Finite element modelling and computer aided design

Objekttyp: **Group**

Zeitschrift: **IABSE reports = Rapports AIPC = IVBH Berichte**

Band (Jahr): **62 (1991)**

PDF erstellt am: **30.06.2024**

Computer Models of Concrete Structures

Modèles informatiques pour les structures en béton

Finite-Element-Modelle von Betontragwerken

Vladimir ČERVENKA
Dr. Eng.
Czech Techn. Univ.
Prague, Czechoslovakia

V. Červenka received his Ph.D. degree in 1970 at the University of Colorado in Boulder. He is active in the research of concrete structures at the Czech Technical University in Prague and at Stuttgart University.

Rolf ELIGEHAUSEN
Prof. Dr.
University Stuttgart
Stuttgart, Germany

R. Eligehausen received his Dr. Ing. degree in 1979 in Stuttgart. Since 1984 he is professor for fastening technique at the Institute of Civil Engineering Materials at Stuttgart University.

Radomír PUKL
Dr. Eng.
Czech Techn. Univ.
Prague, Czechoslovakia

R. Pukl received his Ph.D. degree in 1985 at the Czech Technical University in Prague. He works in the field of computational mechanics at the Czech Technical University in Prague and at Stuttgart University.

SUMMARY

The application of the nonlinear finite element analysis of concrete structures as a design tool, is discussed. A computer program for structures in plane stress state is described and examples of its application in the research of fastening technique, and in engineering practice, are shown.

RÉSUMÉ

On discute ici de l'application, en tant qu'outil de dimensionnement, de l'analyse non-linéaire par éléments finis, et ceci dans le cadre des structures en béton armé. Un programme pour des structures en état plan de contrainte est décrit, ainsi que ses applications dans le domaine de la recherche et de le pratique de l'ingénieur.

ZUSAMMENFASSUNG

Die Anwendung der nichtlinearen Finite Elemente Analyse auf Betontragwerke als Entwurfswerkzeug wird diskutiert. Ein Programm für die Konstruktionen im ebenen Spannungszustand wird beschrieben und Beispiele für die Anwendung in der Forschung der Befestigungstechnik und in der Ingenieurpraxis werden vorgestellt.

INTRODUCTION

Due to the complexity of concrete behavior under various states of stress, the development of rational design models is a difficult task. This development is treated in detail by many authors at this Colloquium. Schlaich explains why the design process requires several "design models", because more general approaches are too complex for designers and serve as "research models". MacGregor and Marti show the recent development of engineering design models which have evolved from simple equilibrium truss analogy into more consistent models which take into account strain fields and the constitutive laws. Further refinement of these models would require better constitutive laws, better modeling of the multiaxial stress states and better discretization. Then, of course, the simple models turn into the complex ones. This complexity can be handeled in a rational manner by the finite element method. FE models of concrete structures have been in development for over 20 years and are now at a stage, that thay can be used as design tools, as demonstrated by Scordelis in this Colloquium. The authors believe, that models of all levels of sophistication have their place in design provided that they are rationally based and verified in practice. It is up to the designer to choose the appropriate model under the given circumstances. However, a unified approach for all design models should be accepted which will assure the compatibility between various levels of sophistication. This is also true for safety concepts, which should be extended to the FEM design models. It should be noted, that the CEB has started a significant effort in this respect. It is the purpose of this paper to demonstrate the capabilities of the non-linear finite element method for the analysis of concrete structures and to show its application as an advanced design model. It will be done on the example of the program SBETA, which was developed by the authors. However, we shall first provide a brief summary of the currently used computational models.

CONSTITUTIVE MODELING

The performance and quality of the non-linear finite element analysis depends on all of it's basic components: constitutive model, finite element discretization, solution technique. Of these components, the constitutive model is the most important since it determines the ability of the analysis to model the specific properties of concrete structures. Therefore, we shall make a brief critical overview of some constitutive models which are important for the development of design models.

In the early stages of development two main classes of constitutive models of concrete were used, namely, the models based on the theory of plasticity and the models based on the non-linear elasticity (hypoelasticity). Each of these approaches can well describe some features of concrete behavior, while other features are modeled poorly. The theory of plasticity is suitable for metals but un-suitable for concrete, which is a quasi-brittle material. Hardening plasticity can model the nonlinear behavior of concrete in compression [13], but cannot model cracking and softening behavior. In that sense, the range of application of the plasticity models in concrete structures is restricted to pre-peak compression. Therefore, the plasticity theory was often combined with the brittle-fracturing model for tension [9,2].

Hypoelastic models have been successfully used by many authors [1,15,19]. The orthotropic hypoelastic models have been criticized for their lack of objectivity [3] in the case of rotation of strain fields. Inspite of this, they have the ability to cover a wide range of the concrete behavior, i.e. tension and compression, cracking, and softening.

All of the models described above have typically been used within the "smeared material" approach with a local formulation of the stress-strain laws, where stresses are related to strains at a material point. This "local concept" cannot describe the size effect which is evident from experiments. Introduction of the size effect can be done by means of the "non-local concept", in which the stresses are related to strains in a certain representative volume [4,5].

Cracking has a dominant effect on the non-linear behavior of concrete structures. Therefore, much research is devoted to improve the fracturing model of concrete, as reported recently during a workshop at Torino [7]. The subject is treated in detail in the papers of Hillerborg and König presented at this Colloquium. It is generally accepted that the tensile toughness of concrete is a material property. It is caused by tension softening response after cracking and is characterized by the fracture energy parameter G_f. In finite element analysis there are two kinds of crack models. In a discrete crack model, a crack is formed by disconnecting the nodes of the finite element mesh and introducing a new boundary. After cracking, re-meshing must be performed in order to adjust the element boundary to the crack path [8,17,18] unless the crack follows a predefined path along the existing element mesh. The softening is modelled by stress-crack-opening law of the crack interface. In a smeared crack model, a band of parallel cracks is formed in the entire element volume under consideration (e.g. volume associated with the integration point). The softening of the crack band is derived from the fracture energy parameter [6,10]. Thus, both approaches have the same theoretical basis and in many cases should give similar results.

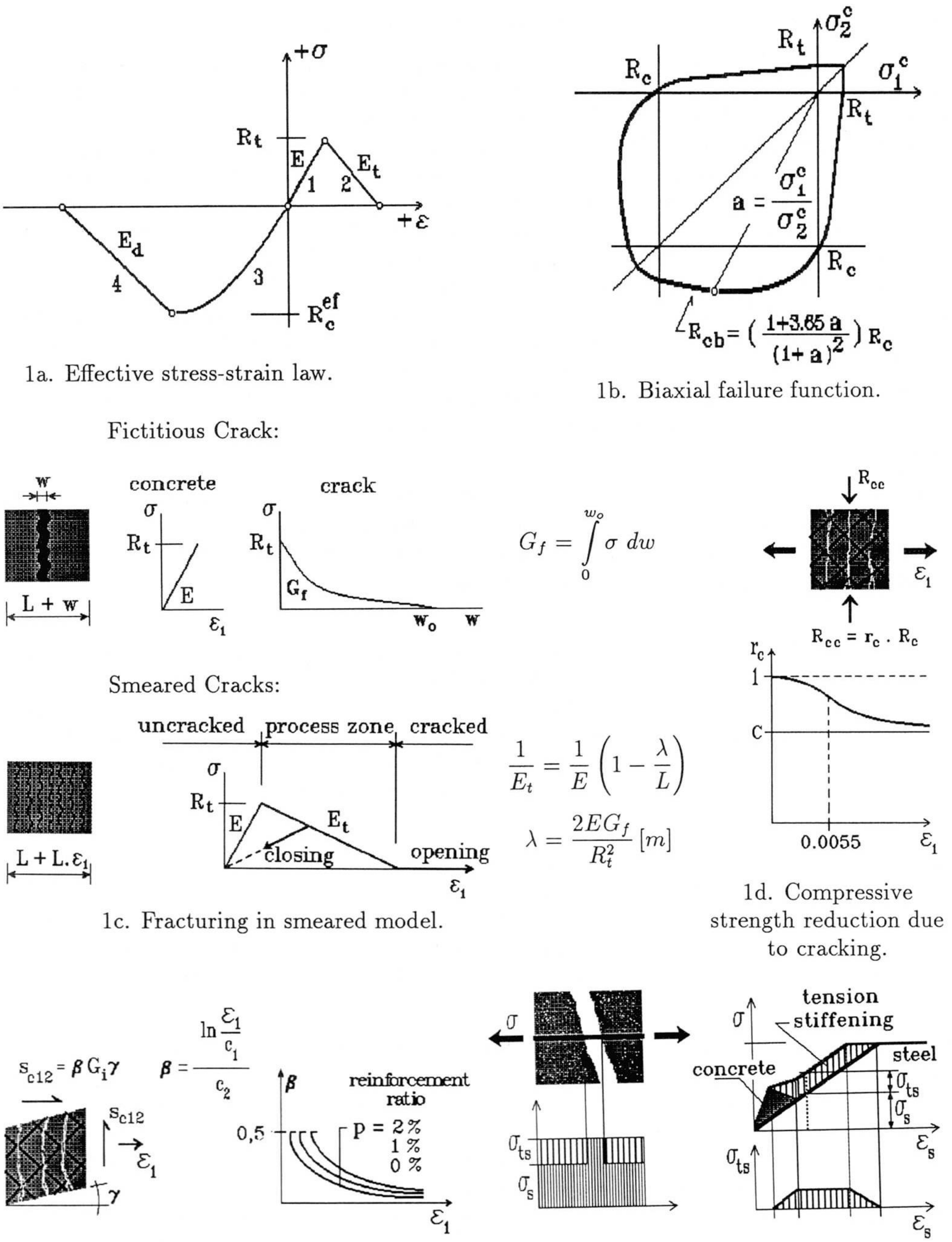
A smeared-crack model based on the orthotropic hypoelastic law can have two basic forms [11]. In a rotated crack model, the axes of principal stress and strain coincide. Rotation of the principal strain axes causes the rotation of material axes (which are coincident with cracks). In a fixed-crack model, the crack direction is determined upon crack initiation, and is kept fixed during the subsequent analysis.

In the fixed-crack model, the crack plane can be subjected to a shear strain and its shear stiffness, representing the aggregate interlock and the dowel action of reinforcement, should therefore be included. This is accomplished by many analysts by means of a shear retention factor, which assignes a constant reduced shear stiffness to the cracked concrete. However, the solution of shear failures is extremely sensitive to the shear retention factor and therefore the use of a constant shear retention factor is not recommended [20]. Improved performance is obtained by decreasing the crack shear stiffness as a function of crack width [11].

Both discrete and smeared crack models have their own merits. The discrete cracks are appropriate for modeling the fracture of plain concrete with one distinct crack, while smeared cracks are more suitable for reinforced concrete. The advantage of the smeared crack model is that it can cover a variety of crack situations ranging from finely distributed cracks in reinforced concrete to a single discrete crack, without modification of the element mesh, as will be demonstrated in this paper.

All previously described smeared models can be classified as macroscopic models. They directly relate stress and strain components. For general stress states and load path situations, they usually require a large number of material parameters. Further improvement can be expected from a microplane model [5,14]. This is a microstructural model in which the material properties, such as the material stiffness matrix, are integrated from elementar behavior of microplanes. It is a three-dimensional model which is unique for all stress states and a wide range of behavior, including cracking, softening, and dilatancy. It is the most general model developed so far for use in the finite element analysis. It is, however, more demanding on computer capacity because the microplanes introduce another level of discretization. The application of the microplane model is presented in a paper by Eligehausen and Ozbolt at this Colloquium.

The above overview is only a brief outline of the present practice with respect to constitutive modeling of concrete structures. Other aspects of FE modeling, such as the method of finite element discretization and solution techniques, shall not be treated here. The interested reader



1a. Effective stress-strain law.

1b. Biaxial failure function.

1c. Fracturing in smeared model.

1d. Compressive strength reduction due to cracking.

1e. Shear stiffness of cracked concrete.

1f. Tension stiffening.

Fig.1 Constitutive model in program SBETA.

can find a number of publications on this subject.

PROGRAM SBETA

Advances in constitutive modeling of concrete and the availability of efficient computers make it possible to produce programs for non-linear finite element analysis which can be used as design tools. Such a program was recently developed by the authors at the Institut für Werkstoffe im Bauwesene at the University of Stuttgart in cooperation with the Building Research Institute of the Czech Technical University in Prague. An overview of the program and examples of it's application in research and engineering practice are presented.

The program SBETA [12] is designed for the analysis of reinforced concrete structures in the plane stress state. It can predict the response of complex concrete structures, with or without reinforcement, in all stages of loading, including failure and post-failure. It can serve as a research tool for the simulation of experiments and for the analysis of experimental results. In design practice it can be used to optimize the geometry and reinforcement detailing and to calculate the load-carrying capacity of structures. It can be also used for the diagnosis of the causes of structural damage or failure.

2a.
2b.
2c.
2d.

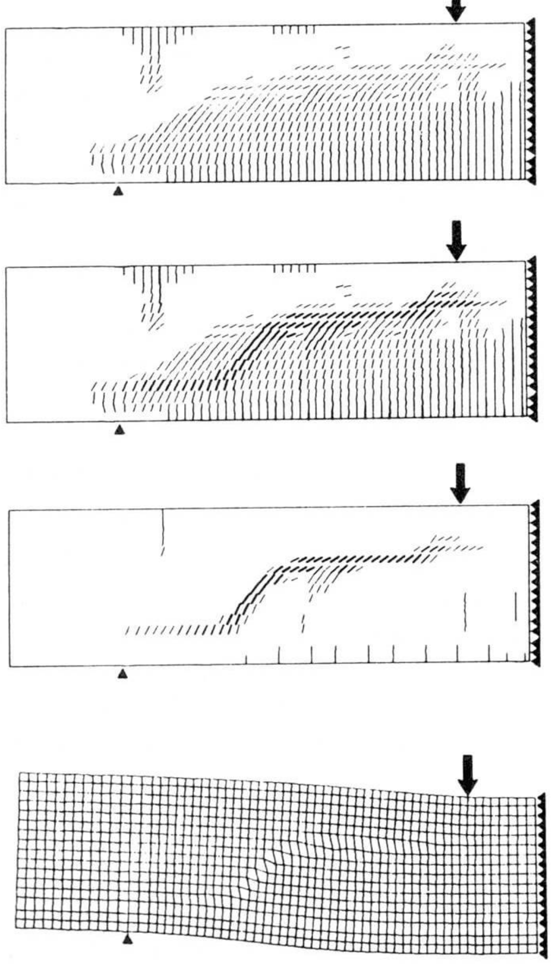


Fig.2 Crack localization in shear.

The constitutive model of the program SBETA is summarized in Fig.1. It is based on the smeared material approach using non-linear elasticity and non-linear fracture mechanics. The behavior of concrete is described by a stress-strain diagram, Fig.1a, which is composed of four branches: non-linear loading in compression, linear loading in tension, and linear softening in both tension and compression. The parameters of this diagram are adjusted in the following way: The peak stress R_c^{ef} is taken from the biaxial failure function of Kupfer, Fig.1b, and the softenning modulus in tension E_t is calculated according to the crack band theory of Bazant [6], Fig.1c.

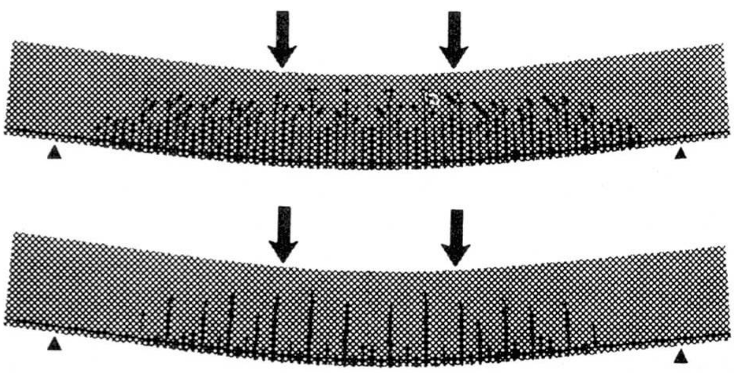
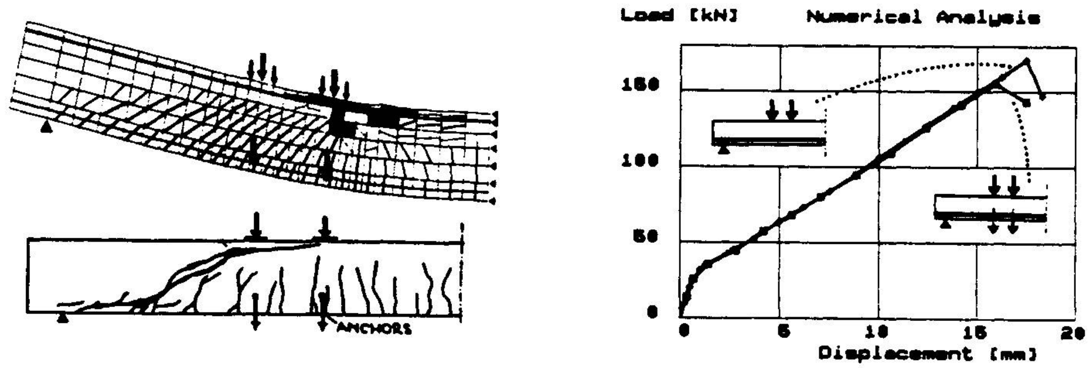


Fig.3 Crack localization in bending.

The modelling of cracked reinforced concrete includes the shear resistance of cracks, Fig.1e, reduction of compressive strength in the direction parallel to the cracks, Fig.1.d and the effect of tension stiffening, Fig.1f. Fixed and rotated crack models are implemented. Reinforcement behavior is bi-linear. A monotonic load history is assumed. A four-node quadrilateral finite element is used for the concrete. The reinforcement can be included either smeared, as a part of the concrete element, or discrete, as a bar element passing through the quadrilateral element. The updated Lagrangean formulation is adopted. The non-linear solution is performed by means of step-wise loading and equilibrium iteration within a load step. Newton-Raphson and arc-length methods are the options for the solution strategy.

The program system SBETA includes a pre-processor, FEM solution program, and an efficient post-processor. A graphical, macro-instruction-based pre-processor generates the FE numerical model. The FEM program can be interactively controlled and runs in several levels of real-time graphics. Thus, the solution process can be observed and solution parameters can be adjusted by the user if necessary. A restart option is available. The dialog-oriented post-processor generates the deformed shapes and images of stress, strain and damage fields (cracking, crushing). An efficient data management (generic names, profile files, etc.) enables the generation of animation sequences , which are important for the detection of failure modes.

A special method has been developed to show crack-localization in the smeared material. Due to strain-softening, deformations localize in narrow bands which indicate the main failure cracks. This is demonstrated with the example of a shear failure of a beam without stirrups. Fig.2a shows the entire crack region (only half of the beam was analyzed), Fig.2b indicates strain-localization within the crack zone, Fig.2c shows the location of the failure crack, and Fig.2d the deformed mesh. Another example of crack-localization for bending is shown in Fig.3.



Analytical and experimental failure patterns. Load-displacement diagrams.

Fig.4 Analysis of the shear resistance of beams with anchors in the tensile zone.

APPLICATIONS IN FASTENING TECHNOLOGY

Fastening technique is a rapidly developing technology in the concrete industry. The load-carrying capacity of concrete anchors rely entirely on the tensile strength and toughness of concrete. In order to understand the mechanics of anchor failure, the authors have performed a number of numerical studies which simulate experimental investigations. In one such investigation [11] a beam with anchors located in the cracked zone was examined, Fig.4. The computer simulation confirmed the experimentally observed reduction of the shear resistance of beams due to the anchor loads introduced into the bottom of the beam. A similar study was conducted by Eligehausen and Kazic for T-beams, using the material model from SBETA in another program, AXIS (see paper presented at this Colloquium).

The concrete cone failure of headed anchor bolts loaded in tension was the subject of an

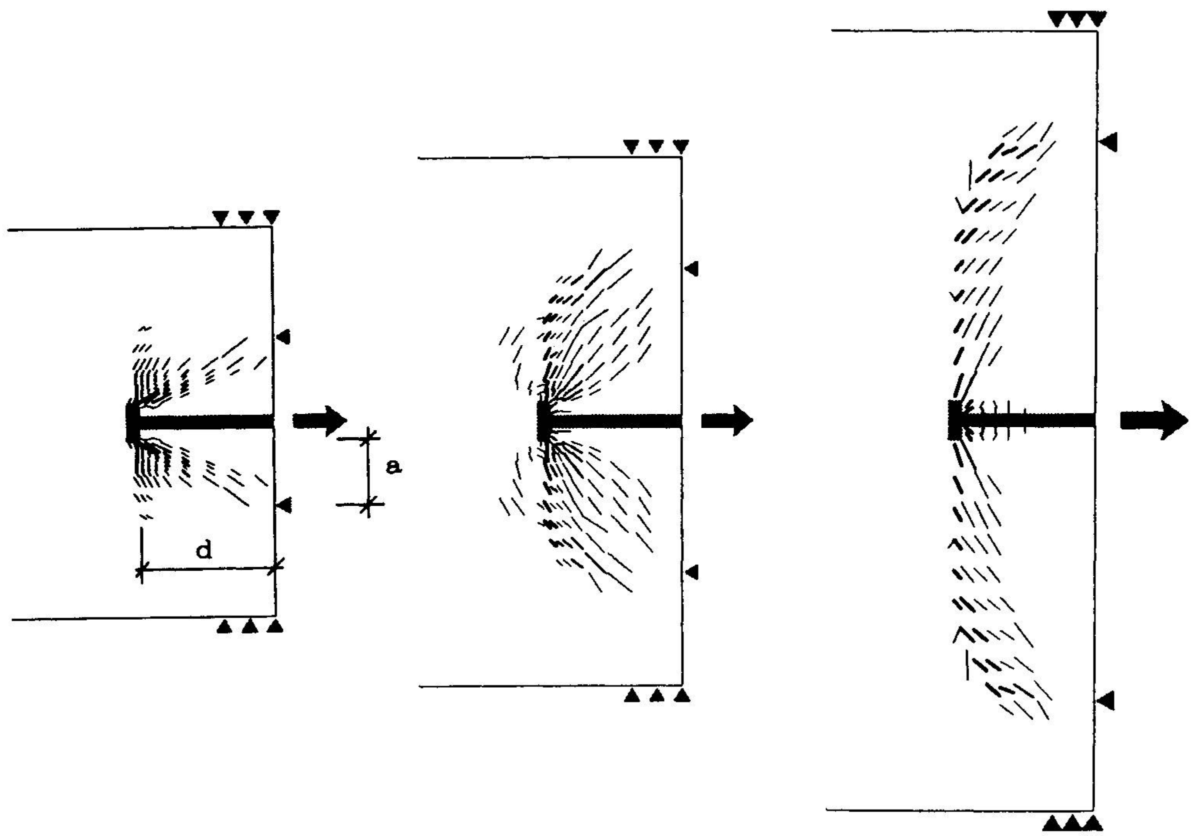


Fig.5 Failure crack patterns of pull-out tests on headed anchors with an embedment depth d=150 mm and three spans a=50, 150, 450 mm.

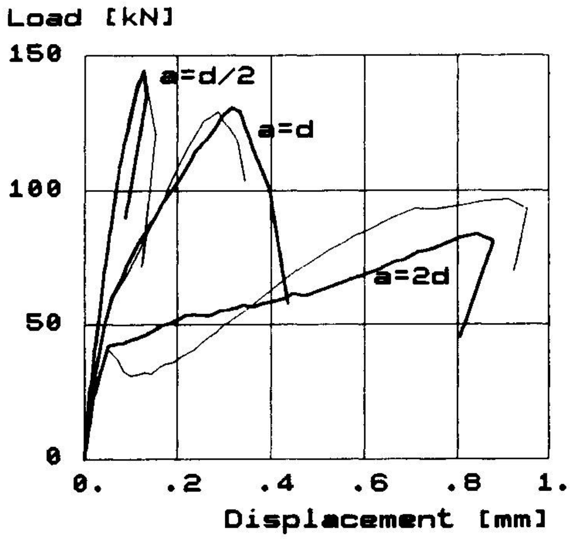


Fig.6 Load-displacement diagrams of pull-out tests for one embedment depth d=150 mm and three spans a=50, 150, 450 mm. Thickness b=100 mm.
thin line - rotated crack model,
thick line - fixed crack model

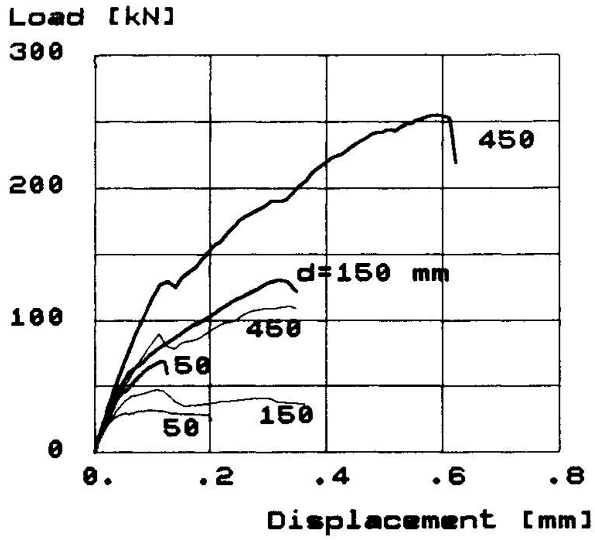
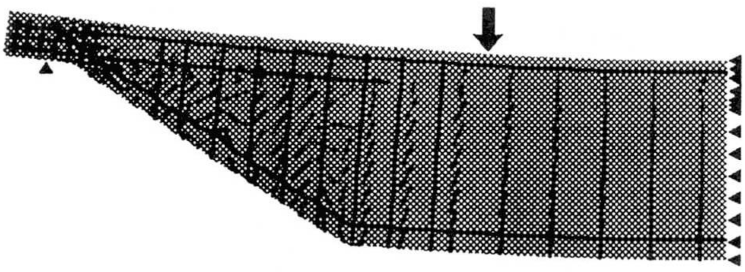
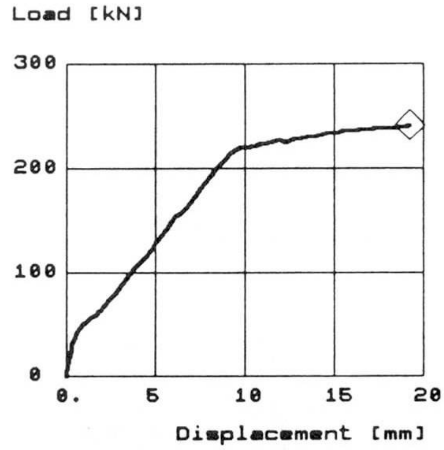


Fig.7 Load-displacement diagrams of pull-out tests for three sizes (d=50, 150, 450 mm) and two lateral constraint conditions. Thickness b=100 mm, a/d=1.
thin line - without constraint,
thick line - with constraint

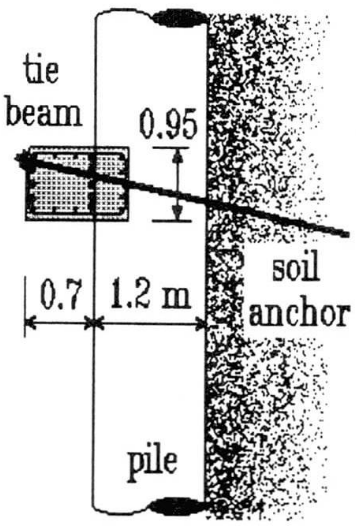


Deformed shape with crack pattern at failure.



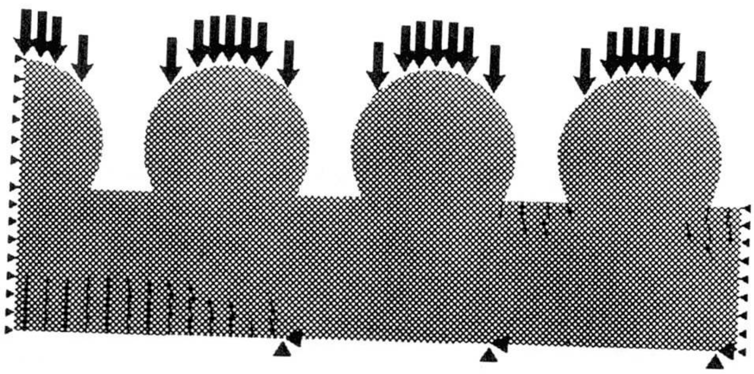
Load-displacement diagram.

Fig.8 Simulation of the ductile failure mode of a tapered beam.
Symetrical half of the beam analyzed.

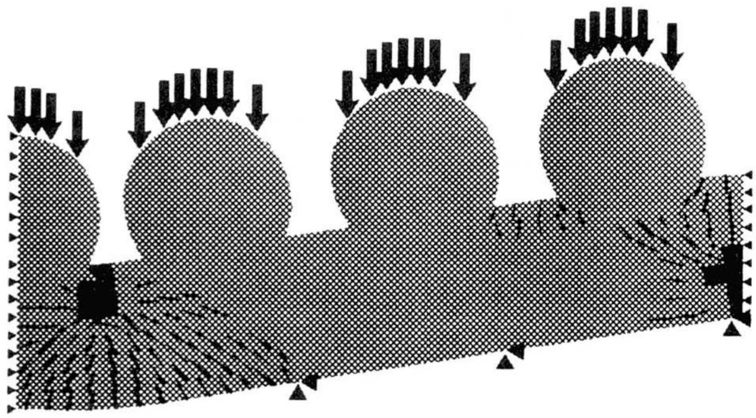


a) Vertical section of retaining wall.

b) p = 467 kN/m



c) failure load p = 974 kN/m



Crack and failure patterns of the tie beam.
Crushed concrete shown by dark shading.

Fig.9 Simulation of the failure of tie beam supported by elastic anchors.

international round-robin analysis organized by the RILEM Commitee on Fracture Mechanics. For this round-robin analysis, the authors have made a parameter study on various 2-D pull-out tests [10]. An example from this study concerning a two-dimensional structure in plane stress state is shown here. The embedment depth d and the shape ratio a/d (a is the support span) were varied. Examples of the failure crack patterns for $d = 150$ mm and three different spans $a = 50, 150, 450$ mm are shown in Fig.5. The load-displacement diagrams for these cases are shown in Fig.6. Fig.7 shows diagrams for $a/d = 1$ using three values for the embedment depth $d = 50, 150, 450$ mm and two assumptions for the lateral constraint (with and without lateral constraint). From these analyses, the influence of the embedment depth (size effect) could be derived . In another application, an SBETA analysis was successfully used to model the behavior of the single anchors and anchor groups subjected to transverse loading [16].

APPLICATIONS IN ENGINEERING PRACTICE

The program SBETA was used at the Prague University for the solution of several practical problems. Two examples are shown here for illustration. In the first example a precast T-beam was analyzed (Fig.8). The web is tapered and the beam is supported by an overhanging flange. The Building Research Institute of T.U. in Prague has performed experimental and numerical studies in order to optimize the reinforcement detailing. Fig.8 shows the failure state of the final solution with a ductile failure mode due to the yielding of reinforcement.

In the second example, a tie beam of a retaining wall was analyzed. The retaining wall consists of vertical reinforced concrete cast-in-place piles which are supported by a horizontal tie beam, Fig.9. The beam is supported by earth anchors which are located between the piles. It was proposed to investigate the cases when several anchors fail. In such a case the tie beam is subjected to bending, while it is laterally constrained. Elastic supports are used to model the anchors. The maximum soil pressures were obtained for various supporting situations. Fig.9 shows two deformed shapes and crack patterns for two load stages. In the failure stage, concrete crushing is also shown. Yielding of reinforcement was also found by the analysis, but it is not shown here.

ROLE OF FEM MODELS IN DESIGN OF CONCRETE STRUCTURES

Non-linear FEM is an advanced tool for modeling the behavior of reinforced concrete structures. It's great potential lies in it's ability to work with steadily developing constitutive laws while satisfying the laws of continuum mechanics and fracture mechanics. As with any model, it is an approximation of reality. However, the degree of approximation can be controlled at all levels of the model. As demonstrated here, these models have their application in situations where simple engineering models are not adequate. In practice they have been successfully used for the design of deep beams, reinforcement detailing (D-regions) and for the diagnosis of the causes of structural failure. In research and development, they have been used for the simulation of experiments, prediction of failure modes and for the analysis of experimental results.

It should be emphasized that a non-linear FE analysis must by supported by efficient graphical tools for pre- and post-processing. Just as drawings are indispensable for structural design, graphics is indispensable for the computer analysis of structural behavior.

References

[1] BATHE, K.J., MEYER, C. - Nonlinear Analysis of R/C Structures in Practice, ASCE Journal of the Structural Division, 108 (ST7), pp. 1605-1622.

[2] BAŽANT, Z.P., KIM, S.S. - Plastic Fracturing Theory for Concrete, ASCE Journal of the Engineering Mechanics Division, 105 (EM3), 1979, pp. 407-428.

[3] BAŽANT, Z.P. - Comment on Orthotropic Models for Concrete and Geomaterials, ASCE Journal of the Engineering Mechanics Division, 109, No.3, June 1983, pp. 849-865.

[4] BAŽANT, Z.P., LIN, F.B. - Nonlocal smeared cracking model for concrete fracture, ASCE Journal of the Engineering Mechanics Division, 114, 1988, pp.2493-2510.

[5] BAŽANT, Z.P., OŽBOLT, J. - Nonlocal microplane model for fracture, damage, and size effect in structures, ASCE Journal of the Engineering Mechanics Division, 116, No.11, Nov. 1990, pp.2485-2505.

[6] BAŽANT, Z.P., OH, B.H. - Crack Band Theory for Fracture of Concrete, Mater. Struct. RILEM, Paris, France, 16, 1983, pp. 155-177.

[7] CARPINTERI, A., Chairman, - International Workshop on the Application of Fracture Mechanics to Reinforced Concrete, Politechnico di Torino, October 6, 1990.

[8] CARPINTERI, A., FERRARA, G., MELCHIORRI, G., VALENTE, S. - The four point shear test on single notched specimens: an experimental and numerical analysis, ECF8 Fracture Behaviour and Design of Materials and Structures, Ed. Firrao, The Politechnico di Torino, Oct.1-5,1990, Chameleon Press Ltd.,London, pp.667-675.

[9] ČERVENKA, V., GERSTLE, K. - Inelastic Analysis of Reinforced Concrete Panels: (1) Theory, (2) Experimental Verification and Application , Publications, IABSE, Zürich, V.31-00,1971, pp.32-45, and V.32-II,1972, pp.26-39.

[10] ČERVENKA, V., PUKL, R., ELIGEHAUSEN, R. - FEM Simulation of Concrete Fracture, ECF8 Fracture Behaviour and Design of Materials and Structures, Ed. Firrao, The Politechnico di Torino, Oct.1-5,1990, Chameleon Press Ltd.,London, pp.667-675.

[11] ČERVENKA, V., PUKL, R., ELIGEHAUSEN, R. - Computer simulation of anchoring technique in reinforced concrete, Proceedings SCI-C 1990, Zell am See, Austria, 4-6 April, 1990, Ed. N.Bićanić, H.Mang, Pineridge Press, Swansea, U.K., pp.1-7.

