PRELIMINARY NUMERICAL ANALYSIS OF A REINFORCED CONCRETE MOCK UP: EFFECTS OF THERMAL BREAKERS AND SHAKING TABLE

S. Crijanovschi[1,2]
(1) Technical University "Gheorghe Asachi", Iasi, Romania
(2) CEA, DEN, DM2S, Laboratoire d'Etudes de Mécanique Sismique, F-91191 Gif-sur-Yvette, France.

B. Richard[1] & T. Chaudat[1]
(1) CEA, DEN, DM2S, Laboratoire d'Etudes de Mécanique Sismique, F-91191 Gif-sur-Yvette, France.

G.M. Atanasiu[1]
(1) Technical University "Gheorghe Asachi", Iasi, Romania

SUMMARY:
Reinforced concrete buildings exhibiting 3D and nonlinear effects are a main concern in the field of earthquake research and regulation. The understanding of the structural behavior of such complex structures under seismic loading as well as of their degradation modes is necessary and still remains a main challenge especially when important torsional effects are expected. To answer to this issue, the European project ENISTAT has been started in 2010. The aims of this project are (i) to provide experimental results for low reinforcement ratio and large crack patterns,(ii) to use new measurement devices such as digital image correlation and (iii) to explore the effects of including new structural components such as thermal breakers. This paper aims at presenting preliminary numerical analyses of the RC mock up. A 3D nonlinear finite element model including the shaking table interaction has been realized estimating the modal frequency shift expected in future experimental campaigns.

Keywords: Nonlinear model; Spectral analysis; Thermal breakers; Shaking table interaction

1. INTRODUCTION

The estimation of tri-dimensional non-linear effects which develop due to seismic loadings in reinforced concrete structures is done through seismic experimental campaigns on reduced scaled models of real buildings or special mock-ups designed especially for study purposes, with application in construction industry. It is necessary to understand the mechanisms behind the crack developing patterns and the failure criteria. For this purpose the ENISTAT numerical research project was created in 2009, derived from "SMART 2008"project (Juster-Lermitte et al. 2009) which was focused on a ¼ scale specimen of a nuclear electric building. The ENISTAT research project aims to study the torsional behavior of reinforced concrete (RC) wall buildings characterized by in-plane irregularity. The project is granted under SERIES project via transnational access to C.E.A/Saclay facilities in France. Extensive numerical studies have been carried out in order to reveal the frequency domain of the structure in the test environment in order to get the expected results in the upcoming experimental campaign. The mock-up will be tested on the AZALEE shaking table at TAMARIS laboratory in C.E.A Saclay using artificial seismic signals defined according to Eurocode 8 (CEN, 2008) regulations. The numerical studies carried out consider the local and general influences on the model which might influence the natural frequencies of the structure.

The fully instrumented specimen will be tested using different levels of artificial seismic excitations, fully fitted on the frequency domain of the building in order to maximize the response of the building. The main objectives of ENISTAT research project focus on the study of non-linear behaviour of reinforced concrete shear wall structures subjected to seismic actions. The use of digital image correlation technique and on crack-tracking techniques will reveal in real time the step-by-step energy dissipation process as well as the failure mechanisms during tests. There will also be monitored the influences of new thermal breaker elements on the overall and local rigidity. These elements will be placed at the connection between slab and shear wall in order to reduce the thermal energy loss at the given level.(Yakut,2010).

This paper presents the custom modal analysis and spectral analysis, considering varying local test conditions revealing the frequency shift which occurs due to local and general boundary conditions. There are performed two sets of numerical analyses in two different computational environments:

- SAP2000: commercial structural analysis software for linear and nonlinear finite element analysis;
- CAST3M: a finite element analysis software designed for research industry and developed by C.E.A. This software is based on the GIBIANE language, a powerful and flexible script allowing refined numerical analyses based on the finite element method.

Considering the advantages and the drawbacks of each software, the numerical analyses that have been carried out can be classified into two categories:

- several modal analyses, with various boundary conditions to quantify their influence on the overall mechanical behaviour;
- a spectral analysis according to specifications of Eurocode 8 (EC8) to verify structural response of the ENISTAT model and validate the design realized by SETEC Industries, the technical designer of the mock-up;
    - o the response spectrum used in the analysis is EC8 spectrum Type 1 with a nominal acceleration of 0.3g scaled down to fit on the frequency domain of the mock-up.

1.1. Objectives of comparative modal analysis

The main objective of the comparative modal analysis between SAP2000 and Cast3M is to compare two different computer software. Three different models are considered in both software to analyze the behaviour of the structure during the seismic testing:

- Simple constituent model without any local influences; steel reinforcements are not considered explicitly
- Model considering influence of shaking table and small deformations which occur due to shaking table interaction; steel reinforcements are not considered explicitly
- Model considering the equivalent steel reinforcement as well as shaking table interaction.

