ASPECTS OF DESIGN AND STRESS CLASSIFICATION OF A PWR SUPPORT STRUCTURE

Julio Ricardo B. Cruz, Miguel Mattar Neto, Carlos Alexandre de J. Miranda, and Luciano M. Bezerra
Structural Mechanics Division
COPES/IPEN-CNEN/SP
São Paulo, Brazil

ABSTRACT

This paper presents the stress analysis of a support structure of a nuclear PWR vessel. Different geometries and thermal boundary conditions are considered to achieve a viable design of the support structure. FE analyses are performed with the ANSYS program. For the stress verification, the ASME Section III requirements are applied. This article also presents: 1) a discussion on the stress classification and linearization, and 2) the jurisdictional boundary between the ASME Subsection NB (Class 1 Components) and Subsection NF (Components Supports).

INTRODUCTION

This paper deals with the stress analysis of a cylindrical skirt designed to support the pressure vessel of a PWR research reactor. The skirt is welded on the flange of the cylindrical body of the vessel, as can be seen schematically in Figure 1. Two main loads drive the skirt design: a severe impulsive dynamic load, and a thermal gradient between the inner surface of the vessel (~280 ^{o}C) and the outer surface of the skirt (~40 ^{o}C). The dynamic stresses were obtained by spectral analysis using a simplified FE model composed of beam elements. The high stresses obtained from the dynamic analysis suggested that the skirt should be very thick. However, increasing the thickness of the skirt leads to higher thermal stresses. Therefore, there is a trade off between the geometrical requirements to support the dynamic load and those to accommodate the thermal effects.

In the initial design concept proposed for the support, the thickness of the skirt was constant, as illustrated in Figure 1, and the detail of the welded joint between the vessel flange and the skirt is shown in Figure 2. Furthermore, the complete outside surface of the skirt was originally bathed by water at 40 ^{o}C. Based on this first idealization, several dynamic and thermal stress analyses were undertaken varying: a) the dimensions of the welded connection between the skirt and the pressure vessel, b) the thickness of the skirt, and c) the level of the water in contact with the outer surface of the skirt. The results obtained in this preliminary study led to the following conclusions:

a) The modification of the weld dimensions by itself did not eliminate the problem of high thermal stresses in the critical area of the joint between skirt and pressure vessel

b) As expected, reducing the support thickness and putting down the level of the water on the outer surface of the skirt showed to be effective in getting lower thermal stresses.

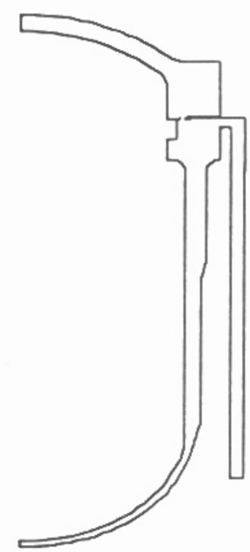


FIGURE 1 - VESSEL AND SKIRT SECTION

From the insights of the preliminary study, a new configuration of the skirt was then proposed. In this new concept (see Figure 3), in order to get lower thermal stresses, the upper region of the skirt was thermally insulated and had its thickness reduced. In

