Experience and lessons learnt from the SMART 2013 benchmark

M.J. Crespo, J. Rodríguez, L. Lacoma, F. Martínez and J. Martí

PRINCIPIA Ingenieros Consultores

Abstract*: The French CEA, together with EDF and the IAEA, recently organised an international benchmark to evaluate the ability to model the mechanical behaviour of a typical nuclear reinforced concrete structure subjected to seismic demands.*

The participants were provided with descriptions of the structure and the testing campaign; they had to propose the numerical model and the material laws for the concrete (stage #1). A mesh of beam and shell elements was generated; for modelling the concrete a damaged plasticity model was used, but a smeared crack model was also investigated.

Some of the initial experimental results, with the mock-up remaining in the elastic range, were provided to the participants for calibrating their models (stage #2). Predictions had to be produced in terms of eigen-frequencies and motion time histories. The calculated frequencies reproduced reasonably the experimental ones; the time histories, calculated by modal response analysis, also reproduced adequately the observed amplifications.

The participants were then expected to predict the structural response under strong ground motions (stage #3), which increased progressively up to a history recorded during the 1994 Northridge earthquake, followed by an aftershock. These results were produced using an explicit solver and a damaged plasticity model for the concrete, although an implicit solver with a smeared crack model was also investigated.

The paper presents the conclusions of the pre-test exercise, as well as some observations from additional simulations conducted after the experimental results were made available.

Keywords*: Concrete, damage, time history.*

1 Introduction

There are considerable uncertainties involved in the prediction of the response of reinforced concrete structures to seismic events. To advance the knowledge and capabilities in that regard, the French Commissariat á l’Énergie Atomique (CEA), Électricité de France (EDF) and the International Atomique

Energy Agency organised the international benchmark named the SMART 2013 project.

The exercise involves a number of shaking table tests carried out in a reinforced concrete mock-up, as well as predictive calculations regarding the outcome of those tests. Principia participated in the project by conducting the necessary calculations for three of the four stages and the present paper summarises the findings.

2 Description of the model (Stage #1)

Both the geometry and material properties assigned to the model were described in Stage #1. However some changes were introduced in the course of the project, as suggested by the results already obtained or by requirements of the analyses.

2.1 Geometry

The structure is a 1/4th scaled mock-up, with a trapezoidal shape, three stories, of reinforced concrete. It is representative of a typical, simplified half part of an electrical nuclear building. The geometry is with the aim of studying the torsional behavior.

The mock up has a total height of 3.1 m, a width of 3 m and the two lateral walls have a width of 2.55 m and 1.05 m respectively. All the lateral walls and the three floors have a thickness of 10 cm. There is also a central pillar with a square section of 10 cm x 10 cm. Figure 1 shows a plan view and a lateral view of the mock-up. This two views together with the 3D ones that will be presented from the finite element mesh provide a complete idea of the mock-up geometry.

