

BULETINUL INSTITUTULUI POLITEHNIC DIN IAȘI
Publicat de
Universitatea Tehnică „Gheorghe Asachi” din Iași
Tomul LVI (LX), Fasc. 4, 2010
Secția
CONSTRUCȚII. ARHITECTURĂ

FLEXURAL BEHAVIOR OF REINFORCED CONCRETE BEAMS USING FINITE ELEMENT ANALYSIS (ELASTIC ANALYSIS)

BY

R. SRINIVASAN and *K. SATHIYA

Abstract. Concrete structural components exist in buildings in different forms. Understanding the response of these components during loading is crucial to the development of an overall efficient and safe structure. Different methods have been utilized to study the response of structural components. Experimental based testing has been widely used as a means to analyse individual elements and the effects of concrete strength under loading. While this is a method that produces real life response, it is extremely time consuming, and the use of materials can be quite costly. The use of finite element analysis to study these components has also been used.

The performed study investigation attempts to compare the results from elastic analysis of a reinforced beam under transverse loading, using an analysis software package to that obtained from a normal theoretical analysis.

Key words: reinforced concrete beams; finite element analysis; ANSYS; reinforcement modeling.

1. Introduction

A concrete beam is a structural element that carries load primarily in bending. Bending causes a beam to go into compression and tension. Beams generally carry vertical gravitational forces but can also be used to carry horizontal loads. The loads carried by a beam are transferred to columns, walls, which is then transferred to foundations. The compression section must be designed to resist buckling and crushing, while the tension section must be able to adequately resist to the tension. Experimental based testing has been widely used as a means to analyse individual elements and the effects of concrete strength under loading. While this is a method that produces real life response,

it is extremely time consuming, and the use of materials can be quite costly. The use of finite element analysis to study these components has also been used.

Unfortunately, early attempts to accomplish this proceeding were also very time consuming and infeasible using existing software and hardware. In recent years, however, the use of finite element analysis has increased due to progressing knowledge and capabilities of computer software and hardware. It has now become the choice method to analyse concrete structural components. The use of computer software to model these elements is much faster and extremely cost-effective.

The goal of this paper is study attempts to compare the results from elastic analysis of a reinforced beam under transverse loading, using an analysis software package to that obtained from a normal theoretical analysis.

2. Finite Element Modeling of Reinforced Concrete Beam

Finite Element Analysis (FEA) represents a numerical method, which provides solution to problems that would otherwise be difficult to obtain. The numerical analysis investigations were performed with commercial software ANSYS. This software is a suite of powerful engineering simulation programs, based on finite element method, which can solve problems ranging from relatively simpler linear analyses to the most challenging non-linear simulations. The analysis of a structure with ANSYS is performed in three stages

- a) Pre-processing P – defining the finite element model and environmental factors to be applied to it.
- b) Analysis solver – solution of finite element model.
- c) Post-processing of results like deformations, contours for displacement, etc., using visualization tools.

Advances in the field of computer-aided engineering during the last two decades have been quite extensive and have led to considerable benefits to many engineering industries. In the building industry, use of advanced finite element tools not only allowed the introduction of innovative and efficient building products, but also the development of accurate design methods. Here SOLID65 and LINK8 elements are used to model RC and LRC beams. Features of these elements are given here and details of the model used in the paper are given in next section.

SOLID65 allows the presence of four different materials within each element, one matrix material (*e.g.* concrete) and a maximum of three independent reinforcing materials. Concrete material is capable of directional integration point cracking and crushing besides incorporating plastic and creep behavior. Reinforcement (which also incorporates creep and plasticity) has uniaxial stiffness only and is assumed to be smeared throughout the element. Directional orientation is accomplished through user specified angles. Concrete material is assumed to be initially isotropic. Element is defined by eight nodes

having three degrees of freedom at each node: translations in the nodal x -, y -, and z -directions. Special features of SOLID65 are: Plasticity, Creep, Cracking, Crushing, Large deflection, and Large strain.

3. Calibration Model

A RCC beam with two point loading case was taken for analysis as shown in the Fig. 1, with: size of the reinforced concrete beam – 100×150 mm; size of loading and support steel plates – 100×100 mm, 6 mm thick; steel reinforcement details: 3 rebars of 12 mm dia at bottom, 2 rebars of 8 mm dia at top, stirrups of 2 legged 6 mm dia at 100 mm c/c.