94(22

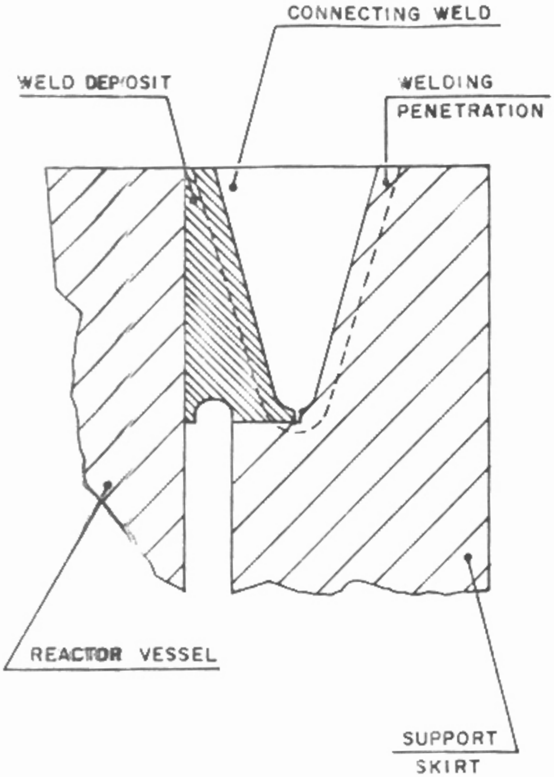


FIGURE 2- WELDED CONNECTION (INITIAL CONCEPTION)

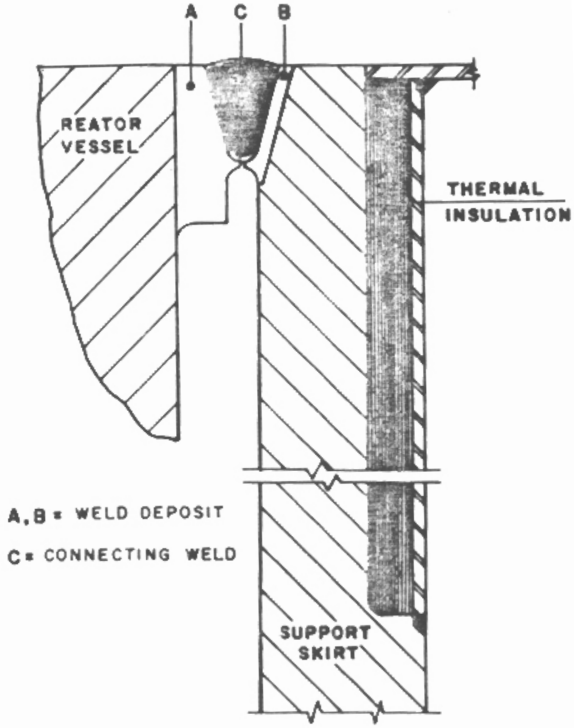


FIGURE 3- NEW CONCEPTION GEOMETRY

addition, some changes were introduced in the weld detail.

We notice that the fabrication process follows the main steps (see Figure 3): (1) a previous machining of the weld deposits A and B; (2) the assemblage of the reactor vessel on the skirt using lugs welded in circumferential locations of the skirt to align the weld edges; and finally, (3) the one side full penetration weld C is executed with care, so that the weld root geometry will be as smooth as possible. This paper will present and discuss the results obtained in the analysis of this new concept of the skirt. It will not describe the dynamic analysis, but will highlight the details of the thermal stress analysis.

FINITE ELEMENT MODEL

The analyses of the skirt were performed using axisymmetric solid finite elements. The same mesh was used either for the thermal analyses and for the stress analysis.

Since the region of interest for the study was confined to the connection between the skirt and the pressure vessel, there was no need to discretize the entire vessel and skirt. The FE mesh is shown in Figures 4 and 5. Figure 5 also indicates the sections AA and BB selected for stress verification.

For the thermal analyses, the isoparametric element with 4 or 3 nodes and 1 degree of freedom (temperature) [1] was used. For the stress analyses, an equivalent structural element with 2 degrees of freedom (2 translations) [1] was used. The vertical straight line in the middle of the head flange (see Figure 4) represents the flanged connection bolts. These bolts were modeled with beam elements [1], on a per radian basis, with equivalent geometric properties to take into account the bolts axisymmetric distribution. The bolt preload was obtained adjusting the initial strain of the equivalent beam element in an iterative process. The contact surface between the head flange and the flange of the cylindrical body of the pressure vessel was represented by gap elements.

We observe that the actual surface in the weld backside will not be as smooth as modeled in Figure 5. However, the FE model adopted is adequate since stress concentration effects are beyond the scope of this paper.