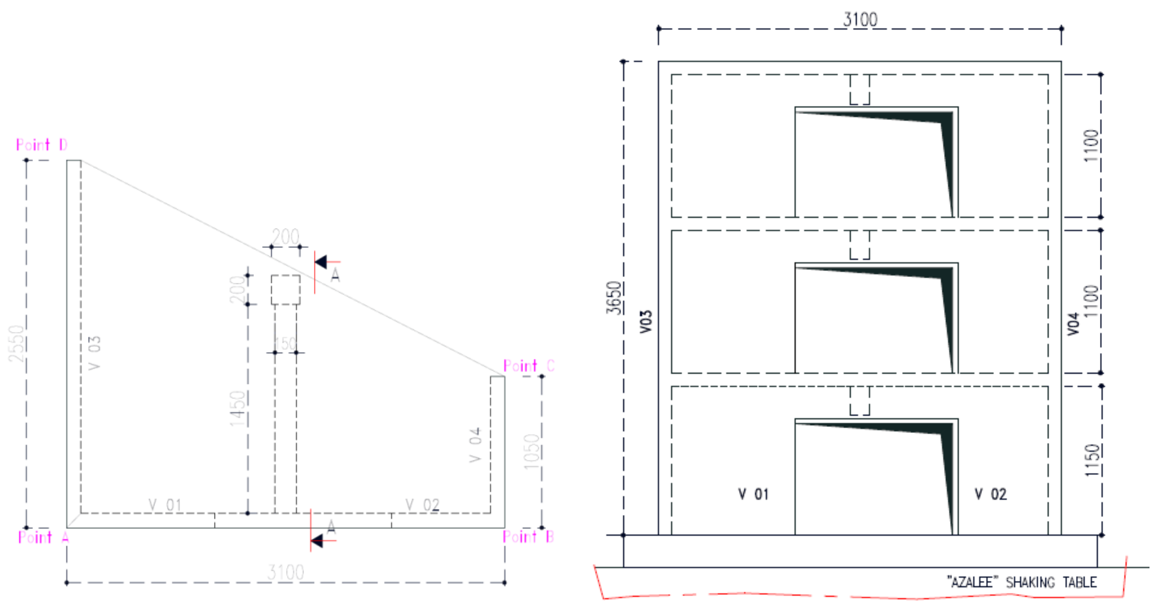

Figure 1 Plan and lateral view of the mock-up.

2.2 Materials

The mock-up involves two materials: concrete and reinforcing steel. In stage #1, the material tests were not yet available and the material properties used in the analyses were conventional design values.

There are several material models in Abaqus [SIM13] that are suitable for modelling concrete. That selected was the "smeared crack concrete model" available in the implicit code Abaqus/Standard. It provides a general capability for modelling concrete in all types of structures, including beams, trusses and solids. The model consists of an isotropically hardening yield surface that is active when the stress is dominantly compressive and an independent "crack detection surface" that determines if a point fails by cracking. It uses oriented damaged elasticity concepts (smeared cracking) to describe the reversible part of the material's response after cracking failure.

A parabolic stress-strain law was used, with an initial modulus of 32 GPa, a peak stress of 30 MPa at 0.22% strain, and an ultimate strain of 0.35%, as presented in Figure 2. The tensile strength is 2.4 MPa with an assumed fracture energy of 180 J/m^2. The resulting biaxial behaviour is shown in Figure 3.

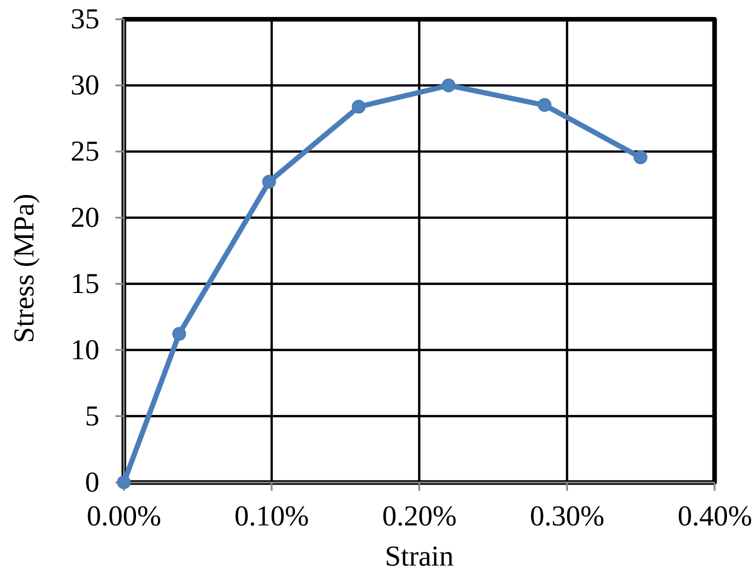

Figure 2 Uniaxial behaviour of concrete

Due to convergence problems experienced in some of the runs in Stage #3, a decision was made to change from implicit to explicit integration, using Abaqus/Explicit. Since this module does not support the former material model, the "concrete damaged plasticity" model was used instead. It combines non-associated multi-hardening plasticity and scalar (isotropic) damaged elasticity to describe the irreversible damage that occurs during the fracturing process. It is meant to provide an accurate approach for modelling cyclic events, allowing the user control over the stiffness recovery effects.

