DYNAMIC ANALYSIS OF VVER-440 NUCLEAR POWER PLANT FOR SEISMIC LOADING CONDITIONS AT PAKS

T. Katona[1], S. Ratkaj[1], L. Turi[1], A.L. Halbritter[2] and N.J. Krutzik[2]

[1]Nuclear Power Plant Paks Ltd., Hungary
[2]Siemens AG Power Generation Group (KWU), Offenbach/M., Germany

ABSTRACT

The dynamic analysis of a VVER-440 nuclear power plant located at Paks (Hungary) was performed for seismic loading conditions. A 3D finite element model was employed to idealize the complete structure. Soil-structure interaction was taken into account. The dynamic characteristics of the structure were also determined experimentally using explosive techniques. A good agreement between analytical and experimental procedures was reached. Floor response spectra and internal forces were calculated and evaluated.

1 INTRODUCTION

The design of the four VVER-440 units in operation at PAKS in Hungary did not take into account important external events.

The seismic analysis is included in a series of safety programs in order to determine the actual seismic resistance capability of the plant and, if necessary, the introduction of improvements and modifications. In order to reduce conservatism analytical and experimental investigations were started simultaneously.

This paper discusses the analytical seismic analysis using a complete 3D finite element model. The dynamic characteristics of the structure which were computed analytically and determined experimentally, are compared.

2 DESCRIPTION OF STRUCTURE

The main building is designed as a rectangular, reinforced-concrete/steel structure, 72 m long and 52 m wide. It is supported on a 1.70 m thick foundation slab. On the same foundation there is a condensing tower having a base surface of 42 x 24 m. The total height of the building is approximately 57 m, of which 49 m are above plant grade (Fig. 1).

Up to the 22 m elevation, the main building is subdivided into two structures: The condensing tower (reinforced concrete) and the main steel structure (reactor hall) enclosing the reactor pressure vessel. Above 22 m, the structure of the main building is composed of reinforced concrete walls and floors. Important instruments are located in the galleries between the main building and the turbine hall and in the control room. Both structures have a width of 12 m and are supported by columns on separate foundations. Coupled to the main buildings is the turbine hall which is composed of steel frames.

The design concept of the turbine hall is shown in Figure 1.

3 IDEALIZATION OF COUPLED STRUCTURES [4]

This plant is characterized by structural elements with different stiffness properties giving a complex mixed structure. For this reason an accurate and detailed model was necessary. In order to ensure an adequate treatment of the interaction effects, all the structures described in the previous section were modeled in only one 3D finite element model (main building-turbine hall, galleries.

The finite element model of a unit of the Paks nuclear power plant has 9930 dynamic degrees of freedom. It comprises 1675 nodal points and 2132 trapezoidal elements, 470 triangular elements and 1005 beam elements.

The soil-structure interaction was considered representing the soil by discrete springs and dampers acting at nodes of the idealized foundation slab. This ensured that not only the elastic deformability of the soil foundation to absorb translational, rocking and torsional motion but also its radiation and hysteretic damping would be properly accounted for.

The adequate stiffnesses and dampings were derived from the impedance functions determined using special soil models.

A general view of the finite element model is shown in Figure 2.

The weight of the structure is composed of the weight of the structure, the weight of mechanical and electrical components as well as different live loads. The total weight of the model is approximately 2200 000 kN.

In order to take into account uncertainities related to the soil properties, a parametric study was performed considering three soil conditions: Gmax, Gave and Gmin.

The dynamic structural interaction between two units was analyzed within the framework of a previous investigation which concluded that it can be neglected.

4 SEISMIC EXCITATION

The seismic excitation was represented using three artificial acceleration time histories (one for each translatory direction) acting simultaneously.

The time histories were generated [1] from the USNRC Design Response Spectra.

The maximum acceleration was at 0.30 g and 0.20 g for the horizontal and the vertical direction respectively. These free-field artificial acceleration time histories formed the basis for the excitations determined at the foundation level.

