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 analyses of the internals of a research PWR vessel: a general overview

Miranda, C.A.J.[1], Mattar Neto, M.[1], Oliveira, C.A.[1], Cruz, J.R.B.[1], Bezerra, L.M.[1], Assis, G.M.V.[2], Aburquerque, L.B.[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 internal structures (or simply Internals) of a research PWR vessel and its analyses are presented. As a first step, some individual models were developed for each structure and modal analyses were performed with some adopted hypotheses on the connections between the structures. In a second step, an integrated 3-D model was developed with superelements using the models of the step before. The superelements were put together with other elements, most of which were gap elements. The behavior of this 3-D model was tested by applying static loads to it. Once the model was working well, a modal spectral analysis was performed, with no active gap. The results have shown that for the seismic loads the structure remains linear. The agreement with the expected behavior was excellent and confirmed that the static analyses considering the isolated structures, and appropriate connections are sufficient to show that the design of the Internals fits the ASME code requirements. All of the analyses were undertaken with the ANSYS program (De Salvo & Gorman, 1989).

1 INTRODUCTION

In the internal structures of the research PWR vessel we analyze here, there has been over 300 gaps, most of them between the fuel elements (FE) themselves and between FE and SHROUDS. Consequently, amplifications may occur due to the impacts between these structures and as a function of the characteristics of the dynamic loads such as the seismic ones.

For a high excitation level the behavior of the Internals can only be well determined if a non-linear analysis using a 3-D model is done.

The scope of this work is to describe the structures and to point out some interesting aspects of its analyses and some of the obtained results. Figure 1 shows, schematically, the internals and their relative positions.

A model was developed for each internal structure, and the models so done were put together to form the 3-D model which was used to analyze the overall structural behavior.

Static and dynamic response spectrum analyses were performed. Appropriate hypotheses on the connections between the internals were used to define the boundary conditions.

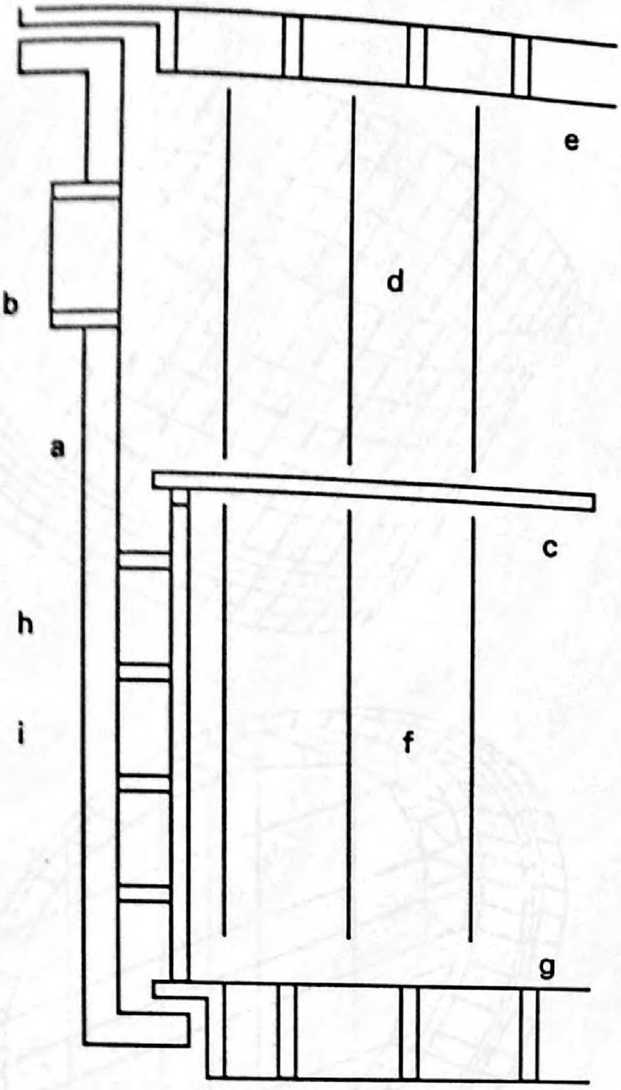
The Reactor Pressure Vessel (RPV) was considered rigid for modeling and analyses purposes.

