A LINEAR ANALYSIS OF A PWR BARREL UNDER A LOCA

S. Marcelino
CNEN/COPESP

ABSTRACT

A three dimensional model using STIF63 [1],[2] shell elements was elaborated to simulate the behavior of a nuclear Pressurized Water Reactor (PWR) core barrel under a time dependent pressure loading due to a hypothetical Loss of Coolant Accident (LOCA) on the primary circuit leg.
The core barrel is represented by a 180° model which accounts to geometric loading and boundary conditions symmetry.
The LOCA loading consists of pressure differences applied to shell elements calculated from the pressure values in outer and inner control volumes around the core barrel, from a previous thermohydraulic analysis. These loadings were computed each millisecond till 500ms and inputed through the PREP6 module.
The model was analyzed with two different versions (4.2B[1] and 4.4A[2]) of the ANSYS Program and several test analyses were performed to evaluate the most appropriate execution option due to software and hardware limitations.
This paper discusses the adequate analyzes for specific large problems including time dependent element pressure loadings.
The displacements and stresses will be analyzed by an adequate post-processing routine.

INTRODUCTION

In this paper we analyze the influence of a sudden loss of pressure inside the core barrel of a nuclear Pressurized Water Reactor (PWR) due to a postulated Loss of Coolant Accident (LOCA) on the cold leg. The illustration of Figure 1 shows some control volumes utilized to describe the termohydraulic analysis from which only those involving the core barrel where computed in the structural analysis. Each curve of Figure 2 contains the differential pressures between the inner and the outer side of the barrel cylinder. Not all volumes are included in this Figure. These pressures were calculated for 0.5s of LOCA and the results were presented for each 0.001s.
A comparison of the Integration Time Step (ITS) calculated trough a detailed Fourier Series Analysis of the loading curve and its evaluation from the natural frequencies of the structure considering "Added Mass" determined the least value of 2.0 E-4s.
The main difficulties found to solve this analysis of LOCA were the size of the model and the largeness of the loading taking into account disk space and CPU time limitations. Our time limitation is about 10 hours of CPU and we are limited to 300,000 blocks of disk space in a VAX 11/785 computer.

