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 cylindrical vessel support saddles

Albuquerque, L.B.
COPESP - Coordenadoria para Projetos Especiais, São Paulo, SP, Brazil
Miranda, C.A.J.
CNEN/SP-IPEN - Comissão Nacional de Energia Nuclear, São Paulo, SP, Brazil

ABSTRACT: The structural analyses of the cradle supports of a large horizontal equipment were presented. The objective of the analyses was to verify the viability of this geometric conception, taking into account only the aspects related to the stress analysis. The ASME NF subsection was used to define the allowable stress limits. The imposed loads were: seismic load, applied as an equivalent static load, and dead weight. The analyses were undertaken by the use of finite element techniques. The supports were discretized with shell elements. The loads to be applied, and the first frequencies were calculated with a beam model. The results showed the possibility of maintaining the stresses below the allowable limits.

1 INTRODUCTION

The analyses presented here are to verifiy the viability of the geometric conception of the cradles supports taking into account only the aspects related to the stress analysis. The other parameters that will contribute to define the viability of this solution, such as post-weld heat treatment, are out of scope of this work. The analyzed cradle supports are welded in a cylinder and are formed by two basic saddle types: one saddle fixed to the base and six other saddles allowing longitudinal displacements to accommodate thermal expansion. Each saddle is formed by one or two principal plates and some transversal ones welded to each other, with stiffeners in the extremities and a base plate. Figure 1 presents the analyzed structure. The loads are: dead weight and an earthquake loading (OBE - Operational Basis Earthquake). The OBE is taken into account considering the earthquake loading as an equivalent static load. Such approach was possible due to the fact that the first frequency of the structure is close to 33 Hz. To verify the stresses in the saddles, it was decided to use the design limits of the ASME code (1989a). Previous simplified calculations were performed with beam model to define the first frequencies and the loads transmitted to each saddle by the supported structure. After that, based on the actual support geometries, two basic saddles were discretized: the fixed type (here called FIXED) and the other most loaded one simple support type (here called UNFIXED). The descritization was performed by the use of shell finite elements of 4 nodes and 6 degrees of freedom each from the ANSYS program (De Salvo & Gorman 1989). With that, it was possible to know in details the stress distribution of the saddles. For the material of the supports, the properties listed in Appendix I of the ASME Code (1989b) corresponding to 100°C design temperature were adopted as: Sm = 120.66 MPa and Sy = 238.56 MPa.