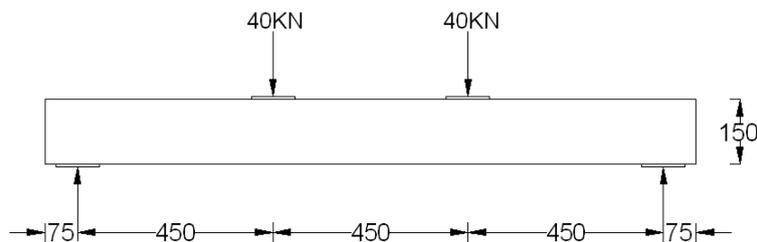


Fig 1 – RCC beam model.

3.1. Steps Involved in the Analysis

A. Step 1: *Pre-Processing*

a) Define the element type, Real constants, Material Models of the concrete model.

b) Create the model.

c) Concrete beam was meshed by mapped mesh.

B. Step 2: *Solution*

a) Apply the boundary conditions and loads.

b) Solve.

C. Step 3: *Post-Processing*

a) Deflections at various locations was obtained.

b) Stress contours were plotted.

4. Modeling Procedure

4.1. Reinforcement Modeling

There are three techniques to model steel reinforcement in finite element models for reinforced concrete (Fig. 2.): the discrete model, the embedded model, and the smeared model. The reinforcement in the discrete model (Fig. 2 a) uses bar or beam elements that are connected to concrete mesh

nodes. Therefore, the concrete and the reinforcement mesh share the same nodes and concrete occupies the same regions as the reinforcement. A drawback to this model is that the concrete mesh is restricted by the location of the reinforcement and the volume of the mild-steel reinforcement is not deduced from the concrete volume.

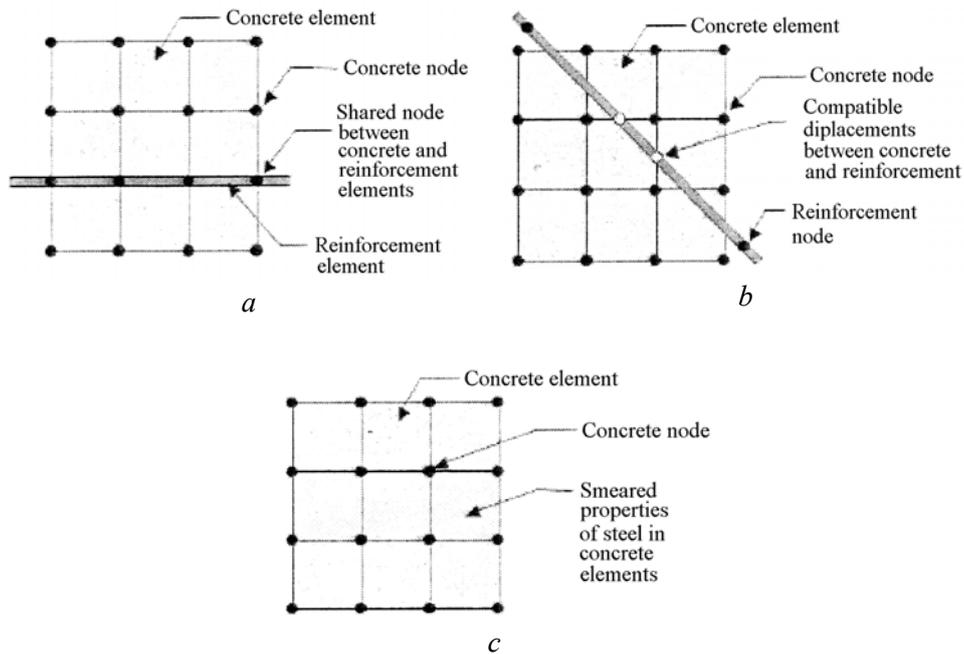


Fig. 2 – Models for reinforcement in reinforced concrete: *a* – discrete; *b* – embedded; *c* – smeared.

The embedded model (Fig. 2 *b*) overcomes the concrete mesh restriction(s).

