Static and dynamic simulation in the seismic behavior of a building structure using ANSYS program

Adriana Ionescu, Madalina Calbureanu, Mihai Negru

***Abstract*—** This paper present the results of seismic behavior simulation obtained in two different methods used to analyze a two stories building. The methods presented in the paper are: the static equivalent forces method and the modal spectral analysis method. Both methods were applied using Finite Elements Method and ANSYS program for the tri-dimensional structure of the building. The study was made in order to determine which type of analysis is more convenient and easier to be applied in ANSYS program for simulating the seismic behavior of a classical type of reinforced concrete building.

***Keywords*—** Modal spectral analysis, seism, Finite Elements Method, ANSYS, building analysis

I. Introduction

IN the Romanian design building codes we have 3 different types of analysis used to determine the seismic behavior of the buildings:

- equivalent static analysis
- modal spectral analysis
- time history analysis

The first two of them are easier to be implemented and they can be applied to a building with a medium complexity of the structure.

ANSYS program is not frequently used in buildings analysis but it is a very complex program which can simulate a wide range of static and dynamic behaviors, and can be used with caution in these types of analysis for buildings.

For the comparison we have choose to analyze a two stories reinforced concrete building with uniform structure and two axis of symmetry, also we have similar stories (fig.2).

The building is subjected to a classical type of loads and the soil foundation is considered rigid.

For this type of analysis are used the bar uni-dimensional finite element for the columns and beams and SHELL finite element for the slabs.

The nodes which connect to soil foundation are considered fixed with all degrees of freedom restrained.

Adriana Ionescu is with University of Craiova, Faculty of Mechanics, Civil Engineering Department (corresponding author to provide phone: 0726930197; e-mail: adita_i@ yahoo.com).

Madalina Calbureanu is with University of Craiova, Faculty of Mechanics, Civil Engineering Department (e-mail: madalina.calbureanu@gmail.com).

Mihai Negru is with University of Craiova, Faculty of Mechanics, Mechanical Engineering Department (e-mail: negrumih@yahoo.com)

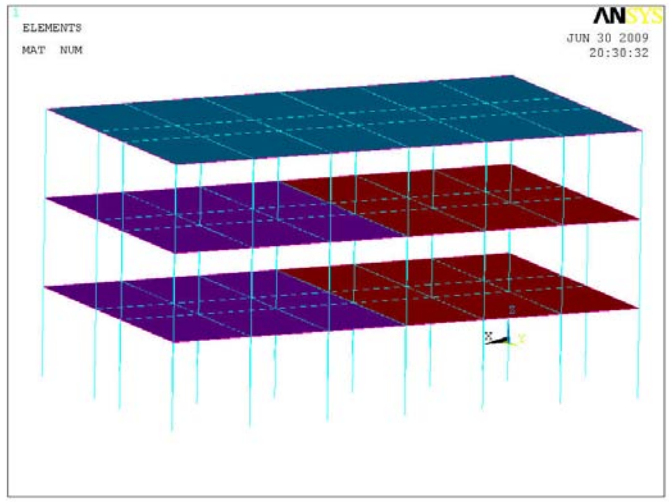

Fig. 1 The finite element model of the structure

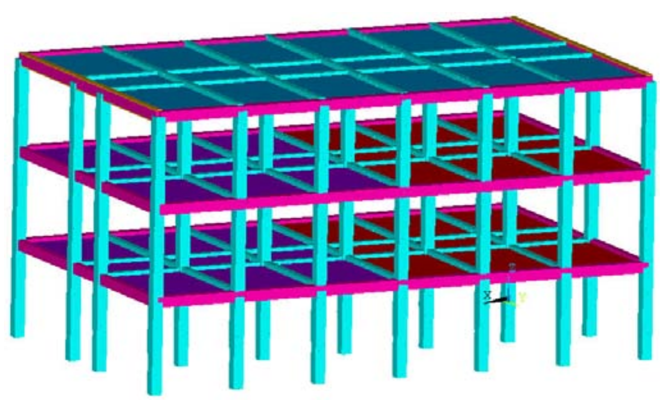

Fig. 2 Resistance structure of the building

II. PROBLEM FORMULATION

The equivalent static method defines a series of forces acting on each floor to represent the effect of earthquake ground motion. These forces are considered to be applied at the mass center of each floor.

$$F_i = F_b \frac{m_i z_i}{\sum_{i=1}^{n} m_i z_i} \tag{1}$$

where: F_i – floor forces, i – floor number, m_i – floor mass; z_i – distance between floor i and ground; F_b – shear base force (depends on building mass):

$$F_b = \gamma_I S_d(T_1) m\lambda \qquad (2)$$

where: γ_I – building importance coefficient,
m – total mass of the building,
λ – correction factor,
$Sd(T_1)$ – response spectrum for T_1

Modal spectral analysis is a linear-dynamic statistical analysis method which measures the contribution from each natural mode of vibration to indicate the maximum seismic response of the structure.

This analysis provides insight into dynamic behavior by measuring pseudo-spectral acceleration, velocity, or displacement as a function of structural period for a given level of damping.

The natural modes which have a superior importance coefficient are considered for modes combination.

The combination of modes can be done in five different ways:

- Complete Quadratic Combination Method (CQC),
- Grouping Method (GRP),
- Double Sum Method (DSUM),
- SRSS Method (SRSS),
- NRL-SUM Method (NRLSUM).

In this paper we have used CQC method.

III. PROBLEM SOLUTION

Both methods for seism simulation: equivalent static and modal structural analysis have been used on the 3d building structure. The results are presented for the central longitudinal and transversal frame.

The moments are presented in the figures in kNm and the shear forces in kN. For a better understanding we have presented the moment diagrams separately for beams and columns for each of the two methods.

