30 Mar 2001, 10:30 am - 12:30 pm

3-D Dynamic Analysis for Stability of Geotechnical Structures

Behrouz Gatmiri
École Nationale des Ponts et Chaussées, France/ University of Tehran, Iran

Kamran-Charles Vossoughi
École Centrale Paris, France

Bahar Jenab-Vossoughi
École Nationale des Ponts et Chaussées, France

3D DYNAMIC ANALYSIS FOR STABILITY OF GEOTECHNICAL STRUCTURES

Behrouz GATMIRI
CERMES
Ecole Nationale des Ponts et Chaussées
77455 Marne - la -Vallée, FRANCE
University of Tehran, Tehran, IRAN

Kamran-Charles VOSSOUGHI
Laboratory of Mechanics (MSSMat)
Ecole Centrale Paris (CNRS UMR 8579)
92295 Châtenay Malabry, FRANCE

Bahar JENAB-VOSSOUGHI
CERMES
Ecole Nationale des Ponts et Chaussées
77455 Marne - la -Vallée, FRANCE

ABSTRACT

This paper presents the interests of using a three-dimensional finite element model with an adequate elasto-plastic constitutive law for analyzing problems related to the construction of a complex geotechnical structure. As displacements and ground surface settlements are the key factors, which cause damages to the structure, these parameters have been carefully investigated. The stability of the whole structure is studied as well as the stability of each component. Both static and dynamic analyses (modal and spectral analysis) are performed.

INTRODUCTION

The use of numerical modeling for the geotechnical structures like earth dams, tunnels and retaining structures becomes a common and unavoidable solution for design purposes especially when different loading cases, construction sequences and soil structure interaction effects are to be considered. The numerical methods such as Finite Element are attractive solutions for the analysis of such geotechnical structures regarding to static and/or dynamic loadings. In this paper we try to present the high capacity of FE method for being applied to the numerical modeling of soil structure interaction problems especially under dynamic loadings. The use of adequate constitutive law for representing the soil and the elements which is capable to consider the interaction aspects, is studied. First the project is described and then the FE model and results are presented.

DESCRIPTION OF PROJECT

The Lar dam, an earth dam situated about 120 km from western Tehran (Iran) is constructed in 1980. The water level in the reservoir never reached the expected level. Insufficient site investigations and preliminary studies for establishing the geological profile of the site seem to be the cause of problems related to water escape from the reservoir after pounding of the dam. The dam is constructed on carstic formations and several holes under the dam's structure and its reservoir cause a large amount of water escape. This problem is so important because till now the dam has never functioned as expected. During the last 20 years many solutions are proposed to improve the situation but up to now none of them has given the expected satisfactory results. The ever-increasing population of Tehran especially during the last two decades made it necessary to find a complementary solution in order to use the maximum quantity of water reserved behind the Lar dam. After several studies and regarding to economical aspects, finally a pump station solution is kept. The project consists of the construction of a pump station including a vertical shaft, two tunnels, service tunnels, control building and etc. These tunnels will let the water passed from the Lar dam's reservoir enter Tehran water facilities by using hydraulic pumps located in the middle of the shaft. The site map of the project is shown in Fig. 1. A part of the dam can be seen in the right side and in the middle the circular shaft and the two tunnels could be found.

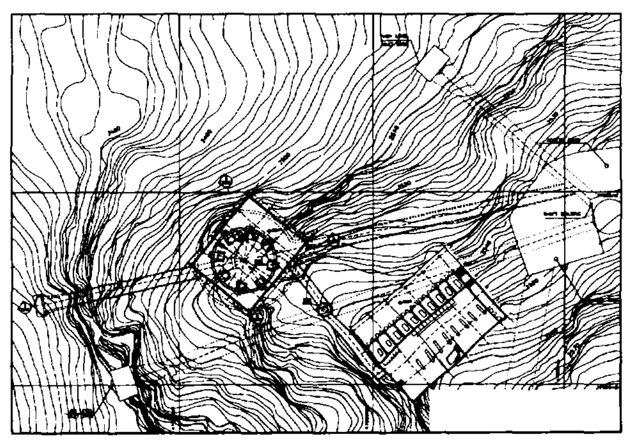

Fig. 1. Site of the project (shaft and two tunnels)

