Transactions of the 15th International Conference on
Structural Mechanics in Reactor Technology (SMiRT-15)
Seoul, Korea, August 15-20, 1999

Ultimate Seismic Analysis of Shear Walls by a FEM Code

Donato Aquaro, Giuseppe Forasassi and Corina Harghel

Universita' di Pisa, Italy

ABSTRACT
This paper deals with the numerical simulation of a shear wall under seismic loads. The considered seismic accelerations have such a level to produce the collapse of the structure. This activity was performed in the frame of the International Standard Problem (ISP) proposed by Nuclear Power Engineering Corporation (NUPEC) and sponsored by OECD/NEA/CSNI. Five artificial acceleration histories of increasing level (called sequentially run_1 to run_5) have been used by NUPEC to test two samples having a 'H' shaped cross section from the elastic range to the ultimate state.
This paper illustrates the results of the run_3 numerical simulation , performed at the Dipartimento di Ingegneria Meccanica, Nucleare e della Produzione (DIMNP) of the University of Pisa (Italy) using a commercial FEM code MARC. The numerical displacements and accelerations in several points of the structure as well as the strains in the rebars are compared with the experimental results obtained by NUPEC.

1.INTRODUCTION

In 1995 Nuclear Power Engineering Corporation (NUPEC) from Japan, proposed an open International Standard Problem (ISP) on the ultimate resistance of concrete shear walls under seismic loads. In this context NUPEC offered its experimental test data to an open ISP. NUPEC carried out experimental vibration tests on a concrete structure having a shaped H cross section. [1]. Five artificial acceleration histories of increasing maximum level were applied sequentially at two samples in 5 runs (run_1 to run_5). The first two runs determined an elastic response of the shear walls while the last (run_5) caused the collapse of the structure.
The International standard Problem was sponsored by OECD/NEA/CSNI.
The main objectives of the ISP were to verify the capability of the computer codes in modeling the seismic shear walls of reactor buildings up to the ultimate state. The Dipartimento di Ingegneria Meccanica, Nucleare e della Produzione (DIMNP) of Pisa University (Italy) was going to partecipate at this ISP using the commercial FEM code MARC. But a severe numerical instability of the used code was discovered during the calculations. The bug was fixed but the simulation was ended too late respect to the scheduled time.
Nevertheless the results obtained at DIMNP have a particular interest because a FEM dynamic analysis was performed and the model was built with an element number much greater than th0se used from the ISP participants. In fact the greatest FEM dynamic model

used in the ISP had 1521 Degree of Freedom (DOF) while the DIMNP model has 17688 DOF [2].
Therefore the numerical results can be compared more accurately with the experimental ones.

2. GEOMETRICAL AND MECHANICAL CHARACTERISTICS OF THE SHEAR WALL

Figure 1 shows the geometry of the specimens experimentally tested. The specimen cross section is 'H' shaped. The web wall is 75 mm thick, 3000 mm long and 2020 mm high. Two layers of steel bars were used for reinforcement in the web (Φ=6mm bars, located horizontally and vertically at a 70 mm pitch) as well as in the flanges (Φ=6mm bars located horizontally at a 70 mm pitch and vertically at a 175 mm pitch. The flange vertical reinforcement has a 70 mm pitch at the intersection with the web. Figures 3-4 show the horizontal and vertical reinforcement bars in the web and in the flanges respectively.