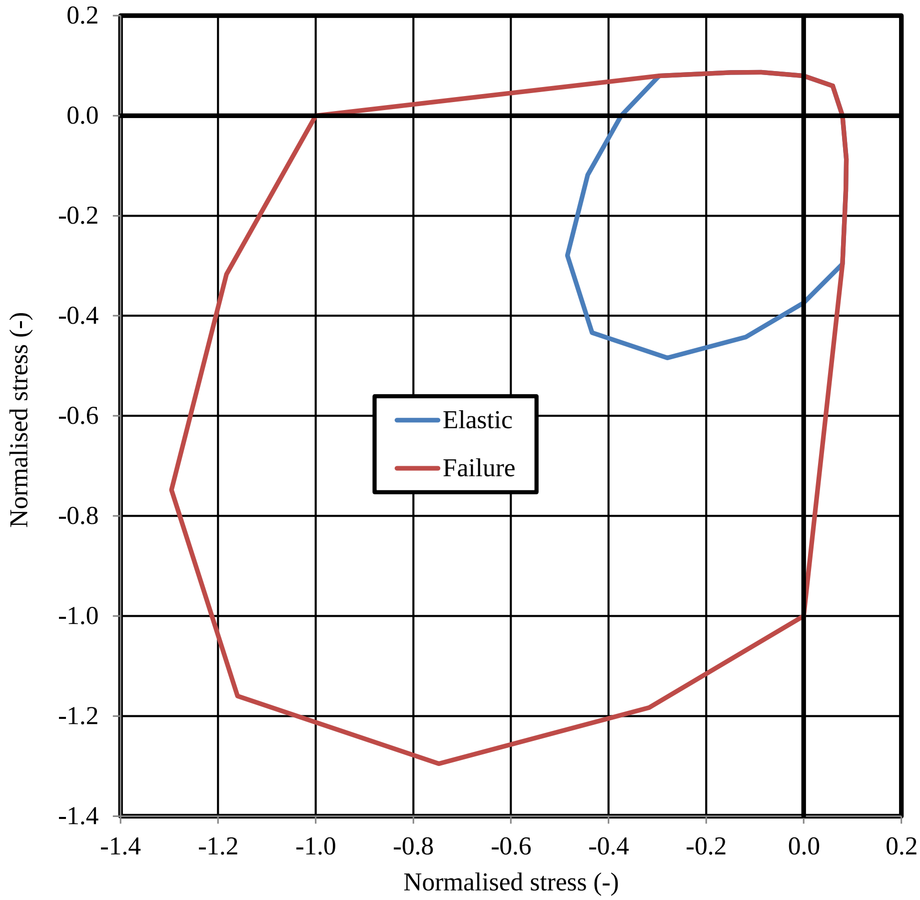

Figure 3 Biaxial behaviour of concrete

The reinforcing steel was modelled as an elastoplastic material with isotropic (von Mises) plasticity. Some experimental information had been provided, namely the Young's modulus, the elastic and shear strengths, and the ultimate strain. Although the simulations of the material tests were performed with reinforcing bars embedded in the concrete elements, those of the mock-up used the "rebar layer" technique, which does not consider individual bars but the averaged stiffness effects on the concrete elements.

2.3 Finite element mesh

The mesh used to represent the mock-up consists mainly of shell elements (8.559); a few (230) linear beam elements were also used for the pillar and the three beams on the floors.

The shaking table has 10,253 elements, as in the mesh provided by the organising committee (OC). The total number of nodes is 20,247. Figure 4 presents two views of the mesh.

