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 research PWR pressure vessel: a general description

Miranda, C.A.J.[1], Mattar Neto, M.[1], Oliveira, C.A.[1], Cruz, J.R.B.[1], Bezerra, L.M.[1], Alburquerque, L.B.[2], Assis, G.M.V.[2]

1) CNEN/SP-IPEN - Comissão Nacional de Energia Nuclear, São Paulo, SP, Brazil

2) COPESP - Coordenadoria para Projetos Especiais, São Paulo, SP, Brazil

ABSTRACT. The stress analyses of a small PWR research pressure vessel for prospective construction in Brazil are presented. Actually the stress analyses of the vessel were concluded for the design conditions and for some dynamic postulated loads. Axisymmetric shell, 3-D shell, and beam finite element models were used. The obtained results were compared to the allowable limits established by the Section III of the ASME code.

1 INTRODUCTION

This paper intends to present a general description of the stress analyses of a small PWR type research reactor that is being designed for prospective construction. The stress analyses, considering the design conditions, were already done for the vessel taking into account some impulsive postulated loads, internal pressure, dead weight, bolt-tightening, and seismic loads. The postulated loads are of impulsive type which produces non-axisymmetric loadings in the vessel. Such loads were modeled by Fourier series. A simplified beam model with concentrated masses, a 3-D model, and an axisymmetric harmonic finite element model were employed for the stress analyses. In the simplified beam model the vessel and its internals were taken into account and the postulated impulsive loading was applied. From the results of the analyses, an acceleration level to be applied in the vessel as a static loading was established. The reactions over the vessel due to the connected structures was also acknowledged. The axisymmetric model was used to analyze the vessel components far from the gross structural discontinuities, including its support skirt. The 3-D models were used to analyze the vessel head with its holes and a localized region in its lower part. The nozzles, in the design condition phase, were analyzed with simplified methods as that proposed by Mershon et al. (1987).

2 GENERAL DESCRIPTION AND REQUIREMENTS

Typically a nuclear reactor pressure vessel consists of a cylindrical body with a spherical bottom head, and a removable torispherical upper head which is flanged to the cylindrical body. Body and upper head are tightened by bolt sets. The cylindrical body of the vessel is provided with inlet and outlet nozzles. The upper head has appropriate openings designed for the control rod drive mechanisms and for the in-core instrumentation tubes. According to the current configuration, the vessel is supported by an upper skirt which is born in the vicinity of the cylinder-head junction. This support is different from the usual support configurations where the reactor is generally supported by the nozzles. The vessel design requires several stress analyses to be undertaken in order to fit the ASME Boiler and Pressure Vessel Code, Section III requirements. To verify the cylindrical body, the flanges, the upper torispherical, and the bottom head of the vessel the ASME Section III, Subsection NB (ASME 1989a) was adopted. Figure 1 sketches the vessel geometric configuration. The ANSYS program (De Salvo & Gorman 1992) was used for all the necessary finite element analyses.

3 ANALYSES

3.1 *Cylindrical body of the vessel, flanges and bottom head*

Due to the impulsive dynamic loads postulated for the design phase of the PWR reactor vessel, an harmonic axisymmetric model was adopted and an equivalent static analysis was performed. The cylindrical body, the flanges, and the bottom head of the vessel were modeled with axisymmetric solid shell elements with harmonic loading capacity. Such elements has 4 nodes and 3 degrees of freedom (dof) per node. In order to have the appropriate stiffness of the parts connected to the vessel, the perforated part of the torispherical head and also the skirt were modeled with the same shell elements. The bolts connecting the vessel flanges were modeled with 2D beam elements with axisymmetric equivalent properties.

The postulated loads over the vessel, the ones which were developed in Fourier series, are shown in Figure 1. These loads are represented by R1, R2 and R3 loads and are related, respectively, with the reactor core reactions over the vessel in the flange, and also over the bottom and the control rod mechanisms guides through the torispherical head. R1, R2 and R3 loads were represented by Fourier series with 5 cosines terms; 32 sines terms, and one sine plus one cosine term, respectively.