Because the stiffness of the reinforcing steel is evaluated separately from the concrete elements the model is built in a way that keeps reinforcing steel displacements compatible with the surrounding concrete elements, when reinforcement is complex,

This model is very advantageous. However, this model increases the number of nodes and degrees of freedom in the model, therefore, increasing the run time and computational cost. The smeared model (Fig. 2 *c*) assumes that reinforcement is uniformly spread throughout the concrete elements in a defined region of the FE mesh. This approach is used for large-scale models where the reinforcement does not significantly contribute to the overall response of the structure. Here, in this problem, the discrete model was used to model reinforcement (Fig. 3).

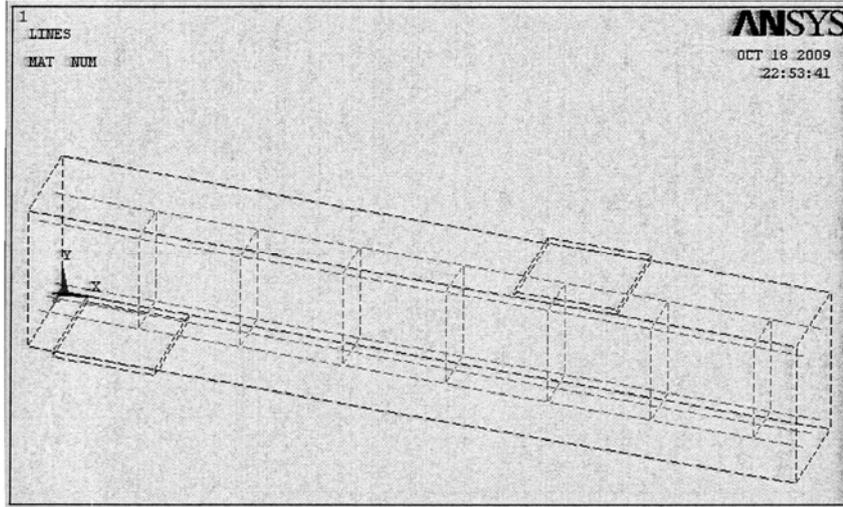


Fig. 3 – Reinforcements modeled in beam.

4.2. Element Types

The Solid65 element was used to model the concrete. This element has eight nodes with three degrees of freedom at each node – translations in the nodal x -, y -, and z -directions. This element is capable of plastic deformation, cracking in three orthogonal directions, and crushing. A schematic of the element is shown in Fig. 4.

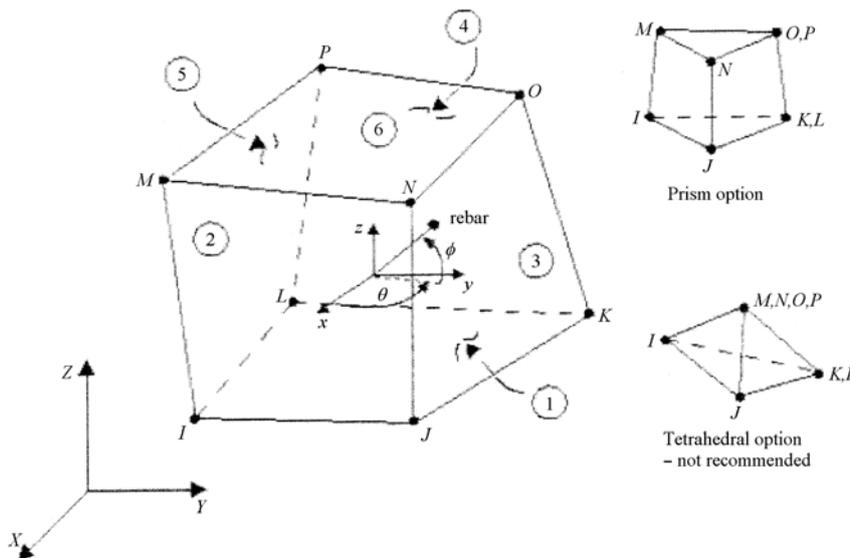


Fig. 4 – Solid65 elements.

A Solid45 element was used for steel plates at the supports and loading of the beam. This element has eight nodes with three degrees of freedom at each node – translations in the nodal x -, y -, and z -directions. The geometry and node locations for this element is shown in Fig. 5.