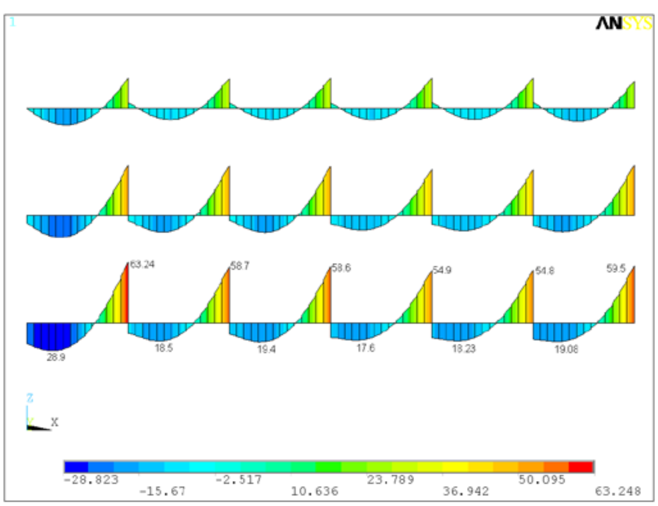

Fig. 3 Beams Moment diagram (first method) on longitudinal frame

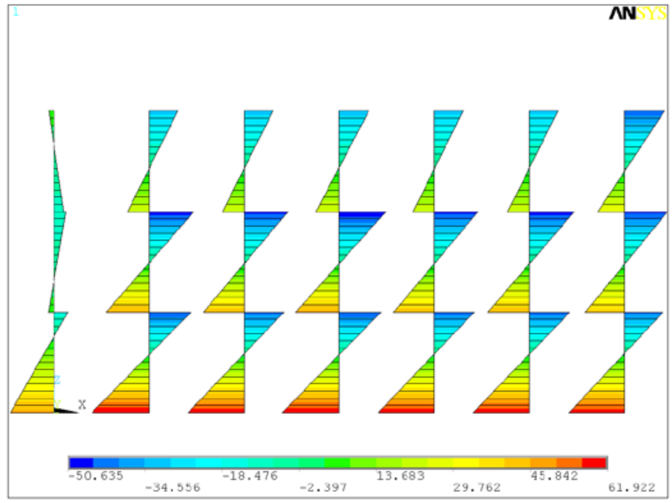

Fig. 4 Columns Moment diagram (first method) on longitudinal frame

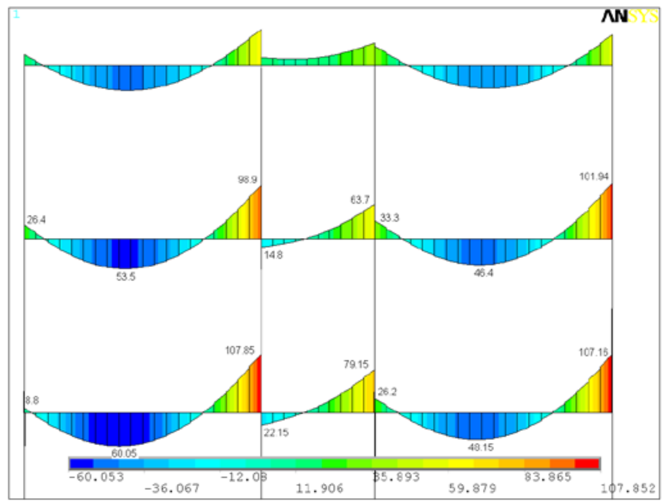

Fig. 5 Beams Moment diagram (first method) on transversal frame

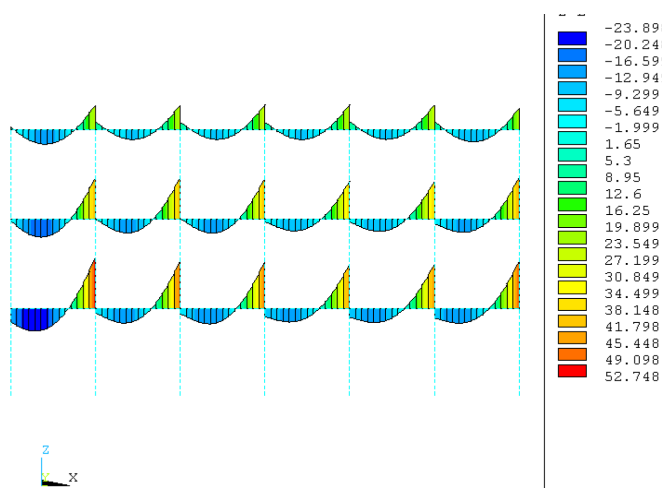

Fig. 6 Beams Moment diagram (second method) on longitudinal frame

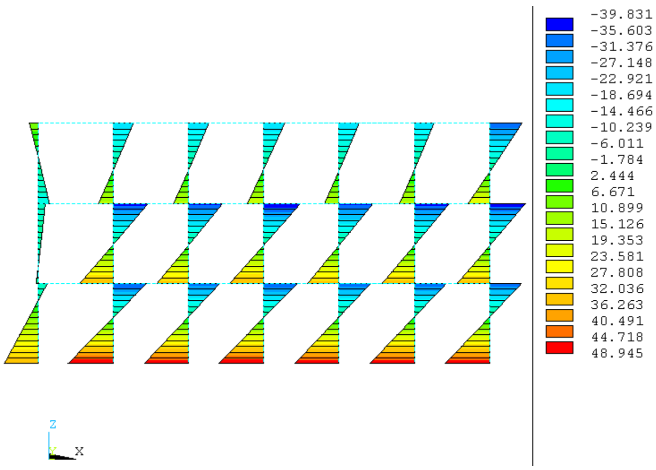

Fig. 7 Columns Moment diagram (second method) on longitudinal frame

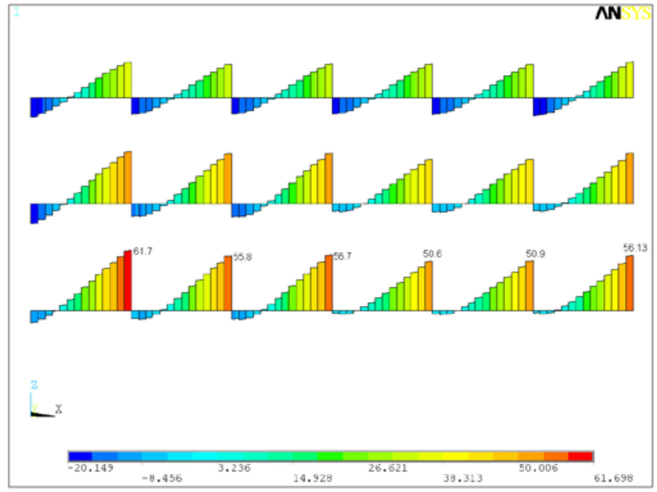

Fig. 10 Beams Shear Force diagram (1st method) on longitudinal frame

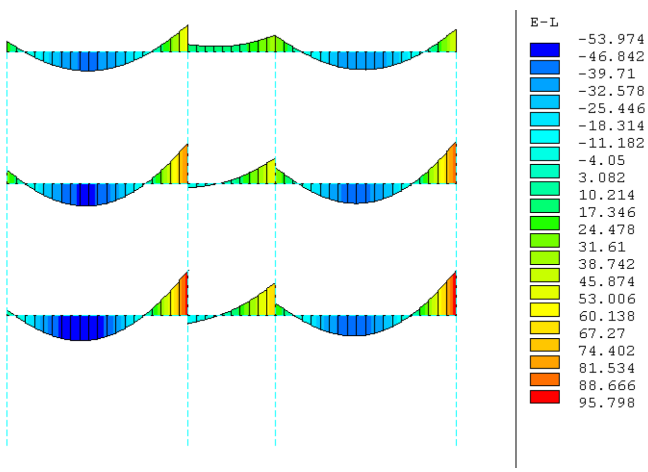

Fig. 8 Beams Moment diagram (second method) on transversal frame.

