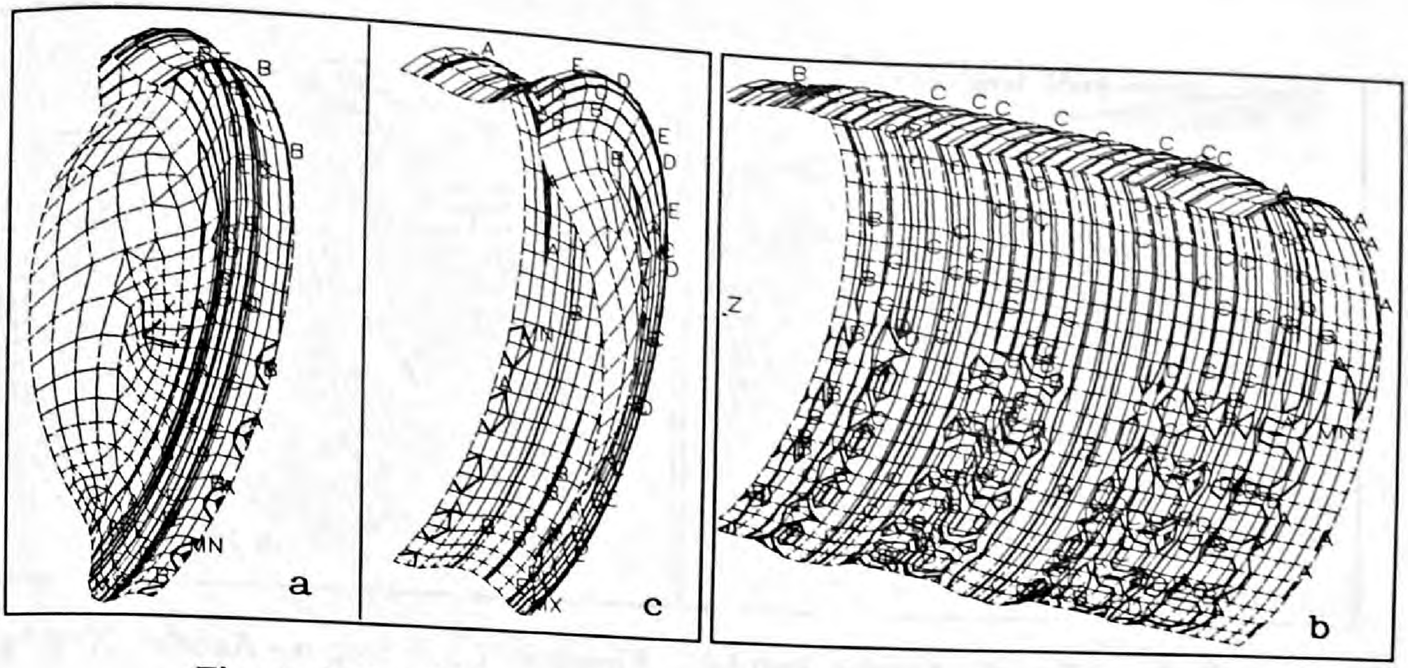

Figure 4 : Load Step 2 - Pressure - SI TOP - Heads and Cylinder

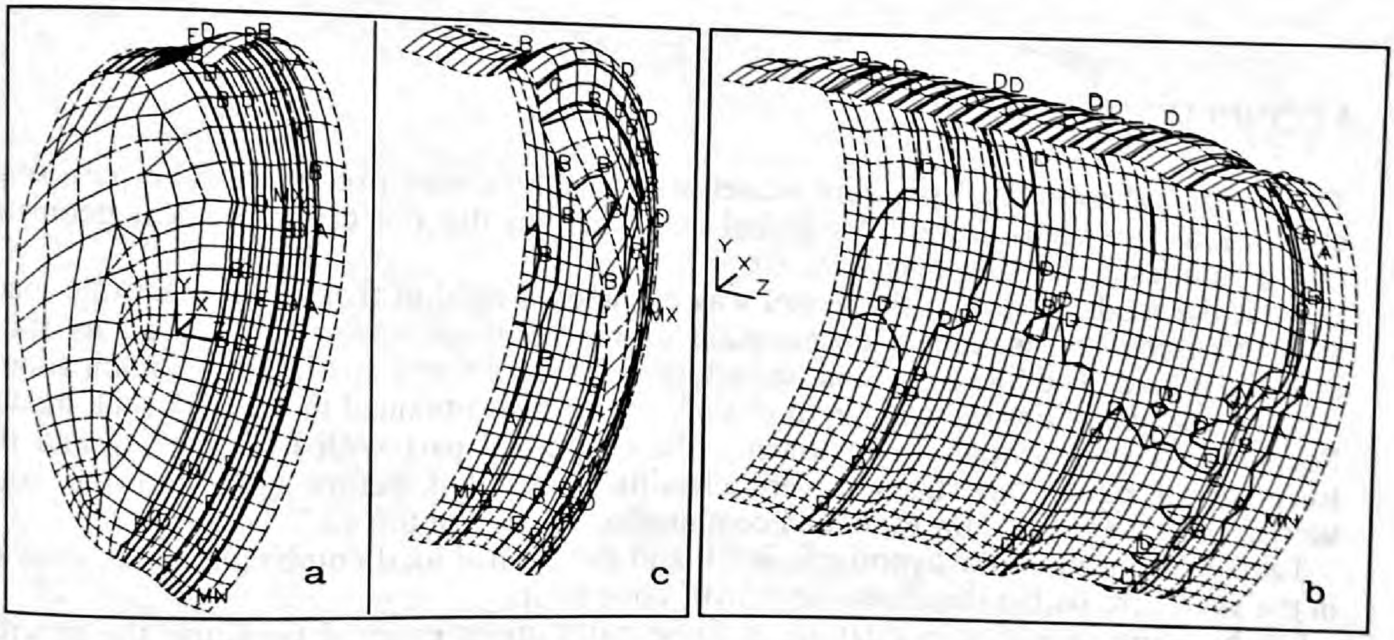

Figure 5 : Load Step 3 - Pressure - SI MIDDLE - Heads and Cylinder

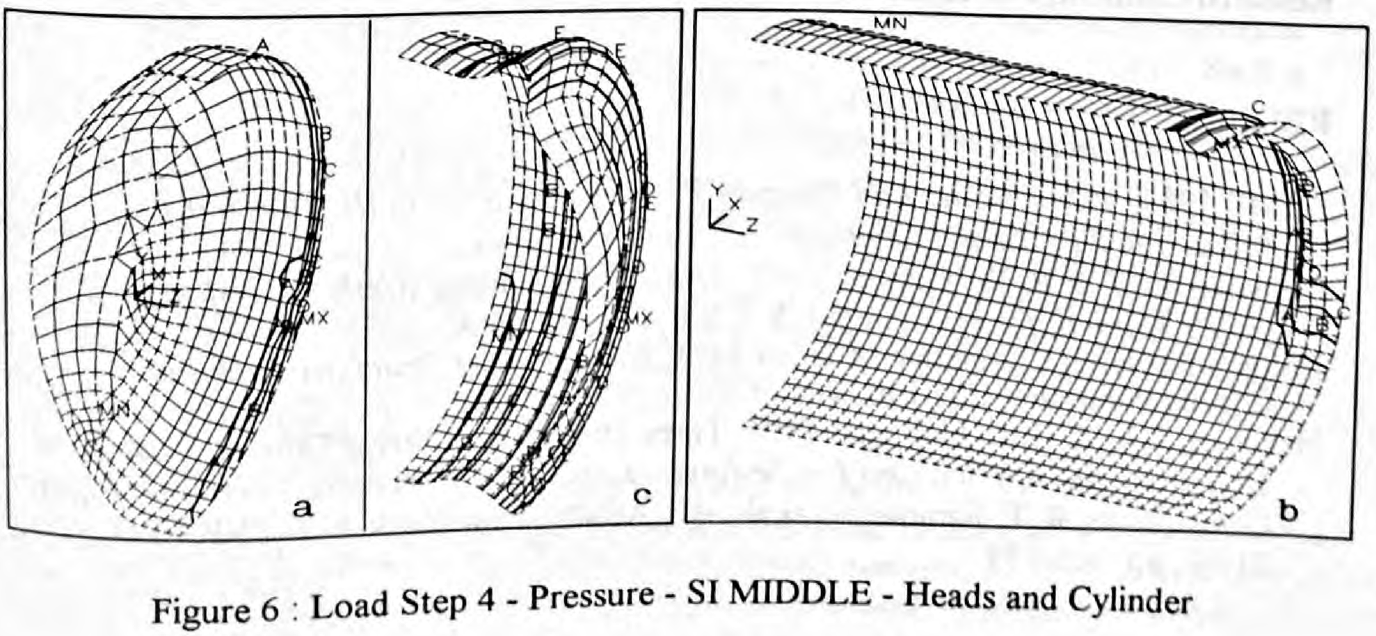

Figure 6 : Load Step 4 - Pressure - SI MIDDLE - Heads and Cylinder

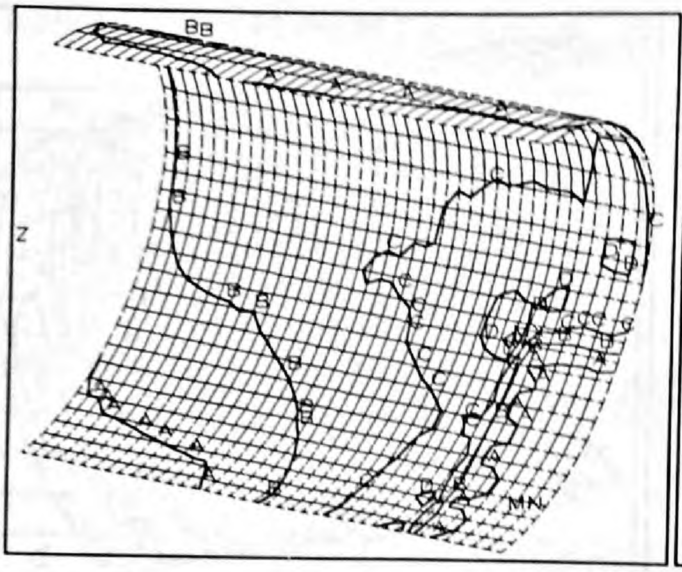

Figure 7 : Load Step 5 - Acceler. Z=0.5g
SI MIDDLE - Cylinder

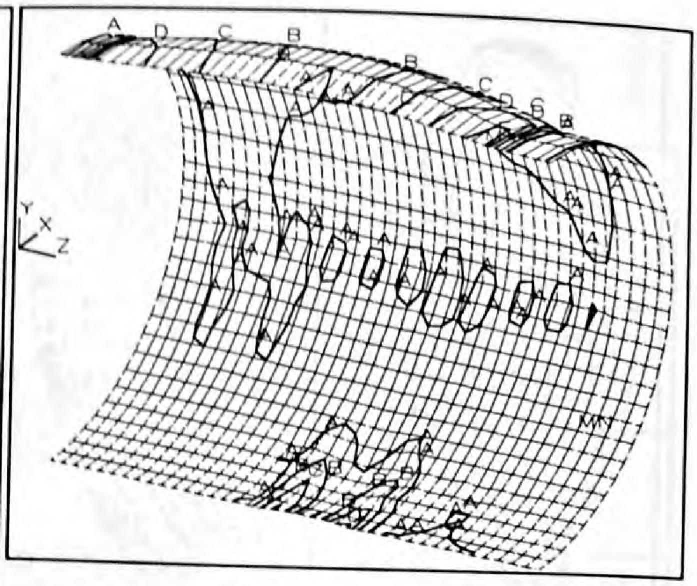

Figure 8 : Load Step 6 - Acceler. X=0.5g
SI MIDDLE - Cylinder

5 CONCLUSIONS

The horizontal cylindrical pressure vessel stress analysis was presented. The performed calculations took into account its global behavior, but did not consider local geometry discontinuities, as flanges, manways, etc.