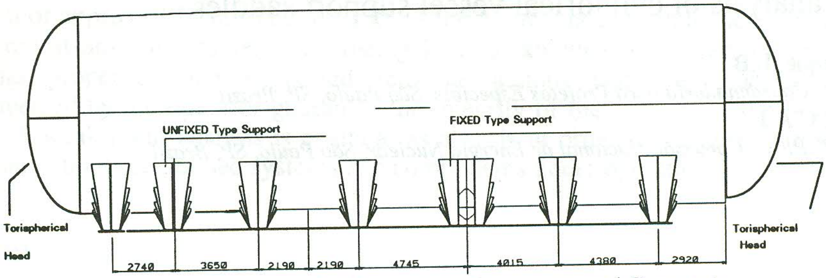

Figure 1a - General View of the Structure and Supports

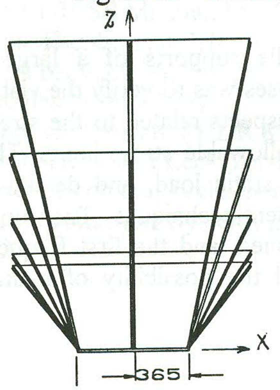

Figure 1b - Unfixed Type Support

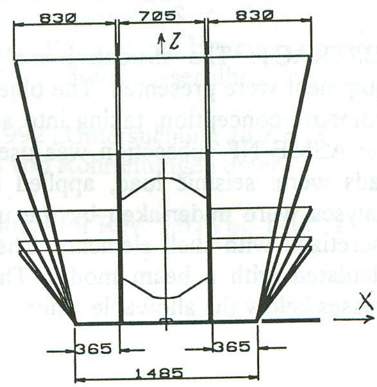

Figure 1c - Fixed Type Support

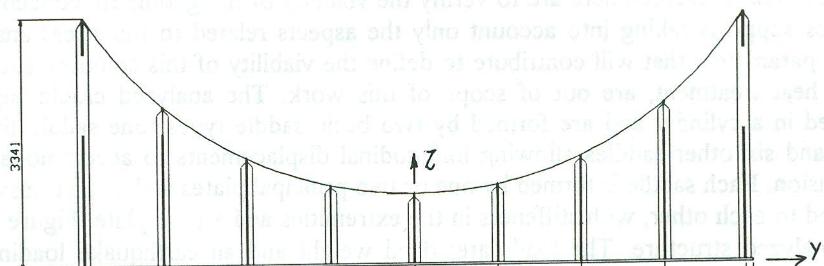

Figure 1d - Detail of the Supports

2 BEAM MODEL

A 2-D beam model was used to represent the cylinder and to obtain the first vibration modes and frequencies. The main internal components of the supported cylinder were represented in the actual positions (longitudinal and vertical) by concentrated mass elements and rigid 2-D beams with no mass. The elements have no rotational inertia. From the modal analysis, it was found, for the first natural frequency, a value near 33 Hz. So one can say that no amplification will occur during the seismic excitation and an equivalent static acceleration can be used to determine the reactions in each saddle support. In order to calculate the reactions in the saddles, the modeled beam was loaded with 0.5g in three load steps. One load step for each direction. Notice that 0.5g is the maximum response spectra acceleration acting in the base level; "g" is the gravity acceleration equal to 9.81 m/s^2. The results, in terms of reactions forces, will be applied in the saddle analyses taking into account its eccentricity. In the 2-D beam model we do not have the transversal degrees of freedom. However, the supports have the same kind of reactions both in the vertical and in the transversal directions.

3 STRESSES IN THE SUPPORTS

Two FE models were developed for the determination of the stresses acting in the supports: one FE model was built up for the FIXED saddle and another FE model for the simple support type (UNFIXED). The last model correponds to the most loaded simple support type. Shell elements with 4 nodes and 6 degree of freedom (dof) per node were used for both FE models. The longitudinal plate has a thickness of 25mm, the transversal plates and its stiffeners are 16 mm thick and the base plate is 50 mm. In each model, a rigid region involving all nodes in its cylindrical upper part was defined. This rigid link forces all dof at the corresponding nodes to behave uniformly. In addition there is a 3-D rigid beam element, with no mass, connected to the central position of the circular supported section (see point 1 in figures 3 and 8). Such point is the place where the reactions were calculated in the previous step (beam model). This 3-D link is also connected to a point in the upper part of the shell model (point 2, figures 3 and 8). This rigid link was conceived to transmit the reactions obtained in the beam model to the shell model.

3.1 *UNFIXED Cradle Model.*

The complete model of this support is shown in figure 3. Note that the central node in the base plate have the UX displacement restrained to eliminate singularities due to the boundary conditions imposed at that region of the model.