The main shaft is a circular one with a diameter of 16 m and a total depth of 75 m. The pump station and the hydraulic system are located in the middle of the shaft and will pump the water from the reservoir's bottom via a tunnel and will send it into Tehran water facilities using the Kalan tunnel.

The two tunnels have a diameter of 3.5 m. The final diameter after lining is 3 m. The first one is situated in the bottom of the shaft and is designed for water entrance. The second tunnel is situated in the middle of the shaft. Figure 2 presents a cross section of the main shaft and the two tunnels.

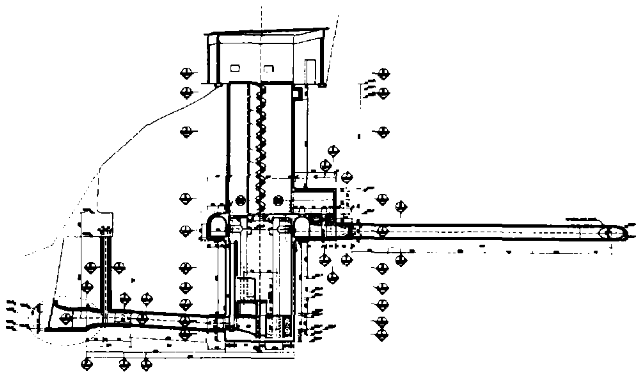

Fig. 2. Main shaft and two tunnels.

FINITE ELEMENT ANALYSIS

Both the static and dynamic behavior of this complex structure using finite element method is examined in this paper. Considering the geological aspects of the region especially the rock, which contains faults, and also the seismic potential of the area, a complete dynamic analysis was performed. For this purpose, a three dimensional dynamic model has been used for studying the behavior of the shaft and tunnels.

Two different analyses, the Modal Analysis and the Spectral Analysis are performed based on the highest spectra observed in the region.

Different loading cases at different steps of construction (excavation, injection, coating) and seasonal changes of the reservoir's water level during construction are considered. For design purposes, different static and dynamic load combinations are taken into consideration.

The first step of simulations was to initialize the at rest stresses on the 3D continuum. This has been done by applying the weight of the soil mass as the only loading. Unit weight of the medium is taken equal to 24 kN/m^3 obtained from laboratory tests. Water level in the reservoir in this case is 2464 m.

The second part consists of the shaft's excavation and cement injection for improving the shaft's walls beginning from the bottom of the shaft to level 2472 m. After injection the lining was executed. The injected zone varies from 4 to 6 m from the shaft's perimeter. For considering a critical state at this step the water level is increased to 2508m. The analysis proved the need for more elaborated soil improvement in this condition. Hence, special considerations should be incorporated. For the hydraulic simulations the water level is at 2508 m which means 2 meters less than the shaft's entrance at 2510m.

Analyses have shown that a two-dimensional plane strain model encounters difficulties with respect to the boundary conditions.

FINITE ELEMNT MODEL

Regarding to the importance of project and the parameters involved in the stability of structure a 3D model has been used. The presumed model consists of a soil mass in pyramidal shape having a 150×150 m base. The model is constructed in such a way that it is in harmony with the mountain shape around the project and is able to consider the excavation effects around the shaft and the tunnels. In the two sides where the mesh is limited to mountains, its dimensions are taken large enough to consider stress variations as representative as possible and are limited to areas where the effects of excavation become negligible. Where the considered soil mass containing the whole structure meets the reservoir, the model follows exactly the mountain's topography. The entire model's height is 125 m having a constant slope of about 45 degrees from the shaft's entrance. At the bottom, this model goes down 40 m beneath the shaft's bottom.