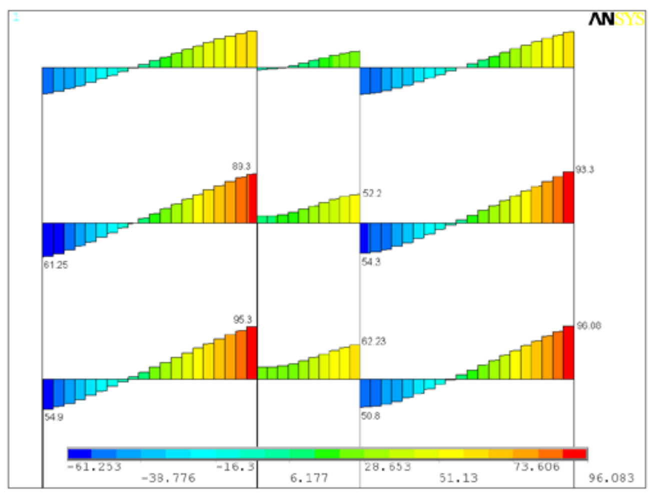

Fig. 11 Columns Shear Force diagram (1st method) on transversal frame

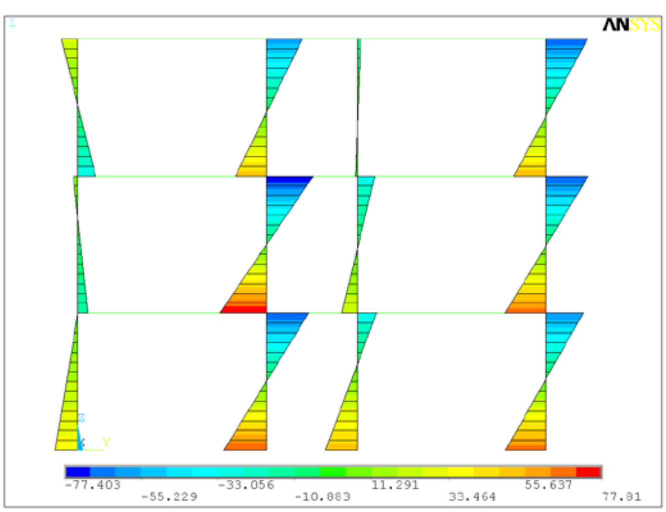

Fig. 9 Columns Moment diagram (1st method) on transversal frame

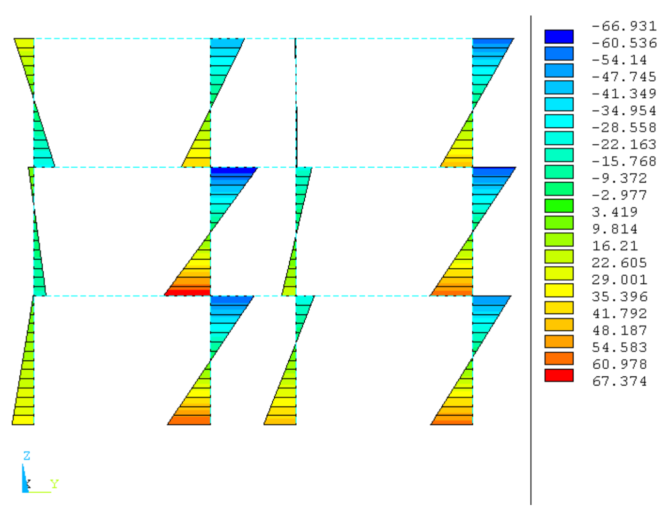

Fig. 12 Columns Moment diagram (2nd method) on transversal frame

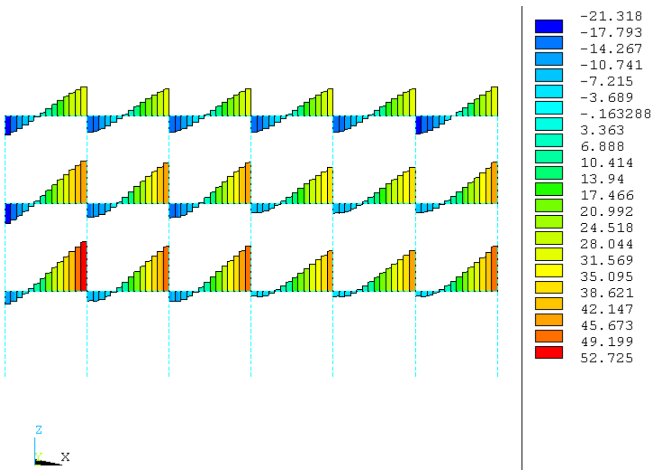

Fig.13 Beams Shear Force diagram (2nd method) on longitudinal frame