The used excitation time histories have a duration of 15 s and were digitized using a time interval of 0.01 s.

5 RESULTS [5]

The dynamic analyses were conducted using the finite element code STRUDYN [2]. Solution of the eigenvalue problem and calculation of the eigenvectors were performed using the iterative LANCZOS procedure.

The mathematical model was excited simultaneously in the three orthogonal directions. Numerical integration of the equation of motion was performed over a period corresponding to the duration of the time histories.

5.1 Modal analysis

The main dynamic characteristics such as natural frequencies, mode shapes, modal dampings and modal masses were determined for each of the soil conditions.

Table 1 shows the modal weights and the calculated and adopted modal dampings (according to KTA rules) for the first modes for a soil condition Gmax.

Table 1: Modal characteristics

Mode No.	Frequency (Hz)		Modal Weight Dir.1	Dir.2	Dir.3	Modal Damping (%) Calculated	Adopted (KTA)
1	1.84	(1.6-1.9)*	6215.	199595.	6.	6.88	6.88
2	2.12	(1.9-2.1)	325114.	1075967.	40.	15.73	15.00
3	2.38	(2.25-2.45)	1238839.	267577.	822.	17.25	15.00
4	2.60		4733.	9035.	10.	3.29	3.29
5	2.82	(3.0)	99.	100947.	7.	6.28	6.28
6	3.34	(3.6)	95679.	6418.	2621.	8.56	8.56
7	3.46		6765.	295.	70.	3.39	3.39
8	3.51		41288.	2.	121.	4.76	4.76
9	3.57		1663.	251.	272.	3.21	3.21
10	3.60		440.	41.	10.	3.13	3.13
11	3.60		360441	17182.	3543.	5.52	5.52
12	3.72		2078.	1893.	4.	3.37	3.37
13	3.73		32.	212.	46.	3.12	3.12
14	3.79		777.	7186.	3139.	3.51	3.51
15	3.82		56.	13944.	8813.	3.84	3.84
16	3.85		79.	35565.	18956.	4.74	4.74
17	3.94		60616.	248051.	34233.	11.26	11.26
18	3.98		1097.	4003.	299.	3.26	3.26
19	4.04		119.	1014.	1179.	3.19	3.19
20	4.16		4955.	13630.	1568292.	37.25	30.00

* Results obtained experimentally [3]

It was found that the sum of the modal weights for each direction up to 19 Hz almost equals the total actual weight of the structure and that therefore the frequency range relevant to the seismic loading condition was adequately enveloped.

The fundamental frequencies of the main building in the horizontal direction were 2.12 Hz and 2.38 Hz and the vertical direction was 4.16 Hz.

The frequencies of the turbine hall and of the reactor hall start at 1.84 Hz. Typical mode shapes are presented in Figures 3 and 4.

5.2 Floor response spectra

The floor response spectra to be used for the verification of the safety of important components were generated at specific points of the structure and for different damping ratios.

Typical response spectra are presented in Figures 5 through 8. (Reactor building level - 6.50 m and 18.90 m).

In addition also maximum accelerations and displacements were determined at relevant points of the structure.

5.3 Internal forces

Internal forces of some selected structural members were also calculated in order to determine the capability of the structure to withstand seismic events. Due to design concept, special attention was paid to the steel structures enclosing the reactor and the turbine halls.

6 CONCLUSIONS

Despite of the complexity of the analyzed structures, reasonable analytical responses were obtained. The interaction between the different structures and also the interaction soil-structure were considered by the analysis.

Partial results such as the dynamic characteristics of the structure were confirmed by means of experimental techniques (Ref.3) showing the adequacy of the analytical methods also in the case of complex structures. The experimental and analytical results are compared in Tab. 1. More detailed information related to this topic will be provided in the framework of the Post-SMiRT 12 Seminar No. 16.

The results of the calculations (displacements, accelerations, response spectra and internal forces), are used to evaluate the capability of the structure and of the important components to withstand seismic events.