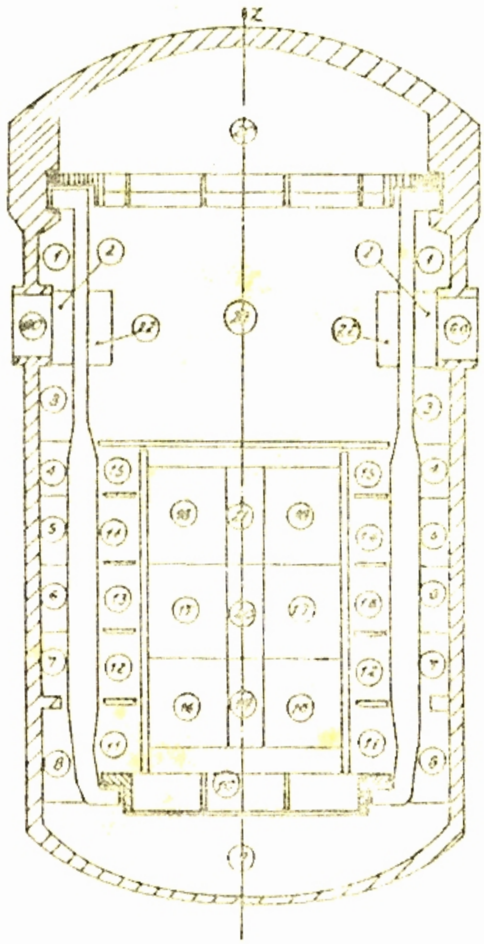

FIGURE 1. CONTROL VOLUME IN THE RPV

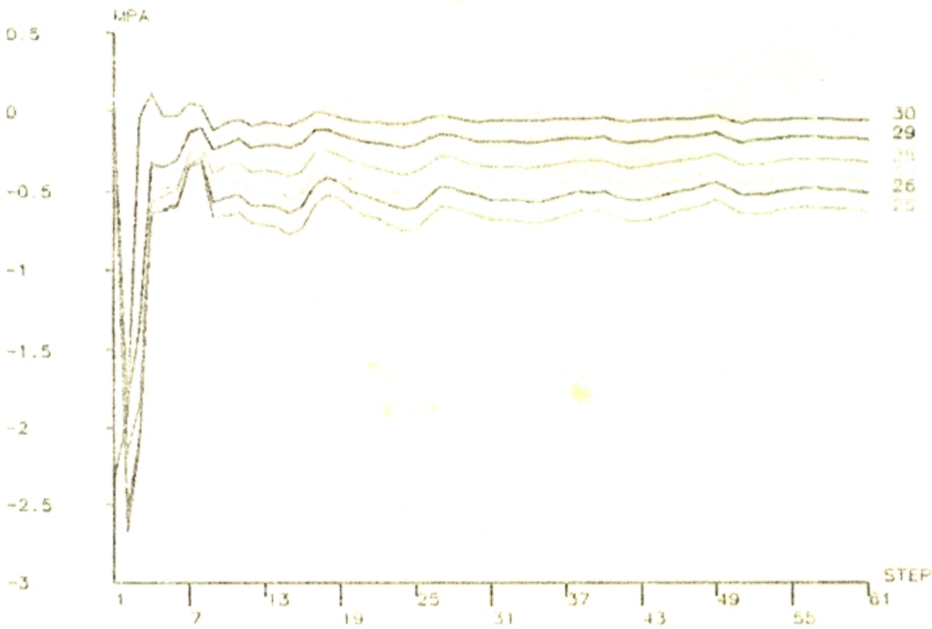

FIGURA 2. TYPICAL CURVES OF PRESSURE DIFFERENCES

MODELLING

The core barrel war represented by cylinder due to its symmetry and modelled by three dimensional shell elements STIF 63. The upper flange was restrained vertically to simulate its supportation by the vessel body.
The four guide supports located at the flange are designed to avoid the relative rotational movement between the barrel and the vessel and were represented by a radial constraint at 90°. The botton of the cylinder has a series of beams to account for the stiffness of the lower grid attached to it.
There were located a series of radial constraints in 90° of the model in the lower part of the cylinder.
The whole model was constructed in order to simulate symmetry in relation to the ZX plane therefore rotation and translation boundary conditions appear along the boundary.
Figure 3 shows a three dimensional view of the model and Figure 4 shows all boundary conditions developed and displayed in a cylindric system.