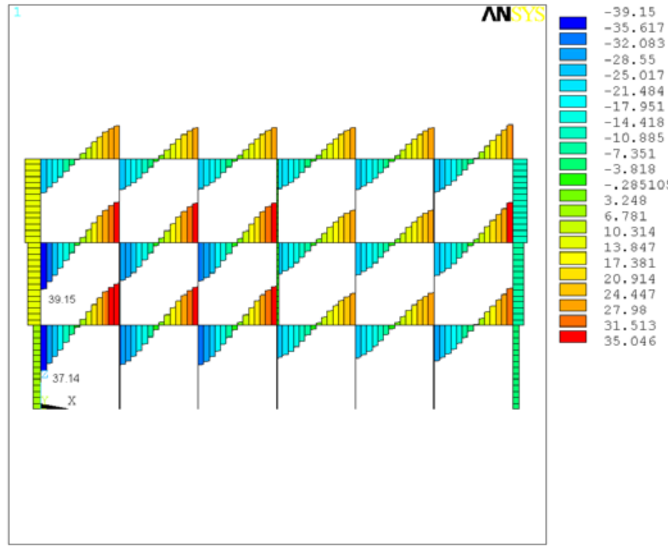

Fig.16 Shear Force diagram (self weight) on longitudinal frame

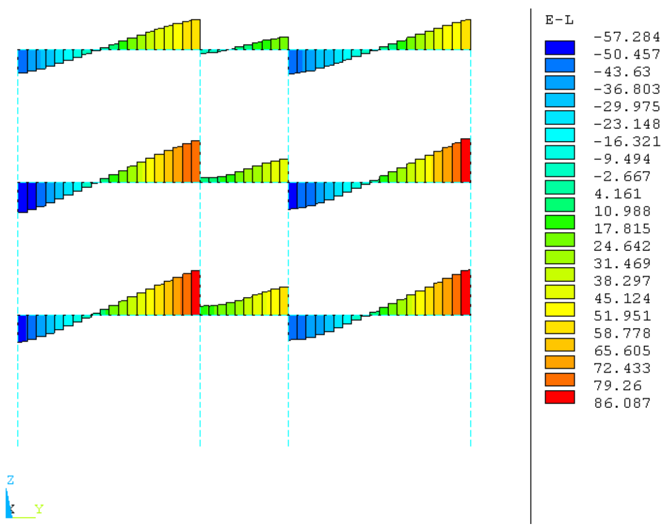

Fig. 14 Beams Shear Force diagram (2nd method) on transversal frame

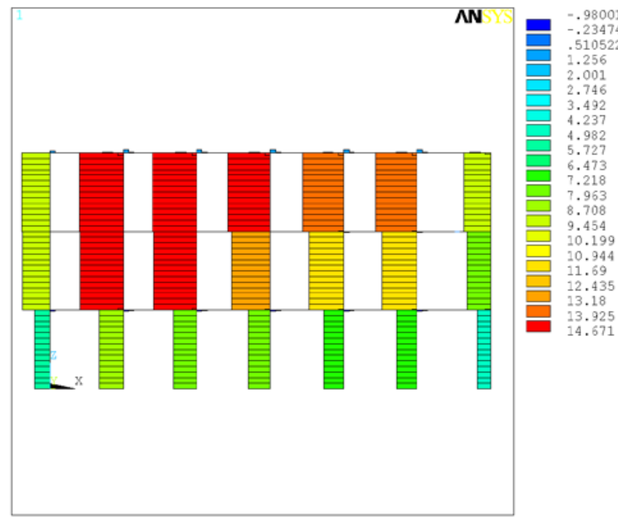

Fig. 17 Columns Shear Forces diagram (self weight) on longitudinal frame

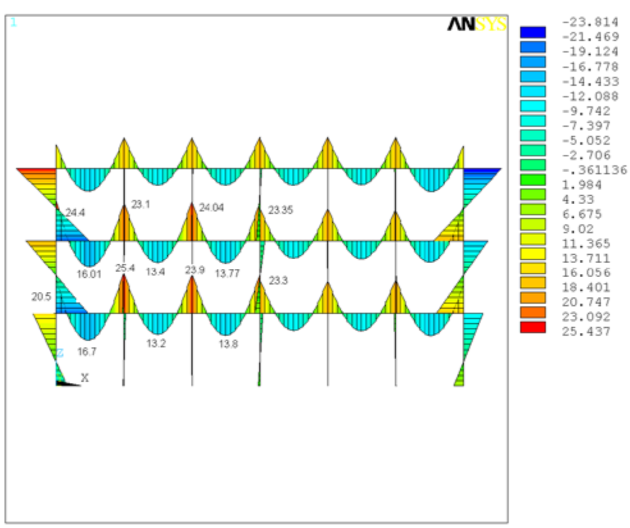

Fig.15 Moment diagram (self weight) on longitudinal frame

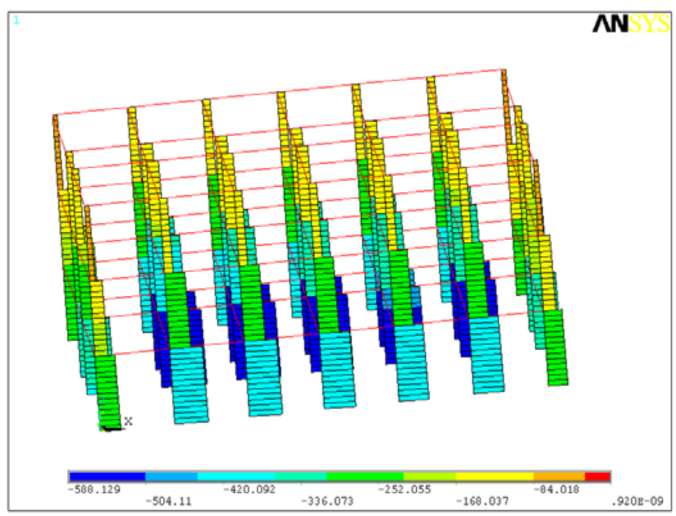