The 3D model is constructed by assembling the 3D sub models. Variations of the shaft's diameter at different levels, the two tunnels with their lining, the improved area using cement injection technique around the shaft, the different soil and rock parameters are integrated in the mesh. In addition to all above-mentioned parameters, other considerations for mesh generation and mesh density are also taken into account. Different element groups are considered to model the soil, the shaft and tunnel's linings, the shaft's foundation and the injected zones.

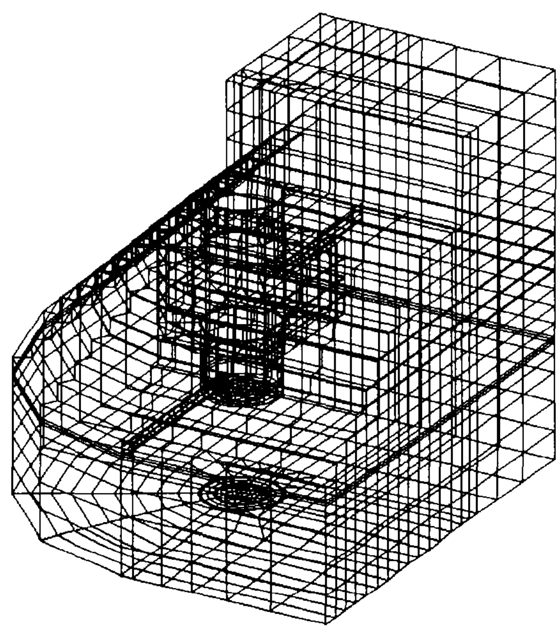

Fig. 3. Main mesh; illustrating the shaft, two tunnels and the injected zone inside the pyramid.

Figure 3 represents the whole mesh. The improved region, the main shaft and tunnels' mesh are illustrated inside the three-dimensional mesh.

Three-dimensional 8 nodes quadrilateral (H8) elements are used to model the soil and rock. Regarding to the problem's shape, three-dimensional 6 node elements (P6) are used where it was necessary. A total number of 6909 elements (4739 H8 and 2170 P6) connected with 6721 nodes are considered in the model. It should be mentioned that because of the large number of degrees of freedom, time of computation has been very long in each case of loading.

All nodes in the bottom of the model are blocked completely in three directions. At the two mountainsides, only vertical displacements are free. Concerning the reservoir side, all nodes are free to move and are submitted to the reservoir's hydraulic loading. The main shaft and tunnels' mesh are presented in Fig.4.

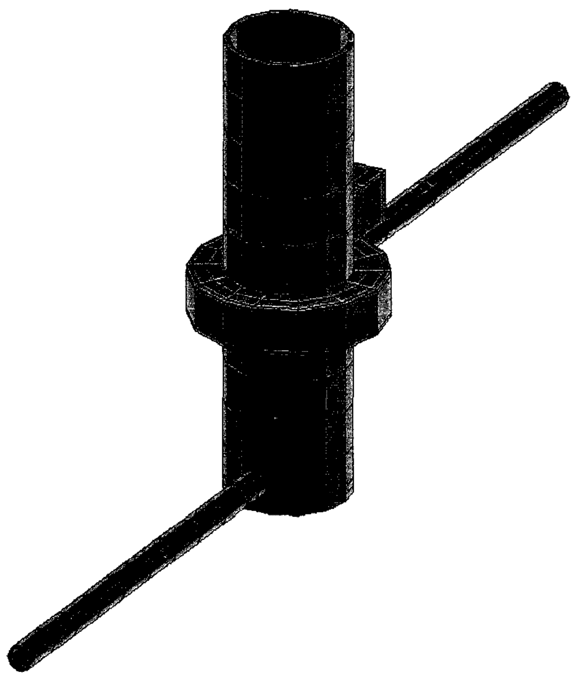

Fig. 4. Main shaft and tunnels' mesh.