[12] ČERVENKA, V., ELIGEHAUSEN, R., PUKL, R. - SBETA Computer Program For Nonlinear Finite Element Analysis of Concrete Structures, Part 1, Program Description, Part 2, User's Manual, Part 3, Examples of Applications, Mitteilungen des Instituts für Werkstoffe im Bauwesen No.1990/1, Universität Stuttgart.

[13] CHEN, W.F. - Plasticity in Reinforced Concrete, McGaw-Hill, N.Y., 1981.

[14] ELIGEHAUSEN, R., OŽBOLT, J. - Size effect in anchorage behavior, ECF8 Fracture Behaviour and Design of Materials and Structures, Ed. Firrao, The Politechnico di Torino, Oct.1-5,1990, Chameleon Press Ltd.,London, pp.721-727.

[15] ELWI, A.A., MURRAY, D.W. - Nonlinear Analysis of Axisymmetric Reinforced Concrete Structures, Structural Engineering Report No.87, University of Alberta, Edmonton, 1980.

[16] FUCHS, W. - Tragverhalten von Befestigungen unter Querlast in ungerissenem Beton, Mitteilungen des Instituts für Werkstoffe im Bauwesen No.1990/2, Universität Stuttgart.

[17] HILLERBORG, A., MODEER, M., PETERSSON, P.E. - Analysis fo crack formation and crack growth in concrete by means of fracture mechanics and finite elements, Cement and Concrete Research, 6,1976, pp.773-182.

[18] INGRAFFEA, A.R., GERSTLE, W.H. - Nonlinear Fracture Models for Discrete Crack Propagation, Application of Fracture Mechanics to Cementitious Composites, Editor: Shah, Martinus Nijhoff Publishers, 1985, pp.247-286.

[19] OTTOSEN, N.S. - Constitutive Model for Short-Time Loading of Concrete, ASCE Journal of the Engineering Mechanics Division, 105, (EM1), 1979, pp.127-141.

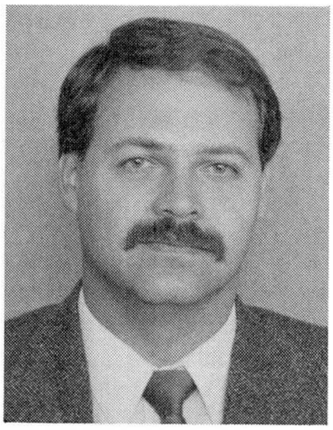
[20] ROTS, J.G, NAUTA, P, KUSTERS, G.M.A., BLAAUWENDRAAD, J. - Smeared crack approach and fracture localization in concrete, HERON, vol.30, No.1, 1985.

Analyses Based on the Modified Compression Field Theory

Analyses basées sur la théorie modifiée du champ de contraintes en compression

Untersuchungen auf Grundlage der modifizierten Druckfeldtheorie

Frank J. VECCHIO
Assoc. Prof.
Univ. of Toronto
Toronto, ON, Canada



Frank J. Vecchio obtained his doctorate in civil engineering at the University of Toronto in 1981, and joined the department in 1985. His research interests relate primarily to the constitutive modelling, nonlinear analysis of reinforced concrete.

SUMMARY

The Modified Compression Field Theory provides a unified, rational approach to the analysis of structural concrete elements under general in-plane stress conditions. Cracked reinforced concrete is treated as a nonlinear elastic orthotropic material based on a smeared, rotating crack assumption. Formulations satisfying equilibrium and compatibility conditions are developed, and new constitutive relations for the component materials are defined. The theory is incorporated into several analytical algorithms. Procedures have been developed for the analysis of membrane structures, beams, plane frames, plates and shells, and three-dimensional solids. Extensive corroborative testing has shown the theory to be able to model accurately the response of structural concrete.

RÉSUMÉ

La théorie du champ de contraintes en compression fournit une approche rationnelle et unifiée à l'analyse des éléments structuraux de béton, sous des conditions de contraintes planes. Le béton armé fissuré est traité comme un matériau orthotropique élastique non-linéaire, se basant sur un concept de fissuration uniforme. Des expressions satisfaisant les conditions d'équilibre ainsi que de comptabilité sont développées, et de nouvelles relations constitutives sont définies pour les matériaux. Des procédures sont développées pour l'analyse des membranes, poutres, cadres plans, plaques, coques et solides tri-dimensionnels. Une série d'essais démontrent que la théorie peut prédire correctement le comportement du béton.

ZUSAMMENFASSUNG

Die modifizierte Druckfeldtheorie stellt eine einheitliche und rationale Methode für die Untersuchung von Konstruktionsbeton-Elementen im ebenen Spannungsfeld dar. Gerissener Stahlbeton wird – basierend auf einem Rissmodell mit verschmierten und rotierenden Rissen – als nichtlineares, elastisches orthotropisches Material behandelt. Es werden Formulierungen, die Gleichgewichts- und Verträglichkeitsbedingungen genügen, entwickelt und neue konstitutive Beziehungen für die Teil-Materialien definiert. Die Theorie wird in mehrere analytische Algorythmen eingebaut. Für die Untersuchung von Membranwerken, Balken, ebenen Rahmen, Platten, Schalen und drei-dimensionalen Festkörpern werden Berechnungsmethoden entwickelt. Zahlreiche Versuche haben gezeigt, dass die Theorie das Verhalten von Konstruktionsbeton genau beschreiben kann.

1. INTRODUCTION

A unified, rational approach to the analysis and design of concrete structures does not currently exist. The difficulty stems from an inability to develop general material behaviour models for structural concrete that are simple, rational, consistent and accurate. This was clearly brought to light in the results of a competition conducted 10 years ago [1].

The international competition was organized to compare analytical methods for predicting the response of reinforced concrete elements subjected to general two-dimensional stress states. Entrants were asked to predict the strength and load-deformation response of four panels tested in a University of Toronto research program. While many of the predictions received were based on analyses conducted using complex nonlinear analysis procedures, a wide scatter in the predictions was evident. Clearly, the ability to accurately model behaviour, using finite element procedures or otherwise, was not very good due to a generally poor state-of-the-art in constitutive modelling.

Much work has been conducted recently at the University of Toronto in an effort to develop improved models. One outcome has been the formulation of the Modified Compression Field Theory.

2. MODIFIED COMPRESSION FIELD THEORY

The Compression Field Theory (CFT) was first developed by Collins and Mitchell [2, 3], and co-workers, for the analysis of beams under combined torsion, shear and flexure. The theory provided a conceptual model for the behaviour of cracked reinforced concrete under two-dimensional stress states, following essentially a smeared, rotating crack idealization. Formulations satisfying conditions of equilibrium and compatibility in a continuum were based on average values of stress and strain in the component materials. It was assumed that the directions of the principal stresses coincided with the directions of the principal strains. Concrete in compression was modelled using the Hognestad parabolic curve; concrete in tension was assumed to carry no stress after cracking.

To develop more accurate constitutive relations for cracked reinforced concrete, a new testing facility was developed and utilized in an extensive experimental investigation. The 'Shear Rig' was capable of loading reinforced concrete panels under general conditions of uniform in-plane stress. In particular, it allowed for the first time anywhere, the testing of reinforced concrete under conditions of pure shear, as well as shear combined with biaxial normal stresses. [More recently, at the University of Toronto, the 'Shell Element Tester' facility was developed permitting the testing of larger elements under conditions involving both in-plane and out-of-plane loading]. In the initial series, 30 panels were tested in the years 1979-1981 [4].

Based on the results of these initial tests, the Modified Compression Field Theory (MCFT) was developed [4]. The refinements introduced by the MCFT related to: i) strain softening of concrete in compression, due to the action of transverse tensile strains; ii) tension stiffening effects in cracked concrete in tension, due to the continued presence of tensile stresses in concrete between cracks; iii) the transfer of stresses across cracks (ie. the need to consider local stress conditions at crack surfaces). These effects were embodied in the analytical model by a new set of constitutive relations.

3. EXPERIMENTAL CORROBORATION

To corroborate and refine the analytical models, and the finite element formulations developed subsequently, several test programs were undertaken involving membrane elements. Bhide and Collins [5] tested a series of thirty-one uniaxially reinforced concrete panels under various combinations of tension and shear. A series of ten panels, with centre perforations, were tested to study behaviour in situations involving stress disturbances due to structural discontinuities [6]. To examine predicted behaviour under conditions where a uniform system of cracks could not be assumed, panels containing a pre-cracked uniaxially reinforced shear plane were tested [7]. The behaviour of prestressed elements was investigated by Marti and Meyboom, through the testing of three large-scale shell elements. To investigate possible scale effects, panels of varying sizes were tested under similar load conditions using the Shear Rig, the Shell Element Tester and facilities elsewhere. The response of high strength concrete elements was investigated by test programs involving four shell elements and two panel elements. Finally, the influence of cyclic loading on the constitutive response of cracked structural concrete was briefly explored via three shell element tests [8]. Several other experimental programs are currently in progress.

Thus, since the development of the MCFT, several test programs have been undertaken involving a total of over one hundred membrane type specimens. The conditions investigated have encompassed a wide range of specimen construction details and loading conditions. In all cases, the MCFT was able to accurately predict behaviour in terms of crack patterns, deformations, reinforcement stresses, ultimate strengths and failure modes. Detailed comparisons of experimental versus theoretical response, for each of the test series, can be found in the references cited.

4. MEMBRANE STRUCTURES

The MCFT was incorporated into nonlinear finite element analysis algorithms, for membrane structures, using several alternative approaches. Adeghe [9] introduced the model's constitutive relations into the standard program ADINA. Stevens et al [8] developed program FIERCM, a nonlinear algorithm based on a tangent stiffness solution procedure and utilizing high-order quadratic strain elements. Cook and Mitchell [10] developed program FIELDS, also a nonlinear finite element algorithm based on a tangent stiffness scheme. Lastly, a secant-stiffness based algorithm was used by Vecchio [11, 12] in developing program TRIX.

In the secant-stiffness formulation, finite elements were developed such as to completely represent the formulations of the MCFT. By incorporating these elements into a iterative linear elastic procedure, nonlinear analysis capability was achieved. The resulting procedure demonstrated good numerical stability and good convergence characteristics. Extensions to the formulations were later developed [12] which permitted the consideration of prestrain effects in the component materials. Prestressing of the reinforcement, shrinkage or expansion of the concrete, or other types of strain offset effects could then be considered.

The accuracy of the finite element analyses were examined by predicting the response of the various membrane elements tested, as well as by modelling several 'benchmark' tests reported in the literature. In general, aspects of response pertaining to strength, stiffness, cracking patterns, reinforcement stresses, concrete distress regions, and failure modes were all predicted with good accuracy [6, 7, 8, 9, 10, 11, 12].

5. BEAM SECTIONS

The MCFT formulations were adapted to the analysis of structural concrete beams subjected to combined shear, flexure and axial loads by developing a layered section analysis procedure (program SMAL) [13]. In the layered procedure, the only sectional compatibility requirement enforced was that plane sections remain plane. Sectional equilibrium requirements included a balancing of the shear flows as well as of the member end actions. Beyond this, uniform stress conditions were assumed to exist in each concrete layer and each longitudinal rebar element. Conditions of compatibility and equilibrium in the concrete layers were enforced according to the formulations of the MCFT. Thus, given the sectional forces, the two-dimensional stress and strain conditions within each layer of the section could be computed. An iterative solution algorithm was employed.

The formulations were found to provide reasonably accurate predictions of the load-deformation response, ultimate load and failure mode of beam specimens. For example, the thirty-five T-beams tested at the University of Washington, under various conditions of shear and axial load, were modelled analytically and very good agreement was obtained. The ratio of experimental to predicted strength had a mean of 1.01 and a coefficient of variation of 15% [13].

6. PLANE FRAMES

A nonlinear frame analysis procedure (ie. program TEMPEST) was developed to model the response of reinforced concrete plane frames subjected to thermal and mechanical loads [14, 15]. The procedure was primarily based on performing rigorous sectional analyses of frame members at several points along their length, and then enforcing these sectional responses in the overall response of the frame. This was done by various means; for example, by defining effective sectional stiffness factors, unbalanced forces, fixed-end forces, or combinations of these. The sectional analyses were performing in the manner previously described for beam sections, but assuming uniform shear flow distributions through member cross-sections. Thus, the multi-layer section analyses incorporated into the frame procedure considered two-dimensional stress conditions in the manner of the MCFT.

Two large-scale frame models were fabricated and tested to corroborate the analysis program [15, 16]. The one-span, two-storey models had a centre-to-centre span of 3.5m and an overall height of 4.6m. The first specimen was subjected to a concentrated transverse load applied at the midspan of the first-storey beam. The second was subjected to constant axial column loads combined with monotonically increasing lateral load applied at the top storey level. The analysis procedure was found to give reasonably accurately predictions of the complex nonlinear response of the test frames. Shear-related effects contributed significantly to the deformations, and membrane action and geometric nonlinearity significantly affected the load capacities. These effects could only be captured in the theoretical analyses by considering two-dimensional stress conditions within the frame members.

7. PLATES AND SHELLS

Analytical procedures based on the MCFT were also developed for modelling the response of reinforced concrete plate and shell elements. Program SEP was initially developed by Kirschner and Collins [17] to predict the response of elements subjected to membrane forces, bending moments and torsional moments. A strain compatibility approach was coupled with a layered element technique. Appropriate assumptions were made regarding the distribution of strains across the thickness of the element. MCFT-derived constitutive relations were then used to calculate stresses within each of the layers. Program SEP was then further developed to account for the influence of out-of-plane shear [17]. This required a consideration of tri-axial stress conditions in each of the layers of the shell element, making the computational algorithm complex, time-consuming, and somewhat unstable. Thus, program SHELL474 was developed by Adebar and Collins [18]. Using the layered element approach, and program SEP as a subroutine, SHELL474 introduced a simplification whereby only the middle layer of the element was analyzed for the three-dimensional stress conditions.

The finite element analysis program APECS was developed to provide global nonlinear analysis capability [19]. The 42-degree-of-freedom quadrilateral shell element formulated allows the modelling of plate or shell structures containing arbitrary in-plane and out-of-plane reinforcement. Also employing a MCFT-based layered-element formulation, the analysis procedure is able to model response to general loading conditions, including a consideration of out-of-plane shear, material prestrains, membrane action, tension stiffening effects, and local stress conditions at crack locations.

Several series of specimens were tested in the 'Shell Element Tester' to corroborate the analytical formulations. Kirschner [17] and Polak [19] tested a total of ten shell elements under various conditions of membrane forces combined with flexure. Adebar [18] tested 9 shell elements and 27 beam elements involving conditions of membrane forces combined with out-of-plane shear. Other test programs are currently underway. In applying the analytical procedures to model the behaviour of the test specimens, good agreement was generally found.

8. THREE-DIMENSIONAL SOLIDS

A nonlinear finite element program (SPARCS) was developed for the analysis of reinforced concrete solids [20]. The program was derived using the analytical models previously described for membrane elements, extrapolated to three dimensions. Thus, an iterative linear elastic formulation was used in which secant moduli were defined and progressively refined according to current local stress/strain states. The three-dimensional constitutive relations incorporated into the formulation were ones extrapolated from the two-dimensional models of the MCFT. An eight-noded regular hexahedral element was formulated accordingly.

To obtain an indication of the potential accuracy of the three-dimensional formulation, torsion beams tested by Onsongo [21] were modelled. The hollow, rectangular beams were subjected to varying conditions of torsion and flexure. They represented a stringent test of the analysis procedure because of the complex loading condition, and because the beams were generally over-reinforced and governed by failure of the concrete. The ultimate load and failure mode of the ten beams tested were predicted very well. The ratio of experimental to theoretical strength had a mean of 0.99 and a coefficient of variation of 6.1%. The load-deformation responses and local strain conditions were also modelled well.

9. DISCUSSION

Breen [22] has pointed to the need for "developing unified, consistent analysis and design approaches" which can be universally applied to "continuum of structural concrete", but has cautioned that "we must dispel the present preoccupation with complex analysis procedures". Further, he has suggested that "as nonlinear analysis packages develop, it is possible nonlinear finite element analysis may be useful" [in this regard]. Scordelis [23] has echoed these feelings, stating that "there is a need to develop a unified approach for the analysis and design of the entire spectrum of ... structural concrete systems which takes advantage of the latest analytical and experimental research, materials, computers, and practical design and construction experience". He adds that "it is desirable to have refined analytical models and methods of analysis which can trace the structural response ... under increasing loads through their elastic, cracking, inelastic, and ultimate ranges".

The MCFT is a conceptual and mathematical model for structural concrete that is consistent with the perceived needs stated by Breen and by Scordelis. As has been shown, the theory presents a unified, rational analysis approach that can be applied to structural concrete in many of its various forms and applications. The theory's formulations are simple, transparent and easy to implement. Further, emphasis is placed on accurately describing the constitutive behaviour of structural concrete, as oppose to developing complex and sophisticated mathematical solutions. The constitutive models incorporated have been corroborated extensively with experimental data. Thus, while MCFT analysis procedures may not always represent the optimal solution to routine analysis and design applications, they do provide for a consistent approach in performing an accurate 'trace of structural response' in situations where it is deemed necessary.

10. CONCLUSIONS

The Modified Compression Field Theory (MCFT) provides a simple, rational framework for modelling the behaviour of cracked reinforced concrete. The theory presents formulations for satisfying conditions of strain compatibility and stress equilibrium in a continuum and, most importantly, defines realistic constitutive relations for the concrete and reinforcement.

The accuracy and range of applicability of the MCFT has been extensively corroborated with additional experimental research since its formulation. The test programs undertaken, conducted on simple panel and shell elements under well controlled conditions, have covered a wide range of structural parameters and loading conditions. The theory, still in essentially its original form, has been found to provide fairly consistent and accurate results in all cases.

The simplicity of the MCFT formulations has allowed them to be easily adopted into various analytical algorithms. Procedures have been developed for the nonlinear analysis of membranes, beams, plane frames, plates and shells, and three-dimensional solids.

In applying the analysis procedures to the modelling of more complex structural systems, generally good correlation was found between predicted and observed responses. The theory was found to provide accurate modelling of crack patterns, deformations, reinforcement stresses, ultimate strengths, and failure modes.

REFERENCES

1. Collins, M.P., Vecchio, F.J., and Mehlhorn, G., "An International Competition to Predict the Response of Reinforced Concrete Panels", Canadian Journal of Civil Engineering, V. 12, No. 3, Sept 1985, pp 626-644.

2. Mitchell, D., and Collins, M.P., "Diagonal Compression Field Theory - A Rational Model for Structural Concrete in Pure Torsion", ACI Journal, V. 71, No. 8, Aug 1974, pp 396-408.

3. Collins, M.P., "Towards a Rational Theory for RC Members in Shear", Proc., ASCE, V. 104, ST4, Apr 1978, pp 649-666.

4. Vecchio, F.J., and Collins, M.P., "The Modified Compression Field Theory for Reinforced Concrete Elements Subjected to Shear", ACI Journal, V. 83, No. 2, 1986, pp 219-231.

5. Bhide, S.B., and Collins, M.P., "Influence of Axial Tension on the Shear Capacity of Reinforced Concrete Members", ACI Structural Journal, V. 86, No. 5, Sept - Oct 1989, pp 570 - 581.

6. Vecchio, F.J., and Chan, C.C.L., "Reinforced Concrete Membrane Elements with Perforations", ASCE Journal of Structural Engineering, V. 116, No. 9, Sept 1990, pp 2344-2360.

7. Vecchio, F.J., and Nieto, M., "Shear-Friction Tests on Reinforced Concrete Panels", ACI Structural Journal, (accepted for publication Sept 1990).

8. Stevens, N.J., Uzumeri, S.M., and Collins, M.P., "Analytical Modelling of Reinforced Concrete Subjected to Monotonic and Reversed Loadings", Publication No. 87-1, Department of Civil Engineering, University of Toronto, Jan 1987, 201 pp.

9. Adeghe, L.N., "A Finite Element Model for Studying Reinforced Concrete Detailing Problems", Ph.D. Thesis, University of Toronto, 1986, 264 pp.

10. Cook, W.D., and Mitchell, D., "Studies of Disturbed Regions near Discontinuities in Reinforced Concrete Members", ACI Structural Journal, V. 85, No. 2, Mar - Apr 1988, pp 206-216.

11. Vecchio, F.J., "Nonlinear Finite Element Analysis of Reinforced Concrete Membranes", ACI Structural Journal, V. 86, No. 1, Jan - Feb 1989, pp 26-35.

12. Vecchio, F.J., "Reinforced Concrete Membrane Element Formulations", ASCE Journal of Structural Engineering, V. 116, No. 3, March 1990, pp 730-750.

13. Vecchio, F.J., and Collins, M.P., "Predicting the Response of Reinforced Concrete Beams Subjected to Shear Using Modified Compression Field Theory", ACI Structural Journal, V. 85, No. 3, May - June 1988, pp 258-268.

14. Vecchio, F.J., "Nonlinear Analysis of Reinforced Concrete Frames Subjected to Thermal and Mechanical Loads", ACI Structural Journal, V. 84, No. 6, Nov - Dec 1987, pp 492-501.

15. Vecchio, F.J., and Emara, M.B., "Shear Deformations in Reinforced Concrete Frames", ACI Structural Journal, (submitted for publication July 1990).

16. Vecchio, F.J., and Balopoulou, S., "On the Nonlinear Behaviour of Reinforced Concrete Frames", Canadian Journal of Civil Engineering, October 1990.

17. Kirschner, U., and Collins, M.P., "Investigating the Behaviour of Reinforced Concrete Shell Elements", No. 86-09, Department of Civil Engineering, University of Toronto, Sept 1986, 83 p.

18. Adebar, P., "Shear Design of Concrete Offshore Structures", Ph.D. Thesis, University of Toronto, 1989, 197 p.

19. Polak, M.A., "Nonlinear Finite Element Analysis of Reinforced Concrete Shells", Ph.D. Thesis (in progress).

20. Vecchio, F.J., and Selby, R.G., "Towards Compression Field Analysis of Reinforced Concrete Solids", ASCE Journal of Structural Engineering, (accepted October 1990).

21. Onsongo, W., "The Diagonal Compression Field Theory for Reinforced Concrete Beams Subjected to Combined Torsion, Flexure and Axial Load", Ph.D. Thesis, Univ. of Toronto, 1978, 246 pp.

22. Breen, J.E., "Why Structural Concrete", Introductory Report, IABSE Colloquium, Stuttgart, 1991.

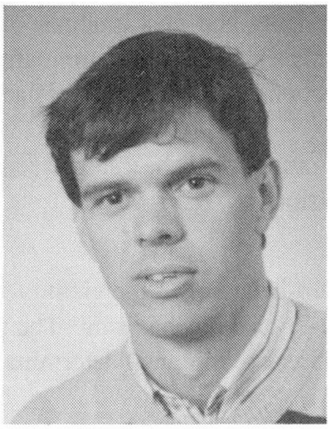
23. Scordelis, A., "Analysis of Structural Concrete Systems", IABSE Colloquium, Stuttgart, 1991.

Computational Bond Models: Three Levels of Accuracy

Modèles de calcul de l'adhérance selon trois niveaux de précision

Rechenmodelle für den Verbund: drei Genauigkeitsstufen

Jan G. ROTS
Research Engineer
Delft Univ. of Technology
Delft, The Netherlands



Jan Rots obtained his civil engineering degree in 1983 and his doctor's degree in 1988, both from Delft University of Technology. He has been a member of the Civil Engineering Department at Delft University of Technology and of the Computational Mechanics Department of TNO Building and Construction Research since 1983.

SUMMARY

The paper reviews three computational bond models of decreasing degree pf precision. Firstly, a detailed resolution of bond-slip is given, which aims at supporting experimental work. Secondly, the bond-slip law is used as input to interface elements with a view to predicting crack spacing and width. Thirdly, embedded reinforcement techniques for global analysis in engineering practice are discussed. Examples demonstrate the capability of nonlinear finite element analysis to simply and correctly reveal «strut-and-tie» systems after stress redistribution.

RÉSUMÉ

Cet article passe en revue des modèles de calcul de l'adhérence, et ceci, selon 3 niveaux décroissants du degré de précision. Premièrement une résolution détaillée est donnée du problème de glissement-adhérence, qui vise à étayer un travail expérimental. Deuxièmement, la loi de glissement-adhérence est introduite dans le cas des interfaces afin de prédire la largeur et l'espacement des fissures. Troisièmement, les techniques d'enrobage des armatures de la pratique sont analysées globalement. Des exemples démontrent les capacités d'une analyse non-linéaire par éléments finis qui révèlent simplement l'efficacité des systèmes basés sur l'analogie du treillis après redistribution des effets.

ZUSAMMENFASSUNG

Der Artikel behandelt Rechenmodelle für den Verbund mit den drei folgenden Stufen abnehmender Genauigkeit. Zuerst wird eine ausführliche Lösung der Verbund-Schlupf-Beziehung gegeben, die zur Ergänzung von Versuchvorhaben dient. Zweitens wird das Verbund-Schlupf-Gesetz als Eingabe für Kontaktflächen-Elemente benutzt, mit der Zielsetzung, Rissabstände und Rissbreiten zu bestimmen. Drittens werden die Verfahren mit in Beton eingebetteten Bewehrungsstäben diskutiert, die zur baupraktischen Berechnung gesamter Systeme dienen. Einige Beispiele demonstrieren die Fähigkeiten nichtlinearer Finite Element Berechnungen, einfach und richtig Stabwerkmodelle nach Spannungsumlagerungen aufzuzeigen.

1. INTRODUCTION

Computational simulators for concrete can be applied in three different ways:

- to support experimental research,
- to develop, verify or validate design rules and hand calculation methods,
- to incidentally analyse particular structures.

The former two types of applications are not meant to be made at the desk of a practising engineer. Instead, these are made by researchers, students and post-graduate students who use the computational tools in a manner similar to experimental tools. In this way, computational models are transferred into practice in an indirect manner. The third type of application is of a direct nature.

In this paper computational bond models will be categorized in the above way. This corresponds to three levels of decreasing degree of precision:

- Resolution of bond-slip.
  This strategy zooms at the micro-behavior in the vicinity of the rebar, where cone-shaped secondary and longitudinal splitting cracks are crucial mechanisms. The method aims at explaining the fundamentals of traction-slip behavior and supports experimental determination of local bond-slip laws.
- Bond-slip interface analysis.
  This approach lumps bond-slip into an interface element, with a view to supporting the derivation of design rules on spacing and width of primary cracks in structural concrete members.
- Embedded reinforcement with tension-stiffening.
  For global analyses even the above approach that zooms at primary cracks becomes too delicate. Instead, the primary cracks are smeared out and techniques of automatic embedment of reinforcing elements and prestressed cables in concrete elements are necessary. In combination with tension-stiffening, these techniques can be directly used in engineering analysis of structural concrete systems.

This paper reviews the three approaches. Most attention will be given to examples of the third category, where results support the 'strut-and-tie' philosophy for structural concrete.

2. DETAILED LEVEL: RESOLUTION OF BOND-SLIP

The first and most sophisticated computational approach to bond-slip is to simulate the behavior in the vicinity around the rebar in detail. The bond-slip, i.e. the tangential relative displacement between the deformed rebar and the concrete (measured some distance away from the rebar), is controlled by four mechanisms [12,17]: (a) elastic deformation, (b) internal conical transverse cracking behind the ribs of the rebar, (c) longitudinal cracking in response to tensile ring-stresses, (d) crushing in compressive cones radiating out from the ribs. Fig. 1 shows a result of an elastic-softening simulation which included elastic deformation, transverse cracking and longitudinal cracking. The structure is a simple reinforced tension bar (tension-pull specimen) modelled in axi-symmetry.

Initially, the external force is transferred from the steel into the concrete primarily via axial tensile stresses. On increasing load, cone-shaped transverse secondary cracks nucleate behind the ribs of a rebar. This cracking starts near the end-face of the specimen and gradually moves inwards, which parallels experimental findings [9]. After transverse secondary cracking, the direct bond action is lost and the transfer of bond forces is subsequently furnished by compressive cones that radiate out from the ribs. The radial components of the compressive cones are balanced by rings of tensile stress [17]. When the ring is stressed to rupture a longitudinal crack arises and the balance against the compressive cones is lost, which causes a further breakdown of bond. These longitudinal splitting cracks are plotted as the shaded area in Fig. 1a.

Aside from giving insight in the basic bond mechanisms, simulations provide quantitative information. Fig. 1c shows a local bond traction-slip curve extracted from the analysis. The curve shows a linear-elastic stage, a stage of decreased stiffness and a softening stage. This trilinear idea of bond curves lends support to analytical bond-models that were derived on the line of argument, and agrees with experimental findings [7]. The slip modulus of the second stage of the curves (appr. 300N/mm^3) falls within experimental scatter [6]. For further results and details the reader is referred to [14,15], where also a numerical contribution is made with regard to the yet unresolved principal issue in bond research, namely the dependence or non-dependence of bond curves on the distance from the primary crack.

In conclusion, a computational resolution of bond-slip contributes to a better understanding of the basic bond mechanisms and works complementary to experimental research in this area. Simulations of this type are not meant to be made at the desk of a practising engineer. Rather, the outcome comes available to practice in an indirect way, via improved bond curves and design rules.

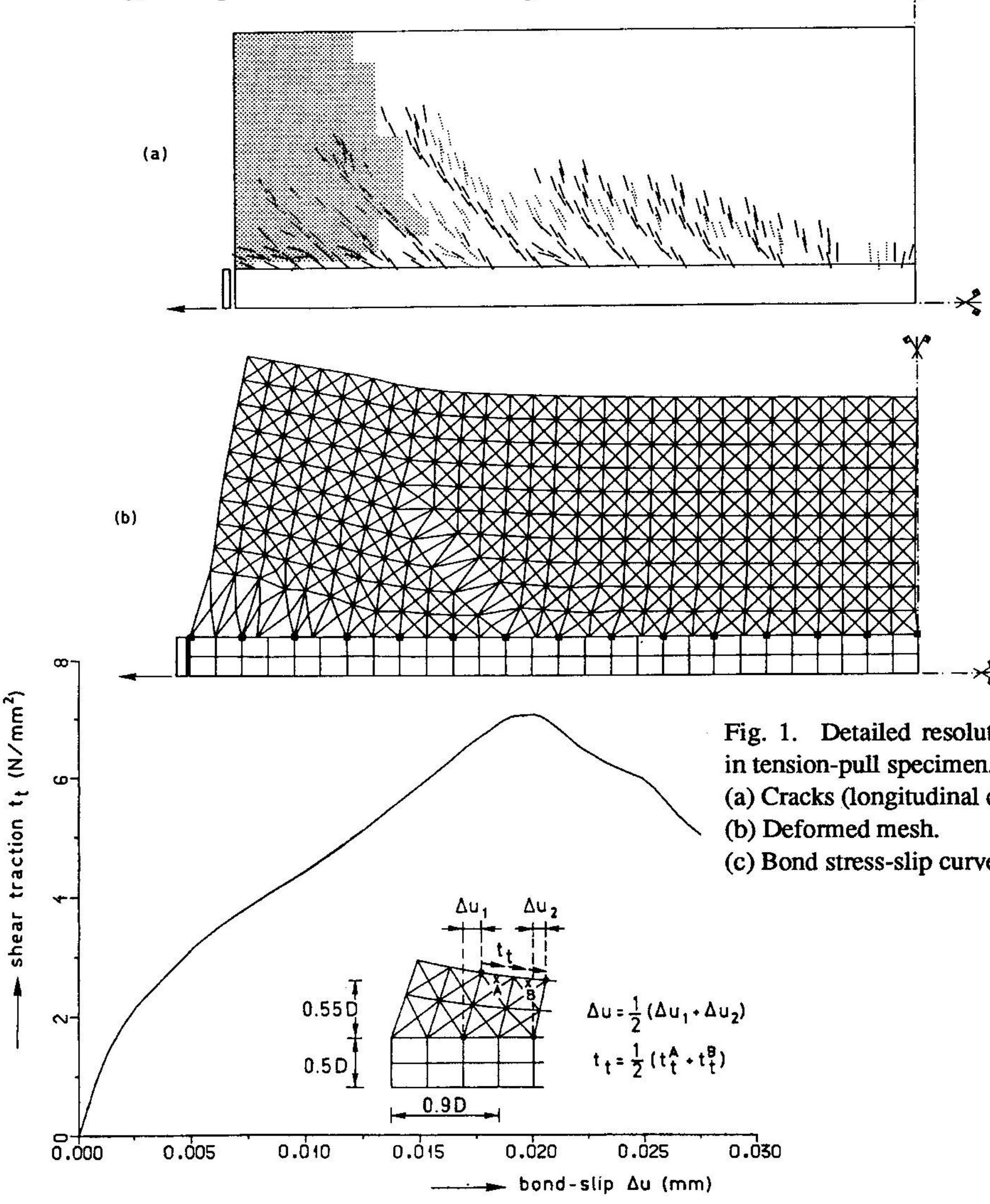


Fig. 1. Detailed resolution of bond-slip in tension-pull specimen.
(a) Cracks (longitudinal cracks shaded).
(b) Deformed mesh.
(c) Bond stress-slip curve computed.

3. INTERMEDIATE LEVEL: BOND–SLIP INTERFACE ANALYSIS

A second approach is to lump the bond-slip behavior into a fictitious interface. Consequently, the traction-slip curve, which was output from the previous analysis, is now input. This technique was introduced by Rehm [12] in order to subsequently evolve into a powerful analytical tool for predicting the spacing and width of primary cracks in structural concrete members, e.g. [11,17,3]. In analytical approaches simplifying assumptions have to be made, like the assumption of a simple and unique bond-slip curve or a sudden stress drop for the concrete after cracking. In computational studies, the procedure can be refined since more advanced bond-slip laws can be inserted for the interface elements and gradual softening curves for the cracks.

Fig. 2 shows an example of a long-embedment tension-pull specimen. The steel is modeled by truss elements, the bond-slip layer by interface elements and the surrounded concrete by continuum elements. The bond traction-slip curve for the interface elements was taken according to [6,10]. The tensile strength for the concrete was assigned a Gaussian distribution, which is consistent with the physical process in heterogeneous materials and essential to prevent a wide band of elements under homogeneous stress from cracking simultaneously. The load-elongation response shows four local maxima corresponding to the successive development of four primary cracks. The serrated type of curve is in agreement with experimental results [8] and justifies engineering models [3]. Beyond formation of the fourth primary crack, the crack pattern was fully developed (stable crack spacing) and the solution could be continued up to yielding of the reinforcement without further physical changes. During that stage only the width of the primary cracks increased.