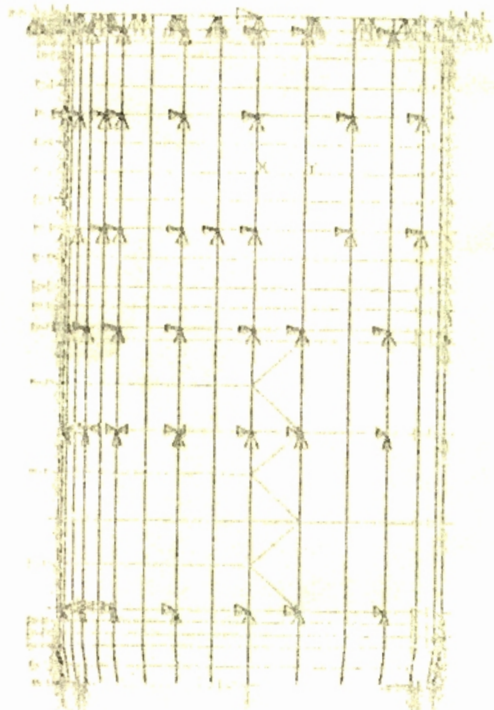

FIGURE 3. CORE BARREL - BOUNDARY CONDITIONS

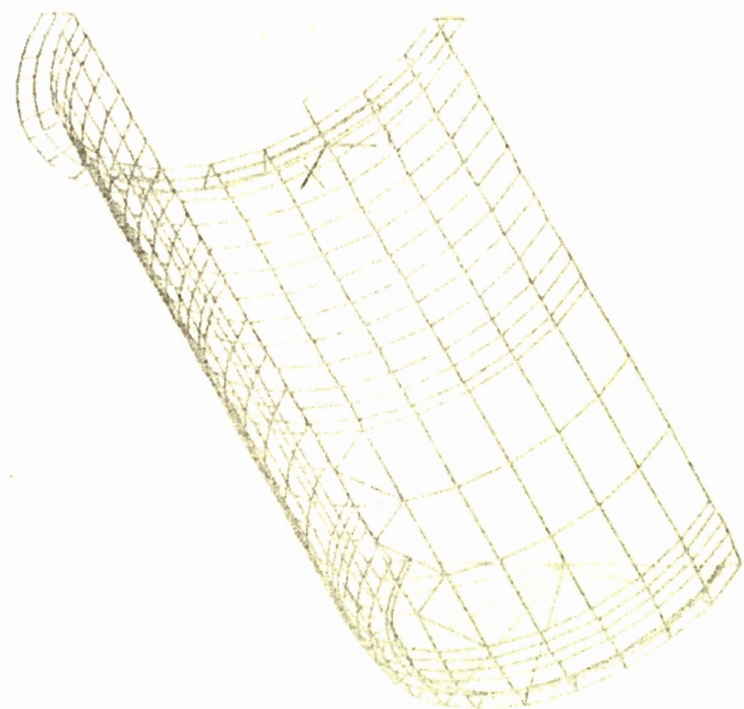

FIGURE 4. CORE BARREL - 3D MODEL

LOADING

Studying the loading curves created by the PREP6 preprocessor for all load steps, we find that significative variations of differential pressures occurs for the first 0.1s shown in Figure 2. After this time the loading shows a slight variation around a constant value responsible for the free vibration. For the analyses itself, from the total of 0.5s of loading duration, we took 0.3s to capture any amplification that could occour during its free vibration period.
The Integration Time Step, and this time interval defined above, determined the number of load steps and iterations used in the items that follow.
In the nonlinear analysis executed in both ANSYS Program versions, there were 300 load steps, with 5 iterations each, defined.
In the nonlinear analyses using superelements, there were used 60 load steps with 25 iterations each.

ANALYSES

Although the reduction of the loading steps had been significative a great deal of analysis stopped due to the reduced CPU time and disk space.
Even though there are no nonlinearities in this analysis, the ANSYS Program doesn't offer a linear dynamic analysis that accepts element pressures as input. The mode superposition linear dynamic analysis requires that the element pressures have a scale factor for its application. So the only aplicable analysis option is the nonlinear dynamics.

Nonlinear Dynamic Analysis Option. A /CHEK analysis demonstrated that a nonlinear dynamic analysis would take at least 40 hours of CPU time. It would force us to use the RESTART procedure due to the files aborted by CPU time expiration, but the several output files (FILE12.DAT) from each execution would take more than the available disk space.

Nonlinear Dynamic Analysis Option Using Substructuring. The Nonlinear Dynamic Analysis with Superelements was the only analysis that allowed us to solve our limitations of CPU time and disk space. But we had to reduce our analysis precision because the Generation Pass has a limit of 60 load steps and in all substructuring the number of MDoF and its locations introduce a certain lack of precision.

RESULTS

The results were obtained using /POST1 postprocessor with the use of MACRO COMMANDS to go through all load steps.
Figure 5 shows the membrane+bending stress intensity(SI) distribution for the inner fiber(BOT) of the shell element along the core barrel. The maximum stress occurs in the second load step, corresponding to 0.005s after the LOCA.
There are two stress concentration regions in the thinner part of the shell, not far from the guide support located in the superior flange.
The maximum values of membrane stress intensity(SI) are distributed along the half cylinder of Figure 6. These stresses occur in the twenty-nineth load step and the maximum nodal stress coincides with the location of the guide suport and around it.