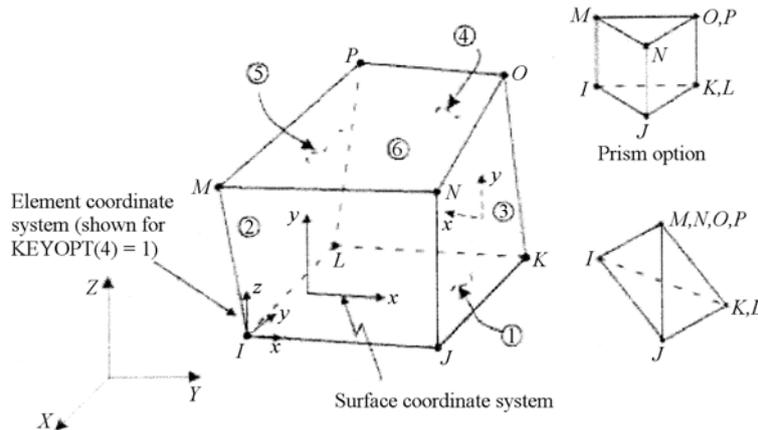


Fig. 5 – Solid45 element.

A Link8 element was used to model steel reinforcement. This element is a 3D spar element and it has two nodes with three degrees of freedom – translations in the nodal x -, y -, and z -directions, being also capable of plastic deformation. This element is shown in Fig. 6.

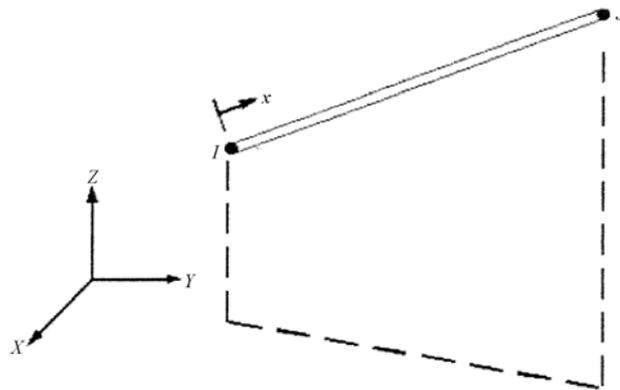


Fig. 6 – Link8 element.

4.3. Real Constants

Individual elements contain different real constants. No real constant set exists for the Solid45 element. Real Constant Set 1 is used for the Solid65 element. It requires real constants for rebar assuming a smeared model. Values

can be entered for material number, volume ratio, and orientation angles. The material number refers to the type of material for the reinforcement. The volume ratio refers to the ratio of steel to concrete in the element. The orientation angles refer to the orientation of the reinforcement in the smeared model.

ANSYS allows the user to enter three rebar materials in the concrete. Each material corresponds to x -, y -, and z -directions in the element. The reinforcement has uniaxial stiffness and the directional orientation is defined by the user. In the present study the beam is modeled using discrete reinforcement. Therefore, a value of zero was entered for all real constants which turned the smeared reinforcement capability of the Solid65 element off. Real Constant Sets 2, 3, 4, and 5 are defined for the Link8 element. Values for cross-sectional area and initial strain were entered

- a) S e t 1: solid 65 all values given zero.
- b) S e t 2. Area 12 mm bar 0.000131 m^2 .
- c) S e t 3. Half 12 mm bar 0.00005655 m^2 .
- d) S e t 4. Area 8 mm bar 0.00005026 m^2 .
- e) S e t 5. Area 6 mm bar 0.00002827 m^2 .

A value of zero was entered for the initial strain because there is no initial stress in the reinforcement.

5. Material Properties

Two material models were given: material 1 for concrete and material 2 for steel, under the linear isotropic material definition.

a) *Material 1: for M20 concrete.*

Modulus of elasticity, $E_c = 5,000\text{sqrt}(20) \text{ N/mm}^2 = 22,360 \text{ N/mm}^2 = 2.236\text{e}+10 \text{ N/m}^2$,

Poisson's ratio, $\nu = 0.25$.

b) *Material 2: for steel.*

Modulus of elasticity, $E_s = 2\text{e}+5 \text{ N/mm}^2 = 2\text{e}+11 \text{ N/m}^2$,

Poisson's ratio, $\nu = 0.30$.

5.1. Meshing

For Solid65 element the mesh was set up such that square or rectangular elements were created. The volume sweep command was used to mesh the steel plate. This properly sets the width and length of elements in the plates to be consistent with the elements and nodes in the concrete portions of the model. The necessary element divisions are noted. The meshing of the reinforcement is a special case compared to the volumes. The reinforcement model was meshed using line elements so that the nodes of the line elements come exactly over the node of the solid elements which are later merged so that both rebar elements and the concrete elements share the same nodes.