From the modal analyses, the vessel was considered rigid in front of the seismic loads and, therefore, the maximum acceleration of the response spectra was used as static equivalent load. The SRSS method was adopted for the seismic spatial load combination.

Some considerations were made to classify the stresses obtained in the 3-D shell model. For all the vessel structural components, the cylindrical part with its stiffeners and the torispherical parts, the stresses were classified as stated before (§4) and the code verifications were made for each load combination defined in the §2.2.

Taking into account the hypothesis in §4, and the defined load combinations, all stresses in the vessel are within the allowable ASME code limits.

For buckling simplified evaluation, in those parts under external pressure, the specific ASME code recommendations, for the pressure load, were followed and additional calculations were made using simplified formulations found in handbook (Column Research Committee of Japan 1971).

REFERENCES

ASME 1989. *ASME Boiler and Pressure Vessel Code, Section III, Division 1, Subsection NB, Class 1 Components*". New York, NY.

De Salvo, G. J. & R. W. Gorman 1992. *ANSYS Engineering Analyses System, User's Manual, for revision 5.0*. S.A.S.I., Houston, PA.

Column Research Committee of Japan 1971. *Handbook of Structural Stability*. Corona Publishing Co., Ltd. Tokyo.

Hechmer, J. L. & G. L. Hollinger 1991. Three Dimensional Stress Criteria. *Codes and Standards and Applications for Design and Analysis of Pressure Vessel and Piping Components*. R. F. Sammataro et al., ed., ASME. New York, NY. PVP - Vol. 210-2, pp. 181-191.