Structural Mechanics

INTERSECTII
INTERSECTIONS
http://www.intersections.ro

# Capacity based earthquake design of reinforced concrete shear wall according to EC8

Laura Fraifeld[1], Ramona Agachi[1], Horst Werkle[2], Franz Zahn[2]
and Gabriela Atanasiu[1]
*[1]Faculty of Civil Engineering, Technical University «Gh. Asachi» Iasi, Roumania*
*[2]Faculty of Civil Engineering, University of Applied Science, Konstanz, Germany*

## Summary

*Any structural design can be considered satisfactory if it meets the three principal goals of structural performance: safety, economy and simplicity of construction. The first two require a sufficient margin of safety against collapse. Also, the construction should be serviceable under working load conditions through its life time.*

*The topic of this paper is the analysis of the seismic response of reinforced concrete building using displacement based design model. For this purpose a typical Romanian residential building has been considered as case study. First, the equivalent static loads acting on the shear walls was determined by using simple hand calculations taking into account linear elastic behavior. An analysis of the most critical shear wall for the determined lateral forces was made by using two different nonlinear analysis programs for comparison, namely the program ADINA [9] and the program ATENA 2D [10]. Seismic demands estimated in this structure using an equivalent single degree of freedom system derived from the first fundamental mode of vibration of the building.*

KEYWORDS: safety against collapse, serviceability, seismic response, linear elastic behavior, nonlinear analysis, seismic demands, single degree of freedom system

## 1. INTRODUCTION

The design of structures for earthquake resistance in engineering practice is generally based on a response spectrum analysis. The ductility of the structure is considered by a behavior factor depending on the construction type of the building. The present simplified approach neglects all the redistribution of the sectional forces due to yielding of structural elements and other nonlinear effects occurring during an earthquake. A rigorous nonlinear computation, how-ever, requires a time history analysis of the building including the effect of material non-linearity. This

ISSN 1582-3024

type of analysis is time consuming and the interpretation of the results is not always obvious.

Capacity design represents a method developed for practical earthquake design, including the overall nonlinear behavior of the structure. It implies some major simplifications, mainly modeling the structure as a single degree of freedom system.

The analysis of a reinforced concrete building according to the capacity design consists in a conventional design by the response spectrum method. The nonlinear behavior of reinforced concrete walls in pushover analysis has been modeled by plane stress elements in the finite element program ADINA [9]. The following steps have to be accomplished:

- -Design of the reinforcement by a RESPONSE SPECTRA ANALYSIS (RSA)
- -Determination of the safety factor according to the capacity design for the reinforcement for response spectra analysis by using ADINA[9] and MathCAD[11].

## 2. ANALYSIS OF A REINFORCED CONCRETE WALL USING A SHELL MODEL

### 2.1. Numerical analysis of reinforced concrete wall. The design of the structural walls

In this work a structural reinforced concrete wall has been analyzed by means of more methods and then the results are compared. The computation of the element took into account a structural wall from the first storey of a building P + 4, denoted Structural Wall TW1.

In the process of design using the multi-storied one-bay frame mode, the horizontal afferent loads to the structural wall had the value $q_{1,\,y} = 3228.08$ daN/m.

### 2.2. Numerical analysis of reinforced concrete walls using ADINA Software

The numerical simulation has been performed using a 3D element acted on top by a force equal to 400000N, at level 3: 0.75 x 400000N, at level 2: 0,5 x 400000N and at level 1 : 0,25 x 400000N. The reinforcement consists of 18ø16 bars disposed on both lateral sides of the wall (9 bars on a part and 9 on the other part). The element collapses at a load of 328200 N. The Program ADINA [9] determines the maximum capacity of the element by several iterations based procedure. The maximum displacement obtained is 0.004675 m.

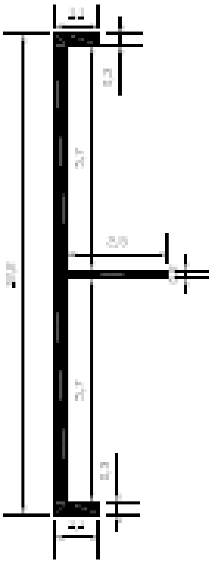


Fig.1. Structural Wall TW1 Topology

## 2.3. Computation of the Structural Wall TW1 using ATENA 2D Software

For the numerical experiments of the wall we use as materials the concrete class C25/30 and reinforcement steel S550. The element was introduced as a bidimensional element having the geometrical configuration presented in Figure 2.

The horizontal and vertical loads are linearly applied in order to avoid the singularity of stresses and displacements.

