VALIDATION OF 3D FINITE ELEMENT MODELS THROUGH SEISMIC TESTS AT THE ENEA "STRUCTURAL DYNAMIC AND VIBRATION CONTROL" LABORATORY

M. Mongelli*, G. De Canio*, N. Ranieri*, G. Fraraccio*, I. Roselli, M. Baldini*, S. Bonifazi*, A. Colucci*, F. Di Biagio*, G. Fabrizi*, A. Picca*

**ENEA Centro Ricerche Casaccia*
Email: marialuisa.mongelli@casaccia.enea.it

ABSTRACT

A general approach to evaluate the seismic behaviour of structures by finite element analyses (FEA) and shaking table experiments is shown in this poster.
The research activities at the FIM-MATQUAL laboratories of the C.R. Casaccia -Rome, Italy, are mainly devoted to numerical simulations and experimental analysis of innovative systems for the seismic isolation and retrofitting of civil, industrial, and historical buildings, together with the seismic tests of sub-structures and scaled mock-ups, in order to evaluate the isolation/dissipation performance of the anti-seismic devices, and the failure modes of the building's structural parts.
Before testing the structures, finite element stress-strain analyses is carried out to understand the dynamic behaviour and to define the best locations of the monitoring sensors such us accelerometers, strain-gage and LVDT.
The material properties and load boundary conditions of the finite element models (FEM) are updated by the experimental results and the FE analyses have been re-executed to optimize the FE models ant to re-align them to the real structure.
The system mode shapes and frequencies which come from the experimental data and the FEA are matched and compared.
Moreover the numerical simulation of different reinforced techniques are obtained introducing in the original FE model new elements which are representative of typical reinforced structures.
The shaking table tests can simulate the real stress fields in the structures due to the dynamic loads induced by wind, traffic and earthquakes.

KEYWORDS: seismic tests, shaking table, structural behaviour, dynamic analysis, FEM

1. INTRODUCTION

The aim of this paper is to fix a methodology to study the effect of seismic events on civil structures and also on historical monumental cultural heritage by finite element analysis. This approach is commonly used in many fields to solve structural analysis problems.
Generally, engineering analysis can be broadly divided into two categories: classical and numerical methods. Classical methods attempts to solve field problems directly by differential equations based on fundamental principals of physics. They have exact solutions and are applied only for simple cases of geometry , loading and boundary conditions.
The finite element method offers virtually the possibility to solve unlimited problems by using many elements of various regular shapes and different properties. All these elements can be combined to approximate any irregular boundary with an assembly of discrete finite elements under general loading and constraint conditions.
The structural behaviour of the models comes from the analysis of the collective response of all elements.
Italian guidelines published for the cultural heritage (Linee Guida per la valutazione e riduzione del rischio sismico del patrimonio culturale published by Gangemi, dicember 2006) support the finite element approach and

suggest linear static, dynamic and non linear analysis to predict the structural behaviour of a model.
Running finite element static analysis, it is possible to make a map of stress and strain distribution to set the position of sensors before structural experimental tests and obtain an early indications about structural behaviour of buildings.
Running finite element modal analysis, frequencies and modal shapes are obtained.
Moreover, the definition about the material properties of the finite element model are very often unknown: they often come from non-destructive tests. For this reason, it is necessary to validate the finite element models to update and refine them step by step by experimental results obtained from shaking tables and from non-destructive tests.

2. STRUCTURAL ANALYSIS BY FINITE ELEMENT METHOD

Some finite element models will be described in the following pages. They were defined for different projects in many civil fields for which ENEA FIM MATQUAL was working on.
They were generally defined to predict their structural behaviour to seismic attack, and more specifically to support new monitoring methodologies, and to test the effect of new devices or techniques to consolidate and protect them.
The performed finite element analysis decrease the number and the cost of prototypes and experimental tests, fix the research activities especially in areas where the stress or strain concentrations can be very dangerous, and allow to compare modal shapes and frequencies numerically obtained with experimental results.

