Seismic Analysis

INTERSECTII
INTERSECTIONS
http://www.intersections.ro

Seismic analysis using Robot Millennium

Mihai Nedelcu

Department of Structural Mechanics , Technical University of Cluj-Napoca, 400020, Romania

Summary

The paper presents the modelling and structural analysis of a complex structure under seismic loads using the Robot Millennium software package. In order to test the output results, the author chose a building already designed in the documentation [1].

The above mentioned documentation was created as a guideline for applying the design standard P100-1/2006. Following the given structure architecture and loads, the author of this paper remodelled the structure using Robot Millennium v.20.1.

The objective of the paper is to show:

1. *the Robot Millennium instruments used for modelling this type of structures*
2. *various modelling ways for the same structure highlighting the most convenient one (in the author's opinion)*
3. *the results accuracy of the analysis in comparison with the output given by the work of Professor Tudor Postelnicu*

KEYWORDS: seismic analysis, modal analysis, stories, rigid links, panel cut, reinforced concrete walls.

INTERSECTII
INTERSECTIONS
http://www.intersections.ro

Mihai Nedelcu

1. INTRODUCTION

The chosen building is fully described in the 3rd example of a very professional and useful documentation: "P100-1/ Building seismic design, volume 2-B. Comments and calculus examples", responsible author: Tudor Postelnicu, Ph.D., Prof., U.T.C.B. For the seismic analysis, the authors of the above mentioned documentation used the lateral force method and 3D linear-elastic computation generated by means of the ETABS program. The author of this paper wanted to see if the Robot Millennium software package could be a good alternative in this kind of structures analysis.

The analyzed building is located in Bucharest and has:

- 3 underground levels (h=3m) + ground floor (h=6m) and 10 stories (h=3m)
- 5 longitudinal spans x 8m and 5 transversal spans 2x7+1x4+2x7m

The structural characteristics:

- R.C. walls (both uncoupled and coupled by spandrel walls), columns and beams
- concrete class C25/30
- steel PC52
- a_g=0.24g, T_c=1.6sec, ductility class H, importance coefficient γ_1=1.2

A current floor plan is shown in Figure 1.

2. MODELLING PRESENTATION

For the seismic analysis, the structure is considered fixed at the ground floor base. Using Robot Millennium v.20.1, the author modeled the structure using bars for columns/beams and panels for slabs/ R.C. walls (see Figure 2). Of course after the mesh generation (even with large finite elements) the great number of equations (of order 10^5) led to a significant slowing down of the analysis. As a good alternative the Rigid Links additional attribute can be used instead of panels for the slabs modeling. By introducing the Membrane rigid link, the user can connect the nodes of each floor according to any DOF, in this case the X,Y displacements and the RZ rotation. And so the slab effect as a rigid body is fully covered. At the same time the beams have to be modeled as T-section, taking into account the corresponding slab rigidity. A slab width of 3xhp (the slab thickness) was taken on each side for the interior beams and of 2xhp on each side for the marginal beams. Having no slab finite elements reduces considerably the analysis time with absolutely no damage to the results. The model is presented in Figure 3.