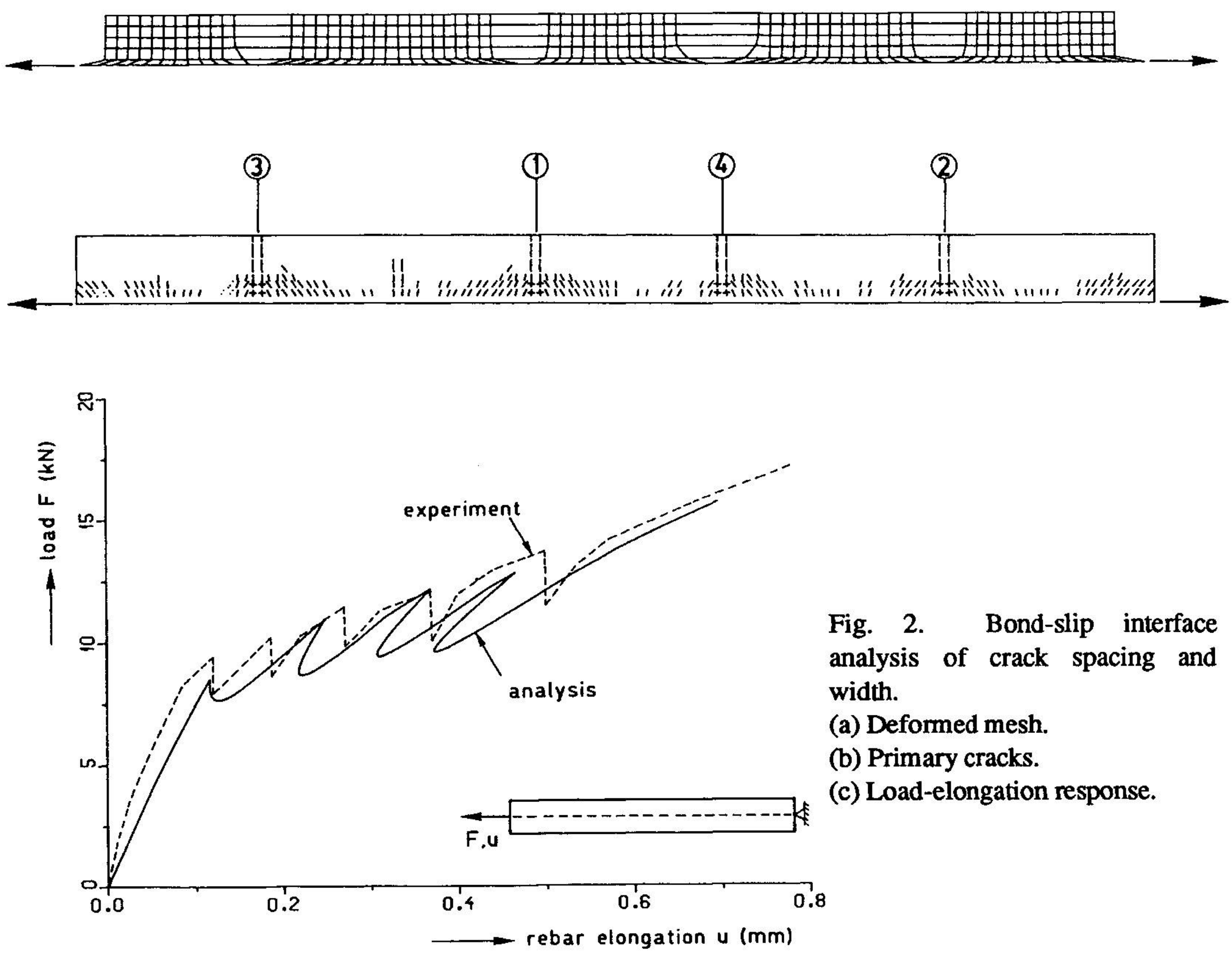


Fig. 2. Bond-slip interface analysis of crack spacing and width.
(a) Deformed mesh.
(b) Primary cracks.
(c) Load-elongation response.

Analyses of this kind apply to cases where information is required regarding crack spacing and/or crack width. This can be the case either in research studies, where adequate design formula for spacing and width are to be developed, or directly in practical analysis of structural details like anchorages, where bond plays a paramount role. An illustrative example of the first category is the combined experimental, computational and analytical study on the control of crack width in deep reinforced concrete beams [2]. Examples of the second category have been given in e.g. [18].

4. GLOBAL LEVEL: EMBEDDED REINFORCEMENT

The third type of approach is full embedment of reinforcements in concrete elements (Fig. 3). With this procedure, the reinforcing elements do not have separate nodes and displacement degrees of freedom, but the strain in the reinforcement is calculated from the nodal displacements of the mother element in which it is embedded. The method generally implies overall perfect bond, i.e. the average strain in the cracked concrete equals the average strain in the reinforcement. For this reason, primary cracks are not treated individually, as though under a magnifying glass, but are smeared out with a view to global analysis of structural concrete systems. The joint action of cracked concrete and reinforcement must then be accounted for via a tension-stiffening model.

The clear advantage is that the lines of the finite element mesh do not need to coincide with the lines of the reinforcement, which is a must when real practical cases with a diffuse set of reinforcing bars, grids and prestressed tendons are analysed.

For practical use of embedded reinforcement techniques in nonlinear finite element analysis, three aspects are essential [4]. Firstly, various embedment combinations must be made available to model the variety of structures and geometries in practice:

- bars in Euler-Bernouilli beam elements
  e.g. 2D and 3D frames
- bars in Mindlin beam elements
  e.g. edge-beams along plates and shells
- bars and grids in plane stress elements
  e.g. panels, deep beams
- bars and grids in axi-symmetric elements
  e.g. storage vessels with radial, tangential and longitudinal reinforcement
- grids in plane-strain elements
  e.g. tunnels
- bars and grids in plate and shell elements
  e.g. slabs, shell roofs, cooling towers
- bars and grids in solid elements
  e.g. massive structures, complicated details

Secondly, options of geometrical preprocessing must be available. It is beyond realism, to let the analyst specify all intersections between reinforcement and element boundaries. Rather, the analyst would like to specify the end-points of the reinforcement, the shape and, depending on the type of shape (straight, parabole, circle etc.) additional information like e.g. a midpoint, see Fig. 3. The preprocessor should then automatically generate all intersections with the lines in the element patch. Once these intersections are known, the reinforcement portions in the elements can be evaluated and the linkage of reinforcement strains to element nodal displacements can be made.

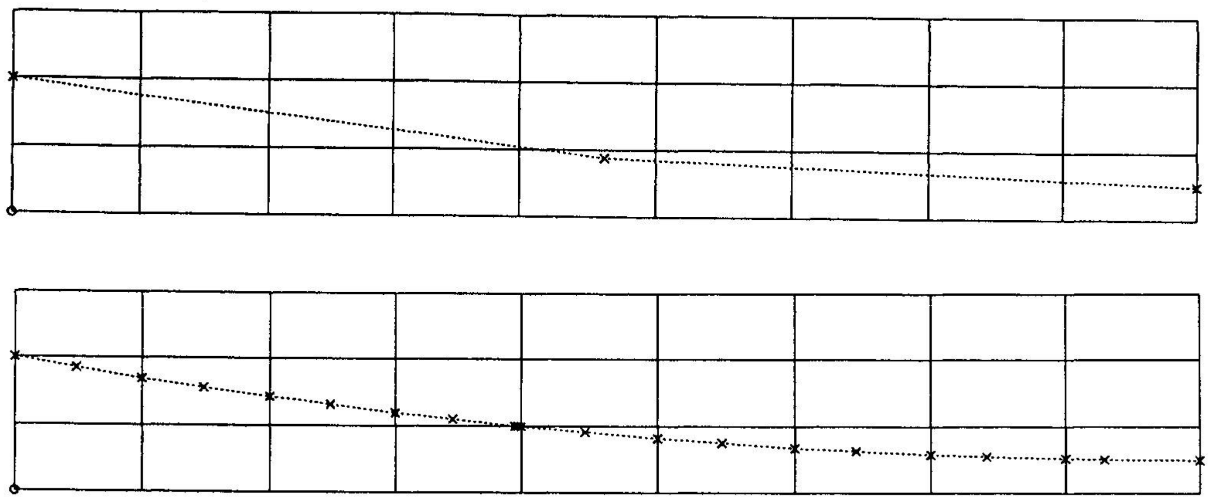


Fig. 3. Embedded reinforcement in finite element analysis.
(a) Simple input of global tendon location, e.g. via three points.
(b) Automatic positioning of reinforcement portions in elements, obtained via geometrical preprocessing routines.

Thirdly, options must be available for prestress, both pre-tensioned and post-tensioned. With pre-tensioning, the reinforcement is assigned an initial stress and the perfect bond assumption is used. With post-tensioning, the reinforcement is given an initial stress distribution (which should guarantee the design value of the jacking force and, simultaneously, account for the losses due to friction, elastic shortening and anchorage slip) and the analysis starts with a no-bond assumption. At this stage, the stiffness of the reinforcement is not included, but only the equivalent nodal element loads due the prestress are placed on the system. After grouting, the no-bond assumption is replaced by perfect bond and the stiffness of the reinforcement is included.

In the latest release of the DIANA finite element program [5] all above listed embedment combinations are available. Furthermore, geometrical preprocessing routines have been included for 2D, $2\frac{1}{2}$D (in-plane and out-of-plane location of curved tendons) and also a limited set for 3D. In the program, the two prestress options are available, whereby it is noted that automatic determination of the initial stress distribution for post-tensioning is currently under development.

5. EXAMPLE 1: CONCRETE WALL IN FLAT

A second example concerns a concrete wall in a twelve-storey appartment building which was already under construction up till the seventh floor when the surveyor noticed that the reinforcement in the 'tie' at the first floor was underdimensioned. To decide on the way of repair, the engineering firm in question called for a review analysis at TNO. Fig. 4 shows some key-results thereof.

The ground level was open and two colums of different dimensions supported the wall. In the mesh, only five storeys were taken into account, while the remaining storeys were represented by a load on top. The figure compares the principal stresses in the wall in the linear-elastic stage and in the final stage at load factor 1.7. It clearly reveals a 'strut-and-tie' system whereby in the linear-elastic stage the tie is formed by the concrete and in the final stage by the reinforcement since the concrete has cracked. Note the change in width and inclination of the strut with increasing load. These are definitely factors that should be accounted for in strut-and-tie design models.

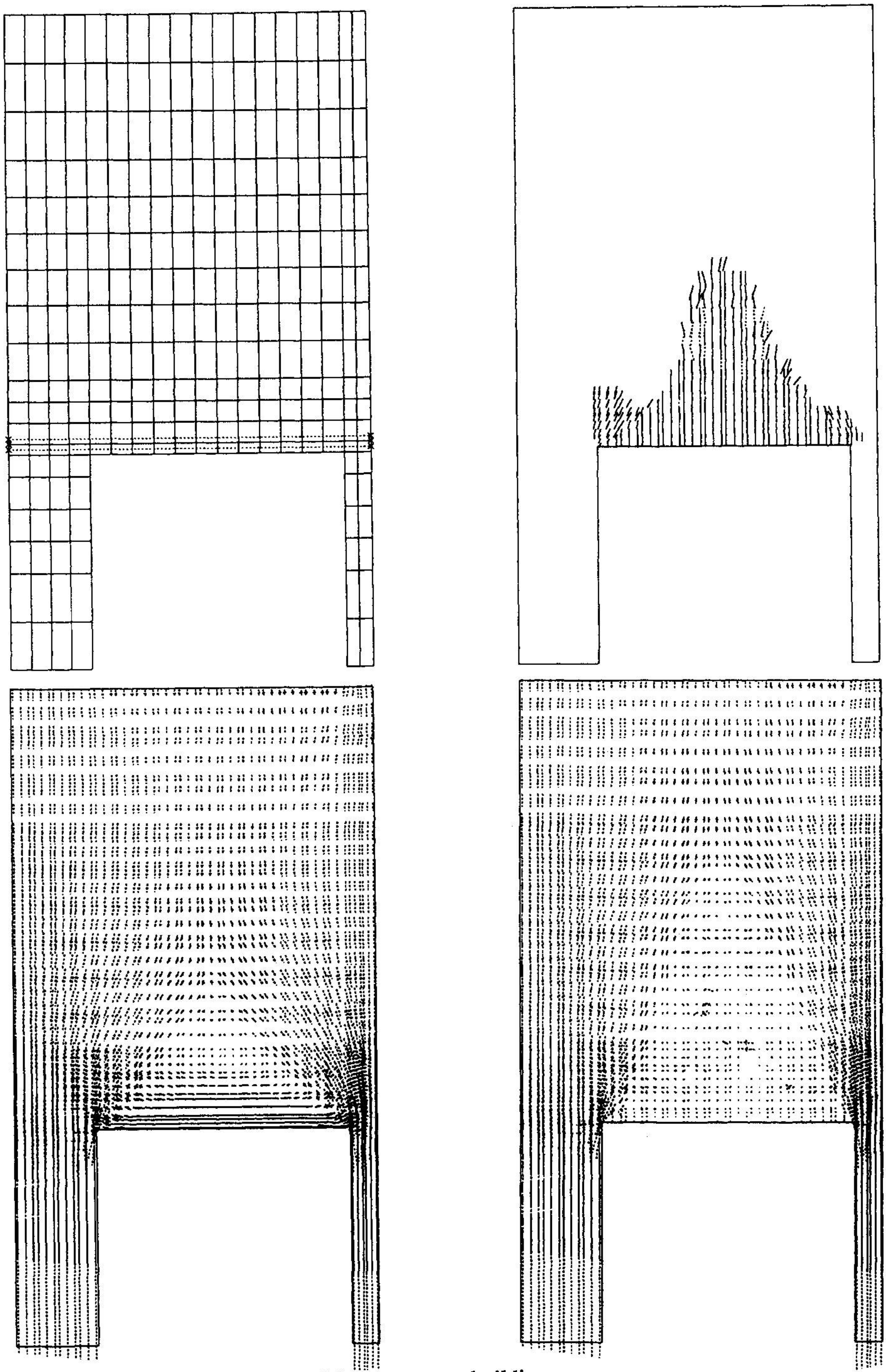


Fig. 4. Review-analysis of concrete wall in appartment building.
(a) Mesh with embedded reinforcement.
(b) Crack pattern in final stage.
(c) Principal stresses in linear stage.
(d) Principal stresses in final stage.

Main conclusions from the analysis were:

- The amount of additional reinforcement in the tie, which was accounted for in the analysis, proved to be sufficient.
- The average crack strain at service load 1.0, divided by an assumed crack spacing, led to crack widths that were well within code limits.
- The maximum compressive stresses in the narrow strut remained below the prescribed strength value in the code.

Based on this review analysis, a low-cost repair could be undertaken, consisting of a limited amount of additional horizontal reinforcement in the top layer at the floor close to the wall. Inclusion of additional vertical reinforcement or local thickening of the wall, options that were originally hinted at by the surveying committee, could be circumvented. Even more important was the fact that due to the quick solution of the problem, construction of the building could be continued without interruption, which saved significant costs.

6. EXAMPLE 2: CROSS SECTION OF TUNNEL STRUCTURE

The second example relates to a damaged part of an existing tunnel structure, which was submitted to consultancy in the Netherlands. This led to a review analysis of the shear capacity and of the bond, the results of which are described in detail in [13]. Herein, only a brief extract of the results is shown, namely the plots of the principal stress trajectories in the linear-elastic stage and the ultimate-load stage (Fig. 5). In the linear-elastic stage the concrete tensile stresses are found to make a substantial contribution to the transfer of the shear force. At ultimate load these tensile stresses have entirely disappeared because of cracking, and a pronounced thrust arch can be observed which is tied by the the midspan tensile reinforcement. The example strengthens the conclusion in the preceding section as to the capability of nonlinear finite element analysis of predicting 'strut-and-tie' systems in structural concrete after significant stress redistribution.

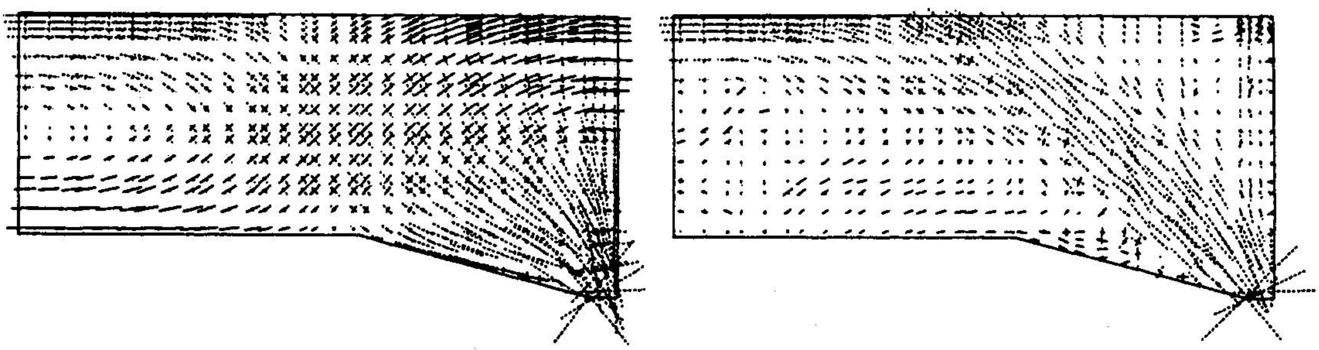


Fig. 5. Transition from linear-elastic behavior to 'strut-and-tie' system in tunnel roof.
(a) Principal stresses in linear-elastic stage.
(b) Principal stresses at ultimate-load stage.

7. EXAMPLE 3: DEEP BEAM

The third example concerns the reinforced deep beam which is also discussed in the key-note paper by Schlaich [16]. In that paper it was proposed to orientate the geometry of a 'strut-and-tie' model at the elastic stress fields, while this orientation can be adjusted upon approaching failure. It is interesting that a somewhat similar procedure was followed in [14] for nonlinear finite element analysis. The beam was first analysed in a global sense using smeared cracks. Subsequently, based on the crack pattern obtained, a simpler model was made with a single predefined discrete crack, incorporated via interface elements. All nonlinearity was lumped into this discrete crack, surrounded by elastic elements for the strut and embedded rebar elements for the tie. Fig. 6 shows results of this 'predictor-corrector' approach, which turned out to work also for shear-critical problems [1]. Assuming sets of predefined discrete cracks in essence comes close to the yield line theory in plasticity where one imagines mechanisms of yield lines. This approach possibly has potential not only for review analysis, but also for dimensioning of so-called D-regions, where D stands for discontinuity, disturbance or detail.

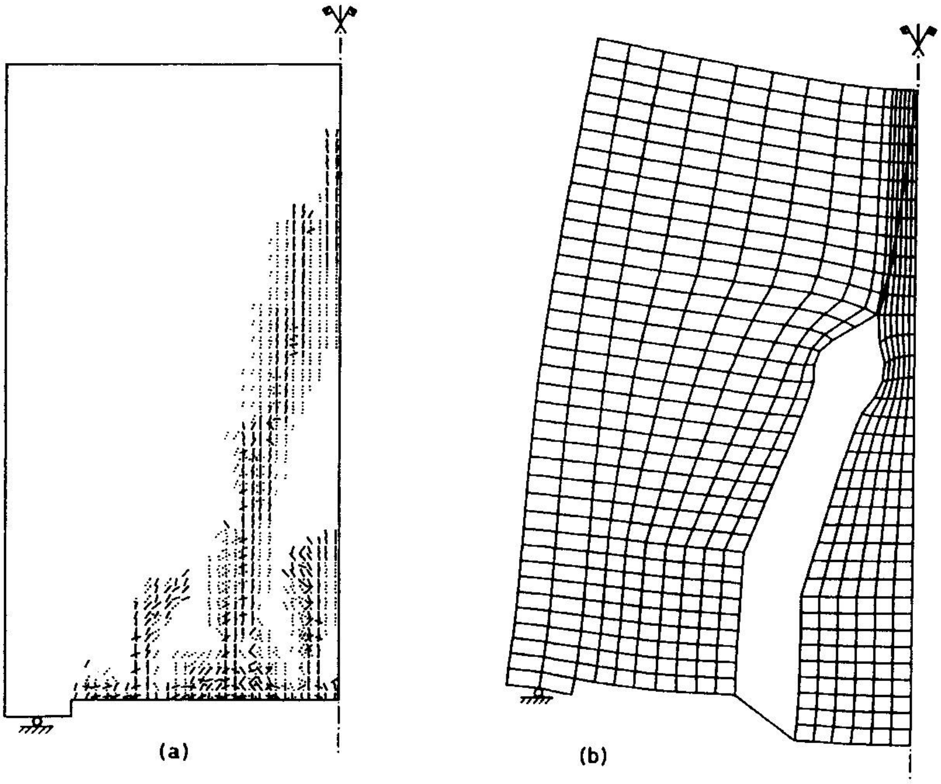


Fig. 6. Deep beam. Two possible strategies in nonlinear finite element analysis.
(a) Smeared crack analysis for prediction of crack path.
(b) Discrete crack analysis with predefined mechanism.

CONCLUDING REMARKS

It is concluded that the strength of very sophisticated bond models lies more in research than in practice. The utility of such models is of an indirect nature. These models for instance lead to a better understanding of basic bond mechanisms and better design rules for crack spacing and width. Global models with embedded reinforcement techniques are directly applicable in engineering practice. Examples have been presented that simply and correctly reveal 'strut-and-tie' systems after stress redistribution.

ACKNOWLEDGEMENTS

The research reported in this paper was based on the DIANA finite element package of TNO Building and Construction Research (formerly TNO-IBBC). Financial support from the Department of Public Works, the Centre for Civil Engineering Research, Codes and Specifications (CUR) and from the Netherlands Academy of Sciences (KNAW) is gratefully acknowledged.

REFERENCES

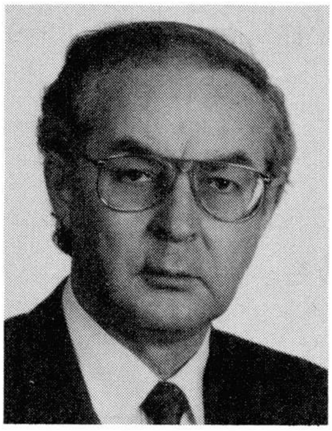
1. BLAAUWENDRAAD J., WANG Q.B., Systematic fracture mechanics study of shear failure in beams under distributed load. IABSE Colloquium Structural Concrete, Stuttgart, 1991.
2. BRAAM C.R., Control of crack width in deep reinforced concrete beams. HERON 35(4), 1990.
3. BRUGGELING A.S.G., Structural concrete: Science into practice. HERON 32(2), 1987.
4. DE BOER A., VISSCHEDIJK M.A.T., ROTS J.G., Embedded reinforcement and prestress. Internal report Bouw9022, Department of Public Works / TNO Building and Construction Research, 1990 (in Dutch).
5. DIANA User's manual – Release 4.1, TNO Building and Construction Research, Delft, 1991.
6. DORR K., Bond behaviour of ribbed reinforcement under transverse pressure. Final Report IASS Symposium Darmstadt, G. Mehlhorn et al. (Eds.), Werner-Verlag, Dusseldorf, Vol. 1, 1978.
7. DRAGOSAVIC M., GROENEVELD H., Concrete mechanics – Local bond, Part I: "Physical behaviour and constitutive consequences", Report BI-87-18, Part II: "Experimental research", Report BI-87-19, TNO Building and Construction Research, Delft, 1988.
8. GIJSBERS F.B.J., HEHEMANN A.A., Some tensile tests on reinforced concrete. Report BI-77-61, TNO Building and Construction Research, Delft, 1977.
9. GOTO Y., Cracks formed in concrete around deformed tension bars, Journal of the American Concrete Institute 68(4), 1971.
10. MEHLHORN G., KEUSER M., Isoparametric contact elements for analysis of reinforced concrete structures. Finite Element Analysis of Reinforced Concrete Structures, C. Meyer, H. Okamura Eds., ASCE, New York, 1985.
11. NOAKOWSKI P., Die Bewehrung von Stahlbetonbauteilen bei Zwangbeanspruchung infolge Temperatur. Deutscher Ausschuss für Stahlbeton 296, 1978.
12. REHM G., On the essentials of bond between concrete and reinforcement. Deutscher Ausschuss für Stahlbeton 138, 1961.
13. ROTS J.G., Tunnel section. HERON 32(3), van Mier J.G.M. (Ed.), 1987.
14. ROTS J.G., Computational modeling of concrete fracture. Dissertation, Delft University of Technology, Delft, 1988.
15. ROTS J.G., Simulation of bond and anchorage: Usefulness of softening fracture mechanics. Applications of Fracture Mechanics to Reinforced Concrete, A. Carpinteri (Ed.), Elsevier, 1991 (in press).
16. SCHLAICH J., Key-note paper on Sub-theme 2.2 "Modelling", IABSE Colloquium Structural Concrete, Stuttgart, 1991.
17. TEPFERS R., Cracking of concrete cover along anchored deformed reinforcing bars. Magazine of Concrete Research 31(106), 1979.
18. VAN MIER J.G.M. (Ed.), Examples of non-linear analysis of reinforced concrete structures with DIANA. HERON 32(3), 1987.

Safety Considerations for Nonlinear Analysis

Concepts de sécurité dans l'approche non-linéaire du calcul statique

Sicherheitsüberlegungen für nichtlineare Bauwerksberechnungen

Josef EIBL
Prof. Dr.
Univ. of Karlsruhe
Karlsruhe, Germany



Director Institut für Massivbau und Baustofftechnologie, Abteilung Massivbau.

SUMMARY
It is shown that the recommended safety concepts as proposed in EC 2 and the CEB Model Code are not rational in the case of safety checks for determining the nonlinear bearing capacities of a structure. A new proposal is given.

RÉSUMÉ
On montre que les concepts de sécurité recommandés dans l'Eurocode 2 et dans le Code Modèle du CEB ne sont pas rationnels pour le contrôle de la sécurité dans le cas des charges critiques non-linéaires. On présente une autre proposition.

ZUSAMMENFASSUNG
Es wird gezeigt, dass die im EC 2 und im CEB-Model Code empfohlenen Sicherheitskonzepte für nichtlineare Traglastermittlungen nicht sinnvoll sind. Deshalb wird ein neuer, abweichender Vorschlag unterbreitet.

1. SAFETY FORMAT AND CURRENT DESIGN

The basis of any rationally founded safety concept must be the probability of failure of a structure. The failure may be defined as a collapse situation, or any other limit state as e.g. the loss of serviceability. In more theoretical terms one asks for the probability of succeeding a limit surface in the space of the different design variable's density functions.

A 'Safety Format' is just one selected method out of an infinite number of simplified methods to guarantee a chosen probability of failure. A special choice is merely justified by reasons of practicability and the unavoidable deviation of the target probability for a whole group of structures. There is no principle advantage of a format using a global safety factor over another with e.g. several splitted partial safety coefficients.

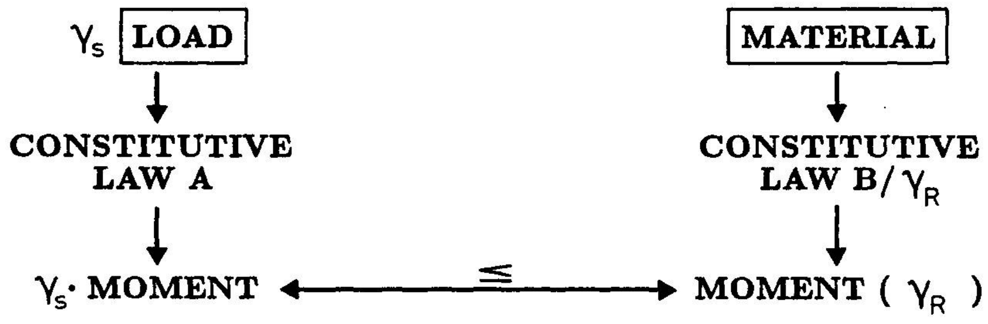


Fig. 1: Linear Analysis / Nonlinear Design

The current design format – called 'Format 1' (Fig.1) – results from the historical development of structural analysis and design. According to this, one first calculates inner forces and moments e.g. by means of the theory of elasticity using a very simple constitutive law A, followed by a so-called cross-section design with a second different set of now nonlinear constitutive laws B for steel and concrete.

Finally on a level of c r o s s - s e c t i o n a l quantities one compares e.g. inner moments increased by a safety factor ≥ 1 to account for the scatter of acting loads with design moments reduced in a n o n l i n e a r manner on the basis of constitutive laws by another safety factor ≥ 1 regarding material defects. Depending on the choice of partial safety coefficients a given minimum probability of failure may be secured.

As due to the different constitutive laws A and B cross-section design is independent of the internal moments and forces, no iteration process including the force distribution is necessary at the design stage.

This is a very simple 'Safety Format' with however strong inconsistencies e.g. that the strains calculated at the first process using material behavior A have nothing to do with the ones which result from the second material law B used for design.

2. BEARING CAPACITY DESIGN

It is however well known that in general local strength values do not control the safety of a structure at least not in case of indeterminate structures (Fig. 2). So e.g. the governing scatter of resistance within the length L of a yield line drops with L

(Fig. 2a), the local value being not decisive. The same is of course known from continuous girders, where yielding of one cross-section does not cause failure of the structure.

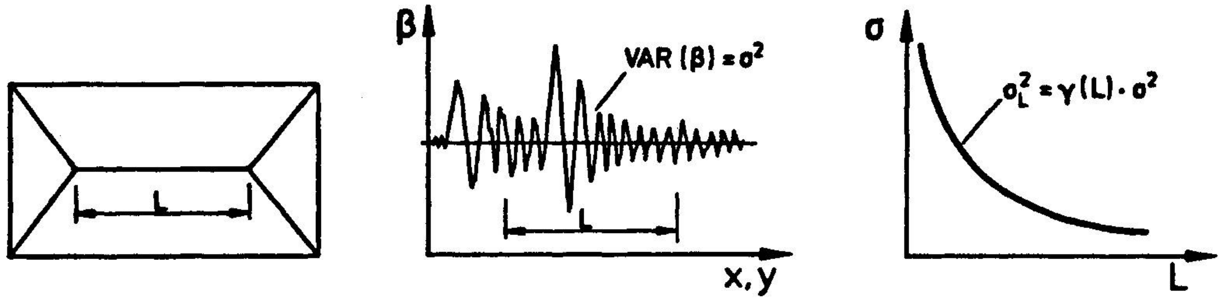


Fig. 2a: Plate as Example

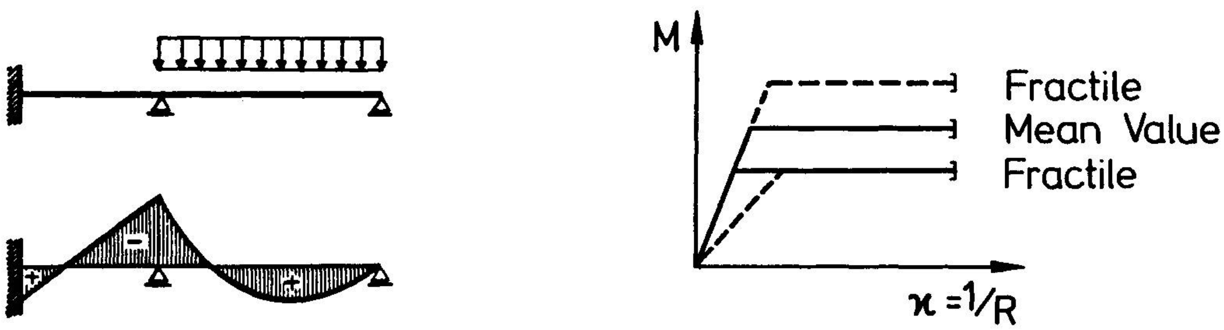


Fig. 2b: Continuous Beam as Example

Now nonlinear analysis allows to calculate the bearing capacity of a structure including local yield without overemphasizing it. This is done on the basis of only o n e physical correct set of constitutive laws A (Fig. 3) and allows a safety cheque comparing bearing l o a d s that can be endured and acting outer loads.

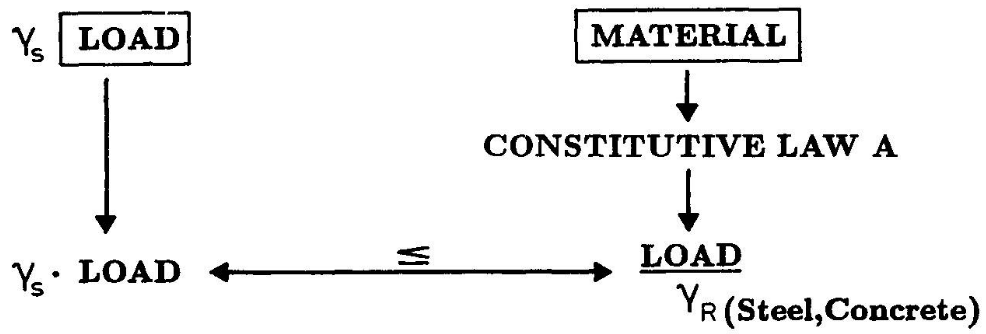


Fig. 3: Nonlinear Analysis and Design

In a 'Safety Format 2' then the bearing capacity is usually calculated by means of material mean values, which will be divided by a partial safety factor to account for materials scatter only at the final level of comparison. Similarly the acting load is increased by another partial load factor.

However instead of dividing the final bearing capacity at the stage of comparison and increasing the acting load both in a linear manner, one may also multiply both safety coefficients together and so end up with a global safety factor.

Proposals made in EC 2 in the CEB-Model Code for a safety concept in nonlinear analysis are not consistent with 'Format 2' and also not very rational.

It is proposed to compute in a nonlinear manner the inner forces of the structure by **m e a n** values of constitutive laws A and to do afterwards a cross-section design using now different constitutive laws B based on **f r a c t i l e** values. The consequence is that the inner forces and moments result from strain states that have nothing to do with the strain state of cross-section design and that the inner forces characterize just one arbitrary state of equilibrium as in 'Format 1'. This approach gives up the the advantage of nonlinear analysis to calculate system bearing capacity and does not justify the much higher amount of work necessary.

The opinion, it would be possible by iteration, to adjust the amount of reinforcement chosen first for the determination of inner forces and the one necessary for cross-section design is wrong of course. It is impossible to be in every cross-section (Fig. 2b) at the horizontal **m e a n** value branch and at the 5 % **f r a c t i l e** branch simultaneously.

When this was acknowledged it was then proposed to do the nonlinear analysis by only fractile laws to be on the 'safe side'. In general this is also not favorable as may be shown e. g. with Fig. 2b where too small support moments may lead to too small shear force at the inner supports. This proposal demands that one always has to decide what the 'safe side' is.

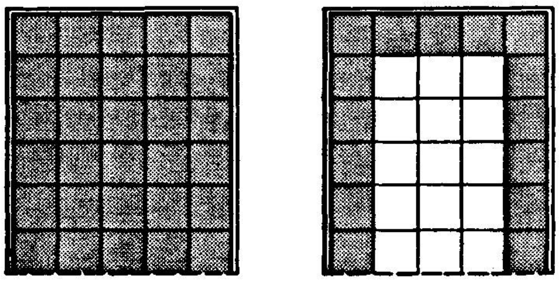


Fig. 4: Plate as Example

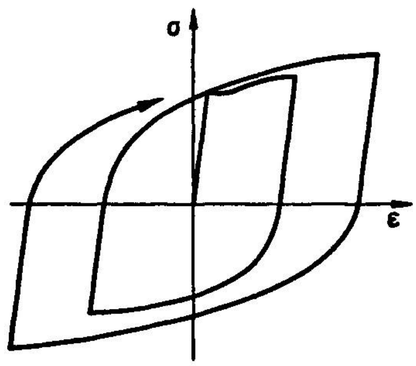


Fig. 5: Cyclic Constitutive Law

To save the concept of two partial safety coefficients, further proposals have been made to manipulate the constitutive laws in such a way as e.g. to take mean values for the first part of a bilinear stress – strain law to define stiffness and combine it with a reduced horiontal fractile branch.

It is obvious that such arbitrary manipulations cannot be a general answer to the problem. How should one manipulate a constitutive law in case of a nonlinear Finite Element computation for a plate (Fig. 4) e.g. to find a 'safe solution' ? If one reduces the whole stress strain law equally in all elements or alternatively only within the boundary elements, one finds quite different moments in the midst of the plate.

