SMiRT-12 / K. Kussmaul (Editor)
1993 Elsevier Science Publishers B.V.





# DYNAMIC ANALYSIS OF A PWR CONTROL ROD MECHANISM

S. Marcelino

IPEN - Travessa R,400, Cidade Universitária, 05508 São Paulo, SP - Brasil

## INTRODUCTION

The stresses and displacements of a nuclear pressurized water research reactor control rod mechanism subjected to a postulated impulsive load superimposed by pressure loading, are analysed.

The control rod mechanism was studied for two different masses, one considering the real masses of the reactor vessel and its supports and the other the effective mass of the whole structure. The last was considered more conservative and was used to build the inpulsive load for the analysis presented here.

The dynamic load was calculated in terms of acceleration and integrated twice to produce a displacement loading curve. This loading curve was introduced into the finite element model of the structure and analyzed by the finite element program ANSYS, version 4.4A [1].

Several stress analyses to evaluate the best position of the support and the influence of some geometry changes were performed.

The comparison of the stress results indicated the elimination of most of the different model options and the recommendation of a severe change in the material properties to comply with the ASME CODE[1].

## MODELLING

The control rod mechanism consists of more than twenty rods attached to the reactor vessel head and surrounded by a cylindrical guide structure.

The finite element model used to represent the whole structure consisted of three different parts:

Part 1-A detailed model of one control rod that represents each change in the geometry and in mass along the structure. The model consists of 57 tube elements (STRAIGHT PIPE-STIF21 [2]) distributed as shown in figure 1. The tube elements have equivalent density to consider the mass of the internal structures;

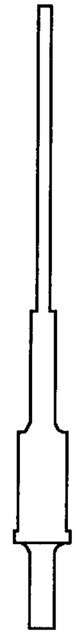

Figure 1: Control Rod Model

Part 2-A simplified model of the whole set of control rods that takes into account its influence on the control rod model described above. It consists of 21 elements 3D beam (BEAM3D-STIF4[2]) with properties that represent all the rods except one;

Part 3-A model using 15 tube elements to represent the guide-structure.

These three parts were fixed at the reactor vessel head and connected to each other through coupling degrees of freedom (CP command [2]), i.e., all coupled degrees of freedom have the same displacement. This connection is done in three levels of the control rod: at the bottom (the reactor vessel head), at an intermediate level and at the top.

**LOADING**

The dynamic load was evaluated considering an effective reduced mass and applied to the node that is fixed to the reactor vessel head and represents the first node of the detailed submodel.

A Fourier analysis of the loading curve and a modal analysis of the structure were performed to evaluate the best integration time step (ITS) used in the analysis.

The loading curve was created by the /PREP6 preprocessor of the structural analysis program ANSYS. The total time of the analysis was 0.53s divided into 4234 steps of 0.00004s (ITS) each.

The loading curve is represented in figure 2.

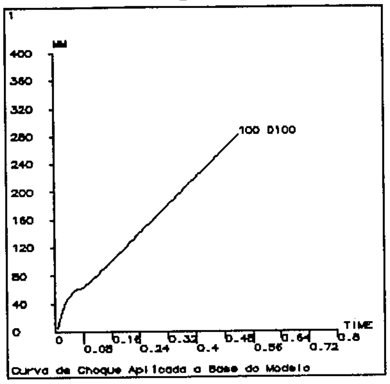


Figure 2 : Displacement Loading Curve

The first five modes of the modal analysis for the support position described in model 1 are shown in figures 3 ,4 and 5.

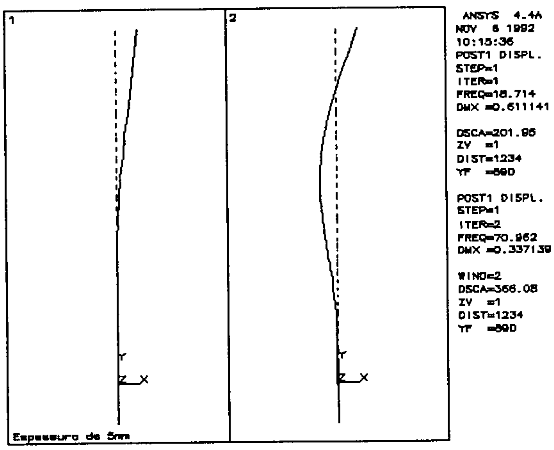


Figure 3:Mode 1 and Mode 2

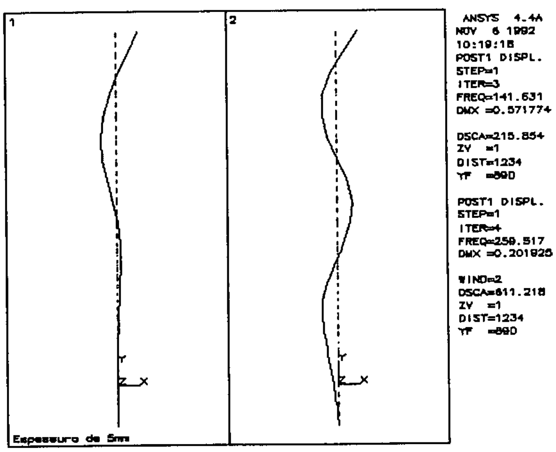


Figure 4:Mode 3 and Mode 4

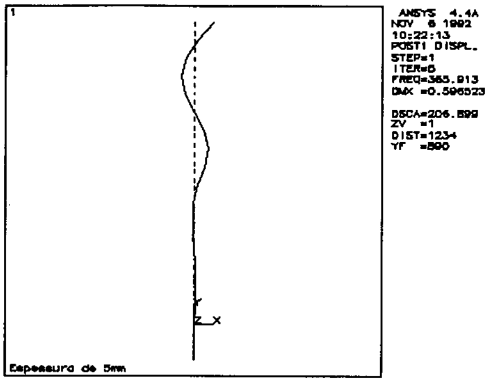


Figure 5: Mode 5

The material properties of the model consider the design temperature of the control rod mechanism.