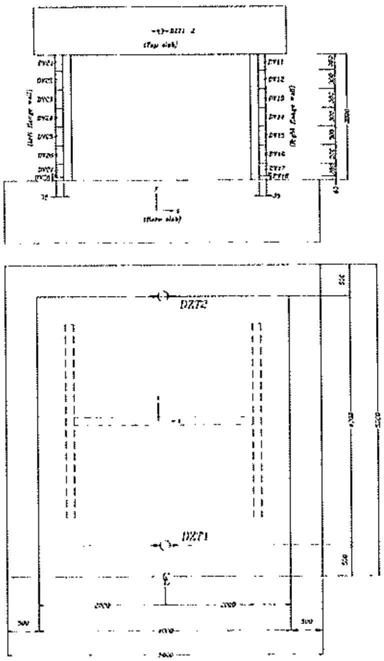


Fig. 1 - Geometry of the shear wall specimen

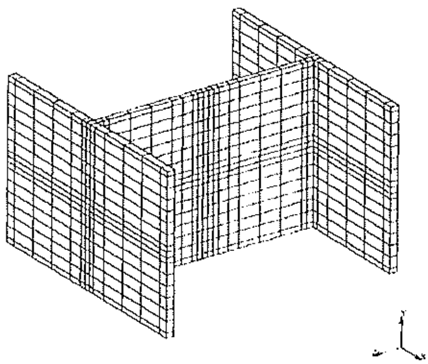


Fig.2 - Shear wall mesh used in the numerical simulations

The specimen is connected on the base at a squared (5 m of side) 1 m thick concrete slab. Similar squared concrete slab (4 m of side and 0.76 m thick) is connected on the top. Moreover the top slab was loaded by 92.9 ton of additional lead masses fixed at the upper and lower surface of the top slab, in order to make the specimen centroid coincident with the top slab geometrical centre of gravity.
The total weight of the upper masses is equal to 122 ton.
The mechanical characteristics of the steel bars and concrete are reported in tab.I. In the tests, the specimens were loaded by accelerations parallel to web plane (z axis in fig.1) applied at

the base slab, using the NUPEC's large scale shaking table at the Tadotsu Engineering Laboratory.

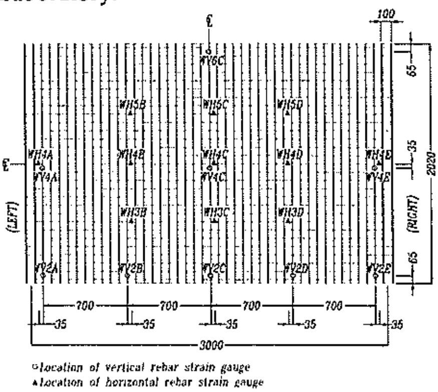


Fig. 3 - Reinforcement steel bars pattern in the web and locations of the bar strain measurements

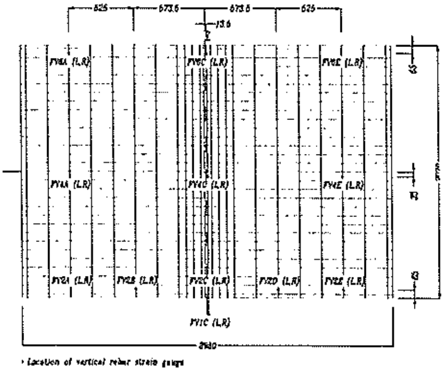


Fig. 4 - Reinforcement steel bars pattern in the flanges and locations of the bar strain measurements

The duration of the acceleration transient was equal to 12 sec.

The top slab horizontal displacement and acceleration have been recorded during the tests. Moreover several displacement transducers were located on the flange external surfaces (in correspondence of the intersections with the web middle plane (Fig.1)) in order to measure the relative vertical displacement between several points along the flange height (displacements indicated with DV11-DV28 in Fig.1).

Several strain gauges have been put on the web and flange steel bars in order to measure the bar strains. Figures 3 and 4 show the positions and the flags of the used strain gauges on the web and flanges respectively.