1.2. Objective of the comparative spectral analysis

The main objective of the comparative spectral analysis is to validate the design procedures of the ENISTAT mock-up provided by SETEC Industries and provide a reference point for future analyses and choice of input signals in accordance with members of ENISTAT research program.

Main input parameters in spectral analysis:

- EC8 Spectrum Type 1 scaled accordingly for ENISTAT specimen (effect of scaling factor time reduced by $\sqrt{2}$);
- Soil type C;
- $a_g = 0.3g$ and $a_{vg} = 0.27g$.

2. DESCRIPTION OF R.C. MOCK-UP

Following the SMART project (high level of reinforcement ratio for a non-symmetric nuclear facility building) for which an international blind benchmark allowed 40 international teams to test their numerical models, this new experiment, based on the SMART geometry will provide experimental results for a different type of structures. Due to the high level of reinforcement, only the lowest levels of nonlinearities and cracking have been reached in SMART 2008 experiments. The ENISTAT experimental campaign will be addressed to classical buildings with low reinforcement ratio specific for civil use. By designing the mock-up with these characteristics there are expected visible development of 3D effects and torsion while in the mean time ensuring a high level of degradation in the structure, (Yakut, 2010).

The ½ scaled model is representative of a three storey office building. The structure is characterized by plane irregularity and has a equal storey height of 1.55 m, as presented in Figure 2.1. The in-plane trapezoidal shape was designed in order to develop torsional behaviour during experimental and to evaluate the effects of torsion on the structure. The scale factor of ½ doesn't imply a change in concrete and steel behaviour laws but it requires placing additional masses on each slab and scaling the design spectrum accordingly.

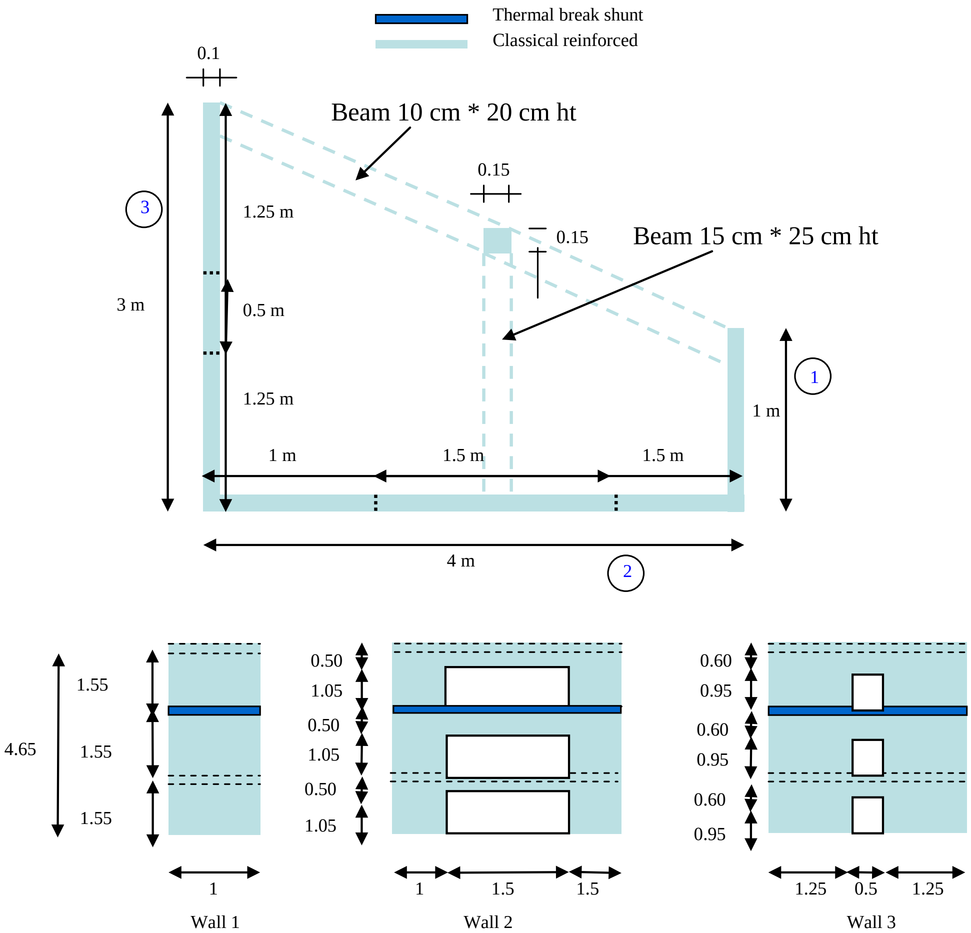


Figure 2.1. Plan and Elevation of ENISTAT specimen (units are in meters).