2 CONNECTIONS

It is supposed that there is a continuity between the Upper Grid (UG) and the Control Rod Guides (CRG). The same is applied to the CRG and Intermediate Plate (IP) connection. The Lower Grid (LG) is connected to the lower flange of the BARREL by pins and bolts, and so continuity is assumed only for displacements. The same is applied to the connections between the Support Plates of the Shrouds (SPS) and the SHROUDS themselves, and between the BARREL and the SPS. There are six spacing grids for each FE assembly. Gap elements are used between the FE grids and SHROUDS, and between FE themselves. Gap elements are also used between the nozzles of the BARREL and the nozzles of the RPV, and between the RPV and the lower end of the BARREL, in the radial direction. The upper flange of the BARREL is supported by the flange of the RPV. The BARREL to RPV to UG connections are made by four locking devices, circumferentially and equally disposed, where the radial displacements are free and the tangential ones are restrained. In addition, the BARREL has four pins, circumferentially positioned, approximately at its medium height, to provide the connection between the BARREL and the IP. Each of these pins has a circumferential null gap and a nominal radial gap of 0.5mm. Pins and springs are used to make the connection between the IP and FE. The lower nozzle of the FE reacts against the LG forming a null gap. There are pins to couple their transverse displacements. The SHROUDS are positioned around the core to provide a guidance to the water flow. There are two basic groups or Types of SHROUDS. They are associated with the respective SPS that are positioned in four levels along the Reactor Core length and in each level they are positioned around the Core, as shown in the Figure 2.e.

3 GENERAL DESCRIPTION and HYPOTHESES

The individual models (used as superelements in the 3-D model) and the 3-D one are described in the following. The CRG were discretized basically with 3-D beam elements. 3-D brick elements with eight nodes were used for discretizing the rings of the upper and lower grids. Four nodes general shell elements were used for modeling the grids, the BARREL, the IP, the SHROUDS and the SPS, whose holes were taken into account. Concentrated mass elements, without rotary inertia, were used in all discretized regions. An element matrix obtained in a separate analysis was used to introduce the stiffness of the FE in the 3-D model. This element matrix does not have a geometric meaning, and is defined by two nodes with six degrees of freedom (DoF) each. The superelement that represents the Core Barrel was divided in two: the Lower Part and the Upper Part. CRG superelements were formed by twenty simple guides and four double ones. Its perforated plates were modeled as rigid regions. For convenience the IP was modeled as two symmetrical parts. FE nozzles were modeled with 3-D beam elements. Each assemblage between the nozzles has its own stiffness matrix given explicitly by the element matrix. The only boundary conditions applied during the superelement definition were the "In-Plane Rotations" for the plane superelements and the Fuel Elements' rotations around their longitudinal axis. No loads were defined. The following modeled superelements are presented in Figure 2: (a) Core Barrel Upper Part, (b) Core Barrel Lower Part, (c) Upper Grid - bottom view, (d) Lower Grid - bottom view, (e) Support Plates of the Shrouds - Type 1 and Type 2 and (f) Intermediate Plate.



a. Core Barrel

b. Nozzle

c. Intermediate Plate

d. Control Rod Guide

e. Upper Grid

f. Fuel Element

g. Lower grid

h. Shrouds

i. Support Plate of the Shrouds

j. Gaps

Figure 1: Relative positions of the Internal Structures

4 ANALYSES OF THE 3-D MODEL

A detailed description of the 3-D model and its analyses can be found in Miranda (1992). This paper is restricted to a discussion on the main aspects involved with the model and analyses.

4.1 *Test Analyses.*

To assure a good behavior of the developed 3-D model, that uses superelements and single elements, some tests had to be done. These tests consisted in static and modal analyses. The analysis of the results showed good agreement with what was expected taking into account the nature of the structures and the applied loads. In order to get a parameter to compare the results, the frequencies of the following individual structures were initially calculated, without considering the influence of the fluid (added mass): FE, CRG, SPS, SHROUDS types 1 and 2.

The applied boundary conditions simulate the connections of the Internals, including the mass contribution of the structures that are supported by other structures. The BARREL was also analyzed, but considering only its upper flange support with two axisymmetric harmonic models: one of them using 2-node shell elements considering "added masses" and the other using 4-node solid elements including the fluid (Miranda 1990).