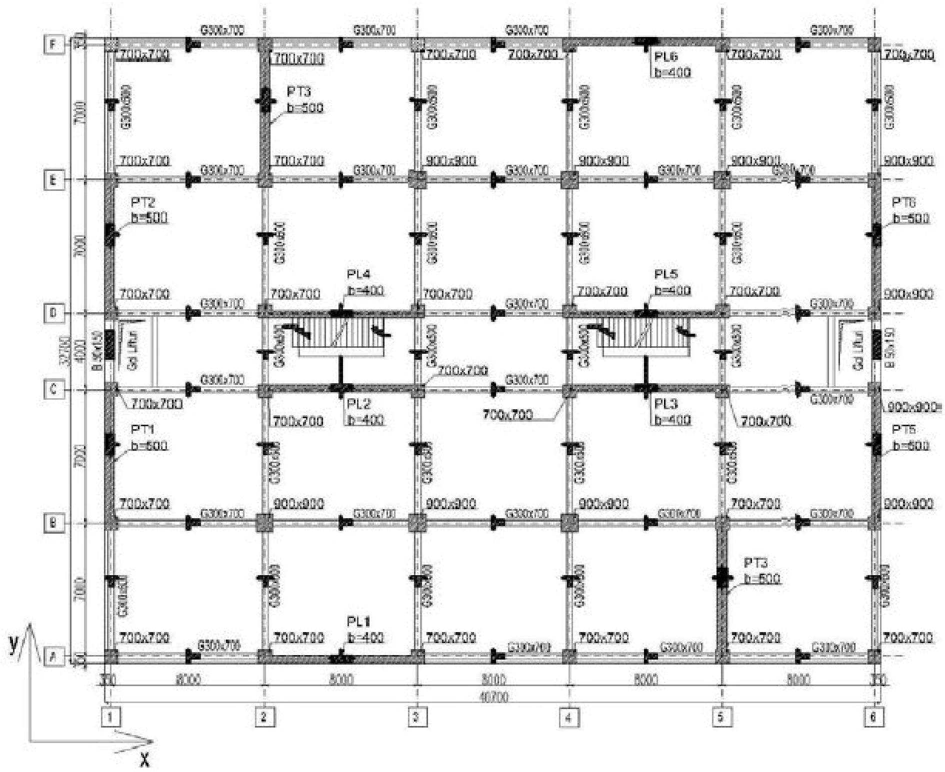


Figure 1. The building current floor plan

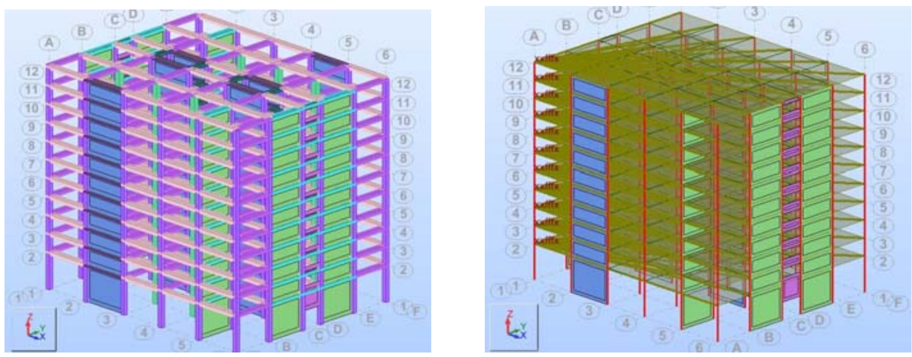


Figure 2. The structure model showing: a) section shapes, b) rigid links

Following the instructions of P100-2006, the rigidity of the elements is taken differently, depending on the type of analysis. First, the goal is to make the modal analysis, in which the rigidity of walls, columns and beams is taken $EI=0.5E_cI_c$. As for the spandrel walls, their Young modulus is $E_{spandrel}=0.4^* E_{walls}$ and the other sectional characteristics: $A=0.2A_c$, $I=0.2I_c$. Reducing the rigidity of the elements, the elements are considered in a plastic stage and so the building behavior during an earthquake is well approximated according to P100-2006. To achieve that the user can define a new material with a modified Young modulus, in this case named C25/30_0.5 (*Job Preferences/Materials/Modification*). The characteristics of a longitudinal wall are set as presented in Figure 3. As for the spandrel walls the double reduction is achieved by multiplying the thickness of 50cm with the 0.2 factor.

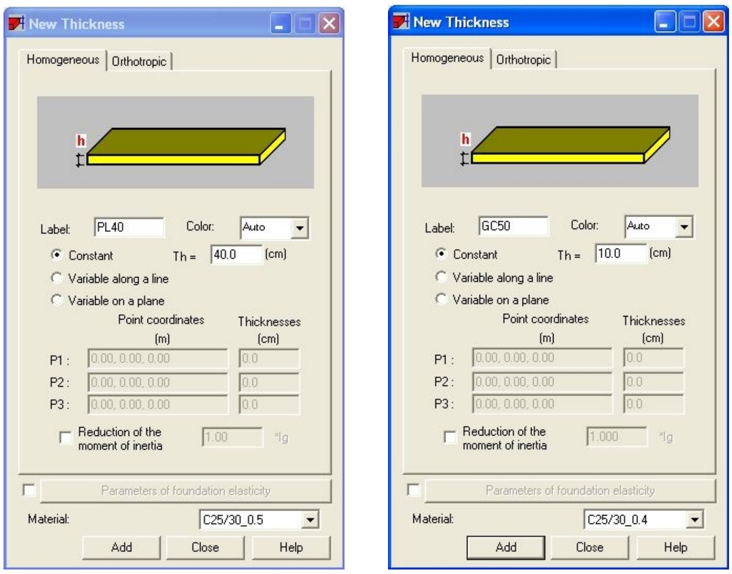
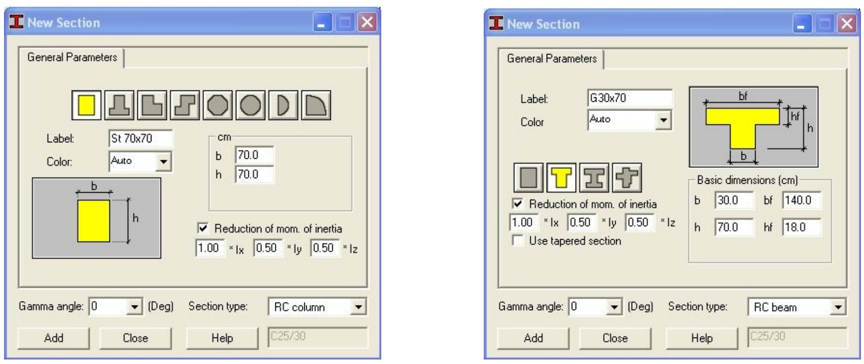