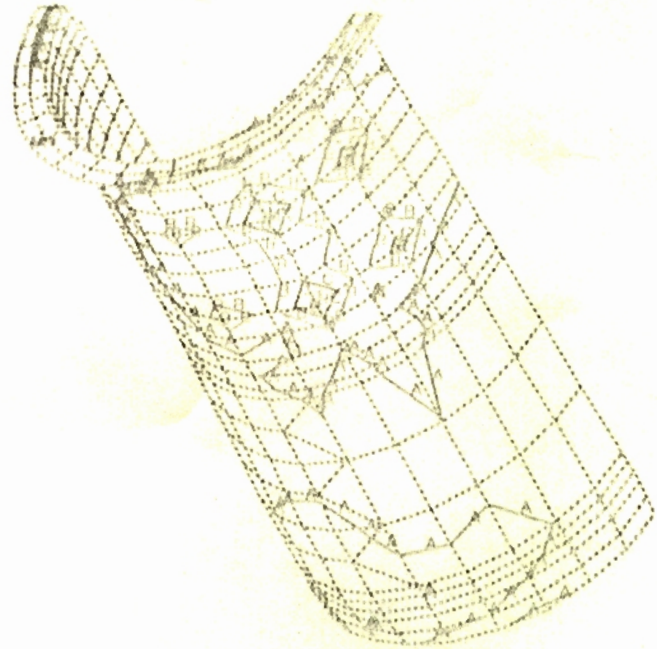

FIGURE 6. PLOT OF THE MAX. MEMBRANE+BENDING - SI

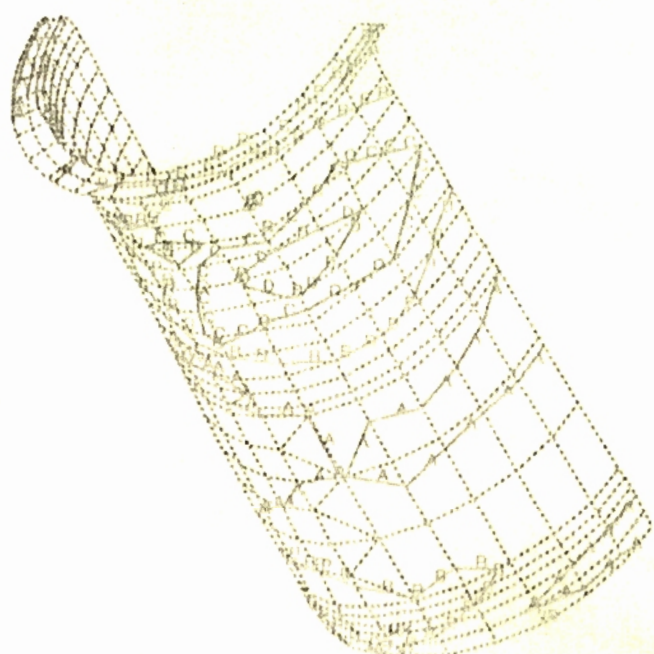

FIGURE 7. PLOT OF THE MAX. MEMBRANE STRESS - SI

CONCLUSIONS

This paper describes some of the difficulties found when large models including element pressure loadings are to be analyzed using ANSYS Program as a Finite Element Modelling(FEM) tool, with certain limitations of CPU time and disk space.
Te clue to the solution was found by the use of substructuring, besides the load step limit of 60.

REFERENCES

[1] Desalvo,G.J., Gorman, R. W., "ANSYS Engineering Analysis System User's Manual," for Revision 4.2B, Swanson Analysis System, Inc., Houston,PA.

[2] Desalvo,G.J., Gorman, R. W., "ANSYS Engineering Analysis System User's Manual," for Revision 4.4A, Swanson Analysis System, Inc., Houston,PA.