ANALYSES RESULTS

The steady-state thermal analyses considered the heat exchange in the inner surfaces of the pressure vessel with a bulk temperature of approximately 280 °C and in the outside surface of the skirt with a bulk temperature of 40 °C. From previous analyses, the heat exchange by radiation between the outer surface of the vessel and the inner surface of the skirt was found to be insignificant and was not taken into account. Finally, the other surfaces of the model were considered adiabatic.

Regarding the stress analysis, the following loads were considered:

a) Nodal temperature distribution
b) Internal operating pressure of the PWR vessel
c) Bolt preload
d) Dead weight
e) Axial force on the thickness of the vessel (this force simulates the effect of the pressure acting on the bottom of the vessel).

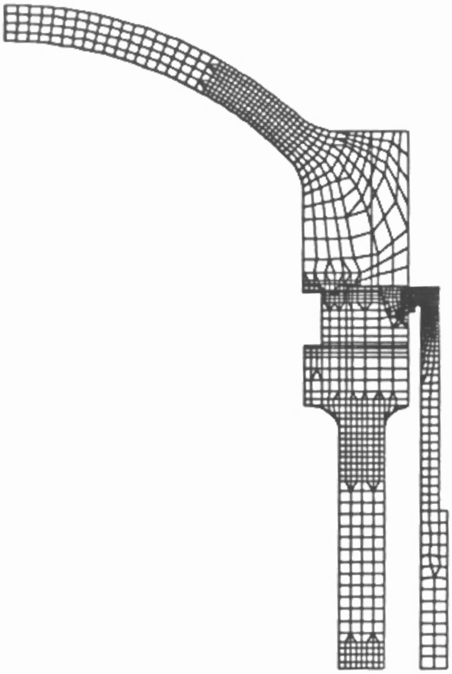


FIGURE 4- FINITE ELEMENT MESH

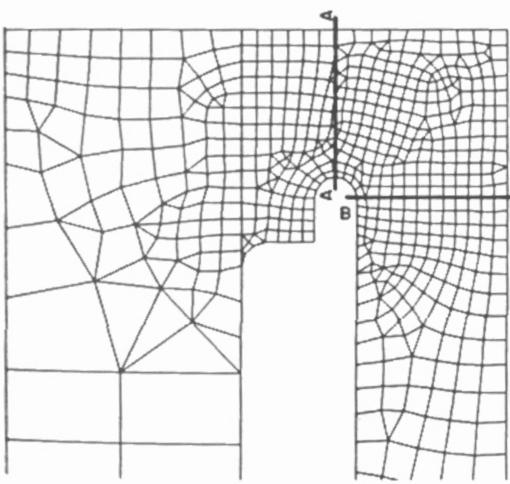


FIGURE 5- ZOOM AT CONNECTION BETWEEN VESSEL AND SKIRT

For the boundary conditions, besides the proper constraints considered on the axisymmetric line of the model, zero axial displacements were imposed at the bottom of the skirt.

The stress results obtained in the analysis of the new conception of the skirt are presented in Table 1. In that table, the relation L/T = 15.7. Four different cases are considered varying the length and/or the thickness of the insulated part of the skirt. For each case the maximum stress intensity value (membrane + bending) in sections AA and BB (shown in Figure 5) are presented.

The two first columns of the table show the stresses due to thermal and mechanical loads, while the next two columns present the stresses coming from the mechanical loads only. The membrane + bending stresses were obtained through the stress linearization procedure of the ANSYS program [1]. This procedure will be discussed in the "Stress Linearization" section of this paper.

The results from Cases 1, 2 and 3 show the influence of the insulated length on the stress level. Comparing the results from Cases 1 and 4 one can see how the thickness of the skirt affects the stresses. The temperature and total SI stress distributions corresponding to Case 2 are shown in Figures 6 and 7, respectively.