Fig. 18 Axial Forcea diagram (self weight) on 3D structure

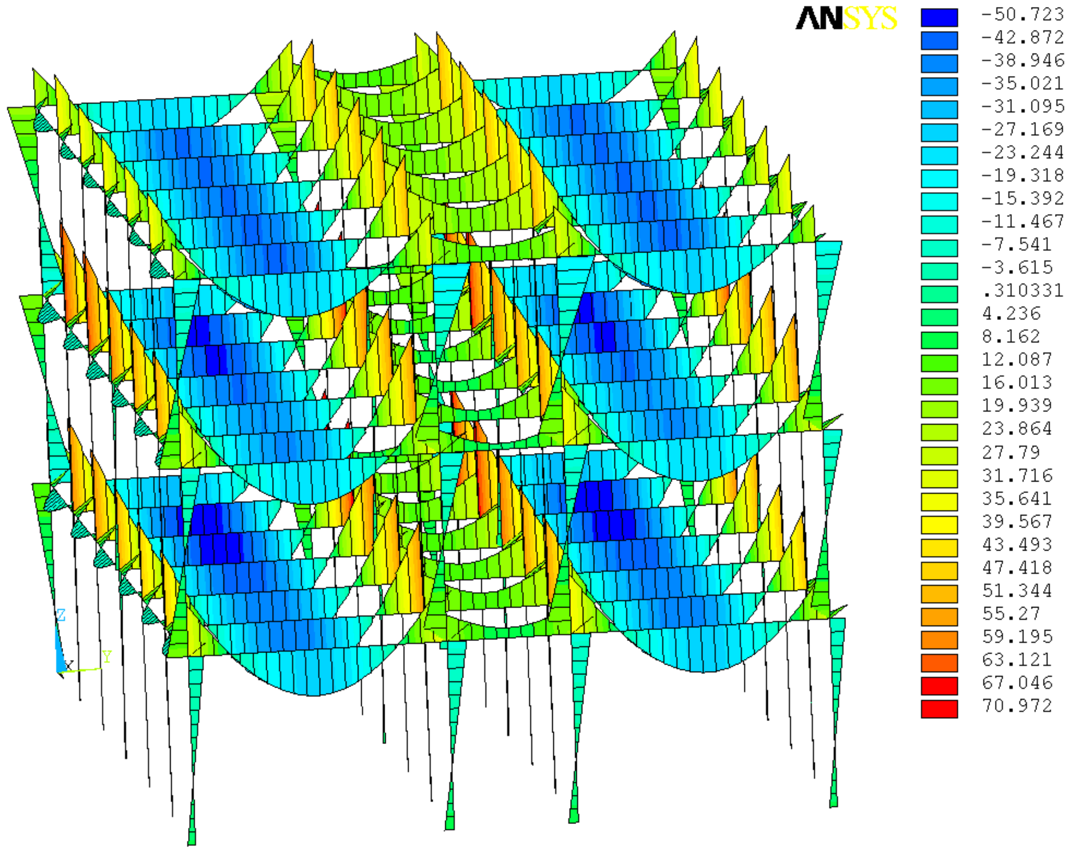

Fig. 19 Moment diagram (self weight) on 3D structure

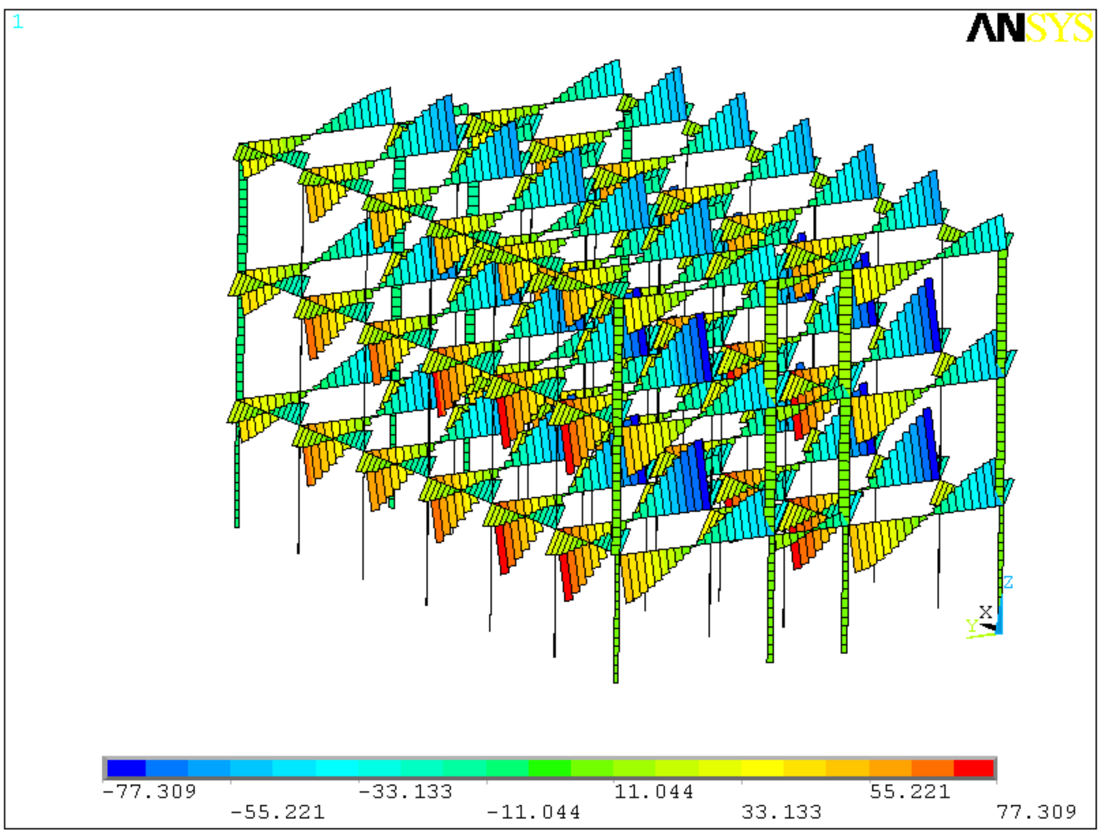

Fig. 20 Shear Forces diagram (self weight) on 3D structure

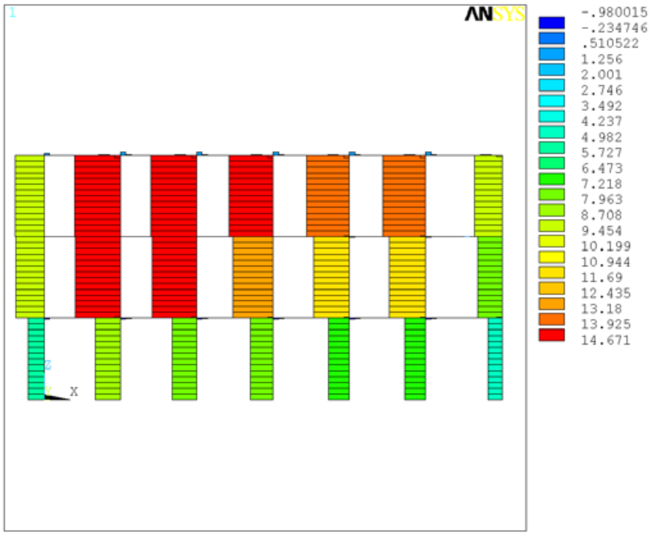

Fig. 21 Columns Shear Forces diagram (self weight) on longitudinal frame

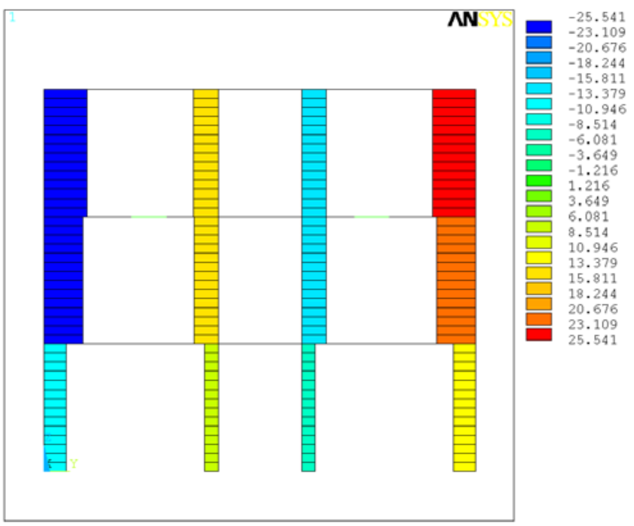