Properties	concrete	steel
Young Modulus (Kg/mm^2)	2337.5	18800
Poisson Modulus	.155	0.3
Yielding stress (Kg/mm^2)	2.8	39.1
Density (Kg/m^3)	2391	7800

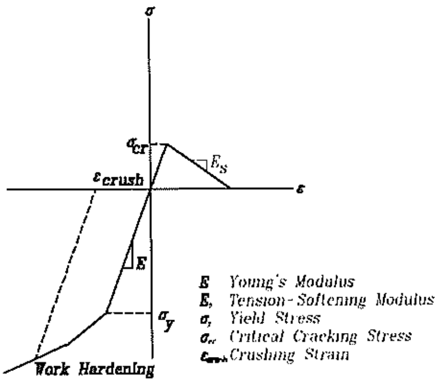
Tab. 1 - Mechanical characteristics of the concrete and the reinforcement bars used for the construction of the shear wall specimens

3. FEM MODEL USED FOR THE SIMULATION OF SHEAR WALL ULTIMATE STATE UNDER SEISMIC LOADS

Figure 2 shows the model used in the simulation of the shear wall ultimate state under seimic loads. The mesh is built by 765 concrete brick elements and 799 reinforcement bar brick elements. Both the elements have 21 nodes. Therefore the model was made by 1564 elements and 5896 nodes. The total DOF number is 17688. The upper masses were simulated by lumped masses tied to shear wall upper nodes and located at such a quote that the centroid of specimen and geometric center of top slab were coincident.

The steel bars were simulated as an isotropic elastic perfectly plastic material using the Von Mises criteria as yielding function and the properties values reported in Tab.I.

The qualitative diagram of concrete constitutive equation is shown in Fig. 5.
The compressive strenght is simulated by means of an elastic-plastic material following the Von Mises criteria and having a strain limit equal to the crushing strain. The material looses its integraty and all load carrying capability when the compressive strain overcomes the crushing strains ,ε_{crush}.
The concrete tensile strength is limited from the critical cracking stress, σ_{cr}, which produces a crack perpendicular to its direction. The material looses the tensile strenght unless a non zero value of tension softening modulus, E_s, is included.



In the numerical analysis, the concrete was simulated as an elastic perfectly plastic material in compression range. Besides the properties value, reported in Tab.I, The following values were used in the analysis:
E_s=0, σ_{cr}=0.29 Kg/mm^2 ,ε_{crush}=2.4 10^{-3}
Shear retention factor (after the cracking closure)= 0.5

Fig. 5 - Constitutive Equation of the concrete

The analysis was performed with the commercial FEM code MARC [3]. The time integration algorithm used was the NewmarK beta algorithm. The Rayleigh damping formulation was used considering only the stiffness matrix factor put equal to β=1.5916 10^{-3}. This figure means that the damping at 15 Hz is equal to ξ=0.075 and the motion at frequency upper than 200 Hz is an aperiodic damped motion.
This choice was suggested from the shear wall natural frequencies calculations performed before starting the transient analysis.

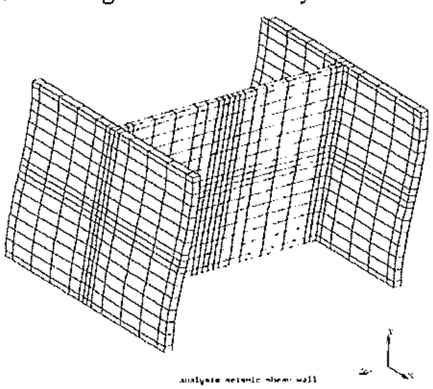


Fig.6 - Characteristic shape correspondent to the 1° natural mode (f=14.85 Hz)

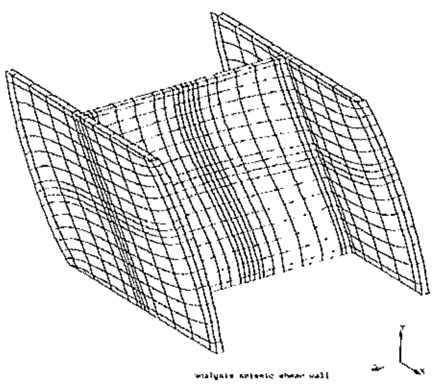


Fig.7 - Characteristic shape correspondent to the 2° natural mode (f=21.37 Hz)

Figures 6-9 illustrate the characteristic shapes correspondent to the first 4 modes. Table II reports the values of the first 5 natural frequencies. The experimental first frequency (13.1-13.2 Hz) is a bit lower than the calculated ones.