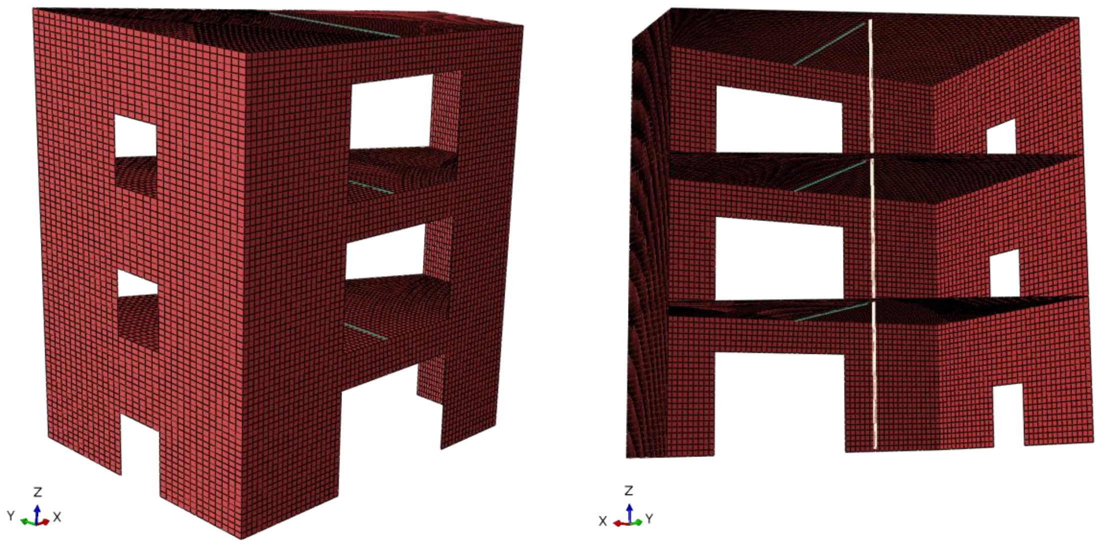

Figure 4 General views of the finite element mesh

3 Linear Dynamic Analysis (Stage #2)

In Stage #2 linear dynamic analyses were performed. The first activity consisted in conducting modal analyses under three different hypotheses:

- Mock-up fixed at its base with no additional masses.
- Mock-up fixed at its base and loaded with additional masses.
- Mock-up fixed to the shaking table and loaded with additional masses.

The frequencies corresponding to the first three natural modes calculated for each of the three above hypotheses appear in Table 1.

The first three modal shapes are similar under the three hypotheses. The first mode is the main one in the X-direction (see axes in Figure 4), the second one is the main one in the Y-direction, and the third one is a torsional mode around the vertical Z-axis. Figure 5 presents the three modal shapes for the third hypothesis.

The frequency differences observed between the first two hypotheses are consistent with the increase in the global mass of the model. The differences between the second and third hypothesis arise from the flexibility of the shaking table.

OC provided the participants with experimentally determined frequencies, which are also indicated in Table 1. The differences between the calculated and experimental frequencies are probably due to the material properties which, at this stage, had to be given standard values; experimentally determined properties only became available later.

Table 1 Calculated and measured frequencies (Hz) under three hypotheses

	Mode 1	Mode 2	Mode 3
Fixed base, no masses	19.9	35.4	65.4
Fixed base, with masses	8.40	15.4	29.7
Fixed to shaking table, with masses	5.65	8.76	19.0
Experimental	6.28	7.86	16.5