Figure 3. Defining the section properties of: a) longitudinal walls, b) spandrel walls

As for the bar elements, there is an alternative. The program allows the reduction of moment of inertia according to local axes x, y, z (see Figure 4).



Seismic analysis using Robot Millennium

Figure 4. Defining the properties of: a) columns 70x70cm, b) beams 30x70cm

Another problem in the modal analysis is the modeling of the additional eccentricities: ± 5% in each direction. The code explains in detail how their effect has to be taken into account. The level seismic forces, computed by hand or by some user defined computer program have to be applied to the resistance elements depending on their rigidity. Because there are 2 main directions, for each one 2 signs and the mass eccentricities can also be positive or negative, there are 8 combinations to be made. This work is time consuming and the probability of user errors is highly increased. A good alternative seems to be the *Modal Analysis Parameters/Definition of mass eccentricities* that the program offers (see Figure 5). The sign of eccentricity can be changed but is impossible to have both signs in the same model. So, different models have to be made. Of course, if the structure is someway symmetrical, the work is much reduced.

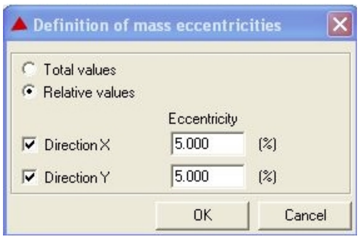


Figure 5. Introducing the additional mass eccentricities

In the 3rd example the loads are generally given. Because of this "generality" the results of this paper are not identical to the original ones, but the differences are acceptable. Given the loads, the load to mass conversion is made automatically. The size of FE-s should be set as big as possible, since from the modal analysis point of view the mesh refinement will not change the results. Doing that the modal analysis was made in a few seconds. The results are compared to the original ones (see Figures 6 and 7). Also the shape of the eigenvectors for the first 3 modes is shown in XY view (see Figure 8). The additional eccentricities effect for the first two modes can be seen.

Mode	Frequency (Hz)	Period (sec)	Rel.mas.UX (%)	Rel.mas.UY (%)	Cur.mas.UX (%)	Cur.mas.UY (%)
1	1.369	0.731	70.757	0.230	70.757	0.230
2	1.552	0.644	71.106	71.161	0.349	70.931
3	2.042	0.490	71.517	73.206	0.412	2.046
4	5.900	0.170	88.945	74.492	17.428	1.285
5	6.115	0.164	90.565	90.305	1.620	15.814
6	7.973	0.125	90.829	90.825	0.264	0.520
7	12.562	0.080	93.355	93.294	2.526	2.469
8	13.144	0.076	95.783	95.461	2.428	2.168
9	13.179	0.076	95.798	95.497	0.015	0.036