Only the last 3 excitations (run_3 to run_5) were simulated in order to reduce the calculation time being elastic the shear wall response to the first two acceleration histories and therefore less interesting.
The time step used in the analysis is equal to 0.01 sec and therefore each 12 sec transient was performed in 1200 increments. The CPU required by each 12 sec transient was about 160 h (on IBM RS6000 hardware with 256 MB of RAM).

	1° mode	2° mode	3° mode	4° mode	5° mode
Frequency(Hz)	14.85	21.37	44.97	68.9	85.15

Tab. II - Natural frequencies of the shear wall specimen

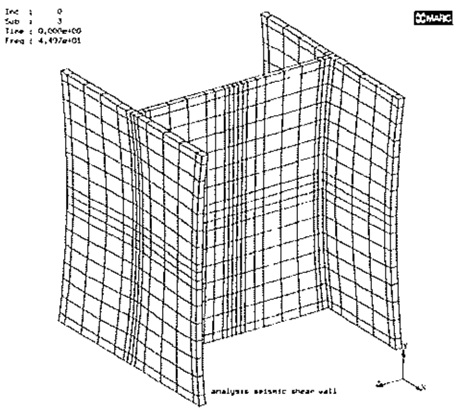


Fig.8 - Characteristic shape correspondent to the 3° natural mode (f= 44.97 Hz)

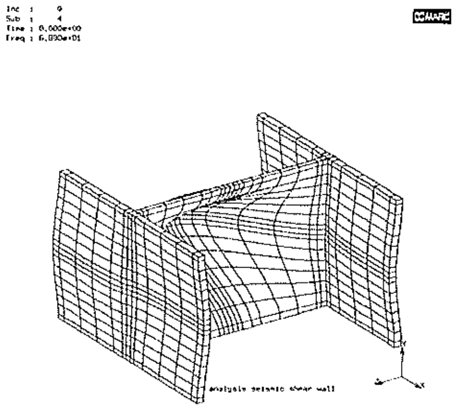


Fig.9 - Characteristic shape correspondent to the 4° natural mode (f=68.9 Hz)

4. - MAIN RESULTS RELATIVE TO RUN_3 TRANSIENT AND COMPARISON WITH THE EXPERIMENTAL DATA

The excitation used in the run_3 transient, shown in fig.10, reaches the maximum value of $a=5765\ mm/s^2$ at t=3.54 sec.
The top slab acceleration obtained in the experimental test and the numerical ones are illustrated in Fig.11 and 12. These signals are compared in the Fig.13 considering only the time range between 2.5 sec and 7 sec (that is the time period characterized by the greatest acceleration values).

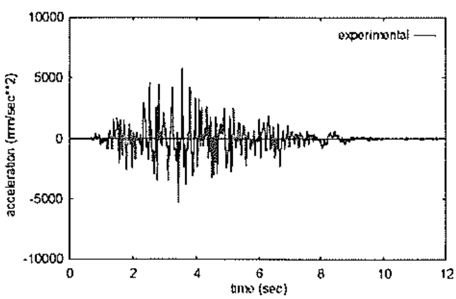


Fig. 10 - Acceleration input of the run_3 excitation

The experimental maximum Z acceleration of the top slab ($a=7043\ mm/s^2$) is obtained at t=2.82 sec , while the numerical ones occurs at the almost same instant (t=2.83 sec) but is 30 % greater ($a=7043\ mm/s^2$).
This discrepancy is strongly reduced in the other instants of the time.
The comparison between the experimental Z relative displacement of the top slab and the calculated one is shown in Fig.14.