STRESS VERIFICATION BASED ON ASME SECTION III

Before making the stress verification, it is necessary to define the jurisdictional boundary between the pressure retaining component (the reactor pressure vessel) and its support. According to subparagraph NB-1132.2 [2], the weld deposit and the connecting weld (see Figure 3) shall conform to Subsection NB. Beyond the connecting weld, the rules of Subsection NF [3] shall apply.

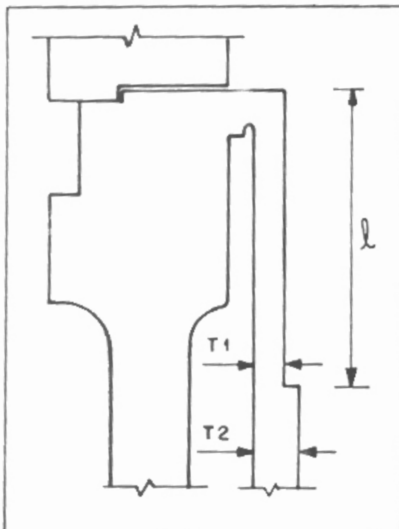
For those sections that goes from the vessel up to the connecting weld, subsection NB (subparagraph NB-3222.2, Level A Service Limits) requires that the following limit be satisfied:

$$P_L + P_b + Q < 3S_m \tag{1}$$

where, $P_L + P_b + Q$ is "derived from the highest value at any point across the thickness of a section of the general or local primary membrane stress, plus primary bending stress plus secondary stress, produced by the specified service pressure and other specified mechanical loads and by general thermal effects associated with normal Service Condition. The allowable value of the maximum range of this stress intensity is $3S_m$" [2]. It should be noted that the limit is applicable to the stress range during the normal operation life of the reactor. Therefore, it would not had been necessary to include the dead weight load in checking this limit, but its influence is irrelevant.

For the material used, $3S_m$ = 481.5 MPa. Therefore, considering normal operating conditions, the stress results obtained in Cases 1, 2 and 4 (see Table 1 for the results related to section AA) comply with the limit expressed in Equation 1. As already mentioned, the design of the vessel skirt was primarily guided by the need to withstand the dynamic and thermal stresses at the same time. As the severe dynamic load is associated with an emergency condition, this implies that primary stress limits have to be verified. Although this paper does not present the verification of the dynamic stresses, it can be asserted that the skirt design based on Cases 1 and 4 of Table 1 turned out to be not viable [4]; Case 1 does not comply with the ASME code limits due to high stresses coming from the dynamic analysis, and Case 4 is not viable from the constructive point of view. Therefore, Case 2 is the only case that meets the requirements for both dynamic and thermal loads.

TABLE 1- MAXIMUM STRESS INTENSITY VALUES (MEMBRANE + BENDING) IN MPa



ANALIZED CASE	$SI_{MÁX.}$ (MEMB + BENDING)			
	THERM.+MECH.		MECH.	
	SEC. AA	SEC. BB	SEC. AA	SEC. BB
CASE 1 ℓ = L T1 = T T2 = 1.5 T	311,4	245,5	41,5	36,5
CASE 2 ℓ = 0.8L T1 = T T2 = 1.5 T	363,8	287,1	41,7	36,6
CASE 3 ℓ = 0.5L T1 = T T2 = 1.5 T	500,8	390,8	42,9	37,6
CASE 4 ℓ = L T1 = 1.25 T T2 = 1.5 T	333,4	250,8	49,2	39,8