Figure 6. The Robot Millennium modal analysis results

Mod propriu	T_k (sec)	directie X	directie Y	directie Rz	cumulat X	cumulat Y	cumulat Rz
		m_k	m_k	m_k	Σm_k	Σm_k	Σm_k
1	**0.728**	**73.53**	0.04	0.00	73.53	0.04	0.00
2	**0.650**	0.04	**75.01**	0.00	73.57	75.05	0.00
3	**0.517**	0.00	0.00	**74.98**	73.57	75.05	74.98
4	0.172	**19.17**	0.03	0.00	92.73	75.08	74.98
5	0.168	0.03	**17.62**	0.00	92.76	92.71	74.98
6	0.134	0.00	0.00	**17.75**	92.76	92.71	92.73
7	0.078	0.01	**5.03**	0.00	92.77	97.73	92.73
8	0.078	**5.05**	0.01	0.00	97.82	97.74	92.73
9	0.063	0.00	0.00	**5.05**	97.82	97.74	97.78

Figure 7. The original modal analysis results

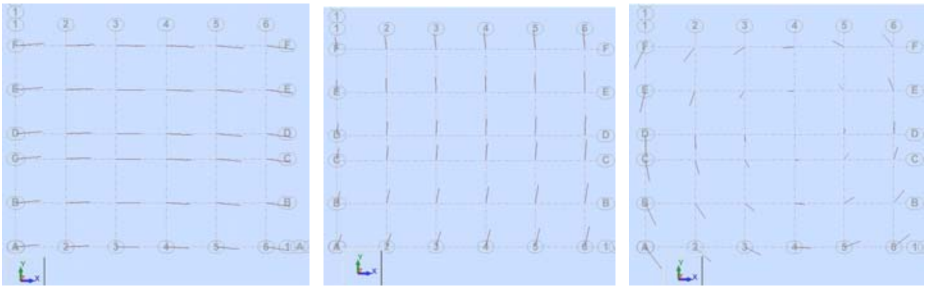


Figure 8. The shape of the eigenvectors for the first 3 modes in XY view

The next step is to define the seismic analysis (see Figure 9). The behavior factor (q) has different values according direction X and Y. For the ductility class H, on X-longitudinal direction, the walls are considered as cantilevers ($q = 4\alpha_u / \alpha_1$) while on Y-transversal direction, the walls are coupled, and so results a higher q ($q = 4\alpha_u / \alpha_1$).

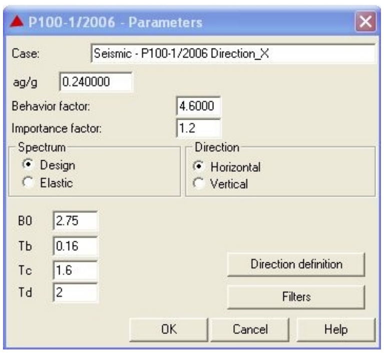


Figure 9. Introducing the seismic analysis parameters

Since the seismic analysis is concerned, only the special load combinations are introduced. To assess the deformations, no changes are made to the FE mesh or to the sectional rigidities. The allowed displacements are given by P100-2006 in relation to the elastic drift.

The program offers another useful tool by allowing the user to define the stories of the building: *Geometry/Code Parameters/Stories.* The elements of each storey are presented in a different color in the Figure 10. The walls are not colored because of the large FE mesh.

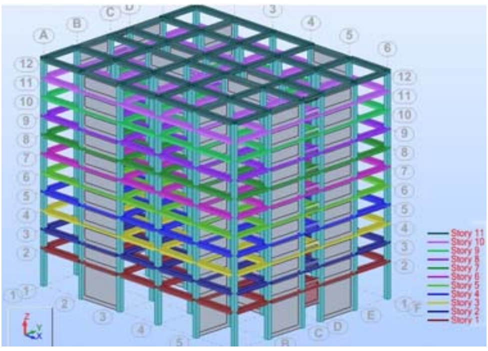


Figure 10. The stories display

The characteristics of each storey can be seen by the *Story Table/Values* option. In the Figure 11, are presented the mass, the gravity and torsion centers, eccentricity e_0 and the applied additional eccentricities e_2.