2.1. Material Proprieties

The materials used in the design and construction of the mock-up are conventional construction materials for a reinforced concrete shear wall structure. The concrete used in the shear walls as well as in the slabs is of grade C25/30 as prescribed by European seismic norms. The reinforcement bars in the concrete elements as well as the thermal breaker elements, which have steel anchorage systems, use the same grade of steel, Fe500-3. There were chosen these conventional materials in order to show their detailed behaviour under high seismic loading, thus allowing a better understanding of non-linearity development in reinforced concrete structures.

Another innovative element of the mock-up is represented by the thermal breakers system Schöck Rutherma DF which connects the slab on the 2nd floor with the shear walls, which is able to transfer high shear load and bending moments as well as provide insulating proprieties, as represented in

Figure 2.2. The thermal breaker system used in the ENISTAT is scaled down with a ½ factor to respect the overall scale of the model. The Schöck Rutherma DF modules are comprised of two main parts, one of polystyrene with thermal insulating proprieties and the other ensuring the anchorage of the element in the reinforced concrete structure. The thermal breaker element is composed for all statical requirements. Therefore it is predestined for additional requirements due to dynamical effects, (Diler, 2010).

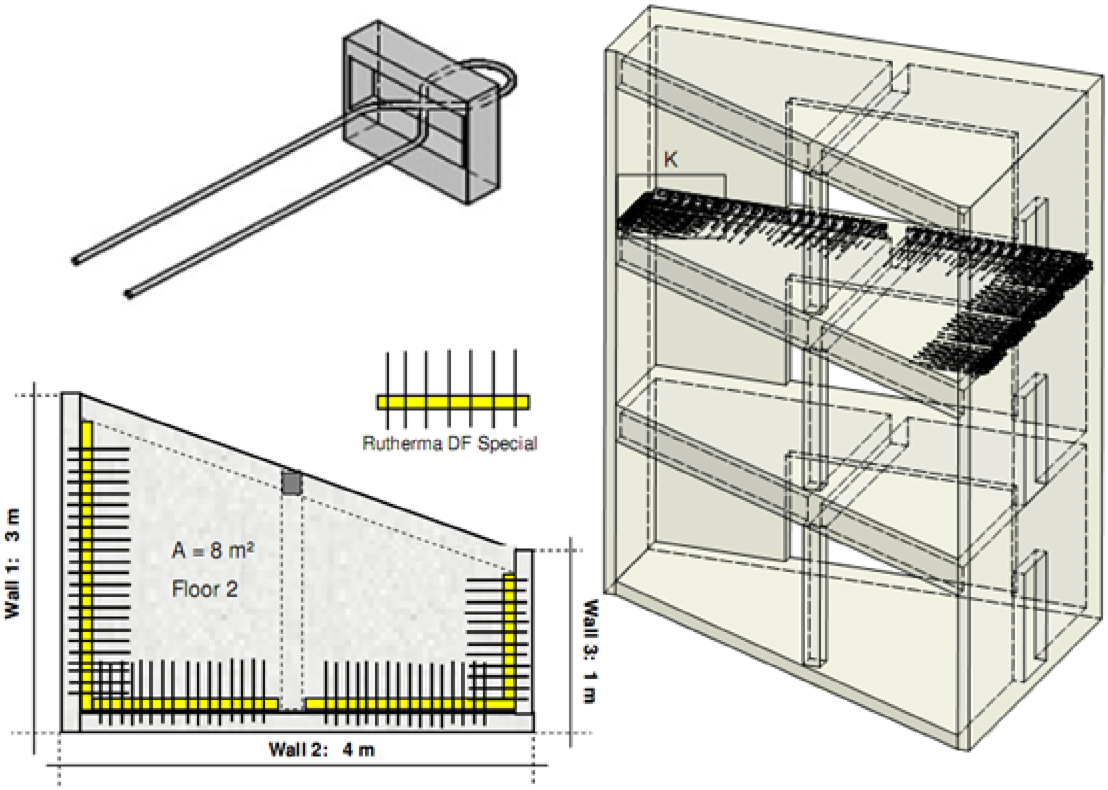


Figure 2.2. Position of thermal breaker elements in ENISTAT mock-up,(Diler, 2010).