7 REFERENCES

[1]   AGA - Generation of Artificial Spectrum Compatible Time Histories.
        Working-Report Siemens KWU R142/183/81.

[2]   STRUDYN - General Computer Program for Linear-Elastic, Static and Dynamic
        Analysis, User Information Manual,VAX-Version 3/1991.

[3]   Katona, T., Turi, L., Halbritter, A., Krutzik, N.: 10 WCEE - Madrid, July 1992,
        Experimental and Analytical Investigation of PAKS Nuclear Power Plant Building
        Structures.

[4]   PAKS II - Main Building. Mathematical (3D) Model for Dynamic Analysis.
        Working-Report Siemens KWU R324/92/E0017

[5]   PAKS-Main Building. Structural Dynamic Analysis for Seismic Loadings.
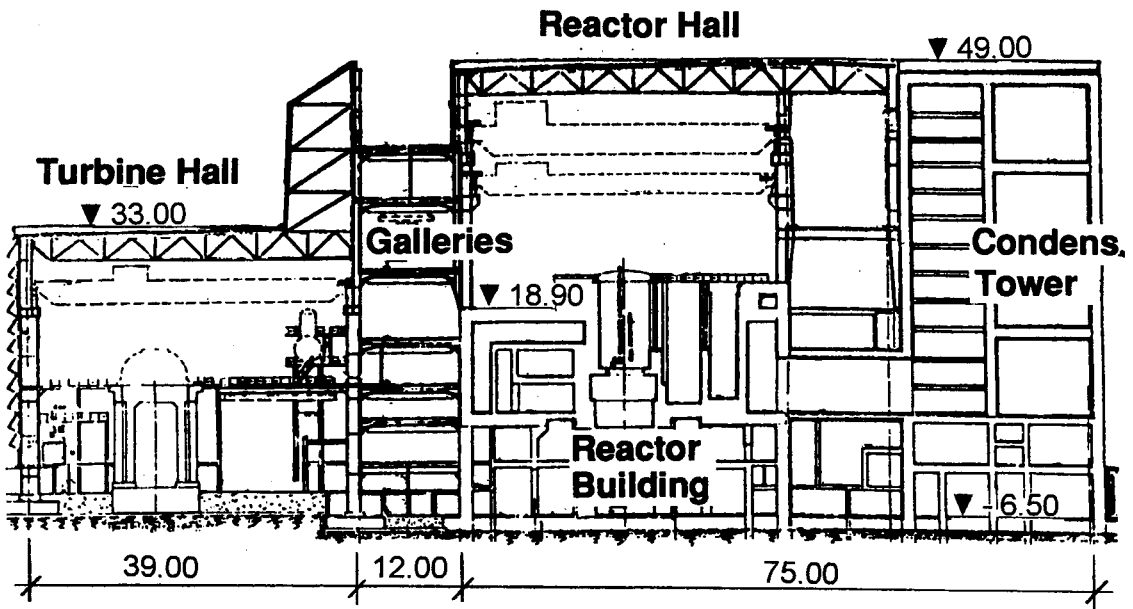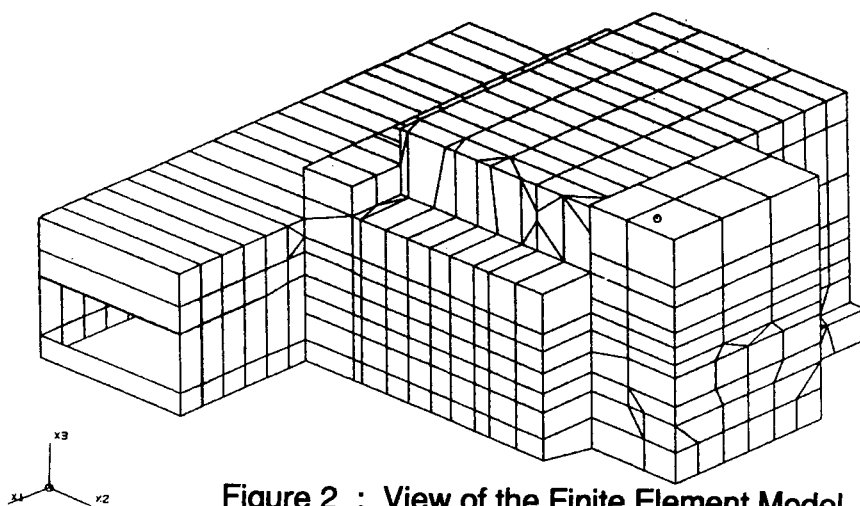   Working-Report Siemens KWU R324/92/E0009