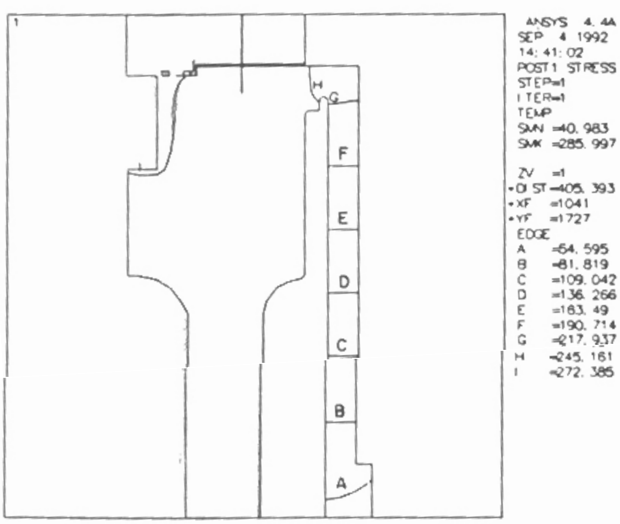


FIGURE 6- TEMPERATURE DISTRIBUTION (CASE 2) IN ^{o}C

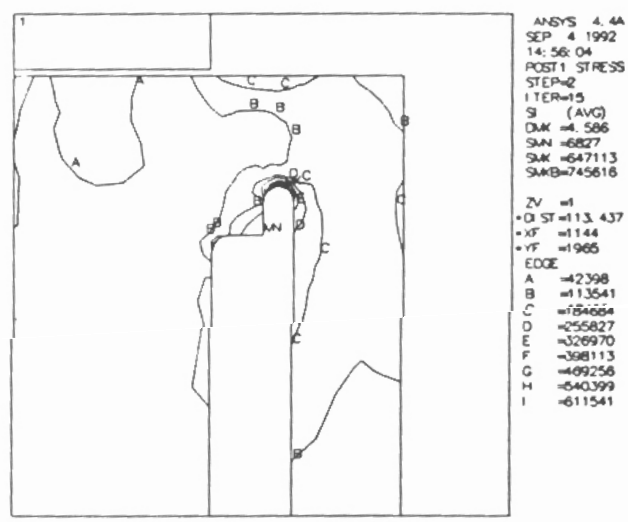


FIGURE 7- STRESS INTENSITY DISTRIBUTION (CASE 2) IN KPa

STRESS LINEARIZATION

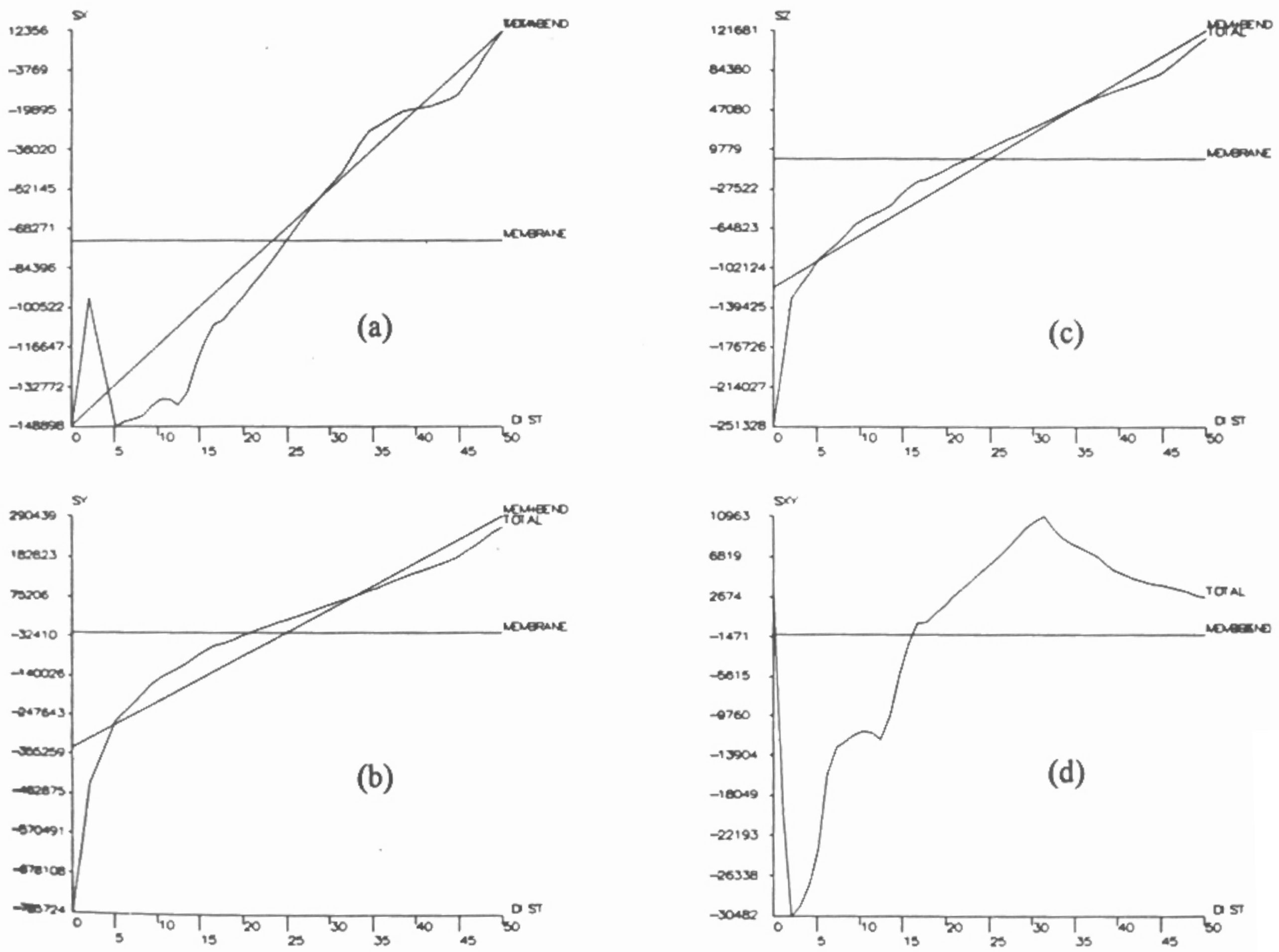
The way of relating FE stress distributions to the ASME failure criteria which impose limits on primary and primary plus secondary stresses is a controversial matter. A recent paper [5] presents a compendium report bringing the opinions of a team of experts on the subject and giving some recommendations to provide assistance to the design-analysis community. However, there are yet several open issues. Some of them involve the linearization process to obtain the membrane and bending stresses.

One questionable issue is related to the choice of which stress components are to be linearized. In 2D axisymmetric analysis there are four stress components: three normal components (hoop, meridional and radial) and one shear component. Regarding the hoop and meridional stresses, there is no doubt that these stresses have to be linearized. The problem involves shear and radial stress linearization. Reference [5] resumes the arguments in favor of and against the linearization of them.

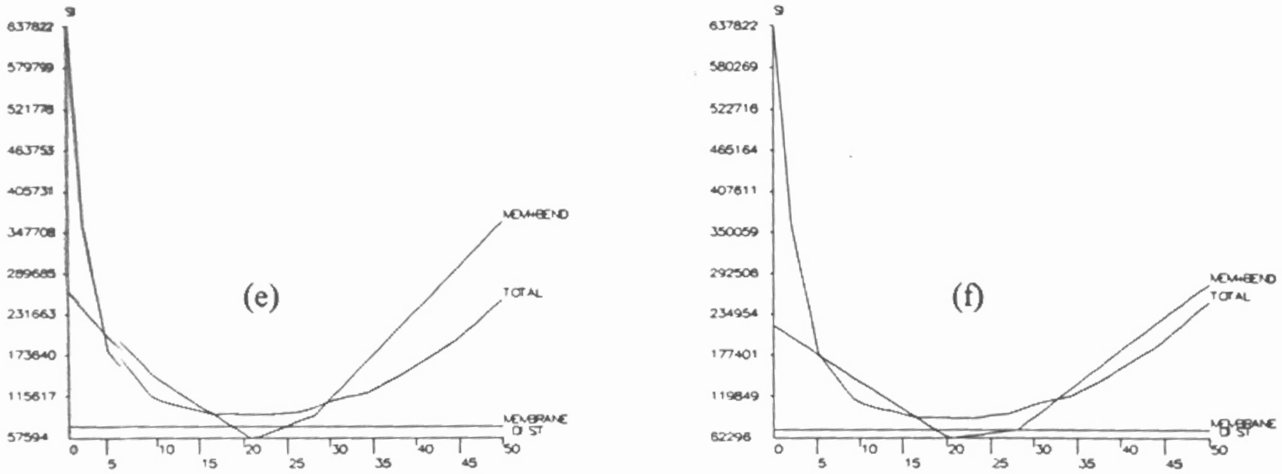
The approach adopted by the ANSYS program considers the linearization of hoop (SZ) and meridional (SY) normal stresses and uses the average shear stress (SXY). Concerning the radial normal stress (through-thickness stress component - SX), the program has two options: (a) to neglect the bending contribution and, therefore, membrane plus bending stress is equal to membrane stress; (b) to assume zero peak stresses on the inner and outer surface points, which means that membrane plus bending stress is equal to total stress at these points.