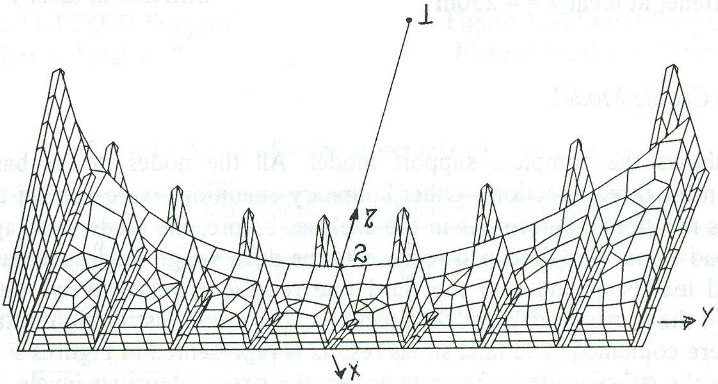

Figure 3 - UNFIXED Support Model

Other boundary conditions were applied to simulate the rigid connections mentioned before. In this model the loads were applied in two load steps. The first load step represents the dead weigh load in the vertical direction corresponding to Z-coordinate. It also includes the dead load of the supported structure (beam reactions) and the dead load of the support itself. The second load step represents the loads (beam reactions) in the transversal direction, Y-coordinate. After the analyses, the stress distributions were appropriately combined. The results of such combinations are represented in Figures 4 to 7. Also Table 1 summarizes the stress values corresponding to the stress intensities levels shown in these figures.

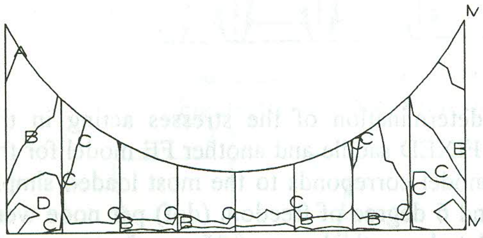

Figure 4-UNFIXED Model Stresses transversal Plate at local x=0

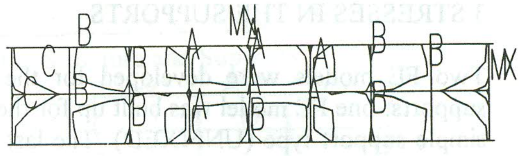

Figure 5-UNFIXED Model Stresses - Base Plate at local z=0

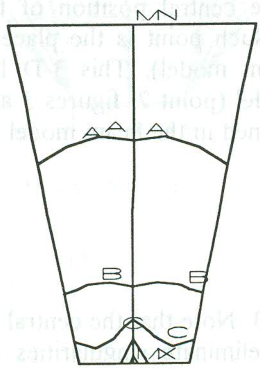

Figure 6-UNFIXED Model Stresses Stiffener at local y = 4.250m

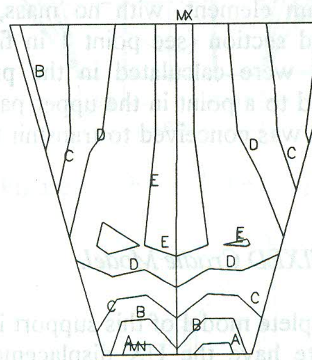

Figure 7-UNFIXED Model Stresses Stiffener at local y = 0

3.2 *FIXED Cradle Model.*

Figure 8 shows the complete support model. All the nodes in the base plate were restrained in the three directions. Other boundary conditions were applied to simulate the connections mentioned before. As in the analyses before, the loads were applied in three different load steps. The first load represents the dead weigh load as already mentioned. The second load step and also the third one represent, respectively, the loads (beam reactions) in the transversal Y and X directions. The stress distribution obtained from the analyses were combined. The final stress results is represented in Figures 9 to 12. Table 1 summarizes the stress values corresponding to the stress intensities levels shown in these figures.