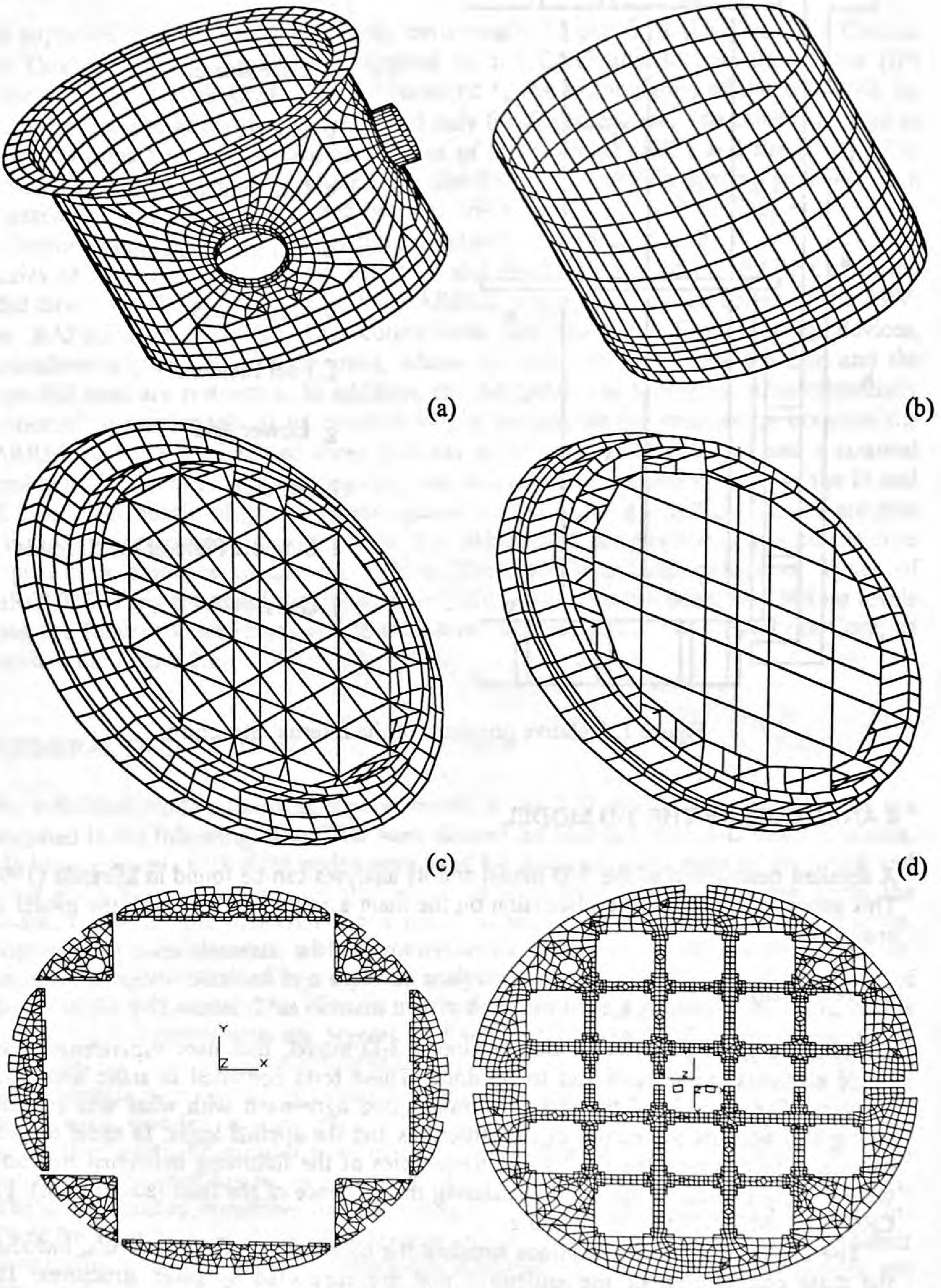


Figure 2: Some of The Modeled Structures/Superelements

For the CRG structure two hypotheses were considered for its lower end: a) only the vertical displacement is free and b) the displacements are free and the rotations restrained. The other structures are considered to be much more rigid than these ones. Some hypotheses were made to take the fluid surrounding the structures into account by simplified analysis, basically using the formulations presented by Blevins (1979). The first frequency of the SPS Type 2, corresponding to a transverse mode, is lower than 33 Hz but close to this limit. Therefore, they can be considered rigid to seismic excitation because the model used to calculate their frequencies is more flexible than they really are (the same applies to the SPS Type 1). For the FE it was found a frequency value of 5.84 Hz (an axial mode associated with its spring in the top) and its first bending mode has a frequency of 27.2 Hz. Since the first real bending mode is less than 33 Hz, a simplified seismic modal spectral analysis was made by using the same model (superelement) and applying the acceleration response spectra given in Figure 3.