3.DESCRIPTION OF NUMERICAL MODELS

Considering the advantages and the drawbacks of each software, the numerical analyses that have been carried out can be classified into two categories:

- several modal analyses, with various boundary conditions to quantify their influence on the overall mechanical behaviour;
- a spectral analysis according to specifications of Eurocode 8 (EC8) to verify structural response of the ENISTAT model and using type 1 response spectrum with 0.3g nominal acceleration.

3.1. SAP2000 Numerical Model

There were considered several numerical models considering various boundary conditions:

- a numerical model considering the ENISTAT structure restrained at the base of the foundation;
- a numerical model considering the ENISTAT structure with shaking table Azalée interaction taken into account;
    - o for this purpose it was considered the original geometry, material characteristics and boundary conditions of the Azalée shaking table, (Le Maoult, 2009);
- a numerical model considering the ENISTAT structure with shaking table Azalée interaction taken into account and influence on overall rigidity of equivalent area of steel;

The numerical models in SAP2000 used shell elements for walls and plates, linear elements for columns and beams and volumetric elements for the foundation.

3.2. CAST3M Numerical Model

In order to validate the numerical results obtained from the numerical models considered in SAP2000, there were considered the same type of models in CAST3M, a finite element software designed for the research industry.

In addition to the aforementioned numerical models, it was developed a model considering the influence of the thermal breakers, (Vassaux, 2011) at the level of connection between the slab on the second floor and the structural walls. This latest model considered also the influence of the Azalée shaking table, (Juster-Lermitte, 2009) and the influence on overall rigidity of equivalent layer of steel. The numerical model considering the influence of thermal brick elements uses FEM designed in the C.E.A. especially to simulate the thermal discontinuity between slab and wall connection and evaluate the effect of these elements on the overall structural behaviour of the structure. The FEM was implemented in the numerical model by developing double nodes with equal coordinates along the thermal break line, where the slab on the second floor connects with the wall, so as to create a line of elements in the mesh that have the same proprieties as the elements designed by SCHOCK Rutherma, the agreed designer of the thermal brick elements.

3.3. Azalée Shaking Table Numerical Model

The numerical model for the Azalée shaking table was designed using the original geometry, constitutive material proprieties and boundary conditions implied by local conditions of the C.E.A Tamaris laboratory. The model was designed for both structural analysis software, SAP2000 and CAST3M using shell elements for steel and aluminium plates and linear elements for the actuators, as displayed in Figure 3.1.

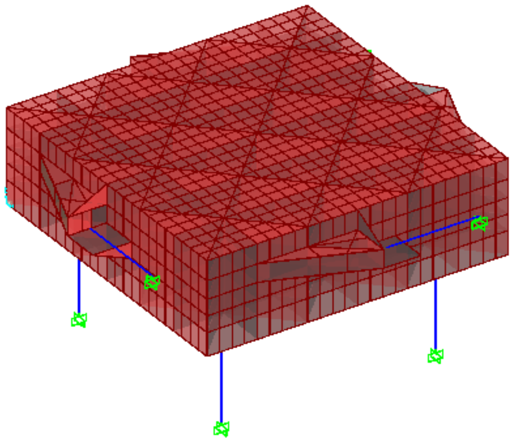


Figure 3.1. Azalée shaking table numerical model developed in SAP2000,(Le Maoult, 2009).

The model of the shaking table was validated through extensive numerical studies under C.E.A internal approval, proving similar frequencies as the ones provided by the initial manufacturer.

4. NUMERICAL RESULTS

4.1. Spectral Analysis

The spectral analysis was performed using SAP2000 vs.14.0 software in order to verify the efforts developed in the model when carrying out a response spectrum analysis from EC8 seismic regulations.

The spectral analysis is based on the EC8 design code. An idealized spectrum is applied on the structure in order to determine the maximum efforts in the structural components with pre established safety and amplification factors. These factors have been specifically chosen for the ENISTAT mock-up. This analysis allows to check if the EC8 requirements are fulfilled.

In order to keep the same acceleration (gravity load cannot be changed) as well as the same material properties, the scaling of ½ of the structure's dimension implies to scaling factors presented in the following table.

Table 4.1. Scaling factors used in ENISTAT research project(Notes d'hypothèses générales indice C, CEA, Project ENISTAT- SERIES FP7)

Specification	Scaling factor between the real structure (scale 1) and the test specimen (scale ½)
Length	2
Mass	4
Time	√2
Acceleration	1
Stress	1
Frequency	1/√2
Force	4
Steel reinforcement area	4

There were defined behaviour coefficients (q) in accordance with the European seismic design norms, Eurocode 8 in two main directions of seismic excitation, horizontal and vertical. For each class of structural elements there were chosen specific behaviour coefficients as follows:

- Behavior coefficient: horizontal seismic action
    - Footings : q = 1;
    - Vertical elements (walls and central column) : q = 2;
    - Slabs: q = 1.
- Behavior factor: vertical seismic action
    - Footings : q = 1;
    - Vertical elements (walls and central column) : q = 1,5;
    - Slabs: q = 1.