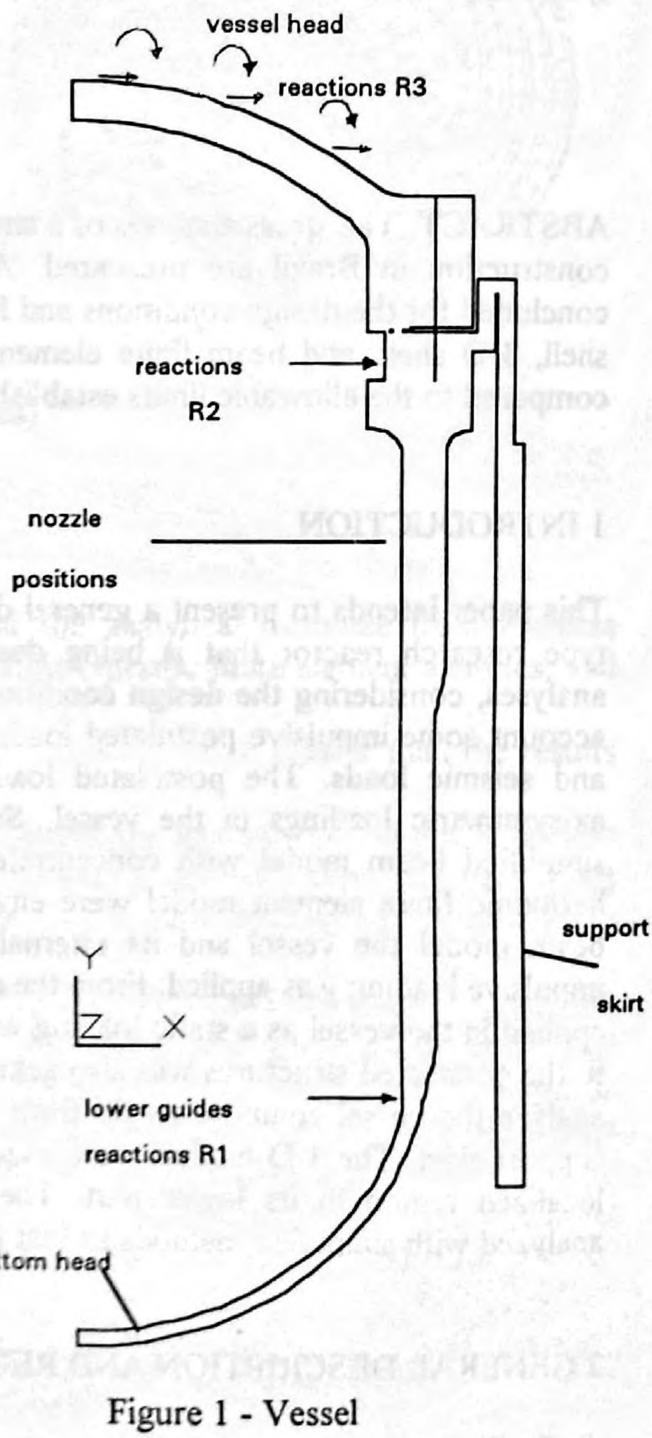

Figure 1 - Vessel

To model the contact between the flanges, a previous axisymmetric analysis with axisymmetric loads only (dead weight, internal pressure, and the bolt-tightening force) was undertaken. For the flange interfaces, gap elements with 2 nodes and 2 dof per node were used. The results of this previous analysis showed which gaps were closed and therefore a "contact area" between the flanges could be estimated. Thus, when the non-axisymmetric (horizontal) loads were applied, the flanges were coupled by the nodes in the "contact area" only.

The skirt nodes at the base had all its dof restrained. The nodes in the symmetry axis had its applied boundary conditions varying according to the MODE parameter command of the ANSYS program. The variation of the MODE parameter happened at each load step. The MODE variation specified the term number of the Fourier series.