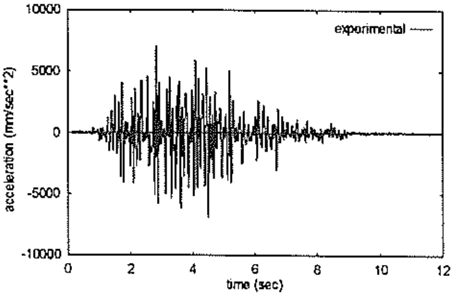


Fig.11 - Experimental Z acceleration of the top slab obtained during the run_3 transient

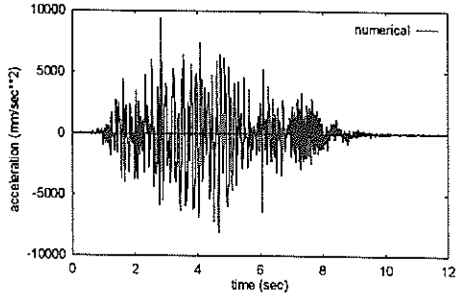


Fig.12 - Calculated Z acceleration of the top slab obtained during the run_3 transient

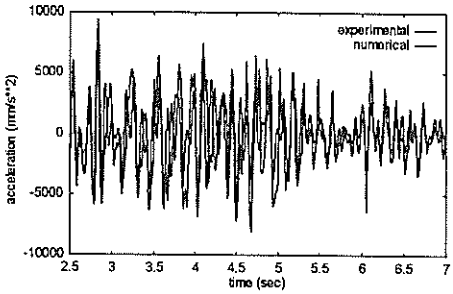


Fig.13 - Comparison between the experimental Z acceleration of the top slab and the calculated ones

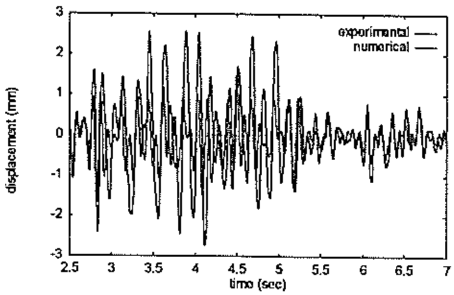


Fig.14 - Comparison between the experimental Z relative displacement of the top slab and the calculated ones

The experimental relative Z displacement of the top slab reaches the minimum value d=-1.63 mm at t=2.82 sec and the maximum value d=1.49 mm at t=4.5 sec. The correspondent calculated values are the following:

- minimum value d= -2.733 mm at t=3.89 sec
- maximum value d= 2.582 mm at t=4.11 sec

The reason of the greater numerical results respect with the experimental ones could be due to a greater stiffness of the numerical model as well as to a numerical damping greater than the actual. In fact after the run_3 was measured a damping equal to 2.5% for the first frequency while the Rayleigh damping used in analysis corresponds to a 7.5% damping.

4.1 - Strain and Stress of the steel reinforced bars

In the experimental tests performed in Japan the strains of the steel reinforced bars were recorded by means strain gauges. Figures 3 and 4 show the measurement points. Figures 15-26 compare the experimental values with the numerical ones. In all the cases but one (Fig.21) the calculated strains are greater than the experimental values in the 3-5 sec range while they are coincident or a bit smaller in the other parts of the transient. The experimental maximum horizontal strains in WH3C point (Fig.15)(on the web axis at 500 mm from the bottom) is $196\ 10^{-6}$ while the correspondent numerical value is equal to $469\ 10^{-6}$. After t=5 sec the coincident value is about 100 10^{-6}. Figure 16 shows the stress in the bar at the same point. The maximum numerical value is 9 Kg/mm.