After combining the seismic actions in all three directions using Newmark's combination methodology there were identified certain areas of main interest in the ENISTAT mock-up. Due to its initial design for torsional behaviour, there were noticed, as presented in Figure 4.1. , high stress concentrations in the shortest side of the building, which allows the structure to twist considerably.

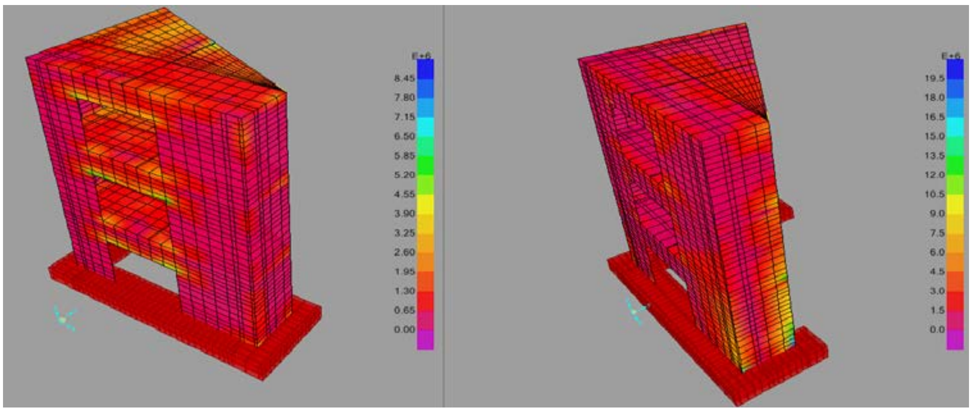
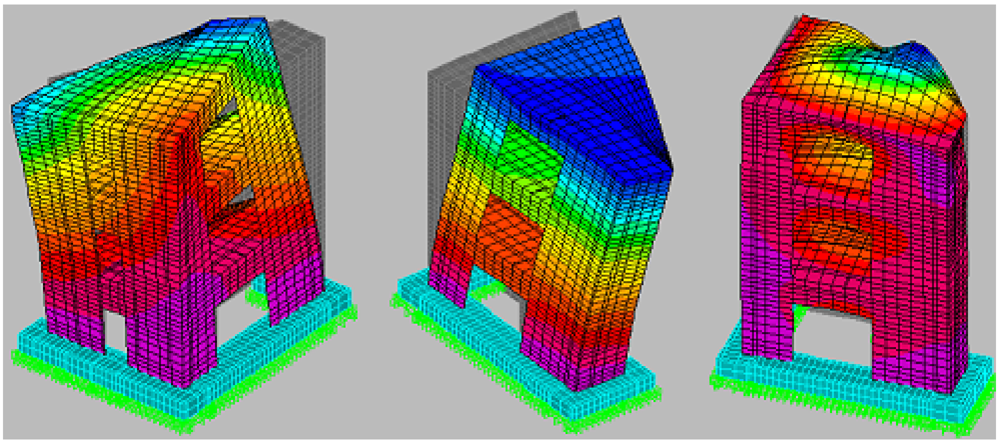


Figure 4.1. Stress distribution in the ENISTAT mock-up under subjected to scaled spectral accelerations in (a) x direction and (b) y direction.

4.2. Modal Analysis

The results of the modal analyses are coming from two distinct computational environments with different market-base application fields, industry and research. However by applying the same behavioural laws and the same boundary conditions on all sets of analyses it is noticed that the results are very close.