2.1. SIMMI project - Railway Infrastructures

The main objective of SIMMI project (**I**ntegrated **S**ystem for the **M**onitoring and the **M**aintenance of railway **I**nfrastructures) was to define a damage definition technique for railway systems avoiding the traditional visual inspection. To establish an effective instrument set-up in particular areas such as in bend, or for tracks which lay on bridges, subjected to severe motion, or in areas close to stations, a finite element model (FEM) as well as an experimental model were be tested. In order to improve the method, a full scale experiment on a small portion of a railway was presented.

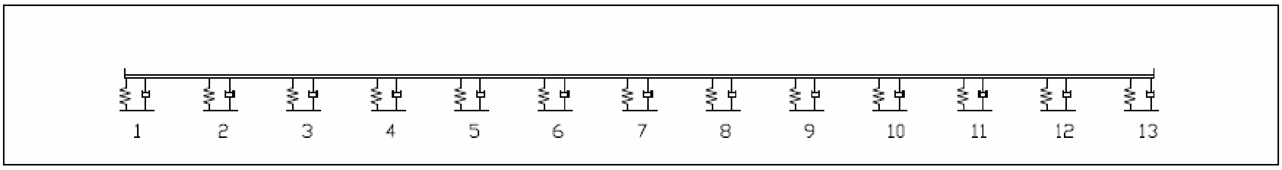


Figure 1 Finite Element Model

A finite element analysis was considered to evaluate the modal behaviour.
The geometry of the realized experimental model represents a 1:1 scaled railway of a total length of 8.60 m and accounts the common measures for the Italian railways. The sleepers are placed at 700 mm, the rails are at 1050 mm, the cross section respects the prescriptive dimensions (UNI 3141).
A finite element model was executed and sleepers and ballast have been modelled by linear springs and viscous dampers (Figure 1). The rail lays on linear springs and dampers representing the rail-pad characteristics. A finite element model with 1D beam elements is constrained to a layer of discrete spring-dampers, represented the interaction between the rail, the pad and the ballast. A simplified cross-section was considered, using a double-tee section. Comparing the rail moment of inertia of the model with the one proposed in the prescriptions, a good agreement between the simplified and the real section is reached. A first order of modes, concerning the full model, which have frequencies under 150 Hz, was obtained by a spring coefficient of 70 MN/m and a damper one of 20KNs/m. A second kind of vibration, concerning single portions of the rail between two adjacent sleepers, called pinned-pinned mode, was found. In order to obtain an estimation of its value, a 70 cm long portion of the

rail was considered, giving a pinned-pinned frequency of 800Hz.
In order to simulate the possible damages by changes in the track, pad and ballast properties many impact test have been executed and different scenarios were considered, tightening bolts in several location with a force lower than the prescriptions. The six accelerations in six different locations have been measured by six accelerometers placed on the track and the structure's frequencies have been evaluated by fast Fourier transform and compared with the frequencies obtained by the finite element modal analysis . Frequencies at about 50-100 Hz are connected with the bending modes of the whole track; while higher frequencies (800 Hz) are connected with the pinned-pinned modes.
The real mode shapes have been used to define if the system is damaged or not by the Modal Assurance Criterion.

2.2. Simmi Project – Civil Building: Bridge of Circunvesuviana in Seiano (Naples)

A finite element modal analysis was realized for a bridge placed in Seiano (Naples) to predict the structural dynamic behaviour and to identify the correct positions for accelerometers and LVDT sensors. The bridge is supported by six pillars and two different railway-sections pinned on (Figure 2).
A finite element 3D model was defined by Nastran solver with exa8 elements and a modal analysis was executed using the material properties shown in Table 1 for the reinforced concrete.