To verify the stresses, according to the ASME code, some critical sections along the vessel were picked up. In these sections the stresses were classified and linearized. The ASME Code offers no standard method to linearize and to classify the stresses. However, the research paper by Hechmer & Hollinger (1991) suggests some practical linearizations procedures and alternatives. For the sake of comparing the different linearization procedures, and just for two typical sections, three different linearization methods were adopted in this analysis. According to the recommendations of Hechmer & Hollinger (1991), these methods differ from each other depending on the way the contribution of the stress components in the computation of the stress intensity SI is taken into account. The results show: a) the section axial stress component (SX in ANSYS) has no significant contribution; b) the bending, in the section, due to the shear stresses components, has no physical meaning but, if considered, its contribution for the total section bending can be significant.

3.2 *Upper Head.*

The reactor pressure vessel upper head was analysed with a 3-D finite element model as shown in Figure 2. The model uses mainly quadrilateral shell elements in the perforated central part of the head and 3-D 8-nodes isoparametric solid elements for the flange. The use of shell elements makes the model simpler than if it was made only with solid elements. Using solid element is also too conservative. This is due to the fact that when shell elements are used, the internal pressure is applied at the medium diameter of the shell. If solid were employed, the internal pressure would have been applied at the actual internal diameter of the head. The flange bolts were discretized with 3-D beam elements, and mass elements were used to represent the structures attached on the head.

This model concerns only the perforated part of the vessel head. In order to make the application of the boundary conditions easily and also to provide stiffness to the shell, the flange ring was also included in the model. The stiffness of the ring was reduced because of the bolt holes. In the model, the lower flange was not represented. To represent the interface between the upper flange and the lower one some gap elements were introduced in the model. Therefore, for each load combination, a computer run was done. Note that all these analyses were basically linear, exception is done at the places of the non-linear gaps. The bolt-tightening effect was applied as an initial deformation imposed on the beam elements. An iterative process was used to find a suitable initial deformation that, once applied to the beam elements, led to the actual bolt-tightening. A typical result is presented in Figure 2b for a specific load combination.

From the obtained analysis results, one could say that all of the stresses are smaller than the stated limits, since local effects due to concentrated loads were neglected. In fact, stress limits are exceeded somewhere, but it happens only at loaded nodes, corresponding therefore to localized effects due to the way the loads are applied. More details of this analysis can be seen in De Oliveira (1994).