Figure 1: Main structures
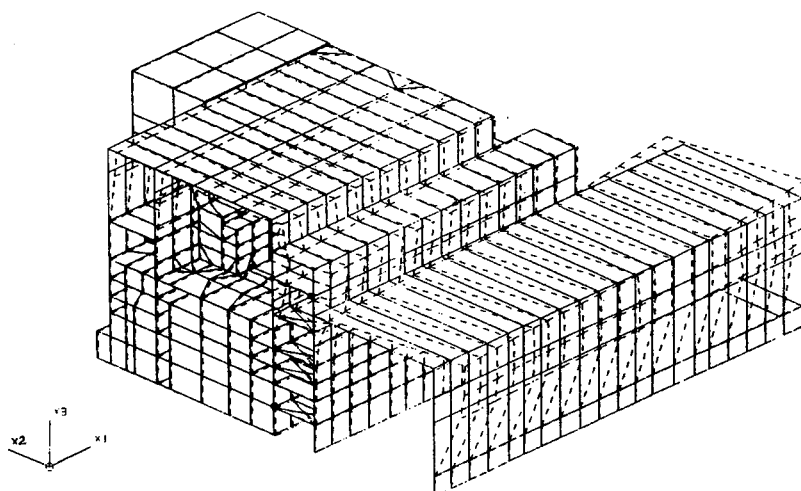
Figure 2 : View of the Finite Element Model

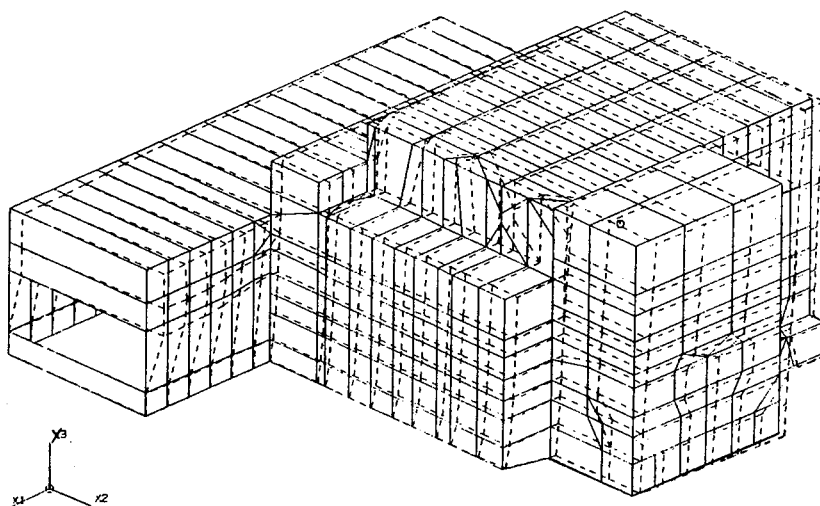Figure 3: Typical Natural Frequency (Reactor Hall and Turbine Hall) F = 1.84 Hz