To illustrate the ANSYS linearization procedure, Figures 8(a) to 8(f) show the stress linearization results on section AA (see Figure 5) relative to Case 2 of Table 1. Figures 8a to 8d refer to the stress components SX, SY, SZ and SXY, respectively. Figures 8e and 8f show the stress intensity SI considering the options (a) and (b), respectively. These figures represent the membrane, membrane plus bending and total stresses.

Comparing the membrane plus bending SI values of Figures 8e and 8f, a sensible difference in the results can be noticed. This difference is directly related to the treatment given to the radial stress (through-thickness stress component) in the linearization process. It is pointed out that the default option of ANSYS is the option (b) mentioned earlier, and in the present case it gives smaller membrane+bending SI values than option (a).



FIGURES 8(a) TO 8(d)- STRESS LINEARIZATION



FIGURES 8(e) TO 8(f)- STRESS LINEARIZATION

CONCLUSIONS

This paper shows the difficulties encountered by the designer in conceiving a PWR skirt support and trying to accommodate, at the same time, the stresses from dynamic and thermal loadings. The present study revealed that a thermal insulation in the critical stress region - close to the welded connection - was necessary. As expected, with the thermal insulation the thermal stresses diminished and it was possible to have the required stiffness characteristics to support the severe dynamic loading.

Another difficulty dealt with in this paper concerns stress linearization. From the discussions in the previous paragraphs, both the hoop and meridional stresses are to be linearized while the linearization of the radial stress (the so called through-thickness stress component) is very questionable and in some situations this issue can not be neglected so easily. In fact, in the cases studied in this paper the thermal loading produces such a stress distribution that the linearization or not of the radial component leads to very different membrane + bending SI values. This does not agree with the comments about linearization of this stress component made in Reference [5], where the authors stated that "for axisymmetric geometries, thermal loads may cause parabolic stress distribution, but their magnitude will be small. Only when linearization is being performed on a line, surface, or plane that is not normal to the inside or outside surface might this stress component be significant". This paper, therefore, claims that linearization or not of certain stress components could make a significant difference and may take an engineer to a non-conservative design.

REFERENCES

[1] DeSalvo, G. J. and Gorman, R. W., 1989, "ANSYS Engineering Analysis System, User's Manual, For Revision 4.4A", S.A.S.I., Houston, PA.

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

[3] ASME, 1989, "ASME Boiler and Pressure Vessel Code, Section III, Division I, Subsection NF, Class 1, Components Supports", New York, NY.

[4] Cruz, J. R. B, Mattar Neto, M. and Miranda, C. A. J., 1992, "Stress Analysis of a PWR Reactor Support", *7th Brazilian Symposium on Piping and Pressure Vessels*, C. A. C. Selke and C. S. Barcellos, ed., ABCM, Florianópolis, SC, Brazil, Vol. I, pp. 379-388, (in Portuguese).

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