Table 1 Reinforced Concrete

E [Pa]	3.8E+10
δ [kg/m^3]	2400
ν	0.2

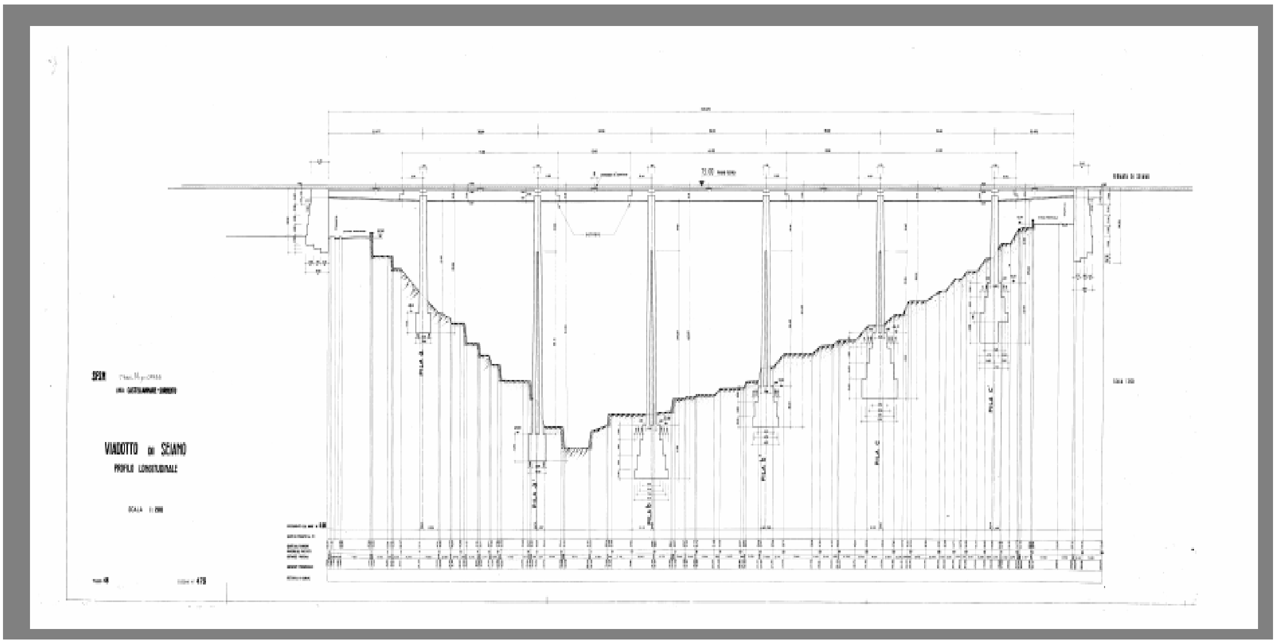


Figure 2 Bridge of Seiano Project

Frequencies and modal shapes are shown in Table 2 and Figure 3, 4, 5, 6

Table 2 Frequencies

Mode	[Hz]
1	1.2424
2	1.4365
3	2.9162
4	3.1727
5	3.8714
6	4.2726

The finite element analysis showed that the most stressed area is located on the fourth pillar from the left side, where the joint was placed. For this reason it is very important to fix the LVDT sensors in these positions and to place the other sensors inside of the two different railway-sections.