Table 1. Material properties.

Soil properties	
E (Young's modulus)	10,000 MPa
ν (Poisson's ratio)	0.3
c (Cohesion)	200 kPa
φ (Friction angle)	35 degrees
ψ (Dilatation angle)	5 degrees
γ (Unit weight)	24 kN/m^3
Concrete properties	
E (Young's modulus)	21000 MPa
ν (Poisson's ratio)	0.2
γ (Unit weight)	25 kN/m^3

MATERIAL PROPERTIES

In order to simplify the model, the soil mass is assumed to contain only one type of material with a non-associated elasto-plastic Mohr-Coulomb constitutive law. The properties of the soil used in the simulations are given in Table 1.

The shaft and tunnel's coatings are considered to be in concrete. Hence, for the lining, a parabolic failure criterion is adopted. It should be mentioned that an injected zone is made during shaft excavation to support the shaft's walls. In some areas rock bolts are put simultaneously. For the injected zone, the same constitutive law is considered with stronger parameters.

STATIC ANALYSIS

The static analysis consists of simulating the shaft during construction, before and after cement injection for soil's improvement around the shaft. The maximum and minimum water levels in shaft are considered to occur in spring and in summer respectively, based on seasonal measures. Hence, hydraulic loads are applied with respect to these parameters.

This type of analysis was also performed for the tunnel's construction. Tunnels drilling and coating execution are modeled using a simplified Confinement-Convergence method (execution sequence simulations). The tunnels both in full of water and in empty conditions are studied with different loading cases. The stability of the whole structure, and each sub-structure is analyzed during these simulations. The results of two critical analyses respectively after shaft and tunnels construction, and with a maximum water level condition are presented in Fig. 8.

DYNAMIC ANALYSIS

Modal Analysis

The main aim of this part is to find the closest p's eigenvalues λ_i to the defined value λ_0 and also the p's associated eigenvectors $\{\phi_i\}$, considering the following equation:

$$[K]\{\phi_i\} = \lambda_i [M]\{\phi_i\}, \qquad i = 1,p \quad (p<n) \qquad (1)$$

in which [K] and [M] are respectively the rigidity and the mass matrices. Also, p is a number taken less than n which is the number of components of the vector $\{\phi_i\}$ or the dimension of [K] and [M] matrices. The sub-space method is used in order to find the eigenvalues. This part's calculations lead to the determination of each eigenvalue and the corresponding displacements. The prescribed precision value for reaching convergence and hence stopping the iterative algorithm is taken equal to 10^{-5}.

Preliminary computational experiences have shown that for a correct modal analysis more than the first five shape modes should be considered. In this way, a complete analysis for which 10 shape modes were needed is performed. Figure 6 presents the third, forth and fifth shape modes found during analysis.

The eigenvalues for each shape modes, the associated frequency and pulsation are presented in Table 2.

Table 2. Shape modes' results.

Mode shape	Eigenvalue	Pulsation (Sec^{-1})	Frequency (Sec^{-1})
1	100.110	10.0055	1.59242
2	205.300	14.3283	2.28042
3	224.744	14.9915	2.38597
4	247.067	15.7184	2.50165
5	269.744	16.4248	2.61409
6	343.653	18.5379	2.95040
7	357.138	18.8981	3.00773
8	390.059	19.7499	3.14329
9	414.917	20.3695	3.24191
10	443.468	21.0587	3.35159

Spectral Analysis

After computing modal responses a spectral analysis was accomplished using design accelerogram of the region. The critical response calculated using this method is presented in Fig.7. The earthquake is applied to the structure for the first series of calculations in the tunnel's direction for the second ones in the direction normal to these tunnels. As the site area is situated in a seismic zone, an adequate spectral acceleration is used for the analysis. The base acceleration for design purposes is assumed to be equal to 0.5 g (g represents the gravity acceleration). The spectral acceleration diagram is shown in Fig. 5. It should be mentioned that the spectrum of earthquake was defined directly for each shape mode.