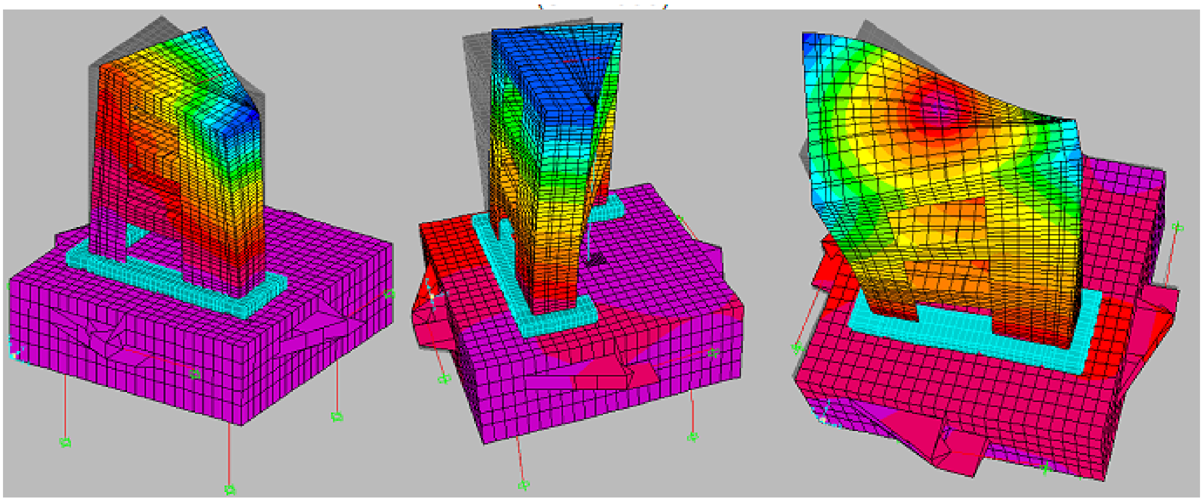
The first mode of vibration of the mock-up model is one which combines translation on y direction with tri-dimensional torsion, due to the in-plane irregular shape, the highest displacements developing at the outer extremity of the short side of the mock-up, as can be observed in Figure 4.2.(a) and in Figure 4.3.(a). The modal shapes are similar, when the shaking table interaction is taken into account, however the frequencies differ significantly.



(a) (b) (c)

Figure 4.2. Modal shapes of ENISTAT mock-up computed in SAP2000 without shaking table interaction.

The second mode of vibration of the structure is one of pure translation along y axis. As expected the highest deformations are developed at the last level of the mock-up, as displayed in Figure 4.2.(b). and Figure 4.3(b). Once again, when we take into account the influence of the shaking table Azalée the frequencies decrease significantly, but the modal deformations have the same patterns.
The third mode of vibration, identified in Figure 4.2.(c) and Figure 4.3.(c). shows differences in modal shape deformation patterns when we take into account the shaking table interaction. When the structure is considered restrained at the base of the foundation, it is observed a excitation of the slabs, as opposed to the case which includes the shaking table, when the structure develops a pure torsional behaviour.



(a) (b) (c)

Figure 4.3. Modal shapes of ENISTAT mock-up computed in SAP2000 with Azalée shaking table interaction.

There were computed the first eight eigen frequencies of the ENISTAT mock-up in both computational environments in order to compare the modal response of the structure in various boundary conditions. The first three frequencies of the mock-up are between 8 Hz and 30 Hz, as we can observe in Table 4.2 and Table 4.3 when the structure is restrained at the base of the foundation. When the shaking table interaction is introduced in the boundary conditions the frequencies decrease considerably due to micro-deformations on the surface of the shaking table and where the actuators are connected to the ears of the shaking table.

Table 4.2. Eigen frequencies of ENISTAT mock-up computed in CAST3M

CAST3M MODELLING															
	ENISTAT RESTRAINED			ENISTAT+AZALEE			ENISTAT+ AZALÈE+ RUTHERMA			ENISTAT+ AZALÈE+ RUP_THER+ STEEL			Frequency shift (Considering all influences)		
Mode	1	7,97	Hz		6,59	Hz	1	6,5303	Hz	1	6,636	Hz	1	16,737	%
	2	15,92		2	10,87		2	10,732		2	10,906		2	31,494	
	3	27,013		3	21,381		3	20,819		3	20,964		3	22,392	
	4	29,66		4	22,917		4	21,428		4	21,621		4	27,103	
	5	32,168		5	29,11		5	25,946		5	25,947		5	19,339	
	6	32,55		6	31,59		6	28,257		6	28,857		6	11,345	
	7	32,97		7	32,05		7	29,853		7	29,838		7	9,499	
	8	33,8		8	33,09		8	31,922		8	31,94		8	5,503	

In the CAST3m modelling it was also considered a model which took into account the influence of the Rutherma thermal breaker system mounted at the second level of the structure in order to create a thermal bridge between the slab and the outer walls. The influence of this system on the overall structural frequencies proved to be very small, as observed in Table 4.2. These factors were not considered in the computations done with SAP2000. Observing the overall shift in frequency from the two tables there are noticed the same behavioural tendencies of the structure, when considering similar boundary conditions.

Table 4.3. Eigen frequencies of ENISTAT mock-up computed in SAP2000 vs14.0.