5.2. Numbering Controls

The command merge items merge separate entities that have the same location. These items will then be merged into single entities. Caution must be taken when merging entities in a model that has already been meshed because the order in which merging occurs is significant. Merging key points before nodes can result in some of the nodes becoming “orphan”; that is, the nodes lose their association with the solid model. The orphan nodes can cause certain operations (such as boundary condition transfers, surface load transfers, and so on) to fail. Care must be taken to always merge in the order that the entities appear. All precautions were taken to ensure that everything was merged in the proper order. Also, the lowest number was retained during merging.

5.3. Loads and Boundary Conditions

Displacement boundary conditions are needed to constrain the model to get a unique solution. To ensure that the model acts the same way as the experimental beam, boundary conditions need to be applied at points of symmetry and where the supports and loadings exist. The symmetry boundary conditions were set first. The model being used is symmetric about one plane. The boundary conditions for both planes of symmetry are shown in Fig. 7.

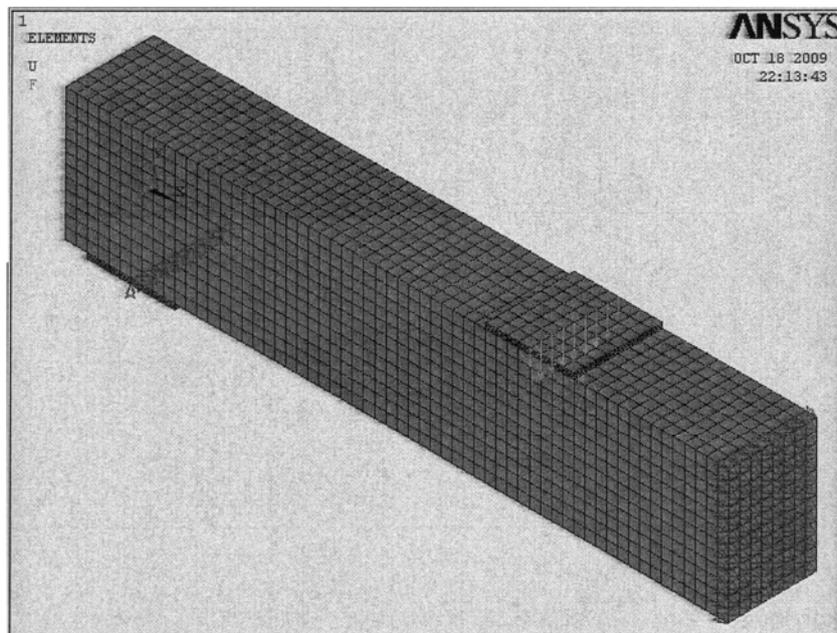


Fig 7– Boundary conditions for planes of symmetry.

Nodes defining a vertical plane through the beam cross-section centroid define a plane of symmetry. To model the symmetry, nodes on this plane must be constrained in the perpendicular direction. Therefore the nodes at the plane of symmetry are given displacement along x -direction, x ; $UX = 0$.

The support was modeled in such a way that a roller was created. A single line of nodes on the plate were given constraint in the UY - and UZ -directions, applied as constant values of 0. By doing this, the beam will be allowed to rotate at the support. The force, P , applied at the steel plate is applied across the entire centerline of the plate.

5.4. Analysis

The finite element model for this analysis is a simple beam under transverse loading. For the purposes of this model, the static analysis type is utilized.

6. Results

a) Stress along X -direction

Maximum tensile stress in concrete, $0.336e+8 \text{ N/m}^2 = 33.6 \text{ N/mm}^2$.

Maximum compressive stress in concrete, $0.388e+8 \text{ N/m}^2 = 38.8 \text{ N/mm}^2$

6.1. Manual Calculation

Maximum bending moment, $40,000 \times 450 = 1.8e+7 \text{ Nmm}$.

Modulus of elasticity of concrete, $E_c = 5,000 \sqrt{2} = 2.236e+4 \text{ N/mm}^2$.

Modulus of elasticity of steel, $E_s = 2e+5 \text{ N/mm}^2$.

Modular ratio, $m = (2e+5)/(2.236e+4) = 8.94$.