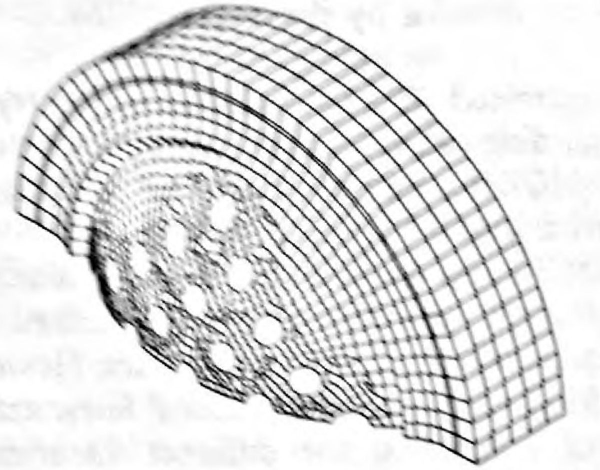

Figure 2.a - Upper head model

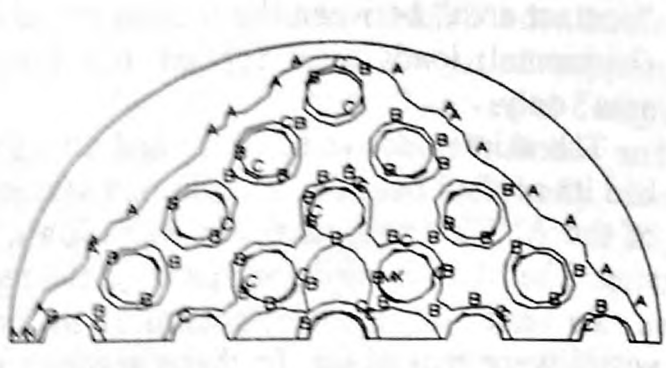

Figure 2b- Upper Head. Membrane SI
A=99.7, B=148.3 and C=196.9 MPa

3.3 *Vessel Support Structure.*

A cylindrical skirt welded on the lower flange of the vessel was designed to support the reactor pressure vessel. The skirt can be seen schematically in Figure 3. Two main aspects drove the skirt design: it has to be thick enough to support the impulsive postulated load, and at the same time it has to be flexible enough not to induce severe thermal effects on the vessel flange. Such thermal effects are due to the thermal gradient between the inner surface of the vessel (~280 ^{o}C) and the outer surface of the skirt (~40 ^{o}C). Different geometries and thermal boundary conditions were considered to achieve a viable design of the support structure.

Figure 3 - Finite Element Mesh

The thermo-stress analyses of the skirt were performed by using an axisymmetric solid finite element model as shown in Figure 3. 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. To reduce the stresses obtained from the dynamic analysis the skirt should be thick. However, increasing the thickness of the skirt led to higher thermal stresses in the vessel flange. Therefore, following ASME (1989b) there was a trade off between the geometrical requirements to support the dynamic load and those to accommodate the thermal effects. Several dynamic and thermal stress analyses were undertaken based on a preliminary configuration of the support skirt and it was concluded that, in order to get lower thermal stresses, the upper region of the skirt needed to be thermally insulated and have its thickness reduced. New cases were then studied to get the best combination of length and thickness of the insulated part of the skirt. The temperature and total SI (Stress Intensity) distributions corresponding to one of the analysed cases are illustrated in Figures 4 and 5 respectively. 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 can be noticed from the results. The studies performed in the skirt support configuration 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. In addition, it was possible to have the required stiffness characteristics to support the severe dynamic postulated loading. More information can be obtained in Cruz et al. (1994).

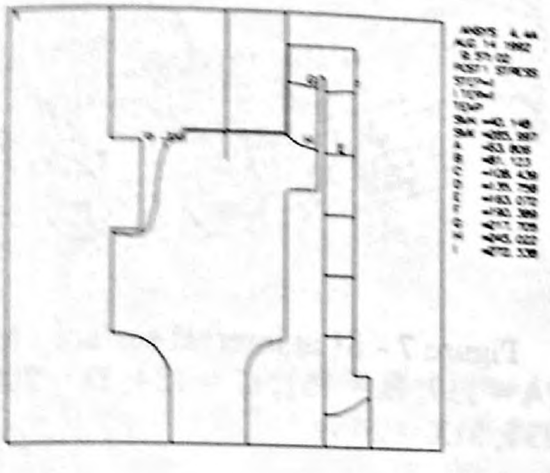

Figure 4 - Temperature distribution (°C)

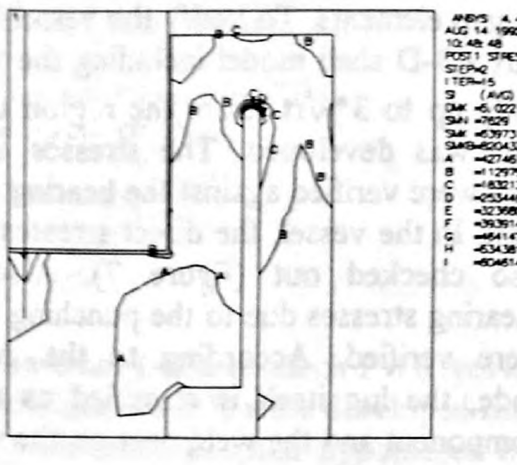

Figure 5 - Stress distribution (Pa)