As horizontal load a load of value of 2500 KN has been applied in the following manner: 10% of the load for the first floor, 15% for the second one, 20% for the third, 25% for the fourth and 30% for the fifth floor in the direction of the line that represents the axis of the slab at the level.

The obtained values are then divided by the length of the wall, respectively of 12.3m, being expressed in [MN/m]. The vertical load is also considered, being linearly applied, thus the values obtained are divided by the length of the wall.

ISSN 1582-3024

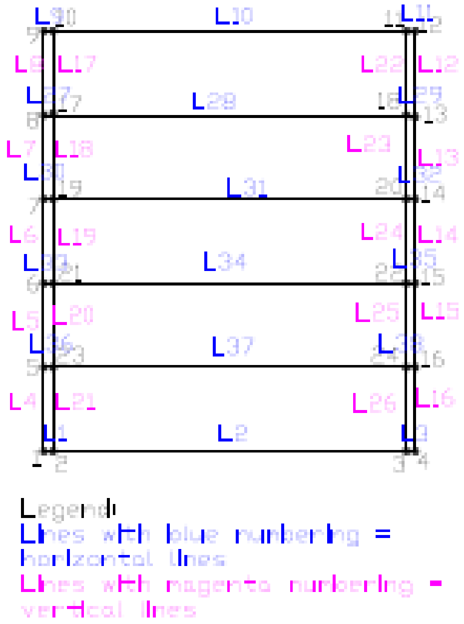


Fig.2. Geometrical Model of the Element

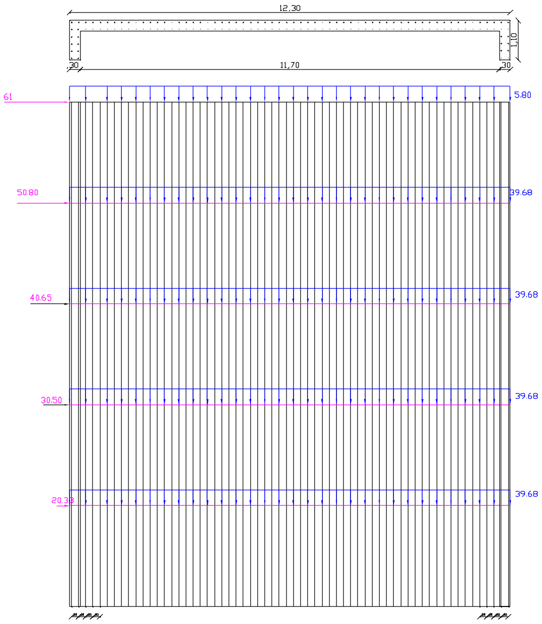


Fig.3. Applying the load on the member

There have been introduced 30 steps with the step multiplier equal to 0.1. In accordance with the Output Data given by ATENA Software, the load capacities of the member with their corresponding displacements have been deducted and are given in Table 1.

Table 1: Load Capacities of the Structural Wall TW1 and the corresponding displacements

| Forces KN | Displacement m | Forces KN | Displacement m |
|---|---|---|---|
| 250 | 1,27E-04 | 4000 | 3,30E-03 |
| 500 | 2,55E-04 | 4250 | 3,62E-03 |
| 750 | 3,82E-04 | 4500 | 3,85E-03 |
| 1000 | 5,10E-04 | 4750 | 4,60E-03 |
| 1250 | 6,38E-04 | 5000 | 4,90E-03 |
| 1500 | 7,66E-04 | 5250 | 6,02E-03 |
| 1750 | 8,94E-04 | 5500 | 7,31E-03 |
| 2000 | 1,02E-03 | 5750 | 9,37E-03 |
| 2250 | 1,39E-03 | 6000 | 1,24E-02 |
| 2500 | 1,56E-03 | 6250 | 1,73E-02 |
| 2750 | 1,88E-03 | 6500 | 2,97E-02 |
| 3000 | 2,07E-03 | 6750 | 6,85E-02 |
| 3250 | 2,26E-03 | 7000 | 1,13E-01 |
| 3500 | 2,67E-03 | 7250 | 1,97E-01 |
| 3750 | 3,08E-03 | 7500 | 3,26E-01 |

## 3. ANALYSIS OF A REINFORCED CONCRETE WALL USING A BEAM MODEL

### 3.1 The moment-curvature analysis

The plastic moment capacity of all ductile concrete members is computed using the moment curvature (M- Φ) analysis based on expected material properties shown in Figure 4.

Moment-curvature analysis derives the curvatures associated with a range of moments for a cross section based on the principles of strain compatibility and equilibrium of forces.