SAP2000 MODELLING												
	ENISTAT RESTRAINED			ENISTAT+AZALEE			ENISTAT + AZALÈE + STEEL			Frequency shift (Considering all influences)		
Mode	1	7,7038	Hz	1	6,3071	Hz	1	6,4785	Hz	1	15,905	%
	2	15,579		2	9,9738		2	10,136		2	34,938	
	3	29,503		3	20,705		3	21,019		3	28,756	
	4	29,886		4	22,563		4	22,601		4	24,37	
	5	32,918		5	28,741		5	29,826		5	9,393	
	6	37,256		6	36,43		6	36,466		6	2,120	
	7	38,807		7	37,789		7	37,684		7	2,893	
	8	39,447		8	38,948		8	38,974		8	1,199	

As it can be observed in the tables above the influence of the local and general boundary conditions on the structural frequencies is quite significant. There are observed shifts in eigen frequencies from 15% up to 35%, results being consistent in both computational environments. The highest shifts in frequency appear in the first four modes of vibration, where most of the modal mass of the mock-up is concentrated.

As expected, most of the influence on structural frequency shift is due to the Azalée shaking table interaction. There are small deformation of the interaction surface between the mock-up and the shaking table, as well as deformations in the anchoring points of the actuators which contribute to these variations in structural frequencies. The ideal case of the structure restrained at the bottom of the foundation requires infinite rigidity on the contact surface of the foundation, which is not realizable in laboratory, experimental environments. This shift of frequency can only be diminished by increasing surface rigidity of the shaking table up to the point where no deformations occur when the seismic platform is loaded.

Related to the influence of the shaking table there were computed the influences of the steel reinforcement on overall rigidity of the structure. A simply designed shell model doesn't consider steel reinforcement influence on the rigidity, so it was considered a multilayered shell structure with equivalent thickness layer, to add local rigidity to the structure. However the influence of the model considering this factor wasn't too significant, as observed in Figure 4.4 and Figure 4.5.

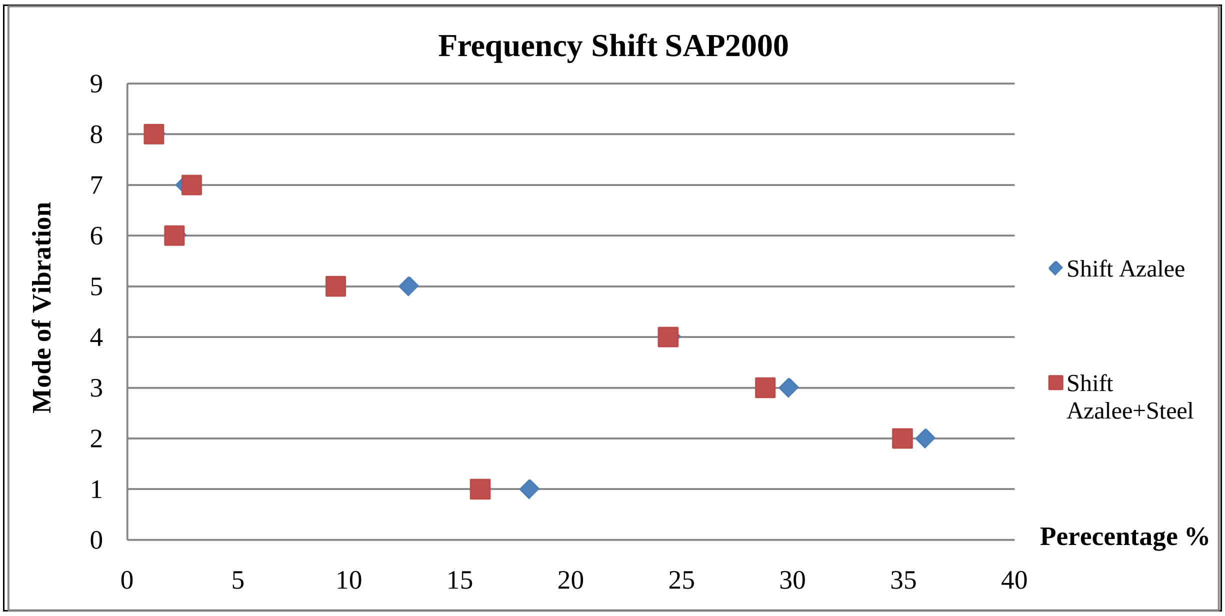


Figure 4.4. Frequency shift in the modal analysis computed in SAP2000 vs14.0.