How does one have to manipulate the cyclic constitutive law (Fig. 5) to find a safe solution for what? These questions show clearly that in general a new approach, that means a new 'Safety Format' is necessary when calculating nonlinear and more realistic bearing capacities.

3. AN ADEQUATE SAFETY CONCEPT FOR NONLINEAR ANALYSIS

In the following a new adequate safety concept is derived by means of the nowadays possible stochastic Finite Element method – not to use it for practical design – for nonlinear analysis and design.

Beginning with the Equilibrium equations:

$$\Big[[\overline{\mathbf{K}}] + [\delta\mathbf{K}]\Big] \cdot \Big\{\{\overline{\mathbf{u}}\} + \{\delta\mathbf{u}\}\Big\} = \{\overline{\mathbf{R}}\}$$

where

K = stiffness matrix
u = deformation
$\overline{(\ )}$ = mean value
R = bearing capacity

one finds after some manipulations (See [6], [7] e.g.):

$$[\mathbf{C_u}] = [\overline{\mathbf{K}}]^{-1} \cdot \left[\frac{\partial\mathbf{K}}{\partial\alpha}\right] \cdot \{\overline{\mathbf{u}}\} \cdot [\mathbf{C}_\alpha] \cdot \left[\frac{\partial\mathbf{K}}{\partial\alpha}\right]^{\mathrm{T}} \cdot \{\overline{\mathbf{u}}\}^{\mathrm{T}} \cdot [\overline{\mathbf{K}}]^{-\mathrm{T}}$$

with

$[\mathbf{C}_\alpha] = \delta\alpha \cdot \delta\alpha^{\mathrm{T}}$ $\qquad$ [ ] = Matrix
$[\mathbf{C_u}] = \delta\mathbf{u} \cdot \delta\mathbf{u}^{\mathrm{T}}$ $\qquad$ {} = Vektor

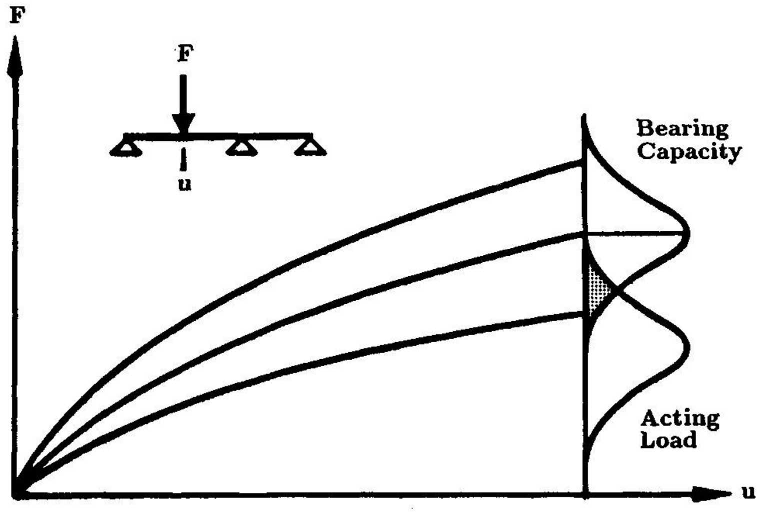


Fig. 6: Nonlinear Analysis including Probabilistic Approach

saying that from the given covariance matrix $\mathbf{C}_\alpha$ of the scattering basic variables α one can easily compute the covariance matrix $\mathbf{C_u}$ of the deformations $\mathbf{u}$ and by means of $\delta\mathbf{u}$

finally the values $\overline{\mathbf{R}}$ and the scatter of $\mathbf{R}$. So even in nonlinear analysis the scatter of the bearing capacity may be given in a $\mathbf{R} - \mathbf{u}$ plot (Fig. 6).

From these argumentations the following conclusions may be drawn :

What is necessary to achieve a desired probability of failure is just one **g l o b a l** safety factor to determine the distance of the acting load density function to the density function of the bearing capacity computed by mean material values. This may be a different factor in cases of steel or concrete failure of course. Even the different safety behavior of structural determinate and indeterminate structures is then inherently covered.

4. REFERENCES

[1] Vanmarcke, E.: Random Fields: Analysis and Synthesis. The MIT Press, Cambridge, Massachusetts, London, England, 1983.

[2] Vanmarcke, E.: Stochastic Finite Element Analysis. Probabilistic Methods in Structural Engineering, London, England, 1983.

[3] Belytschko, T., Liu, W. K.: Random Field Finite Elements. International Journal for Numerical Methods in Engineering, Vol. 23, 1986.

[4] Eibl, J.: Ableitung von Sicherheitsfaktoren. Beitrag zum Sicherheitsseminar des Instituts für Bautechnik, Berlin, 1977.

[5] Eibl, J., Stempniewski, L.: Zur Anwendung finiter Elemente bei der stochastischen Berechnung von Plattentragwerken, Veröffentlichung in Vorbereitung, Institut für Massivbau und Baustofftechnologie, Universität Karlsruhe.

[6] Eibl, J., Stempniewski, L.: Fragen zum Nachweis der Tragsicherheit bei Berücksichtigung nichtlinearer Stoffgesetze. Tagung 'Sicherheit von Bauwerken — Günter Breitschaft Symposium', Eigenverlag des IfBt, Berlin, März 1989.

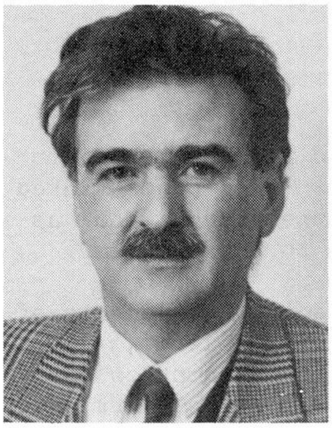
[7] Eibl, J.: Open Problems and Identification of Possible Future Work within CEB. Contribution to the 26th CEB Plenary Session, Dubrovnik, Sep. 1988, Bulletin d'Information / Comité Euro-International du Béton (CEB), Nr. 194, Lausanne 1990.

Plane Elements Analysed Via a Simple Microplane Model

Analyse de structures planes selon un modèle simple par microplans

Analyse von Stahlbetonscheibenelementen mit einem einfachen Mikroebenen-Modell

Gian M. DONIDA
Prof. of Struct. Eng.
Univ. of Parma
Parma, Italy



Gian M. Donida, born 1942, received his degree in Civil Engineering in 1968 at Trieste University. He has been a Professor since 1986. Major interests: Computational Mechanics (Finite Elements) and Concrete Mechanics.

Pietro G. GAMBAROVA
Prof. of Struct. Eng.
Politecnico di Milano
Milan, Italy

Pietro G. Gambarova, born 1941, received his degree in Aeronautical Eng. in 1955 at Milan Univ. of Technology. He has been a Professor since 1980. Major interests: Stress Analysis and Concrete Mechanics (shear, bond, const. laws).

SUMMARY

Panels subjected to monotonically increasing loads and deep beams with and without cutouts are analysed assuming a simple microplane model for concrete. The microplane model is incorporated into a finite element program based on an incremental-iterative procedure, which is well suited to the description of the highly non-linear behaviour of reinforced concrete elements. The reinforcement is smeared; bond-induced stiffening effects are included.

RÉSUMÉ

Des parois en bèton armé ainsi que des murs porteurs soumis à des charges continument croissantes sont examinés en recourant pour le béton à un modèle simplifié par microplans; ce dernier fait partie d'un programme par éléments finis basé sur une procédure itérative pas-à-pas. L'armature est disposée sur chaque élément, de même que l'on tient compte de l'adhérence acier-béton et de ses effets raidisseurs. Le modèle de microplans étudié s'adapte de façon satisfaisante aux résultats expérimentaux obtenus.

ZUSAMMENFASSUNG

Stahlbetonscheiben bei stetig ansteigender Belastung und wandartige Träger mit Öffnungen werden mit Hilfe eines vereinfachten Mikroebenen-Modells für den Beton untersucht. Dieses Modell ist in ein Finite Elemente Programm eingebaut, das auf einem schrittweise wiederholenden Verfahren beruht. Die Bewehrung ist über jedes Element verschmiert und auch der Verbund zwischen Stahl und Beton ist mit seiner Steifigkeit vorhanden.

1. INTRODUCTION

The so-called "local models" for the description of the multiaxial behavior of concrete have become very appealing lately due to their intrinsic simplicity and to their strict connection with the micromechanics of aggregate materials.
Among the local models, the Microplane Model (Bazant and Oh [1], Gambarova and Floris [2]) has enjoyed special attention, since it seems well suited to the modellization of a variety of loading conditions, monotonic as well as cyclic (Bazant and Prat [3]).
Here an initial and relatively simple version for 2-D problems [2] is introduced into a pre-existing F.E. code and is applied to the analysis of several RC plane structures, such as Collins' panels [4], Cervenka and Gerstle's ribbed panels [5] and Kong's deep beams [6,7], in order to assess the ability of the model to describe the structural behavior (load-displacement response, cracking and path-dependency).
In a previous paper (Donida et al. [8]), Maier and Thürlimann's shear walls were successfully analysed with the proposed model.

2. MICROPLANE MODEL FOR CONCRETE SUBJECTED TO PLANE STRESSES

Concrete is considered as a system of randomly oriented planes (the microplanes), in which the elastic and inelastic deformations are concentrated (Figs.1a,b). In the simplified formulation adopted here, three fundamental assumptions are introduced, with a fourth assumption referring to 2-D problems:

a) the local strains, acting on each microplane, are the resolved components of the applied strains (macroscopic strain tensor): $\varepsilon_n = n_i\, n_j\, \varepsilon_{ij}$, with $i,j=1,2$.
b) the shear stiffness of the microplanes is neglected: this assumption has a physical explanation [1,2], but has been introduced mostly for its intrinsic simplicity, and may be dropped in a more general approach [3];
c) only the normal stiffness of the microplanes is introduced, and the coupling between the normal microstress σ_n and the shear strain γ_{nt} is disregarded. Both the elastic and inelastic behavior of the concrete is described by assuming that σ_n is a function of ε_n: $\sigma_n = F(\varepsilon_n)\ \varepsilon_n$;
d) in 2-D problems only the microplanes at right angles to the reference plane are considered (Fig.1b).

In order to work out the coefficients of concrete stiffness matrix it is necessary to formulate the constitutive law for the microplanes: then, by suitably superimposing the contributions from all microplanes, the stiffness characteristics of the concrete can be obtained [2], as well as the increments of the stresses in the general reference system. Under increasing loads, it suffices to specify the stress-strain relations for loading, unloading and reloading in tension and compression (Fig.1c). The model is -by its very nature- a kind of "rotating crack model" (crack planes coincide with the microplanes exhibiting the maximum normal strain in tension) and is path-dependent (the microplanes are activated independently of each other).

3. F.E. PROGRAM IMPLEMENTATION WITH THE MICROPLANE MODEL

A suitable F.E. code based on an incremental-iterative procedure, has been implemented with the microplane model. Quadrangular 4-node elements are used (Fig.3), a fifth inner auxiliary node being provided in order to subdivide each element into four constant-stress triangular elements. The fifth node does not contribute to the degrees of freedom of the structure, because it is removed before the stiffness matrix of the structure is assembled, by means of a condensation process [8]. The introduction of the microplane model, as well as the

evaluation and updating of concrete stiffness matrix, are worked out in a first subroutine dealing with the triangular elements; a system of 12 microplanes suffices for concrete description. The stiffness matrix of the quadrangular elements is evaluated and assembled in a second subroutine.
General properties of the F.E. code are: (a) the stiffness coefficients are formulated by the "direct method"; (b) the solution is based on the Gauss-Doolittle method; (c) the shape functions are of the polynomial type; (d) at the moment only monotonic load histories can be studied; (e) the reinforcement is smeared in two directions at right angles to each other.
Bond-induced tension stiffening effects are introduced by modifying the stress-strain law of the reinforcement: to this purpose, crack orientation, spacing and opening have to be evaluated at each load step and in each triangular element (Fig.2a). Once cracks are formed, their orientation remains fixed. In order to evaluate crack spacing, an "equivalent" steel ratio (Fig.2b) has to be defined [8]: the Young's modulus E_s' of the steel is a function of the average steel strain according to two different bond-stress situations (Fig.2c,d and Fig.4).
Finally, a very simple failure criterion has been introduced for the microplane system: as soon as 2/3 of the microplanes reach a prefixed limit strain in compression and/or in tension (where strain softening automatically diminishes the stiffness), the stiffness of the material is put to zero; subsequently, as soon as the solution process no longer converges, the whole structure fails.

4. FITTING OF TEST DATA

Nine different cases are here examined (Figs.5,6 and 7) and no detailed comments are necessary, since the results are mostly self-explanatory; the principal material properties and the size of test specimens are reported in the figures or in the captions below. For further details see the references, which are easily

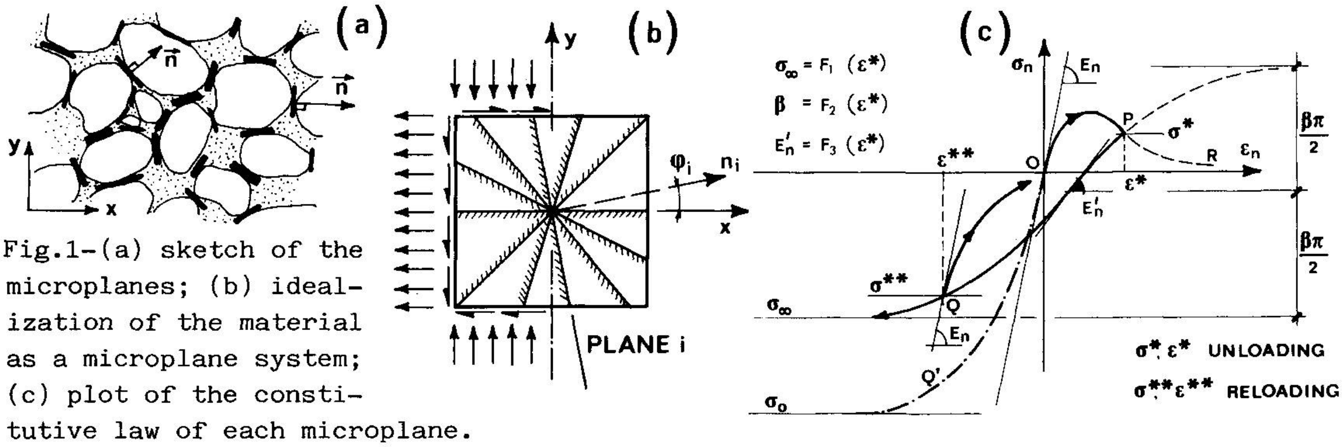


Fig.1-(a) sketch of the microplanes; (b) idealization of the material as a microplane system; (c) plot of the constitutive law of each microplane.

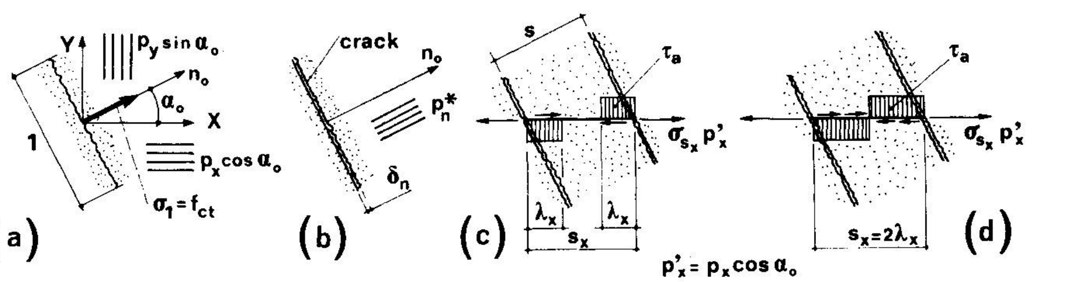


Fig.2 - Formation of a crack (a); equivalent steel ratio (b); bond stresses for mixed bond conditions (chemical adhesion and mechanical interaction) (c); bond stresses after the loss of chemical adhesion (mechanical interaction only) (d).

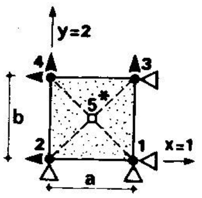


Fig. 3 - Typical 4-node element used in F.E. analysis; (*)means node condensation.

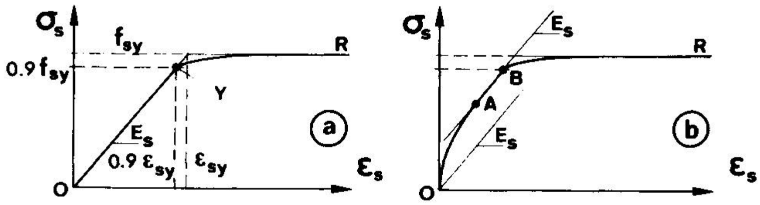


Fig. 4 - Constitutive law of the reinforcement without tension stiffening (no bond) (a) and with tension stiffening (embedded bars) (b).

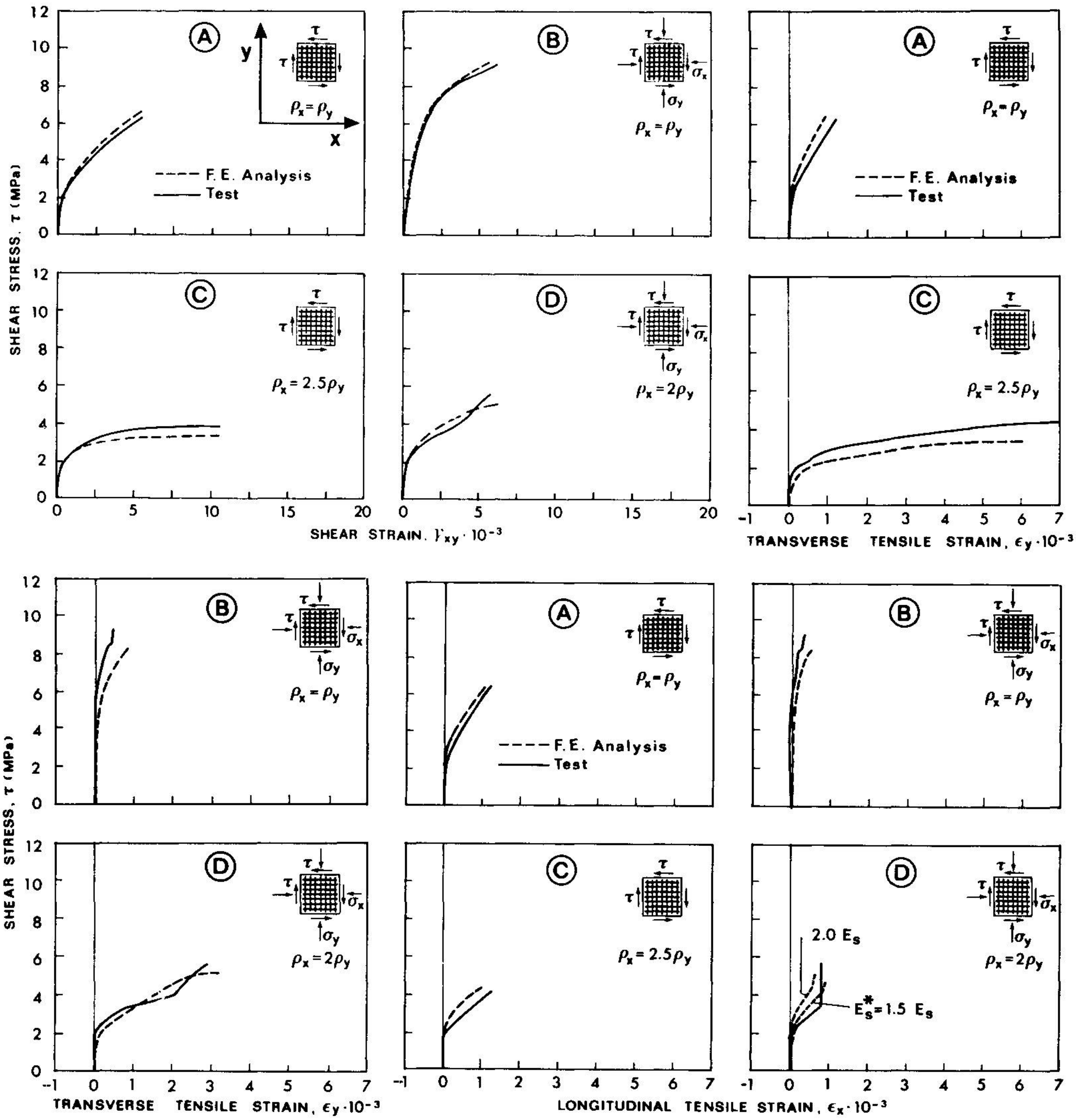


Fig. 5 - Fits of Collins' test results [4] : shear stress $\tau = 0 - \tau_u$; $\rho_x = 0.01785$. Panel A : $f'_c = 20.5$ MPa, $f_{sy} = 442$ MPa, $\sigma_x = \sigma_y = 0$; Panel B : $f'_c = 19.3$ MPa, $f_{sy} = 466$ MPa, $\sigma_x = \sigma_y = -0.7\tau$; Panel C : $f'_c = 19.0$ MPa, $f_{sy} = 458$ MPa for x-bars and 299 MPa for y-bars, $\sigma_x = \sigma_y = 0$; Panel D : $f'_c = 21.7$ MPa, $f_{sy} = 441$ MPa for x-bars and 324 MPa for y-bars, $\sigma_x = \sigma_y = 0$ for $\tau \leq 3.9$ MPa and $\sigma_x = \sigma_y = -(\tau - 3.9)$ MPa for $\tau > 3.9$ MPa. Panel size : 890 x 890 x 70 mm. F.E. discretization: 9 square elements.

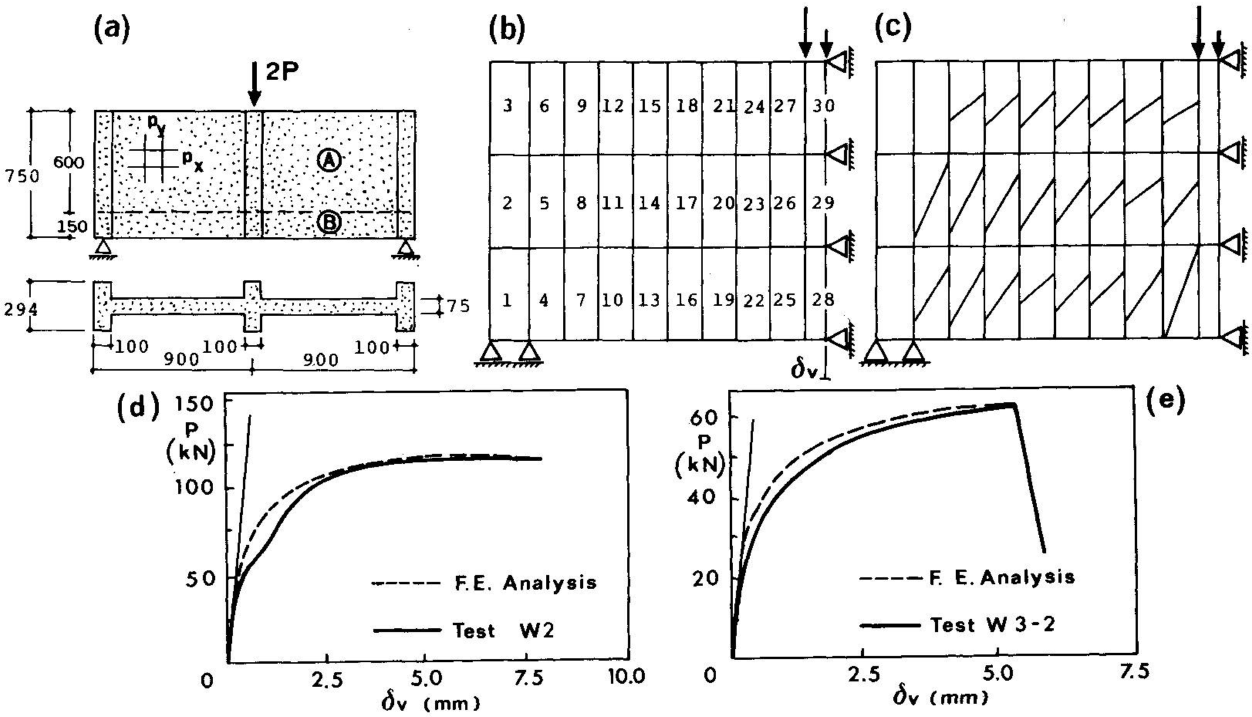


Fig. 6 - Fits of Cervenka and Gerstle's test results [5]: Test W2 (a,c,d) web thickness $t = 75$ mm, rib thickness $t' = 294$ mm, Zone A with $\rho_x = \rho_y = 0.00916$, Zone B with $\rho_x = 0.01832$, $\rho_y = 0.00916$; Test W3-2 (a,e) $t = 50$ mm, $t' = 269$ mm, Zones A and B with $\rho_x = 0.0123$, $\rho_y = 0$; (b) FE mesh, loads and boundary conditions; (c) directions of the principal compressive strain, at collapse; (d,e) load-displacement curves. $f'_c = 27.4$ MPa, $f_{sy} = 362$ MPa.

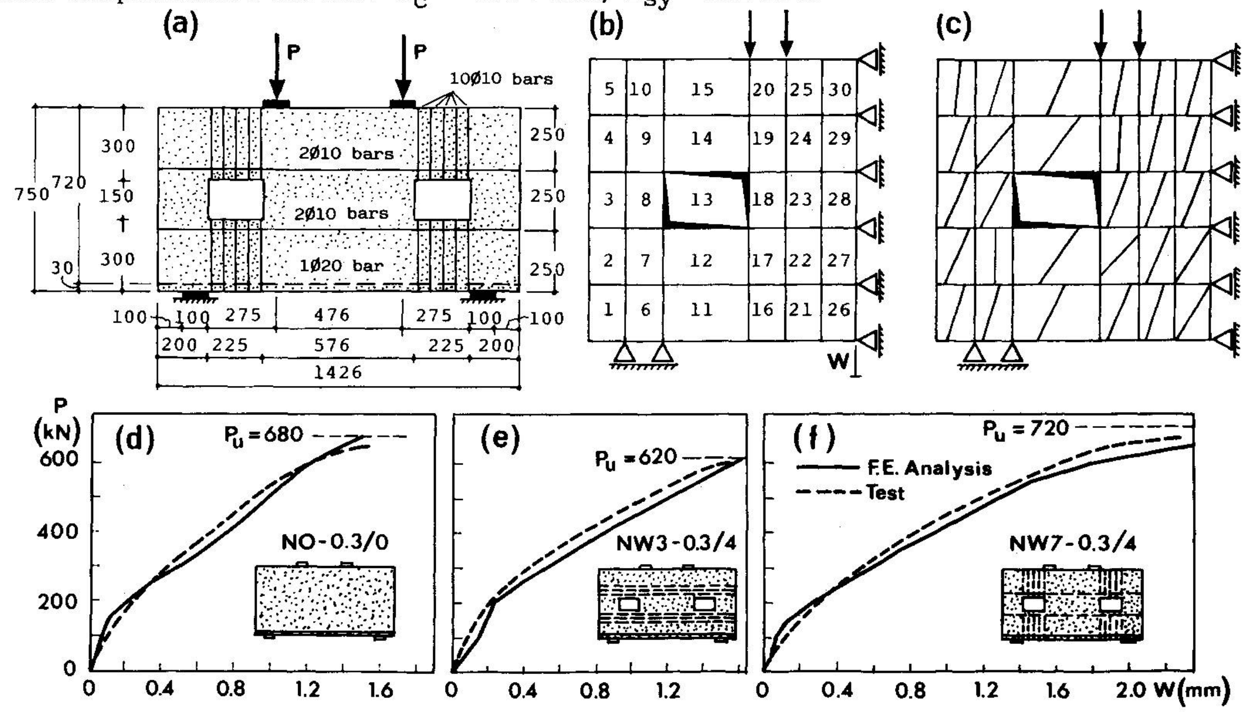


Fig. 7 - Fits of Kong et al. test results [6,7]: Beam NW7-0.3/4 (a) geometry, (b) FE discretization and (c) directions of the principal compressive strain at collapse; (d,e,f) load-displacement curves of Beams NO-0.3/0 ($f'_c = 44.8$ MPa), NW3-0.3/4 ($f'_c = 46.2$ MPa) and NW7-0.3/4 ($f'_c = 42.9$ MPa); $f_{ct} = 3.75$ MPa, $f_{sy} = 430$-450 MPa. Beam thickness = 100 mm.

found in the literature. As a rule, the rebars were smeared in one or two directions over the elements they go through; consequently, the "steel density" is often quite far from the real situation, but this fact does not seem to have a major impact on the results of the analysis. In Panel D tested by Collins ([4] Fig.5) significant unloading occurs after compressive stresses are applied; as a result, the "effective Young's modulus" of the embedded steel during unloading plays a relevant role ($E_s^* = 1.5\text{-}2.0\ E_s$).
On the whole the fits are more than satisfactory, but a more refined analysis with a better topological description of the steel arrangement is in progress.

5. CONCLUDING REMARKS

1. The Microplane Model can be introduced easily into available F.E. codes.
2. The Microplane Model can describe in a relatively simple way a few complex aspects of concrete mechanics, such as multiaxial behavior, path-dependency and cracking.
3. The necessity of storing a few data on the history of each microplane is offset by the limited number of parameters required by the formulation of the microplane constitutive relations.
4. The Microplane Model may be easily improved in order to describe concrete behavior under variable loads (for instance, cyclic and fatigue loads).

ACKNOWLEDGEMENTS

The financial support from the Italian Ministry for University and Research-MURST (Funds 40% granted to the 2nd author, 1988-89) and from the University of Parma (Funds 60% granted to the 1st author, 1989) is gratefully acknowledged.

REFERENCES

1. BAZANT Z.P. and OH B.H., Microplane Model for Progressive Fracture of Concrete and Rock. Journal of Engineering Mechanics, ASCE, Vol.111, No.4, April 1985, pp.559-582.

2. GAMBAROVA P.G. and FLORIS C., Microplane Model for Concrete Subject to Plane Stresses. Nuclear Engineering and Design 97 (1986), pp.31-48.

3. BAZANT Z.P. and PRAT P.C., Microplane Model for Brittle-Plastic Material: I.Theory-II. Verification. Journal of Engineering Mechanics, ASCE, Vol.114, No.10, October 1988, pp.1672-1688 and 1689-1699.

4. COLLINS M.P., VECCHIO F. and MEHLHORN G., An international competition to predict the response of reinforced concrete panels. Festschrift: Prof. Dr. Bruno Thürlimann zum 60. Geburstag, 1983, pp.471-491.

5. CERVENKA V. and GERSTLE K.H., Inelastic Analysis of Reinforced Concrete Panels: Experimental Verification and Application. AIPC - Ass. Int. des Ponts et Charpentes, Mémoires, 32-II, 1972, pp.25-39.

6. KONG F.K., SHARP G.R., APPLETON S.C., BEAUMONT C.J. and KUBIK L.A., Structural idealization for deep beams with web openings: further evidence. Magazine of Concrete Research, Vol.30, No.103, June 1978, pp.89-95.

7. BEAUMONT C.J., An Investigation of the Behavior of Reinforced Concrete Deep Beams. M.A. Thesis, University of Cambridge, Cambridge - U.K., 1975.

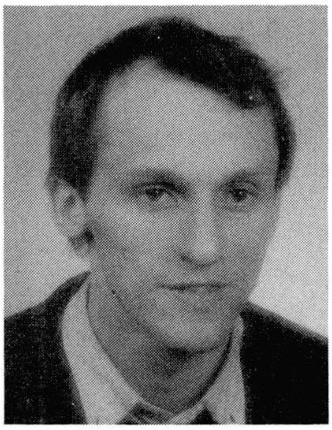
8. DONIDA G.M., FLORIS C. and GAMBAROVA P.G., Non Linear Analysis of Concrete Structures: the Microplane Model (in Italian). "On the Fiftieth Anniversary of the Faculty of Architecture of Naples (1936-1986): Prof. Franco Iossa and His Works", Editor: U.Carputi, Department of Building Research, University of Naples, 1988, pp.371-395.

Evaluation of the Safety of a Cracked Concrete Cooling Tower

Estimation de la sécurité d'une tour de refroidissement dont le béton est fissuré

Untersuchung der Sicherheit eines gerissenen Stahlbetonkühlturmes

G. MESCHKE
Univ. Assist.
Technical Univ. Vienna
Vienna, Austria



Günther Meschke, born 1958, graduated as Dipl. Ing. in 1983 and as Dr. techn. in 1989, both from TU Vienna. Since 1983 he has been involved in research projects in the field of computational mechanics of concrete.

H. A. MANG
Professor
Technical Univ. Vienna
Vienna, Austria

Herbert A. Mang, born 1942, graduated as Dr. techn. (1970) from TU Vienna, Ph. D. (1974) from Texas Techn. Univ.; University Assistant (1967), Professor (1983), both at TU Vienna; Corresp. Member of the Austrian Academy of Sciences.

SUMMARY
The limit of serviceability and the residual safety against structural collapse of a cracked concrete cooling tower, which was built 25 years ago, is evaluated numerically by the finite element method. The process of damage is simulated by means of different thermal load histories. Several assumptions concerning the degree of corrosion are made.

RÉSUMÉ
La méthode des éléments finis a permis d'estimer numériquement la limite de service et la réserve de sécurité vis-à-vis d'une ruine de la structure, dans le cas d'une tour de refroidissement construire il y a 25 ans, dont le béton est fissuré. Le processus de détérioration est simulé à partir de considérations concernant différentes applications de charges de type thermique, ainsi que selon différentes hypothèses sur l'état de corrosion des armatures.

ZUSAMMENFASSUNG
Mit Hilfe der Methode der Finiten Elemente wird die Gebrauchssicherheit sowie die Traglast eines 25 Jahre alten, durch Risse beschädigten Kühlturms bestimmt. Der Schädigungsprozess – Ausbildung vertikaler Risse infolge von Temperaturspannungen – wird mit verschiedenen Annahmen für die Abfolge der Temperaturbeanspruchung sowie für die Entwicklung der Korrosion des Betonstahles nachvollzogen.

1. PRELIMINARY REMARKS

Concrete structures which were designed and built in the 1950's and 1960's frequently show signs of damage such as cracks, caused, e.g., by thermally induced stresses, which usually were not considered adequately by the design provisons of that time [1]. This paper demonstrates the role of nonlinear finite element analyses in the context of the evaluation of the residual safety coefficient of a cracked natural draught concrete cooling tower, which was built 25 years ago. With regards to "safety", loss of serviceability is considered as well as the limit state, characterized by the ultimate load.

The cooling tower Ptolemaïs-III is part of a 125 MW power station located in Ptolemaïs, Greece. The present state of the cooling tower shell is characterized by an approximately uniform distribution of long meridional cracks of crack width up to ~ 1 cm, which are distributed more or less evenly along the cirumference (Fig.1a).