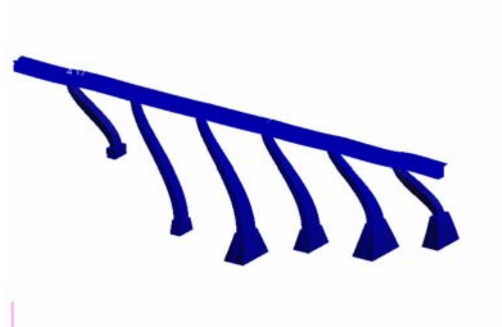


Figure 3 First Modal Shape 1

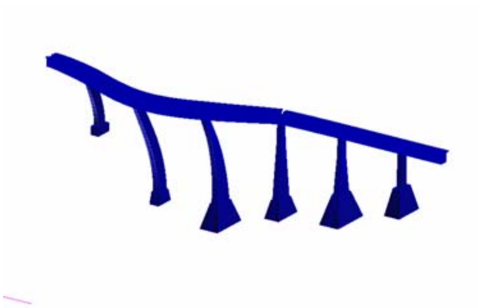


Figure 4 Second Modal Shape

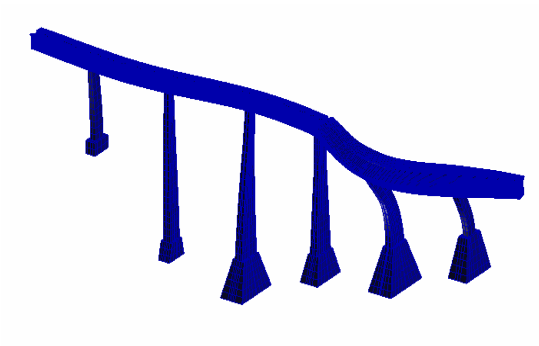


Figure 5 Third Modal Shape

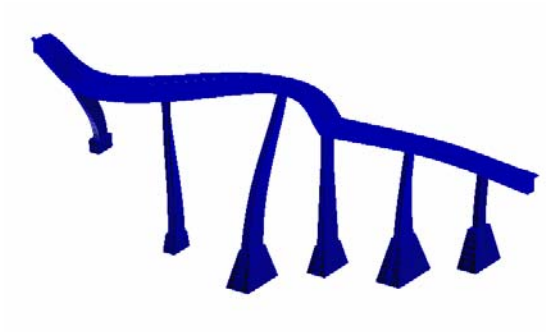


Figure 6 Fourth Modal Shape

2.3. TREMA project: Masonry Buildings

A finite element model was defined before testing two scaled typically Italian masonry buildings (Figure 7) by the shaking table at the ENEA FIM MATQUAL laboratories. The finite element model was realized by solid elements such as in Figure8. The material properties are illustrated in Table 3.
A static and a modal analysis were executed and it was possible to identify the more stressed and strained areas, frequencies and modal shapes (Table 4, Figure 9)

Figure 7 Two historical masonry buildings (scaled 2:3)

Table 3 Materials properties

	Timber beams (red)	Basis (blue)	test 1 Masonry (green)	test 2 Masonry (green)	test 3 Masonry (green)
E [Pa]	1E+10	3.8E+10	9.0 E+8	1.7E+9	3.0E+9
ρ [kg/m3]	700	2400	1500	1600	1700
ν	0.2	0.2	0.2	0.2	0.2

