INTRODUCTION

Within the Fourth ICOLD Workshop on Numerical Analysis of Dams, a benchmark problem has been proposed to study the seismic response of an arch dam. The problem focuses on the dependence of this response on the opening condition of a contraction joint.

It has been recognized that during an earthquake this kind of joints may open, changing the load path from that of a monolithic structure. Actually, analytical models assuming a monolithic dam usually predict the development of tensile stresses in the arch which cannot be transferred across the joints.

In recent years, several approaches have been developed, and are still being investigated, for taking into account the nonlinear effects of joint opening. From a computational point of view, the main concerns in this area are related to the strong non-linearity of the contact constraint and to the presence of unrealistically high vibration frequencies induced numerically by the impacts between the two faces of the joint.

The purpose of the benchmark problem is to compare numerical techniques developed by several authors, with the final aim of validating reliable analysis procedures.

The rest of the present paper summarizes the work carried out by Principia-EQE regarding this benchmark problem. First, the problem is briefly introduced. Then, a section is devoted to describe the specific aspects of the methodology employed. Finally, the results requested
by the specification of the benchmark are presented. The tables with the numerical results are gathered in Annex 1.

DESCRIPTION OF THE PROBLEM

The problem is that stated in the documentation package provided by the Workshop’s organizers. The dam is a single curvature arch dam. The upstream face is a vertical cylindrical surface, with a radius of 23.5 m. The total height is 37.5 m, including a freeboard of 1.5 m. The downstream radius is 22.0 m in the upper 8.8 m and, from this point downwards, decreases linearly up to a value of 16.5 m at the base. As a consequence, the thickness is 1.5 m at the crest and 7.0 m at the base.

A vertical joint exists in the central part of the dam. This joint extends from the crest down over a length of 12.0 m.

The suggested finite element mesh is shown in figure 1. Two similar meshes are provided within the documentation package, one with the joint and the other one without it.

The foundation rock is assumed to be rigid. Other mechanical parameters are as follows:

- Concrete of the dam:
  - Young’s modulus = 30 GPa
  - Poisson’s ratio = 0.20
  - Density = 2450 kg/m³

- Water in reservoir:
  - Density = 1000 kg/m³

For the purposes of the benchmark, water has been treated as an incompressible fluid. Also, in studying the vibrations, an equivalent viscous damping ratio of 2% has been assumed for all frequencies.

Three acceleration time histories are provided, corresponding to the three directions of motion along the axes used to define the finite element mesh. These accelerograms should be applied to all nodes along the dam-rock interface. The peak ground acceleration in the upstream-downstream direction is about 0.3g and the strong motion has a duration of about 10 s.
Four analyses have to be carried out. The conditions for each of these cases are summarized in table 1. In all cases the dead weight of the dam and the prescribed seismic motion must be considered.

<table>
<thead>
<tr>
<th>Case</th>
<th>Joint</th>
<th>Water in reservoir</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>No</td>
<td>No</td>
</tr>
<tr>
<td>2</td>
<td>No</td>
<td>Maximum level</td>
</tr>
<tr>
<td>3</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>4</td>
<td>Yes</td>
<td>Maximum level</td>
</tr>
</tbody>
</table>

Table 1. Requested analyses

In the cases including the joint, the latter is assumed to be closed prior to the application of the loads. The friction coefficient for the contact between the two faces of the joint is given as 0.75.

The requested results comprise the maximum values of the response (relative displacements and velocities, absolute accelerations, principal stresses) at certain gauge points, and the acceleration response spectra computed from the time histories obtained at certain locations.

**METHODOLOGY**

The methodology employed is based on the the finite element method. A commercial finite element code, ABAQUS/Standard (HKS, 1995), has been used in all the computations performed.

In order to facilitate the comparison with the results produced by others, the meshes suggested in the problem specification have been utilized. In both meshes the body of the dam is discretized by means of 162 standard 20-node isoparametric (serendipity) brick elements; these are supplemented with 28 standard 15-node wedge elements, which are used as mesh fillers at the rock-dam interface. The mesh representing the monolithic dam has a total of 1163 nodes, whereas that of the jointed dam has 1195 nodes. A linear elastic model has been used to represent the behaviour of the concrete.

The loads have been applied in two steps. In the first one, the weight of the dam and the hydrostatic pressures are applied statically. The resulting stress state provides the initial condition for the second step, in which the seismic ground motions are applied at the rock-dam interface.
For the first step, boundary conditions are specified as a null displacement of all nodes at the rock-dam interface. These nodes, a total of 191, have been taken from the list given in the information package provided by the organizers.

The contact constraint at the joint has been modelled using a “soft” contact approach. In this approach the contact pressure starts at zero for a certain gap size and increases exponentially as the gap size reduces. In the present case, it has been considered that the contact pressure starts developing when the gap is 1 mm and reaches a value of 1 MPa when the gap vanishes. This approach is convenient for avoiding the high frequencies associated with contact “chatter”.

Dynamic dam-fluid interaction has been taken into account in the two cases with water in the reservoir. An “added mass” approach has been followed to simulate the effects of horizontal movement. In this manner, mass elements have been introduced at the corner nodes of element faces on the upstream face of the dam. The added mass for corner nodes has been computed using Westergaard’s formula (ICOLD-27, 1975). The total added mass is 18.1x10^6 kg, compared to a concrete mass in the dam of 8.7x10^6 kg. The mass matrix of these mass elements is constructed in such a way that, in a reference system with an axis normal to the surface of the dam, the only non-zero component is the diagonal term corresponding to this axis. As a consequence, horizontal movements parallel to the tangent plane do not generate hydrodynamic pressures.