Fig. 22 Columns Shear Forces diagram (self weight) on transversal frame

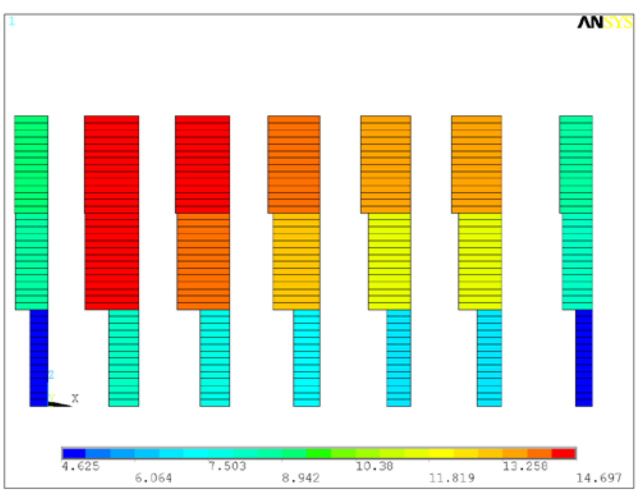

Fig. 23 Columns Shear Forces diagram (seism) on longitudinal frame first method

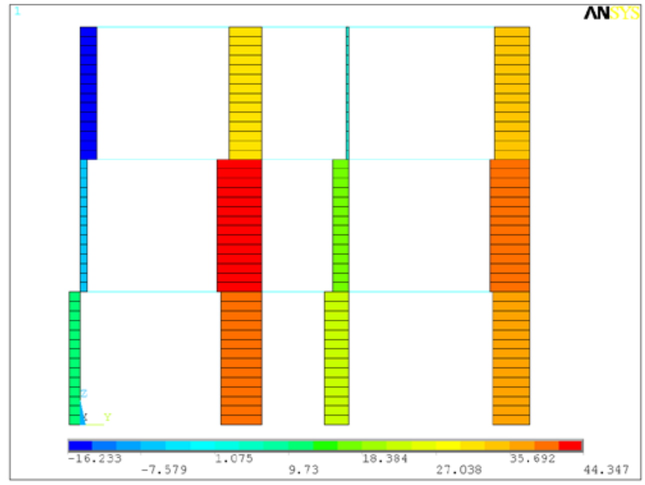

Fig. 24 Columns Shear Forces diagram (seism) on transversal frame first method

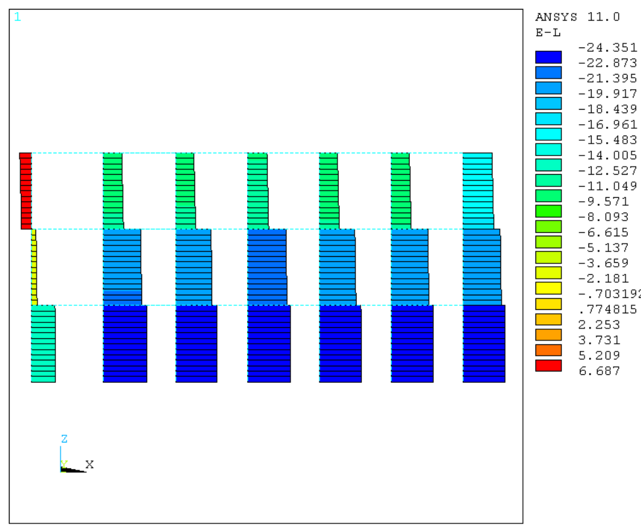

Fig. 25 Columns Shear Forces diagram (seism) on longitudinal frame second method

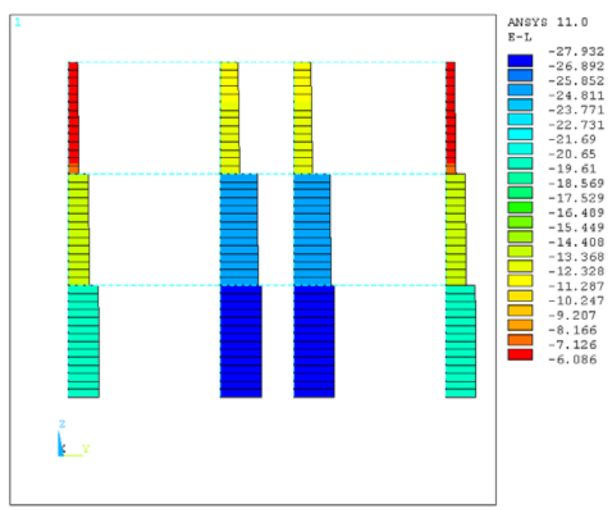