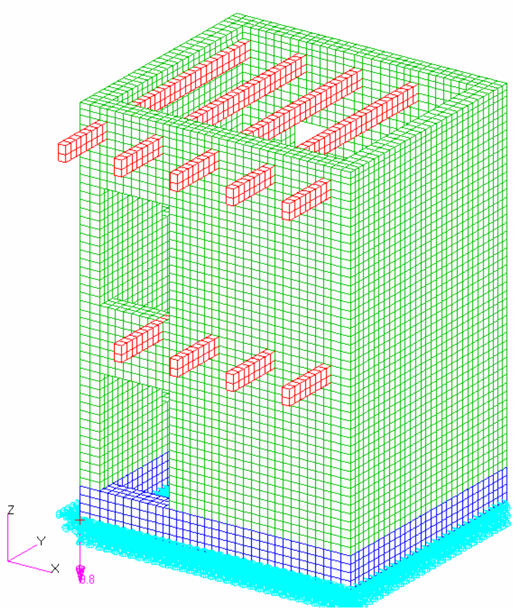
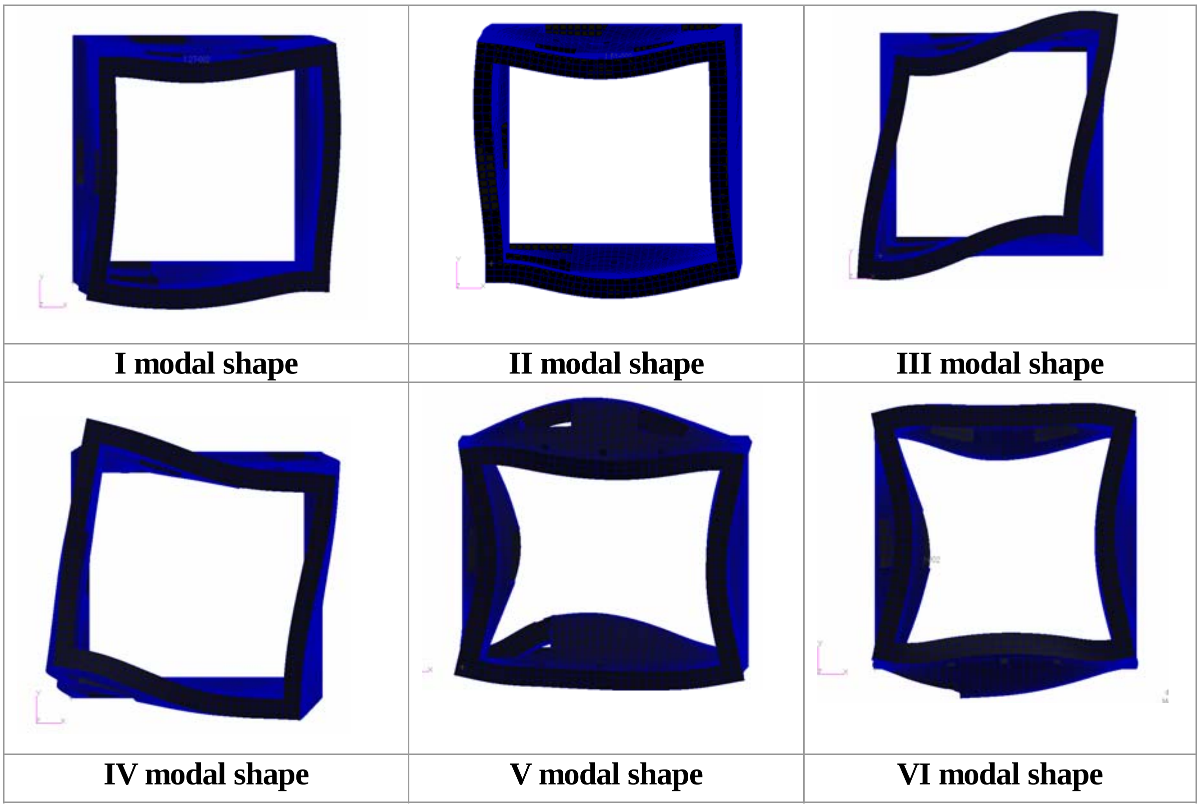


Figure 8 The finite element model

Table 4 Frequencies [Hz]

Mode	test 1	test 2	test 3
1	10.8	14.2	18.0
2	11.2	14.8	18.8
3	12.9	16.8	21.0
4	17.9	23.7	30.3
5	20.6	26.9	34.0
6	21.8	28.1	35.0
7	24.3	31.3	39.2
8	26.2	34.0	42.6
9	29.4	38.0	47.2
10	30.8	39.2	48.0



I modal shape | **II modal shape** | **III modal shape**

IV modal shape | **V modal shape** | **VI modal shape**

Figure 9 modal shapes

3. FUTURE DEVELOPMENTS: INNOVATIVE APPROACH TO ACQUIRE DATA BY EXPERIMENTAL VIBRATION ANALYSIS AND COMPARIDSON WITH FINITE ELEMENT MODELING

An innovative approach to acquire data during the experimental tests on shaking tables was considered at the ENEA FIM-MATQUAL Casaccia laboratories. It consists of a constellation of infrared cameras named 3DVISION which are able of measuring the X, Y, Z, trajectories of retroreflective markers located in specific areas of the structures. The position of the markers is defined by a predictive finite element modal and static analysis that describes the behaviour of the structures.