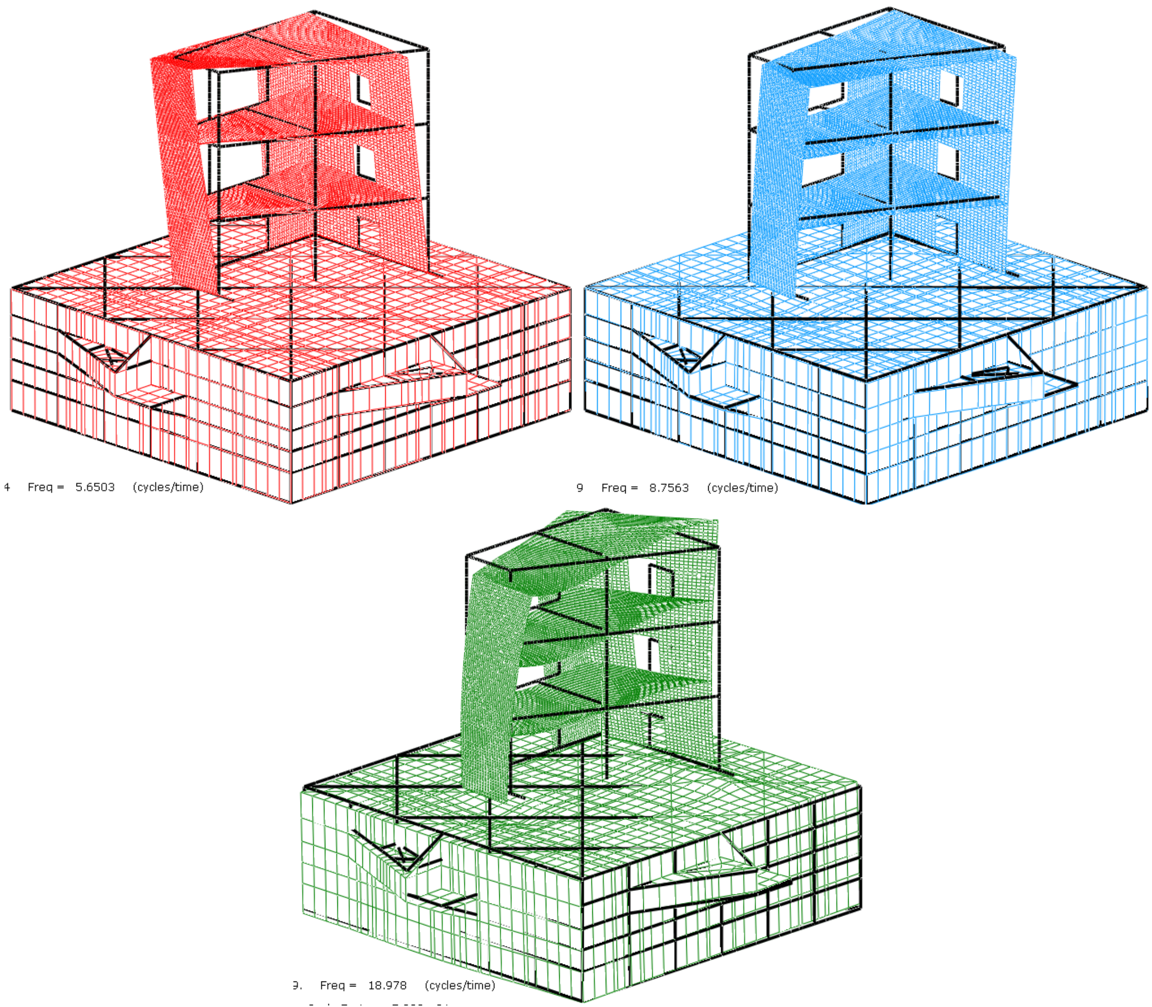

Figure 5 Modes of the mock-up fixed to shaking table and loaded with masses

This stage also included a linear time domain analysis for two time histories (identified as Runs 6 and 7), both with a PGA of 0.10g. Run 6 is a synthetic white noise, while Run 7 is a scaled design signal.

The input motion was introduced in the calculation by imposing displacements at the actuators, i.e.: creating an additional node linked with a distributing coupling to the four nodes of the shaking table surrounding the actuator. This type of connection distributes forces but does not introduce any stiffness.

Modal dynamic analyses were conducted for the different base motions and associated input motions at the actuators. The damping was assigned on a mode-by-mode basis, using the information provided in the experimental report by OC: 2.6% for the first mode, 4.2% for the second, and 5.5% for the third one. A damping ratio of 6% was assigned to all the higher modes. The results were presented in terms of absolute displacements and accelerations.

In general, for this stage the results obtained were of the same order as those provided by OC. Space limitations do not allow an exhaustive comparison, but a sample result has been selected for display, which is the horizontal X-displacement of point A in floor 3. Figure 6 provides the comparison of calculated and measured results for Run 6. The rest of the comparisons presented in the paper correspond to that same location.

4 Nonlinear Dynamic Analysis (Stage #3)

Convergence problems were experienced with Abaqus/Standard for some of the runs contemplated in this stage. The concrete cracking caused a sudden release of the internal energy inducing a reduction of the time increment to the order of 10 μs. The convergence could be improved by changing the concrete fracture law or by adding numerical damping but, since such strategies would alter the structural behaviour, a decision was made to change to the explicit solver in Abaqus/Explicit [SIM13]. The time increment is now around 2 μs, smaller than in Abaqus/Standard but much faster per increment since no matrix needs to be inverted or factorised. Besides, the increment is almost not reduced during the simulation.

As mentioned earlier, this required changing the material model. The main problems posed by the new formulation of the concrete behaviour are that the reversals are somewhat stiffer than expected and that, with increasing numbers of cycles, the equivalent plastic strain may ratchet up with little physical basis.

Some results are presented below for all the mandatory runs, which are no. 9, 13, 17, 19, and 23. All the results included here correspond to the same X-displacement of point A in floor 3. For each of the five runs mentioned, the comparisons appear in Figure 7 through Figure 11.

The large displacement part of the response is considered to be generally satisfactory in all cases, perhaps particularly in Runs no. 13, 17 and 19. However, there are important differences in the amplitudes of the oscillations during the second half of the history which, although their significance may not be very important, they are still not easy to explain.