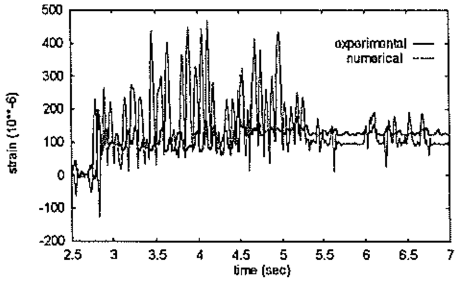


Fig. 15 - Steel bars strains in the Wh3c point

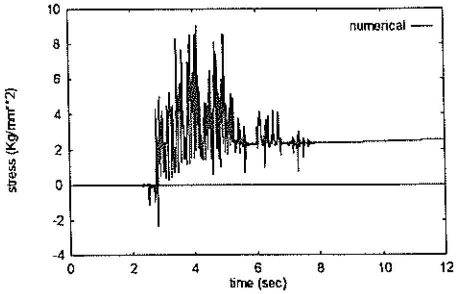


Fig. 16 - Steel bars stress in the Wh3c point

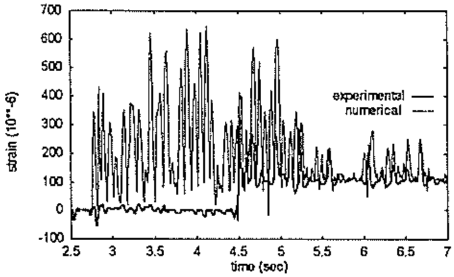


Fig. 17 - Steel bars strains in the Wh4d point

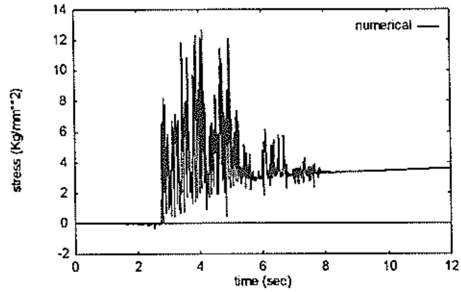


Fig. 18 - Steel bars stress in the Wh4d point

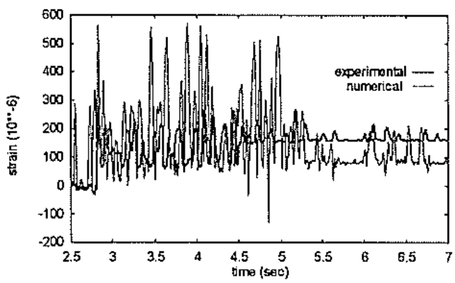


Fig. 19 - Steel bars strains in the Wh5b point

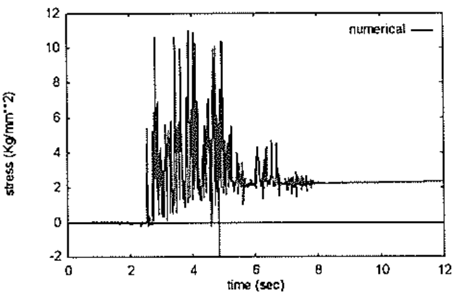
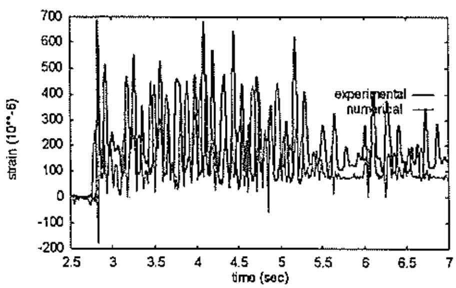


Fig. 20 - Steel bar stress in the Wh5b point

Figure 21 illustrates the horizontal strain diagram of the Wh5c point. The experimental and numerical maximum values are 682 10^{-6}. and 527 10^{-6}, respectively . The strains in the symmetric point Wh5d would be practically similar (as occurs in the numerical analysis(see Fig.23)but the experimental values in this point are about five times smaller.
Figures 25-26 illustrate the steel bars vertical strains in the flanges (point Fv2cr).
Also in this case the numerical results overestimate the experimental values. The Fv2cr is located at 65 mm from the bottom nodes were the accelerations have been applied. Therefore the numerical results depend strongly on the nearness of load application points. But NUPEC gave only this signal for strains in the left as well as right flange, therefore it isn't possible to perform a more complete comparison.