3.4 *Flange bolts.*

The pressure vessel upper and lower flange rings are joined by means of bolts. These bolts are threaded to the lower flange and, once the upper head is well-positioned on the lower flange, nuts are threaded to the bolts thus closing the vessel. The contact surfaces between the nuts and their corresponding washers are spherical, what facilitates relative displacements between them caused by the bending of the bolts due to the flange rotation under the applied loading. The vessel bolts were firstly dimensioned with the aid of the ASME recommendations (ASME 1989a) for the design pressure load. The bolts were submitted to the effects of the following loads: bolt-tightening, internal pressure, various effects of accelerations, and the impulsive postulated load. For the set of axisymmetric loadings acting on the pressure vessel, the stresses on the bolts were obtained throughout a finite element analyses. For the non-axisymmetric loadings generated by earthquake and postulated dynamic loads, simplified calculations were adopted with beam models. The stresses obtained from the analysis were verified for the specific ASME limits.

3.5 *Nozzles.*

To verify the vessel nozzles, in this phase of the project, the simplified recommendations and methodology presented by Mershon et al (1987) were adopted.

3.6 *Vessel Lower Part - Localized Analysis.*

Between the core barrel lower part and the vessel there are 16 lugs (with square section) welded in the vessel at every 22.5° along the circumference. There is low clearance values between the lugs and the barrel during the hot condition. Under the postulated impulsive loads the barrel can hit the lugs and transmit forces to the vessel. The interaction forces between the barrel and that region of the vessel were obtained from a simplified 3-D model where the barrel was modeled with shell elements and concentrated mass elements were used to simulate the vessel internals.

The interface between the barrel and the vessel in the lugs positions was represented by gap elements. To verify the vessel lower part a 3-D shell model including the vessel body up to $3^*\sqrt{rt}$ from the region of the lugs was developed. The stresses at the lugs were verified against the bearing stress limit. In the vessel, the direct stresses were also checked out (Figure 7). Also the shearing stresses due to the punching effect were verified. According to the ASME code, the lug itself is classified as a NG component and the weldment on the vessel as NB.

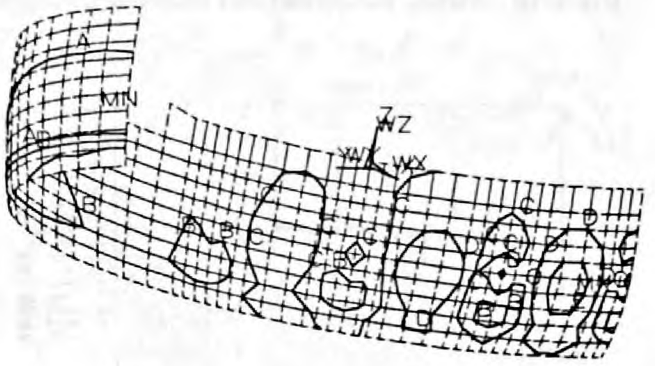

Figure 7 - SI at internal surface - MPa (A = 137; B = 161; C = 184; D = 208; E = 231; MX = 243)

4 CONCLUSIONS

This paper presented a general description of the FE models and analyses undertaken to qualify the pressure vessel of a small research PWR according to the ASME code. It is observed that the design satisfied all the code requirements regarding the "Design Conditions". Moreover, this paper described the main difficult aspects of the project.

REFERENCES

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

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

Cruz, J. R. B., M. Mattar Neto, C. A. J. de Miranda, and L. M. Bezerra 1994. Aspects of design and stress classification of a PWR support structure. PVP - vol. 277.

De Oliveira, C. A. 1992. 3-D Models to Analyse a PWR Pressure Vessel Head under Non-Axisymmetric Loadings. *7TH Brazilian Symposium on Piping and Pressure Vessels.* Florianópolis, SC - Brazil.

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.

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 1991.* R. F. Sammataro et al., ed., ASME. New York, NY. PVP - VOL. 210-2, PP. 181-191.

Mershon, J.L., K. Moktarian, G.V. Ranjan and E.C. Rodabough 1987. Local Stresses in Cylindrical Shells due to External Loadings on Nozzles - Supplement to WRC Bulletin #107 - (Revision 1). New York, Welding Research Council.