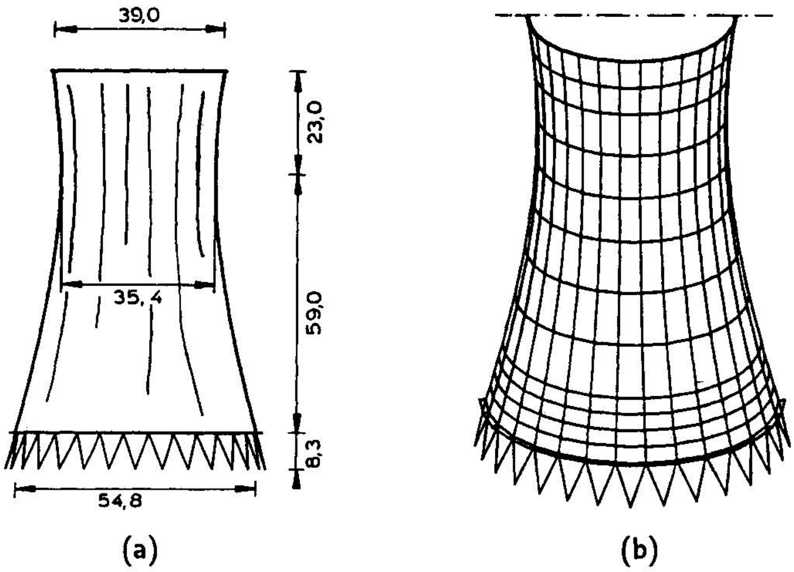


Figure 1: Meridional cracks, geometry, and finite element discretization of the Ptolemaïs cooling tower; (a) meridional cracks and geometry; (b) finite element discretization

The shell is supported by 30 pairs of concrete columns. Except for a small zone of ~ 9.3 m heigth at the lower part of the shell, the structure is reinforced by only one layer of reinforcement located in the middle of the shell. The thickness of the shell is 10 cm with the exception of the aforementioned lower part of the shell, where the thickness is gradually increasing from 10 cm to 50 cm towards the base.

Following a detailed in-situ inspection it was concluded (having rejected improbable causes of the damage) that the temperature gradient between the inside and the outside surface of the cooling tower shell according to winter conditions is the most likely cause for the development of the observed meridional cracks.

2. OUTLINE OF THE NUMERICAL PROCEDURE

2.1 Finite Element Model

The numerical investigation is based on an incremental – iterative procedure with the step size being adapted to the degree of nonlinearity of the structural behaviour. Fig.1b shows the chosen finite element mesh. Because of symmetry of the wind loading, only one half of the shell needs to be considered for the analysis. The shell and the stiffening rings located at the crown and at the bottom of the shell, respectively, are discretized by 225 quadrilateral shell elements [6]. Each element is subdivided into 13 layers such that approximately a plane state of stress may be assumed in each layer. As has been verified by a mesh sensitivity study based on a consistent refinement of the mesh [2], the selected discretization is sufficiently fine. The supporting columns are treated as beam elements (Fig. 1b) with axial as well as bending stiffness.

2.2 Constitutive Model

The numerical representation of cracked concrete is based on the "smeared-crack" approach. Cracks will open normal to the direction of the maximum principal stress when this stress exceeds the tensile strength f_{tu}. After crack initiation, tensile stresses are gradually released according to a linear post-peak stress-strain-relationship defined by a constant softening modulus E_S [2], [4]. The residual interface shear transfer across cracks, resulting from the roughness of the crack face and the dowel action of the reinforcing bars, is considered by means of a crack-strain dependent shear modulus G_c [2]. Tension stiffening is disregarded.

A linearly elastic - ideally plastic constitutive law is assumed for the reinforcement steel. The meridional and the circumferential reinforcement bars are smeared to mechanically equivalent, thin layers of steel which only have axial stiffness in the respective direction (orthotropic material). Corrosion is considered by multiplying the diameter of the reinforcement bars located near both faces in the lower part of the shell by 0.9.

Table 1: Material parameters

CONCRETE		
E_c= 2600 kN/cm^2	Tensile Strength	f_{tu}= 0.18 kN/cm^2
ν_c= 0.20	Compressive Strength	f_{cu}= 2.25 kN/cm^2
E_S= 2000 kN/cm^2	Coeff. of Therm. Exp.	$\alpha = 10^{-5}/^\circ C$
STEEL		
E_{ST}= 206000 kN/cm^2	Yield Stress	σ_y^{ST}= 40.0 kN/cm^2

2.3 Considered Load Cases

The investigation of the influence of the thermal preloading on the limit of serviceability and on the ultimate load of the structure comprises several thermal load histories. Two load cases refer to winter conditions. They are characterized by the incremental increase of the temperature difference ΔT_W - the subscript "W" stands for "winter" - between the inside and the outside surface up to 45°C (LOAD CASE II) and, for one of these two load cases (LOAD CASE III), by subsequent thermal unloading. Thereafter, the wind load w according to [3] is applied incrementally. These load histories may be written symbolically as

$$g + \Delta T_W/h + \lambda w \quad (LOAD \quad CASE \quad II), \tag{1}$$

$$g + \Delta T_W/h - \Delta T_W/h + \lambda w \quad (LOAD \quad CASE \quad III), \tag{2}$$

where g represents the dead load, h is the thickness of the shell and λ is a dimensionless parameter ($\lambda \geq 0$). For the purpose of including a possible weakening of the structure due to (micro)cracks on the inner surface, one additional load case (LOAD CASE IV) was considered. It allows simulation of a winter-summer cycle ($\Delta T_W = 45^oC, \Delta T_S = -20^oC$) with subsequent thermal unloading prior to the incremental application of the wind load:

$$g + \Delta T_W/h - \Delta T_W/h + \Delta T_S/h - \Delta T_S/h + \lambda w. \tag{3}$$

In order to assess the stiffness reduction due to the observed cracks, an ultimate load analysis of the originally uncracked shell was performed (LOAD CASE I):

$$g + \lambda w. \tag{4}$$

3. INFLUENCE OF THE THERMAL LOAD HISTORY

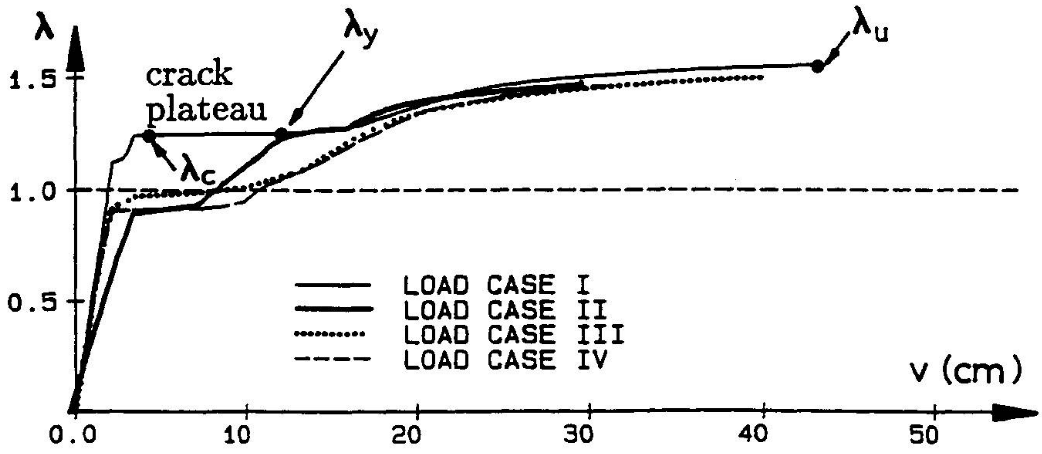


Figure 2: Load-displacement curves for load cases I, II, III and IV

In Fig.2, the horizontal displacement at the windward meridian, at the crown of the shell, parallel to the axis of symmetry (see Fig. 1b) is plotted as a function of the dimensionless factor λ for the load cases I, II, III and IV, respectively. Obviously, the damage induced by thermal preloading (LOAD CASES II, III and IV) has a significant influence on the level of the "crack plateau" (λ_c). With respect to the ultimate load level (λ_u), however, the influence of these temperature cracks is insignificant. The beginning of yielding of the reinforcement (λ_y) may be considered as a *sufficiently conservative limit of the serviceability* of the structure. Compared to the uncracked structure, the values for λ_y obtained from analyses of the damaged shell (λ_y = 1.24 for LOAD CASES II and III and λ_y= 1.20 for LOAD CASE IV) are only sligthly lower than λ_y= 1.29, corresponding to LOAD CASE I. Table 2 summarizes the values for λ_c, λ_y and λ_u resulting from the numerical investigation.

Table 2: λ_c, λ_y and λ_u for load cases I, II, III and IV

Load Case	λ_c	λ_y	λ_u
I	1.26	1.29	1.56
II	0.92	1.24	1.495
III	0.98	1.24	1.515
IV	0.915	1.20	1.47

It is noteworthy that the mode of considering the history of the temperature loading does not have a great influence on the structural resistance of the shell when being subjected to wind loading.

4. INFLUENCE OF CORROSION

In the investigation described so far, corrosion of the reinforcement was considered by multiplying the diameter of those reinforcement bars by 0.9, which are located near both surfaces at the lower part of the shell. This assumption was regarded as not sufficiently conservative. Therefore, three different

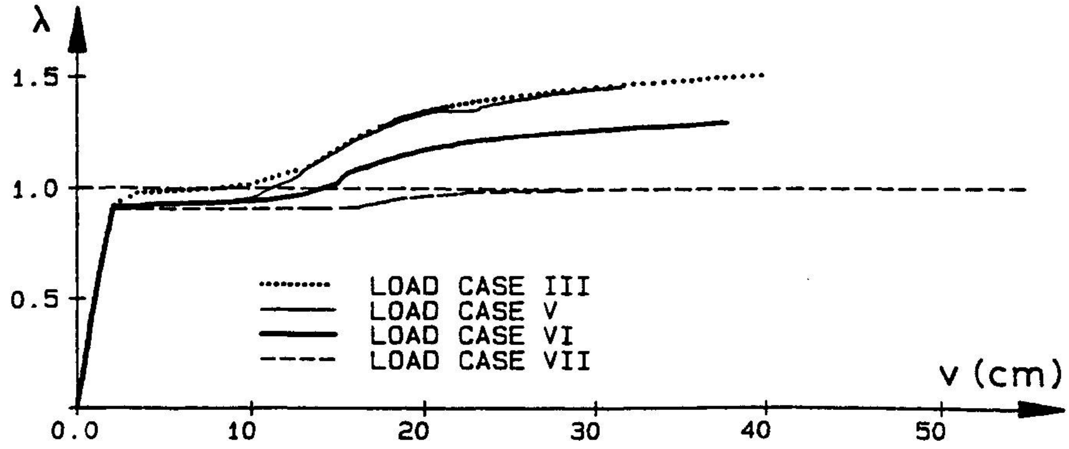


Figure 3: Load-displacement curves for load cases III, V, VI and VII

assumptions were made concerning the amount of corrosion that may occur in the remaining lifetime of the cooling tower. (The temperature loading process was chosen according to LOAD CASE III). In LOAD CASE V, only corrosion of the reinforcement bars located near both faces of the lower part of the shell is considered. According to [5], a reduction of the diameter of the reinforcement bars by 0.1 mm per year is a realistic assumption. Hence, after 25 years, the total reduction of the diameter will be 2.5 mm. In LOAD CASE VI, also the corrosion of the reinforcement bars located in the middle of the shell is taken into account. The diameter of these bars is multiplied by the factor 0.9. LOAD CASE VII is based on the worst assumption. Corrosion is considered by a 2.5 mm reduction of the diameter of *all* reinforcement bars.

Fig.3 contains load-displacement diagrams obtained from LOAD CASES III to VII. The respective values for λ_c, λ_y and λ_u are summarized in Table 3.

Table 3: λ_c, λ_y and λ_u for load cases III, V, VI and VII

Load Case	λ_c	λ_y	λ_u
III	0.98	1.24	1.515
V	0.925	1.21	1.465
VI	0.93	1.07	1.30
VII	0.905	0.915	0.995

For LOAD CASE VI, λ_y and λ_u are obtained as 1.07 and 1.30, respectively, as compared to 1.24 and 1.515 for LOAD CASE III. For LOAD CASE VI the ratio $\lambda_u/\ \lambda_y$, is equal to 1.22. In view of the character of this ratio as a "residual safety factor", the reduction of the diameters of the reinforcement through multiplication of the diameter by 0.9, as considered in LOAD CASE VI, can be regarded as a tolerable *upper limit* for the corrosion. For a significantly worse state of corrosion, as represented by LOAD CASE VII, almost no safety margin between the cracking plateau and the onset of yielding exists. This load case represents a critical state of corrosion, where neither safety against loss of serviceability nor against structural failure is guaranteed! Such a critical state of corrosion of the reinforcement located in the middle of the shell, however, is unlikely to occur.

5. INFLUENCE OF THE TENSILE STRENGTH

The sensitivity of the structural response with respect to the assumed value for the direct tensile strength f_{tu} is demonstrated by reinvestigating LOAD CASE VI, on the basis of the experimentally obtained value $f_{tu} = 0.26\,kN/cm^2$ (LOAD CASE VIII) instead of the original value $f_{tu} = 0.18\,kN/cm^2$. Table 4 contains the values for λ_c, λ_y and λ_u for the two load uses. For an increase of the tensile strength by 44.4 %, λ_c increases by 28 %, λ_y by 13 % and λ_u by 2.7 % !

Table 4: λ_c, λ_y and λ_u for Load Cases VI and IVII

Load Case	λ_c	λ_y	λ_u
VI	0.93	1.07	1.30
VIII	1.205	1.21	1.335

6. CONCLUSIONS

It is concluded that the investigated cooling tower shell will be sufficiently safe against structural failure even if no provisions for a repair of the concrete shell are taken, provided the reduction of the diameter of the reinforcement bars will be less than 10 % within the remaining life-time of the structure. The degree of corrosion of the reinforcement and, consequently, the efficiency of the precautions taken to protect the reinforcement from further corrosion are the relevant criteria for the safety of the cooling tower against loss of serviceability and structural collapse.

REFERENCES

[1] BERRA, M., BETTINALI, F. and FATTICCIONI, A., Survey and analysis of an old Italian cooling tower. Proc. Third Int. Symp. on Natural Draught Cooling Towers, Paris, France, Alsthom Etablissement SCAM, Paris, 1989.

[2] MESCHKE, G., MANG, H.A., KOSZA, P., Finite Element Analyses of a Cracked Cooling Tower Shell. ASCE, Structural Mechanics Division, submitted for publication.

[3] VGB-FACHAUSSCHUSS, Bautechnik bei Kühltürmen, Teil 2. Bautechnische Richtlinien, 1980.

[4] MILFORD R.V.; SCHNOBRICH W.C., The Effect of Cracking on the Ultimate Load of Reinforced Concrete Cooling Towers. Proceedings of the 2nd Intern. Symposium on Natural Draught Cooling Towers, Bochum, Germany, Sept. 1984.

[5] MOLL, H.L., Über die Korrosion von Stahl im Beton. Deutscher Ausschuß für Stahlbeton, Heft 169, Berlin 1964.

[6] MARC Analysis Research Corporation: "MARC General Purpose Finite Element Program, Vol.B, Element Library", MARC Analysis Research Corporation, Palo Alto, USA, Rev. K.3, 1988

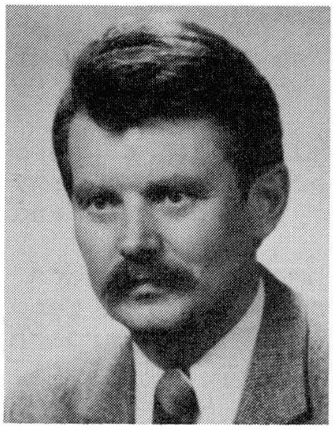
Remarks on Failure of a Reinforced Concrete Cooling Tower

Considérations sur la ruine d'une tour de refroidissement en béton armé

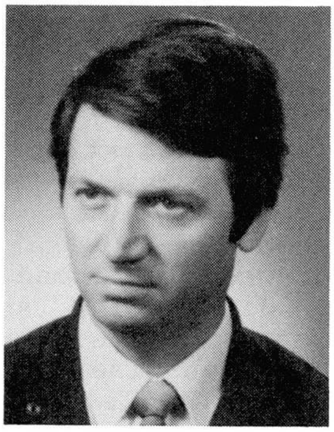
Bemerkungen über den Einsturz eines Stahlbetonkühlturms

Marian PERSONA
Assist. Prof.
Wroclaw Techn. Univ.
Wroclaw, Poland

Mariusz SZECHINSKI
Assist. Prof.
Wroclaw Techn. Univ.
Wroclaw, Poland



Marian Persona, born 1936, graduated in Civil Engineering and took his Dr. techn. degree at the Wroclaw Techn. University. He worked as an Assistant and later as an Assistant Professor at the Institute of Building Science. He published up to 63 papers and was involved in more than 120 consulting works for industry.



Mariusz Szechinski, born 1947, graduated in Civil Engineering and took his Dr. techn. degree at Wroclaw Technical University. Since 1970 he worked (later as an Assistant Professor) at the Institute of Building Science. He published 34 papers on the theory and practice of Concrete Structures and was involved in more than 200 consulting works for industry.

SUMMARY

In February 1987, a reinforced concrete cooling tower, 100 m high, collapsed at the power station Turow in the Lower Silesian region. As the process of the collapse, its effects and the results of later research on its cause revealed a series of interesting findings and conclusions, we decided to describe them in our paper.

RÉSUMÉ

C'est en février 1987 que le tour de refroidissement en béton armé de la centrale thermique de Turow (Basse Silésie, Pologne), haute de 100 m, s'effondra brusquement. Le processus de l'écroulement, ses effets et les résultats de la recherche qui s'ensuivit afin d'en déterminer les raisons, ont apporté une série de remarques intéressantes ainsi que certaines conclusions. Ces considérations font l'objet du présent article.

ZUSAMMENFASSUNG

Im Februar 1987 ist ein 100 m hoher Stahlbetonkühlturm des Kraftwerks Turow eingebrochen. Der Verlauf des Unfalles, seine Auswirkungen und die Ergebnisse späterer Forschungen über die Ursachen waren so interessant, dass wir uns entschlossen haben, sie in dieser Arbeit zu beschreiben.

1. Introduction.

Increasing in the last years computing abilities and intensive research in modeling and structure analysis, brought the formulation of the new idea of concrete structure definition a term called structural concrete. The term called total structural designing is tightly with this term connected. What is understood by these terms, we may find in the works of Bruggeling [1], Breen [2] and others.
First it is necessary to introduce a definition of the structural concrete. Bruggeling [1] gives the following definition of this term; "Structural concrete" refers to any structure built from concrete, and non-prestressed and/or prestressed reinforcement which can resist, in controlled way, all the actions exercised on these structures by loads, imposed deformations and other influences. Moreover, these structures must be constructed in the safe and economical way. In that definition, the article in a controlled way says about our possibility to control, for instance: some deformations, cracks, or durability during the designing process.
Breen [2] says that a structural concrete that is the term describing a wide range of concretes used for building any structure, involving plane concretes, normally reinforced and prestressed concretes.
The basic idea of what is termed the "Structural Concrete Approach" is to eliminate distracting and artificial barriers which tend to compartmentalize the designer's thinking. In the new approach it's necessary to emphasise more global attention to total load paths and resisting elements.That means that at the base of that definitions there lies a wish at overcoming the existing division of concrete structures on normally reinforced and prestressed. In other words it is a wish to enable an analysis of freely chosen structure independently of the way of its reinforcing and a type of acting loads.
Introduction of these definitions and satisfactionary description of computing methods opens the way to the total structural design. Under this term it's necessary to take into account, building of special kind of structure model, (it may particularly refer to the concrete structure), connected with environment through all existing influences. Model like that may be totally analyzed,and the picture we receive of its internal work, gives the possibility of its effective and active forming and constructing. This process maybe repeated up to the moment of receiving optimal characteristic of designed structure. Especially important is the fact, that using this method, a designer is reducing (minimizing) a material usage and is controlling the work of the structure all the time. Using this method, it is possible to calculate and design any type of structure independently of the type of loads.So the principle of total design lies down now in looking on the structure as a whole, not as at the system of separated elements.
Scordelis [3] describes a process of total designing that may be fulfilled and presented as a collection of the following stages:
- first stage it is a conceptual stage,
- second is a predesignig,
- third: structure analysis,
- fourth: structure synthesis,
- fifth: a drafting stage.

Especially important is the third stage that means observations of

structure behaviour under the wide range of influences. Designing process on this stage may be realized in two ways:
First relays on calculating the internal forces and further on using the received results in detailing of structure. This way is a kind of imitation of traditional methods, fulfilled at the general look at the designing structure. Second way relays on examining the internal forces flow in structures involved in the wide range of influences connected with active and effective forming of this structure during the process.
There opens a wide range of possibilities and it is enough to show here only one of them that is possibility of modeling the process of structure collapse.
As an example the case of cooling tower collapse was analyzed.

2. THE COLLAPSE AND ITS EFFECTS.

On the 7th of February 1987 at the power station TUROW in Lower Silesia Region in Poland, there collapsed one of nineexisting there cooling towers. On the hight around 43 m over the ground level, that means between 38 and 39 m of tower shell, almost horizontally, the shell was cut and fell down into the tower.
The construction of tower and its remains after the collapse are shown in fig. 1.

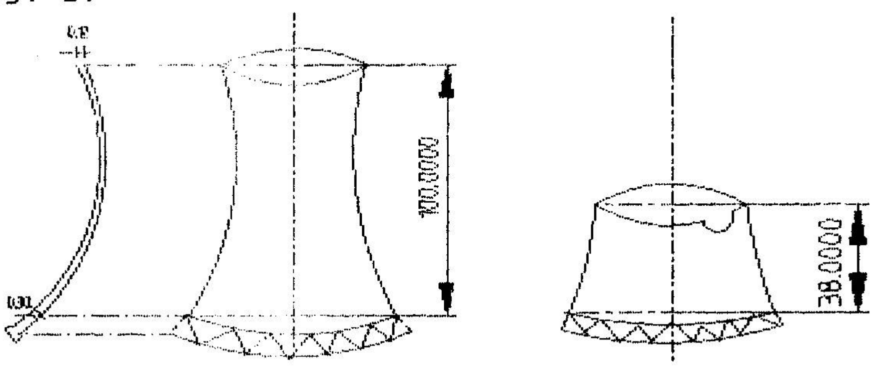


Fig.1
Cooling tower before and after collapse.

The discussed cooling tower was built in 1963. The Main part of this structure make the reinforced concrete double curved shell, dimensioned as follows: total high 100 m (cover shell 95 m), cover shell dimension 74.6 m, shell thickness in the lower ring 0.30 m and it is decreasing to 0.12 on the level of 30 m, further thickness is constant. The shell is reinforced by nets of bars on the distance 0.20 x0.20 m.
For over 25 years of exploitation, the discussed structure was very often controlled and examined and never any damages were found.That is why the researches on the collapse were very difficult.
There were many tests made of which most important were:
- tests of strengths characteristic for used materials,
- tests of physical and chemical properties of used materials,
- geodesic tests of the real geometry of the remaining part of the shell and comparison of the results with the ideal shell geometry,
- analysis of changes in the shell state during exploitation caused by the failures during the shell building,

- analysis of the influence of super loads on the shell work,
- analysis of correctness of designing solutions applicable to the tested shell.

Generally it was established that:
- concrete was generally technically good, that means its strength and quality coefficients were containing in the range demanded by codes of practice,
- position and quality of reinforcement was good,
- most sensitive points of the shell were the places of work breaks made during the shell concreting. It was established that reinforcement bond was decreased in these places. Also the corrosion of concrete developed there and it was increased by diffusion of the water through concrete. The weakest point developed in the region of 38th stripe of concreting. Moreover the stripe of the weak concrete developed on the great length. The increasing lixiviation of cement from concrete caused probably some rising of horizontal slit in the shell. This led further to a change of the static scheme, local overload of the structure and finally to a collapse.

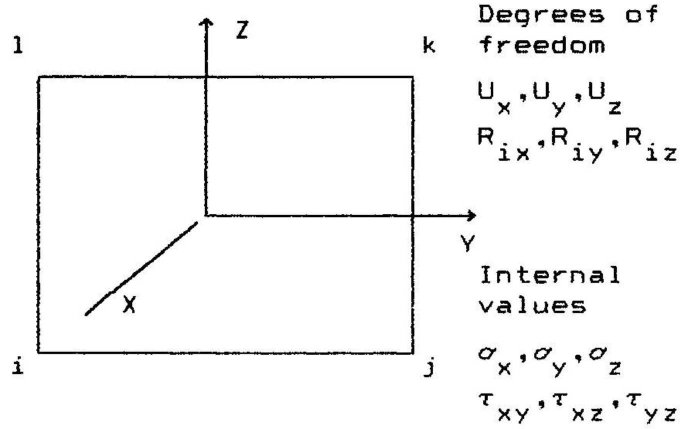
The following hypothesis about the mechanism and causes of collapse were formulated:
- the post building imperfections of the wall-shell,
- the technological and technical disadvantages referring to the concreting process,
- a very bad quality of technological breaks during the concreting,
- the destructive influence of the environment eq.
  - strong wind blasts,
  - local erthquake caused by mine exploitation existing around,
  - thermal influences etc.

After the detailed analysis of the causes listed above, the main reason of the collapse was found. It was the destruction of concrete on the 30 m long stripe of shell circumference, caused by long term lixiviation of cement from concrete. There appeared a long slit in the wall, which changed the static model of the shell structure.

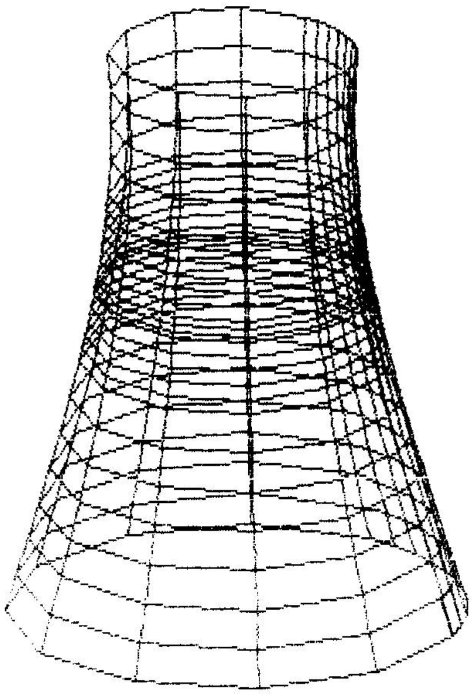
3. MODELING OF THE SHELL WORK.

To check the hypotheses about the cooling tower collapse, mentioned in the previous section, the analytical model of structure was built. There was A series of Computer Programs for Static Finite Element Analysis of Structures called STRAINS used [4]. System STRAINS works on the base of the following assumptions:
- there exists the continuity of the matter,
- stresses in the structures develop in the moment applying some forces,
- the matter is linear,
- the state of stresses in examined point is marked by a state of deformations in this point,
- displacements and deformations are small,
- it is possible to use Saint-Venan's principle.

To build the model of the discussed structure thre were used plane surface elements as shown in. Fig. 2a . The received node mesh is shown in fig. 2b.
On the shell surface, the horizontal slit was modelled. The length of the slit was changeable. The model was loaded by wind forces and several times examined while changing the slit length.
The results of the calculations are shown in figure 3a,b,c.

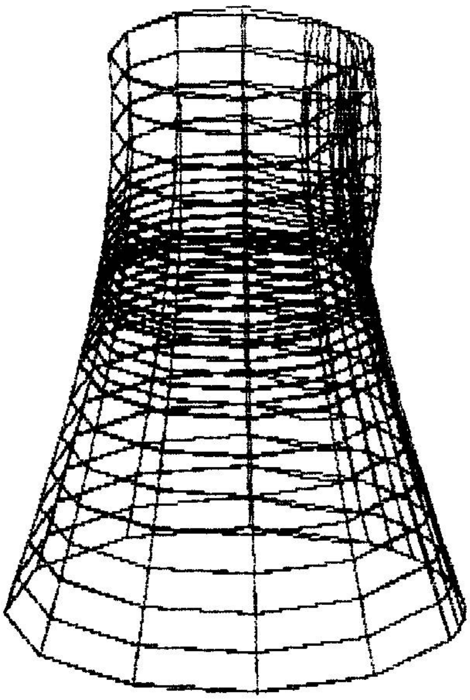


a. Used shell element

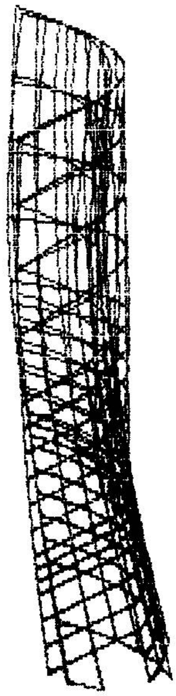


b. FEM modell of the shell

Fig. 2 a,b.
Finite Element model of the discussed cooling tower.



a.Displacements of the shell under the wind load. Length of the slit 40 m



b. Zoom of the deformed place

fig. 3 a,b
Results of the computer calculations.

On the base of the upper listed results it was possible to establish the limited length of the slit which gave enough values of the displacements to cause destruction of the shell.

During the calculation process the limitations of used computer program were seen. Mainly they may be listed as follows:

- it was possible to use (in the applied system STRAINS) only 500 elements. It was only enough to cover the tower coat, by the mesh dimensioned 6x6m. Only around the slit, the mesh had smaller dimensions 0.20x0.2 m. The mesh like that was to rare and received results were only a far approximation of reality,
- there were used only isotropic elements (concrete) to build model of the reinforced shell. In the used system there were no special reinforced concrete elements,
- the used system was't connected with dimensioning systems, so it was necessary to transfer data handy.

Instead so many limitations and disadvantages the gained results make the evidence that such method of structure analysis has significant future.

4. CONCLUSIONS

The presented way of the total structural design of structures, lead to a wider evaluation of their work. That is a way of connecting in the one system, on the same level, methods of structure analysis and synthesis, with the possibility of the data exchange. During the process of the structural designing it is possible to take into account many new aspects and observe a structure response. Also it's possible to design wanted features of structure, e.g, material, shape or ways of reinforcing.

In the presented example the structural analysis of work of the cooling tower shell work with slit permitted to check several hypotheses about collapse reasons.

This way of structure analysis is possible only while using computers. Growing calculation power of computers gives a lot of hope about progress in that field. This trend is bound to continue.

REFERENCES,

[1] Bruggeling A.S.G., A new Approach to Concrete Structures. Classification and Design, Polish Academy of Science, Warsaw, 1989,

[2] Breen J.E.; Why Structural Concrete. Preproceedings on IABSE Colloquimum ,Structural Concrete: Stuttgart, 1991.

[3] Scordelis A.; Analysis of Structural Concrete Systems, Preproceedings on IABSE Colloquim , Structural Concrete, Stuttgart, 1991,

[4] Gryszkiewicz M., STRAINS - A series of Computer Programs for Static Finite Element Analysis of Structures. Instruction, Warsaw , 1990.

Cracking Analysis of a Prestressed Concrete Containment Structure

Analyse des fissures dans un réservoir en béton précontraint

Analyse der Rissbildung bei einem Spannbetonbehälter

Frano B. DAMJANIC
Prof. of Civil Eng.
Univ. of Ljubljana
Ljubljana, Yugoslavia

Joaquim A. FIGUEIRAS
Assoc. Prof.
Univ. of Porto
Porto, Portugal

Rui H.C.F. POVOAS
Assist. Prof.
Univ. of Porto
Porto, Portugal

Marjan STANEK
Senior Lecturer
Univ. of Ljubljana
Ljubljana, Yugoslavia

SUMMARY

A practical problem-related to the assessment of structural cracking is considered. The structure is analysed using two finite element models, i.e. two separate computer codes. Although the same numerical model for simulation of nonlinear concrete behaviour is employed, certain discrepancies in the numerical results are observed. This is mainly due to different modelling of boundary conditions and prestressing. Regardless of the problems mentioned, FEM is proving to be a most powerful engineering tool.

RÉSUMÉ

L'article traite de l'évaluation des fissures dans une construction. La construction a été analysée par deux modèles d'éléments finis. Bien qu'on applique le même modèle numérique pour la situation du comportement non-linéaire du béton, une divergence des résultats numériques a été observée. C'est dû surtout aux modèles différents des conditions limites et des forces de précontrainte. Cependant, la méthode est efficace pour l'analyse structurale.

ZUSAMMENFASSUNG

Der Beitrag behandelt die rechnerische Erfassung der Rissbildung eines Spannbetonbehälters. Bei der Analyse der Konstruktion wurden zwei entwickelte Modelle der Methode der finiten Elemente und das zugehörige Rechenprogramm verwendet. Obwohl das nichtlineare Verhalten des Betons in beiden Modellen gleich simuliert wurde, konnten Differenzen in den numerischen Ergebnissen festgestellt werden. Die Ursache liegt in den verschieden modellierten Randbedingungen und in der Vorspannung. Trotz dieser Probleme hat sich die Methode der finiten Elemente als ein sehr erfolgreiches Ingenieurmittel erwiesen.

1. INTRODUCTION

The 1st Conference on Computer-Aided Analysis and Design of Concrete Structures [1] held in Split in 1984 successfuly summarized a large number of numerical models and techniques, particularly these based on the finite element method, for the analysis of concrete structures. Since then, numerical modelling and solution techniques have been significantly improved. On the other hand, the advent of powerful computers, as well as microcomputers with parallel processing, endowed with the exciting prospect of developing an interactive design system gives the possibility that such techniques can be efficiently employed to the solution of complex practical problems. However, it is true to say that the numerical capability is nowadays in advance of the knowledge of concrete constitutive behaviour. The knovledge of concrete behaviour under triaxial stress states is incomplete; or at least no consistent constitutive relation has yet been established. Nevertheless, in spite of this fact, satisfactory results can be achieved in engineering practice. The 2nd ICC conference [2], together with another recent conference has proved this trend. However, having experience in both the fields, practical structural concrete design and finite element analysis, we are aware of all the problems, or quoting Ref. [3], "the current inconsinstencies" well summarized in Section 3 of the mentioned colloquium's introductory report. In this paper showing results of F.E. analysis of a practical problem we intend to point out certain issues primarely related to F.E.M. as an efficient designe-oriented analysis tool.