Fig.26 Columns Shear Forces diagram (seism) on transversal frame second method

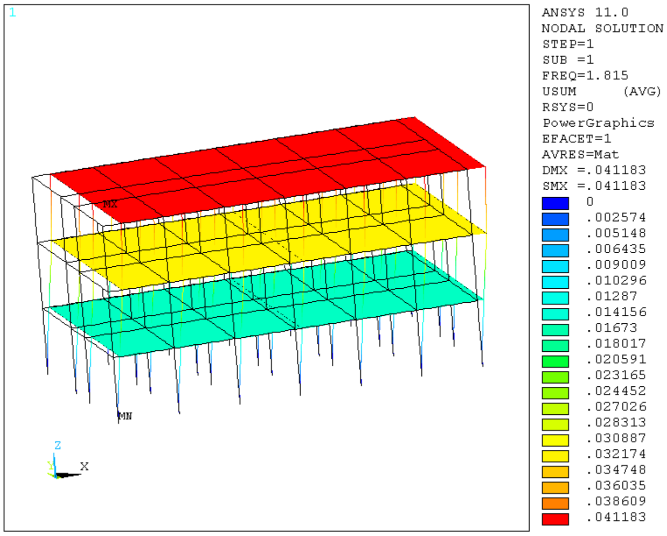

Fig. 27 Vibration Mode no. 1 (f = 1,815 Hz)

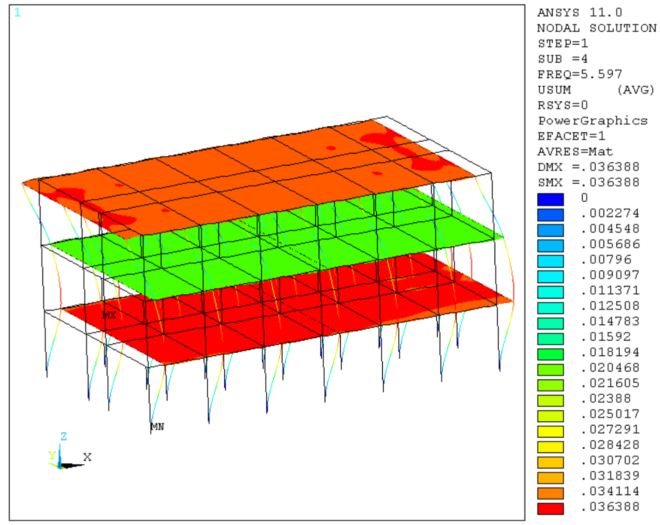

Fig. 30 Vibration Mode no. 4 (f = 5,597 Hz)

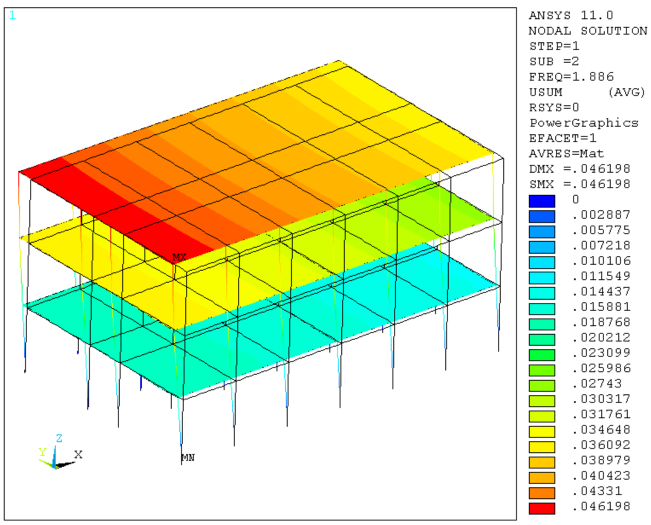

Fig. 28 Vibration Mode no. 2 (f = 1,886 Hz)

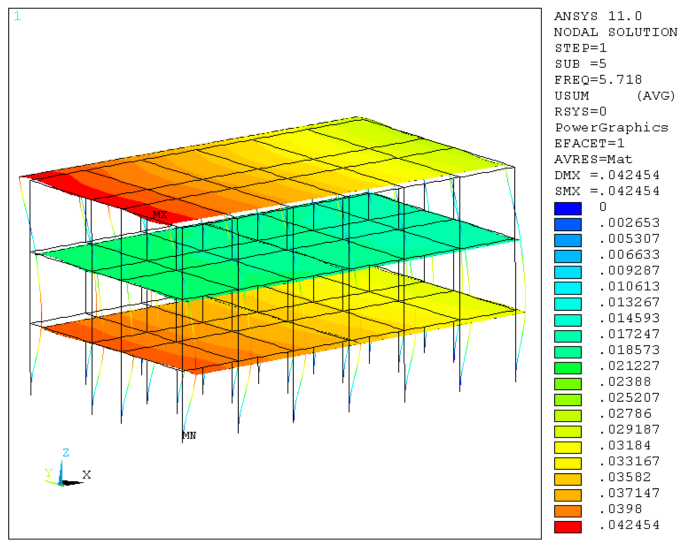

Fig. 31 Vibration Mode no. 5 (f = 5,718 Hz)