The (M- Φ) curve can be idealized with an elastic perfectly plastic response to estimate the plastic moment capacity of a member’s cross section. The elastic portion of the idealized curve should pass through the point marking the first reinforcing bar yield.

The idealized plastic moment capacity is obtained by balancing the areas between the actual and the idealized (M- Φ) curves beyond the first reinforcing bar yield point.

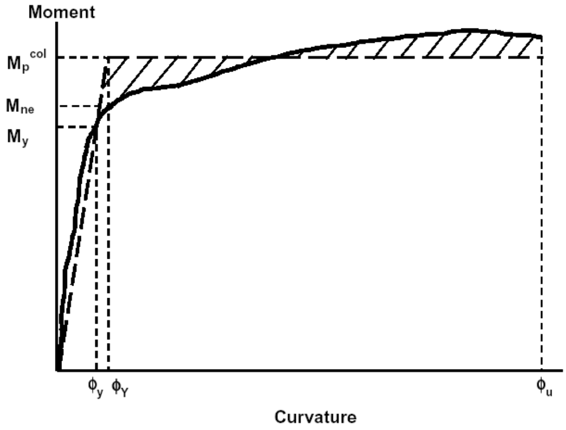


Fig.4. Moment-Curvature Representation

### 3.2 Determination of plastic hinge

The plastic hinge length is required to convert the moment-curvature relationship into a moment-plastic rotation relationship for the nonlinear pushover analysis.

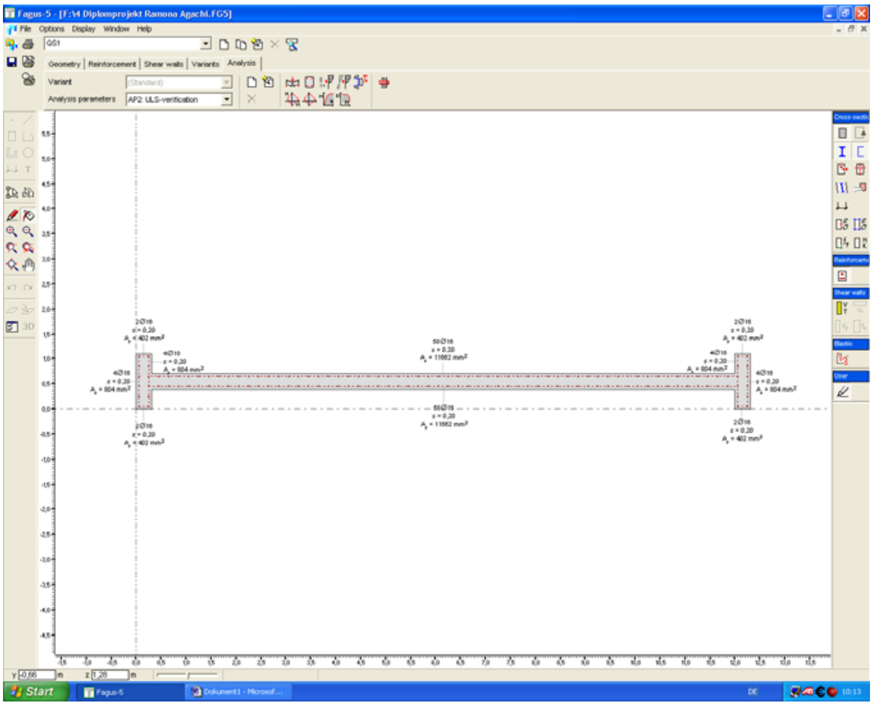
The plastic rotation, $\theta_P$, can be determined from the following Equation 1, by using moment-curvature analysis and applicable strain limitations. The plastic rotation is:

$$\theta_p = L_p\phi_p = L_p\left(\phi_m - \phi_y\right) \tag{1}$$

Where: $L_p$ is the plastic hinge length, $\Phi_p$, the plastic curvature, $\Phi_m$, the maximum curvature and $\Phi_y$ the yielding curvature.

### 3.3 Beam bending

The wall TW1 has been loaded with a force N equal to 2023.5 KN. In the Figure 5 the cross-section of the shear wall TW1 is shown, as an extract from FAGUS-4 Software.



Fagus-5 - [F:\4 Diplomprojekt Ramona Agachi.FG5]

QS1

Variant (Standard)

Analysis parameters AP2: ULS-verification

Fig.5. Cross-section of the shear wall TW1

The relationship (M- Φ) has been obtained by using the software FAGUS-4 [12] shown in Fig. 6.