In the following example, the input time history was derived from an Italian real earthquake accelerogram (Colfiorito) and it was apply to the above described TREMA models of paragraph 2.3.

It was registered by the 3DVISION markers placed on the shaking table in terms of displacement and it was inserted in the load boundary conditions of finite element model. A linear transient analysis was executed and the displacements were post processed for the nodes in Figure 9 (left). In the same figure are shown the combined X,Y acceleration Modulus (absolute value) on the nodes at the top of the building and at the base.

In the figure 10 (left) is reported the time history at node 15305 (base), the maximum value of the Strain energy density shown in Figure 10 (right) occur at time t = 2.6 seconds, when the maximum ground peak acceleration occur. It is evident the hamming effect of the first and second floors, generating the first path of fractures. This innovative approach will be developed to compare numerical and real results by a common user-friendly interface. On one hand, this interface will be able to visualize the finite element model key nodes in the structure to be monitored means of markers located in the corresponding point on the real structure. On the other hand, 3Dvision acquired data can be use as a guide to improve and update the finite element model.

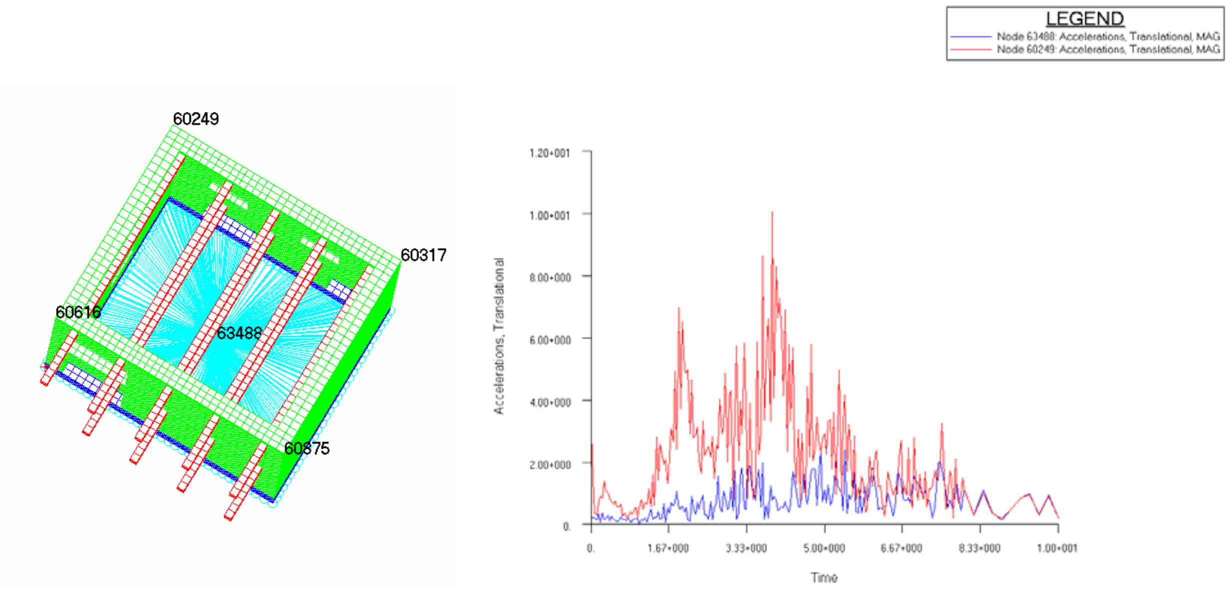


Figure 9 Input time history (left) and position of output requested nodes (right)