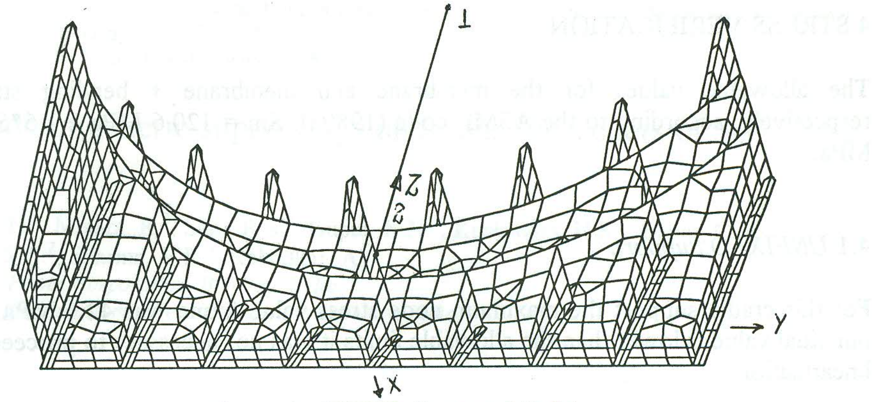

Figure 8 - FIXED Support Model

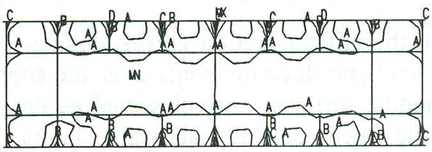

Figure 9-FIXED Support
Base Plate at local z = 0

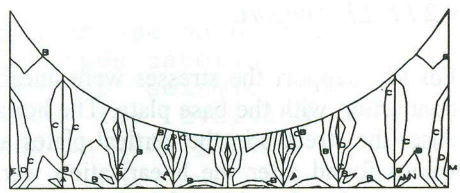

Figure 10 - FIXED Support
Plate at local x = 0.365m

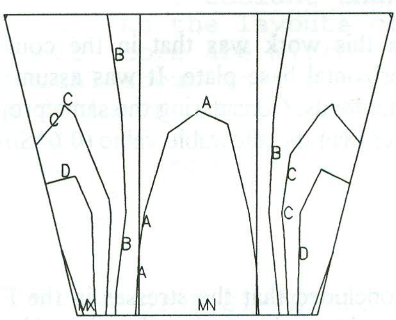

Figure.11-FIXED Support
Plate at local x= 0

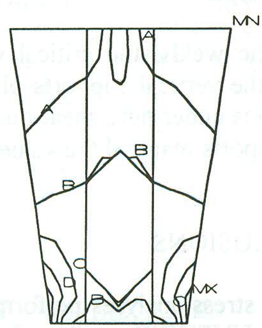

Figure 12-FIXED Support
Plate at local y = 4.250m

Table 1 - Values for the stress intensities levels

Figure #	Values of the Stress Intensities Level (MPa)				
	A	B	C	D	E
4, 5 e 6	8.5	16.0	23.6	31.2	38.8
7	10.1	13.3	16.3	19.4	22.5
9	79.7	214.7	352.8	490.8	628.9
10	41.3	69.2	97.1	124.9	152.8
11	91.5	226.2	361.0	495.7	630.5
12	56.9	137.5	218.1	298.7	379.3

4 STRESS VERIFICATION

The allowable values for the membrane and membrane + bending stresses are, respectively, according to the ASME code (1989a): Sm = 120.6 MPa e 1.5*Sm = 180.9 MPa.

4.1 *UNFIXED Support.*

For this cradle support the maximum node stress value found was 42.6 MPa. Since this punctual value is lower than the allowable limits it was not necessary to proceed the stress linearization.

4.2 *FIXED Support.*

For this support the stresses were linearized in the critical section corresponding to the connection with the base plate. The horizontal loads produce moments over the support. Also, the stresses in the vertical plates at its middle surfaces were classified as Pm. The values found after the linearization were a little lower than the allowable limits. This suggests that the width of the base must be extended.

4.3 *Weldments.*

Among the welds, the critical weld verified in this work was that in the connection between the vertical supports plates and the horizontal base plate. It was assumed that such weld is under pure shear due to the horizontal loads. Considering the same properties of the supports material the value found was lower than the allowable value (0.6*Sm).

5 CONCLUSIONS

From the stress analyses performed, it can be concluded that the stresses in the FIXED and in the UNFIXED cradles of the supported cylinder are lower than the allowable limits. Therefore, from the stress verification point of view, it is possible to qualify these supports.

REFERENCES

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

ASME 1989b. *ASME Boiler and Pressure Vessel Code, Section III, Division 1, Appendices.* New York, NY.

De Salvo, G. J. & R. W. Gorman 1989. *ANSYS Engineering Analyses System, User's Manual for revision 4.4a.* S.A.S.I., Houston, PA