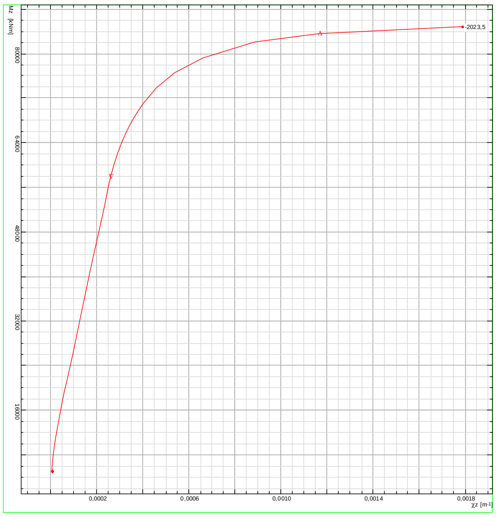


Fig.6. Moment-Curvature Diagram

From the moment-curvature diagram, trying to equal the area cut off and added, the following results for the ultimate moment, yielding moment and yielding curvature are obtained: $\Phi u = 0.0020\ m^{-1}$, $\Phi y = 0.00048\ m^{-1}$ and My = 84000 KN*m.

### 3.4 Displacement computational aspects

The base shear force of the wall resulted is: $V_b = 8000$ KN. The actual moment distribution will be somewhere between parabolic and linear, and a better approximation would be to use the average of 1/4 (for parabolic) and 1/3 (for linear curvature distribution). The length of the plastic hinge is: $L_P$=1.52 m.

The displacement corresponding to the yielding moment is determined using Equation 2:

$$\Delta y := \int_{0}^{h} \phi y(x) \cdot M(x)\ dx \qquad (2)$$

After yielding the wall will be very heavily cracked and therefore there will also be a plastic (additional) shear displacement.

The plastic curvature is computed with the following equation:

$$p = \Phi u - \Phi y \qquad (3)$$

The plastic rotation of the plastic hinge is:

$$\theta p = Lp * \Phi p \qquad (4)$$

The plastic displacement is given by the following relationship:

$$\theta \Delta p = \theta p * (h - Lp/2) \qquad (5)$$

Table 2: Displacement values of the shear wall TW1

| No. | Type of displacement | Displacement value (m) |
|---|---|---|
| 1 | Yield displacement($\Delta y$) | 0.02744 |
| 2 | Shear displacement ($\Delta_{shear}$) | 0.0044 |
| 3. Total yield displacement ($\Delta_{yieldtot}$) (1+2) | | 0.03184 |
| 4 | Additional shear displacement ($\Delta_{asdy}$) | 0.035 |
| 5 | Plastic displacement ($\Delta p$) | 0.03059 |
| 6. Ultimate displacement ($\Delta u$) (3+4+5) | | *0.09743* |

## 4. RESULTS INTERPRETATION

For the capacity curve of the wall, a bilinear approximation has been used considering a linear elastic part up to the point where the shear capacity of the wall $V_b$. This point was reached and a perfectly plastic part with zero stiffness. The bilinear capacity curve of a wall has been defined by three parameters, the shear capacity of the wall $V_b$, the nominal yield displacement Δy and the nominal ultimate displacement at the top of the wall Δu. The comparison made with the results given by ATENA Software is graphically represented in Fig.7.

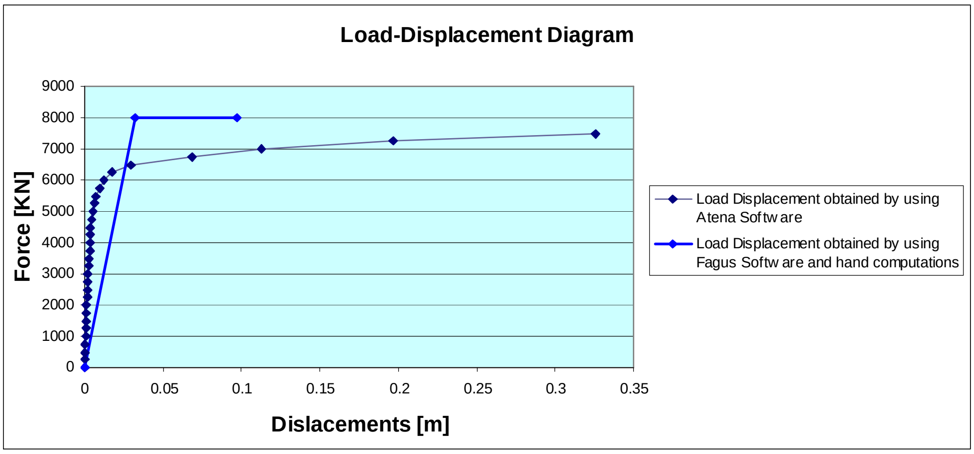


Fig.7. Numerical interpretation of displacement – shear force relationship

## 5. FINAL REMARKS

Being a benchmarking methodology, the paper presented the comparison between two methods of analysis of the shear wall nonlinear behavior.