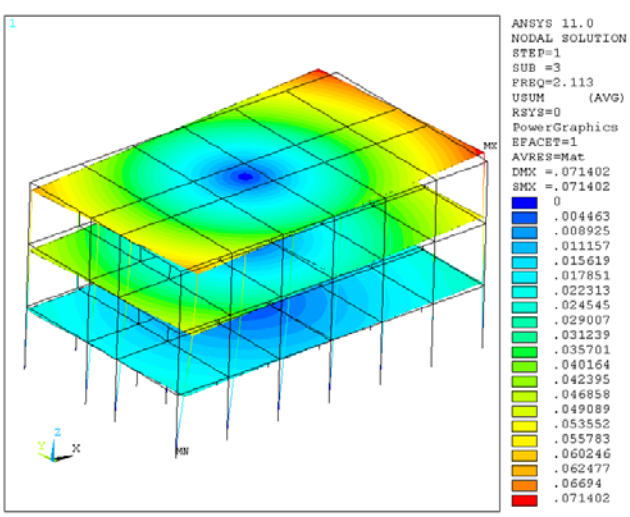

Fig. 29 Vibration Mode no. 3 (f = 2,13 Hz)

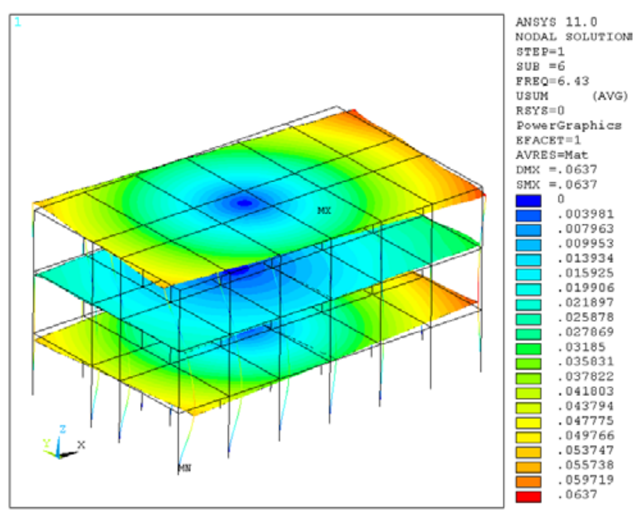

Fig. 32 Vibration Mode no. 6 (f = 5,718 Hz)

In Ansys for the beams and columns of the structure has been used 3DBeam finite elements and for the floors has been used SHELL elements. The supports of the structure has being considered rigid.

For the equivalent static analysis, were calculated all Fi forces and were applied at the mass centers of each floor and for the modal spectral analysis was used a corresponding spectrum.

I. CONCLUSIONS

The equivalent static analysis implies using the master and slave nodes in order to achieve the diaphragm hypothesis for the floors. In this case the master nodes are the nodes at the mass center of each floor in which we have applied the Fi equivalent forces and the slave nodes are all other nodes on that floor.

The modal spectral analysis is a two step analysis, first is necessary to perform a modal analysis then the spectrum analysis.

As conclusion, in Ansys the preparation of the model for analysis is quicker for modal spectral analysis then for the equivalent static analysis.

Analyzing the diagrams results we can see that the shape of the diagrams is almost identical and the values are similar. The maximal differences between the results obtained with these two methods are as follows:

- 14% for beams moments - longitudinal frame;
- 16% for columns moments – longitudinal frame;
- 7 % for beams shear forces – longitudinal frame;
- 10% for beams moments - transversal frame;
- 14% for columns moments – transversal frame;
- 10 % for beams shear forces – transversal frame.

In conclusion, in ANSYS program is easier to determine the seismic behavior of the buildings using modal spectral analysis and the results of both methods are very similar, the average difference between results was 11,8 %.

REFERENCES

[1] Negru M., Rinderu P., Manea I, "Spectrum analysis of the SF6 High Voltage Circuit Breaker using Finite Element Method", *The XIII International Congress on Sound and Vibration, ICSV13,* CD-ROM Proceedings, Vienna 2006, Austria.

[2] M. Negru, Dumitru N., Rinderu P., "Modal analysis of the SF6 high voltage circuit breaker using finite element method", *The XIII International Congress on Sound and Vibration, ICSV 12*, CD-ROM Proceedings, Lisbon 2005, Portugal.

[3] O.C. Zienkiewicz, R.L. Taylor, *The Finite Element Method*, Fourth edition, V.2, McGraw- Hill.

[4] *Ansys User's Guide*.

[5] Negru M., Rinderu P., Curtu I. – "Modal analysis of the portal plane milling machine optimized resistance structure by using F.E.M." , *The XII International Congress on Sound and Vibration, ICSV12*, 11-14 july, Lisabona, Portugalia, 2005.

[6] Negru M., Bolcu D.,Manea I., "Earthquake simulation of the SMEP 400kV high voltage disconnector structure using Finite Element Method", *11th WSEAS International Conference on COMPUTERS* (July 26-28, 2007) Crete Island, Greece.

[7] Saeed Moaveni., *Finite Element Analysis*, Prentice Hall Inc., 1999.

[8] Adriana Ionescu, Mihai Marin, Madalina Olga C. Gugila, Mihai M. Negru, "Arch steel building optimization used in civil engineering, by Finite Element Method", *First International Conference for PhD students in Civil Engineering CE-PhD* 2012, 4-7 November 2012,Cluj-Napoca, Romania.

[9] Negru M., Georgescu I., Albota E., "Optimization of a large steel truss structure used in civil engineering, by finite element method", *1st WSEAS International Conference On Finite Differences - Finite Elements - Finite Volumes - Boundary Elements*, 2008.

[10] Negru M., Bolcu D., Manea I., "Finite Element Method in modal analysis of the IO 400kV high voltage Circuit Breaker structure", *11th WSEAS International Conference on COMPUTERS* (July 26-28, 2007) Crete Island, Greece, section: Advanced Computational Techniques, Algorithms and Numerical Methods for Modelling, Simulation and Optimization.

[11] Ionescu A., Calbureanu M., Negru M., "Boussinesq method in seism analysis of a building structure using ANSYS program" – *WSEAS International Conference*, Vouliagmeni, Athens, Greece, may 14-16, 2013.