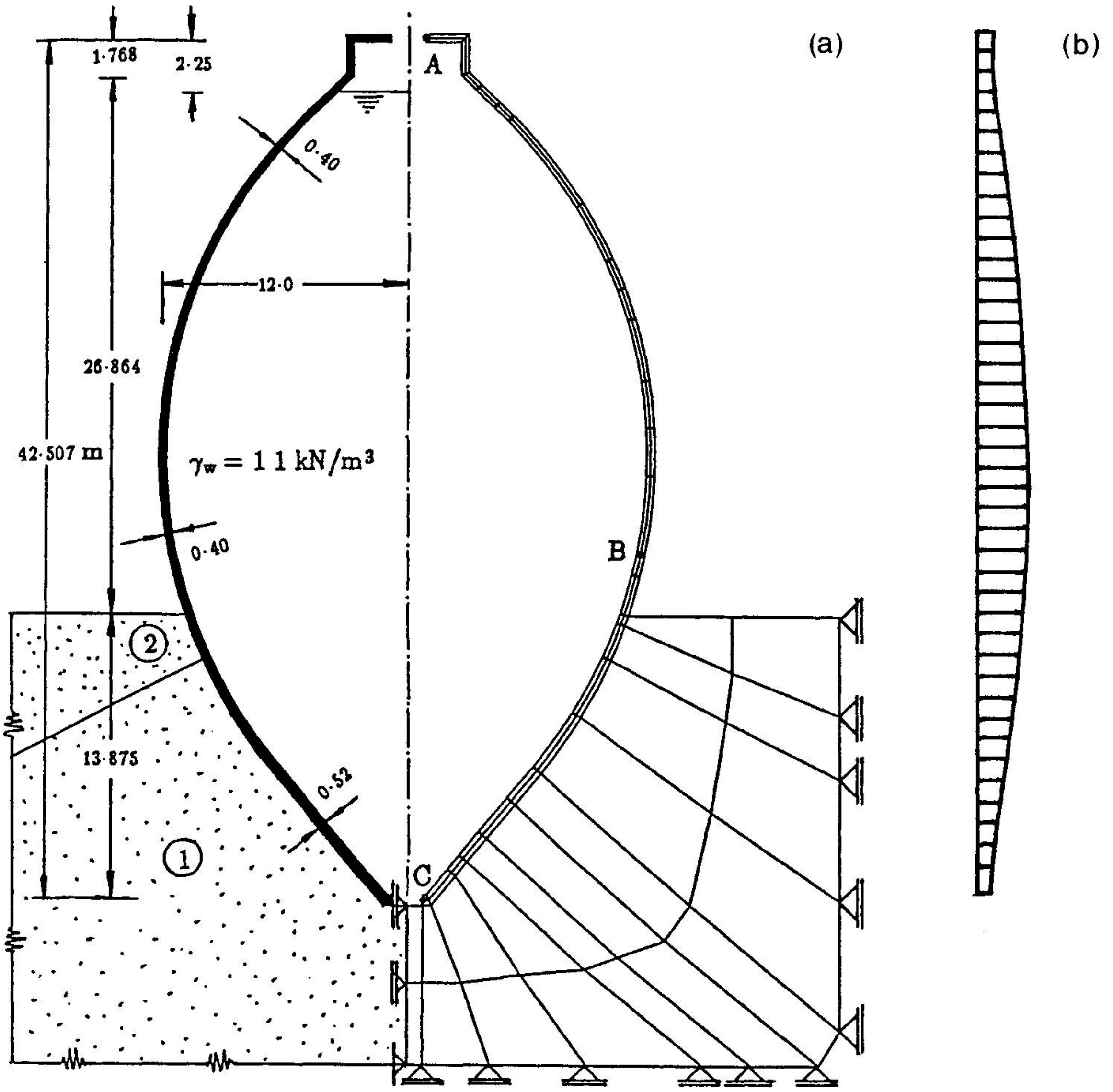


Fig.1 Prestressed concrete septic containment showing problem details and finite element idealisations: (a) axisymmetric mesh, and (b) shell element mesh.

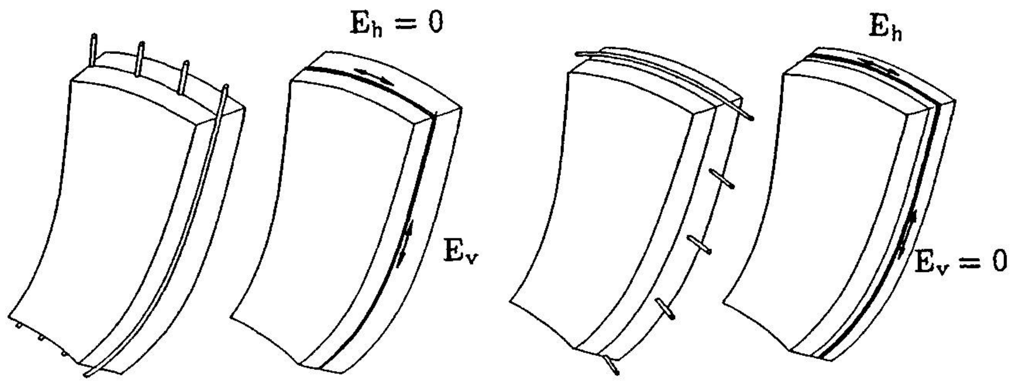


Fig.2 Prestressing cable position and finite element idealisation (Axi. elements).

The example here presented is related to the assessment of structural cracking of a prestressed septic containment under normal service loading. As a matter of fact, cracking of concrete structures very often occurs even far bellow service load conditions, and represents probably the main, and also undesirable, feature of concrete behaviour. Permiting insight into the cracking phenomenon (initiation, spreading and closing of cracks) the finite element method makes possible a rational analysis of concrete structures.

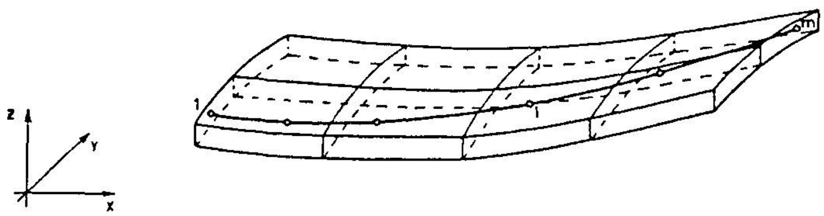


Fig.3 Geometry of the prestressing tendon (Shell elements).

The structure is analysed using our two finite element programmes (the first one based on the 2D/axisymmetric formulation [4] and the second based on the shell formulation [5]) for the non-linear analysis of reinforced and prestressed concrete structures. Numerical model employed in the programmes accounts for the most dominant nonlinear behaviour of concrete, e.g. multiaxial elastoplastic compressive behaviour including crushing, initiation and spreading of cracks etc. The prestressing cables are represented with discrete elements allowing uniaxial elastoplastic modelling of steel behaviour. The model is earlier developed [6-8] and further improved and extended. Results of several engineering studies [9-13] illustrate the applicability and practical merit of both the model and the codes.

2. ILLUSTRATIVE EXAMPLE

2.1 Problem description

The containment, a prestressed concrete structure shaped in an axial symmetric "amphora" form, was earlier designed for the sewage treatment plant of the city of Ljubljana. The design followed a "standard" engineering manner; a linear elastic finite element code was employed to determine internal forces, than reinforcement arrangement was defined. Essential details of the structure designed are given in Fig.1. Approximatively one third of the structure, the lower conical part, is "implanted" into the soil. No additional structural foundation elements are designed. The structure is reinforced with vertical (meridional) prestressing cables located in the middle of the structural wall cross section, and horizontal (circumferential) prestressing cables are close to the outer surface of the wall. There are 96 meridional cables all together. However, the lenght of 24 cables is practically from the top to the bottom of the structure. The rest of them are located in the mid-hight of the structure to reinforce the widest part of the structure. The total number of circumferential cables is 142. The effective prestress forces (after initial prestressing losses) of 600 kN/cable and 500 kN/cable are expected to act in meridional and cicrumferential cables respectively. One of the major requirements for the structure is that structural cracking is not permitted due to very agressive contents of the waste water. Since the structure was designed assuming prestressed concrete as a homogeneous isotropic linear elastic material we reanalysed the containment using two non-linear F.E. codes (Axisymmetric and shell formulation). Finite element mesh discretizing the structure and a part of surrounding soil for the axisymmetric analysis is shown in Fig.1(a). The prestressing cables are represented by a pair of membranes of equivalent thickness (Fig.2) and the influence of the prestress forces due to curved shape of the structure by external equivalent pressure (as suggested in Ref.(14). For the shell analysis a vertical section (6 degrees) of the structure is discretized with 44 shell elements shown in Fig.1(b). Prestressing cables are modelled as indicated in Fig.3 using a tendon formulation [12]. Soil influence is simulated by equivalent springs at the appropriate nodes.

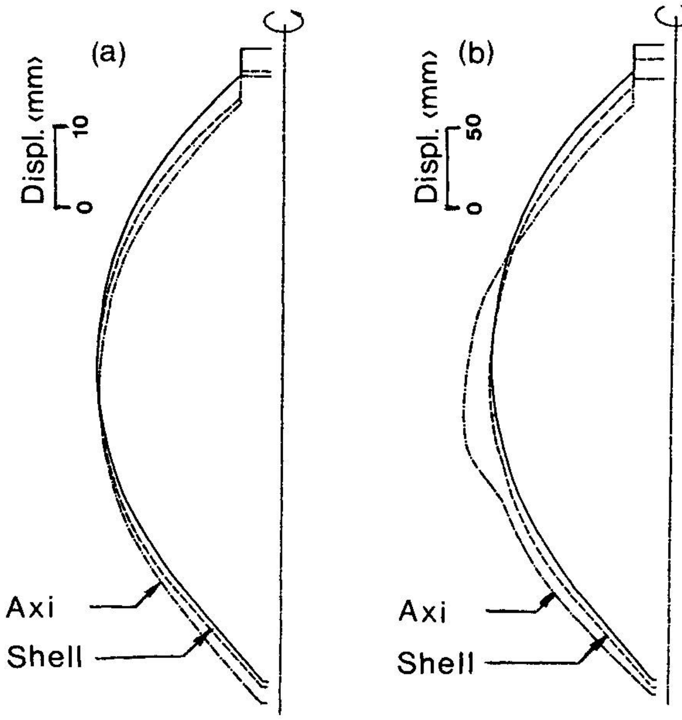


Fig.4 Deformed shapes due to: (a) Self-weight and prestressing, and (b) servis load.

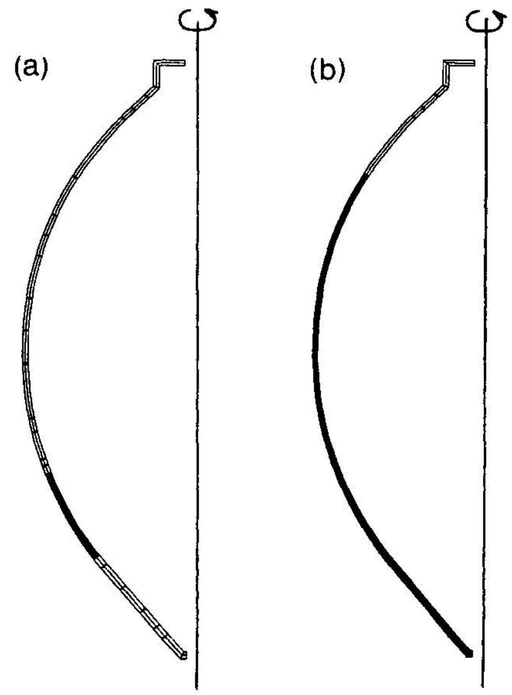


Fig.5 Tensile cracking zones using: (a) Shell formulation, (b) Axisymmetric formulation.

2.2 Numerical results

The structure is analysed under the conditions of prestress and internal hydrostatic pressure equivalent to the containment filled up to the maximum level (See Fig.1). The deformed structure profiles, i.e. the prestressed configuration and the configuration at the full service load compared to the initial configuration of the structure are illustrated in Fig.3. No cracks are predicted under the prestressing conditions. However, vertical cracks throughout the thickness of the structure are predicted at the service load conditions. The zones of vertical cracks are indicated in Fig 5. The max. crack width (supposing a crack distribution at the position of meridional cables only) is estimated in the range of 1.13 mm (axisymmetric analysis) and 0.5 mm (shell analysis). Displacements obtained using the two codes are compared in Table 1. Significant stresses in cables and in concrete are listed in Tables 2 and 3 respectively. Considering two set of the results it is clearly seen that: (a) vertical cracks will occur, and (b) the shell formulation predicts a rather stiffer structural response. The differences in the results are primarely due to different modelling of: (a) prestressing and (b) influence of the surrounding soil. More realistic prestress modelling is applied in the shell analysis (Equivalent load in the axisymmetric). On the other hand, soil interaction in the axisymmetric is modelled in a more appropriate way analysis.

Displacements [mm]		SHELL	AXISYMMETRIC
Load case 1: g + p	top	u_r= 0.004 u_z=-3.08	u_r= ≈ 0 u_z=-3.59
	bottom	u_r=-0.016 u_z=-0.837	u_r=-0.005 u_z=-3.03
Load case 2: g + p + w	top	u_r= 0.016 u_z=-9.88	u_r= ≈ 0 u_z=-22.84
	bottom	u_r=-0.912 u_z=-5.17	u_r= 0.271 u_z=-8.86

Table 1: Radial and vertical displacements.

Min&Max stresses in prestressing cables $*10^6$ [kN/m^2]		SHELL	AXISYMMETRIC
Load case 2: g + p + w	circum.	0.90 - 1.04	0.89 - 1.23
	meridional	1.07 - 1.08	1.07 - 1.50

Table 2: Extreme stresses in the cables.

Max. compressive stress in concrete [kN/m^2]		SHELL	AXISYMMETRIC
Load case 1: g + p	circum.	-8720	-735
	in plane	-3201	-1138
Load case 2: g + p + w	circum.	-3596	-486
	in plane	-2926	-3891

Table 3: Max. principle compressive stress.

3. DISCUSSION AND CONCLUSIONS

The brief description of the results obtained analysing one single structure using non-linear FE analysis and applying practicaly the same consitutive model for the simulation of R.C. behaviour, but based on two different formulation indicates the following:

* Non-linear FE analysis is (or if there is any doubt, it will be definitively very soon) one efficient reliable and practical design-oriented analysis tool which is perfectly consistent with "strut-and-tie model" (STM) concept.

* It seems that using FE structural analysis a step-by-step procedure (Firstly linear FE analysis, dimensioning, and than fully non-linear FE "control") has to be used.

* Although the same concrete constitutive model is employed in the two analyses presented the results are a rather different. This is not due to two different formulations applied, but primarely due to different boundary condition interpretations.

* In additon to the above, we can conclude that nowadays a large number of concrete constitutive models, from a very simple one (if we do not take into account the "historical" linear elastic) to very sophisticated are spreaded and applied in practice. In this case, applying different models included in a FEA, a spectrum of different results for the same example would be obtained. An immediate question is: What is

a real "truth"? Which model is the most reliable, appropriate, or correct? This is a natural question or dilemma of any researcher and designer. From the FEA aspects we may conclude, as we mentioned earlier, that the numerical capability is in advance of the konwledge of structural concrete behaviour.

Therefore we strongly support the initiative to consider the list of "challenges" (Section 5, Ref. [3]) in the colloquium. Furthermore, we expect from the colloquium to make a general, but an organized step forward in structural concrete design.

REFERENCES

1. Proc. 1st Int. Conf. Computer-Aided Analysis and Design of Concrete Structures, Eds. F.B.Damjanic et al., Pineridge Press, UK, 1984.
2. Proc. 2nd Int. Conf. Computer-Aided Analysis and Design of Concrete Structures, Eds. N.Bicanic, H.Mang, Pineridge Press, UK, 1990.
3. BREEN J.E., Why Structural Concrete. Preliminary invited lecture at IABSE Colloquium "Structural Concrete", Stuttgart, 1991.
4. DAMJANIC F., A Finite Element Technique for Analysis of Reinforced and Prestressed Concrete Structures. Finite Element Methods for Nonlinear Problems, Eds. P.Bergan, K.J.Bathe, W.Wunderlich, Springer, Berlin, 1986, p.623-637.
5. OWEN D.R.J., FIGUEIRAS J.A., Ultimate Load Analysis of Reinforced Concrete Plates and Shells Including Geometric Nonlinear Effects. Finite Element Software for Plates and Shells, Eds. E.Hinton, D.R.J.Owen, Pineridge Press, UK, 1984, p.235-326.
6. DAMJANIC F.B., Reinforced Concrete Failure Prediction under both Static and Transient Conditions, Ph.D. Thesis, University of Wales, 1983.
7. FIGUEIRAS J.A., Ultimate load analysis of anisotropic and reinforced concrete plates and shells. Ph. D.Thesis, University of Wales, 1983.
8. OWEN D.R.J., FIGUEIRAS J.A., DAMJANIC, F., Finite element analysis of reinforced and presstresed concrete structures including thermal loading. Comp. Meth. Appl. Mech. Engng., 41, 1983, p.323-366.
9. FIGUEIRAS J.A., OWEN D.R.J., Nonlinear Analysis of Reinforced Concrete Structures. Computer-Aided Analysis and Design of Concrete Structures, Eds. F.B.Damjanic et al., Pineridge Press, UK, 1984, p.509-532.
10. DAMJANIC F., Finite Element Analysis of Buried Reinforced Concrete Pipes under Internal Pressure. NUMETA/85, Eds. J.Middleton, G.Pande, Balkema, Rotterdam, Netherlands, Part 2, 1985, p.679-686.
11. FIGUEIRAS J.A., Design of RC Shells by Using a Finite Element Nonlinear Model. Numerical Methods for Non-linear Problems, Eds. C.Taylor et al., Pineridge Press, UK, Vol.3, 1986, p.1276-1286.
12. POVOAS R.H.C.F., FIGUEIRAS J.A., Nonlinear Analysis of Prestressed Concrete Shells: Tendon Formulation. Computational Plasticity: Models, Software and Applications, Eds. D.R.J. Owen et al, Pineridge Press, Swansea, 1989, p.1379-1394.
13. STANEK M., DAMJANIC F.B., CELCER V., Cracking Prediction of a Prestressed Concrete Septic Containment. Computer-Aided Analysis and Design of Concrete Structures, Eds. N.Bicanic, H.Mang, Pineridge Press, UK, 1990, p.623-631.
14. BRUGGELING A.S.G., An Engineeringg Model of Structural Concrete. Preliminary invited lecture at IABSE Colloquium "Structural Concrete", Stuttgart, 1991.

Serviceability Analysis of Reinforced Concrete Slabs

Analyse à l'état de service de dalles en béton armé

Berechnungsverfahren für die Gebrauchsfähigkeit von Stahlbetonplatten

Jana MARGOLDOVÁ
Dr. Eng.
Czech Techn. Univ.
Prague, Czechoslovakia

J. Margoldová received her Ph.D. degree in 1987 at the Czech Technical University in Prague. She works as a research engineer at the Building Research Institute of the Czech Technical University in Prague.

Radomír PUKL
Dr. Eng.
Czech Techn. Univ.
Prague, Czechoslovakia

R. Pukl received his Ph.D. degree in 1985 at the Czech Technical University in Prague. He works in the field of computational mechanics at the Czech Technical University in Prague and at the Stuttgart University.

Vladimír ČERVENKA
Dr. Eng.
Czech Techn. Univ.
Prague, Czechoslovakia

V. Červenka received his Ph.D. degree in 1970 at the University of Colorado in Boulder. He is active in the research of concrete structures at the Czech Technical University in Prague and at Stuttgart University.

SUMMARY

Proposed analysis of reinforced concrete slabs in the serviceability limit states is based on the finite element method and the effective stiffness model of the plate cross-section. The nonlinear effects of concrete cracking, tension stiffening, and simplified creep behaviour are included. Examples of the analysis of real structures are described.

RÉSUMÉ

La méthode proposée pour l'analyse de dalles en béton armé à l'état de limite de service est basée sur la méthode des éléments finis, ainsi que sur un modèle de rigidité effective de la section transversale de la dalle. Les effets non-linéaires de la fissuration du béton, ainsi qu'un modèle simplifié pour la fissure sont également inclus. Des exemples d'analyse de structures réelles illustrent ces propos.

ZUSAMMENFASSUNG

Das vorgeschlagene Berechnungsverfahren für Stahlbetonplatten im Gebrauchszustand beruht auf der Finite Elemente Methode und dem Modell effektiver Querschnittssteifigkeiten. Es werden die nichtlinearen Effekte der Rissbildung, der tension-stiffening-Effekt und, auf einfache Weise, das Kriechen des Betons berücksichtigt. Es werden Beispiele für die Berechnung realistischer Bauwerke gegeben.

INTRODUCTION

The authors have developed a method for the computer analysis of reinforced concrete plates under serviceability conditions. The purpose of this work was to provide an efficient tool for practical design which would enhance current engineering practice. It is known that deflection of reinforced concrete structures is greatly affected by concrete cracking and by the long-term effect of creep. However, such effects are non-linear and they can complicate analysis to the extent that it becomes impractical. Therefore, it was decided to simplify the material and numerical model as much as possible and to focus on efficiency at the expense of unnecessary accuracy.

In view of the above considerations the following assumptions are made:

1. Thin-plate theory based on the Kirchhoff hypothesis is used for plate mechanics. This theory is based on the action of bending moments only. Normal forces are not considered and shear forces must be derived from moments.

2. An effective stiffness model is used in which the material behavior is defined for the whole cross-section of the plate. The constitutive law is defined for moments and curvatures.

The work presented here is based on the first author's Ph.D. dissertation [1].

CONSTITUTIVE MODEL

A smeared approach is used to model the material non-homogeneity, i.e., the material properties are constant within a considered element volume but can vary between the elements. The moment-curvature relation for the plate bending element is as follows:

$$\mathbf{m} = \mathbf{D}.\mathbf{k} \tag{1}$$

where $\mathbf{m} = \{m_x, m_y, m_{xy}\}$ is the vector of internal moments per unit width and $\mathbf{k} = \{\kappa_x, \kappa_y, \kappa_{xy}\}$ is the vector of corresponding curvatures. The constitutive matrix $\mathbf{D}$ for uncracked concrete has the form of the elastic matrix for isotropic plates. The cracking criterion is given by the cracking moment $m_c = h^2 R_t/6$, where R_t is the tensile strength of concrete. When the principal moment reaches the cracking moment, the constitutive matrix takes the form of the orthortopic elastic plate matrix. The axes of orthotropy are aligned with the crack direction, Fig.1:

$$\mathbf{D} = \begin{bmatrix} d_{nn} & d_{nt} & 0 \\ d_{tn} & d_{tt} & 0 \\ 0 & 0 & d_{qq} \end{bmatrix} \tag{2}$$

The coefficients of the constitutive matrix are calculated according to Jofriet et al. [2] :

$$d_{nn} = \frac{B_n}{1-\nu^2} \qquad d_{nt} = d_{tn} = \nu\sqrt{d_{nn}d_{tt}} \tag{3}$$
$$d_{tt} = \frac{B_t}{1-\nu^2} \qquad d_{qq} = \tfrac{1-\nu}{2}\sqrt{d_{nn}d_{tt}}$$

where B_n, B_t are the flexural stiffnesses normal and parallel to the cracks, respectively, and ν is the Poisson's ratio. The flexural stiffness of the cracked cross section is modeled according to Hajek [3] as:

$$B_r = \frac{h_o z}{\frac{\psi_s}{E_s A_{st}} + \frac{2\psi_b}{E_c A_{ci}}} \qquad r = n, t \tag{4}$$

in which n, t are indices for the directions normal and parallel to cracks. E_s and E_c are the elastic moduli of the steel and concrete, respectively; A_{st} and A_{ci} are the areas of steel and

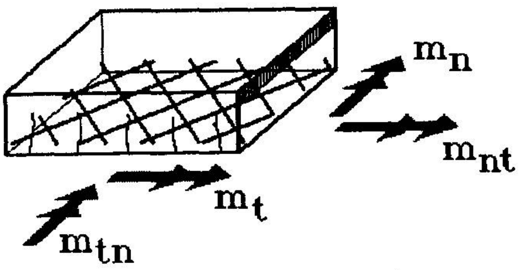


Fig.1 Moments on the cracked plate element.

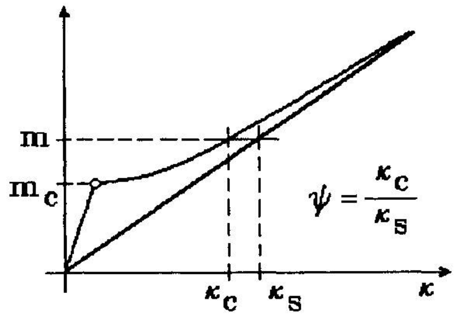


Fig.2 Moment-curvature diagram. Meanig of tension stiffening coefficient ψ.

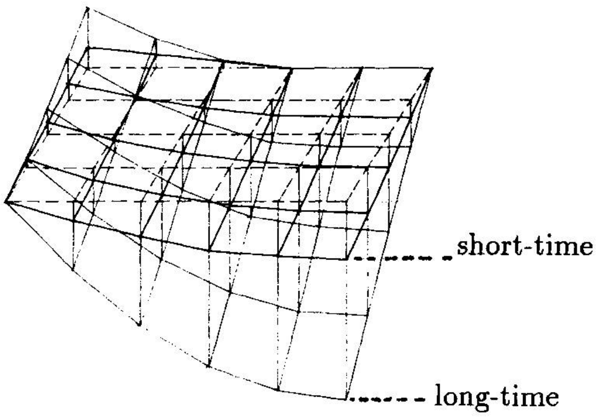


Fig.3 Long- and short-time deflection shapes in Hajek's experiments [3].

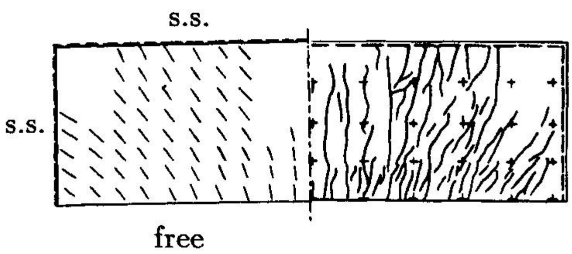


Fig.4 Comparison of experimental and analytical crack patterns.
Legend: free - free edge
s.s. - simply supported edge

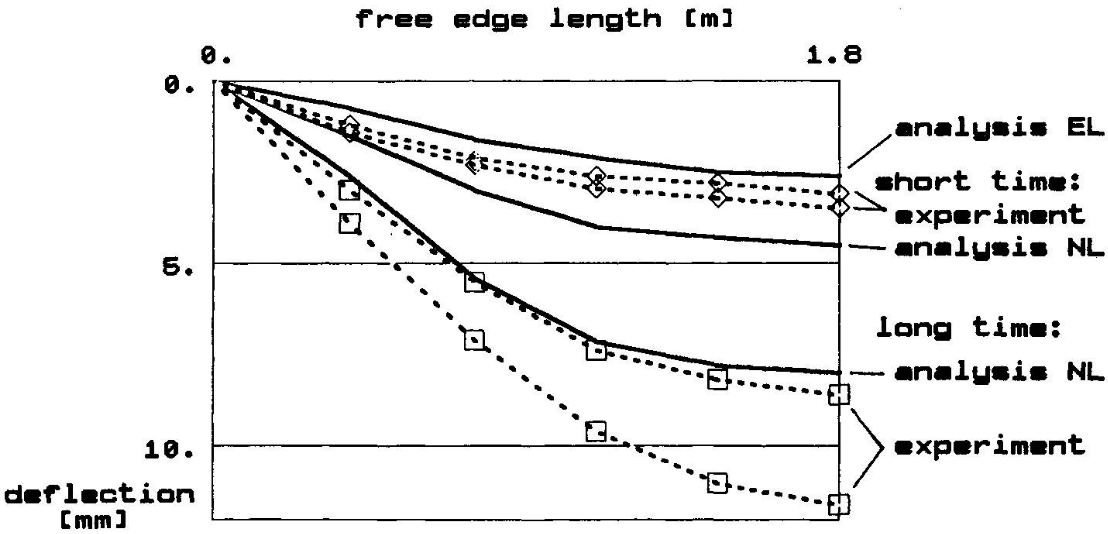


Fig.5 Comparison of analytical and experimental deflections of the free edge for Hajek's slab [3].

concrete, respectively; h_o is the effective depth and z is the lever arm of the internal forces. The tension-stiffening coefficient ψ_s represents the reduction of steel strains due to concrete tensile stiffness after cracking. A similar effect in the compression zone is described by ψ_c. These coefficients are defined as follows:

$$\psi_i = 1 - \rho\left(1 - \psi_{ir}\right), \quad i = s, c, \qquad \rho = \frac{1}{4}\left(5\frac{m_c}{m} - 1\right) \tag{5}$$

$$\psi_{sr} = \frac{6}{0,85} n \frac{A_s\left(2h_o - h\right)z}{bh^3}, \qquad b = 1m \tag{6}$$

$$\psi_{cr} = \frac{6}{1,7}\frac{A_{ci}\, z}{bh^2}, \qquad n = \frac{E_s}{E_c} \tag{7}$$

in which the indices s, b refer to steel and concrete, respectively; ρ is an interpolation parameter for tension stiffening and ψ_{sr}, ψ_{cr} are initial values of ψ_i after cracking. The moment-curvature diagram resulting from this model is schematically shown in Fig.2. In the case of an inclined crack, the effective reinforcement area, A_s, is calculated using appropriate transformation of the two reinforcing directions passing through the cracked cross-section. Three crack modes are recognized: 1. one crack, 2. two orthogonal cracks on the same surface, 3. two orthogonal cracks on opposite surfaces.

The time effect is included as a simplified method based on an extensive experimental investigation by Hajek of real plates [3]. He found an affinity relationship between the short-term and the long-term deflected shapes of plates (Fig.3). The deflection $w(t, x, y)$ at the time t is governed by the simple formula

$$w(t, x, y) = (1 + \beta).w(in, x, y) \tag{8}$$

in which $w(in, x, y)$ is the initial deflected surface of the plate and β is the creep coefficient. The creep function is taken from the Czechoslovak National Standard for concrete structures as follows:

$$\beta = \left(0,15 + 0,008e^{-0,015t_1}\right)\left(1 - e^{-0,07\sqrt{t-t_1}}\right)\phi \tag{9}$$

where t is the age of concrete at the considered time, t_1 is the age of concrete at the time of load application, and ϕ is the coefficient of creep.

NUMERICAL SOLUTION

The numerical solution of the plate analysis is performed using the finite element method. The Clough-Felippa [4] quadrilateral finite element is used. The element is composed of four linear-strain subtriangles. It has four external nodes with three degrees of freedom in each node, one deflection and two slopes. This element has piece-wise linear moment distribution and thus gives quite good results even for coarse meshes. The non-linear solution is done by the secant stiffness method. The total loading is applied, then iteration is performed until the constitutive laws and equilibrium are satisfied. Finally, the creep effect is included by scaling the deflected shape using the creep law, Eq.9.

The program can handle irregular geometrical shapes, internal hinges, elastic supports, elastic foundations, concentrated and distributed loadings and other features.

The program is written in FORTAN77 and can be installed on various computer systems (mainframes, workstations, personal computers). It is equipped with efficient pre- and post-processors by means of which numerical results can be presented on grahical terminals, plotters, or printers.

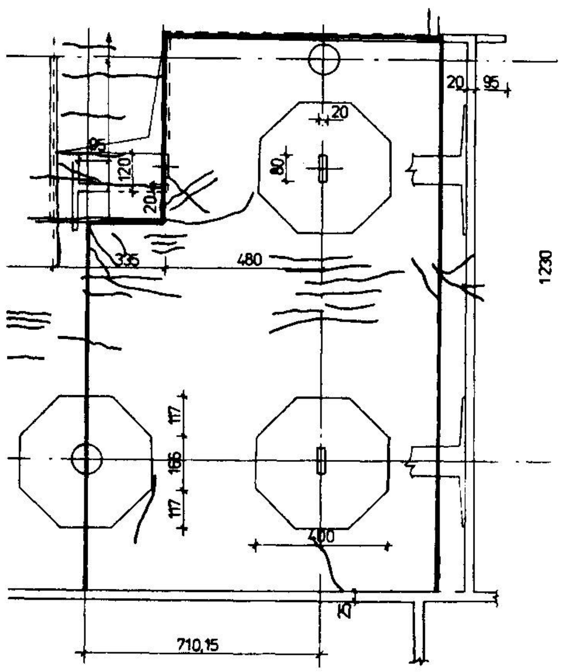


Fig.6 Garage slab. Plan view of the slab section with observed cracks.

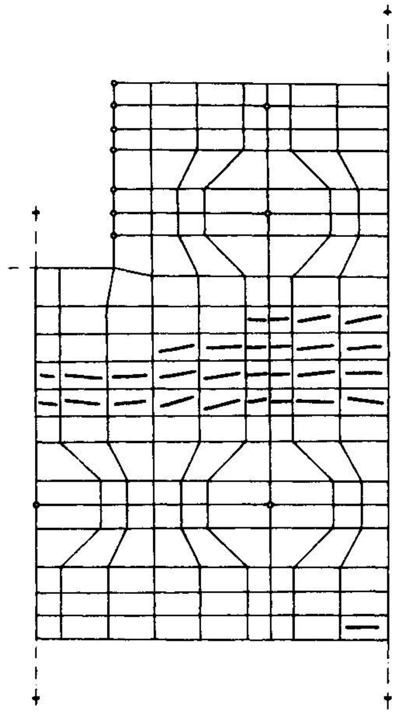


Fig.7 Garage slab. Finite element model with calculated crack pattern.

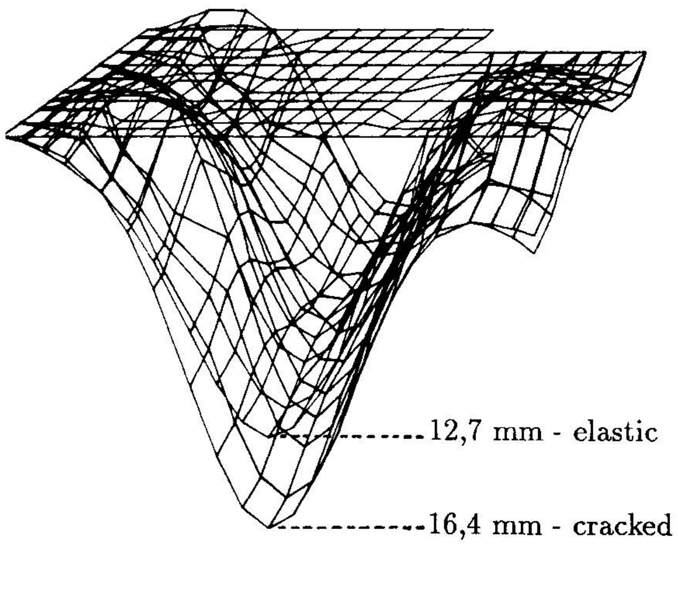


Fig.8 Calculated deflection surfaces.

COMPARISON WITH HAJEK'S EXPERIMENTS

The experimental slabs in ref.[3] were supported on three sides. The slab dimensions were 3,6 x 1,2 x 0,12 m. They were reinforced by mesh with reinforcing ratios in both directions equal to 0,0112. The concrete quality was 30 MPa. The slabs were loaded 28 days after casting by a uniform load of 14,7 kN/m^2. The load was applied by plastic bags under air pressure, and was kept constant during the entire loading history. The slabs were kept under controlled laboratory conditions where deflections under sustained load were monitored during a period exceeding one year. An example of measured deformed shapes is shown in Fig.3. Long term deformations are considered at 400 days.

The slab was analyzed by the above described program. The deflections of the free edge are compared in Fig.5 with experimental results for short-term and long-term effects. The results from two nominally identical experimental slabs A, B are shown to indicate the scatter of the experimental results. The experimental and analytical crack patterns are compared in Fig.4. Good agreement of the analysis with the experiment is found.

EXAMPLE OF A SLAB IN PARKING GARAGE

The program was successfully used for checking deflections of several slab structures in design practice. One example is shown here. It concerns an existing floor structure of a parking garage which did not performed satisfactorily after the first few years of service life. It was required to check the deflections of the slab. The project also included the design of repair provisions. Here we are showing only partial resuts for illustration. Fig.6 shows a section of the floor structure comprised of an irregular slab, supported by columns. The slab, 20 cm thick, is cast-in-place and the octagonal precast shear heads are 35 cm thick. The existing cracks are also recorded. Fig.7 shows the finite element model and crack pattern as calculated by analysis. The calculated deflected shapes are shown in Fig.8. The elastic and cracked shapes are compared. The analysis was used to check the serviceabilty performance of the slab before and after the designed reconstruction.