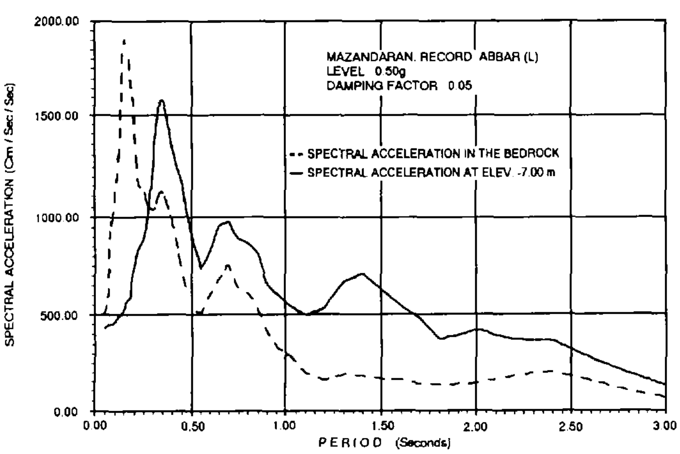

Fig. 5. Spectral acceleration diagram of the region at the level 0.5 g.

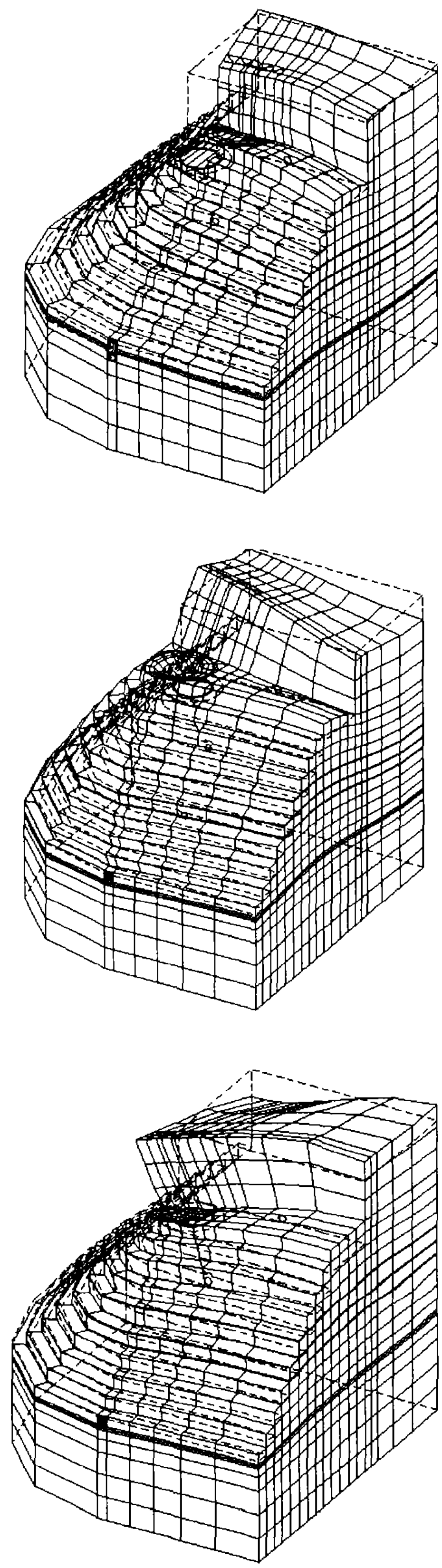

Fig. 6. Shape modes.