Using ADINA [9] the maximum displacement obtained for the base shear force equal to 328200 N is 0.004675 m. By using ATENA [10], for a base shear force of 7500 KN it results a displacement of 0.0326 m.

Considering the 2nd method based on FAGUS[12] software and hand computations, the ultimate displacement obtained for a shear force equal to 8000 KN is equal to 0.09743 m.

The difference of numerical results between the displacements obtained in the analyses made with ATENA [10] and the displacements by means of the second method consists in the methodology accuracy. In the first case, the base shear force is of a smaller value than the value obtained using the second method.

ADINA [9], a very complex program, was used for analysis of structures using many fields of study, while ATENA [10] is specialized in analysis of construction works. By comparing the numerical results obtained, one considers that ADINA [9] is a program with insufficiencies regarding the civil engineering field. ATENA [10] is therefore easier to approach and for purpose of obtaining numerical results following a simple analysis of structures, as the case of the analyzed concrete wall. ADINA [9] is more suitable for very complicated structures, as in the case of the San Francisco Oakland Bay Bridge or the performed crush analysis test of automobiles for Ford Windstar. Using ATENA [10] software one can analyse only bi-dimensional elements, while with ADINA [9] is suitable to perform 3D analyses of various structural elements. The software gives appropriate results, illustrated by the moment – curvature analysis obtained in the final step of computation.

As a final conclusion, the results of the numerical experiments done with ADINA [9], FAGUS [12] and MATHCAD [11] software shows give the corresponding results of load – displacement curve for the analyzed structural reinforced concrete wall. The main difference consists in the way the software is suitable for numerical simulations in the construction field analyses.

## References

1. Werkle, H., *Finite Elemente in der Baustatik, Statik und Dynamik der Stab- und Flächentragwerke,* Vieweg Verlag 2001
2. Werkle, H, Avak, R., *Mathcad in der Tragwerksplanung* , Vieweg Verlag 2002
3. Atanasiu, G., *Structural Dynamics*, 2nd version, Vasile Goldis University Press, Arad ,2005
4. Atanasiu, G., Vlad, I. Bratianu, C., *Finite Elements in Structural Analysis*, Cermi, Iasi, 2005
5. Bindseil, P., *Massivbau Bemessung im Stahlbetonbau*, Vieweg Verlag 2005
6. Schneider Bautabellen, *Bautabellen für Ingenieure*, Auflage 2004
7. Wommelsdorf, O., *Stahlbetonbau Teil 1: Grundlagen – Biegebeanspruchte Bauteile*, Werner Verlag, 2004
8. Wommelsdorf, O., *Stahlbetonbau Teil 2: Stützen und Sondergebiete des Stahlbetonbaus Bemessung und Konstruktion,* Werner Verlag, 2004
9. ADINA, *Finite Element Software*, Registered Trademark of K.J. Bathe/ADINA R & D Inc. USA, version 2002
10. ATENA 2D*, Finite Element Software*, version SBETA, 2001
11. Mathcad, Mathematical Computations Software, version 2001 Professional
12. FAGUS Software, Statical Computations Software, version 4, 2004

## Appendix

- Finite Element Analysis – a computer simulation technique used in engineering analysis, which uses a numerical technique called the Finite Element Method (FEM).

- Finite Element Method (FEM) – a structural mechanics method in which the object is represented by a geometrically similar model consisting of multiple, linked, simplified representations of discrete regions – i.e. finite elements.

- Response Spectrum – a plot of the peak or steady state response (displacement, velocity or acceleration) of a series of oscillators of varying natural frequencies, that are forced into motion by the same base vibration.

- Modal Analysis – represents the study of the dynamic properties of structures under vibration excitation.

- Pushover analysis - is a non-linear static analysis carried out under conditions of constant gravity loads and monotonically increasing horizontal loads.

- Linear Dynamic Analysis – is the analysis in which the response of the structure to the ground motion is computed in the time domain, and all the phase information is therefore maintained.

- Moment curvature analysis - derives the curvatures associated with a range of moments for a cross section based on the principles of strain compatibility and equilibrium of forces.

- The length of plastic hinge - required to convert the moment-curvature relationship into a moment-plastic rotation relationship for the nonlinear pushover analysis.