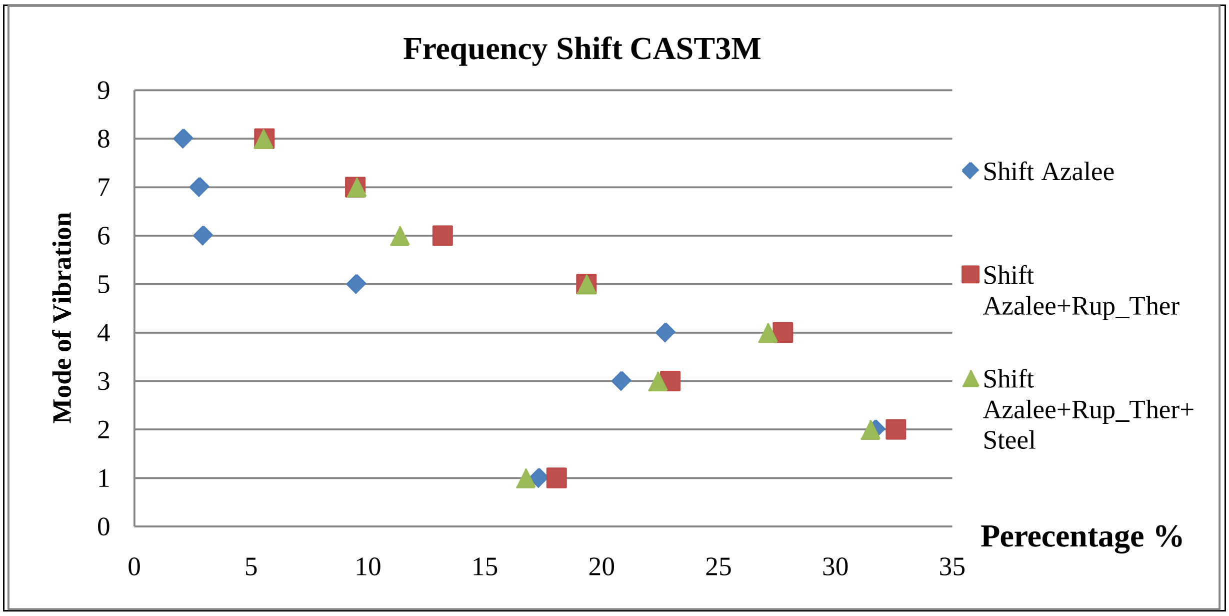


Figure 4.5. Frequency shift in the modal analysis computed in CAST3M.

The model considering the effects of thermal breakers on the overall structural rigidity is very little especially in the first modes of vibration, as we can observe in Figure 4.5. This influence increases slightly in higher modes of vibration where the difference between the model considering only the shaking table interaction and the model which also takes into account the thermal breakers is 5%-10%. This slight increase in frequency shift is due to significant excitation of the modal mass of the slabs, in report to the whole modal mass of the structure.

5. CONCLUSIONS AND PERSPECTIVES

The importance of considering all these local and general influences on boundary conditions is crucial when choosing a seismic signal for the experimental campaign. Considering the fact that this research project aims to study high levels of cracking due to seismic action, the frequency content of the input accelerograms must fit the frequency range of the mock-up.
One of the main conclusions is that the most important contributing factor responsible for the frequency shift is the shaking table (15-35%). The main perspective of this work is to extend the model for nonlinear time-history analysis. This work is ongoing at CEA/EMSI and a predictive model should be operational shortly in order to compare numerical results to the ones available in the experimental campaign.

ACKNOWLEDGEMENT
This project is granted under SERIES project, GA No.227887/2009, within FP7 2007-2013, via Work Package related to Transnational Access, Task TA1, CEA Task leader.

6. REFERENCES

Le Maoult, A. , Bairrao, R. , and Queval J.-C. (2009). Dynamic Interaction between the Shaking Table and the Specimen during Earthquake Tests. *Proceedings of 20th International Conference on Structural Mechanics in Reactor Technology (SMiRT 20),* Finland.
Yakut, A. , Juster-Lermitte, S. , Chaudat, T. , Kazaz, İ. , Ragueneau, F. , Delaplace, A. , Atanasiu, G. M. , Crijanovschi, S. , Scheer, S. , Diler, S. (2010). Experimental and NumericalInvestigation of RC Wall Building With Torsional Effects. *Proceedings of 14th European Conference on Earthquake Engineering(14ECEE),* Paper 811, Macedonia.
Juster-Lermitte S., Chaudat Th. & Courtois A. (2009). SMART 2008 Project experimental tests on a reinforced concrete building subject to torsion, *Proceedings of 20th International Conference on Structural Mechanics in Reactor Technology (SMiRT 20),* Finland.
Eurocode 8, (2006): Design *of structures for earthquake resistance, Part 3: Strengthening and repair of buildings.* Brussels: CEN.
Diler, S. (2011). RUTHERMA DF Thermal break elements. *Design of test specimen for ENISTAT*. Société Schöck France – R&D Internal Report.
Computers and Structures Incorporated, 2010, SAP2000 vs. 14.0, CSI Berkley , California.
CAST3M – http://www.cast3m-cea.fr.