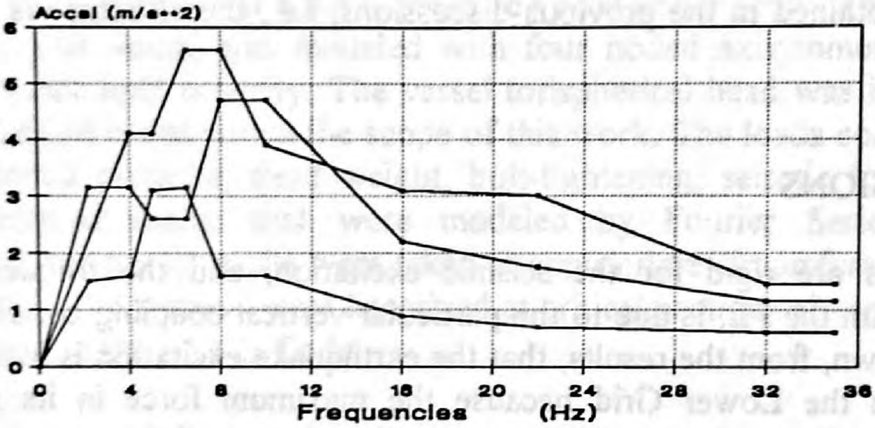


Figure 3: Acceleration Response Spectra

The first two transverse modes (the most significant in the transverse direction) have 36% and 43% of the FE mass associated to them, respectively . In Z direction, as it was expected, the first mode (5.84 Hz) has all the FE mass associated with it. This is due because in this analysis/model the FE in Z direction is connected only by its upper part.

4.2 *Dynamic Analyses. 3-D model.*

For the entire 3-D model the overall frequencies were calculated in two test analyses defining the master DoF at first only in Z direction, and thereafter in X direction, with no masters in the IP, SHROUDS and SPS and with 120 master DoF defined, in each analysis, at the same nodes. The results show good agreement with the expected ones. Some of the frequencies are repeated or are close to those obtained for the isolated structures and some of the calculated modes present a multiplicity of 21, again as expected. So, excluding the FE, all the Internals can be considered rigid for the seismic excitation. Therefore, a dynamic seismic (spectral) analysis was made to calculate the behavior of the structures and to verify if they remain linear under this load.

4.3 *Seismic Analysis. 3-D model.*

The spectra in Figure 3 were applied in the model that has no gaps.

360 master DoF were defined on the same 120 nodes chosen during the tests. The defined masters capture about 90% of the total mass of the 3-D model. To combine the modal results of each load step and the very load steps among themselves, the SRSS - Square Root of the Sum of the Squares - method was chosen. The displacements were investigated before the STRESS calculation phase that should be applied over each expanded mode and each superelement, to verify if the maximums of them in the X and Y directions, in the gap positions, were lower or greater than the existing gaps. If the obtained displacements were greater than the gap values the analysis ought to be changed to a non-linear one with the gap elements activated and the application of the displacement pulse derived from the accelerograms obtained from the seismic acceleration response spectra. If they were smaller than the gap values the spectral analysis should continue to obtain the displacements and stresses inside the superelements for each significant mode and, then, be combined properly. The resultant analysis showed that the maximum displacements are lower than half of the gap values, so the gaps remain open and the structure remains linear during the seismic excitation. These results confirm the conclusion obtained in the previous discussions, i.e., the structure is rigid to the seismic excitation.

5 CONCLUSIONS

The Internals are rigid for the seismic excitation, and the frequency below 33 Hz, associated with the FE, is due to the particular vertical coupling condition assumed for it. It can be shown, from the results, that the earthquake excitation is insufficient to separate the FE from the Lower Grid because the maximum force in its spring, due to the earthquake, is lower than the pre-compressing forces of the spring. Consequently, the design for the seismic loads can be made with equivalent static analyses. In other words, the analyses made with equivalent static loads corresponding to accelerations greater than the maximums given in Figure 3, are sufficient to demonstrate the good design of the internal structures under the seismic loads. The works whose beginning is described here have been continued by doing some other analyses involving other loadings, such as LOCA (Loss of Coolant Accident) ones (de Oliveira & de Noronha 1993).

6 REFERENCES

Miranda, C. A. de J. 1992. A 3-D Model with Superelements for Dynamic Analysis of a PWR Vessel Internals. *ANSYS Technology Conference*. Pittsburgh, PA

Miranda, C. A. de J. 1990. Interação Fluido-estrutura: Estudo Comparativo entre Modelos com Elementos Finitos de Casca com Added Mass e Modelos com Elementos Sólidos e de Fluido 2-D Axissimétricos. *XI Congresso Latino-ibero-americano sobre Métodos Computacionais para Engenharia*

Blevins, R. D. 1979. *Formulas for Natural Frequency and Mode Shape*. van Nostrand Reinhold Co.

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

De Oliveira, C. A. & R. F. de Noronha, R. F. 1993. Analysis of the Interaction between Fuel Elements and Supporting Grid in LOCA Conditions. *12th International Conference on Structural Mechanics in Reactor Technology*. Stuttgart, Germany