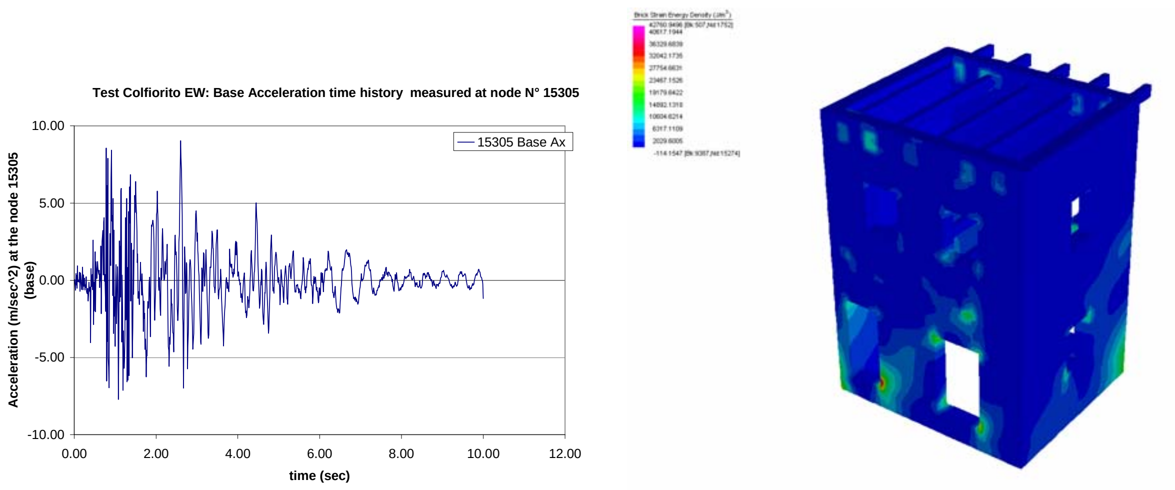


Figure 10 base time history (left) and Strain energy density at time 2.6 s (right)

4. CONCLUSIONS

Italian guidelines published for the cultural heritage support the finite element methodology and suggest linear static, dynamic and non linear analysis to predict the structural behaviour of a civil building.
For this reason same finite element models was described in this paper with the goal to predict the dynamic behaviour of a structure before experimental tests and to define the best locations for the monitoring sensors such us accelerometers, strain-gage and LVDT. About the innovative 3D VISION system, it is also possible to fix the position of retroreflective markers of in specific stressed and strained areas by the finite element analysis. Vice versa by the obtained experimental results it is possible to re-align and update the model such as regards materials properties and load and boundary conditions.

REFERENCES

Munjal M.L., Henckel M.1982,Vibration of a periodic rail-sleeper system excited by an oscillating stationary transverse force, *Journal of sound and vibration* 81,491-500

Garcia Marquez F.P., Schmid F., Collado J.C.,2003, A reliability approach to remote condition monitoring. A railway point case study, *Reliability Engineering and system safety*, 80,33-40

Lombaert G., Degrande G., Kogut J., Francosi S.,2006,The experimental validation of numerical model for the prediction of railway induced vibration, *Journal of sound and vibration,* 29, 512-535.

R. Pinho, C. Casarotti, S. Antoniou, 2007, A comparison of single-run pushover analysis techniques for seismic assessment of bridges**,** Earthquake *Engineering & Structural Dynamics,* **36:10**, 1347-1362

Daigoro Isobe, Michihiro Tsuda, 2003, Seismic collapse analysis of reinforced concrete framed structures using the finite element method, *Earthquake Engineering & Structural Dynamics*,**32:13**, 2027-2046

Lee D.G., Kim H.S., Chun M.H.,2002, Efficient seismic analysis of high-rise building structures with the effects of floor slabs, *Engineering Structures*, 24, 613-623.

Laurent Pasticier, Claudio Amadio, Massimo Fragiacomo, 2008, Non-linear seismic analysis and vulnerability evaluation of a masonry building by means of the SAP2000 V.10 code,
Earthquake Engineering & Structural Dynamics,**37:3**,467-485

Michael N Fardis, 1997, Seismic analysis of RC structures, *Progress in Structural Engineering and Materials*,**1:3**, 57-67