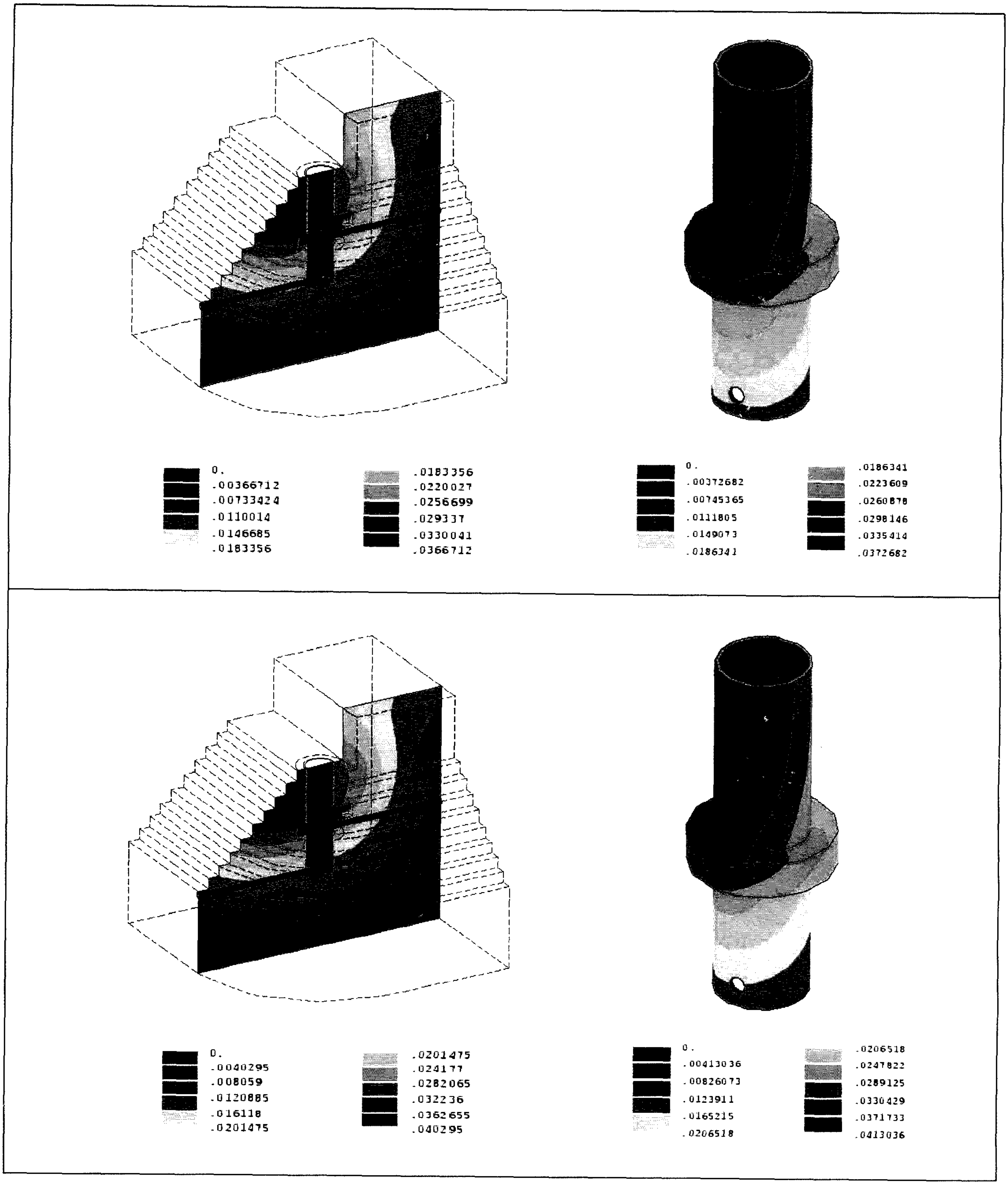

Fig 7. Ground movement results for the earthquake applied in two directions - Displacement contours in meters.