CONCLUSION

The proposed analysis of reinforced concrete plates based on the finite element method and the effective stiffness of the slab cross section gives a fair approximation of the real performance of slabs under serviceability conditions. It can model complex geometrical shapes and important effects of cracking, tension stiffening, and creep.

References

[1] MARGOLDOVÁ, J. - Physically Nonlinear Analysis Of Reinforced Concrete Slabs, Ph.D.Thesis, SÚ ČVUT in Prague, August 1987, (in Czech).

[2] JOFRIET, J.C., McNIECE, G.M. - Finite Element Analysis Of Reinforced Concrete Slabs, Journal of the Structural Division, Proceeding of ASCE, March 1971, pp.785-807.

[3] HÁJEK, J. - Deformation Of Reinforced Concrete Two-Way Slabs Under Long-Term Loading, Research Report, USTARCH SAV, Bratislava 1985 (in Slovak).

[4] FELIPPA, C.A. - Refined Finite Element Analysis Of Linear And Nonlinear Two-Dimensional Structures, Ph.D.Thesis, University of California, Berkeley 1969.

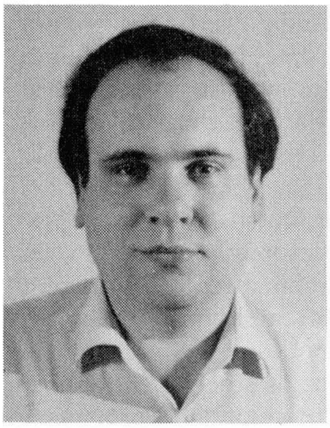
Computer-Aided Automatic Construction of Strut-and-Tie Models

Génération automatique de modèles conformes à l'analogie du treillis

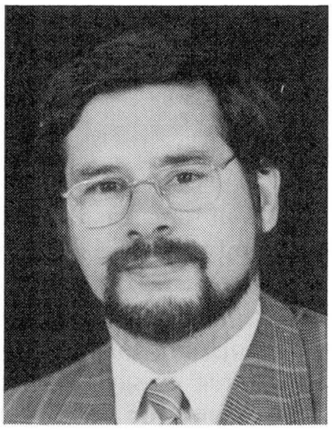
Automatisches Entwickeln von Stabwerkmodellen

Argiris HARISIS
Consulting Engineer
University of Patras
Athens, Greece

Michael N. FARDIS
Prof. of Civil Eng.
University of Patras
Patras, Greece



Argiris Harisis, born 1958, received his Civil Engineering Diploma from the University of Patras, and his M.Sc. in Civil Engineering from the University of Toronto (1986). He is now a consulting engineer specializing in computer applications, and a Ph.D. candidate in Structural Engineering at the University of Patras.



Michael N. Fardis, born 1949, holds an M.Sc. and a Ph.D. from the Civil Engineering Department at the Massachusetts Institute of Technology, where he served as Assistant and Associate Professor from 1979 to 1983. He is now a Professor of Concrete Structures at the University of Patras.

SUMMARY

A multi-step algorithm is presented for the automatic construction of strut-and-tie models of in-plane loaded structural concrete plates. The algorithm is implemented in a user-friendly post-processing mode into a plane-stress linear elastic finite element program, and results are compared to manually constructed strut-and-tie models.

RÉSUMÉ

Cet article présente un algorithme destiné à générer automatiquement des treillis conformes à l'analogie, en vue de l'étude de structures planes en béton. Pour faciliter l'utilisateur, l'algorithme est muni d'un système de post-processing qui l'amène dans un programme d'éléments finis linéaires-élastiques exprimant les contraintes dans le plan. Les résultats obtenus sont comparés à des modèles de treillis dessinés sans l'intervention de l'ordinateur.

ZUSAMMENFASSUNG

Ein Algorhythmus zur automatischen Entwicklung des Stabwerkmodells einer Stahlbetonscheibe wird vorgestellt. Dieser Algorhythmus ist in benutzerfreundlicher Weise in einem linear-elastisch Finite Elemente Programm zur Berechnung der ebenen Beanspruchung eingebaut. Die Ergebnisse werden mit manuell aufgestellten Stabwerkmodellen verglichen.

1. INTRODUCTION

Recent years have seen the emergence of Strut-and-Tie models as a powerful general approach for the rational and consistent design of structural concrete plates and of two-dimensional regions of static or geometric discontinuity (the so-called D-regions) [1], [2]. Strut-and-Tie models have been originally proposed and developed as a hand-calculation design procedure, in which the structural engineer uses his experience and intuition to draw load paths through the structure in the form of a (usually statically determinate) truss, which is then analysed for the design loads and proportioned according to the applicable Code and to other appropriate rules of practice. In the manual exercise of the construction of the Strut-and-Tie model the engineer may be aided by knowledge of the magnitude and of the directions of principal stresses, obtained by a linear Elastic plane-stress Finite Element analysis of the structural concrete plate or D-region under the design loads. The Struts-and-Ties of the model may then be drawn collinear to the resultants of the principal stresses. However, even when such Finite Element results are available, development of a Strut-and-Tie model for a nontrivial case requires from the designer not only a certain experience and expertise but also considerable time. This may work against the wide acceptance of this new and powerful design tool by the Structural Engineering community, and in favor of the old-fashioned detailing rules-of-thumb advocated by traditional Codes of practice. This is more so as a high design cost for structural concrete plates and discontinuity regions is not economically justifiable, as they are of relatively low cost and of the one-of-a-kind type.

Development of computational tools for the construction of Strut-and-Tie models will reduce the total design time and cost, and therefore may contribute to their more widespread application. To date, and despite the fact that Strut-and-Tie models have drawn considerable attention in recent years, progress in this direction has been limited. As a notable exception, researchers at ETH have developed computational algorithms for the automatic verification of the nodes of a Strut-and-Tie model [3], and for the selection of such a model so that the total weight of steel in the ties is minimized [4]. Nevertheless, selection of the topology of the Strut-and-Tie model still remains a manual task. In the present paper an attempt is made to develop and apply a computational algorithm that automatically generates the topology of Strut-and-Tie models. Development of such an algorithm is facilitated by the fact that the primary information normally used by the engineer to sketch a Strut-and-Tie model, i.e. the stresses obtained from a Finite Element Analysis, is in a systematic digital form that can be directly processed by the computer for further utilization.

2. ALGORITHM FOR THE SELECTION OF THE STRUT-AND-TIE TOPOLOGY

The first stage of the proposed algorithm consists of a linear Elastic Finite Element Analysis of the two-dimensional element or region, subjected to the force and displacement Boundary Conditions of the problem. The Finite Element mesh should be relatively fine and is defined by automatically generated nodes with user-selected constant spacing in the two orthogonal directions x and y. The Analysis yields nodal stresses, by averaging over the neighboring elements, and from them computes (and plots, if the user so desires) the magnitude and the direction of the principal stresses, σ_1 and σ_2, at the nodes. Two databases are then formed, one referring to the positive principal stresses and the other to the negative ones. The generic record in each database includes the coordinates x and y of the node, the value of σ_1 or σ_2 there, and the angle θ_1 and θ_2 between its direction and axis x. Using these databases, the program performs the following tasks, separately for the positive and the negative stresses :

1) It identifies the group of nodal points where the value of the principal stress of interest (positive or negative) lies within a certain user-specified range of values with respect to the mean value of this stress over the entire two-dimensional region, e.g. the group of points where this stress exceeds the mean value by a user-specified multiple of the standard deviation of the stress in question (positive or negative) within the two-dimensional region, but not by more than another such user-specified multiple. In this way the points with relatively high nodal stresses are identified, irrespective of their location in the two-dimensional region.

2) The points in the group identified in 1) above are ordered according to the magnitude of the corresponding angle θ_i($i=1$ or 2), as this angle varies between 0^o to 180^o. In other words, the sample histogram (cumulative distribution function) of θ_i is constructed. Next we compute the difference between successive angles in this ordering, which is inversely proportional to the derivative of the histogram (i.e. to the probability density function) of θ_i. A range of angles where this difference is small (i.e. where the probability density function assumes high values) consists of points where the principal stress in question (positive or negative) not only assumes relatively high values but also has nearly parallel directions. So, if they are close, such points may form a strut (for negative stresses) or a tie (for positive). A user-specified number N of groups is formed, covering the entire range of angles between 0^o and 180^o, each group bounded by two successive local maxima of the difference between the angles θ_i (i.e. by two successive local minima of the probability density function of θ_i). To avoid cases of very small subgroups between two local maxima at almost equal values of θ_i, the variation of these differences is smoothed, by taking their 10-point moving average. In this way closely spaced local maxima merge into a single one.
3) All pairs of points in each group of nearly parallel principal stresses formed in 2) above, are examined for geometric proximity, by comparing the distance of the two points in the group to a certain percentage of the maximum size of the two-dimensional region. In this way each group is partitioned into subgroups of neighboring points with nearly parallel principal stress directions.
4) Provided that it contains a minimum number of points, each subgroup is replaced by a provisional straight-line strut or tie. This straight line passes through the center of gravity of the group points weighted by the value of the principal stress of interest (i.e. positive or negative). Its direction is chosen to coincide with the average principal stress direction of the points in the group, again weighted as above. An option has been included for turning the direction of the ties (or of some of them) to the closest one among a set of user-preferred reinforcement directions (e.g. at 0^o, 45^o, 90^o, or 135^o to the x-axis).
5) The provisional struts and ties are replaced by final ones, which form a statically determinate truss consisting of triangles. This is accomplished by finding triads of neighboring points of intersection of three different struts or ties, merging these three points into a single one, and shifting accordingly the corresponding struts or ties. Ties which have originally been arranged to be parallel to one of the pre-defined directions mentioned above (i.e. at 0^o, 45^o, 90^o, or 135^o), may be excepted from this shifting operation. Concentrated applied loads or reactions are also included in this part of the algorithm as struts or ties of fixed direction and position.

The algorithm has been implemented in a FORTRAN program that runs under DOS. Both the algorithm and the Finite Element program have been installed on IBM-compatible PC/XT's, and PC/AT's.

3. APPLICATION EXAMPLES

The computational procedure described above is applied to three simple examples: a) A simply supported deep beam with $l/h=2.0$, subjected to a concentrated force at midspan (Fig.1); b) a simply supported deep beam with $l/h=1.56$, with a square perforation extending over $h/3$ near the left support, subjected to a concentrated force at a distance $l/7$ to the right of midspan (Fig. 2, and [1], [4]); and c) the D-region of a stepped beam (Fig. 3) extending to both sides of the step by one beam depth, subjected to pure bending [1]. The Finite Elements used for the analysis were 8-noded square ones, with side about equal to one tenth of the maximum height of the two-dimensional region.

In the first problem the primal group of principal tensile stresses formed by step 1 of the algorithm was chosen to include all stresses exceeding the mean value of the principal tensile stresses by two standard deviations, whereas that of principal compressive stresses was selected to cover all compressive stresses from half a standard deviation below the mean (principal compressive) stress to infinity. For a user-specified maximum number of compressive and tensile stress subgroups equal to 3 each, the Strut-and-Tie model in Fig. 1 was finally obtained. This figure, and the ones to follow, shows the struts by dotted lines, and the ties by continuous ones. In addition, it shows as points the centroids of the stresses which correspond to the nearby strut or tie.

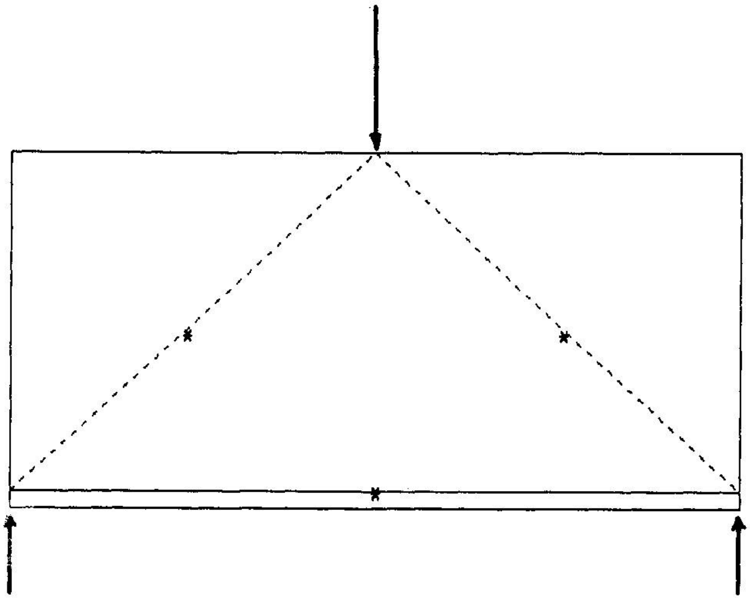


Fig.1 Strut-and-Tie Model for a deep beam

In the second problem the primal groups of principal compressive stresses has been selected as in the first problem, whereas that of the principal tensile stresses has been chosen to include all stresses exceeding the corresponding mean value. For a user-specified maximum number of subgroups of compressive and tensile stresses equal to 4 and 3 respectively, the Strut-and-Tie model is as shown in Fig. 2a. This model is almost identical to the simplest Strut-and-Tie models manually drawn for this problem in [1] and [4].

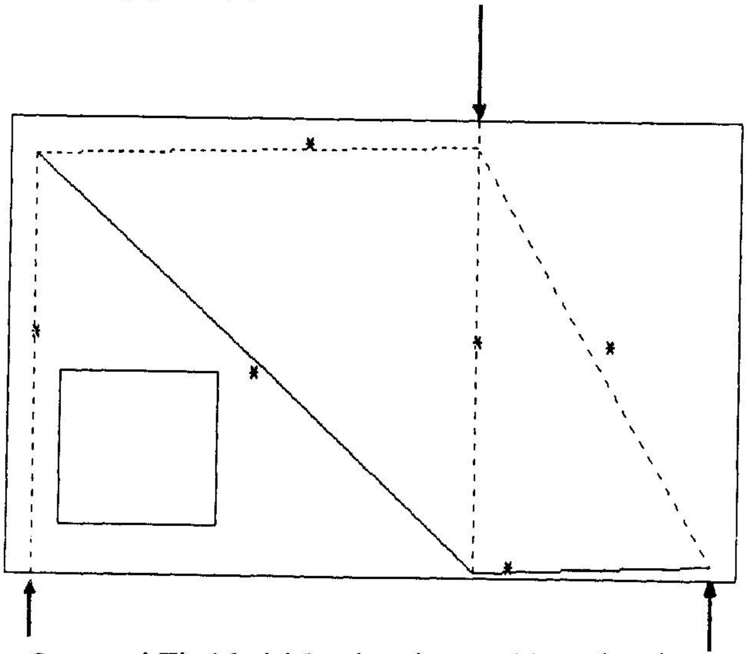


Fig.2 Strut-and-Tie Model for deep beam with perforation

In the third problem, the primal group of principal compressive stresses was chosen as in the previous two problems. A similar selection was made for the group of principal tensile stresses. The partitioning user had specified each of these groups into a maximum number of 4 subgroups of the type of step 2) of the algorithm. This latter step produced only 2 such subgroups for each type of stresses, which were further partitioned into 3 subgroups of neighboring points in Step 3) of the algorithm. The final Strut-and-Tie model, shown in Fig. 3a, is stable only for the specific loads of the problem. The same holds for the more refined Strut-and-Tie model proposed for this case in [1]. So, this Strut-and-Tie model can be analysed only by equilibrium of the nodes, and not by a general purpose computer program based on the Direct Stiffness Displacement approach. To make the truss stable, the user should insert additional Struts-and-Ties to the model. An extension of the present algorithm to automatically augment the Strut-and-Tie model in such cases is currently under development.

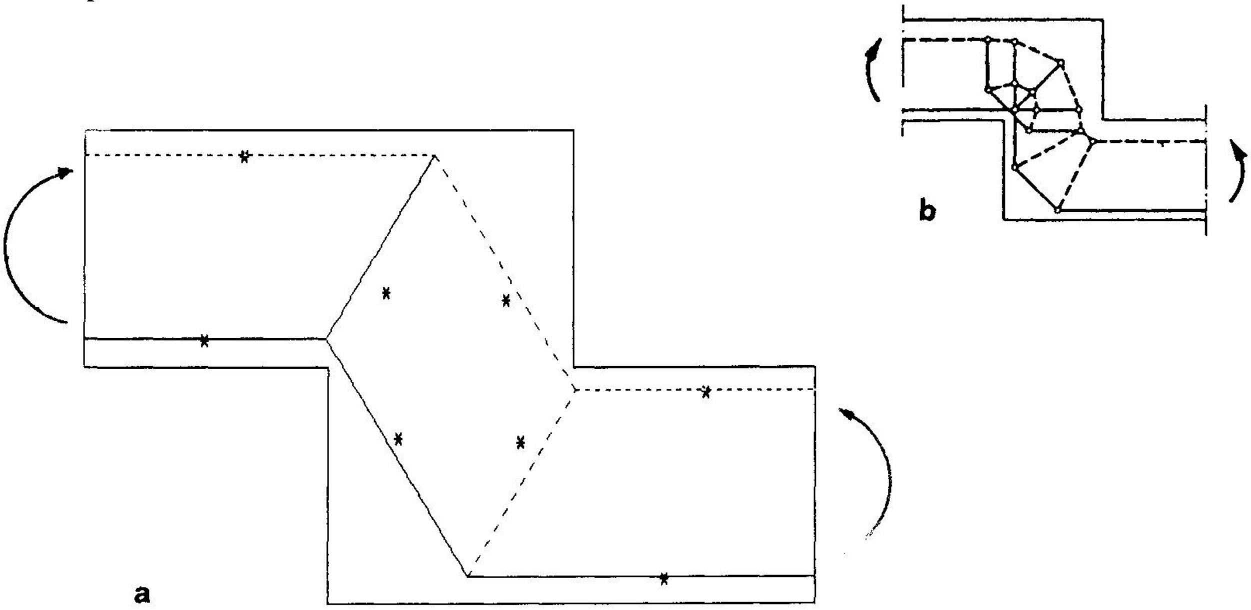


Fig.3 a) Strut-and-Tie Model for stepped beam; b) Strut-and-Tie Model after [1].

The automatic construction of the Strut-and-Tie model in each one of the 3 examples herein required less than 1 min. of computer time on a PC/AT. (To be compared to the few minutes required for the Finite Element Analysis of each problem on the same computer). This time is relatively short, allowing the engineer to try various alternative selections of user-specified parameters in order to improve or refine the Strut-and-Tie model.

REFERENCES

1. SCHLAICH J., SCHÄFER K., and JENNEWEIN M., "Toward a Consistent Design of Structural Concrete", PCI Journal, 32, No 3, 1987, 74-150.

2. BREEN J.E., "Why Structural Concrete", Introductory Report, IABSE Colloquium "Structural Concrete", Stuttgart, April 1991.

3. ANDERHEGEN E., and SCHLAICH M., "Computer-aided Design of Reinforced Concrete Structures Using the Truss Model Approach", Proc., 2nd Int. Conf. on Computer-Aided Analysis and Design of Reinforced Concrete Structures, Zell-am-See Austria, 539-550, Apr. 1990,

4. SCHLAICH M., and ANAGNOSTOU G., "Stress Fields for the Nodes of Strut-and-Tie Models", Jour. of Struct. Division, ASCE, V. 116, No. 1, Jan. 1990, 13-23.

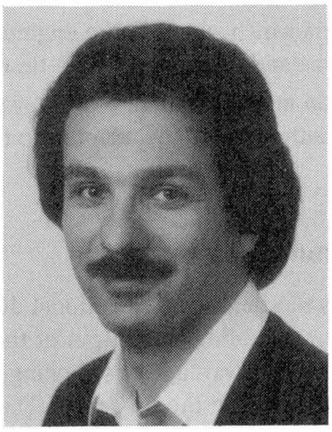
Leere Seite
Blank page
Page vide

Design and Analysis with Strut-and-Tie-Models – Computer-Aided Methods

Méthodes assistées par ordinateur pour la conception basée sur l'analogie du treillis

Methode der Stabwerkmodelle: Umsetzung in einem Computerprogramm

Klaus J. RÜCKERT
Dr. Eng.
Schlaich Bergermann & Partner
Stuttgart, Germany



Klaus J. Rückert, born 1955, obtained his degree at the University of Stuttgart and at UC Berkeley, USA. He was employed in consulting offices in South Africa and San Francisco, USA before he joined the Institute for Structural Design at the University of Stuttgart. He is now a structural engineer with Schlaich, Bergermann & Partner in Stuttgart.

SUMMARY

The design method for strut-and-tie models was implemented in a computer program both to increase user-friendliness and to use the graphical and numerical capabilities of the computer. Therefore, a program system was developed which consists of several independent modules, which are linked through interprocess-communication and which run under a uniform user interface. The strut-and-tie model method was extended especially in the fields of: Modelling (strut-and-tie models can automatically be constructed out of trajectory fields) and Analysis (the use of a nonlinear analysis program together with nonlinear material laws and an algorithm for optimization, allows to calculate forces and displacements for the design as well as for the serviceability state).

RÉSUMÉ

La méthode de conception qui se base sur l'analogie du treillis a été traduite par un programme d'ordinateur afin d'augmenter le confort de l'utilisateur et de profiter des capacités graphiques et numériques de la machine. Le programme qui a été développé dans ce but comprend plusieurs modules indépendants liés à une méthode de communication interactive faisant l'interface avec l'utilisateur. L'analogie du treillis a été spécialement étendue dans les domaines de la modélisation (à partir du champs des trajectoires, on peut construire automatiquement des modèles de treillis) et de l'analyse (un programme d'analyse non-linéaire permet, conjointement aux lois non-linéaires de comportement des matériaux et d'un algorithme d'optimisation, de calculer forces et déplacement nécessaires au projet, aussi bien que pour l'évaluation de l'état de service).

ZUSAMMENFASSUNG

Die Methode der Stabwerkmodelle wurde in ein Computerprogramm umgesetzt, um die Benutzerfreundlichkeit zu erhöhen und die grafischen und numerischen Möglichkeiten des Rechners auszunützen. Dafür wurde ein Programmsystem entwickelt, das aus mehreren unabhängigen Modulen besteht, die durch Interprozess-Kommunikation gekoppelt sind und unter einer einheitlichen CAD Benutzeroberfläche ablaufen. Die Methode der Stabwerkmodelle wurde insbesondere erweitert auf den Gebieten der Modellfindung (aus Trajektorienfeldern können automatisch Stabwerkmodelle abgeleitet werden) und der Berechnung (durch Verwendung eines nichtlinearen Rechenprogrammes zusammen mit nichtlinearen Materialgesetzen und einem Optimierungsalgorhythmus können realistische Kräfte und Verformungen sowohl für die Bemessung als auch für den Gebrauchszustand berechnet werden).

1. Introduction

During the last years the strut-and-tie model (STM) design was developed as a method to unify the design of structural concrete for all kinds of concrete members and details /1,2,3/. One can consider this method as a combination of graphical and analytical techniques which had traditionally been applied 'by hand'. As the development and application of hard- and software has proceeded rapidly in the last years and a computer is on nearly every desk, it became clear that this method should also benefit from the graphical and numerical capabilities of the computer /7,9/ since a programm based on a consistent design concept would be more logical for a CAD program than the effort to program only codes. Developments took place at different locations in various directions, i.e. based on theory of plasticity /5/.

The goal of bringing the strut-and-tie models 'onto the computer' included the following two tasks:

- Development of a program system which supports the engineer in an easy to use, graphical manner and allows to display, input, edit, analyze and design strut-and-tie models on a workstation.
- Enhancement of the strut-and-tie model method, especially in the fields of model finding and calculation of forces and displacements. The method had to be adapted to the computer, but this yielded also new possibilities.

2. Development of the Program System

The overall design process includes the fields of conceptional design, idealization of the structural model, analysis, design of the structural members, detailing and output of the drawings. Specialized programs exist for each of this fields with the user having the disadvantage of switching between different programs and the difficulty of transferring and translating the data between the programs. There was no integrated system which supports not only single steps but the whole process. To fulfil this requirement the developed integrated system needed the following modules within 'one' program:

- A CAD program to draw and edit the models and drawings and to display the results of the calculations.
- A FE program to analyze the structures.
- Programs to design the structural members with the STM method.
- A database to save all input and calculated data for easy access through the other modules.
- A interactive graphical user interface and control program which allows ease of use, portability and expandability.

An 'automatic' program was surely not the goal, as '.. it is imperative that an experienced and qualified engineer be involved in the interpretation of the results using their knowledge of structural behavior ..' /9/. Therefore the program was developed as a toolbox or method-base of application programs which is used similar to a database. The user communicates with the program through a uniform user interface (the CAD program) and can choose among several ways (simple to more demanding) and modules according to the specific needs of the problem. The results of each step can be checked as they are immediately displayed on the screen and if needed can be calculated repeatedly. The user must not be concerned about the compatibility of the program modules and the integrity and transfer of the data as the program system takes care of this.

It was unreasonable to combine everything in one program because of program size, complexity and expandability with other modules. Therefore programming interfaces were developed for graphics, database and interprocess communication (IPC). All stand-alone programs (CAD, FE, design, ...) were only extended with these interfaces. The CAD program with its graphical capabilities for input, editing and output was used as the main and control program. The others, which can run invisible in different processes or windows, were linked to it through IPC and can exchange commands and data very quickly. By this way the programs can be developed (and used) separately and new or improved modules can be implemented easily. This structure is invisible to the user as he has the impression of only one single menudriven program. An overview of the program system is shown in Fig: 1 .

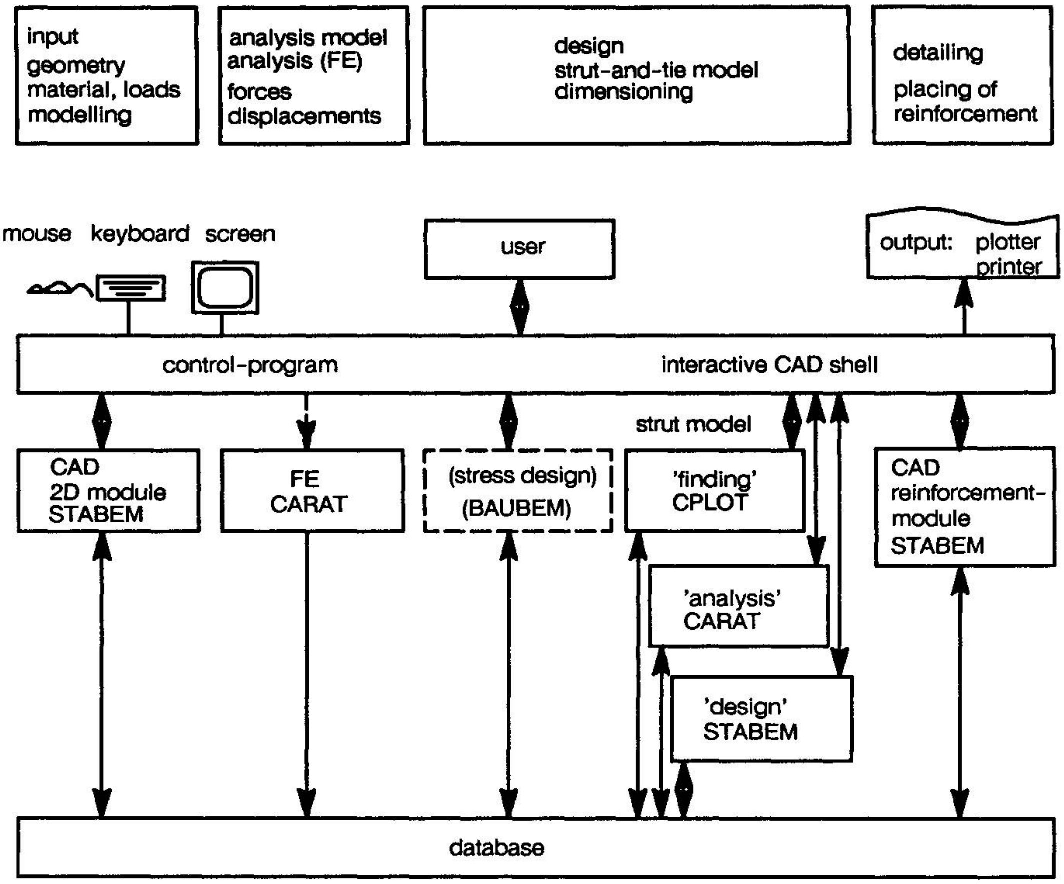


Fig. 1: Structogram of the program system

3. Strut-and-tie model design

3.1. Elements

The struts as basic elements represent both the resultant forces and the corresponding stress fields. In the design program they are therefore defined as truss elements with varying shapes (Fig: 2).

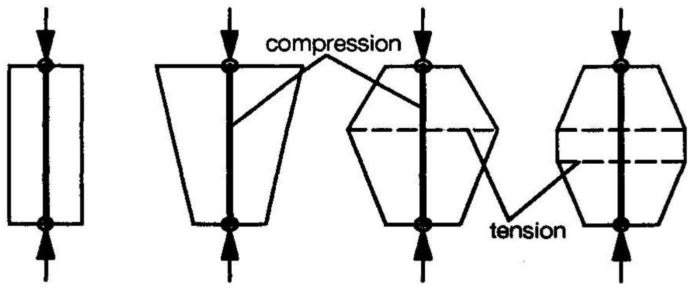


Fig. 2: Elements

The struts have dimensions according to the thickness of the structure and the width of the stress fields. This together with realistic material laws for concrete (compression and tension) and reinforced concrete (tension) allows the calculation of the nonlinear behavior (stress, strain, displacement, energy) of struts and ties.

3.2. Modelling

The finding of a suitable model is one of the most important points for the design as the model has to represent the actual flow of forces in the structure and is the starting point for all following calculations.

The simple modelling methods:
- 'free hand' drawing of simple models
- adaptation of known typical models to a specific case

are supported by the computer through the graphical drawing and editing possibilities and a 'database' of known models.

A simple preliminary elastic FE analysis of the structure helps the modelling methods:
- orientation of the model at the linear theory of elasticity,
- finding of the 'load path '

by underlaying the principle stress fields and possible calculations of stress resultants anywhere in the structure.

The most interesting and newly developed part is the model finding with trajectory fields. Trajectories can be generated at a desired density (which can but need not increase accuracy) and automatically be transformed into a strut-and-tie model (Fig:3). This represents a model which is closely oriented at the theory of elasticity and is inherently in a state of equilibrium.

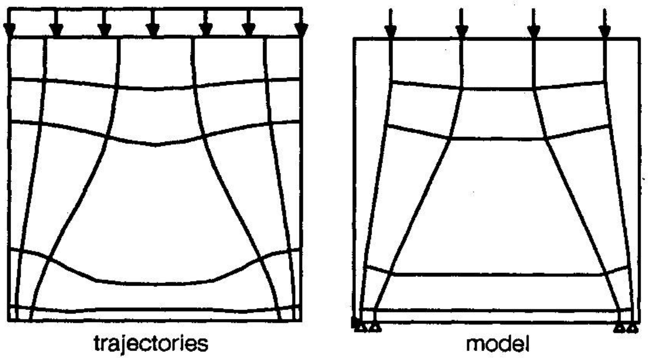


Fig. 3 : Strut model out of trajectories

The net of trajectories is equivalent to a 'shear free' net of (FE–) elements which have only biaxial normal stresses. It is possible to calculate the forces in a 'strip' bordered by two trajectories (Fig: 4). Therewith the 'flow' of forces in the structure (e.g. from the load to the support and perpendicular) becomes visible. These forces and their direction in the continuum are then equivalent to the forces and directions of the struts resp. stress fields and therewith the analogy between continuum and strut model becomes obvious.

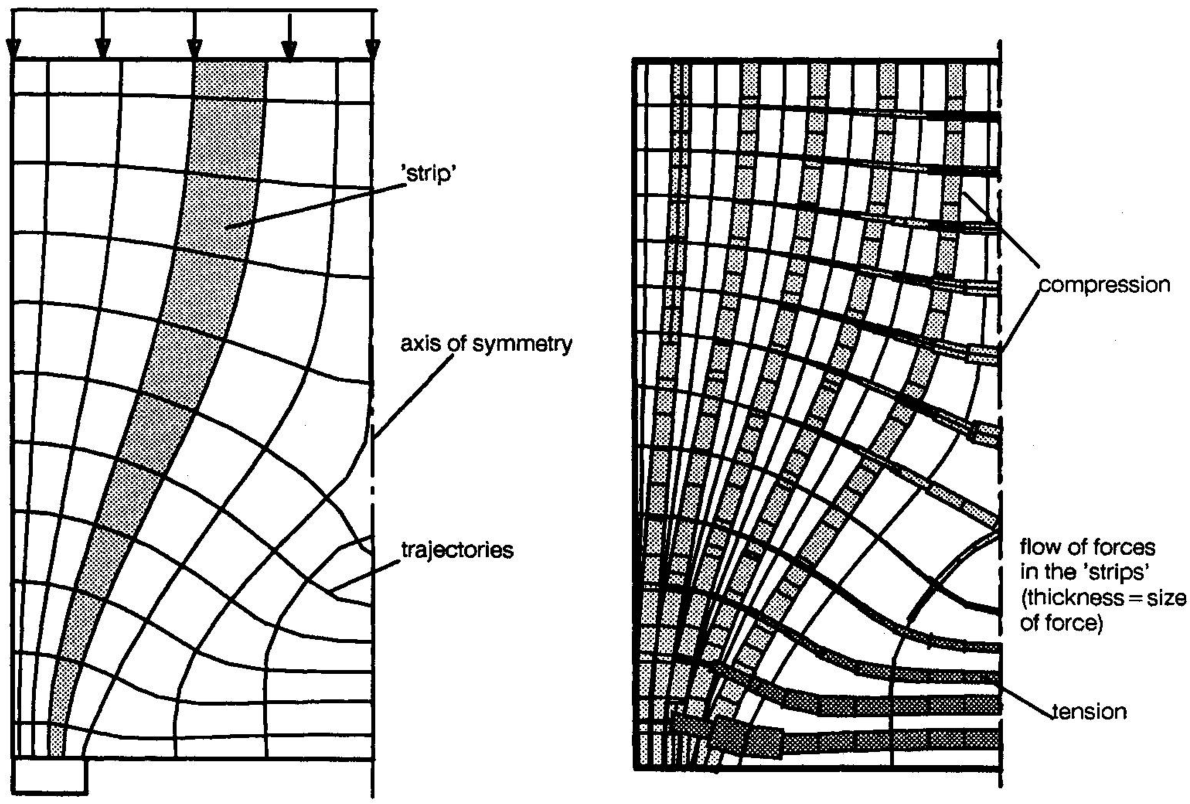


Fig. 4 : 'Trjectories and flow of forces'

The 'density' of the model is up to the designer and depends on the design stage /9/. Often it is sufficient for design purposes to use a simple model (Fig: 3) and leave the less important trajectories out. The program provides numerous possibilities to edit the automatically created model and to adapt it to the specific needs, as for example given positions and directions of reinforcement.

3.3. Analysis

For statically determinate and indeterminate models it is difficult to verify that they represent the load bearing of the structure as the geometry can be (almost) arbitrarily chosen. For kinematic models there is (for a given load) only one stable geometry which then reflects the load bearing.