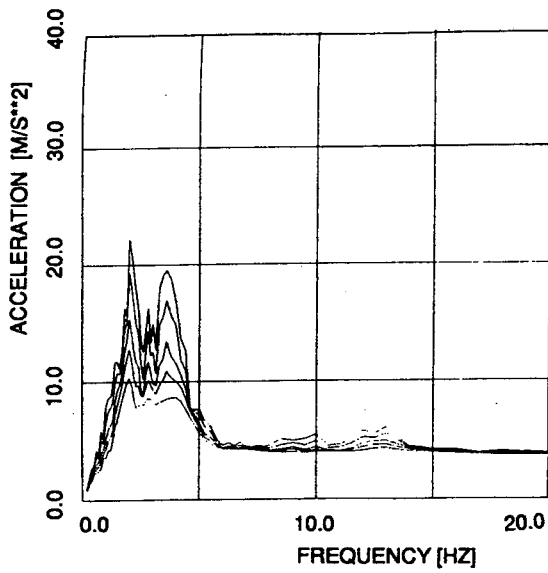Figure 4: Typical Natural Frequency (Global Horizontal) F = 2.38 Hz

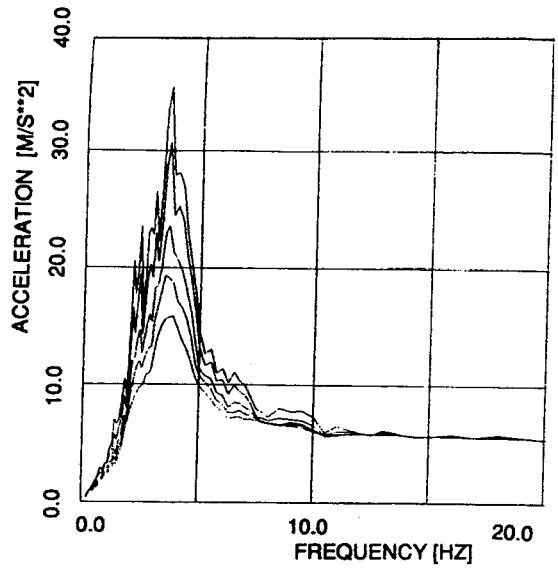Figure 5: Horizontal response spectra
- foundation (level - 6.50 m)

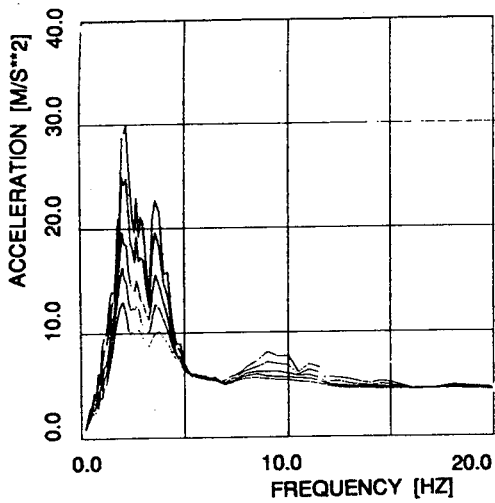Figure 6: Vertical response spectra
- foundation (level -6.50 m)
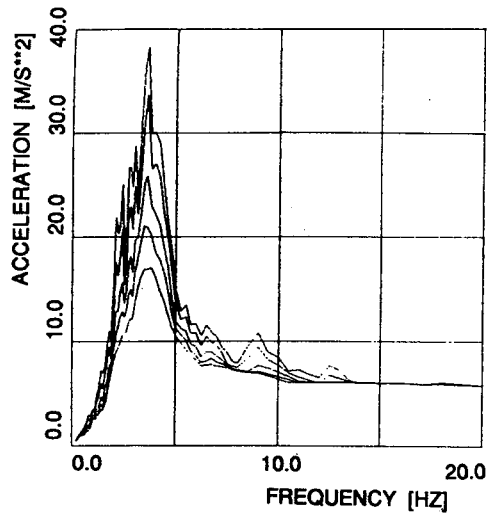
Figure 7: Horizontal response spectra
(level 18.90 m)

Figure 8: Vertical response spectra
(level 18.90 m)

Damping ratio values: 2, 3, 5, 7 and 10 %