Equivalent increased area of concrete at tension zone (3...12 mm dia bars)

$$A_{st} = (8.94 - 1)(3 \times 113) = 2,691 \text{ mm}^2$$

Equivalent increased area of concrete at compression zone (2...8 mm dia bars)

$$A_{sc} = (8.94 - 1)(2 \times 50) = 794 \text{ mm}^2$$

Neutral axis depth,

$$y = (15,000 \times 75 + 2,691 \times 125 + 794 \times 25) / (15,000 + 2,691 + 794) = 80.13 \text{ mm}$$

Gross moment of inertia,

$$bd^3/12 = 100 \times 150^3/12 = 2.815e+7$$

Transformed moment of inertia,

$$I_{tr} = [(2.815e+7) + 15,000 \times 5.13^2] + (2,691 \times 44.87^2) + (794 \times 55.13^2) = 3.637e+7 \text{ mm}^4$$

Maximum compressive stress in concrete,

$$f_c = M y_c / I_{tr} = [(1.8e+7) \times 80.13] / (3.637e+7) = 39.66 \text{ N/mm}^2.$$

Maximum tensile stress in concrete,

$$f_t = M y_t / I_{tr} = [(1.8e+7) \times 69.87] / (3.637e+7) = 34.58 \text{ N/mm}^2.$$

7. Conclusions

The results obtained with Ansys analysis were the following:

1. Maximum compressive stress in concrete, 38.80 N/mm².
2. Maximum tensile stress in concrete, 33.60 N/mm².

These are similar to the result obtained from analysis by strength of materials

1. Maximum compressive stress in concrete, 39.66 N/mm².
2. Maximum tensile stress in concrete, 34.58 N/mm².

Thus the validity of Ansys analysis was satisfactory when compared to the theoretical results of the elastic analysis. The minor difference in results may be due to the fault of the modelling, such as lack of deduction of the volume of the mild-steel reinforcement from the concrete volume.

Received, May 20, 2010

Tamilnadu College of Engineering, India
 Department of Civil Engineering
 e-mail: srinivasan_civil@rediffmail.com
 and
 *Avinachilingam University for Women,
 Department of Civil Engineering

REFERENCES

1. Davison J.S., Fisher J.W., Hammons M.I., Portor J.R., Dinan R.J., *Failure Mechanisms of Polymer Reinforced Concrete Masonry Walls Subjected to Blast*. J. of Struct. Engng., **131**, 8, 1194-1205 (2005).
2. Luccioni B.M., Luege M., *Concrete Pavements Slab under Blast Loads*. IJIE, **32**, 8, 1248-1266 (2006).
3. Mougín J.-P., Perrotin P., Mommessin M., Tonnelo J., Agbossou A., *Rock Fall Impact on Reinforced Concrete Slab: an Experimental Approach*. IJIE, **31**, 2, 169-184 (2005).
4. Dancygier A.N., Yankelevsky D.Z., Jaegermann C., *Response of High Performance Concrete Plates to Impact on Non-Deforming Projectiles*. IJIE, **34**, 12, 1768-1779 (2007).
5. Elfahal M.M., Krauthammer T., Ohno T., Beppu M., Minders S., *Size Effect for Normal Strength Concrete Cylinder Subjected to Axial Impact*. IJIE, **31**, 4, 461-481 (2005).

STUDIUL COMPORTĂRII LA ÎNCOVOIERE A GRINZILOR DIN BETON
ARMAT UTILIZÂND METODA ELEMENTELOR FINITE (ANALIZĂ ELASTICĂ)

(Rezumat)

Componentele structurale din beton iau diferite forme în alcătuirea structurală a unei clădiri. Înțelegerea răspunsului acestor componente sub acțiunea încărcărilor este crucială pentru dezvoltarea unei structuri globale, eficientă și sigură. Diferite metode au fost folosite pentru a studia răspunsul elementelor structurale. Încercările experimentale sunt utilizate pe scară largă ca un mijloc de analiză a elementelor structurale individuale, dar sunt extrem de consumatoare de timp și destul de costisitoare. Utilizarea analizei cu elemente finite devine, astfel, o modalitate atractivă pentru studiul răspunsului sub încărcări a elementelor structurale din beton. Studiul de față compară rezultatele unei analize prin elemente finite cu cele obținute în urma unei analize teoretice convenționale, pentru o grindă din beton armat încărcată transversal.