The difficulty of analyzing a kinematic model was solved by using a geometric nonlinear program which was extended with initial stiffnesses to calculate 'kinematic cable structures' and which can therefore also be used to find a stable geometry (Fig: 5). If the model is generated through trajectory fields, the advantage is that it is already in a stable position and also represents the actual flow of forces.

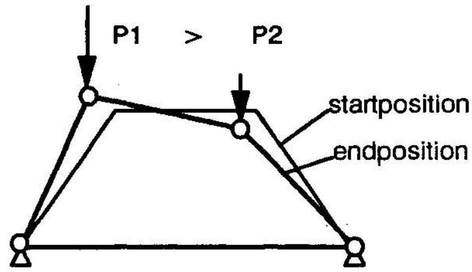


Fig. 5 : Analysis

The dimensioning of concrete and reinforcement is a relatively simple task and could for every model also be done by the theory of plasticity /4,5/. A model which is oriented at the theory of elasticity is both well suited for the design of struts and ties, as the calculated amount of reinforcement and concrete stresses are on the safe side according to the lower theorem of plasticity, and the requirements for compatibility and serviceability are also approximately fulfilled.

If however, one wants to know the state of stress or the displacements for any loading stage from cracking up to ultimate load, the geometry of the model has to be adapted to the load bearing behavior resp. 'load path' of this state. The struts must also have the according properties (width, material, etc.). The implemented strut elements with dimensions corresponding to the stress fields and nonlinear material laws allow the calculation of forces and stresses as well as displacements (Fig: 6)

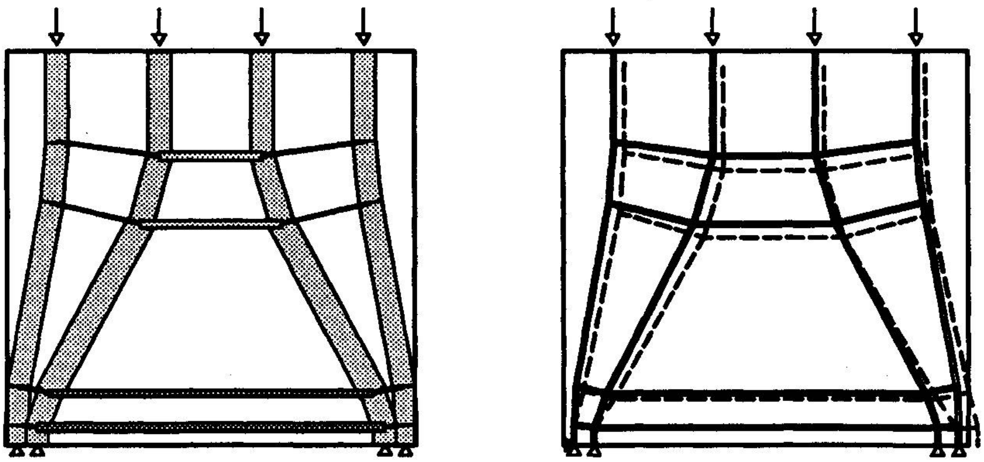


Fig. 6 : Forces and displacements

This is done with the criterion of optimizing/minimizing the internal energy of the total system as it tends to undergo the smallest possible stresses and strains. This together with an iterative nonlinear analysis allows to adapt the model, i.e. find the right geometry, for increasing loads as shown in Fig: 7 . Comparisons with tests show satisfying results.
This is a tool for the experienced engineer to achieve results for the design and its verfication in a rather simple and quick way. An additional nonlinear FE analysis for the verification (but not for the design itself) could still be done, but to achieve possibly better results it has to be much more elaborate.

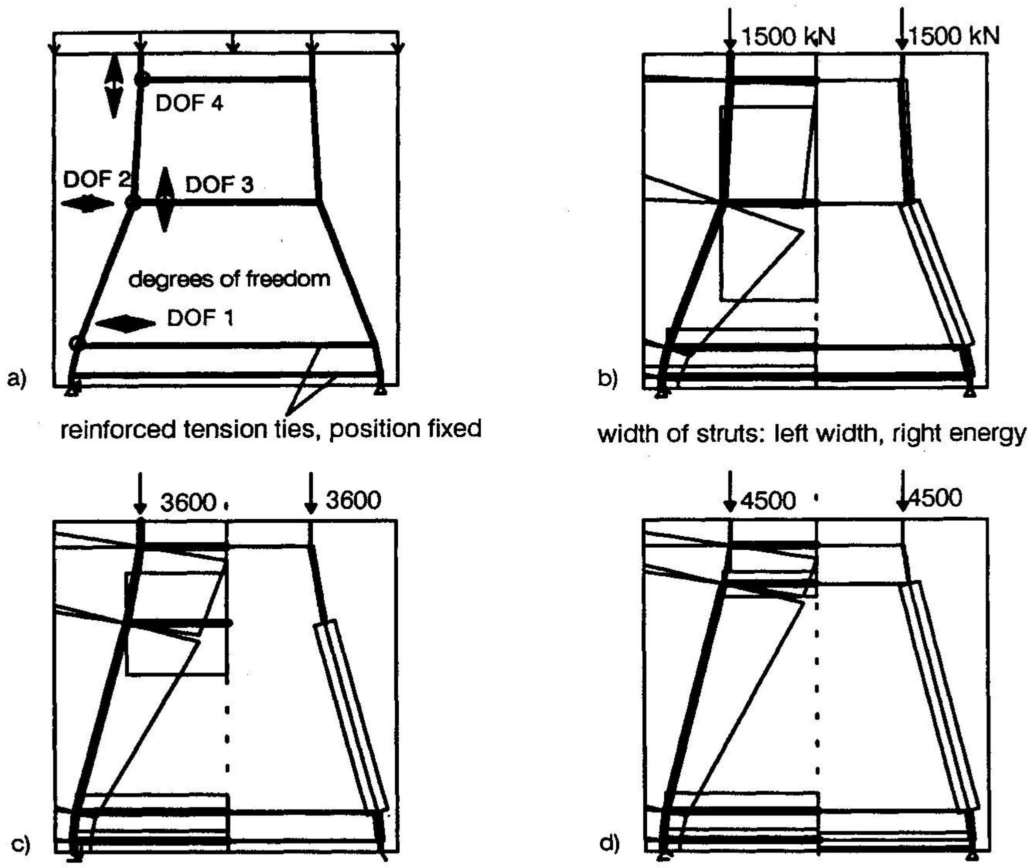


Fig. 7 : Adaptation of model under increasing load

References

1. SCHLAICH J., WEISCHEDE D., A practical method for the design and detailing of structural concrete. Bulletin d'Information No. 150. CEB, Paris, March 1982.
2. SCHLAICH J., SCHÄFER K., JENNEWEIN M., Toward a consistent design of structural concrete. PCI Journal Vol. 32 No. 31, May/June 1987.
3. SCHLAICH J., SCHÄFER K., Konstruieren im Stahlbetonbau. Betonkalender 1987.
4. MARTI P., Truss Models in Detailing, Concrete International, Vol. 7, No. 12, Dec. 1985.
5. SCHLAICH M., Computerunterstützte Bemessung von Stahlbetonscheiben mit Fachwerkmodellen, Bericht 1, Professur f. Informatik ETH Zürich, 1989.
6. RÜCKERT K., SCHREIBER V., Baucad Handbuch (user manual). Institut für Tragwerksentwurf und -konstruktion, Universität Stuttgart, 1989.
7. BREEN J., Why structural concrete. Introductory lecture, IABSE Colloquium 'Structural Concrete', Stuttgart April 1991.
8. SCHLAICH J., 'Modelling'. Introductory lecture, IABSE Colloquium 'Structural Concrete', Stuttgart April 1991.
9. SCORDELIS A., Analysis of Structural Concrete Systems. Introductory lecture, IABSE Colloquium 'Structural Concrete', Stuttgart April 1991.
10. SCHLAICH J., SCHÄFER K., JENNEWEIN M., Strut-and-Tie Modelling of Structural Concrete. IABSE Colloquium 'Structural Concrete', Stuttgart April 1991.

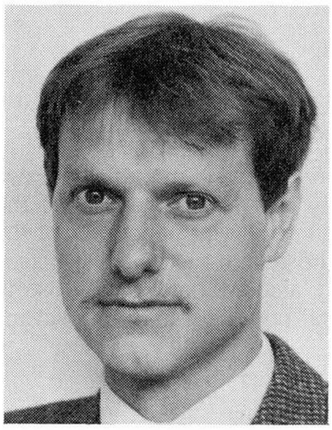
Nonlinear Behaviour of Deep Beams

Comportement non-linéaire d'un élément porteur de type cloison

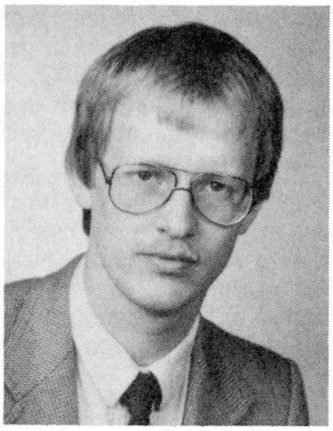
Nichtlineares Tragverhalten von Scheiben

Wolfgang SUNDERMANN
Dipl.-Ing.
Univ. of Stuttgart
Stuttgart, Germany

Peter MUTSCHER
Dipl.-Ing.
Univ. of Stuttgart
Stuttgart, Germany



Wolfgang Sundermann, born in 1957 in Wuppertal, studied at the RWTH Aachen and subsequently worked for 1½ years in a company. Since 1985 he is a Research Assistant at the Inst. for Structural Design, Univ. of Stuttgart.



Peter Mutscher, born in 1960 in Solingen, studied at the University of Wuppertal. Since 1986 he is a Research Assistant at the Institute for Structural Design, University of Stuttgart.

SUMMARY

This paper deals with an analytical method, which is an extension of the method of Strut-and-Tie-Models. It enables the design engineer to calculate the nonlinear behaviour of deep beams in a simple way. It yields approximate values for the ultimate load, the support reactions of statical indeterminate systems, the important stresses and strains and the load deflection behaviour, respectively. Additionally, a simultaneous control of locally high stressed regions, e.g. near supports or loadings, is possible. The results of this analysis can be used also as an independent verification of the results of a nonlinear finite analysis.

RÉSUMÉ

On présente dans ce rapport une méthode dérivée de la modélisation par bielles (analogie du treillis) qui permet à l'ingénieur de calculer simplement le comportement non-linéaire d'éléments porteurs de type cloison. On obtient ainsi des valeurs approchées de la charge ultime, des réactions d'appui de systèmes hyperstatiques, des contraintes et des déformations déterminantes ainsi que des diagrammes force-déplacement. Le contrôle de régions particulièrement sollicitées, comme le voisinage des appuis ou le point d'application d'une force concentrée est de plus rendu possible. Les résultats obtenus par cette méthode peuvent être employés pour vérifier une analyse non-linéaire par éléments finis.

ZUSAMMENFASSUNG

In dieser Abhandlung wird ein Verfahren vorgestellt, das eine Erweiterung der Methode der Stabwerkmodelle darstellt und dem entwerfenden Ingenieur ermöglicht, das nichtlineare Tragverhalten seines Tragwerks vereinfacht zu untersuchen. Dabei können z.B. die tatsächlich erreichbare Bruchlast, Auflagerkräfte statisch unbestimmter Systeme, massgebende Spannungen, Dehnungen und Last-Verschiebungskurven ermittelt werden. Zusätzlich ist eine Kontrolle lokal hoch beanspruchter Knotenbereiche, z.B. im Auflager- oder Krafteinleitungsbereich, möglich. Das Verfahren kann z.B. auch als ingenieurmässiges unabhängiges Kontrollinstrument für eine nichtlineare FEA verwendet werden.

1. INTRODUCTION

For the analysis and dimensioning of deep beams of reinforced concrete, especially for statical indeterminate structures, the theory of elasticity still is the main basis. However, the real behaviour of these structures under increasing load is determined by the nonlinear behaviour of the materials. This has a major influence on the real ultimate load capacity and the load-deflection-response, respectively.

Up to now the calculation of such effects is only possible with the Nonlinear Finite Element Analysis (NLFEA) and therefore requires an enormous amount of computing. Moreover the results are very difficult to check. Additionally the sophisticated calculation of stresses and strains in every point of the structure is often of no interest (/1/).

This paper presents a practical tool, which enables the design engineer to analyse his concrete structure with respect to some interesting points, e.g. (s. also Fig.1):

- Magnitude of the real ultimate load after the mobilizing of all the bearing capacities within the structure;
- Determination of the structural elements which are probably responsible for the failure of the whole structure, e.g. node regions, parts of reinforcement etc.;
- Load-Deflection-Behaviour with increasing load;
- Distribution of the support reactions in statical indeterminate structures;
- Sensitivity of a structure to restraint;
- Response of the structure to variations in amount and arrangement of reinforcement (e.g. partially prestressed ties, redistribution of reinforcement between span and support-regions etc.).

The presented method is an extension of the Strut-and-Tie-Model (STM) analysis and therefore draws attention to those elements, which mainly influence the load bearing capacity and the behaviour of the structure. The assumptions for the numerical calculation, e.g. for the strength and nonlinear behaviour of the materials, are always present because of the small number of elements. So the response of the structure remains transparent and the results are easy to control. Therefore this tool may also become a very useful educational aid (/3/). Because of the permanent check of the highly stressed local regions this tool performs both at global as well as at local level, which is strictly demanded in /4/.

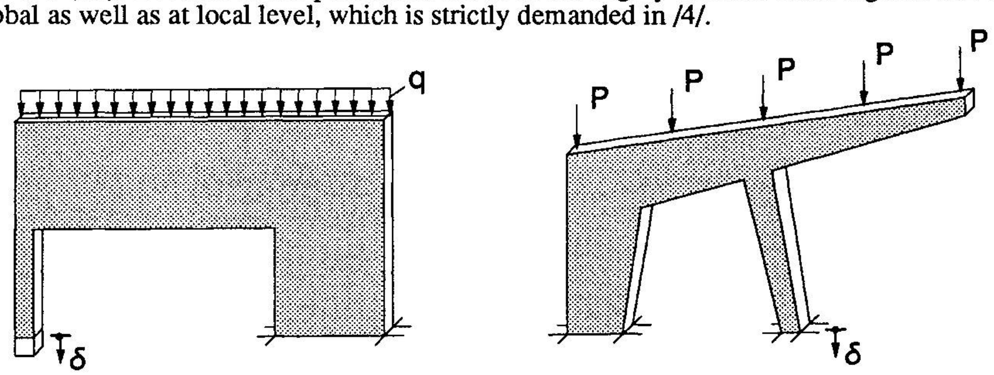


Fig.1 Two typical deep beams, for which a study of the load carrying behaviour due to load (q and P) and settlement (δ) may lead to a better estimation of the real structural safety.

2. FORMULATION OF THE METHOD

The **Flow of Forces** in the structure can be simply modeled either by hand ("Load Path Method", /2/) or by a more refined analysis of the structure with a Linear Finite Element Analysis (/5/). The result is a STM with struts, reinforced ties and nodes, s. Fig 2(a).

The **Strut-and-Tie-Net** separates the predominantly one-dimensional stress fields of the struts and ties from the two-dimensional stress fields of the nodes. With the fixed netpoints and the assumption of some effective widths of the struts, s. fig 2(b), the geometry of the Strut-and-Tie-Net can be found as a result of the current geometry of the STM, s. fig 2(c). Now the effective widths of the struts and ties and consequently their stresses, which govern the capacity and behaviour of these elements, can be computed.

The bearing capacity of the individual **Ties** is determined by the amount and strength of the reinforcement. The nonlinear behaviour can be calculated either according to the well-known regulations in codes (e.g. MC 90, EC 2 etc.) or according to appropriate publications. In this paper a simple formulation is used, which is an extension of the tension stiffening relations given in /7/. With its help the number of cracks, the crack-width and the average deformation of the ties can be calculated. Additionally an early state of cracking can be assumed. The different kinds of reinforcing-steel (e.g. in a partially prestressed ties) can be taken into account by their stress-strain-curves including the strain hardening, s. fig. 3(a).

The capacity of the **Struts** is governed by the stresses at the borderline between the node and strut regions.

For simplification the stresses are calculated in the so called "Transition-point", s. fig. 4. The description of the nonlinear behaviour follows a modified rule in the MC 90 and the DIN 1056, respectively. With the help of 3 factors, which determine the actual strength (α_f), the tangent modulus at the origin (α_E) and the strain reached at ultimate stress (α_ε) a wide range of nonlinear response of the struts can be covered, s. fig. 3(b). The values of these factors depend e.g. on the geometrical form of the struts (fan- or bottle-shaped) and their structural design (longitudinal or transversal reinforcement).

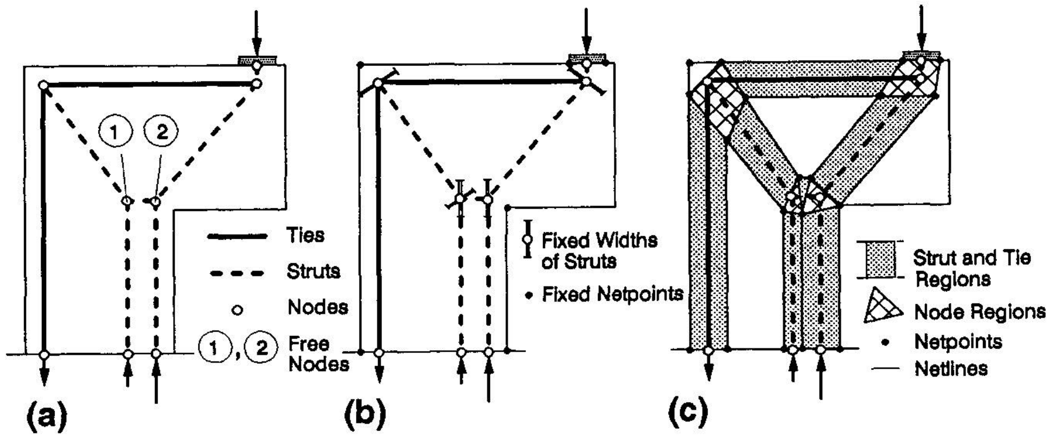


Fig.2 Evaluation of the Strut-and-Tie-Net demonstrated for a cantilever wall:
(a) The structure with its borderline, external forces and the STM
(b) Determination of the fixed netpoints and some widths of struts
(c) Complete Strut-and-Tie-Net with the effective idealized fields of struts, ties and nodes

The bearing capacity of the **Nodes** is also determined by the stresses in the "Transition Point", s. fig. 4. It depends on the kind of node (e.g. pure compression- or bond-node) and its detailing (e.g. kind of anchorage, amount of transverse reinforcement etc.) and is described by a strength factor (α_n), too.

The **time dependent behaviour** is determined by the creep coefficient, which can be adopted from code instructions. For this method the behaviour of struts and ties is considered differently: the struts have a creep strain, whereas the cracked ties have a time dependent decrease of the bond-stresses.

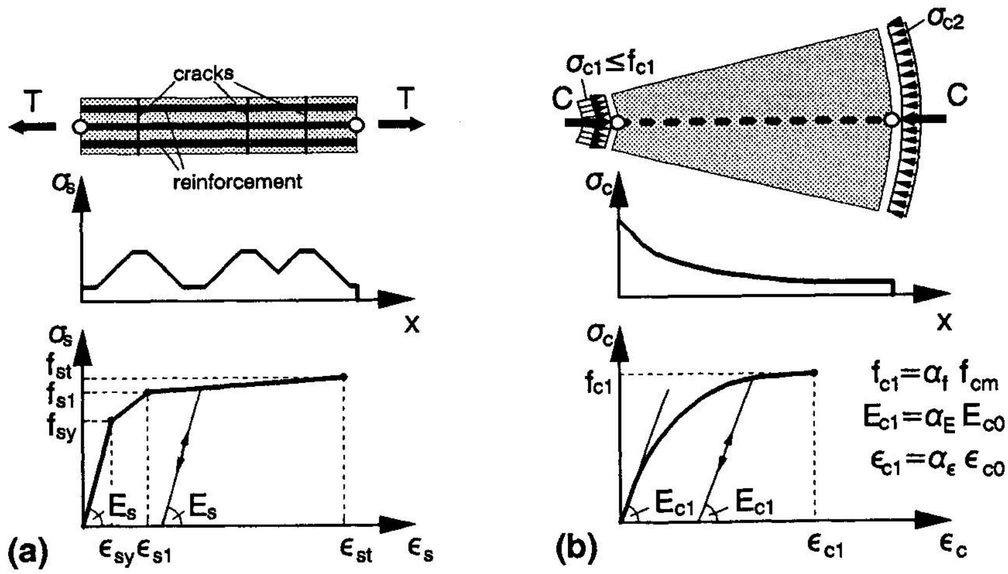


Fig.3 Calculation of the nonlinear behaviour of Ties (a) and Struts (b)

The positions of the **Free Nodes** are determined by applying an energy-criterion. This yields the best simulation of the real load bearing behaviour regarding to the accuracy of the chosen STM. The applied energy-criterion of the extreme value of the overall potential is computed by the internal deformation energy and the potential of external forces and support settlements. The compatibility of this equilibrium state is thus automatically guaranteed.

For the **numerical calculation** the loading, settlements and creep coefficients can be increased step by step, whereby the best position of the free nodes is computed in every loading step. If some struts or ties start to fail near the ultimate load, the increments are reduced to 1/5 of the original value. This ensures a sufficient redistribution of the inner forces. The results of the computation are directly presented in diagrams, which allow a fast and engineering analysis of the structure (e.g. load deflection curves, important strains and stresses, crack-widths, support reactions etc.).

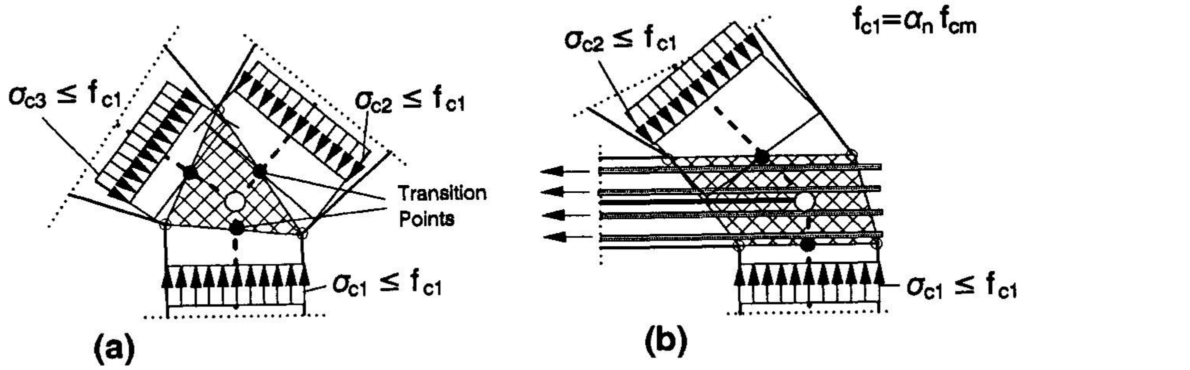
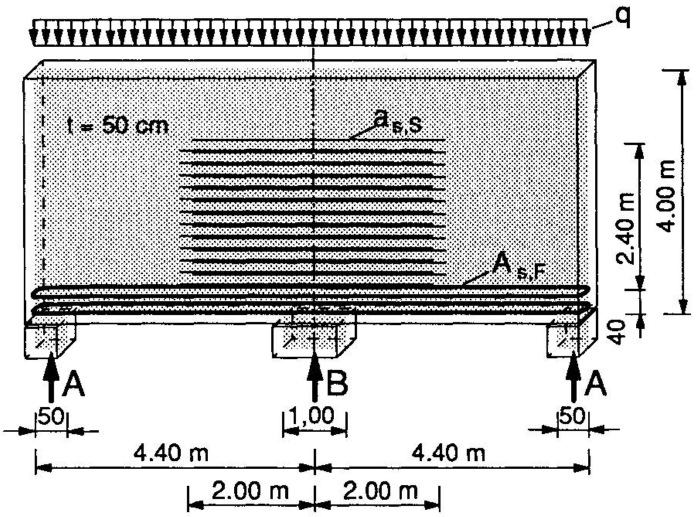


Fig. 4 Two examples for node regions and their governing stresses
(a) Pure compression node (CCC)
(b) Node with deviation of struts due to the anchorage of a tie (CTC)

3. ANALYSIS OF AN EXAMPLE

In fig. 5 a deep beam with 2 spans is shown with its geometry, loading and two cases of reinforcement. In case 1 the reinforcement is chosen according to a linear-elastic analysis. In case 2 the amount of the span-reinforcement is substantially increased, while simultaneously the reinforcement at support is reduced as against case 1. The load carrying behaviour for both cases now will be examined with the above presented method.



Concrete: f_{cm} = 36.0 MN/m^2
f_{ctm} = 3.20 MN/m^2
E_{co} = 37,000 MN/m^2
ε_{co} = 2.2 %.

Reinforcement:
f_{sy} = 500 MN/m^2
f_{st} = 550 MN/m^2
E_s = 210,000 MN/m^2

Case 1: Reinforcement due to linear-elastic analysis
$A_{s,F}$ = 8 ϕ 20 = 25.12 cm^2
$a_{s,S}$ = ϕ 12/20 cm = 11.3 cm^2/m

Case 2: Increased span reinforcement
$A_{s,F}$ = 12 ϕ 25 = 58.92 cm^2
$a_{s,S}$ = ϕ 10/25 cm = 6.28 cm^2/m

Loading: q = 1.0 MN/m (service load)

Fig.5 Geometry, reinforcement and loading of the example

The structure is modeled with a very simple STM, s. fig. 6(a), which was found by the "Load Path Method". The geometry of this STM determines the initial position of the free nodes for both cases. In order to leave open the relations of the support reactions (**A** and **B**) and also the internal lever arm in the span (distance between tie 1 and strut 13) the free nodes 7 to 10 can vary in any direction. To allow a free distribution of the tie forces 7 and 17, the nodes 5 and 6 can vary in their x-coordinates. Regarding the symmetry and the internal coupling of nodes there are altogether 3 degrees of freedom for the whole system. The ultimate stresses at the supports are set to $\mathbf{f_{c1} = 1.20\ f_{cm}}$ ($\mathbf{\alpha_n=1.20}$) because of the very proper node design with loops, while all other factors (α_f, α_E and α_ε) are simply set to **1.0**. To avoid a contribution of the concrete tensile strength over the middle support, the Ties 7 and 17 are assumed to be precracked with one crack, whereby the tie 17 is assumed to be reinforced like tie 7. The load increment is Δq = 0.50 MN/m.

In the first load step (q = **0.50 MN/m**) the structure behaves in a linear-elastic mode. The position of the free nodes, calculated by the energy-criterion, lead to the geometry of the STM shown in fig. 6(b). The internal lever arm and the distribution of stresses over the middle support agree very well with a linear-elastic FEA.

With increasing load the geometry of the STM changes, which is caused by a redistribution of the inner forces. The internal lever arm increases in both cases, until the geometries for the STMs at the ultimate loads are obtained, see fig. 6(c). In case 1 the ultimate load is **q_u = 3.60 MN/m** and the failure of the structure is initiated by a simultaneous failure of the reinforcement in the span and at support. In case 2 the ultimate load is **q_u = 4.60 MN/m** and the failure of the structure is caused by a failure of the nodes at the supports, which have nearly all the same pressures at the ultimate load.

The relation of **B/A**, see fig 6(d), shows a distinct increase for the wall in case 1. This is due to a redistribution of forces from the span to the support region because of the relatively weak stiffness of the tie in the span. In case 2 the ratio B/A remains nearly constant, it even deminishes nearby the ultimate load.

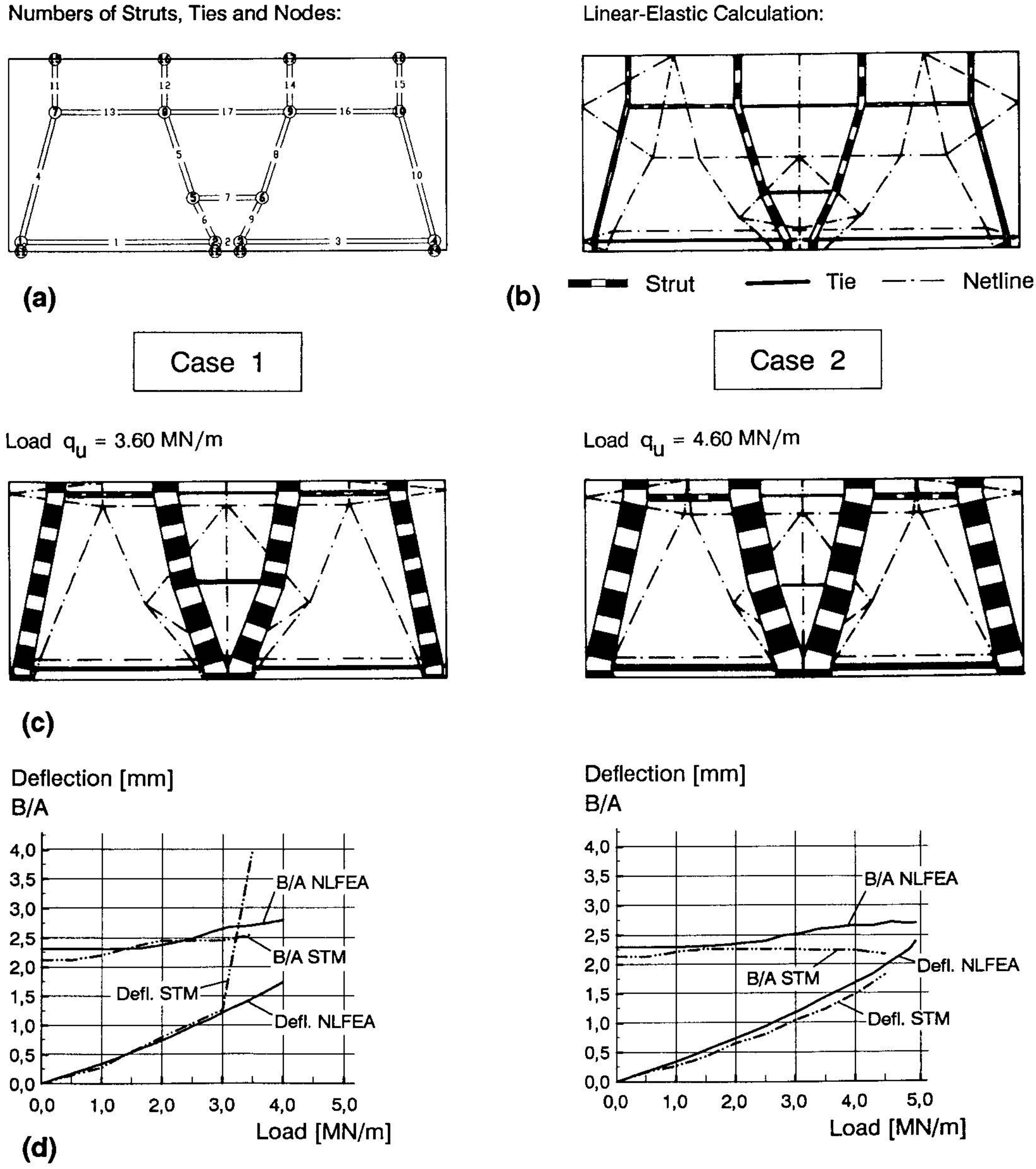


Fig. 6 Response of the structure

The **load-deflection-curves** in fig. 6(d) show a quite linear increase of the deflections because of the overall stiffness of the structure. Only in case 1 the deflection rapidly increases shortly before the ultimate load is reached. This is due to the yielding of the span- and support-reinforcement.

Additionally a NLFEA was carried out using the program SBETA described in /8/. The aim was to proof the reliability of the demonstrated STM method. Some results are shown in the diagrams (s. fig. 6(d)) in comparison to the calculations with the STMs. The ultimate loads of the NLFEA (q_u = **4.10 MN/m** for case 1 and q_u = **5.20 MN/m** for case 2) are slightly higher than the values of the STMs. In case 1 the increase of **B/A** occurs at higher loads in the NLFEA than in the STMs. This is due to the uncracked state, which is conserved up to higher loads in the NLFEA. In case 2 the relation B/A is higher in the NLFEA, which is also caused by uncracked areas especially at the top region over the middle support. The **load-deflection-curves** are in good agreement, apart from the case 1, where a difference occures near the ultimate load, because of a greater stiffness of the wall in the NLFEA. A more refined STM would of course improve the simulation of this deep beam. More details about the non-linear calculations with STM and the NLFEA including further examples can be found in /6/.

4. CONCLUSIONS

It could be demonstrated that the overall load carrying behaviour is simulated quite well with the presented tool. The discussed method especially provides the following advantages:

- The flow of forces remains transparent throughout the whole nonlinear analysis because of the small number of load bearing elements.
- The capacity of locally high stressed regions can be adapted separately according to their structural design.
- The whole numerical analysis runs on a PC with small equipment and takes only a very short time.
- This tool enables the engineer to analyse the behaviour of his structure under varying boundary conditions (e.g. with or without concrete tensile strength, with or without creep effects etc.).
- An adaptation of this model to future code regulations is easily possible.

When applying this method, the following points should be paid attention to:

- The chosen STM must be able to follow all the possible internal redistributions of the forces.
- The method cannot cover all structural effects, especially in the uncracked state, because of the simple modelling.

This practical tool promotes a very good understanding of the whole structure and the interaction of its components. This leads to a better design and to a more reliable estimation of the safety of the analysed structure.

REFERENCES

/1/ SCHLAICH, J., On the Relevance of Sophisticated Modelling Techniques on the Design of Real Concrete Structures. Proceedings of the 2nd International Conference of Computer Aided Analysis and Design of Concrete Structures, Zell am See, Vol.1, edited by N. Bicanic and H. Mang, 1990.

/2/ SCHLAICH, J., SCHÄFER, K., Konstruieren im Stahlbetonbau. Beitrag im Betonkalender 1989, Bd. II, Verlag Ernst & Sohn,Berlin.

/3/ BREEN, J.E., Why Structural Concrete. Introductory Report, IABSE Colloquium "Structural Concrete", Stuttgart, April 1991.

/4/ SCORDELIS, A., Analysis of Structural Concrete Systems. Introductory Report, IABSE Colloquium "Structural Concrete", Stuttgart, April 1991.

/5/ RÜCKERT, K., Computerunterstütztes Bemessen von Stahlbetonbauteilen mit Stabwerkmodellen, Thesis in Preparation, Institute for Structural Design, University of Stuttgart.

/6/ SUNDERMANN, W., Untersuchungen zur Tragfähigkeit und zum Verformungsverhalten von Stahlbetonscheiben mit Hilfe von Stabwerkmodellen. Thesis in Preparation, Institute for Structural Design, University of Stuttgart.

/7/ SCHOBER, H., Ein Modell zur Berechnung des Verbundes und der Risse im Stahl- und Spannbeton. Thesis at the Institute for Structural Design, University of Stuttgart, 1984.

/8/ CERVENKA, V., ELIGEHAUSEN, R., PUKL, R., SBETA Computer Program For Nonlinear Finite Element Analysis of Reinforced Concrete Structures. IWB Mitteilungen 1990/1, Institut für Werkstoffe im Bauwesen, University of Stuttgart, 1990.