**ANALYSIS**

Three linear dynamic analysis of the structure was executed considering the following arrangements:

Model 1 - The detailed part 1 alone fixed at the bottom and free at the top.
Model 2 - The whole model, parts 1,2 and 3, fixed at the bottom and at the intermediate level. It represents a rigid connection between the control rods at the intermediate level.
Model 3 - The whole model fixed at the bottom, at the intermediate level and at the top. It considers also a rigid plate connecting the control rods and the guide-structure at the top.

**RESULTS**

The obtained results were analyzed using the /POST26 postprocessor of the ANSYS Code[2]. The largest stress intensity values (SI) in a 0.53s analysis were calculated for each element along the detailed model. These stresses were added to those from the internal pressure and compared with the ASME limits.

The most strained element for each analysis had its values of stress versus time plotted in figures 6,7 and 8.

The relative displacement of the control rod top was calculated subtracting the displacement of the basis. Figures 9 and 10 shows the relative displacement versus time for Model 1 and Model 2 analysis.

The maximum stresses for most of the analyses were concentrated around the two regions of geometry transition.

Due to the dynamic behavior of the structure, it wasn't possible to reduce the stresses to an acceptable value only increasing the thickness of the most affected regions.

From the results of the three Models analysed, the best solution was the Model 1 analysis alone with fixed support at the bottom and free at the top. Even for this solution the material used in the upper part of the control rod had to be changed to one with yield stress at least two times larger.

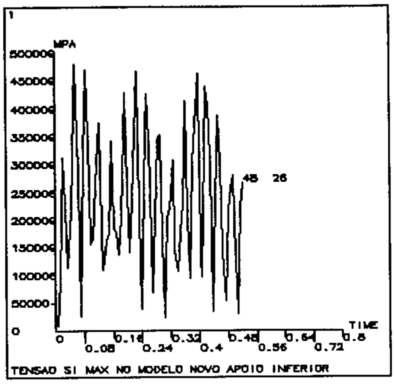


Figure 6: Max. SI Stress for Model 1 Analysis

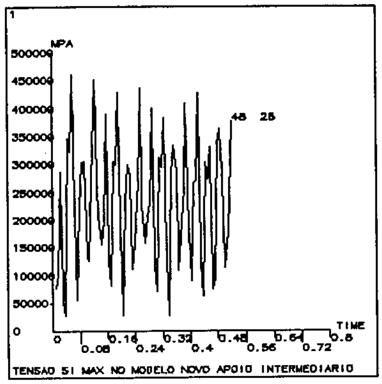


Figure 7: Max.SI Stress for Model 2 Analysis

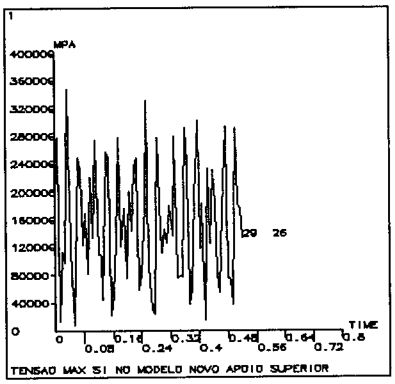


Figure 8: Max.SI Stress for Model 3 Analysis.

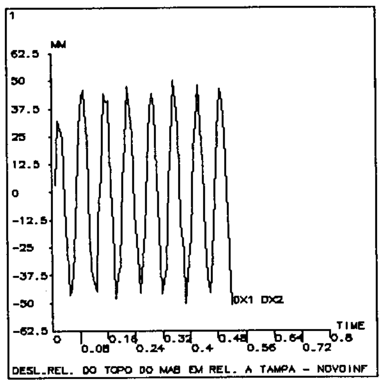


Figure 9: Relative Displ. for Model 1 Analysis.

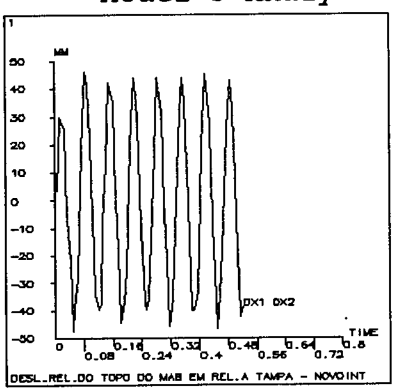


Figure 10: Relative Displ.for Model 2 Analysis.

**REFERENCES**

[1] ASME, Section III, Division I, Subsection NB,1989;

[2] De Salvo, J. S.; ANSYS; Swanson Analysis System; User's Manual, Version 4.4A 1990;