Name	Mass (kg)	G (x,y,z) (m)	T (x,y,z) (m)	Ix (kgm2)	Iy (kgm2)	Iz (kgm2)	ex0 (m)	ey0 (m)	ex2 (m)	ey2 (m)
Story 1	589008.071	20.00 16.00 5.6	20.00 16.00 5.6	48458578.237	137822167.335	185487768.660	0.0	0.00	2.00	1.60
Story 2	481217.949	20.00 16.00 9.0	20.00 16.00 9.0	44481317.600	94352002.734	138747026.140	0.0	0.00	2.00	1.60
Story 3	481217.949	20.00 16.00 12.	20.00 16.00 12.	44483548.229	94354233.363	138747026.140	0.0	0.00	2.00	1.60
Story 4	481217.949	20.00 16.00 15.	20.00 16.00 15.	44485778.858	94356463.992	138747026.140	0.00	0.00	2.00	1.60
Story 5	481217.949	20.00 16.00 18.	20.00 16.00 18.	44488009.488	94358694.622	138747026.140	0.0	0.00	2.00	1.60
Story 6	481217.949	20.00 16.00 21.	20.00 16.00 21.	44490240.117	94360925.251	138747026.140	0.0	0.00	2.00	1.60
Story 7	481217.949	20.00 16.00 24.	20.00 16.00 24.	44492470.746	94363155.880	138747026.140	0.0	0.00	2.00	1.60
Story 8	481217.949	20.00 16.00 27.	20.00 16.00 27.	44494701.375	94365386.509	138747026.140	0.0	0.00	2.00	1.60
Story 9	481217.949	20.00 16.00 30.	20.00 16.00 30.	44496932.005	94367617.139	138747026.140	0.0	0.00	2.00	1.60
Story 10	481217.949	20.00 16.00 33.	20.00 16.00 33.	44499162.634	94369847.768	138747026.140	0.00	0.00	2.00	1.60
Story 11	433533.572	20.00 16.00 35.	20.00 16.00 35.	42832327.656	75282000.722	118019847.868	0.0	0.00	2.00	1.60

Figure 11. Story Table: Values

Using *Story Table/Displacements* option, the stories maximum absolute and relative displacements (d_{re}) can be seen (Figure 12).

ISSN 1582-3024

INTERSECTII
http://www.intersections.ro

	MAX UX (cm)	Node	MAX UY (cm)	Node	dr UX (cm)	dr UY (cm)	MIN UX (cm)	Node	MIN UY (cm)	Node
Case 7	seismX									
Story 1	0.3073	563	0.0279	563	0.2436	0.0594	0.0637	1206	-0.0315	580
Story 2	0.5406	627	0.0464	627	0.1864	0.0985	0.3543	579	-0.0521	644
Story 3	0.8121	691	0.0670	691	0.2295	0.1420	0.5826	619	-0.0750	708
Story 4	1.1158	755	0.0890	755	0.2664	0.1886	0.8494	683	-0.0997	772
Story 5	1.4414	819	0.1117	819	0.2986	0.2368	1.1428	747	-0.1251	836
Story 6	1.7808	883	0.1345	883	0.3257	0.2850	1.4551	811	-0.1505	900
Story 7	2.1265	947	0.1568	947	0.3478	0.3323	1.7786	875	-0.1755	964
Story 8	2.4725	1011	0.1781	1011	0.3655	0.3776	2.1070	939	-0.1995	1028
Story 9	2.8125	1075	0.1983	1075	0.3785	0.4204	2.4340	1003	-0.2221	1092
Story 10	3.1507	1139	0.2171	1139	0.3929	0.4604	2.7577	1091	-0.2433	1147
Story 11	3.4344	10	0.2288	393	0.3672	0.4854	3.0671	1131	-0.2566	12

Figure 11. Story Table: Values

And so the elastic and inelastic drift can be computed. Table 1 is showing a comparison between the results of this paper and the original ones for X direction.

Table 1. Name of the table

	Robot Millennium			Example 3	
Level	dreX	driftSLS	driftULS	driftSLS	driftULS
	(cm)	<0.005	<0.025	<0.005	<0.025
GF	0.1470	0.0009	0.0042	0.0009	0.0040
E1	0.1309	0.0016	0.0075	0.0015	0.0070
E2	0.1590	0.0020	0.0091	0.0018	0.0084
E3	0.1801	0.0022	0.0103	0.0020	0.0095
E4	0.1948	0.0024	0.0111	0.0022	0.0102
E5	0.2041	0.0025	0.0116	0.0023	0.0105
E6	0.2086	0.0026	0.0119	0.0023	0.0107
E7	0.2090	0.0026	0.0119	0.0023	0.0106
E8	0.2063	0.0025	0.0118	0.0022	0.0104
E9	0.2018	0.0025	0.0115	0.0022	0.0102
E10	0.1940	0.0024	0.0111	0.0021	0.0098

The next step is to calculate the internal forces. The elements rigidities are to be changed. For walls and bar elements CR 2-1-1.1 gives EI= E_cI_c and for the spandrel walls EI= 0.4(E_cI_c). As the building flexibility is decreasing, the internal forces will be calculated on the hypothesis of the uncracked sections and so they are higher then the most probable ones. The FE mesh of the walls was now refined up to a 50cm size. Further refinement gave no substantial modification of the results. Of course the challenge is to find the internal forces concerning the walls and the spandrel walls. For this reason the program has the option *Reduced Results for Panels*. With the sign convention from Figure 12 the user can choose the section (the cut) where moments, shear forces and stresses will be displayed (see Figure 13).