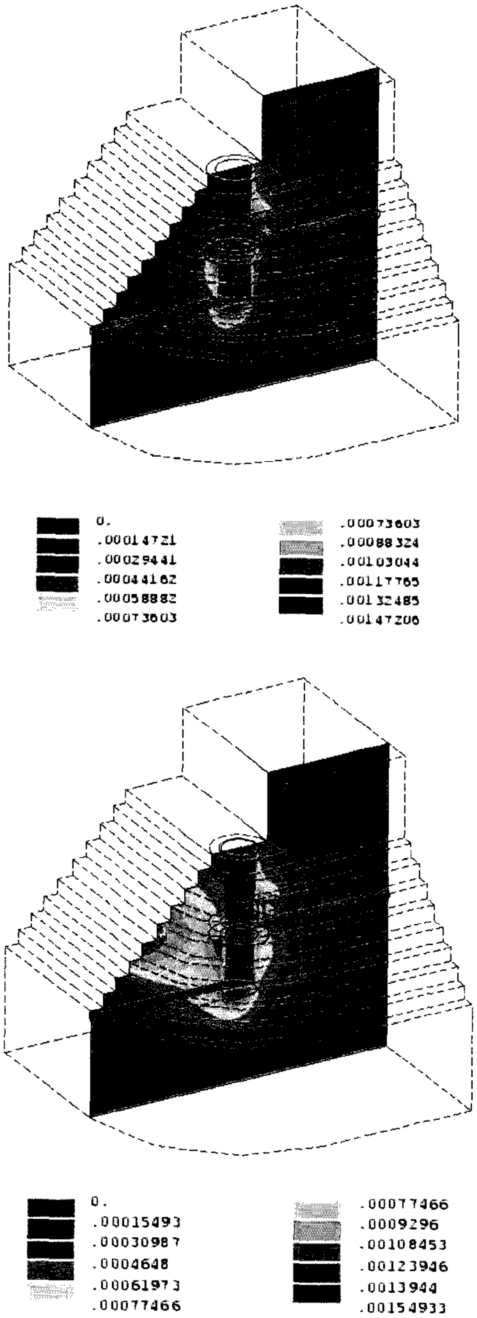

Fig. 8. Ground movement results for static simulations, Up: Water level at 2464m; Down: Water level at 2508m - Displacement contours in meters.

TIME OF PROCESSING AND THE CAPACITY OF MACHINE

Two main difficulties encountered during the three-dimensional analysis are the capacity of machine and processing time. The model used in this analysis contains about 6700 nodes and 6900 three-dimensional 8 and 6 nodes elements. The dynamic analyses were performed linking two hard disks between two machines. The processor used about 600 Mo bytes of the hard disk capacity for the IO processing during analysis.

Regarding to the great number of degrees of freedom in the model, computation times were relatively long. The dynamic modal which consisted of 10 shape modes took about 44 hours on a SUN (SPARK Station20).

CONCLUSION

The performance of the finite element method on a three dimensional model is shown in this study. Analyses shows the high capacity and the potential of the FE method for application in design purposes. Results obtained by the three-dimensional model demonstrate the advantages of such a real model in comparison with a two-dimensional plane strain and/or plane stress model.

On the other hand, the modal analysis shows the importance of number of shape modes needed for a complete analysis of the dynamic response of the structure in order to predict its behavior during an earthquake.

The soil-structure interaction aspects are properly modeled on the interfaces between the shaft and tunnels and the soil. In regions where interaction forces should be simulated, some special measures are to be taken into account like the mesh size. Among the problems which may be encountered for the application of the FE method to complex structures are high computation times and limited capacities of computers. These problems are to be solved by the progress in technology and the development and integration of special mathematical methods in the FE method.

REFERENCES

[1998]. Lar dam pump station project - Structural calculations and underground body @ Site Investigations and Preliminary Studies Report, Tehran.

[1994]. CESAR - LCPC 3.1. - Le code de calcul CESAR @ Presse de Laboratoire Central des Ponts et Chaussées, Paris.