The hydrodynamic effects due to the vertical movement of the ground have been taken into account by introducing instantaneous variations of hydrostatic pressure. These variations are computed from the ground vertical acceleration history. It is considered that an upwards ground acceleration \( a \) increases the hydrostatic pressure by a factor of \( \left(1 + a/g\right) \), where \( g \) is the acceleration of gravity.

Physical damping has been modelled using the Rayleigh approximation. Factors \( \alpha_R \) and \( \beta_R \) have been computed in each case, so that an equivalent viscous damping ratio between 1.8% and 2.0% is obtained for the first ten natural frequencies of each finite element model.

The equations of motion have been integrated in the time domain using a direct implicit scheme. In particular, the Hilber-Hughes-Taylor operator with a little numerical damping (\( \alpha = -0.05 \)) has been chosen. This scheme is unconditionally stable and a fixed time step of 0.01 s has been used. This is the same time step with which the ground motion was specified.
RESULTS

Eigenvalue analyses

With the purpose of adjusting the coefficients of Rayleigh damping and to gain a first understanding of the dynamic behaviour of the dam, the natural frequencies and mode shapes corresponding to the four finite element models were computed. The fundamental frequencies of the models are summarized in table 2. Figures 2 and 3 show the shape of the first mode of vibration in cases 1 and 3.

<table>
<thead>
<tr>
<th>Case</th>
<th>Joint</th>
<th>Water in reservoir</th>
<th>Fundamental frequency (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>No</td>
<td>No</td>
<td>13.2</td>
</tr>
<tr>
<td>2</td>
<td>No</td>
<td>Maximum level</td>
<td>9.3</td>
</tr>
<tr>
<td>3</td>
<td>Yes</td>
<td>No</td>
<td>12.4</td>
</tr>
<tr>
<td>4</td>
<td>Yes</td>
<td>Maximum level</td>
<td>8.7</td>
</tr>
</tbody>
</table>

Table 2. Fundamental frequencies of FE models

It can be seen that the models without the joint are slightly stiffer and that the introduction of the added mass of the water causes an important reduction of the fundamental frequency. In general, the dam is considered to be fairly rigid from a seismic standpoint (fundamental frequencies above 8 Hz). Nevertheless, in the cases with water in the reservoir, significant dynamic amplifications should be expected since the response spectrum of the ground motion has a peak at around 9 Hz (figure 4).

Time history analyses

For each case examined, the following results are provided on paper (Annex 1) and in a diskette:

- Maximum relative displacements, relative velocities and absolute accelerations, together with their time of occurrence, at the specified points. The relative displacements are measured from the state at the beginning of the ground motion.
• Response spectra for 2% damping of acceleration time histories in the three coordinate directions, at the specified points.

• Maximum principal stresses and their time of occurrence, at the specified points.

• For cases 3 and 4, maximum relative displacements of nodes across the joint.

All results are given in SI units. The maximum values of the response always take place between 2 and 5 s from the beginning, which corresponds to the interval when the shaking is more severe.

In general, the peak response values requested are between 2 and 3 times larger in the cases with joint than without it. For instance, the maximum displacement at the crest in the upstream-downstream direction is 0.86 mm in Case 1 and 2.6 mm in Case 3; and it is 5.6 mm in Case 2 and 9.5 mm in Case 4. It is worth mentioning that the maximum deflection caused by hydrostatic pressure is on the order of 1 mm.

The maximum responses in the cases with water are about 4 to 5 times larger than the corresponding cases without water. This is because the fundamental frequencies with water are closer to the frequency of the peak of the input spectrum (figure 4) and, therefore, yield a significantly higher dynamic amplification.

Maximum absolute accelerations at the centre of the crest (point 6) are very high for the cases with water. They reach almost 5.5g in Case 4 and near 2.2g in Case 2.

Regarding maximum principal stresses, it should be noted that the values reported here are values extrapolated and averaged at nodes. Therefore, they should be interpreted with care, since the finite element mesh is not particularly fine. Besides, it should be mentioned that the stresses include those due to the weight of the concrete as if it had been applied instantaneously at the end of the construction.

In the cases without water, the maximum principal stresses are always below 1 MPa in absolute value. At specified points, the maximum tensile stress as are 0.69 MPa in Case 1 (point 36) and 0.94 MPa in Case 3 (point 33).
Stresses are significantly larger in the cases with water. Maximum tensile stresses reach 3.4 MPa in Case 2 (point 36) and 2.8 MPa in Case 4 (point 33).

In the cases with joint, the maximum opening at the top of the joint (crest) during the earthquake is 2.1 mm (Case 3) and 4.7 mm (Case 4).

Computer execution times for a 10 s simulation vary from case to case. The shortest time is obtained for Case 1, where no joint exists and there are no added masses. For running this Case, roughly 3 hours of CPU were needed in an HP 9000/C100 workstation. On the other hand, the longest time was obtained for Case 4, with the joint and the added masses; in this Case, more than 17 hours of CPU were needed. As might be expected, the cases with simulation of contacts (cases 3 and 4) require considerably greater resources. Case 3 required almost 15 hours of CPU time, whereas Case 2, without contacts, used just above 7 hours.

REFERENCES


Figure 1. Finite element model
Figure 2. Case 1. First natural mode of vibration
Figure 3. Case 3. First natural mode of vibration
Figure 4. Ground motion response spectrum in the upstream-downstream direction