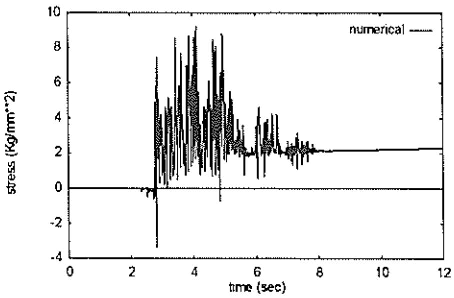
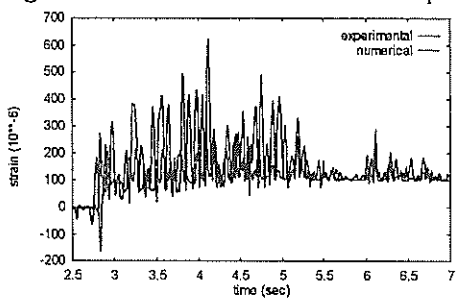


Fig. 21 - Steel bars strains in the Wh5c point Fig. 22 - Steel bar Stress in the Wh5c point



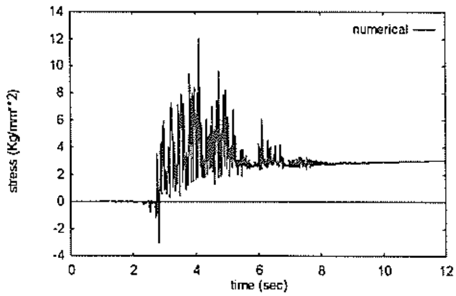
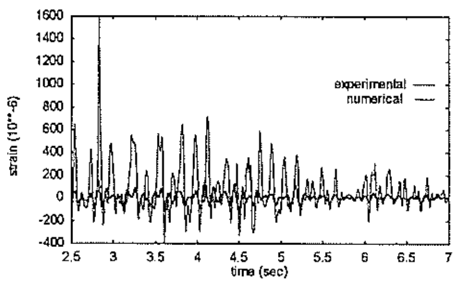


Fig. 23 - Steel bars strains in the Wh5d point Fig. 24 - Steel bar stress in the Wh5d point



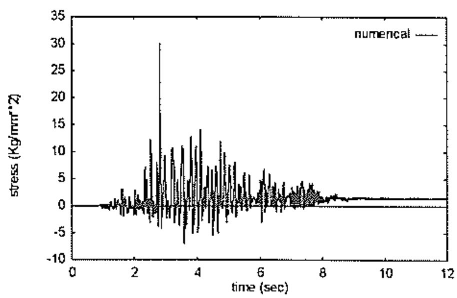


Fig. 25 - Steel bar vertical strains in the Fv2cr flange point

Fig. 26 - Steel bar stress in the Fv2cr flange point

5. - CONCLUSIONS

The shear wall numerical simulation of the run_3 excitation has shown a sufficient agreement with the experimental results in terms of global behaviour of the specimen and the acceleration and displacement values. The greater calculated figures are perhaps due to a greater stiffness of the model and a greater damping used in the analysis. The strain of the steel bars are perfectly coincident after t=5 sec but greater in the first part of transient. The simulation required a lot of calculation time (about 160 h per transient).

6 - REFERENCES

[1]- NUPEC-'Spec.Rep.of SSWISP on NUPEC's Seismic ultimate dynamic response test-OECD NEA- Aug.94

[2]-Kitada Y.et alii'Report on SSWISP organized by OECD/NEA/CSNI- 14th In.Con.SMiRT -vol5-Lyon 1997

[3] Marc Analysis Corp. ' User Manual ' Vol. A - Palo Alto Ca - 1998