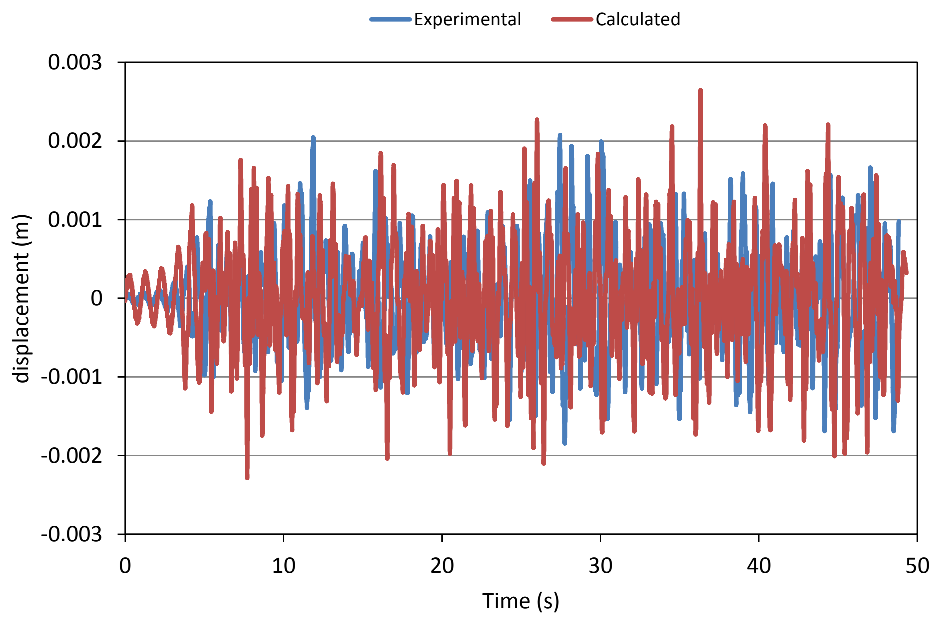

Figure 6 Calculated and measured X-displacements for point A, floor 3, Run 6

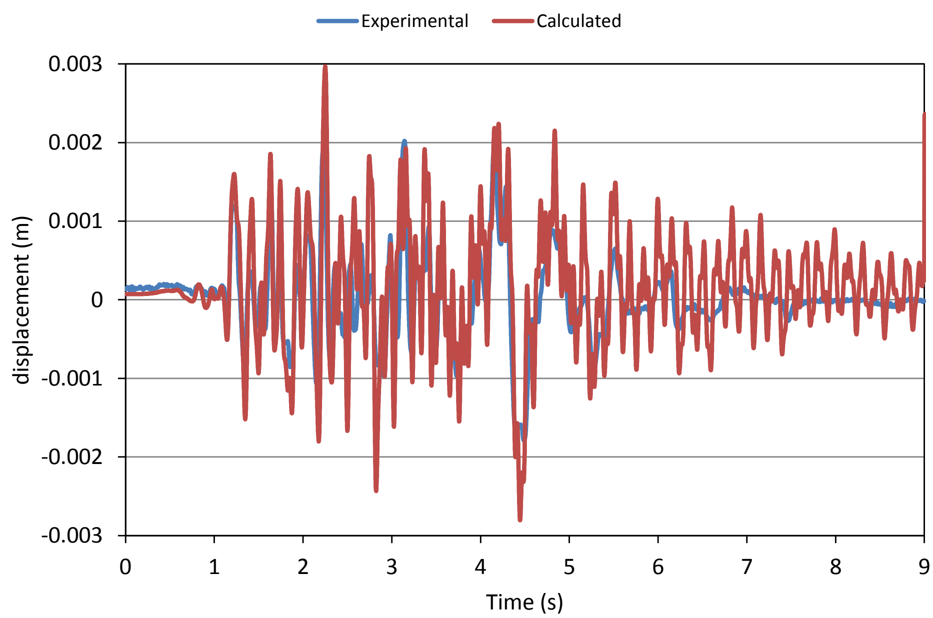

Figure 7 Calculated and measured X-displacements for point A, floor 3, Run 9

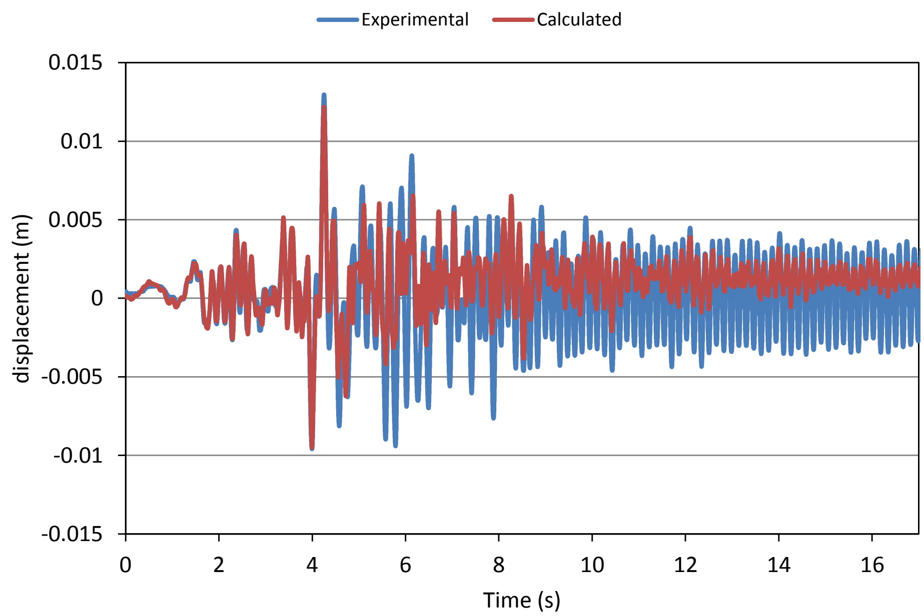

Figure 8 Calculated and measured X-displacements for point A, floor 3, Run 13

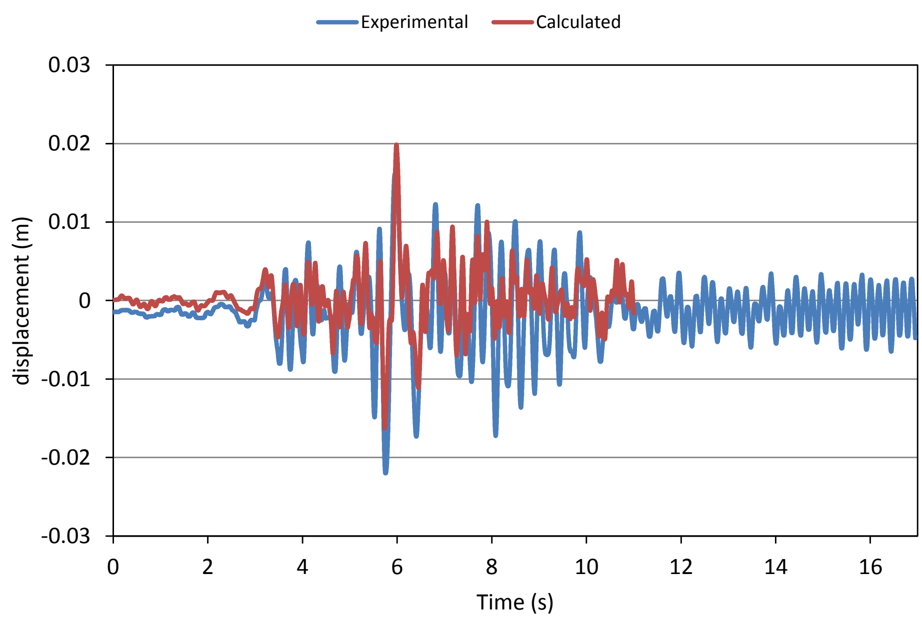

Figure 9 Calculated and measured X-displacements for point A, floor 3, Run 17

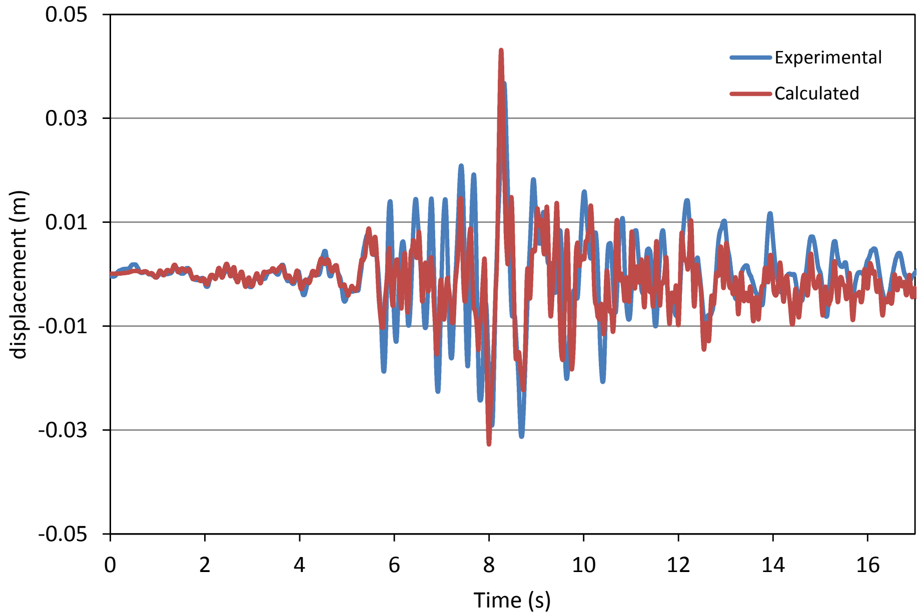

Figure 10 Calculated and measured X-displacements for point A, floor 3, Run 19

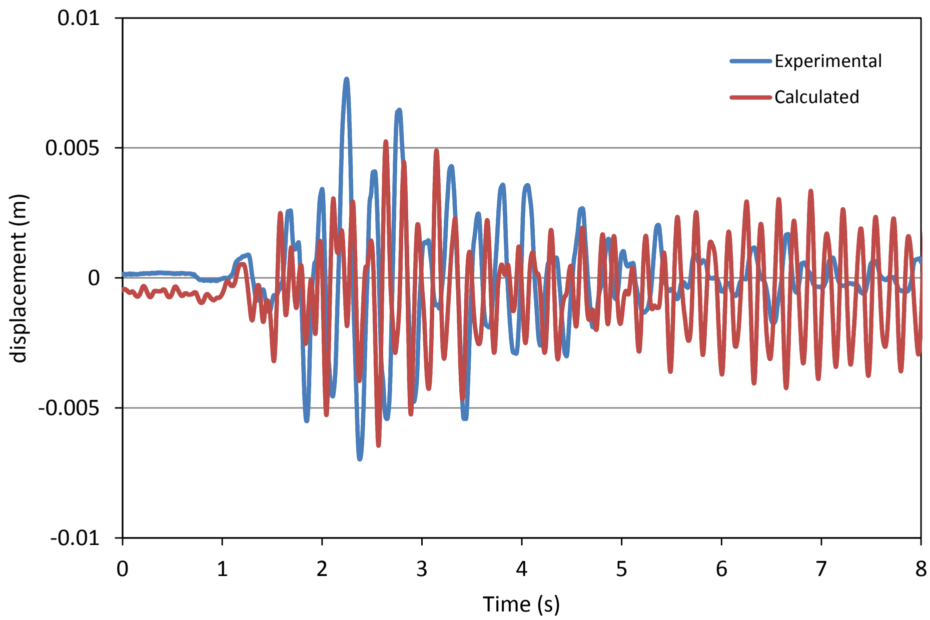

Figure 11 Calculated and measured X-displacements for point A, floor 3, Run 23

5 Conclusions