ISSN 1582-3024

Seismic analysis using Robot Millennium

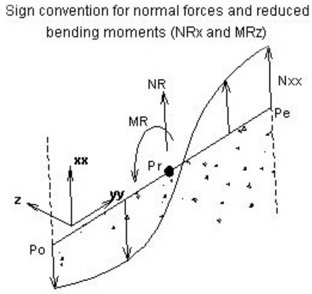
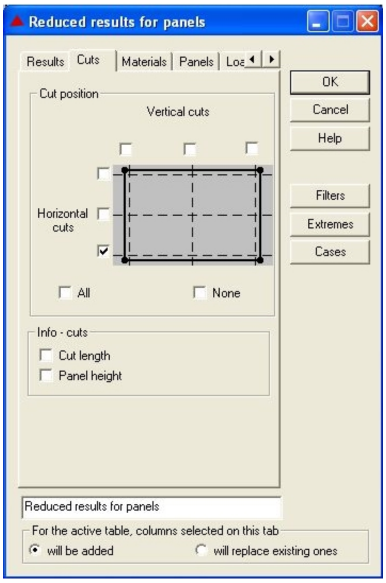


Figure 12. Sign convention



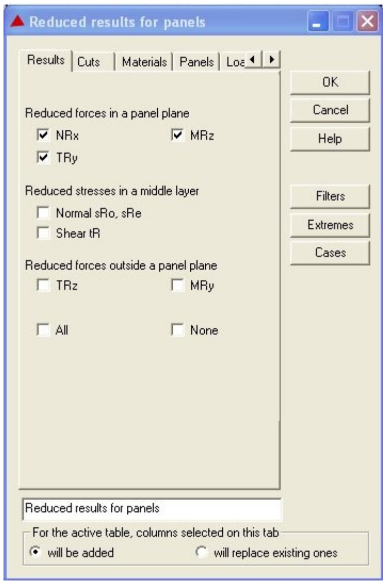


Figure 13. Choosing a) the cut position, b) the reduced forces to be displayed

There are two possibilities for modeling the wall columns. Until now they were modeled as bars. In this case, for a RC wall the internal forces of both wall and wall columns must be combined in order to have the result for the entire wall section. A second possibility is to eliminate the wall columns and to use an equivalent wall section from the primary moment of inertia point of view. The building stiffness will slightly decrease due to the lack of compression/tension absorbed by the wall columns. Comparison of the 2 modeling methods with the original results is given for a longitudinal wall (Table 2). The forces are calculated at the ground floor level from one seismic combination on X direction. The difference between the axial forces has to come from the lack of knowledge in the loads distribution over each storey since the evaluation of the building total weight was checked and almost perfect similarity was achieved.

ISSN 1582-3024

Table 2. Final internal forces for a longitudinal wall

Wall PL1	N (kN)	M (kNm)	T (kN)
Wall	-7741.22	-32648.3	2986.15
Wall column 1	-2760.13	-178.81	324
Wall column 2	5175.29	-251.61	389
Final forces	-10156.4	64389.96	3699.15
Wall without columns	-9939.21	-63955.5	3468.02
Example 3	-11456	65463	3778

3. CONCLUSIONS

The Robot Millennium software package gives us a reliable and elegant alternative for seismic analysis. Its features fully cover the demands of P100-2006. However, careful examination of its tools has to be done in order to choose the optimal modeling strategy.

References

1. **Postelnicu Tudor, P100-1/Proiectarea Seismica a cladirilor vol.2-B. Comentarii si exemple de calcul** – *ctr. U.T.C.B. nr. 158/02.08.2005 M.T.C.T.*
2. **Robot Millennium v.20.1**. *Autodesk Inc, 2008*
3. **CR 2-1-1.1**, *Cod de proiectare a constructiilor cu pereti structurali de beton armat*
4. **P100-1/2006**, *Cod de proiectare seismica*