Calculations were conducted for the first three stages of the International Benchmark SMART 2013, which involve predictions of the response of a mock-up in a series of shaking table tests. As a result of the work conducted, the following conclusions can be offered:

1. The calculated modal frequencies reproduce only moderately those determined experimentally. The differences are partly attributed to the fact that, at the time, standard rather than experimentally determined material properties had to be used.
2. The low-amplitude time history analysis did provide a reasonable match of the experimentally determined response.
3. The strong non-linearities and softening caused by the large-amplitude motions in stage #3 gave rise to convergence problems in Abaqus/Standard which could not be satisfactorily solved while maintaining the material description. This caused a migration to Abaqus/Explicit as solver, which also entailed moving from a smeared crack model to a damage plasticity model for the concrete.
4. Finally, the calculations performed in stage #3 agreed reasonably well with the experimental results at least for the large displacement part of the response, though the amplitude of the oscillations in the second half of the history is somewhat less satisfactory.

6 References

[SIM13] SIMULIA (2013) “Abaqus Analysis User’s Guide”, Version 6.13, Providence, Rhode Island.

[RIC13] Richard, B. and Chaudat, T. “Presentation of the SMART 2013 International Benchmark”, Document no. DEN/DANS/DM2S/SEMT/EMSI/ST/12-017/E, 9 September.